Copyright Statement

The digital copy of this thesis is protected by the Copyright Act 1994 (New Zealand).

This thesis may be consulted by you, provided you comply with the provisions of the Act and the following conditions of use:

- Any use you make of these documents or images must be for research or private study purposes only, and you may not make them available to any other person.
- Authors control the copyright of their thesis. You will recognise the author's right to be identified as the author of this thesis, and due acknowledgement will be made to the author where appropriate.
- You will obtain the author's permission before publishing any material from their thesis.

To request permissions please use the Feedback form on our webpage. [http://researchspace.auckland.ac.nz/feedback](http://researchspace.auckland.ac.nz/feedback)

General copyright and disclaimer

In addition to the above conditions, authors give their consent for the digital copy of their work to be used subject to the conditions specified on the Library Thesis Consent Form.
Application of Computational Fluid Dynamics to Two-Dimensional Downwind Sail Flows

Stephen Collie
Departments of Mechanical Engineering and Engineering Science
School of Engineering
The University of Auckland
August 2006
Abstract

The research detailed in this thesis investigates the practical application of Computational Fluid Dynamics (CFD) to downwind sail design. Simulations were performed using CFX-5, an unstructured commercial CFD package. The research focuses on the performance of the SST and k-ω turbulence models which were judged to be CFX-5’s most appropriate turbulence models for downwind sail flows. Two-equation turbulence models are viewed as the most appropriate model type for sail simulations, they capture a significant amount of the flow physics whilst providing turnaround times for sail simulations of less than one day.

CFD simulations were compared with experimental data for a flat plate at shallow angles of incidence. This test case holds particular relevance to sail flows since both flows are affected by leading edge separation bubbles which form due to knife-edge separation at sharp leading edges. The CFD captures this leading edge bubble well, with the SST model predicting the length of the bubble with 7% of the experimental value.

Wind tunnel data was gathered for two-dimensional downwind sail sections for the purpose of CFD validation. A preliminary wind tunnel study was carried out using a low aspect ratio model. The tests were prone to three-dimensional effects and only three-dimensional CFD simulations were capable of successfully reproducing the flow. High aspect ratio wind tunnel test results were also conducted in an effort to obtain nominally two-dimensional wind tunnel data. Surface pressures were measured using Pressure Sensitive Paint (PSP), however due to the low dynamic pressure of the tests error appeared in the data and comparison with the CFD was poor. Results show that CFD is capable of qualitatively reproducing downwind sail flows, the leading and trailing edge separation regions were captured and the CFD results compared well with wind tunnel flow visualisation.

Finally, CFD simulations were used to investigate the two-dimensional downwind sail design space through a parametric study of sail draft and camber. Results show that increasing camber increases both lift and drag a trend that also is evident in three-dimensional sail designs. It is also shown that gains can be made by using designs with draft values as far aft as 60% which helps reduce the extent of trailing edge separation. This parametric design study illustrates how CFD can be used successfully to analyse design trends and rank designs.

The research presented illustrates how CFD can be used in the design process but also that care must be made in validating the method. Through this study the relative strengths and weaknesses of the turbulence models are better understood. Whilst CFD cannot yet be reliably used for downwind sail performance prediction, it is still a useful tool for investigating the flow structure which leads to better understanding of the design space.
Acknowledgements

My greatest appreciation goes out to Professor Margot Gerritsen whose investment in me over many years has been considerable. Through undergraduate, Masters and Ph.D. theses Margot has supported me and has provided a solid base for my academic development. Her kindness in continuing to advise me after her move from Auckland to Stanford University was vital to the course of my Ph.D. Margot has readily gone out of her way for me and I appreciate that greatly. Moreover, her advise is always well thought out and clear whilst also provides room for discovery and individual thought. Margot, I thank you and I look forward to continuing doing research with you.

Much thanks goes to Professor Peter Jackson whose supervision in this Ph.D. was extremely valued and reflective of his considerable experience and knowledge. Peter’s thoughtful questions and answers always motivated me to explore the topic deeply and challenged me to question results from new directions.

Burns Fallow has been my mentor in the sail design industry for many years. From giving me the opportunity as an undergraduate to perform my work experience at North Sails, through to his valued advice on this thesis. Burns has provided me with many valuable experiences and shared much of his esteemed knowledge of sail design.

Acknowledgement must also be provided to Technology New Zealand who supported me thought TIF fellowship contract NSLX9901. Similarly thanks goes to The University of Auckland’s Doctoral Scholarship program.

Finally I would like to thank my friends and family who have always provided me with a warm and exciting life outside of my studies. Sharon, Dave, Jocelyn, Heather and Philippa, your love, support and sacrifice is cherished.
# Contents

Abstract iii

Acknowledgements iv

1 Introduction 1

1.1 Motivation 1

1.2 Previous aerodynamic studies related to yacht sails 4

1.3 Present contributions 8

1.4 Thesis outline 9

2 Introduction to Sailing, Sails and Sail Flows 11

2.1 Introduction 11

2.2 The America’s Cup 11

2.2.1 The history of the Cup 11

2.2.2 The America’s Cup Rule 12

2.2.3 America’s Cup Yacht Racing 13

2.3 Upwind Sailing 13

2.4 Downwind sailing 16

2.4.1 Wind tunnel testing of downwind sails 17

2.4.2 Sail shapes 18

2.4.3 Flow structure for two-dimensional downwind sail sections 19

2.4.4 Flow structure for three-dimensional downwind sails 20

2.4.5 What is the maximum possible lift for a sail section? 24

2.5 Computational issues for sail flow modelling 26

2.6 Aerelasticity of sails 26

3 Computational Approach 28

3.1 Introduction 28

3.2 Turbulence and turbulence modelling 28

3.2.1 Turbulent boundary layers 30

3.2.2 Reynolds averaging and the RANS equations 32

3.2.3 The Boussinesq approximation 33

3.2.4 Summary of two-equation models 34
3.3 The turbulence models .................................................. 35
  3.3.1 The standard $k - \epsilon$ model .................................. 35
  3.3.2 Low Reynolds Number (LRN) modifications for the $k - \epsilon$ model .................................................. 38
  3.3.3 The $k - \omega$ model ............................................. 38
  3.3.4 Comparing the $k - \epsilon$ and $k - \omega$ models ............ 40
  3.3.5 Menter's BSL and SST models .................................. 41

3.4 Turbulence modelling issues for downwind sail flows .......... 44
  3.4.1 Validation of turbulence models for high-lift configurations .................................................. 44
  3.4.2 Validation of the CFX-5 turbulence models for the NACA 4412 airfoil at maximum lift .................... 45
  3.4.3 Unsteady RANS .................................................... 47
  3.4.4 Suitability of the CFX-5 two-equation turbulence models for downwind sail flows ................... 48

3.5 Description of the CFD software ................................... 49
  3.5.1 Grid generation .................................................. 49
  3.5.2 Pre-processing ................................................... 50
  3.5.3 The solver ....................................................... 54

4 The Flat Plate at Shallow Incidence ................................. 58
  4.1 Introduction ......................................................... 58
    4.1.1 Flow structure .................................................. 59
    4.1.2 The thin airfoil bubble ....................................... 61
    4.1.3 The short bubble .............................................. 62
    4.1.4 Experimental data ............................................ 63
    4.1.5 The CFD model .................................................. 64
  4.2 Results .............................................................. 66
    4.2.1 Grid convergence study ....................................... 66
    4.2.2 Comparison at $\alpha = 1^\circ$ ................................ 67
    4.2.3 Comparison at $\alpha = 3^\circ$ ................................ 74
  4.3 Summary ............................................................ 80
    4.3.1 Experiments vs SST and $k - \omega$ simulations ........... 80
    4.3.2 SST vs $k - \omega$ ............................................ 81
  4.4 Conclusions ........................................................ 82

5 Preliminary Wind Tunnel and CFD Investigations .................. 83
  5.1 Introduction ........................................................ 83
    5.1.1 Wind tunnel setup ............................................ 83
    5.1.2 The CFD Model .................................................. 85
  5.2 Results .............................................................. 86
    5.2.1 Convergence studies ........................................... 86
    5.2.2 Wind tunnel - CFD comparison .............................. 87
  5.3 Conclusions ........................................................ 92
List of Tables

4.1 Reattachment lengths for the flat plate at $\alpha = 1^\circ$ ........................................... 69
4.2 Reattachment lengths for the flat plate at $\alpha = 3^\circ$ ........................................... 74

6.1 Chordwise positioning of the 5 boundary layer measurement stations within each flow region.119
6.2 Position of the reattachment and separation points for the SST and $k-\omega$ models at $\alpha = 20^\circ$.119
# List of Figures

1.1 Americas cup yachts sailing downwind under spinnaker. The large foresails - which are symmetrical in cross-section - are known as spinnakers. Asymmetrical downwind sails are known as gennakers. ................................................................. 1

1.2 Wind tunnel testing of a model gennaker in The University of Auckland’s Twisted Flow Wind Tunnel. ......................................................... 2

1.3 Examples of flow visualisation techniques. a. Wind tunnel smoke stream visualisation. b. Streamlines and surface pressures plotted from CFD results. ................................................................. 3

1.4 Wilkinson’s universal pressure distribution (Wilkinson, 1984). ................................................................. 6

2.1 The America’s Cup. ................................................................. 12

2.2 The America’s Cup Course. ................................................................. 13

2.3 The aerodynamic forces on a yacht sailing upwind. ................................................................. 14

2.4 The hydrodynamic forces on a yacht sailing upwind. ................................................................. 14

2.5 The twisted wind profile. ................................................................. 15

2.6 The aerodynamic forces on a yacht sailing downwind at 90 degrees apparent (average conditions for an ACC yacht). ................................................................. 16

2.7 A schematic of the University of Auckland’s Twisted Flow Wind Tunnel. ................................................................. 17

2.8 A sail section with its defining geometry. ................................................................. 18

2.9 A two-dimensional downwind sail flow. ................................................................. 19

2.10 Schematic of the three-dimensional flow past a gennaker at typical trim. ................................................................. 20

2.11 Smoke-flow visualisation for a fibreglass model of an ACC spinnaker in the Twisted Flow Wind Tunnel. ................................................................. 21

2.12 Lift coefficient versus angle of attack (relative to the chord line of the sail) for a solid model spinnaker. The lift coefficients were calculated using the projected sail area. ................................................................. 23

3.1 Schematic of large eddies in a turbulent boundary layer. The flow above the boundary layer has a velocity $U$; the turbulent eddies move at a velocity scale, $u_{mix}$, which is of the order of a tenth of $U$. The largest eddy size, $l_{mix}$, is comparable to the boundary layer thickness ($\delta$) (Wilcox, 1998). ................................................................. 30

3.2 Subdivisions of the near-wall region. ................................................................. 31

3.3 Streamlines for the flow past the NACA 4412 airfoil at maximum lift (13.87°). The simulation was computed by the Author using CFX with the SST model. ................................................................. 45
3.4 Comparison of experimental results of the pressure coefficient, $C_p$, with different turbulence models for the NACA 4412 airfoil (Carrega-Ferreira, Holzwarth, Menter, Esch and Luu, 2001). ......................................................... 46
3.5 Trailing edge detail of the pressure coefficient for the NACA 4412 airfoil (Carrega-Ferreira et al., 2001). ........................................................................................................... 46
3.6 Streamwise velocity profiles computed using several different turbulence models for the NACA 4412 airfoil (Carrega-Ferreira et al., 2001). ........................................................................ 47
3.7 Positions of the six boundary layer traverses for the NACA 4412 test case. .............................................................. 47
3.8 The ICEM-HEXA grid generation technique. .......................................................... 50
3.9 A schematic of the wall boundary treatment. .......................................................... 52
3.10 The control volume approach. ............................................................................. 54
4.1 Schematic of the flow past a flat plate at shallow incidence. .................................... 59
4.2 Typical pressure coefficient plots. ......................................................................... 60
4.3 Schematic of the leading edge bubble illustrating the secondary bubble near the leading edge. .............................................................. 62
4.4 The short airfoil bubble (bubble size exaggerated). .............................................. 62
4.5 Flat plate dimensions. .......................................................................................... 64
4.6 Details of the domain for the flat plate. .................................................................. 65
4.7 Computational grid for the flat plate (medium resolution). .................................... 65
4.8 Grid convergence study of the lift and drag coefficients ($\alpha = 3^\circ$). .............. 66
4.9 Grid convergence of the surface pressure coefficients ($\alpha = 3^\circ$). .................. 67
4.10 Velocity contours ($\alpha = 1^\circ$). .......................................................................... 68
4.12 Chordwise velocity profiles within the leading edge bubble ($\alpha = 1^\circ$). ....... 69
4.11 Flow streamlines and the measurement stations for the flat plate at $\alpha = 1^\circ$ (SST model). .......................................................... 69
4.13 Near-wall chordwise velocity profiles within the leading edge bubble ($\alpha = 1^\circ$). .......................................................... 70
4.14 Chordwise velocity profiles downstream of reattachment ($\alpha = 1^\circ$). .......... 71
4.15 Chordwise velocity profiles downstream of reattachment (log scale, $\alpha = 1^\circ$). .......................................................... 72
4.16 Pressure coefficient plot ($\alpha = 1^\circ$). ............................................................. 74
4.17 Streamwise velocity contours ($\alpha = 3^\circ$). ....................................................... 75
4.18 Chordwise velocity profiles within the leading edge bubble ($\alpha = 3^\circ$). ....... 76
4.19 Near wall chordwise velocity profiles at $x/c = 0.031$ ($\alpha = 3^\circ$). ............... 77
4.20 Turbulent kinetic energy profiles within the leading edge bubble ($\alpha = 3^\circ$). .......................................................... 78
4.21 Turbulent kinetic energy contours around the leading edge ($\alpha = 3^\circ$). ....... 79
5.1 The wind tunnel model. ....................................................................................... 83
5.2 The wind tunnel model setup (from above, not to scale). ..................................... 84
5.3 Details of the domain for the preliminary study. .................................................. 85
5.4 The coarse grid. .................................................................................................... 86
5.5 Grid convergence of the time-averaged lift and drag coefficients. ....................... 87
5.7 Velocity contours and streamlines ($\alpha = 15^\circ$). ............................................... 88
5.6 Force coefficients versus angle of attack ($\alpha = 15^\circ$). —— Data, $c$ SST, $+ k - \omega, \times k - \epsilon, \Delta$
3D solution (SST). ......................................................... 88
5.8 Wind tunnel flow visualisation overlaid with streamlines computed using CFD. .... 89
5.9 The wake structure. ....................................................... 91
5.10 The wake structure computed by the 3D model, $\alpha = 15^\circ$ (image reflected through symmetry plane). ................................................................. 91
5.11 Suction side shear stress and pressure coefficient surface plots computed using the 3D model. ..................... 91

6.1 CFD surface pressure and wall shear ($\tau_w$) plots for the three-dimensional analysis of the wind tunnel model (tunnel wall at bottom of figure, tunnel centerline at top). ......... 95
6.2 CFD streamlines and surface pressures for the three-dimensional analysis of the wind tunnel model (tunnel wall at figure left, tunnel centerline at figure right). ......... 96
6.3 Shaping the model using tension. ........................................ 97
6.4 A schematic of the tensioning system (not to scale). ......... 97
6.5 Photographs illustrating the tensioning system and the mounting of the model to the tunnel frame. ..................... 98
6.6 The model set up in the $7 \times 10$ foot subsonic wind tunnel. ......... 99
6.7 The clamps and stay setup that was used to increase the torsional rigidity of the model. ............. 99
6.8 Illustration of the reference targets on the PSP and the positions of the test and calibration regions. ........... 101
6.9 Comparison between the raw $C_p$ data for a single row of pixels and the smoothed data ($65 \times 7$ pixel averaging stencil) for the suction side of the model at $\alpha = 20^\circ$. Note that the raw data comes from an average image produced from a series of images taken during the test. ..................... 102
6.10 Details of the medium sized domain (not to scale). ........... 102
6.11 The computational grid (medium grid density, medium domain size). ............. 103
6.12 Flow visualisation experiments for the 25% camber circular arc at $\alpha = 20^\circ$. ......... 105
6.13 Close up of the surface flow visualisation illustrating the presence of a secondary recirculation bubble within the leading edge bubble. ..................... 105
6.14 Comparison between CFD and PSP runs at $\alpha = 20^\circ$. ......... 106
6.15 Lift coefficient versus domain length for the SST model at $\alpha=20$ degrees. ......... 108
6.16 Time step convergence of the lift and drag coefficients ($\alpha = 15^\circ$). ......... 109
6.17 Grid convergence of the lift and drag coefficients ($\alpha = 20^\circ$). ......... 110
6.18 Computed lift coefficients near maximum lift ......... 110
6.19 Evaluation of the fitted polynomial for the SST lift polar. ......... 111
6.20 Comparison between the PSP data and CFD at $\alpha = 20^\circ$. ......... 112
6.21 Lift and drag coefficients plotted against angle of attack for the 25% camber arc. These coefficients were computed using the SST and $k - \omega$ turbulence models. ......... 112
6.22 Lift/Drag polars for the 25% camber arc computed using the SST and $k - \omega$ models. ......... 113
6.23 Time-averaged $C_p$ plots for the 25% camber arc at various angles of attack. All plots were computed using the SST turbulence model. ......... 114
6.24 A schematic of the flow past the 25% camber arc at low angles of attack ($\alpha \approx 5^\circ$).  
6.25 A schematic of the flow past the 25% camber arc at ideal angle of attack ($\alpha \approx 9^\circ$).  
6.26 A schematic of the flow past the 25% camber arc at maximum lift ($\alpha \approx 20^\circ$).  
6.27 A schematic of the post-stall flow past the 25% camber arc ($\alpha \approx 25^\circ$).  
6.28 Time history of the force coefficients for the 25% camber arc at $\alpha = 27.5^\circ$ (SST model).  
6.29 Time-averaged reattachment lengths ($X_R$) of the leading edge bubble as a function of angle of attack ($\alpha$).  
6.30 Time-averaged separation points ($X_S$) of the trailing edge separation region as a function of angle of attack ($\alpha$).  
6.31 Flow streamlines and normalised velocity contours ($U/U_\infty$) for the 25% camber arc showing the evolution of the wake through one period of vortex shedding. The simulation was computed using the SST turbulence model and the angle of attack is 20°.  
6.32 Lift and drag time-history for the 25% camber arc at $\alpha = 20^\circ$.  
6.33 Measurement stations for the boundary layer profiles.  
6.34 Chordwise velocity profiles within the leading edge bubble ($\alpha = 20^\circ$).  
6.35 Chordwise velocity profiles in the attached flow region ($\alpha = 20^\circ$).  
6.36 Chordwise velocity profiles within the trailing edge separation region ($\alpha = 20^\circ$).  
6.37 Turbulent kinetic energy profiles within the leading edge bubble ($\alpha = 20^\circ$).  
6.38 Turbulent kinetic energy profiles within the attached flow region ($\alpha = 20^\circ$).  

7.1 Comparison of sails of varying camber (21%, 23%, 25%, 27%, 29%, 31%) with draft fixed at 45%.  
7.2 Comparison of sails of varying draft (40%, 45%, 50%, 55%) with camber fixed at 23%.  
7.3 Forces on a downwind sail.  
7.4 Details of the domain for the design study.  
7.5 The medium grid from the grid convergence study (domain dimensions = 15m x 8m, chord length = 1m and near wall grid spacing = 6.25 x 10^{-5} m).  
7.6 Time step convergence of the lift and drag coefficients ($\alpha = 15^\circ$).  
7.7 Grid convergence of the lift and drag coefficients ($\alpha = 20^\circ$).  
7.8 Lift versus angle of attack for the 2345 section.  
7.9 Comparison between the 2345 section (in red) and a 23% camber circular arc (in blue).  
7.10 Typical induced downwash distribution (in the streamwise direction) due to three-dimensional tip effects (Marchaj, 1979).  
7.11 The computational grid for the gennaker/mainsail configuration (Close up of the region around the sails).  
7.12 Comparison of the flow streamlines and velocity contours for the gennaker / mainsail configuration (a) and the gennaker without the mainsail present (b) ($\alpha = 20^\circ$). The simulations are unsteady and the plots presented are at the same phase angle (180 degrees).  
7.13 Pressure coefficient plots for the gennaker with and without the mainsail present ($\alpha = 20$).  
7.14 A schematic of the circulation field of the mainsail and its influence on the gennaker.  
7.15 Lift versus angle of attack for the gennaker/mainsail configuration and the gennaker by itself.  

xii
7.16 $C_{L_{\text{max}}}$ versus camber (where $C_{L_{\text{max}}}$ is averaged across the different drafts). .......................................................... 141
7.17 $C_{L_{\text{S_{max}}}}$ versus camber (where $C_{L_{S_{\text{max}}}}$ is scaled by the arc length and averaged across the different draft values). .......................................................... 142
7.18 $C_{DS}$ at $C_{L_{\text{max}}}$ versus camber (where $C_{DS}$ is scaled by the arc length and averaged across the different draft values). .......................................................... 143
7.20 The influence of draft position on the flow. .......................................................... 144
7.19 $C_{L_{S_{\text{max}}}}$ versus draft (where $C_{L_{S_{\text{max}}}}$ is scaled by the arc length and averaged across the different camber values). .......................................................... 144
7.21 $C_{L_{\text{S_{max}}}}$ versus draft for each camber value. .......................................................... 145
7.22 Driving force coefficient for the base section shape (section 2345). .......................................................... 147
7.23 Heeling force coefficient for the base section shape (section 2345). .......................................................... 148
7.24 Driving force coefficient for section 3155. .......................................................... 149
7.25 Driving force polar for the base sail shape (section 2345) based on a estimated ACC apparent wind speeds. .......................................................... 149
List of Symbols and Abbreviations

Chapter 1

\( a \)  
mean-line designation; fraction of the chord from leading edge over which loading is uniform at the ideal angle of attack

ACC  
America’s Cup Class

CFD  
Computational Fluid Dynamics

FEA  
Finite Element Analysis

LDA  
Laser Doppler Anemometry

NASA  
National Aeronautics and Space Administration

PC  
Personal Computer

RANS  
Reynolds Averaged Navier-Stokes

RMS  
Root Mean Squared

SYR  
Stanford Yacht Research

TIF  
Technology for Industry Fellowship

TFWT  
Twisted Flow Wind Tunnel

VPP  
Velocity Prediction Program

Chapter 2

\( \alpha_i \)  
induced reduction of the angle of attack due to three-dimensional effects

\( \beta \)  
apparent wind angle

\( \Gamma \)  
circulation

\( \lambda \)  
leeway angle

\( A \)  
wing / sail area

\( AR \)  
aspect ratio

\( c \)  
chord length

\( C_{L(2D)} \)  
lift coefficient (two-dimensional flow)

\( C_{L(3D)} \)  
lift coefficient (three-dimensional flow)

\( DSP \)  
displacement of an ACC yacht

\( D_{Aero} \)  
aerodynamic drag force
\( F_D \)  
\begin{align*} 
\text{driving force} \\
\end{align*}

\( F_H \)  
\begin{align*} 
\text{heeling force} \\
\end{align*}

\( F_S \)  
\begin{align*} 
\text{hydrodynamic side force} \\
\end{align*}

\( F_{T(Aero)} \)  
\begin{align*} 
\text{total aerodynamic force} \\
\end{align*}

\( F_{T(Hydro)} \)  
\begin{align*} 
\text{total hydrodynamic force} \\
\end{align*}

\( h \)  
\begin{align*} 
\text{wing / sail span} \\
\end{align*}

\( L \)  
\begin{align*} 
\text{measured length of an ACC yacht} \\
\end{align*}

\( L_{Aero} \)  
\begin{align*} 
\text{aerodynamic lift force} \\
\end{align*}

\( O_{CE} \)  
\begin{align*} 
\text{Center of Effort} \\
\end{align*}

\( R \)  
\begin{align*} 
\text{resistance} \\
\end{align*}

\( S \)  
\begin{align*} 
\text{measured upwind sail area of an ACC yacht} \\
\end{align*}

\( V_A \)  
\begin{align*} 
\text{apparent wind velocity} \\
\end{align*}

\( V_S \)  
\begin{align*} 
\text{boat velocity} \\
\end{align*}

\( V_T \)  
\begin{align*} 
\text{true wind velocity} \\
\end{align*}

\( y_{back} \)  
\begin{align*} 
\text{the perpendicular distance between the sail and the chordline halfway between } x_d \text{ and the trailing edge} \\
\end{align*}

\( y_{front} \)  
\begin{align*} 
\text{the perpendicular distance between the sail and the chordline halfway between the leading edge and } x_d \\
\end{align*}

\( y_{max} \)  
\begin{align*} 
\text{the greatest perpendicular distance between the sail and the chordline} \\
\end{align*}

\( x_d \)  
\begin{align*} 
\text{the chordwise location of } y_{max} \\
\end{align*}

\( \text{IACC} \)  
\begin{align*} 
\text{International America’s Cup Class} \\
\end{align*}

\( \text{RNG} \)  
\begin{align*} 
\text{Renormalisation Group} \\
\end{align*}

**Chapter 3**

\( \alpha \)  
\begin{align*} 
\text{closure coefficient for the production of } \omega \\
\end{align*}

\( \beta \)  
\begin{align*} 
\text{closure coefficient for the dissipation of } \omega \\
\end{align*}

\( \beta^* \)  
\begin{align*} 
\text{closure coefficient for the dissipation of } k \\
\end{align*}

\( \psi \)  
\begin{align*} 
\text{blending parameter for the NAC term} \\
\end{align*}

\( \delta \)  
\begin{align*} 
\text{displacement thickness} \\
\end{align*}

\( \delta_{ij} \)  
\begin{align*} 
\text{dirac delta function} \\
\end{align*}

\( \epsilon \)  
\begin{align*} 
\text{dissipation of turbulent kinetic energy per unit mass} \\
\end{align*}

\( \phi \)  
\begin{align*} 
\text{unknown variable} \\
\end{align*}

\( \kappa \)  
\begin{align*} 
\text{Von Karman’s constant} \\
\end{align*}

\( \nu \)  
\begin{align*} 
\text{kinematic viscosity, } \nu = \frac{\mu}{\rho} \\
\end{align*}

\( \nu_T \)  
\begin{align*} 
\text{eddy viscosity} \\
\end{align*}

\( \mu \)  
\begin{align*} 
\text{dynamic viscosity} \\
\end{align*}

\( \rho \)  
\begin{align*} 
\text{density} \\
\end{align*}

\( \sigma \)  
\begin{align*} 
\text{closure coefficient for the turbulent transport} \\
\end{align*}

\( \sigma_d \)  
\begin{align*} 
\text{closure coefficient for cross diffusion} \\
\end{align*}
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\sigma_{\epsilon}$</td>
<td>closure coefficient for the turbulent transport of $\epsilon$</td>
</tr>
<tr>
<td>$\sigma_k$</td>
<td>closure coefficient for the turbulent transport of $k$</td>
</tr>
<tr>
<td>$\sigma_T$</td>
<td>closure coefficient for the turbulent transport of $T$</td>
</tr>
<tr>
<td>$\tau$</td>
<td>Reynolds shear stress</td>
</tr>
<tr>
<td>$\tau_{ij}$</td>
<td>Reynolds (or turbulent) stress tensor</td>
</tr>
<tr>
<td>$\tau_w$</td>
<td>shear stress at the wall</td>
</tr>
<tr>
<td>$\omega$</td>
<td>specific rate of dissipation of turbulent kinetic energy</td>
</tr>
<tr>
<td>$a$</td>
<td>finite volume coefficients</td>
</tr>
<tr>
<td>$A$</td>
<td>face area</td>
</tr>
<tr>
<td>$a_1$</td>
<td>Bradshaw's coefficient</td>
</tr>
<tr>
<td>$b$</td>
<td>boundary condition vector</td>
</tr>
<tr>
<td>$B$</td>
<td>log-law constant</td>
</tr>
<tr>
<td>$C_{\epsilon 1}$</td>
<td>closure coefficient for the production term in the $\epsilon$-equation</td>
</tr>
<tr>
<td>$C_{\epsilon 2}$</td>
<td>closure coefficient for the dissipation term in the $\epsilon$-equation</td>
</tr>
<tr>
<td>$C_{\mu}$</td>
<td>closure coefficient for the eddy viscosity</td>
</tr>
<tr>
<td>$f$</td>
<td>current face of the control volume</td>
</tr>
<tr>
<td>$F_1$</td>
<td>cross-diffusion blending function</td>
</tr>
<tr>
<td>$F_2$</td>
<td>SST blending function</td>
</tr>
<tr>
<td>$I$</td>
<td>turbulence intensity</td>
</tr>
<tr>
<td>$i_p$</td>
<td>integration point</td>
</tr>
<tr>
<td>$k$</td>
<td>turbulent kinetic energy</td>
</tr>
<tr>
<td>$L$</td>
<td>characteristic mean-flow length scale</td>
</tr>
<tr>
<td>$l_{mix}$</td>
<td>mixing length of the turbulent eddies</td>
</tr>
<tr>
<td>$N$</td>
<td>the Navier-Stokes operator</td>
</tr>
<tr>
<td>$n$</td>
<td>upstream node</td>
</tr>
<tr>
<td>$n_b$</td>
<td>the set of neighboring nodes</td>
</tr>
<tr>
<td>$p$</td>
<td>instantaneous pressure</td>
</tr>
<tr>
<td>$p_t$</td>
<td>total pressure</td>
</tr>
<tr>
<td>$p_d$</td>
<td>dynamic pressure</td>
</tr>
<tr>
<td>$P$</td>
<td>mean-flow pressure</td>
</tr>
<tr>
<td>$p'$</td>
<td>turbulent pressure fluctuations</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number, $Re = \frac{u_L}{\nu}$</td>
</tr>
<tr>
<td>$S_{ij}$</td>
<td>mean rate-of-strain tensor</td>
</tr>
<tr>
<td>$t$</td>
<td>time</td>
</tr>
<tr>
<td>$t_{ij}$</td>
<td>viscous stress tensor</td>
</tr>
<tr>
<td>$T$</td>
<td>time-scale for Reynolds averaging</td>
</tr>
<tr>
<td>$u$</td>
<td>instantaneous velocity</td>
</tr>
<tr>
<td>$U$</td>
<td>mean-flow velocity</td>
</tr>
<tr>
<td>$u'$</td>
<td>turbulent velocity fluctuation</td>
</tr>
<tr>
<td>$U^+$</td>
<td>dimensionless velocity, $U^+ = \frac{U}{u_\tau}$</td>
</tr>
<tr>
<td>$u_\tau$</td>
<td>friction velocity, $u_\tau = (\tau_w/\rho)^{1/2}$</td>
</tr>
</tbody>
</table>
\( u_{\text{mix}} \) mixing velocity of the turbulent eddies
\( U_\infty \) freestream velocity magnitude
\( V \) volume of the control volume
\( \Delta y \) perpendicular distance between the wall and the first grid point
\( y \) distance to the nearest wall
\( y^+ \) non-dimensional wall distance, \( y^+ = \frac{u^+y}{\nu} \)

AIAA American Institute of Aeronautics and Astronautics
AMG Algebraic Multigrid
ASM Algebraic Stress Models
BSL the Baseline model
CTR Centre for Turbulence Research
DES Detached Eddy Simulation
DNS Direct Numerical Simulation
ILU Incomplete Lower Upper factorisation
LES Large Eddy Simulation
LRN Low Reynolds Number
NAC Numerical Advection Control
PDE Partial Differential Equation
SST Shear Stress Transport

Chapter 4

\( \alpha \) angle of attack

\( A \) surface area
\( C_D \) drag coefficient, \( C_D = \frac{D}{1/2 \rho U_\infty^2 A} \)
\( C_L \) lift coefficient, \( C_L = \frac{L}{1/2 \rho U_\infty^2 A} \)
\( C_P \) pressure coefficient, \( C_P = \frac{P}{1/2 \rho U_\infty^2} \)
\( D \) drag force
\( L \) lift force
\( N \) non-dimensional grid spacing
\( u \) velocity in the chordwise direction
\( u' \) turbulent velocity fluctuations in the chordwise direction
\( u_{\text{RMS}} \) RMS of the \( u \) velocity fluctuations
\( v \) velocity in the direction perpendicular to the plate
\( v' \) turbulent velocity fluctuations in the direction perpendicular to the plate
\( v_{\text{RMS}} \) RMS of the \( v \) velocity fluctuations
\( w \) velocity in the spanwise direction
\( w' \) turbulent velocity fluctuations in the spanwise direction
\( w_{\text{RMS}} \) RMS of the \( w \) velocity fluctuations
$x$  chordwise dimension

$X_R$  reattachment length

ZPG  Zero Pressure Gradient
Chapter 5

2D  Two Dimensional
3D  Three Dimensional

Chapter 6

\[ \begin{align*}
A & \quad \text{PSP calibration coefficient} \\
B & \quad \text{PSP calibration coefficient} \\
\phi & \quad \text{Phase angle} \\
I & \quad \text{light intensity at pressure } p \\
I_0 & \quad \text{light intensity at pressure } p_0 \\
L_* & \quad \text{Length of a particular flow region (leading edge bubble, recovery region or trailing edge separation region)} \\
p_0 & \quad \text{wind-off pressure} \\
X_R & \quad \text{Reattachment length of the leading edge bubble (%c)} \\
X_S & \quad \text{Trailing edge separation position (%c)} \\
PSP & \quad \text{Pressure Sensitive Paint}
\end{align*} \]

Chapter 7

\[ \begin{align*}
\beta & \quad \text{apparent wind angle} \\
\varepsilon_A & \quad \text{aerodynamic drag angle} \\
C_T & \quad \text{total force coefficient} \\
C_{DF} & \quad \text{driving force coefficient} \\
C_{DS} & \quad \text{drag coefficient scaled by the arc length} \\
C_{HF} & \quad \text{heeling force coefficient} \\
C_{L_{max}} & \quad \text{maximum lift coefficient} \\
C_{L_{S_{max}}} & \quad \text{maximum lift coefficient scaled by the arc length} \\
s & \quad \text{arc length} \\
V_{MG} & \quad \text{velocity made good (the yacht's velocity component in the direction of the next mark)} \\
SF & \quad \text{foot length} \\
SLE & \quad \text{leech length} \\
SLU & \quad \text{luff length} \\
SMG & \quad \text{mid girth length} \\
SSA & \quad \text{measured downwind sail area}
\end{align*} \]
Chapter 1

Introduction

1.1 Motivation

With the rapid advancement and depreciating cost of high-end computers, engineers and designers are increasingly tempted to invest in computational methods. However, unlike many other aspects of yacht and sail design, downwind sail development is yet to make regular use of Computational Fluid Dynamics (CFD) tools. The flow field around downwind sails is complex and involves extensive regions of separated flow. Consequently, the downwind sail design process is largely empirical, based around incremental developments and experiences with past designs. Recently wind tunnel testing has become an integral part of high performance downwind sail development but with the current rate of increase of CFD technology and computer power the inclusion of CFD in the design process is inevitable.

Figure 1.1: Americas cup yachts sailing downwind under spinnaker. The large foresails - which are symmetrical in cross-section - are known as spinnakers. Asymmetrical downwind sails are known as gennakers.
In contrast to downwind sails, it is common for sail designers to rely almost entirely on computational approaches in the design of upwind sails. Wind tunnel tests are less accurate for upwind studies due to difficulties in producing a realistic onset flow and in trimming the sails accurately. Upwind sails involve predominantly attached flow for which computational methods are well established. Such aerodynamic design codes are most commonly based on the vortex-lattice method (Katz and Plotkin, 1991; Fiddes and Gaydon, 1996) and can provide three-dimensional solutions for the flow past genoa/mainsail configurations in less than a minute on a common desktop personal computer (PC). Consequently, extensive parametric design studies can be carried out using a large number of design variables. However inviscid panel methods are only valid for close-hauled sailing conditions where the flow remains attached. In fact, their inability to predict leading edge and trailing edge separation makes the application of panel methods for performance prediction questionable even in close-hauled conditions. As a result they are seldom relied upon for aerodynamic input into velocity prediction programs (VPPs).

There are several clear advantages of CFD over wind tunnel testing. Wind tunnel testing is plagued by inaccuracies in the model construction and in obtaining the correct sail trim. In a CFD simulation the geometry is fixed and can be taken either from digitised photographic data, or from an aeroelastic analysis. Furthermore, since wind tunnel models are much smaller than real sails, scaling errors are introduced, both in terms of the inviscid/viscid behavior (Reynolds number scaling) and in the deformation of the model under load (strain scaling). Difficulties are also found in creating realistic flow conditions since the problem is complicated by the twisted apparent wind profile that is created as the yacht travels within the atmospheric boundary layer. At present only three wind tunnels in the world are known to have the

Figure 1.2: Wind tunnel testing of a model gennaker in The University of Auckland’s Twisted Flow Wind Tunnel.
facility to reproduce twisted flow for sailing yachts; the twisted flow wind tunnel (TFWT) (Flay, 1996) at The University of Auckland's Yacht Research Unit (see Figure 1.2) and similar tunnels in California and Italy commissioned by Americas Cup syndicates Oracle BMW racing and Prada respectively.

Perhaps the most significant advantage of CFD is in visualisation of the flow. Flow visualisation techniques in the wind tunnel (see Figure 1.3a) are time-consuming, expensive and inaccurate, whereas with a computer model the flow can be examined in detail with streamlines, velocity vectors and contours plotted with relative ease (see Figure 1.3b). CFD enables a more comprehensive analysis of design changes and a better understanding of the physics involved in the flow. By using CFD, designers can learn not just about the performance of different sail shapes, but they can also begin to understand why a sail performs well - or poorly - and possibly make further improvements based on these findings.

The prospect of obtaining accurate numerical solutions for the flow past downwind sails is enticing for a sail designer. However, there are also many downsides to computational modelling and consequently - for downwind sails at least - CFD is yet to play a significant role in the design process. The computational demands for a CFD model of a three-dimensional sail are considerable. Due to the high Reynolds numbers of sail flows ($1 \times 10^6 - 1 \times 10^7$) the flow will always be turbulent. Grid accurate analysis requires computational grids in excess of 10 million nodes and consequently many gigabytes of computer memory are required. For downwind sails the angle of attack is large (approximately $25^\circ - 30^\circ$ to the chord line) compared to upwind sails and the presence of large and unsteady separated regions makes the solution process difficult. Due to the unsteady nature of downwind sail flows the simulations must use a transient solver with time steps small enough to resolve the unsteady behavior. Even with state-of-the-art supercomputers, simulations of this flow problem inevitably take days, or even weeks, to solve.

Considering the computational demands of a single simulation of the flow past a downwind sail configuration, performing a comprehensive validation study with suitable grid and time-step convergence studies for a three-dimensional case would require exorbitant computer resources and inappropriate solution times considering the time frame of this Ph.D. project. Therefore since three-dimensional solutions were impractical it was decided to study flows around two-dimensional downwind sail sections, in particular; to investigate the performance of turbulence models for flows around thin, highly cambered sections. Two-dimensional simulations require only a fraction of the computer resources and involve the essence of
the physics of three-dimensional flow problem. Flow separation is the most demanding feature of the flow and simplifying the problem from three to two-dimensions does not make the prediction of separation any easier. Further discussion of the differences and similarities between two- and three-dimensional sail flows is provided in chapter 2.

Perhaps the largest obstacle to the accurate simulation of sail flows is the inevitable presence of turbulence, a phenomenon that cannot be simulated exactly (even with the most powerful supercomputers of the present) and must be modelled using a turbulence model. Turbulence models are notoriously unreliable for flows that involve flow separation and numerous validation studies have illustrated that dramatically different results can be achieved depending on the choice of turbulence model (Menter, 1996; Wilcox, 1998). Consequently it is unwise to embark upon a computational design program for a flow as complex as a downwind sail configuration without first performing an in depth investigation into the relative performances of the available turbulence models. In order to be confident of results and to understand their limitations it is necessary to first validate the method carefully and to develop a good understanding of the physics involved. This is the essence of the motivation for the thesis; to gain an in-depth understanding of the physics involved in two-dimensional downwind sail flows and to determine the most suitable computational method for downwind sail analysis.

1.2 Previous aerodynamic studies related to yacht sails

There have been no publications to date on the validation of computational methods for the flow past highly cambered foils that are representative of downwind sail flows. In fact it is difficult to find a flow case that involves all the typical flow features of downwind sail flows. Other high-lift foils such as aircraft take-off and landing configurations and front and rear wings on race cars involve the same design goal of maximising lift while ignoring the influence of drag, however these foils make use of multiple elements, slots and vortex generators to help sustain attached flow. Downwind sail flows - with their large unsteady wakes - resemble flows past circular cylinders more closely than the flow past most high-lift airfoils.

In order to simulate viscous flows involving flow separation it is necessary to model the development of any attached boundary layers that appear in the flow. Bailey (1999) developed upon the work of Jackson and Fiddes (1995) using a two-dimensional panel code coupled to integral boundary layer methods to compute two-dimensional sail flows. Bailey’s model was capable of predicting turbulent transition, leading edge separation and boundary layer separation. However it was found that the methods used were only applicable to low camber sails at moderate angles of attack. Unfortunately extension of such a method to three dimensions is a difficult and as yet unexplored task.

The work of Jackson and Fiddes (1995) highlighted the importance of accurately representing the leading edge bubble that often forms as air passes the luff (leading edge) of a sail. Jackson and Fiddes modelled the leading edge bubble using a geometric approximation that assumed that the bubble shape was elliptical. This approximation is inadequate since leading edge bubbles have complicated shapes that will not necessarily collapse into a generic function. Jackson and Fiddes also expressed the need for good experimental data on the problem in order to provide a better understanding of the bubble shape and its dynamics.

Following Jackson and Fiddes’ request for experimental data on leading edge bubbles, Crompton
(2001) performed wind tunnel tests on a flat plate at shallow angles of incidence. Surface pressures were recorded using pressure tappings, and laser Doppler anemometry (LDA) was used to measure both the mean-flow and RMS velocities in the leading edge bubble and in the developing turbulent boundary layer downstream of the point where the leading edge bubble reattaches. This study answers many questions with regard to the shape and structure of the leading edge bubble, but unfortunately it still provides little insight into the behavior of trailing edge separation of the type that occurs in downwind sail flows.

Whilst a wealth of experimental data exists for two-dimensional airfoil sections, there is little data available for solid thin sections representative of two-dimensional sails shapes. There have been just two significant investigations into the behavior of the flow past solid two-dimensional sail sections, the water tunnel tests of Milgram (1971, 1978) and the wind tunnel tests of Wilkinson (1984, 1989, 1990).

Milgram (1971) performed water tunnel tests for thin cambered plates shaped to the NACA 65 and \( a = 0.8 \) mean lines. The data is in the form of lift, drag and pitching moment, all measured using a force balance. The experiments used Reynolds numbers of \( 6 \times 10^5, 9 \times 10^5 \) and \( 12 \times 10^5 \) and the models had cambers 12%, 12.9%, 15% and 18%. Large chord lengths were used relative to the width of the tunnel so that high Reynolds numbers could be achieved, consequently the aspect ratio of the models was just 2.2. For such low aspect ratio sections and large camber ratios spanwise three-dimensionalities are likely and Milgram was unable to assess the extent of their influence (Milgram, 1971). In 1978 Milgram tested the NACA \( a = 0.8 \) mean line with camber values of 12% and 15% in the same fashion as the 1971 tests, however this time he also included circular and elliptical masts of different diameters in front of his sail models (Milgram, 1978).

Wilkinson (1984) tested the NACA 63 and \( a = 0.8 \) mean lines with camber values ranging from 7.5% up to 17.5% at Reynolds numbers ranging from \( 3.5 \times 10^5 \) to \( 16 \times 10^5 \). Wilkinson’s models had a circular mast attached in the same manner as Milgram, however unlike Milgram he also measured the surface pressure distributions and boundary layer profiles (Wilkinson, 1989; Wilkinson, 1990). Wilkinson was the first to paint a detailed description of the behavior of the flow past sails with reference to a universal description of the surface pressure distribution (see Figure 1.4). The models used by Wilkinson all had aspect ratios of 3.0 and it is likely that some degree of error was introduced due to three-dimensional effects, especially for the high camber models.

Regular vortex shedding from a three-dimensional America’s Cup Class (ACC) mainsail (in a downwind configuration) has been witnessed by the author using smoke flow visualisation in the University of Auckland’s twisted flow wind tunnel. Spinnakers and gennakers do not exhibit structured vortex shedding, but rather a chaotic and unsteady wake region. The lack of any obvious regular vortex shedding for these sails is thought to be due to the very low aspect ratios of these sails and the presence of large turbulent structures. Initial 2D CFD investigations carried out in 2000 were unsteady with periodic vortex shedding and it is expected that wind tunnel tests of two-dimensional downwind sail sections will also exhibit periodic vortex shedding (Collie, 2000). The largest camber values tested by Milgram and Wilkinson were 18% and 17.5% respectively and at these high cambers it is surprising that vortex shedding was not present. However the frequency of vortex shedding for the Wilkinson and Milgram tests would have been between 6 and 40Hz assuming that the Strouhal number was between 0.6 and 0.8 for the tests (these Strouhal numbers have been estimated based on CFD results computed during the course of the current project). Therefore it is possible that vortex shedding was present, but was not
Figure 1.4: Wilkinson’s universal pressure distribution (Wilkinson, 1984).
recorded, due to low sample rates.

Unfortunately, due to errors in the experiments and the low camber values used, the available experimental data for two-dimensional sail sections is insufficient both in terms of quality and detail if one’s goal is to reliably validate a computational method for downwind sail flows. In order to discern differences in the performance of different turbulence models it is necessary to have data that is firstly accurate enough so that the differences between the different turbulence models are more significant than the errors in the data itself. Secondly it is important to have a range of different flow variables to examine in order to enable better understanding of the behavior of the models.

One aerodynamic device that holds many similarities to downwind sails is the parachute. Parachutes are generally designed to provide safe descent of bodies (people, stores or vehicles) or the deceleration of surface vehicles. Therefore they need to inflate reliably and efficiently, and they need to provide significant drag force (Peterson and Strickland, 1996). Such parachutes are typically hemispherical in shape, and in steady descent the flow is separated with a large unsteady wake. Attempts to simulate the aeroelastic behavior of parachutes generally rely on simple aerodynamic models, and as yet no complete computational model of the aerodynamics has been developed or validated (Peterson and Strickland, 1996).

Three-dimensional simulations have been carried out for gliding ram-air parachutes (parafoils) which are designed to provide large lift-to-drag ratios to allow controlled descent (Kalro and Tezduyar, 2000). The aerodynamic model that was used solved the Navier-Stokes equations with a simple (algebraic) turbulence model. The study simulated the fluid-structure interaction of the parafoil with a finite element analysis (FEA) of the membrane deformation. However unfortunately no validation was carried out and no comparisons with experimental results were made.

Whilst parafoils are membrane structures designed to generate lift in a similar fashion to yacht sails, there are significant differences between the two flows. Most notably parafoils use a two layer construction and make use of stagnation pressure to inflate the parachute via small openings at the leading edge. This introduces thickness to the shape and consequently parafoils have a rounded leading edge around which it is possible to sustain attached flow. Some parafoils also make use of vents to help alleviate boundary layer separation on the suction surface in a similar fashion to multi-element high-lift airfoils. Such developments are outlawed in most sailing classes.

There have been few publications to date on three-dimensional CFD simulations of downwind sails. Hedges, Richards and Mallinson (1996) computed the flow past a spinnaker/mainsail configuration at a range of apparent wind angles using the commercial package CFDS-FLOW3D. Richter, Horrigan and Braun (2003) describe a suite of software packages designed to predict the aeroelastic behavior of downwind sails. In their simulations the aerodynamics are tackled using FLUENT 6.0, a commercial CFD package, and the structural behavior is calculated using MemBrain, an in-house FEA package developed by North Sails International. Unfortunately in neither of the studies by Richter et al. or Hedges et al. were grid or time-step convergence studies carried out, and no comparisons were made with experimental data. Therefore the validity of both sets of results is difficult to establish.

The field of sail shape optimisation is in its infancy, yet CFD-based optimisation tools will undoubtedly have a positive impact on sail design in the near future. Work presented by Stanford Yacht Research (SYR) at the recent High Performance Yacht Design Conference in Auckland, New Zealand, represents
CHAPTER 1. INTRODUCTION

the forefront of sail shape optimisation and is detailed by Shankaran, Doyle, Gerritsen, Iaccarino and Jameson (2002). The optimisation methods used by SYR are evolutionary algorithms (Bueche, Stoll and Koumoutsakos, 2001), gradient-based cost function minimisation (Mohammadi and Pironneau, 2001) and the adjoint method (Kim, Alonso and Jameson, 2002). The adjoint method in particular shows considerable promise and the most significant work carried out is the optimisation of an ACC upwind sail plan, accounting for the deformation of the sail under load (Shankaran, 2003). The adjoint method allows for efficient calculation of the cost function sensitivities of the design variables with the computational demand being independent of the number of design variables. The sensitivities are calculated from a single solution to the adjoint equations which are derived from the flow equations, thus the computational demands are equivalent to that of one solution to the flow equations. One additional evaluation of the flow equations themselves is also required per iteration. The optimisation simulations presented by Shankaran et al. (2002) and Shankaran (2003) are inviscid and are based upon the Euler equations, however extension of their solver to account for viscous flow is currently under development.

1.3 Present contributions

The overall purpose of the current work is to gain an in depth understanding of the physics involved in downwind sail flows and to determine the most suitable computational method for downwind sail analysis. In order to achieve this goal the research pursues three specific aims:

- Firstly to assess the performance of different turbulence models for leading edge bubbles of the type found in downwind sail flows.
- The second aim was to perform wind tunnel experiments for several two-dimensional downwind sail sections in order to develop further understanding of the flow features and to provide a database of results suitable for turbulence model validation.
- The final aim was to assess the performance of different turbulence models for our downwind sail test suite from the wind tunnel experiments.

The research was funded by Technology New Zealand under a Technology for Industry Fellowship (TIF). The purpose of this fellowship scheme is to encourage and support academic research and development within New Zealand companies. The project was sponsored by North Sails New Zealand Limited and the work was carried out at Team New Zealand Limited with the research tailored towards the America’s Cup market. The initial goal of the project was to develop a CFD based downwind sail design package and integrate the software into the design procedures of North Sails.

The present research began in 2000 with a study of the current downwind sail design methodology and procedures of North Sails New Zealand and Team New Zealand. This involved investigation into the design parameters and goals, wind tunnel testing, the computational tools - both aerodynamics and structural mechanics (FEA) - and the manufacturing process itself. The result of the study was a list of specifications for the basic input and output parameters for a CFD program and details as to how such a tool would fit into the design process. Following on from this study work was carried out investigating the suitability of different computational methods for downwind sail analysis and design. Initially a wide
range of different methods were investigated from panel codes to advanced turbulence simulations and it became apparent that CFD codes based on the Reynolds Averaged Navier-Stokes (RANS) equations were the most applicable.

Initial attempts were made to develop OVERTURE - an open-source overlapping grid CFD code - for sail flow simulations. However, the state of this software was well below that suitable for commercial use, and it was clear that the project was becoming an exercise in computer programming rather than one of fluid dynamics and sail design. Fortunately, Team New Zealand secured a sponsorship arrangement with AEA Technology for their commercial CFD suite CFX, which was made available for the project.

A literature review was carried out investigating a wide range of RANS based turbulence models which considered the suitability of the models based both on accuracy and simulation efficiency. The review focused on validation studies that have been carried out for flows involving features that are relevant to sail flows, e.g., airfoils near maximum lift, flows involving adverse pressure gradients and boundary layer separation, flows with high streamline curvature, unsteady vortex shedding and three-dimensional effects such as tip vortices. This literature review is documented in Collie, Gerritsen and Jackson (2001) and the key aspects of this study are also presented in chapter 3.

Subsequent work specifically addressed the three primary aims listed above: The validation of CFD methods for the leading edge bubble for flat plates; wind tunnel testing of downwind sail sections; and comparison of CFD results with the wind tunnel results. As an extension to the project a parametric design study was carried out looking at a range of downwind section designs. This study was used to illustrate how CFD can be employed in the downwind sail design process and to gain a better understanding of the influence of different design parameters.

### 1.4 Thesis outline

A description of the basic principles of sailing and the physics involved with sails and yachts is presented in chapter 2. This chapter is intended to provide the reader with insight into the design goals and obstacles related to downwind sails and to provide familiarity to the flow problem. The chapter is targeted towards America’s Cup Class (ACC) yachts and America’s Cup racing, however much of the discussion is relevant to sailing yachts in general.

Chapter 3 motivates, presents and discusses the computational approach that was used in the CFD simulations that were carried out during the research. The issues related to turbulence and turbulence models are discussed, and the results from a review of literature related to turbulence model validation are presented. The chapter also describes the numerical method used in the CFD simulations and the key aspects of the flow solver and grid generation techniques.

A validation study investigating the ability of different turbulence models to reproduce the flow over flat plates at small angles of incidence is presented in chapter 4. CFD simulations are computed and the results are compared with the experimental work of Crompton (2001). The test case is particularly relevant due to the existence of a leading bubbles similar to that found in downwind sail flows.

Chapter 5 documents a preliminary wind tunnel and CFD study of the flow past a two-dimensional downwind sail section. This study details initial efforts to gauge the performance of different turbulence models for downwind sail flows and to determine the nature of complications that are inevitable whenever
wind tunnel experiments are carried out. It was concluded that three-dimensional effects due to the low aspect ratio of the wind tunnel model were significant, and as a result the experimental data was of insufficient quality.

The wind tunnel experiments were repeated in a larger wind tunnel at NASA Ames Research Center, California, in conjunction with Stanford Yacht Research. The larger tunnel accommodated a model with an aspect ratio of 15 and consequently three-dimensional effects were significantly reduced with the flow at midspan being nominally two-dimensional. The results from the wind tunnel tests are presented in chapter 6. The chapter goes on to present the CFD results for the test case and make comparisons with the experimental data. The relative strengths and weaknesses of the different turbulence models are discussed and the most suitable turbulence model is selected.

Chapter 7 presents results from an parametric design study for downwind sail shapes. The study investigates the influence of the draft and camber of the sail sections and attempts to answer the question: What is the maximum lift coefficient attainable from a two-dimensional sail section? The chapter also investigates the influence that the mainsail has on the flow past the gennaker in an effort to establish whether or not downwind sails can be designed without considering the presence of the mainsail.

The final chapter (chapter 8) summarises the thesis, draws conclusions and suggests opportunities for further research.
Chapter 2

Introduction to Sailing, Sails and Sail Flows

2.1 Introduction

The purpose of this chapter is to provide the reader with a background to the physics of sail flows and some insight into sailing and the America's Cup. In particular the challenges involved with sail performance analysis and computer modelling are highlighted. Whilst the discussion is primarily aimed towards America's Cup Class (ACC) yachts and America's Cup racing, much of the discussion is relevant to sailing yachts in general. However this chapter is by no means intended to represent an exhaustive work on the science of sailing. For more detailed discussion on sail design and the naval architecture of sailing yachts the reader is referred to the works of Marchaj (Marchaj, 1964; Marchaj, 1979) or the more modern works of Larsson and Eliasson (1995), Claughton, Shenoi and Wellicome (1998a) and Claughton, Shenoi and Wellicome (1998b).

2.2 The America's Cup

2.2.1 The history of the Cup

The America's Cup was created in 1848 and it is the world's oldest sporting trophy. The Cup was originally referred to as the 100 Guinea Cup and was presented to the Royal Yacht Squadron as a prize for the club's annual regatta. In 1851 the Squadron put up the Cup for a contest between the 101-foot New York Yacht Club schooner America and a fleet of Royal Yacht Squadron vessels in a race clockwise around the Isle of Wight. America was a formidable opponent for the British: She was designed by George Steers, a leading marine designer, with the objective to create a "yacht that would be the fastest afloat". Whether or not this objective was met, America was certainly fast enough to lead the fifteen British yachts around the Isle of Wight to claim the 100 Guinea Cup for the United States. America's victory had more significance than the prestige of winning the world's first international yachting regatta; it illustrated the emergence of the United States of America as a technological power. Since then the America's Cup has been as much about technology as it has been about sailing.
The 100 Guinea Cup was donated to the New York Yacht Club and subsequently was renamed after the winning schooner as the America's Cup. The donation of the Cup came with a list of racing conditions detailed under "the deed of gift" which is still the seminary document that governs the America's Cup. The deed was designed to set up the America's Cup "as a perpetual Challenge Cup for friendly competition between foreign countries".

Between 1870 and 1983 the New York Yacht Club had the longest winning streak in sailing history, successfully defending the cup 24 times before it first left American shores after victory by the Australian Challenge in 1983. Dennis Conner won back the Cup for the United States of America in 1987, and it stayed in San Diego until it was won by Team New Zealand in 1995. In the year 2000 Team New Zealand became the first team to successfully defend the America's Cup outside the United States of America. 2003 saw the Cup return to Europe with Alinghi of Switzerland's victory over Team New Zealand.

2.2.2 The America's Cup Rule

The America's Cup Class (ACC) (formerly the International America's Cup Class or IACC) was initially developed in 1989 by a group of the world's top yacht designers and sailing administrators. It is the purpose of the rule "to produce wholesome day sailing monohulls of similar performance while fostering design developments that will flow through to the mainstream of yachting" (America's Cup Class Rule, Version 4.0, 2000). The ACC class itself is based upon the following rule relating boat length, sail area and displacement,

\[ \frac{L + 1.25 \sqrt{S} - 9.8 \sqrt{DSP}}{0.679} \leq 24, \]

where \( L \) is the rated length of the boat in meters, \( S \) is the rated upwind sail area in meters squared and the displacement, \( DSP \), is the weight of the yacht in meters cubed divided by 1025. Further details for the measurement of \( L, S \) and \( DSP \) can be found in the America's Cup Class rule, version 4.0 (America's Cup Class Rule, Version 4.0, 2000). The purpose of the ACC rule is to create a trade off between the yacht's displacement and its length and sail area so that if a boat is designed to be light, length or sail area must be sacrificed. In general heavy boats have been favoured due to the additional stability.
provided by placing extra weight in the keel.

### 2.2.3 America's Cup Yacht Racing

The course for an America's cup race is illustrated in Figure 2.2. The race consists of four legs, two of which are sailed into the wind (upwind) and two with the wind (downwind). The Cup is contended as a match race, one-on-one between the best challenging syndicate and the Cup holders. In recent years the Cup has been decided over a series of up to nine races.

![Figure 2.2: The America's Cup Course.](image)

### 2.3 Upwind Sailing

The ability of a yacht to sail efficiently to windward relies on an equilibrium between the aerodynamic forces of the rig and sails and the hydrodynamic forces on the hull and its appendages. Yachts sail at an angle to the wind so that the sails can generate lift and provide driving force for the yacht. America's Cup yachts do this efficiently (i.e. they point close to the wind compared to most classes of yachts) and will typically sail at between 30 degrees to the wind in heavy winds up to 45 degrees in light airs. The aerodynamic forces on a yacht sailing upwind (see Figure 2.3) are often resolved into either lift and drag forces relative to the apparent wind angle, $\beta$, or into the driving ($F_D$) and heeling forces ($F_H$) of the yacht. To an aerodynamicist forces are most commonly considered in terms of lift and drag, however for a yacht designer the driving and side forces are often more important since they relate more intuitively to the performance of the yacht.

Generally upwind sails behave much like airfoils; they operate at low angles of attack (relative to the chord line of the sail) and the flow is almost entirely attached. Upwind sails are designed to be efficient, with the design objective being to produce a required driving force (usually specified by the naval architect), with minimum possible heeling force and heeling moment. Increased lift can potentially
CHAPTER 2. INTRODUCTION TO SAILING, SAILS AND SAIL FLOWS

Figure 2.3: The aerodynamic forces on a yacht sailing upwind.

Figure 2.4: The hydrodynamic forces on a yacht sailing upwind.
be generated by trimming the sails so that they have more camber, and this extra lift could potentially be used to increase the driving force of the yacht. However this will always come with an increase in heeling force which is undesirable for two reasons: Firstly a large heeling force tips (heels) the yacht over decreasing the efficiency of the sails and often also the hull. Secondly, the heeling force much be balanced by an equivalent sideways force (the hydrodynamic side force, \( F_S \)) on the hull and appendages and increasing this side force always comes with an associated increase in hydrodynamic drag.

In order to achieve hydrodynamic side force the yacht must travel at an angle of leeway, \( \lambda \), to provide an angle of attack to the hull and appendages so that they can generate lift in the opposite direction to the aerodynamic heeling force. America’s Cup keels use an adjustable trailing edge flap (trim tab) to help generate this lift efficiently and consequently they can sail at leeway angles as low as 1° or less. The hydrodynamic forces are illustrated in Figure 2.4.

The apparent wind velocity, \( V_A \), illustrated in Figures 2.3 and 2.4 is the wind velocity relative to the yacht, i.e. \( V_A = V_T - V_S \). The apparent wind speed, \( V_A \), and angle, \( \beta \), increase up the height of the mast due to the atmospheric boundary layer of the true wind (\( V_T \)) above the sea surface. Typically this boundary layer is between 200 and 400 meters thick and the wind speed at 10m above sea level is approximately 80% of the speed at the top of the mast of an ACC yacht (approximately 34m). Wind twist is illustrated in Figure 2.5 which shows the relationship between the true wind (\( V_T \)), the boat velocity (\( V_S \)) and the apparent wind velocity (\( V_A \)) in three dimensions.

![Figure 2.5: The twisted wind profile.](image)

When sailing upwind the angle of attack at deck level can be 10-15 degrees lower than the angle of attack at masthead. Consequently sail designers must pay attention to wind shear whilst designing upwind sails; the sails must twist off up the rig so that reasonably constant (and efficient) angles of attack (to the chord line) are achieved up the height of the rig.
2.4 Downwind sailing

For many sailing vessels the optimal course downwind is to sail directly with the wind so that the boat is driven by drag. However for high performance yachts (such as ACC yachts) it is advantageous to sail at an angle from the wind so that the lift on the sails also contributes to the driving force. In this way much higher apparent wind speeds can be obtained enabling the yacht to potentially sail faster than the wind. America’s Cup yachts typically sail at true wind angles of between 135 degrees in light airs up to 165 degrees in heavy winds. These true wind angles correspond to a range in apparent wind angles of 50 degrees up to 150 degrees. Here the apparent wind measurements are taken at the masthead and halfway down the mast the apparent wind direction is typically 5-10 degrees smaller due to wind shear.

The average apparent wind angle for an ACC yacht is around 90 degrees and in moderate sailing conditions (9 - 14 knots of true wind speed) the apparent wind angle typically varies from 70 to 130 degrees. According to Team New Zealand’s Sail Coordinator, Fallow (2003), during the last two America’s Cup regattas - that were held on the Hauraki Gulf, Auckland - such conditions were the most common and it was typical for America’s Cup teams to spend approximately 75% of their time sailing in these moderate conditions.

![Diagram of aerodynamic forces on a yacht sailing downwind at 90 degrees apparent (average conditions for an ACC yacht).](image)

The situation with the apparent wind at 90 degrees is particularly interesting since at this angle the lift force coincides with the driving force and the drag force coincides with the heeling force. On a downwind course the side force on an ACC yacht is small compared with upwind configurations. Since
ACC yachts have large amounts of ballast and efficient keels the yacht’s side force and heeling moment are relatively insignificant and can be ignored when designing downwind sails. Consequently, the design objective for downwind sails at an apparent wind angle of 90 degrees is the same as for a high-lift airfoil; we want to produce maximum possible lift whilst paying little consideration to the amount of drag that is generated. A yacht sailing on a course with an apparent wind of 90 degrees is illustrated in Figure 2.6.

2.4.1 Wind tunnel testing of downwind sails

The primary obstacle for wind tunnel testing of downwind sails is the phenomenon of wind twist. Real genoakers are designed so that the sail twists off 20-40 degrees from the foot to the head to account for wind shear. Consequently testing such a sail in a wind tunnel with uniform flow would produce poor results, with either the foot overtrimmed or the head undertrimmed and unstable. As a result, downwind sails are tested primarily at full scale - on the water - through two boat testing.

![Diagram of Twisted Flow Wind Tunnel](image)

Figure 2.7: A schematic of the University of Auckland’s Twisted Flow Wind Tunnel.

The development of the Twisted Flow Wind Tunnel (TFWT) (see Figure 2.7) at the University of Auckland’s Yacht Research Unit has revolutionised the way downwind sails are tested. The TFWT is a low-speed open-jet tunnel with a cross-section of $7.2m \times 3.6m$. The air is sucked into the tunnel by two 3m diameter, 45 kW fans. The air initially passes through a series of turbulence screens and a flow straightening section before a 15m long section for boundary layer development. At the end of the tunnel the flow is twisted by a row of flexible vanes that can be adjusted to provide the desired twist profile.
The center of the model is situated on a turntable 2115 mm downstream of the tunnel opening and the model loads are carried through the turntable to the force balance. The tunnel itself is situated in a large warehouse through which the air is free to recirculate. For further details of the University of Auckland’s Twisted Flow Wind Tunnel the reader is referred to Flay (1996).

From the time that the wind tunnel was built in 1994, Team New Zealand has built and tested over a thousand sails and improvement in excess of 20% (in total driving force) has been achieved across the entire range of downwind sails though testing carried out at the TFWT (Fallow, 2003). However, as the development curve levels out, designers are looking for increasingly subtle performance gains and it is often difficult to discern between different sail designs due to inconsistency and inaccuracy of the tunnel. Irregularity of the return flow, due to obstacles and the size of the building in which the tunnel is housed, is thought to be the primary source of error. Other sources of error are difficulties in trimming the sails consistently (and optimally), the pressure drop created by the vanes (which leads to a non-uniform transverse pressure profile), the turbulent structures in the wakes of the vanes and issues with Reynolds number scaling and strain scaling.

2.4.2 Sail shapes

Typically sail designers build three-dimensional sail shapes from a series of two-dimensional cross-sections. Figure 2.8 illustrates the method for defining the sail cross-sections that is used by North Sails New Zealand Limited. The shapes are described by the draft, camber, front percentage, back percentage, leading edge angle and trailing edge angle. These parameters are defined as follows:

\[ \text{Camber} = \frac{y_{\text{max}}}{c}, \]  
\[ \text{Draft} = \frac{x_d}{c}, \]  
\[ \text{Front percentage} = \frac{y_{\text{front}}}{y_{\text{max}}} \times 100, \]  
\[ \text{Back percentage} = \frac{y_{\text{back}}}{y_{\text{max}}} \times 100, \]

where \( y_{\text{max}} \) is the greatest perpendicular distance between the sail and the chord line, \( y_{\text{front}} \) is the perpendicular distance between the sail and the chord line halfway between the leading edge and \( x_d \), and \( y_{\text{back}} \) is the perpendicular distance between the sail and the chord line halfway between \( x_d \) and the trailing edge.

![Figure 2.8: A sail section with its defining geometry.](image-url)
Leading edge angle is the slope of the tangent to the sail at the leading edge.

Trailing edge angle is the slope of the tangent to the sail at the trailing edge.

2.4.3 Flow structure for two-dimensional downwind sail sections

Downwind sails are designed to maximise the driving force. Aerodynamic lift is the primary contributor to this force and little attention is paid to reducing the drag (in fact drag often contributes to driving force). Consequently downwind sails typically have a large amount of camber (20-30%) and operate at high angles of attack (25-35 degrees to the chord line of the sail) and as a result there is extensive trailing edge separation.

A schematic of the flow past a two dimensional downwind sail near maximum lift is illustrated in Figure 2.9. In the figure the forward stagnation point rests on the windward (pressure) side of the sail just back from the leading edge (luff). This is necessary in order to support the sail surface which is a flexible membrane. If the flow stagnates on the leeward (suction) surface of the sail then the luff of the sail will collapse (for non-rigid sails) due to the pressure difference across the sail. When real sails are trimmed they are initially sheeted out (i.e. the sheets are let out to lower the angle of attack) until the luff of the sail begins to collapse, the sail is then sheeted in slightly until the luff resets. In this fashion the sails are set to the lowest possible angle where the sail remains inflated and stable. It is impossible to set a sail so that the stagnation point rests exactly on the leading edge due to lack of luff tension and the unsteady nature of the flow. Consequently for real sail flows the stagnation point always rests on the windward side of the sail, just back from the leading edge.

A particularly interesting feature of sail flows is the existence of a thin leading edge bubble on the leeward surface of the sail. This bubble is formed as the flow separates whilst attempting to round the sharp leading edge. The leading edge bubble typically reattaches within the first 10% of the chord length, $c$, and downstream of reattachment a turbulent boundary layer develops. Trailing edge separation generally occurs soon after the position of maximum thickness (the draft position) due to an adverse pressure gradient that typically extends across the rear half of the sail.
Simulations have shown that the trailing edge separation region is unsteady and periodic with counter rotating vortices being shed alternately from the windward and leeward surfaces of the sail. This unsteady process has a considerable effect on the loading of the sail with the forces oscillating in a sinusoidal fashion. Since the circulation, $\Gamma$, of the sail is changing the angle of incidence to the leading edge also oscillates, hence the positions of stagnation, reattachment and separation points must also vary in a periodic manner.

As will be shown in this thesis, the maximum lift for two-dimensional ACC downwind sail sections occurs at an angle of attack of $18.5^\circ \pm 3^\circ$ depending on the shape of the sail. As stall is reached the trailing edge separation point moves forward whilst the leading edge bubble also grows in size. At approximately 5 degrees above maximum lift the leading and trailing edge bubbles meet and the flow becomes fully separated with vortices being shed alternately from the luff and leech.

**2.4.4 Flow structure for three-dimensional downwind sails**

The flow structure described in the previous section assumes that there is no flow in the third dimension (i.e. vertically). For real three-dimensional sails this is not the case since the sails have significant curvature in the spanwise direction and they typically have small aspect ratios which results in significant pressure leakage at the tips. Consequently three-dimensional effects are considerable in downwind sail flows. Through wind tunnel flow visualisation smoke-streams have been observed to pass a gennaker to windward approximately 25% of the height up the luff, then drop and exit approximately midway along the foot. To leeward the opposite occurs; smoke streams passing the luff near the foot leave the sail well up the leech. Even near mid height there is considerable crossflow. A schematic of the flow past a gennaker is provided in Figure 2.10 which illustrates crossflow on the sail surface and the development of tip vortices around the head and foot.

![Figure 2.10: Schematic of the three-dimensional flow past a gennaker at typical trim.](image)
The degree of three-dimensionality of downwind sail flows has a large influence on performance. Downwash due to the tip vortices is considerable and the sectional pressure distributions for a real sail have different characteristics than two-dimensional sails. An investigation into stall behavior of three-dimensional solid downwind sails was carried out by the author at the University of Auckland’s Twisted Flow Wind Tunnel (TFWT). Figure 2.11 illustrates the wind tunnel setup and the smoke-streams which were created using a heated wire filament, coated with oil and positioned upstream of the model. The fiberglass model that was used was developed by McLean (2001) and Prentice (2001) for an investigation into downwind sail shapes. By using sails molded out of thin fiberglass McLean and Prentice were able to test sails of fixed geometry with the flying shape almost identical to the design shape. For flexible cloth sails - which are more commonly used in the wind tunnel - the shape that the model forms under load is often quite different to the design shape and the shape is quite sensitive to the trim settings. For the present study an additional advantage of using solid sails is that the sail retains its shape even at low angles for which cloth sails would collapse due to stagnation on the leeward surface.

Figure 2.11: Smoke-flow visualisation for a fibreglass model of an ACC spinnaker in the Twisted Flow Wind Tunnel.

Smoke-flow visualisation showed that there is significant interaction between the leading edge and trailing edge separation regions. The angle of attack where the local flow is at incidence with the leading edge and there are no leading edge bubbles is defined as the ideal angle of attack. For the present tests
this situation occurred at an angle of attack of approximately 25 degrees. Below the ideal angle of attack a leading edge bubble forms on the windward side of the sail and at 5 degrees below the ideal angle of attack the leading edge bubble extends over the forward half of the windward surface. Below and at the ideal angle of attack the flow is almost fully attached on the leeward surface as can be seen in Figure 2.11. For these angles the flow near mid height of the sail is quite two-dimensional with the windward and leeward streams exiting the leech with little spanwise separation.

As the angle of attack is increased above the ideal angle a leading edge bubble begins to form on the leeward surface and the trailing edge separation point moves forward abruptly. It was found that increasing the angle of attack by just a few degrees above the ideal angle of attack caused the trailing edge separation point to move forward by 30%—40%. It was also noticed that above the ideal angle of attack there was considerably more spanwise separation between the windward and leeward smoke-streams as they passed the leech.

The adverse effect that the leading edge bubble has on the trailing edge separation region is thought to be due to a combination of two effects. Firstly downstream of the leading edge bubble the boundary layer is in a mode of recovery and has a profile that is atypical of turbulent boundary layers. Instead of the large near-wall velocity gradient that is characteristic of a turbulent boundary layer the near wall velocities have lower velocity in the near-wall region and hence also less kinetic energy. The displacement thickness of the boundary layer is also increased due to the presence of the leading edge bubble. Consequently the recovering boundary layer is sluggish and prone to separate under the influence of an adverse pressure gradient such as that which typically extends over the aft half of a downwind sail. However secondly, and perhaps more significant is the interaction between the leading edge bubble and the tip vortices at the head and foot of the sail. As the leading edge bubble forms the pressure distribution changes with suction increasing in the immediate vicinity of the leading edge bubble. This change in the flow topology has an effect on the both the chordwise and spanwise loading of the sail and hence also the downwash distribution. Moreover, inside the leading edge bubble the velocity magnitudes are small and consequently significant spanwise flow occurs within the bubble. This effect is most notable near the head of the sail where the afterwards sweep of the luff encourages upwards flow within the bubble. This effect has been observed in smoke flow visualisation and it is hypothesised that the cross flow in the leading edge bubble feeds into the tip vortex at the head. This process alters the sail’s downwash distribution, which increases the amount of trailing edge separation and initiates stall.
CHAPTER 2. INTRODUCTION TO SAILING, SAILS AND SAIL FLOWS

The lift polar from the solid sail TFWT experiments is provided in Figure 2.12. Maximum lift occurs at an angle of attack of approximately 25 degrees which coincides with the ideal angle of attack. Stall occurs beyond the ideal angle of attack due to the formation of the leading edge bubble and the rapid increase in the amount of trailing edge separation. At angles of attack of 35 degrees and above the flow on the leeward surface was fully separated.

In Figure 2.12 there are two sharp changes in the lift coefficient - one between 25 and 27.5 degrees and the other between 32.5 and 35 degrees - both of which are caused by significant changes in the flow topology. The first change is due to formation of the leading edge bubble and the subsequent increase in the amount of trailing edge separation. The second drop in lift coefficient is due to the flow entering a fully stalled state where there is no attached flow on the leeward surface. Another notable feature of Figure 2.12 is the particularly large lift values with the results indicating a maximum lift coefficient of approximately 1.81.

Through comparison between the current tests and past experience with smoke-flow visualisation for conventional wind tunnel models (i.e. cloth sails) the flow at around 30 degrees appeared to be the most similar to the position where real sails are trimmed. It appears that real, flexible sails need to be sheeted at approximately 5 degrees above the ideal angle of attack in order to support a stable luff. This extra 5 degrees constitutes approximately a 3% loss in lift which in the realm of America’s Cup sail design is significant. Therefore sail designs that can retain a stable luff whilst keeping the leading edge bubble small are attractive. Sails with inflatable luffs, or slots - emulating the leading edge slats that are found on high-lift airfoils - are attractive design prospects, however unfortunately such concepts are forbidden by the design rules for most sailing classes including the ACC (America’s Cup Class Rule, Version 4.0, 2000).

The flow situation at 30 degrees has a leading edge bubble that extends approximately 6%c downstream of the leading edge and trailing edge separation occurs approximately halfway along the sail. This sectional flow behavior is believed to be typical of downwind sails - at mid height at least - and is similar to the two-dimensional flow presented in Figure 2.9. The flow situation is likely to change approaching the head and foot of the sail and the degree of flow separation is also dependent on the draft and camber of the sail.
The maximum lift coefficient from Figure 2.12 occurs at an angle of attack 5-10 degrees higher than that for the maximum lift for two-dimensional downwind sail sections. Real sails are trimmed to even higher angles in order to keep the luff inflated. In the next section it is shown that this shift is due to the effects of downwash which lowers the effective angle of attack in the vicinity of the luff by at least 10 degrees. Downstream of the luff downwash increases as the tip vortices develop and gather more vorticity. This leads to the trailing edge separation bubble being slightly smaller in size compared with that observed in two-dimensional sail flows with comparable leading edge bubbles.

Two-dimensional sail flow solutions are still valuable to a sail designer despite the fact that real sail flows are strongly three-dimensional. Two-dimensional models provide a useful tool for a designer to explore the influence of the design parameters that describe the sectional sail shape. Most sail designers build up three dimensional designs as a combination of two-dimensional sections up the height of the sail. Also some Velocity Prediction Programs (VPPs) formulate aerodynamic force coefficients based upon series of sectional coefficients up the height of each sail. Therefore looking at the performance of the individual sail sections must have some relevance to the behavior of the real, three-dimensional sail.

Two-dimensional simulations are also valuable for validation of computational methods that are intended for use in three-dimensional sail simulations. From a turbulence modelling perspective the two-dimensional simulations present many of the same difficulties as the three-dimensional problem: leading edge separation, reattachment, boundary layer recovery, trailing edge separation and the formation of an unsteady wake. The only additional complexity in the three-dimensional case is crossflow in the separated zones and the development tip vortices that may involve significant streamline curvature. However for validation of the ability of turbulence models to predict separation and the resulting unsteady wake, two-dimensional sail flows provide a good test case. If a method was deemed suitable for computing two-dimensional sail simulations then the step to three-dimensional simulations would be largely elementary, with grid generation posing a more significant obstacle than the turbulence modelling.

2.4.5 What is the maximum possible lift for a sail section?

Multi-element airfoils can produce lift coefficients as high as 5 (Smith, 1975) and through boundary layer control techniques such as tangential blowing and vortex generators even higher lift coefficients are possible. However conventional single-element airfoils seldom have maximum lift coefficients greater than 2 which makes the values presented in Figure 2.12 look dubious, especially considering the relatively low aspect ratio of the model. The reason such high lift coefficients can be obtained for downwind sails is the relative freedom in the design goal which is solely to maximise lift. Conventional airfoils generally are designed for cruise conditions where the goal is to minimise drag for a set amount of lift. Such airfoils have only small amounts of camber and never exhibit separated flow. Typical lift to drag ratios for conventional airfoil sections are in the vicinity of 50-100 and the entire Boeing 747 aircraft has a lift to drag ratio of 17.7. In comparison the downwind sail tested in the TFWT has a lift to drag ratio of 2.24 at maximum lift.

Liebeck (1973) set out to answer the question: "What is the maximum lift that can be developed by a single-element foil in attached flow...?" Using inverse airfoil design techniques he developed a high-lift airfoil that has a theoretical lift coefficient of 3.06, and for similar airfoils wind tunnel data closely matched the theoretical values. The Liebeck airfoil is designed solely for maximum lift, it is highly cambered and
has almost negligible thickness, consequently away from its design conditions the airfoil performs poorly. The Liebeck airfoil was designed under the constraint that the flow remains attached all the way to the trailing edge, remove this constraint and even higher lift coefficients are possible. It is likely that such airfoils would still be thin sections but with higher camber, much like downwind sail sections.

The effect of aspect ratio on the performance of downwind sails and the relationship between the two-dimensional and three-dimensional lift coefficients can be estimated using Lanchester-Prandtl wing theory (Lanchester, 1907; Prandtl, 1918a; Prandtl, 1918b). This theory assumes attached flow and an elliptical lift distribution hence its use here provides only a guide rather than quantitative predictions. The three-dimensional lift coefficient is related to the two-dimensional coefficient and the aspect ratio of the wing by the following equation,

$$C_{L(3D)} = \frac{C_{L(2D)}}{1 + \frac{2}{AR}},$$

(2.2) where $C_{L(3D)}$ is the three-dimensional lift coefficient, $C_{L(2D)}$ is the two-dimensional lift coefficient and $AR$ is the aspect ratio of the wing which is defined as,

$$AR = \frac{h^2}{A},$$

(2.3) where $h$ is the span of the wing and $A$ is the wing area. The aspect ratio of the model used in the wind tunnel tests was 1.98, and hence equation (2.2) suggests that the three-dimensional lift coefficient is approximately half as large as the average lift coefficient of the two-dimensional sections that make up the three-dimensional model. Figure 2.12 gives the maximum lift of the three-dimensional model to be 1.81, suggesting that the two-dimensional $C_{L(\text{max})}$ could be as high as 3.62. However this argument is only valid well below stall where the lift increases linearly with angle of attack. Therefore the two-dimensional $C_{L(\text{max})}$ is likely to be considerably less than 3.62. At 25 degrees, where maximum lift occurs for the three-dimensional model, an equivalent two-dimensional would be stalled since the three-dimensional model sees a reduction in the effective angle of attack due to downwash caused by three-dimensional effects. This induced downwash angle can be calculated using wing theory, i.e.,

$$\alpha_i = \frac{C_{L(3D)}}{\pi AR},$$

(2.4) where $\alpha_i$ is the induced angle of attack. Using the figures quoted above the induced angle of attack is calculated to be 16.7°, i.e. the angle of attack for the flow past a three-dimensional downwind sail should be 16.7° higher than the angle where the equivalent two-dimensional flow occurs. Wing theory assumes an elliptical spanwise lift distribution which provides a constant induced angle along the span of the wing. Since our model is likely to have a lift distribution that is not elliptic the induced angle cannot be expected to be constant across the span of the model and the figure of 16.7° serves only as a guide.

As mentioned in the previous section the maximum lift angle in three-dimensions is only 5-10 degrees higher than the angle of maximum lift for typical two-dimensional sections. However for two-dimensional sections the maximum lift coefficient occurs at an angle where the leading edge bubble is present on the leeward side and the ideal angle of attack occurs approximately 10 degrees below maximum lift. Therefore 16.7° is not an unreasonable estimation of $\alpha_i$ considering the coarse approximation to wing theory, and it certainly provides a good guide as to the extent of the three-dimensional effects.
2.5 Computational issues for sail flow modelling

Complete CFD simulations of three-dimensional sail simulations pose large demands on current state-of-the-art computers. In order to adequately predict flow separation it is necessary to use a computational grid that captures the flow right down to the sail surface including the viscous sublayer, the innermost region of the boundary layer. Generating grids for a pair of sails is particularly painstaking and since most sails are triangular it is very difficult to map block-structured or overlapping (chimera) grids to the sail surfaces. Instead, an unstructured approach with hexahedral or prismatic cells in the boundary layer is generally favoured at the expense of having to use greater cell densities near the sail surfaces to avoid low aspect ratio tetrahedral cells at the boundary layer edge. The close proximity of the two sails near the head of the genoa (upwind) or the head regions of both the genaker/spinnaker and mainsail (downwind) requires additional refinement of the grid. Inclusion of the mast, spreaders, shrouds, boom and the yacht itself pose additional demands on the grid generation process. Unfortunately these extra components of the geometry can have significant influence on the flow field and often cannot be ignored. The resultant grid sizes for both upwind and downwind sail configurations are typically in excess of 10 million nodes. Grid sizes can be reduced significantly for the upwind case through the use of wall functions. However since even the upwind case often involves flow separation - particularly near the head of the mainsail - the wall function approach may instigate misleading results.

The downwind case is further complicated by the unsteady nature of the flow. Full scale America’s Cup sails exhibit vortex shedding over a range of frequencies typically around 1 – 5Hz (Fallow, 2003). The timestep size for CFD simulations must be small enough to capture the shedding frequency which typically leads to timesteps that are much shorter than would be required by the stability criterion of the numerical scheme. Many timesteps are required in order to achieve convergence to a shedding cycle with a consistent period where the flow variables oscillate with constant mean, amplitude and period. Even upwind simulations are likely to experience some form of unsteadiness due to regions of separated flow. As a result, convergence to a steady state is difficult and is often only achievable (falsely) with a large amount of numerical diffusion.

2.6 Aeroelasticity of sails

Aerodynamics and structural engineering go hand in hand as the key aspects of sail design. Producing a sail that sets in the desired aerodynamic shape requires an understanding of the stresses and strains within the sail as it comes under load. For downwind sails in particular the strains are large and the flying shape of the sail (the deformed shape of the sail under load) can be quite different to its design shape (the unloaded shape of the sail). Similarly the production of a sail with sufficient structural integrity and stability, whilst keeping weight to a minimum, requires an understanding of the aerodynamic pressure loading of the sail. Consequently, the overall development of the flying shape of a particular sail is an exercise in aeroelasticity.

There have been many publications regarding the development of mathematical methods to describe the aeroelastic behavior of flexible sails. Early two-dimensional models assumed that the flow on the leeward surface was either fully separated (Cisotti, 1932; Dugan, 1970) or fully attached (Voelz, 1950; Thwaites, 1961) and approximated the tension as constant along the entire length of the sail. The attached
CHAPTER 2. INTRODUCTION TO SAILING, SAILS AND SAIL FLOWS

flow model that was developed independently by Voelz (1950) and Thwaites (1961) used a potential flow solution based upon a distribution of vortices along the sail surface. This formulation formed the basis for the modern three-dimensional panel methods that are used by most sail designers.

More recent developments include iterative methods with non-uniform tension (Jackson, 1984) and analysis of three-dimensional membrane wings (Jackson and Christie, 1987). Limited publications have investigated the aeroelastic behavior of sails with trailing edge separation. Cyr and Newman (1996) use a boundary integral method to predict the trailing edge separation point and model the separation region itself using a single point source on the sail surface near the trailing edge. Lorillu, Weber and Hureau (2002) used a similar method to Cyr and Newman (1996) but made use of experimental data to predict the point of separation. Unfortunately none of these methods consider leading edge bubbles which have been shown to have a significant effect on the aerodynamics (Bailey, 1999).

In modern design lofts the typical evolution of a sail design most commonly relies on an iterative analysis of the aerodynamic and structural behavior of the sail (Fallow, 1996). Initially a design shape is created and the aerodynamics tested, usually with a potential flow solver. A finite element analysis (FEA) is then carried out to calculate the deformation of the sail based upon input from the aerodynamic pressure loads. Once a deformed sail shape is determined from the FEA the aerodynamic loads are recalculated for the new shape and the process is repeated until the sail shape converges to a realistic flying shape. The performance of the final shape is then quantified and a designer may choose to make improvements in the sails structural or aerodynamic design and then repeat the aeroelastic analysis.

At present such a design cycle is more common for upwind sails where the computational methods for the aerodynamics are well established. Downwind aeroelastic analysis is sometimes performed using estimates of the surface pressure field, however without accurate CFD or wind tunnel results the process can be imprecise. Yet downwind sail design is perhaps where the most advantage can be gained from aeroelastic analysis. Upwind designs are relatively stable and hence the aero-elastic situation can be approximated well with a steady state analysis. Downwind sails are held at just three points (head, tack and clew) and consequently have more geometric degrees-of-freedom compared with upwind sails which are fixed along the luff and in the case of the mainsail also the foot. The unsteadiness of downwind aerodynamics also contributes to the dynamic aeroelastic response of the sails. Spinnakers and gennakers will deform in response to the unsteady aerodynamic loads and if the load frequencies match the resonant frequencies of the sails supporting structure (the mast, halyards, spinnaker pole and sheets) then large motions may be experienced that are detrimental to the performance of the yacht and possibly even the structural integrity of the rig and yacht itself. This type of effect is particularly noticeable when downwind sails stall and the system develops a response analogous to bending-torsion stall flutter found in airplane dynamics. As further evidence of the unsteady aeroelastic behavior of downwind sails, when a spinnaker or gennaker is trimmed aggressively the luff can often be seen to inflate and collapse at a frequency dictated by the shedding of trailing edge vortices.

Aeroelastic analysis is not considered in this work and instead the computational method is intended for use in assessing flying shapes only, the philosophy being that it is necessary to first understand what sail shapes provide the best aerodynamic performance. Creating the design shapes that will produce the desired flying shape is something that we can be concerned about later. In any case coupling of the CFD analysis to structural codes is common and can be achieved without the need for large amounts of code.
Chapter 3

Computational Approach

3.1 Introduction

In this chapter the computational approach used for modelling the flow past downwind sail sections is motivated, presented and discussed. Firstly, a discussion of the turbulence modelling issues and the preferred approach is provided. Turbulence modelling is seen as the primary obstacle to the accurate computation of sail flows and consequently a background to turbulence modelling is provided. Then, the turbulence modelling issues that are related to sail flows are discussed and reference is made to validation studies from the literature that are pertinent to sail flows. A more extensive literature review on the topic of turbulence modelling for sail flows was completed by Collie et al. (2001) and much of the wider information has not been repeated here. In the literature review a wide range of turbulence models were presented and discussed whereas here we introduce only the models that were used directly in this research. Finally, a brief outline of the numerical solver, CFX-5, is given along with a description of the grid generation software, ICEM-HEXA, and the techniques used in the development of the computational grids.

3.2 Turbulence and turbulence modelling

Unfortunately for sail designers, sail flows are at Reynolds numbers where the flow will always involve turbulence. For the flow past downwind sails at least, turbulence cannot be ignored since the overall topology of the flow is strongly dependent on the turbulent behavior.

Turbulence is commonly regarded as one of the foremost unsolved problem of classical physics (Speziale, 1991). But why is the computation of turbulent flows so difficult? One of the most notable features of turbulence is the chaotic fluctuation of velocities and pressure through space and time, a phenomenon which is often seen as swirling structures in the flow known as turbulent eddies. The turbulent eddies extend down to a minute size and in the near-wall region of a turbulent boundary layer even the largest eddies are extremely small (in fact the eddy size tends to zero at the wall). Resolving turbulent eddies within a boundary layer at Reynolds numbers typical of sail flows ($10^6 < \text{Re} < 10^7$) would require minute grid sizes and solving the three-dimensional flow past yacht sails using the pure Navier-Stokes equations would require many gigabytes of memory just to store the computational grid. Therefore turbulence must be modelled through approximations.
The scope of turbulence models available today is extensive. However, despite over a century of
research in the field of turbulence, there are no turbulence models that can be appropriately applied to a
universal range of flows. Turbulence modelling can be divided into the following primary fields (ordered
by increasing computational demand):

1. Integral boundary layer methods, where empirical equations are used to describe the boundary
layer growth within a potential flow code. These methods use empirical relations to describe the
viscous effects within the boundary layer including transition to turbulent flow and (sometimes)
the effect of boundary layer separation. These methods are limited to simple boundary layer flows
and often require tuning for each new application.

2. Reynolds Averaged Navier-Stokes (RANS) modelling, where turbulence is modelled using the
Reynolds averaged Navier-Stokes equations (equation (3.7)). The RANS equations are derived
from ensemble-averages of the Navier-Stokes and continuity equations. The core element of RANS
modelling is the representation the Reynolds or turbulent stresses which describe the effects of the
turbulent fluctuations of pressure and velocities.

3. Large Eddy Simulation (LES), where the large, energy containing eddies are computed directly and
the effect of the small-scale eddies are modelled using sub-grid models (Ferziger, 1996; Rodi, 1997;

4. Direct Numerical Simulation (DNS), where the pure Navier-Stokes equations are solved on grids
fine enough, and with time-steps small enough, to capture the full range of turbulent scales (Moin
and Mahesh, 1998).

Only RANS modelling was considered in this review since the other methods are currently either too
computationally expensive for practical engineering applications, or in the case of integral boundary layer
methods, too simple and unreliable (especially for flows with significant regions of separated flow). DNS
requires extremely high grid resolution and minute time steps, and is thus not practical for application
to sail flows. Presently DNS modelling is limited to simple low Reynolds number flows and is predomin-
antly used for generating turbulent flow data for simple problems in order to help develop and validate
turbulence models.

LES models use a mathematical model for the small eddies (a sub-grid model) and hence a lower
grid resolution can be used compared with DNS. The sub-grid model is used to represent the eddies that
are too small to be resolved by the grid and the larger eddies are fully captured in the same fashion
as DNS. The small eddies that are modelled are likely to be less anisotropic than the large, energy
containing eddies and hence can be captured more appropriately with a mathematical model. Typically
for non-wall-bounded flows the grid resolution for LES is required to be approximately an order of
magnitude lower than if the same simulation was performed using DNS. This is still not practical for
most engineering problems and is certainly unsuitable for sail flows. For wall bounded flows - such as
sail flows - the computational requirements for LES are particularly high since close to the wall even
the largest eddies are very small. At Stanford University’s Centre for Turbulence Research (CTR) LES
has been used for many high-Reynolds number reacting and non-reacting flows in complex geometries
(Mahesh, Constantinescu, Apte, Iaccarino, Ham and Moin, 2002) including the flow around the circular cylinder at a Reynolds number of 1 million (Catalano, Wang, Iaccarino and Moin, 2003).

A relatively new field of research is Detached Eddy Simulation (DES) where LES is used in regions outside the near-wall which itself is captured using a RANS model (Spalart, Jou, Strelets and Allmaras, 1997; Strelets, 2001). For industrial applications involving wall boundaries, DES provides considerable promise and despite being at the infant stage of development good results have been obtained, particularly for wall bounded flows involving massive separation and unsteady phenomena (Spalart et al., 1997).

3.2.1 Turbulent boundary layers

For most aerodynamic flows it is the boundary layer that is the source of turbulence in the flow. High mean-shear within the boundary layer drives transition and the production of turbulent energy. Figure 3.1 presents a smoke-flow visualisation of a turbulent boundary layer illustrating the turbulent eddies within the boundary layer.

![Figure 3.1: Schematic of large eddies in a turbulent boundary layer. The flow above the boundary layer has a velocity $U$; the turbulent eddies move at a velocity scale, $u_{mix}$, which is of the order of a tenth of $U$. The largest eddy size, $l_{mix}$ is comparable to the boundary layer thickness ($\delta$) (Wilcox, 1998).](image)

Experiments have shown that a boundary layer in turbulent equilibrium (where production of turbulent energy is balanced by its dissipation) can be divided up into three layers as depicted in figure 3.2. In the inner layer, which is known as the viscous sublayer, molecular viscosity plays a dominant role in momentum transport.
CHAPTER 3. COMPUTATIONAL APPROACH

Figure 3.2: Subdivisions of the near-wall region.

Here viscous damping of the turbulent fluctuations makes turbulence modelling difficult. In the outer layer, called the defect layer, turbulence plays the dominant role in momentum transport and the effect of molecular viscosity is negligible. Finally, in between these two regions, both turbulence and molecular viscosity play a part in the flow dynamics. In this region, which is known as the log-law region, velocity profiles have been shown to follow a logarithmic profile referred to as the log-law of the wall. The log law of the wall for the tangential velocity in a zero pressure gradient boundary layer is as follows,

$$ U^+ \simeq \frac{1}{\kappa} \ln y^+ + B, $$

where $U^+$ is the dimensionless velocity and $y^+$ is the dimensionless distance to the wall. These are defined as,

$$ U^+ = \frac{U}{u^*} \quad \text{and} \quad y^+ = \frac{u^* y}{\nu}, $$

where the friction velocity, $u^* = (\tau_w/\rho)^{1/2}$, and $\tau_w$ is the shear stress at the wall. Coles and Hirst (1969) found estimates of $\kappa$ and $B$ from correlation with a number of attached, incompressible boundary layer experiments. The values are:

$$ \kappa = 0.41 \quad \text{and} \quad B = 5.0. $$

The log-law of the wall (equation (3.1)) can be used to describe the flow in the region of $20 < y^+ < 300$ for most simple boundary layer flows, so long as there are no large pressure gradients. A common implementation of RANS models makes use of the log-law of the wall to formulate off-the-wall boundary conditions known as wall functions. This approach is particularly popular as it reduces the near-wall grid resolution requirements considerably. However, for more complex adverse pressure gradient flows and separated flows the boundary layer profile in the log-law region has been shown to depart significantly from equation (3.1) and wall functions are often blamed for the poor performance of RANS models (Patel, Chon, and Yoon, 1991; Schofield, 1986; Wilcox, 1998).

The velocity profile in the sublayer behaves in a linear fashion and can be described using the sublayer equation,

$$ U^+ = y^+. $$
3.2.2 Reynolds averaging and the RANS equations

In RANS modelling the velocities and pressure are divided into their mean and fluctuating parts, so that:

\[ u_i = U_i + u'_i, \quad \text{and} \quad p = P + p'. \] (3.5)

Here \( u_i \) and \( p \) are the instantaneous variables we are decomposing, \( U_i \) and \( P \) are the mean-flow values, and \( u'_i \) and \( p' \) represent the turbulent fluctuations. It is important to note that by “mean-flow” we do not mean a long term spectral average since \( U_i \) and \( P \) may still vary through time. More specifically, \( U_i \) is the non-turbulent behavior of \( u_i \), and \( P \) is the non-turbulent behavior of \( p \). \( U_i \) can be evaluated from the instantaneous velocity using a time-average of the form,

\[ U_i(x,t) = \int_{t_1}^{t+T} u_i(x,t) \, dt, \quad T_1 << T << T_2, \] (3.6)

where \( T_1 \) is the time-scale of the turbulent fluctuations and \( T_2 \) is the time-scale of any mean-flow unsteadiness. Equation (3.6) relies upon the mean-flow behavior operating at a time-scale several orders of magnitude higher than that of the turbulent behavior.

The Reynolds averaged Navier-Stokes (RANS) equations can be derived by substituting the flow variables in the form given by expression (3.5) into the incompressible Navier-Stokes equations and making use of equation (3.6). The RANS equations are as follows:

\[ \frac{\partial U_i}{\partial x_i} = 0, \] (3.7)

\[ \frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( t_{ij} - \overline{u'_i u'_j} \right), \] (3.8)

where \( \rho \) is the density of the fluid and the tensor, \( t_{ij} \), is the viscous stress tensor defined as,

\[ t_{ij} = 2\nu S_{ij}. \] (3.9)

Here \( \nu \) is the kinematic viscosity of the fluid and \( S_{ij} \) is the mean rate-of-strain tensor,

\[ S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right). \] (3.10)

The RANS equations have an almost identical appearance to the instantaneous Navier-Stokes equations. The dependant variables in the RANS equations are now the mean velocities and mean pressure instead of the instantaneous values. The other difference is the appearance of the \( -\overline{u'_i u'_j} \) terms. This tensor is known as the Reynolds stress tensor, \( \tau_{ij} \), (also known as the turbulent stress tensor) which represents the influence of turbulent fluctuations on the mean flow. The Reynolds stress tensor is symmetric and so by Reynolds averaging we have created 6 additional unknowns, however since no extra equations have been generated the system is as yet not closed. It is possible to derive differential equations for \( \tau_{ij} \) (the Reynolds stress equations), however this process in turn creates additional unknowns involving triple correlations of the turbulent velocities and pressure. In order to close the system of equations we need
to introduce additional physical principles rather than just performing mathematical operations and in this closure process we must make assumptions and simplify the Reynolds stress tensor.

RANS modelling can be divided into two classes of models: eddy-viscosity models and second order closure models. Eddy-viscosity models approximate the Reynolds stress tensor as a function of the eddy viscosity and the mean rate-of-strain tensor, \( S_{ij} \), whereas second order closure models solve simplified versions of the Reynolds stress equations that are closed using various approximations and assumptions. For a review of second order closure models see Speziale (1991).

### 3.2.3 The Boussinesq approximation

The most common approach to eddy-viscosity modelling is known as the Boussinesq approximation. The Boussinesq approximation assumes that the Reynolds stress tensor is proportional to the mean rate-of-strain tensor \( S_{ij} \), i.e.,

\[
\tau_{ij} = 2\nu_T S_{ij}. \tag{3.11}
\]

In this relationship the constant of proportionality is known as the eddy viscosity, \( \nu_T \). It is important to note here that the eddy viscosity is not a property of the fluid like molecular viscosity, but a property of the flow field and hence will vary throughout the flow domain.

The basis for the Boussinesq approximation comes from analogy with molecular transport where the viscous stresses are proportional to the local velocity gradients (equation 3.9). However whilst the magnitudes of the turbulent stresses definitely depend on the local mean-strain, there is no reason to suppose that the relationship should be so simple. Equation 3.9 is based upon the random motions of the fluid molecules whereas the turbulent eddies that are being modelled with the Boussinesq approximation are fundamentally different and their motions are not completely random, they interact and cannot be described uniquely by the local mean-flow behavior. Moreover, the true representation of the viscous stresses, \( t_{ij} \), involves higher order terms that we have ignored under the assumption that the length scale of molecular motion is very small relative to the mean-flow. However the basis for ignoring the \( O(S_{ij}^2), O(S_{ij}^3), \ldots \) terms from the Boussinesq approximation is less justified since the length scale of the turbulent eddies can be of comparable magnitude to the mean-flow motions. Nevertheless the Boussinesq approximation provides a simple but effective representation of the Reynolds stress tensor that is suitable for a wide range of flows.

Before the Boussinesq approximation can be applied we need to find a prescription for the eddy viscosity. Prandtl (1925) developed the first formation of the eddy viscosity with his mixing-length hypothesis. From analogy with molecular transport he assumed that the eddy viscosity was proportional to a length and a velocity scale, i.e.,

\[
\nu_T = u_{mix} \ell_{mix}. \tag{3.12}
\]

where the mixing length, \( \ell_{mix} \), is a scale for the distance over which the turbulent eddies retain their momentum and the mixing velocity, \( u_{mix} \), can be interpreted as the characteristic velocity of the turbulent eddies.

The three main types of eddy viscosity models are algebraic (or zero-equation) models, one-equation models and two-equation models, all of which provide prescriptions for \( u_{mix} \) and \( \ell_{mix} \) in equation (3.12). Algebraic models use an algebraic specification for the eddy viscosity that is related to mean flow and
geometric properties. One-equation models solve a single partial differential equation (PDE) that is used to describe the transport of a single turbulent scale used in the evaluation of the eddy viscosity often in conjunction with a second scale that is found using algebraic relations. In two-equation models the second turbulence scale is found using a second transport equation.

The Boussinesq approximation has frequently been criticised as a major limitation of eddy-viscosity models. This is largely because it assumes that the Reynolds stress tensor is isotropic, i.e., its axes align with the mean rate-of-strain tensor. However in many flow situations anisotropy effects cannot be neglected and a full second-order closure is necessary. For anisotropic flows many authors have proposed nonlinear modifications to the Boussinesq approximation (Speziale, 1987; Wilcox and Rubesin, 1980). Also, advanced constitutive relationships for the Reynolds stresses have been derived directly from second-order closure models. Such models are called Algebraic Stress Models (ASM) (Abid, Rumsey and Gatski, 1995; Gatski and Speziale, 1993; Speziale, 1997). ASM require only slightly more computational effort than models based on the Boussinesq approximation yet in certain flow situations they can provide considerable improvement when the flow is anisotropic. These flow situations include non-equilibrium flows, separated flows and flows with streamline curvature. These flow features all occur for many sail flows.

3.2.4 Summary of two-equation models

Two-equation models provide an attractive balance between simplicity and accuracy, and in the opinion of the author, these models are the most suitable for the computation of most sail flows. They are the simplest complete models that can be suitably applied to a full range of different flow types. Since there is no immediate requirement for specification of turbulent length scales or calibration of algebraic functions it is easy to apply two-equation models without considering the suitability of the application. However, this may be a mistake since most two-equation models have been designed with a particular application in mind. Consequently they often work well for applications they are calibrated for but are just as likely to present unsuitable results for extraneous applications.

The standard $k-\varepsilon$ model (Jones and Launder, 1972; Launder and Sharma, 1974) has been calibrated using simple flows such as decaying homogeneous turbulence and flat plate boundary layer flows. Consequently it performs well for a wide range of simple flows. However care must be taken when using the model for more difficult flows, especially flows that exhibit adverse pressure gradients and separated regions.

The standard $k-\omega$ model (Wilcox, 1988) has the advantage that it can be integrated through boundary layers without modifications being made to the model. It also performs well for adverse pressure gradients. However, the model is unsuitable for free shear flows (and to a lesser extent also boundary layer flows) due to the dependence of the model on freestream boundary condition for $\omega$ (Wilcox, 1991). In fact, care must be taken when applying freestream boundary conditions with this model for all flows.

Menter's BSL and SST models (Menter, 1993; Menter, 1994) avoid the problems that the standard $k-\omega$ model has with freestream dependency by blending from the $k-\omega$ model near walls into the $k-\varepsilon$ model outside the boundary layer. These models perform well for wall bounded flows while retaining the $k-\varepsilon$ model's average performance for free shear flows. The SST model is designed for adverse pressure gradients and separated flows and consequently performs well for these difficult flow types.
3.3 The turbulence models

3.3.1 The standard \( k-\epsilon \) model

The most commonly used two-equation turbulence model over the last three decades has been the standard \( k-\epsilon \) model (Jones and Launder, 1972; Launder and Sharma, 1974). The model solves two transport equations for the turbulent kinetic energy, \( k \), and the dissipation of turbulent kinetic energy per unit mass, \( \epsilon \). Turbulent kinetic energy describes the kinetic energy of the large-scale eddies and is defined as

\[
k = \frac{1}{2} u_i' u_i' = \frac{1}{2} \left( \overline{u'^2} + \overline{v'^2} + \overline{w'^2} \right).
\]  

(3.13)

The dimensions of \( k \) are \( m^2.s^{-2} \) and thus the turbulent velocity scale of the flow can be represented as \( \sqrt{k} \). The exact equation for \( k \) can be obtained by taking the trace of the Reynolds-stress equations, since

\[
\tau_{ii} = -u_i' u_i' = -2k.
\]

This equation for \( k \) is:

\[
\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \epsilon + \frac{\partial}{\partial x_j} \left[ \nu \frac{\partial k}{\partial x_j} - \frac{1}{2} \overline{u_i' u_i'} - \frac{1}{2} \overline{v_j' v_j'} \right],
\]

(3.14)

where the dissipation per unit mass, \( \epsilon \), is defined by the correlation,

\[
\epsilon = \nu \frac{\partial u_i'}{\partial x_i} \frac{\partial u_i'}{\partial x_k}.
\]

(3.15)

The terms in equation (3.14) can be interpreted as follows:

\[
\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} \text{: The unsteady term and the convective term are collectively known as the material derivative of } k. \text{ This can be described as the rate of change of } k \text{ along a streamline.}
\]

\[
\tau_{ij} \frac{\partial U_i}{\partial x_j} \text{: Production of } k. \text{ The rate at which turbulent kinetic energy is generated due to work done by the mean-flow.}
\]

\[
\epsilon \text{: Dissipation of } k. \text{ The rate at which turbulent kinetic energy is converted to thermal energy through the action of viscosity.}
\]

\[
\frac{\partial}{\partial x_j} \left( \nu \frac{\partial k}{\partial x_j} \right) \text{: Molecular Diffusion of } k. \text{ The diffusion of } k \text{ by the fluids natural molecular transport process.}
\]

\[
\frac{\partial}{\partial x_j} \left( \frac{1}{2} \overline{u_i' u_i'} \right) \text{: Turbulent Transport of } k. \text{ The rate at which turbulent kinetic energy is transported through the fluid by the turbulent fluctuations.}
\]

\[
\frac{\partial}{\partial x_j} \left( \frac{1}{\rho} \overline{v_j' v_j'} \right) \text{: Pressure Diffusion of } k. \text{ Turbulent transport of } k \text{ due to the correlation between velocity fluctuations and pressure fluctuations.}
\]

In order to close equation (3.14) it is necessary to model the unknown correlations of turbulent fluctuations that appear in the turbulent transport and pressure diffusion term. This is achieved in the same fashion as was used in the Boussinesq approximation using the gradient-diffusion approximation which assumes that the turbulent transport of a scalar property relates proportionally to the local mean-flow.
gradient of that property, i.e., $-\frac{1}{2} u_i^j \phi^{ij} \simeq \nu_T \partial \phi / \partial x_j$. Unfortunately there is no equivalent approximation for the pressure-diffusion term and consequently it has become common practice to group these terms together as:

$$\frac{1}{2} u_i^j u_j^i + \frac{1}{\rho} \rho' u_j' = -\frac{\nu_T}{\sigma_k} \frac{\partial k}{\partial x_j},$$

(3.16)

where $\sigma_k$ is a closure coefficient calibrated to simple homogeneous flows. Using this approximation the $k$-equation reduces to:

$$\frac{\partial k}{\partial t} + U_i \frac{\partial k}{\partial x_i} = \tau \frac{\partial U_i}{\partial x_j} - \tau + \frac{\partial}{\partial x_j} \left[ (\nu + \nu_T/\sigma_k) \frac{\partial k}{\partial x_j} \right].$$

(3.17)

The second transport equation that is solved in the standard $k - \epsilon$ model is for the dissipation of turbulent kinetic energy per unit mass, $\epsilon$. The $\epsilon$-equation is based on the exact equation for $\epsilon$, which is found by taking the following moment of the Navier-Stokes equations,

$$2\nu \frac{\partial u_i}{\partial x_j} \frac{\partial}{\partial x_j} [N(u_i)] = 0,$$

(3.18)

where $N(u_i)$ is the Navier-Stokes operator, defined by:

$$N(u_i) = \rho \frac{\partial u_i}{\partial t} + \rho u_k \frac{\partial u_i}{\partial x_k} + \frac{\partial p}{\partial x_i} - \mu \frac{\partial^2 u_i}{\partial x_k \partial x_k}.$$

(3.19)

After considerable algebra the exact equation for $\epsilon$ is obtained:

$$\frac{\partial \epsilon}{\partial t} + U_j \frac{\partial \epsilon}{\partial x_j} = -2\nu \left[ \frac{u_j u_k u_{ij}}{u_{ik}} + \frac{u_k u_{ij}}{u_{ik}} \right] \frac{\partial U_i}{\partial x_j} - 2\nu u_k^j u_{ij} \frac{\partial^2 U_i}{\partial x_j \partial x_k} - 2\nu u_k^j u_{ij} u_{ik} \frac{\partial^2 U_i}{\partial x_j \partial x_k} - 2\nu^2 u_k^j u_{ij} u_{ik} \frac{\partial^2 U_i}{\partial x_j \partial x_k} - 2\nu \frac{\partial^2 u_i}{\partial x_j \partial x_k} + \frac{\partial}{\partial x_j} \left[ \frac{\nu}{\partial x_j} - \nu u_j u_{i,m} u_{i,m} - \frac{2\nu}{\rho} \frac{\partial p}{\partial x_j} u_{ij,m} \right].$$

(3.20)

This equation is much more complicated than the exact equation for $k$ and involves many new unknown second and third-order correlations of fluctuating velocities, pressure and velocity gradients. The dissipation of turbulent kinetic energy occurs in the small eddies where the kinetic energy of smallest motions is converted to thermal energy by the action of molecular viscosity. Hence the exact $\epsilon$-equation describes processes of the small eddies yet we use $\epsilon$ to determine the eddy viscosity, which should really be defined by large-eddy scales. Moreover, in the modelled $\epsilon$-equation, the unknown correlations from equation 3.20 are approximated using expressions based upon the motions of the large eddies, not the small eddies. Therefore we are really using the modelled $\epsilon$-equation to describe the rate of energy transfer from the large eddies to the small eddies. This is suitable since the rate of dissipation to heat is set by the rate at which energy is handed down the eddy cascade. Since the $\epsilon$-equation is really modelling the energy transfer in the large eddy-scales it is not surprising that the modelled equation for $\epsilon$ bears little relation to its exact equation. Consequently, the poor performance of the $k - \epsilon$ model is often blamed on this transport equation for $\epsilon$. Many researchers have searched for adaptations to the $\epsilon$-equation with little success. Historically, researchers that have used alternative forms (using a different turbulence scale) for the second transport equation have received much better results for a wider range of flows. The standard
k – \epsilon model as described by Jones and Launder (1972), is as follows:

**Eddy viscosity:**

\[ \nu_T = C \mu k^2 / \epsilon. \] (3.21)

**Turbulent kinetic energy:**

\[ \frac{\partial k}{\partial t} + U_i \frac{\partial k}{\partial x_i} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \epsilon + \frac{\partial}{\partial x_j} \left[ (\nu + \nu_T) \frac{\partial k}{\partial x_j} \right]. \] (3.22)

**Dissipation rate:**

\[ \frac{\partial \epsilon}{\partial t} + U_i \frac{\partial \epsilon}{\partial x_i} = C_{\epsilon 1} \frac{\epsilon}{k} \tau_{ij} \frac{\partial U_i}{\partial x_j} - C_{\epsilon 2} \frac{\epsilon}{k} + \frac{\partial}{\partial x_j} \left[ (\nu + \nu_T) \frac{\partial \epsilon}{\partial x_j} \right]. \] (3.23)

**Closure coefficients and auxiliary relations:**

\[ C_{\epsilon 1} = 1.44, \quad C_{\epsilon 2} = 1.92, \quad C_{\mu} = 0.09, \quad \sigma_k = 1.0, \quad \sigma_\epsilon = 1.3, \]
\[ \omega = \epsilon / (C_\mu k), \quad \epsilon_{\text{mix}} = C \mu k^{3/2} / \epsilon, \quad u_{\text{mix}} = \sqrt{k}. \] (3.24)

These closure coefficients are calibrated from comparison to free shear flows and decaying isotropic turbulence.

The Reynolds stress tensor, \( \tau_{ij} \), is computed using a slightly different form of the Boussinesq approximation to that given in equation (3.11), here we use:

\[ \tau_{ij} = 2 \nu_T S_{ij} - \frac{2}{3} k \delta_{ij}, \] (3.25)

where \( \delta_{ij} \) is the Dirac delta function (\( \delta_{ij} = 1, \forall i = j, \delta_{ij} = 0 \) otherwise). In equation (3.25) the second term, \( -\frac{2}{3} k \delta_{ij} \), is required to obtain the proper trace of \( \tau_{ij} \) (i.e., \( \tau_{ij} = -2k \)). This is the form of the Boussinesq approximation that is generally used for two-equation models including each of the models that are presented in this thesis.

This model has produced good results for a wide range of simple flows. However, due to its extensive use its deficiencies are well known. The most notable of these is that the standard \( k – \epsilon \) model is inaccurate in the presence of adverse pressure gradients (Wilcox, 1998). For flows that are subject to adverse pressure gradients the \( k – \epsilon \) model is notorious for overpredicting the turbulent length scale within the outer region of the boundary layer (the defect layer) and hence the model also overpredicts the eddy viscosity. As a consequence, separation is often delayed or prevented completely. For aerodynamic flows in high-lift situations this problem can lead to an over-prediction of lift and an under-prediction of drag, thus exaggerating the performance of the airfoils. Considering the poor performance of the standard \( k – \epsilon \) model for adverse pressure gradient flows it is surprising the model is still as popular as it is and has not yet been superseded by a more suitable model.
3.3.2 Low Reynolds Number (LRN) modifications for the $k - \epsilon$ model

The standard $k - \epsilon$ model also fails to give accurate solutions in the viscous sublayer (i.e., the low Reynolds number region of the flow). When near-wall solutions computed using the standard $k - \epsilon$ model are compared to the log-law of the wall (equation (3.1)) it is evident that the model underpredicts the constant $B$. Very close to walls, molecular viscosity damps the tangential velocity fluctuations, while normal fluctuations are reduced directly by the wall itself.

The standard $k - \epsilon$ model requires modifications to account for the influence of the wall and many low-Reynolds-number adaptations have been made to the standard $k - \epsilon$ model (Chien, 1982; Jones and Launder, 1972; Lam and Bremhorst, 1981; Launder and Sharma, 1974). It is important not to confuse these models as models designed specifically for flows at low Reynolds numbers; they are really models that are designed to account for viscous effects near walls where the local turbulent Reynolds number is low. However poor performance of the standard $k - \epsilon$ model is more likely at low Reynolds numbers where the log-law of the wall (equation (3.1)) is less appropriate.

LRN $k - \epsilon$ models use viscous damping functions for the closure coefficients in the $\epsilon$-equations, as well as for the eddy-viscosity itself. These damping functions change the values of the closure coefficients in the near wall region. The damping functions are calibrated in order to reproduce the appropriate value of 5.0 for the constant $B$ in the log-law of the wall (equation (3.1)) and to achieve asymptotic consistency of $k, \epsilon$ and the Reynolds shear-stress in the viscous sublayer region. Effectively, they allow the $k - \epsilon$ model to be integrated through the viscous sublayer. However, they do not directly improve solutions for adverse pressure gradient flows since it is the defect layer where problems arise for the $k - \epsilon$ model, and in the defect layer these LRN adaptations are negligible.

3.3.3 The $k - \omega$ model

In $k - \omega$ models the modelled $k$-equation is solved together with an equation for the specific rate of dissipation of turbulent kinetic energy, $\omega = \epsilon/k$. Commonly $\omega$ is thought of as the characteristic frequency of the turbulent decay process, however more correctly the reciprocal of $\omega$ is the time scale on which dissipation of the turbulent energy occurs. Although dissipation occurs at a molecular level, its actual rate is set by the rate of transfer of energy down the eddy spectrum and therefore $\omega$ is set by the large-scale motions and is closely related to mean-flow properties.

The first ever two-equation turbulence model was the $k - \omega$ model of Kolmogorov (1942). Whilst the Kolmogorov model was a pioneering step in turbulence modelling, his $\omega$-equation possesses several flaws, the most notable being that the model does not contain a term to describe the production of $\omega$. Production of $\omega$ can be interpreted as the effect that the work done by the mean-flow on the turbulent eddy spectrum has on the specific dissipation rate.
The most popular $k-\omega$ model is that of Wilcox (1988) which is commonly referred to as the standard $k-\omega$ model:

**Eddy viscosity:**

$$\nu_T = \frac{k}{\omega}. \quad (3.26)$$

**Turbulent kinetic energy:**

$$\frac{\partial k}{\partial t} + U_i \frac{\partial k}{\partial x_i} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[ (\nu + \sigma_k \nu_T) \frac{\partial k}{\partial x_j} \right]. \quad (3.27)$$

**Specific dissipation rate:**

$$\frac{\partial \omega}{\partial t} + U_i \frac{\partial \omega}{\partial x_i} = \frac{\omega}{k} \frac{\partial U_i}{\partial x_j} - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[ (\nu + \sigma_\omega \nu_T) \frac{\partial \omega}{\partial x_j} \right]. \quad (3.28)$$

**Closure coefficients and auxiliary relations:**

$$\alpha = \frac{5}{9}, \quad \beta = 0.075, \quad \beta^* = 0.09, \quad \sigma_k = 0.5, \quad \sigma_\omega = 0.5, \quad \epsilon = \beta^* \omega k, \quad \ell_{\text{mix}} = k^{1/2}/\omega. \quad (3.29)$$

Unlike the $\epsilon$-equation of the standard $k-\epsilon$ model, the $\omega$-equation in the Wilcox $k-\omega$ model was not derived from an exact equation but rather from dimensional analysis and physical reasoning. Wilcox has illustrated that the model not only performs well for free shear flows and flat plate boundary layer flows, but also for more complicated adverse pressure gradient flows and separated flows (Wilcox, 1998).

The major downfall of the standard $k-\omega$ model is that it has a large dependence on the freestream boundary condition for $\omega$ (Wilcox, 1991; Menter, 1992b). It has been shown that if the value of $\omega$ at turbulent / non-turbulent interfaces is too low then spurious amounts of eddy viscosity appear at the interface, smearing out the turbulence profile (which should be a sharp interface). Reducing the freestream value of $\omega$ towards zero can alter the eddy viscosity within a flat plate boundary layer by as much as 100% (Kok, 1999; Kok, 2000).

The freestream dependence problem can be avoided by applying sufficiently large freestream boundary conditions for $\omega$ or by enforcing a minimum value of $\omega$ throughout the domain. This method works well for boundary layer simulations where the near-wall values of $\omega$ approach infinity and therefore will be not affected by the $\omega$ values in the non-turbulent region. However, this is not a valid solution to the problem for free-shear flows where the $\omega$ values within the turbulent region can be of comparable magnitude to the required $\omega$ values in the non-turbulent region. The topic of freestream dependency will be further discussed in sections 3.3.4 and 3.3.5 where some solutions to the problem are provided.
3.3.4 Comparing the $k - \epsilon$ and $k - \omega$ models

If the modelled $\epsilon$-equation from the standard $k - \epsilon$ model is written in terms of $\omega$ then it can be seen that the $k - \epsilon$ and $k - \omega$ models are similar. The $\epsilon$-equation written in terms of $\omega$ is given as follows:

$$
\frac{\partial \omega}{\partial t} + U_i \frac{\partial \omega}{\partial x_i} = \alpha \frac{\omega}{k} \frac{\partial U_i}{\partial x_j} - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[ (\nu + \sigma \nu_T) \frac{\partial \omega}{\partial x_j} \right] + 2 \left( \frac{\nu + \sigma \nu_T}{k} \right) \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}
$$

where $\beta = C_\mu$ and $\alpha, \beta, \sigma$ and $\sigma^*$ are simple functions of the $k - \epsilon$ model's closure functions. If for simplicity we only consider the behavior of the model outside of viscous regions (where $\nu_T >> \nu$), then the only difference between the $k - \epsilon$ and $k - \omega$ models is the term,

$$
\frac{2 \sigma \nu_T}{k} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}
$$

This term is a cross diffusion term and is often written in the form (Wilcox, 1998):

$$
\frac{1}{\sigma_d} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}
$$

where $\sigma_d$ is a constant. Cross diffusion does not represent any natural flow process and hence the omission of this term from the $k - \omega$ model cannot be viewed as unphysical. Cross diffusion is merely a term that comes about due to a formal change of variables. Several authors have used cross diffusion terms to blend between the $k - \epsilon$ and $k - \omega$ models. For example, in the next section we will see how Menter uses cross diffusion to create a model that behaves like the $k - \omega$ model close to walls and the $k - \epsilon$ model outside the boundary layer. By using cross diffusion away from wall boundaries Menter successfully eliminates the freestream dependency of the $k - \omega$ model.

Wilcox (1998) presents a new $k - \omega$ model that improves upon the standard $k - \omega$ model's accuracy for free-shear flows and reduces the freestream dependency of the model so long as the $\omega$ values at freestream boundaries are close to zero. The new $k - \omega$ model includes new coefficients for the dissipation terms in the $k$- and $\omega$-equations. The new coefficient for the dissipation of $k$ is based upon the cross-diffusion term (equation 3.34) and is designed to enhance the dissipation of $k$ in free-shear and non-turbulent regions of the flow. This serves to remove spurious eddy viscosity levels at turbulent/non-turbulent interfaces such as the outer edge of the boundary layer. Within boundary layers the alterations are designed to have negligible influence. The $k - \omega$ model that is currently implemented in CFX-5 is the standard $k - \omega$ model and so details of the newer version are not presented here.

The standard $k - \omega$ model holds several crucial advantages over the standard $k - \epsilon$ model:

1. It is difficult to specify boundary conditions for both $\epsilon$ and $\omega$ at solid boundaries where $\omega$ is singular and $\epsilon$ is undefined. However, accurate solutions for $\omega$ can still be found when a finite (but large) value of $\omega$ is specified at the wall. This is not possible for the $\epsilon$-equation as there is no obvious algebraic relationship describing the near-wall behavior of $\epsilon$. The application of these wall boundary conditions will be discussed in more detail in section 3.5.2.
2. The $\omega$-equation, as it stands in the standard $k - \omega$ model, holds throughout a turbulent boundary layer and provides an accurate prediction of the constant, $B$, in the low-law of the wall (equation (3.1)). The $\epsilon$-equation requires modification (viscous damping-functions (Lauder and Sharma, 1974)) for accurate integration through the viscous sublayer.

3. The effects of surface roughness can be conveniently incorporated using the wall boundary condition for $\omega$. This will be discussed in more detail in section 3.5.2.

4. The $\omega$-equation gives a much improved defect-layer solution for flows subject to adverse pressure gradients. The $k - \epsilon$ model over-predicts the turbulent length scale in this region and hence also over-predicts the eddy viscosity. This is the reason for the poor prediction of flow separation that is typical for $k - \epsilon$ models.

5. In many flow situations the $\omega$-equation is more stable numerically and less stiff compared with the $\epsilon$-equation, allowing for larger time-steps.

The first two deficiencies of the $\epsilon$-equation are often treated using various near wall treatments. The most common of these are wall functions, where no computations are performed in the near-wall region and instead the solution is forced to asymptote to the log-law of the wall. Wall functions will be discussed further in section 3.5.2. Another common approach is to model the near-wall region with a one-equation model that solves a transport equation for $k$ and finds $\epsilon$ using algebraic functions based on the distance to the wall.

### 3.3.5 Menter's BSL and SST models

Menter (1992$b$) has shown that introducing cross diffusion in the form of equation (3.34) can reduce the freestream dependency of the $k - \omega$ model. The principal effect of cross diffusion in free-shear flows is to enhance the production of $\omega$, which consequently increases the dissipation of $k$. Menter chose to introduce the cross diffusion in a non-uniform fashion by multiplying the cross diffusion term with "blending functions" based upon the distance to the nearest wall (Menter, 1993; Menter, 1994). These functions are zero at the inner edge of a turbulent boundary layer and blend to a unitary value at the outer edge of the layer. Consequently the model behaves like the $k - \epsilon$ model away from walls and the $k - \omega$ model in the near-wall region. In this fashion the good boundary layer performance of the standard $k - \omega$ model is retained whilst gaining the $k - \epsilon$ model's more reliable results for free-shear flows.

Menter developed two new turbulence models based on this zonal approach; the Baseline (BSL) model and the Shear-Stress Transport (SST) model (Menter, 1993; Menter, 1994). The BSL model is identical to the standard $k - \omega$ model in the inner 50% of the boundary layer but changes gradually to the $k - \epsilon$ model (in a $k - \omega$ formulation) over the outer half of the layer. The SST model has identical transport equations to the BSL model, but uses a new definition of the eddy-viscosity that is believed to provide a better representation of the transport of turbulent shear-stress in adverse pressure gradient boundary layers (Menter, 1993). Menter's SST eddy-viscosity relation is based on the assumption that in boundary layer flows the Reynolds shear stress is proportional to the turbulent kinetic energy (Townsend, 1962), i.e.,

$$\tau = a_1 k,$$

(3.35)
where \(a_1 = 0.3\), and \(\tau\) is the Reynolds shear-stress. This relation has been confirmed for a large number of boundary layer experiments (Townsend, 1962). For most two-equation models the relation for the Reynolds stress tensor is given by

\[
\tau_{ij} = 2C_{\mu} \frac{k^2}{\varepsilon} S_{ij} - \frac{2}{3} \frac{k}{\omega} \delta_{ij} = 2 \frac{k}{\omega} S_{ij} - \frac{2}{3} k \delta_{ij}.
\]  (3.36)

This can be rewritten to give the Reynolds shear stress,

\[
\tau = \sqrt{\frac{\text{Production}_k}{\text{Dissipation}_k} a_k k},
\]  (3.37)

since

\[
\text{Production}_k = \tau_{ij} \frac{\partial U_i}{\partial x_j} = 4 \frac{k}{\omega} S_{ij}^2,
\]  (3.38)

and

\[
\text{Dissipation}_k = \beta^* k \omega = a_k^2 k \omega.
\]  (3.39)

For equilibrium boundary layers, production of turbulent kinetic energy matches its dissipation and therefore two-equation models predict the Reynolds shear-stress to be equivalent to equation (3.35). However, for adverse pressure gradients the ratio of production to dissipation in the \(k\)-equation can be significantly larger than one. There is no experimental evidence that the ratio of \(|\tau|/k\) increases to values higher than \(a_1\), therefore equation (3.36) is unsuitable for adverse pressure gradient flows. Consequently, Menter postulated that a more appropriate form for the eddy viscosity is to use equation (3.35) in adverse pressure gradients, and the standard relation (equation (3.36)) otherwise. This led to the SST expression (equation (3.41)) in the SST model.

The BSL and SST models are given by:

**Eddy viscosity:**

- **BSL:** \(\nu_T = \frac{k}{\omega}\).  
- **SST:** \(\nu_T = \frac{a_k k}{\max(a_1 \omega; |S_{ij}|/F_2)}\).  

**Turbulent kinetic energy:**

\[
\frac{\partial k}{\partial t} + U_i \frac{\partial k}{\partial x_i} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[ (\nu + \sigma_k \nu_T) \frac{\partial k}{\partial x_j} \right].
\]  (3.42)

**Specific dissipation rate:**

\[
\frac{\partial \omega}{\partial t} + U_i \frac{\partial \omega}{\partial x_i} = \alpha \frac{\tau_{ij}}{k} \frac{\partial U_i}{\partial x_j} - \beta \omega^3 + \frac{\partial}{\partial x_j} \left[ (\nu + \sigma_\omega \nu_T) \frac{\partial \omega}{\partial x_j} \right] + (1 - F_1) 2 \sigma_d \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}.
\]  (3.43)

**Closure coefficient and auxiliary relations:**

Let \(\phi\) represent the set of closure constants for the BSL and SST model and let \(\phi_1\) and \(\phi_2\) represent the constants from the standard \(k-\omega\) and \(k-\varepsilon\) models respectively.


Set 1, \( \phi_1 \) (standard \( k - \omega \)):

\[
\beta_1 = 0.075, \quad \beta^* = 0.09, \quad \sigma_{k1} = 0.5, \quad \sigma_{\omega 1} = 0.5, \quad \sigma_{d1} = 0.856, \quad \kappa = 0.41, \quad \alpha_1 = 5/9.
\] (3.44)

Set 2, \( \phi_2 \) (standard \( k - \epsilon \)):

\[
\beta_2 = 0.0828, \quad \beta^* = 0.09, \quad \sigma_{k2} = 1, \quad \sigma_{\omega 2} = 0.856, \quad \sigma_{d2} = 0.856, \quad \kappa = 0.41, \quad \alpha_2 = 0.44.
\] (3.46)

(3.47)

In Menter's models the constants \( \phi \) are calculated using the following blend between the constants \( \phi_1 \) (\( k - \omega \)) and \( \phi_2 \) (\( k - \epsilon \)):

\[
\phi = F_1 \phi_1 + (1 - F_1) \phi_2, \quad \ldots \quad (3.48)
\]

where

\[
F_1 = \tanh (\text{arg}_1^1), \quad \ldots \quad (3.49)
\]

and

\[
\text{arg}_1 = \min \left[ \frac{\sqrt{k}}{\beta^* \omega y} \cdot \frac{500 \nu}{y^2 \omega} \cdot \left( \frac{4 \rho \sigma_{\omega 2} k}{C D_{k \omega} y^2} \right) \right]. \quad \ldots \quad (3.50)
\]

Here \( y \) is the distance to the nearest surface and \( C D_{k \omega} \) is the positive portion of the cross diffusion term, i.e.,

\[
C D_{k \omega} = \max \left( 2 \rho \sigma_d \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} ; 10^{-10} \right). \quad \ldots \quad (3.51)
\]

The blending function for the eddy-viscosity relation in the SST model is defined,

\[
F_2 = \tanh (\text{arg}_2^2), \quad \ldots \quad (3.52)
\]

where

\[
\text{arg}_2 = \max \left( 2 \frac{\sqrt{k}}{\beta^* \omega y} \cdot \frac{500 \nu}{y^2 \omega} \right). \quad \ldots \quad (3.53)
\]

These blending functions have been derived through calibration with free-shear and boundary layer flows.

Compared with the standard \( k - \omega \) model, the BSL and SST models have been shown to have considerably decreased dependency on freestream boundary conditions (Menter, 1993; Menter, 1994; Menter, 1996). The SST model in particular has gained good results for flows involving adverse pressure gradients and separation (Menter, 1993; Menter, 1994; Menter, 1996). This is primarily due to the modified eddy-viscosity relation (equation (3.41)), which is the only difference between the SST and BSL models.
3.4 Turbulence modelling issues for downwind sail flows

The features of downwind sail flows that present notable difficulties for turbulence models are:

1. The leading edge separation bubble.
2. Transition (this occurs in the outer shear layer of the leading edge bubble).
3. Recovery of the turbulent boundary layer downstream of reattachment of the leading edge bubble.
4. Trailing edge separation due to the adverse pressure gradient on the suction surface of the genaker.
5. Unsteady vortex shedding within the wake.

Downwind sails operate like high-lift airfoils and in their design the lift force is maximised whilst little attention is paid to the drag force. Modelling flows past high-lift foils is particularly difficult due to the high angles of attack that they operate at, and the adverse pressure gradients and separated zones that are commonly present. High-lift airfoils are commonly used for landing and take-off configurations of aeroplanes and for rear wings on racing cars. Consequently, there have been many studies regarding the performance of turbulence models for high-lift airfoils.

3.4.1 Validation of turbulence models for high-lift configurations

For high-lift applications the one-equation Spalart-Allmaras model (Spalart and Allmaras, 1994) and Menter's SST model have frequently stood out amongst RANS models. These two models seem to model adverse pressure gradients and separation better than most other models. Godin, Zingg and Nelson (1997) compared these two models using several high-lift flows. They discovered that for attached flows both models gave excellent results, however for separated flows the SST model was able to model the recirculation region with higher accuracy. The SST model is a two-equation model and can therefore be expected to suitably model a wider range of flow situations than the one-equation Spalart-Allmaras model. They conclude that the Spalart-Allmaras model is more robust, especially for attached flow over multi-element foils. This is due to the transport equation for the eddy-viscosity which has the convenient wall boundary condition, $\nu_T = 0$, and hence does not involve the surface boundary condition problems of $\epsilon$ and $\omega$. Therefore, the Spalart-Allmaras model could possibly be more suitable for upwind sail flows where the flow remains predominantly attached.

Kim, Kim and Rho (2000) computed flows over single and multi-element foils using the standard $k-\epsilon$, standard $k-\omega$ and SST models. The SST model consistently outperformed the other models and in particular the model gave much better velocity profiles, especially within separated regions.

Recently several Algebraic Stress Models (ASM) have been used for high-lift flows, with moderate success. Abid et al. (1995) computed several high-lift flows using the explicit algebraic stress equation derived by Gatski and Speziale (1993) and made comparison to solutions computed with the standard $k-\epsilon$ and standard $k-\omega$ models. The ASM models were found to provide slight improvement for high-lift flows which was attributed to their ability to account for anisotropy. However in this study the choice of the form of the two-equation model (i.e. $k-\epsilon$ or $k-\omega$) had a more dramatic influence on the results, with the $k-\omega$ based ASM outperforming the $k-\epsilon$ ASM. Rumsey, Gatski, Ying and Bertelrud (1998)
compared the Spalart-Allmaras model, the SST model and the explicit ASM of Speziale (1997) (in a $k-\omega$ formulation) on two multi-element high-lift airfoil systems. Although some variations between the models are evident, the non-linear ASM predicts similar lift and drag coefficients to the linear eddy-viscosity models. These airfoils exhibit no boundary layer separation, and the effects of anisotropy of the Reynolds stress tensor can be expected to be minimal. For flows at much higher angles of attack with high streamline curvature and large separation regions the anisotropy of the Reynolds stress tensor is more likely. Such a situation occurs in downwind sailing and where ASM may provide some improvement over the SST and Spalart-Allmaras models, however any improvement would be small and are unlikely to outweigh the added complexity involved with the ASM formulation.

High-lift flows are demanding and challenge the assumptions used in the derivation of two-equation models. Downwind sail flows are even more challenging. Whilst both types of flow operate at similar high angles of attack, downwind sails generally have much larger camber and do not have separate elements and slots to help sustain attached flow. Consequently, the flow around downwind sails is more likely to be in non-equilibrium and to be anisotropic. Therefore a minimum requirement for a turbulence model that is to be used for downwind sails is that it should be accurate for high-lift flows. However even models - such as the SST model - that perform well for these flows may struggle to resolve the complicated unsteady flow field around downwind sails.

### 3.4.2 Validation of the CFX-5 turbulence models for the NACA 4412 airfoil at maximum lift

Validation studies carried out by CFX International have illustrated that there is a wide range in the performance of the available turbulence models (Carrega-Ferreira et al., 2001; Langtry, Kuntz and Menter, 2003). Of particular interest is the validation for the NACA 4412 airfoil at maximum lift (see Figure 3.3). This test case is commonly used for turbulence model validation due to the availability of good experimental results (Coles and Wadcock, 1979) and the complexity of the flow which involves an adverse pressure gradient on the suction side of the airfoil and flow separation near the trailing edge.

![Figure 3.3: Streamlines for the flow past the NACA 4412 airfoil at maximum lift (13.87°). The simulation was computed by the Author using CFX with the SST model.](image)

A validation study (Carrega-Ferreira et al., 2001) for the NACA 4412 test case has been conducted by CFX comparing the SST model with the standard $k-\omega$ model and the Algebraic Stress Model (ASM) of Abid et al. (1995). In Figure 3.4 the pressure coefficient plots from experimental results are compared with the CFD results.
Figure 3.4: Comparison of experimental results of the pressure coefficient, $C_p$, with different turbulence models for the NACA 4412 airfoil (Carrega-Ferreira et al., 2001).

The CFD results are all in good agreement with the experimental data except near the trailing edge where flow separation occurs. In this region the models all overpredict the pressure coefficient with the standard $k - \epsilon$ model predicting larger pressure coefficient values than the other models. Detail of the trailing edge region is provided in Figure 3.5.

Figure 3.5: Trailing edge detail of the pressure coefficient for the NACA 4412 airfoil (Carrega-Ferreira et al., 2001).

Further insight into the performance of the turbulence models can be gained by looking at the velocity profiles illustrated in Figure 3.6. A schematic illustrating the position of the velocity profiles from Figure
3.6 is illustrated in Figure 3.7.

Figure 3.6: Streamwise velocity profiles computed using several different turbulence models for the NACA 4412 airfoil (Carrega-Ferreira et al., 2001).

Figure 3.7: Positions of the six boundary layer traverses for the NACA 4412 test case.

In Figure 3.6 the differences between the turbulence models is quite clear; the SST model accurately predicts the location of the separation point and does a good job in reproducing the velocity profiles in the separation zone, whereas the $k - \varepsilon$ model omits separation entirely. The performance of the ASM model lies between the results for the other two models.

### 3.4.3 Unsteady RANS

The application of RANS modelling to unsteady flows such as downwind sail flows is trivial yet often misunderstood. The common misconception is that because we are dealing with averaged equations there can be no unsteadiness. However the vortical structures shed from downwind sails and other bluff bodies are not at all turbulent phenomena but rather are statistical mean-flow phenomena and so there is no reason why the RANS equations are unsuitable for unsteady flows. When we simulate using the RANS we are not calculating the true flow field or time history, but instead we are simulating a history of the mean flow including the effects of turbulence. If you were to look at the frequency spectrum of an unsteady flow there would be a wide band of noise that is the turbulence as well as spikes that represent...
the periodic unsteadiness of the mean-flow. If the mean-flow unsteadiness is of the same order as the turbulence motions then it will be get damped by the turbulent stresses. In such a situation LES models are more appropriate. However for many turbulent flows including sail flows the mean-flow unsteadiness occurs at a scale that can be captured with time-steps and grid sizes still significantly greater than the relevant scales of the turbulence spectrum.

An example of an unsteady flow that has commonly been simulated inappropriately using steady state simulations is the surface mounted cube (Iaccarino, Ooi, Durbin and Behnia, 2003). This test case was selected for the "Workshop on Large Eddy Simulation of Flows Past Bluff Bodies" (Rod, 1997) where LES calculations were shown to provide significantly better results than most RANS simulations. However in those cases RANS simulations were computed to an artificial steady state by damping the unsteady structures using unsuitably large time-steps. Iaccarino et al. (2003) showed that by appropriately selecting the time-step sizes and time stepping methods, unsteady solutions can be produced for this test case that significantly improve on steady state RANS simulations and in fact rival the LES results in accuracy. When simulating unsteady flows with the RANS equations it is imperative that a suitable time-step analysis is employed to ensure that simulation accuracy is independent of the time-step size.

3.4.4 Suitability of the CFX-5 two-equation turbulence models for downwind sail flows

There are five two-equation turbulence models currently implemented in CFX-5. They are:

- Standard $k - \epsilon$
- RNG $k - \epsilon$
- Standard $k - \omega$
- BSL $k - \omega$
- SST $k - \omega$

Of these models only the RNG $k - \epsilon$ model has not as yet been mentioned. The RNG $k - \epsilon$ model was developed by Yakhot and Orszag (1986) using renormalisation group (RNG) analysis of the Navier-Stokes equations. The model is essentially the same standard $k - \epsilon$ model, however the RNG version uses slightly different closure coefficients and an additional source term that is dependent on the magnitude of the local mean strain-rate. The RNG $k - \epsilon$ model has been shown to provide small improvements upon the standard $k - \epsilon$ model for some flows, however the model cannot be considered as a significant advance upon the standard $k - \epsilon$ model (Collie, 2000). Consequently, the RNG $k - \epsilon$ model was not considered in the current research.

Similarly, the BSL $k - \omega$ model was not considered since it is very similar to the SST $k - \omega$ model, and particularly for flows involving adverse pressure gradients and flow separation, the SST model has been shown to outperform the BSL model (Menter, 1994). The standard $k - \epsilon$ model, the standard $k - \omega$ and the SST $k - \omega$ model are quite different from each other, which warrants their comparison for downwind sail flows. Whilst the SST model has been shown through the literature to be the most reliable
performer for separated flows, it is also the most complicated model, and it is necessary to determine how well the simpler $k - \varepsilon$ and $k - \omega$ models compare. Unfortunately there is no low Reynolds number implementation of the $k - \varepsilon$ model in CFX-5, and consequently the standard $k - \varepsilon$ model must be used with wall functions. The standard $k - \varepsilon$ model is not suitable for modelling downwind sail flows, and its inclusion in the current research serves purely as a reference point for the other models.

### 3.5 Description of the CFD software

The CFD simulations carried out in this research were computed using CFX-5, an unstructured commercial CFD package supplied by ANSYS Inc (CFX-International, 2003). The computational grids were generated in ICEM-HEXA, a block-structured grid generation package also provided by ANSYS Inc (ICEM-CFD-Engineering, n.d.).

#### 3.5.1 Grid generation

The base computer aided design (CAD) geometry for all grids used in this project was generated in Solidworks, a solid modelling package that has a direct interface with ICEM. In Solidworks surfaces are created to describe the objects being modeled (i.e. the sail surfaces), the domain boundaries and the internal block boundaries. In all cases the base geometry was carefully developed to ensure the flow was as close as possible to being orthogonal to the grid throughout the domain. In particular the block boundaries that intersect sail geometry and the external boundaries were kept orthogonal in order to minimise the numerical error in these regions.

The CAD surfaces are imported to ICEM-HEXA where the grids were developed. Inside ICEM-HEXA a global block is automatically generated to encompass the CAD. This global block is then subdivided into smaller blocks through a series of cuts. Once the grid is divided into the desired block structure it is mapped to the CAD geometry by associating vertices, edges and surfaces in the grid to their corresponding entities in the geometry. This technique is illustrated in Figure 3.8, where a schematic of a sail grid is presented.

The number of nodes and the distribution of those nodes are specified along each block-boundary. This process is done carefully to ensure adequate grid densities in important regions of the flow (such as in boundary layers and wakes), whilst keeping expansion ratios small and skewness to a minimum. The grids are then generated automatically with structured grids created in each block and the entire grid patched together to form a global block-structured grid. In the case of the sail grids the global grid was itself structured and in ICEM it was possible to convert the multiple-block grid to a single-block grid and re-order the storage of the nodes in order to improve computational efficiency. However this procedure would have not provided any improvement in solver efficiency since CFX-5 is an unstructured solver and so CFX itself would always convert the grid storage back to an unstructured array.

CFX-5 is a three-dimensional solver and hence the grids that were used were required to be three-dimensional. Most of the simulations carried out in this project were for two-dimensional flows and correspondingly the grids used were essentially two-dimensional. These grids have just one cell in the third dimension with symmetry conditions prescribed on the top and bottom boundaries of the domain.
One particularly attractive advantage in using *ICEM* was that the grid structure and CAD geometry are stored as separate entities. This allowed rapid regeneration of grids when small changes were made in the geometry. For example, in the design study 24 different sail shapes were generated but the base block structure remained the same. Instead of regenerating grids for each new sail it was possible to merely alter the sail geometry in *Solidworks* and then map the original grid structure to the new geometry. This worked well provided the geometry did not vary greatly and degrade the quality of the grid. Occasionally the node spacings and expansion ratios had to be adjusted in order to restore the grid quality for a new sail shape.

### 3.5.2 Pre-processing

Boundary conditions, initial conditions, fluid properties and the solver setup were implemented in *CFX-Pre*, the preprocessor for *CFX-5.6*. Prior to the release of *CFX-Pre (beta)* in January 2003, preprocessing was performed in *CFX-Build*, the grid generator and preprocessor for *CFX-5.5*.

**BOUNDARY CONDITIONS**

**Inlet**

Cartesian velocity components can be specified directly at boundaries, i.e.,

\[
U_{\text{Inlet}} = U_{\text{specified}} + V_{\text{specified}} + W_{\text{specified}} k. \tag{3.54}
\]

This approach was used initially, largely because it is the most straightforward and obvious. However, it was found that since the pressure, \( p \), at the boundary remains a component of the solution it was possible to get a non-uniform total pressure at the boundary and consequently the incorrect freestream Reynolds number if the boundary was positioned too close to the sail. For highly cambered downwind sail sections the influence of the sail on the pressure field was significant several chord lengths (as many
as ten chord lengths) upstream of the model. Consequently it was found to be more suitable to specify the total pressure and the flow direction at the inlet boundary, i.e.,

$$p_i = p + p_d,$$  
(3.55)

where $p_i$ is the total pressure, specified at the boundary, $p$ is the static pressure of the fluid which is part of the solution at the boundary, and $p_d$ is the fluids dynamic pressure,

$$p_d = \frac{1}{2} \rho |U|^2,$$  
(3.56)

which is also part of the solution at the boundary. An additional constraint is applied upon the solution that forces the velocity of the fluid to point in a specified direction. Therefore a direction and the total pressure need to be specified, and in all cases $p_i$ was calculated from the dynamic pressure in the freestream where the static pressure is zero (i.e. $p_{i(\text{inlet})} = p_{d(\text{freestream})}$).

It was found that solutions were significantly less dependent on domain size when using the total pressure boundary condition. When the Cartesian velocity component method was used the stagnation pressure was equal to the total pressure at the point where the stagnating streamline crossed the inlet boundary. Since the static pressure is not guaranteed to be zero at the inlet this method could potentially produce stagnation pressures higher than the desired freestream dynamic pressure, resulting in $C_P > 1$ at the stagnation point.

Specification of the turbulence quantities, $k$ and $\omega$ are also required at the inlet. However in all the simulations computed during this research the background turbulence levels were assumed to be low. In regions of zero mean-strain the eddy-viscosity will naturally decay and therefore introducing background turbulence is difficult and generally requires modification to the production terms in the turbulence models themselves. In the case of our simulations, if the turbulent kinetic energy at the inlet is sufficiently small then it will always decay to zero by the time the flow reaches the sail. Setting high levels of turbulence at inlets is unwise since it will set up a gradient of background turbulence along the length of the domain making it difficult to determine the actual background turbulence levels in the vicinity of the object being modelled.

In all simulations carried out during this research the turbulence quantities were set via the specification of the turbulence intensity, $I$, and the turbulence length scale, $l$ (from which $k, \omega$ and $\varepsilon$ are calculated). For the simulations performed that emulate wind tunnel experiments the turbulence intensity was set at the turbulence intensity of the tunnel and the length scale was set based on the mesh size of the turbulence screens. In all cases the turbulence intensity was small ($<1\%$) and in this range the solutions were independent of the actual values of the turbulence quantities at the boundary.

**Outlet**

At all outlets the static pressure is set at zero. A constant gradient constraint is imposed on turbulence quantities and the velocity components at the boundary remain part of the solution.

**Wall**

All sail surfaces are modelled as hydraulically smooth with zero thickness and zero permeability. For
viscous flows the wall boundaries must have the no-slip condition and hence,

$$U_{wall} = 0 \text{ and } k_{wall} = 0,$$  \hspace{1cm} (3.57)

must be enforced at the boundary nodes.

The behavior of the dissipation of turbulent kinetic energy, $\omega$, in the viscous sublayer, as derived from a perturbation analysis of the boundary layer (Wilcox, 1998) is:

$$\omega = \frac{6\nu}{\beta y^2}, \quad \text{for } y^+ < 2.5,$$  \hspace{1cm} (3.58)

where $\beta = 0.075$. It is difficult to pose a wall boundary condition on $\omega$ since it is proportional to $1/y^2$ which suggests that $\omega$ is infinite at the boundary. The most common approach to circumvent this problem of singular wall values is to use wall functions where the log-law of the wall (equation (3.1)) is used to define the velocity and turbulence conditions at the first grid point out from the wall. Wall function implementations generally remove the wall neighboring cells from the finite volume computation, thus eliminating the need for wall boundary conditions.

The CFX-5 implementation follows the approach of Menter (1992a) where the wall values of $\omega$ are set according to:

$$\omega_{wall} = \frac{6\nu}{\beta (\Delta y)^2} \quad \text{at } y = 0,$$  \hspace{1cm} (3.59)

where $\Delta y$ is the normal distance between the wall and the first grid point out from the wall. This formulation sets the wall value of $\omega$ at the same value as $\omega$ should be at the first grid point out from the wall according to the analytic sublayer solution for $\omega$ (equation (3.58)). Alternatively, this implementation can be interpreted as placing the physical position of the wall a distance $\Delta y$, inside the grid boundary and the grid boundary is interpreted by the solver as being positioned at $y = \Delta y$. This concept is illustrated in Figure 3.9. For small $\Delta y$ this is suitable as it leads to a large wall value of $\omega$ (close to infinity) and merely causes a very small error ($O(\Delta y)$) in the displacement thickness. Across the first control volume out from the wall the equation for $\omega$ is deleted and replaced by the analytic sublayer equation to ensure that the correct near-wall behavior is enforced.

![Figure 3.9: A schematic of the wall boundary treatment.](image)

The CFX-5 wall boundary condition implementation provides a convenient method of blending to a
wall function approach for grids with large $y^+$ values. In regions of the grid where the first grid point is positioned outside the viscous sublayer wall function boundary conditions are prescribed. In this fashion a grid can be designed so that the solver will integrate through the viscous sublayer in some regions of the flow and use wall functions in other regions (e.g. where the pressure gradients are small). The switch is made to wall functions at $y^+ = 11$ which is the intersection between the viscous sublayer profile $(U^+ = y^+)$ and the log-law of the wall $(U^+ = \frac{1}{k} \ln y^+ + B)$. Results have shown that this is the lowest $y^+$ value for which wall functions can be applied reliably (Grotjans and Menter, 1998).

For grids (or in regions of the grid) where $y^+ > 11$, the geometric wall nodes are treated by the solver as being at $y^+ = 11$, i.e., the boundary nodes are treated as if they were at the outer edge of the viscous sublayer, and the variable values ($U$, $k$ and $\omega$) are applied using a wall function approach. Therefore even if the first grid point is positioned further out than $y^+ = 11$ the wall function model still treats the wall boundary as being at the outer edge of the viscous sublayer. The CFX-5 documentation recommends that the first grid point out from the wall should be positioned so that $20 \leq y^+ \leq 100$ when wall functions are used and $y^+ \leq 2$ when a low Reynolds number formulation is desired (CFX-International, 2003).

This method of applying the wall functions has the advantage that the boundary condition is independent of grid spacing provided the near wall grid spacing is greater than $y^+ = 11$. However for grids with near wall spacings of $y^+ < 11$ the boundary condition is dependent on the grid due to the appearance of $\Delta y$ in equation (3.58). For all simulations used in this research the grids used have $y^+ \approx 1$ and at such low $y^+$ values the near-wall spacing has negligible impact on the solution. It has been shown that for rough surfaces the effect of roughness can be accounted for in the solution by using finite values of $\omega$ at the surface (Saffman, 1970). Higher values of $\omega_{wall}$ are equivalent to smoother surfaces and the limit $\omega_{wall} \to \infty$ corresponds to perfectly smooth walls. Therefore boundary conditions that use sufficiently high $\omega_{wall}$ can be used to represent hydraulically smooth surfaces for which changes in roughness have negligible influence on the flow (Schlichting, 1997). Internal validation studies carried out at CFX (Carrega-Ferreira et al., 2001; Menter, 2003) as well as validation studies performed for the 2nd AIAA Drag Prediction workshop (Langtry et al., 2003) have indicated that the wall boundary condition implementation in CFX-5 does not introduce any spurious grid dependency provided a low value of $y^+$ is used (i.e. $y^+ < 2$). For all simulations carried out in this work grid convergence studies were carried out in order to quantify the error introduced due to the grid and the numerical method.

**INITIAL CONDITIONS**

For all simulations the solutions were either started from quiescent flow conditions or initial solutions were interpolated from a simulation with similar conditions, e.g. solutions using a different grid resolution, turbulence model or a different angle of attack.

**FLUID CONDITIONS**

The fluid properties used in all simulations were constant with:

$$p = 1.2047 \text{kg.m}^{-3} \quad \text{and} \quad \mu = 1.8205 \times 10^{-5} \text{kg.m}^{-1} \text{s}^{-1}. \quad (3.60)$$
These values correspond to that of air at atmospheric pressure and a temperature of 20°C. In all cases the appropriate Reynolds number for the flow simulation was set via the velocity or total pressure at the inlet.

3.5.3 The solver

The finite volume approach

In the CFX-5 solver the RANS equations are linearised and discretised using a conservative and time-implicit finite volume method (CFX-International, 2003). The approach is fully unstructured with the control volumes constructed around each grid node, \( i \), where the flow variables are stored. The momentum, mass and turbulent fluxes are calculated at the centre of each face (the integration points), and the discrete equations are formulated as a balance of the fluxes into each control volume. The control volume approach used in CFX-5 is illustrated in Figure 3.10. Note that Figure 3.10 is a two-dimensional view of what is really a three-dimensional control volume.

![Figure 3.10: The control volume approach.](image)

The linear equation solver

The linear system of equations that arises from the sum of the discrete equations from all of the control volumes in the grid can be written in the form:

\[
\sum_{nb_i} a_i^{nb} \phi_i = b_i, \tag{3.61}
\]

where \( \phi \) is the solution vector, \( b \) is the right hand side (known values generated from the boundary conditions), \( a \) represents the coefficients from the discrete control volume equations, \( i \) is the identifier for the current control volume or node and \( nb \) represents the group of neighboring nodes that are involved in the discretisation stencil of \( \phi_i \). For the coupled three-dimensional system of equations each \( a_i^{nb} \), \( \phi_i \) and
have multiple components, i.e.,

$$a_{tb}^{nb} = \begin{bmatrix} a_{uu} & a_{uv} & a_{uw} & a_{up} \\ a_{uv} & a_{vv} & a_{uw} & a_{vp} \\ a_{uw} & a_{uw} & a_{ww} & a_{wp} \\ a_{pu} & a_{pu} & a_{pw} & a_{pp} \end{bmatrix},$$

(3.62)

$$\phi_i = \begin{bmatrix} u \\ v \\ w \\ p \end{bmatrix},$$

(3.63)

and

$$b_i = \begin{bmatrix} b_u \\ b_v \\ b_w \\ b_p \end{bmatrix},$$

(3.64)

The transport equations for the turbulence variables are coupled together, but are solved uncoupled from the mass and momentum equations. The turbulence equations are only weakly linked to the mass and momentum equations (via $\nu_T$), whilst being more strongly linked to each other. Therefore, this technique produces only a small loss in robustness and efficiency of the solution process, whilst reducing the memory requirements of the linear equation solver significantly.

Equation (3.61) is solved using an iterative solver accelerated using an incomplete lower upper (ILU) factorisation technique and an additive correction algebraic multigrid (AMG) method. Details of these techniques and their implementation can be found in (Raw, 1996).

**Interpolation schemes**

The finite volume discretisation of the governing equations involves the calculation of the fluxes across each face of the control volume, however the variables are only stored at the element nodes. Therefore to complete the discretisation we need to interpolate $\phi_{ip}$, the value of $\phi$ at each of the integration points from the values at the elements nodes, $\phi_n$. For the diffusion terms this is achieved using a second-order accurate central interpolation between the current node ($\phi_i$ i.e. $\phi$ at the element node at the centre of the current control volume) and the nearest neighboring node ($\phi_n$) to the integration point.

For the advection terms the interpolation of $\phi_{ip}$ is biased towards the upstream nodes taking advantage of the fact that transported variables convect with the flow direction. The most basic interpolation scheme for advection terms is the first-order upwind scheme where the variable value at the integration point is taken to be the same as the value at the upstream node, $n$, i.e.,

$$\phi_{ip} = \phi_n.$$

(3.65)

By directly linking the integration point with a solitary upstream node stability is obtained at the expense of accuracy. Higher order accuracy can be gained by using interpolation functions that depend
on more nodes (i.e. a wider discretisation stencil). For example second-order accuracy can be obtained by interpolating \( \phi_{ip} \) from \( \phi_n \) with a correction based upon the gradient of \( \phi \) at \( n \) (i.e., \( \nabla \phi_n \)). The interpolated is carried out in the following form,

\[
\phi_{ip} = \phi_n + \psi \phi_{NAC},
\]

where \( \psi \) is a blending parameter and \( \phi_{NAC} \) is the Numerical Advection Correction (NAC) term.

\[
\phi_{NAC} = \nabla \phi_n \cdot \Delta s.
\]

In the NAC term \( \Delta s \) is the distance between the upstream node, \( n \) and the integration point, \( ip \), and \( \nabla \phi_n \) is the gradient of \( \phi \) at \( n \),

\[
\nabla \phi_n = \frac{1}{V} \sum_f \phi_f A,
\]

where \( \phi_f \) is the face value of \( \phi \) computed using an average of \( \phi \) at the nodes neighboring the face, \( f \), \( A \) is the area of that face and \( V \) is the volume of the control volume surrounding \( \phi_n \).

CFX-5 allows the blending parameter, \( \psi \), to be set by the user. Setting \( \psi = 0 \) corresponds to the upwind scheme with first order accuracy and \( \psi = 1.0 \) corresponds to a second-order accurate interpolation scheme. The second-order accurate scheme is able to capture sharp gradients with less numerical smoothing (since there is less artificial viscosity in a second-order scheme), but is more prone to instability and dispersive errors.

CFX-5 also offers a non-linear interpolation scheme where \( \psi \) is non-uniform and controlled by a limiter function that bounds the NAC term to prevent any new maxima or minima being introduced to the solution. In this way the interpolation scheme retains second-order accuracy in regions where the flow gradients are small but reverts to the monotonic upwind scheme (which is first-order accurate) in the vicinity of shocks. This interpolation scheme is referred to as the high resolution scheme and is similar to that of Barth and Jesperson (1989).

**Solver summary**

The sequence of the solution process is carried out as follows:

1. The mass and momentum equations are discretised and linearised.
2. The system of linear equations for mass and momentum are solved.
3. The turbulence equations are discretised and linearised.
4. The system of linear equations for turbulence are solved.
5. The eddy viscosity, \( \nu_T \), is calculated for the next iteration (for eddy viscosity models).
6. The solution is advanced in time.
The steady and unsteady solution techniques

Steady state simulations are performed by advancing the solution through time until the flow converges to a steady state. Such a simulation does not represent the true transient behavior since the equations solved at each timestep are linearised. CFX-5 provides the option to use a transient solver for unsteady problems. The transient solver performs a series of inner loop iterations (i.e. the coefficient loop, steps 1-5) in order to update the non-linear coefficients before the solution is advanced in time (the outer loop). The CFX documentation recommends the use of 3 coefficient loop iterations for each outer loop iteration (CFX-International, 2003). In the initial stages of the project a convergence study was carried out in order to determine the most appropriate number of coefficient loops for unsteady downwind sail simulations. Simulations were carried out using 2, 4, 8 and 16 coefficient loops and it was found that there was little advantage gained from using more than 4 coefficient loops and that refining the timestep size proved to be a more efficient means of reducing non-linearity errors. For all transient simulations performed in this research 4 coefficient loops were used.
Chapter 4

The Flat Plate at Shallow Incidence

4.1 Introduction

This validation study investigates the performance of the standard $k-\omega$ and SST $k-\omega$ turbulence models for the two-dimensional flow past the flat plate at shallow angles of incidence. The flow past the flat plate poses significant relevance to sail design due to the formation of a separation bubble at the leading edge of the type that regularly appears in flows past sails. This type of leading edge bubble is most commonly found in flows past yacht sails but is also often present for flows past some thin airfoils such as those used for turbine blades.

In this study computational results computed with CFX-5 are compared with wind tunnel results compiled by Crompton (2001) and particular attention is paid to the structure of the leading edge bubble and the recovery of the turbulent boundary layer downstream of the reattachment point. The relatively recent experimental data compiled by Crompton presents a new and demanding test case for turbulence models. The notable features of the flow field are unsteady shear layer flapping, shear layer transition, relaminarisation, secondary recirculation, flow reattachment, and post-reattachment boundary layer recovery. Each of these flow features directly challenges the assumptions from which many turbulence models are derived.

Leading edge bubbles of the type found in Crompton’s flat plate flow will usually exist for sails that are not supported by a mast (i.e. headsails such as genoas and jibs, or downwind sails such as spinnakers and gennakers), and these bubbles have significant influence on the pressure distribution and lift generated by such sails. The shape and structure of the separation region itself has significant impact on the pressure peak near the leading edge which is the primary contributor to the lift on a sail. The influence of the leading edge bubble extends well downstream of the reattachment point and the interaction between the leading edge bubble and the trailing edge separation region is poorly understood. It is hoped that this study will shed light onto the susceptibility of recovering boundary layers to adverse pressure gradient driven separation.

Whilst extensive CFD validation studies have been carried out for pressure gradient driven separation, there have been no investigations into the relative performance of various turbulence models for leading edge bubbles of the type found in the flow past flat plates. Nor has much thought been paid to the suitability of CFD methods for predicting adverse pressure gradient driven separation of a boundary
layer that is in a mode of post-reattachment recovery. Through this comparison between CFD solutions and experimental data for the inclined flat plate it is hoped that a better understanding can be gained of the performance of turbulence models for flows involving leading edge bubbles.

### 4.1.1 Flow structure

![Diagram of the flow past a flat plate at shallow incidence.]

The most interesting feature of the flow past a flat plate is the formation of the leading edge bubble due to knife-edge separation at the leading edge (see Figure 4.1). The shear layer that separates from the leading edge is immediately unstable with vortices being shed periodically from the tip of the plate and advecting along the shear layer. Driver, Seegmiller and Marvin (1987) observed this type of vortex shedding in the mixing layer behind a backward facing step and referred to the phenomenon as shear layer flapping. Both Driver's backward facing step flow and the flow past the flat plate at shallow incidence involve a turbulent shear layer that separates from a point and reattaches some distance downstream.

In Crompton's experiments the separated shear layer was found to undergo transition immediately after it leaves the plate. Downstream of transition, turbulent kinetic energy is entrained rapidly due to high mean-shear across the shear layer. This entrainment of turbulent kinetic energy causes the shear layer to thicken and curve back toward the surface of the plate. As the shear layer approaches the surface of the plate, the velocity fluctuations in the direction normal to the plate are damped. Energy from the normal fluctuations is converted to the other components of the Reynolds stress tensor with the turbulent eddies stretching in the spanwise and streamwise directions. The flow reattaches some distance, \( X_R \), downstream of the leading edge. Here, the flow bifurcates with the bulk of the flow continuing along the length of the plate, whilst a smaller fraction is driven back towards the leading edge to complete the leading edge bubble. The boundary layer that forms downstream of the reattachment point is embedded with turbulent structures that are a legacy from the leading edge bubble. These mixing layer structures advect downstream and interact with the development of the boundary layer.

The recovering turbulent boundary layer has a unique structure that is atypical of turbulent boundary layers; it is sluggish and prone to boundary layer separation in adverse pressure gradient flows. At the reattachment point the velocity gradient \( \partial U/\partial y \) must be zero due to the bifurcation of the flow. As
a result of this condition it is some distance downstream of the reattachment point before the boundary layer can obtain the steep near wall velocity gradients typical of boundary layers in turbulent equilibrium. Naturally, the structure of the boundary layer during recovery dictates the size and structure of any trailing edge separation region that might form downstream. Consequently, the nature of the boundary layer after reattachment can have a significant impact on the lift coefficient. Naturally, the shape and structure of the separation region itself affects the pressure peak near the leading edge which is the primary contributor to lift. The interaction between the leading edge bubble and the trailing edge separation region is as yet not fully understood.

A typical pressure coefficient distribution for a flat plate at shallow incidence is illustrated in Figure 4.2. Also presented are typical pressure coefficient plots for a generic airfoil and a thick airfoil with a leading edge separation bubble. The flat plate experiences a significantly smaller suction peak at the leading edge compared with the airfoils due to its inability to sustain any attached flow in this region. Reattachment occurs approximately at the conclusion of the adverse pressure gradient that exists over the aft half of the bubble.

![Typical pressure coefficient plots](image)

Figure 4.2: Typical pressure coefficient plots.
4.1.2 The thin airfoil bubble

The leading edge bubble that occurs due to knife-edge separation from flat plates is known as the "thin airfoil bubble". Crompton (2001) reported that in all cases transition occurred upstream of \( x/c = 0.025 \) and was found to occur at the same chordwise location in experiments carried out at Reynolds numbers of \( 0.53 \times 10^5 \) and \( 2.13 \times 10^5 \). Consequently it is hypothesised that eddy viscosity models (which assume fully turbulent flow) can be applied much more suitably to flows involving thin airfoil bubbles where transition occurs in close proximity to the leading edge than to flows that involve shear layers with larger regions of laminar flow.

Crompton (2001) found Reynolds number insensitivity of the reattachment length above a Reynolds number of approximately \( 10^5 \). This limit increased slightly as the incidence reduced due to viscous effects being more dominant in the shallower separation bubbles. The reattachment length is primarily dependent on the rate of turbulent entrainment in the shear layer. At low Reynolds numbers as the Reynolds number is increased so does the entrainment rate and hence the reattachment length shortens. As the Reynolds number is increased further the maximum entrainment rate is reached as the shear layer enters a state of turbulent equilibrium. At this point the reattachment length begins to lengthen again due to a reduction of shear layer thickness associated with increasing Reynolds number (due to lower diffusive effects resulting in lower shear layer spreading rates). At approximately \( \text{Re} = 10^5 \) this thickness reduction becomes negligible and the shear layer profile (and hence also the reattachment length) becomes independent of Reynolds number.

Since the reattachment process is always a turbulent one, the downstream recovery of the turbulent boundary layer can be expected to be only weakly dependent on Reynolds number. However within the separation bubble Reynolds number dependence can be expected. As the shear layer bends towards the surface the normal velocity fluctuations are damped by the adjacent wall. Also as the shear layer bifurcates at the reattachment point the large eddies are split into pairs of smaller eddies. These two effects contribute to a drop in turbulent shear stress around the reattachment point. The reversing fluid is subject to a strong favourable pressure gradient and subsequently it accelerates and reaches a maximum reversed velocity of approximately \(-0.4U_\infty\) at approximately half way back along the length of the bubble. This favourable pressure gradient has a stabilising effect on the reverse flow boundary layer and a significant drop in turbulence intensity was recorded over this region (Crompton, 2001). Corresponding to this drop in turbulence intensity the velocity gradients near the surface reduce and the boundary layer profiles become more laminar-like; the favourable pressure gradient is initiating relaminarisation (reverse transition).

Near the front of the separation bubble the pressure gradient is adverse to the reversing flow and consequently the laminar-like boundary layer is prone to separate. Crompton observed a small secondary separation bubble in his experiments near the leading edge of the plate. The likely form of this secondary separation bubble is illustrated in Figure 4.3. The length of this separation bubble varies with both angle of incidence and Reynolds number. The Reynolds number dependence is caused by the relaminarisation process in the reversed flow boundary layer which itself is dependent on the local velocity and hence also the freestream Reynolds number.
CHAPTER 4. THE FLAT PLATE AT SHALLOW INCIDENCE

4.1.3 The short bubble

Airfoils with rounded leading edges may also exhibit separation near the leading edge (as in the airfoil case in Figure 4.2). These bubbles form due to separation of the laminar boundary layer at the onset of an adverse pressure gradient near the leading edge. Such bubbles are referred to as "short bubbles" and they have been reported to have distinctly different characteristics from thin airfoil bubbles (Crabtree, 1957). A typical short bubble is illustrated in Figure 4.4. In the short bubble a laminar shear layer typically extends across more than half the length of the bubble and after transition the shear layer thickens rapidly and curves back towards the airfoil surface where it reattaches. Consequently, the short bubble has a quite different shape to the thin airfoil bubble, and as its name suggests it is relatively short in the streamwise direction.

Flows involving the short bubble pose several difficulties for turbulence models. The boundary layer that exists between the forward stagnation point and the separation bubble is laminar and generally turbulence models predict early transition to turbulent flow in this region. As an example, for a flat plate boundary layer under zero pressure gradient, turbulence models will predict transition to occur at a Reynolds number that is approximately an order of magnitude lower than experimental evidence suggests (Wilcox, 1998). Therefore for airfoil simulations involving short separation bubbles the calculated
boundary layer is likely to be turbulent upstream of the separation point. Consequently the solution will exhibit an unrealistic increase in skin friction due to the high shear of turbulent boundary layers compared with laminar ones. More troublesome is the fact that a turbulent boundary layer has much steeper velocity gradients near the surface and is hence more resistant to flow separation than a laminar layer. Therefore separation of the short bubble is likely to be delayed - if not omitted entirely - in CFD simulations. If the boundary layer in the CFD simulation were to remain laminar up until separation then it is likely that transition would occur in the shear layer almost immediately after detachment, upstream of the desired location. Consequently the shear layer will grow much faster which would cause the shear layer to reattach much earlier than in reality where the shear layer is laminar over a significant portion of the bubble. Transition models, such as the transition version of the \( k - \omega \) model designed by Wilcox (1994), have had moderate success at predicting the correct transition location for some simple flows. Transition models typically use different model coefficients for the turbulent transport equations in the three separate regions of the flow; the laminar region, transition region and the turbulent region. Tuning such a model is an arduous task and the position where transition occurs can be quite unpredictable between different flows due to the influence of the local pressure gradient and the different flow mechanisms that may initiate transition. In the case of the short bubble, transition actually occurs within the shear layer which poses additional difficulties to tuning the model since the coefficients used for a transition model are usually calibrated using boundary layer flows. It is also likely that within short separation bubbles some degree of relaminarisation will occur because the reversed flow is driven by a favourable pressure gradient. This relaminarisation process is another phenomenon that turbulence models cannot be expected to predict.

4.1.4 Experimental data

Of the experimental results concerned with the flat plate at shallow incidence, the work by Crompton and Barrett (Crompton and Barrett, 2000; Crompton, 2001) is the most comprehensive. Crompton presents a detailed examination of the bubble structure which extends upon earlier work by Gault (Gault, 1957) and Newman and Tse (Newman and Tse, 1992). Through the use of Laser Doppler Anemometry (LDA), detailed velocity and turbulence statistics were measured for the leading edge bubble. LDA measurements of the developing turbulent boundary layer in the recovery region downstream of the reattachment point were also taken. Surface pressures were recorded through static pressure tappings. Flow visualisation techniques (surface tufts, china clay and oil streakline) were used to complement the LDA and static pressure data. Unfortunately no skin friction measurements were conducted in the experiments.

The wind tunnel model used in the experiments had a chord length of 160mm and a span of 800mm giving an aspect ratio of 5 which was demonstrated (using oil-streakline flow visualisation) to be sufficient to provide nominally two-dimensional flow. In order to provide enough stiffness to minimise deflection and to facilitate the pressure tappings, the plate was constructed out of 6mm steel plate, giving a thickness to chord ratio of 3.75%. The leading edge of the plate was chamfered at 20 degrees in order to provide a sharp leading edge. The geometry of the leading edge can have significant influence on the separated shear layer. Ideally for a sharp leading edge the flow will separate at any departure from zero incidence. However in reality there must be some roundedness to the leading edge which may allow the boundary layer to remain attached through small departures from zero incidence. A cross-section of the model is illustrated in Figure 4.5.


Figure 4.5: Flat plate dimensions.

The majority of Crompton’s wind tunnel tests were carried out at a Reynolds number of $2.13 \times 10^5$ and it is at this Reynolds number that comparison between the experiments and CFD have been made. The experimental data is provided for angles of attack, $\alpha = 1^\circ$ to $5^\circ$, in $1^\circ$ intervals. At $\alpha = 1^\circ$ the leading edge bubble is small and similar in length to chord ratio to leading edge bubbles observed for downwind sail flows in the University of Auckland’s twisted flow wind tunnel. At $\alpha = 5^\circ$ the flow is separated for the majority of the length of the plate and at $\alpha = 6^\circ$ the shear layer fails to reattach. The asymmetry of the plate creates a lift force even at zero degrees of incidence which causes the flow to curve upwards (in the direction of the lift axis) near the leading edge of the plate and consequently a small leading edge separation bubble forms at zero angle of attack. The so-called ‘ideal’ angle of incidence, where laminar boundary layers are able to develop on both surfaces without leading edge separation, occurs at a small negative angle.

4.1.5 The CFD model

Crompton’s flat plate was modelled using the same geometry and dimensions that were used in the experiments (Figure 4.5). The thickness and asymmetry of the plate influences the flow to a degree where it is necessary to use the true geometry, rather than an infinitely thin plate for which grid generation would be simpler. The computational domain is illustrated in Figure 4.6. Simulations performed using a domain of twice the length and height verified that the domain dimensions of Figure 4.6 were adequate with the proximity of the far-field boundaries having negligible influence on the near-field. At the inlet Cartesian velocity components are specified according to the angle of incidence. Freestream turbulence intensity was set at $0.05\%$ which was the maximum value measured in the tunnel while the experiments were carried out (Crompton, 2001). The turbulent length scale was set at $0.001m$, a typical value for low turbulence wind tunnels.

An illustration of the computational grid used in this study is presented in Figure 4.7. Three grid resolutions were used, and the grids are referred to as coarse, medium and fine with 11920, 49625 and 202435 cells respectively. A grid convergence study is presented in section 4.2.1 and at each refinement level the grid spacing was halved in both directions. In order to achieve a $y^+$ of approximately 1.0 the near wall spacing was set at $1.0 \times 10^{-5}m$. Particular care was taken to generate orthogonal cells, with low aspect ratios near the leading edge and at the very leading edge the cells have an aspect ratio of $1:1$. The tip region is illustrated in the close-up view in Figure 4.7. Further down the chord high aspect ratio cells were used in order to resolve the large flow gradients normal to the wall and at the trailing edge the aspect ratio reduces to $1:10$.

Spatial interpolation was carried out using the second-order upwind advection scheme (as described in chapter 3) and all CFX simulations converged to a steady state with the residuals of each flow variable decreasing by at least 4 orders of magnitude.
CHAPTER 4. THE FLAT PLATE AT SHALLOW INCIDENCE

Figure 4.6: Details of the domain for the flat plate.

Figure 4.7: Computational grid for the flat plate (medium resolution).
Results are presented for the SST and standard $k - \omega$ turbulence models. Simulations were also carried out using the standard $k - \varepsilon$ model for comparative purposes. However, it was found that the algebraic wall functions used to capture the inner portion of the boundary layer in these models do not perform well for this flat plate flow. The wall functions assume that the inner boundary layer region is in turbulent equilibrium, which is not the case here, especially within the leading edge bubble. The $k - \varepsilon$ model was found to diverge for the fine grids that were used in this study and convergence was obtained only on grids much coarser than desired. Consequently, only results computed using the $k - \omega$ and SST models are presented in this chapter.

4.2 Results

This section initially focuses on the case at $\alpha = 1^\circ$ since it is at this angle that the flow past the flat plate looks most similar to sail flows. Here the leading edge bubble is of similar length to chord ratio to typical downwind sail flows and the pressure distribution has similar characteristics to pressure distributions from sail flows. At this angle the leading edge separation bubble stretches across the first 14% of the chord length (i.e. $X_R/c = 0.14$). Since the bubble is quite small fewer velocity measurements were taken within the bubble at this angle than at higher angles of attack. Consequently, a decision was made to also investigate the case of $\alpha = 3^\circ$ in order to gain a better understanding of the flow within the separation bubble. At this higher angle of incidence the reattachment length, $X_R/c = 0.47$, i.e. the flow is separated over almost half of the plate.

4.2.1 Grid convergence study

The grid convergence study was performed solely at $\alpha = 3^\circ$ and only with the SST turbulence model. At $\alpha = 1^\circ$ the results are not as grid dependent since there the flow separation is much less severe than at $\alpha = 3^\circ$. Lift and drag coefficients are presented in Figure 4.8 for the sequence of grids. The force coefficients are plotted against $1/N$ which is a non-dimensional measure of the grid spacing where $N^2$ is the total number of cells in the grid.

As the grid is refined there is a clear convergence of the lift and drag coefficients. The lift from
the medium grid solution is within 0.015% of the fine grid solution and drag is within 0.0055%. This indicates that the solutions are have negligible dependence grid spacing, and subsequent refinement is unlikely to result in any increased accuracy of the solution. Since forces are integrated properties there is a possibility that the grid convergence illustrated in Figure 4.8 could have occurred fortuitously. To confirm that this is not the case the surface pressure distributions for the three grids are presented in Figure 4.9. Note that the spike in the pressure coefficient on the pressure side of the plate is caused by the shoulder at the end of the chamfer.

The $C_p$ curves for the medium and fine grid lie almost directly upon one another, the fine grid solution has slightly less suction at the leading edge, however this effect is minimal. Since the fine grid and medium grid solutions are virtually identical the medium grid was chosen as the most suitable grid, and it has been used for all subsequent simulations.

4.2.2 Comparison at $\alpha = 1^\circ$

Velocity contours

Figure 4.10 presents the velocity contours for both the experimental and CFD results at $\alpha = 1^\circ$. The CFD models capture the size and shape of the leading edge bubble well and in general the results are in good agreement with the wind tunnel experiments.

Boundary layer profiles in the leading edge bubble

Boundary layer velocity profiles were measured by Crompton at 9 different chordwise locations along the plate using LDA. The measurement stations are located at $x/c = \{0.031, 0.125, 0.25, 0.375, 0.5, 0.625, 0.75, 0.875, 1.0\}$. Figure 4.11 shows the position of these measurement stations in relation to the flow field at $\alpha = 1^\circ$. 
Figure 4.10: Velocity contours ($\alpha = 1^\circ$).
CHAPTER 4. THE FLAT PLATE AT SHALLOW INCIDENCE

Figure 4.12: Chordwise velocity profiles within the leading edge bubble ($\alpha = 1^\circ$).

Figure 4.11: Flow streamlines and the measurement stations for the flat plate at $\alpha = 1^\circ$ (SST model).

<table>
<thead>
<tr>
<th></th>
<th>$X_R$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Crompton</td>
<td>0.14</td>
</tr>
<tr>
<td>SST</td>
<td>0.1486</td>
</tr>
<tr>
<td>$k - \omega$</td>
<td>0.1841</td>
</tr>
</tbody>
</table>

Table 4.1: Reattachment lengths for the flat plate at $\alpha = 1^\circ$.

Table 4.1 shows that the reattachment lengths computed with the SST and $k - \omega$ models differ from the experimental value. The SST model overpredicts $X_R$ by 5.8% and the $k - \omega$ model overpredicts $X_R$ by 24.0%. To allow a better comparison between the boundary layer profiles the measurement stations are scaled by $X_R$ so that each station is in the same position relative to the reattachment point. For $\alpha = 1^\circ$ only the first two measurements stations are within the leading edge bubble. The profiles at these stations are shown in Figure 4.12.

The portion of the fluid forced upstream at the reattachment point is accelerated by the favorable pressure gradient and forms a reversed boundary layer. Crompton reported a maximum reversed velocity of $-0.4U_\infty$ approximately midway along the bubble. The CFD results show lower reversed velocities at the same position along the bubble with the SST model predicting $-0.290U_\infty$ and the $k - \omega$ model predicting $-0.284U_\infty$. The measured velocities within the leading edge bubble are larger than usually experienced.
within short airfoil bubbles, for which the reversed flow velocity is typically below \(-0.2U_\infty\) (Crompton, 2001). For the thin airfoil bubble, however, early transition results in a high rate of entrainment of turbulent kinetic into the bubble. Subsequently, greater velocities in both the outer and inner shear layers are observed. The turbulence models underpredict the entrainment rate and hence also underpredict the velocity magnitude throughout the leading edge bubble. This will be discussed further in section 4.2.3, where the kinetic energy profiles within the separation bubble are presented.

Crompton reported that the reversed flow within the leading edge bubble experienced relaminarisation, encouraged by the strong favorable pressure gradient, and that the inner shear layer began to show very laminar-like features. The velocity profiles within the separation bubble are presented in greater detail in Figure 4.13. At \(x/c = 0.031\) in particular, the experimental data show a much more laminar profile than those provided by the CFD. To emulate the relaminarisation process an appropriate transition model is required which is not provided by the turbulence models investigated.

One of the direct consequences of the more turbulent-like boundary layer profiles predicted by the CFD is their resistance to separation. In the simulations the reversed boundary layer remains attached all the way to the leading edge and there is no secondary separation of the type reported by Crompton. Crompton suggested that the form of this second bubble has significant influence on the development of the shear layer near the leading edge and consequently its influence is likely to spread well downstream (Crompton, 2001).

**Boundary layer profiles downstream of reattachment**

A comparison of the boundary layer profiles downstream of the reattachment point is presented in Figure 4.14. In general the CFD results compare well with the boundary layer profiles measured by Crompton. From Figure 4.14 it is evident that the wind tunnel results give slightly fuller boundary layer results than the CFD. However, this effect is difficult to quantify and it is hard to assess the structure of the different boundary layers in the form they are presented in Figure 4.14.

In Figure 4.15 the boundary layer profiles at \(x/c = 0.25\) and 0.875 are presented using a log scale for \(y^+\). Also plotted is a typical turbulent boundary layer at zero pressure gradient (ZPG) (Österlund, 1999). The wall shear stress, \(\tau_w\), was not provided in the experimental results and therefore it was difficult to

---

**Figure 4.13:** Near-wall chordwise velocity profiles within the leading edge bubble \((\alpha = 1^\circ)\).
Figure 4.14: Chordwise velocity profiles downstream of reattachment (α = 1°).
Figure 4.15: Chordwise velocity profiles downstream of reattachment (log scale, $\alpha = 1^\circ$).
non-dimensionalise the Crompton data. Here, \( y^+ \) and \( u^+ \) are calculated based upon the \( \tau_w \) values from the CFD data computed using the SST model. Any errors introduced by this approximation are likely to be small. The non-dimensionalised plots in Figure 4.15 follow the same trends as data compared using the raw velocity, \( U \), and normal distance, \( y \).

At \( x/c = 0.25 \) the boundary layer profiles given by the experiments and the CFD lie below the ZPG profile. This is expected because of the slight adverse pressure gradient the boundary layer works against. The velocities predicted by the CFD are smaller than those measured. We attribute this difference to the reattachment process.

The boundary layer profile at reattachment has a shape distinctive of reattaching turbulent shear layers. These layers are characterized by a thick sublayer which extends well out towards the boundary layer edge (Horton, 1969). The figures show the inner structure of the recovering boundary layer developing just downstream of the reattachment point. The layer is accelerated through the inwards entrainment of momentum from the outer region. Both experimental and CFD results show the sublayer developing with its characteristic thin region of steep velocity gradients. Moving further aft from the reattachment point, the boundary layer starts to show a typical turbulent boundary layer profile: the sublayer thicken-ens and the mixing layer structures in the outer region of the boundary layer spread and dissipate. At \( x/c = 0.875 \) the inner region of the boundary layer appears to have recovered in the experiments and shows a profile similar to ZPG up to \( y^+ \approx 50 \). The outer region of the boundary layer - whilst showing its characteristic defect layer shape - still predicts low velocities and is yet to recovery fully.

The delayed post reattachment recovery of the boundary layers predicted by the CFD was also observed by Lai and Yoo (So and Lai, 1988) for flow past a backward facing step. In their experiments wall function based models produced better results in the recovery region than low Reynolds number turbulence models. This is not surprising because wall functions, which are based on local-equilibrium arguments, force the flow to asymptote to the log law-of-wall.

As a final note on the boundary layer profiles, Figure 4.15 shows that the \( k - \omega \) model predicts profiles that are slightly closer to the measured profiles than those predicted by the SST model. This is believed to be the consequence of the SST limiter on the eddy viscosity which leads to lower production of turbulent kinetic energy.

**Pressure distributions** The pressure coefficient plot for the flat plate at \( \alpha = 1^\circ \) is presented in Figure 4.16. In the experimental data the pressure decreases downstream of the leading edge and reaches a minimum at \( x/c \approx 0.045 \). The CFD results show a smaller and flatter suction peak. They do not experience the gradual decrease in pressure over the forward half of the separation bubble that is seen in the experimental result because of the inability of the turbulence models to predict the correct transition location and resolve the secondary recirculation region. The experiments show a leading edge bubble that is shorter and fatter indicating greater shear layer curvature. Within the leading edge bubble the pressures, which are determined predominantly by the curvature of the outer shear layer, are lower in the experimental data compared with the CFD. The shear layer is initially laminar and able to sustain a positive pressure gradient before transition. It is possible that background turbulence in the experiments may have helped to increase the curvature of the shear layer, however the freestream turbulence intensity in the experiments was less than 0.25% and so this is unlikely.
As shown in Figure 4.16, the $k - \omega$ model predicts lower pressures than the SST model and the experimental data between $x/c = 0.1$ and $x/c = 0.2$. This corresponds to the model overpredicting the length of the separation bubble.

When the $C_P$ curves are normalized based on reattachment length, the pressure profiles compare well. The only significant differences are a larger suction at the leading edge in the experimental data, and a particularly low suction peak predicted by the $k - \omega$ model. Downstream of reattachment, the pressure coefficients in the experimental results are slightly lower than in the CFD results. This is directly related to the greater velocities in both the leading edge bubble and the boundary layer in this flow region.

![Figure 4.16: Pressure coefficient plot ($\alpha = 1^\circ$).](image)

4.2.3 Comparison at $\alpha = 3^\circ$

Velocity contours

Figure 4.17 presents the velocity contours for both the experimental and CFD results at $\alpha = 3^\circ$. Again the CFD models capture the size and shape of the leading edge bubble well and in general the results are in good agreement with the wind tunnel experiments.

Boundary layer profiles in the leading edge bubble

<table>
<thead>
<tr>
<th>Model</th>
<th>$X_R$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Crompton</td>
<td>0.47</td>
</tr>
<tr>
<td>SST</td>
<td>0.4374</td>
</tr>
<tr>
<td>$k - \omega$</td>
<td>0.5096</td>
</tr>
</tbody>
</table>

Table 4.2: Reattachment lengths for the flat plate at $\alpha = 3^\circ$.

The reattachment lengths for the $\alpha = 3^\circ$ case are presented in Table 4.2. The SST model underpredicts $X_R$ by 6.4% and the $k - \omega$ model overpredicts $X_R$ by 8.4%. As for the $\alpha = 1^\circ$ case, the chordwise positions of the boundary layer profiles are scaled by $X_R$.  

---

---

---
Figure 4.17: Streamwise velocity contours ($\alpha = 3^\circ$).
Figure 4.18: Chordwise velocity profiles within the leading edge bubble ($\alpha = 3^\circ$).
CHAPTER 4. THE FLAT PLATE AT SHALLOW INCIDENCE

Figure 4.19: Near wall chordwise velocity profiles at $x/c = 0.031$ ($\alpha = 3^\circ$).

The velocity profiles within the separation bubble are presented in Figure 4.18. In this discussion the analysis works from the rear of the bubble forward, starting with the measurement station closest to the reattachment point (i.e., $x/c = 0.375$). At this station the CFD results show much lower velocities than the experimental results in both the outer and inner shear layers. It appears that as the shear layer curves down towards the wall the velocities in the chordwise direction are damped more in the CFD results, compared with the experiments. This is related to the turbulent behavior of the outer shear layer which is discussed in the next section.

The measurement station at $x/c = 0.25$ is approximately mid length of the leading edge bubble. Here the velocities are at their maximum, both at the outside edge of the shear layer and in the reversed flow region. In the inner shear layer the maximum reversed velocity is $-0.393U_\infty$ for the experimental results, whereas the SST model predicts $-0.252U_\infty$ and the $k-\omega$ model predicts $-0.271U_\infty$. The reversed boundary layer is notably thicker in the experiments than in the CFD results. Moving towards the leading edge, we see that at $x/c = 0.125$ the reversed flow weakens primarily due to mass flow in the direction normal to the plate. In the experiments, the presence of an adverse pressure gradient also has a significant effect on the reversed flow.

At $x/c = 0.031$ the secondary separation bubble is evident in the experimental data as shown in Figure 4.19. The CFD simulations do not show this feature and the near-wall flow is reversed all the way to the leading edge. A close-up of the near-wall region, showing the secondary bubble (in the experimental results), is presented in figure 4.19.

Turbulent kinetic energy profiles in the leading edge bubble

In this section we compare the turbulent kinetic energy profiles calculated by the CFD with the profiles from the experiments. Crompton’s data was supplied in terms of $u_{RMS}$, $v_{RMS}$ and $w_{RMS}$, where RMS represents the root mean square of the velocity fluctuations. From this data the turbulent kinetic energy was calculated using $k = 1/2 \left( \bar{u}' \bar{u}' + \bar{v}' \bar{v}' + \bar{w}' \bar{w}' \right)$, where the normal turbulent stresses are calculated from the RMS values as $\bar{u}' \bar{u}' = (u_{RMS})^2$. The RMS measurements include both the turbulent fluctuations and also the mean-flow unsteadiness that was observed in the shear layer. Therefore, the turbulent kinetic energy for the experimental data presented in Figure 4.20 is not a true representation of the
turbulence since it is made up of both turbulent and mean-flow kinetic energy. Crompton reported that the mean-flow unsteady structures seen in the laminar shear layer were absorbed into the turbulent eddy spectrum downstream of transition. Therefore the mean-flow kinetic energy associated with shear layer flapping is eventually converted into turbulent kinetic energy. Periodicity in the RMS velocities (indicating mean-flow unsteadiness) was observed as far back as \( x/c = 0.375 \) for the \( \alpha = 3^\circ \) case. At each of the measurement stations it is impossible to determine the amount of energy that has been converted to turbulent kinetic energy and hence the experimental data in Figure 4.20 may well be overvalued.

At \( x/c = 0.031 \) the experimental turbulent kinetic energy profile shows two peaks. The first, and lower, peak is associated with the secondary separation bubble. At the outer edge of this bubble a region of high mean-shear exists which increases production of turbulent kinetic energy. The second peak occurs at the point of inflection in the velocity profile in the outer shear layer where the mean-shear is at its maximum. It is stronger than the first peak as it is driven by the large velocities around the leading edge of the plate. The maximum turbulent kinetic energy observed in the experiments is 2.5 times that predicted by the SST model and 4.0 times the maximum predicted by the \( k-\omega \) model. Apart from the above mentioned mean-flow unsteadiness there are other effects that may contribute to high turbulent kinetic energy in this region. Firstly, the unsteady shear layer flapping at the leading edge causes additional entrainment of kinetic energy. Secondly, the small secondary recirculation bubble at the leading edge enhances the mean-shear in the leading edge region which increases the production of

---

**Figure 4.20:** Turbulent kinetic energy profiles within the leading edge bubble (\( \alpha = 3^\circ \)).

---

At \( x/c = 0.031 \) the experimental turbulent kinetic energy profile shows two peaks. The first, and lower, peak is associated with the secondary separation bubble. At the outer edge of this bubble a region of high mean-shear exists which increases production of turbulent kinetic energy. The second peak occurs at the point of inflection in the velocity profile in the outer shear layer where the mean-shear is at its maximum. It is stronger than the first peak as it is driven by the large velocities around the leading edge of the plate. The maximum turbulent kinetic energy observed in the experiments is 2.5 times that predicted by the SST model and 4.0 times the maximum predicted by the \( k-\omega \) model. Apart from the above mentioned mean-flow unsteadiness there are other effects that may contribute to high turbulent kinetic energy in this region. Firstly, the unsteady shear layer flapping at the leading edge causes additional entrainment of kinetic energy. Secondly, the small secondary recirculation bubble at the leading edge enhances the mean-shear in the leading edge region which increases the production of
turbulent kinetic energy. These effects can not be adequately resolved by the turbulence models tested.

The turbulent kinetic energy contours for the leading edge are illustrated in Figure 4.21. Here it can be seen that in the SST simulations the turbulent kinetic energy levels begin to be amplified slightly further upstream than they do for the \( k - \omega \) model. Whilst we have said that these turbulence models are not transition models it is not true to say that they do not model transition at all. The simulations still pass from a region where the turbulent kinetic energy is zero (i.e. a laminar region) through to a turbulent region where production terms in both the \( k \) and \( \omega \) equations are positive, but they do not model the transition process appropriately. Typically, for transitional boundary layer flows standard two-equation models will predict transition to occur at a Reynolds number at least an order of magnitude too small (Wilcox, 1998). Similarly in this test case transition occurs immediately after separation and much further upstream than the figure of 2.5% reported by Crompton.

At \( x/c = 0.125 \) the experiments give a peak magnitude of turbulent kinetic energy in the shear layer that is slightly lower than at the upstream station. This is due to the diminished effect of the secondary separation bubble which extends only to \( x/c = 0.045 \). This reduction in turbulent kinetic energy may also be associated with the diffusion of the unsteady structures that were observed in the shear layer near the leading edge. The turbulence models are in better agreement with the experiments than upstream with the SST model outperforming \( k - \omega \). In both the CFD and the experimental results the peak in turbulent kinetic energy has widened due to the thickening of the shear layer. Near the wall the experimental results show lower levels of turbulent kinetic energy than the CFD which is a result of relaminarisation of the reversed boundary layer.

Between \( x/c = 0.125 \) and \( x/c = 0.25 \) the experimental results indicate that the kinetic energy in the shear layer is still increasing and that the maximum entrainment rate has not yet been reached. The shear layer predicted by the SST model reaches local equilibrium prematurely because the SST limiter reduces the amount of turbulent shear stress and hence production of turbulent kinetic energy.

Between \( x/c = 0.250 \) and \( x/c = 0.375 \) both the SST and \( k - \omega \) models predict a decrease in turbulent kinetic energy, whereas in the experimental results \( k \) continues to increase. In this region, the shear layer curves back towards the wall and decelerates as it approaches reattachment. The experimental results show a drop in vertical velocity fluctuations \( (u') \). This energy is mostly converted to transverse \( (u') \) and streamwise \( (u') \) velocity fluctuations and thus the kinetic energy levels are retained. The SST

---

**Figure 4.21**: Turbulent kinetic energy contours around the leading edge \( (\alpha = 3^\circ) \).
and $k - \omega$ turbulence models cannot emulate this transfer of energy from the vertical to transverse and streamwise fluctuations because they assume isotropy of the Reynolds stress tensor. In order to model this process a more advanced anisotropic turbulence model is required, such as an algebraic stress model or a second-order closure model.

At $x/c = 0.375$ both the SST model and the $k - \omega$ model predict peaks in turbulent kinetic energy that are approximately 1.7 times smaller than in the experimental results. In the experimental results the kinetic energy peak is wider, which indicates that the shear layer thickness is underpredicted by the CFD.

**Downstream of reattachment**

For the $\alpha = 3^\circ$ case the downstream velocity profiles are quite similar in shape to those found in the $\alpha = 1^\circ$ case. Again, in this region the velocities predicted by the CFD are lower than those measured in the experiments. It is likely that in the experiments the outer turbulent structures, which originate in the shear layer of the leading edge bubble, feed energy into the inner region and in this way accelerate the development of turbulence. Without these increased levels of turbulent kinetic energy the recovery of the boundary occurs over a larger downstream distance in the CFD. It is hoped that future LES simulations will be able to help us to improve our understanding of this apparent slow post-reattachment recovery of the two-equation turbulence models.

In the recovery region the $k - \omega$ model predicts greater turbulent kinetic energy compared with SST. Consequently the $k - \omega$ model predicts slightly steeper near wall velocity gradients and the turbulent boundary layer is able to recover more rapidly. The lower values of turbulent kinetic energy in the SST results are due to the SST limiter which becomes active in regions of large mean-strain.

### 4.3 Summary

#### 4.3.1 Experiments vs SST and $k - \omega$ simulations

The experimental results show greater growth and curvature of the outer shear layer of the leading edge bubble. This is due to several phenomena which are not adequately captured by the SST and $k - \omega$ turbulence models: Firstly, there is increased entrainment in the experimental results at the leading edge due to unsteady shear layer flapping. Also production of turbulent kinetic energy dominates dissipation and $k$ continues to increase along the length of the shear layer in the experiments. The lower shear layer growth predicted by the CFD causes reattachment to be delayed and the pressure coefficient to be overpredicted within the leading edge bubble.

The reversed flow in the leading edge bubble is subject to a favorable pressure gradient which causes relaminarisation, an effect which is not captured by the turbulence models because they are designed for fully turbulent flows and are incapable of modelling transitional effects. Consequently the models also fail to predict the secondary bubble near the leading edge. The omission of the secondary bubble in the CFD results is not believed to be an artifact of the numerics since the solutions showed excellent grid convergence and the secondary bubble was not present in the fine grid simulations.

Approaching the reattachment point both turbulence models artificially damp turbulent kinetic energy.
due to their isotropic representation of the Reynolds stress tensor. In the experiments the turbulent fluctuations normal to the wall were damped, however the energy from this was fed into the spanwise and chordwise turbulence. The $k-\omega$ and SST models use an isotropic formulation of the Reynolds stress tensor and are thus unable to model this anisotropic effect.

Downstream of reattachment the CFD profiles recover towards turbulent equilibrium slowly compared with the experiments. In the observed recovering boundary layer the eddies in the outer region of the near wall originate in the separated shear layer, whereas the scales in the inner region are associated with the development of the new boundary layer. The two-equation turbulence models cannot be expected to emulate this complicated recovery process involving an irregular eddy spectrum. Also, the anisotropic effects that occur as the shear layer approaches reattachment, which leads to transfer of turbulent kinetic energy from the normal to the transverse and streamwise velocity fluctuations, cannot be captured by the turbulence models. The models instead damp the turbulent kinetic energy associated with the velocity fluctuations normal to the plate near the reattachment point. The CFD predicts a slower recovery of the boundary layer downstream of reattachment.

4.3.2 SST vs $k-\omega$

The SST model captures the leading edge bubble more accurately than the $k-\omega$ model. It predicts greater turbulent kinetic energy levels near the leading edge and therefore predicts greater shear layer growth. As a result its predictions of the reattachment lengths are in closer agreement with the observed lengths (within 7\% for both angles of attack that were investigated ($\alpha = 1^\circ$ and $\alpha = 3^\circ$)), and the pressure coefficient in the vicinity of the leading edge bubble is more accurate. The shear layer predicted by the SST model has more curvature and bends back towards the plate earlier than the layer predicted by the $k-\omega$ model. However, the SST model returns lower turbulent kinetic energy values in the rear half of the leading edge bubble. This is due to the SST limiter. As a consequence the SST model produces lower velocities in both the outer and inner shear layers compared with both the $k-\omega$ model and the experimental data.

Downstream of reattachment the $k-\omega$ model predicts a more rapid boundary layer recovery than the SST model, but the recovery is still slower than observed in the experiments. The differences between the velocity profiles predicted by the two turbulence models are small and are a consequence of the SST limiter.
4.4 Conclusions

The flow past the flat plate at shallow incidence is a complex and challenging assignment for two-equation turbulence models. The overall flow topology of the leading edge bubble was predicted with reasonable accuracy by both the $k - \omega$ and SST turbulence models despite the fact that both turbulence models did not capture the secondary bubble. Both the $k - \omega$ and SST models underpredict the production of turbulent kinetic energy within the leading edge bubble which resulted in lower velocities in both the outer and inner shear layers.

Proper prediction of the recovering boundary layer downstream is particularly difficult for such turbulence models. We expect that CFD boundary layer profiles will be more susceptible to flow separation when an adverse pressure gradient is present, as is the case, for example, for curved sails. A question certainly remains over how the unsteady structures in the turbulent shear layer interact with the turbulence within the shear layer and how they affect the downstream behavior of the flow. These structures are damped by turbulence models because of the high eddy viscosities around stagnation and the leading edge. LES results would be useful here as a tool to aid the understanding of the transient behavior.

For the validation of sail flow simulations several results from this chapter are encouraging. Firstly the overall flow topology of the leading edge bubble and recovering boundary layer is predicted reasonably well and the pressure coefficient plots are close to the experimental data, especially for the SST model. Secondly, despite using turbulence models that assume fully turbulent flow, the leading edge bubble was adequately captured. This is largely because the leading edge bubble involves little laminar flow with the shear layer undergoing transition within the first 2.5% of the length of the plate. Therefore, turbulence models that do not account for transition can be much more suitably applied to the thin airfoil bubble than the short bubble where extensive regions of laminar flow exist, both upstream of the bubble and within the separated region itself.

Correct prediction of the pressure distribution is the most important criterion for evaluating turbulence models for sail flow simulations since almost all of the lift and drag is generated from the pressure forces. Based upon this method of evaluation, the SST model is preferred over the $k - \omega$ model. Considering the complexity of this flow and just how different it is to flows that modelers usually investigate and tune their models, the results are actually quite pleasing.
Chapter 5

Preliminary Wind Tunnel and CFD Investigations

5.1 Introduction

The review of sail flow research presented in chapter 1 highlighted the lack of available data for sail sections and the limited understanding of the flow features. Moreover, there have been no wind tunnel tests conducted for downwind sail shapes (camber > 20%). This chapter describes preliminary wind tunnel tests and CFD results for a two-dimensional downwind sail section. The goal of the study was to obtain insight into the behavior of two-dimensional sail flows and to gain an initial gauge of the performance of different turbulence models. This preliminary investigation was used to form the direction and goals of the subsequent research.

5.1.1 Wind tunnel setup

Wind tunnel experiments were carried out at the De Bray wind tunnel in the University of Auckland’s Yacht Research Unit. This wind tunnel is a low-speed, closed loop facility with a test section of dimensions 768mm × 615mm, a maximum flow speed of 60m.s⁻¹ and a maximum turbulence intensity of 1%. Figure 5.1 illustrates the model which is positioned on aerodynamic struts above the force balance which itself is mounted horizontally below the test section.

![Figure 5.1: The wind tunnel model.](image)
From analysis of ACC sail shapes it was evident that the circular arc represents a good approximation to a typical downwind sail section whilst providing a shape that is simple to manufacture. For this initial study a circular arc section was used with 24.7% camber, a radius of 200mm and a chord length of 319mm. Steel plate of 1mm thickness was rolled to the desired shape with a radial tolerance of less than 3mm which is equivalent to a maximum camber variation of 0.6%. Steel strengthening sections were soldered to the underside of the model in order to minimize deflection under load and to provide an attachment point for the struts. Pitch was controlled using the rear strut and a range of different angles were investigated from 5° through to 30° at 2.5° intervals.

The arrangement of the wind tunnel model within the tunnel is illustrated in Figure 5.2. At each end of the model end plates were positioned that span the height of the tunnel. These end plates were positioned 165.5mm inside the side walls of the tunnel to allow the boundary layer on the tunnel walls to pass without influencing the model. The leading edges of the end plates were fixed at 448mm in front of the model and extend 711mm past the trailing edge. The model itself is 435mm (1.37c) wide and small gaps (<2mm) exist between the tips of the model and the end plates. Tape and petroleum jelly were used to limit tip leakage through these gaps. Particular care was taken to ensure that friction between the model and the side walls did not influence force measurements. This was verified by measuring the forces when fixed weights were applied to the model.

The model was tested at an inlet speed of $25 \text{m.s}^{-1}$ which corresponds to a Reynolds number of $5.25 \times 10^5$. This Reynolds number is significantly lower than the typical Reynolds number range of 1 – 10 million for ACC sail flows. However it is believed that the flow should be reasonably independent of Reynolds number and validation of CFD models at the Reynolds number used in the wind tunnel should also apply to full scale sail flows. The main reason that Reynolds number insensitivity can be expected is the presence of the leading edge separation bubble. As the shear layer separates at the leading edge transition occurs rapidly and consequently the boundary layer reattachment process is always turbulent and independent of Reynolds number. In Crompton’s experimental study of the flat plate at shallow
incidence reattachment of the leading edge shear layer was found to be independent of Reynolds number above \( \text{Re} = 10^5 \) (Crompton, 2001).

### 5.1.2 The CFD Model

A schematic of the computational domain is illustrated in Figure 5.3. Due to the size of the model in relation to the wind tunnel dimensions it was felt that it was necessary to include the side walls in the CFD model. Simulations were carried out using both non-slip and free-slip boundary conditions on the side walls and there was found to be no significant loss in accuracy when free-slip conditions were used. Consequently free-slip boundary conditions were used for all of the simulations that are presented in this chapter. The turbulence intensity and length scale were set at 1\% and 0.001\( m \) respectively.

The structured grids used in this study were generated using \( \text{ICEM-HEXA} \). To ensure that the grids themselves were not a source of error a grid refinement study was conducted (section 5.2.1). Three levels of refinement were used with the grids referred to as coarse, medium and fine with 8496, 28912 and 120482 cells respectively and the coarse grid is illustrated in Figure 5.4. At each refinement level the number of nodes along every block boundary was doubled. In order to achieve a \( y^+ \) of approximately 1.0 the near wall spacing was set at \( 1.275 \times 10^{-5} \)c. Particular care was taken to generate orthogonal cells, with reasonably low aspect ratios near the leading edge and at the very leading edge the cells have an aspect ratio of 1 : 2.5. Away from the leading and trailing edges high aspect ratio cells are used in order to resolve the large flow gradients normal to the sail. Unfortunately, since the wind tunnel model rotates in relation to the tunnel walls a new grid was required for each new angle of attack.

For all cases spatial interpolation was carried out using a bounded high-resolution advection scheme, details of which can be found in chapter 3. Time integration was carried out using second-order backward Euler time-stepping and a series of 4 inner-iterations within each time-step to update the non-linear coefficients.
5.2 Results

5.2.1 Convergence studies

Figure 5.5 shows the time-averaged force coefficients for the three grids at an angle of attack of 15 degrees which was felt to be a good representation of downwind sail flows. The force coefficients are plotted against $1/N$ which is a non-dimensional measure of the grid spacing where $N^2$ is the total number of cells in the grid. The medium grid produced a lift coefficient that was 1.6% smaller than the fine grid result. For the purpose of this preliminary study, the medium grid was judged to be acceptable, and the solutions are within 3% of the extrapolated value for lift.
A convergence study was also carried out on the time step. Solutions were compared using time steps of 0.025s, 0.0125s and 0.00625s. The lift coefficient calculated using the medium time-step was just 0.65% lower than the lift coefficient calculated using the short time step. For all simulations the 0.0125s was used for the time-step which corresponds to approximately 32 iterations per period of vortex shedding for the simulation at 15 degrees. In all cases the simulations were run until the solutions had settled into regular periodic vortex shedding, i.e., the forces had constant means and amplitudes and the period of vortex shedding was stable.

5.2.2 Wind tunnel - CFD comparison

The time-averaged force coefficients for the CFD and wind tunnel results are presented in Figure 5.6. In the experimental data a peak in the lift coefficient exists at 10 degrees which is associated with the so-called ideal angle of incidence where there is no leading edge bubble disrupting the flow. There is a drop off in lift above 10 degrees associated with the formation of the leading edge bubble. Below 10 degrees the flow stagnates on the leeward side and a separation bubble develops on the windward side of the sail and for flexible sails in this situation the luff would collapse. Above 15 degrees the lift starts to increase again. At an angle of attack of 30 degrees the model is fully separated yet there is little sign of the lift coefficient stalling. Also the lift coefficient values at 30 degrees are very high with the wind tunnel measurement above 3.5. These high lift coefficient values are due to the influence of the wind tunnel walls which enables large suction values on the leeward surface, they are also the reason why the sail seems to be resisting stall.
Similar trends are evident in the CFD results. The ideal angle of incidence is slightly lower (around 8 degrees) in the CFD, and at 30 degrees the lift curve for the SST model is starting to level off. At this angle of attack the flow is fully separated with leading edge bubble detaching and reattaching in a semi-chaotic manner. At lower angles of attack vortex shedding is regular and a vortex street exist in the wake (see Figure 5.7).

The general structure of the two-dimensional flows - both in the wind tunnel and from the CFD - compares well with streamlines visualised on three-dimensional model sails near mid-span. Figure 5.8 illustrates streamlines computed using CFD overlaid upon a photo of smoke-lines from a three-dimensional wind tunnel test. Observations from smoke visualisation indicate that three-dimensional downwind sails generally separate at approximately 50%-60% along the chord and two-dimensional CFD results (without mainsail) indicate that angles of attack of between 20 and 25 degrees are required in order to obtain this degree of flow separation.
In general lift forces are overpredicted by CFD which is a commonly reported shortcoming of turbulence models. Overprediction of lift by CFD codes is typically due to spuriously large values of the eddy viscosity causing the boundary layer to remain attached in adverse pressure gradients that would normally cause separation. This effect has been documented for many similar flows such as the NACA 4412 airfoil at maximum lift (Menter, 1996) (see section 3.4.2). However in such high-lift validation studies comparisons between CFD and experiment have shown better agreement than the current comparison.

In our case the primary source of the error is attributed to three-dimensionality in the wind tunnel tests. Oil-streakline and surface-tuft flow-visualisation was performed which illustrated spanwise variation of the separation point and significant cross-flow in the separated region near the trailing edge. Upstream of separation the flow was nominally two-dimensional, although the oil streams did bend slightly immediately upstream of the separation line. The three-dimensional structures present in the wake originate at the tunnel walls where the vorticity within the boundary layer on the wind tunnel wall interacts with the boundary layer on the sail creating a horseshoe vortex around the leading edge of the sail model. Consequently, there is a greater degree of flow separation near the tunnel walls, and this effect influences the flow across the entire span of the model.

A basic sketch of the flow structure is illustrated in Figure 5.9. On the near side of the foil the flow visualisation results are summarised in schematic form with the arrows representing the directions of surface tufts. Oil streaklines are also shown, upstream of the separation line. On the far side of the foil a schematic of the wake structure is illustrated. At approximately quarter and three-quarter span longitudinal vortices exist in the wake. This vorticity develops from the second (lower) separation bubble of the vortex pair which rolls up and separates into the wake.

This wake structure was confirmed using a three-dimensional CFD model and the streamlines are illustrated in Figure 5.10. The grid that was used was based upon the coarse grid from the two-dimensional
study with 60 grid points in the spanwise direction and a symmetry plane at the midspan of the model. The surface plots of pressure coefficient and skin friction in Figure 5.11 illustrate spanwise variations and the skin friction contours that closely follow the separation line patterns observed in the surface oil-flow visualisation.

The presence of the longitudinal counter rotating vortex pair causes downwash near mid span and upwash near the tunnel walls. Through examination of the streamlines upstream of the foil it is clear that the net effect of the three-dimensional wake is to reduce the effective angle of attack to the leading edge across the majority of the span. The leading edge bubble in the three-dimensional simulation is nominally two-dimensional and of almost constant length across 90% of the foil’s span (the bubble breaks down near the side walls). At this angle of attack (15°) the leading edge bubble reattaches at 0.033c and for the equivalent two-dimensional simulation reattachment occurred at 0.077c. The smaller leading edge separation bubble in the three-dimensional simulation is due to the lower effective angle of attack imposed due to downwash induced by the three-dimensionality of the wake.

From Figure 5.6 it is evident that the wind tunnel experiments predict the ideal angle of attack to be approximately 2 degrees higher than the CFD results indicate. This higher ideal angle of attack is due to downwash induced by the three-dimensional wake. Despite using tape to seal of the tips between the model the side walls, there is still likely to be a small amount of tip leakage. Flow visualisation was performed with the gaps at the tips left unsealed and the flow was observed to be considerably more three-dimensional. Force measurements were also taken with the tape removed and the discrepancies with the CFD results were greater. Notably the ideal angle of attack was approximately 2.5 degrees higher than the previous wind tunnel experiments.

Unfortunately the errors caused by the three-dimensionality of the wind tunnel results are too large to allow reliable evaluation of the relative performances of the turbulence models. The forces from the three-dimensional CFD results (at 15°) were in better agreement with the wind tunnel experiments than the two-dimensional simulations. The lift was just 2.5% lower than the experimental results and the drag was 9.3% lower. However the grid was relatively coarse and no refinement studies were performed. The grid refinement study in Figure 5.5 indicates that the forces can be expected to rise with grid refinement suggesting that a grid independent solution is likely to be even closer to the force values measured in the experiments. The three-dimensional simulations were performed primarily for visualisation and to aid understanding of this complex three-dimensional flow. To draw stronger conclusions would require a more in-depth study of the three-dimensional flow.

An attempt was made to record the Strouhal number in the wind tunnel using a hot-wire anemometer positioned in the wake. However, Fourier analysis showed only low frequency modes which were attributed to fluctuations of three-dimensional wake structures rather than unsteady vortex shedding. In order to determine the Strouhal number - or even prove that unsteady vortex shedding is present for this flow case - it is necessary to have nominally two-dimensional flow.
Figure 5.9: The wake structure.

Figure 5.10: The wake structure computed by the 3D model, \( \alpha = 15^\circ \) (image reflected through symmetry plane).

Figure 5.11: Suction side shear stress and pressure coefficient surface plots computed using the 3D model.
5.3 Conclusions

The main goal of the preliminary study was to obtain insight into the behavior of two-dimensional sail flows and to gain an initial gauge of the performance of different turbulence models. Direct comparison between CFD and the wind tunnel tests was compromised by the presence of significant crossflow in the separated region of the wind tunnel model. Visualisation techniques were used to show that three-dimensional structures in the wake were dramatically altering the flow and inducing downwash across the span of the model. Consequently the drag in the wind tunnel tests was increased and the lift decreased. A three-dimensional CFD simulation was performed in order to help visualise the wake structure and the computational model agreed well with the flow visualisation performed in the wind tunnel. CFD proved to be a useful tool in gaining understanding of a particularly complicated flow problem.

Due to the inability to produce nominally two-dimensional flow, the preliminary wind tunnel tests proved to be a poor test case for the validation of turbulence models. The error associated with three-dimensional effects in the wind tunnel outweighed the differences between the CFD results making it impossible to discern between the performances of the turbulence models tested. Whilst the literature suggests that the SST model is the most suitable two-equation turbulence model for flows with mild separation, little can be said here about its suitability for highly separated flows.

In view of these results, it is highly likely that the sail section data presented by Milgram (1971, 1978) is affected by similar three-dimensional effects. Milgram notes that wall effects are neglected and that it was impossible to estimate the influence that non-uniform flow separation would have on the results. Milgram's model had an aspect ratio of just 2.2 and there were 1.6mm gaps at the tips of the model, hence it is likely that three-dimensional effects were considerable.

The models tested by Wilkinson (1984, 1989, 1990) had an aspect ratio of 3.0 and hence, for the high camber sections (up to 17.5% camber) at least, some degree of three-dimensionality must have existed in the wake. However, Wilkinson's data was in the form of pressure coefficient measurements and boundary layer profiles at the midspan of the model and it is possible that three-dimensional effects are isolated near the walls and the flow at midspan was nominally two-dimensional. Wilkinson confirmed this by comparing pressure distributions three different spanwise locations. Unfortunately the data recorded by Wilkinson does not include velocity and turbulence measurements within the leading and trailing edge separation regions. Without such data it is difficult to compare turbulence models and to understand the physical and mathematical reasons for their relative performance. Also the models tested by Wilkinson were derived from upwind sail designs and have less camber than downwind sails, consequently the data compiled by Wilkinson is less than ideal for the validation of turbulence models for downwind sail design.

The wind tunnel tests presented in this chapter are insufficient for validation of CFD results and it was evident at the completion of the project that further wind tunnel tests were required. From the results of this preliminary study it was concluded that significant changes to the setup and measurement techniques would be necessary for subsequent wind tunnel tests; they are:

1. High aspect ratio models are required in order to minimise three-dimensional effects. CFD simulations can be used to determine suitable aspect ratios.

2. A larger wind tunnel is required in order to obtain higher aspect ratios whilst maintaining practical chord lengths for resolution of measurement techniques and to maintain a convenient Reynolds
number.

3. Measurements need to be conducted at midspan in order to remove errors associated with three-dimensional effects at the model/wall junction, i.e., no span-averaged measurements of lift, drag etc.

4. Velocity and turbulent stress measurements are required in order to provide a more detailed understanding of sail flows. In particular the physics of the leading and trailing edge separation regions as well as the boundary layer leading up to the trailing edge separation point need to be understood better.

These conclusions were used to form the strategy for the wind tunnel tests carried out at NASA Ames in conjunction with Stanford Yacht Research and the Center for Turbulence Research at Stanford University. The results of these tests are presented in the next chapter.
Chapter 6

Validation Study

6.1 Introduction

The preliminary wind tunnel and CFD study (Chapter 5) illustrated the need for wind tunnel tests of downwind sail sections using high aspect ratio models. To facilitate a large aspect ratio either a larger tunnel or a pressurised (variable density) tunnel was required in order to keep the Reynolds number at a reasonable level. Through collaboration with Stanford Yacht Research arrangements were made to conduct the tests at NASA Ames Research Center, Moffet Field, California in a 7 x 10 foot subsonic wind tunnel. These tests were conducted with the help of Professor Margot Gerritsen and Dr. Andrew Crook and Stanford Yacht Research as well as Dr. James Bell and Dr. Edward Schairer at NASA Ames.

6.2 Wind tunnel experiments

6.2.1 Model Design

A circular arc was once again chosen to represent our two-dimensional downwind sail. This time an arc with a camber of 25% was used which is just slightly higher than the 24.7% camber arc that was used in the preliminary study. Three-dimensional CFD analysis of a model with an aspect ratio of 15 showed nominally two-dimensional flow over the middle 50% of the span. Figure 6.1 illustrates the surface pressures and wall shear computed in the three-dimensional CFD analysis. The simulation used a symmetry plane at the midspan of the model and whilst significant three-dimensional effects are evident at the wall junction, the flow near midspan is nominally two-dimensional. Figure 6.2 illustrates the flow field using streamlines across the span of the model. The effect of the horseshoe vortex at the wall junction is clear with a large degree of separation near the wall, however the effect of the wall is isolated to the outer ends of the model and near midspan there is no evidence of crossflow.

Whilst analysis showed that an aspect ratio of 10 would be reasonable it was decided that there was significant risk that three-dimensional effects may still impair the flow quality. The CFD analysis used grid resolutions equivalent to the coarse grid that was used in the preliminary study and it is possible that numerical diffusion may have reduced the extent of the three-dimensional effects. Consequently the aspect ratio 15 model was selected for the wind tunnel tests.

The initial concept of the design was to force the model to conform to the desired shapes by applying
Figure 6.1: CFD surface pressure and wall shear ($\tau_w$) plots for the three-dimensional analysis of the wind tunnel model (tunnel wall at bottom of figure, tunnel centerline at top).
Figure 6.2: CFD streamlines and surface pressures for the three-dimensional analysis of the wind tunnel model (tunnel wall at figure left, tunnel centerline at figure right).

tension to a thin steel sheet that was clamped to the sail shape at either end. This concept is illustrated in Figure 6.3 and one of its main advantages is its potential to generate different sail shapes using the same steel sheet. Unfortunately safety requirements enforced a thickness on the steel sheet of 0.75mm and initial tests showed that this thickness prevented the model from conforming to a consistent shape over the span of the tunnel.

Whilst a redesign of the model setup would have been desirable after the initial test-of-concept it was decided that the period of time before the scheduled test dates was too short and that the most simple solution was to use the same setup but to pre-shape the steel sheet. This approach requires a different steel sheet for each different sail shape tested.

### 6.2.2 Model Setup

The model was mounted vertically in the tunnel and was supported between the force balance under the tunnel and the tunnel frame above the tunnel. This allowed the tension force to be measured via the force balance. Tension was applied using two steel plates connected via 4 threaded rods as is illustrated in Figure 6.4. The top tensioning plate was fixed to the tunnel frame while the model itself was allowed to rotate with the turntable through the use of a tapered roller bearing mounted on the lower tensioning plate. A tension force of approximately 1 tonne was applied by turning the 4 threaded rods simultaneously using a timing chain. Photographs of the tensioning system are provided in Figure 6.5.

The model itself has dimensions 2616.2mm x 142.2mm however only 7 feet, or 2133.6mm of the span of the model was within the tunnel. Splitter plates were used to reduce the influence of the vorticity of the tunnel wall boundary layer. These splitter plates were positioned 50mm inside the tunnel walls and
Figure 6.3: Shaping the model using tension.

Figure 6.4: A schematic of the tensioning system (not to scale).
Figure 6.5: Photographs illustrating the tensioning system and the mounting of the model to the tunnel frame.

were attached to the turntables so that they could rotate with the model. The set-up of the model within the tunnel is illustrated in Figure 6.6.

Initial tests revealed an aerelastic instability when the wind speed rose above 30m.s$^{-1}$. The instability was found to be a flutter mode caused by torsional divergence of the model. The flexural axis of the model was positioned behind the center of aerodynamic pressure and consequently as the tunnel speed was increased the model twisted so that the angle of attack increased. At 30m.s$^{-1}$ the model stalls which leads to a reduction of lift and causes the center of pressure to move aft. As the center of pressure moves aft the torsional loads decrease and the model twists back (reducing angle of attack) until the cycle repeats and the aerelastic instability is created.

Ideally the rigidity of the model could have been increased by increasing the tension however this option was not allowed due to safety restrictions imposed by NASA Ames. Instead the instability was removed by clamping the model at 1/3 and 2/3 span and running stays to the floor and ceiling of the tunnel as illustrated in Figure 6.7. Flow visualisation presented in section 6.3 shows that the clamps and stays did not disrupt the flow near the midspan of the model.

6.2.3 Test Conditions

The model has a chord length of 142.24mm and the tunnel was run at 40m.s$^{-1}$ which gives a Reynolds number of approximately $3.77 \times 10^5$. Whilst the tunnel was capable of speeds up to 110m.s$^{-1}$, 40m.s$^{-1}$ was determined to be the maximum speed achievable without risk of compressibility errors. The tunnel has a series of turbulence-reducing screens that reduce the turbulence intensity to a maximum of 0.25% in the test section.
Figure 6.6: The model set up in the 7 × 10 foot subsonic wind tunnel.

Figure 6.7: The clamps and stay setup that was used to increase the torsional rigidity of the model.
6.2.4 Pressure measurement setup

Surface pressure measurements were taken using pressure sensitive paint (PSP). Most of the PSP setup and processing was carried out by James Bell and Edward Schairer of NASA Ames Research Center (see Bell, Schairer, Hand and Mehta (2001) for a review of PSP measurement techniques). PSP was seen as an attractive means of gathering pressure data since it is noninvasive. Accurate static pressure tappings would require mounting the pressure tubing inside the model which was undesirable due to the model thickness that would be required to facilitate the tubing.

Pressure sensitive paints are in fact sensitive to the presence of oxygen (Liu, Cambell, Burns and Sullivan, 1997). They are typically comprised of two parts, an oxygen sensitive fluorescent molecule and an oxygen permeable binder. When molecules absorb a photon they are excited to a new energy state and will recover to their ground state though the emission of a photon of a longer wavelength. Oxygen sensitive molecules behave differently, the presence of oxygen aids the transition to the ground state though a process known as oxygen quenching. Thus the intensity of the light emitted is reduced and molecules under high oxygen pressure will emit light of lower intensity than molecules under low pressure. The intensity of the light is measured using digital photography and the intensity field is converted to static pressures using calibration data for paints at known pressures. The brightness of the pressure sensitive paint is converted to pressure using the relation,

\[
\frac{I_0}{I} = A + B \frac{p}{p_0},
\]

where \( I \) and \( I_0 \) are the light intensities emitted at pressures \( p \) and \( p_0 \) respectively and \( A \) and \( B \) are calibration coefficients determined through reference to either static pressure tappings on the model or calibration tests for the paint at set temperature and pressure.

PSP techniques were originally developed for transonic flows and it is difficult to obtain accurate pressure data at low speeds. This is because it is the absolute pressure that is measured by the paint and consequently noise becomes dominant when the dynamic pressure is low. However, Bell (2002) has illustrated that through careful experimental technique and the use of biluminophor paints (i.e. paints with both pressure sensitive and insensitive components) and data reduction, PSP can be useful at flow speeds as low as 17m.s.\(^{-1}\).

Images are taken with the tunnel running at the test condition ("wind-on") and with the tunnel off ("wind-off"). Bias error due to pixel-to-pixel variations is then reduced by subtracting images taken with the shutter closed ("dark-images") from the raw images. The "wind-on" and "wind-off" images are then ratioed in order to determine the ratio \( I_0/I \) in equation (6.1). Since the model or camera may shift between the "wind-on" and "wind-off" states the "wind-on" images must be mapped onto the "wind-off" image. This is achieved using algorithms that search for dark reference targets on the model (see Figure 6.8) and then transform the wind-on image using a biquadratic interpolation (Bell and McLachlan, 1996). Using the reference targets the images are mapped onto a three-dimensional surface grid that accounts for the camber of the model. The pressures can then be determined from equation (6.1) in a pixel-by-pixel manner and since the pixel intensities are ratioed between the "wind-on" and "wind-off" runs any error associated with uneven paint thickness or inconsistent lighting is removed.

The data is further improved through the biluminophor technique which uses a mixture of two paints,
one that is sensitive to pressure and a reference paint that is insensitive to pressure (Bell, 2002). Two cameras positioned side-by-side capture the emitted light of each paint simultaneously; one camera uses a filter to isolate the wavelength from the reference paint, the other captures the wavelength of the pressure-sensitive paint. The pressure-insensitive images are taken in both "wind-on" and "wind-off" states in the same fashion as the pressure-sensitive images. Since the pressure-insensitive images should be identical between the "wind-on" and "wind-off" states the images can be correlated to improve the pressure-sensitive image data by reducing error associated with uneven lighting between the "wind-on" and "wind-off" states.

The quality of the data can be further improved through the use of smart choices for the length of the sample times (i.e. the shutter speed) and the number of samples and by carefully monitoring the test conditions. By using long sample times (allowing the camera to receive more photons) and many samples from which to obtain an average image the random noise can be reduced significantly. When testing at low dynamic pressures noise can be significant and may dominate the pressure field (see Figure 6.9). Averaging over a large number of samples can remove this source of error, but only at the expense of increased bias error.

Unfortunately pressure sensitive paint is sensitive to temperature as well as pressure and consequently if the tests are long and the temperature is not consistent the sensitivity of the paint will drift between the wind-on and the wind-off condition. Consequently a compromise needs to be made between the amount of bias error and the amount of noise. Bell (2002) presents a study investigating when reduction in bias error or random error is more desirable for improving overall PSP accuracy. Error levels as low as 22 Pa were obtained on a small wind tunnel model at a flow speed of 17 m/s \(^{-1}\) \((p_d \approx 170 Pa)\) (Bell, 2002). The technique presented in Bell’s study was followed in the current tests under the guidance of James Bell.
Figure 6.9: Comparison between the raw $C_p$ data for a single row of pixels and the smoothed data (65 x 7 pixel averaging stencil) for the suction side of the model at $\alpha = 20^\circ$. Note that the raw data comes from an average image produced from a series of images taken during the test.

The wind tunnel used for the current tests does not have temperature control and is hence more prone to bias error due to temperature variations than the isothermal tunnel used by Bell (2002). However these two-dimensional tests have the advantage that only two-dimensional data are required for the pressure coefficient plots and consequently noise can be reduced by averaging over a range of pixels in the spanwise direction. The effectiveness of this approach is illustrated in Figure 6.9 where the data is smoothed using a 65 x 7 pixel stencil, i.e. the data is averaged across 65 pixels (63.5mm) in the spanwise direction and 7 pixels (6.5mm) in the chordwise direction.

### 6.2.5 The CFD model

Four different domains were used in this study in order to assess the influence of the freestream boundaries. These domains have dimensions 15c x 8c, 29c x 16c, 57c x 32c and 113c x 64c and the 29c x 16c domain is illustrated in Figure 6.10.

![Diagram of the CFD model](image_url)
For all simulations a chord length of 1m was used, and the Reynolds number based on chord length was set at approximately $3.77 \times 10^5$ in order to match the Reynolds number used in the experiments. At the inlet the total pressure and the angle of incidence were set to match the experiments. The freestream turbulence conditions were also set at the inlet using a turbulent intensity of 0.25% and a length scale of 0.001m, however as mentioned in chapter 3 the simulations are independent of these values.

An illustration of the computational grid used in this study is presented in Figure 6.11. A grid convergence study is presented in section 6.4.3 using three grids that are referred to as coarse, medium and fine with 25760, 100100 and 406260 cells respectively. The $y^+$ values are approximately 0.2, 0.1 and 0.05 respectively for the coarse, medium and fine grids. The medium grid is illustrated in Figure 6.11. Particular care was taken to generate orthogonal cells, with low aspect ratios near the leading edge and at the very leading edge the cells have an aspect ratio of $1:1$. The leading edge region is illustrated in the close-up view in Figure 6.11. Further along the sail high aspect ratio cells are used in order to resolve the large flow gradients normal to the wall.

Figure 6.11: The computational grid (medium grid density, medium domain size).
CHAPTER 6. VALIDATION STUDY

For all cases spatial interpolation for the convective terms was carried out using a bounded high-resolution advection scheme, details of which can be found in chapter 3. Time integration was carried out using second-order backward Euler time-stepping and a series of 4 inner-iterations within each time-step to update the non-linear coefficients.

It is important to note that downwind sails exhibit periodic vortex shedding which is a mean-flow phenomenon. This vortex shedding is not a turbulent process and can be suitably resolved by the RANS equations which average out the turbulent behavior. Simulation of unsteady vortex shedding should not be confused with Large Eddy Simulation (LES) where turbulent eddies of sufficient size to be captured by the grid are resolved using the Navier-Stokes equations, whilst the smaller eddies are accounted for using sub-grid models.

In all cases the simulations are run until the mean and peak forces remain stationary and the period is steady. In the results of this chapter the flow variables are averaged over the periodic cycle to give time-averaged results. Here the averaging is performed as the simulations are being carried out, i.e. the simulations are run until a periodically steady state is reached and then the simulations are run for several more periods while a running average is calculated for each flow variable.

6.3 Wind Tunnel Results

6.3.1 Surface flow visualisation

Surface flow visualisation was performed in order to validate the two-dimensionality of the experiments. A mixture of florescent paint pigment and kerosene was carefully and evenly applied to the surface of the model. As the tunnel is run up to speed shear stress on the surface causes the kerosene-paint mixture to run and after several minutes the kerosene fully evaporates leaving an imprint of the flow on the model (see Figure 6.12). The mixture coagulates in regions of low or zero shear such as separation and reattachment lines. These are clear in Figure 6.12 with a reattachment line approximately 10% downstream of the leading edge and a separation line shortly downstream of mid chord. A secondary separation bubble is also evident within the leading edge bubble, this is illustrated in greater detail in Figure 6.13.
Figure 6.12: Flow visualisation experiments for the 25% camber circular arc at $\alpha = 20^\circ$.

Figure 6.13: Close up of the surface flow visualisation illustrating the presence of a secondary recirculation bubble within the leading edge bubble.

The flow visualisation experiments illustrate that the flow is nominally two-dimensional at mid span. The leading edge reattachment length is particularly straight with only a slight decrease in reattachment length near the clamps. However, the trailing edge separation line is not as straight and is significantly affected by streamwise vorticity created by the clamps. At the clamps there is an abrupt change in lift which causes a streamwise vortex to be shed from the spiral node illustrated in Figure 6.12. This change in vorticity creates a downwash effect near the clamp which locally reduces the amount of separation. These three-dimensional effects extend spanwise several chord lengths from the clamps, however around the tunnel centerline the separation line is straight and the flow is nominally two dimensional.
6.3.2 Surface pressure measurements

Figure 6.14 presents a comparison between two separate PSP runs and the CFD results at \( \alpha = 20^\circ \). The CFD and PSP pressure coefficient distributions have the same general shape indicating a suction peak associated with the leading edge bubble, a recovery region along which suction increases followed by an adverse pressure gradient and subsequent boundary layer separation. It is pleasing to note that the trough at the end of the leading edge bubble is located at the same chordwise location in both the PSP and CFD results. This indicates that both methods are predict similar leading edge bubble lengths. Moreover in both the PSP and CFD data the adverse pressure gradient ends in approximately the same chordwise position and the pressure coefficients are comparable along the constant pressure region. This indicates that the trailing edge separation region is of similar size and shape in both sets of data.

However there is a discrepancy in the absolute magnitude of the pressures between the CFD and PSP. Also the two PSP runs present different pressure distributions, and the discrepancy is worse near the leading edge of the model.

In the calibration of the PSP there was a poor correlation between the pressure tapping data and the PSP measurements. Initially there were just two pressure tappings per side of the model and an extra tapping was positioned near the leading edge in an attempt to improve the calibration of the PSP data. However with the extra pressure tappings the correlation with the PSP data was worse which indicates that the bias errors are not uniform across the surface of the model. Moreover the pressure distribution derived from the PSP data seems questionable near the leading edge bubble. The data suggests that the pressure coefficient actually decreases from the leading edge to the reattachment point at approximately \( x/c = 0.12 \). Naturally the pressure should increase along the separation bubble and the CFD pressures follow a shape that is similar to the pressure distributions found in the flat plate experiments by Crompton and Barrett (Crompton and Barrett, 2000; Crompton, 2001). Unfortunately this leading edge region provides a significant contribution to the lift force and is of considerable interest.

```
Figure 6.14: Comparison between CFD and PSP runs at \( \alpha = 20^\circ \).
```
for our validation exercise.

It appears that there were several issues with the pressure sensitive paint that caused unsatisfactory results:

1. Change in paint temperature between the wind-on and wind-off runs and through the runs themselves. In "run 1" the wind-off runs were performed immediately after the corresponding wind-on run whereas in "run 2" a series of wind-on runs were performed before the corresponding wind-off runs were performed. Consequently the PSP error is greater for "run 2" compared with "run 1". This temperature change is likely to have been non-uniform across the surface of the model. Unfortunately almost all the runs performed in the testing period were conducted following the procedure of "run 2".

2. High curvature of the model creates an error due to the large variation of surface incidence. PSP paints emit light equally in all directions, however since two cameras were used side by side (one for the pressure sensitive paint and one for the reference paint) there was an error created by the different perspectives. This error is more pronounced at the leading edge where the cameras were at a particularly oblique angle.

3. The tests were conducted at a dynamic pressure ($p_D$) of 960 Pa, which is less that 1% of the atmospheric pressure and consequently a large amount of noise was present in the data. Large sample times were used to account for the low signal-noise ratio and this in turn led to greater bias error due to temperature changes.

Following the experiments it was concluded that the following measures could be made to significantly improve results from future tests:

1. Usage of an isothermal wind tunnel.
2. Usage of a model made out of a better thermal insulator than steel (e.g. carbon fibre or fiberglass composites).
3. Performing wind-off measurements taken before and after each wind-on test so that the average wind-off measurement better represents the average test conditions.
4. Shorter sample times to reduce bias error, whilst performing a larger number of samples and averaging to reduce noise.
5. Cameras mounted above one another rather than side-by-side to reduce errors associated with surface incidence.
6. An extra camera pair to further reduce error associated with surface incidence.

The issues with the PSP data were not recognised during the testing. In fact since it takes considerable time to post-process the PSP data it was not until after the testing period that any inaccuracies in the PSP data were identified. The experiments were to be repeated in the next available wind tunnel slot, however regrettably the NASA AMES 7 × 10 foot subsonic wind tunnel was closed shortly after the
CHAPTER 6. VALIDATION STUDY

2.36
2.32
2.28
2.24
0 15 30 45 60 75 90 105 120

Domain Length (c)

Figure 6.15: Lift coefficient versus domain length for the SST model at $\alpha=20$ degrees.

conclusion of the first testing period. Attempts were made to arrange subsequent testing in other wind tunnels and a composite model was constructed complete with pressure tappings, however unfortunately there was not enough funding to complete the wind tunnel project. It is still hoped that some time in the near future the experiments will be repeated and suitable data will be obtained for this demanding experiment.

6.4 CFD Results

6.4.1 Domain size verification

Determining the correct domain size is particularly important for downwind sails due to their bluff nature. Unlike more streamlined airfoils, downwind sails typically have a region of positive pressure that extends well upstream of the sail. For simulations computed using inlet conditions where the velocity is specified this can present a problem. If the pressure field ($p$) of the sail reaches as far upstream as the inlet boundary then the total pressure of the boundary is greater than the inlet dynamic pressure ($p_t = p + p_d = p + 1/2pU^2$). Elevated total pressure at the inlet is undesirable since effectively more energy is supplied to the system than is desired and consequently the forces will be spuriously larger. If the total pressure at the inlet boundary is greater than the dynamic pressure then the stagnation pressure on the sail will also be spuriously high. Consequently pressure coefficients greater than 1.0 can appear if the dynamic pressure is used to non-dimensionalise the pressure coefficient. As mentioned in chapter 3 it was found that this problem can be alleviated by specifying the total pressure and a direction at the inlet instead of directly specifying the inlet velocity.

Figure (6.15) illustrates the relationship between domain size and lift coefficient for our 25% camber circular arc sail at an angle of attack of 20 degrees. For this analysis the domain dimensions were repeatedly doubled until the lift coefficient became stationary. Only with the very large domain (domain length = 113c, inlet 32c upstream of the sail) did the solution show clear independence from the domain size. However for the purpose of this study the domain length of 29c is suitable, for this domain the lift coefficient is within 0.41% of the value for the largest domain.
CHAPTER 6. VALIDATION STUDY

6.4.2 Time step convergence study

Results from chapter 5 indicated that time-step sizes at least as low as 0.008s were required in order to adequately capture the transient behavior. The influence of the time-step size for this case was investigated by starting with a time step of 0.008s and then repeating the simulations with the time step halved and then halved again, i.e. the time step sizes used are 0.008s, 0.004s and 0.002s. Simulations were performed at each angle of attack ($\alpha = 15^\circ, 17.5^\circ, 20^\circ$ and $22.5^\circ$), however only the results for $\alpha = 15^\circ$ are presented since this is the case with the fewest time steps per shedding cycle. For larger angles of attack the time step convergence was better than the $\alpha = 15^\circ$ case due to the greater resolution of time steps per shedding cycle. In Figure 6.16 the mean lift and drag coefficients are presented for the sequence of time step sizes.

As the time-step size is reduced both the lift and drag coefficients level off indicating satisfactory time-step size independence. The lift for the solution using the medium time step is within 0.035% of the lift computed using the short time step and the drag is within 0.15%. The medium time step size corresponds to approximately 58 time steps per shedding cycle for the $\alpha = 15^\circ$ case and 91 time steps per shedding cycle for the $\alpha = 22.5^\circ$ case. In all remaining simulations the medium time-step ($t = 0.004s$) was used.

6.4.3 Grid convergence study

The grid convergence study was performed at $\alpha = 20^\circ$ which is approximately where maximum lift occurs and is consequently the angle of most interest. Mean lift and drag coefficients are presented in Figure 6.17 for the sequence of grids. The force coefficients are plotted against $1/N$ which is a non-dimensional measure of the grid spacing where $N^2$ is the total number of cells in the grid.

The lift from the medium grid solution is within 0.8% of the fine grid solution and the drag is within 2.3%. The accuracy of the medium grid solution is not entirely pleasing. The lift coefficient curve does not seem to be leveling off as the grid is refined and the 2.3% difference in drag coefficient between the medium and fine grids is significant. However the lift coefficient for the medium grid is still within 0.8% of the fine grid solution and it was judged that using a more refined grid would be impractical. All subsequent simulations were computed using the medium grid resolution.
The slight grid dependence that was found for this case is due to the very fine near wall spacing. The same near wall spacing was used for both this validation case and the design study presented in the next chapter. However, the Reynolds number for this case is almost 10 times smaller than the Reynolds number in the design study and consequently the $y^+$ values are also almost 10 times smaller. In retrospect a grid with fewer cells in the near wall and more cells in the adjacent regions would have been more suitable and may have demonstrated superior grid independence.

### 6.4.4 Maximum lift

The computed lift coefficients near maximum lift are presented in Figure 6.18. The data points ($\alpha = 15^\circ, 17.5^\circ, 20^\circ, 22.5^\circ$) are fitted using a cubic polynomial (the plotted curve) which is differentiated to obtain the angle of maximum lift. Using this method the SST and $k - \omega$ models were found to predict maximum lift coefficients of 2.35 and 2.52 respectively. For the SST model the maximum lift coefficient is reached at $\alpha = 21.19^\circ$, whereas the $k - \omega$ model predicts the angle of maximum lift as 19.22°. In Figure 6.19 the lift coefficient versus angle of attack is plotted again for the SST model with additional points at $\alpha = 18.75^\circ$ and 21.25° included. These two points lie close to the polynomial fit that was used to determine the maximum lift. Therefore the polynomial approximation of the lift coefficient provides a suitable approximation in the vicinity of maximum lift.
6.4.5 Pressure coefficient comparison

A comparison between the time-averaged pressure coefficient for the experimental results and the CFD at $\alpha = 20^\circ$ is presented in Figure 6.20. At this angle of attack both the SST and $k - \omega$ models are close to maximum lift, as can be seen in the plots of lift coefficient versus angle of attack (Figures 6.18a and 6.18b).

The pressure coefficient plots predicted by the two turbulence models - which are in reasonably close agreement with each other - are significantly different to the pressure distribution measured in the experiments.

6.4.6 Force coefficient plots

The lift and drag coefficients for the 25% camber arc are presented in Figures 6.21 and 6.22. Between $\alpha = 5^\circ$ and $\alpha = 10^\circ$ Figure 6.21 shows a sharp and approximately linear increase in lift. In fact the slope of the lift curve in this region is more than 2 times as steep as the value, $2\pi$, predicted by inviscid thin airfoil theory. This suggests that severe viscous effects are in operation and indeed they are. Even at these low angles of attack the sail section is separated over approximately the final 30% of the leeward surface (suction side). However, and perhaps more significantly, there is considerable separation on the windward surface (pressure side). This can be seen in the pressure coefficient plots presented in Figure 6.23. For the case at $\alpha = 5^\circ$ (yellow) there is a negative pressure coefficient over almost the entire length of the windward surface suggesting that a large separation bubble is present. In thin airfoil theory the downwash (and hence also the circulation and lift) is calculated by assuming that the flow leaves the trailing edge in a direction tangential to the surface which is a good approximation for low camber airfoils at low angles of attack. However in our situation the flow near the trailing edge is severely influenced by viscous regions of the flow and so the downwash is not closely related to the angle of the trailing edge. This is illustrated in more detail in Figures 6.24 and 6.25 where it can be seen that the downwash at the trailing edge increases significantly as the windward separation bubble disappears.

As the angle of attack is increased above $\alpha = 5^\circ$ the windward surface separation bubble diminishes.

Figure 6.19: Evaluation of the fitted polynomial for the SST lift polar.
Figure 6.20: Comparison between the PSP data and CFD at $\alpha = 20^\circ$.

Figure 6.21: Lift and drag coefficients plotted against angle of attack for the 25% camber arc. These coefficients were computed using the SST and $k - \omega$ turbulence models.
rapidly. It disappears entirely at around $\alpha = 9^\circ$, the ideal angle of attack (Figure 6.25) where there are no leading edge separation bubbles on either the windward or leeward surface. This ideal angle of attack coincides with the maximum lift to drag ratio which is illustrated in Figure 6.22. As the windward separation bubble disappears between $\alpha = 5^\circ$ and $\alpha = 9^\circ$, the downwash angle at the trailing edge increases dramatically and hence so does the lift. This explains the very large slope in the lift coefficient (Figure 6.21) between $\alpha = 5^\circ$ and $\alpha = 10^\circ$.

In Figure 6.23 we can see the pressure coefficient plot at $\alpha = 10^\circ$ (red), where the angle of attack is slightly greater than ideal. At the leading edge there is only a small pressure difference between leeward and windward surfaces which coincides with a very small suction side leading edge bubble. Without a large leading edge bubble to disrupt the leading edge the flow is able to remain attached for a significant distance along the sail. Note that the suction peak near the middle of the sail section is greatest for this angle of attack and that the adverse pressure gradient is more severe for this case as compared with the higher angles of attack, yet here the trailing edge separation region is smaller. This illustrates the aerodynamic advantage of minimising the size of the leading edge bubble.

Between $\alpha = 10^\circ$ and $\alpha = 20^\circ$ both the leading edge bubble and the trailing edge separation regions grow in size as the angle of attack increases. This increase is approximately linear with a slope much lower than $2\pi$. This low increase in lift (compared with that predicted by thin airfoil theory) is due to the formation and growth of the leading edge bubble and the forward movement of the trailing edge separation region. At maximum lift there is a significant suction peak at the leading edge which is a source of much of the lift of the sail section. Here the aft suction peak in the attached flow region ($0.2 < \alpha/c < 0.4$) is much smaller and the disruption of the leading edge bubble is clearly felt. As will be discussed further in section 6.4.8 the boundary layer downstream of the leading edge reattachment point is in a mode of recovery and consequently has trouble resisting the adverse pressure gradient over the aft.
Figure 6.23: Time-averaged $C_P$ plots for the 25% camber arc at various angles of attack. All plots were computed using the SST turbulence model.

half of this suction peak.

Both the $k - \omega$ and SST models predict stall at an angle of attack of approximately 20°. The $k - \omega$ model predicts a maximum lift of 2.52 at $\alpha = 19.22^\circ$ compared to a $C_{L(\text{max})}$ of 2.36 for the SST model at an angle of attack of 21.19°. Beyond these angles the lift slowly decreases and the drag starts to increase more rapidly. Whilst these effects are certainly undesirable, stall for downwind sail section is not as abrupt as that found for airfoils or low camber sails that exhibit leading edge stall. In those cases the loss of lift and increase in drag is abrupt and often difficult to foresee. Since downwind sails exhibit such a soft stall, when trimming downwind sails it is difficult to tell whether the sail is over trimmed or not.

By $\alpha = 25^\circ$ the 25% camber arc is fully stalled. Figure 6.21 illustrates a clear drop off in lift and the pressure coefficient plot in Figure 6.23 has a very flat curve on the leeward side. Stall occurs as both the trailing and leading edge separation regions begin to grow at an increasing rate. It should be noted that the lift starts decreasing at $\alpha \approx 20^\circ$ (SST model) and at this stage the flow is not fully detached. It appears that the leading edge reattachment point and the trailing edge separation point meet for the first time somewhere between $\alpha = 22.5^\circ$ and $\alpha = 25^\circ$, at which point there is a more abrupt decrease in the lift coefficient.

In the stalled flow regime ($\alpha > 22.5^\circ$) the leading edge bubble is alternately detaching and reattaching. Through this cycle the lift (and consequently also the circulation and upwash) is going through significant changes in magnitude. For example the SST model predicts $C_L$ variations of ±7.5%. At $\alpha = 27.5^\circ$ the shedding cycle appears irregular with several distinct modes although the flow does remain periodic. This can be seen in Figure 6.28 where a portion of the time-history for the transient lift coefficient is presented. In Figure 6.21 it can be seen that the lift coefficient actually increases between $\alpha = 25^\circ$ and $\alpha = 27.5^\circ$, however the $\alpha = 27.5^\circ$ result is questionable; it would be quite surprising if the SST turbulence model was able to appropriately predict fully separated vortex shedding.
Figure 6.24: A schematic of the flow past the 25% camber arc at low angles of attack ($\alpha \approx 5^\circ$).

Figure 6.25: A schematic of the flow past the 25% camber arc at ideal angle of attack ($\alpha \approx 9^\circ$).

Figure 6.26: A schematic of the flow past the 25% camber arc at maximum lift ($\alpha \approx 20^\circ$).

Figure 6.27: A schematic of the post-stall flow past the 25% camber arc ($\alpha \approx 25^\circ$).
Figure 6.28: Time history of the force coefficients for the 25% camber arc at $\alpha = 27.5^\circ$ (SST model).

6.4.7 General comparisons between the turbulence models

In the force plots presented in Figure 6.21 it can be seen that the $k - \omega$ model is predicting greater lift coefficients and lower drag. Also, as mentioned in section 6.4.4 the $k - \omega$ model predicts maximum lift at a lower angle of attack ($\alpha = 19.22^\circ$ compared with $\alpha = 21.19^\circ$ for the SST model). The variation in lift coefficients between the $k - \omega$ and SST models can be more closely examined by inspecting the behavior of the leading edge reattachment lengths and the trailing edge separation points.
Leading edge bubble length

Figure 6.29: Time-averaged reattachment lengths ($X_R$) of the leading edge bubble as a function of angle of attack ($\alpha$).

Figure 6.29 presents the reattachment ($X_R$) positions of the leading edge bubble as a function of angle of attack ($\alpha$).

In this Figure it can be seen that the $k - \omega$ model predicts a longer leading edge bubble consistently from $\alpha = 15^\circ$ to $\alpha = 20^\circ$ and as the angle of attack increases the difference in the reattachment length increases. The $k - \omega$ model predicts larger lift coefficients than the SST model and consequently predicts greater circulation and therefore also upwash to the leading edge. Consequently the leading edge effectively sees a greater angle of incidence which is part of the reason for the $k - \omega$ model predicting larger reattachment lengths.

In the validation study for the flat plate that was presented in chapter 4 the $k - \omega$ model was also found to predict a longer leading edge bubble than the SST model. The $k - \omega$ model was shown to predict lower turbulent kinetic energy levels in the outer shear layer of the leading edge bubble. Consequently in the $k - \omega$ model the shear layer did not grow as rapidly as the SST model (or the experimental results) and reattachment was delayed. In section (6.4.8) the boundary layer profiles are presented for the SST and $k - \omega$ model at $\alpha = 20^\circ$ and it will be shown that the leading edge shear layer behavior for the 25% camber arc is similar to the results presented for the flat plate.
Figure 6.30: Time-averaged separation points (\(X_s\)) of the trailing edge separation region as a function of angle of attack (\(\alpha\)).

Figure 6.30 presents the trailing edge separation position for the \(k-\omega\) and SST models. At \(\alpha = 15^\circ\) and \(\alpha = 17.5^\circ\) the SST model predicts slightly more trailing edge separation than the \(k-\omega\) model which is part of the reason why the SST model predicts lower lift coefficients. At \(\alpha = 20^\circ\) the \(k-\omega\) model predicts slightly more trailing edge separation and at \(\alpha = 22.5^\circ\) the separation point is 2.8\% forward of the separation point predicted by the \(k-\omega\) model. However the differences between the separation positions are small and it is likely that the shape and curvature of the shear layer also contribute to the discrepancies in the lift coefficient. The relationship between the wake and the leading edge bubble is complex. The shape of the wake influences the local velocities at the leading edge and therefore also affect the shape and size of the leading edge bubble. This in turn affects the development of the downstream boundary layer and the position of the rear separation point.

The \(k-\omega\) model reaches stall at a lower angle of attack than the SST model (\(\alpha = 19.22^\circ\) compared with \(\alpha = 21.19^\circ\) for the SST model) which is largely due to the longer leading edge predicted by the \(k-\omega\) model. Downstream of the leading edge reattachment point the attached boundary is in a recovery mode with the inner region of the boundary layer retarded and prone to separation under an adverse pressure gradient. As the leading edge bubble grows the downstream boundary layers become increasingly prone to separation and consequently the separation point moves further forward. It appears that for two-dimensional downwind sail flows stall occurs primarily due to growth of the leading edge bubble. Therefore designs such as leading edge slats or local leading edge thickening (inflatable luffs for example) that serve to reduce the extent of leading edge separation are very attractive. However such concepts are also illegal in most classes of yacht racing including the Americas Cup. When trimming downwind sails it is very important to maintain a stable luff (leading edge) region that is closely aligned to the onset flow.

Above \(\alpha = 20^\circ\) the time-averaged trailing edge separation point remains quite stationary for the SST model. The separation point is calculated from the time-averaged flow-field and at these angles (above \(\alpha = 20^\circ\)) the separation point is varying dramatically and by \(\alpha = 25^\circ\) the flow is oscillating between a full detached state and an attached one. Unfortunately transient results were not recorded for this case and so little can be said about the transient movement of the trailing edge separation point at these high
angles of attack.

6.4.8 Examination of the flow near maximum lift ($\alpha = 20^\circ$)

Unsteady flow visualisation

Figure 6.31 presents a sequence of flow streamlines and velocity contours evenly spaced across a single shedding cycle. These images correspond to the force coefficient time-histories presented in Figure 6.32, i.e. $\phi = 90^\circ$ corresponds to the maximum lift position (approximately $t = 0.08s$) and $\phi = 270^\circ$ corresponds to the minimum lift condition (approximately $t = 0.2s$). The sequence of images clearly shows a vortex street in the wake of the sail section, with vortices being shed alternately from the upper and lower surfaces of the sail.

Boundary layer velocity profiles

In this section suction side boundary layer velocity profiles are presented for the downwind sail test case at $\alpha = 20^\circ$. The sail surface was divided into 3 sections, the leading edge bubble region, the attached flow region and the trailing edge separation region. Within each of these sections 5 boundary layer profiles were investigated with the measurement stations located divided up according to table 6.1. Here $L_*$ is the chordwise length of the particular region, i.e. for the leading edge bubble $L_* = X_R$, for the attached flow region $L_* = X_S - X_R$ and for the trailing edge separation region $L_* = c - X_S$. The values of $X_R$ and $X_S$ are presented in table 6.2 and the size and positions of the measurement stations are illustrated in Figure 6.33.

<table>
<thead>
<tr>
<th>Measurement Station</th>
<th>$%L_*$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.5</td>
</tr>
<tr>
<td>2</td>
<td>12.5</td>
</tr>
<tr>
<td>3</td>
<td>37.5</td>
</tr>
<tr>
<td>4</td>
<td>62.5</td>
</tr>
<tr>
<td>5</td>
<td>87.5</td>
</tr>
</tbody>
</table>

Table 6.1: Chordwise positioning of the 5 boundary layer measurement stations within each flow region.

<table>
<thead>
<tr>
<th></th>
<th>$X_R$</th>
<th>$X_S$</th>
</tr>
</thead>
<tbody>
<tr>
<td>SST</td>
<td>0.10346</td>
<td>0.53719</td>
</tr>
<tr>
<td>$k - \omega$</td>
<td>0.11581</td>
<td>0.53582</td>
</tr>
</tbody>
</table>

Table 6.2: Position of the reattachment and separation points for the SST and $k - \omega$ models at $\alpha = 20^\circ$.

Figure 6.34 presents the time-averaged velocity profiles in the leading edge bubble. These profiles hold considerable resemblance to the flat plate leading edge bubble profiles that were presented in chapter 4. The only notable difference with the flat plate leading edge bubble is that in this case the velocities are larger, both in the outer and inner shear layers. It should also be noted that neither the SST or $k - \omega$ model predict a secondary bubble although the flow visualisation experiments (section 1.3) indicated
Figure 6.31: Flow streamlines and normalised velocity contours ($U/U_\infty$) for the 25% camber arc showing the evolution of the wake through one period of vortex shedding. The simulation was computed using the SST turbulence model and the angle of attack is 20°.
that a secondary bubble does exist. The reason for this is the same as in the flat plate test case; the turbulence models cannot capture relaminarisation of the recirculating shear layer.

Figure 6.35 presents the time-averaged velocity profiles downstream of the reattachment point. Again the profiles look quite similar to the data from the flat plate test case. The boundary layer rapidly recovers a steep near-wall velocity gradient, whilst the middle region of the boundary layer (which should have the log-law profile) remains very linear, a legacy of the shear layer behavior at reattachment. It is expected that the CFD results would experience a slower recovery than experimental results, as was the case in the flat plate test case. At the last station the inner region of the boundary layer is resisting a strong adverse pressure gradient and consequently the near-wall velocities are reduced. Throughout this region the SST model predicts slightly larger velocities. At this angle of attack \((\alpha = 20^\circ)\) the SST model is yet to reach maximum lift whereas the \(k - \omega\) model is just beginning to stall.

The velocity profiles in the trailing edge separation region are plotted in Figure 6.36. Throughout this region the SST and \(k - \omega\) models predict very similar velocity profiles.
Figure 6.34: Chordwise velocity profiles within the leading edge bubble ($\alpha = 20^\circ$).
Figure 6.35: Chordwise velocity profiles in the attached flow region ($\alpha = 20^\circ$).
Figure 6.36: Chordwise velocity profiles within the trailing edge separation region ($\alpha = 20^\circ$).
Turbulent kinetic energy profiles

Figure 6.37 presents the turbulent kinetic energy profiles within the leading edge bubble. Once again the profiles are very similar to those presented in chapter 4. The SST model predicts greater production of turbulent kinetic energy in the outer shear layer compared with the $k - \omega$ model. Consequently the SST model sustains slightly larger velocities in both the inner and outer shear layers. As the shear layer approaches the reattachment point the turbulent kinetic energy levels drop in the same fashion as was found with the flat plate. This effect is more pronounced for the SST model. However, unlike in the flat plate case the SST model still predicts greater turbulent kinetic energy levels right up to the reattachment point. It would appear that in this case the SST limiter is less active compared with in the flat plate case. This is due to the large curvature of the sail section which creates a large suction pressure near the leading edge. This pressure pulls the shear layer towards the sail so that it reattaches before the shear layer fully develops. In the case of the flat plate shear layer growth had a more significant role in the reattachment of the shear layer whereas for downwind sails reattachment is driven primarily by suction forces. In the flat plate at $\alpha = 1^\circ$ the reattachment length was 0.15c for the SST model and 0.18c $k - \omega$ model whereas for the 25% camber arc at $\alpha = 20^\circ$ the reattachment lengths are 0.10c and 0.12c for the SST and $k - \omega$ models respectively. However, in the case of the 25% camber arc the angle of attack is approximately 11\(^\circ\) above the ideal angle of attack. In the flat plate at $\alpha = 1^\circ$ the minimum pressure coefficient over the leading edge bubble is -0.885 for the SST model and -0.775 for the $k - \omega$ model, and for the 25% camber arc at $\alpha = 20^\circ$ the minimum pressure is -3.45 for both the SST and $k - \omega$ models.

The turbulent kinetic energy profiles downstream of the reattachment point are presented in Figure 6.38. Here the boundary layer is dominated by the advection of turbulent kinetic energy from the upstream leading edge bubble. As we move downstream the shear layer thickens and the turbulent kinetic energy begins to dissipate. This is more pronounced in the SST model than the $k - \omega$ model, at the reattachment point the SST model is predicting greater turbulent kinetic energy levels, however by $x/c = X_R + 0.125(X_S - X_R)$ the $k - \omega$ model has surpassed the SST model’s turbulent kinetic energy levels. In the flat plate test case the $k - \omega$ model was already predicting greater turbulent kinetic energy than the SST at reattachment. In this case the reduction of turbulent shear in the SST model by the action of the SST limiter occurs further aft in respect to the reattachment point. In the 25% camber arc there is a slight positive pressure gradient acting between $x \approx 0.15c$ and $x = 0.3c$ which has a stabilising effect on the boundary layer and accelerates its recovery towards turbulent equilibrium.
Figure 6.37: Turbulent kinetic energy profiles within the leading edge bubble ($\alpha = 20^\circ$).
Figure 6.38: Turbulent kinetic energy profiles within the attached flow region ($\alpha = 20^\circ$).
6.5 Conclusions

Investigation of the flow past downwind sail sections is a challenging assignment for both CFD and wind tunnel methods. Wind tunnel models should be of thin construction in order to appropriately model the shape of real sails, this leads to models that are vulnerable to flutter and have insufficient volume to house pressure tappings. Pressure sensitive paint was used as an alternative to pressure tappings, however results proved to be inadequate due to excessive noise and in particular bias errors associated with inconsistencies in temperature and surface incidence.

The unsteady CFD results paint a good overall picture of the flow and capture the leading edge bubble, reattachment zone and unsteady wake well. Comparison with the PSP results and surface flow visualisation results indicate that the reattachment and separation points are located with good accuracy in the CFD simulations, although it was impossible to pinpoint these locations in the experimental data.

Downwind sail flows exhibit a periodically unsteady wake with force coefficients oscillating in a sinuous fashion. To capture this transient behavior simulations must be computed using appropriately small time-steps and proper sensitivity studies should be carried out. The RANS equations - which average out the turbulent behavior of the flow - are suitable for these unsteady flows and the unsteady vortex shedding should not be confused as turbulence.

The ideal angle of attack for our 25% camber arc was found to occur at an angle of attack of approximately 9°. Below this angle a separation bubble exists on the pressure side of the sail and for membrane sails the negative pressure difference near the leading edge will cause the luff to deflate. Above the ideal angle of attack a leading edge bubble is formed on the suction side due to knife-edge flow separation at the leading edge. This separation bubble is similar to the leading edge bubble that was found for the inclined flat plate which was studied in chapter 4 and it is likely that the turbulence models are performing in a similar way. In the case of the flat plate study the $k - \omega$ model overpredicted the bubble length $X_R$ by 16-24%, whereas the SST model was within 7%.

The stall behavior of two-dimensional downwind sail sections is governed by a complex relationship between the leading and trailing edge separation regions. The size of the wake and the curvature of the shear layer dictates the circulation and lift of the section. For the $k - \omega$ model - which predicts less trailing edge separation at angles of attack below maximum lift - there is more lift and consequently more upwash to the leading edge. This increased upwash contributes to the larger leading edge reattachment lengths, $X_R$, predicted by the $k - \omega$ model. These larger leading edge bubbles inhibit the development of the downstream boundary layer and lead to the $k - \omega$ model predicting maximum lift at $\alpha = 19.22^\circ$ compared with $\alpha = 21.19^\circ$ for the SST model. At maximum lift $k - \omega$ predicted a lift coefficient of 2.52 and the SST model predicted a lift coefficient of 2.35.
Chapter 7

Two-Dimensional CFD Analysis of Downwind Sail Designs

7.1 Introduction

In this chapter CFD is used to examine the performance of different two-dimensional downwind sail designs with an overall goal to estimate the maximum lift coefficient that can be generated from a two-dimensional sail. This study serves as an initial examination of the downwind design space, with a goal to obtain an indication of the maximum lift coefficient that can be generated from a single thin-membrane airfoil. By varying draft and camber a range of sail shapes are generated that cover the typical range of an ACC downwind sail inventory. In order to reduce simulation times only a solitary downwind sail is used, i.e., the influence of the mainsail is ignored. One sail shape is also analysed with a mainsail included in an attempt to establish the impact of the mainsail’s presence and to determine whether or not downwind sails can be designed - in two-dimensions at least - whilst ignoring the influence the mainsail.

7.1.1 The sail sections

As described in chapter 2 sail cross-sections are defined by draft, camber, front percentage, back percentage, leading edge angle and trailing edge angle. Using these parameters the sail shapes are generated using two fourth-order Bezier curves. The two curves meet at the maximum draft position $(x_d, y_{max})$, and at this point the sail’s curvature and the first derivative of curvature are forced to be continuous.

Six different camber values were tested: 21%, 23%, 25%, 27%, 29% and 31%. For each camber the draft was set at 40%, 45%, 50% and 55% which in total gave 24 different section shapes. The design with 23% camber and 45% draft was based upon a horizontal slice from the flying shape of one of Team New Zealand’s mid-range gennakers. The range of cambers investigated is illustrated in Figure 7.1 where six sail shapes with 21%, 23%, 25%, 27%, 29% and 31% camber are plotted, each with 45% draft. In Figure 7.2 four sail shapes are plotted with 40%, 45%, 50% and 55% draft, each with 23% camber. The sails shapes are identified using the naming convention XXYY, where XX is the camber and YY is the draft, i.e., section 2345 has 23% camber and 45% draft.
7.1.2 Downwind performance analysis and design goals

As mentioned in chapter 2 in average downwind sailing conditions the apparent wind angle, $\beta$, is usually around 90°. Consequently the lift force is the primary contributor to the driving force of the yacht and the drag force has more influence on the yacht’s heeling moment and heeling force. Unlike when sailing upwind, the side force on a yacht travelling downwind has little influence on performance since the loads on the keel are far less and the yacht heels only slightly. Therefore downwind sails are generally designed and trimmed in order to maximise lift whilst paying little consideration to drag. Even when sailing at deep angles where the drag contributes to driving force (i.e. above 90°), the increase in drag that is experienced at angles above the maximum lift angle (i.e. post stall) does not successfully compensate for the loss of lift due to stall. Consequently downwind sails are nearly always trimmed to maximise lift.

As a yacht sails downwind the wind strength and direction is usually changing and consequently both the yacht’s heading and the sail trim must be continually adjusted. Through all these changes the sails are still trimmed to achieve maximum lift and hence provided the section shapes of the sails remain

---

Figure 7.1: Comparison of sails of varying camber (21%, 23%, 25%, 27%, 29%, 31%) with draft fixed at 45%.

Figure 7.2: Comparison of sails of varying draft (40%, 45%, 50%, 55%) with camber fixed at 23%.

For all sails tested in this study the leading and trailing edge angles were fixed at 60 and 50 degrees respectively, and the front and back percentages were set at 82.45% and 79.87% respectively, which were the values from the base design. Leading and trailing edge angle as well as front and back percentage are parameters which certainly influence the performance of downwind sails and ideally these parameters should be included in any parametric studies. However in order to reduce the number of simulations required in this study a decision was made to limit the design variables to the three parameters that have the most influence on the design, namely camber, draft and angle of attack.
constant (or a sail change does not take place) the angle of attack of the wind to the chord line of the sail remains relatively constant. That is, as the apparent wind changes, the position of the sails relative to the apparent wind direction typically remains quite steady. Consequently, it is possible to get a good idea of the performance of a particular sail shape through a range of conditions by determining its lift and drag coefficient at maximum lift and resolving the drive and side forces across a range of apparent wind angles. This is suitable since changes in apparent wind direction effectively result (after the sails have been trimmed) in a solid-body rotation of the sails about the mast. Therefore, the shape of the sails remains remarkably constant relative to the apparent wind direction and one can view the boat as rotating (relative to the apparent wind) underneath the rig. Similarly we can assume that the lift and drag forces on the sail remain constant (relative to the apparent wind) and it is the driving/heeling force axes that rotate through a change in apparent wind angle.

Consider Figure 7.3, which illustrates the forces on a downwind sail resolved into both lift and drag coefficients ($C_L$ and $C_D$) and driving and heeling force coefficients ($C_{DF}$ and $C_{HF}$).

![Figure 7.3: Forces on a downwind sail.](image)

In Figure 7.3 $\delta_A$ is the aerodynamic drag angle ($\delta_A = \tan^{-1}(C_D/C_L)$), $\beta$ is the apparent wind angle and $C_T$ is the total force coefficient ($C_T = \sqrt{C_L^2 + C_D^2}$). The driving and heeling force coefficients can be derived from the lift and drag coefficients, i.e.,

\[ C_{DF} = C_L \sin \beta - C_D \cos \beta \]  
\[ C_{HF} = C_L \cos \beta + C_D \sin \beta. \]  

If we assume that the sails are trimmed to maximise lift then we can evaluate the driving and heeling
force coefficients across a range of apparent wind angles using equations (7.1) and (7.2) with $C_L$ set to $C_{L_{\text{max}}}$ and $C_D$ set according to the drag at maximum lift. Alternatively we can trim the configuration to optimise the driving force coefficient for a given apparent wind angle if we have expressions for the lift and drag coefficients as a function of angle of attack. Both these approaches have been compared and the results are presented in section (7.2.7).

For the current study the force coefficients were measured at four different angles of attack around the angle of maximum lift. The CFD results computed in this study suggest that the sail sections experience maximum lift at an angle of attack of 18.5° ±3° and accordingly the forces were evaluated at α =15°, 17.5°, 20° and 22.5°. The forces were then fitted with a cubic polynomial and the lift and drag coefficients at maximum lift were determined from this fit. In order to evaluate the accuracy of this method simulations were performed at several other angles of attack for the base design (camber = 23%, draft = 45%). The results of this investigation are presented in section 7.2.3.

7.1.3 The CFD model

The domain used for all simulations in this chapter is the small domain (see Figure 7.4) that was used in chapter 6. Recall that in the validation study a larger domain was eventually used to reduce the influence of the upstream boundary. However, for the design study, where many more simulations were required, the smaller domain was preferred in order to reduce simulation times. For all simulations the chord length was set at 1m and the Reynolds number based on chord length was set at approximately $3.31 \times 10^6$. For a typical downwind sail (chord length $\approx 14m$ at mid girth) this Reynolds number corresponds to apparent wind speed of approximately $3.5 m.s^{-1}$ or 7 knots which is a typical apparent wind speed for an ACC yacht. At the inlet Cartesian velocity components are specified according to the angle of incidence and the Reynolds number. At the inlet the freestream turbulence intensity was set at 1% with a length scale of 0.001m, however as mentioned in chapter 3 the simulations are independent of these values.

The computational grids used in this study are all block-structured and were generated in ICEM-HEXA (ICEM-CFD-Engineering, n.d.). A grid convergence study for the 2345 section is presented in section 7.2.2 using three grids that are referred to as coarse, medium and fine with 13200, 55380 and 225940 cells respectively. The $y^+$ values of the grids are approximately 2, 1 and 0.5 from the coarse to fine grids. The medium grid is illustrated in Figure 7.5. In order to achieve a $y^+$ of approximately 1.0
the near wall spacing was set at $6.25 \times 10^{-5} \text{m}$. Particular care was taken to generate orthogonal cells, with low aspect ratios near the leading edge and at the very leading edge the cells have an aspect ratio of 1:1. The leading edge region is illustrated in the close-up view in Figure 7.5. Further along the sail very high aspect ratio cells are used in order to resolve the large flow gradients normal to the wall.

For all simulations the SST turbulence model was used since this model was the best performer in the validation study. Spatial interpolation of the convective terms was carried out using a bounded high-resolution advection scheme, details of which can be found in chapter 3. Time integration was carried out using second-order backward Euler time-stepping and a series of 4 inner-iterations within each time-step to update the non-linear coefficients.
CHAPTER 7. TWO-DIMENSIONAL CFD ANALYSIS OF DOWNWIND SAIL DESIGNS

7.2 Results

7.2.1 Time step convergence study

Initial simulations indicated that time-steps as low as 0.00125s were required in order to adequately capture the transient behavior. Time step convergence was investigated by starting with a time step of 0.00125s and then repeating the simulations with the time step halved and then halved again, i.e. the time step sizes used are 0.00125s, 0.000625s and 0.0003125s. Simulations were performed at each angle of attack ($\alpha = 15^\circ$, $17.5^\circ$, $20^\circ$ and $22.5^\circ$), however only the results for $\alpha = 15^\circ$ are presented since this is the case with the fewest time steps per shedding cycle. For larger angles of attack the time step convergence was better than the $\alpha = 15^\circ$ case. In Figure 7.6 the mean lift and drag coefficients are presented for the sequence of time step sizes.

As the timestep size is reduced both the lift and drag coefficients level off indicating satisfactory time step convergence. The lift for the solution using the medium time step is within 0.075% of the lift computed using the short time step and the drag is within 0.19%. The medium time step size corresponds to approximately 33 time steps per shedding cycle for the $\alpha = 15^\circ$ case and 54 time steps per shedding cycle for the $\alpha = 22.5^\circ$ case.

7.2.2 Grid convergence study

The grid convergence study was performed at $\alpha = 20^\circ$ which is approximately the angle where maximum lift occurs. Mean lift and drag coefficients are presented in Figure 7.7 for the sequence of grids. The force coefficients are plotted against $1/N$ which is a non-dimensional measure of the grid spacing where $N^2$ is the total number of cells in the grid.

As the grid is refined there is a clear convergence of the lift and drag coefficients. The lift from the medium grid solution is within 0.9% of the fine grid solution and the drag is within 3.33%. The accuracy of the medium grid solution is not entirely pleasing with the drag in particular showing evidence of grid dependence. However, the drag has little influence on the performance of a sail section and since many simulations were required in this study, using the fine grid would have been too time consuming. Consequently, it was decided that the medium grid was the most suitable grid, and that it would at least provide sufficient accuracy for qualitative analysis of the design space.
Chapter 7. Two-Dimensional CFD Analysis of Downwind Sail Designs

Figure 7.7: Grid convergence of the lift and drag coefficients ($\alpha = 20^\circ$).

Figure 7.8: Lift versus angle of attack for the 2345 section.

7.2.3 Performance of the base sail section

The time-averaged lift coefficient versus angle of attack for the 2345 section is plotted in Figure 7.8. Also plotted in red is a polynomial fit through the data points at 15°, 17.5°, 20° and 22.5° (highlighted in blue). The fitted curve estimates the maximum lift coefficient to be 2.26 and to occur at 19.8°. As can be seen in Figure 7.8b the fitted curve agrees well with the data points at 18.75° and 21.25° and therefore it should also provide a suitable prediction for $C_{L_{\text{max}}}$.

7.2.4 Mainsail influence

Mainsail/headsail interaction is a well covered and debated subject in upwind sail design (see Marchaj (1979) and Whidden and Levitt (1990)), however little work has gone into such interactions for downwind sailing. Wind tunnel experiments have been carried out at the University of Auckland’s Yacht Research Unit investigating gennaker/mainsail interaction by measuring the sail forces independently (Johnson and Stanton, 1999; Cazala, 2002). Results showed that, when tested both in isolation and together, the total lift of the gennaker/mainsail configuration was less than the sum of its parts. Results showed that the forces on the mainsail were adversely influenced by the presence of the gennaker. However, nothing
conclusive could be said about the influence of the mainsail on the gennaker, it was difficult to distinguish between the lift and drag forces measured on the gennaker with and without the mainsail present. Richards, Johnson and Stanton (2001) suggest that since the forces on the gennaker are not significantly affected by the presence of the mainsail "it is possible to consider their optimisation independently". However, this assumption does not consider that the mainsail may alter the appearance of the flow over the gennaker without significantly changing the force coefficients. Consequently, design rankings may well not be preserved between runs performed with and without the mainsail present.

In the present study the flow past the 2345 gennaker is modeled in conjunction with a mainsail with shape and position determined using overhead photography from the wind tunnel for an apparent wind direction of 90°. The geometry corresponds to a two-dimensional slice taken approximately midway up the mainsail. Initial simulations for this configuration showed the mainsail to be well overtrimmed and fully stalled. It is believed that in the real three-dimensional situation the mainsail can be sheeted to a higher angle of attack (compared with our two-dimensional model) due to downwash from three-dimensional effects. As mentioned in chapter 2, three-dimensional effects must reduce the effective angle of attack to the leading edge of the gennaker by 10-15 degrees since the ideal angle of attack for real three-dimensional sail flows occurs 10 -15 degrees higher than that found in two-dimensional flows. As illustrated in Figure 7.10 downwash increases downstream as the tip vortices develop and hence the mainsail can be rotated clockwise (i.e. sheeted in) further in the real three-dimensional situation compared with our two-dimensional simulations. Accordingly for the current simulations the mainsail geometry was rotated anti-clockwise by 10° about the leading edge in an attempt to provide a more realistic flow situation. The grid for this configuration is illustrated in Figure 7.11. The grid has 130685 cells and is based upon the medium single sail grid (i.e., the same grid spacings are used streamwise along each sail with the same grid densities and expansion ratios normal to each sail).
Figure 7.10: Typical induced downwash distribution (in the streamwise direction) due to three-dimensional tip effects (Marchaj, 1979).

Figure 7.11: The computational grid for the gennaker/mainsail configuration (Close up of the region around the sails).

Figure 7.12 presents a comparison between the flow field for the gennaker/mainsail configuration (Figure 7.12a) and the flow field for the solitary gennaker (Figure 7.12b), with both configurations at an angle of attack of 20°. The main difference that can be seen for gennaker/mainsail configuration is that streamlines have more upwards curvature upstream of the gennaker (Figure 7.12a), an interaction that has been well documented (see Marchaj (1979) and Whidden and Levitt (1990)). The effect is partially due to the circulation from the mainsail providing upwash, however the effect of the slot (the gap between the gennaker and the mainsail) between the two sails has a more pronounced contribution. Within the slot the individual circulation fields around the gennaker and mainsail oppose each other and the flow is not significantly accelerated by the venturi-like contraction that the slot creates (although the balance of circulation fields of the two sails may well accelerate the flow within the slot). Instead much of the flow upstream of the slot is diverted around the sails, i.e. to leeward of the gennaker and to windward of the mainsail. Consequently the luff of the gennaker sees a higher angle of attack than it would if the
Figure 7.12: Comparison of the flow streamlines and velocity contours for the gennaker / mainsail configuration (a) and the gennaker without the mainsail present (b) ($\alpha = 20^\circ$). The simulations are unsteady and the plots presented are at the same phase angle (180 degrees).
mainsail was not present and the mainsail sees a lower angle of attack than it would if the gennaker was not present.

The increased angle of attack upstream of the gennaker leads to a larger leading edge bubble and higher velocities (and hence also more suction) around the leading edge in Figure 7.12a compared with Figure 7.12b. It is also noticeable that with the mainsail present the gennaker has a larger trailing edge separation region. These effects can also be seen in the pressure coefficient plot in Figure 7.13. The gennaker/mainsail configuration has a larger suction peak at the leading edge and the pressure coefficient flattens out (indicating trailing edge separation) earlier than in the simulation without the mainsail.

Whilst the flow upstream of the gennaker is clearly rotated upwards due to the influence of the mainsail, the pressure coefficient plots do not compare well with those for the gennaker in isolation at higher angles of attack. For the single sail simulations if the angle of attack is increased beyond 20 degrees the sail begins to stall. The amount of trailing edge separation increases rapidly until the sail fully separates at approximately 25 degrees. In fact, it is impossible to obtain a suction peak of $C_p = -4$ for the single sail simulation at any angle of attack. For the gennaker/mainsail configuration the mainsail enables the gennaker to sustain a higher suction peak without stalling. This is because the influence of the mainsail is not uniform along the length of the gennaker (see Figure 7.14). Towards the trailing edge of the gennaker the circulation field of the mainsail helps to accelerate the flow in the streamwise direction which aids the attached flow on the leeward side of the gennaker and effectively delays separation relative to the size of the suction peak. Consequently, higher lift coefficients are obtainable for the gennaker with the mainsail present than without it. For the current study the $C_{L_{\text{max}}}$ of the gennaker was 11.9% higher for the gennaker/mainsail configuration than for the gennaker by itself. For the gennaker/mainsail configuration, $C_{L_{\text{max}}}$ of the gennaker occurs at an angle of attack of 17.12° compared with 19.8° for the gennaker by itself. The lift versus angle of attack curve for the gennaker/mainsail configuration is plotted in Figure 7.15 along with the lift for the gennaker by itself.
Near the leech of the gennaker the circulation field from the main accelerates the flow in the streamwise direction and helps delay trailing edge separation.

Near the luff of the gennaker, the circulation field from the main provides upwash.

Figure 7.14: A schematic of the circulation field of the mainsail and its influence on the gennaker.

Figure 7.15: Lift versus angle of attack for the gennaker/mainsail configuration and the gennaker by itself.

The shedding frequency of the gennaker (calculated using a Fourier transform of the force history) for the gennaker/mainsail simulation at $\alpha = 20^\circ$ is $29.1 \, Hz$ and the Strouhal number is 0.58. The simulation for the gennaker by itself had a shedding frequency that was 18% higher than the gennaker/mainsail configuration, which is due the larger wake that exists when the mainsail is present. The shedding frequency for the gennaker/mainsail configuration at $17.5^\circ$ is similar to that of the gennaker by itself at $20^\circ$. This is not surprising since at these angles both configurations are close to maximum lift where the trailing edge separation regions are expected to be of similar size.

This study illustrates that the mainsail does have significant influence on the flow over the gennaker and that ideally any optimisation of downwind sail shapes should be carried out with the mainsail present. It is likely that the performance ranking of the designs would not be preserved between studies conducted with and without the mainsail present. Nevertheless, for the remainder of the study the
downwind sail shapes were tested in isolation since it is felt that initial validation of any design method should be performed in the simplest possible configuration to allow a more rapid exploration of the design space. The study provides a qualitative indication of the influence of draft and camber on downwind sail performance. Also, one of the main purposes of the study was to answer the question of how much lift can potentially be generated by a single thin-membrane airfoil.

The difficulty experienced in obtaining the correct mainsail trim illustrates the significance of three-dimensional effects on downwind sail aerodynamics. The three-dimensional design problem is far removed from the simple two-dimensional case that is modelled in this study. Consequently, the results of this study cannot be used for quantitative analysis of real sail designs. This study is intended to provide an indication of what two-dimensional sections provide the largest lift coefficients. Predicting fast three-dimensional sail shapes using CFD would require a more extensive study including three-dimensional simulations.

7.2.5 Effects of sail camber

For most airfoils, increasing the camber results in an increase in lift and an associated increase in drag due to steeper pressure recovery (and possibly separation) on the suction side of the foil. Thin airfoil theory predicts that lift will increase linearly with camber at a slope of $2\pi$. However, thin airfoil theory is only valid for small camber values and small angles of attack, and downwind sails operate well outside the range of validity of this theory. In this section we look at the relationship between camber and maximum lift at a camber range much higher than that of conventional airfoils. A plot of maximum lift versus camber for our downwind sail shapes is illustrated in Figure 7.16. Recall that for each camber setting four different sails were analysed, each with a different draft percentage. The lift coefficient presented in Figure 7.16 is an average across the range of draft positions. The chart illustrates that even at these high camber values $C_{L_{\text{max}}}$ continues to increase with camber and reaches a maximum at 30.44% camber, above which $C_{L_{\text{max}}}$ begins to decrease.

![Diagram](image)

Figure 7.16: $C_{L_{\text{max}}}$ versus camber (where $C_{L_{\text{max}}}$ is averaged across the different drafts).

Figure 7.16 suggests that the optimal camber for a downwind sail should be 30.44% if maximising $C_{L_{\text{max}}}$ is the design goal, however 25-26% camber is more typical for up-range America’s Cup sails. The reason for this is largely due to the way ACC sails are measured which penalises high camber sails. The
sail area is defined as

\[ SSA = (SLU + SLE) \times \left( \frac{SF}{12} + \frac{SMG}{3} \right), \]  

(7.3)

where SSA \([m^2]\) is the measured downwind sail area, SLU is the length of the luff, SLE is the length of the leech, SF is foot length and SMG is the sail’s mid girth, defined as the arc length from the mid point of the luff to the mid point of the leech. Both the foot length and the mid girth measurement incorporate camber since it is the actual arc length that is measured at these sections rather than the chord length. Therefore, in order to make the current results relevant to America’s Cup design the arc length of the sail sections must be used as the length scale in the calculation of the lift coefficient. A plot of the scaled lift versus camber is given in Figure 7.17, where \(C_{LS}^{max}\), the maximum lift coefficient scaled by arc length, \(s\), is now the variable on the \(y\)-axis.

![Figure 7.17: \(C_{LS}^{max}\) versus camber (where \(C_{LS}^{max}\) is scaled by the arc length and averaged across the different draft values).](image)

From Figure 7.17 the optimal camber is now 27.03% which is in much better agreement with America’s Cup designs. Typically, up-range downwind sails have 25-26% camber in the mid girth and sails that are designed for light airs may have considerably less camber. Often downwind sails have more camber near the head of the sail since this region is unmeasured and hence it is possible to gain sail area for free. So a sail designer might favour a slightly smaller mid girth camber in order to lower the measured sail area.

Another reason for low camber sails being favoured for America’s Cup downwind sails is the effect of induced drag. The most apparent effect of three-dimensional effects is a rotation of the lift-drag axis due to downwash. Consequently drag is increased and lift is decreased relative to the two-dimensional coefficients. Therefore if the heeling force is positive (i.e. the boat is heeling to leeward) then any increase in induced drag is directly diminishing the driving force. This is the typical situation in light to medium wind strengths, or when the yacht is pointing high (i.e. gybing through large angles). In the design of sails for such situations designers look to reduce both induced drag and profile drag by building flat sails with large span. In heavy airs yachts often heel to windward and induced drag rotates the total force closer to the direction the yacht is travelling. Therefore one could make the mistake of assuming that three-dimensional effects are improving the performance of the yacht. It is true that induced drag is less important in heavy airs, however three-dimensional effects do still reduce the performance of the sails. The total force coefficient for a three-dimensional sail flow is considerably lower than that achievable in purely two-dimensional flow due to pressure leakage at the tips. Therefore designers are always looking
to reduce three-dimensional effects and consequently it is not surprising that America's Cup sails seldom have camber as high as the optimal predicted by this two-dimensional study.

The relationship between the drag at maximum lift and camber is illustrated in Figure 7.18. This plot does not represent a direct relationship between drag and camber since we are looking at the drag at maximum lift. At the low end of the camber range as the camber is increased the maximum lift peak occurs at lower and lower angles of attack and hence the drag at maximum lift also decreases. At higher cambers the drag at maximum lift begins to increase as the sail section shapes become increasingly inefficient.

![Figure 7.18: $C_{DS}$ at $C_{L_{\text{max}}}$ versus camber (where $C_{DS}$ is scaled by the arc length and averaged across the different draft values).](image)

7.2.6 Effects of sail draft

Until recently the most common downwind America's Cup sails were spinnakers; sails that are symmetrical about the mitre so that through a gybe the sail is not required to invert. Asymmetrical spinnakers (gennakers) were first seen in the America's Cup on board J-boats in the 1930's and were known as "balloon" genoas. These were really just large, deep genoas capable of sailing deep on a broad reach. Extensive development of gennakers was carried out by Julian Bethwaite in the Australian skiff classes of the late 1970's with considerable success. These sails were well suited to the skiff classes that are capable of reaching at high angles (low apparent wind angle) when sailing downwind. Asymmetrical spinnakers reemerged in the America's Cup in the 1970's on board Intrepid during the America's Cup defense trials. These sails were the first to be flown from a spinnaker pole and during a gybe the sail had to be inverted around the front of the forestay making gybing a difficult process.

Despite being more difficult to gybe, asymmetrical sails (gennakers) have often proven to be more effective than symmetrical spinnakers, most noticeably at lower wind strengths. Gennakers were developed primarily to allow the luff to be longer than leech, thus providing more sail area in the forward section of the sail from where most of the lift force is generated (especially at small apparent wind angles). Asymmetrical sails also permit the draft position to be moved away from the middle of the sail (mitre) and it was found that gennakers with the draft forward of 50% were favoured, particularly in light airs. Gennakers are used more and more in the America's Cup and in the 2003 regatta many syndicates used them in winds of 15kts and above, wind strengths where spinnakers had previously been used exclusively.
In this study we look at the influence of the draft position on lift coefficient for our two-dimensional sail sections. Figure 7.19 presents the results for the scaled maximum lift coefficient, $C_{LS_{\text{max}}}$, versus draft, where $C_{LS_{\text{max}}}$ is averaged across the six camber values tested. The plot shows the maximum lift coefficient increasing with draft and indicates that the optimal draft position is 54.66%. This result was initially surprising since real sails are always either symmetric (spinnakers) or have the draft forward of 50% (gennakers).

Gennakers with the draft position pushed back perform well since they experience less trailing edge separation. Such sails have a longer region downstream of the leading edge reattachment point where the sail is flat and the boundary layer can recover and build speed (see Figure 7.20). Consequently there is a greater pressure difference across the sail in the area between 30-60% $x/c$ than can be achieved for sails with the draft position further forward.
CHAPTER 7. TWO-DIMENSIONAL CFD ANALYSIS OF DOWNWIND SAIL DESIGNS

The high lift coefficients obtained for sail sections with the draft position at 55% can also be explained through analogy to airfoils with trailing edge flaps. The region near the trailing edge of an airfoil's camber line has the greatest effect on the lift. By increasing the curvature of the camber-line in this region the circulation of the airfoil is increased which causes more upwash upstream of the airfoil, and higher velocities around the leading edge. This means that airfoils with trailing edge flaps are prone to flow separation at the leading edge when the flap is deployed. Consequently such airfoils often have drooped leading edges or leading edge slats that can be drooped as the flap angle is increased. Therefore it is likely that additional improvements in maximum lift could be made by increasing the leading edge angle and the curvature of the sail near the leading edge, thus emulating drooped nose airfoils.

The relationship between draft and performance is also dependent on camber and wind strength. As mentioned, in light airs draft (both induced and profile draft) is undesirable and consequently designers favour low camber genmakers with the draft well forward (as low as 40% draft). As the wind strength increases, reducing drag becomes less important and consequently camber and draft increase, until at some point spinnakers become favoured due their ease of use and efficiency though gybes. Therefore it is necessary to look at the relationship between maximum lift and draft over a range of different cambers as is illustrated in Figure 7.21. From Figure 7.21 it is evident that it is the higher camber sections that perform better with the draft pushed aft. For the 21% camber sails the optimum draft position predicted by the study is 47.48% whereas for the 31% camber sails the optimum draft position is 60.09% based on an extrapolation from Figure 7.21. Highly cambered sails are more susceptible to early trailing edge separation than low camber sails, consequently as the camber is increased it becomes favourable to push the draft further aft in order to prevent premature stall.

Figure 7.21: $C_{LS\text{max}}$ versus draft for each camber value.

For cambers of 25% and above Figure 7.21 shows an increase in maximum lift between 50-55% draft suggesting that draft values of 60% and beyond should be included in the study. However increasing the draft further would begin to create sail shapes that are not physically attainable with a trailing edge angle of 50°. Sails are supported only at their corners and rely on a positive pressure difference to inflate. Therefore the windward surface of the sail must be concave. In this study the trailing edge angle is fixed at 50° and consequently as the draft position moves aft a point of inflection begins to appear near the leech. If we were to look at sails with the draft further aft than 55% then it would be necessary to also
increase the trailing edge angle.

In real sail designs the draft is often further forward than the optimal predicted for the lowest camber tested in this study which is due to both three-dimensional effects and the influence of the mainsail. Results from the current study indicate that the trailing edge separation point (at $C_{L_{\text{max}}}$) follows the draft position closely with the flow typically separating immediately aft of the draft position. Consequently, by pushing the draft position forward, the separation point also moves forward which can decrease the performance of the sail. For three-dimensional sails, downwash from both three-dimensional effects and the influence of the mainsail diminishes the adverse pressure gradient and the flow is able to remain attached further aft compared with a two-dimensional section. As a result, sail designers can get away with pushing the draft position forward whilst keeping the rear separation point further aft compared with two-dimensional sail sections. Having the draft position forward is desirable when sailing at small apparent wind angles since it increases the curvature - and hence also the pressure difference - in the region close to the luff.

In this study the leading and trailing edge angles were kept constant, whereas in reality gennakers that have a draft position well forward will have greater leading edge angles than spinnakers. By leaving the leading edge angle fixed we are limiting the potential increase in curvature around the luff that can be gained by shifting the draft position forward. Perhaps if the leading edge angle had been increased on the sails with the smaller draft values then higher lift coefficients would have been gained for these sails. However, leading edge angle is a design variable in its own right and including leading edge angle in the parametric study would have increased the number of simulations considerably. Whilst any further two-dimensional design studies should investigate leading edge angle (and possibly front and back percentage), it was felt that such as study was unnecessary for this initial exploration of the design space.

Gennakers designed for use at low wind speeds (up to 10kts) are not built to the maximum sail area allowed under the rule due to difficulties in keeping the leech flat and the sail stable. At the clew the trailing edge angle must be tangential to the angle of the sheet which can create an undesirable increase in curvature (known as hook) near the leech. Consequently, for light air sails, designers will favour a sail with a shorter foot since this reduces the sheeting angle and with it the unwanted curvature near the leech. Reducing sail area is also favoured since full size gennakers will need to be sheeted in harder in order to keep the sail inflated which also closes the slot. Both these effects contribute to drag which is undesirable at small apparent wind angles. Consequently undersize gennakers are favoured since they allow the sail to be eased more resulting in lower drag and better $V_{MC}$ (velocity component in the direction of the next mark). Since sails designed for low wind speeds are already below maximum sail area there is no penalty in choosing sections with larger arc lengths. Consequently in light airs we are looking to maximise $C_{L_{\text{max}}}$ rather than $C_{L_{\text{Smax}}}$ and therefore there is no penalty associated with increasing the arc length if the draft position is moved forward. Plotting draft against $C_{L_{\text{max}}}$ rather than $C_{L_{\text{Smax}}}$ indicates that indeed the optimal draft position is further forward, however the effect is small.

This study mimics real trends in sail design with the draft position shifting aft as camber (and wind strength) is increased. However in real sail inventories a choice is generally made to switch to symmetrical spinnakers at wind speeds over a certain strength and consequently sails with the draft position aft of 50% are not used. This study raises the question of whether gennakers with the draft position shifted aft of 50% could provide a performance gain in higher wind strengths, and whether symmetrical and
draft aft gennakers could eventually replace spinnakers altogether. Several questions remain unanswered. Firstly, will the same trends be witnessed for three-dimensional sails where the stall behavior may be quite different? Also, can we even generate a flying shape with maximum draft aft of the mitre? When a sail design comes under load the draft tends to be pushed forward under strain and so designing a shape that will end up lying in the desired shape is not an elementary task. Finally, whilst driving force is the obvious measure of sail performance there are other factors such as the stability of the sail and its ease of trimming and manoeuvring that are also important.

7.2.7 Driving and heeling force polars

In this section the lift and drag forces on the sails are resolved into driving and heeling force coefficients over a range of apparent wind angles. The driving force coefficient polar for the base sail shape (section 2345) is presented in Figure 7.22. The plot presents two curves, one for the sail with the angle of attack to the chord line fixed to provide maximum lift, the other with the trim set to maximise the driving force coefficient. The trim optimisation is achieved with a simple conjugate gradient search that adjusts the angle of attack at each apparent wind angle using our polynomial fit for the force coefficients. The greatest difference in driving force coefficient between the two trimming techniques is 1.5% which occurs at the maximum apparent wind angle (120°). At this angle the sail is set at an angle of attack 23.3° for the driving force optimisation. At the lowest apparent wind angle the angle of attack is reduced down to 15.0° for only a 2.1% gain in driving force over the case with the sail trimmed to maximum lift. For this sail shape maximum lift occurs at 19.8°.

![Driving force coefficient for the base section shape (section 2345).](image)

Figure 7.22: Driving force coefficient for the base section shape (section 2345).

The heeling force coefficient polar for the base section shape is presented in Figure 7.23. Both the optimised and maximum lift trim settings are given, and as was found for the driving force, both curves are similar with the optimised trim providing only minimal reduction in heeling force. When sailing at small apparent wind angles the heeling force is positive (i.e. the total force points to leeward of the bow) and the boat heels to leeward. At approximately 100° of apparent wind angle the total force points in the same direction as driving force (i.e. all of the aerodynamic force is driving the yacht) and the boat is at zero heel. Above this angle the boat heels to weather. ACC boats have zero heel at around 110 - 120
degrees which is a slightly higher angle than this study suggests. This is due to three-dimensional effects which cause the lift-drag axis to rotate to leeward.

Whilst quite a large range of angles of attack (15.0° – 23.3°) was covered in the optimised trim, little gain was made in driving force. This is because our lift polar has a soft stall and there is little change in total force with angle of attack. Real downwind sails are almost always trimmed to maximum lift which suggests that stall is more dramatic and that over-trimming the sails (past stall) results in poor boat speed. Similarly under-trimming is undesirable due the inefficiency of having an unstable luff. In three-dimensional sail flows downwash effects cause the ideal angle of attack (i.e. the angle where there is no leading edge bubble) to occur at higher angles of attack than in two dimensions. Stall for two-dimensional sails occurs as the rear separation point moves forward and meets the leading edge bubble; this happens slowly resulting in a soft stall pattern. For three-dimensional sail flows there is less trailing edge separation at ideal angle of attack due to downwash induced by three-dimensional effects. However, as explained in chapter 2, when the angle of attack is increased past the ideal angle of attack the leading edge bubble develops on the leeward side and interacts with the tip vortices at the head and foot of the sail. This in turn has a pronounced influence on the downwash distribution over the sail and the trailing edge separation point to moves forward dramatically causing the sail to stall. As a result maximum lift occurs at the ideal angle of attack for real sail flows and the only reason for overtrimming the sails is to keep the luff region stable.

Figure 7.24 presents the driving force coefficient polar for the 3155 section which is the sail shape that produces the most lift (using chord length in the calculation of lift coefficient). For this sail shape even less gain is made by optimising the trim than was achieved for the base section shape. In fact for all sail shapes there is little to be gained by setting the angle of attack to anything but maximum lift. Consequently for all apparent wind angles the sail section with the highest maximum lift coefficient provides the most driving force. Therefore the 3155 section is the best sail shape if we determine the driving force coefficient based on chord length and the 2755 section is the best performer based on arc length.
The driving force polars presented thus far are non-dimensional and the real driving force depends on the apparent wind speed. Figure 7.25 presents the driving force polar for the 2345 section based on apparent wind speed calculated by Team New Zealand’s VPP. The VPP provides the optimum (target) apparent wind speed and direction across the range of true wind speeds. Figure 7.25 illustrates that at small apparent wind angles greater driving forces are generated (despite lower driving force coefficients) due to larger apparent wind speeds achieved by sailing high. For an ACC yacht the driving force does not follow the same trend as Figure 7.25 and instead the driving force increases between 60° and 120° of apparent wind angle. This is due to sail selection. Figure 7.25 resolves force coefficients from a single sail shape across the apparent wind range whereas in reality a range of sail shapes are used. In light airs small, flat sails are used with considerably less total force than that generated by the high camber spinnakers used for running in strong wind conditions.

Figure 7.24: Driving force coefficient for section 3155.

Figure 7.25: Driving force polar for the base sail shape (section 2345) based on estimated ACC apparent wind speeds.
7.3 Conclusions

Sail design is largely based upon minor adjustments and refinements to existing designs with the objective of creating sails that are fast and stable. Unfortunately it is often difficult to determine whether one sail is performing better than another and hence frequently sails are subjectively selected based on appearance. Wind tunnel testing has helped introduce objectivity into the process by providing a means to quantify the performance of different sails. This has revolutionised the way sails are designed and as a consequence many current designs are not the aesthetic creations that have prevailed in the past. This work illustrates how CFD can be utilised in the design cycle, not just for performance prediction but also as a means to obtain better understanding of the flow topology and the behavior of different sail designs.

Results from the current study suggest that lift coefficients of the order of 2.4 – 2.5 are obtainable for two-dimensional downwind sail sections. However the present study imposes limitations on the design space by fixing several of the design variables (front percentage, back percentage, leading edge angle and trailing edge angle) and hence lift coefficients of above 2.5 may very well be possible. Also, downwind sail sections that are aided by the presence of a mainsail can achieve even higher lift coefficients.

The two-dimensional lift coefficients predicted in this study are considerably higher than lift coefficient values (1.0-1.7) typically used in Velocity Prediction Programs (VPPs) indicating that real downwind sails experience three-dimensional effects (primarily due to strong tip vortices) that hinder their performance considerably. Consequently, two-dimensional design studies are unsuitable for refinement of three-dimensional sail designs. Studies such as this one are more suited for initial exploration of the design space, and to help the user understand how different design parameters influence the flow.

In the current, two-dimensional study the relationship between camber and maximum lift follows the same trend as found in three-dimensional sails; increasing in camber produces increases in both lift and drag. The relationship between draft and maximum lift indicates that - for two-dimensions sections at least - there is merit in using sections with the draft values greater than 50\% in an effort to delay trailing edge separation for up-range sail designs. Whether this theory will also hold for three-dimensional sails is yet to be determined and any advantages may well be lost due to three-dimensional effects, the influence of the mainsail and issues in constructing and trimming such a sail.

Simulations carried out including the mainsail indicate that the mainsail has a pronounced effect on the flow over the gennaker. With the mainsail present the lift on the gennaker was 12\% higher than without it. The circulation field of the mainsail has a non-uniform influence on the flow over the gennaker and it is not possible to merely consider the gennaker as seeing an increased angle of attack due to upwash provided by the mainsail. Contribution of the mainsail itself to the yachts driving force cannot be ignored and it is necessary to also consider the influence that the gennaker has on the mainsail. Whilst outside the scope of the current project a two-dimensional investigation of the influence of size of the slot and overlap between the gennaker and mainsail could provide useful results. The relationship between the tip vortices shed from the tips of both sails also needs to be taken into consideration for three-dimensional designs.
Chapter 8

Conclusions

Sail designers are faced with many challenges in assessing the performance of downwind sail designs. Wind tunnel and full scale testing results are compromised with uncertainties related to human error, variations of conditions and geometry and difficulties in gathering complete and precise data. Consequently design changes are generally assessed through repeated full-scale testing. CFD can provide much more complete data in precise testing conditions and thus has the potential to allow a much more thorough assessment of sail performance. Visualisation of CFD results can provide the designer with valuable insight into the flow physics. Furthermore, there is the prospect of computer simulated design optimisation, using techniques such as gradient-based cost function minimisation, evolutionary algorithms or the adjoint method.

Therefore the prospect of obtaining CFD results for downwind sails is enticing for the sail designer. However, as discussed in this thesis, the turbulent characteristics of sail flows makes the computation of CFD simulations difficult. Obtaining reliable and accurate CFD solutions is no simple task and the sail designer must take care to ensure they are not mislead by poor simulations.

To fully resolve turbulence down to the smallest scales would require immense grid resolution and minute time-steps for ACC sail simulations where Reynolds numbers are typically in excess of a million. Such direct numerical simulations are not realistically solvable even on today’s fastest computers. Modeling turbulence to a level where sail simulations can be carried out on today’s computers requires considerable simplification of the physics and consequently a selection of turbulence models are likely to give a range of varying results for the same flow problem. As discussed in Chapter 3, many turbulence models are tailored to be more suitable for different flow cases and consequently it is important to understand the strengths and weaknesses of a turbulence model rather than trusting it implicitly for a particular application.

The flow of air around yacht sails involves several features that pose particular trouble for turbulence models. As the flow rounds the sharp leading edge it is forced to separate and form a leading edge bubble of the thin airfoil kind as discussed in Chapter 4. The leading edge bubble reattaches a short distance along the sail however its influence extends well downstream. The boundary layer near the reattachment point is dominated by viscous effects and turbulent structures that have been advected from the outer shear layer of leading edge bubble. Downstream of the reattachment point the shear layer gradually recovers and starts to exhibit properties typical of boundary layers in turbulent equilibrium. As discussed in Chapter 4 these non-equilibrium effects are considerable and consequently simplified
turbulent boundary layer treatments such as wall-functions are unsuitable and are likely to produce inappropriate results.

Downwind sails always exhibit trailing edge flow separation due to their high camber which produces a strong adverse pressure gradient on the suction surface. Typically these separation bubbles extend over almost half of the sail and for this reason downwind sails are often inappropriately regarded as bluff-body drag devices rather than high lift airfoils. The trailing edge separation bubble is unsteady with counter rotating vortices being shed in a similar fashion to the Von Karman vortex street. Accurate prediction of trailing edge separation is a difficult and much researched aspect of high-lift turbulence models. Many turbulence models (e.g. $k - \omega$, SST, Spalart Almaras) have been developed with high-lift applications in mind. However in the case of downwind sails the problem of predicting trailing edge separation is compounded by the fact that the boundary layer upstream of the separation point is recovering from the leading edge bubble as well as the fact that the wake is unsteady.

This research seeks to investigate the suitability of various turbulence models for application to downwind sail flows. A detailed investigation into the suitability of the standard $k - \omega$ and SST $k - \omega$ turbulence models for flows involving leading edge bubbles of the thin airfoil type was carried out and the results are presented in Chapter 4. The overall topology of the flow and the surface pressure coefficient plots were predicted well with both models. The SST model in particular performed well, predicting the length of the separation bubble within 7% of the experimental results. Both turbulence models failed to predict the secondary separation bubble that was evident in the experimental results. This is due to the inability of these turbulence models to predict relaminarisation of the inner shear layer which leads to the development of the secondary recirculation bubble. The omission of this flow feature does not seem to have a significant effect on the overall quality of the results for this flow case.

It should also be noted that despite having no predictive capability for transition, the turbulence models capture the outer shear layer of the leading edge bubble well. This is because the shear layer is inviscidly unstable and subject to a large amount of shear as the flow separates from the leading edge. For the flat plate transition occurs within the first 2.5%c and it is expected that for sails the transition location would be similar. In fact, for ACC sails, transition is likely to be even more abrupt since the Reynolds number is an order of magnitude higher. Therefore models such as the standard $k - \omega$ and SST models, that are only capable of predicting fully turbulent flows are quite suitable for flows with this type of transition.

Downstream of the reattachment point for the leading edge bubble both turbulence models predict slower recovery of the turbulent boundary layer than the experiments indicate. The turbulence models cannot fully capture the effect of the outer turbulent structures that are advected from the leading edge bubble. These structures provide energy to the boundary layer which accelerates the development of the inner-most region of the boundary layer. This slow recovery of the boundary layer is a concern for sail flows simulations since it could lead to premature trailing edge separation.

To complete the validation of CFD for downwind sail design comparisons with experimental data for sail sections were necessary. Two-dimensional sail flows incorporate the essence of the physics of the three dimensional case whilst being significantly easier and cheaper to study, both in the wind tunnel and through computer simulation. To the knowledge of the author there are no suitable results available from wind tunnel tests of sections that are characteristic of downwind sails. Preliminary results were
CHAPTER 8. CONCLUSIONS

compared with wind tunnel tests conducted by the author at the University of Auckland using a low aspect ratio model of a 24.7% camber circular arc which is representative of a typical downwind sail section. Unfortunately, due to the low aspect ratio of the model, three-dimensional effects associated with the junction between the model and the wind tunnel walls made the results unsuitable for comparison with two-dimensional CFD. Three-dimensional CFD was used to reproduce the full flow pattern and results showed better agreement with the wind tunnel results. Visualisation of the computer simulated flow field proved to be useful in understanding the flow topography and comparison was made with the wind tunnel tests.

In the pursuit of obtaining good experimental data for downwind sail sections for validation of CFD models, wind tunnel tests were carried out at NASA Ames, California in conjunction with Stanford Yacht Research (SYR). The 7 by 10 foot test section meant that a model with a much larger aspect ratio of 15 could be used, compared to the aspect ratio of 1.4 that was used in the previous tests. An attempt was made to determine the surface pressures using pressure sensitive paint however the dynamic pressure proved to be too low to provide accurate results. Comparison with the PSP results and surface flow visualisation results indicate that the reattachment and separation points are located with good accuracy in the CFD simulations, although it was impossible to pinpoint these locations in the experimental data. Particle image velocimetry (PIV) was also planned and a new model with pressure tappings was built by staff at Stanford Yacht Research. Unfortunately due to funding issues the SYR wind tunnel project was abandoned. It is hoped that eventually the wind tunnel tests will be completed for comparison with the CFD results.

Although the research does not provide a complete assessment of the suitability of CFD simulations for downwind sail performance analysis, it does present a significant step forward in the understanding of the behavior of these complex turbulent flows. It also provides qualitative indication that turbulence models are functioning well for this flow case and it establishes the relative strengths and weaknesses of the turbulence models. From the results it appears that the SST $k-\omega$ model is the most suitable turbulence model in CFX-5 for downwind sail flows. It is expected that this model will provide the most accurate pressure predictions and will capture the unsteady separated flow regions more appropriately than other $k-\omega$ and $k-\epsilon$ turbulence models.

In Chapter 7 results are presented from a parametric design study for downwind sail shapes. The design study is carried out using the SST $k-\omega$ model and illustrates how CFD can be used to provide useful insight into design optimisation. The results illustrate how CFD can be used to rank designs and establish trends. For such applications CFD is a useful design tool since although for performance prediction some error is likely, designs will still rank correctly in most cases. The results for this design study also indicate that very high lift coefficients in the order of 2.5 are obtainable with two-dimensional downwind sail sections. Due to the impact of three-dimensional flow effects, two-dimensional design studies are unsuitable for detailed design refinement and performance prediction. However such studies do provide the designer with valuable insight into the design space and the general influence of the design parameters.

Research is currently being carried out at Stanford University’s Center of Turbulence Research looking at detached and large eddy simulations (DES and LES) of the inclined flat plate test case and results will be used to further validate the results presented in this thesis. DES and LES are not currently viewed
upon as practical design tools, however by capturing more of the turbulent physics these simulations have the potential to provide much information about the functionality of the lower order models. In the near future the DES and LES simulations will be extended to include downwind sail sections and it is hoped that these simulations will provide good data to aid the validation of turbulence models for downwind sail flows.

In summary, three-dimensional CFD simulations are potentially a valuable tool for the design of high performance downwind sails. However care must be taken to ensure correct grid sizes and time-steps are used; steady state simulations on coarse grids are unsuitable. Suitable grid and time-step sensitivity studies are strongly advised. Simulations using two-equation turbulence models may be inadequate for accurate performance analysis or for input into VPP software, however they still provide the designer with considerable information about downwind sail flows. Visualisation of CFD simulations has the potential to provide the designer with powerful insight into the flow properties. Also, whilst not providing accurate quantitative data, CFD should in most cases correctly rank designs and show general trends and relationships between parameters such as camber and performance. Though post-processing much can be learnt about the flow structure and this leads to better understanding of the design space. CFD is a powerful tool, provided its limitations and weaknesses are well understood.
Glossary of Sailing Terms

This glossary is intended to provide the reader with a brief explanation of the sailing terms that have been used in this thesis. For additional sailing terms that have not been provided here, the reader is referred to http://www.sailboat-technology.com (Fassardi, 2003). For more complete explanations of these terms and additional information about sailing, the reader is referred to Marchaj (1979).

<table>
<thead>
<tr>
<th>Term</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Apparent wind</td>
<td>The wind relative to the yacht. The resultant of the true wind and the wind induced by the motion of the boat.</td>
</tr>
<tr>
<td>Boom</td>
<td>A horizontal spar connected to the mast for extending the foot of the mainsail.</td>
</tr>
<tr>
<td>Bow</td>
<td>The forward end of the yacht.</td>
</tr>
<tr>
<td>Clew</td>
<td>A term used to describe the intersection of the leech and foot of a sail.</td>
</tr>
<tr>
<td>Close hauled</td>
<td>Sailing on a course as close as possible to the wind.</td>
</tr>
<tr>
<td>Downwind</td>
<td>The direction the wind is blowing to.</td>
</tr>
<tr>
<td>Draft</td>
<td>The maximum depth of a yacht under the water. Or when referring to sails, the chordwise position of maximum depth.</td>
</tr>
<tr>
<td>Foot</td>
<td>The bottom of a sail.</td>
</tr>
<tr>
<td>Foresail</td>
<td>A sail placed forward of the mast, also known as a headsail.</td>
</tr>
<tr>
<td>Forestay</td>
<td>A stay from the top of the mast to the bow to prevent the mast from falling back, also known as a headstay.</td>
</tr>
<tr>
<td>Gennaker</td>
<td>A large asymmetric foresail used for sailing downwind or reaching.</td>
</tr>
<tr>
<td>Genoa</td>
<td>A foresail that overlaps the mainsail, generally used for sailing upwind.</td>
</tr>
<tr>
<td>Gybe</td>
<td>To change tack whilst sailing downwind.</td>
</tr>
<tr>
<td>Head</td>
<td>The top of a sail.</td>
</tr>
<tr>
<td>Headfoil</td>
<td>A luff groove device used for attaching the luff of a foresail to the forestay.</td>
</tr>
<tr>
<td>Headsail</td>
<td>See foresail.</td>
</tr>
<tr>
<td>Heel</td>
<td>The inclination of a yacht to one side.</td>
</tr>
<tr>
<td>Hull</td>
<td>The body of a yacht.</td>
</tr>
<tr>
<td>Jib</td>
<td>A foresail that does not overlap the mainsail, generally used for sailing upwind.</td>
</tr>
<tr>
<td>Keel</td>
<td>A submerged vertical hydrofoil that provides lateral resistance to the sideways force produced by the sails as well as righting moment to reduce heel angle.</td>
</tr>
<tr>
<td>Leech</td>
<td>The trailing edge of a sail.</td>
</tr>
<tr>
<td>Leeward</td>
<td>The direction the wind is going. Also used to describe the side of yacht that is sheltered from the wind direction.</td>
</tr>
<tr>
<td>Leeway</td>
<td>The sideways movement of a sailboat away from the wind.</td>
</tr>
<tr>
<td>Term</td>
<td>Definition</td>
</tr>
<tr>
<td>----------</td>
<td>------------</td>
</tr>
<tr>
<td>Luff</td>
<td>The leading edge of a sail.</td>
</tr>
<tr>
<td>Mainsail</td>
<td>The largest upwind sail on a multiple sail boat.</td>
</tr>
<tr>
<td>Mast</td>
<td>A vertical spar used to support the sails.</td>
</tr>
<tr>
<td>Masthead</td>
<td>The top of the mast.</td>
</tr>
<tr>
<td>Mid girth</td>
<td>Arc length of a sail between the mid points of the luff and leech.</td>
</tr>
<tr>
<td>Mitre</td>
<td>The chordwise midline of a sail.</td>
</tr>
<tr>
<td>Rig</td>
<td>The arrangement of a yacht's mast, sails and spars.</td>
</tr>
<tr>
<td>Rigging</td>
<td>The cables and lines that support or control a yacht's rig.</td>
</tr>
<tr>
<td>Upwind</td>
<td>The direction the wind is blowing from.</td>
</tr>
<tr>
<td>Sail</td>
<td>A cloth used to catch the wind and propel a vessel.</td>
</tr>
<tr>
<td>Schooner</td>
<td>A boat with two or more masts with the main mast aft.</td>
</tr>
<tr>
<td>Shroud</td>
<td>A sideways support for the mast running from the masthead to the side of the boat.</td>
</tr>
<tr>
<td>Spar</td>
<td>The term for the structures that support the sails, e.g. the mast and boom.</td>
</tr>
<tr>
<td>Spinnaker</td>
<td>A large symmetric foresail used for sailing downwind.</td>
</tr>
<tr>
<td>Stay</td>
<td>Wires or ropes used to support the mast.</td>
</tr>
<tr>
<td>Stern</td>
<td>The rear end of a yacht.</td>
</tr>
<tr>
<td>Trim</td>
<td>The act of adjusting the shape of a sail whilst sailing.</td>
</tr>
<tr>
<td>Tack</td>
<td>To change direction when sailing upwind and take the wind on the other side of the boat. Also a term used to describe the intersection of the luff and foot of a sail.</td>
</tr>
<tr>
<td>Windward</td>
<td>The direction the wind is coming from. Also used to describe the side of a yacht closest to the wind.</td>
</tr>
</tbody>
</table>
Bibliography


*America’s Cup Class Rule, Version 4.0* (2000). Challenger of Record and the Defender for America’s Cup XXXI.


Barth, T. J. and Jesperson, D. C. (1989), The design and application of upwind schemes on unstructured meshes, Technical Report AIAA paper 89-0366, AIAA.


Fallow, J. B. (2003), Personal communication.


Menter, F. R. (2003), Personal communication.


Prandtl, L. (1925), ‘Uber die ausgebildete turbulenz’, ZAMM 5, 136–139.


