Modelling Leading Edge Separation on a Flat Plate and Yacht Sails using LES

Stefano Nava\textsuperscript{a}, John Cater\textsuperscript{b}, Stuart Norris\textsuperscript{a}

\textsuperscript{a}Department of Mechanical Engineering, University of Auckland, 20 Symonds Street, Auckland, 1010, New Zealand
\textsuperscript{b}Department of Engineering Science, University of Auckland, 70 Symonds Street, Auckland, 1010, New Zealand

Abstract

This work presents an investigation into the aerodynamics of upwind sailing using different methods of modelling turbulence, comparing Large Eddy Simulation (LES) and Reynolds-Averaged Navier-Stokes (RANS) methods. A preliminary simulation of a flat plate at an angle of attack was performed in order to investigate the potential for LES in modelling flows around a sharp leading edge. Comparison with existing experimental data highlighted the formation of a double recirculation bubble structure at the leading edge that only LES was able to predict. However, a fine mesh and small timestep are required to correctly reproduce the pressure distribution on the flat plate. The RANS and LES models were subsequently used to model an upwind sail geometry tested in the wind tunnel. The CFD results showed good agreement with the experimental data and in particular highlighted the superior performance of the LES model in predicting the leading edge recirculation structure compared to RANS. The solution was sensitive to the mesh and timestep used. A mesh size of order $y^+ \sim 1$ and a Courant number $\leq 1$ provided results with RMS errors less than 0.1 between the simulation and the experimental values of the coefficients of pressure at the top of the mainsail.

Keywords: LES, Upwind sailing aerodynamics, Leading edge separation bubble

1. INTRODUCTION

Computational methods are a key component of the design process for a high performance sailing yacht. The fluid-dynamics that characterises sailing yachts is extremely complex, due to the fact that the yacht is partly immersed in water and partly in air, with the flow being three-dimensional and turbulent. Experimental and numerical studies have created a large body of knowledge of the physics of the problem, but at the same time have highlighted the limits to the predictive methodologies available for designers. The need to develop, improve and validate numerical models is the reason for this work.

There have been a number of studies of the aerodynamics of upwind sails using RANS (Reynolds-Averaged Navier-Stokes equations). Early studies only had experimental force data for validation. Miyata & Lee (1999) presented the results of several simulations made on an IACC-class yacht sailing upwind, testing the effectiveness of a range of turbulence models. The Baldwin-Lomax model was found to perform the best, having an error of less than 10% in the lift force prediction. Subsequently Yoo et al. (2006) used a RANS code with the standard $k-\varepsilon$ turbulence model on a structured mesh of nearly 1.7 million cells, with a $y^+$, the non-dimensional distance of the first mesh node from the wall, between 50 and 230. The poor prediction of separation led to inaccurate results with the predicted force coefficients being incorrect by 15% for the lift and 59% for the drag.

Querard & Wilson (2007), used the SST turbulence model, and reduced the mesh spacing in the normal direction to the wall to $y^+ \sim 10$ over the entire surface of the sails. This resulted in a more accurate computation of the flow, reducing error in the computed force coefficients to less than 12% for the lift and 24% for the drag. Masuyama et al. (2007), also using the SST model, further reduced the wall normal spacing to $y^+ \sim 1$ for a mesh of approximately $5 \times 10^5$ cells, achieving error less than 3% in lift and a systematic overestimation of drag of nearly 6%.
The previously discussed studies were based on validation against global force data. In contrast, Fluck (2010) measured the pressure distribution on a set of fiberglass sails reproducing a hypothetical AC33 class yacht design. Viola et al. (2013) used a steady RANS solver with the SST turbulence model, and reproduced the pressure distribution over both of the sails. However, he was unable to match the pressure distribution at the top of the mainsail where the flow was detached.

More recently, Queutey et al. (2015) applied different turbulence models to model the Viola et al. (2011) experiment. They attributed their poor prediction of pressure distribution in the separated flow regions to their omission of the sail thickness in the computational model, and to differences in the onset flow between computation and experiment. Interestingly, a result calculated using the DES turbulence model did not show any improvement in the prediction of the pressure distribution over two equation methods. Viola et al. (2013) and Queutey et al. (2015) correctly predicted the flow over most of the sails, but did not correctly model the separated flow at the top of the mainsail.

In this study Large Eddy Simulation (LES) is used in order to evaluate its ability to model this separated flow. The ability of LES to model separation on airfoils has been demonstrated by previous authors, modelling an A-airfoil at a high angle of attack. For example, Frohlich & Mellen (2001) used the Smagorinsky sub-grid scale (SGS) model and found that the model predicted the presence of the trailing edge separation only when it correctly computed the transition position after the leading edge separation bubble. Sagaut & Mary (2001) used the selective mixed scale model. The trailing edge separation area was correctly predicted and the velocity profiles matched the experiment. However, the Reynolds stresses were overestimated which the authors attributed to the short width of the domain in the spanwise direction. The authors suggested that to correctly capture the flow features, the grid size should not exceed the following values in the airfoil proximity: $\Delta y^+ = 2, \Delta x^+ = 100, \Delta z^+ = 20$, where $y$ is the normal to the wall direction, $x$ is the streamwise direction and $z$ is the spanwise direction.

LES has also been shown to be able to model leading edge separation. Sampaio et al. (2014) modelled the flat plate experiment of Crompton & Barret (2000), using RANS and LES. Comparison between RANS and LES solution showed that the RANS solver, using the SST turbulence model, failed to predict the presence of the secondary recirculation bubble, while it was predicted by the LES solution using the Dynamic Smagorinsky SGS model. This is consistent with a study by Collie & Gerritsen (2006). An analysis of the Reynolds stresses showed good agreement between LES and experiments, while the RANS calculations gave results far from the correct values. Sampaio et al. (2014)’s investigation is particularly relevant for upwind sailing modelling, as the flat plate leading edge bubble features characteristics similar to the separation bubble forming at the sail leading edge.

To date there are no published results for LES in sailing aerodynamics, and so the application of the methodology to model a well-documented experiment (Fluck (2010) and Viola et al. (2011)) allows testing of its utility to this field. A high resolution grid was used so as to capture the fine flow structures at the leading edge, and to correctly predict the pressure distribution on the sails, especially in regions of flow separation. Care was taken to replicate the experimental test conditions and to evaluate the influence that different models and boundary conditions have on the calculated results. Simulations were performed using RANS and LES on the same mesh, allowing a direct comparison between the two methods.

In the next section the numerical methodology will be discussed; the subsequent sections will focus on modelling the flat plate experiment of Crompton & Barret (2000), with a comparison between the RANS, LES and experimental results. The later section will describe the modelling of the Viola et al. (2011) upwind sail experiment. The final section will analyse the model of the leading edge separation bubble forming at the top of the mainsail.

2. NUMERICAL METHODOLOGY

The computational modelling was performed using ANSYS Fluent v.16 (2015). This software solves the Reynolds Averaged Navier Stokes equations, or the spatially filtered Navier-Stokes equations used in Large Eddy Simulation. Block structured hexahedral meshes were used for all the models, which were generated with ANSYS ICEM-CFD. This type of mesh was preferred since it allowed fine control over cell dimensions, enabling the specification of different refinement levels in different axes, and it generated a highly refined computational grid in regions of interest.
The RANS calculations were performed using the SST turbulence model with a SIMPLE solver scheme as recommended by Masuyama et al. (2007) and Viola et al. (2013). A second order upwind differencing was used for the momentum equations while the turbulence scalars were discretized using first order upwinding, since higher order schemes did not improve the results.

For stability, the LES calculations used the PISO solver with three iterations per timestep (Sampaio et al., 2014) for the simulations with Courant number higher than one, and the more efficient fractional step solver for the simulations with a Courant number lower than one. The momentum equations were discretized with the second order central difference scheme, whilst time stepping used a second order formulation. The timestep size was based on the smallest dimension of the smallest cell and different values were used in order to investigate the influence of this parameter on the results; the tested values correspond to the maximum calculated Courant number varying between 0.5 to 5. The subgrid scale turbulence was modelled using the dynamic Smagorinsky-Lilly model as adopted by Sampaio et al. (2014).

For the LES simulations, onset turbulence characteristics were determined through the generation of fluctuating inlet velocity profiles. The spectral synthesizer method described in ANSYS Fluent v.16 (2015) was adopted and reproduced the desired turbulence profiles in terms of turbulence intensity and turbulence length scale.

All the simulations were executed until the residual values converged, evaluated as a scaled $L^1$-norm of the equation residuals (ANSYS Fluent v.16 (2015)). The residual values for the momentum equations reached a threshold of $10^{-6}$ while the continuity equation residual was $10^{-4}$. It was possible for the continuity residuals to be smaller than $10^{-6}$ only for the low Courant number LES simulations. For the LES calculations, the flow variables were averaged over a period of 0.4 s for the flat plate case and 4 s for the upwind sail case, after an initial period of 0.2 s and 2 s respectively, to allow the flow field to develop. All LES simulations were initialised using results from RANS calculations. The calculations were performed on an Intel-based high performance computational cluster, with two Intel Xeon E5-2680 Sandy Bridge 2.70 GHz processors per node, with an infiniband interconnection. The wallclock time for the RANS calculations varied between 1 hour for the flat plate simulations to a maximum of 6 hours for the upwind sailing simulation, using 64 cores. For the LES simulations 30 to 200 hours were required for the flat plate simulations, while 60 to 200 hours were needed for the upwind cases, using 128 cores.

3. FLAT PLATE EXPERIMENTS

The Crompton & Barret (2000) experiment of a flat plate at a low angle of attack was initially used to determine the applicability of the LES methodology to modelling flows dominated by leading edge separation. This particular experiment has separated flow at the leading edge, which is typical of upwind sails, but it has a simple and well-defined geometry.

The flat plate used in the experiments was 0.16 m long, with a thickness to chord ratio of 3.75% and an aspect ratio of 5. The plate was chamfered at the leading edge forming a sharp angle of 20°. The flat plate was mounted inside the test section of the Bristol low turbulence wind tunnel and was tested for angles of attack between 1° and 5°. The tunnel speed was 20 m/s corresponding to a Reynolds number of $2.13 \times 10^5$, which is comparable to the upwind sail experiment of Fluck (2010) carried out at Reynolds number $2.6 \times 10^5$. The inlet flow was uniform, and the upstream turbulence intensity was less than 0.05%. The experiments present data for the pressure distribution from pressure taps on the top surface of the plate and velocity profiles obtained with the Laser Doppler Anemometry technique.

4. FLAT PLATE COMPUTATIONAL MODEL

The computational domain was a rectangular box representing the test chamber of the Bristol low turbulence wind tunnel, with the flat plate located as shown in Figure 1, at 1° angle of attack.

Figure 1: Longitudinal section of the flat plate computational domain.
The inlet boundary had a velocity of 20 m/s, with a turbulence intensity of 0.05%, while the turbulence length scale was set to 1 m. This was not measured in the experiment, but variation of the value from 0.001 m to 1 m, was found to have a negligible effect on the computed pressure distribution and velocity field. The free-stream turbulence intensity was three orders of magnitude smaller than the turbulence produced at the leading edge of the plate, and did not influence the results. The top and bottom boundaries were set as free-slip walls to reproduce the walls constraint on the flow, the outlet boundary was a constant pressure outlet, while a periodic boundary condition was assigned to the lateral bounding walls in order to avoid span-wise velocity constraints. The domain has a span of $c/4$, where $c$ is the plate chord length, in order to reduce computational effort.

A block structured mesh of approximately 2.8 M cells was used, as shown in Figure 2. The mesh size was chosen based on the results of Sampaio et al. (2014). A mesh resolution study showed a variation of less than 1% in the computed minimum $C_p$. The spacing normal to the wall was set to 25 $\mu$m corresponding to a value of $y^+$ less than 1, as prescribed by Sagaut & Mary (2001) and Sampaio et al. (2014). In order to capture the separation bubble occurring at the leading edge, the length of the cell in the streamwise direction was set to 25 $\mu$m at the leading edge, growing to 1.5 mm at the trailing edge, where the flow was attached. The growth ratio both tangential and normal to the plate was 1% near the plate and 10% away from it. Finally, a uniform discretization was used in the spanwise direction, with 17 nodes, ensuring enough spatial resolution to resolve the spanwise turbulent structures as used by Sampaio et al. (2014).

The results presented are calculated with a timestep size of 0.62 $\mu$s which yielded a maximum Courant number smaller than 1. The simulation was run for 960,000 timesteps to ensure convergence of the calculation, according to the criterion described in section 2.

5. FLAT PLATE RESULTS

The flow was modelled using both RANS and LES. Unless stated otherwise, the LES results were calculated using a maximum Courant number of 1. Streamlines and contours of velocity are shown in Figure 3 revealing a large region of recirculating flow located at the leading edge, within which the fluid rotates in the clockwise direction. The streamwise velocity contours indicate that the backflow inside the leading edge recirculation bubble has a high velocity, over 25% of the nominal onset velocity. Figure 3b, which shows the LES simulation, indicates that the boundary layer separates a second time close to the leading edge, forming a counter rotating secondary recirculation bubble due to the adverse pressure gradient. This secondary bubble is found in Crompton’s experimental data (Crompton & Barret, 2000).

Figure 3 highlights a major difference between the RANS and the LES models, with the former unable to predict the secondary recirculation bubble, which is consistent with the findings of Collie & Gerritsen (2006). The prediction of the secondary bubble is important since it moves the centre of the primary recirculation bubble further downstream.

The presence of a separated region at the leading edge of the plate is reflected in the experimental pressure distribution with the pressure coefficients ($C_p$) in the first 5% of the chord length only varying by a small amount, decreasing from approximately -0.85 to -0.9 (Figure 4a). Downstream, the boundary layer reattaches to the plate surface and the pressure increases to a pressure coefficient of -0.1 (Figure 4b).

The differences in the prediction of the leading edge separation by RANS and LES models are highlighted in Figure 4a. While there is general agreement between the calculated pressure coefficients and the experimental data, the RANS model predicts a shorter constant pressure region and a more gradual pressure recovery. The impact of the secondary recirculation bubble modelled by the LES is evident, with the primary recirculation bubble moving downstream, increasing the length of the constant pressure area and delaying the
pressure recovery compared to the RANS results. LES accurately predicts the length of the constant pressure region, the values of the pressure coefficient inside the recirculation area, and the boundary layer reattachment point.

Three simulations were performed with three different values of the timestep size in order to investigate the influence of the Courant number on the LES predictions. Figure 5 shows the pressure coefficient distributions in the leading edge region for different values of the maximum Courant number. The graph indicates a trend in the results with an increase in the Courant number resulting in the model underestimating the length of the recirculation bubble. Moreover, at high Courant numbers, the pressure inside the separated region decreases, with a minimum pressure coefficient value that

Figure 3: Contours of streamwise velocity and velocity streamlines at the leading edge of the flat plate.

Figure 4: Pressure coefficient distribution on the upper surface of a flat plate at 1° angle of attack: (○) experiments (Crompton & Barret, 2000), (∙∙∙) RANS, (—) LES.

Figure 5: Pressure coefficient distributions in the leading edge region for different values of the maximum Courant number. The graph indicates a trend in the results with an increase in the Courant number resulting in the model underestimating the length of the recirculation bubble. Moreover, at high Courant numbers, the pressure inside the separated region decreases, with a minimum pressure coefficient value that...
decreases with the increasing timestep, over-predicting
the pressure coefficient peak by 30%. For maximum
Courant numbers less than 1, the model is insensitive
to the timestep size, with solution for Courant number
1 and Courant number 0.5 being approximately equal.

Figure 5: The effect of Courant number variation on the
LES pressure distribution on the upper surface of a flat
plate at 1° angle of attack: (○) experiments (Crompton & Barret, 2000), (- -) max Courant number=3, ( - -) max
Courant number=1, (—) max Courant number=0.5.

This reduction in length of the separation bubble
is shown in Figure 6, which should be compared
with Figure 3. When the maximum Courant number
exceeds 1 the computational model is not able to
resolve the flow inside the separation area and therefore
underestimates the length of the primary recirculation
bubble. Although the presence of the secondary recir-
culation bubbles is still detected, its centre of rotation
moves close to the leading edge, and the size is reduced.

6. UPWIND SAILS EXPERIMENTS

The most detailed set of experimental data for
upwind sails are those of Fluck (2010). The pressure
distributions on the sails are summarised in Fluck et al.
(2010). Successively, Viola et al. (2011) tested different
sail shapes and trims with the same model-scale sails
and experimental setup developed by Fluck (2010).
The tested geometries and the measured pressure
distributions are available on Viola et al. (2013).

Figure 6: Leading edge separation bubble calculated
with maximum Courant number=3.

The experiment used two thick fiberglass sails to model
a 1/15th scale upwind sail plan of a hypothetical AC33
class yacht design. The distance between the top of the
mainsail and the foot of the headsail was 2.25 m with
the maximum chord-length of the head and mainsails
being 0.7 m and 0.65 m respectively. The sails were
approximately 4 mm thick with a blunt leading edge.
The mast and hull were not included in the experimental
model, which was lifted 0.6 m above the floor to avoid
the wind tunnel floor boundary layer, and mounted on a
square baseplate. Twist vanes were not used to ensure
the sails were tested in a straight uniform onset flow
(Figure 7).

Figure 7: University of Auckland Twisted Flow Wind
Tunnel experimental apparatus.

The head and mainsails were both equipped with pres-
sure taps, which were distributed in eight rows, four on the headsail and four on the mainsail, as shown in Figure 8. The wind tunnel onset flow was 7.4 m/s, with the turbulence intensity at the model sails location being approximately 1.5%. The sails and base were oriented to give an apparent wind angle of 18.7°, and the two sails were each trimmed in four different positions, giving 16 different sailing configurations. The maximum driving force was generated with a sheeting angle of 7.9° for the headsail and −2° for the mainsail and this configuration (termed G3M2 by Viola et al. (2013)) has been chosen as the reference case for this investigation.

7. UPWIND SAILS COMPUTATIONAL MODEL

The geometry for the model was obtained from Viola (2013). The model centreline was rotated by an angle of 18.7° to match the experiment, and the effects on the flow field generated by the square baseplate were reproduced by including a square no-slip surface underneath the sails in the computational model. Instead of lifting the baseplate and the sails 0.6 m above the floor as in the experiments, the computational domain was cut at the height of the baseplate with a free-slip boundary condition being applied to the lower surface. This configuration was used successfully by Viola et al. (2013) and Queutey et al. (2015) and provided satisfactory results.

The sails were modelled as thick surfaces and although the inclusion of thickness resulted in a small increase in the number of cells in the mesh, it improved the prediction of the location of the stagnation point at the leading edge of the sails. Different leading edge shapes with constant thickness were tested, including squared, chamfered and rounded geometries. Since no significant differences were observed in the resulting pressure distributions on the sails, a squared geometry was chosen, which simplified the meshing process.

Previous studies (Viola et al., 2013; Queutey et al., 2015; Nava et al., 2016) found that the choice of domain size and boundary conditions greatly affected the solution. In the wind tunnel the sails are in an open test section and an open jet flows around the sails, rather than the constrained flow of a closed test section. This must be reproduced in order to reproduce the experiment. Therefore, an outer box 13 m wide and 15 m long was used with the sails placed at the centre. The height of 3.1 m matched that of the wind tunnel, the inlet only covered a portion of the front surface, while all the other bounding surfaces were set to an constant pressure outlet boundary condition (Figure 9). The wind tunnel walls were modelled as free-slip walls as was the roof, in order to minimise the computational effort in calculating the flow behaviour close to these surfaces. Finally the sails were given a no-slip wall boundary condition.

Figure 9: Computational domain and boundary conditions used to reproduce the open jet wind tunnel.
simulation. Although the flat plate results highlighted the need for a highly refined mesh, the large size of the upwind sail domain made it infeasible to reproduce the same mesh refinement, due to the high computational cost. However, care was taken when meshing the top section of the mainsail, where the experiment showed the formation of a leading edge separation bubble. The fine mesh resolution at the top of the mainsail was achieved by reducing the mesh density in other area of the domain, such as the jib, to control the overall mesh size. The spacing normal to the wall was set to 0.1 mm giving a $y^+$ less than 1. The streamwise direction cell length was set to 1 mm in the leading edge area, with the length increasing to 3 mm at the trailing edge. The growth ratio normal to the wall was approximately 5% on the leeward surfaces and 15% on the windward surfaces, while it was 10% in the streamwise direction near the leading edge and 20% towards the trailing edge. Finally, an approximately uniform discretization was used in the spanwise direction, with the nodes separated by approximately 20 mm. This value was decreased near the edges to 5 mm.

The inlet flow velocity was set to a uniform 7.4 m/s as used in the experiments. The inlet flow turbulence intensity was set to 3% and this decayed to turbulence intensities of 1% and 0.6% at the location of the sails for the RANS and LES models respectively. These values are on the order of the turbulent intensity of the tunnel which is approximately 1.5%. The turbulence length scale was set to 0.4 m which is three times the mesh size at the inlet section, allowing sufficient nodes for the LES model to resolve the large scale turbulent structures.

The timestep size for the results in the next section was 10 $\mu$s corresponding to a maximum Courant number smaller than 1. The simulation was run for 600,000 timesteps to let the solution converge as described in section 2.

8. PRESSURE DISTRIBUTION AND FLOW FIELD

As with the flat plate model, both RANS and LES solutions were computed and are compared. The wall shear streamlines shown in Figure 11 for the leeward surfaces of the sails indicate that the flow was mostly attached for the RANS but regions of separation are observed in the LES. The contours of pressure coefficient show the maximum suction occurs on the headsail, typically occurring near the leading edge and near the position of maximum camber. The pressure distribution over the mainsail has a lower suction than the headsail, partly due to the reduced camber of the sail and partly because of a local smaller angle of attack due to the downwash generated by the headsail.

The influence of the headsail on the mainsail flow is noticeable looking at the drastic change in the flow structure between the sections above and below the hounds at the top of the headsail. The top of the mainsail is not affected by the downwash generated by the headsail, resulting in a high apparent angle of attack and flow separation.

The predicted amount of separation differs between the calculation methods. As shown in the pressure coefficient ($C_p$) distribution for the head of the mainsail labelled M4 in Figure 12, the LES results show a similar pressure distribution to the flat plate case (c.f. Figure 4a), with an approximately constant pressure extending for the length of the leading edge separation bubble showed in Figure 13b. However, in the RANS solution the section is fully stalled (see also Figure 13a) with the air backflowing from the trailing edge to the leading edge. In contrast, the LES simulation (see Figure 13b) predicts the reattachment of the leading edge separation bubble on that section at approximately 60% of the chordlength. The pressure distribution inside the bubble varies for the different simulations as shown in Figure 12, with the LES showing the best agreement with the experimental data inside the

![Figure 10: Detail of the mesh around the sail location used for the upwind sails model.](image)
Figure 11: Wall shear streamlines and contours of pressure coefficient on the sails leeward (suction) surface. Flow from left to right.

Significant differences between the RANS and LES solutions can also be observed in the mainsail trailing edge flow along the span of the sail. The RANS simulations predict the flow to be fully attached, while the LES shows a region of separated flow, extending for nearly a quarter of the length of the chord. Despite the differences between flow structure in the solutions, the pressure distributions are similar at the trailing edge, and both are in good agreement with the experimental data.

For the headsail, the solutions for the RANS and LES are similar at the lower sections of the sail and agree with the experiment. The wall shear stress streamlines indicate that the flow near to the surface has an upward direction, calculated differently by the RANS and LES. Moreover, the predicted flow over the head of the headsail differs for the two models. The RANS simulations show the flow as attached, with the $C_p$ matching the experiment. In contrast, the LES simulation shows separated flow moving from the trailing edge towards the leading edge. The different flow field computated is reflected in the pressure distribution on the two upper sections which over-predict the suction by approximately 10% compared to the experimental results.

The differences between RANS and LES in reproducing the experimental flow field are summarized in Figure 14, which shows the values for the averaged RMS errors for each section of the sails in the pressure prediction on the suction side, for both RANS and LES. The RMS values are defined using the following formula:

$$RMS_{error} = \sqrt{\sum_i \left( C_{p,i}^{CFD} - C_{p,i}^{EXP} \right)^2 \cdot \frac{\Delta x_i}{c}}$$

where $\Delta x_i$ is the portion of the chord associated with each pressure tap and $c$ is the chord length of each sail section. The RANS prediction for the headsail is closer to the experiments, as highlighted by the pressure coefficients distributions in Figure 12, with RMS error values around 0.1, while the LES shows higher RMS errors, with the maximum being 0.15 at the third sail section from the bottom. On the mainsail the trend is reversed, with the LES errors significantly smaller than the values affecting the RANS. In particular, the greatest difference between LES and RANS is visible at the top section of the mainsail (section in Figure 14), where the RMS error for the LES is 0.08 in accordance
with the other sections, while the RANS RMS error is nearly 0.25, considerably higher than the LES and the other RANS sections.

9. LEADING EDGE SEPARATION ON THE MAINSAIL

The results presented in the previous section show that the top of the mainsail is the most difficult region to model using RANS. The reason for this is the development of a leading edge separation bubble due to the sharp edge at the luff of the sail.

The velocity streamlines shown in Figure 13 present data for a RANS and a LES computation on the same mesh (4.7 million cell mesh) and show that the flow detaches at the edge of the leeward side of the sail, increases speed along the length of the recirculation bubble and bends toward the sail surface at approximately mid-chord. Here, the streamlines divide, flowing to the trailing edge and forward towards the leading edge, feeding the recirculation bubble. The contours of streamwise velocity in Figure 13 indicate that the velocity field downstream varies significantly for the different models.

In the RANS solution the centre of the bubble is located at 15% of the chord, while in the LES results it is at almost 40% of the chord. For the RANS model, even though the velocity streamlines suggest flow reattachment downstream of the bubble center location, careful examination and a comparison with the 3D flow field discussed in section 8 indicate that the flow is running vertically in the 3D vortex structure highlighted by the streamlines in Figure 11a, with small or negative values of streamwise velocity extending past the trailing edge. In contrast, the velocity contours for the LES solution shows a rapid recovery with values of velocity 3 to 5 times higher than the RANS solution.

An explanation for why the recirculation bubble is resolved differently by the two models can be found from careful examination of the flow at the leading edge of the sail (Figure 13c and 13d). The LES velocity vectors and streamwise velocity contours predict the development of a secondary separation bubble not present in the RANS solution. For the LES solution, the backflow in the leading edge separation bubble has a high velocity, approximately one quarter of the freestream velocity. This flow decelerates near the leading edge and it separates due to the high positive pressure gradient forming a secondary separation bubble. This is in agreement with the model of the flat plate in section 5, although the presence of a secondary recirculation bubble can not be confirmed by the experiment as the only data available is the pressure distribution recorded with the pressure taps.

As in the flat plate case, the size of the timestep limits the accuracy of the solution. Figure 15 shows the pressure distribution at the head of the mainsail for two different Courant numbers. In the maximum Courant number 5 case, the constant pressure region is replaced with a suction peak that overestimates the experimental data and the low Courant number solution by approximately 30%. Moreover, the pressure recovery is located earlier along the chordlength for the larger timestep simulation while the short timestep calculation yields an accurate prediction of its position. This behaviour matches that seen for the flat plate in Figure 5.

10. CONCLUSIONS

The ability to predict the Leading Edge Separation Bubble on a flat plate and on upwind sails has been tested using RANS and LES, with the results validated against published experimental data. Over most of the sail surface the methods agree equally well with the experimental data. Small differences are seen between the results of the RANS and LES models of the upwind sails, with the latter predicting flow separation on part of the mainsail trailing edge and at the top region of the headsail.

However, a substantial difference is seen between the methods in the prediction of the leading edge separation bubble. The LES model correctly predicts the formation of a secondary recirculation bubble on the flat plate, and also predicts its presence on the upwind sails, while the RANS does not. In both cases the LES model has been shown to be able to predict the correct pressure distribution for this separated flow, while the RANS simulations shows less accurate results.

There are substantial differences between the flow features predicted by the solvers for both the flat plate and sail flows. However, there is no significant differences between the pressure distributions, with both the computational models in good agreement with each other and the experimental data, in the attached zone of the flow. Conversely, large differences between the predicted pressure distribution between RANS, LES
Figure 12: Distribution of pressure coefficient on the four headsail sections (left) and on the four mainsail sections (right): (○) experiments (Fluck, 2010), (- -) RANS, (—) LES.
and experiment appear in the areas of flow separation.

Finally, the computational effort required by the LES simulation is much larger than the RANS by a factor of 50 for the coarser meshes and by a factor of 100 for the finer meshes.

11. ACKNOWLEDGEMENTS

The authors wish to acknowledge the contribution of NeSI high-performance computing facilities to the results of this research. NZ’s national facilities are provided by the NZ eScience Infrastructure and funded jointly by NeSI’s collaborator institution and through the Ministry of Business, Innovation & Employment’s Research Infrastructure programme. URL http://www.nesi.org.nz.

REFERENCES

Masuyama Y., Tahara Y., Fukasawa T., Maeda N.: Database of Sail Shapes vs. Sail Performance and Validation of Numerical Calculation for Upwind Condition, in the proceedings of the 18th Chesapeake Sailing Yacht Symposium, Annapolis, Maryland, USA. (2007).
Nava S., Cater J., Norris S.: A Comparison of RANS and LES for Upwind Sailing Aerodynamics, in the proceedings of the 22nd Chesapeake Sailing Yacht Symposium, Annapolis, Maryland, USA. (2016).


Figure 14: Force averaged values for the sectional RMS errors in the pressure distribution calculation: (■) RANS, (■) LES.

Figure 15: LES pressure coefficient distribution on the mainsail top section computed with two different Courant numbers: (○) experiments (Fluck, 2010), (—) max Courant number=5, (—) max Courant number=1.