DELAYED DETACHED EDDY SIMULATION OF SAILING YACHT SAILS

IM Viola, Yacht and Superyacht Research Group, School of Marine Science and Technology, Newcastle University, UK, (corresponding author) ignazio.viola@ncl.ac.uk
S Bartesaghi, Mechanical Department, Politecnico di Milano, Italy.
T Van-Renterghem, Yacht and Superyacht Research Group, School of Marine Science and Technology, Newcastle University, UK.
R Ponzini, CINECA, SuperComputing Applications and Innovation Department (SCAI), Italy.

Wind tunnel experiments on a 1:15\textsuperscript{th} model-scale AC33-class yacht were modelled with Reynolds-average Navier-Stokes simulations (RANS) and Delayed Detached Eddy Simulations (DDES). Numerical simulations were performed with two different grids, where the node distance was halved from the coarser to the finer grid, and with three different time steps, where the smallest one was 1/4\textsuperscript{th} of the largest one. High-grid-resolution DDES allowed drawing the topology of the turbulent structures in the sail wake and discovering new flow features, which were hardly detectable with low-grid-resolution DDES and, particularly, with RANS. It was found that the span-wise twist of the spinnaker leads to a mid-span helicoidal vortex, which has a horizontal axis almost parallel to the apparent wind and rotates in the same direction of the tip vortex generated from the head of the sail. Vortical span-wise tubes are released from the trailing edges of the mainsail and the spinnaker and, while convecting downstream, these structures roll around the tip and mid-span vortices of the spinnaker. Vortical tubes are also detached intermittently from the sails’ feet and these break down into smaller and smaller structures while convecting downstream.

1 INTRODUCTION

Sailing yacht sails are efficient aerodynamic fins, which operate at low Reynolds numbers ($Re$). In particular, full-scale sails operate at $Re$ of the order of $10^6$ (based on the aerodynamic sail chords) but sails are commonly tested and optimised at $Re$ of the order of $10^5$ using model-scale wind tunnel tests. Traditionally, low-$Re$ aerodynamics ($Re \approx 10^4 - 10^5$) has received somewhat less attention than high-speed aerodynamics ($Re \geq 10^6$), though today there is an unmet need for fluid-dynamic efficiency in emerging applications where fins operate at low $Re$, such as autonomous underwater vehicles, micro aerial vehicles and small renewable energy converters. These applications could benefit from the research on sail aerodynamics and, particularly, on the analysis of some characteristic flow features of highly cambered twisted sails leading to good aerodynamic performance.

On conventional thick airfoils at high $Re$ the laminar-to-turbulent transition occurs near the leading edge. Therefore, the boundary layer is mostly turbulent allowing large entraining momentum from the outer layer and making it able to tolerate adverse pressure gradients due to the airfoil curvature. Conversely, on the suction side of airfoils at $Re$ between roughly $5 \cdot 10^4$ and $3 \cdot 10^6$ \cite{1}, a laminar boundary layer develops from the leading edge until separation occurs due to the adverse pressure gradient; then the unstable separated shear layer triggers the laminar-to-turbulent transition and reattachment occurs, leading to the ‘laminar separation bubble’ and to a turbulent boundary layer downstream the bubble. At low $Re$, the performance of conventional thick airfoils designed for high $Re$ is poor and thinner airfoils may allow much higher maximum lift and lift/drag ratio than thick airfoils \cite{2,3}. Sails are very thin airfoils and the flow separates at the leading edge due to the sharpness of the edge, leading to a high suction peak \cite{4} (Figure 1). The laminar-to-turbulent transition occurs in the separated shear layer, leading to reattachment and then to the development of a turbulent boundary layer. Further downstream along the chord, the sail curvature leads to a second suction peak. Highly cambered sails show significant trailing edge separation due to the adverse pressure gradient correlated with the sail curvature, but allow a very high driving force. The sharp leading edge and the second suction peak due to the sail curvature are typical of sails and unusual on airfoils. Figure 1 shows the typical flow and pressure fields when the complementary angle between the true wind velocity and the boat velocity is larger than 90°, leading the boat to experience a wind coming between roughly 45° and 90° from her bow. In these conditions, modern sailing yachts use a highly cambered foresail, namely the spinnaker, and flatter and smaller aft sail, namely the mainsail.

Spinnaker (foresail) and mainsail (aft sail) can be compared with the two superimposed wings used by biplanes. The chord and span of the fins of an aircraft and a yacht are of the same order of magnitude but the thickness and the $Re$ of yacht fins are more than one order of magnitude smaller than those of aircrafts. Differently from aircraft wings, sails are significantly twisted and cambered both chord-wise and span-wise. For instance, the spinnaker analysed in this paper has an aspect ratio (span/mean-chord) of 1.8 (2326mm/1297mm), a span-wise twist angle (horizontal angle between the lowest and highest chords) of more than 17°.
a chord-wise camber of the chord, and a span-wise camber of the span. The sail twist moderates the increase of angle of attack due to the twist of the onset flow experienced by a sailing yacht, namely the apparent wind. In fact, the apparent wind is the vectorial difference between the true wind and the boat velocity, and it increases and rotates from the bow towards the stern of the boat with the height (Figure 2). The bi-cambered (chord-wise and span-wise) twisted geometry of the sails leads to a characteristic wake.

![Pressure coefficient graph]

Figure 1: Typical flow and pressure distributions on sails

![Twist of the apparent wind experienced by a sailing yacht]

Figure 2: Twist of the apparent wind experienced by a sailing yacht

The study of sails’ wake can be performed experimentally or numerically. Full-scale experiments are very complicated for wake measurements and have never been attempted. However, forces [5-9] and pressures [10-16] were measured in full scale, though the non-controllable and unmeasured atmospheric boundary layer profile limited the measurement accuracy. Model scale sails are normally tested in wind tunnels, where flexible sails with low thickness/chord ratio can be used (for instance, [17]). However, PIV and LDV measurements are difficult in wind tunnels due to the need for inseminations and only unpublished smoke observations were performed. Flow visualisation is easier in water tunnels, where unfortunately thin models are used with difficulty due to the very high hydrodynamic loads. A sensible way to study sail wakes is using numerical simulations. Potential flow codes cannot model viscous effects, which are dominant in the wake and, therefore, Navier-Stokes solvers must be used. The relatively high Re and the complex 3D geometry make Direct Numerical Simulations (DNS) unfeasible and turbulence must be modelled with turbulence models or subgrid models. Reynolds-averaged Navier-Stokes simulations (RANS) have been performed since 1996 on downwind sails [18] and, since then the agreement between numerical and experimental forces has increased in parallel with the growth of computational resources. The number of grid cells increased by about one order of magnitude every three years; Hedges et al. [18] used a number of grid cells of the order 10^7, three years later Miyata and Lee [19] used a number of grid cells of the order 10^8, and ten years later Viola [20] used a number of grid cells of the order 10^10. Richards and Lasher [21] and Viola and Flay [15] compared pressure distributions computed with RANS to those measured in wind tunnels. They found good numerical-experimental agreement on the mid sections of the sails but larger differences on the highest sail sections, where the suction peak near the leading edge was under-predicted by CFD.

As far as known by the authors, the present paper presents the first published investigation on sail aerodynamics performed with Detached Eddy Simulations (DES). However, it must be noted that Braun and Imas [22] stated that DES was used in the design process of an ACC-V5-class yacht for the 32nd America’s Cup, though no results were presented; and Wright et al. [23] presented few results achieved with DES but no details were provided to verify the validity of the simulation. In the present paper, the wind tunnel test on a spinnaker with both RANS and DES, using different grids and time steps, are presented.

The paper is structured as follows: in the Method section, the experimental tests are introduced and the numerical simulations modelling the experiments are described, including details of the equations solved, the boundary conditions, the grids and the time steps tested, and the hardware used to run the simulations. The procedure used to assess the numerical uncertainty in the computation of forces and pressures is also presented. In the Results section, the general flow field computed with the numerical simulations is presented, and details of the
near-wall region and of the sail wake are discussed. Forces and pressures computed with the different simulations are compared with the experimental data. In the Conclusions section, the key findings of the research are summarised.

2 METHOD

2.1 WIND TUNNEL TESTS WITH FLEXIBLE SAILS

A 1:15 th model-scale AC33-class yacht equipped with flexible sails was tested at the Auckland University wind tunnel. Figure 3 (left) shows the model during the wind tunnel test. The tunnel has a 3.5-m-high and 7-m-wide open jet section, where the floor and the roof extend downstream for 5.1m and 4.8m, respectively. The 2.3-m-high model was placed on the wind tunnel floor at 2.7m downstream from the open jet section. A flexible spinnaker and mainsail were mounted on a model scale yacht, which included the hull and the rigging, at an apparent wind angle and a heel angle. Viola and Flay reported the force [24] and pressure [25] measurements. Forces were measured using a 6-component balance placed underneath the wind tunnel floor, and sail surface pressures were measured using pressure taps attached to the sails. Pressure taps were 20-mm long, 10-mm wide and 4-mm height, attached to the sail on the opposite side to that under investigation, and a 1-mm-diameter hole was made in the sail to allow pressure transmission to the tap. PVC tubes with a 1-mm internal diameter, suspended from the sail to the boat mast, carried the pressures from the tap to the pressure transducers located on the boat deck. Pressure taps were placed on 5 horizontal sections at heights of 1/8, 1/4, 1/2, 3/4 and 7/8 of the mitre, which is the line on the sail surface equally far from the leech and the luff. The far-field static pressure was computed by the difference of the total and dynamic pressures measured by a Pitot static probe located approximately 10 m upstream at the top-mast height. The pressure transducers measured the difference between the sail surface pressure and a reference pressure, which was time-averaged over a period of 70 s and was about 7.5 Pa. Forces were measured at 200 Hz and averaged over the same period of 70 s. Uncertainties in the measurement of forces were estimated to be about 1% for the leeward and windward sides, respectively. Several photographs were taken during the tests and were used to detect the flying shapes of the two flexible sails in order to make a mathematical model, which was used to perform CFD simulations and, successively, to build rigid sails for further tests.

2.2 WIND TUNNEL TESTS WITH RIGID SAILS

The mathematical model of the flying shapes was used to build a CAD/CAM wooden mould, which, in turn, was used to build rigid sails with fibreglass and a sandwich structure [26]. The sails were less than 4-mm thick, mainly due to the thickness of the core, with the external fibreglass layer of negligible thickness. The thickness/chord ratio was less than 1%. The core was made of extruded polypropylene, resulting in parallel square tubes. These were used to carry the pressure from 1-mm-diameter holes on the sail surface to the trailing edge, where 1-mm internal-diameter PVC tubes, gathered together along the trailing edge towards the sail foot, carried the pressure to the pressure transducers located on the boat deck. Figure 3 (right) shows the model during the wind tunnel tests. The same testing setup as the one adopted with flexible sails was used: pressures were measured at the same sail sections, forces and pressures were measured with the same instrumentation, at the same frequency and averaged over 70 seconds. Uncertainties in the measurement of forces were estimated to be the same as for flexible sails.

Figure 3: Wind tunnel tests performed with flexible sails (left) and rigid sails (right).
2.3 COMPUTATIONAL DOMAIN AND BOUNDARY CONDITIONS

The detected flying shapes of the sails were used to perform the numerical simulations. Sails, mast, boom (horizontal mast at the mainsail foot) and hull were modelled with non-slip condition. A prismatic computational domain 3-m high, 6.2-m wide and 18.4-m long was used to model the wind tunnel (Figure 4). The domain length is equivalent to eight times the boat height ($h = 2.3$ m). A smaller test section than the physical one was modelled (3x6.2 m instead of 3.5x7 m) in order to avoid solving the sidewall and roof boundary layers. Slip condition was used on these boundaries from the inlet to the open jet section, while pressure outlet was used downstream from the open jet section. The experimental blockage is almost negligible and mostly due to the wind tunnel sidewalls which decrease the deflection of the streamlines upstream the model. This effect is taken into account using slip conditions, though it has an almost negligible effect being the open jet section wider than 1.5$h$ and more than $h$ upstream from the model. The onset vertical velocity profile measured in the wind tunnel experiment was used as inlet condition. The wind direction was uniform on the test section (un-twisted flow), while the wind speed presented a boundary layer profile on the floor. Therefore, non-slip condition was used on the floor boundary, which extend 3.5$m$ downstream from the model. The mean velocity of the onset flow was $U_\infty = 3.5$ m/s (7.5 Pa), the turbulent intensity was set to $T_u = 3\%$, as measured in the wind tunnel, while the turbulent length scale was assumed to be $l_t = 0.01m(\approx h/200)$. The computational domain extended downstream further than the end of the physical roof and floor, therefore pressure outlet conditions were used on these boundaries.

2.4 GRIDS

Two non-structured hexahedra grids were build with Pointwise version 16.04 R1. The coarse grid was made of four million cells (4M). Figures 5 shows the surface grid on the spinnaker (left) and a grid section at 1/2 of the spinnaker’s mitre height (right). The 4M-cell grid allowed modelling the spinnaker with about 60 cells chord-wise and about 64 cells span-wise, with $y^+$ ranging from 0.01 to 10. A finer grid was achieved using the hanging node function of Ansys Fluent version 13.0.0, which split every cell in eight cells leading to a 32-million-cells grid (32M). Table 1 shows the maximum and minimum $y^+$ computed by the different simulations on the suction side of the spinnaker.

2.5 REYNOLDS-AVERAGED NAVIER-STOKES

The incompressible steady RANS equations for Newtonian fluids were solved with the finite-volume pressure-based solver of Ansys Fluent version 13.0.0. The Spalart-Allmaras turbulence model was used to model the turbulence. This one-equation model was preferred to more accurate two-equations models in order to decrease the computational time. The production term of the modified turbulent viscosity $\overline{\nu}$ was computed with a vorticity-based approach, and at the inlet it was set as follows: $\overline{\nu} = 3^{0.5} 2^{-0.5} U_\infty T_u l_t$. A SIMPLEC scheme was used to couple velocity and pressure. A second-order accurate centred discretization algorithm was used for the pressure, while second-order-accurate upwind algorithms were used for momentum and modified turbulent viscosity.

![Figure 4: Computational domain and boundary conditions.](image)

![Table 1: $y^+$ for the two grids computed with RANS and DDES](image)

<table>
<thead>
<tr>
<th>Grid Type</th>
<th>Min</th>
<th>Max</th>
</tr>
</thead>
<tbody>
<tr>
<td>4M RANS</td>
<td>0.58</td>
<td>8.56</td>
</tr>
<tr>
<td>4M DDES</td>
<td>0.76</td>
<td>9.05</td>
</tr>
<tr>
<td>32M DDES</td>
<td>0.19</td>
<td>5.54</td>
</tr>
</tbody>
</table>

2.6 DELAYED DETACHED EDDY SIMULATION

The transient Navier-Stokes equations were solved with a DES approach. A Spalart-Allmaras turbulence model, with a vorticity-based production term, was used to model the turbulence in the RANS region. In order to preserve the RANS model throughout the whole boundary layer, the DES length scale was modified as suggested by Spalart et al. [27] for the Delayed Detached Eddy Simulation approach. A SIMPLEC scheme was used to couple velocity and pressure. Second order accuracy discretization algorithm was used for the pressure, while second order central difference algorithms were used for momentum and modified turbulent viscosity. The fluctuating velocity components at the inlet were computed by synthesizing a divergence-free velocity-vector field from the summation of 100 Fourier harmonics. More details on the numerics can be found in the User Manual of Ansys Fluent [28].

2.7 TEST MATRIX

A RANS simulation was performed on the 4M-cell grid, while DDES simulations were performed on both the 4M-cell grid and the 32M-cell grid. On the coarser grid,
three different time steps were tested, 0.001 s, 0.002 s and 0.0005 s, in order to estimate the uncertainty due to the time discretisation, while the maximum number of iterations per time step was kept constant to 20, allowing convergence at each time step. All these time steps allowed Courant numbers in the sails’ wake lower than one. For instance, with a time step of 0.001 s and the 4M-cells grid, the Courant number ranged from $10^{-5}$ to $10^{-2}$. On the 32M-cell grid, only the intermediate time step (0.001 s) was used with 20 iterations per time step. Table 2 summarises the numerical simulations performed.

All the numerical simulations ran until convergence was achieved for the aerodynamic forces. In particular, lift, drag and heeling moment were monitored. Forces, pressure and velocity fields computed with DDES were averaged over a period of 10 s. For example, Figure 6 (left) shows the convergence of the drag coefficient $C_D$ of the two sails (hull and rigging excluded) for the DDES simulations performed with high grid resolution.

Table 2: Test matrix of the numerical simulations

<table>
<thead>
<tr>
<th>Method</th>
<th>Space discretisation</th>
<th>Time discretisation</th>
</tr>
</thead>
<tbody>
<tr>
<td>RANS</td>
<td>4M-cell grid</td>
<td>Steady</td>
</tr>
<tr>
<td>DDES</td>
<td>4M-cell grid</td>
<td>0.0005 s</td>
</tr>
<tr>
<td>DDES</td>
<td>4M-cell grid</td>
<td>0.0010 s</td>
</tr>
<tr>
<td>DDES</td>
<td>4M-cell grid</td>
<td>0.0020 s</td>
</tr>
<tr>
<td>DDES</td>
<td>32M-cell grid</td>
<td>0.0010 s</td>
</tr>
</tbody>
</table>

2.8 HARDWARE

All simulations were performed in double precision on a 64-bit Hewlett-Packard Linux cluster made of 336 nodes HP 2x220 2xIntel Exa-cores 3.166 GHz — 24Gb RAM per node interconnected with Infiniband QDR and a node HP DL980 8 CPU Intel E5420 — 512 GB RAM for post-processing and results visualization. In order to take advantage of the High Performance Computing system, a preliminary scalability test using the smallest grid was performed. According to the scalability results the calculations on the different grid sizes have been performed using up to 256 computational cores.

2.9 VERIFICATION

Different time and grid resolutions allowed estimating the numerical uncertainty for forces and pressures with DDES. This estimate is only approximate; in fact DDES does not necessarily show asymptotic convergence with increasing resolution [29].

The uncertainty at 95% confidence level was computed following the guidelines of Viola et al. [30]. For example, the uncertainty $U_{tCD}$ due to the time step for the $C_D$ were estimated using Equations (1):

$$ U_{tCD} = 1.5 \frac{C_{D_{MAX}} - C_{D_{MIN}}}{1 - \frac{0.0005}{0.002}} $$

where $C_{D_{MAX}}$ and $C_{D_{MIN}}$ are the maximum and the minimum $C_D$, respectively, between those computed with time steps 0.0005 s, 0.001 s and 0.002 s.

The uncertainty $U_{CD}$ due to the grid for the $C_D$ were estimated using Equations (2):

$$ U_{CD} = 1.5 \frac{C_{D_{MAX}} - C_{D_{MIN}}}{1 - \left(\frac{\sqrt{4}}{\sqrt{32}}\right)^{\frac{1}{4}}} $$

where $C_{D_{MAX}}$ and $C_{D_{MIN}}$ are the maximum and the minimum $C_D$, respectively, between those computed with the 4M-cells grid and the 32M-cells grid, respectively.

The convergence uncertainty $U_c$ was estimated as two times the standard deviation of the time history of each quantity $\Phi$. For instance, Figure 6 (right) shows the mean (dotted line) and the uncertainty (error bar) of the drag coefficient $\Phi = C_D$. The convergence uncertainties for $C_D$ and $C_L$ were $U_C D = 0.016 C_D$ and $U_C L = 0.010 C_L$, respectively.

The numerical uncertainty was then computed as the L2-norm of the uncertainties due to the time step and due to the grid, plus the convergence uncertainty, which is not under the square root because it is not independent from the other two uncertainties (Equation 3):

$$ U_\Phi = \sqrt{U_{t\Phi}^2 + U_{g\Phi}^2 + U_c^2} \quad (3) $$

The resulting uncertainties for the aerodynamic forces were $U_{C_D} = 0.11 C_D$ and $U_{C_L} = 0.096 C_L$; while the largest numerical uncertainty for the pressure coefficient was $U_{CP}^{max} = 1.6$. 

The Third International Conference on Innovation in High Performance Sailing Yachts, Lorient, France

INNOV\textsc{sail} 26\textsc{th} - 28\textsc{th} June, 2013
3. RESULTS

The DES approach allowed the identification of flow structures that have never been solved with a RANS approach so far. The key findings of this research are the identification of these structures, and the analysis of their effect on the sails’ mean pressures. In the next subsections, firstly we provide an overview of the general flow field, then we show where the flow separates and reattaches along the spinnaker surface, and then we discuss the different flow structures in the sail wake. We then discuss similarities and differences between the forces and the pressure distributions computed with RANS, DES and measured experimentally.

3.1 GENERAL FLOW FIELD

Figure 7 shows the general flow field around the yacht computed with RANS using the 4M-cell grid. Pathlines are coloured by flow velocity. The two sails behave like tandem wings where the spinnaker is larger and more cambered than the mainsail. The grey scale shows the pressure difference across the sail surface. The larger delta pressures on the spinnaker than on the mainsail are due to the favourable upwash of the mainsail, while the mainsail experiences the unfavourable downwash of the spinnaker.

The flow is attached on the leeward (suction) side of the spinnaker near the leading edge, while near the trailing...
edge separation occurs. Streamlines from the leading edge converge towards two vortical structures: the tip vortex at the head of the sails and a parallel vortex at mid-span height. As far as is known by the present authors, this mid-span vortex, which will be discussed in sub-sections, has never been identified before. It is due to the span-wise camber, which leads to convergence of streamlines toward mid-span. Most of the separated flow, downstream of the trailing edge separation, is convected into this vortex.

On the windward (pressure) side, the flow is attached and the streamlines, which are not showed in Figure 7, are slightly deflected upwards. This is due to the trailing edge being somewhat higher than the leading edge. In fact, the lower corner of the trailing edge, namely the clew (Figure 5), is higher than the lower corner of the leading edge, namely the tack. Only those streamlines which are near the sail foot are attracted by the suction on the leeward side and are thus deflected downward convecting into the separated flow region downstream the sail foot.

3.2 NEAR-WALL REGION

Figure 8 shows skin friction lines on the leeward surface of the spinnaker computed with the 4M-cells-grid RANS (left), the 4M-cells-grid DDES (centre), and the 32M-cells-grid DDES (right). Results for the two DDES simulations are achieved with a time step of 0.001 s and we showed the instantaneous solution at 30 s. Mean skin friction lines for DDES were not computed within the timeframe of this research project. Representative skin friction lines, highlighted with a solid red line, show that the flow is mainly attached in the region near the leading edge, while trailing edge separation occurs (dash-dotted line) somewhere on the second half of the chord. As a reference, several fractions of the spinnaker mitre (the line equidistant from the leading and trailing edge) are showed on the right-end side of Figure 8. Between ½ and ¾ of the mitre, the flow is mostly horizontal before trailing edge separation occurs. Conversely, below ½ of the mitre, the attached boundary layer is deflected upwards. In the separated region downstream of the trailing edge separation, the flow from the lower region moves upwards and converges towards the trailing edge separation line (dash-dotted line) between ½ and ¾ of the mitre. It is interesting to note that the flow field near the spinnaker’s clew is computed differently with low and high-grid resolution. Only DDES with high grid resolution predicts a clear trailing-edge separation from ¾ of the mitre to the sail foot, while RANS and DDES computed with low grid resolution do not show a continuous trailing-edge separation line.

Near the sail foot, the flow from the leading edge is deflected downwards due to the low pressure associated with the highly curved streamlines coming from the windward side and rolling over the sail foot.

Near the leading edge, a laminar-separation bubble occurs. In sail aerodynamics the separation is associated with the sharp leading edge and it is continuous along all the leading edge from the head to the foot. In conventional wings, such as those used in aeronautics, the laminar separation bubble occurs only in the middle of the wing and not near the root and the tip. For this reason it is called laminar-separation “bubble”. Therefore, in sail aerodynamics, it may be more appropriate to use laminar-separation “tube”. The laminar-separation tube (LST) is smaller near the sail foot and becomes progressively larger towards the sail’s head. The flow within the LST has a strong vertical component, as observed also by Viola et al [15], transferring kinetic energy from the lower sections to the tip vortex.

The 32M-DDES results are in very good agreement with the visual observations performed in the wind tunnel with rigid sails. In particular, the position of separation and reattachment lines were qualitatively confirmed using a stick with a wool tail. However, the vertical flow component of the flow in the region around mid-chord at ¼ of the mitre height seemed over predicted.

3.3 WAKE

Figure 9 shows iso-surfaces of Q-criterion [31] equal to 500. The higher the Q-criterion, the more the flow rotation dominates the strain and the shear of the flow, therefore it can be interpreted as an index of the coherency of the flow structure. Iso-surfaces are coloured by the sign of the helicity, red being positive and blue negative. Helicity is computed with reference to the right-handed (positive) Cartesian coordinate system, where the x, y, z axes are the longitudinal, transverse and vertical axes of the wind tunnel, positive towards the inlet, towards leeward and upwards, respectively.

On the left in Figure 9 the results for the 4M-cell grid solved with a RANS approach are presented. The leeward side of the spinnaker is mostly covered by an iso-surface with negative helicity. The negative helicity is due to the negative span-wise vorticity of the boundary layer. Near the trailing edge, separation occurs leading to less coherent flow structures and lower values of the Q-criterion. The tip vortex from the spinnaker’s head is the larger visible flow structure. It converts along an axis which is almost aligned with the wind direction. A similar vortex develops from the spinnaker’s clew (lower corner of the trailing edge), and rotates in the opposite direction than the head vortex. Interestingly, the mid-span vortex is not visible, meaning that its coherency is weaker than those of the visualised structures.

In the centre of Figure 9, the same grid is solved with a DDES approach. Despite the low grid resolution (4M-cells), LES allows solving these flow structures with a much greater extent than RANS. In particular, we found that the tip vortex generated from the head of the
mainsail rolls around the spinnaker’s tip vortex. Also, small flow structures, which become more visible with the 32M-cells-grid DDES (right in Figure 9), appear near the sail foot. These are chord-wise-stretched vortices generated from the spinnaker’s foot and convected downstream intermittently, breaking down into smaller and smaller structures.

Span-wise-stretched vortices are generated from the trailing edge with a significantly lower frequency than those from the sail foot. The few periods computed with the simulations did not allow an accurate measurement of these frequencies. Decreasing the Q-criterion from 500 to 100, it is possible to see that these vortices do not break down as quickly as those from the foot but, conversely, are stretched between the tip vortex and the mid-span vortex. Figure 10 shows the same comparison between different simulations as Figure 9 but with a different prospective view and decreasing the Q-criterion to 100. In order to allow the spinnaker to be visible, the iso-surface of Q-criterion is hidden in a near-wall region. While the mid-span vortex is hardly visible for the RANS simulation, it appears clearly in the two DDES simulations. In particular, with low grid resolution (centre in Figure 10), the mid-span vortex is showed by a continuous vortical tube while its complicated structure is revealed using higher grid resolution.

Figure 11 shows four views of the Q-criterion iso-surfaces computed with the 32M-cells-grid DDES. In the four different views, only the flow structures upstream of section A, B, C and D (Figure 10), respectively, are shown. This sequence allows the visualisation of the correlation between the various flow structures in the sail wake. The vertically stretched trailing edge vortex rolls around the tip and the mid-span vortices, which both have horizontal axes and rotate clock-wise. Therefore, the trailing edge vortex, which is a tube parallel to the trailing edge when detached form the sail, assumes an ‘S’ shape while convecting downstream. The ‘S’ shape is schematically showed with a solid yellow line in Figure 12 (right), while dotted lines show the two axes of the tip and mid-span vortices. The weaker trailing edge vortex of the mainsail also rolls around the tip and mid-span vortices, but due to its windward position with respect to the mid-span vortex, it is broken down into two vortices schematically showed by two white solid lines in Figure 12 (right).

Figure 12 shows the differences between 4M-RANS, 4M-DDES and 32M-DDES in modelling the evolution of the spinnaker and mainsail trailing edge vortices. In particular, the same view as Figure 11(C) is used in Figure 12. The axes of the tip and mid-span vortices computed with high grid resolution are superimposed for comparison on the low grid-resolution RANS and DES, revealing that the lower grid resolution leads also to different directions of the axes. Videos of the simulations, which are available on the webpage of the first author [www.ignazioviola.com], show that the directions of these axes are stationary but different for the two DDES simulations.

Figure 7: Pathlines computed with RANS on the 4M-cell grid.
Figure 8: Skin friction lines on the leeward side of the spinnaker computed by RANS and DDES with the 4M-cell and the 32M-cell grids.

Figure 9: Iso-surfaces of Q-criterion 500 coloured by helicity computed by RANS and DDES with the 4M-cell and the 32M-cell grids

Figure 10: Iso-surfaces of Q-criterion 100 coloured by helicity computed by RANS and DDES with the 4M-cell and the 32M-cell grids
3.4 FORCES

The forces measured with the two experiments showed significant differences and the numerical results of the different simulations are mostly in between the experimental ranges. Figure 13 show the drag and lift coefficients ($C_D$ and $C_L$, respectively) experimentally measured and numerically computed. Coefficients are defined as the total aerodynamic force acting on the sails, rigging and hull, divided by the far field dynamic pressure $q_m$ and the sail surface. The experimental $C_D$ ranges between 0.52 for the rigid sails to 0.64 for the flexible sails, while the $C_D$ computed with the different DDES simulations ranges between 0.52 and 0.56. Similarly, experimental $C_L$ ranges between 1.31 for rigid sails to 1.51 for flexible sails, while $C_L$ computed with the different DDES simulations ranges between 1.43 and 1.46. $C_D$ and $C_L$ computed with RANS show the maximum differences with the experimental data. In particular, while $C_D$ is between the maximum and minimum experimental $C_D$, while $C_L$ is 1% higher than the largest experimental $C_L$ (flexible sails). $C_D$ and $C_L$ computed with DDES are lower than those computed with RANS, though their trends are to increase with the time and the space resolution. However, different resolutions lead to small differences. In particular, differences are smaller than 1% and 3% for $C_D$ and $C_L$, respectively. Interestingly, RANS and DDES with the same grid resolution show larger differences than two DDES simulations where the grid resolution is doubled.

Figure 14 shows the breakdown of the aerodynamic coefficients for the spinnaker, the mainsail and the two sails combined but without hull and rigging. For the three cases, the coefficients were computed using only the sail area of the spinnaker, mainsail and the two sails together, respectively. These broken-down coefficients, which are achieved with difficulty with experimental tests, show that the spinnaker is significantly more efficient than the mainsail, having higher $C_L$ and lower $C_D$, despite its aspect ratio is about half the one of the mainsail. This is largely due to the upwash and downwash experienced by spinnaker and mainsail, respectively.
3.5 PRESSURES

Figure 15 shows the pressure distributions on five sail sections of the spinnaker: 7/8, 3/4, 1/2, 1/4 and 1/8 of the mitre respectively. The pressure coefficient is defined as

\[ C_p = \frac{p - p_	ext{in}}{\frac{1}{2} \rho V^2} \]

where \( p \) is the pressure measured on the sail surface, \( p_	ext{in} \) on both the windward and leeward pressure sides are presented versus the non-dimensional chord-wise coordinate \( c/c_l \). On the left in Figure 15, computed with RANS, 4M-DDES and 32M-DDES are presented. The two DDES simulations are performed with a time step of 0.001 s. Also, measured experimentally with both flexible and rigid sails are presented for comparison. Error bars for the 32M-cells grid show the estimated numerical uncertainties. On the right in Figure 15, computed with 4M-DDES and three different time steps of 0.0005 s, 0.001 s and 0.002 s, respectively, are presented.

The pressure distributions show that sails operate very close to the ideal angle of attack, meaning that the flow at the leading edge is parallel to the local sail surface. In this condition, on the leeward side of the sail, the pressure decreases gradually from the leading edge to the point of maximum sail curvature. At angles of attack just above the ideal one, a very sharp leading edge suction peak occurs, which is clearly visible on the lowest sections of the sail. On the highest section of the sail, the angle of attack is higher, leading to a larger leading-edge suction peak but also to a wider pressure plateau near the trailing edge due to trailing edge separation. On the highest sections, where the LST fails to reattach, the suction peak is very close to the leading edge and does not show the suction peak due to the sail curvature.

The larger differences between the numerical and experimental are near the leading edge on the highest section. In this region, the differences between numerical simulations performed with different grid and time-step resolutions showed large differences. Therefore, it seems that the spatial and time resolution used to model the tip vortex is critical to the correct computation of the sail surface pressures. The different trends on the 7/8 section are reflected on the 3/4 section, while differences are small on the lowest sections. The computed base pressure of the pressure plateau near the trailing edge is also quite different from the one measured experimentally. Both the differences noted on the highest sections and those near the trailing edge suggest that flow separation was under-predicted by the numerical simulations.

As a confirmation of the trends showed by the forces in Figure 13, computed with RANS and DDES show larger differences than computed with different resolutions. Particularly, larger differences occur near the head and foot of the sail, while on the mid section of the spinnaker differences are smaller. On the lowest sections, RANS predicts a later trailing edge separation than DDES and thus a larger suction peak correlated with the sail curvature. On the highest section, where the LST fails to reattach, the suction on the leeward side of the sail is quite sensitive to the different time steps tested with DDES, leading to higher numerical uncertainty and thus to larger error bars. Using different time and grid resolutions, the same pressure trend is computed near the trailing edge and on the windward side of the sail.
The Third International Conference on Innovation in High Performance Sailing Yachts, Lorient, France

The experimental tests presented in this paper are the first of their kind and the large differences between $C_p$ measured with flexible and rigid sails show that the level of accuracy of these tests is still quite poor. The differences between the two measurements are probably due to differences in the sail shapes used during the two experiments. In fact, on one hand the shape of flexible sails is measured with difficulty, and on the other hand rigid sails may experience deformations due to their own weight, being suspended only from the head and tack corners. The numerical simulations are based on the flexible-sail flying shapes, which were also used to build the mould for the rigid sails. Further investigations are in progress in order to establish if the main source of inaccuracy is the photogrammetric reconstruction of the flexible sails or the deformation of the rigid sails. In the first case, the geometry modelled numerically would be more similar to the rigid sails, while in the second case it would be more similar to the flexible sails.

4 CONCLUSIONS

In the present work, wind tunnel experiments on a 1:15$^{th}$ model-scale sailing yacht were modelled with RANS (Reynolds-average Navier-Stokes simulations) and DDES (Delayed Detached Eddy Simulations), allowing new insights on the aerodynamics of sails. In particular, sails are efficient aerodynamic fins, which operate at low Reynolds numbers. The tested configuration foresaw two sails in tandem where the spinnaker (foresail) had larger sail area, low aspect ratio and high camber, while the mainsail (aftsail) had smaller sail area, higher aspect ratio and less camber. Most of the aerodynamic load was carried by the spinnaker, which experienced the upwash of the mainsail.

Experiments were performed with both flexible and rigid sails, and both global aerodynamic forces and pressure distributions on sails were measured. Numerical simulations were performed with two different grids, where the node distance was halved from the coarser to the finer grid, and with three different time steps, where the smallest one was 1/4 of the largest one.

The high grid and space resolution allowed modelling the flow near the sails with high accuracy. An attached boundary layer was found on the windward side (pressure side) of the sails while the flow separates on the leeward side (suction side) along all the leading edge of the spinnaker. Laminar to turbulent transition occurs on the separated shear layer and the flow reattaches on most of the sail but not on the highest region, creating a span-wise-axis laminar separation tube. The reattached turbulent boundary layer grows along the sail chord for more than half chord, when trailing edge separation occurs.

High-grid-resolution DDES allowed drawing the topology of the sail’s wake and discovering new flow features, which were barely detectable with low-grid-resolution DDES and, particularly, with RANS. A helicoidal tip vortex is generated from the head of the spinnaker and convects downstream in the direction of the far field velocity. The tip vortex from the head of the mainsail rolls around the former one. The span-wise twist of the spinnaker also leads to a mid-span helicoidal vortex having a horizontal axis and rotating in the same direction of the tip vortex. It should be noted that the mid-span vortex has never been reported by previous authors, and its role on the aerodynamic performance of the sail should be further explored. Vortical span-wise tubes are released from the trailing edges of the mainsail and the spinnaker and, while convecting downstream, these structures roll around the tip and mid-span vortices of the spinnaker. Vortical tubes are also detached intermittently from the sails’ feet and these break down into smaller and smaller structures while convecting downstream.

The comparison between the different numerical models showed that DDES allow a step change in the understanding of the sails’ wake topology. Importantly, the more resolved sail wake lead to differences on the pressure distributions on the sails and thus on the global aerodynamic performances. Forces and surface pressures computed with DDES were in better agreement with the experimental data than those computed with RANS, though significant differences between the measurements performed with flexible and rigid sails did not allow a proper verification of the numerical simulations.

DDES with different time and space resolutions led to similar forces and pressure distributions, while RANS led to significantly different pressure distributions and, particularly to higher suction on the leeward side on the lowest sections of the spinnaker, leading to larger global aerodynamic forces. While the forces predicted by DDES were between the maximum and the minimum forces measured with flexible and rigid sails, RANS predicted a lift force 1% and 17% larger than the those measured with flexible and rigid sails, respectively. Therefore DDES seems to be able to predict sail performance more accurately than RANS. Forces and pressures were almost independent from the time and space resolutions tested in the present work. The largest differences were observed on the suction side of the spinnaker in the region of separated flow: on the highest sections near the leading edge and downstream from the trailing edge separation.

5 ACKNOWLEDGEMENTS

This research was supported in part by CILEA Interuniversity Consortium (Italy), CFD Technologies (UK) and ANSYS (Italy), who kindly provided HPC resources, licences of Pointwise and Fluent, respectively.
Figure 15: $C_p$ versus $x/c$ on five horizontal sail sections computed with different simulations and measured with the two experimental tests.
6 REFERENCES


AUTHORS BIOGRAPHY

Ignazio Maria Viola, PhD, is Lecturer in Naval Architecture at the School of Marine Science and Technology of Newcastle University, UK. He has a background in applied fluid dynamics and a specialist expertise in yacht engineering. His previous experience includes a Post Doctoral Fellowship at the Yacht Research Unit (University of Auckland), which was Scientific Advisor of the America’s Cup team Emirates Team New Zealand, and a PhD (Politecnico di Milano) on experimental and numerical modelling of the aerodynamics of sailing yachts, sponsored by the America’s Cup team Luna Rossa. Ignazio is Group Leader of the Yacht and Superyacht Research Group at Newcastle University, he serves in several international committees including the CFD Specialist Committee of the ITTC, he is Member of the Editorial Board of the Journal of Small Craft Technology, Reviewer for more than ten international journals and has written more than 50 peer-reviewed publications since 2008.

Simone Bartesaghi, PhD, is a former PhD student of the Politecnico di Milano (Italy) who joined the Yacht and Superyacht Research Group for an internship of six months under the supervision of Dr Viola. Simone has a research interest on Computational Fluid Dynamics and, particularly, on its applications to yacht engineering. His previous experience includes Master in Yacht Design (110/110) at Politecnico di Milano and Università degli Studi di Genova. Other projects include consultancies for the small craft industry and yacht designers. In close collaboration with PortoRicerca snc, he was in the design team as CFD RANS analyst for the new design VOR70’s CAMPER/Emirates Team New Zealand Volvo Ocean Race 2011-2012 campaign, 2nd overall and 24h-speed record.

Thomas Van-Renteghem is a former student of the engineering school Arts et Métiers Paris Tech, who joined the Yacht and Superyacht Research Group for an internship of nine months under the supervision of Dr Viola. Simone has a research interest on fluid dynamics and, particularly, on its applications to yacht engineering and aeronautics. Thomas is currently employed by Airbus (Toulouse).

Raffaele Ponzini, PhD, is a member of the Super-Computing Applications and Innovation Department of CINECA, which is the largest Italian supercomputer centre. Raffaele, who was awarded a PhD in Bioengineering at the Politecnico di Milano in 2007, has a specialist expertise in High Performance Computing and Computational Fluid Dynamics. His research interests also include multiscale models in hemodynamics, and scientific visualization.