Numerical Simulations of a Surface Piercing A-Class Catamaran Hydrofoil and Comparison against Model Tests

Thilo Keller, TU Berlin, Berlin, Germany
Juryk Henrichs, DNV GL SE, Potsdam, Germany
Karsten Hochkirch, DNV GL SE, Potsdam, Germany
Andrés Cura Hochbaum, TU Berlin, Berlin, Germany

ABSTRACT

Hydrofoil supported sailing vessels gained more and more importance within the last years. Due to new processes of manufacturing it is possible to build slender section foils with low drag coefficients and heave stable hydrofoil geometries are becoming possible to construct. These surface piercing foils often tend to ventilate and undergo cavitation at high speeds. The aim of this work is to define a setup to calculate the hydrodynamic forces on such foils with a RANS based CFD method and to investigate whether the onset of ventilation and cavitation can be predicted with sufficient accuracy.

Therefore, a surface piercing hydrofoil of an A-Class catamaran is simulated by using the RANS code FineMarine with its volume of fluid method for predicting two phase flows. The C-shaped hydrofoil is analysed for one speed at Froude Number 7.9 and various angles of attack (AoA) by varying rake and leeway angle in ranges actually used while sailing. In addition model tests were carried out in the K27 cavitation tunnel of the Technical University of Berlin, for the given hydrofoil and in the same conditions as simulated with CFD to provide data for validation.

Based on the CFD calculation this paper shows how the rake and leeway angles influence the foil’s lift to drag ratio.
The simulations have been verified by extensive analyses, including domain size verification for unrestricted water, mesh refinement and \( y^+ \) verification. The influence of the dimensions of the cavitation tunnel on the flow around the hydrofoil and the wave system is also considered, as the test section of the K27 influences the flow around the foil, the forces and the wave elevation.

Finally the CFD results are compared with experiments conducted in the K27 in order to validate the used method.

**NOTATION**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \alpha )</td>
<td>Angle of attack (AoA)</td>
<td>(^{\circ})</td>
</tr>
<tr>
<td>( \beta )</td>
<td>Leeway angle (in figures: L)</td>
<td>(^{\circ})</td>
</tr>
<tr>
<td>( \varphi )</td>
<td>Rake angle</td>
<td>(^{\circ})</td>
</tr>
<tr>
<td>( \omega )</td>
<td>Cant angle</td>
<td>(^{\circ})</td>
</tr>
<tr>
<td>( \rho )</td>
<td>Density of water</td>
<td>[kg/m³]</td>
</tr>
<tr>
<td>( \Lambda )</td>
<td>Aspect ratio</td>
<td>[-]</td>
</tr>
<tr>
<td>( c )</td>
<td>Chord length</td>
<td>[m]</td>
</tr>
<tr>
<td>( m )</td>
<td>Tank blockage coefficient</td>
<td>[-]</td>
</tr>
<tr>
<td>( s )</td>
<td>Span</td>
<td>[m]</td>
</tr>
<tr>
<td>( t )</td>
<td>Acceleration time</td>
<td>[s]</td>
</tr>
<tr>
<td>( v )</td>
<td>Speed</td>
<td>[m/s]</td>
</tr>
<tr>
<td>( A )</td>
<td>Projected foil area</td>
<td>[m²]</td>
</tr>
<tr>
<td>( A_{x} )</td>
<td>Area of tank cross section</td>
<td>[m²]</td>
</tr>
<tr>
<td>( AoA )</td>
<td>Angle of Attack for 2D</td>
<td>(^{\circ})</td>
</tr>
<tr>
<td>( Fr )</td>
<td>Froude number</td>
<td>[-]</td>
</tr>
<tr>
<td>( L_{ref} )</td>
<td>Reference length</td>
<td>[m]</td>
</tr>
<tr>
<td>( Re )</td>
<td>Reynolds number</td>
<td>[-]</td>
</tr>
<tr>
<td>( T )</td>
<td>Draft</td>
<td>[m]</td>
</tr>
</tbody>
</table>

**INTRODUCTION**

Within the last years we have seen more and more sailing vessels using the technology of hydrofoils, either to support the hull or to use them as a fully foiling system. Especially the 34th America’s Cup has shown how much potential this technology offers. It shall be noted that there are several different systems developed to get boats fully foiling. The widely used Moth’s technology with a T-foil setup and an adjustable flap is probably the most often built foiling system. But on catamarans like the A-Class the rule book forbids foils like the T-foils on the Moth. The A-Class rules [1] only allow foils to be inserted from top and with a minimum distance between the foils of 1.5m under the waterline. That is why only surface piercing lifting foils are seen nowadays in this class. This type of foil shows some additional challenges in the design process. First of all they show a significant amount of wave making which is causing extra drag compared to deeply submerged foils. Secondly, ventilation and cavitation are likely to occur at increasing speeds and larger AoA’s. Ventilation means that air is sucked down from the free water surface towards the tip of the foil. This leads to a sudden loss of lift, which typically cannot be compensated fast enough to continue a controlled flight.

Within this work RANS calculations have been carried out for a representative segment of an A-Class catamaran lifting foil, to investigate if the hydrodynamic forces on such foils can be correctly predicted with a CFD code in a computationally cost effective setup and if the onset of cavitation and ventilation can be predicted with acceptable accuracy. In order to validate the calculations experimental investigations have been carried out for the same geometry and conditions in the K27 cavitation tank of the TU Berlin.

All computations have been done blind, i.e. they have been carried out prior to the model tests and were preceded by intensive studies on the numerical effects of e.g. computational domain and grid size as well as time step size.

To allow for the most straight forward comparison between experiment and CFD and to avoid any uncertainties due to scale effects, a segment of the A-Cat foil small enough to fit in the test section of the K27 was used.

For the numerical simulations the well-known software package FineMarine was chosen. This software package includes the meshing tool Hexpress to generate unstructured 3D grids needed for the RANS simulation.

The aim of this paper is to present the methods used as well as the results derived from experiment and CFD in order to find a feasible approach for this kind of calculations.

**THE TEST OBJECT**

As test object, an A-Class catamaran hydrofoil has been chosen. Typically these boats sail with both foils immersed at the same time. Due to the large distance between the two foils no direct interaction is expected and therefore only the starboard foil has been investigated.

The foil itself has a span wise bend with a constant radius which makes it a so called C-foil. The chord and section is constant within that bended span region. Towards the tip, the foil is straight over the last 0.3m and shows an elliptical taper on the leading edge with a straight trailing edge (cf. Figure 1). The straight part of the foil is angled 30° (cant angle) to the vertical. The chord length of the foil is \( c=0.16 \text{m} \) tapering to \( c_{T_{tip}}=0.06 \text{m} \) at the tip.

In order to fit the foil in the measuring section of the K27 cavitation tank only a relatively small immersion of \( T=0.3 \text{m} \) could be tested, which represents a rather limiting operational case.

The resulting aspect ratio of only \( A=2.645 \) is rather small and significantly changes the overall performance and behavior of the foil as well as its stall characteristics.

With an immersion of \( T=0.3 \text{m} \) the span equals \( s=0.34 \text{m} \) and projected foil area is \( A=0.0437 \text{m}^2 \).
The section of the foil is a Selig/Donovan SD7032-099-88 and is kept the same over the whole span. It is designed for low Reynolds numbers. The main characteristics of the section are: Thickness 10% of $c$ at 26.6% of $c$ and a camber of 3.4% of $c$ at 45.1% of $c$ (cf. Figure 2). The angle of attack $\alpha$ for zero lift is $\alpha_{CL0} = -4.08^\circ$.

Figure 2 – Foil section [2]

THE TEST CASES

For all CFD simulations the draft $T=0.3\text{m}$, the cant angle $\omega=30^\circ$ and the speed $v=10\text{m/s}$ were kept constant to simplify the test matrix. The rake $\varphi$ was varied between $+2^\circ < \varphi < -8^\circ$ in steps of $2^\circ$ resembling the rake adjustment in the center board case of the boat. The leeway angle $\beta$ was varied from $-2^\circ < \beta < +20^\circ$ in steps of $4^\circ$.

The results, however, will be presented with respect to the resulting 2-dimensional AoA the straight part of the foil will experience and a Cross-Flow-Angle indicating the amount of flow along the span.

THE TEST FACILITY

The K27 cavitation tank of Technical University of Berlin offers ideal conditions for the experimental investigations of the hydrofoil. It provides high enough flow speeds (with a maximum speed of 12 m/s) while offering a free surface at the same time and a comparatively large test section, big enough to fit the chosen segment of the foil in full scale.

Figure 3 shows the principle layout of the K27 facility. The test section is 3.5m long with a rectangular cross section of 0.6m depth (to the free surface) and 0.6m width. The foil is placed in the center plane of the test section 0.3m from the inlet.

The forces on the foil were measured using a 6 component force balance. The balance itself can be rotated around the $y$-axis to change the rake of the foil. To adjust the leeway angle an additional adapter was applied which allowed a rotation around the pitched $z$-axis. The foil was fixed to that adapter with a cant angle of $30^\circ$ to the normal of the adapter plate.

It has to be noted that the rotation sequence is different to that typically used in ship dynamics. To sum up the performed rotation: 1$^{st}$ the rake angle was applied to the foil, 2$^{nd}$ the leeway angle and 3$^{rd}$ the cant angle.
are spaced every 50mm from the tip and the sections every 20mm from the trailing edge.

**Force reference point**

The intersection of the trailing edge with the free surface was chosen as reference point for the force vector and as origin for the CFD computations. The \(x\)-axis is facing forward against the free stream, the \(y\)-axis is facing to port and the \(z\)-axis is facing upwards (see Figure 5).

**NUMERICAL COMPUTATION**

The numerical simulation was performed using the commercial software package FineMarine Version 3.1-3 which comprises the RANS CFD solver ISIS, the grid generator Hexpress and the post-processing tool CFView.

**Domain**

CFD calculations for the foil have been carried out for two situations: placed in the measuring section of the towing tank and, to determine the effect of blockage, in unrestricted waters.

In case of the latter a domain size study has been carried out varying domain width and length to make sure the domain size does not affect the flow and resulting forces. Independence from the domain size was found with the outlet placed 25 chord length behind and the inlet placed 4 chord length forward of the foil trailing edge. The domain sides needed to be at a width of 12 chord length to each side of the foil.

Figures 5 and 6 show the domain for the computations in the cavitation tunnel (Figure 5) and for the open water case (Figure 6).

**Grid Generation**

Hexpress generates non-conformal strictly hexahedral unstructured meshes. Mesh generation is based on an initial background grid which is gradually refined towards surface patches, curves, or control volumes. The level of refinement determines the number of subdivisions of the initial grid.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Ref. side</th>
<th>Ref. LE/TE</th>
<th>Ref. LE-curve</th>
<th>Cell size on surface [m]</th>
<th>Nb. of cells in mesh [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Very coarse</td>
<td>5</td>
<td>6</td>
<td>0</td>
<td>0.02</td>
<td>3285245</td>
</tr>
<tr>
<td>Coarse</td>
<td>6</td>
<td>7</td>
<td>0</td>
<td>0.01</td>
<td>3679304</td>
</tr>
<tr>
<td>Medium</td>
<td>7</td>
<td>8</td>
<td>10</td>
<td>0.005</td>
<td>3775107</td>
</tr>
<tr>
<td>Fine</td>
<td>8</td>
<td>9</td>
<td>12</td>
<td>0.0025</td>
<td>4790642</td>
</tr>
<tr>
<td>Very fine</td>
<td>9</td>
<td>10</td>
<td>14</td>
<td>0.00125</td>
<td>9216295</td>
</tr>
</tbody>
</table>

Table 1 shows the grid refinements on the various surface patches of the foil (see also Figure 7) as well as the resulting cells size on the surface and the number cells for different mesh sizes investigated.

To capture the high pressure and velocity gradients near the leading and trailing edge as well as on the tip more refinements have been applied here compared to the rest of the foil. Figure 7 shows some mesh details for the “fine” mesh.

**Figure 5 – Foil in the test section of the K27 and origin**

**Figure 6 – Domain length variation**

**Figure 7 – Mesh refinements on the foil**

Figure 8 shows the change in side force of the foil for different mesh refinements. Grid independency is achieved for cell sizes smaller 0.02c=0.0025m (fine and very fine grid). The settings of the fine mesh were therefore chosen for the actual simulations.
For a sufficient resolution of the free surface two different refinements were applied to the static water plane, first the $z$-direction was refined to have a sharp borderline between the two fluids and secondly a patch was refined in $x$- and $y$-direction to allow for a better capturing of the wake behind the foil. Some details of the mesh used for the simulations in the K27 are shown in Figure 9.

To capture the boundary layer on the foil as well as on the walls of the cavitation tunnel, a high Reynolds number turbulence model with wall functions was chosen. This requires the non-dimensional distance of the first cell layer normal to the wall ($y^+$) to be roughly about 100 [4].

Figure 10 shows the achieved $y^+$ distribution for various target values of $y^+$ (following the Blasius boundary layer velocity profile) on the pressure side of the foil (top row) and of the suction side of the foil (bottom row). For a target value of $y^+ = 50$, the achieved $y^+$ near the leading edge falls below a value of 30. To ensure the correct application of the wall functions a target value of $y^+ = 150$ was chosen. For the fine grid this resulted in the insertion of 3 viscous layers on the foil sides and 4 layers at the leading and trailing edge.

For the walls of the test section of the K27 the target value of $y^+$ was set to 300, without intending to resolve the boundary layer, to reduce mesh size.

Turbulence Model

The Menter-$k$-$\omega$-SST-Model was chosen for turbulence closure showing generally a good accuracy while keeping computation times small. It is a two equation turbulence model using a blending between near wall flows (using $k$-$\omega$-model) and far away from the wall flows (using $k$-$\varepsilon$-model). The turbulent kinetic energy $k$ and the turbulence frequency $\omega$ can be preset in FineMarine. Since no data was available for the turbulence intensity in the K27, the recommended values have been used [4].

Boundary Conditions

For the simulations in the K27 the walls and the bottom of the K27, as well as the foil itself, have been set as “Solid Wall”, which means that the velocities are zero there. The inlet and outlet have been set to “Far Field”, which is a Neumann-Boundary-Condition, stating that the gradient of the flow variables is equal to zero. The top was set as “Updated Hydrostatic Pressure” which is the only pressure boundary condition used. This is a Dirichlet-Boundary-Condition constantly reinitializing the pressure.

Initial Condition and Time Step

To accelerate the convergence of the forces, the calculations are started at rest and the flow is accelerated to target speed following a quarter-sinus function. The acceleration time is calculated from equation (2), which is a rule of thumb. $L_{ref}$ is the reference length of the simulated body, in this case the chord length, while $\tau$ represents the
target speed of the free flow.

\[ t = 20 \frac{L_{ref}}{v} \]  

(2)

The simulation is performed as pseudo unsteady simulation using a fixed time step. In order to reduce the simulation time a variation of different time steps has been investigated. The largest possible time step allowing convergence of the simulation is 0.01s (cf. Figure 11). Some of the simulations performed with a bigger time step of 0.05s diverged.

\[
\begin{align*}
\text{Figure 11 - Time step variation}
\end{align*}
\]

**CFD RESULTS**

All results will be presented in terms of the dimensionless lift and drag coefficients (Equations 3 and 4) and are based on the 2-dimensional AoA.

\[
\begin{align*}
c_L &= \frac{L}{\rho \frac{1}{2} v^2 A} \\
c_D &= \frac{F_X}{\rho \frac{1}{2} v^2 A}
\end{align*}
\]

(3)

(4)

**Comparison K27 versus open water**

To quantify a possible blockage effect CFD calculations have been carried out for unrestricted open water and considering the actual geometry of the K27 measurement section.

Table 2 shows the blockage \( m = \frac{Ax}{A} \) for a series of different leeway angles at a constant rake of \( \varphi = -8^\circ \). At a leeway angle of \( \beta = 20^\circ \), the blockage has a value of 0.077. This means that the projected area of the foil is 7.7% of the cross sectional area of the measurement section.

Figure 12 shows the lift coefficient \( C_L \) and the lift/drag ratio \( C_L/C_D \) for this series - for open water (ow) and in the K27. The deviation in \( C_L \) between the runs for open water and in the K27 is increasing with increasing \( \text{AoA} \). This is most probably a consequence of the increasing blockage effect for increasing \( \text{AoA} \).

\[
\begin{align*}
\text{Table 2 – Blockage of the K27 for different leeway angles}
\end{align*}
\]

<table>
<thead>
<tr>
<th>Rake ( \beta )</th>
<th>2D AoA</th>
<th>Cross flow</th>
<th>( m = \frac{Ax}{A} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( ^\circ )</td>
<td>( ^\circ )</td>
<td>( ^\circ )</td>
<td>[-]</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>8</td>
<td>6.9</td>
<td>-4.0</td>
</tr>
<tr>
<td>0</td>
<td>20</td>
<td>17.5</td>
<td>-10.3</td>
</tr>
</tbody>
</table>

At small \( \text{AoA} \)’s only \( C_L \) increases as an effect of the blockage while \( C_D \) is nearly unaffected. This results in a noticeable increase of \( C_L/C_D \) for K27 compared to open water.

With larger \( \text{AoA} \)’s and hence a larger blockage the effect on \( C_D \) increases as well. Therefore \( C_L/C_D \) at larger \( \text{AoA} \)’s is similar for K27 and open water.

\[
\begin{align*}
\text{Figure 12 – Effect of blockage on } C_L \text{ and } C_D
\end{align*}
\]

Figure 13 shows the wave pattern in unrestricted open water (left) and in the K27 (right) for \( \beta=0^\circ \) (top row) and \( \beta=8^\circ \) (bottom row). It can be seen the blockage effect on wave making is significantly larger at higher \( \text{AoA} \)’s.

This indicates that the effect of blockage on \( C_D \) is mainly contributed to an effect on wave making resistance and induced resistance is not affected.

As this was not the focus here, it will be left to future work to investigate whether or not typical corrections for blockage in towing tank or wind tunnel experiments can be applied for this kind of investigation.
are closing in to each other and will cross each other if extrapolated to smaller values of rake and therefore $C_L$. This indicates that there is a limit to increasing leeway if this is traded off by too large a decrease in rake.

Figure 15 shows the lift coefficient over the AoA compared to the 2D lift characteristic of the used section. As expected for a low aspect ratio foil ($\Lambda=2.645$), the gradient of the 3D curve is significantly smaller and the stall angle is significantly higher than those of the 2D curve [5]. Furthermore the gradient is not linear for small AoA in the 3D case. These effects are the result of the induced drag due to the tip vortex, which does not exist in the 2D case and whose influence increases with decreasing aspect ratio.

**Ventilation and cavitation inception**

For almost all AoA’s from 10° onwards (all rake variations for leeway angles $\beta \geq 12^\circ$) regions were found near the leading edge and the tip where the absolute pressure value drops below the vapor pressure of water. This is a strong indication that for those cases cavitation is likely to occur during the experimental investigations.

However, no clear indication for the onset of ventilation was found.

Ventilation inception requires three relevant pre-requisites to exist: (1) a region of sub-atmospheric pressure, (2) a path of low impedance to air flow from the free surface into this region (e.g. separated flow) and (3) a disturbance in the free surface [6][7].

Regions of sub-atmospheric pressure are present due to flow acceleration near the position of maximum thickness of the profile and along the span. But only at leeway angles of $\beta > 20^\circ$, significant amounts of flow separation were observed, fulfilling the second prerequisite. The location of these regions was close to the tip rather than to the free surface, hence not providing for the third prerequisite.

Additionally mesh size and resolution were chosen to balance accuracy and computational effort. Although a comparatively large mesh was applied, the resolution was certainly not fine enough to capture effects as e.g. the
formation of free surface filament vortices which could initiate ventilation [6].

K27 RESULTS

The experiments have been conducted following the setup described above.

Ventilation inception

While it was the intention to investigate the same rake and leeway combinations at the same speed as in CFD, it was found, that ventilation at \( v=10 \) m/s flow speed occurred already at \( \text{AoA}'s \) of approximately 3° (or at leeway angles of 4°). Only very few measurement points are therefore available at this speed. In order to avoid ventilation and to determine the lift and drag characteristic of the foil for varying rake and leeway angles smaller flow speeds of 5m/s and 7.5m/s have been chosen for additional measurements.

Figure 16 shows the ventilation on the foil and the size of the ventilated cavity. The ventilation was very stable in position and size, but for a given \( \text{AoA} \) it was not reproducible at which flow speed ventilation occurred, when accelerating to the target flow speed. However, a clear dependency on the amount of acceleration was found and for very careful acceleration ventilation did sometimes not occur at all.

![Figure 16 - Ventilated foil in K27](image)

The early inception of ventilation in the K27 was not predicted by CFD but seems to be a result of the quality of the free surface in the K27. For higher speeds the free surface became very blurred showing a layer of white water with a thickness of up to 60mm (cf. Figure 16). This provided a significant disturbance to the free surface (3rd prerequisite for ventilation inception) as well as local separation close to the free surface (2nd prerequisite).

Three different regimes can be distinguished with respect to ventilation: fully attached or fully wetted (FW), instable or partially ventilated (PV) and fully ventilated (FV) [7],[8].

As indicated by [7] the FV regime is marked by the stall angle, which is reached at about 20° AoA due to the small aspect ratio. Hence the PV or instable regime applies for most of the measured conditions, which explains why ventilation inception was experienced rather random. The dependency to flow acceleration must be contributed to dynamic interactions of the flow and the tank geometry leading to larger perturbations.

Ventilation can occur in a process of self-stabilization in which the required flow separation does not need to exist initially over the entire surface but is induced by the ventilated cavity itself during ventilation inception [7]. This may be the primary inception mechanism as observed in the K27. Figure 17 shows a typical transition process from FW to FV. Clearly ventilation starts at the free surface and stabilizes in a cavity which covers the entire foil.

![Figure 17 – Ventilation inception (sequence in transition from FW to FV regime)](image)

**Polars**

Eventually model tests have been performed for speeds of 5m/s, 7.5m/s and 10m/s. Figure 18 shows the lift coefficient over the \( \text{AoA} \) for all measured points. For a leeway of \( \beta = -2° \) (L-2) and \( \beta = 4° \) (L+4) the whole range of rakes could be measured for the speeds of 5m/s and 7.5m/s. The red circles mark those rake and leeway combinations which could be measured at all three speeds including 10m/s.

It is found that the lift coefficient decreases with increasing speed. This effect is larger for small \( \text{AoA} \) and decreases for larger \( \text{AoA}'s \). Additionally the zero lift angle becomes larger with increasing speed.

The change in wave elevation due to the increased speed might explain these effects. The higher the flow speed is, the lower is the pressure on the suction side and the deeper is also the resulting wave trough. This in return decreases the wetted surface and reduces the available area for lift generation.
Corrections of the experimental data

Two effects have been observed during the tests which affect the results of the measurements and need to be corrected for a sensible comparison with the CFD results. These are:

- A reduction in wetted surface due to a blurred interface between air and water
- A change in AoA due to flexibility of the foil and the mounting frame.

As Figure 16 shows there is a noticeable layer of white water at the free surface. In order to correct the area in the calculation of $C_L$ and $C_D$ (Equations 3 and 4), additional measurements have been carried out at zero rake and leeway and at 10m/s: one with increased draft of 350mm and one with a decreased draft of 250mm. The result of the 250mm draft run compared well with the corresponding CFD results (at 300mm draft). Therefore all $C_L$ and $C_D$ values determined from the K27 measurements are calculated with the reduced projected area corresponding to the 250mm draft.

During the CFD simulations the structure of the foil was assumed to be infinitely stiff. However, the real foil showed some significant amount of flexibility. For the highest lift conditions (maximum AoA without ventilation) a sideward deflection of the foil tip of up to 60mm was estimated. In order to correct for this flexibility an offset depending on $C_L$ was applied to the cant angle which was used to calculate the 2D-AoA of a section and the cross-flow-angle.

As Figure 16 shows there is a noticeable layer of white water at the free surface. In order to correct the area in the calculation of $C_L$ and $C_D$ (Equations 3 and 4), additional measurements have been carried out at zero rake and leeway and at 10m/s: one with increased draft of 350mm and one with a decreased draft of 250mm. The result of the 250mm draft run compared well with the corresponding CFD results (at 300mm draft). Therefore all $C_L$ and $C_D$ values determined from the K27 measurements are calculated with the reduced projected area corresponding to the 250mm draft.

During the CFD simulations the structure of the foil was assumed to be infinitely stiff. However, the real foil showed some significant amount of flexibility. For the highest lift conditions (maximum AoA without ventilation) a sideward deflection of the foil tip of up to 60mm was estimated. In order to correct for this flexibility an offset depending on $C_L$ was applied to the cant angle which was used to calculate the 2D-AoA of a section and the cross-flow-angle.

Polars

The full range of rake angles could be measured for only two leeway angles ($\beta = -2^\circ$ and $\beta = 0^\circ$) in the K27 cavitation tank at a speed of 10m/s. These two are compared to the CFD results after application of the above corrections.

Figure 19 shows the lift coefficient over the 2D-AoA comparing both, the EFD and CFD results. The overall values match reasonably well for higher AoA’s with a deviation of 5% or less. For AoAs smaller than 1° (and respective small values of $C_L$) however the difference increases up to 44%. Additionally the EFD polars also show a slightly steeper slope than those of the CFD.

Unfortunately these effects could not be evaluated for higher angles of attack, but it is likely this trend will persist.

While the correction of the wetted area results in a vertical shift of the EFD polars (increasing $C_L$) and a decrease of the slope, the correction of the cant angle affects the slope of the polars only. Since the deflection of the foil was not accurately measured there is some uncertainty in the amount of correction necessary. However, values which would allow for a better agreement between the EFD and CFD results are unreasonably high. Hence other effects may exist which are not covered by the above corrections.

Figure 20 shows the comparison of the drag coefficient plotted over the AoA for the EFD and CFD results. A large difference can be observed between the EFD and CFD results, especially for AoA’s larger than -1° (for instance 25% at 2°). The data sets for the two CFD cases and for the two EFD cases are close together. Therefore another systematic error is suspected to be present either in the experiment or in the CFD simulation.
CONCLUSION

The comparison of the results derived from experimental investigations and RANS CFD computations for an A-Class catamaran hydrofoil show a good agreement for $C_L$ for relevant AoA’s but significant deviations in $C_D$. The latter may be a result of the assumption of fully turbulent flow in the CFD calculations or some difference in the respective setups of calculation and experiment. The section in use is a so called laminar profile intended to decrease frictional drag by shifting the laminar-turbulent transition of the boundary layer aft and thus providing a significant amount of laminar flow along the section. A laminar-turbulent transition model was not available in Fine/Marine 3.1-3. However, future investigations should investigate this effect.

The experiments revealed a significant amount of flexibility in the section of the hydrofoil. Future experiments should either be conducted on a sufficiently stiff test object or provide measurements of the foil deflection. For numerical investigations, Fluid-Structure-Interaction provides means to capture flexibility of the foils.

No indication was found from the CFD simulations for the early ventilation inception observed in the K27 cavitation tunnel. To capture the flow in the necessary detail to allow for ventilation, substantially finer and hence larger meshes are needed [7][9]. This is computationally expensive and unattractive if looking at series of runs for different operational conditions. As an alternative, adaptive grid refinement [9] may provide means to balance effort for future investigations.

The free surface in the K27 was highly disturbed and is likely to have caused the early ventilation inception. This disturbance is not given in CFD. It would be worthwhile to investigate if a disturbance with a similar effect could be applied in CFD to induce ventilation inception – an adequate grid resolution provided.

Alternatively, it would be interesting to repeat the experiments in a towing tank with an undisturbed free surface. This would probably be closer to real life condition where a much smoother free surface can be expected than in the circulating tank and it is also likely to delay ventilation inception to larger AoA’s.

However, this might be difficult since towing tank carriages capable of reaching 10m/s speed are similarly rare as cavitation tanks with the possibility to investigate a free surface.

REFERENCES


