Super-Cavitating Profiles for Ultra High Speed Hydrofoils: a Hybrid CFD Design Approach

Stefano Brizzolara¹ and Alessandro Federici²

Marine CFD Group, Genova - Italy
¹ University of Genova, Italy, Department of Naval Architecture, Head of Marine CFD Group
² University Pole for Yacht Design of La Spezia, Italy

ABSTRACT:
The paper presents the main results obtained from a systematic assessment about the current possibility to simulate hydrodynamic characteristics of super-cavitating hydrofoils with state of the art CFD methods. First a systematic validation of the finite volume RANSE solver with volume of fluid method for the multiphase flow and a simple Rayleigh-Plesset model for bubble dynamics. Sensitivity of the solver to various parameters which affect the cavitation model is verified as well as the integration time-step, the number of inner iterations, which influence the unsteady calculations. The final best configurations of the CFD model are used for its validation against experimental results on a reference two terms super-cavitating profile at a typical design angle of attack and for the complete range of cavitation indexes. After this preliminary study, the paper continues with the verification of the performance in case of different super-cavitating hydrofoils with two/three and five terms face shapes, designed using a classical asymptotic theory for the face and the back shapes chosen on the basis of cavity shapes predicted RANSE simulations. The RANSE method proves to be effective to design the hydrofoil back side for finite cavitation numbers and to assess its performance of for the next 3D hydrofoil design.

1 NOMENCLATURE

\begin{align*}
B_{OA} & \quad \text{vessel breadth over all} \quad \text{[m]} \\
\sigma_{\text{i}}, \sigma_{\text{o}} & \quad 2(p_{\text{i}}-p_{\text{o}})/(\rho V^2), \text{inflow speed cavitation number} \\
\rho & \quad \text{water mass density} \quad \text{[kg m}^{-3}\text{]} \\
p_0 & \quad \text{hydrostatic (ambient) pressure at foil depth} \quad \text{[Pa]} \\
p_v & \quad \text{vapour tension at given water temperature} \quad \text{[Pa]} \\
V & \quad \text{flow speed} \quad \text{[m/s]} \\
\text{VoF} & \quad \text{volume of fluid} \\
C_{L,D} & \quad \text{Lift or Drag Coefficient} \quad 2(L, D)/(\rho c V^2) \\
c & \quad \text{hydrofoil chord length} \quad \text{[m]} 
\end{align*}

2 INTRODUCTION

Results presented in the paper are those obtained in the preliminary phase of more ample research project dedicated to the concept design and hydrodynamic optimization of a super-high speed autonomous surface vehicle developed by the Marine CFD Group of the University of Genoa in a research program sponsored by the Office of Naval Research Global. As from general specifications, the unmanned vehicle is meant to reach a top speed of 120 knots in sea state two, but is also requested to loiter at slow speed for days. To this scope, an unconventional SWATH hull has been designed for loitering mode and optimized for minimum resistance at take-off speed, on the basis of the previous experience in the design of unconventional SWATH vessels (S. Brizzolara, 2004) (Brizzolara & Bruzzone, 2007) also in case of unmanned vehicles (Brizzolara et al. 2011). For the operations at super high speed, the unconventional displacement hull can be converted into a hybrid hydrofoil-wig craft by means of two pairs of retractable super-cavitating hydrofoils. The initial concept design of the vessel is presented in the two picture of Figure 1, which shows 3D renderings of the two operating modes: hullborne and foilborne.

The design of the surface piercing hydrofoils turns to be one of the most challenging design topics of the project connected with the ultra-high speed
requirement. It is well known, in fact, that cavitation becomes almost unavoidable at speed greater than 50/60 knots, above which super-cavitating hydrofoil shapes have to be used. The major hydrofoil studies and scientific advancements were done between the ‘60s and the ‘80s and indeed few example of hydrofoils vessels with top speed higher than 60 knots can be found from those years, as for instance super-cavitating submerged hydrofoils TAP-1 or TAP-2 designed by Boeing (Baker E., 1975).

The major hydrofoil studies and scientific advancements were done between the ‘60s and the ‘80s and indeed few example of hydrofoils vessels with top speed higher than 60 knots can be found from those years, as for instance super-cavitating submerged hydrofoils TAP-1 or TAP-2 designed by Boeing (Baker E., 1975).

The major hydrofoil studies and scientific advancements were done between the ‘60s and the ‘80s and indeed few example of hydrofoils vessels with top speed higher than 60 knots can be found from those years, as for instance super-cavitating submerged hydrofoils TAP-1 or TAP-2 designed by Boeing (Baker E., 1975).

The major hydrofoil studies and scientific advancements were done between the ‘60s and the ‘80s and indeed few example of hydrofoils vessels with top speed higher than 60 knots can be found from those years, as for instance super-cavitating submerged hydrofoils TAP-1 or TAP-2 designed by Boeing (Baker E., 1975).

The major hydrofoil studies and scientific advancements were done between the ‘60s and the ‘80s and indeed few example of hydrofoils vessels with top speed higher than 60 knots can be found from those years, as for instance super-cavitating submerged hydrofoils TAP-1 or TAP-2 designed by Boeing (Baker E., 1975).

The major hydrofoil studies and scientific advancements were done between the ‘60s and the ‘80s and indeed few example of hydrofoils vessels with top speed higher than 60 knots can be found from those years, as for instance super-cavitating submerged hydrofoils TAP-1 or TAP-2 designed by Boeing (Baker E., 1975).

The major hydrofoil studies and scientific advancements were done between the ‘60s and the ‘80s and indeed few example of hydrofoils vessels with top speed higher than 60 knots can be found from those years, as for instance super-cavitating submerged hydrofoils TAP-1 or TAP-2 designed by Boeing (Baker E., 1975).

The major hydrofoil studies and scientific advancements were done between the ‘60s and the ‘80s and indeed few example of hydrofoils vessels with top speed higher than 60 knots can be found from those years, as for instance super-cavitating submerged hydrofoils TAP-1 or TAP-2 designed by Boeing (Baker E., 1975).

The design methods for super-cavitating hydrofoils are still nowadays primarily based on the linearised asymptotic theories developed in the mid of last century by Tulin and Burkart (1955), generalized to the design of 3D hydrofoils near the free surface by Virgil Johson (1957). This theory is valid only in the limit of zero cavitation number and is able to give the shape of only the face of the profile (since the back is supposed to lie inside the supercavity, by definition). Actual profiles for the previously mentioned super-high speed vehicle have to be optimized for a finite cavitation number ($\sigma_0=0.05$ at 120 knots) and their back face need to lie inside the cavity while ensuring a sufficient thickness for strength issues.

In this respect, the authors have decided to explore a mixed design approach: the linearised theory to design the face and a CFD approach to verify the hydrodynamic characteristics and design the back side with a trial and error procedure. Hence the topic of RANSE solvers of multiphase cavitating flow.

The solution of the flow field around super-cavitating hydrofoils is rather complex because of the critical and unstable nature of cavitation. Modern volume finite volume unsteady RANS solvers promise to be a valid tool to study this complex phenomenon, but the theoretical models need to be set up correctly for what regards the parameters that control the physical models and numerical approximation schemes.

After a concise description of the theoretical and physical models used by the numerical method, the paper will continue with the description of its validation on a set of experimental results for a 2D super-cavitating foil in section 4 and finally will illustrate the design method followed to design the basis 2D section of the surface piercing super-cavitating hydrofoils of the autonomous vessel.

Figure 1 – Initial concept (patent pending) of the hybrid Hydrofoil-SWATH-WIG in foilborne (up) and hullborne (bottom) operating modes

### 3 MULTIPHASE VISCOUS FLOW SOLVER

#### 3.1 RANSE solver with multiphase VoF model

A state of art of RANSE solver with VoF method for representing the free surface has been selected. The software suite (CD-Adapco, 2009) has the capability of solving model or full scale turbulent flows around a body in non-stationary conditions, with the VoF method for predicting the free surface around it, with a simplified adiabatic cavitation model, described in the next paragraph.

The solver is applied to the following group of equations which express the mass and momentum balance with an Eulerian approach and Reynolds time-Average approach with the needed boundary conditions valid for this type of CFD simulation. The RANS equations can be expressed, in our case, for an incompressible flow as shown in Eq. 2:

\[
\begin{align*}
\nabla \cdot \mathbf{V} &= 0 \\
\rho \mathbf{V} &= -\nabla P + \mu \Delta \mathbf{V} + \mathbf{V} \cdot \mathbf{T}_\text{Re} + \mathbf{S}_M
\end{align*}
\]

where $\mathbf{V}$ is the Reynolds averaged flow velocity vector, $P$ is the average pressure field, $\mu$ is the dynamic viscosity, $\mathbf{T}_\text{Re}$ is the tensor of Reynolds stresses and $\mathbf{S}_M$ is the vector of momentum sources. The component of $\mathbf{T}_\text{Re}$ is computed in agreement with the standard k-\varepsilon turbulence model selected for this application, in agreement with the Boussinesq hypothesis.
\[ e_{ij}^{Re} = \mu_t \left( \frac{\partial V_i}{\partial x_j} + \frac{\partial V_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij} = 2 \mu_t D_{ij} - \frac{2}{3} \rho k \delta_{ij} \] (2)

Where \( \mu_t \) is the turbulent viscosity, \( k \) is the turbulent kinetic energy. The realizable k-\( \varepsilon \) turbulence model was selected to close the hydrodynamic problem. To save cells close to the hull surface, an analytical wall function has been adopted to the velocity vector and all other scalar quantities are extrapolated from the known quantities on the wall boundary surface.

As regards the wall function, on the cell closest to the profile, a two layer model approach has been applied. The two layer wall function model is a model that imposes a first thin laminar layer near the wall and a second logarithmic layer over the first; this model assumes that the centroid of the first cell near the wall lies within the logarithmic region of the boundary layer. The wall treatment is optimized to approximate boundary layer effects with a dimensionless wall distance \( \gamma^+ < 100 \), but a much smaller value has been used in this study in order to accurately solve the cavity thickness growth along the wall.

The RANS solver is based on a Finite-Volume method to discretize the physical domain. The equation for an incompressible multiphase fluid is used in the simulation, with one more transport equation for the VoF, shown in Eq. 3, which represents the fraction of water inside each cell:

\[ \frac{\partial \text{VoF}}{\partial t} + \nabla (\text{VoF} \cdot \mathbf{U}) = 0 \] (3)

This equation ensures to find the correct shape of the free surface between water and air, this method is powerful for solving problems when the wave breaking may occurred, or when the air effects are important on the free surface shape.

To solve the time-marching equations, an implicit unsteady solver is used.

All hydrodynamic unknown quantities in the field are solved at each time step using an iterative method. In particular, many inner iterative steps are needed to reach a good prediction of the unsteady cavity evolution at each time step. The software uses a SIMPLE method to conjugate pressure field and velocity field, and a AMG (Algebraic Multi-Grid) solver to accelerate the convergence of the solution. A more complete description can be found in the technical reference manual (CD-Adapco, 2009).

### 3.2 Cavitation Model

The Cavitation model is based on the flow mixture concept and the Rayleigh-Plessset simple model to simulate bubbles dynamics. With this simplified model one can define the cavitation inception and bubble dynamics by setting the size and number of micro-bubble seeds (cavitation nuclei), and their stochastic spectral distribution in the considered. These are approximately represented in the RANSE solver through an average seed radius \( R_0 \) and an average seed density \( n_0 \). The latter is a constant strongly dependent on water quality and it is defined as the number of cavitation seeds (nuclei) per unit volume of liquid (Sauer 2000).

To simplify the problem, the vapor bubbles in a control volume are represented in an average sense by an homogeneous distribution of seeds with the same mean radius. This assumption allows to describe the bubble distribution by a single scalar field, the vapor volume fraction \( \alpha_v \), which influences directly the VoF variable solved in (3).

Assuming that only one liquid phase and the corresponding liquid-vapor phase can occupy the control volume where cavitation takes place:

\[ \alpha_v = \frac{\text{Vol}_v}{\text{Vol}_l + \text{Vol}_v} = \frac{n_o \frac{4}{3} \pi R^3}{1 + n_o \frac{4}{3} \pi R^3} \] (4)

where \( \text{Vol}_v \) is the volume occupied by the vapor, \( \text{Vol}_l \) the volume occupied by the liquid and \( N_{\text{bub}} \) the number of vapor bubbles in the control volume. This vapor volume fraction changes, as all phases, due to convective transport but also to bubble growing and collapsing.

The modeling of bubble grow rate is based on a Lagrangian observation of a cloud of bubbles, using the Rayleigh-Plasset equation to describe the average bubble radius time evolution \( R(t) \):

\[ R \frac{d^2 R}{dt^2} + 3 \left( \frac{dR}{dt} \right)^2 = \frac{p_i - p_0}{\rho_l} - \frac{2\gamma}{\rho_l R} - 4 \frac{\mu_t}{\rho_l R} \frac{dR}{dt} \] (5)

where \( p_i \) is the saturation pressure corresponding to the temperature at the bubble surface and \( \rho_l \), \( \mu_t \) are the water density and viscosity respectively, and \( \gamma \) is the surface tension coefficient.

The model is simplified and the final equation for the bubble growth, obtained neglecting the inertia, viscous and surface tension terms in (5) is given by the following expression:

\[ \left( \frac{dR}{dt} \right)_{\text{Ray}} = \frac{2}{3} \frac{p_i - p_0}{\rho_l} \] (6)
4 ACCURACY OF THE RANSE MODEL

Different numerical models have been generated to be used with the RANSE solver described in the previous section, by systematically varying geometrical mesh parameters as well as cavitation model constants, to be validated against the experimental results on a super-cavitating hydrofoil of Waid & Lindberg (1957).

The systematic validation study has been performed for different angles of attack of the profile (6, 4 and 2 degrees), for each of them exploring a quite wide cavitation index range, by keeping constant the flow speed and opportunely changing the ambient pressure. Only the results of the 6 degree are reported in this article, as most representative of a typical design conditions.

The general approach, derived from experience in order to cope with particular instability issues of the unsteady cavitation model, is first to converge on the steady non-cavitating viscous flow field around the airfoil and then to gradually increase the vapor pressure to its correct value from an initial (virtual) very negative one. This procedure is necessary to avoid the cavitation model to be applied on the initial unrealistic negative pressures field which rise for the sudden acceleration of the flow around the foil in the first instants, during the solution of the virtual transient to converge on the steady flow field; these unrealistic negative pressure values, which gradually disappear until convergence on the steady non-cavitating flow field around the foil, would cause the unrealistic formation of large vapor bubbles around the foil and would lead the solution to divergence.

4.1 Physical model set up

The set-up of the parameters which rule the physical model has been also systematically investigated.

First, as perhaps obvious, for the inherent unsteady nature of cavitation an unsteady solution must be searched for in order to find a feasible solution, although the final global parameters compared with the experimental tests are the mean forces components. Also the integration time-step of the solver was studied: a constant value of $1 \times 10^{-3}$ s was assumed as the minimum for stability and accurate results. In fact, higher time-step values brought to inaccurate results, while smaller values made the convergence property of the solution very weak being too sensible to the turbulent fluctuation of the flow around the foil.

As regards parameters of the cavitation model, in a particular case for $\sigma_0=0.48$ (i.e. the region where the highest differences between experimental and numerical results are registered) it was verified the effect of a variation in the seed diameter of the cavitation nuclei (chapter 3.2) from 1e-6 to 1e-4 meter; this test is carried out with the mesh type Q2 (see chapter 4.2). The results are quite interesting: as the seed diameter increases, the lift coefficient $C_L$ decreases coming closer to the experimental value, on the other hand $C_D$ instead diverges. A good compromise seems to lay around $5 \times 10^{-5}$ m, in fact the accuracy on $C_L$ increases of about 4% while that on $C_D$ is worse of about 2%. Also the ratio $L/D$ at this point is quite in line with the tunnel tests results.

Not being convinced about the obtained trends and being uncertain of the tunnel tests at this particular cavitation index and about the effect of this parameter for all the other $\sigma_0$, the more typical seed diameter value of $5 \times 10^{-5}$ m was assumed, eventually privileging a smaller error on drag than that on lift. On the other hand, specific measurement of the cavitation nuclei size statistical distribution were not made in the reference tests.

Other sensibility tests varying the seed number $N_{hub}$ were performed, but this parameter does not appreciably influence the results, so the standard value $10^{12}$ nuclei per cubic meter of water was assumed for all the simulations.

4.2 Mesh sensitivity – First Systematic Analysis

As a result of this systematic analysis a domain spanning of 3 chord lengths upstream of the foil leading edge, 15 chord lengths aft of the trailing edge and 5 chord lengths from the face and back sides of the foil itself has been chosen, as illustrated in Figure 2. This with the scope of eliminating the influence of the boundaries (especially the inlet and top and bottom slipping walls) on the solution. Since results reported from the tests in the water tunnel on a small model (3x3 inches) were not corrected blockage and upper/lower wall effects, some
uncertainty remains when comparing the forces which needs to be discussed case by case.

To find the correct mesh refinement, two different sensitivity analyses were performed: the first with a closer inlet, about 1 chord lengths upstream of the foil (usually sufficient for standard profiles), the second with 3 chord lengths; two different refinements have been applied to both mesh sizes.

Figure 2 – Fluid domain, final sizes (mesh Q2+I)

Figure 3 gives the idea about how large must be the refinement around the foil in order to correctly capture the shape of the cavity. In fact, a thick prism layer of very thin cells is needed over the suction side of the hydrofoil.

Figure 3 - Mesh Q2+I around the foil

Figure 4 – Close up on prism layer mesh near the leading edge of the foil and results of the VoF of the liquid phase.

The close up of Figure 4 renders the quality of the solution of the vapor volume fraction in terms of free surface tracking capability along the prismatic cells generated on the suction side of the foil.

The thickness of the prism layer plays an important role: this must be greater than the cavitation bubble height in the condition that has to be solved. Figure 5 shows the difference between two different prism layer sizes at the same condition, the difference in results are quite important: not only we have a reduction of the cavity length but also a different unsteady behavior of the bubble (and induced forces) with time. This problem is rather complex and is further detailed in paragraph 4.3.

Figure 5 – Effect of two different prism layer thicknesses on the predicted cavity shape (same condition)

The development of the cavity as a function of the cavitation index is given in the pictures of Figure 21 in the Appendix. The effect of the mesh refinement is also important when the cavity overpass the foil length (super-cavitating conditions): a proper refinement behind the trailing edge is also needed to catch the correct cavity free surface and the unsteady turbulent flow in the cavity closure region. Even at the highest cavitation indexes ($\sigma_0=1.7\div1.3$), as soon as the partial cavity is forming and growing at the leading edge, it gradually (and significantly) affects the hydrodynamic predicted forces, since it alters considerably the distribution of pressure and flow field in the surrounding areas (see Figure 19 and 20). In fact, both lift and drag force components are influenced by the partial cavity development and as from Graph 3 and Graph 4. In these graphs four different numerical curves are plot corresponding to for four different meshes considered in the sensitivity analysis. The experimental values are taken from the tests of Waid & Lindberg (1957) on a 3x3 inches wing spanning the full width of the cavitation tunnel test section and without any
correction for blockage or top and bottom wall reflection effects.

The first mesh type, named Q1 corresponds to the lighter mesh with the lowest number of prism layer cells from the wall and a refinement region of one chord behind foil trailing edge. The second mesh type, Q2, is more refined but has the same extensions of mesh Q1.

As expected, the main differences between the global forces predicted in both cases are around $\sigma_0 = 0.5$, where the cavity is partially interesting the profile and particularly unsteady. Indeed, it is troublesome to solve the flow in these conditions: due to the inherent instability of the phenomenon, in fact, the bubble grows up, bursts and regenerates itself continuously and this high frequency behavior is not simple to follow numerically. More details will be given in the next paragraph.

At lowest cavitation indexes, i.e. in supercavitating conditions ($\sigma_0 < 0.45$ at the considered angle of attack of 6 degree), the predicted forces are well following the trends of the experimental measurements, still showing considerable errors on experimental $C_L$ (about $+15\%$) and on $C_D$ (around $22\%$).

### 4.3 Mesh Sensitivity - Second Systematic Analysis

After assessing the influence of the mesh refinement from first sensitivity analysis (in particular on the pressure field around the foil) a larger domain extension was tested since some influence of the boundary conditions on the local solution around the foil was noted. The new domain size corresponds to the extensions quoted in Figure 2.

So also in this second sensitivity analysis the two different mesh resolutions were investigated but with a larger domain size: the new codes for these two meshes are Q1+1 (larger size but lower refinement) and Q2+1 (larger size and higher refinement). The forces obtained in these cases are represented in Graph 3 and Graph 4, with orange and red lines, respectively.

As clear the difference in lift and drag coefficient in non cavitating conditions ($\sigma_0 = 2.95$) has been solved. The shift of the inlet from one to three chord lengths upstream of the profile is the main responsible of the good correlation with the experimental lift force, regardless of the mesh refinement. For the drag, instead, also the mesh refinement influences the predicted force and strangely the coarser mesh is closer to the experimental results in non-cavitating or partially cavitating conditions than the finest one.

It is to be noted that also experimental measurements show some unexpected result: in fact for $\sigma_0 \geq 1.5$ when there is still no cavitation on the foil, the measured forces should be constant, while in fact they have a fluctuation both on lift and drag components. Perhaps the order of magnitude of the measurement precision is in this range of these fluctuations. Focusing the attention to the fully cavitating condition ($\sigma_0 < 0.6$), the best results are achieved with the Q2+1 mesh type, both for $C_L$ and $C_D$. The average error in this regime being an overestimation of about $8\%$ on $C_L$ and $12\%$ on $C_D$. As anticipated in the previous paragraph, the problematic zone is between $\sigma_0 = 0.5$ and $\sigma_0 = 1.0$; A deeper analysis of the problem revealed that in these cavitating conditions the cavity forming on the foil suction side is not stable, but it is growing and collapsing at a certain frequency.

![Graph 2 - $C_L$ Time history for two different mesh refinements](image)

Graph 2 presents the time histories of the lift coefficient predicted with two different mesh refinements at $\sigma_Y = 0.744$. The minimum cell size condition the frequency with which the lift force signal oscillates (and so the cavity on the back). With a low refinement, after an initial period of time in which the lift is growing similarly to the more refined mesh, we note a certain quiescent period in which the force remain almost constant (3.2s to 5.0s). This phenomenon is differently predicted with the higher resolution mesh, for which the cavity is always contained within the prism layer height during its growth. The finer resolution mesh is able to capture a periodical growth and collapse of the cavity (so the lift) on the hydrofoil suction side, being more adherent to reality (Brennen, 1995). This different behavior with time obviously influence also the mean value of the force when averaged over a period of about 5 seconds, which is the value compared with the experiments. So the apparently better correlation of $C_L$ with the Q1 mesh is due to this unphysically predicted lift time history and not to a real improvement in the model accuracy.

The high frequency oscillations of the partial cavitation stops at lower cavitation numbers, as soon as the cavitation interests the complete length of the profile.
Graph 3 - Lift Coefficient sensitivity to different mesh types and resolution

Graph 4 - Drag Coefficient sensitivity to different mesh types and resolution
A similar behavior is predicted for the drag coefficient (Graph 5), which can be subject to the same error in averaging the unsteady signal. The peaks and through of lift are synchronized with those of drag, so the cavity affects the two force components in the same way.

Graph 5 - $C_D$ time history for two different mesh refinements

Additional Figures of the results obtained from the systematic CFD calculations on the mesh Q2+I are reported in the Appendix, where Figure 19 presents streamlines and pressure field around the foil and Figure 20 the flow velocity magnitude. From these figures it clearly distinguishable the separated zone aft of the blunt trailing edge which are interested by recirculating flow in wetted as well as cavitating conditions. Also it is possible to note the constant pressure inside the vapor cavity and behind the blunt trailing edge of the hydrofoils.

5 DESIGN & OPTIMIZATION OF A NEW FOIL

After this initial investigation devoted to the tune and set-up of the CFD model, the study continued with the design and optimization of the 2D hydrofoil to be used as a base section for the 3D supercavitating hydrofoils which sustain the vessel at high speed.

The approach followed was a hybrid approach: first the face of the supercavitating hydrofoil was designed with traditional linear asymptotic theories (V. Johnson, 1957) with the desired $C_L$ value. In fact, in fully cavitating condition, only the face carry out the lift, because the back and the trailing edge are normally covered by the cavity vapor bubble.

The theory developed by Johnson (1957) is based on the conformal mapping method of Wu, as linearised developed by Tulin and Burkart (1955), generalizing the series expansion which defines the vorticity distribution along the chord of the profile (in the imaginary plane) to higher order and so opening the possibility to obtain higher efficiencies with respect to standard circular arc or Tulin-Burkart profiles, but at the cost of more complex line shapes.

As presented in Graph 6, in fact, the three and five terms shapes have a higher maximum camber that is moved further aft with respect to the corresponding two terms shape and even an initial negative camber more pronounced in the five terms shape.

Graph 6 – Face shapes obtained for different super-cavitating profiles shapes at the given design $C_L=0.25$.

As a result of the conformal mapping techniques extended with higher order terms the maximum theoretical efficiency which can be obtained in ideal conditions (zero angle of attack) for the two terms profile (also called Tulin-Burkart) is:

$$\frac{L}{D} = \frac{25}{4} \frac{\pi}{2C_L}$$

(7)

while the Johnson three terms profiles have 1.44 times higher efficiency than that (7) of the Tulin-Burkart shapes and the five terms profile can reach 1.78 higher efficiency at the same lift coefficient.

The cited design method developed by Johnson extends even further these asymptotic methods valid for 2D sections which are valid for infinite water depth, by considering also free surface effects, and the three dimensional effects caused by the finite aspect ratio of a 3D hydrofoil and that due to the the cross flow.

Based on the cited theory, three different 2D face shapes have been designed with a 2D-$C_L=0.42$, corresponding to a 3D-$C_L=0.25$ for the surface piercing foil, according the approximate corrections. The design angle of attack of 5 degree (higher than the ideal one) was selected in order to have a sufficient cavity thickness at the design operating point. After having defined the face shape, also the shape of the back have been designed with the aid of RANSE simulations by a trial and error method, better described herein after. Moreover RANSE simulations allow also to verify the order of approximation of the asymptotic theory in evaluating the theoretical lift coefficient for $\sigma_0=0$,
instead of the actual design cavitation number \( \sigma_0 = 0.05 \) (at the top speed of 120 knots). At finite cavitation indexes, the real cavity thickness is important and the shape of the back must be designed in order to obtain the maximum profile thickness at each chord station (to increase the section area modulus, for strength reasons) while ensuring that the back line is still laying inside the vapor cavity. From the comparison of the designed face camber lines (Graph 6), it was decided to discard the Johnson 5-terms line, because of its strong thickness reduction of the profile area close to the trailing edge, resulting in a rather poor sectional modulus of inertia. The largest inertia modulus for the profile is ensured by the Tulin-Burkhart camber line, so this type of face line was the first to be simulated. As in other conventional supercavitating profiles, the trailing edge was sharply truncated and with a trial and error procedure the thickness distribution along the chord was calibrated in order to obtain the maximum sectional area still remaining in supercavitating conditions, as from Figure 6.

With respect to the design 2D-\( C_L = 0.42 \) the obtained T-B (Tulin-Burkhart) profile presented in Figure 6 (which the vapor phase is represented in blue and the liquid phase in red) is only \( C_L = 0.36 \) with a \( C_D = 0.035 \) and an efficiency of \( L/D = 10.3 \), too low with respect to the ideal one for our scope. It was then decided to go on with the Johnson Three Terms (J3T) camber line for the face, designed for the same bi-dimensional lift coefficient. The first designed J3T thickness distribution (v.1), as shown in Figure 7, was not a valid solution since the vapor cavity does not start from the leading edge of the profile and it is only formed at the maximum curvature point of the back. Consequently the lift predicted values are in good agreement with the design values. In fact RANSE simulations predict:

\[ v1: \quad C_L = 0.405 \quad ; \quad C_D = 0.040 \quad ; \quad L/D = 10.1 \]

(non optimal)

So a closer agreement of the lift value with the theoretical (design) one, but still with a poor efficiency similar to the T-B, for sure due to the partly wetted back surface, which increases the friction resistance. Following Figure 8 and 9 show the pressure and velocity distributions around the foil with the streamlines.
A new tentative design (v.2) was then performed as shown in Figure 10, in which the back line was modified to increase the camber starting from mid-chord in order to anticipate the cavity detachment point. Moreover the thickness at the leading edge was also increased to achieve a larger area in this region. Also in this case, the results are not satisfactory: the cavity actually detaches earlier on the back, but still is not created at the nose. Predicted hydrodynamic characteristics for the solution v.2 are:

\[ v2: \ C_L = 0.378 ; \ C_D = 0.044 ; \ L/D = 8.6. \]

Notice that not the \( C_D \) increases and \( C_L \) decreases with respect to the previous version v.1. This is because of the larger profile thickness at the leading edge not balanced by a pressure recovery at the trailing edge of the profile (which is inside the cavity). The increase of curvature from mid-chord station was not beneficial.

The fourth design tentative (v.4), shown in Figure 16, the back line was brought to the limit as regards local strength. Indeed with this line a thin cavity is obtained that spans the entire back of the profile, but the thickness of the profile in the first half chord is very low. Global results results:

\[ v4: \ C_L = 0.432 ; \ C_D = 0.035 ; \ L/D = 12.2, \]

resulting in a drastic reduction lift, simply explained by the change of curvature the initial part of the back.

The third design(v.3), shown in Figure 13, returned to use the back shape of v.1 with a further decrease of thickness on the back at the leading edge, in order to cause the cavity detachment; but this was not achieved, according RANSE calculations because the thickness of the foil is still too high at the nose and the local pressure does not go under the vapor tension.

Estimated hydrodynamic characteristics:

\[ v3: \ C_L = 0.365 ; \ C_D = 0.045 ; \ L/D = 8.1 \]

No appreciable separation is noted on the back of this profile, as from Figure 18, at the contrary of what can be noted for the other versions v.2 and v.3. in Figure 12 and 15 respectively.

Diagrams in Graph 7 shows the pressure distribution on the face and the back of v.1 in red, v.3 in green and v.4 in blue. The main difference is of course in the back distribution where the only one showing constant \( C_P = \sigma_0 \) is the v.4, while the other two tentative designs create a pressure recovery area on
the back, which in the case of v.3 is able to influence also the face pressure distribution close to the nose.

Graph 7 - $C_P$ distributions on the different J3T foil designs

6 CONCLUSIONS

Calibration and validation of a state of the art finite volume RANSE solver based on the volume of fluid method for the solution of the turbulent viscous flow with vapor/water mixture and a simplified Rayleigh-Plesset model to control the phase changes and fraction distribution in each cell have been presented and discussed in the paper. The best accuracy obtained from the RANSE simulation in terms of global force components is in the order of 10% against the experimental values, which is not particularly satisfactory, but still good for engineering purposes and also taking also into account the uncertainties levels of the experimental results.

Further investigations need to be done including the effect of surface tension (Weber number) and a even further mesh refinement level. Some newer reference experimental results would be also deemed in order to avoid uncertainties due blockage and symmetry effect which were not corrected in the available experimental results.

Finally the paper also presents and offers the results achieved by an hybrid method to effectively design 2D super-cavitating profiles at small finite cavitation numbers (such as $\sigma_0=0.05$ in our case). The approach is based on mixed use of the Johnson asymptotic linear theory to design the face line and a DOE (Design Of Experience) approach for the back shape using systematic RANSE flow simulations.

The application of this new design approach leads to very satisfactory results in terms of the lift and hydrodynamic efficiency (L/D) predicted by the RANSE calibrated model with respect to the best achievable ideal performance.

7 AKNOWLEDGEMENTS

The present work has been performed in the framework of the research grant N62909-10-1-7116 of the Office of Naval Research Global, dedicated to the “Hydrodynamic Design and Assessment by CFD Methods of Hybrid SWATH/Hydrofoil Hulls for a Super High Speed USV”.

8 REFERENCES

CD-Adapco (2009), Star-CCM+ User and Theory Manual, version 4.04.011
APPENDIX – RESULTS OF VALIDATION ON FOIL TESTED BY WAID & LINDBERG (1957)

Figure 19 – Pressure coefficients around the 2 terms supercavitating foil (Waid and Lindberg, 1957) predicted by the RANSE model at different cav. numbers and $\alpha=6$ degree.

Figure 20 – Velocity magnitude around the supercavitating foil, predicted by the RANSE model at different cav. numbers and $\alpha=6$ degree (inlet flow speed of 9.14 m/s).
Figure 21 – Predicted cavity shapes on the 2 terms supercavitating foil of Waid and Lindberg (1957) used in the validation study of the RANSE model.