Three Phases RANSE Calculations for Surface-Piercing Super-Cavitating Hydrofoils

Stefano Brizzolara Department of Naval Architecture, Marine and Electrical Engineering, Genova, Italy currently Visiting Peabody Associate Professor at MIT MechE, Cambridge, MA, USA <u>stebriz@mit.edu</u>

SUMMARY

A new Super-Cavitating Surface-Piercing hydrofoil has been designed as the main component of a Super High Speed SWATH (Small Waterplane Area Twin Hull) hybrid hydrofoil craft. The design speed of the vessel is 120 knots, corresponding to a cavitation index σ =0.05 for the hydrofoil.

A finite volume RANSE solver, based on the flow mixture concept and volume of fluid technique to deal with multi-phase flows and cavitation model, is tested to model the complex cavitation and ventilation mechanisms the interest the are validated hydrofoil. Numerical results against hydrodynamic lift force measured in a preliminary series of model tests performed in a depressurized cavitation tunnel with free surface flow on a 1:6 scale model. Limits and performance of the three phases flow mixture model are evidenced and discussed in the paper by comparison with the cavitationventilation phenomena observed in the water tunnel. Different numerical algorithms to solve the advection and dispersion of each fluid phase have been tested and their influence on the final solution has been proven to be important.

INTRODUCTION

A new a Super High Speed unmanned surface vehicle (Brizzolara, 2011) has been designed (Figure 1) for dual model operational profile: hullborne mode in which the vessel navigate as a SWATH (Small Waterplane Area Twin Hull) propelled by a diesel-electric propulsion system; and a foilborne mode in which the vessel, after deploying in water two pairs of surface piercing hydrofoils and igniting its airborne turbojet engines, reaches the speed of 120 knots as a supercavitating hydrofoil craft with a partial contribution of the wing shaped superstructure to the lift force.

An extensive aero-hydrodynamic study (Brizzolara, 2011) based on systematic CFD calculations, has been conducted to optimize the drag of the underwater SWATH hull forms at takeoff speed, resulting in the unconventional hull form and arrangement of struts shown in Figure 1 (Brizzolara et al. 2011) Diego Villa Department of Naval Architecture, Marine and Electrical Engineering, Genova, Italy Diego.Villa@unige.it

as well as to optimize the drag and lift contribution of the wing in air at top speed.



Figure 1 - 120 knots HYSWATH Autonomous Surface Vehicle with four super-cavitating hydrofoil in their deployed position

The Super-Cavitating (SC) Surface-Piercing (SP) hydrofoils, though, remain the most particular element of the vessel to be optimized. To design supercavitating sections, a hybrid CFD-theoretical design method has been developed (Brizzolara & Federici, 2011): it is based on the Johnson's asymptotical theory for drawing the face line of the foil and a trial and error method based on CFD simulations to find the shape of the back that is dully included inside the cavity, while ensuring the correct modulus of inertia to the foil.

When the hydrofoil operates intersecting the free surface, ventilation from the free surface, i.e. suction of air from the above the free surface, can occur and interact with the (super) cavitation.

Indeed, the mechanism of combined cavitation and ventilation on surface piercing supercavitating hydrofoils has not yet completely characterized at very high speed. While a number of studies, especially on physical models, were done

Proceedings of the 8th International Symposium on Cavitation CAV2012 – Paper No. 90 August 14-16, 2012, Singapore

during years 50's to 80's on fully submerged SC hydrofoils (Baker, 1975), few ones are known on surface piercing hydrofoils. Among this last category of hydrofoils few studies were done at such low cavitation numbers as those corresponding to speeds in excess of 100 knots. One of the few known examples was done by NASA, in occasion of a study on the hydrodynamic planing characteristics in water of a supersonic water-based airplane (McKann et al., 1962). In this occasion, a systematic series of tests were conducted on three different hydrofoils with simple supercavitating circular arc face profile, not really optimized for a design operational point.

Being this the state of the art in the field, it is clear that some more studies and investigations are needed to understand the hydrodynamic behavior of high speed supercavitating surface piercing hydrofoils, as the one to be used on the mentioned USV. The study on combined effects of cavitation and ventilation on surface piercing hydrofoils, moreover, is of interest also for other kind of fast vehicles: such as fast planing crafts and fast hydrofoils and in general marine sea based vehicles that are designed to operate at speeds higher than about 50/60 knots that need some form of.

The particular study presented here regards a preliminary investigation on the capabilities of a state of the art finite volume CFD solver in predicting the complex physics and the value of the hydrodynamic forces acting on the hydrofoil.

A theoretical background on the idealized physical models and of the numerical CFD solver used is given in the next two paragraphs. The physical scaled model used in the experiments and the virtual numerical model created for the CFD simulations is described in the following chapter.

THEORETICAL MODELS

The cavitation is an unsteady phenomenon, that involves a compressible fluids where its phase could change, in the domain, due to a changing of the pressure fields acting on its. So to simulate this kind of phenomenon should be used a solver that is able to take in to account: the compressibility of the two phase, the condensation and evaporation rate, the interface between the two phases and the kinematics of all these phenomena. For engineering application this detail level could be not so useful, so could be better take in to account only the principle effects. Additionally, for this particular problem, the free surface between the air and water should be taken into account. For that reasons should be take in to account three different fluids, where two of them can react together. The most straightforward and easy method to implement into a generic RANSE solver to obtain a solution to this complex problem is the multi-fluids continuum theory. Although especially air should be treated as a compressible fluid, a further simplification has been assumed by considering all the three fluids as incompressible (this allows to avoid to solve for the the acoustic scale of the phenomena witch in first approximation can be neglected). Moreover the fluid is considered isothermal and immiscible except for the water that is modeled with two reacting phase. The phase-change has been solved with a interface capturing method based on the VoF (Volume of Fluids) theory.

$$\nabla \cdot U = \frac{\dot{m}}{\rho_l}$$

$$\frac{D}{Dt} (\rho_{mix}U) - \nabla \cdot (\mu_{mix}\nabla U) - (\nabla U \cdot \nabla \mu_{mix}) = \nabla \cdot T^{Rn} - \nabla p$$

$$\frac{\partial \alpha_l}{\partial t} + \nabla \cdot (\alpha_l U) = \frac{1}{\rho_l} - \frac{1}{\rho_v} \dot{m}$$

$$\frac{\partial \alpha_a}{\partial t} + \nabla \cdot (\alpha_a U) = 0$$
(1)

The classical Navier-Stokes equations (1) for the fluid mixture are used together a modified version of the continuity equation to take in to account the rate of phase-change between the liquid and the vapor. These equations moreover have been solved together the two volume fractions equation witch govern the quantity of each phase inside a single cell. The variable α_l represents the concentration of liquid phase, α_a represents the air phase, and α_v represents the vapor phase. It should be noted that the vapor fraction does not have an explicit equation. This because the following conservation equation (2) is always valid:

$$\alpha_l + \alpha_v + \alpha_a = 1 \tag{2}$$

Moreover, for this kind of problem the correct evaluation of the phase-change rate (source and sink of vapor phase) is important. The solver used is based on the Schnerr-Sauer model, which starting from the well know Rayleigh-Plesset equation (3):

$$R_{B}\frac{d^{2}R_{B}}{dt^{2}} + \frac{3}{2}\frac{dR_{B}^{2}}{dt} + \frac{2\sigma}{R_{B}} = \frac{p_{v} - p}{\rho_{l}}$$
(3)

that describes the dynamic equilibrium of a spherical vapor bubble of a radius R_B subject to an external pressure p and a surface tension σ . This simple dynamic equation with the necessary simplifications can be converted in the final differential equation used in the RANSE solver to model the vapor phase production and condensation:

$$\dot{\mathbf{m}} = -3\rho_{v} \sqrt[3]{n_{0}\frac{4}{3}\pi(\alpha^{2} - \alpha^{3}(1 - \frac{\rho_{v}}{\rho_{l}}))} \cdot sign(p_{v} - p) \\ \cdot \sqrt{\frac{2}{3}\frac{|p_{v} - p|}{\rho_{l}}}$$
(4)

Where n_0 represents the initial mean diameter of a bubble inside the fluid.

NUMERICAL METHODS

To solve the previous mathematical model the StarCCM+ suite has been used. The solver is able to solve unsteady RANS equations of compressible and incompressible fluids with a finite volume method on unstructured polyhedral meshes.

The solver uses an implicit first order approach for the time derivative to guarantee the stability of the solution (at the cost of a smaller computational time step). Moreover, the transport equation for VoF is a solved with a second order scheme with a sharpening factor algorithm (Ferziger & Peric, 2002).

All the equations are solved with the segregated flow technique, and the coupling between the continuity equation and the momentum equation (pressure-velocity link) is solved through the well know SIMPLE algorithm, in conjunction with an AMG linear solver to speed-up the convergence of the solution during each iteration. Turbulence has been modeled with the realizable k- ε model, extrapolated at the solid boundaries of the domain (hydrofoil) with a two layer all y+ wall function.

The gravity acceleration has been imposed as a body force to solve for the free surface waves, and the problem has been solved in terms of absolute pressure, normalized with respect to that specified for an undisturbed point in the air at the free surface (reference value). Other specific details on the solver can be found in the User Manual (CD-Adapco, 2012).

PHYSICAL AND NUMERICAL MODEL

The supercavitating surface piercing hydrofoil presented in Figure 3 is designed on the basis of a new two dimension foil represented in Figure 2. This new supercavitating foil shape has been developed in order to achieve good hydrodynamic efficiencies, in terms of lift to drag ratio, the performance of the foil for a dual mode operational points; at full speed (120 knots), where the cavitation index

$$\sigma_0 = \sigma = (p_{atm} - p_v)/(1/2\rho V^2) \tag{5}$$

is very low (0.05) typical a sure supercavitating regime and a take-off speeds (18-25 knots), corresponding to cavitation indexes σ in excess of 1.1 at which the foil operates in a fully wetted regime.

In supercavitating regimes the new foil shape (Figure 2), composed of a main body (up to the sharp edges) and an annex (downstream of the two sharp edges) behaves like a very efficient SC profile while in fully wetted flows the new profile which is not truncated as conventional SC profiles maintains a very good lift force and a low drag to lift ratio (below 0.05), resembling more a conventional NACA profile. This dual mode operation is important to ensure a good efficiency at take-off.

For the design of the 3D hydrofoil a simple trapezoidal shape has been chosen for the part submerged at high speeds with a tip winglet to reduce the induced drag.



Figure 2: Water/vapour fraction distribution (red water, blue vapor) around the new SC profile, operating at design condition (α =5 deg, σ =0.05, Re=32·10⁶).

A variable distribution of angles of attack along the span (twist) was calculated according predictions obtained from a standard lifting line code, recently reformulated (Vernengo & Brizzolara, 2011) to correctly consider the vorticity created by supercavitating profiles, the proximity effect of the free surface, the effect of sweep angle and negative dihedral angle which were grossly approximated, instead, in the preliminary methods used to design this hydrofoil. The resulting geometrical angles of attack vary along the span from about 9 degree at the foil tip, at the root of the winglet, to about 3.5 degrees at upward end of the trapezoidal part of the foil, around an average value of about 5 degrees, which is the design angle of attack of the 2D profile.



Figure 3 – SC SP Hydrofoil Model Planview Shape and distribution of chord length and angle of attack along the span

A 1:6 scale model has been designed and built in high strength aluminum alloy (Ergal). Main dimensions and particulars of the hydrofoil model are given in the planform view of Figure 3. The design submersion to which all the calculation presented in this model refer is 100mm, measured along the span of the foil from the tip of the trapezoidal region (at the root of the winglet) with an inclination (dihedral) angle to the free surface of 40 degrees. The model has been tested in the free surface cavitation tunnel of the Technical University of Berlin (TUB) and the results and observation will serve to validate the CFD simulations presented in this paper. Different cavitation numbers during the experiments have been obtained by varying the tunnel internal pressure p_{atm} according formula (5), keeping the inflow speed equal to the maximum achievable value of 11 m/s. The CFD calculations were prepare to replicate the model tests, with same dimensions and physical parameters. Vapor tension of water was set to 3170 Pa.

VOLUME MESH

The selected finite volume solver can generate different types of meshes, either structured or unstructured, and different types of elements, from simple tetrahedral to any kind of irregular polyhedral. In particular, for the surface piercing super-cavitating hydrofoil, in similitude with the planing hull forms (Brizzolara & Villa, 2010), the most appropriate mesh choice is: Cartesian prismatic cells in the far field, a number of prism layers cells obtained by extrusion of the surface mesh of the foil in normal direction to its surface, in order to accurately resolve the boundary layer and cavity development and a region with (polyhedral) trimmed cells to efficiently transition from the Cartesian cells to the prism layers cells. Figure 1 presents a longitudinal section of the volume mesh close to the hydrofoil tip.

Anisotropic refinements have been used for cells close to the undisturbed free surface, and an isotropic one was used to increase cell density near the body. These refinements guarantee a good mesh quality near the body and to capture the typical long waves generated by the surface piercing body, while minimizing the total number of cells needed to solve the entire problem.



Figure 4 - Mesh in the far field and on the hydrofoil

For each combination of angle of attack and submergence, a different mesh has to be generated. In general, all the meshes used have a total number of cells between 1.7 M and 2.2

millions. The prismatic cells region around the body is made up of 5 layers of cells, in vertical direction, with a total thickness of 1 mm in order to achieve a mean value of y+ of about 40 and a maximum value of about 80 in the conditions presented.

ANALYSIS AND VALIDATION OF NUMERICAL RESULTS

Model scale tests have considered only the similitude of the cavitation number calculated at the free surface that is σ =0.05 (full scale speed of 120 knots), by lowering the internal pressure of tunnel. The maximum Froude number, referred to the mean chord reached in model scale was about Fn=11, against its corresponding full scale value of 25.4 at top speed. Also the Reynolds and Weber numbers, of course, are not respected. Scaling effects are then present in the tests and indeed could be studied by CFD, but this will be a future scope of the research.

Systematic CFD calculation have been done in model scale by varying the angle of attack around the design value adding and subtracting three degrees. Then a systematic change of cavitation number (pressure in air) and submergence of the foil has been operated, exactly like in the cavitation tunnel.



Figure 5 – Type of ventilated cavity development in the wake of a surface piercing strut (Thomsen & Rubin, 1963).

CFD simulations offer an interesting possibility to analyze the mechanism of ventilation that naturally occurs at low cavitation numbers or at high angles of attack on the foil.

In general, the air is naturally drawn down on the back of the profile from the free surface due to the low absolute pressure created by the high speed flow on the suction side of the hydrofoil. This phenomenon happens also at low speeds (or high cavitation numbers) when cavitation is hardly present. Thomsen & Rubin (1963) after a number of tests on different kind of describe three possible states of cavitation and ventilation: the cavitation inception and pre-base ventilation state, the base ventilation state, and the post-base ventilation state, as schematically presented in Figure 5. In the inception and pre-base ventilation state, the ventilated cavity is building up behind the strut with increasing speed, reaching down to a depth somewhere above the strut base. In the base ventilation state, the bottom of the ventilated cavity springs from the strut base. In the post-base ventilation state, the cavity behind the

Proceedings of the 8th International Symposium on Cavitation CAV2012 – Paper No. 90 August 14-16, 2012, Singapore

strut is sealed off at the free surface by the flow, its contents being vapor.



Figure 6 - Free surface elevation predicted by the three phase RANSE model at design condition: σ =0.05, α =0.



Figure 7 – Closer view of the cavity opening to the free surface for the hydrofoil working at design condition (α =0, σ =0.05)

It is interesting to compare this consolidated interpretation of the physics of cavitation/ventilation interactions with what is predicted by the RANSE model.

Figure 6 presents the wave elevation predicted at design condition, trough colored wave contours: the lifting surfacepiercing foil produces a long divergent-like (single) wave, which has a fine crest on the downwash side of the foil and a through on the back side. The crest and through of the wave are connected by a very steep surface which actually contains the vorticity shed by the lifting profile. This vorticity, in fact, is what determines the steep discontinuity of the free surface in the wake of the foil. The wave crest is forming on the face of the foil by the piling up of water in this high pressure region which forms also a spray jet flow on the foil face surface. This spray flow, which has many analogies with the transverse flow on planing hulls, can be well solved by volume of fluid solvers, as demonstrated for instance in Brizzolara & Villa (2010). From the close-up view of Figure 7 that again shows the air/water free surface but this time colored by the absolute pressure (undisturbed pressure in air at this cavitation number is equal to about 6100Pa), one can notice the opening of the supercavity to the free surface (cyan colored). Through this opening the air has access to the lower cavity that originates at the leading edge of the foil as a cavitating bubble initially fulfilled with vapor, as evidenced by the red colored surface in Figure 8. The cavity aperture to the free surface is sealed after about one chord length.



Figure 8 – Cavity and free surface elevation predicted at design condition using the interface capturing method without free surface sharpening algorithm. Color scale indicates the absolute pressure on the water/air interface. Red colored cavity indicates the predicted water/vapor interface.

Experimental evidence obtained from early tests on different kind of vertical struts suggests the cavitationventilation scheme of Figure 5. Comparing this with the (super) cavity predicted by the RANSE solver on the new hydrofoil, it is clear that the regime of operation in the so defined "postbased" cavitation state: in fact the cavity is spanning the full length of the foil and its lower part has a shape very similar to that sketched in the figure. In fact, the reduction in cavity length toward the tip of the foil is due to the increase of local cavitation number, due to hydrostatic pressure rise: at a given angle of attack, the higher the cavitation number the lower the cavity length. The closure on the opposite side, i.e. toward the free surface, is due to the free surface effect that tends to bring circulation to zero on the foil sections close to the free surface, by reducing the angle of attack, lift and consequently cavity length.

The main difference between what is found by the CFD model on this new hydrofoil (Figure 8) and the cavity observed in other cases at the post-base ventilation state are twofold:

The existence of two cavities: one just below the free surface, starting at back midchord, long and short, fully ventilated; the other, starting just below the free surface, has a similar shape of that described in - The lower cavity is partly cavitating and partly ventilated: in fact the CFD solver predicts a first portion of the lower cavity full of vapor, colored in the red in Figure 8 and second portion of it partly ventilated, colored with the local absolute pressure;

Corresponding evidence from model tests are captured by the two picture of Figure 9. Unfortunately a foamy sheet of water at the free surface just close to the transparent window of the tunnel is preventing the observation of the cavity shape at the free surface. The hydrofoil effectively results supercavitating state, but a single super-cavity exists and it persists for a longer length aft of the hydrofoil. Moreover from the aspects the cavity seems thicker and fully ventilated up to its leading edge.



Figure 9 – Cavitation pattern on the back and face of the hydrofoil at design condition (α =0, σ =0.05)

In general, then, model tests show stronger natural ventilation, than what predicted by the numerical model. The longer length and thickness of the cavity are actually consistent with this interpretation.

It is important, then, to analyze in more detail the flow predicted by the numerical model, in order to identify the reasons of the underestimation of the ventilation phenomena. The mechanism of ventilation predicted by the RANSE model can be well explained through analysis of the cross flow on the sequence of transverse sections in Figure 13.

It is noted that already at the first section the upper cavity on the back is open to the free surface and is drawing air inside, as expected. But, according also Figure 8, it is sealed on its lower edge where a strip of wetted flow is predicted on the foil back, so that the sucked air cannot ventilate the lower cavity which in fact results still full of vapor at this station. The next station sectioning the annex has again a similar type of flow, although the local thickness of foil has diminished and the lower cavity thickness results larger. The third section just aft the foil trailing edge shows the cavity completely open and in fact a copious air-flow is predicted. In fact the cavity surface at this location is a limiting surface of the air/water phase, as predicted by the RANSE model.

Downstream of the third section, the predicted cavity is fully ventilated and it is interesting to see how the numerical model predicts the closure of the cavity. From station 3 to 7 the strip of water that was sealing the top ventilated cavity in the first two sections is closing back into opposed water face previously opened by the foil pressure side. The closing effect due to gravity is accelerated by the low pressure caused by the venture effect of the air sucked into the cavity (sections 4 and 5), that is actually causing a jet flow into the lower cavity (sections 5 to 7) after that the upper part is sealed.



Figure 10 – Same condition of Figure 8 but solved with interface sharpening numerical algorithm with a sharpening factor 0.8.

Though the physics is very similar to what was observed in other cases, still the ventilation phenomena is underestimated. The principal causes are thought to be:

- the lower resolution of the mesh used one chord length downstream of the foil to limit the cell number. This creates a premature closure of the cavity by creating an artificial numerical diffusion of the volume of fluid of water in the region;
- the numerical integration of the last two equations of (1) that govern the volume of fluids of the three phases;

A test was done repeating the calculations with no free surface sharpening algorithm or with a sharpening factor equal to 0.8.

The result of this last case is given in Figure 10 which be compared directly with the results previously commented (Figure 8). Generally the ventilated part of the cavity is more extended, both in the upper cavity directly in contact with the free surface and the lower cavity, where the air phase can flow upstream in the cavity, reaching the leading edge at the connection with the winglet. From the comparison it is clear that the numerical solution is very sensible to the numerical algorithm used to describe the transport of the air phase and its mixture with the vapor phase. If the mathematical model used works well for immiscible phases, such as air and water, for which a sharp interface really exists, they seem not perfectly adapted to two miscible gaseous phases such as air and vapor. Further work is planned on this subject.

A systematic series of calculation has been done, anyhow, with the non-sharp interface algorithm, varying the angles of attack , the submersion and the cavitation number. Figure 12 shows the results obtained at the design cavitation number σ =0.05 and design submergence S=110mm, for three different angles of attack: -3deg, 0deg (design) and +3deg. Overall it is

noted that an increase of incidence causes a major ventilation in of the cavity, as expected, while at -3 deg, the vapor cavity detaches on the back at the max thickness line and it is almost all cavitating. The cavity length is also generally increasing with the angle of attack, despite the rather low resolution of the mesh in the wake.

The predicted lift force, as presented in Figure 11 against experimental values, is rather satisfactory. In general the force is overestimated when the cavity in the experiments is fully ventilated and underestimated at the lowest of attack when the the back is almost fully wetted.



Figure 11 - CFD and Experimental lift coefficients at design condition, different angles of attack

CONCLUSIONS AND FUTURE PLANS

RANSE finite volume solver with a volume of fluid method to allow for different fluid phases has been applied to simulate the main phenomena of mixed cavitation/ventilation around surface-piercing super-cavitating hydrofoils. The ability of the method to consider simultaneous cavitation and ventilation has been demonstrated with satisfactory results.

Accuracy in the prediction of forces is in general on a level of 10% on different angles of attack, but numerical results are rather sensible to the free surface sharpening algorithm used, as previously recalled. Main issue relies in the prediction of the air flow into the cavity initially full of vapor.

Additional investigations, in fact, are needed about the modeling and numerical solution of the transport equations that describe the advection and diffusion of the air and vapor phases. The ventilated portion of the cavity predicted by the numerical method is very sensible to the numeric schemes used to solve these transport equations. A different kind of numerical scheme would be perhaps the ultimate solution to accurately solve this complex phenomena involving interactions between water vapor and air fluids.

ACKNOWLEDGMENTS

The authors are grateful to the Office of Naval Research (ONR) and ONR Global for their financial support through grant no. N62909-11-1-7007.

REFERENCES

- Baker E.S. (1975). *Review of Supercavitating Hydrofoil Experiments 1955 through 1972*. David Taylor NSRDC, report SPD-567-01.
- Brizzolara S., Young J. (2012) Physical and Theoretical Modeling of Surface-Piercing Hydrofoils for a High-Speed Unmanned Surface Vessel, to appear on Proceedings of the ASME 2012 31st International Conference on Ocean, Offshore and Arctic Engineering, June 2012, Rio de Janeiro, Brazil.
- Brizzolara S., Bovio M., Federici A., Vernengo G. (2011). Hydrodynamic Design of a Family of Hybrid SWATH Unmanned Surface Vehicles. 11th International Conference on Fast Sea Transportation. Honolulu, Hawaii (USA), 26-29 sept. 2011.
- Brizzolara S., Federici A., (2011). Super-Cavitating Profiles for Ultra High Speed Hydrofoils: a Hybrid CFD Design Approach. 9th Symposium on High Speed Marine Vehicles, HSMV 2011. Naples, March 2011, vol. 1, p. 1-13, ISBN/ISSN: 978889061120
- Brizzolara S. (2011). ONR USV HY2-SWATH. Hydrodynamic Design and Assessment by CFD Methods of HYbrid HYdrofoil / SWATH hulls for a Super High Speed USV. Final report of ONR grant # ONRG-N62909-10-1-7116.
- Brizzolara S., Villa D. (2010). CFD simulations of Planing Hulls. In: Seventh International Conference on High-Performance Marine Vehicles, HIPER 2010. Melbourne, Florida, USA, 11-15 Oct. 2010. ISBN: 9781450732314.
- Sauer J., 2000, Instationaer kavitierende Stroemungen Ein neues Modell, basierend auf Fron Capturing VOF und Blasendynamik, Ph.D. Dissertation, Universitaet Karlsruhe.
- McKann R.E., Blanchard U.J., Pearson A.O. (1960) Hydrodynamic and Aerodynamic Characteristics of a Model of a Supersonic Multijet WaterBased Aircraft Equipped with Supercavitating Hydrofoils. NASA TM-191.
- Thomsen P., Rubin H. (1963). *Cavity shape and Drag in ventilated Flow*. TRG Incorporated Report TRG-156-SR-2.

Proceedings of the 8th International Symposium on Cavitation CAV2012 – Paper No. 90 August 14-16, 2012, Singapore







Figure 13 - Evolution of the volume fraction of water/air-vapor as captured at different transverse sections at and aft the hydrofoil. The presented sections are ordered from left to right, top to bottom, and correspond to the transverse planes indicated in the first two figures.