AN EXPERIMENTAL AND NUMERICAL STUDY ON CAVITATION OF HULL APPENDAGES

Luca Savio, Department of Naval Architecture and Marine Engineering (DINAV), Genoa University, Italy
Chiara Pittaluga, Hydrodynamics Department, CETENA, Italy
Michele Viviani, Department of Naval Architecture and Marine Engineering (DINAV), Genoa University, Italy
Marco Ferrando, Department of Naval Architecture and Marine Engineering (DINAV), Genoa University, Italy
Francesco Conti, Fincantieri C.N.I. Spa – Naval Vessel Business Unit - Genoa, Italy

SUMMARY

In the present work a study on cavitation of hull appendages, such as stabilizer fins or rudders, is presented. The attention is focused on tip related cavitation and especially on tip vortex cavitation. Devices, such as end plates and tip fairing, commonly adopted to reduce this phenomenon are analyzed through experiments and numerical calculations. Various solutions are compared underlining their advantages and shortcomings, considering the effect on cavitation inception and vortex intensity. With this aim, a model of control surface was fitted with different end plates and with a tip fairing and tested at various angles of attack. Experiments were carried out in the cavitation tunnel of the University of Genoa while CFD computations were performed by CETENA. Moreover a comparison between experiments and numerical results is presented showing the relation existing between the two different approaches. As a result a simplified technique to predict cavitation phenomenon by means of numerical simulations calibrated with experimental results is outlined.

1. INTRODUCTION

Cavitation is one of the most important issues in propeller and hull appendages design. Among various types, vortex cavitation, in particular its inception, is still nowadays considered far from being completely understood [1]. With the increase in ships performance, more strict requirements in terms of environmental impact of ships and inboard comfort, this particular type of cavitation is gaining more and more relevance. Vortex cavitation is a source of radiated noise, but also of pressure pulses, and being able to propagate far downstream can interact with appendages (for example propeller tip and hub vortex with rudder).

From the design point of view what is more needed is to achieve a technique to predict the inception of vortex cavitation, because procedure to scale model test to full size are well known and reliable, see for instance reference [2]. In this scenario CFD computations may be a feasible way to solve the problem.

For the above reasons Cetena has been involved in CFD calculations for long time, dealing with different problems, among which evaluation of cavitation inception. As an example in the activities of the EU funded research project Leading Edge attention was focused on tip vortex cavitation inception on propellers. The study was mainly carried out with numerical simulations, but besides achievement obtained in this project the need of comparison between numerical solutions and experimental tests was stressed. This need was mainly linked to difficulty of computations in fully predicting tip vortex cavitation inception. [3]

The present study deals with this complicated topic partly simplifying the problem, by considering tip cavitation on a typical hull appendage, such as a stabilizer fin or a rudder in open water, thus dealing with a simpler geometry with respect to the propeller case. The wing dimensions were chosen to be as close as possible to an average hull appendage in order to add value to results. The low aspect ratio of the tested wing and its tip shape led to the identification of two different tip cavitation types, as described in the following paragraphs. This paper therefore focuses also on their interaction.

This work was fully supported by Fincantieri C.N.I, who also supplied the geometry of the wing.

2. MODELS AND TEST DESCRIPTION

In the following, the experimental set-up is briefly described; in particular, in paragraph 2.1 different models geometries are reported, in paragraph 2.2 the experimental facility is described and in paragraph 2.3 the tested conditions are listed.

2.1 MODELS

As anticipated, three different appendage models have been considered in the present study; in particular, all models are derived from the same initial geometry, with modification of the tip shape, and in particular:

- Squared tip (model 1)
- Fairied tip (model 2)
- Squared tip + end plate (model 3)

The choice of utilizing the same geometry (in terms of macroscopic characteristics, i.e. aspect ratio, taper ratio, thickness/chord ratio and sectional profiles) is linked to the wish to analyse specifically the influence of the tip shapes on the tip vortices inception, without taking into account other factors such as load distribution on the appendage.

It has to be noted, from this point of view, that the appendage might be indifferently a fin or a rudder, for
which application of different tip shapes has been documented in literature [4][5][6].

In the following table 1, main non-dimensional characteristics of the appendage are reported (where t/c is thickness / chord ratio, a.r. is geometrical aspect ratio and \( \lambda \) is the taper ratio), while in following figures 1-4 photographs of the models tested are reported.

<table>
<thead>
<tr>
<th>t/c</th>
<th>0.18</th>
</tr>
</thead>
<tbody>
<tr>
<td>a.r.</td>
<td>1.10</td>
</tr>
<tr>
<td>( \lambda )</td>
<td>0.80</td>
</tr>
</tbody>
</table>

Table 1: Main non-dimensional characteristics

Fig. 1: Overall view of the appendage

Fig. 2: Model 1 tip (squared)

As it can be seen from figure 4, the end plate utilized is somehow different from conventional ones, being applied only in the forward part of the appendage and not up to the trailing edge. This choice is due to the interest in analyzing end plates which can be easily adopted both to appendages with and without flaps, therefore its extension has been chosen in order to avoid covering the possible movable flap part.

Fig. 3: Model 2 tip (faired)

Fig. 4: Model 3 tip (squared + end plate)

In order to analyse also the possible disturbance of a flap actuator, this has been schematized with two cylinders in the aft part of the root section, even if no flap has been included in the present analyses for the sake of simplicity.

2.2 EXPERIMENTAL FACILITY

The experiments are carried out at the Cavitation Tunnel facility of the Department of Naval Architecture and Marine Engineering of the University of Genoa (DINAV), represented in figure 5.

The facility is a Kempf & Remmers closed water circuit tunnel with a squared testing section of 0.57 m x 0.57 m, having a total length of 2 m. Optical access to the testing section is possible through large windows. The nozzle contraction ratio is 4.6:1, and the maximum flow speed in the testing section is 8.5 m/s. Vertical distance between horizontal ducts is 4.54 m, while horizontal distance between vertical ducts is 8.15 m. Flow speed in the testing section is measured by means of a differential venturi-meter with two pressure plugs immediately upstream and downstream of the converging part.
A depressurization system allows obtaining an atmospheric pressure in the circuit near to vacuum, in order to simulate the correct cavitation index for propellers and profiles (2D and 3D). In the present experimental campaign, the 3D appendage has been connected to one of the tunnel windows (simulating a completely flat hull surface) with a low gap between the root section and the wall (about 4 mm), and with the possibility of setting the angle of attack to different values.

The appendage dimensions have been chosen in order to allow a good visualization of cavitation phenomena, in particular the span is 200 mm. Cavitation phenomena visualization in the testing section has been made with two Allied Vision Tech Marlin F145B2 Firewire Cameras, with a resolution of 1392 x 1040 pixels and a frame rate up to 10 fps.

In the following Figure 6, the testing setup is presented, while in following Figures 7 and 8 two typical pictures from the two cameras are provided.

For what regards forces evaluation, this was not performed in the present testing campaign since the tunnel is not equipped with a measuring equipment; it has to be pointed out once more that main objective of this experimental campaign has been the evaluation of tip vortices inception.

### 2.3 TESTED CONDITIONS

For all appendage configurations, a series of tests at different cavitation index values have been performed. In particular, cavitation index values tested range from 5.5 to 1. Moreover, all tip configurations have been tested in correspondence to an angle of attack of 5° and 10°. The limitation of the angles to these two values is due to the particular interest of the industrial partner to these configurations, which correspond to the range utilised for the greatest part of operating time.

The flow speed has been kept as high as possible; in order to have a sufficiently high Reynolds number (over $1.0 \times 10^6$ for all tests); in particular, these tests have been carried out at a 8 m/s flow speed for the 5° configuration, whereas the flow speed has been reduced to 6 m/s for the 10° configuration, in order to limit lift forces considering the experimental set-up.

Regarding cavitation index, the formulation used is obviously the one related to the flow speed, as follows:

$$\sigma_0 = \frac{P_0 - P_v}{\frac{1}{2} \rho V^2}$$  \hspace{1cm} (1)
where $p_0$ is the pressure in correspondence to the appendage, considering atmospheric pressure and hydrostatic pressure, $p_v$ and $\rho$ are water vapour pressure and density and $V$ is flow speed.

In particular, apart making visual observations of cavitation phenomena (and their extent) in correspondence to different cavitation index values, inception values have been recorded for both vortex related phenomena experienced (see next paragraph 3.1).

3. EXPERIMENTAL RESULTS

3.1 ANALYSED PHENOMENA

As already anticipated, main aim of this study has been the analysis of vortex related phenomena. In particular, two different vortices have been evidenced during trials, one which detaches from the sheet cavitation which develops on the tip section in correspondence to the sharp angle at leading edge (see Figures 9 and 10) and one which detaches from about midchord location (model 1 and 3) or trailing edge (model 2) on the suction side of the appendage (see Figures 11, 12 and 13).

As it can be seen, the first phenomenon is present only in correspondence to squared tip configuration (as expected), both with and without endplate, and is strictly
connected to the sharp angle at the leading edge, not present in the faired tip configuration.

On the contrary, the second phenomenon is present in correspondence to all configurations tested; for model 1 and 3 (squared shapes), the inception of this phenomenon is anyway in correspondence to lower values of the cavitation index. When both vortices are present, they tend to collapse into each other forming a unique vortex which continues downstream. This phenomenon is clearly visible in following Figure 14.

Previous figures are all referred to an angle of attack of 5°, but the above reported considerations can be extended also to the higher value. Only inception values experience a considerable modification from 5° to 10° angle of attack, as it is reported in the next paragraph.

During various tests performed, also other cavitation phenomena have been experienced (both bubble cavitation at midchord and sheet cavitation), as represented in following figures 15 and 16.

Nevertheless, their detailed analysis is not reported in the present paper since main interest has been devoted to tip vortices. This choice was also due to the different nature of these phenomena, being tip vortex cavitation highly related to viscous effect, while sheet and bubble cavitation do not show this strong dependence. Vortex cavitation occurs at lower cavitation index in model scale with respect to full scale, therefore phenomena such as sheet and bubble cavitation may be present in tunnel test when studying tip vortex cavitation, but they are not present in full size at same cavitation index.

3.2 INCEPTION VALUES

In figures 17 and 18, inception values recorded during tests for both vortex-related phenomena are reported in correspondence to the angle of attack of 5° and 10° respectively.

As it can be seen, in correspondence to different values of the angle of attack, different behaviours have been experienced.

In particular, at the lower angle, the faired tip configuration appears as the best one, eliminating completely the “sheet cavitation vortex” and delaying the “midchord vortex” more than the other configurations.
This behaviour is considerably modified in correspondence to the higher angle, where the faired tip configuration still eliminates the “sheet cavitation vortex”, but fails to delay the “midchord vortex” more than the other configurations. Regarding the sheet cavitation, inception values are similar for both configurations for which the phenomenon is present, and they are not influenced strongly by the angle of attack, with a slight increase in correspondence to the higher value.

4. DESCRIPTION OF NUMERICAL METHOD

4.1 RANSE SOLVER

The complexity of numerical prediction of vortex flow structure shed by body submerged and the related possibility of cavitation is mainly connected to the physics of the viscous phenomena in turbulence dominated flows. Regarding naval appendages, they are usually positioned in a region of fully turbulent flow. Despite the important development gained by researches in the potential flow, these methods cannot properly describe local three-dimensional structure such as the tip vortex, in which viscosity plays a fundamental role.

The Navier-Stokes equations describe theoretically the viscous flow but unfortunately, no rigorous theoretical treatment of turbulence is nowadays possible, this means that empirical turbulence models have to be used and results have to be tested against model experiments.

The approach adopted is the Reynolds Navier Stokes equations, commonly referred to as RANSE. These equations describe the transfer of mass and momentum in a viscous flow and can be coupled with additional equations reproducing specific phenomena such as water cavitation and transport of scalar quantities. Tip vortex cavitation is an extremely complex phenomena and RANSE supply a new tool for the control and the study of vortexes that originate partly at the pressure side and detach at suction side of the tip.

The numerical model used to solve the RANSE is the finite volume method. This method requires the discretisation of the region of fluid of interest, called ‘fluid domain’ in a set of small volumes, the ‘finite volumes’ or ‘cells’, that make up a tridimensional grid in the fluid domain. The RANSE equations are defined and solved at each cell. The dimensions of the cells depend on the scale of the phenomena to be investigated and have to be small enough to make the calculation results independent of the calculation grid.

4.2 NUMERICAL SET-UP

For the present work the commercial code ANSYS-CFX 11, has been used, together with the mesh generator code ANSYS-ICEMCFD 11.

Solution of the RANSE in a fluid domain requires that proper boundary conditions are set at the domain boundaries. The boundary conditions are known properties of the flow that the RANSE solution is required to satisfy at the boundaries. Because of the need to set boundary conditions, the definition of the fluid domain is a delicate issue: the domain has to be large enough for the boundaries to be at locations where the fluid has a known behaviour, but not too large as this way the solution of the RANSE would require an unaffordable computational effort because of the large number of cells.

The fluid domain adopted is shaped as the cavitation tunnel layout with a section 0.57 m with rounded corners, the inlet in the tunnel has been set 2m in front of the models and the outflow at a distance of 6m in order to satisfy the condition of unperturbed flow (see figure 19).

The boundary conditions applied to the domain borders are as follows:
- at the inflow section the undisturbed velocity is applied
- at the outflow section a “zero” pressure gradient is given
- tunnel walls and the fins blade are no-slip walls with a scalable wall function.
to properly define the grid dimensions, prior to any calculations a sensitivity analysis was carried out, consisting in solving the RANSE with different grids of increasing fineness, until independence of the solution is achieved.

The mesh is a structured multi-block grid; therefore the decomposition of domain has been studied in order to refine the grid around the appendage model, in the tip wake region and around the tunnel walls in order to properly describe the boundary layer of the walls and the pressure and velocity distribution in the tip vortex core.

A H-Grid topology is applied to split the volume in the 3 Cartesian directions, so the initial block is subdivided in 9 blocks; then into the inner domain and around the blade, an O-Grid topology is applied with the generation of two new blocks.

Fig. 20: Domain decomposition, surface mesh at the tunnel and in the longitudinal mid section.

A total of 4369444 nodes have been used. Figure 20 presents the block structure, together with the node distribution around the boundary of the domain; in the longitudinal mid section.

The node distribution for model 1, squared tip, is given in figure 21. Note that the node distribution are refined at the leading/trailing edge and particular attention is given to capture the flow at the fin extremity.

The node distribution for model 2, faired tip, is given in figure 22. Note that the surface mesh structure has been modified in order to capture the flow detachment at the closure of the fairing at the trailing edge.

The node distribution for model 3, squared tip with plate, is given in figure 23. The mesh from the model 1 has been modified in order to take into account for the plate where it has been largely increased.
The K-ε turbulence model is used, the convergence has been achieved with about \(10^3\) iterations and has been carried out using different resolution scheme up to high resolution, RMS < 10^{-6}.

It was decided not to adopt any cavitation scheme in calculations in order to keep them as simple as possible. In particular, calculation for each configuration tested was completed in about 3 hours with a 4 processor cluster, which is considered a time compatible with design needs.

5. NUMERICAL RESULTS

The RANSE equations are defined and solved at each cell, their solution produces the pressure rate (total pressure, dynamic pressure, static pressure and gravity pressure) and the velocity field at each node of the grid.

In order to be able to compare the results with the experiments the numerical data are presented in terms of pressure coefficient:

\[
C_p = \frac{P - P_\infty}{0.5 \rho V_\infty^2}
\]

\(V_\infty\) is the velocity of the main flow at the inlet and \(P_\infty\) is the pressure value of the main flow at the outlet section.

The study of the vortex detachment has been analysed by 3D streamlines that represents the path of a particle that detach, as an example, from the fin surface and is driven in the fluid domain in addition to the visualisation of the vorticity and helicity in the wake.

The vorticity in the axial direction is obtained from the differences between the derivatives of the transversal velocity therefore it points out the tendency of the flow to rotate in the direction of the main motion.

\[
\omega = \frac{\partial V_z}{\partial y} - \frac{\partial V_y}{\partial z}
\]

Moreover the helicity pattern has been analysed, which is a scalar variable representing the intensity of the vorticity along the main flow direction.

\[
Helicity = (\mathbf{V} \cdot \nabla \times \mathbf{V})
\]

Therefore the Helicity is zero if a vortex move perpendicular to the main flow direction and higher in the flow direction (due to the velocity component of the flow). Due to its mathematical formulation the Helicity pattern seems to be the best compromise to point out a vortex in the ship wake that in the stern zone has the tendency to close the flow at the propeller region and as a consequence of the transversal velocity is no more aligned with the main flow direction.

In formula the Helicity is obtained by the scalar product between velocity and vorticity:

\[
V_z \left( \frac{\partial V_z}{\partial y} - \frac{\partial V_y}{\partial z} \right) + V_y \left( \frac{\partial V_y}{\partial z} - \frac{\partial V_z}{\partial x} \right) + V_z \left( \frac{\partial V_z}{\partial x} - \frac{\partial V_x}{\partial y} \right)
\]

5.1 VORTICES CAPTURING ABILITY

In order to better understand the concepts introduced in the above section, in figures 24, 25 and 26 a detail of the streamlines detachment from the tip region and, at a transversal plane 0.08m astern the model, the velocity vectors and the Helicity contour plot are presented. For all the cases the incidence of the flow is 5°.

![Fig. 24: Helicity along a transversal plane 1m astern the model 1 squared tip, flow incidence 5°](image1)

The flow astern the squared model is dominated by the main tip vortex but still the strength of the vortex that detach at the leading edge from sheet cavitation inception is well captured. The main tip vortex detaches at midchord; at a distance of 0.08m from the model the two vortexes are not completely merged.

![Fig. 25 Helicity along a transversal plane 1m astern the model 2 faired tip, flow incidence 5°](image2)

Regarding the flow separation from the faired tip model, only one vortex structure detaches at the tip region, close to the trailing edge, as a result only one vortex core is captured at the transversal section.
The flow astern the squared model with end plate is dominated from the main midchord tip vortex and from the vortex that detaches from sheet cavitation inception.

At a distance of 0.08m from the model the two vortexes core are completely independent. Again it can be noticed that the main tip vortex detaches from midchord, moreover the strength of the vortex from sheet cavitation inception is increased, compared to the model 1 results.

In the following figures rendering camera location is chosen to be similar to the physical camera setup used during experiments. In particular vortexes trajectories are compared to streamlines showing a good agreement with experiments.

As described above when the two vortical structures are present they tend to collapse into a unique vortex. It was also noted that the effect of the end plate was not only of postponing the inception of the midchord vortex, but also to move downstream the coalescence point.

Another phenomenon analyzed is the tip fairing effect of moving the starting point of the vortex propagating downstream from the midchord to the trailing edge. All these findings are showed in the following figures, where it can be also noted the good capability of the solver to describe these phenomena. Figures 27 to 38 refer to an angle of attack of 5 degrees, for all configurations considered.

For what concern the 10 degrees configuration the effect is to move upstream the coalescence point and form more complicated trajectories. This will appear clearly considering the figures 39-42, referred to the squared tip configuration.
Fig. 31: Squared tip + endplate– $\alpha = 5^\circ$ - Horizontal view - Experimental

Fig. 32: Squared tip + end plate-- $\alpha = 5^\circ$ - Horizontal view – Numerical

Fig. 33: Squared tip + endplate– $\alpha = 5^\circ$ - Longitudinal view - Experimental

Fig. 34: Squared tip + endplate– $\alpha = 5^\circ$ - Longitudinal view - Numerical

Fig. 35: Faired tip– $\alpha = 5^\circ$ - Horizontal view - Experimental

Fig. 36: Faired tip– $\alpha = 5^\circ$ - Horizontal view - Numerical

Fig. 37: Faired tip– $\alpha = 5^\circ$ - Longitudinal view – Experimental

Fig. 38: Faired tip– $\alpha = 5^\circ$ - Longitudinal view - Numerical
5.2 VORTEX INCEPTION

In the previous paragraph a qualitative analysis of the flow capturing ability of the solver has been presented. In order to get a further insight in the phenomenon, an analysis of the possibility to predict the vortices inception has been made.

As already remarked, vortex inception prediction is still nowadays a problem in the design phase, in which a tool able to assess the merits of different solutions is strongly needed. In this paper, in particular, focus has been posed on the application of a RANSE solver; in order to simplify as much as possible the problem (with the aim of keeping computational times not too high and compatible with design procedures needs) RANS equations without cavitation model have been used. Moreover, the number of cell adopted for the computational grid has been limited to a value which, on the basis of previous experiences, is still sufficient for the vortex capturing.

Keeping this in mind, two different procedures to assess vortex inception have been adopted.

The first procedure consists in looking at different isosurfaces with pressure coefficient $C_P$ as parameter, reducing progressively the modulus of the $C_P$ value itself until an initial area in the vortex inception region is obtained (see figures 43 and 44 for the vortex from sheet cavitation and the midchord vortex respectively); the vortex inception region is localised clearly from the streamlines previously represented, and the values recorded for the $C_P$ are used as an index of cavitation inception.

The second procedure is similar to the first one, but the value of interest is recorded not when an initial area is found for the isosurfaces, but when a larger region appears, with an initial vortex-like area, as represented in figures 45 and 46 for the two phenomena analysed.
It could be argued that this second procedure presents margins of error related to the operator recognisation of the “initial vortex-like area”, but it is believed that, once the operator remains the same and the criteria adopted (e.g. extension of the vortex like area) is not varied from configuration to configuration, a rather objective procedure results. Moreover, it has to be noted that small differences in the extent of the area usually do not result in significant modifications of values recorded, thus ensuring an implicitly robust procedure.

On the other hand, moreover, it has to be underlined that the first procedure, despite seeming more objective, is more affected by possible local problems of the grid, which can result in problems for the assessment of the real merits of different configurations, as it will be reported in the following.

Finally, it has to be noticed that, in order to make a wider analysis, the two procedures have been applied not only considering iso-$C_p$ surfaces, but also iso-helicity surfaces. This additional analysis has been made because, as already remarked, helicity is a very good parameter to highlight vortices, resulting even better than vorticity in some cases. The concept underlying in this choice is to link the vortex strength (represented by helicity) to the cavitation inception.

In the following figures 47 and 48, values of $C_p$ recorded, obtained with the first and second procedure respectively, for all configurations considered and for the two phenomena analysed are reported on the y axis, while on the x axis the correspondent experimental value of the cavitation index is reported. Angle of attack considered is $5^\circ$.
As it can be seen, the application of the first procedure does not allow obtaining a merit index among the various configurations, for which a similar value of the $C_P$ index is found. The reason for this is probably linked to local effects of the grid, which should be further refined to capture correctly the phenomenon.

On the contrary, the second procedure allows to find a clear (and correct) tendency for both phenomena analysed, thus confirming its supposed capability of avoiding local numerical problems.

It has to be noticed, however, that none of the procedures allows to get the absolute inception value ($C_P$ value should result equal to cavitation index), but only the relative merits of different solutions. On the basis of previous experience, it is believed that this problem could be overcome by means of a much finer local grid, but this would result in a much higher computational effort (and time), in contrast with the initial scope of this analysis.

In the following figures 49 and 50, similar results are reported, representing in this case nondimensional values of the helicity instead of pressure coefficient values.

In this case, it can be noticed that both procedures are able to capture the order of merit between different configurations for both phenomena, thus confirming the vortex related phenomena capturing ability of the helicity parameter. Nevertheless, it has to be remarked that this method allows providing a merit order for the two phenomena separated from each other, while it would not be possible to define which is the first phenomenon present. It has also to be noted, however, that the stronger interest from the designer point of view is linked to the midchord vortex phenomenon which, despite having a delayed inception with respect to the other, is the one which is present downstream of the wing, thus being a more probable source of problems.

In the following figures 51-52 and 53-54, same analysis has been repeated for the higher angle of attack considered ($10^\circ$). Similar conclusions can arise also from the analysis of this series of figures; in particular, the second procedure allows again finding the merit among different configurations, adopting both indices introduced. It has to be noticed that this means that the procedure has been capable of observing the modification in the merit order, with the faired tip configuration becoming worse than the others.
6. CONCLUSIONS
In the present paper, some comparisons between numerical results obtained with a commercial RANS solver and experimental observations about the vortex-related phenomena of a wing have been presented, considering the vortical phenomena themselves and the related cavitation inception.

The solver shows a remarkable capability in capturing the two different vortical structures present in correspondence to different configurations and angles of attack.

Regarding cavitation inception, two simplified procedures for the assessment of merit order among different configurations have been presented together with the correspondent results. In general, these two procedures (especially the second one) allow to obtain the merit order correctly in correspondence to all configurations tested without requiring a too high computational time, representing a possible useful tool in the design phase in order to compare a range of different configurations with localised geometrical variations.

It is believed, anyway, that the need for a final verification by means of model tests is still present: main reason for this is the current impossibility of assessing the absolute value of cavitation index in correspondence to the phenomenon inception with the limitations in computer burden chosen for the present study.

Further analyses will be made in order to have a clearer insight in the additional time (or computational capability) needed for a more precise assessment of the cavitation index values in correspondence to phenomenon inception and in correspondence of different geometries like those tested in the present study.

REFERENCES
[1] Keller A.P. “Cavitation scale effects. Empirically found relations and the correlation of cavitation number and hydrodynamics coefficients” Cav 2001