Numerical simulation of the cavitating KP505 propeller working in open water conditions

A Lungu
Department of Naval Architecture, “Dunarea de Jos” University of Galati,
47 Domneasca Street, 800008 Galati, Romania

E-mail: adrian.lungu@ugal.ro

Abstract. The present paper proposes a numerical study, fully experimentally validated, usable for the optimal design of the naval propulsion features. The marine propeller KP 505 is a representative propeller geometry with experimental available detailed, so this is chosen as a reference for probing the accuracy of the present study. Data obtained from this model is extensively used to provide validation for unsteady RANS calculations, with specific interest in predicting tip-vortex flow characteristics. The method used is based on the fundamental conservation equations. Since DNS modeling requires excessive computational effort due to the very dense discretization of computational domain and very limited time step, it is assumed that the influence of turbulence on the flow is derived with sufficient accuracy based on the averaged flow parameters. In the present computations the EASM model is applied, which proved to be a stable and accurate model. The computed overall accuracy of the proposed model is validated by an extensive comparison with experimental data from the free running tests performed by NMRI Tokyo and HSVA Hamburg. The main features of the flow field, with particular attention to the vortical structures detached from the blades are comparatively presented in an attempt to validate the technique proposed.

1. Preamble
Cavitation on ship propellers can cause the propeller thrust breakdown, noise, vibrations and serious erosion. It is of a primary importance for the naval architect to reduce the risks of cavitation occurrence by adjusting properly the propeller geometry i.e. the chord length and the blade area ratio. For that purpose, one of the tools still remains the experiment performed in a cavitation tunnel. Since the experimental study is highly complex, expensive, difficult and time consuming, the numerical simulation became a trustful alternative. With the advances of the hardware performances flowing down from the IT development technology, more and more of the propeller cavitation analysis are done by means of the CFD tools. The numerical analysis of a propeller in an unsteady flow by using the wide offer of the URANS solution methods is therefore required as a first step in a thoroughly numerical conducted investigation rather than to the tedious cavitation experiments.

Computational methods for cavitation have been developed for almost of four decades. The beginnings may be set by the comprehensive study of Lee [1], who proposed a study on marine propeller cavitation in 1979 in a widely referenced Ph.D. thesis. Basically, the theoretical methods can be largely categorized into two groups: single-phase modeling with cavitation interface tracking and multiphase modeling with an embedded cavitation interface. The former approach, i.e., single-phase modeling with cavitation interface tracking, has been widely adopted for inviscid flow solution methods, such as potential flow boundary element methods [2] and Euler equation solvers [3]. These methods
have evolved significantly and many successful application results have been presented, as reviewed in [4]. Still in many cases they require cumbersome iterative procedures and a considerable amount of preliminary knowledge, such as cavity closure conditions.

The multi-phase modelling with an embedded cavitation interface can be used for more general viscous flow solution methods, such as the unsteady RANS solvers. This approach can include the effects of compressible vapour and gas, turbulence fluctuations, and a bubbly phase within the mixture, although a closure equation is required to connect the density to the other variables. Two types of techniques are available for this approach: a mixture fluid method, with separate continuity equations for multi-phase analysis [5], [6], and a full multi-fluid method with separate conservation equations for each phase [7]. Compressibility effects have been incorporated and several positive achievements have been reported [8], as various improvement efforts are still underway.

A step ahead was made in [9] where the solutions for the well-known E779A benchmark test-case [10] computed with five different CFD codes. Both open water flow and the flow behind a prescribed non-uniform velocity field were considered and the numerical solutions were compared against BEM results and experimental data. Later, implicit LES results for this case were reported in [11], where only performance characteristics and qualitative comparisons of cavity extents were made for a simplified geometrical setup. For the Potsdam Propeller Test Case extensive comparisons of the open water cavitating flow performance predictions were done in [12] when several commercial codes were used to describe the flow features.

Present paper proposes a numerical study, experimentally validated, devised for the optimal design of the KP 505 propeller. The model is a representative propeller geometry for which experimental detailed velocity field LDV measurements and cavitation inception measurement are public, therefore the model been chosen as a reference for probing the accuracy of the CFD solver. Computed data are used to validate the RANSE calculations, with specific interest in predicting tip-vortex flow characteristics. The theoretical approach is based on the fundamental conservation equations. Since the DNS, LES and DES techniques require excessive computational effort due to the very dense discretization of computational domain and very limited time step, it is assumed that the influence of turbulence may be derived with sufficient accuracy based on the averaged flow parameters. In the present computations the EASM model is applied, which proved as being an evoluted model of high stability and accuracy levels. The overall accuracy of the proposed model is validated by an extensive comparison with experimental data from the free running tests performed by some of the most prestigious ship-hydrodynamic towing tanks, i.e. NMRI Tokyo and HSVA Hamburg [13], [14]. The main features of the flow field, with particular attention to the vortical structures detached from the blades are discussed in an attempt to clarify the cavitation occurrence mechanism.

2. Cavitation milestones
Cavitation phenomenon may occur behind the blade of a fast or overloaded rotating propeller, i.e. when the total blade area is insufficient. It may also occur when air is sucked into the disk while a ship is sharply turning, when the propeller gets out of the water or when too much power causes bubbles of vacuum and several effects such as thrust reduction, noise and visible mechanical damage manifest. In order for cavitation to occur, the bubbles generally need a surface on which they can nucleate, which is where the propeller surface comes into play. Depending on the either side of the propeller, the pressure can be higher or lower. Cavitation bubbles form on either the suction side at as low as 80°C or on the discharge side at as high as 150°C. As the water rushes in to fill in the vacuum, it vaporizes due to the lowered pressure. It implodes against the blades therefore leads to a loss of balance between the forces developed at the shaft. When the cavitation occurs, depending upon its extent and severity, the propeller may suffer from performance breakdown, noise, vibration and erosion. Partial cavitation on a propeller blade does not affect its thrust. A small amount of cavitation determines the augmentation of the blade camber and hence, may increase the thrust. However, when it covers more than 20–25% of the blade area both thrust and torque drop. As thrust decreases more rapidly than torque, the propeller efficiency drops as well.
One of the most important determinants for cavitation to occur is the existence of nuclei and nucleation sites. The prediction and control of nucleation mechanisms are very unclear even for water. Usually, a liquid contains dissolved air within miniscule gaseous and/or vaporous bubbles, which may serve as cavitation nuclei. When these nuclei enter inside a low-pressure region, cavitation starts. Consequently, bubbles appear on different spots in the low-pressure regions. In most liquids, the reduction in volumetric stability is facilitated by the presence of external surfaces, which usually take the form of various admixtures such as dissolved or suspended impurities, particularly those at a sub-microscopic level. Basically, there are two types of nucleation modes: homogeneous and heterogeneous. Homogeneous nucleation takes place when there is no prior presence of additional gaseous phases in the liquid phase due to the distribution of thermal energy among the molecules in the liquid volume. Due to the difference in the level of excitation of the molecule, the cavitation nuclei are very unstable. A gas bubble will easily dissolve in an under-saturated solution, while the effect of surface tension will cause it to dissolve in a saturated solution.

Once the pressure in the liquid is reduced to a certain level, cavities become the repository of vapour or dissolved gases. This change in the hydrodynamic pressure causes phase transition in the liquid, forming a two-phase flow mainly composed of the liquid and its vapour. The immediate result of this condition is the rapid increase in the cavity size. When this larger cavity enters a zone of higher pressure, there is a reduction in its size as a result of condensation of the vapours in the liquid causing collapse of the bubble. During the collapse, particles of liquid surrounding the bubble quickly move to its centre. Kinetic energy from these particles create local water hammers of high intensity, which grow as the particles progress towards the centre of the bubble. However, due to the rise in the pressure inside the bubble, the contraction of the bubble eventually slows down and stops. After the collapse, the bubble reappears in the form of a rebound bubble of smaller size. The rebound happens due to gas trapped in the bubble at the collapse. The cavitation bubbles are filled with a mixture of vapour and non-condensable gas, the origin of which is still unclear, but is thought to come from a combination of gas dissolved in the water, and gas created at the energy deposition due to the generation of the bubbles. The quantity of these non-condensable gas in the bubble influences the maximum radius of the rebound bubble. When collapsing, the shape of the bubble is distorted since the Rayleigh-Taylor instability theorem states the existence of the instability at the interface between fluids of two different densities that occurs when the fluid of lower density tries to accelerate through the fluid of higher density.

Usually, the propeller cavitation is classified depending on its physical appearance in: bubble, sheet, tip and root vortices cavitation. Since the last two are beyond the scope of our research, they will be simply skipped in the present analysis, although some references will be made in respect to them. In the sheet cavitation, which is the main interest of the present work, the cavities form on a solid boundary and remain attached as long as the conditions that caused their formation do not change. Sheet cavitation, which is regarded as a steady-state cavitation, is a region of vapour that remains approximately at the same position relative to the profile or propeller blade after its complete development. Sheet cavitation occurs when the pressure distribution has a strong adverse gradient and the flow separates from the blade surface. Sheet cavitation initially becomes apparent at the leading edge of the blade when the blade sections are working at large positive angles of attack as it will be proven in the following sections. Sheet cavitation can have considerable volume of cavitation bubbles on the propeller blade whose dynamic behaviour may generate strong pressure fluctuations at frequencies lower than the noise frequencies.

3. Computational conditions

The present paper represents a first attempt in assessing the hydrodynamic features of the flow around a propeller working under both cavitating and non-cavitating conditions. Since an important prerequisite for any self-propulsion computation is the propeller open water test (POW hereafter), the work reported in here will refer only at this subject. The hull chosen for the present study is the KRISO Container Ship, which is a preliminary design of a modern surface container vessel, accessible for
fundamental research purposes to the worldwide community with interests in the subject. The ship geometry is shown in figure 1 that depicts the bulbous hull appended with a five-blade propeller and a semi-suspended rudder. The length between perpendiculars of the model is 7.2786 m, while the maximum beam of the waterline, the depth and the draught are 1.019 m, 0.6013 m and 0.3418 m, respectively. The KP505 propeller model has a diameter of 0.25 m. The geometry of the hull, propeller and rudder are those officially provided in [13] and [14].

Figure 1. KCS container ship hull.

ISIS-CFD solver available in the FineTM/Marine package is employed in the present study. The solver is based on the finite-volume method to build the spatial discretization of the transport equations on unstructured grids. The simulation is accomplished in a global approach in which the momentum and mass conservation equations written in respect to a Cartesian system of coordinates are solved. Since no coordinate transformation is done in the solving algorithm, the efficiency of the numerical approach may be considered as being suited to the purpose. Dependent variables of the equations set are the velocity and pressure. Closure to the turbulence is achieved through the EASM model. The face-based method is generalized to unstructured meshes composed of arbitrary volume shapes. Fluxes are built using the higher-order bounded difference scheme AVLSMART. Velocity field is obtained from the momentum conservation equation and the pressure field is extracted from the mass conservation constraint transformed into a pressure equation. Picard’s procedure is used for the linearization of the equations. The whole discretization is fully implicit in space and time and it is formally second order accurate. The flow around the 250 mm diameter propeller model rotating at 1920 rpm is simulated at first for a set of 20 incoming flow velocities so that the resulted values of the advance coefficient be between 0 and 1. The computational domain consists of a cylinder, whose diameter is 4 times the propeller diameter, and length is 8 times the propeller diameter, as depicted in figure 2, which shows the computational domain and boundary conditions formulation.

Figure 2. Computational domain and boundary conditions.

Figure 3. Computational mesh.
4. Results and discussions

Having the given computational domain size given, three different grids denoted by \( G_1 \), \( G_2 \) and \( G_3 \) were generated for the POW grid convergence test. \( G_1 \) consists of 7.12 M cells, \( G_2 \) of 15 M cells, whereas \( G_3 \) shown in figure 3 consists of 30.1 M cells. The resulting number of faces for each blade is approximately 50000 for the finest mesh. A viscous layer consisting on 18 cells is introduced on each solid boundary to keep the \( y^+ \) below unity. The numerical solution was computed on an HPC, mostly over 120 processors for a CPU cost per time step of around 94 seconds, which means an average of about 0.33 msec. per grid point. The test is performed for a set of three high-load advance coefficient, namely, \( J=0.2 \), \( J=0.3 \) and \( J=0.4 \). The time step for all computations was set at a constant value of \( \Delta t=0.0001 \) s and the computation was performed for 25 seconds. The grid convergence test is done in terms of the thrust and torque coefficients, which are compared with those measured and reported in [13] and [14], as table 1 shows. The relative error varies from 1.4% to 5.99% for the thrust coefficient and from 0.16% to 3.87% for the torque coefficient, respectively. Based on the grid convergence test, the \( G_3 \) grid is further chosen for the POW simulation for reasons related to the level of the numerical error, in spite of the higher CPU cost implied by the large number of the discretization cells.

| \( J \) | \( K_T \) | \( 10^{-3} K_Q \) |
|-------|-------|-------|
| EFD   |       |       |
| 0.2   | 0.435 | 0.387 |
| 0.3   | 0.336 | 0.622 |
| 0.4   | 0.557 | 0.497 |
| G1    | 0.45  | 0.40  |
|   e%  | 2.90  | 3.01  |
| 0.3   | 0.57  | 0.52  |
| 0.4   | 2.96  | 2.45  |
| 0.3   | 0.64  | 0.57  |
| 0.4   | 0.55  | 0.57  |
| CFD   |       |       |
| 0.43  | 0.52  |
| 0.38  | 0.54  |
| 0.35  | 0.50  |
| 0.32  | 0.51  |
| 0.30  | 0.53  |
| 0.25  | 1.94  |
| 0.22  | 2.55  |
| 0.50  | 1.94  |
| 0.55  | 0.51  |
| 0.32  | 0.51  |
| 0.30  | 0.50  |
| 0.35  | 1.94  |
| 0.40  | 2.55  |

For the sake of having a basepoint for the present analysis, the data for the open propeller diagrams are computed at first, therefore at this point, all the computations were performed for the non-cavitating regime. The EASM is used as the turbulence closure model. Computed performances are compared with measurements [13], [14] in figure 4. The overall agreement between calculations and experiments is satisfactory. For a relatively wide range of advance coefficient, the predicted performances in terms of thrust and torque coefficients differ from the measurements by less than 6% for the lower values of advance coefficient, where the numerical predictions show a rather different slope of the performance curves. The data shown in figure 4 will serve further for a comparison with the corresponding values computed for the cavitating regime, as it will be discussed in the followings.

![Figure 4. Open water propeller diagrams for the non-cavitating regime.](image-url)
The numerical modeling of cavitation is based on the resolution of a transport equation in which source terms are added to model the vaporization and condensation of the liquid/vapour phases. The Sauer, Merkle and Kunz models are implemented in the ISIS-CFD solver. Since in the Sauer model the nuclei density which is unknown has to be provided the model has been disregarded. The justification for the choice resides in the fact that the accuracy of the inputs are very much subjected to a fortunate choice rather than to a consistently motivated one. Under such circumstances, the only remaining choice was between the Merkle and Kunz models for which the input data are the liquid product coefficient, liquid destination coefficient and cavitation reference length. Based on previous numerical simulation achievements [15], [16], the Kunz model is chosen herein for the numerical simulation.

In brief, the mathematical model used for the cavitation simulation is described in the followings. The ISIS-CFD code treats the cavitation as being based on the resolution of a transport equation similarly to what is done for free surface, although the significance of the sheet surface is completely different to that which concerns the water waves. In this particular case the source terms are used in modeling the vaporization and condensation of the two phases. If \( C_i \) is the volume fraction used to compute the evolution of the free surface, the parameter gives the volume fraction of air. Let \( C_{av} \) be a variable that defines the vapour fraction or the liquid fraction. Assuming that \( \alpha_a, \alpha_v \) and \( \alpha_l \) are the volume fractions of air, vapour and liquid, the following equation may be simply written, equation (1):

\[
\alpha_a \rho_a + \alpha_v \rho_v + \alpha_l \rho_l = \rho
\]

and the equation of mass conservation becomes, equation (2):

\[
\frac{\partial \rho}{\partial t} + \text{div}(\rho \vec{U}) = \rho_a \left( \frac{\partial \alpha_a}{\partial t} + \text{div}(\alpha_a \vec{U}) \right) + \rho_v \left( \frac{\partial \alpha_v}{\partial t} + \text{div}(\alpha_v \vec{U}) \right) + \rho_l \left( \frac{\partial \alpha_l}{\partial t} + \text{div}(\alpha_l \vec{U}) \right)
\]

Supposing that \( (1 - C_i - C_{av}) \) represents the vapour fraction \( \alpha_v \), then \( C_i = \alpha_l + (1 - C_i - C_{av}) \) represents the volume fraction of both liquid and vapour. The volume fraction of liquid is given by \( \alpha_l = C_i + C_{av} - 1 \), which may result in equation (3):

\[
\text{div}(\vec{U}) = -\left( \frac{\rho_l - \rho_v}{\rho_v} \right) \vec{n}_i = \left( \frac{\rho_l - \rho_v}{\rho_l} \right) \vec{n}_v
\]

or in equation (4):

\[
\vec{n}_v = -\frac{\rho_l}{\rho_v} \vec{n}_i
\]

where \( \vec{n}_l \) and \( \vec{n}_v \) represent the vaporization of the liquid and the condensation of the vapour, whereas \( \rho_l \) and \( \rho_v \) are the densities for the liquid and vapor, respectively. With all these particular specifications the Kunz model finally reads as follows, equation (5) and equation (6):

\[
\frac{\partial \alpha_a}{\partial t} + \text{div}(\alpha_a \vec{U}) = \bar{m}_a = \frac{C_{\text{dest}}}{0.5 \rho_l U_{\infty}^2 T_\infty} \rho_v \text{Min}(P - P_v, 0)(\alpha_i) + \frac{C_{\text{dest}}}{T_\infty} \frac{\rho_v}{\rho_l} (\alpha_l)^2 (1 - \alpha_l)
\]

\[
\frac{\partial \alpha_v}{\partial t} + \text{div}(\alpha_v \vec{U}) = \bar{m}_v = \frac{C_{\text{prod}}}{0.5 \rho_l U_{\infty}^2 T_\infty} \text{Min}(P - P_v, 0)(\alpha_j - 1) + \frac{C_{\text{dest}}}{T_\infty} \frac{\rho_v}{\rho_l} (1 - \alpha_v)^2 \alpha_v
\]
Based on [17], the default values for the parameter coefficients are 4100 for the liquid destruction term and 455 for the liquid production term.

From the numerical point of view, the computation of the cavitating propeller follows the one performed for the non-cavitating regime i.e. the cavitation is triggered by switching to the Kuntz model for a certain simulation time. In this particular case, the computation is performed for seven and a half more seconds after those 25 of the non-cavitating propeller simulation and only for three high-load advance coefficients ($J=0.2$, $J=0.3$ and $J=0.4$, respectively) at which the cavitation is likely to occur. Its inception takes place a little after 0.7 seconds after the cavitation is triggered and it originates simultaneously at the blade tips and on the leading edge of the root where fairly small nuclei were observed. The cavity evolution in time is rather fast, so that after 1 second its extension looks as depicted figure 5 that shows the coverage after a second whereas in figure 6 is represented the cavity after 1.25 seconds. Comparing the two one may easily see that the incipient cloud in the tip wake vanishes as the root cavities do. They rapidly aggregate either on the leading or on the trailing edge for a short time till the flow completely develops. Although these instabilities were also reported during the experiments in the cavitation tunnel, it is still unclear whether their computed extension has a physical explanation.

![Figure 5. Cavitation inception at t=26s, J=0.2.](image)

![Figure 6. Cavitation evolution at t=26.25s, J=0.2.](image)

Obviously, after its location stabilizes as shown in figure 7, the cavitation sheet develops in the radial direction, constantly in time, up to a certain extent from which its growth stops. This may be seen in figure 8 that bears out the outmost extension of the cavitation sheet computed after 27.5 seconds. Systematic recording of the solution proved that the additional cavitation seeds developed in the root area and shown in figure 8 appear and vanish periodically every 0.012 second. Seemingly, this is explained by the fact that the energetic conditions for the formation of new cavitation bubbles are not sufficient for their stability, therefore they cannot survive.

![Figure 7. Cavitation evolution at t=26.5s, J=0.2.](image)

![Figure 8. Cavitation development at t=27.5s, J=0.2.](image)
On the contrary, when the numerical solution stabilizes, the development of the cavitation sheet does not evolve anymore and the flow parameters remain almost constant on the propeller blade, regardless the moment the registration is made. This is depicted in figure 9 that shows the cavitation complete development after 32.5 seconds. A comparison between figure 9 and figure 8 does not reveal major evolutions although the represented solutions are computed at five seconds difference. The cavitation sheet extension covers about 32% of the total area of the blades, a fact that severely influences the energetic performances as it may be seen in figure 10 that bears out a comparison between the propeller diagrams computed for the non-cavitating and cavitation regimes. The first set of diagrams are drawn by lines doubled by symbols drawn for the experimental data, whereas the three values that corresponds to the cavitation regime are represented in coloured lines. The efficiency drop varies from 23.1% for $J=0.2$ and 5.9% for $J=0.4$, a fact that explains why the cavitation should be prevented.

**Figure 9.** Cavitation complete development at $t=32.5s$, $J=0.2$.

**Figure 10.** Open water propeller diagrams drawn under the cavitation condition.

The flow field around a propeller working in open water is characterized, as expected, by the formation and convection of strong vortices. To visualize these vortices, the isosurface of the second largest invariant computed for $J=0.2$ at $t=25$ seconds is drawn in figure 11. Four sets of vortical structures are coexisting: two tip vortex sets, a root vortex one and, of course, the straight hub vortex. The vortices are washed away in the wake and their evolution can be captured as long as the resolution of the computational grid is reasonably good. On the opposite, they are rapidly damped when the grids stretches towards the outflow. In the figure are also visible the vortices that form at the root of the blades and eventually merge into the hub vortex. After a slight initial reduction of the radial position of the vortex cores, caused by the acceleration of the flow behind the propeller, the helices formed by the tip vortices remain located on a cylinder of almost constant radius up to the end of the grid refinement block. However, when the cavitation model is triggered, things changes substantially as figure 12 shows. Figure 12 depicts the vortical structures computed at $J=0.2$ at $t=26$ seconds.

**Figure 11.** Vortical structures in the propeller wake computed for $J=0.2$ at $T=25$ seconds.

**Figure 12.** Vortical structures in the propeller wake computed for $J=0.2$ at $T=26$ seconds.

It is worth mentioning that the cavitation sheet development on the blades determines several changes that take place in the propeller wake. Instead of having two sets of tip vortices, it remains only one,
which becomes thicker and shorter. This is because of the energy loss due to the cavitation. Similarly, the root vortices undergo the same modification. On the contrary, the hub vortex gains in thickness and get twisted. This behaviour is rather unstable in time and apparently follows the fast changes produced by the cavitation evolution, as figures 13 and 14 show. At $t=26.5$ sec all the three vortical systems become weaker and the blade-produced helices suffer an increase of the pitch, as figure 11 bears out. The energy loss will further create all the conditions for the vortices to be shed in the wake. Once this happens, another triple vortical system generates, see figure 14, and the process described above continues. The numerical reconstruction of the unsteady computed solution recorded every time step has proven that the generation-destruction process is periodic having a frequency that depends on the propeller rpm rate.

![Figure 13. Vortical structures in the propeller wake computed for $J=0.2$ at $t=26.5$ seconds.](image)

![Figure 14. Vortical structures in the propeller wake computed for $J=0.2$ at $t=27$ seconds.](image)

To have at least a qualitative measure of the level of resolution obtained with the grid used for the simulations, the amount of resolved kinetic energy is computed in the wake of the propeller to get an estimation of the numerical solution accuracy, knowing that in the Boussinesq’s approximation of turbulent stresses, the modelled kinetic energy is absorbed in the pressure terms and cannot be discerned from it unless the discretization in space and time is extremely fine. For that purpose, figure 15 shows the turbulent kinetic energy (TKE hereafter) contours drawn in propeller and the symmetry planes of the computational domain. The figure reveals not only the high intensity of the TKE peaks just behind the propeller that are in phase with the vortices shed in the downstream, but also their strong diffusion in the flow. From this point of view, the author believes that the method still has a promising potential for studying, within the range of scales resolved by RANS. Nevertheless, because of the averaging, the unsteady effects do not seem be completely resolved with the RANS approach, therefore a DES or LES-based simulation may be a better alternative despite their prohibitive CPU costs.

Same conclusion may be withdrawn from figure 16 which shows the axial velocity contours drawn in the same cutting planes as in figure 15. Worth mentioning that there is no phase shift between the velocity peaks and the corresponding vortex cores, a fact that may suggest the physical consistency of the solution. The vortical structure of the flow has an important influence not only on the propeller itself, but also on the rudder efficiency since the control surface of the governing system is working in the propeller wake therefore the fluctuating velocity and pressure fields in the wake will determine fluctuations in the lift force and moment at the rudder stock.
5. Conclusions
The paper describes a numerical method suited for studying the flow around the five-blade KP505 propeller model working under the cavitation assumption. Several POW computations for the non-cavitating regime are performed at first for verification and validation purposes and the comparisons with the experimental available data revealed a good agreement.

The use of the Kuntz method of the ISIS-CFD code made possible not only to simulate the flow under the cavitation conditions, but also to capture the extension of the cavitation sheet on the blades. Moreover, the numerical solution allowed the study of the instability of the vortical structures behind the propeller, a fact that may prove the robustness of the proposed method. Given the complexity of the phenomena, more detailed studies are necessary to completely clarify the physics behind the particular conditions of such a multi-phase flow.

6. References
[1] Lee C S 1979 Prediction of steady and unsteady performance of marine propellers with or without cavitation by numerical lifting surface theory, Ph.D. dissertation, Massachusetts Institute of Technology
[2] Kinnas S A and Fine N E 1993 A numerical nonlinear analysis of the flow around two- and three-dimensional partially cavitating hydrofoils J. Fluid Mech. 254 151–181
[3] Deshpande M., Feng J and Merkle C L, 1994, Cavity flow predictions based on the Euler equations J. Fluids Eng. 116(1) 36–44
[4] Kinnas S A, Choi J-K, Lee H, Young Y L, Gu, H, Kakar K and Natarajan S 2002, Prediction of cavitation performance of single or multi-component propulsors and their interaction with the hull Transactions of Soc. Nav. Archit. Mar. Eng. 110 215–244
[5] Senocak I and Shyy W 2002 A pressure-based method for turbulent cavitating flow computations J. Comput. Phys. 176 363–383
[6] Singhal A K, Athavale M M, Li, H Y and Jiang Y. 2002 Mathematical basis and validation of the full cavitation model J. Fluids Eng. 124(3) 617–624
[7] Venkateswaran S, Lindau J W, Kunz R F and Merkle C L 2002 Computation of multiphase mixture flows with compressibility Effects J. Comput. Phys. 180 54–77
[8] Lindau J W, Venkateswaran S, Kunz R F and Merkle C L 2003 Computation of compressible multiphase flows AIAA paper 2003-1285, Proc. 41st AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV
[9] Salvatore F, Streckwall H, and van Terwisga T, 2009 Propeller Cavitation Modelling by CFD - Results from the VIRTUE 2008 Rome workshop Proc. Int. Symp. on Marine Propulsors SMP09 Trondheim 362-372
[10] Pereira F, Salvatore F and Di Felice F 2004 Measurement and modelling of propeller cavitation in uniform inflow J. Fluids Eng. 126 671–679
[11] Bensow R E and Bark G 2010 Implicit LES predictions of the cavitating flow on a propeller J. Fluids Eng. 132 1–10
[12] Klasson O and Huuva T 2011 Potsdam propeller test case (PPTC) Proc. Int. Symp. on Marine Propulsors SMP2011 Hamburg 43-50
[13] Larsson L, Stern L, and Vissonneau M 2013 Numerical Ship Hydrodynamics: An assessment of the Gothenburg 2010 Workshop Springer
[14] National Maritime Research Institute (NMRI) 2015 Tokyo 2015: A Workshop on CFD in Ship Hydrodynamics http://www.t2015.nmri.go.jp
[15] Lungu A and Ungureanu C 2008 Numerical study of a 3-D juncture flow AIP Conference Proceedings 1048(1) pp 839-842
[16] Ungureanu C and Lungu A 2009 Numerical simulation of the turbulent flow around a strut mounted on a plate AIP Conference Proceedings 1168(1) 689-692
[17] Morgut M, Nobile E, Bilus I 2011 Comparison of mass transfer models for the numerical prediction of sheet cavitation around a hydrofoil Int. J. of Multi-phase Flow 37(6) 620-626