Numerical estimation of cavitation intensity

L Krumenacker 1, R Fortes-Patella 1, A Archer 2

1 Univ. Grenoble Alpes, LEGI, F-38000 Grenoble, France
2 EDF R&D, Département MFEE, 6 quai Watier 78401 Chatou CEDEX, France
E-mail: laurent.krumenacker@legi.grenoble-inp.fr

Abstract. Cavitation may appear in turbomachinery and in hydraulic orifices, venturis or valves, leading to performance losses, vibrations and material erosion. This study propose a new method to predict the cavitation intensity of the flow, based on a post-processing of unsteady CFD calculations.

The paper presents the analyses of cavitating structures’ evolution at two different scales:
• A macroscopic one in which the growth of cavitating structures is calculated using an URANS software based on a homogeneous model. Simulations of cavitating flows are computed using a barotropic law considering presence of air and interfacial tension, and Reboud’s correction on the turbulence model.
• Then a small one where a Rayleigh-Plesset software calculates the acoustic energy generated by the implosion of the vapor/gas bubbles with input parameters from macroscopic scale.

The volume damage rate of the material during incubation time is supposed to be a part of the cumulated acoustic energy received by the solid wall.

The proposed analysis method is applied to calculations on hydrofoil and orifice geometries. Comparisons between model results and experimental works concerning flow characteristic (size of cavity, pressure, velocity) as well as pitting (erosion area, relative cavitation intensity) are presented.

1. Introduction
In industrial conditions, cavitation may occur in orifices, venturis, valves or turbomachinery. It leads to vibration, performance loss and can damage the material. A reliable numerical prediction of the cavitating flow intensity could help improving the turbomachinery design and/or identifying the locations requiring a specify surface treatment, to avoid mass loss, performance decrease and maintenance costs.

The cavitation erosion phenomenon is induced by fast and strong implosions of cavitating structures near the solid. Generally, these implosions occur at time and spatial scales far lower than turbomachinery flow simulations ones. As a matter of fact, URANS codes, used in the industry nowadays and based on averaged methods, are not able to simulate both phenomena simultaneously. Furthermore, the pressure waves responsible for the erosion depend on the partial pressure of non condensable gases in bubbles. The flow is not only a two phases system (gas
and liquid water, for example) but contains also multiple species (such air).

Despite this complexity, several authors have proposed methods to predict cavitation erosion from numerical simulations for industrial purposes. We have classified these methods in three different methodologies:

**Energetic approach:**
Fortes-Patella [1] and Nohmi [2] have proposed an energetic approach to conciliate both scales of the flow. The results obtained by URANS simulations can be used to define the energy contained in the gaseous phase. The temporal variation of this energy is supposed to be proportional to the acoustic energy propagating in the flow.

**Refined simulations:**
In the CATUM code developed by Thalhamer [3], a three equations system based on a compressible Euler model and using explicit time discretization was implemented. It simulates cavitating flows and the propagation of emitted pressure waves through the fluid. The temporal and spatial discretization are much refined than classical URANS simulations and can take into account bubble implosions phenomenon.

**Two simulations approach:**
Ochiai and Van Loo have proposed methods in which URANS softwares simulate the flow around the geometry meanwhile cavitation intensity is estimated from bubble simulations.

- In Ochiai’s work [4], this method is based on a one way coupling system between both scales of the flow. The URANS software uses a five equations models (conservation of mass, momentum, energy of an homogeneous fluid and the continuity equations of the mixture gas and non condensable gas of air). At small scales, bubbles are convected within a motion equation. A cavitation erosion intensity is calculated through Keller’s equation [5].
- Van Loo [6] uses an in-house solver, FRESCO +, to compute the macroscopic flow. It uses isothermal Navier-Stokes equations coupled with an Euler-Lagrangian approach for the transport of gaseous structures in the fluid [7]. At each time step, the software calculates the acoustic pressure waves in the fluid and characterizes the cavitation erosion intensity.

Such methods propose different solutions to predict the cavitation erosion. Furthermore, none of them consider the surface tension at macroscopic scale. The presence of non condensable gases, is also not always modeled. This study is part of the third approach, aiming to propose a new method considering both of these phenomena from isothermal Navier-Stokes equations by using homogeneous fluid and a barotropic law.

### 2. Method Presentation

The presented method can be described by three steps. Each step depends on the previous one without changing it.

(i) The macroscopic flow is calculated through a URANS software considering air content and surface tension.

(ii) Afterward, in each cell of the mesh, and at each time step, the solution of the macroscopic flow provides initial and boundary conditions for bubbles simulations. From these calculations, acoustic energies propagating in the flow could be evaluated.

(iii) Finally, these acoustic energies reaching the solid surface are accumulated and provide a surface acoustic power characterizing the cavitation erosion intensity.
2.1. Macroscopic simulation

In this model, non-condensable gases are supposed represented by only one specie considered as air. It coexists with vapor and water respectively as gas and dissolved component. With these considerations, four fields of density and velocity, as well as two temperature fields and an interface motion have to be computed to solve the equation system.

To simplify the system, we will consider a mixture and an homogeneous fluid including respectively the two species and the two phases of the flow. For the notations, the signs $a$, $w$ and $M$ design respectively the air, water species and the mixture fluid, and the exponent $l$ and $g$ design the liquid and the gaseous phase.

2.1.1. Physical models

To respect the conservation of mass and of momentum, the density, $\rho^k_M$, and the velocity, $u^k_M$, of the mixture fluid is defined as the sum of those of each species and can be written as a function of the mass ratios $Y^k_i$:

$$\rho^k_M = \rho^k_a + \rho^k_w, \quad Y^k_i = \frac{\rho^k_i}{\rho^k_M}, \quad u^k_M = Y^k_w u^k_w + Y^k_a u^k_a, \quad k = \{l, g\} \quad i = \{a, w\} \quad (1)$$

Moreover, the density $\rho_M$, the velocity $\mathbf{u}_M$ and the pressure $p_M$, of the homogeneous fluid are determined by the presence ratio, $\alpha_k = \frac{V^k_k}{V_{tot}}$, and the pressure $p^k_M$ of each phase:

$$\rho_M = \alpha_g \rho^g_M + \alpha_l \rho^l_M, \quad \rho_M \mathbf{u}_M = \alpha_g \rho^g_M \mathbf{u}^g_M + \alpha_l \rho^l_M \mathbf{u}^l_M, \quad p_M = \alpha_g p^g_M + \alpha_l p^l_M \quad (2)$$

To make the system computable for industrial purposes, we apply several physical models at the macroscopic scale.

- First of all, we consider that there is not sliding velocity at the interface, meaning, the averaged velocities of the two phases are equals.
- The isothermal Navier-Stokes equations are used. It implies that the effect of the compression and of the mass transfer at the interface on the temperature field can be neglected.
- Considering that mass transfers and the diffusion in each phase are faster than the convection of the bubbles, we can establish the partial pressure of the species in the gaseous phase. The equilibrium state of this model is based on:
  - the saturation vapor pressure for the water
  - the Henry’s law for the air ($H_e$ is the Henry constant)
\[ p^g_w = p_{sat} \quad p^g_a = \frac{\rho^l_a}{H_e} \]  

• The perfect gas law is used for gaseous phase. Moreover, the gas pressure can be defined by the sum of the partial pressures. \((r_i\) being the specific gas constant for each species) 

\[ p^g_i = \rho_i r_i T^g \quad i = \{a, w\} \quad p^g_M = p^g_a + p^g_w \]  

• According to the Tait law, the variation of the density of the water in the liquid in used range at macroscopic scale (1 - 10^6 Pa) is very low and can be assumed as constant. 

\[ \rho^l_w = \text{constante} \]  

• Considering that the mass fraction of air, \(Y_a\), is constant at initial condition and at the boundaries of the simulation (where the fluid is supposed totally liquid), the physical models used in this method imply that it is constant in the whole field. The parameter value depends on the water quality. 

\[ Y_a = \left( \frac{\rho^l_a}{\rho^l_a + \rho^l_w} \right)_{\text{inlet}} = \text{constante} \]  

This paper presents results for a molecular concentration of air of 15 ppm corresponding to \(Y_a = 2.3 \times 10^{-5}\).

2.1.2. Interfacial consideration

Most of homogeneous models and especially those used in methods presented previously, consider equality between the pressure of both phases in the mixture area. This assumption cancels the interface’s influence in the equation of momentum, and so prevent the determination of bubble sizes from macroscopic simulations. This work proposes to consider a pressure field \(p_{MI}\), including the pressure of the both phases and interfacial forces \(f_I\). \(<.>\) and \(\delta_I\) corresponding respectively to the average operator used in URANS simulation and to a Dirac distribution representing the interface

\[ \nabla p_{MI} = \nabla p_M - <f_I \delta_I> \]  

Interface forces can be related to the mean curvature of the interface, \(H^g_I\), and to the surface tension, \(\sigma\). Furthermore, according to Ishii’s work, presented in [8], the average of the normal vector at interface \((n^g_I)\) can be related to the gradient of void ratio \(\alpha_g\).

\[ f_I = 2\sigma H^g_I n^g_I \quad , \quad \nabla \alpha_g = -<n^g_I \delta_I> \]  

Neglecting the viscosity and the effect of the mass transfer in the momentum jump condition, the mean curvature of the interface can be related to the pressure on both sides and provides a link between the macroscopic and small scales through the shape of the gaseous structures.

\[ 2\sigma H^g_I = p^l_M - p^g_M \quad \nabla p_{MI} = \alpha_g \nabla p^g_M + \alpha_l \nabla p^l_M \]  

2.1.3. Presentation of the barotropic law

In this work, an equation based on the barotropic law proposed by Delannoy [9] is used to determine the void ratio to the pressure field \(P_{MI}\). It is characterized by two parameters, the
equilibrium vapor pressure $p_{sat}$ and the minimum speed of sound in the homogeneous flow, $c_{min}$, an adjustment parameter of the law, ($c_{min} = 1 m.s^{-1}$ in the presented work).

$$\alpha_g = \begin{cases} 
\frac{1}{2} \left[ 1 - \sin \left( \frac{p_{MI} - p_{law}}{c_{min}^2} \frac{2}{\rho_{M0}^g - \rho_{M0}^l} \right) \right] & \text{if } p_{M}^{inf} < p_{MI} < p_{M}^{sup} \\
0 & \text{if } p_{MI} \geq p_{M}^{sup} \\
1 & \text{if } p_{MI} \leq p_{M}^{inf}
\end{cases} \tag{10}$$

With:

- $\rho_{M0}^{l} = \rho_{law}^{l}$ and $\rho_{M0}^{g} = \rho_{law}^{g}$, the densities of the pure phases.
- $p_{M}^{inf} = p_{sat} \left[ 1 + \frac{Y_a \cdot r_a}{Y_w \cdot r_w} \right]$ and $p_{M}^{sup} = p_{M}^{inf} + \frac{\pi}{2} \left( \rho_{M0}^{l} - \rho_{M0}^{g} \right) c_{min}^2.$
- $p_{law} = p_{M}^{inf} + \frac{\pi}{4} \left( \rho_{M0}^{l} - \rho_{M0}^{g} \right) c_{min}^2$ and $p_{M}^{sup} = p_{M}^{inf} + \frac{\pi}{2} \left( \rho_{M0}^{l} - \rho_{M0}^{g} \right) c_{min}^2.$

The definition of the homogeneous fluid density (Eq. 2) and the barotropic law (Eq. 10) gives a direct relation between the density and the pressure of the homogeneous fluid (Fig. 2).

The barotropic law proposed by Delannoy assumes that condensation and evaporation are two symmetrical processes with respect of saturation pressure. However, to ensure the continuity of densities and pressure with the physical models used in this work, the metastable zone has to be shifted above the saturation pressure, considering that when pressure is bigger than saturation pressure, all the liquid has been vaporized. The influence of the air concentration is very weak in our application ($Y_a \approx 10^{-5}$).

![Figure 2](image-url)  
**Figure 2.** Relation between the density and the pressure for applied homogeneous barotropic cavitation model and for barotropic law proposed by Delannoy

All of these physical considerations reduce the system at macroscopic scale, to only two equations: an equation of mass conservation (Eq. 11) and an equation of conservation of momentum (Eq. 12).

$$\frac{\partial}{\partial t} [\rho_{M}] + \nabla \cdot (\rho_{M} \mathbf{u}_{M}) = 0 \tag{11}$$

$$\frac{\partial}{\partial t} [\rho_{M} \mathbf{u}_{M}] + \nabla \cdot [\rho_{M} \mathbf{u}_{M} \mathbf{u}_{M}] = -\nabla (p_{MI}) + \nabla (\mathbf{\tau}) + \nabla (\mathbf{\tau}) \tag{12}$$

$\mathbf{\tau}$, $\mathbf{\tau}$ represent respectively the viscous constraint tensor and the Reynolds tensor.
2.2. Small scales
The proposed method considers that phenomenon producing the material damage of erosion comes from fast and strong pressure variations due to bubbles cloud collapses, generally occurring in the end of cavitation sheet and during cavitation shedding [10]. To characterize the aggressiveness of the flow, we can simulate the dynamic of the flow at small scales in these sensitive areas, i.e., where the Lagrangian variation of pressure is high. The bubbles simulations are performed in the mixture area, where the Lagrangian time differentiation of liquid pressure is positive \( \frac{dp_{l}}{dt} > 0 \), i.e., where the bubbles undergo implosions.

In each cell of the computational mesh and for each time step, the macroscopic simulation provides initial conditions for the small scales computations (cf table 1). Initial bubble radius \( R_0 \) (opposite of mean curvature’s inverse \( H^{-1} \)), and number of bubbles \( N_B \) can be computed from those parameters.

\[
R_0 = \frac{2\sigma}{p_{M}^I - p_{M}^I} \quad N_B = \frac{\alpha_g V_{cell}}{\frac{4}{3} \pi R_0^3}
\]

The barotropic law makes it possible to relate all these parameters to the void ratio (Fig. 3).

![Figure 3. Pressure in each phase and in the homogeneous flow and bubble radius depending on the void ratio for \( Y_a = 2 \cdot 3 \cdot 10^{-5} \)](image)

Several authors, like Lord Rayleigh [11] or Keller [5], proposed equations system to characterize the behavior of an isolated spherical bubble. The proposed method uses Keller’s equation to determine the pressure wave created during the collapse of a bubble. The compressibility of the liquid essential to the acoustic wave propagation is taken into account.
at small scales. Moreover, the air transfer through the interface is neglected.

\[
\left(1 - \frac{1}{c_\infty} \frac{dR}{dt}\right) R \frac{d^2 R}{dt^2} + \frac{3}{2} \left(1 - \frac{1}{3c_\infty} \frac{dR}{dt}\right) \left(\frac{dR}{dt}\right)^2 = \left(1 + \frac{1}{c_\infty} \frac{dR}{dt}\right) \frac{p_I^l - p_\infty^l}{\rho_\infty^l c_\infty^l} + \frac{R}{\rho_\infty^l c_\infty^l} \frac{d}{dt} \left(\frac{p_I^l - p_\infty^l}{\rho_\infty^l c_\infty^l}\right)
\]

(14)

The liquid pressure at the interface \(p_I^l\) is defined as:

\[
p_I^l = p_{sat} + p_{g0}^a \left(\frac{R}{R_0}\right)^{3\kappa} - \frac{2\sigma}{R} - \frac{4\mu_M}{R} \frac{dR}{dt}
\]

(15)

With \(R\) the radius of the bubble, \(\mu_M\) the dynamic viscosity of the liquid, \(\sigma\) the surface tension, \(\kappa\) the ratio of specific heats and, \(c_\infty\) the sound celerity in the liquid.

This system provides the pressure in all the field around the bubble.

2.3. Propagation of acoustic power

The acoustic energy emitted by a single bubble can be determined by the temporal integration of the acoustic intensity calculated by Keller’s equation on a closed surface surrounding the bubble (\(n_S\) being the normal vector of this surface). The fluid is supposed isentropic so the energy calculated does not depend on the surface \(S\) chosen.

\[
E_{ac} = \int_0^\infty \left( \int_S \left( p_I^l - p_\infty^l \right) u_M^l \cdot n_S dS \right) dt
\]

(16)

The acoustic energy emitted by the cloud is supposed proportional to the one emitted by a single bubble. This assumption does not take into account collective effect which can appear in bubble cloud implosions but works in progress should be able to improve this aspect. The number of bubbles undergoing the pressure variation by second is estimated by the number of bubbles leaving the cell by second, \(\dot{N}_{imp}^B\) and depends on the macroscopic velocities of the fluid and the mesh of the URANS computation (\(n_c\) being the normal vector of a surface of the considered cell).

\[
\dot{N}_{imp}^B = \frac{N_B}{V_{cell}} \int_{\partial cell} \max(0, u_M \cdot n_c) dS
\]

(17)

For each cell of the mesh, an acoustic power, assumed propagating spherically and in all directions, can be determined.

\[
P_{ac}(cell) = \dot{N}_{imp}^B E_{ac}
\]

(18)

Solid angle is used to determine the part of this energy reaching a solid wall surface \(S_w\) (see Fig. 4). The accumulation of acoustical energies emitted from all the cells of the mesh and reaching this surface represents the cavitation intensity of this method, \(\dot{W}_{CI}\).

\[
\dot{W}_{CI}(S_w) = \sum_{cell} \frac{P_{ac}(cell)}{S_w}
\]

(19)
3. Results
This method has been used on different geometries. This paper presents the results on a NACA and an orifice geometry tested respectively by the EPFL, École Polytechnique Fédérale de Lausanne and by EDF R&D. The URANS simulations are computed by two different codes: Fine-Turbo, a commercial software developed by Numeca International and IZ, a in-house software, developed in LEGI for the CNES. The description of cavitation module of these CFD softwares are available respectively in [12] and in [13]. Both of them are used with a $k - \epsilon$ model coupled with the Reboud’s correction, with the parameter $n = 10$ [14], for the turbulence model.

Both experimental studies are based on the same principle: samples of different materials are disposed on the studied geometry surface and exposed to a cavitating flow for a short time period, $t_{\text{exp}}$. Afterward, a 3D laser profilometry technique is used to measure the total volume pitted $V_{\text{pit}}$ on a surface $S_{\text{exp}}$ and a local volume damage rate $V_d$ can be established [15].

$$ V_d = \frac{V_{\text{pit}}}{t_{\text{exp}} S_{\text{exp}}} $$

(20)

In the present study, only qualitative appreciations are made between simulations and experiments. Indeed, at this time, only the energy received by the wall is determined by the model and no material properties are taken into account.

3.1. EPFL’s NACA profil
The considered hydrofoil is a NACA 65012 with a chord of 100 mm length and a span of 150 mm placed with a 6 degrees incidence. The experimental tests have been made by Pereira [10] in the EPFL. This paper presents the results for an inlet velocity of $u_{\text{in}} = 15 m.s^{-1}$ and for two cavitation parameters $\sigma_c \approx \{1.5, 1.8\}$. Apparition of cloud shedding in simulations can be observed on figure 5.

$$ \sigma_c = \frac{p_{\text{in}} - p_{\text{sat}}}{\frac{1}{2} \rho_M u_{\text{in}}^2} $$

(21)
Figure 5. Cloud shedding above the EPFL profil for $\sigma_c = 1.5$, at a certain time

Figure 6 shows the cavitation erosion intensity obtained from IZ simulations. The damaged area evaluated from the IZ simulations are well positioned and the quantitative variations between the two different cavitation numbers are respected between the experiments and the simulations.

Figure 6. Numerical estimation of the time averaged acoustic power applied on the wall and experimental volume damage rate measured along chord of the NACA profil

3.2. EDF R&D’s orifice
The geometry of an orifice has been also studied with the Fine-Turbo software: the orifice internal diameter is $D = 200 mm$ and the ratio between orifice’s diameter and the internal diameter is 0.4. It is mounted in the middle of a long straight pipe (42D long). The flow is assumed to be axisymmetric and a two-dimensional cylindrical axisymmetric meshing is used. The experimental tests have been performed using the EPOCA facility at EDF R&D [16]. The samples are placed on the internal surface downstream of the orifice (Fig. 7). The test conditions are controlled using two independent parameters, the flow rate and the outlet pressure of the computation. The results for two different conditions are presented.
The formation of cavitation shedding observed in experiments is obtained by the simulations. The predicted damaged areas match in both cases with the experimental results (Fig. 8). Moreover, the quantitative variations also correspond between the experiments and the simulation for the analyzed cases.

4. Conclusions
This method of prediction of the cavitation intensity presents the interest of taking into account the surface tension and the air content in the bubble. However, the use of state of laws limits the physic phenomena and does not consider the latency occurring in mass transfers or in molecular diffusion. Comparisons with simulation using the void ratio transport equation should be an interesting perspective.
Concerning the cavitation erosion, the prediction of the damaged area’s location depends mainly of the reliability of the macroscopic flow simulation. In simulations where the topology of the mainstream is respected (cavitation sheet, shedding,... Readers may refer to [17]) qualitative results are quite satisfying. However, this method does not take into account the amplification effect caused by the presence of several bubbles. A multi-bubbles software is being developed and should be able to estimate the amplification of the pressure wave in bubble clouds collapse.

Regarding the quantitative cavitation intensity, future work has to be done to link the acoustic energy received to pits formed on the surface considering material properties.

Acknowledgments
Authors want to thanks the EDF R&D and CETIM to finance this project and the CNES for the use of the code IZ. The laboratory LEGI is part of the LabEx Tec 21 (Investissements d’Avenir - grant agreement n°ANR-11-LABX-0030)

References
[1] Fortes-Patella R., Archer A., Flageul C. 2012, Numerical and experimental investigations on cavitation erosion, Proceeding of 26th IAHR
[2] Nohmi M., Ikoagi T., Iga Y., 2008, Numerical prediction method of cavitation erosion, Proceedings of FEDSM2008, Jacksonville, USA
[3] Thalhamer M., Schmidt S., Mihauch M. and Al, 2012, Numerical simulation of sheet and cloud cavitation and detection of cavitation erosion, 14th Int. Symp. on Transport Phenomena and Dynamics of Rotating Machinery, ISROMAC-14
[4] Ochiai N., Iga Y., Nohmi M and al., 2010, Numerical prediction of cavitation erosion intensity in cavitating Flows around a Clark Y 11.7% Hydrofoil, Journal of Fluid Science and Technology 5-3, 416-31
[5] Keller J. B. and Kolodner LL, 1956, Damping of underwater explosion bubble oscillations, Journal of Applied Physics, Vol. 27-10 1152-61
[6] Van Loo S., Van Terwisga T.J.C., Hoeijmakers H.W.M. and al, 2012, numerical study on collapse of a cavitating cloud of bubbles,Proc. of the 8th int. symp. on cavitation, CAV-2012
[7] Yakubov S, Cankurt B. and al., An advanced Euler-Lagrange approach to numerical simulation of cavitation engineering flows, Proc. of the 8th int. symp. on cavitation, CAV-2012
[8] Yoon, Ishii and al., 2006, Choking flow modeling with mechanical and thermal non-equilibrium, Int. Journal of Heat and Mass Transfer, 49, 171-86
[9] Delannoy Y, 1989, Modélisation d’écoulements instationnaires et cavitants, Phd thesis of the Institut National Polytechnique de Grenoble
[10] Pereira F, 1997, Prédiction de l’érosion de cavitation: Méthode énergétique, Phd thesis of école polytechnique fédérale de Lausanne
[11] Rayleigh, 1917, On pressure developed in the fluid during the collapse of a spherical cavity, Philosophical Magazine,34-200.
[12] Pouffary B., 2004, Simulation numérique d’écoulements 2D/3D cavitants, stationnaires et instationnaires: Analyse spécifique pour les turbomachines, Phd thesis of the Institut National Polytechnique de Grenoble
[13] Coutier-Delgosha O., Fortes-Patella R., Reboud J.L., 2003, Evaluation of the turbulence model influence on the numerical simulations of unsteady cavitation, J of Fluids Eng., 125 38-45.
[14] Reboud J.L., Stutz B. and Coutier O., 1998, Two-phase flow structure of cavitation : experiment and modeling of unsteady effects, Proc. of the 3rd int. symp. on cavitation, CAV-1998
[15] Fortes Patella R., Reboud J.L., Archer A., 2000, Cavitation damage measurements by 3D laser profilometry, Wear 246 5967.
[16] Archer A., 2002, A predictive model for cavitation erosion downstream orifices, Proceedings of ASME, Montreal
[17] Reboud J.L., Coutier-Delgosha O. and Al., Numerical simulation of unsteady cavitating flows: Some applications and open problems, Proc. of the 5th int. symp. on cavitation, CAV-2003