Numerical simulation of unsteady cavitating flows using a fractional step method preserving the minimum/maximum principle for the void fraction.

R Chebli¹,², O Coutier-Delgosha¹ and B Audebert ²

¹ Laboratoire de Mécanique de Lille (LML, UMR 8107) / Arts et Métiers ParisTech, 8 Boulevard Louis XIV, 59046 Lille, France.
² EDF R&D, Département Mécanique des Fluides Energies et Environnement, 6, quai Watier, BP 49, 78401 Chatou, France.

Abstract. Cavitation is one of the most demanding physical phenomena influencing the performance of hydraulic machines. It is therefore important to predict correctly its inception and development, in order to quantify the performance drop it induces, and also to characterize the resulting flow instabilities. The aim of this work is to develop an algorithm for the numerical simulation of cavitation in an industrial CFD code (Code_saturne). It is based on a fractional step method which preserves the minimum/maximum principle of the void fraction. An implicit solver, based on a transport equation of the void fraction coupled with the Navier-Stokes equations is proposed. A specific numerical treatment of the cavitation source term provides physical values of the void fraction (between 0 and 1) without including any artificial numerical limitation. The influence of RANS turbulence models on the simulation of cavitation on a 2D Venturi type geometry is then studied. It confirms the capability of the k-ε model and the k-ω SST model with the modification proposed by Reboud et al. (1998) to reproduce the main features of the unsteady sheet cavity behaviour.

1. Introduction
Cavitation consists in the appearance of vapour in a liquid subjected to a local depression. An algorithm based on a fractional step method implemented in an industrial CFD code (Code_saturne, Archambeau et al. 2004), which preserves the minimum/maximum principle for the void fraction, is presented in this paper. An implicit solver, based on a transport equation of the void fraction coupled with the Navier-Stokes equations for a single homogeneous fluid, is proposed. A specific treatment of the cavitation source term provides physical values of the void fraction (between 0 and 1) without adding any artificial numerical limitation. The influence of the turbulence modelling is then studied. For that purpose, several turbulence models based on the RANS approach are tested and compared.

2. Cavitation modelling
The internal structure of the two-phase flow is extremely complex and variable, particularly in the case of cavitation. The precise motion and size of small bubbles are often unknown. In addition, capturing numerically all bubble interfaces would require a huge number of cells and thus unaffordable computational resources. This is why the structure of both phases is considered at a macroscopic scale (Coutier-Delgosha et al. 2007). It means that each control volume Ω contains a volume fraction of...
liquid and a volume fraction of vapour. The proportion of vapour in this cell is defined macroscopically by a void fraction \( \alpha \):

\[
\alpha = \frac{\text{Vapour volume in } \Omega}{\text{Total volume of } \Omega}.
\]

(2.1)

The fluid is thus considered as homogeneous at the scale of the mesh size, with a mixture density \( \rho \) is given by:

\[
\rho = \alpha \rho_v + (1 - \alpha) \rho_l
\]

(2.2)

where \( \rho_v \) and \( \rho_l \) are densities of vapour and liquid phases respectively.

Under the assumption of non-compressibility of pure vapour and pure liquid phases (\( \rho_l \) and \( \rho_v \) are considered as constants \( \rho_l^0 \) and \( \rho_v^0 \)), the velocity divergence may be written as a function of the cavitation source term. The system to be solved is then:

\[
\text{div}(\vec{U}) = \Gamma \left( \frac{1}{\rho_v^0} - \frac{1}{\rho_l^0} \right).
\]

(2.3)

\[
\frac{\partial}{\partial t}(\rho \vec{U}) + \text{div}(\rho \vec{U} \otimes \vec{U}) = -\text{grad} P + \text{div} \vec{\tau},
\]

(2.4)

\[
\frac{\partial \vec{u}}{\partial t} + \text{div}(\vec{u} \otimes \vec{u}) = -\frac{\Gamma}{\rho_l^0},
\]

(2.5)

\( P \) is the mixture pressure of the liquid/vapour mixture, \( \vec{U} \) is the mixture velocity, \( \vec{\tau} = \mu(\text{grad} \vec{U} + \frac{2}{3} \mu \text{div} \vec{U}) \) is the stress tensor with the mixture dynamic viscosity defined by \( \mu = \alpha \mu_v^0 + (1 - \alpha) \mu_l^0 \), where \( \mu_v^0 \) and \( \mu_l^0 \) are the two constant dynamic viscosities of the pure vapour and liquid phases respectively. \( \Gamma \) is the cavitation source term and in this study the cavitation model given by Merkle is used in the numerical applications (Merkle et al, 1998).

3. Resolution method

The proposed algorithm is based on a rather classical fractional step method. The numerical resolution of the Navier-Stokes equations is performed by decomposing the operators into less complex ones using sub-intermediate steps in the same physical time step (Guerrmond et al, 2006). Two sub-steps are performed: the first one treats the convective and the diffusive parts, as well as the possible source terms of the momentum equation. It constitutes the velocity prediction step. The second sub-step, known as the pressure correction or the velocity projection step, resolves the mass equation. The boundary conditions are not considered hereafter in the discretization for clarity.

3.1. Velocity prediction step

The first step consists in solving the momentum equation. Shifted discretization with a first-order temporal scheme is applied to the unsteady term, while the pressure gradient is treated explicitly.

\[
\frac{[\Omega_i]}{\Delta t} (\rho^n \vec{U}^*_{i} - \rho^{n-1} \vec{U}^*_{i}) + \sum_{j \in \text{neighbours}(i)} \left[ (\rho^n \vec{U} \cdot \vec{n})^n \vec{U}^*_{ij} \right] S_{ij} - \sum_{j \in \text{neighbours}(i)} \left[ \mu^n \text{grad} \vec{U}^* \cdot \vec{n} \right] S_{ij} = - (\text{grad} P^n) \frac{[\Omega_i]}{\Delta t} + \sum_{j \in \text{neighbours}(i)} \left[ \mu^n \text{grad} \vec{U} \cdot \vec{n} \right] S_{ij}.
\]

(3.1)

where \( [\Omega_i] \) is the volume of the cell (I), \( \vec{U}^* \) is the predicted mixture velocity, \( S_{ij} \) is the common surface between cells (I) and (J), \( \vec{n}_{ij} \) is the normal vector to the common surface between cells (I) and (J), pointing outward the cell (I), \( \Delta t \) is the time step. For the sake of simplicity, the term \(-\frac{2}{3} \mu \text{div} \vec{U} \) of the diffusive operator is not discretized here even if it is discretized in the code (see Archambeau, 2004).
The index \( n \) refers to the previous time step. An upwind scheme with respect to the mass flux sign is used for the convective flux. The elliptic operator for the implicit part of the diffusion term is discretized with a two point flux approximation and gradient reconstructions to handle non orthogonalities, and is solved thanks to a classical deferred correction method (see Archambeau et al. 2004) for more details on the spatial discretization used in Code_saturne and in particular for the spatial discretization of the pressure gradient).

### 3.2. Pressure correction step

The equation for pressure correction is obtained by subtracting the equation of velocity prediction, already resolved at previous step, from the momentum equation written at time \( n+1 \) with an implicit pressure gradient:

\[
\sum_{j \in \text{neighbours}(i)} \frac{\Delta t}{|\Omega_i|} \frac{1}{\rho} \text{grad} (\delta P) \cdot \bar{n} \bigg] \cdot S_{ij} = \sum_{j \in \text{neighbours}(i)} \bigg[ \bar{U}^{n+1} \cdot \bar{n} \bigg]_{ij} S_{ij} - \sum_{j \in \text{neighbours}(i)} \bigg[ \bar{U}^* \cdot \bar{n} \bigg]_{ij} S_{ij}, \tag{3.2}
\]

where \( (\delta P) \) stands for the pressure increment, which is equal to the pressure difference between the times \( n \) and \( n+1 \). The mass conservation at time \( n+1 \) is used to evaluate the first term of the right hand side.

A Rhie & Chow filter is then applied to avoid the decoupling of even and odd cells on a Cartesian grid, which is due to the presence of the cell pressure gradient in the predicted mass flow and which is inherited from the prediction velocity step (Rhie & Chow, 1983). In practice, the predicted mass flux is corrected thanks to this explicit filter (see Archambeau et al. 2004) for more details).

\[
\sum_{j \in \text{neighbours}(i)} \frac{\Delta t}{|\Omega_i|} \frac{1}{\rho} \text{grad} (\delta P) \cdot \bar{n} \bigg] \cdot S_{ij} = \sum_{j \in \text{neighbours}(i)} \bigg[ \bar{U}^* \cdot \bar{n} \bigg]_{ij} S_{ij} + \Gamma_1 \left( \frac{1}{\rho^0_v} - \frac{1}{\rho^0_l} \right) |\Omega_i|. \tag{3.3}
\]

The elliptic operator for the pressure increment is also discretized with a two point flux approximation and gradient reconstructions to handle non orthogonalities and is solved with a classical deferred correction method (see Archambeau et al. 2004). The inverse of the density at the faces is discretized thanks to an harmonic interpolation in order to ensure a good precision for the resolution of the equation (3.3). The discretization of the cavitation source term \( \Gamma_1 \) is discussed hereafter.

Discretization of the cavitation source term \( \Gamma_1 \) :

In most previous works (see for example Kunz et al. 1999 and Senocak & Shyy, 2004), the cavitation source term \( \Gamma \) is expressed with a vapourisation and a condensation term, \( \dot{m}^+ \) and \( \dot{m}^- \) respectively. These terms are derived either from the local pressure difference \( P - P_{\text{vap}} \), with \( P_{\text{vap}} \) the vapour pressure, or from a simplified form of the Rayleigh-Plesset equation, which describes the dynamics of a single bubble in a varying pressure field. They depend thus on the pressure and also usually on the void fraction \( \alpha \).

\[
\Gamma = \dot{m}^- + \dot{m}^+ = f(\alpha, P). \tag{3.4}
\]

Temporal discretization of the source term \( \Gamma \) plays a major role in the convergence, the stability of the calculation and the global mass conservativity of the method. An explicit treatment of the pressure generates intense instabilities which lead to the divergence of the calculation. To address this issue, the source term is treated implicitly with respect to the pressure in the correction step using a Taylor expansion. The aim of this treatment is to damp the instabilities observed during the resolution of the pressure correction equation:
A new term containing the pressure increment $\delta P$ appears then in the right hand side of the pressure equation which depends on $\delta P_i$. This term will be implicated and the corresponding multiplicative term will be added to the resolution matrix diagonal. The equation (3.3) can finally be solved implicitly with respect to the pressure.

**3.3. Solving of the void fraction transport equation**

The final step of the algorithm is the resolution of the void fraction transport equation. In this study, it is assumed that the multiplicative term $[\alpha \times (1 - \alpha)]$ can be extracted in front of the cavitation source term, so that it can be written in the form $\Gamma = \alpha \times (1 - \alpha) \Gamma'$. The fact that this factorization is possible for any cavitation source term is a sufficient condition to show that the minimum/maximum principle for the void fraction is preserved at the continuous level. Indeed, in this case the source term is bounded on $\Omega \times [0,T]$ with $\Omega$ the spatial domain of interest and $[0,T]$ the time interval of integration. It is then easy to show that, with positive initial and boundary conditions, the void fraction remains in the interval $[0,1]$ on $\Omega \times [0,T]$ (this classical demonstration is left to the reader and makes use of the Grönwall lemma and the Hölder’s inequality). With this writing of the source term, the application of an upwind convection scheme and the use of the discretization (3.5) for the source term result in the following two conditions on the time step, in order to preserve the discrete minimum/maximum principle for the void fraction:

$$1 + \Delta t \left( \frac{1 - a^n_i}{\rho_v} \right) \left[ \Gamma_i^{n+1} + \frac{\partial \Gamma_i^{n+1}(P_i^n, a_i^n)}{\partial P} \delta P_i \right] \geq 0, \quad (3.6)$$

$$1 - \Delta t \frac{a^n_i}{\rho_v} \left[ \Gamma_i^{n+1} + \frac{\partial \Gamma_i^{n+1}(P_i^n, a_i^n)}{\partial P} \delta P_i \right] \geq 0. \quad (3.7)$$

It should be noted that with the Merkle source term used here for the numerical applications, it is not possible to extract the multiplicative term $[\alpha \times (1 - \alpha)]$. Nonetheless, this modification is not necessary, and the original writing of this source term also preserves the minimum/maximum principle at the continuous level (the demonstration is the same and is also left to the reader). However, we will consider this modification in the following and we let to the reader the writing of the time step constraints for the particular case of the original Merkle source term.

The equation to solve for the void fraction is as follows:

$$\frac{\Omega_i}{\Delta t} \left( a^{n+1}_{i} - a^n_i \right) + \sum_{j \in \text{neighbours}(i)} \left[ \left( \bar{U} \cdot \bar{n} \right)^{n+1} a^{n+1}_j \right] s_{ij} = \frac{\Omega_i}{\rho_v} \left[ \Gamma_i^n + \frac{\partial \Gamma_i^n}{\partial P} \delta P_i \right], \quad (3.8)$$

with the mass flux $\left( \bar{U} \cdot \bar{n} \right)^{n+1}_j$ inherited from the previous pressure correction step and an upwind discretization, with respect to the mass flux sign, for the convective flux.

Finally, the key features of the proposed algorithm are the following:

- Global mass conservation thanks to the identical discretization (3.5) of the cavitation source term in equations (3.3) and (3.8).
  - It is indeed easy to check that the sum of the discrete equations for the void fraction $\alpha$ and for $1 - \alpha$, with the use of the discretization of the mass flux deduced from the correction step, gives the discrete global mass conservation for the mixture density $\rho$.
- The discrete minimum/maximum principle preservation for the void fraction $\alpha$ thanks to the two constraints (3.6) and (3.7) on the time step and an upwind scheme for the convective flux of the void fraction equation.
4. Validation test cases

The proposed algorithm is validated on a 2D Venturi type geometry. The Venturi is 1.272 m long, its inlet section is 0.044 m wide and 0.05 m high, and its divergence angle is 8°. Validation of the results is based on comparisons with pressure and optical probe measurement performed previously (see Stutz and Reboud, 1997a, b, 2000). A periodical unsteady sheet cavitating behaviour including large vapour shedding is expected, so the attention will focus hereafter on the oscillation frequency, the mean attached cavity length and the time-averaged profiles of velocity and void fraction. Two turbulence models, based on the Reynolds-Averaged Navier-Stokes equations (RANS) approach, are tested to study their influence on the results. The first one is a high Reynolds $k-\varepsilon$ model (with scalable wall function is used for low Reynolds type meshes), while the second one is a low Reynolds $k-\omega$ SST model. The influence of the mesh size on the Strouhal number $Str = f L/U_{inlet}$ has been studied to perform mesh-independent calculations (where $f$ is the vapour shedding oscillation frequency, $L$ is the mean sheet cavity length, and $U_{inlet}$ is the velocity upstream from the Venturi). 2D simulations were performed. The final mesh for the $k-\varepsilon$ model is composed of 99000 elements (including 77500 elements in the cavitation region) with a non-dimensional wall distance $Y^+$ lying in the interval [10, 60]. The final mesh for the $k-\omega$ SST model composed of 135000 elements (including 84000 elements in the cavitation region) and $Y^+$ lies in the interval [0.01, 1.25]. It is now well known that these models are not able to reproduce the unsteady character of the flow; they have thus been modified using the correction proposed by Reboud et al. (1998), which consists in reducing the turbulent viscosity in the mixture where it is often overestimated. For the boundary conditions, it is assumed that no vapour crosses the inlet or outlet limits of the domain. An inlet velocity $U_{inlet} = 7.2 \text{ m/s}$ and an outlet cavitation number, $\sigma = 2.6$ when using the $k-\varepsilon$ model and $\sigma = 2.8$ for the $k-\omega$ SST model, are imposed, to obtain in both cases a mean sheet cavity length in agreement with the experimental conditions. The cavitation number is defined as: $\sigma = \frac{P_{outlet} - P_{vap}}{0.5 \rho \frac{f}{U_{inlet}}}$, where $P_{vap}$ is the vapour pressure at 20°C. In this study, $\rho_0^v$ and $\rho_0^l$ are set to 1 [kg/m$^3$] and 1000 [kg/m$^3$] respectively.

5. Results and discussion

First, it is checked that a constant time step $\Delta t = 10^{-5} s$ satisfies the two conditions given in the section (3.3) for both turbulence models and thus maintains the values of the void fraction in the physical range [0,1]. The simulation using the modified $k-\varepsilon$ model gives a frequency $f = 39 \text{ Hz}$ in fair agreement with the experiments (45 Hz). The resulting Strouhal number $Str = 0.29$ is in close agreement with the experimental value 0.3. The modified $k-\omega$ SST model leads to a very similar result ($f = 40 \text{ Hz}, Str = 0.31$). The standard $k-\varepsilon$ and $k-\omega$ SST models give a steady behaviour as expected.

![Figure 1](a) Fast Fourier transform applied to the input pressure signal, (b) instantaneous void fraction.

![Figure 2](a) Cavitation flow simulation, $\sigma_{outlet} = 2.6$, $U_{inlet} = 7.2 \text{ m/s}$, modified $k-\varepsilon$ turbulence model. (b) Cavitation flow cycle, $\sigma_{outlet} = 2.6$, $U_{inlet} = 7.2 \text{ m/s}$, modified $k-\varepsilon$ turbulence model.

In order to compare the experiments and the simulations, numerical results and optical probes measurements are plotted and compared. Figures (3) and (4) display time-averaged void fraction and
velocities, respectively, for the modified k−ε turbulence model, along 4 profiles downstream from the Venturi throat. A fair agreement between simulations and experiments is obtained.

Figure 3. Time-averaged void fraction profiles (comparison between experiments (in black) and simulations (in red)), \( \sigma_{\text{outlet}} = 2.6 \), \( U_{\text{inlet}} = 7.2 \text{ m/s} \), modified k−ε turbulence model.

Figure 4. Time-averaged velocity profiles (comparison between experiments (in black) and simulations (in red)), \( \sigma_{\text{outlet}} = 2.6 \), \( U_{\text{inlet}} = 7.2 \text{ m/s} \), modified k−ε turbulence model.

6. Conclusion
A numerical model based on a fractional step method has been developed for the simulation of cavitating flows. The originality of the present work is the application of two conditions on the time step in order to keep the variations of the void fraction in the physical range [0, 1] without any artificial numerical limitation. The results obtained with both RANS turbulence models confirm the capability of the modified k−ε model initially proposed by Reboud et al. (1998), and the modified k−ω SST model, to reproduce the unsteady periodical behaviour of the sheet cavity.

References
[1] Rhie C M, Chow W 1983 AIAA J. 21 1525-32.
[2] Merkle C L, Feng J Z and Buelow P E 1998 Computational modeling of the dynamics of sheet cavitation 3rd Int. Symp. on cavitation (Grenoble, FRANCE, 7-10 April 1998) p 307-311.
[3] Coutier-Delgosha O, Stutz B, Vabre A and Legoupil S 2007 J. F. Mech. 578 171-222.
[4] Coutier-Delgosha O 2001 Modélisation des écoulements cavitants: Etude des comportements instationnaires et application tridimensionnelle aux turbomachines (Grenoble: INPG).
[5] Delannoy Y, Kueny J L 1994 Journal of Fluid Engineering 116(1) 36-44.
[6] Guermond J L, Minev P and Shen J 2006 Comput. Methods Appl. Mech. Engrg. 195 6011-45.
[7] Senocak I and Shyy W 2004 International journal for numerical methods in fluids 44 975-995.
[8] Reboud J L, Stutz B and Coutier O 1998 Two-phase flow structure of cavitation: experiment and modelling of unsteady effects (Grenoble, FRANCE, 7-10 April 1998).
[9] Kunz R F, Boger D A, Chyczewski T S, Stinebring D R, Gibeling H J and Govindan T R 1999 multi-phase cfd analysis of natural and ventilated cavitation about submerged bodies 3rd ASME/JSME Joint Fluids Engineering Conference (San Francisco, California, 18-23 July 1999).
[10] Kunz R F, Boger D A, Stinebring D R, et al 2000 Computers & Fluids 29 849-875.
[11] Archambeau F, Mehitoua N and Sakiz M 2004 Code_Saturne: a finite volume. code for the computation of turbulent incompressible flows International Journal on Finite Volumes 1(1) (available online).
[12] Stutz B and Reboud J L 1997 *Experiments in Fluids* 22 191-8.
[13] Stutz B and Reboud J L 1997 *Phys. Fluids* 9(12) 3678
[14] Stutz B and Reboud J L 2000 *Exp. In Fluids* 29 545-52.