Comparison of Numerical Solvers for Cavitating Flows
Eric Goncalvès da Silva, Maxime Champagnac, Régiane Fortes Patella

To cite this version:

HAL Id: hal-00534695
https://hal.archives-ouvertes.fr/hal-00534695
Submitted on 20 Dec 2012
RESEARCH ARTICLE

Comparison of Numerical Solvers for Cavitating Flows

E. Goncalves \textsuperscript{a,*}, M. Champagnac \textsuperscript{a} and R. Fortes Patella\textsuperscript{a}

\textsuperscript{a}LEGI, Grenoble-INP, 1025 rue de la Piscine, 38400 St Martin d’Heres, France

(Received 00 Month 200x; final version received 00 Month 200x)

Different computational fluid dynamics (CFD) strategies have been developed to simulate, analyze and better understand cavitating flows. Based on homogeneous models, two numerical approaches using compressible and incompressible codes, are applied to capture large density variations and unsteady behaviours of cavitating flows. Simulations are performed on two-dimensional Venturi geometries and compared with experimental data. Local and global analyses are proposed and the necessity to account for compressibility phenomena is discussed.

Keywords: Two-phase flow; cavitation; homogeneous models; numerical schemes

Nomenclature

\begin{itemize}
  \item $c$: speed of sound
  \item $C_P, C_V$: thermal capacities
  \item $E$: total energy
  \item $e$: internal energy
  \item $F_c, F_d$: convective and diffusive flux densities
  \item $H$: total enthalpy
  \item $h$: enthalpy
  \item $k$: turbulent kinetic energy
  \item $M$: local Mach number
  \item $M\infty$: infinite Mach number
  \item $n$: outer normal to a surface
  \item $P$: static pressure
  \item $P_t$: stagnation temperature
  \item $P_{vap}$: vapour pressure
  \item $P_r, P_{rt}$: Prandtl numbers
  \item $P_{\infty}$: reference pressure
  \item $q$: total heat flux
  \item $q_{ref}$: energy of fluid at a given reference state
  \item $Re_L$: Reynolds number based on the length L
  \item $T$: mean static temperature
  \item $T_i$: stagnation temperature
  \item $T_{ref}$: reference temperature
  \item $W$: conservative variables
\end{itemize}

\textsuperscript{*}Corresponding author. Email: eric.goncalves@legi.grenoble-inp.fr
\(\alpha\) void ratio
\(\beta\) parameter of the preconditioning method
\(\gamma\) ratio of thermal capacities
\(\lambda_\pm\) eigenvalues of the preconditioned inviscid system
\(\delta(\cdot)_{l+1/2}\) difference operator for the \(l\) direction: \((\cdot)_{l+1} - (\cdot)_{l}\)
\(\mu(\cdot)_{l+1/2}\) averaged operator for the \(l\) direction: \(((\cdot)_{l+1} + (\cdot)_{l})/2\)
\(\varepsilon\) dissipation rate
\(\mu, \mu_t\) molecular and eddy viscosity
\(\rho\) density
\(\rho(A)\) spectral radius of matrix \(A\)
\(\rho^V\) viscous spectral radius
\(\sigma\) cavitation number
\(\tau\) total stress tensor
\(\Omega\) cell volume
\(\cdot\)\(_L\) liquid value
\(\cdot\)\(_V\) vapour value
\(\cdot\)\(_n\) value at temporal iteration \(n\)

1 INTRODUCTION

The capability for simulation and prediction of cavitating flows is of critical importance in the design of equipment such as turbomachinery, turbopumps in rocket propulsion systems, hydrofoils, fuel injectors, marine propellers, nozzles and underwater bodies. In most cases, cavitation is an undesirable phenomenon, causing significant degradation in the performance: reduced flow rates, lower pressure increases in pumps, load asymmetry, vibrations, noise and erosion. In most industrial applications, cavitating flows are turbulent and the dynamics of the formed interface involves complex interactions between vapour and liquid phases. These interactions are not well understood in the closure region of cavities where a distinct interface may not exist and where the flow is unsteady.

A number of physical and numerical models have been developed to investigate cavitating flows. Two-phase flow is often treated as a homogeneous phase mixture. A homogeneous mixture consists of a single fluid of varying density \(\rho\). In the mixture, the phases are assumed to be in kinematic, mechanical and thermal equilibrium: they share the same velocity, pressure and temperature. Moreover, vaporization or condensation processes are assumed to be instantaneous. Hence, this approach cannot reproduce strong thermodynamic or kinetic non-equilibrium effects. Density variation is determined by a cavitation model. In the literature, cavitation modelling is often based on either "equation of state (EOS)" models or "void ratio transport equation" models. In the EOS method, to close the system, the density is related to the pressure and the temperature (Delannoy and Kueny 1990, Song 2002, Liu et al. 2004, Moreau et al. 2004, Sinibaldi et al. 2006, Pouffary et al. 2008, Goncalves and Fortes-Patella 2009). In the second approach, the system is closed by adding a transport equation for the void ratio including a cavitation source term (Kunz et al. 2000, Ahuja et al. 2001, Senocak and Shyy 2002, Singhal et al. 2002, Wu et al. 2005, Fortes-Patella et al. 2006).

The crucial requirement of multiphase codes is the ability to accurately and efficiently span both incompressible and compressible flow regimes. Cavitating flows are characterized by large variation of the local Mach number. Indeed the speed of sound can be several orders of magnitude higher in the liquid phase than in the two-phase mixture (supersonic regime). Thus for low speed applications, numerical methods must be able to properly simulate simultaneously
nearly-incompressible and highly-compressible flow areas. In the literature, both incompressible and compressible solvers have been used. The first category must include density variations for cavitating applications; for example with SIMPLE or PISO pressure-based algorithms (Ventikos and Tzabiras 2000, Senocak and Shyy 2002, Singhal et al. 2002, Coutier-Delgosha et al. 2003, Wu et al. 2005, Fortes-Patella et al. 2006). On the other hand, compressible codes require an appropriate equation of state to cover all possible fluid states (pure liquid, two-phase mixture and pure vapour). A well known difficulty concerns the stiffness on the solution convergence when the Mach number becomes low. In this situation, the dominance of convection terms renders the system stiff and compressible solvers converge slowly. To overcome this difficulty, a preconditioned method is necessary. The physical acoustic waves are replaced by pseudo-acoustic modes much closer to the advective velocity, reducing stiffness and enhancing the convergence. A number of studies have been carried out with different preconditioned compressible solvers (Edwards and Franklin 2000, Kunz et al. 2000, Ahuja et al. 2001, Schmidt et al. 2006, Sinibaldi et al. 2006, Goncalves and Fortes-Patella 2009). Both approaches seem to have strong and weak points. According to (Ahuja et al. 2001), incompressible codes can lead to an erroneous acoustic speed in the mixture, particularly in the interface region, which may be not appropriate for unsteady simulations. Both CFD algorithms have been compared in (Venkateswaran et al. 2002) and applied to unsteady cavitating ogive flows. Computational results showed distinct behaviours between codes regarding the re-entrant jet phenomenon: with the incompressible code, the phenomenon was more developed and reached the leading edge of the cavity. The authors concluded that compressibility effects may need to be taken into account to correctly describe the cavity dynamics. Finally, a third method combining the advantages of the first two approaches has been explored (Rossow 2007, Housman 2009) and provided interesting results.

In the present study, we present two in-house finite-volume codes solving the one-fluid RANS equations with an EOS cavitation model. In the mixture, a barotropic EOS is used to close the system for both codes. Two numerical approaches have been tested. The first code is an incompressible pressure-based solver modified to consider density variations. Both pure phases are assumed incompressible. The second approach is based on a compressible density-based code with a preconditioning method. Both pure phases are compressible and follow the stiffened gas EOS.

The models and numerical techniques are evaluated by comparing the numerical results with experimental data on two Venturi geometries with cold water (Fortes-Patella et al. 2006, Barre et al. 2009). In the following, we will first summarize the essential elements of the governing equations, modelling concepts and numerical schemes before presenting the computational results.

2 EQUATION OF STATE: THE BAROTROPIC LAW

The cavitation model applied in the present work is based on a sinusoidal barotropic law (Delannoy and Kueny 1990) (Figure 1). This law is characterized by an adjustable parameter $c_{\text{min}}$ which can be interpreted as the minimum speed of sound in the mixture. This parameter is set to 0.86 m/s for both codes.

When the pressure is between $P_{\text{vap}} + \Delta P$ and $P_{\text{vap}} - \Delta P$, the following relationship applies, function of the void ratio $\alpha$:

$$P(\alpha) = P_{\text{vap}} + \left(\frac{\rho_{L}^{\text{sat}} - \rho_{V}^{\text{sat}}}{2}\right) c_{\text{min}}^2 \arcsin (1 - 2\alpha) \quad \text{and} \quad \alpha = \frac{\rho - \rho_{L}^{\text{sat}}}{\rho_{V}^{\text{sat}} - \rho_{L}^{\text{sat}}}$$  \hspace{1cm} (1)

where $\Delta P$ represents the pressure width of the law, $\rho_{L}^{\text{sat}}$ and $\rho_{V}^{\text{sat}}$ are the saturation values of
liquid and vapour densities. For a void ratio value of 0.5, the pressure is equal to the saturation pressure $P_{vap}$.

For cold water, the cavitation phenomenon is assumed to be isothermal. The speed of sound can be easily computed:

$$c^2 = \left( \frac{\partial P}{\partial \rho} \right)_s = \left( \frac{\partial P}{\partial \rho} \right)_T = \frac{c_{\min}^2}{\sqrt{1 - (1 - 2\alpha)^2}}$$  \hspace{1cm} (2)

This speed is always strictly positive and thus the inviscid compressible system is hyperbolic. Moreover, this mixture EOS is convex for the $c_{\min}$ value considered (i.e. the Hessian matrix of internal energy has to be positive definite) and respects entropy conditions (Goncalves and Fortes-Patella 2009).

### 3 THE INCOMPRESSIBLE SOLVER

The IZ code, supported by the French space agency CNES, is based on an incompressible approach adapted to two-dimensional cavitating flows. The numerical resolution is a pressure-correction method derived from the SIMPLE algorithm. It uses a finite-volume discretization applied to structured orthogonal meshes with staggered grids. All computations are time-dependent to take into account the variation of density. Other applications are presented in (Coutier-Delgosha et al. 2002, 2003, Fortes-Patella et al. 2006).

#### 3.1 Governing equations

The RANS equations coupled with a two-equation $k – \varepsilon$ turbulence model are solved in the orthogonal frame of curvilinear coordinates $(\xi, \eta)$. The system can be expressed as:

$$\frac{\partial W}{\partial t} + \frac{1}{J} \frac{\partial}{\partial \xi} \left[ h_2 \left( uW - \frac{F_W}{h_1} \frac{\partial(W/\rho)}{\partial \xi} \right) \right] + \frac{1}{J} \frac{\partial}{\partial \eta} \left[ h_1 \left( vW - \frac{F_W}{h_2} \frac{\partial(W/\rho)}{\partial \eta} \right) \right] = S_W$$  \hspace{1cm} (3)

\[ W = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho k \\ \rho \varepsilon \end{pmatrix}, \quad F_W = \begin{pmatrix} 0 \\ \mu + \mu_t \\ \mu + \mu_t \\ \mu_t/\sigma_k \\ \mu_t/\sigma_\varepsilon \end{pmatrix}, \quad S_W = \begin{pmatrix} 0 \\ -1 \frac{\partial (h_2 P)}{\partial \xi} \\ -1 \frac{\partial (h_1 P)}{\partial \eta} \\ \frac{\mu_t}{J} \left[ \frac{1}{J} \frac{\partial (h_2 v)}{\partial \xi} \right]^2 + \left[ \frac{1}{J} \frac{\partial (h_1 u)}{\partial \eta} \right]^2 \rho \varepsilon \\ C_1 k \left[ \frac{1}{J} \frac{\partial (h_2 v)}{\partial \xi} \right]^2 + \left[ \frac{1}{J} \frac{\partial (h_1 u)}{\partial \eta} \right]^2 \end{pmatrix} \]

where $W$ denotes the conservative variables, $h_1$ and $h_2$ the scale factors (the square roots of the diagonal components of the metric tensor) and $J = h_1 \times h_2$, $u$ and $v$ the velocity components along coordinates $\xi$ and $\eta$ respectively. The quantities $\sigma_k$, $\sigma_\varepsilon$, $C_1$ and $S_\varepsilon$ depend on the formulation of the turbulence model.
3.2 Spatial discretization

The finite volume method is applied for the space discretization. Each equation is integrated locally on its own control volume based on the staggered grid to avoid pressure oscillations. The pressure and the density are calculated at the center of the cells, while the velocity components \( u \) and \( v \) are located respectively on the western and the southern faces of each cell. The diffusive terms are discretized with a second-order space-centered scheme, while the convection terms are estimated through the second-order HLPA (Hybrid Linear Parabolic Approximation) scheme (Zhu 1991). This is a second-order scheme, which locally switches to first order, to prevent numerical oscillations in critical high pressure or high density gradient areas.

3.3 Temporal discretization

Several first and second order numerical schemes are available. An unconditionally stable second-order implicit scheme is applied to compute values at iteration \( n + 1 \):

\[
\frac{\partial W}{\partial t} = \frac{3W^{n+1} - 4W^n + W^{n-1}}{2\Delta t} \quad (4)
\]

3.4 Turbulence model

A \( k - \varepsilon \) RNG turbulent closure model (Yakhot et al. 1992), modified for two-phase flow simulations by adding a turbulent viscosity limiting function \( f(\rho) \), is used. The turbulent viscosity satisfies:

\[
\mu_t = f(\rho)C_\mu k^2 \quad \text{where} \quad C_\mu = 0.085 \quad (5)
\]

In pure phases, \( f(\rho) = \rho \). In the two-phase mixture regions, an arbitrary decrease of the turbulent viscosity was proposed by Reboud (Reboud et al. 1998, Coutier-Delgosha et al. 2002):

\[
f(\rho) = \rho v + (1 - \alpha)^n (\rho_L - \rho_v) \quad (6)
\]

where \( n \) is a parameter greater than 1. In the present study, we have chosen the value \( n = 10 \). The turbulence model is coupled with standard wall functions.

3.5 Iterative procedure

If the flow is incompressible, we use a classical pressure-correction method proposed by Patankar (Patankar and Spalding 1972). It is based on the SIMPLE algorithm, modified to include the cavitation treatment. The resolution of each time-step is divided into several iterations, until convergence is reached. Each iteration is composed of successive steps. The velocities are first estimated from the momentum equations with a pressure field. Then, values of velocities are corrected by solving a Poisson pressure correction equation in order to satisfy continuity. The cavitation process induces the following modifications:

- The density is first calculated with the barotropic state law after solving the momentum equations, as well as its derivative \( \frac{\partial \rho}{\partial P} \).
- When the pressure correction \( \delta P \) is obtained, the density values are corrected using the relation: \( \delta \rho = \left( \frac{\partial \rho}{\partial P} \right) \delta P \)

The main steps of a single iteration are listed next to determine values at iteration \( n + 1 \):
a) Resolution of the transport equations for the turbulent variables and calculation of the
turbulent viscosity.
b) Calculation of the estimated velocities $U^*$ from the momentum balance equations.
c) Calculation of the intermediate density $\rho^*$ and its derivative with respect to the pressure,
according to the equation of state.
d) Resolution of the pressure correction equation, which involves not only velocity variations
$dU$ but also supplementary density variations terms $d\rho$.
e) Computation of the new velocity $U^{n+1}$, pressure $P^{n+1}$ and density $\rho^{n+1}$ fields.

A loop is added to the pressure correction step inside each iteration to check that the void ratio
remains between 0 and 1. Moreover, the density, velocity components and turbulent variables
can be under-relaxed if convergence is difficult to obtain. The pressure correction is not relaxed.

3.6 Inlet and outlet conditions
The boundary conditions are based on a system of dummy cells. Classical incompressible types
of boundary conditions are applied:
- at the inlet, the velocity component $u$ is imposed and the component $v$ is set to 0.
- at the outlet, a static pressure is imposed, and if the velocity component $u < 0$, the gradient
$\frac{\partial v}{\partial x}$ is set to 0.
We assume that inlet and outlet areas are in a pure liquid region. No cavitation appears in these
boundaries.

4 THE COMPRESSIBLE SOLVER
The CaviFlow code solves the compressible one-fluid RANS system for multi-domain structured
meshes. It is based on a cell-centered finite-volume discretization. More details concerning the
code are given in (Goncalves and Fortes-Patella 2009).

4.1 Preconditioning method for the Euler system
The preconditioning method is based on the modification of the derivative term by a pre-
multiplication with a suitable preconditioning matrix $P_c$. We used the formulation proposed by
(Turkel 1987) with the primitive variables pressure, velocity and entropy :

$$P_c = \begin{bmatrix} \beta^2 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

$\beta$ is a parameter of the order of the Mach number. In our study, we have chosen the form
given by (Choi and Merkle 1993):

$$\beta^2 = \min \left[ \max \left( M^2, KM_{\infty}^2 \right), 1 \right]$$

(7)
This form implies that there is no preconditioning used in transonic and supersonic flow regions
(in the mixture). When $\beta^2 = 1$, the preconditioning matrix becomes the identity matrix and the
system returns to its classical non-preconditioned form. Moreover, for very small flow-velocity,
$\beta^2$ is not allowed to be less than a given percentage of the freestream velocity, determined by
the coefficient $K$. For inviscid computations, $K$ is near unity.
4.2 The compressible RANS system

The inviscid preconditioner presented previously is used (no viscous terms are introduced). The compressible RANS equations coupled with a two-equation turbulence model can be expressed as:

$$P_c^{-1} \frac{\partial W}{\partial t} + \text{div} (F_c - F_v) = S$$

(8)

$$W = \begin{pmatrix} \rho \\ \rho V \\ \rho E \\ \rho k \\ \rho \Psi \end{pmatrix}; \quad F_c = \begin{pmatrix} \rho V \\ \rho V \otimes V + \mathbb{P}I \rho E + P \mathbb{I} \\ (\rho E + P) V \\ \rho k V \\ \rho \Psi V \end{pmatrix}; \quad F_v = \begin{pmatrix} 0 \\ \mathbb{T}_V V - q \\ (\mu + \mu_t/\sigma_k) \text{grad} k \\ (\mu + \mu_t/\sigma_\Psi) \text{grad} \Psi \end{pmatrix}$$

where $W$ denotes the conservative variables, $V$ the mean velocity vector, $E$ the total energy, $F_c$ and $F_v$ the convective and viscous flux densities and $S$ the source terms which concern only the transport equations. $\Psi$ is a turbulent variable.

The exact expression of the eddy viscosity $\mu_t$ and the source terms depends on the turbulence model as well as constants $\sigma_k$ and $\sigma_\Psi$. $\mathbb{T}_V$ is the total stress tensor and $q$ the total heat flux vector, evaluated with the Newtonian law, the Fourier law and the Boussinesq assumption.

For pure phases, we use the convex stiffened gas equation of state (Le Metayer et al. 2004):

$$P(\rho, e) = (\gamma - 1) \rho (e - q_{ref}) - \gamma P_\infty$$

(9)

$$P(\rho, T) = \rho (\gamma - 1) C_v T - P_\infty$$

(10)

$$T(\rho, h) = \frac{h - q_{ref}}{C_p}$$

(11)

where $\gamma = C_p/C_v$ is the polytropic coefficient, $C_p$ and $C_v$ are thermal capacities, $q_{ref}$ the energy of the fluid at a given reference state and $P_\infty$ a constant reference pressure. $e$ and $h$ denote the internal energy and enthalpy respectively.

4.3 Spatial discretization

Using the finite-volume technique for space discretization, a semi-discrete form of equation (8) may be written for a computational cell of volume $\Omega$ limited by a surface $\Sigma$ and with an outer normal $n$:

$$P_c^{-1} \Omega \frac{\partial W}{\partial t} + \sum_{\text{all faces}} (F_c - F_v)_n \Sigma = \Omega \hat{S}$$

(12)

For the mean flow, the convective flux density vector on a cell face is computed with the space-centered Jameson scheme stabilized by scalar artificial dissipation (Jameson et al. 1981).

Discretization of the viscous terms is performed by a second-order space-centered scheme. For the turbulence transport equations, the upwind Roe scheme (Roe 1981) was used to obtain a more robust method. The second-order accuracy was obtained by introducing a flux-limited dissipation.
4.4 Temporal discretization

Time integration is achieved through a low-cost implicit method (Luo et al. 1998). The implicit method consists in solving, at each time step, a system of equations arising from the linearization of a fully implicit scheme. The main advantage of this method is that the storage of the Jacobian matrix is completely eliminated, which leads to a low-storage algorithm. The viscous flux Jacobian matrices are replaced by their spectral radii. The Jacobian matrices, which appear from the linearization of the centered fluxes, are approximated with the numerical fluxes. The following system is obtained, for iteration $n$:

$$\Delta W^n + \sum_{l=i,j,k} \left[ \sigma_l \delta_l (\Delta f_l^n) - \sigma_l (\rho_l^V \Delta W^n) - 0.5 \sigma_l (P^{-1} \rho (PA_l) \delta_l \Delta W^n) \right] = R^n_{\text{expl}}$$

(13)

where $\Delta W^n = W^{n+1} - W^n$, $\sigma_l = \Delta t / \Delta x_l$, and $R^n_{\text{expl}}$ represents the explicit conservative residuals. For each direction $l$, $\rho_l^V$ is the viscous spectral radius, $\mu_l$ the averaged operator, $\delta_l$ the difference operator, $A_l$ the Jacobian matrix of the convective flux for the direction $l$ and $\rho(PA_l)$ the spectral radius of the matrix $PA_l$.

The implicit time-integration procedure leads to a system which can be solved directly or iteratively. The direct inversion can be memory intensive and computationally expensive. Therefore, an implicit relaxation procedure is preferred and the point Jacobi relaxation algorithm was chosen.

For unsteady computations, the dual time stepping method, proposed by (Jameson 1991), was used to tackle the lack of numerical efficiency of the global time stepping approach. The derivative with respect to the physical time is discretized by a second-order formula. The initialization of the derivative with respect to the physical time was performed with a first-order formula.

4.5 Turbulence modelling

The considered model is the Jones-Launder $k - \varepsilon$ model (Jones and Launder 1972). In two-phase areas, the Reboud limiter is applied with the same constant $n = 10$. The turbulence model is coupled with standard wall functions. Computations with both codes were not performed rigorously with the same turbulence model, but we assumed that the Jones-Launder and the RNG $k - \varepsilon$ models roughly provided similar solutions.

4.6 Inlet and outlet boundary conditions

The numerical treatment of the boundary conditions is based on the use of the preconditioned characteristic relations. The number of variable to impose at boundaries is given by the number of positive characteristics. The characteristic relations obtained for the preconditioned system, in two-dimensional flows, are:

$$- c^2 (\rho^e - \rho^s) + (P^c - P^s) = 0$$

(14)

$$V^c_t - V^s_t = 0$$

(15)

$$(\lambda_+ - V_n)(P^c - P^s) + \rho \beta^2 c^2 (V^c_n - V^s_n) = 0$$

(16)

$$(\lambda_- - V_n)(P^c - P^s) + \rho \beta^2 c^2 (V^c_n - V^s_n) = 0$$

(17)
The variables with superscript $c$ are those to be computed at the boundary. Variables with superscript $s$ are obtained by the current numerical scheme. $V_t$ and $V_n$ are the tangential and normal components of the mean velocity respectively. $\lambda_+$ and $\lambda_-$ are the eigenvalues of the preconditioned inviscid system, given by:

\[
\lambda_{\pm} = \frac{1}{2} \left[ u(1 + \beta^2) \pm \sqrt{(\beta^2 - 1)^2 u^2 + 4 \beta^2 c^2} \right] \tag{18}
\]

At inflow, four variables have to be imposed. We have chosen the stagnation pressure $P_0$, the stagnation temperature $T_0$, and the direction of the velocity. A Newton algorithm can be used to compute the pressure with the characteristic relation (17) and the conservative variables can be evaluated at the boundary.

At outflow, only one variable is imposed, which we have chosen to be the static pressure. The conservative variables are computed with three characteristic relations (14)-(16).

We assume that inlet and outlet areas are in a pure liquid region. No cavitation appears in these boundaries.

5 COMPUTATIONS AND COMPARISONS: OPEN CAVITY

The first case is an open cavity on a Venturi geometry, following the classification of partial cavity flows (Laberteaux and Ceccio 2001).

5.1 Experimental geometry and conditions

The tested Venturi is characterized by a divergence angle of $4^\circ$ (see Figure 2). The selected operating point is characterized by (Barre et al. 2009):

- $U_{\text{inlet}} = 10.8$ m/s: the inlet velocity
- $P_{\text{inlet}} = 35000$ Pa: the pressure in the inlet section
- $T_{\text{ref}} = 293$ K: the reference temperature
- $\sigma_{\text{inlet}} = \frac{P_{\text{inlet}} - P_{\text{vap}}}{0.5\rho U_{\text{inlet}}^2} \simeq 0.55$: the cavitation parameter in the inlet section
- $Re_{L_{\text{ref}}} = \frac{U_{\text{inlet}} L_{\text{ref}}}{\nu} = 2.710^6$: the Reynolds number

With these parameters, a cavity length $L$ ranging between 70 mm and 85 mm has been obtained.

The experimental void ratio and velocity profiles are obtained for five stations by a double optical probe (Figures 2 and 3). The velocity is evaluated as the most probable value and the void ratio is obtained from the signal of the double optical probe using a post processing algorithm.

The experimental views for this geometry show a relatively stable cavity behaviour (Figure 4). It is characterized by an almost constant length, although the closure region always fluctuates, with the existence of a re-entrant jet and little vapour cloud shedding. The re-entrant jet is mainly composed of liquid which flows upstream along the solid surface. For this geometry, no periodic cycles with large shedding were observed.

5.2 Mesh

The grid is an H-type topology. It contains 251 nodes in the flow direction and 62 in the orthogonal direction. A special contraction of the mesh is applied in the main flow direction just after the throat to better simulate two-phase flow area (Figure 5). The $y^+$ values of the mesh, at the center of the first cell, vary between 12 and 27, for a non-cavitating computation.
According to the study of mesh dependence presented in (Coutier-Delgosha et al. 2005), this grid size is adequate to simulate cavitation sheets in such Venturi type section.

5.3 Numerical parameters

For the non-cavitating regime, computations are started from an uniform flow-field. For the cavitating regime, we used for both codes:
- the vapour pressure, \( P_{vap} = 2339 \text{Pa} \)
- the saturation values of liquid/vapour density, \( \rho_{sat}^L = 1000 \text{kg/m}^3 \) and \( \rho_{sat}^V = 0.02 \text{kg/m}^3 \)
- the farfield value of turbulent kinetic energy, \( k_\infty = 0.0045 \text{ m}^2/\text{s}^2 \)
- the dimensionless time step for unsteady computations, \( \Delta t^* = \frac{\Delta t U_{inlet}}{L_{ref}} = 0.015 \).

The numerical parameters used for simulations are summarized in Table (1) and Table (2).

5.4 Comparison of the two solvers

Two series of solutions are presented and compared: steady and unsteady solutions.

5.4.1 Steady solutions

First, the goal was to obtain a steady solution with a stable cavitation sheet with a length close to 80 mm. The defined sheet length \( L_{cav} \) used for representing the computational results has been determined by the length of the iso-line corresponding to a void ratio \( \alpha = 0.2 \).

With the compressible solver, steady computations were carried out with a local time stepping method whereas, with the incompressible solver, unsteady simulations were performed without reducing the turbulent viscosity. The overall results are given in Table (3). The \( \sigma_{inlet} \) value obtained with the compressible code is over-estimated in comparison with both the experimental value and the numerical ”incompressible” value.

Figure 6 shows the longitudinal velocity (left) and void ratio (right) profiles for the present experiments and numerical simulations. The overall agreement seems good between the experimental data and simulations. This is especially true for stations 1 and 2 where no re-entrant jet phenomena occur. Further downstream, experimental observation indicates a recirculating behaviour with a re-entrant jet extending roughly through half the sheet thickness. No relevant influence of the numerical schemes and turbulence models is observed.

The wall pressure profiles are plotted in Figure 7 versus the distance \( x - x_{inlet} \). The first five data are located inside the cavity (where void ratio and velocity profiles are measured). For all computations, the pressure remains at an almost constant value in the cavity. Downstream, discrepancies are notable due to the re-entrant jet. In comparison with experimental data, the re-compression is a little under-evaluated with the compressible code and over-estimated with the incompressible code.

5.4.2 Unsteady solutions

Next, unsteady calculations were performed with the goal of obtaining a quasi-stable cavitation sheet with a significant re-entrant jet. With the compressible solver, unsteady computations were carried out with the dual time stepping method and the Rebound viscosity limiter. With the incompressible solver, unsteady simulations were performed with the same turbulent viscosity limiter. The overall results are given in Table 4; they show the better agreement obtained with the compressible code for the \( \sigma_{inlet} \) value.

The numerical solutions obtained from both codes present some marked differences. The incompressible code captured low frequency periodic oscillations of the cavitation sheet (around
6 Hz). A view of the cavity (contours of the void ratio) obtained with the IZ code over one period is presented in Figure 8. Oscillations of the cavity are clearly exhibited. On the other hand, the compressible code captured a quasi-stable behaviour in good agreement with the experimental view. No frequencies are revealed by a direct Fourier transformation of the vapour volume. A view of the cavity at five instants is plotted in Figure 9. We can observe the attached cavity from the throat of the Venturi up to $x = 0.3\text{m}$. Downstream, the recirculating area with two-phase structure shedding is well illustrated.

Figure 10 shows the longitudinal velocity (left) and void ratio (right) profiles for experiments and unsteady numerical results. The numerical values are obtained by a time-averaged treatment.

At stations 3, 4 and 5, the recirculating behaviour with a re-entrant jet is well simulated by the compressible code. On the other hand, the incompressible code largely under-estimated the re-entrant jet intensity and the thickness of the recirculating area (at station 3, it is not captured).

For the void ratio profiles, both codes provided a similar solution at station 3, in good agreement with the experimental data. At station 4, with the incompressible code, a liquid re-entrant jet ($\alpha = 0$) is observed close to the wall, up to a distance $y = 0.001\text{m}$, whereas the re-entrant jet captured by the compressible calculation is not composed of pure liquid (at the wall, the void ratio value is around 0.35 instead of 0.15). At station 5, the length of the sheet obtained with the incompressible code is shorter than the experimental value. With the compressible code, the void ratio is in very close agreement with the experimental data.

The wall pressure is plotted in Figure 11 versus the distance $x - x_{\text{inlet}}$. Theses values are obtained by a time-averaged treatment for both experiments and calculations. With the incompressible code, the computed pressure inside the cavity is smaller than the vapour pressure. After station 3, we observe an area where the time-averaged pressure is above the vapour pressure, due to the oscillating behaviour of the cavity. With the compressible code, the pressure inside the cavity is in good agreement with the experimental data. Downstream, the re-compression is slightly under-evaluated, as previously observed with the steady simulations.

Comparisons with measurements data suggest that the compressible formulation provides an improved description of the cavitation dynamics compared with incompressible computations.

6 COMPUTATIONS AND COMPARISONS: UNSTEADY RE-ENTRANT CAVITY

The second case is an unsteady re-entrant cavity with quasi periodic break-off and roll-up of a portion of the cavity.

6.1 Experimental geometry and conditions

This Venturi is characterized by a divergence angle of $8^\circ$, illustrated in Fig. 12. The geometrical data are:

- Inlet section: $S_i = 50 \times 44 \text{ mm}^2$ (where the reference pressure is measured);
- Throat section: $S_{\text{throat}} = 33.5 \times 44 \text{ mm}^2$;
- Length of the test section (chord): $L_{\text{ref}} = 224 \text{ mm}$.

The selected operating point is characterized by the following physical parameters defined by Fortes-Patella et al. (2006):

- $U_{\text{inlet}} = 7.04 \text{ m/s}$: the inlet velocity
- $Q = 0.0155 \text{ m}^3/\text{s}$: the flow imposed in the circuit
\( P_{\text{inlet}} = 55000 \text{ Pa} \): the pressure in the inlet section
\( \sigma_{\text{inlet}} = \frac{P_{\text{inlet}} - P_{\text{vap}}}{0.5\rho U_{\text{inlet}}^2} \simeq 2.15 \pm 0.06 \): the cavitation parameter in the inlet section
\( Re = \frac{U_{\text{inlet}}L_{\text{ref}}}{\nu} = 1.57 \times 10^6 \): the Reynolds number

With these parameters, an unsteady cavity with quasi-periodic fluctuations of the attached sheet and vapour clouds shedding has been obtained. Fig. 13 shows an instantaneous photograph of the cavity with a large structure shedding. The maximum length of the attached cavity is about 45 mm. The vapour shedding frequency is close to 45 Hz, estimated from Fourier transformations of the unsteady pressure signal.

This geometry is equipped with three probing holes to make various measurements such as the local void ratio and instantaneous local velocity (Fig. 14).

### 6.2 Mesh

The grid is an H-type topology. It contains 250 nodes in the flow direction and 61 in the orthogonal direction. A special contraction of the mesh is applied in the main flow direction just after the throat to better simulate the two-phase flow area (Fig. 15). The \( y^+ \) values of the mesh, at the center of the first cell, vary between 9 and 31 for a non cavitating computation.

### 6.3 Numerical parameters

For the cavitating regime, the dimensionless time step is: \( \Delta^* = \frac{\Delta t U_{\text{inlet}}}{L_{\text{ref}}} = 6 \times 10^{-3} \).

Numerical parameters used for simulations are similar to those presented previously.

### 6.4 Global analyses

Different calculations were performed, summarized in Table (5). The goal was to obtain a periodic cavitation sheet with a frequency close to 45 Hz.

The vapour volume changes are plotted in Fig. 16 for both codes. The incompressible solution seems more regular. Moreover, the values are quite different: the ratio between compressible and incompressible values is around 2.

A direct Fourier transformation (DFT) of the vapour volume was performed to evaluate the frequency. Fig. 17 presents the frequency results for the incompressible code (left) and the compressible code (right). A peak is clearly visible for both codes, with better agreement for the incompressible code.

A view of the cavity (contours of the void ratio) obtained with the incompressible IZ code, over one period, is presented in Figure 18. Large-scale two-phase structure shedding is clearly observed. The re-entrant jet is very intense and flows upstream to the throat of the Venturi, leading to the disappearance of the attached cavity. These results corroborate conclusions presented in (Venkateswaran et al. 2002).

The solution obtained with the compressible code presents differences. A view of the cavity over one period is plotted in Figure 19. A small cavity attached to the throat is always present. Two-phase structures convected in the flow are smaller and condense rapidly, in comparison with the incompressible solution.
6.5 Velocity and void ratio profiles

Figure 20 illustrates the longitudinal velocity (left) and void ratio (right) profiles for experiments and unsteady numerical results. The numerical values are obtained by a time-averaged treatment.

At station 1, with the compressible code, the recirculating behaviour is very well captured, yet the void ratio is largely over-predicted. For the incompressible code, the void ratio is slightly under-estimated due to the disappearance of the attached cavity.

At station 2, both codes under-evaluated the recirculating zone thickness. For the void ratio profile, values obtained with the incompressible code are largely over-predicted, whereas values obtained by the compressible code are in better agreement with the experimental data.

At station 3, no experimental velocity profiles are usable. For the void ratio profile, the compressible code computed a pure liquid zone: two-phase structures shedding is not enough extended.

On the other hand, the incompressible code captured large-scale cavitation structures, but with excessive void ratio values.

7 CONCLUSION

Two different cavitating one-fluid CFD code have been presented and applied to Venturi flows. The first is based on a pressure-based incompressible algorithm modified to take into account density variations. The second uses a density-based compressible approach including a preconditioning algorithm to treat the pure liquid incompressible region.

Both codes were compared on cavitating flows using the same cavitation model, the same mesh, a $k-\varepsilon$ model with the same viscosity limiter and the same numerical time step. Two different cavitation cases were tested: an aperiodic quasi-stable cavity sheet and a quasi-periodic unsteady re-entrant cavity. Global and local analysis of flows based on void ratio and velocity profiles were carried out.

For the first case, the compressible code captured a solution in close agreement with the experimental data, both for the global behaviour and the local analysis. On the other hand, the incompressible code provided a different solution with a periodic low frequency oscillating sheet. For this case, it seems necessary to account for compressibility effects to correctly describe the cavity dynamics.

For the second case, the re-entrant jet phenomenon obtained with both codes presents some marked differences. The compressible code under-predicted large-scale cavitation structure shedding. The opposite situation is observed with the incompressible code: a very intense re-entrant jet is captured, which flows upstream to the throat and completely eliminates the attached cavity. These results corroborate conclusions presented in (Venkateswaran et al. 2002).

Additional investigations are necessary to clearly conclude on the importance of the compressibility effects. Additional computations are in progress to analyze the influence of the turbulence model for unsteady computations and to pursue comparative studies between numerical approaches.

References


REFERENCES


List of Figures

Fig1: The barotropic equation of state
Fig2: Schematic view of the 4° Venturi
Fig3: View of the cavity and the five stations, 4° Venturi
Fig4: Photograph of the cavity, 4° Venturi
Fig5: Enlargement of the mesh near the throat, 4° Venturi
Fig6: Velocity (left) and void ratio (right) profiles, steady solutions, 4° Venturi
Fig7: Wall pressure evolution, steady solutions, 4° Venturi
Fig8: Contours of the void ratio over one period, incompressible code, 4° Venturi
Fig9: Contours of the void ratio at five instants, compressible code, 4° Venturi
Fig10: Time-averaged velocity (left) and void ratio (right) profiles, unsteady solutions, 4° Venturi
Fig11: Time-averaged wall pressure versus distance, unsteady solutions, 4° Venturi
Fig12: Schematic view of the 8° Venturi
Fig13: Photograph of the cavity, 8° Venturi
Fig14: View of the three stations of measurement, 8° Venturi
Fig15: Enlargement of the mesh near the throat, 8° Venturi
Fig16: Evolution of the vapour volume, incompressible code (left) and compressible code (right), 8° Venturi
Fig17: Fourier transformation of the vapour volume, incompressible code (left) and compressible code (right), 8° Venturi
Fig18: Contours of void ratio over one period, incompressible code, 8° Venturi
Fig19: Contours of void ratio over one period, compressible code, 8° Venturi
Fig20: Time-averaged velocity (left) and void ratio (right) profiles, 8° Venturi

List of Tables

Table1: Numerical parameters, IZ
Table 2: Numerical parameters, CaviFlow

Table 3: Sigma inlet values, steady solutions, 4° Venturi

Table 4: Sigma inlet most probable values, unsteady solutions, 4° Venturi

Table 5: Sigma inlet most probable values, unsteady solutions, 8° Venturi
<table>
<thead>
<tr>
<th>IZ parameters</th>
<th>all computations</th>
</tr>
</thead>
<tbody>
<tr>
<td>maximum of SIMPLE iterations / physical time step</td>
<td>1500</td>
</tr>
<tr>
<td>convergence criterion for the density $\rho$</td>
<td>$10^{-6}$</td>
</tr>
<tr>
<td>under-relaxation coefficient for $\rho$</td>
<td>0.2</td>
</tr>
<tr>
<td>under-relaxation coefficient for $u$ and $v$</td>
<td>0.7</td>
</tr>
<tr>
<td>under-relaxation coefficient for $k$ and $\varepsilon$</td>
<td>0.4</td>
</tr>
<tr>
<td>CaviFlow parameters</td>
<td>steady non-cav</td>
</tr>
<tr>
<td>----------------------------------</td>
<td>----------------</td>
</tr>
<tr>
<td>CFL number</td>
<td>10</td>
</tr>
<tr>
<td>preconditioning parameter $K$</td>
<td>1</td>
</tr>
<tr>
<td>implicit Jacobi iterations</td>
<td>15</td>
</tr>
<tr>
<td>2nd and 4th order dissipation parameter</td>
<td>0 ; 0.032</td>
</tr>
<tr>
<td>dual time stepping sub-iterations</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>$\sigma_{\text{inlet}}$</td>
</tr>
<tr>
<td>-------------------</td>
<td>-------------------------</td>
</tr>
<tr>
<td>experimental</td>
<td>$\geq 0.55$</td>
</tr>
<tr>
<td>incompressible solver</td>
<td>0.585</td>
</tr>
<tr>
<td>compressible solver</td>
<td>0.63</td>
</tr>
</tbody>
</table>
Table 4. Sigma inlet most probable values, unsteady solutions, 4° Venturi

<table>
<thead>
<tr>
<th>Code</th>
<th>$\sigma_{\text{inlet}}$</th>
<th>$L_{\text{cav}}$ (mm)</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Incompressible solver</td>
<td>0.60</td>
<td>40 to 74</td>
<td>periodic oscillations of the sheet: 6Hz</td>
</tr>
<tr>
<td>Compressible solver</td>
<td>0.56</td>
<td>70</td>
<td>quasi-stable sheet with re-entrant jet</td>
</tr>
</tbody>
</table>
Table 5. Sigma inlet most probable values, 8° Venturi

<table>
<thead>
<tr>
<th>code</th>
<th>$\sigma_{\text{inlet}}$</th>
<th>comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>experimental</td>
<td>$\approx 2.15$</td>
<td>periodic sheet: 45Hz</td>
</tr>
<tr>
<td>incompressible solver</td>
<td>2.11</td>
<td>periodic sheet: 48Hz</td>
</tr>
<tr>
<td>compressible solver</td>
<td>2.09</td>
<td>periodic sheet: 40 Hz</td>
</tr>
</tbody>
</table>
Figure 1. The barotropic equation of state
Figure 2. Schematic view of the 4° Venturi
Figure 3. View of the cavity and the five stations, $4^\circ$ Venturi
Figure 4. Photograph of the cavity, 4° Venturi
Figure 5. Enlargement of the mesh near the throat, 4° Venturi
Figure 6. Velocity (left) and void ratio (right) profiles, steady solutions, 4° Venturi
Figure 7. Wall pressure evolution, steady solutions, 4° Venturi
Figure 8. Contours of the void ratio over one period, incompressible code, 4° Venturi
Figure 9. Contours of the void ratio at five instants, compressible code, 4° Venturi
Figure 10. Time-averaged velocity (left) and void ratio (right) profiles, unsteady solutions, 4° Venturi
Figure 11. Time-averaged wall pressure evolution, unsteady solutions, 4° Venturi
Figure 12. Schematic view of the $8^\circ$ Venturi
Figure 13. Photograph of the cavity, 8° Venturi
Figure 14. View of three stations of measurement, $8^\circ$ Venturi
Figure 15. Enlargement of the mesh near the throat, $8^\circ$ Venturi
Figure 16. Evolution of the vapour volume, incompressible code (left) and compressible code (right), 8° Venturi
Figure 17. Fourier transformation of the vapour volume, incompressible code (left) and compressible code (right), 8° Venturi
Figure 18. Contours of the void ratio over one period, incompressible code, 8° Venturi
Figure 19. Contours of the void ratio over one period, compressible code, $8^\circ$ Venturi
Figure 20. Time-averaged velocity (left) and void ratio (right) profiles, 8° Venturi