NUMERICAL SIMULATION OF UNSTEADY CAVITATING FLOWS:
SOME APPLICATIONS AND OPEN PROBLEMS

JL. Reboud
LEGI, Grenoble, France
now: CNRS-LEMD, University of Grenoble, France
jean-luc.reboud@grenoble.cnrs.fr

O. Coutier-Delgosha
LEGI, Grenoble, France
now: ENSTA – UER de Mécanique, Palaiseau, France
coutier@ensta-bretagne.fr

B. Pouffary
LEGI, Grenoble, France
pouffary@hmg.inpg.fr

R. Fortes – Patella
LEGI, Grenoble, France
fortes@hmg.inpg.fr

ABSTRACT

The Turbomachinery and Cavitation team of LEGI (Grenoble) develops numerical models to take into account the steady state and unsteady effects of cavitating flows. That work is performed in collaboration with the French Space Agency CNES and the Rocket Engine Division of SNECMA-Moteurs, to predict the cavitation behaviour of turbomachinery. The final aim is to provide assistance to the design and revision of operating range of rocket engine turbopumps.

In this presentation we summarize different types of problems encountered, and illustrate possible applications of the numerical models. This part is widely based on previous papers. We give also an overview of some open problems concerning unsteady cavitation modelling, such as the coupling between a cavitating section and the inlet and outlet pipes or the comparison and validation of different models.

INTRODUCTION

As the first attempt to simulate unsteady cavitation phenomena Furness and Hutton [1] consider the deformation of the liquid-vapour interfaces to describe the early stage of re-entrant jet formation. For the simulation of complete cyclic behaviour, including vapour cloud shedding, the tracking of the liquid-vapour interface becomes a quite impossible challenge, because of multiple splitting of the main vapour structures and very quick vaporisation and condensation phenomena. An alternative approach is proposed for about 15 years (Delannoy and Kueny, 1990 [2]. Kubota et al., 1992 [3]). It consists in modelling the cavitating liquid as a homogeneous two-phase mixture of liquid and vapour. One main assumption in this case is to neglect the possible slip between the two phases, which leads to a single-phase fluid whose density may vary over a large range from pure liquid to pure vapour. This approach has been investigated in different ways:

- Delannoy and Kueny [2] proposed a formulation that strongly links the mixture density to the static pressure: they use a barotropic law $\rho(P)$, which describes the mixture density both in the incompressible parts of the flow field and in the transition zone. This kind of model has been applied during the last decade by other researchers with different state laws (Song and He [4], Shin and Ikohagi [5], Ventikos and Tzabiras [6]...).

- Kubota et al. [3] proposed to relate the density evolution to the motion of bubbles in the flow. A given number of bubbles are settled at the inlet, and their evolution is governed by the Rayleigh-Plesset equation according to the pressure field. It is assumed that the bubbles are spherical, and remain not too close from each other. The void fraction is thus theoretically limited to a small value. However, that approach has been widely developed since that first work, for example by Chen and Heister [7], Grogger and Alajbegovic [8], Bunnel and Heister [9] or Sauer and Schnerr [10].

- Different authors proposed more recently to consider a transport equation model for the void ratio, with vaporization/condensation source terms to control the mass transfer between the two phases (Shingal et al. [11], Merkle et al. [12], Kunz et al. [13], Senocak and Shyy [14]...). This method has the advantage that it can take into account the time influence on the mass transfer phenomena, through empirical laws for the source term.

The main numerical problem in multidimensional simulations is the simultaneous treatment of two very different flow conditions: two almost incompressible ones (pure liquid and pure vapour), and a highly compressible one in the
transition between vapour and liquid. Most of the methods have serious difficulties when the ratio \( \rho_v/\rho_l \) is lowered. Mainly two numerical methods have been developed to simulate unsteady cavitation:

- The first method adapted to incompressible fluids computations, is the pressure correction method, based on the SIMPLE scheme. Delannoy and Kueny [2], adapted this algorithm to cavitation by considering the mass equation as a transport equation for density, which depends on the pressure through a barotropic state law. This method, first developed for inviscid fluids, considers a physical speed of sound in the vapour/liquid transition and thus captures the very strong density gradient in the mixture [6, 14].

- The second method is based on the adaptation of compressible time-marching algorithms to low Mach numbers: artificial compressibility method. This kind of resolution was originally devoted to highly compressible flows. In the case of low-compressible or incompressible simulations the efficiency of the numerical scheme decreases dramatically. That problem was solved (Turkel [15]) by multiplying the pseudo-time derivatives by a preconditioning matrix that modifies the equations and accelerates the convergence, without altering the result accuracy provided each time step is correctly converged. Choi and Merkle [16] implemented it in an implicit algorithm, and they obtained satisfactory results, either with a barotropic law or a two mass equations model. Kunz et al. [17] presented results obtained with this algorithm, using a three-fluid method based on two mass transfer equations and including the presence of a non-condensable gas.

The Turbomachinery and Cavitation team of LEGI (Grenoble-France) develops models of unsteady cavitating flows for many years, with the constant support of the French Space Agency CNES – Centre National d’Etudes Spatiales. The initial model "IZ" was based on the Euler equations and developed for 2D flows [2]. Successive improvements of the 2D model "IZ" and applications to different flow configurations were performed during the last decade [18-26]. In parallel, the cavitation model was implemented in the 3D RANS code FINE/TURBO™, for applications to cavitating inducers, in collaboration with CNES and the rocket engine division of SNECMA-Moteurs [27-33]. That work, still in progress, has the final objective to provide assistance to the design and prevision of operating range of rocket engine turbopumps, taking into account the steady state and unsteady effects of cavitation. Other applications of unsteady cavitation modelling are also investigated in the research team, for example in the field of cavitation erosion.

In this lecture, we will summarize types of problems encountered during the model development and illustrate different possible applications of the numerical models. This part is widely based on previous papers. We will give also an overview of some open problems concerning unsteady cavitation modelling, as the coupling between a cavitating section and the inlet and outlet pipes or the comparison between different models and their experimental validation.

### NOMENCLATURE

- **c_{min}**: minimum speed of sound in the medium (m/s)
- **Cp**: \((P-P_{ref})/(\rho_{ref}V_{ref}^2/2)\) pressure coefficient
- **k**: turbulent kinetic energy per unit mass (m² s⁻²)
- **L_{ref}**: geometry reference length (m)
- **P**: local static pressure (Pa)
- **P_{ref}**: reference pressure (Pa)
- **P_v**: vapour pressure (Pa)
- **V_{ref}**: reference velocity (\(=V_{upstream}\)) (ms⁻¹)
- **\(\alpha\)**: void ratio
- **\(\epsilon\)**: turbulence dissipation per unit mass (m²s⁻³)
- **\(\mu_{t}, \mu_{l}\)**: laminar and turbulent dynamic viscosity (Pa.s)
- **\(\rho\)**: mixture density (kg/m³)
- **\(\rho_{liq}, \rho_{vap}\)**: liquid (=ref), vapor density (kg/m³)
- **\(\sigma:\ (P_{ref}P_{v})/(\rho_{ref}^2)\)**: cavitating number
- **\(\omega\)**: specific dissipation rate (s⁻¹)

### MODELS BASES

We use a single fluid model to describe the liquid-vapour mixture, with a mixture density \(\rho\) varying in the flow field between the vapour density and the liquid density. The equivalent void ratio \(\alpha\) of the liquid-vapour mixture relates to the varying specific mass by \(\rho = \alpha \rho_v + (1 - \alpha) \rho_l\). In the mixture, the velocities of liquid and vapour phases are the same and we obtain only one set of equations for the mixture mass, momentum, temperature or turbulence (k-\(\epsilon\) or k-\(\omega\)), written in their conservative form.

The void ratio \(\alpha\) of the mixture depends of the local static pressure. The main work was performed in our laboratory, as initially proposed by Delannoy and Kueny, [2], using a barotropic state law \(\rho(P)\) to manage the relation between pressure and mixture density. A smooth arbitrary law was chosen, \(\rho\) rapidly varying between liquid density \(\rho_l\) and vapour density \(\rho_v\) when the local static pressure \(P\) is around the vapour pressure \(P_v\). The law is characterised by its maximum slope at \(P = P_v\), which is related to the minimum speed of sound \(c_{min}\) in the two-phase homogeneous medium. No significant effects of the ratio \(\rho_v/\rho_l\) is observed, provided its value is imposed smaller than 0.01. \(c_{min}\) is then the only adjustable parameter of the model, found from comparisons with the experimental results of Stutz and Reboud [31, 32] close to 1 to 2 ms⁻¹. That order of magnitude is kept constant for the different applications with water.
vapour and unsteady effects have to be taken into account in
is based on an upstream pressure measurement as reference
to the vapour pressure \( P_v \). Because the experimental value of
spontaneously in the regions where the pressure decreases close
to the cavitating number \( \sigma \). Phase change then occurs
oscillation behaviour \( [35, 37] \), with quasi-periodic
fluctuations. For a cavitation number \( \sigma \) of about 2.4 (based on
the time-averaged upstream pressure) and an inlet velocity \( V_{inf} = 7.2 \) m/s, the vapour shedding frequency observed
experimentally is about 50 Hz (\( \approx 1.5 \) \( V_{inf}/L_{cav} \)) for a maximum
attached cavity length of 45mm (\( \approx 0.2 \) \( L_{cav} \))\( \pm \) 5mm. That gives
the classical Strouhal number, based on the cavity length: \( St = 0.3 \).

3D Model

To calculate 3D cavitating flows the barotropic cavitation
model was implemented in the commercial code FINE/TURBO\(^\text{TM} \) developed by NUMECA International.
FINE/TURBO\(^\text{TM} \) is a three-dimensional structured mesh code
that solves the time dependent Reynolds-averaged Navier-
Stokes equations, with artificial compressibility method. A
detailed description of the code is given in Hakimi [40]. Time
accurate resolutions use the dual time stepping approach.
Pseudo-time derivative terms are added to the equations. They
march the solution towards convergence at each physical time
time. The code resorts to a multigrid strategy to accelerate the
convergence in non-cavitating conditions, associated with a
local time stepping and an implicit residual smoothing. In the
case of low-compressible or incompressible flows, the pseudo-
time derivatives are multiplied by a preconditioning matrix
(Hakimi [40]) based on the studies of Turkel, [15], Choi and
Merkle, [16].

The space discretization is based on a finite volume
approach. A second order central scheme is associated with two
artificial dissipation terms, respectively of second and fourth
order. The first one is activated in the strong pressure and
density gradient areas. The other one is used in the whole
domain, and it results in a second order space accuracy. The
pseudo-time integration is made by a four-step Runge-Kutta
procedure. The physical time-derivative terms are discretized
with a second order backward difference scheme that ensures a
second order accuracy in time.

EXAMPLES OF APPLICATIONS AND OPEN
PROBLEMS

Self oscillation behaviour in 2D venturi : influence of the
turbulence model

Many numerical simulations were performed on a Venturi
type section whose convergent and divergent angles are
respectively about 18° and 8°. The shape of the Venturi bottom
downstream from the throat simulates an inducer blade suction
side with a bevelled leading edge geometry and a chord length
\( L_{cav} = 224 \) mm. According to experimental observations in this
gamma cavitation sheets develop from the throat and show a
typical self-oscillation behaviour [35, 37], with quasi-periodic
fluctuations. For a cavitation number \( \sigma \) of about 2.4 (based on
the time-averaged upstream pressure) and an inlet velocity \( V_{inf} = 7.2 \) m/s, the vapour shedding frequency observed
experimentally is about 50 Hz (\( \approx 1.5 \) \( V_{inf}/L_{cav} \)) for a maximum
attached cavity length of 45mm (\( \approx 0.2 \) \( L_{cav} \))\( \pm \) 5mm. That gives
the classical Strouhal number, based on the cavity length: \( St = 0.3 \).
Four turbulence models have been compared to simulate unsteady cavitating flows in the Venturi type section [24]. Results obtained with the standard \( k-\varepsilon \) RNG model and with the \( k-\omega \) model [39] without taking into account compressibility effects are in poor agreement with experimental observations: the models lead to a stable cavitating sheet, as illustrated by figure 3 and fail to reproduce the vapour cloud shedding behaviour observed experimentally. On the other hand, an empirical modification of the \( k-\varepsilon \) RNG model, consisting in reducing the turbulent viscosity in the mixture regions [19, 24], and the \( k-\omega \) model including compressibility effects [39] lead to a reliable simulation of the Venturi unsteady cavitation behaviour (figure 4).

![Figure 2: Curvilinear-orthogonal mesh of the Venturi type section (160x50 cells). \( L_{ref} = 224 \text{ mm} \)](image1)

![Figure 3: Time evolution of the cavity length. Calculation conditions: \( \sigma = 2.4; V_{ref} = 7.2 \text{ m/s}; mesh = 160x50 \) - time step \( \Delta t = 0.005T_{ref} \) \( (T_{ref} = L_{ref}/V_{ref}) \), standard \( k-\varepsilon \) RNG model.](image2)

![Figure 4: Time evolution (in abscissa) of the cavity length (graduated in ordinate). Instantaneous density distribution of attached and cloud cavities are drawn on the left at \( T = 11T_{ref} \). Calculation conditions: \( \sigma = 2.4; V_{ref} = 7.2 \text{ m/s}; mesh = 160x50 \) - time step \( \Delta t = 0.005T_{ref} \) \( (T_{ref} = L_{ref}/V_{ref}) \), modified \( k-\varepsilon \) RNG model.](image3)

![Figure 5: Time averaged values and standard deviation of velocity (1, 2) and mean void ratio (3). Numerical results with modified \( k-\varepsilon \) (lines) and optical probe measurements [37].](image4)
The satisfactory results obtained with the modified k-ε RNG model have been confirmed by simulations performed in other geometries, such as a hydrofoil [20], a foil cascade [23], and another Venturi type section leading to a more stable cavitation sheet [25]. In all cases, the general experimental behaviour is correctly obtained, and oscillation frequencies are well predicted in unsteady configurations. Moreover, velocity and void ratio distribution within the cavity compare favourably with optical probes measurements performed in the same geometry [19, 25, 35-37] (figure 5).

According to the numerical calculations, the fluid compressibility in the mixture regions seems to play a significant role in the turbulence structure, and must be taken into account to simulate the local unsteadiness generated by cavitating flows. The physical analysis and modelling of the local interaction between unsteady flows, vapour structures and turbulence, remains a wide field of investigation for researchers.

**Complex interaction between cavitation and unsteady flow**

In the scope of the European Research Program PROCOPE, researchers of the LEGI and of the Chair of Turbomachinery and Fluid Power at TU Darmstadt (Germany) worked together in order to improve the understanding of the unsteady behaviour of cavitating flows and the related erosive aggressiveness. Cavitation was studied in a cascade of three hydrofoils mounted in a cavitation tunnel (Lohrbeg et al. [23]).

**Figure 6 : Cavitation tunnel and hydrofoils cascade geometry, from [23]**

**Figure 7. Time evolution of the cavitation structures on the three hydrofoils (Vref = 14m/s, \( \sigma = 1.95 \))**

**Density field at \( t = 160 \) and \( 172 \) ms (=45 and 48 tref)**

**Variation with time (Tref = Lref/Vref = 3.6 ms) of the size of the cavitation structures in the channels.**
At given cavitation number observations showed two dominant frequencies, whose estimation needed a special experimental treatment in the autocorrelation domain [23]. On a wide range of cavitation numbers, the lower frequency is about 40 Hz and does not depend on the cavitation number. The higher frequency corresponds to a Strouhal number close to 0.3 for the upper channel, where typical self-oscillating cloud cavitation is observed. At phase angles, when the cavitation in the upper channel is long, the cavitation structures in the other channels are short and vice versa.

Numerical simulation of that cascade configuration shows similar behaviour (figure 7). According to the computation results, the highest frequency correspond to the self-oscillating behaviour of the cavity in the upper channel, while the lowest frequency appears to be strongly linked to the coupling of the three channels. Analyses indicate that vortices generated by the cloud cavitation phenomena in the upper channel are convected by the mean flow to the trailing edge. The flow repartition in the different channels appears strongly affected by the interaction between the travelling vortices and the trailing edge flow of the upper foil, what seems to be the origin of the low frequency phenomenon.

The good agreement obtained between experimental and numerical results (figure 8) indicates that the proposed model well describes the interaction between unsteady cavitation behaviour and associated hydrodynamic effects. The model can then be used as an interesting tool to understand the mechanisms of complex hydrodynamic interactions in unsteady cavitating flows.

In the applications presented, the circuit impedance is ignored by the steady boundary conditions imposed at the tunnel upstream and downstream ends. One consequence is that the pressure and flow rate fluctuations generated during the cavitation cycles are not properly distributed on both sides. One effect can be illustrated below:
- when the volume of vapour of a cavitation structure (attached sheet or cloud) decreases, the instantaneous mass flow rate downstream of the cavity is reduced with respect to the value imposed as upstream boundary condition.
- when the vapour structure finally collapses, the mass flow rate increases then quite instantaneously in the whole region downstream because of the very low compressibility of the liquid.
- the pressure then rises in the cavitation region, with respect to the inertia effects and the fixed static pressure imposed as outlet boundary condition. That global increase of static pressure coupled with the barotropic law causes the total disappearance of any vapour structures during some time steps.

The result of such sequences is an artificial increase of the amplitudes of the pressure fluctuations upstream, of the vapour volume variations and of the flow rate fluctuations downstream.

To solve that problem, we are testing two modifications of the model:
- the barotropic approach can be replaced by the transport equation for the void ratio presented previously. Because it takes into account a characteristic time limiting the phase change, a sudden increase of pressure does not lead so quickly to the total disappearance of all the vapour structures.
- the effect of the hydro-elastic behaviour of upstream and downstream pipes can be taken into account. Because the computation is purely transient, we use a simplified 1D hydro-elastic model, that solves the Allievi's equations taking into account the speed of sound in the pipes and their diameter (Longatte [41]). Non-reflecting boundary conditions are applied at one end of each pipe, the other end being coupled with the 2D computational domain of the cavitation tunnel. So pressure and flow rate fluctuations can leave the cavitation tunnel on either side, with respect to the hydro-acoustic characteristic of the connected pipes. Slow pressure or flow rate variations (i.e. with a characteristic time much larger than the cavitation cycle period) are corrected, to impose a given time-averaged value of flow rate and cavitation number.

An example of such modifications is illustrated in figure 9. It can be noticed than the instantaneous void ratio is always smaller than with the barotropic state law and the fixed upstream and downstream conditions (figure 4). With the parameters chosen (pipes diameter = 0.25 m, speed of sound = 1000 m/s, transport equation of $\alpha : C_{1V}/\tau = 7, C_{V}/\tau = 3, \rho V/p_l = 10^{-3}$) the averaged vapour volume is about 1/3 smaller, the cycle frequency about 20% higher. Pressure fluctuations are reduced from a factor 2. Influences of the parameters for the transport equation of the void ratio and the pipes characteristics are under investigation.

**Figure 8. Main frequency of the low and high pass filtered pressure signal as a function of the cavitation number $\sigma$, comparison between experimental measurements and numerical simulation. $V_{ref} = 14$ m/s (From [23]).**

**Influence of the upstream and downstream conditions and of the cavitation model**
Figure 9: Time evolution (in abscissa) of the cavity length (graduated in ordinate). Instantaneous density distribution of attached and cloud cavities are drawn on the left at $T = 20.1T_{\text{ref}}$. Calculation conditions: $\sigma = 2.4$; $V_{\text{ref}} = 7.2$ m/s; mesh = 160x50 - time step $\Delta t = 0.005T_{\text{ref}}$ ($T_{\text{ref}} = L_{\text{ref}} / V_{\text{ref}}$), modified $k$-$\varepsilon$ RNG model.

1D Pipes : $\emptyset$ 0.25 m, speed of sound 1000 m/s
Transport equation of $\alpha$: $C_{Lx}/t = 7.5$, $C_{Lx}/t = 3$, $pv/p\varepsilon = 10^{-3}$

Comparisons with experiments remain an important challenge, more particularly in unsteady configurations. Computations provide the complete time-dependant fields of velocity, pressure and mixture density. Besides, empirical parameters and crude assumptions are used and need to be validated for various flow configurations. Future collaborations with experimentalists are therefore strongly needed.

Applications to 2D inducer blade cascades.

One source of unsteadiness in turbopump inducers consists in rotating cavitation behaviour, characterized by different cavity shapes on the different blades, and leading to super or sub synchronous disturbances. We intend to simulate this phenomenon in the case of 2D blade cascades corresponding to typical four-blade inducers [22, 26]. Therefore, the numerical model of 2D unsteady cavitating flows was adapted to take into account non-matching connections and periodicity conditions [21].

Single-channel and four-channel computations were performed. In the second case non-symmetrical unstable flow patterns were obtained for certain operating conditions (illustration of rotating cavitation result is given in figures 11 and 12). Limits of stability according to the mass flow rate and the cavitation number are obtained [22, 26].

Illustrations and comment of recent results are given in CAV2003 paper by Coutier-Delgosha et al. [26].

Figure 10: Cascade geometry with non-matching cells at the connections.

Blades highly inclined because of the inducer small axial length

High curvature $\rightarrow$ non matching cells at the connections

Figure 11: Cavitation behaviour at reference flow rate and $\sigma = 0.15$ (ratio 5:1 between horizontal and vertical scales)
3D steady cavitating flows in turbomachinery

3D computations of steady cavitating flows in turbomachinery have already been performed, with a first version of the cavitation model implemented in FINE/TURBO™. They concern the pure radial pump designed and tested at TU Darmstadt [32], the SHF centrifugal pump tested at Electricité de France [30,31], and turbopump inducers designed by SNECMA [28]. Cavitating behaviours obtained by the computations were compared to experimental measurements and visualizations. A reliable agreement is obtained for the radial and centrifugal pumps concerning both the head drop charts and the development of the vapour structures (examples in figure 13-16).

Figure 14: Head-drop curves of the TUD pump
Comparison at 0.8Qn, Qn and 1.08Qn

Figure 15: Comparison experiment/calculation of the suction side cavity extension for several NPSH values, SHF pump,
Experimental visualizations by EDF [30].
Because the head drop of a cavitating pump leads to very complicated flows, work is still in progress to assure a good convergence of the numerical scheme even during the last stages of the NPSH decrease. Moreover convergence difficulties related to the very complex inlet flow are also under investigations in inducer geometry [33].

Improvement and validation of the numerical model for unsteady cavitating flows is performed in 2D geometry [34] and despite the very large computation time needed application to 3D flows is planed.

ACKNOWLEDGMENTS

The authors wish to express their gratitude to the French space agency CNES and SNECMA-Moteurs for their support.

REFERENCES

[37] Stutz, B., Reboud, J.L (2000), "Measurements within unsteady cavitation", Experiments in Fluids n°29 pp545-552