A cavitation model based on Eulerian stochastic fields

F Magagnato¹ and J Dumond²

¹Institute of Fluid Mechanics, Karlsruhe Institute of Technology, Karlsruhe D-76131, Germany
²AREVA GmbH, Erlangen, Paul-Gossen-Straße 100, Erlangen D-91052, Germany

E-mail: franco.magagnato@kit.edu

Abstract. Non-linear phenomena can often be described using probability density functions (pdf) and pdf transport models. Traditionally the simulation of pdf transport requires Monte-Carlo codes based on Lagrangian “particles” or prescribed pdf assumptions including binning techniques. Recently, in the field of combustion, a novel formulation called the stochastic-field method solving pdf transport based on Eulerian fields has been proposed which eliminates the necessity to mix Eulerian and Lagrangian techniques or prescribed pdf assumptions. In the present work, for the first time the stochastic-field method is applied to multi-phase flow and in particular to cavitating flow. To validate the proposed stochastic-field cavitation model, two applications are considered. Firstly, sheet cavitation is simulated in a Venturi-type nozzle. The second application is an innovative fluidic diode which exhibits coolant flashing. Agreement with experimental results is obtained for both applications with a fixed set of model constants. The stochastic-field cavitation model captures the wide range of pdf shapes present at different locations.

1. Introduction

The vaporization of a fluid following a sudden static pressure decrease is called cavitation. This phenomenon was already discussed by Lord Rayleigh in 1917 [1]. Cavitation starts at favorable places in the flow, called nuclei. In engineering applications, the nuclei typically consist of air and vapor bubbles in the bulk or trapped in crevices of impurities and walls (Figure 1.). The fluid vaporizes at these nuclei in regions where the pressure is below the saturation pressure. These nuclei become larger cavities, which then collapse in regions where the pressure is above saturation pressure. Meanwhile, the cavities are transported and interact with the surrounding fluid and the surrounding cavities. They are submitted to forces, heat exchange, coalescence, and breakup.

Cavitation is a complex process which can be exploited in many engineering fields. In chemical or power generation industries, accidental loss of pressurized fluids can be reduced when cavitation is enforced upstream of choking cross sections; in medicine [2,3], textile manufacturing [4,5], or water treatment [6], strong pressure waves and high velocity jets resulting from bubble collapse can be used to fragment and destruct molecules. In hydraulic systems, pumps and turbines, cavitation is often undesirable. Enhanced total pressure losses reduce efficiency of machinery [1,7-10]; reduced sound speed leads to blocking in pipes or valves [11,12]; collapse of bubbles accompanied with intense microjets damages surfaces [1]. Furthermore, two-phase flow instabilities may be encountered [13].
1.1. Challenge of cavitation modeling

The capability to investigate cavitation with a numerical code is of great benefit in the design and operation of many engineering systems. However, the simulation of cavitating flow is a challenging problem both in terms of developing robust numerical methodologies and modeling the physics. Cavitating flows are characterized by considerable variations of the local density and speed of sound; both having a strong impact on the stability and efficiency of a numerical code. Cavitating flows are also inherently stochastic [9] as a result of both water quality [nuclei-size [14] and nuclei number [15] probability density function (pdf) have large variance (Figure 2)] and turbulence-cavitation interaction [Nuclei activation and bubble growth are influenced by turbulent pressure fluctuations [16]; breakup [17] and coalescence [18] are affected by turbulent velocity fluctuations; growth and collapse of cavities initiate disturbances while cavities may damp turbulent fluctuations]. Finally, most phenomena occurring in cavitating flows are highly non-linear and predominantly controlled by the size and number of the vaporeous cavities.

For example, the mass transfer occurring between the phases in a volume $V$ at time $t$ is the mass transfer $\dot{M}(t)$ occurring between the liquid and the $N(t,V)$ cavities present in this volume:

$$\dot{M}(t) = \sum_{i=1}^{N(t,V)} \dot{m}_i(t)$$

(1)

$\dot{m}_i(t)$ being the mass transfer between the liquid and the $i^{th}$ cavity. This interfacial mass transfer is highly non-linear since vaporization occurs [$\dot{m}_i(t) \neq 0$] only for those nuclei which have a size $R(t)$ larger than a critical radius $R_{cr}$. This critical radius depends on the local fluid pressure $p(t)$ and temperature $T(t)$ and can be written a

$$R_{cr}(p,T,t) = \frac{4\sigma}{3(p_{sat}[T(t)] - p(t))}$$

(2)
$p_{sat}$ being the saturation pressure and $\sigma$ the surface tension.

In engineering applications where knowledge on mean total pressure loss, mass flow rates etc. is needed, a statistical approach is sufficient (and required): the flow property of interest is not the instantaneous number and size of bubbles at any location (which are beyond current computing and experimental capacities anyway) but their statistical distribution in any discretization volume centered at $x$, i.e. the bubble number density and bubble-size joint-pdf $f_{n,R}(n,R;x)$. A numerical code able to reproduce this statistical property yields accurate mean flow properties since average interfacial transfers appear in exact form. For instance, the average interfacial mass transfer is exactly:

$$
\langle M' \rangle = \lim_{T \to \infty} \frac{1}{T} \int_0^T \! \int_0^\infty \! \int_0^\infty \! f_{n,R}(n,R;x) m(n,R) dR dn
t \quad \quad (3)
$$

While models are available for $m(n,R)$ the mass transfer per unit volume of $n$ bubbles of radius $R$, the modeling of the joint-pdf $f_{n,R}(n,R;x)$ resulting from water quality and turbulence–disperse-phase interaction still remains one of the major difficulties of cavitation modeling, and more generally of multiphase modeling.

1.2. Review of existing cavitation models

1.2.1. Standard approach: The Eulerian formulation

The Eulerian formulation (Euler - 1757) is the standard approach to simulate cavitating flows (e.g. Delannoy and Kueny [19]) since Eulerian solvers can efficiently and robustly handle compressibility and large density ratios. Water and vapor are usually assumed to be in mechanical and thermal equilibrium since bubbles are finely dispersed in the liquid phase. Water and vapor share same velocity, pressure and temperature. They are treated as a continuum and an Eulerian solver solves the Navier-Stokes equations of the mixture.

In this framework, two approaches exist to describe the water-vapor mixture: the drift flux model and the homogeneous model. The distinction between both models relies on the treatment of the thermodynamic behavior of the mixture. The drift flux accounts for the thermodynamic non-equilibrium (metastable conditions can be described) by solving the transport equation of the vapor mass fraction $Y$ (Figure 3) whereas the homogeneous model is purely empirical and estimates the vapor mass fraction from mixture properties (energy [20], density [21], etc.). In the first approach, the vapor mass fraction transport equation reads:

$$
\frac{\partial}{\partial t} (\rho Y) + \nabla \cdot (\rho u Y) = -\nabla \cdot (j) + S \quad \quad (4)
$$

where $j$ is the mass flux vector and $S$ is the interfacial mass transfer.

Since the length scales of the bubbles and turbulent fluctuations range down to some micrometers, their resolution is beyond current computing capability in engineering applications: The instantaneous flow properties (pressure, temperature, bubble-size, void fraction etc.) cannot be solved directly and only average flow properties are available to model average phenomena (e.g. $S$ represents the average interfacial mass transfer).

Standard drift flux models introduce algebraic models based on average vapor mass fraction, density and pressure to model the average interfacial mass transfer. This modeling often fails to universally describe cavitating flows: The fluctuations have a significant effect on the average mass transfer but cannot be deduced from average flow properties. For instance, the interfacial mass transfer significantly differs between a liquid containing a few large nuclei (liquid i, Figure 4.) and another liquid containing a lot of very small nuclei (liquid ii, Figure 4.); even though average properties (void
fraction, pressure, etc.) were equal. In this example, all nuclei of the liquid i would grow while interfacial mass transfer would be absent in liquid ii.

![Diagram](image)

**Figure 3.** Representation of the Eulerian approach. The Navier-Stokes equations of the mixture and vapor mass fraction transport equation (Drift-flux formulation) are solved on an Eulerian numerical mesh (a). Only average flow properties [e.g. the average vapor mass fraction (b)] are known. No information on nuclei number density (c) and size pdf (d) are available.

![Diagram](image)

**Figure 4.** Schematic representation of probability density functions (pdf) of nuclei number density (b) and cavitation nuclei size (b) of two liquids (i and ii) with different nuclei distribution but identical void fraction; \( R_{cr} \) being the critical radius [eq. (2)].

This explains why the source terms of the vapor mass fraction equation found in the literature (e.g., [22-24]) may differ by several orders of magnitude; each model being calibrated to reproduce experimental results and, thus, adapted for specific conditions and geometry.

1.2.2. Lagrangian-Eulerian formulation

An improvement of the standard modeling of cavitating flows consists in coupling a Lagrangian (Lagrange - 1788) solver to an Eulerian solver. This approach is the so-called Lagrangian-Eulerian formulation (e.g. Meyer et al. [25]). The Lagrangian solver tracks the vaporous cavities: The trajectory equation, the momentum balance and the bubble dynamic, typically the Rayleigh-Plesset equation [eq. (32)] or one of its modified forms (e.g., [26]), are solved for individual bubbles. The Eulerian solver solves the Navier-Stokes equations of the continuous phase and evaluates additional flow variables, e.g. turbulence and vapor mass fraction. By recording the size of the bubble present in the Eulerian cells, statistical information on the bubbles is obtained (Figure 5) and all interfacial transfers are well represented.

However, this technique is balanced by the typical Eulerian-Lagrangian solver coupling difficulties: computational demand; complex algorithms and interpolation errors. For instance, the number of bubbles within each Eulerian computational cell must be sufficient to obtain a good statistic and convergence [27]. For these reasons, this method is limited to low void fractions and does not prevail in cavitation modeling.
Figure 5. Illustration of the Eulerian-Lagrangian approach in cavitating flows. Bubble’s position \( x_p \) is tracked; bubble’s velocity \( u_p \), radius \( R \) and growth \( \dot{R} \) are calculated along bubble’s path with a Lagrangian solver (a). An Euler solver solves the Navier-Stokes equations of the liquid phase (b). With this technique, the bubble density number (c) and size (d) pdf can be modeled.

1.2.2.1. Direct quadrature method of moment (DQMOM) Recently Bannari [28] applied the direct quadrature method of moment (DQMOM) to model the fluctuations of the bubble-size in cavitating flows. Originated from [29], the DQMOM is an Eulerian technique whose basic idea is to approximate the pdf by a summation of \( N \) Dirac delta functions [30] (Figure 6). Transport equations are solved to determine the position and length of these Dirac functions.

Figure 6. Illustration of the DQMOM method in cavitating flows: Approximation of a bubble size pdf by a summation of 3 Dirac delta functions.

Since the fluctuations of the cavity size are modelled, the interfacial transfers are much better described than in the standard modelling approach of cavitating flows. However, this technique requires the use of an external solver which has to be coupled with the Eulerian solver. Furthermore, although the fluctuations of the bubble-size can be modelled, the modelling of the interaction turbulence-cavitation is not straightforward.

1.2.2.2. Probability density function approach To avoid the use of an external solver, and thus, remaining in a fully Eulerian framework and model turbulence-cavitation interaction, we can model the highly non-linear cavitation process using probability density functions transport models (Figure 7.). Pdf models are common to describe stochastic processes such as the random walk of gas molecules. As an example, relaxation to equilibrium of a fluid consisting of gas molecules is obtained from statistical mechanics in the so-called Boltzmann equation which was derived by Boltzmann in 1872. This Boltzmann equation represents the prototype of a pdf transport equation. In chemical reactions, pdf methods are commonly used to model the interaction chemical species – turbulence, and
thus, to accurately describe the highly non-linear chemical source term. For this purpose, Dopazo and O’Brien [31] introduced the transport equation of the composition joint-pdf
\[ f(\psi; x, t) \cdot f(\psi; x, t) \] are the probability of the event:
\[ \{ \psi_\alpha \leq \tilde{\phi}_\alpha(x, t) \leq \psi_\alpha + d\psi_\alpha, \ \alpha = 1..N_\phi \} \] (5)
for \( d\psi \rightarrow 0 \) where \( \psi = (\psi_\alpha), \ \alpha = 1..N_\phi \); at position \( x \) and time \( t \). \( \tilde{\phi}(x, t) \) is the scalar property vector, each of its components \( \tilde{\phi}_\alpha(x, t), \ \alpha = 1..N_\phi \), \( N_\phi \) being the number of scalars, is a stochastic process. In modeled form, the composition pdf transport equation reads [32]:
\[
\frac{\partial \rho f_\phi}{\partial t} + \frac{\partial \rho u_\phi f_\phi}{\partial x_i} + \frac{\partial \rho S_\phi(\psi) f_\phi}{\partial \psi_{\phi}} = \frac{\partial}{\partial x_i} \left[ D'_\alpha \frac{\partial \rho f_\phi}{\partial x_i} \right] + \frac{1}{2} \frac{\partial}{\partial \psi_{\phi}} \left[ \frac{\psi_\alpha - \langle \tilde{\phi}_\alpha \rangle}{\tau_\phi} \right] \rho f_\phi
\] (6)
Where \( u = (u_i) \) is the mixture velocity, \( \rho \) is the mixture density, \( D'_\alpha \) is the combined molecular and turbulent scalar diffusivity, \( \langle \tilde{\phi}_\alpha \rangle \) is the mean value of the components \( \tilde{\phi}_\alpha \). In eq. (6) term 1 represents the change of rate and convection of the pdf; term 2 is the source term of the pdf; term 3 accounts for the turbulent transport of the pdf and term 4 represents the micromixing in the composition space using the interaction by exchange with the mean model [33]. Note, that the dimensions of this equation include time, space and sample space, yielding \( 1+3+N_\phi \) independent variables. Note the analogy with the Boltzmann equation where gas molecule velocity is introduced as independent variable.

Although the pdf transport equations are derived in an Eulerian framework, their solution generally is achieved by following particles with a Lagrangian solver in combination with Monte-Carlo methods (e.g. Pope [34] in combustion). These particles carry information on the flow properties. An Eulerian solver is needed to exploit particles’ statistical information. In combustion, typically an Eulerian solver is used for the flow equations, e.g. for the pressure equation. The coupling in an Eulerian-Lagrangian solver is complex due to the necessity to locate Lagrangian particles in the Eulerian numerical mesh. Moreover the Lagrangian method is very expensive for a good statistic and convergence. To avoid the cost and complexity of Euler-Lagrangian techniques, Valiño [35] developed the stochastic-field method to solve the pdf in a full Eulerian framework. This new formulation has recently gained substantial interest (e.g., [36-38]).

In this paper, a novel cavitation model based on the stochastic-field method is derived to simulate cavitating flows. As it will be explained in the following, this method allows modeling the fluctuations of the bubble-size and turbulence-cavitation interaction in a full Eulerian framework avoiding the complex and expensive coupling of an external solver (e.g. Lagrangian). In our simulations we make a first attempt to describe transsonic cavitation with the stochastic field method. Therefore aspects of LES-PDF coupling [39] do not represent the primary focus of the publication. In the practical simulations joint unsteady evolution of the LES and the Y-filter density function (FDF) is considered, and no LES-FDF iterations are performed.

In the present work, the method is used to model the bubble-size fdf, though its extension to the bubble density number – bubble-size joint-fdf is straightforward and will be the purpose of future work. Extension of the method to the velocity fdf should be more difficult since a density-based solver is used. The present paper describes the mathematical modeling of this innovative cavitation model in chapter mathematical modeling. Then, it demonstrates the capability of this cavitation model to model the various shapes of the bubble spectrum, and thus, to obtain good agreement with experimental results for validation cases characterized by very differing flow conditions. Two applications are considered in chapter validation cases. Firstly, sheet cavitation is simulated in a Venturi-type nozzle where detailed experimental results are available. These experimental results are exploited to calibrate...
the model constants representing water quality and inertial effects. The second application is an innovative fluidic diode which exhibits coolant flashing in one flow direction. Agreement with experimental results is obtained for both applications with a fixed set of model constants.

Figure 7. Historical context of the proposed stochastic-field cavitation model: In statistical mechanics the pdf ansatz was introduced in the 19th century by Boltzmann so that pdf transport models and corresponding numerical techniques have matured in this field. Cavitation originates from an Euler-Lagrangian point of view and currently develops in the direction of pure Eulerian pdf models. Eulerian and Lagrangian models are framed in blue and red, respectively.

2. Mathematical modeling

2.1. Navier-Stokes equation of the mixture

The proposed modeling is an extension of the standard cavitation modeling approach. Water and vapor are assumed to be in mechanical and thermal equilibrium since bubbles are finely dispersed in the liquid phase. They share same velocity, pressure and temperature and, thus, are treated as a continuum. A density-based Eulerian solver solves the Navier-Stokes equations of the mixture:

Continuity equation:

$$\frac{\partial}{\partial t} \left( \rho \right) + \nabla \cdot \left( \rho \mathbf{u} \right) = 0$$  \hspace{1cm} (7)

Momentum equation:

$$\frac{\partial}{\partial t} \left( \rho \mathbf{u} \right) + \nabla \cdot \left( \rho \mathbf{u} \cdot \mathbf{u} \right) - \nabla \cdot \mathbf{T} = 0$$  \hspace{1cm} (8)

Specific total energy equation:

$$\frac{\partial}{\partial t} \left( \rho E \right) + \nabla \cdot \left( \rho \mathbf{u} E \right) - \nabla \cdot \left( \mathbf{T} \cdot \mathbf{u} \right) + \nabla \cdot \mathbf{q} = 0$$  \hspace{1cm} (9)
In eq. (7) to (9), $u$ is the mixture velocity, $\rho$ is the mixture density, $E = e + 1/2\|u\|^2$ is the mixture specific total energy, $\bar{T}$ is the mixture stress tensor and $q$ is the mixture heat flux density.

2.2. Vapor mass fraction $\text{f}_{\text{df}}$

The consideration of the thermodynamic non-equilibrium is essential in cavitating flows, especially where coolant flashing occurs (e.g. the second validation case presented in this paper). To account for the finite time of cavities' growth, the standard drift flux method solves the transport equation of the vapor mass fraction $Y$ [eq.(4)]. As already discussed in the chapter introduction, no information is available on the fluctuations with this standard modeling approach and, thus, it is very difficult to model the highly non-linear source term $S$. Therefore, in the proposed modeling approach, the transport equation of the vapor mass fraction $\text{f}_{\text{df}}$ is solved instead:

$$
\frac{\partial \rho \text{f}_{\text{df}}}{\partial t} + \frac{\partial \rho u \text{f}_{\text{df}}}{\partial x} + \frac{\partial \rho S(y)\text{f}_{\text{df}}}{\partial y} = \frac{\partial}{\partial x} \left( D_y \frac{\partial \rho \text{f}_{\text{df}}}{\partial x} \right) + \frac{1}{2} \frac{\partial}{\partial y} \left[ \left( y-y'_y \right) \rho \text{f}_{\text{df}} \right] - \rho \text{f}_{\text{df}}
$$

Eq. (10) is derived in analogy to the modeled composition joint-$\text{f}_{\text{df}}$ introduced in combustion [eq. (6)], since the vapor mass fraction is also a transported scalar property. In eq. (10), $u = \left( u_i \right)$ is the mean velocity, $D_y$ is the effective diffusivity coefficient combining molecular and turbulent transport; $S(y)$ is mass transfer associated to $y$ being the sample space at every position $x = (x_i)$ and time $t$; and $\langle Y \rangle$ is the mean value of $Y$. The last term in eq. (10) represents the micromixing of the vapor mass fraction in the composition space using the interaction by exchange with the mean model [33]. Note, that the dimensions of this equation include time, space and sample space, yielding $1+3+1=5$ independent variables.

2.3. Equations of state

A density-based code is used to solve the system of governing equations. In the simulation of cavitating flows, the equations of state (eos) basically are required to calculate the pressure, the temperature and the mean vapor mass fraction $\langle Y \rangle$ from the conserved variables. These primitive flow variables are required in the transport equations and for the calculation of the constitutive properties and speed of sound. Since the calculation of primitive variables is performed at any iteration in any cells, simple analytical eos are advantageous. Large simulation speed up and improved code stability are expected in comparison to techniques which access tabulated eos and liquid-vapor thermodynamic tables.

However, simple analytical eos covering the wide range of pressures (0-7.5 MPa) and temperatures (293-575K) involved in our validation cases are not available in the literature. Furthermore caloric eos are erroneous in most cavitation models. The error associated to these caloric eos does not have much influence on the results at low pressures but yields unphysical results at higher pressures. Thus, one part of the development of the cavitation model is to collect in the literature or derive simple analytical eos covering the range of our flow conditions. Flow properties whose variation significantly impacts the numerical results are also implemented as a function of the temperature.

2.4. Vapor

Similarly to other cavitation models, vapor is modeled as an ideal gas. Its thermal eos is given by:

$$
p = \rho_g R_g T;
$$

with $R_g = 461.5$ J/(kg K). The caloric eos of vapor is expressed in the form:
where $K_{v,v}(T)$ is a simple bijective function of $T$:

$$K_{v,v}(T) = K_{v,1} \sqrt{T_c - T} + K_{v,3}.$$  

(13)

$T_c$ is the critical temperature; $K_{v,1}$ and $K_{v,3}$ are two constants chosen such that eq. (12) provides a good approximation of $e_g$ in the domain of interest. The vapor heat capacity $\left(c_p\right)_g$ is assumed to be constant.

2.5. Water

The eos of water is described by the model of Tamman [4]:

$$p + p_T = \rho_l K_T (T + T_T)$$  

(14)

In the present work, model constants $p_T$ and $T_T$ are set to their standard values derived by Tamman [4]:

$$p_T = 1944.61 \text{ MPa} ; \quad T_T = 3837 \text{ K}.$$  

The model constant $K_T$ is modified to fit water properties (obtained from IAPWS [40]) over large pressure and temperature ranges (until at least 7.5 MPa):

$$K_T (T \text{°C}) = \begin{cases} 
1.6 \cdot 10^{-3} T^2 - 5.78 \cdot 10^{-2} T + 472.22 & \text{if } T < 220 \text{°C} \\
9.893 \cdot 10^{-1} T + 315.61 & \text{otherwise}
\end{cases}$$  

A simple bijective function $K_{v,l}(T)$ of $T$:

$$K_{v,l}(T) = K_{v,2} \sqrt{T_c - T} + K_{v,3}$$  

(15)

is introduced to approximate liquid specific internal energy $e_l$:

$$e_l = K_{v,l}(T)$$  

(16)

Constants in this equation are chosen such as they provide a good approximation of $e_l$ in the domain of interest. Similarly to $\left(c_p\right)_g$, water heat capacity $\left(c_p\right)_l$ is assumed to be constant.

2.6. Mixture

The mixture density is expressed as a linear combination of the density, $\rho_l$ and $\rho_g$, of water and vapor respectively, with the void fraction $\alpha$:

$$\rho = \alpha \rho_g + (1 - \alpha) \rho_l.$$  

(17)

$\alpha$ is calculated according to the mean void fraction $\langle Y \rangle$:

$$\alpha = \frac{\rho \langle Y \rangle}{\rho_g}.$$  

(18)
Following the work of [41], the eq. (19) of a locally homogeneous gas-liquid medium is written with eq. (11), (14) and (17) as:

\[
\rho = \frac{p(p + p_l)}{K_l(1 - \langle Y \rangle)p(T + T_l) + R_g \langle Y \rangle(p + p_l)T}.
\]  

(19)

The speed of sound in the mixture is derived from this equation [41]. This model is compared to experimental results in [42] and show remarkable agreement. Saturation pressures and temperatures are calculated with the polynomials of [43]. The contribution of the surface tension energy to the specific internal energy of the system \(e\) is negligible in comparison to the contribution of liquid and vapor specific internal energies, \(e_l\) and \(e_g\), respectively. Under this assumption, the specific internal energy \(e\) of the system consists of the specific internal energy \(e_l\) and \(e_g\) of the liquid and gas phase, respectively; in proportion to their respective mass fraction:

\[
e = \langle Y \rangle e_g + (1 - \langle Y \rangle)e_l.
\]  

(20)

Vapor heat capacity \(c_p\) and water heat capacity \(c_{pl}\) are mass weighted to obtain the heat capacity \(c_p\) of the mixture:

\[
c_p = \langle Y \rangle(c_{pg}) + (1 - \langle Y \rangle)(c_{pl}).
\]  

(21)

The water-vapor mixture is assumed to be a Newtonian fluid. In this work, the molecular viscosity is expressed according to the formula found in [44]:

\[
\eta = (1 - \alpha)(1 + 2.5\alpha)\eta_l + \alpha\eta_g.
\]  

(22)

The heat flux density \(q\) in eq. (9) is modeled by Fourier’s law. The mixture thermal conductivity \(\lambda_t\) is given by:

\[
\lambda_t = \frac{c_p\eta}{Pr};
\]  

(23)

where \(Pr\) is the mixture Prandtl number defined as:

\[
Pr = \langle Y \rangle Pr_g + (1 - \langle Y \rangle)Pr_l;
\]  

(24)

with \(Pr_g\) and \(Pr_l\) being vapor and liquid Prandtl numbers, respectively.

2.7. Large eddy simulation

In the present paper, the numerical simulations are large eddy simulations (LES). The Navier-Stokes equations [eq. (7) to (9)] are spatially filtered: only the turbulent fluctuations which are larger than the grid size are resolved. To model the diffusive effect of the unresolved turbulent fluctuations on the momentum, both the monotone integrated large eddy simulation (MILES [45]) and High Pass Filter (HPF) Smagorinsky [46] approaches are employed in the following validation cases. In the MILES approach, artificial numerical dissipation obtained from the discretization scheme, here [47], represents the diffusive effect of the subgrid-scale fluctuations while an algebraic model is introduced within the HPF model. The stochastic field method also requires information on the turbulent fluctuations. This information appears through the diffusivity coefficient \(D_t^l\) and the turbulent
relaxation time \( \tau \) (section stochastic field method). In both MILES and HPF calculations, these flow properties are defined according to the following equations:

- \( \tau_y \) scales with the turbulent time scale \( \tau_t \) with a coefficient \( C_y = 2.0 \):
  \[
  \tau_y = \frac{\tau_t}{C_y} ;
  \]  
  (25)

where \( \tau_t \) is defined as [48]:
  \[
  \tau_t = \frac{\rho \Delta^2}{\eta + \eta_{sgs}} ;
  \]  
  (26)

- \( D_y \) scales with the subgrid viscosity \( \eta_{sgs} \):
  \[
  (D_y)_{sgs} = \frac{\eta_{sgs}}{\rho (\sigma_y)_{sgs}} .
  \]  
  (27)

In these expressions, \( (\sigma_y)_{sgs} \) is the sub-grid Schmidt number, \( \Delta \) is the grid size and the subgrid viscosity is defined according to the HPF algebraic model.

2.8. Stochastic field method
The stochastic field method (SFM) of Valiño [35] is a specific technique to account for the additional dimensionality of composition joint-fdf transport equations [e.g. eq. (10)]. The basic idea of the SFM is to represent the fdf \( f_y \) by \( N \) stochastic fields \( Y^k \), \( k \in 1..N \), continuous and differentiable in space and continuous in time [35]:
  \[
  f_y (y; x, t) = \frac{1}{N} \sum_{k=1}^{N} \delta [y - Y^k (x, t)] \equiv \langle \delta [y - Y^k (x, t)] \rangle .
  \]  
  (28)

Quoting Valiño [35], “those \( N \) stochastic fields are not any particular realization of the real field, but constitute a stochastic system allegedly equivalent” to eq. (10). “They represent possible particular scalar values for each value of \( x \) and \( t \).” The approximation of the fdf is exact for an infinite number of stochastic fields \( Y^k \), or at least a very large number of fields, provided an iterative method exists to determine the value of each of these fields. This is far too complex and expensive. Therefore, instead, the stochastic-field method introduces a supplementary dimension, a pseudo-time [Figure 8, top]. The stochastic fields evolve according to eq.(29) during the pseudo-time [see Figure 8, bottom (b)]. At each time and in each cell, the stochastic fields represent realizations of the fdf.

In steady state RANS simulations pseudo-time would be introduced to obtain good statistics with a small number of stochastic fields. In the present study LES is applied in numerical simulations and joint evolution of velocity fields and stochastic Y fields is considered. Thus, no distinction between iteration time step and pseudo time step is foreseen. Both time steps have the same meaning in the following. From a conceptual point of view, though, it would be interesting to explore, if distinguished time steps and specific iteration schemes yield a faster statistical convergence compared to our most simple joint LES-FDF unsteady evolution approach. This will be the purpose of future work. Exactly as in conventional Eulerian-Lagrangian method, probability of fields to have a given value in a finite interval is estimated by binning [see Figure 8, bottom (a), (e) and (d)]. Each realization at each time step and in each cell has a certain probability to occur. The statistics of the vapor mass fraction fluctuations in each cell is obtained for a sufficiently long evolution time in pseudo time [see Figure 8, bottom (d)].
Figure 8. Illustration of the stochastic-field method. In each cell at each time, the fdf is approximated with a finite number of stochastic Eulerian fields [here 8, bottom (a) and (c)]. The statistical convergence is obtained, in similarity to an Eulerian-Lagrangian code, by time averaging (here pseudo time averaging). The fields evolve in time [bottom (b)] to reproduce the statistic of the fluctuations in each Eulerian cell [bottom (d)].

The number of fields should ensure a good approximation of the mean mass vapor fraction $Y$ at each time, and thus a good convergence of the Eulerian solver. According to experience in combustion [49], 8 fields represent a good compromise between stability and efficiency. The transport equation of the stochastic fields is derived in exact form from the fdf transport equation eq. (10) using standard stochastic techniques (see e.g. [50]). For the scalar $Y^k$, $k = 1..N$, it is given by [35]:

$$dY^k = -u_i \frac{\partial Y^k}{\partial x_i} dt + S(Y^k) dt + \frac{\partial}{\partial x_i} \left( D'_Y \frac{\partial Y^k}{\partial x_i} \right) dt + \sqrt{2D'_Y} \frac{\partial Y^k}{\partial x_i} dW^k_i - 1/2 \frac{\left(Y^k - \langle Y \rangle\right)}{\tau_Y} dt.$$  \hspace{1cm} (29)

In eq.(29), $W^k$ is a Wiener process, independent for each spatial component $i$ but constant in space; and $\langle Y \rangle$ is the mean value of the $N$ scalars $Y^k$. $D'_Y$ and $\tau_Y$ are obtained appropriately from a turbulence model and may include molecular transport, e.g. in LES. Note, that the time derivative of a Wiener process is a white noise centered on zero with unity variance, so that $dW^k_i$ can be obtained from a random number generator. Random number usage is the basic idea of Monte-Carlo methods, and therefore the SFM is called a Monte-Carlo Eulerian technique. Eq. (29) is the Ito-formulation of the corresponding fdf transport equation eq. (10). The so-called Stratonovitch calculus [51] would be an alternative formulation but it requires specific boundary treatment and, thus, has not been implemented in the present code.

Practically, the stochastic field equations [eq.(29)] are solved such as standard scalar transport equations; although the discretization scheme should be consistent with the Ito formulation [52]. The cavitation model proposed in the framework of this paper uses the SFM technique in two different ways to improve the accuracy of the Eulerian code.

First, in analogy to eq. (10), the source term $S(Y^k)$ in eq. (29) is the mass transfer associated to the particular value $Y^k$ of the vapor mass fraction. In a fdf model, mass transfer is a closed term so that $S(Y^k)$ does not include any ambiguity resulting from fluctuations. Since non-linear source terms are closed in SFM it proved successful in other applications (e.g. in reacting flows).
Second, the bubble-size fdf is obtained by geometrical consideration from the vapor mass fraction \( f_{df} \). For this purpose, the vapor is assumed to be finely dispersed in the continuous water phase in form of a finite number of \( n \) spherical cavities. This assumption is commonly made in cavitating flows. Thus, the vapor mass fraction can be interpreted as being the vapor mass fraction of one particular bubble: Considering a fluid containing \( n \) cavities per unit volume, it is always possible to divide each unit volume into \( n \) volumes containing exactly one bubble.

Then, each stochastic field represents the vapor mass fraction of one of these volumes, i.e. of one of the bubbles present in the unit volume. Within this framework, the radius \( R^k \) of each of these “representative” bubbles can easily be derived from the mass fraction \( Y^k \) and the number \( n \) of cavities per unit volume:

\[
R^k = \left( \frac{3 \rho Y^k}{4 \pi \rho_y n} \right)^{\frac{1}{3}}.
\]

The fdf of the size of all possible particular bubbles for each value of \( x \) and \( t \) is obtained for a sufficient number of realizations. Here \( n \) is a property of the fluid, which can be modeled as a function of vapor mass fraction to account for the reduction of the number of cavities in case of high vapor content. In this paper, the number \( n \) of cavities per unit volume is related to the average void fraction \( \alpha \) to simulate coalescence and breakup with limiting values \( n_0 \) and unity:

\[
n(\alpha) = \frac{n_0 + 1}{2} - \frac{n_0 - 1}{2} \tanh(5(\alpha - 0.4)).
\]

In eq. (31), \( n_0 \) is the initial number of nuclei in the fluid and \( \alpha \) calculated with eq. (18).

### 2.9. Interfacial mass transfer

The cavity-size fdf can be exploited at any temporal and spatial position to accurately calculate physical processes such as coalescence occurring in cavitating flows. Typically, the proposed cavitation model uses the bubble-size fdf to compute in closed form the mass transfer. This mass transfer is based on a modified form of the Rayleigh-Plesset equation which takes into consideration thermal, water quality and inertial effects.

The Rayleigh-Plesset equation describes accurately the variation of the size \( R \) of a vaporous cavity, and thus the phase-change process, with a varying liquid pressure \( p(t) \). It reads [53]:

\[
\frac{p_{sat}(T_x) - p(t)}{\rho_l} + \frac{p_{sat}(T_b) - p_{sat}(T_x)}{\rho_l} + \frac{m_g T_b R_g}{4 \pi / 3 R^3} = 3 \left( \frac{dR}{dt} \right)^2 + R \frac{d^2R}{dt^2} + \frac{4\nu_l}{R} \frac{dR}{dt} + \frac{2\sigma}{\rho_l R}.
\]

In eq. (32), \( p_{sat} \) is the saturation pressure, \( T_x \) is the temperature of the fluid, \( \rho_l \) is the liquid density, \( T_b \) is the bubble temperature, \( m_g \) is the mass of dissolved gas, \( R_g \) is the gas constant, \( \nu_l \) is the fluid viscosity and \( \sigma \) is the surface tension. On the left hand side, phase exchange is modeled where the first term is the driving pressure difference, the second accounts for thermal non-equilibrium of gas and liquid phase, and the third term for dissolved gases. On the right hand side, the growth or shrinkage of a spherical bubble is corrected by accounting for inertia (second term), viscous forces (third term) and surface tension (fourth term).

In the proposed model, to reduce the computation time, the Rayleigh-Plesset equation is not directly solved since the time constant of the bubble dynamics is much smaller than the turbulent fluctuations time constant. Instead, inertial effects are represented by a model constant \( C_{cond} \) in the condensation...
term. Further model constants (the initial number of nuclei $n_0$ and the equilibrium vapor mass fraction $Y_e$) represent water quality effects. The source term also accounts for the initial nuclei distribution sampling the minimum size of the cavities from a prescribed lognormal distribution. In this manner, the fluctuations of the bubble-size originating from water quality are also included in the model.

Per definition, the source term $S(Y^k)$ of the stochastic-field vapor mass fraction equation is related to the growth of the cavity $R^k$ as follows:

$$S(Y^k) = \frac{D}{Dt} \left( \frac{\rho_g}{\rho} n 4\pi (R^k)^3 \right) = \frac{\rho_g}{\rho} 4\pi n (R^k)^2 \frac{DR^k}{Dt}; \quad (33)$$

when the temporal variation of the cavity number and densities are neglected. In the proposed modeling, the mass transfer $S(Y^k)$ is decomposed into a condensation part $S_c(Y^k)$ and a vaporization part $S_v(Y^k)$. Using eq.(11), (30) and (33), the expression for $S_c(Y^k)$ is:

$$S_c(Y^k) = (36\pi n)^{1/3} (\rho_g)^{1/3} (\rho Y^k)\frac{2}{3\rho_l} \sqrt{\max \left[ p_{sat}(T) - p - \frac{4\sigma}{3R^k}, 0 \right]} . \quad (34)$$

With a Heavyside function $H$ the expression $S_v(Y^k) = S_v^1(Y^k) + H(Y - Y_{eq}) (S_v^2(Y^k) - S_v^1(Y^k))$ switches between two cases:

$$S_v^1(Y^k) = \frac{\rho}{\tau_{nuc}} (Y_e - Y^k); \quad (35)$$

$$S_v^2(Y^k) = -C_{cond} (36\pi n)^{1/3} (\rho_g)^{1/3} (\rho Y^k)\frac{2}{3\rho_l} \sqrt{\max [p - p_{sat}(T), 0]}, \quad (36)$$

where $\tau_{nuc}$ is a model time-constant assuring that vaporous cavities do not become smaller than the initial nuclei after collapse.

Other non-linear phenomena such as drift flux could be estimated with the radius fdf. The fdf can also be applied to model the transfer occurring between the different bubbles, e.g., the stochastic implementation of break-up and coalescence only requires estimation of break-up and coalescence frequencies. All existing physical models available for Lagrangian techniques, presumed fdf or binning methods can be easily adapted to the stochastic-field formulation. In the SFM, the number of samples is identical for all locations (in contrast to Lagrangian techniques) and no a priori knowledge on the expected fdf must be supplied (as would be needed with presumed fdf techniques or binning).

2.10. Solver and numerical scheme

This new concept in two-phase flows is implemented into the state-of-the-art compressible code SPARC [54]. This density-based code can handle large density ratios and high Mach numbers as encountered in cavitating flows. The discretization in time is performed with the four stage Runge-Kutta explicit scheme. This scheme is fourth order accurate in time. The central difference scheme with artificial dissipation SWITCH [47] is used for the spatial discretization. This numerical scheme is second order accurate in space everywhere except at large density jumps (A modification of the sensor was needed to ensure code stability). The fourth order Runge-Kutta scheme was not chosen due to its high numerical accuracy but due to its robustness. In fact it is not consistent with Ito calculus. Since the first predictor step in the Runge-Kutta scheme is an Euler forward step which is consistent with Ito calculus, we expect a numerical solution of first order accuracy in time. The inconsistency with
respect to Ito calculus was considered acceptable, since other assumptions (modeling of the Rayleigh-Plesset equation, turbulence modeling etc.) are supposed to be more restrictive. Nevertheless, future work will focus on implementing a time integration scheme consistent with Ito calculus.

In the following paragraphs, our Monte-Carlo Eulerian cavitation model is shown to simulate the flow behavior under very different cavitating flow conditions in two distinct geometries at affordable computational resources. The calibration of model constants, representing water quality and inertial effects, is achieved with detailed experimental results (Barre et al. [55]; sect. 3. ). Model constants are frozen for further simulations.

3. Validation cases

3.1 Venturi-like nozzle

Experimental results of Barre et al. [55] are utilized to validate and calibrate the novel cavitation model. In these experiments, an attached cavitation sheet develops in the Venturi-type test section CREMHYG (Institut national polytechnique Grenoble). The test section is equipped with pressure and temperature sensors as well as a double optical probe. The double optical probe is used to evaluate the void fraction and velocity field inside the cavity at five horizontal positions. For the selected operating point, the pressure in the tank is \( p = 0.0713 \) MPa and the volume flow rate is \( \dot{V} = 23.75 \text{ dm}^3/\text{s} \). A cavity develops downstream of the contraction. The length \( L_{\text{cav}} \) of this cavity is about 80 mm.

Figure 9. represents the computational domain and boundary conditions. Adiabatic non-slip walls are assumed. Temperature and velocity components are imposed at the inlet, where velocity is constant and normal to the boundary. Its magnitude is calculated according to the experimental volume flow rate. Outlet pressure is varied until the size of the cavitation sheet meets experimental observation since no experimental data of the outlet pressure is reported.

3.2 Numerical details

The test section is modeled with block-structured meshes. The meshes are refined at the contraction; downstream of the contraction; and at the walls. Far upstream and downstream of the contraction, the cells are stretched out to increase the damping of the disturbances. In this manner, increased code stability and faster statistical convergence are obtained.

By means of the full multigrid method, it was possible to study the effect of the grid refinement and increase the speed of convergence of the solution. The calculations are performed on 4 meshes characterized by different grid sizes. The damping of the pressure waves occurs principally on the coarser meshes. Once a statistically steady solution was obtained, the solution was interpolated on a finer mesh. This finer mesh contains 8 times more cells than the coarse mesh since the cells are doubled in all directions. A new calculation is performed until again a statistically steady state is obtained. This procedure is performed until the mesh was sufficiently fine to resolve the small fluctuating scales in the bulk. This mesh has 1.2 million cells. Although the \( x^+, y^+ \) and \( z^+ \) values of the finest mesh correspond to the usually recommended values in resolved (down to the wall) LES calculations [56], the cells were pretty large in the bulk. Therefore, the artificial dissipation was reduced to a very low value (1.5e-4) and the MILES approach was used.
3.3 Model constants

Model constants $n_0$ and $Y_e$ were first evaluated according to available experimental results on nuclei distributions (delivered by coulter counter [57, 58], acoustic and light scattering techniques [59], liquid holograms [58] or cavitation susceptibility meter [60]). These experimental results indicate that the number of nuclei encountered in engineering facilities often is about $n_0 = 10^8\text{ m}^{-3}$. Thus, $n_0$ was set to $10^8\text{ m}^{-3}$ in the first calculation. The nuclei distribution is logarithmic with a mean radius of the order of several micrometers and has a large variance. Thus, $Y_e$ was set to $10^{-7}$ which corresponds to a nucleus of about 8 micrometers in the first calculation.

In contrast, no experimental value is available for $C_{cond}$: $C_{cond}$ replaces the inertial terms in the Rayleigh-Plesset equation which have a much smaller time constant than the flow (also turbulent fluctuations). Thus, $C_{cond}$ is first evaluated with a one dimensional case to $1/500$. Numerical results of the first simulation of the cavitation sheet indicate that cavitation inception occurs further downstream in the numerical simulation than in the experiment. This observation suggests that the nuclei are smaller in the simulation than in reality. In order to palliate this inaccuracy, the number of nuclei was modified to $n_0 = 10^7\text{ m}^{-3}$. With this value, cavitation inception occurs at the same location in the simulation and experiment. The effect of the nuclei distribution ($Y_e, n_0$) was not further investigated in the present work.

The constant $C_{cond}$ has a strong impact on the numerical results (velocity and void fraction profiles, length and stability of the cavitation sheet); especially on the behavior of the flow properties at the rear part of the cavitation sheet. A high $C_{cond}$ value leads to a small unstable cavitation sheet where the vapor condenses as soon as the pressure is above saturation pressure and the strong condensation causes strong instabilities at the rear end of the cavity (pressure waves). In contrast, a small $C_{cond}$ value leads to a long cavitation sheet without flow detachment at the rear part. None of these characteristics is observed in the experiments. Following several try and errors, a value of $C_{cond} = 1/750$ shows good agreement with experimental results.

3.4. Qualitative behavior of results

As in the experiments, an attached stable cavitation sheet is observed in the Venturi-type test section. The intensity of the density field is fluctuating but the length of the cavitation sheet remains approximately constant (Figure 10 and Figure 11).

**Figure 10.** Representation of a density iso-surface ($\rho = 900\text{ kg/m}^3$) obtained in the LES calculation of the stable attached cavitation sheet. The flow conditions and the geometry are given in [55].

**Figure 11.** Numerical results of the LES exhibit an attached cavitation sheet. The cavitation sheet is characterized by the instantaneous density (a) and average density (b) of the vapor-water mixture. A density below 900 kg/m$^3$ indicates the presence of vapor. The vertical lines represent experimental measurement positions.
3.5. Quantitative behavior of results
The calculated ensemble-average velocity profiles at the five longitudinal positions indicated by vertical lines in Figure 11 are in agreement with the experimental results of Barre et al. [55] and Stutz et al. [61]. The calculated velocity profiles are represented by solid lines in Figure 12 and the experimental results by triangles.

![Figure 12. Measured (triangles: Barre et al. [55] and Stutz et al. [61]) and calculated (solid lines) velocity profiles inside the cavitation sheet at the five longitudinal positions indicated by solid lines in Figure 11.](image)

The calculated ensemble-average void profiles at the five longitudinal positions indicated by vertical lines in Figure 11 are in good agreement with the experimental results of Barre et al. [55]. The calculated void profiles are represented by solid lines in Figure 13 and the experimental results by triangles. The void fraction is slightly over-predicted at the rear part of the cavitation sheet.

![Figure 13. Measured (triangles: Barre et al. [55]) and calculated (solid lines) void fraction profiles inside the cavitation sheet at the five longitudinal positions indicated by solid lines in Figure 11.](image)

3.6. Fluidic diode
In the KERENA boiling water reactor developed by AREVA a fluidic diode, called the passive outflow reducer (POR), is required to limit the loss of coolant following a postulated break of a pipe connected to the reactor pressure vessel at low elevation. This flow direction is called the backward direction below. In opposite flow direction, denoted the forward direction, the flow resistance in the POR must be low to guarantee emergency core cooling. An innovative design composed of 37 parallel double-Venturi channels is developed (Figure 14). These channels are shaped to achieve the following requirements: In the backward direction (red arrow on Figure 14), cavitation and energy dissipation are enforced in the first Venturi and choking flow in the second Venturi, where low sound speed and density are present. In the forward direction, (blue arrow on Figure 14), the flow resistance is very low.

![Figure 14. Longitudinal section of POR - 37 channels (blue); material (grey).](image)
3.7. Model constants
The calibrated model constants (section Venturi-like nozzle) are used to simulate the flow behavior of the fluidic diode, i.e., the number density is \( n_0 = 10^7 \text{ m}^{-3} \), the equilibrium vapor mass fraction is \( Y_e = 10^{-7} \), and the scaling factor of the condensation source term is \( C_{\text{cond}} = 1/750 \).

3.8. Backward direction
An experimental campaign was carried out in Karlstein to validate the innovative fluidic diode. During this campaign, a single double-Venturi channel of the POR is tested under realistic plant conditions. Here, experimental data of two tests are exploited to validate the code and analyze the flow behavior of the POR in backward direction.

3.9. Numerical details
Two LES are performed with the HPF model. Adiabatic non-slip walls are assumed. The total pressure and the temperature at the inlet and the static pressure at the outlet of the double Venturi are given by the experiments and imposed in the simulation (Figure 15).

![Figure 15. Computational domain and boundary conditions for backward-direction simulations. Origin \( x_0 = 0 \) is the position of geometry inlet.](image)

One POR channel is represented by a three dimensional block-structured computational mesh containing about 550 000 cells. Far upstream and downstream of the contractions, the cells are stretched out to increase the damping of the disturbances. In this manner, increased code stability and faster statistical convergence are obtained. The mesh was sufficiently fine to resolve the small fluctuating scales in the bulk. The maximal ratio between the sub-grid scale viscosity and the laminar viscosity was about 7 in the region of interest. In spite of the mesh refinement at the walls, the \( y^+ \) values are larger than those usually recommended in resolved LES calculations [37]. For example, \( y^+ \) values up to 115 may be encountered locally. The mean \( y^+ \) value is about 40.

In a first validation case, the coolant is characterized by a subcooling of 11 K. The total pressure at the inlet is 7.03 MPa and the static pressure at the outlet is 2.22 MPa. The resulting mass flow rate is \( \dot{M} = 4.2 \text{ kg} / \text{s} \). Measured pressure along the POR channel during the experiment is represented by squares, and the pressure distribution obtained in the bulk with the proposed cavitation model is the blue solid line on Figure 16, respectively.

![Figure 16. Validation of numerical against experimental pressure distribution along a single POR channel in case of moderately subcooled (11 K) coolant in backward direction.](image)
Good agreement between the numerical analysis and experimental results is found. The deviation between measured and calculated mass flow rates is less than 2% as the cavitation model predicts a mass flow rate of $\dot{M} = 4.28 \, \text{kg/s}$. In the second validation test, the coolant is characterized by a subcooling of 4K. Inlet pressure is 6.57 MPa and outlet pressure is 1.69 MPa. The resulting mass flow rate is $\dot{M} = 3.10 \, \text{kg/s}$. Also in that case, the agreement between the numerical and experimental results is good. The pressure distribution along the POR design obtained in the calculation (solid line on Figure 17) is consistent with experimental results (squares on Figure 17). The numerical code slightly overestimates the mass flow rate ($\dot{M} = 3.38 \, \text{kg/s}$). Therefore, the static pressure decrease at the first Venturi nozzle is slightly overpredicted. This static pressure decrease primarily is induced by an increase of the dynamic pressure.

Some results on the flow behavior of the fluidic diode in the backward direction are given in the following. These results are obtained in case of moderately subcooled coolant (11 K subcooling).

3.10. General flow behavior

Figure 18 represents the mean pressure distribution (blue solid line), saturation pressure (black solid line) and vapor mass fraction (green solid line) in the bulk of the POR channel.

In the first section (up to $x_1$), the wall shear friction slightly decreases the static pressure. A larger static pressure drop occurs in the contraction ($x_2$) mainly due to a dynamic pressure rise. Downstream of the expansion (between $x_2$ and $x_3$), the pressure recovery occurs only partially. The flow detaches from the wall and large amounts of mechanical energy dissipate in the strongly fluctuating jet (Figure 19).

In average, the reattachment point is situated at a small distance from the enlargement (Figure 20). The pressure recovery is maximal near this reattachment point.
In the second contraction of the double-nozzle design a self-sustained mechanism is observed. The coolant drastically vaporizes (Figure 18 and Figure 26) yielding a significant velocity increase (Figure 20 and Figure 21) which in turn amplifies the static pressure fall and, thus, the vaporization. The vaporization of the coolant also has an effect on the speed of sound in the mixture (black solid line, Figure 21). This speed of sound drastically reduces with a vapor content rise (green solid line). In the second nozzle, the velocity of the coolant (blue solid line) reaches the speed of sound. The flow chokes. Note that pressure below saturation pressure is observed indicating thermodynamic non-equilibrium.

3.10.1. Choking  Figure 22 shows the instantaneous Mach number in the second nozzle. The Mach number reaches one at the smallest cross section and increases downstream up to about 1.4 before it decreases to subsonic conditions at the outlet. In contrast to other flow properties, the location of the choking cross section does not fluctuate much.

A close up on the second smallest cross section is given in Figure 23. The choking cross section does not coincide with the smallest cross section but rather with an iso-surface of the void fraction.
3.11 Void generation and transport
The coolant is characterized by a moderate subcooling (11 K). In the first nozzle, vaporization only occurs in the core of the largest turbulent eddies (Figure 24). In Figure 24, the Q-criterion of Hunt [62] is used to represent these turbulent structures.

In the core of these turbulent eddies, the pressure may be below the saturation pressure due to their rotation. As an example, Figure 25 shows the largest eddies on (a), the pressure field on (b) and the void fraction on (c). The largest eddies are selected here by increasing the values of the Q criterion. The pressure field is blue when it is below the saturation pressure. Minimum pressure regions and largest eddies exactly coincide. They are denoted by white circles in Figure 25 (a) and (b). In contrast, the regions with higher vapor content coincide only with some of these eddies [denoted by white circles on Figure 25, (c)]. There, the pressure is minimal. In the other eddies [denoted by gray circles on Figure 25, (c)], the pressure is not sufficient to induce the vaporization of the coolant. Once the vapor is generated inside the intense eddies, the vapor content seems to follow another dynamic than that of the turbulent eddies.
The turbulent eddies are generated in the shear layers at the wall and at the jet shear layers. Due to the strong fluctuations [Figure 24 and Figure 26, (a) to (c)], the mixing of the vapor is large. The vapor is transported transversally [Figure 26, (a) to (c)], and reaches the zone with high streamwise velocities. In average [Figure 26, (c)], the vapor diffuses in the transversal direction and is advected down to the second nozzle.

The turbulent eddies dissipate at a short distance from the enlargement (Figure 24) and the mean pressure increases with the reattachment of the flow. The pressure is everywhere above the saturation pressure and the vapor starts to condensate.

![Figure 26. Instantaneous vapor mass fraction at three different times (a), (b), (c) and mean vapor mass fraction (d) in a double-nozzle channel in case of moderately subcooled (11 K) coolant in the backward direction.](image)

3.12. Bubble-size fdf

The proposed stochastic-field cavitation model enables to capture bubble-size fdf in all computational cells. For illustration, bubble-size fdfs in case of moderately subcooled (11 K) coolant at the locations 1, 2 and 3 marked by a cross on Figure 27 are displayed on Figure 28.

![Figure 27. Illustration of a POR channel with backward flow direction. Bubble size fdf are numerically sampled at the locations 1, 2 and 3.](image)

![Figure 28. Calculated bubble-size fdf in a POR channel in case of moderately subcooled (11 K) coolant in the backward direction at locations 1 (a), 2 (b) and 3 (c) marked in Figure 27.](image)

At location 1, the bubbles have collapsed due to the pressure recovery and remain very small. In contrast, at location 2 and 3, the coolant vaporizes. The bubbles grow and are characterized by large radii. Representation of the same data on another scale is given on Figure 29. Downstream of the second Venturi nozzle, the bubble-size fdf becomes nearly Gaussian.

These fdf are obtained from the radius-size $R^k$ pseudo-time history of the 8 stochastic fields calculated at these locations. The pseudo-time history of the 8 radius-size $R^k$ in case of moderately subcooled (11 K) coolant at location 1, 2 and 3 are shown on Figure 30. In backward direction,
turbulence intensity is small and local tension \( p_{sat} - p \) is very large at the considered locations. All nuclei are activated and evolve very similarly. The fluctuation of the bubble-size on the sub-grid scale is small.

![Figure 29](image)

**Figure 29.** Calculated bubble size pdf in a POR channel in case of moderately subcooled (11 K) coolant in the backward direction at locations 1 (a), 2 (b) and 3 (c) marked in Figure 27.

![Figure 30](image)

**Figure 30.** Radius-size pseudo-time history of the 8 stochastic fields calculated at locations 1 (a), 2 (b) and 3 (c) marked in Figure 27 in case of moderately subcooled (11 K) coolant in the backward direction.

### 3.13. Forward direction

In forward direction, experiments indicate that cavitation only occurs beyond POR operating conditions. Nevertheless, one LES is performed to show the flow behavior of the POR under cavitating conditions and illustrate the generality of the stochastic-field method.

#### 3.13.1 Numerical details

One POR channel is represented by a three dimensional block-structured computational mesh consisting of about 2.8 million cells. In spite of the mesh refinement at the walls, the \( y^+ \) values are larger than those usually recommended in resolved LES calculations [56]. For example, \( y^+ \) values up to 130 may be encountered locally. The mean \( y^+ \) value is about 60. In contrast, the mesh is sufficiently fine to resolve the small fluctuating scales in the bulk. The maximal ratio between the sub-grid scale viscosity and the laminar viscosity is about 7 in the computational domain. Far upstream and downstream of the smallest cross sections; the cells are stretched out to increase the damping of the disturbances. In this manner, increased code stability and faster statistical convergence are obtained. A mass flow of 1.8 kg/s is imposed at the inlet of the computational domain. There, the flow direction is assumed to be normal to the boundary and the temperature of the coolant is imposed. The pressure known from the experiments is prescribed at the outlet. These boundary conditions are shown on Figure 31.

![Figure 31](image)

**Figure 31.** Representation of the computational domain and boundary conditions in
the forward direction. \( x_0 = 0 \) is the position of geometry inlet.

3.13.2 General flow behavior  
Figure 32 represents the calculated pressure (blue solid line), the calculated saturation pressure (black solid line) and the vapor mass fraction (green solid line) in the bulk along the POR channel in the forward direction in case of coolant near saturation.

The static pressure strongly decreases in the smallest cross sections due to the acceleration of the fluid (see Figure 33).

When the coolant is near saturation at the inlet of the POR, the static pressure in the bulk reaches the saturation pressure at both smaller cross sections and both POR nozzles are filled with vapor (Figure 34).

The speed of sound decreases in both nozzles [Figure 33 (b)] but, under these flow conditions, the velocity is significantly below the speed of sound in the mixture. The flow does not choke. Acceleration and static pressure drop accentuate in the second nozzle due to intense coolant vaporization. The velocity in the second nozzle reaches 27 m/s while it is only 20 m/s in the first nozzle (see Figure 33). Following the flow deceleration, the static pressure recovers in the diffuser and the pipes. The coolant condenses. This condensation is enhanced by the thermal effects: The temperature, and thus the saturation pressure, decreases (Figure 32) as a result of coolant vaporization.
3.13.3. Flow resistance  The two-phase mixture does not choke. However, the total pressure loss significantly increases in comparison to single-phase conditions. For same mass flow and pressure, the total pressure loss amounts 0.12 MPa while it is only 0.03 MPa (available from experiments) when no vaporization occurs. The total pressure distribution in the double-nozzle design is represented on Figure 35. Energy is principally dissipated in the vicinity of the smallest cross sections and in the diffusers while dissipation is negligible in the straight pipes and contractions.

![Figure 35. Total pressure in one POR channel (a) in case of saturated coolant in the forward direction; and zoom on the first nozzle (b) and second nozzle (c).](image)

Increased energy dissipation results from larger velocities and increased fluctuations. Larger velocities occur in regions with higher vapor content where the density is lower. As an example, the maximal velocity in the second nozzle is 27 m/s instead of 20 m/s in the first nozzle. This mechanism has a direct impact on the flow resistance: the total pressure loss is higher in the second nozzle (0.07 MPa) than in the first nozzle (0.05 MPa).

Larger velocities are also observed in the vicinity of the smallest cross sections. There, the growth and collapse of the vaporous region has a destabilizing effect on the incoming flow. The flow detaches, the effective cross section reduces, and a jet with higher velocities develops. The larger fluctuations (Figure 36) are intimately related to this flow detachment since they occur primarily in the bounding region of the jet.

![Figure 36. Instantaneous velocity field in the second nozzle of one POR channel in case of saturated coolant in the forward direction.](image)

3.13.4. Bubble-size fdf  The radius-size pseudo-time history of the 8 stochastic fields in case of saturated coolant at three positions in the double Venturi is represented in Figure 37.

![Figure 37. Radius-size $R^k$ pseudo-time history of the 8 stochastic fields calculated at location](image)
1 (a), 2 (b) and 3 (c) marked in Figure 38 in case of saturated coolant in forward direction.

These three locations are marked on Figure 38. Position 1 is situated in the cavitation core of the first Venturi; position 2 is at the inlet of the second Venturi and position 3 is located in the cavitation core of the second Venturi.

Figure 38. Positions in the double-Venturi design where the fdf are sampled numerically in the forward direction. Dashed lines delimitate the section of the double-Venturi channel considered on Figure 39.

In the first Venturi nozzle, the pressure decrease is large. All nuclei similarly grow so that fluctuation of the bubble size on the sub-grid scale is small. The stochastic fields behave very similarly at location 1 [Figure 37 (a)]. In location 2 and 3 we find a fdf behavior which is qualitatively different from any behavior reported above. In the straight pipe connecting the two Venturi, the instantaneous static pressure decreases below the saturation pressure exclusively in the core of turbulent eddies. There, the turbulent fluctuations are low and, thus, the local tension $P_{sat} - p$ is small. As an example, Figure 39 (a) shows the most intense eddies present in the straight pipe visualized with the Q criterion [62] at arbitrary pseudo time. The corresponding pressure field is represented on the same Figure 39 (b). Regions in the pipe where the pressure falls below saturation pressure are colored in blue.

Figure 39. Contour plots of Q criterion (a) and pressure (b) in the straight pipe connecting the two Venturis in case of saturated coolant in forward direction. Regions in the pipe where the pressure is below saturation pressure are colored in blue.

Since the local tension is small, only the largest nuclei are activated, subsequently grow and finally collapse. Other nuclei remain inactive. Thus, bubble-size fluctuations on the sub-grid scale are large. Individual stochastic fields evolve very differently at location 2 [Figure 37, (b)]. These fluctuations are yet more accentuated in the second Venturi nozzle [Figure 37 (c)]. The corresponding bubble size fdf is shown in Figure 40.

Figure 40. Calculated bubble size fdf in a double-Venturi design at location 1 (a), 2 (b) and 3 (c) marked in Figure 38 in case of saturated coolant in forward direction.

Corresponding instantaneous and averaged contour plots of some of these stochastic fields are shown in Figure 41 and indicate that the cavity may qualitatively change its appearance.
4. Conclusion

In statistical mechanics the fdf ansatz was introduced in the 19th century by Boltzmann leading to mature fdf transport models and corresponding numerical techniques for gas dynamics and gaseous combustion. Early cavitation models were proposed by Lord Rayleigh. These cavitation models describe the physics based on an Euler-Lagrangian point of view. The present theory uses for the first time the stochastic-field method for multiphase-flow modeling and thus combines the fdf ansatz with a pure Eulerian description.

In conclusion, the stochastic-field method adapted to cavitation shows very encouraging results. The agreement between numerical and experimental results is good under very differing flow conditions. Results demonstrate that at large local tension, $p_{\text{sat}} - p$, all nuclei are activated. These large local tensions are typically encountered in the backward direction of the passive outflow reducer but also in the sheet cavitation of the validation case. At small local tension, $p_{\text{sat}} - p$, polydisperse cavities are found since large nuclei are exclusively activated. This phenomenon occurs for instance in the pipe between the two Venturi nozzles of the passive outflow reducer operated in forward direction. The wide range of fdf shapes present at different locations in cavitating flows can be described without excessive computational demand thereby enabling to account for stochastic processes and highly non-linear, radius dependent phenomena. Thus, the method is also very attractive for other multi-phase flows which in general possess similar characteristics (stochastic processes, highly non-linear interfacial exchanges, crucial dependence on interfacial area). The method is compatible with finite-volume codes where all existing physical models available for Lagrangian techniques, presumed fdf or binning methods can be easily extended to the stochastic field formulation.

In spite of the encouraging results, implementation of a high-order numerical scheme consistent with Ito calculus as well as investigation of interaction fdf-les will be the purposes of future work.

Acknowledgement

This work is supported by the AREVA Nuclear Professional School. We thank the unknown reviewers for constructive comments with regard to the numerical scheme and LES-PDF interactions which will be subject of future investigations.

References

[1] Rayleigh L 1917 The London, Edinburgh, and Dublin Philosophical Magazine and Journal of Science 34(200) 94-8
[2] Bailey M, Khokhlova V, Sapožnikov O, Kargl S, and Crum L 2003 Acoustical Physics 49(4)
[3] Chaussy C H, Brendel W, and Schmiedt E 1980 The Lancet 316(8207) 1265-8
[4] Amy S 2003 Agricultural Research 51(2) 10
[5] Amy S 2003 Agricultural Research(Ultrasound) 51(2) 10
[6] Chakinala A G, Gogate P R, Burgess A E, and Bremner D H 2008 Ultrasonics Sonochemistry 15(1) 49-54
[7] Bilus I and Predin A 2009 International Journal of Numerical Methods for Heat & Fluid Flow 19(7) 818-34
[8] Kim J H, Ishizaki M and Enomoto N, et al. 2004 Nihon Kikai Gakkai Ryutai Kogaku Bumon Koenkai Koen Ronbunshu 82 131
[9] Arndt R E A 1981 Annual Review of Fluid Mechanics 13(1) 273-326
[10] Satoshi W, Naoki E, Koichi I, Akinori F, and Kim J H 2004 Turbomachinery 32(2) 94-100
[11] Bernad S, Susan-Resiga R, Muntean S, et al. 2007 Cavitation phenomena in hydraulic valves. Numerical modeling Proceeding of the Romanian Academy, Series A 8(2)
[12] Tullis J P 1971 Journal of the Hydraulics Division 97(2) 1931-45
[13] Franc J P and Michel J M 2004 Fundamentals Of Cavitation (Berlin: Springer)
[14] Brennen C E 1995 Cavitation and Bubble Dynamics (London: Oxford University Press)
[15] Kamp A M, Chesters A K, Colin C, and Fabre 2001 International Journal of Multiphase Flow 27(8) 1363-96
[16] Ran B and Katz J 1994 Journal of Fluid Mechanics 262 223-63
[17] Martinez-Bazán C, Montanes J L, and Lasheiras J C 1999 Journal of Fluid Mechanics 401(1) 157-82
[18] Thomas R M 1981 International Journal of Multiphase Flow 7(6) 709-17
[19] Delannoy Y and Kueny J L 1990 Cavitation and multiphase flow forum 98 153–8
[20] Goncalves E and Patella R F 2009 Computers & Fluids 38(9) 1682-96
[21] Hickel S, Mihatsch M, and Schmidt S 2011 Implicit Large Eddy Simulation of Cavitation in Micro Channel Flows (WIMRC: University of Warwick)
[22] Kunz R F, Boger D A, Stonebring D R, et al. 2000 Computers & Fluids 29(8) 849-75
[23] Yuan W, Sauer J, and Schnerr G H 2001 Mecanique & Industries 2(5) 383-94.,
[24] Singhal A K, Athavale M M, Li H, and Jiang Y 2002 Journal of Fluids Engineering 124(3) 617
[25] Meyer R S, Billet M L, and Holl J W 1992 Journal Fluids Engineering 114(4) 672-9
[26] Kubota A, Kato H, and Yamaguchi H 1992 Journal of Fluid Mechanics 240(1) 59-96
[27] Durst F, Milioievic D, and Schöning B 1984 Applied Mathematical Modelling 82(2) 101-15
[28] Bannari R 2011 Cavitation Modelling Based on Eulerian-Eulerian Multiphase Flow (Canada: Université de Sherbrooke)
[29] McGraw R 1997 Aerosol Science and Technology 27(2) 255-65
[30] Marchisio D L and Fox R O 2005 Journal of Aerosol Science 36(1) 43-73.
[31] Dopazo C and O'Brien E E 1974 Acta Astronautica 1(9-10) 1239-66
[32] Haworth D C 2010 Progress in Energy and Combustion Science 36(2)168-259
[33] Tavare N S 1995 AIChE Journal 41(12) 2537-48
[34] Pope S B 1985 Progress in Energy and Combustion Science 11(2) 119-92
[35] Valiño L 1998 Flow Turbulence and Combustion 60(2) 157-72
[36] Garmory A, Richardson E S, and Mastorakos E 2006 Atmospheric Environment 40(6) 1078-91
[37] Jones W P and Tyliszczak A 2010 Flow Turbulence and Combustion 85(3–4) 711-34
[38] Bini M, Jones W P, and Lettieri C 2009 Large Eddy Simulation of Spray Atomization with Stochastic Modelling of Breakup Proceedings of the European Combustion Meeting (Vienna, Austria, 14-17 Apr 2009)
[39] Pope S B 2010 Journal of Fluid Mechanics 652 139-69
[40] Harvey A H and Friend D G 2004 Physical properties of water, in Aqueous Systems at Elevated Temperatures and Pressures (London: Academic Press) Chapter 1 pp 1-27
[41] Saito Y, Takami R, Nakamori I, and Ikohagi T 2007 Computational Mechanics 40(1) 85-96
[42] Shamsborhan H, Coutier-Delgosha O, Caignaert G and Abdel Nour F 2010 Experiments in Fluids 49(6) 1359-73
[43] Garland W J and Hand B J 1989 Nuclear Engineering and Design 113(1) 21-34
[44] Balasubramaniam R, Ramé E and Kizito J, et al. 2006 Two phase flow modeling: Summary of flow regimes and pressure drop correlations in reduced and partial gravity (USA: NASA Glenn Research Center)
[45] Grinstein F F and Fureby C 2005 On Monotonically Integrated Large Eddy Simulation of Turbulent Flows Based on FCT Algorithms//Flux-Corrected Transport ((Berlin: Springer Berlin Heidelberg) pp 79-104
[46] Friedrich R, Geurts B, and Métais O 2004 Direct and Large-Eddy Simulation V (Berlin: Springer)
[47] Jameson A, Schmidt W, and Turkel E 1981 Numerical solution of the Euler equations by finite volume methods using Runge Kutta time stepping schemes 14th AIAA Fluid and Plasma Dynamics Conference (Palo Alto, CA, 23-25 June 1981)
[48] Jones W P and Prasad V N 2010 Combustion and Flame 157(9) 1621-36
[49] Jones W P and Navarro-Martinez S 2009 Large eddy simulation and the filtered probability density function method AIP Conference Proceedings (Sochi, Russia, 28 Sep-2 Oct 2009)
[50] Gardiner C W 1985 Handbook of Stochastic Methods for Physics, Chemistry, and the Natural Sciences (Berlin: Springer-Verlag)
[51] Soulard O and Sabel’nikov V A 2006 Combustion, Explosion and Shock Waves 42(6) 753-762.
[52] Minier J P and Peirano E 2001 Physics Reports 352(1–3) 1-214
[53] Plesset M S and Prosperetti A 1977 Annual Review of Fluid Mechanics 9(1) 145-85
[54] Magagnato F, Fritz B and Gabi M 2005 Journal of Thermal Sciences 14(2) 156-61
[55] Barre S, Rolland J, Boitel G, Goncalves E and Patella R F 2009 European Journal of Mechanics - B/Fluids 28(3) 444-64
[56] Fröhlich J 2006 Large Eddy Simulation turbulenter Strömungen (Berlin: Springer DE)
[57] Dzubur A, Bajic B and Jovanovic I 1993 Intl. Shipbuilding Progress 40(422) 165-75
[58] O'Hern T J, d'Agostino L and Acosta A J 1988 J. Fluids Eng. 110 200-7
[59] Yilmaz E, Hammitt F G and Keller A 1976 The Journal of the Acoustical Society of America 59(2) 329-38
[60] d'Agostino L and Acosta A J 1991 J. fluids Eng. 113(2) 270-7
[61] Stutz B and Reboud J L 2000 Experiments in Fluids 29(6) 545-52
[62] Hunt J C R 1987 Canadian Society for Mechanical Engineering, Transactions 11(1) 21–35