

Direct Numerical Simulation of Flow Induced Cavitation in Orifices

Giacomo Falcucci and Elio Jannelli
Univ. of Naples Parthenope

Stefano Ubertini
Univ. of Tuscia

Gino Bella
Univ. of Rome Tor Vergata

ABSTRACT

In this paper, a multiphase Lattice Boltzmann approach is adopted to directly simulate flow conditions that lead to the inception of cavitation in an orifice.

Different values of fluid surface tension are considered, which play a dramatic role in the evolution of vapour cavity, as well as different inlet velocities at the computational domain boundary.

The results of the flow simulations in terms of density and velocity magnitude fields are examined, with special focus on the components of the stress tensor inside the cavitating region: a comparison with cavitation inception criteria known from literature is proposed, highlighting the good agreement between our direct numerical simulations and theoretical predictions.

CITATION: Falcucci, G., Jannelli, E., Ubertini, S., and Bella, G., "Direct Numerical Simulation of Flow Induced Cavitation in Orifices," *SAE Int. J. Fuels Lubr.* 6(3):2013, doi:10.4271/2013-24-0005.

INTRODUCTION

With *cavitation* we refer to the phase-change that takes place in a liquid as a consequence of the drop in static pressure caused by mechanical changes in the flow configuration, i.e. fluid flowing through orifices, or experiencing sharp changes in flow direction. This sudden and local phase-change may be an aggressive phenomenon, causing brittling and erosion to the hardware: this is the reason why great attention is given to flow conditions leading to cavitation in many engineering applications, such as turbines and propellers, [1, 2, 3]. This aggressiveness derives from the collapse of cavitation bubbles, giving rise to violent implosions with emission of shock and light waves, [4, 5, 6, 7], focusing substantial amounts of fluid energy within narrow regions of space. For what concerns the field of internal combustion engines, the occurrence of cavitation in orifices and nozzles significantly influences the atomisation of the fuel liquid jets, making it a fundamental topic in spray

dynamics and in the development of combustion, [8, 9, 10, 11].

Unlike *boiling*, which involves a given amount of a liquid as a whole, cavitation is a strongly off-equilibrium non-thermal phase-transition, which makes it the subject of interest for fundamental studies of non-equilibrium thermodynamics, [1, 12, 13]. This peculiar characteristic of being a non-thermal process, makes the response of cavitating flows to external perturbations, for instance a sudden drop in the local pressure, significantly different from the case of standard multiphase flows. Cavitating flows, in fact, are subject to rapid changes (with respect to typical hydrodynamic scales) in the local pressure distribution, which may lead to potential instabilities, such as growth in surface oscillation, eventually leading to interface rupture and/or bubble implosion (collapse).

Capturing *direct* experimental evidence of the morphology of real-scale cavitating flows is a major challenge, due to the fast transient dynamics at very short

spatial scales, besides the practical difficulty of using non-intrusive probes. This is the reason why the presence of a cavitation region is typically deduced “ex-post”, measuring and detecting its effects, such as spray morphology, break-up regime, vibrations and acoustic emissions, [14, 15, 16]. Also, the visualization of the light reflected by the interface using transparent liquids is employed, [17], but only in much larger nozzles than the real-scale injectors, with very significant scale effects, [18].

Cavitating flows represent a major challenge for numerical simulations, as well. The difficulties for solvers based on Navier-Stokes equations are related to the dynamical tracking of the evolution of the interface between the two fluids.

Different cavitation models are available in literature, such as the tracking model of cavitation bubble by Gavaises et al. [3], but cavitation is also studied by means of advanced single-phase approaches, [10].

Generally, such models add a surface-tension forcing term acting upon the unknown interface to the Navier-Stokes (NS) equations. The interface itself is located by means of semi-empirical criteria based on the space-time distribution of the pressure or other thermodynamic parameters, a methodology that one may call “cavitation modeling”, [19, 20, 17, 21, 22].

Adopting the Lattice Boltzmann (LB) approach, no criteria are required, since the liquid-gas interface emerges spontaneously from of the underlying intermolecular interactions, [23, 24, 25], driven by the corresponding non-ideal equation of state and surface tension. The LB method has been used with success for a wide range of multiphase and multicomponent flow applications, but its viability for cavitation studies has been addressed only very recently, [26, 27, 28, 29, 30].

In this work, results of direct numerical simulations (DNS) of cavitation flows, obtained by means of the lattice Boltzmann Method (LBM) are presented. Different non-dimensional Cavitation Numbers (CN) are considered, proving that (CN) is not an unequivocal predictor of cavitation inception and development; the effects of surface tension on the dynamic morphology of the cavitating bubbles are explored, as well. In our simulations, we find that Joseph's minimum tension criterion, [19, 20] correctly captures the onset of incipient cavitation, provided the cavitation threshold is corrected by means of surface-tension terms, as proposed by Brennen, [1], and Plesset, [31, 32].

LATTICE BOLTZMANN FOR NON-IDEAL FLUIDS: THE TWO-BELT APPROACH

The Lattice Boltzmann Method (LBM) derives from Boltzmann's kinetic equation and is a numerical technique for fluid dynamics investigation, characterized by a statistical discrete approach rather than by the continuum assumption underlying all numerical solvers based on Navier-Stokes equations, [24]. In recent years, LBM has been successfully

employed in the field of non-ideal fluids, [33, 34, 35, 36, 37, 38, 39, 28, 40], proving to be a robust, versatile and efficient alternative to the NS approach.

The standard LB equation with pseudo-potential interaction reads as follows (with unit time step for simplicity), [24, 25]:

$$f_i(\mathbf{x} + \mathbf{c}_i, t + 1) - f_i(\mathbf{x}, t) = -\omega(f_i - f_i^{eq}) + F_i(\mathbf{x}), \quad (1)$$

where f_i is the probability density function of finding a particle at site $\mathbf{x}$ at time t , moving along the i -th lattice direction defined by the discrete speeds $\mathbf{c}_i$ with $i = 0, \dots, b$. The left-hand side of Eq. (1) represents molecular free-streaming, whereas the right-hand side is characterized by relaxation in time towards local Maxwellian equilibrium, due to collisions.

Once the populations f_i are determined by solving Eq. (1), macroscopic density ρ and velocity $\mathbf{u}$ are given by $\rho(\mathbf{x}, t) = \sum_{i=0}^b f_i$ and $\rho(\mathbf{x}, t)\mathbf{u}(\mathbf{x}, t) = \sum_{i=0}^b \mathbf{c}_i f_i$, [25, 24].

Equation 1 highlights the main characteristics of LBM, as compared to NS-based solvers: the nonlinear convective term $\mathbf{u}\mathbf{u}$ is not present and there is no need to solve the Poisson equation to have pressure, since in LBM the communication is always local and pressure is retrieved by means of the Equation of State (EOS). Besides this, the treatment of boundary conditions is very easy and accurate in LBM and Eq. (1) is suitable for being easily and efficiently parallelized, resulting in overall high computational efficiency, [24].

In Eq. (1), F_i represents the volumetric force acting between the different phases, which is directly related to the adopted EOS. In this work, F_i is represented by the sum of the two forces proposed in [35], acting on the two belts of nearest lattice sites for each node, see Fig. 1, $F(\mathbf{x}, t) = F_1(\mathbf{x}, t) + F_2(\mathbf{x}, t)$, defined as follows:

$$F_1(\mathbf{x}, t) = G_1 \psi(\mathbf{x}; t) \sum_{i=0}^{b_1} w_i \psi(\mathbf{x}_{1i}, t) \mathbf{c}_{1i} \Delta t \quad (2)$$

$$F_2(\mathbf{x}, t) = G_2 \psi(\mathbf{x}; t) \sum_{i=0}^{b_1} p_{1i} \psi(\mathbf{x}_{1i}, t) \mathbf{c}_{2i} \Delta t + G_2 \psi(\mathbf{x}; t) \sum_{i=0}^{b_2} p_{2i} \psi(\mathbf{x}_{2i}, t) \mathbf{c}_{2i} \Delta t \quad (3)$$

with $\psi(\rho)$ the local parameter called *pseudopotential*, governing the interaction between the phases and given by $\psi = \sqrt{\rho_0}(1 - e^{-\rho/\rho_0})$. The coupling parameters G_1 and G_2 have to be chosen in order to produce the spinodal decomposition lying at the basis of phase separation, [24, 39]; according to our previous works, we chose G_1 and G_2 in order to realize a density jump between the phases of about

1 : 30. For detail discussion on the choice of G_1 and G_2 parameters, the Reader is addressed to [35, 36].

In Eqs. 2 and 3, the indices $k=1, 2$ refer to the first and second Brillouin zones in the lattice (belts, see Fig. 1), and c_{ki} , p_{ki} are the corresponding discrete speeds and associated weights; $x_{ki} \equiv x + c_{ki}\Delta t$ are the displacements along the i -th direction in the k -th belt.

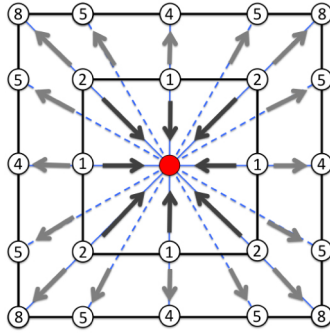


Figure 1. Sketch of the forces acting on the lattice Brillouin zones in the 2-belt approach: the attractive force $F_1(x, t)$ acts on the first belt, composed of $b_1 = 8$ neighbours; the repulsive force $F_2(x, t)$ acts on both belts, with $b_2 = 24$ neighbours.

The first belt is discretized with 9 speeds ($b_1 = 8$), while the second with 16 ($b_2 = 16$) and the weights are chosen in order to fulfil the following relations, [41, 36, 38]:

$$\sum_{i=0}^{b_1} w_i = \sum_{i=0}^{b_1} p_{i1} + \sum_{i=0}^{b_2} p_{i2} = 1,$$

$$\sum_{i=0}^{b_1} w_i c_i^2 = \sum_{i=0}^{b_1} p_{i1} c_{i1}^2 + \sum_{i=0}^{b_2} p_{i2} c_{i2}^2 = c_s^2, c_s^2 = 1/3$$

being the lattice sound speed.

Taylor expansion of eqs. (2) and (3) to second-order delivers the following non-ideal equation of state (EOS)

$$p \equiv P/c_s^2 = \rho + \frac{(G_1 + G_2)}{2} c_s^2 \psi^2(x, t) \quad (4)$$

Further expansion to fourth-order provides the following expressions for the surface tension

$$\gamma = -\frac{(G_1 + \xi G_2)}{2} c_s^4 \int_{-\infty}^{\infty} |\partial_y \psi|^2 dy \quad (5)$$

where y runs across the interface. Among the features allowed by the choice of the double potential, we have the possibility to change the density ratio between the phases and the surface tension independently. In particular, since $\xi = 12/7 > 1$ the effect of the repulsive belt is to lower the surface tension at a given value of $G_1 + G_2$.

The EOS in Eq. 4 can be shown to support a liquid-vapour phase-transition at the critical values given by $\rho_{cr} = \ln 2$ and $P_{cr} = \frac{1}{3}(\ln 2 - \frac{1}{2})$ for $G_{tot} = G_1 + G_2 < G_{tot,cr} = -4.0$,

all in lattice units. In the above, the subscript cr indicates the critical point in the phase diagram.

To be noted that the non-ideal part of the EOS depends on one single parameter because the model is purely attractive ($G < 0$).

Table 1. Main thermodynamic characteristics of the simulations. G_1 and G_2 represent the strength of non-ideal interactions in the first and second shell of lattice neighbours, respectively and P_L^* is the corresponding excess pressure in the liquid phase at coexistence.

ρ_L	ρ_V	ρ_L/ρ_V	
1.9	0.068	27.94	
G_1	G_2	P_L^*	σ
-5.0	0.0	-0.602	0.0565
-10	5.0	-0.602	0.0482
-15	10.0	-0.602	0.0412

However, due to the density dependence of $\psi(\rho)$, this EOS generates the coexistence between the liquid and vapour phases. The critical density ρ_{cr} and the critical coupling strength $G_{tot,cr}$ are computed based on the usual requirement $\frac{\partial p}{\partial \rho} = 0$ and $\frac{\partial^2 p}{\partial^2 \rho} = 0$. The former delivers $\rho_{cr} = \ln 2$ and inserting this in the latter, we obtain $G_{tot,cr} = -4.0$, from which the critical pressure follows directly from the Equation of State. From this discussion, we see that $G_{tot,cr}$ plays the role of an effective critical temperature, in the sense that for $G < G_{tot,cr}$ the EOS provides a Van-der-Waals-like loop.

In order to convert from physical to lattice undimensional unites, the procedure is the following.

- Length: $L_{phys} = L_{lb} \Delta x$, where L_{phys} is the physical length, [m], Δx is the Lattice Spacing in physical units and L_{lb} is the length in non-dimensional (lattice) units.
- Time units are fixed based on $t_{phys} = t_{lb} \Delta t$, where Δt is the time step in physical units, [s], and it is computed based on the viscosity relation, $\nu_{phys} = \nu_{lb} \Delta x^2 / \Delta t$.
- Finally, the mass units are such that the Lattice Boltzmann critical density, $\rho_{cr} = \ln 2$ corresponds to the critical density of the fluid under investigation.

SIMULATIONS, RESULTS & DISCUSSION

We present the results of simulations of a multiphase, single component fluid flowing through an orifice of height $h = 400$ lattice points (see Fig. 2) for a 2D 2000×2000 domain. At the inlet, a constant density and velocity profile are imposed, while a free-flow condition and constant density are imposed at the outlet. We considered the total pressure, given by the sum of the inlet pressure and kinetic energy at the inlet section, as the reference condition. Different values of the inlet fluid speed u_{in} are accounted for, in order to scan