Numerical study of dissolved gas release induced by cavitation in a high speed channel flow

U Iben¹, A Makhnov²,³ and A Schmidt²,³

¹ Robert Bosch GmbH, Corporate Research, St. Petersburg, Marshala Govorova 49A, 198095 Russia
² Peter the Great Saint Petersburg Polytechnic University, St. Petersburg, Polytechnicheskaya 29, 195251 Russia
³ Ioffe Institute, St. Petersburg, Polytechnicheskaya 26, 194021 Russia

e-mail: a.makhnov@mail.ru

Abstract. Cavitation often leads to performance degradation of hydraulic devices and, in some cases, to damage of the elements of their constructions (cavitation erosion). Liquids, that are used in technical devices, often contain dissolved gas. Such liquids are of particular interest because cavitation causes release of the dissolved gas into the cavities which can significantly change the flow structure. Mathematical models based on detailed description of all the underlying mechanisms still need development. The approach that was proposed and analyzed in the present study combines a simplified equilibrium model of liquid-vapor phase transition with a non-equilibrium diffusion model describing gas release process. The distributions of the hydrodynamic parameters and the gaseous phase fraction obtained show significant influence of the dissolved gas on evolution of cavitation structures. This is in qualitative agreement with experimental observations regarding the main features of cavitating flows and the behavior of the degassed air. It demonstrates robustness of the proposed simulation algorithm which means that its further development can be a promising way to create a universal model for reliable quantitative prediction of dissolved gas release.

1. Introduction

For the time being, several approaches are proposed in literature for mathematical description of liquid flows in which the release of dissolved gas occurs. However, these approaches are simplified and were developed by the authors of the corresponding studies to predict liquid degassing only for specific conditions observed in concrete hydraulic devices. In the study [1], the existing models used in various industrial applications are summarized and divided into several groups depending on the assumptions and limitations.

The simplest models presented in the literature are based on the assumption of a fixed content of released gas in a liquid. Changes of the released gas concentration in the liquid due to gas release and dissolution are not taken into account. Instead of this, it is assumed that the liquid contains a small (but constant) amount of the released gas in form of small bubbles which finite sizes can be neglected. The results obtained in [2] show that the use of such an approach can be enough for qualitative analysis of...
the effect of the released gas on the flow structure and parameters, including cavitation inception and development. But it is important to understand that this simplified model does not describe the process of dissolved gas diffusion through the liquid-gas phase boundary which is the fundamental mechanism of liquid degassing in reality.

Another alternative option that can be found in literature is the use of equilibrium models based on the Henry’s law. According to the Henry’s law, the concentration of the released gas and the rate of its release (dissolution) processes depend on the pressure of the medium, on the gas content in the liquid, and on its solubility which is determined by the value of the Henry’s constant. These dependences are only partially taken into account in equilibrium models. On one hand, the amount of the released gas is considered as a function of the external pressure based on the Henry’s law. On the other hand, it is assumed that diffusion of the gas molecules from the liquid towards the phase boundary takes place instantaneously and so the amount of the released gas is always “adjusted” to the changing conditions. In other words, the concentration of gas is always at the equilibrium value which exactly satisfies the Henry’s law. The unsteady evolution from the initial value to the equilibrium one is not considered.

However, in reality, the release and dissolution of gas in a liquid are non-equilibrium processes that take place at finite rates. In conditions of rapid pressure drops typical for the flows of working fluids in real hydraulic devices, this circumstance plays an especially important role. When such flows are studied by numerical simulation, it is fundamentally wrong to consider changes in the fractions of the dissolved and released gas simply as transitions between the equilibrium states. It is necessary to take into account that these changes have non-equilibrium nature, and to consider the finite rates of diffusion processes. The use of the equilibrium assumption can only be reasonable if liquid degassing under sufficiently slowly changing external conditions is investigated.

The approach proposed in the present study is based on description of transport of the dissolved gas due to diffusion through the liquid-gas interface. It is important to note that the entire two phase media is considered as a quasi-homogeneous mixture which means that the liquid-gas interface itself is not resolved. This also automatically means that the interface area and the dissolved gas concentration gradient at the interface cannot be computed precisely. But in spite of this simplification it is expected that the basic idea to consider the diffusion process will allow to predict the rates of gas release and dissolution in cavitating flows at least on the level of the orders of magnitude.

2. Mathematical model
The modeling approach used in the present study is based on solution of the Navier-Stokes equations for the heterogeneous mixture which includes a liquid phase and a gaseous one. The liquid phase is considered as a homogeneous mixture of the liquid and the dissolved gas always occupying the same volume. The gaseous phase is assumed to be a homogeneous gas mixture containing the vapor and the released gas. As a result, the set of the governing equations of the mathematical model includes:

1) Mass conservation:

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0 \]  \hspace{1cm} (1)

Here \( \rho \) is the total mixture density which depends on the densities and the volume fractions of the mixture components:

\[ \rho = \alpha_{\text{liquid}} (\rho_{\text{liquid}} + \rho_{\text{gas,dissolved}}) + \alpha_{\text{gaseous}} (\rho_{\text{vapor}} + \rho_{\text{gas,released}}) \]  \hspace{1cm} (2)

The density of each component is a function of the pressure given by the corresponding equation of state; \( \vec{V} \) is the velocity field of the flow which is assumed to be the same for both phases.
2) Momentum conservation:

\[ \frac{\partial}{\partial t} (\rho \vec{V}) + \nabla \cdot (\rho \vec{V} \vec{V}) - \nabla \cdot (\rho v \nabla \vec{V}) = - \nabla p \]  

(3)

Here \( p \) denotes the pressure distribution in the flow. Viscosity of both liquid and gaseous phases is taken into account, \( \nu \) is for the kinematic viscosity coefficient of the mixture. The heat transfer effects are not assumed to significantly influence the processes under study, therefore the energy equation is not solved and the constant temperature is specified.

3) Equations of state of the liquid and gaseous phases:

The liquid is described as a weakly compressible medium with the constant speed of sound \((c_i)\):

\[ \rho_{\text{liquid}} = \rho_{\text{liquid,s}} + \frac{1}{c_i^2} (p - p_s) \]  

(4)

The isothermal ideal gas law is used as the equation of state for the vapor and the released gas.

The initial density of the dissolved gas is governed by the Henry's law and depends on the pressure of initial gas dissolution. In the present study the gas is assumed to have been previously dissolved at the normal conditions. Then, in course of pressure variations, the density of this gas remains proportional to the liquid density:

\[ \rho_{\text{gas, dissolved}} = C_{\text{He}} p_{nc} \frac{\rho_{\text{liquid}}}{\rho_{\text{liquid,nc}}} \]  

(5)

The subscripts "s" and "nc" denote the conditions of saturated vapor formation and the normal conditions, respectively; \( C_{\text{He}} \) is the Henry's law constant.

4) Cavitation modeling:

The vapor mass fraction, \( \mu \), is computed as a function of the pressure based on the barotropic model which has been already used in previous studies on purely vapor cavitation \[3\]. The equation of this cavitation model is derived from a simplified version of the First Law of Thermodynamics for the equilibrium liquid-vapor mixture. The model is formulated for an adiabatic system and the dissipation effects are neglected.

\[ \mu (p) \approx \min \left( \max \left( K (p_s - p), 0 \right), 1 \right), p \leq p_s \]  

(6)

where \( K \) is the model coefficient which is assumed to be constant.

5) Diffusion equation describing release of the dissolved gas:

The mass fraction of the released gas, \( x \), is obtained from the ordinary differential equation with a source term derived based on consideration of the diffusion process governed by the Fick’s law:

\[ \frac{dx}{dt} = \frac{1}{\rho V_c} \left[ D \frac{\rho_{\text{gas, dissolved}} - C_{\text{He}} \rho_{\text{gas, released}}}{V_c - \frac{4}{3} V_g} \left( \frac{V_g}{C} \right)^{2/3} \right] - \frac{x}{1/c^2} \frac{dp}{dt} \]  

(7)

Here \( V_c \) is a fixed elementary volume containing both liquid and gaseous phases (or the volume of a current computational cell in numerical simulations); \( D \) is the diffusivity of the dissolved gas in the liquid; \( V_s \) is the volume occupied by the gaseous phase (in other words, the volume occupied by the vapor and the released gas both separately and together); \( c \) is the speed of sound of the mixture. To derive an equation in terms of the mass fraction \( x \), it is needed to take into account that the total mass
of the elementary volume is not constant. This leads to presence of the second term containing the temporal derivative of the mixture density in the right hand side of the equation.

3. Numerical approach
The formulated models of cavitation and dissolved gas release have been implemented into the open-source CFD toolbox OpenFOAM [4] and combined with the existing finite volume solvers of the unsteady Navier-Stokes equations. The main advantage of the OpenFOAM toolbox is open access to the source codes which provides an opportunity to implement original models and algorithms. In the present study, two different numerical solvers for compressible flows were used as a base for further implementation of the cavitation and gas release models. The first solver, rhoCentralFoam, is based on an explicit density-based algorithm involving the Kurganov-Tadmor scheme [5]. The second one, cavitatingFoam, is a pressure-based solver in which the SIMPLE method [6] is used. In order to analyze the difference between the results of both solvers and to choose the more suitable one for further simulation of cavitation flows, the flow of a one phase liquid in the rectangular channel was considered (the description of the problem setup is presented in the next section). The simulation was carried out in a 2D arrangement. The main idea was that, on one hand, the solver must provide stable simulations, but, on the other hand, highly unsteady behavior of the mass flow rate through the channel typical for turbulent regime of the flow must be reproduced. Figure 1 compares the mass flow rate evolution curves obtained using both of the aforementioned numerical solvers.

![Figure 1. Left: the results of the Kurganov-Tadmor solver. Right: the results of the implicit solver.](image)

The mass flow rate evolution obtained using the Kurganov-Tadmor solver represents a monotonic growth until reaching some constant value or, depending on the flux limiter (the van Leer limiter and the Minmod one are compared), regular oscillations with the nearly constant frequency. It can be seen from the comparison that the features of turbulent regime of the flow are captured noticeably better if the implicit pressure-based solver is used, which means that the latter solver contains less numerical dissipation. This is the main reason why the implicit solver was chosen as more suitable for simulation of unsteady flows with high local gradients of the hydrodynamic parameters. This capability of the numerical solver is especially important if the goal is to study cavitation inception and development in such flows and accompanying phenomena.

Regarding the settings of the implicit solver specified in OpenFOAM, the central scheme with the van Leer TVD flux limiter (Gauss vanLeer) was used to compute the convective terms in the transport equations. The diffusive terms are computed using the central scheme (Gauss linear). The specified time step ensures the maximal value of the CFL number around 0.6. Mesh sensitivity was also studied, and it was shown that the obtained unsteady flow regime is not influenced by spatial resolution. Regarding the mesh topology, a regular mesh with rectangular computational cells was used. The total number of cells was approximately 15000, the number of cells across the throttle was 60.
4. Results and discussion

Unsteady cavitating flow in a micro-channel was numerically investigated. The flow structure and the distributions of the hydrodynamic parameters obtained demonstrate formation of vapor cavities, their evolution including collapses, and, as a result, formation and propagation of the pressure waves.

The mass flow rate through the channel corresponds to the Reynolds number of the problem equal to 28000 which means turbulent regime of the flow. Cavitation number is equal to 0.498.

![Figure 2](https://example.com/fig2.png)

**Figure 2.** Schematic diagram of the channel and cavitation structures observed in the experiment [7].

$L = 2$ mm, $l = 1$ mm, $H = 3$ mm, $h = 0.3$ mm, $\delta = 0.3$ mm.

The initial parameters in the region 1 are: $p = 3 \times 10^7$ Pa, $T = 353K$, $\rho = 800.32kg/m^3$.

The initial parameters in the region 2 are: $p = 1.14 \times 10^7$ Pa, $T = 353K$, $\rho = 786.2kg/m^3$.

![Figure 3](https://example.com/fig3.png)

**Figure 3.** The pressure and the X-component of the velocity vector at the moment $t = 4.5 \times 10^{-5}$ s.

The results obtained with and without considering release of the dissolved gas are compared.

Figure 2 shows a schematic diagram of the micro-channel under study and the initial conditions. This geometry is used commonly as a test bench for cavitation modeling applied to Diesel injector passages (see, for example, [7]). Two rectangular chambers are connected by a channel of 0.3mm height and 1mm length with sharp inlet and outlet edges (see figure 2). The working fluid is the diesel liquid with the saturation pressure at 9745.4Pa, the density of the liquid at saturation conditions is 777.56kg/m$^3$. The operating temperature is assumed to be constant and equal to 353K. Experimental observations [7] by means of shadowgraphy imaging showed that gaseous phase occurs inside the channel and in the right chamber both in form of small cavities and in form of large cavitation clouds. The same flow features were qualitatively demonstrated in simulations of a cavitating liquid flow at the conditions under study based on the purely equilibrium approach without a gas release model [8].

The boundary conditions for the present simulations were specified to reproduce the arrangement of the experiment [7] as close as possible. The condition of the fixed total pressure ($3 \times 10^7$ Pa) was used for the channel inlet, and the non-reflective condition in combination with the fixed far-field pressure ($1.14 \times 10^7$ Pa) was used for the outlet. $L = 2$ mm was enough to avoid the influence of the boundaries.

The simulation results showed that the channel flow in conditions under study has a complicated spatial structure. This can be seen in figure 3 showing the velocity field and the pressure distribution in the channel. The asymmetry of the flow is related to its complex inhomogeneous structure containing
macroscopic vortices and cavities filled with the vapor and the released gas. Depending on the flow parameters, these cavities can grow or collapse, which leads to high pressure variations (leading, in turn, to propagation of pressure waves and to erosion of construction elements which is an important problem in many industrial applications). Furthermore, the results show that the flow (and therefore cavitation and accompanying erosion as well) can be influenced by presence of the released gas.

Figure 4. The mass fractions of vapor (left) and released gas (right) at the moment \( t = 4.5 \times 10^{-5} \) s.

Qualitatively different character of formation of the vapor and the released gas was observed in the simulations and can be seen in figure 4 which shows the corresponding mass fractions. While the vapor is present locally in regions of pressure decrease below the saturation one and does not occupy the entire volume of the gaseous phase, the released gas is being dissolved back into the liquid at a finite (and rather low) rate and is therefore present in a significant part of the channel volume.

5. Conclusions

An equilibrium model of cavitating liquid flow was extended by introducing a description of release of the dissolved gas based on the diffusion model. An OpenFOAM algorithm is developed and applied for numerical simulations of unsteady cavitating flow in the channel. Initially the dissolved gas is released inside vapor cavities, but since the process of gas diffusion has a finite rate, the domains of gas release are transported by the flow and tend to occupy larger volume in the channel independently on possible vapor condensation which takes place instantaneously.

The results obtained are in qualitative agreement with the experimental observations regarding the main features of cavitating flows and the behavior of the released gas. It demonstrates adequacy and robustness of the proposed simulation algorithm which means that its further development can be a promising way to create a universal model for reliable quantitative prediction of dissolved gas release.

References

[1] Zhou J, Vacca A and Manhartsgruber B 2013 Journal of Fluids Engineering 135(9) 091305
[2] Battistoni M, Duke D J, Swantek A B, Tilocco F Z, Powell C F and Som S 2015 Atomization and Sprays 25(6) 453-83
[3] Iben U 2015 Ein- und Mehrphasenströmungen - Modellierung und Anwendungen in der Industrie. (Stuttgart: Institut für Aerodynamik und Gasdynamik, Universität Stuttgart)
[4] OpenFOAM: the open source CFD toolbox // URL: https://www.openfoam.com/
[5] Kurganov A and Tadmor E 2000 Journal of Computational Physics 160 241-82
[6] Patankar S V and Spalding D B 1972 Int. J. Heat Mass Transfer 15(10) 1787-806
[7] Skoda R, Iben U, Morozov A, Mihatsch M, Schmidt S J and Adams N A 2011 Proc. WIMRC 3rd International Cavitation Forum (Warwick, UK) vol 1 (e-publication) pp 1-10
[8] Iben U, Makhnov A and Schmidt A 2018 The Eighth Polyakov’s Reading: Proc. International Scientific Conference on Mechanics (St. Petersburg, Russia) vol 1959(1) (Maryland: AIP Conference Proceedings) 050013