Numerical Study of a Vapor Bubble Collapse near a Solid Wall

U Iben¹, A Makhnov² and A Schmidt², ³
¹ Robert Bosch GmbH, Germany
² Peter the Great Saint-Petersburg Polytechnic University, Russia
³ Ioffe Institute, Russia

E-mail: a_makhnov@mail.ru

Abstract. Cavitation is the reason for degradation of the performance of technical devices that involve high speed liquid flows. Detailed information on structure of cavitating liquid flow is of key importance for design of such devices. But experimental study of these flows is a severe problem due to extreme flow parameters and very small spatial and temporal scales. That is why numerical simulation became an efficient tool of investigation of the detailed flow structure. Modelling of the collapse of a bubble located in the proximity of a solid wall is of particular interest from the point of view of cavitation erosion analysis. A quasi-homogeneous model of cavitating flows is formulated based on the assumptions of mechanical and thermodynamic equilibrium between the liquid and vapor phases. Numerical investigation is carried out for the unsteady compressible flow developing in the process of single vapor bubble collapse. The proposed algorithm demonstrates adequacy and robustness.

Introduction
Theoretically liquids can sustain rather high negative pressures, in fact it concerns only specially prepared liquids which do not contain any inhomogeneities. In practice, liquids contain some impurities like microbubbles or solid particles. Presence of these inhomogeneities significantly reduces the tensile strength of the liquid. The growth of generated cavitation bubbles can be described by the Rayleigh-Plesset equation which usually demonstrates oscillating nature of bubble behavior. Growth of the bubble is followed by its collapse.

Collapse of bubbles in cavitating liquids can generate compression waves [1] and cumulative jets [2] which in course of fluid-structure interaction result in erosion of construction of corresponding devices and, in turn, in degradation of their performance and in damage up to full destruction. But there are also positive effects caused by cavitation which are utilized in cleaning technology and in biomedical applications [3].

Lauterborn and co-workers [4] were, probably, the first who carried out a detailed experimental investigation of bubble collapse near a solid wall. They used high-speed photography to visualize the propagation of waves and the development of jets. Experimental study of these processes is very difficult because of extremely small spatial and temporal scales. That is why numerical simulation became an efficient tool of investigation of the detailed flow structure.

The first numerical results on the collapse of a single spherical bubble near a solid wall were presented by Plesset and Chapman [5]. Later, the results from this work were used as a benchmark. These studies did not take into account the compressibility of the liquid phase which did not allow to observe the compression waves propagating in the liquid. Johnsen and Colonius [6] considered compressible liquid and studied structure of the flow induced by bubble collapse, but they neglected phase transition effect.

In the present study both compressibility of the liquid and phase transition are taken into account. The goal is to get deep insight into the key mechanisms of cavitation bubble collapse near a solid wall. The implemented approach is based on the following main assumptions. It is assumed that cavitating liquid represents a homogeneous barotropic mixture. Interface between liquid and vapor phases is not captured, and the mass and momentum conservation equations are solved for the entire two-phase
mixture. Heat transfer is neglected and the energy equation is not solved. Phase transition is described as an equilibrium process which takes place instantaneously at equilibrium saturation conditions. Applied to the problem of cavitating flow in micro-channel, a model based on the same principles was implemented in [8].

On one hand, this approach has its significant limitations. The content of generated vapor in each computational cell can be analyzed by means of the mass and volume fractions only. If the value of the vapor fraction is between 0 and 1, then it is clear that cavitation occurs but no information about the liquid/vapor interface can be derived. It is automatically impossible to get any understanding about the size, quantity and the evolution of generated cavitation bubbles. But on the other hand, all the mechanisms, that are neglected by using this simplified approach, are expected to be not critical for the dynamics of a collapsing vapor bubble. The dominating mechanism of the collapse itself is the focusing of mass and momentum at the collapse center. The dominating mechanisms important for the wall loading induced by collapse are also described by the convection terms of the conservation equations. Moreover, it is possible to study the single bubble collapse based on the proposed approach because the initial bubble can be specified as the region where the vapor fraction is equal to 1.

1. Mathematical model
A mathematical model developed and tested in the present study is based on the Navier-Stokes equations combined with the equations of state of liquid and vapor and with the cavitation model. The resulting system of the governing equations includes:

1) Mass conservation:

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

2) Momentum conservation:

\[ \frac{\partial}{\partial t} \rho \mathbf{V} + \nabla \cdot (\rho \mathbf{V} \mathbf{V}) - \nabla \cdot (\mu \nabla \mathbf{V}) = -\nabla p \] \hspace{1cm} (2)

Here \( \rho, \mathbf{V} \) and \( p \) are for the density, for the velocity and for the pressure fields, respectively. Viscosity of both liquid and vapor phases is taken into account, \( \mu \) is for the mixture viscosity. Heat transfer is not considered, therefore the energy equation is not included. The two-phase mixture of liquid and vapor is assumed to have constant and uniform temperature.

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

Liquid phase is considered to be a weakly compressible fluid, vapor phase is assumed to follow the ideal gas law.

\[ \rho \left| p \right| = \rho_0 + \frac{1}{c_L^2} \left| p - p_v \right| \]  \hspace{1cm} (3)

\[ \frac{p_v}{\rho_v} = R_v T_{\text{ambient}} \] \hspace{1cm} (4)

Here \( \rho_0 \) is for the liquid density under saturation conditions and \( p_{\text{sat}} \) is the saturation pressure. The speed of sound in the liquid phase, \( c_L \), is assumed to be constant. The subscript "v" means vapor, and \( T_{\text{ambient}} \) is the temperature of the ambient medium.
4) Cavitation model:

Description of cavitation is based on the homogeneous equilibrium approach and on the barotropic
equation of state. When the pressure of the mixture decreases below the saturated vapor pressure, the
vapor mass fraction is calculated using the barotropic model [7]. This model provides a coupling
relation between the local pressure and the mass fraction of generated vapor:

\[ m \left| p \right| \approx \min \left(-\frac{1}{r_{\text{evap}}} \frac{dh'}{dp} - \frac{1}{\rho_0}, 1\right) \approx \min \left(-K \left(p - p_v\right), 1\right), p \leq p_v \]  \hspace{1cm} (5)

Here \( m \) is for the vapor mass fraction, \( p \) is the pressure, \( r_{\text{evap}} \) is for the evaporation enthalpy, \( h' \) is
the enthalpy of the liquid phase. Estimations show that in the problem under study the coefficient \( K \) is
approximately constant. Formula (5) is derived from a simplified version of The First Law of
Thermodynamics for the considered two-phase mixture. Simplifications are based on the assumptions
that the considered system is adiabatic and the entropy change due to dissipation is negligible.

2. Approach to solution of model equations
The open-source computational fluid dynamics toolbox OpenFOAM [9] is used. The main advantage
of the OpenFOAM toolbox is open access to the source codes which provides an opportunity to
implement original models and algorithms. The following settings of the numerical algorithm were
specified in OpenFOAM:

- PISO scheme for pressure-velocity coupling;
- Approximation of the convective terms using the bounded central scheme (Gauss
  limitedLinear);
- Approximation of the diffusive terms using the central scheme (Gauss linear);
- Gradients calculation using the Green-Gauss method.

3. Results and discussion
Collapse of a spherical vapor bubble near a solid wall is investigated. The computational domain and
the vapor bubble in the proximity of the wall (both the initial topology and the configuration after the
time of 2.10e-6 s) are shown in Fig. 1. The problem is solved in axisymmetric arrangement.

Fig. 1. Single vapor bubble collapse near a wall: the density distributions at \( t = 0 \) (the left picture) and
\( t = 2.10\text{e-6} \) s (the right picture).
Fluid properties correspond to water at the temperature of 25°C: \( \rho_0 = 997.01 \text{ kg/m}^3, \) \( p_{\text{sat}} = 3149 \text{ Pa} \) (saturated water vapor at 25°C), \( c_L = 1498 \text{ m/s} \). The initial bubble radius \( R \) is equal to 10 μm. Vapor inside the bubble is initially at saturation conditions. Both liquid and vapor are initially at rest, and the initial density field is derived from the pressure using the equation of state. The initial pressure of ambient water is equal to 101 325 Pa (1 atmosphere).

Important parameter of the problem is the distance, \( h \), from the bubble center to the wall. In the present study we consider \( h = 1.2R \). Flow, generated in this case, possesses all the key features characteristic to bubble collapse in the proximity of the wall.

In the framework of axisymmetric arrangement the computational domain is a two-dimensional rectangle. The following boundary conditions are stated: (1) on the solid wall this is no-slip condition for the velocity, the symmetry condition is specified for the pressure; (2) on the right and upper boundaries (see Fig. 1) these are non-reflective conditions for both velocity and pressure.

The size of the domain used in the present simulations is \( 4R \times 4R \). The computational mesh consists of 200 cells in both directions. Uniform mesh with square cells is used. The time step is determined by the CFL condition. The simulations were carried out with several different levels of spatial resolution, and reliability of the results in terms of grid independence has been achieved. Another one methodological issue that was studied is the influence of the boundary conditions used on the right and upper boundaries. It has been shown that using of the symmetry boundary condition leads to reflection of the compression and rarefaction waves. This is the reason why the non-reflective “waveTransmissive” condition available in the OpenFOAM toolbox was chosen in the end. The size of the computational domain was chosen to satisfy two important requirements. The first requirement is that the domain should be not too large but still enough to obtain the domain-independent solution. The second one is that domain should be large enough to analyze the flow structure and waves propagation not only near the wall and directly around the bubble, but also in the bulk of the liquid at least 2R away from the bubble. Based on this logic it was finally decided to use the \( 4R \times 4R \) domain.

Fig. 2 shows the pressure distributions and profiles along the wall for different time moments. Initially the pressure decreases due to rarefaction wave propagating from the bubble and becomes equal to the saturation pressure (cavitation occurs), then liquid is accelerated towards the bubble and the pressure increases significantly. Finally, the collapse of the bubble results in propagation of the compression wave. At first the wave reaches the epicenter, and then the pressure peak shifts to the periphery. In context of the present study the word “epicenter” means the point on the wall directly under the center of the bubble (in other words, the projection of the bubble center onto the wall). A fragment of the computational domain in the vicinity of the bubble is presented.

Fig. 2. Pressure distributions (the left picture, 1: \( t = 2.240 \times 10^{-6} \text{ s} \), 2: \( t = 2.248 \times 10^{-6} \text{ s} \)) and profiles along the wall (the right picture) for different time moments. A fragment of the computational domain is shown.
To get insight into the flow structure, scalar and vector fields of the velocity were analyzed. Fig. 3 shows the distributions of the velocity magnitude taken at two different time moments. A fragment of the computational domain in the vicinity of the bubble is presented.

At the first moment (left half of the picture, $t = 2.10 \times 10^{-6}$ s) liquid is flowing from all directions towards the bubble center. The stage of the bubble collapse that corresponds to this time moment is shown in Fig. 1 (right picture). Significant asymmetry is observed in the distribution of the velocity magnitude which has a maximal value above the bubble. This asymmetric development of the flow results in generation of the cumulative jet which is illustrated on the right half of the picture ($t = 2.24 \times 10^{-6}$ s). Maximal velocity inside this jet is observed in the vicinity of the center of collapse. The velocity close to the wall is not that high, because there is a significant damping effect of the initial liquid gap between the bubble and the solid wall.

The plot presented in Fig. 4 shows the temporal evolution of the pressure taken at the epicenter. Initially, the wall loading is equal to the initial liquid pressure around the bubble (101 325 Pa). Then the rarefaction wave propagates from the bubble, and the reflected rarefaction wave leads to generation of a vapor cavity attached to the wall. Condensation of this vapor cavity results in the first increase of the wall loading, and then the compression wave from the collapse of the initial spherical bubble reaches the wall and causes further pressure increase. Non-monotonic evolution of the wall loading is connected with complex unsteady behavior of the cumulative jet and its interaction with the falling and reflected compression waves.

![Fig. 3. Distributions of the velocity magnitude (m/s) and vectors for two time moments, 1: $t = 2.10 \times 10^{-6}$ s, 2: $t = 2.24 \times 10^{-6}$ s. A fragment of the computational domain is shown.](image)

![Fig. 4. Wall loading at the epicenter over time.](image)
4. Conclusions
A model of cavitating flows was formulated based on the assumptions of mechanical and thermodynamic equilibrium between the liquid and vapor phases. Phase transition was taken into account in the equation of state for the two-phase mixture. Numerical study has been carried out for the unsteady compressible flow developing in course of a single vapor bubble collapse near a solid wall. Non-monotonic profiles of the wall pressure were obtained and mechanisms of formation of such profiles were elucidated.

Formation of the cumulative jet was reproduced. The velocity distributions demonstrated the expected asymmetric scenario of bubble collapse near the wall. The time moment of the pressure peak at the epicenter corresponds to moment when the jet is impinging onto the wall. Both the jet and the compression waves contribute into the resulting wall loading. Detailed parameter study including variation of the initial distance from the bubble center to the wall could provide more information and is therefore considered as a possible next step.

The developed algorithm demonstrates adequacy and robustness.

Acknowledgements
Andrei Makhnov thanks the German-Russian Interdisciplinary Science Center (G-RISC) funded by the German Federal Foreign Office via the German Academic Exchange Service (DAAD) for the support of these investigations.

References
[1] Rayleigh L 1917 *Phil. Mag.*. 34
[2] Kornfeld M and Suvorov L 1944 *J. Appl. Phys.* 15
[3] Jamaluddin A R, Ball G J, Turangan C K and Leighton T G 2011 *J. Fluid Mech.* 677 305–41
[4] Vogel A, Lauterborn W and Timm R 1989 *J. Fluid Mech.* 206 299–338
[5] Plesset M S and Chapman R B 1971 *J. Fluid Mech.* 47 283–90
[6] Johnsen E and Colonius T 2008 *J. Acoust. Soc. Am.* 124(4): 2011–20
[7] Iben U 2015 *Ein- und Mehrphasenströmungen - Modellierung und Anwendungen in der Industrie*. (Stuttgart: Institut für Aerodynamik und Gasdynamik, Universität Stuttgart)
[8] Iben U, Makhnov A and Schmidt A 2018 Three-dimensional numerical simulations of turbulent cavitating flow in a rectangular channel *AIP Conference Proceedings* 1959 050013
[9] OpenFOAM: the open source CFD toolbox // URL: https://www.openfoam.com/