Numerical Investigation of Cavitating Flows with Liquid Degassing

U Iben\textsuperscript{1}, A Makhnov\textsuperscript{2} and A Schmidt\textsuperscript{2,3}
\textsuperscript{1} Robert Bosch GmbH, Germany
\textsuperscript{2} Peter the Great Saint-Petersburg Polytechnic University, Russia
\textsuperscript{3} Ioffe Physical and Technical Institute, Russia

E-mail: a_makhnov@mail.ru

Abstract. Cavitation is a phenomenon of formation of bubbles (cavities) in liquid as a result of pressure drop. Cavitation plays an important role in a wide range of applications. For example, cavitation is one of the key problems of design and manufacturing of pumps, hydraulic turbines, ship’s propellers, etc. Special attention is paid to cavitation erosion and to performance degradation of hydraulic devices (noise, fluctuation of the mass flow rate, etc.) caused by formation of a two phase system with an increased compressibility. One more important problem accompanying cavitation is liquid degassing due to diffusion of the dissolved gas into the cavities. Determination of the degassed air content is important for the performance forecast of devices. Mathematical models of the degassing in cavitating flows are not yet developed and presented in the literature satisfactorily. Therefore, development of a model for simultaneous cavitation and liquid degassing is an important fundamental and applied task. To validate the algorithm simulations of unsteady flow of a cavitating liquid in a channel have been conducted. The obtained results are in satisfactory agreement with the experimental data and demonstrate the efficiency and robustness of the formulated model and the algorithm.

Introduction
Cavitation means formation of bubbles (cavities) as a result of pressure drop. This phenomenon has been a subject of many experimental and theoretical investigations for more than one hundred years. The first fundamental studies on cavitation date back to the beginning of the 20\textsuperscript{th} century and belong to Lord Rayleigh. Cavitation plays a very important role in power engineering (pumps, hydraulic turbines), in various liquid injection systems, in shipbuilding and in many other applications. Formation of a two phase system with an increased compressibility can lead to performance degradation of hydraulic devices (noise, fluctuation of the mass flow rate, etc.). Collapse of cavitation bubbles and clouds often leads to structure erosion. Therefore, the cavitation phenomenon greatly affects performance and lifetime of hydraulic devices.

All liquids contain some amount of a dissolved gas. It is known that the concentration of dissolved air in water at normal conditions varies from 8 vol \% to 14 vol \% \cite{1}. Typical mineral-oil-based hydraulic liquids can also dissolve high amounts of air (7-12 vol \%) \cite{2,3}. The dissolved gas transported from the liquid into growing cavitation bubbles due to diffusion through the interfaces is a reason of the higher bubble volume fraction \cite{4}.

Therefore, physically cavitation is a superposition of two mechanisms: phase transition and vapor bubbles formation and release of the dissolved gas into the bubbles. Thus, the bubbles formed during cavitation contain both vapor and gas. Nowadays, quite a few is known about the interaction of these two processes. Mathematical description of these processes is not completed yet. The goal of the present study is development and validation of a combined mathematical model for numerical (CFD) simulation of cavitating flows with liquid degassing.

The present paper is divided into four sections. In the first section, a description of the developed mathematical model is given. The second section is dedicated to the numerical approach that is used in simulations. In the third section, the most important results of the study are presented and discussed. The last section summarizes the main conclusions drown from the obtained results.
1. Mathematical model
The model developed and validated in the present study includes a model of vapor cavitation as well as a model of liquid degassing due to gas diffusion into the vapor cavities. The total system of governing equations (1) is a combination of the Navier-Stokes equations, the developed cavitation model, and the developed degassing model:

\[
\begin{align*}
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) &= 0 \\
\frac{\partial}{\partial t} (\rho \mathbf{u}) + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) - \nabla \cdot (\mu \nabla \mathbf{u}) &= -\nabla p
\end{align*}
\]

Here \( \rho, \mathbf{u} \) and \( p \) are for the density, the velocity and the pressure fields, respectively.

Viscosity of both liquid and vapor phases is taken into account (\( \mu \) is the dynamic viscosity coefficient of the mixture). Heat transfer is not considered, therefore the energy equation is not solved. The two-phase mixture of liquid and vapor is assumed to have constant and uniform temperature.

1.1. Model of vapor cavitation
A vapor cavitation model formulated in the present study is based on the homogeneous equilibrium approach and barotropic equation of state. Liquid phase is described using a weakly compressible fluid linearization (2), vapor phase is assumed to follow the ideal gas law.

\[ \rho(p) = \rho_0 + \frac{1}{c_L^2}(p - p_v) \]  

Here \( \rho_0 \) is for the liquid density under saturation conditions and \( p_v \) is the saturation pressure. The speed of sound in the liquid phase, \( c_L \), is assumed to be constant.

When the pressure of the two-phase mixture of liquid and vapor becomes less than the saturation pressure, the vapor mass fraction is calculated using the barotropic model [5]. This model provides a coupling between the vapor mass fraction and the pressure:

\[ m(p) = -\frac{1}{r_{e_{\text{evap}}}} \left( \frac{dh'}{dp} - \frac{1}{\rho_0} \right) (p - p_v) = -K(p - p_v) \]  

Here \( m \) is for the vapor mass fraction, \( p \) is the pressure, \( r_{e_{\text{evap}}} \) is for the evaporation enthalpy, \( h' \) is the enthalpy of the liquid phase. The coefficient \( K \) is assumed to be constant in the present study.

Formula (3) is derived from a simplified version of The First Law of Thermodynamics for the considered two-phase mixture. Simplifications are based on the assumption that, firstly, the considered system is adiabatic, and, secondly, entropy change due to dissipation can be neglected.

1.2. Model of degassing
A degassing model formulated in the present study provides a coupling between the dissolved gas diffusion process and the local characteristics of the cavitating flow (pressure and vapor volume fraction). The model is based on an analytical solution of the diffusion equation in a spherical cell corresponding to a single bubble:

\[ D \frac{1}{r^2} \frac{\partial}{\partial r} \left( r^2 \frac{\partial c_s}{\partial r} \right) = 0 \]  

Here \( D \) is for the dissolved gas diffusivity, \( c_s \) is for the dissolved gas concentration, \( r \) is the radial coordinate (distance from the bubble center).

The Henry's law is used to formulate a boundary condition for gas concentration at the bubble surface.
Equation (4) describes stationary diffusion of the dissolved gas. Transient term is neglected because validation cases considered in the present study are characterized by much smaller time scales (by 2-3 orders of magnitude) in comparison with the diffusion time scale.

2. Numerical method

The system of governing equations is solved numerically based on a finite volume approach. Open-source CFD toolbox OpenFOAM [6] is used in the present study for simulations. The main advantage of the OpenFOAM toolbox is open access to codes which provides possibility to implement original models and algorithms.

The following settings of the algorithm were specified:
- PISO scheme for pressure-velocity coupling;
- Approximation of convective terms using Gauss limitedLinear 1 scheme;
- Approximation of diffusive terms using central scheme (Gauss linear).

3. Results and discussion

The developed algorithm has been validated on two test cases. The first test case is simulation of a single cylindrical bubble collapse. The second test case is the cavitating flow in the planar throttle for which the experimental data is available.

3.1. Single cylindrical bubble collapse

The main goal of this test case is to study the capability of the solver to capture a complex structure of interacting compression and rarefaction waves. Initial conditions correspond to cylindrical vapor domain located in a square liquid domain (see Fig. 1). Boundaries of the square liquid domain are solid walls with no-slip condition for the velocity and symmetry boundary condition for the pressure.

Size of the square domain is 100 mm x 100 mm. Radius of the cylindrical vapor domain is 0.0015 mm. Fluid properties correspond to water at the temperature of 20°C: \( \rho_0 = 998 \text{ kg/m}^3 \), \( p_v = 2330 \text{ Pa} \), \( c_L = 1485 \text{ m/s} \). Initial pressure inside the vapor and liquid regions is specified to 2000 Pa and \( 2\cdot10^7 \text{ Pa} \), respectively.

The computational mesh consists of 1000 cells at both directions. The mesh is uniform, and all the cells have a shape of square. The time step is 10 ns.

![Fig. 1a. Pressure distribution at t = 7 μs, t = 16 μs and t = 17.5 μs.](image-url)
Fig. 1b. Pressure distribution at $t = 25 \mu s$, $t = 37.5 \mu s$ and $t = 50 \mu s$.

Pressure distributions at different time moments are presented in Fig. 1a and 1b. The first set of pictures (Fig. 1a) illustrates two processes. On one hand, a rarefaction wave is propagating from the vapor region into the liquid. On the other hand, the vapor region in the center is being compressed, liquid is accelerated towards the center and finally this leads to the collapse. The second set of pictures (Fig. 1b) visualizes the propagation of a pressure wave generated by the collapse. Complex wave structure of the flow formed by interaction of compression and rarefaction waves is seen in Fig. 1b at time $t = 50 \mu s$.

3.2. Cavitating flow in a planar throttle

The main goal of the simulations presented in this subsection is to validate the implemented solver on the experimental data. Geometry of the channel and fluid properties were selected in accordance with the experiment [7]. Fig. 2 shows a scheme of the considered channel. Inlet total pressure is 300 bar, outlet static pressure is 114 bar.

The most important problems caused by cavitation in hydraulic devices are mass flow rate fluctuations and cavitation erosion. That is why special attention in the present study is paid to the mass flow rate and to the wall loading. Left plot in Fig. 3 demonstrates oscillations of the mass flow rate with high amplitude. Right plot shows that despite the decrease of the average wall pressure due to cavitation, formation and collapse of cavities lead to local pressure peaks up to 1000 bar. Such pressure peaks can result in erosion.

Fig. 2. Experimental throttle geometry [7] and the computational domain.
Fig. 3. The mass flow rate (left) and the throttle wall loading (right).

Fig. 4. Density profiles inside the throttle and qualitative comparison with the experiment [7].

Simulations enabled to observe inception and development of the cavitation zones. Comparison shows that the predicted cavities are in qualitative agreement with experimental observations [7].

4. Conclusions
A model for cavitating flows with stationary degassing is formulated based on the Euler-Euler description and barotropic equation of state for equilibrium two-phase mixture. Results of the validation show the capability of the model to predict inception of cavitation, cavity development and collapse. Comparison with the experimental data demonstrates the efficiency and robustness of the formulated model and algorithm.
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] Brennen C E 1995 *Cavitation and Bubble Dynamics* (Oxford University Press)
[2] Bock W, Braun J, Puhl N, and Heinemann H 2010 *Air Release Properties of Hydraulic Fluids* (Internationals Fluidtechnisches Kolloquium (IFK), Vol. 7)
[3] Iben U, Wolf F, Freudigmann H-A, Froehlich J, and Heller W 2015 *Optical Measurements of Gas Bubbles in Oil Behind a Cavitating Micro-Orifice Flow* (Exp. Fluids, 56(6))
[4] Crowe C T 2006 *Multiphase Flow Handbook* (Taylor & Francis Group, LLC)
[5] Iben U 2015 *Vorlesungsskript Ein- und Mehrphasenströmungen - Modellierung und Anwendung in der Industrie. Institut für Aerodynamik und Gasdynamik* (Universität Stuttgart)
[6] OpenFOAM: the open source CFD toolbox // URL: https://www.openfoam.com/
[7] Skoda R, Iben U, Morozov A, Mihatsch M, Schmidt S J, and Adams N A 2011 *Numerical Simulation of Collapse Induced Shock Dynamics for the Prediction of the Geometry, Pressure and Temperature Impact on the Cavitation Erosion in Micro Channels* (Warwick: WIMRC 3rd International Cavitation Forum 2011)