DOI: <https://doi.org/10.14311/TPFM.2021.005>

NUMERICAL SIMULATION OF THE PROBLEM OF BLOWING GAS INTO A LIQUID FLOW

Natalia Dimitrieva^{1,2}

¹ Department of Hydrobionics and Boundary Layer Control, Institute of Hydromechanics of the National Academy of Sciences of Ukraine, 8/4 Zheliabova St., 03057 Kyiv, Ukraine

² Institute of Physics and Technology, National Technical University of Ukraine "Igor Sikorsky Kyiv Polytechnic Institute", Kyiv, Peremohy ave., 37, 03056 Kyiv, Ukraine

Abstract

The non-stationary 3D problem of the formation of a gas cavity in a moving liquid was studied. Based on the open source software OpenFOAM numerical simulations of two-phase flows have been performed. A standard InterFoam solver for two isothermal incompressible media without phase change has been proposed. It has shown good results at low velocity when natural cavitation (evaporation) is negligible. The mesh is constructed by the stepwise cell thickening method using the snappyHexMesh utility, which takes into account small-scale flow structures in the phase transition zone and near the streamlined body. The influence of gas blowing parameters on the formation of an air cavity, size, shape and stability has been investigated. Good qualitative agreement with experimental data was obtained.

Keywords: two-phase flow, cavity, numerical simulation, volume of fluid

1 Introduction

Currently, the problem of flow control and drag reduction remains relevant. One of the methods to reduce drag is to create a system of cavities on the streamlined surface filled with vapor or air [1, 2] which is called in [3] as supercavitation.

The phenomenon of natural hydrodynamic cavitation is local in nature and occurs only where there are special conditions for the evaporation of liquid. Such a condition can be, for example, a local decrease in pressure in a liquid with an increase in velocity behind a streamlined body or in a pipe with a variable diameter. Natural cavitation is difficult to control. To create and maintain a stable cavity, air is often forcibly blown into the vapor-containing cavity.

Ventilated cavities behind a streamlined body at a relatively low flow velocity are examined in this paper. At this stage of research, the problem was simplified by eliminating the phenomenon of evaporation.

The purpose of this study is to develop a technique for the numerical calculation of the non-stationary problem of the formation of a gas cavity in a fluid flow. Of scientific and practical interest is the analysis of geometric and dynamic parameters that affect the formation and development of the cavity, its size, shape and stability.

2 Mathematical Modelling

A two-phase immiscible medium water-air is studied. At a low flow velocity, the effect of liquid phase evaporation can be neglected.

Volume of Fluid (VOF) method is used to solve the two-phase flow problem. The two immiscible media are considered one effective fluid all over the domain. Mathematical modelling of the problem is based on the equation set of incompressible fluid mechanics which includes equations of continuity, Navier-Stokes, diffusion of phase volume fraction.

2.1 Volume of Fluid Method

VOF is a method that is based on determining the indicator function for a cell of the calculation grid [4]. The indicator function $\alpha(x, t)$ is defined at all mesh points in following way: Function α is equal one if the point location is at time t occupied by fluid and function α is equal zero if there is an air. Then the function α is evaluated in the mesh cells as the average of all points located at the cell, i.e. it represents

the amount of fluid in the cell. Cells with a value of α different from 0 to 1 must have a free interfacial surface. When calculating the model equations, the derivatives of α can then be used to determine the normal to the cell face. The fluid motion with its interface is determined by the transport equation for the phase fraction α :

$$\frac{\partial \alpha}{\partial t} + \frac{\partial}{\partial x_j} (\alpha u_j) = 0 \quad (1)$$

One of the most important questions in the numerical simulation of free-surface flows using the VOF model is the conservation of the phase fraction. Accurate calculation of the phase fraction distribution is crucial for the correct assessment of the surface curvature. It is necessary to determine the force of surface tension and the corresponding pressure gradient across the free surface. The surface area between the two phases is very sensitive to the mesh resolution.

Thus, the VOF provides a simple and economical free surface detection method in 2D or 3D meshes. This method can be used to detect discontinuity surfaces of the material.

2.2 Governing Equations

The equations of continuity and momentum is defined as follows:

$$\frac{\partial u_j}{\partial x_j} = 0, \quad (2)$$

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\mu \frac{\partial u_i}{\partial x_j} \right) + f_{\sigma i}, \quad (3)$$

where density ρ is calculated as weighted averages based on the fraction distribution of the fluid, i.e.

$$\rho = \alpha \rho_1 + (1 - \alpha) \rho_2, \quad (4)$$

Here, u_i is component of the velocity field, ρ is density, ρ_1 is water density, ρ_2 is air density, p is the pressure, μ is the dynamic viscosity coefficient, t is time. Surface tension $f_{\sigma i}$ is modeled as the surface force [5, 6], which is calculated as follows:

$$f_{\sigma i} = \sigma \kappa \frac{\partial \alpha}{\partial x_i}, \quad \kappa = -\frac{\partial n_i}{\partial x_i} = -\frac{\partial}{\partial x_i} \left(\frac{\partial \alpha / \partial x_i}{|\partial \alpha / \partial x_i|} \right)$$

where σ is the surface tension constant, κ is the curvature [6, 7]. The two immiscible media are considered one effective fluid all over the domain. Its physical properties are calculated as weighted averages based on the fraction distribution of the fluid.

3 Numerical Modeling

Numerical modeling of equations (1-4) is carried out within the framework of the open-source software OpenFOAM (www.openfoam.com) by the finite volume method [8].

3.1 Solver

A standard numerical model interFoam of the OpenFOAM package is used to solve this problem corresponding to the equation system (1-4). The interFoam solver for two incompressible, isothermal media without a phase transition uses the volume of fluid method. Turbulence models were not applied to address this problem. Testing of the interFoam model, which was carried out in [5], showed that:

- The mass conservation and acceptable advection errors were performed perfectly.
- The solver has shown high computational accuracy for inertia-dominated flows. Even with modest mesh resolutions, accurate physics has been fixed for non-stationary problems in the high Weber number regime.
- The momentum formulation of interFoam helps to yield a discrete balance between pressure gradient and surface tension.
- Accuracy of curvature computation was also evaluated through a verification test involving a stationary 2D droplet. The finding is that the average curvature and the average pressure jump across the interface good agreement with experiments was obtained.

- InterFoam was able to show a good result even with modest levels of grid resolution in the test cases for a standing capillary wave, Rayleigh breakup of a laminar jet and capillary retraction of a liquid jet.

3.2 Geometry and Mesh

The considered domain has dimensions of 30x30x110 mm. At a distance of 15 mm from the initial section is a hemispherical body (cavitator) with a diameter of 5 mm.

In this paper, it is proposed to use the snappyHexMesh technique intended for automatic generation of three-dimensional unstructured meshes with a predominance of hexahedral elements. The utility snappyHexMesh automatically generates three-dimensional grids of hexaelements (hex) and split hexaelements (split-hex) based on the triangulated geometry of the studied body in the format of Stereolithography (STL).

As a result, the mesh is constructed that takes into account small-scale flow elements in the interfacial zone and near the streamlined body. The 3D mesh shown in Fig. 1 consists of about 2.5 million cells, 98 % of which are hexaelements. The minimum cell has a volume of the order of 10^{-5} mm^3 .

3.3 Boundary Conditions

The constant fluid velocity $u_z = U_0$ is set at the inlet. No-slip conditions are used on the solid wall of the cavitator. As for the pressure, the boundary condition fixedFluxPressure on the cavitator wall was used. It is more acceptable than the standard zeroGradient when additional forces such as gravity and surface tension are included in the equations. Pressure is set to a constant (1 bar) at the outlet.

At the initial time $t = 0$, the liquid phase $\alpha = 1$ is set over the entire domain except for the air blowing region. Blowing parameters are set using the preprocessing utility, setFields. The zone with special boundary conditions (hole) is defined on the rear flat wall of the hemisphere. The absence of a liquid phase $\alpha = 0$ and a constant velocity $u_z = U_c$ are prescribed within this zone. Using setFields utility, we didn't need to rebuild the mesh when changing the hole parameters.

Figure 1: Mesh

4 Calculation Results

This section presents the results of numerical calculations. It is of interest to analyze the parameters that affect the formation and development of the cavity, its size, shape, and stability. The external liquid flow velocity was fixed, $U_0 = 1.0 \text{ m/s}$. The volumetric gas flow rate through the hole, $Q = U_c S$, was changed by changing the blowing velocity, U_c , and the area of the hole, S .

4.1 Visualization and Comparison with Experimental Data

Test numerical experiments showed that a stable air cavity is formed in the wake of the body (Fig.2). The thickness of the ventilated cavity is determined by the diameter of the cavitator, which is consistent with the data of many physical experiments [1, 3]. The calculation results confirmed that the stability of the cavity shape depends on the velocity of the surrounding liquid and the air flow rate. At low flow rates, the gaseous cavity has an unstable shape and volume, and a nonstationary unstable bubble structure is observed in the wake.

Fig. 2, a) presents the results of photoregistration of the ventilated cavity which were obtained in the hydrodynamic tubes of the Institute of Hydromechanics of the NAS of Ukraine. Numerical results (Fig. 2, b) showed a more symmetrical cylindrical shape of the gaseous cavity. First of all, this is due to the fact that in the calculations the force of gravity was neglected in order to study the process of the formation of the cavity depending on the velocity and geometry of the gas blowing hole without the influence of external factors. The experimental data showed the curvature of the cavity in the vertical direction due to the influence of gravity and the emersion of bubbles in the wake (Fig. 2, a).

Figure 2: Momentary patterns of the formed ventilated cavity: experiment (a), calculations (b)

The most significant differences between the results of numerical calculations and experimental data are observed behind the air cavity. Experimental data showed a gradual decrease in diameter along the cavity. As a result of calculations the cavity is closed in the form of a corona and bubbles periodically come off the sharp edges of the phase interface. You can notice that the separated bubbles are carried further into the liquid flow, twisting like a spiral, which qualitatively coincides with the experimental data of photographic recording (Fig. 3).

Figure 3: Bubbles behind the air cavity: experiment (a), calculations (b)

4.2 Cavity Formation

Of interest is the process of formation of a gaseous cavity in the wake of the body in a liquid flow, shown in Fig. 4. This case was calculated when blowing air velocity U_c exceeded external fluid velocity U_0 twice. Fig. 5 illustrates the velocity field in the central section at the respective times.

When the air flow begins to flow out of the hole, it spreads over the entire back wall, filling all the space behind the body. This can be explained by the no slip conditions. Thus, the gaseous flow receives a radial component losing velocity as can be seen in Fig. 4, a) and Fig. 5, a). When the air bubble has reached the diameter of the body, a hemisphere, it becomes cylindrical with an inwardly curved edge in the area of reverse flow (Fig. 4, b and Fig. 5, b). Due to the difference in velocity the trailing edge gradually sharpens acquiring a corona-like structure, as shown in Fig. 4, c).

The formed corona has an unsteady structure, from which bubbles begin to break off and be carried further into the fluid flow (Fig. 4, c-e). Over time, the corona becomes more complex, pointed and unstable. As we can see in fig. 4, e), the bubble emissions period gradually decreases until it becomes quasi-stationary. Behind the formed gaseous cavity, the flow of bubbles swirls in the flow and goes further into the flow in a spiral (Fig. 3, b).

It should be noted that the shape of the developed cavity weakly depends on the blowing velocity or air flow rate. The cavity diameter depends only on the diameter of the streamlined body independently of the size of the blowing hole. Air flow rate mainly affects the length and stability of the cavity.

4.3 Analysis of Physical Characteristics in the Air Cavity

I would draw particular attention to the result that the pressure field in the formed stable air cavity and in the surrounding liquid is practically the same. This is confirmed by numerous theoretical and

experimental data. Fig. 6 shows a momentary pattern of the distribution of the phase fraction in the central section for the case when the velocity of gas blowing out is the same with the velocity of the surrounding liquid flow. The gaseous phase is indicated in blue, and the liquid one – in red.

Fig. 7 illustrates the pressure field at the time corresponding to Fig. 6. You can see that the pressure field generally changes little in the wake behind the streamlined body, except for the phase change regions. Detailed analysis showed that when the phases change, there are jumps in pressure values of not more than 0.2 %.

Figure 4: Air cavity formation process:

a) $t = 0.004$ s, b) $t = 0.008$ s, c) $t = 0.018$ s, d) $t = 0.025$ s, e) $t = 0.055$ s, f) $t = 0.100$ s
 $(U_0 = 1$ m/s, $U_c = 2$ m/s)

Figure 5: Velocity field formation process:

a) $t = 0.004$ s, b) $t = 0.008$ s, c) $t = 0.018$ s, d) $t = 0.025$ s, e) $t = 0.055$ s, f) $t = 0.100$ s
 $(U_0 = 1$ m/s, $U_c = 2$ m/s)

Figure 6: The phase fraction α in the central section. The gas is in blue, the liquid is in red.
($U_0 = 1$ m/s, $U_c = 1$ m/s)

Figure 7: Pressure field in the central section.
($U_0 = 1$ m/s, $U_c = 1$ m/s)

Within the framework of this investigation, the influence of various parameters on the structure of a two-phase flow was studied. The structure of the air cavity with a bubble trail for the above cases can be seen in Fig. 8. Fig. 8, a) represents the case when the diameter of the air blowing hole coincided with the diameter of the cavitator. Fig. 8, a) and Fig. 8, b) correspond to the same values of gas flow rate Q due to changes in the thickness of the hole and the blowing velocity. For cases in Fig. 8, b) and Fig. 8, c) the size of the hole was the same, and the blowing velocity and flow rate differed twice.

The results of numerical calculations showed that the diameter of the air cavity does not depend on the size of the blowing hole, but is completely determined by the diameter of the cavitator. When the blowing velocity is increased, the length of the cavity increases. Reducing the blowing velocity has a negative effect on the stability of the gaseous cavity. A bubble trail is formed in the separation zone, which is carried further into the flow and twists in a spiral.

Figure 8: Momentary patterns of the 3D air cavity structure: a) $U_c = 0.2$ m/s, $S = 20 \cdot 10^{-6}$ m², $Q = 4 \cdot 10^{-6}$ m³/s; b) $U_c = 1$ m/s, $S = 4 \cdot 10^{-6}$ m², $Q = 4 \cdot 10^{-6}$ m³/s; c) $U_c = 2$ m/s $S = 4 \cdot 10^{-6}$ m², $Q = 8 \cdot 10^{-6}$ m³/s

Fig. 9, a) shows the pressure distribution along the z axis at different values of the blowing velocity U_c . The calculation results show that in the initial part of the cavity there is a uniform pressure regardless of the parameters of gas blowing. The length of this zone depends mainly on the gas flow rate; the size of the hole has little effect on the length of the cavity. Further downstream, the pressure should have a discontinuous structure in the zones where the bubbles are located.

The distribution of the longitudinal velocity U_z along the z axis shows a sharp decrease of the value in the area of a stable air cavity and a change in the direction of movement regardless of the blowing parameters (Fig. 9, b). Jumps of velocity values can be seen in the non-stationary zone of separation of bubbles from the cavity. Next, the velocity of the two-phase mixture gradually increases to the value of the velocity of the main fluid flow U_0 .

Figure 9: Pressure p (a) and longitudinal velocity component U_z (b): 1) $U_c = 0.2$ m/s, $S = 20 \cdot 10^{-6}$ m², $Q = 4 \cdot 10^{-6}$ m³/s; 2) $U_c = 1$ m/s, $S = 4 \cdot 10^{-6}$ m², $Q = 4 \cdot 10^{-6}$ m³/s; 3) $U_c = 2$ m/s, $S = 4 \cdot 10^{-6}$ m², $Q = 8 \cdot 10^{-6}$ m³/s

5 Conclusions

A technique for numerical calculations of two-phase flows using the open source software OpenFoam is proposed. The interFoam model is used for two incompressible isothermal media without a phase transition. The mesh was constructed by the method of stepwise cell thickening using the snappyHexMesh utility to generate 3D unstructured mesh with a predominance of hexahedral elements. It takes into account small-scale flow structures in the phase transition zone and near the streamlined body. Calculations of the formation of an air cavity in a moving fluid were performed. It was determined that air blowing in the tail part of the hemispherical body forms a stable cylindrical cavity in the wake. Bubbles are periodically separated from the air cavity and are carried further into the fluid flow with a spiral swirl. The influence of geometric and dynamic parameters on the formation and development of an air cavity, its size, shape and stability has been investigated. The thickness of the air cavity depends on the diameter of the cavitator and does not depend on the diameter of the blowing hole. Increasing the velocity or flow rate of gas has a positive effect on the length and stability of the formed cavity. Good qualitative agreement with experimental data was obtained.

References

- [1] Wang B., Wang J., Chen D., Sun N., Wang T.: Experimental Investigation on Underwater Drag Reduction Using Partial Cavitation. *Chin. Phys. B*, vol. 26, no.5 (2017) pp. 054701.
- [2] Amromin E., Mizine I.: Partial Cavitation as Drag Reduction Technique and Problem of Active Flow Control. *Marine Technology*, vol. 40, no. 3 (2003) pp. 181–188.
- [3] Supercavitation: Advances and Perspectives. Ed. I.Nesteruk. Springer-Verlag, Berlin and Heidelberg, 2012. ISBN 978-3-642-23656-3.
- [4] Hirt C.W., Nichols B.D.: Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries, *J. Comp. Phys.*, vol. 39, no. 1 (1981) pp. 201-225.
- [5] Deshpande S. S., Anumolu L., Trujillo M. F.: Evaluating the performance of the two-phase flow solver interFoam *Computational science & discovery*, vol. 5, no 1 (2012) pp. 014016.
- [6] Heyns J.A., Oxtoby O.F.: Modelling surface tension dominated multiphase flows using the VOF approach *6th European Conference on Computational Fluid Dynamics* (2014).
- [7] Brackbill J. U., Kothe D. B., Zemach C.: A continuum method for modeling surface tension *J. Comp. Phys.*, vol. 100, no. 2 (1992) pp. 335-354.
- [8] Damián S.M.: Description and utilization of interFoam multiphase solver *Comp.Fluid Dyn.* (2012).