DOI: https://doi.org/10.14311/TPFM.2025.007

VORTICES INSIDE A VENTILATED CAVITY

N. F. Dimitrieva

Institute of Hydromechanics of the National Academy of Sciences of Ukraine, 8/4 Marii Kapnist St., 03057 Kyiv, Ukraine

Abstract

This study is devoted to the study of complex vortex flows inside the cavity and the influence of injection parameters on the restructuring of the vortex structure. The calculations are performed using the OpenFoam software. The interFoam model is used for two incompressible isothermal media without phase transition. The mesh is constructed by the stepwise cell refinement method using the snappyHexMesh utility. A complex vortex structure inside a ventilated cavity is described. Three zones are distinguished: constant pressure, viscous diffusion and two-phase mixing layer. The dimensions of the cavity parts depend on the velocity of the main fluid flow and the flow rate of the injected gas. Good agreement with theoretical and experimental data was obtained.

Keywords: two-phase flow, cavity, vortex, computational fluid dynamics, volume of fluid,

1 Introduction

The problem of supercavitation as a method of flow control has been known since the middle of the 20th century [1]. Supercavitation is essentially the process of formation of a single gas bubble (cavity) behind a streamlined object. 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. If a body is placed behind the cavitator, a gas layer is created between the solid surface and the liquid. Since the density and viscosity in such a gas layer are significantly lower, a drag reduction can be achieved.

Natural cavitation is difficult to control. Therefore, to create and maintain a stable cavity, air is injected into the vapor-containing cavity.

There are a lot of investigations in field of supercavitation [2, 3]. Logvinovich's team made a significant contribution to the theoretical and experimental study of supercavitation [3-5]. The integrated approach allowed both to develop the experimental methodology and to develop analytical and empirical methods for calculating the supercavity in a real liquid.

However, the internal structure of the flows inside the cavity, which determines its shape, size and stability in the flow, remains poorly understood. This study is devoted to the study of complex vortex flows inside the cavity and the influence of injection parameters on the restructuring of the vortex structure.

Today, there are many approaches to modeling two-phase flows. However, this remains one of the difficult problems in computational fluid dynamics. A complex multiphase system of equations, taking into account external forces and interphase transitions requires accurate solution methods and significant computing power.

This study proposes to use the volume of fluid (VOF) approach. Early calculations of two-phase fluids for various applications have shown that VOF provides a simple and cost-effective method for detecting free surfaces in 3D meshes [6-8].

2 Preliminary Research

Test calculations of the formation of a vapor cavity in a moving liquid have been carried out using a InterPhaseChangeFoam solver. Visualization of the interphase surface of the cavity is shown in Fig. 1, a).

The cavity has a finite size. It depends on the flow rate at a fixed cavitator size. The cavity particles are not carried away in the wake. The vapor cavity exists exclusively in the low-pressure zone according to the phase diagram of water. A classic closed system of vortices is observed inside the cavity in the wake of the body (Fig. 1, b). The reverse flow limits the length of the cavity.

A series of calculations showed that natural (water vapor) supercavitation is observed at a flow velocity > 15 m/s for a cavitator with a diameter of 5 mm.

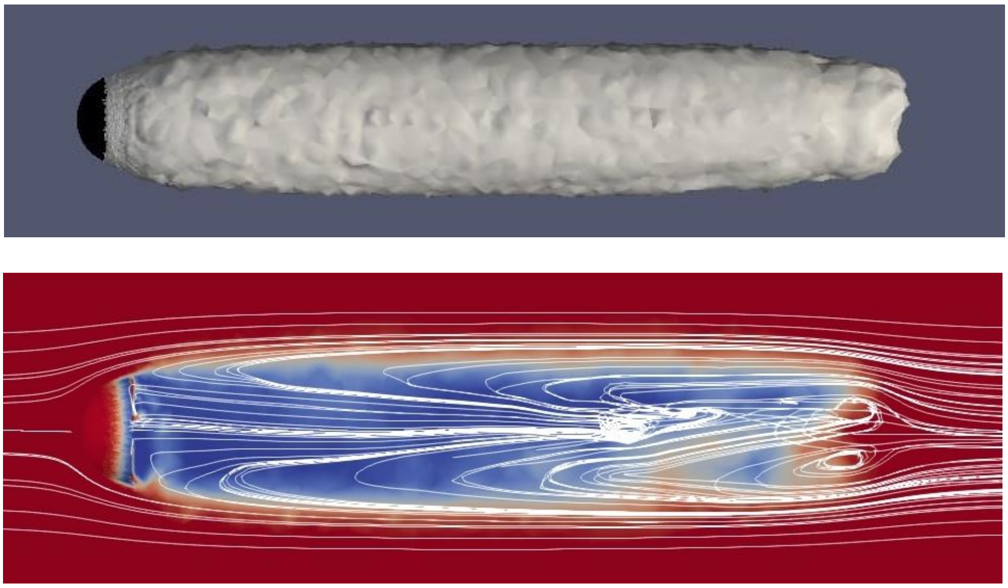

a)
b)

Figure 1: Vapor cavity at U_w = 20 m/s: a) visualization of the interfacial surface; b) streamlines in central section

At a low flow velocity, the effect of liquid phase evaporation can be neglected. Also, early studies of the ventilated cavity found that the contribution of compressibility and the turbulent model to the equation set was negligible for the range of parameters studied [7]. Adding additional equations and turbulence models complicated the calculations but had no effect on the results.

The study of a forced ventilated cavity without phase transition in an incompressible fluid is presented below.

3 Mathematical Modelling

A two-phase immiscible medium water-air is studied.

Volume of Fluid (VOF) method is used to solve the two-phase flow problem [6]. 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:

$$\frac{\partial u_j}{\partial x_j} = 0, \tag{1}$$

$$\frac{\partial (u_i)}{\partial t} + u_j \frac{\partial (u_i)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \upsilon \frac{\partial^2 u_i}{\partial x_j^2} + f_{\sigma i}, \tag{2}$$

$$\frac{\partial \alpha}{\partial t} + u_j \frac{\partial \alpha}{\partial x_j} = 0, \tag{3}$$

where i, j = 1, 2, 3, u_i is component of the velocity field, p is the pressure, υ is the kinematic viscosity coefficient, t is time, density ρ is calculated as weighted averages based on the fraction distribution of the fluid, i.e.

$$\rho = \alpha \rho_w + (1-\alpha)\rho_a, \tag{4}$$

Here, u_i is component of the velocity field, ρ is density, ρ_w is water density, ρ_a is air density. Surface tension $f_{\sigma i}$ is modeled as the surface force [9], which is calculated as follows:

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

where σ is the surface tension constant, κ is the curvature [10]. 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.

The phase fraction α is defined at all mesh points in following way: α = 1 if the cell location is occupied by fluid and function α = 0 if there is an air. Cells with a value of α different from 0 to 1 have a free interfacial surface. 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 sensitive to the mesh resolution.

4 Numerical Modeling

Numerical modeling is carried out within the framework of the open-source software OpenFOAM by the finite volume method [9]. 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.

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.

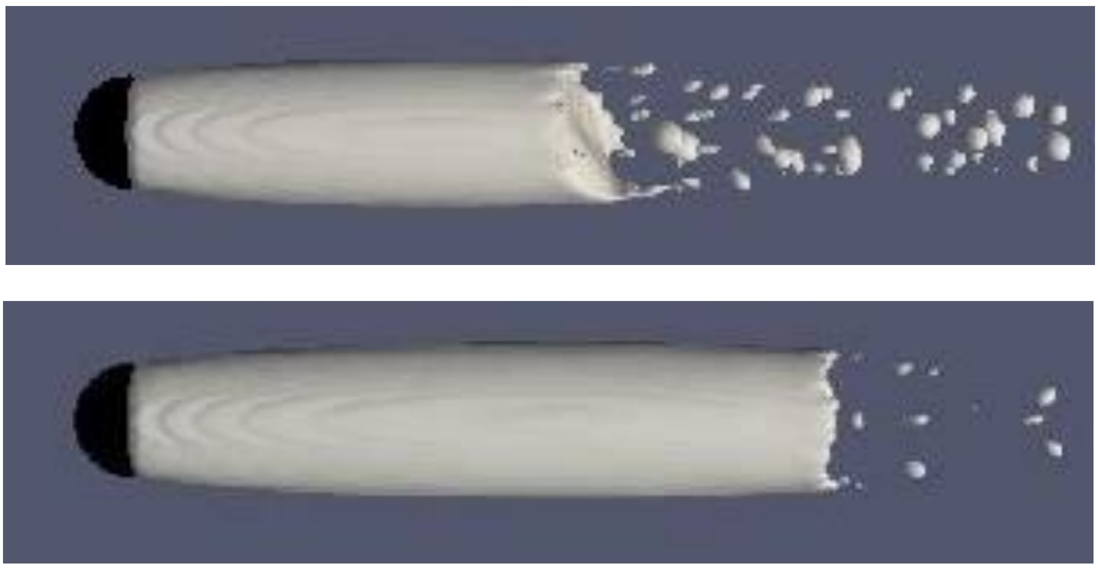
As a result, the mesh is constructed that takes into account small-scale flow elements in the interfacial zone and near the streamlined body [11]. The 3D mesh consists of about 2.5 million cells, 98 % of which are hexaelements. The minimum cell has a volume of the order of 10^{-5} mm^3.

The constant fluid velocity U_w 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 surface tension are included in the equations. The outlet pressure is set to zero.

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 hole with an area of $S = 4 \cdot 10^{-6}$ m^2 is defined on the rear flat wall of the hemisphere. The absence of a liquid phase $\alpha = 0$ and a constant velocity U_a are prescribed within this zone. Using setFields utility, we didn't need to rebuild the mesh when changing the hole parameters.

5 Calculation Results

Numerical results showed that a stable air cavity is formed in the wake of the body (Fig.2). The calculation results confirmed that the stability of the cavity shape depends on the velocity of the surrounding liquid and the air flow rate.

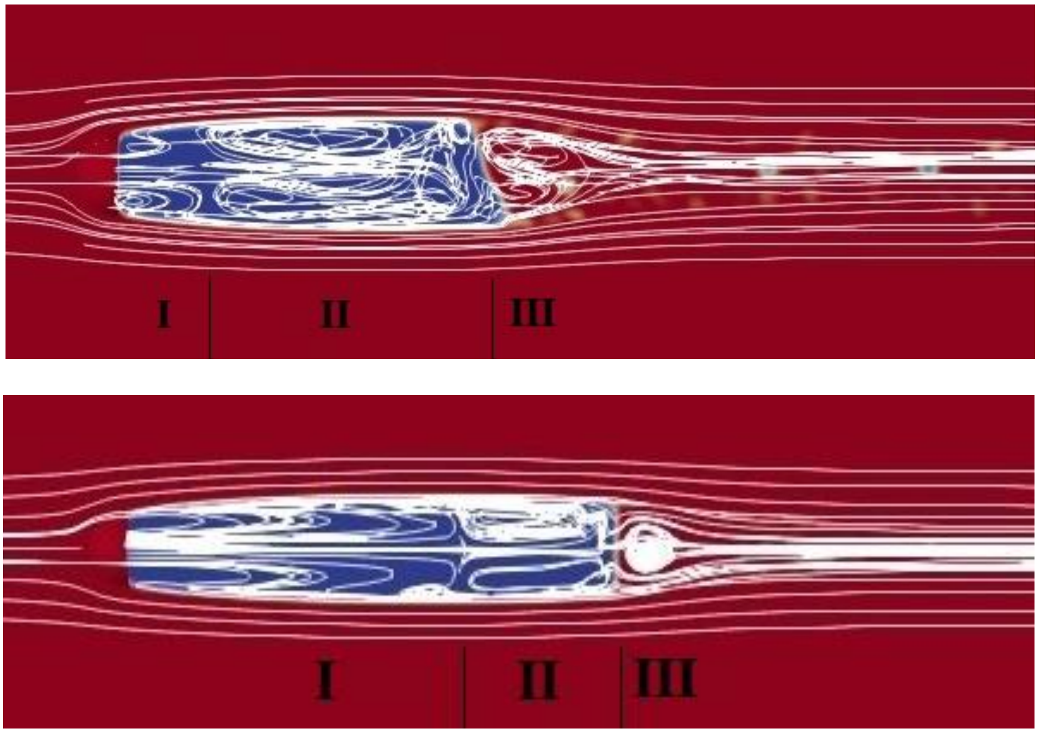


a)

b)

Figure 2: Visualization of the air cavity. Isosurfaces of the phase fraction α:
a) U_w = 1 m/s, U_a = 1 m/s; b) U_w = 1 m/s, U_a = 2 m/s

Good qualitative agreement with experimental data was obtained [3, 11]. Unlike the vapor cavity (Fig. 1, a), we can see a bubble trail behind the stable part. The proportion of gas in the studied volume does not change because the gas is blown in forcibly and there is no phase transition from gas to liquid.
A more detailed study of the internal structure of the flows inside the cavity revealed a complex vortex set. Fig. 3 shows the instantaneous vortex structure in the central longitudinal section. The air phase is shown in blue, water in red, and the streamlines in white.



a)
b)

Fig. 3. Central section: air is in blue, water is in red, streamlines are in white
a) U_w = 1 m/s, U_a = 1 m/s; b) U_w = 1 m/s, U_a = 2 m/s

The developed cavity of high elongation can be conditionally divided into three zones. The first zone is characterized by constant pressure. It is quite accurately determined by the limit of the molecular layer of the air-water interface and a thin layer of viscous attraction of the cavity gas. The second zone is characterized by a layer of viscous entrainment, which is filled with small water droplets due to viscous diffusion, dynamic interaction at the interface of the two media, and incoming air from the cavitator to maintain constant pressure. The third zone is characterized by a two-phase mixing layer reaching the axis of the cavity. It closes the cavity and forms a two-phase long trace behind the cavity.

This is in good agreement with earlier theoretical studies [12].

Note that without forced gas injection the first zone is absent as we can see in Fig. 1 for the case of natural supercavitation. In addition, the third zone of two-phase mixing is absent in the steam cavity. The reverse phase transition from water vapor to water prevents the entrainment of gas in the wake.

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 air flow rate has a positive effect on the length and stability of the resulting cavity.

Fig. 4 shows the pressure distribution along the longitudinal axis at different values of the air blowing velocity U_a. The calculation results show that in the first zone of the cavity there is a uniform pressure. The length of this zone depends mainly on the gas flow rate. Further downstream, the pressure should have a discontinuous structure in the zones where the bubbles are located.

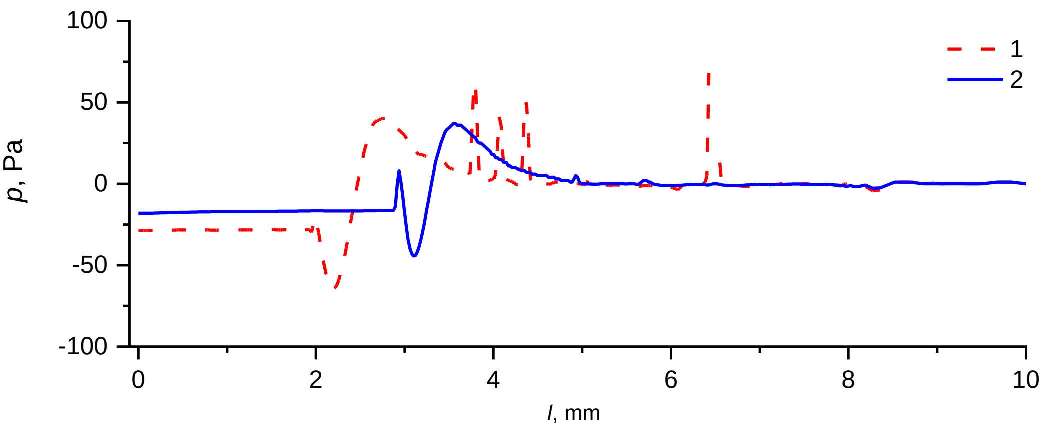


Fig. 4: Longitudinal axis pressure distribution: 1 – U_w = 1 m/s, U_a = 1 m/s; 2 – U_w = 1 m/s, U_a = 2 m/s

The longitudinal velocity component in the central section is shown in Fig. 5. It also illustrates the internal structure of the ventilated air cavity. We can see three clearly defined zones indicated in Fig. 3.

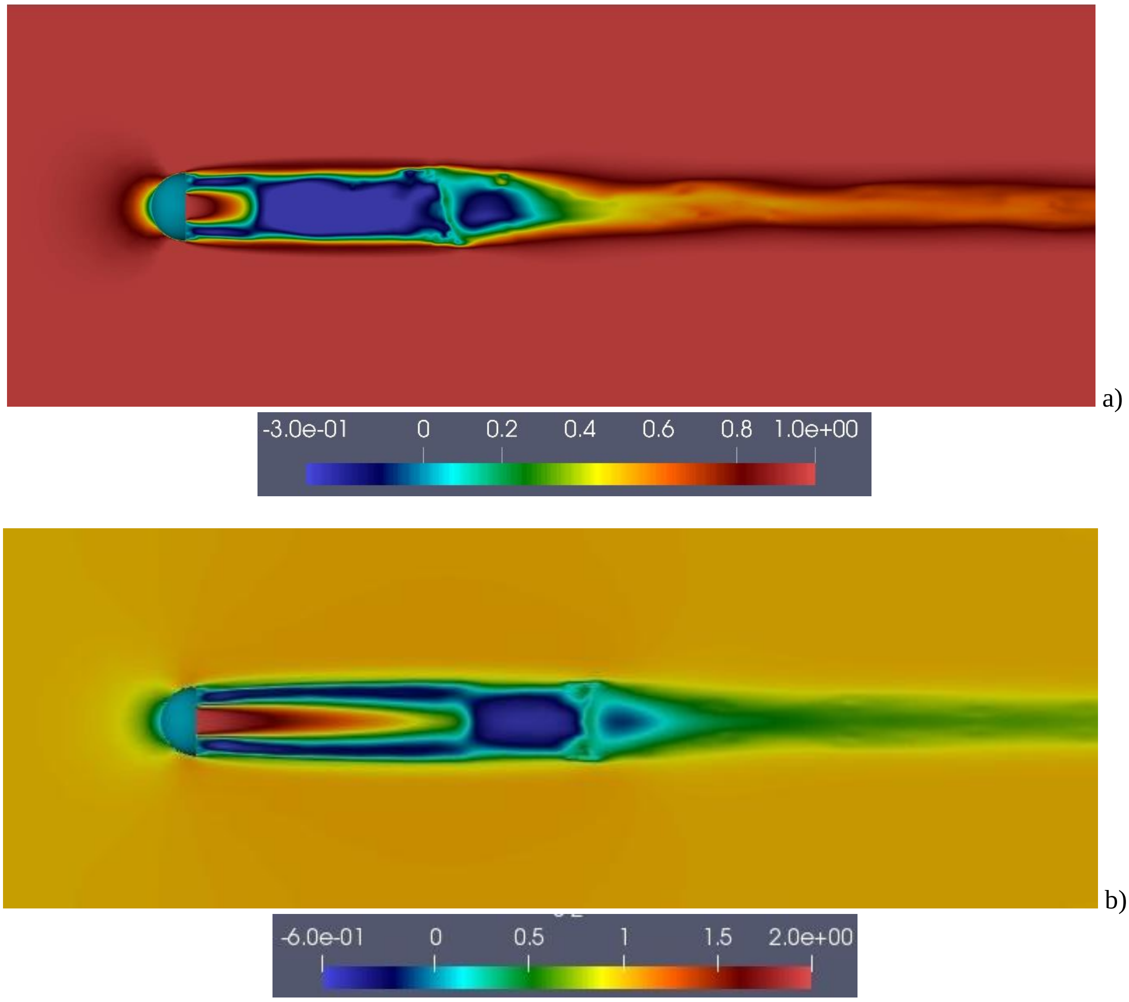


Figure 5: Longitudinal component of velocity in the central section:
a) U_w = 1 m/s, U_a = 1 m/s; b) U_w = 1 m/s, U_a = 2 m/s

At constant pressure, part of the air volume of the cavity is involved in the movement of the surrounding liquid and is carried away by the flow to the rear part of the cavity. There, a turning jet is formed along the axis of the cavity, forming a circulation vortex zone. This structure of gas flow in the cavity is possible when the thickness of the internal boundary layer is significantly less than the radius of the cavity and there is no mass exchange through the phase boundary surface.

6 Conclusions

The calculations of air cavity formation in a moving liquid are performed using the open source software OpenFoam. The interFoam model is used for two incompressible isothermal media without phase transition. The mesh is constructed by the stepwise cell refinement method using the snappyHexMesh utility for generating a three-dimensional unstructured grid with a predominance of hexahedral elements. It takes into account small-scale flow structures in the phase transition zone and near the streamlined body.

A complex vortex structure inside a ventilated cavity is described. Three zones are distinguished: constant pressure, viscous diffusion and two-phase mixing layer. The dimensions of the cavity parts depend on the velocity of the main fluid flow and the flow rate of the injected gas.
Good agreement with theoretical and experimental data was obtained.

References

[1] Tulin, M. P.: Supercavitating flows. Small perturbation theory. *Journal of Ship Research*, vol. 8 (1964) pp. 16–37.

[2] Franc, J.-P. & Michel, J.-M.: Supercavitation. *In: Fundamentals of Cavitation. Fluid Mechanics and Its Applications*, Springer, Dordrecht, vol. 76: (2005) pp. 97-130.

[3] Supercavitation: Advances and Perspectives A collection dedicated to the 70th jubilee of Yu.N. Savchenko. Ed. I.Nesteruk. Springer-Verlag, Berlin and Heidelberg, (2012) 230 p.

[4] Logvinovich, G. V. & Serebryakov, V. V.: The methods for the calculation of the shape of slender axisymmetric cavities. *Hydromechanics*, issue 32 (1975) pp. 47-54.

[5] Savchenko, Yu. & Semenenko, V. & Savchenko G.: Peculiarities of supercavitating vehicles' maneuvering. *International Journal of Fluid Mechanics Research*, vol. 46, no. 4: (2019) pp. 309-323.

[6] Hirt, C. W. & Nichols, B. D.: Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries, *Journal of Computational Physics*, vol. 39, no. 1 (1981) pp. 201-225.

[7] Koval, S. O. & Dimitrieva, N. F. & Voropaiev, G. O.: Calculation of a Ventilated Cavity in OpenFOAM. *In Proc. Topical Problems of Fluid Mechanics, ed. D.Šimurda and T.Bodnár,* Prague: (2023) pp. 88-95.

[8] Koval, S. O. & Dimitrieva, N. F.: Two-Phase 3D Modeling of an Airlift Pump in Openfoam. *In Proc. Topical Problems of Fluid Mechanics, ed. D.Šimurda and T.Bodnár,* Prague: (2024) pp. 101-108.

[9] 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.

[10] Brackbill, J. U. & Kothe, D. B. & Zemach, C.: A continuum method for modeling surface tension *Journal of Computational Physics*, vol. 100, no. 2 (1992) pp. 335-354.

[11] Voropaiev, G. O. & Korobov V. I. & Dimitrieva N. F.: Modeling of a ventilated cavity behind a streamlined body. *Journal of Computational and Applied Mathematics*, vol. 135, no. 1: (2021) pp. 73–79.

[12] Voropaiev, G. O.: Viscous entrainment of gas in a ventilated cavity of the prescribed shape. *Applied Hydromechanics*, vol. 15, no. 1: (2013) pp. 10-23.