Numerical simulation of the process of gas outflow into an open pipe with an obstacle filled with a liquid (water, lead)

I S Vozhakov and S I Lezhnin
Kutateladze Institute of Thermophysics SB RAS, Novosibirsk, 630090, 1 Ac. Lavrentyev Ave., Russia
E-mail: vozhakov@gmail.com

Abstract. Submerged gas jets find a wide variety of industrial applications, and their behavior is characterized by the ratio of inertia to buoyancy and can vary from the emergence of individual bubbles to stable jets. A numerical study of the high-speed outflow of gas under a pressure of 18 MPa into a cavity with an obstacle filled with a liquid under a pressure of 2 MPa is carried out. The simulation is performed using the VOF method in conjunction with the k-ε turbulence model. The calculations are realized for three distances between the outflow hole and the obstacle: 100, 200, and 300 mm. Principal scenarios of gas jet evolution and characteristic expiration times are obtained.

1. Introduction
The submerged gas jet is used in various technological processes in metallurgy, chemical and food industries. In addition, the plume is generated by a subsea release due to a variety of different causes, including drilling, bursting pipelines, failure of subsea equipment, and gas leaks from the ocean floor. In the nuclear industry, this phenomenon is inextricably linked with the safety of power equipment using a liquid metal coolant.

Gas jets submerged in water are characterized by a ratio of inertia to buoyancy and exhibit a variety of behaviors, from bubbling plume to steady jets. At low flow rates, there is a bubbling regime during which bubbles are formed near the hole. Rayleigh was one of the first who studied their behaviors [1]. The formation of bubbles was found to generate a sound spectrum that can be used to determine their size [2].

At higher flow rates, a gas jet is formed and it remains stable under certain conditions. In this mode, the formation of bubbles occurs only far from the hole. Mori et al [3] were among the first to experimentally investigate the efflux of nitrogen into mercury and to quantitatively describe the point of the jet decay into bubbles. Recent works by Weiland [4] and Harby et al. [5] experimentally studied the flow structure of a submerged high-speed gas jet.

In [6], a numerical study of the unsteady air flow into water with a flow rate from 58 to 108 m/s was carried out. The simulation was performed using the VOF method, supplemented by the RANS turbulence model. As a result, an idea was obtained about the relationship between the flow structure and the acoustic field.

In [7], both experimental and numerical studies of gas jets flowing into water under a pressure of 1-3 MPa were carried out. The jet interface was found to exhibit continuous vibrations and strong mixing in the conical regions. This led to a violation of the interface and the formation of a large number of small gas bubbles, intensifying mixing.
Previously, the authors carried out a numerical simulation of gas outflow from a high-pressure vessel into an open tube region [8]. It was shown that for liquids with different densities (water and liquid lead), different scenarios for the development of the gas outflow were realized. The problem statement in the form of a gas outflow into a free tube region was only an initial approximation, which is rarely realized in real technical applications. The geometry of the working sections of technical devices filled with liquid also contained structural elements (pipes, spacer grids, bushings, racks). The presence of structural elements can significantly change the very process of gas outflow into a liquid, while it is important to know the force effect on the structural elements.

The aim of this work is to study the injection of gas into a liquid (water, lead) in an open pipe region with an obstacle inside it.

![Figure 1](image)

**Figure 1.** Computational domain: 1 – receiver with high pressure air, 2 – injector, 3 – working volume, \( W \) – wall type condition, \( P_i \) – the condition of inflow-outflow at constant pressure, \( P_o \) – the condition of inflow-outflow at constant pressure, \( ax \) – axis symmetry condition, \( h \) – the liquid level, \( H \) – height of the working volume, \( r \) – radius of the gas volume, \( L \) – length of the gas volume, \( r_i \) – radius of the injector, \( L_i \) – length of injector, \( r_d \) – radius of disk, \( L_d \) – width of disk, \( h_d \) - installation level of disk.

2. **Methods**

To simulate the outflow of air into a pipe filled with liquid (liquid lead, water), we use the compressible InterFoam solver of the OpenFOAM package based on solving the Navier-Stokes equations for a compressible medium supplemented by the k-\( \varepsilon \) turbulence model, and track the interface using the VOF method. The computational domain is built on an axisymmetric geometry: the inner diameter of the pipe region is 0.2 m, and the length is 0.5 m. The liquid level is set at a height of 0.4 m. The remaining space is filled with air. On the right surface of the computational domain, the boundary condition of free outflow is set. The initial pressure in the working area is \( 2 \times 10^5 \) Pa. Gas outflow into the working area occurs through a nozzle with a diameter of 0.02 m. The initial air pressure in the cavity, from which air is supplied, is \( 180 \times 10^5 \) Pa. The initial temperature of air and liquid lead is 650 K and that of water is 300 K. The simulation is carried out according to three different scenarios, in which a disk barrier with a diameter of 0.1 m and a thickness of 0.01 m is installed at different distances from the hole: 100, 200 and 300 mm. A detailed description of the calculation methodology is presented in previous works [8, 9].

3. **Results**

Figures 2, 3 and 4 show the volume fractions of the liquid and the velocity modulus for air outflow in lead with an obstacle at a distance of 100 mm, 200 mm and 300 mm, respectively. The outflow into lead is characterized by a transonic gas velocity in the center of the jet and low velocities near the interface because of high lead density. As a result, the evolution of the gas bubble in the area of the
obstacle changes little as the obstacle moves away from the holes. In the region before the obstacle, the evolution of the bubble differs more strongly. Thus, at a small distance to the obstacle, there is a complex structure of flows, which leads to a strong instability of the interface and the formation of transverse jets and bubbles. With an increase in the distance to the obstacle, the amplitude of instability decreases, and the growth of the bubble occurs more smoothly.

![Figure 2](image1.png)

**Figure 2.** Distribution of the volumetric liquid content (a) and the velocity modulus (b) at different points in time when air flows into lead with an obstacle at a distance of 100 mm.

With an increase in the distance to the obstacle, the gas-dynamic structure inside the gas bubble changes significantly. So, at a distance of 100 mm, the flow is blocked, and the gas-dynamic structures forming inside the bubble are unstable. The maximum gas velocity does not exceed 790 m/s. With an increase in the distance to the obstacle to 200 mm, stable gas-dynamic structures with a maximum gas velocity of 810 m/s are formed inside the bubble. Gas-dynamic structures are also stable at a distance of 300 mm to the obstacle. The maximum velocity increases to 840 m/s. It can be seen that the displacement of liquid lead occurs faster at a greater distance to the target. At a small distance (100 mm), a film of liquid lead flows down from the walls and the gas jet is blocked.

![Figure 3](image2.png)

**Figure 3.** Distribution of the volumetric liquid content (a) and the velocity modulus (b) at different points in time when air flows into lead with an obstacle at a distance of 200 mm.

![Figure 4](image3.png)

**Figure 4.** Distribution of the volumetric liquid content (a) and the velocity modulus (b) at different points in time when air flows into lead with an obstacle at a distance of 300 mm.
Figures 5, 6 and 7 show the distribution of the volume fraction of the liquid and the velocity modulus when air flows into water with an obstacle at a distance of 100 mm, 200 mm and 300 mm, respectively. In this case, the lower inertia of the water leads to a faster growth of the bubble, the formation of vortex flows, and the formation of drops and bubbles. An increase in the distance to the disk obstacle leads to a change in the gas outflow regime from jet to jet-slug.

Unlike liquid lead, gas-dynamic structures upon gas injection into water are not stable at any distance to the obstacle. The maximum gas velocity corresponds to 920, 910, and 950 m/s for distances to the disc of 100, 200, and 300 mm, respectively. For liquid lead, the displacement of the liquid by the gas occurs in the form of a projectile, which occupies almost the entire volume section up to the obstacle. In water, the displacement of liquid by gas occurs in the form of a jet. Therefore, a large amount of liquid remains in the volume between the disk and the injector, which blocks the gas jet from the unstable interface.
Conclusions
A numerical study of the high-speed outflow of an air jet in liquids with different densities (water, lead) at different distances from the obstacle to the hole has been carried out. At the smallest distance to the obstacle (100 mm), a strong reverse gas flow is observed in the region up to the obstacle, which leads to the instability of the jet and the formation of additional bubbles, as in the cases of lead and water. For lead at a distance of 100 mm, the gas-dynamic structures formed inside the bubble are unstable; the maximum outflow velocity is 790 m/s. At distances of 200 and 300 mm, stable gas-dynamic structures with a higher gas velocity of up to 840 m/s are formed inside the bubble. When the obstacle is removed from the wall, stabilization of the outflowing jets is observed. So, for lead, this results in a smoother growth of a single projectile, and for water, in a shift in the spray area of the jet. Gas-dynamic structures during gas injection into water are unstable at any distance from the obstacle. The maximum gas velocity corresponds to 920, 910, and 950 m/s for distances to the disc of 100, 200, and 300 mm, respectively. At the maximum distance (300 mm) from the obstacle in the case of lead, bubble growth appears to be stable; however, for water, due to its low inertia, a complex flow structure is observed both in the region in front of the obstacle and after it.

Acknowledgments
The work was performed under the state contract with the IT SB RAS (121032200034-4).

References
[1] Rayleigh L 1917 Philos. Mag. 34 94–8
[2] Manasseh R, Lafontaine R F, Davy J, Shepherd I and Zhu Y G 2001 Exp. Fluids 30 672–82
[3] Mori K, Ozawa Y and Sano M 1982 Trans. of the Iron and Steel Inst. of Japan 22 377
[4] Chris W and Pavlos P V 2013 Int. J. Multiphase Flow 48 46–57
[5] Harby K, Chiva S and Munoz-Cobo J L 2014 Exp. Thermal Fluid Science 53 26–39
[6] Liu J T, Qin S J, Miao T C and Wu D Z 2015 IOP CS: Material Science and Engin. 72(4) 042038
[7] Zhang X, Li S, Yang B and Wang N 2020 Ocean Engin. 213 107611
[8] Alekseev M V, Vozhakov I S and Lezhnin S I 2019 Thermoph. and Aeromech. 26(5) 781–4
[9] Alekseev M V, Vozhakov I S and Lezhnin S I 2019 Multiphase Systems 14(2) 82–8