Numerical simulation and experimental research of cavitation nozzle based on equation curve
Lifu Wang, Dongyan Shi, Zhixun Yang, Guangliang Li, Chunlong Ma, Dongze He and Liang Yan

ABSTRACT

To further investigate and improve the cleaning ability of the cavitation nozzle, this paper proposes a new model that is based on the Helmholtz nozzle and with the quadratic equation curve as the outer contour of the cavitation chamber. First, the numerical simulation of the flow field in the nozzle chamber was conducted using FLUENT software to analyze and compare the impact of the curve parameters and Reynolds number on the cleaning effect. Next, the flow field was captured by a high-speed camera in order to study the cavitation cycle and evolution process. Then, experiments were performed to compare the cleaning effect of the new nozzle with that of the Helmholtz nozzle. The study results demonstrate that effective cavitation does not occur when the diameter of the cavitation chamber is too large. For the new nozzle, with the increase of the Reynolds number, the degree of cavitation in the chamber first increases and then decreases; the cleaning effect is much better than that of a traditional Helmholtz nozzle under the same conditions; the nozzle has the best cleaning effect for the stand-off distance of 300 mm.

Key words | cavitation jet, cavitation model, cavitation nozzle, impact performance, numerical simulation

HIGHLIGHTS

● A new type of cavitation nozzle based on the one-dimensional quadratic equation curve.
● Through the orthogonalization experiment the optimal solution was obtained.
● Explore the cavitation period and the change of bubble diameter and liquid volume fraction.
● Explore the cleaning target distance and cleaning effect.
● Verify the performance of new nozzles and traditional nozzles through experiments.

INTRODUCTION

Cavitation occurs when local pressure in the liquid is less than its saturated vapor pressure. It is a process that involves the generation, development, and collapse of cavitation inside the liquid (Knapprt et al. 1970). Cavitation also commonly occurs in nature; for instance, some creatures even hunt prey by cavitation. Versluis et al. (2000), (Lohse et al. 2003) studied the predation of pistol shrimp. They found that when the large claw of pistol shrimp closes at a high speed of up to 100 km/h, cavitation bubbles are formed in front of the large claw, and the high pressure of the cavitation bubbles is used to hunt prey. Meanwhile, the cavitation bubble releases a significant amount of
energy at the moment of collapse, which strengthens the liquid flow and increases energy efficiency. Therefore, the cavitation jet has been applied in many industrial fields such as rock breaking and cleaning. Yuan et al. (2020) proposed a composite nozzle combining a Venturi tube structure and a Helmholtz resonant cavity and compared it with a traditional Helmholtz nozzle. Duret et al. (2018) studied the impact of the cavitation in turbulent two-phase flow on the hydrodynamics using the Coupled Level Set and Volume of Fluid method (CLSVOF) for interface capturing. Based on the interface capturing method, Pivello et al. (2014) proposed a new computational method for simulating 3-dimentional (3D) two-phase flows, which can greatly reduce the difficulty of handling 3D interfaces.

Cai et al. (Cai et al. 2020) studied the Strouhal number by varying the nozzle structure parameters; the Strouhal numbers showed qualitatively similar behaviors. Their overall trend is rising rapidly with a relatively small angle, declining after the surge, and then increasing slightly. They also inferred that self-resonance would not occur if the straight line is too long. Fang et al. Fang et al. (2019) conducted numerical simulation and experimental research to further examine the performance of the Helmholtz self-oscillating water jet. They compared the morphological characteristics of the erosion surfaces of the cone nozzle and the Helmholtz nozzle and obtained the evolution law of cavitation. Liu et al. (Liu et al. 2017) evaluated the erosion performance of the 120° impact edge Helmholtz nozzle based on previous studies and analyzed the vibration mechanism through numerical simulations and experiments. Wu et al. (Wu et al. 2019) captured the details of the cavitation flow under different nozzle structures and working pressures using high-speed photography. By quantitatively evaluating the cavitation jet generated by the Helmholtz nozzle, they discovered that the cavitation intensity produced by a Helmholtz nozzle is significantly higher than that by a traditional conical nozzle, and the geometry of the nozzle has a great influence on the length of the cavitation cloud and the shedding period. Lee et al. (2018) studied the impact of the orifice inlet geometry on the flow rate and cavitation characteristics of high-temperature hydrocarbon liquid jets and correlated the flow characteristics with the Reynolds number and the cavitation number in a macroscopic view. Alehossein et al. (Alehossein & Qin 2007) simulated the formation and collapse of cavitation bubbles in the jet by solving the Rayleigh-Plesset equation and concluded that cavitation bubbles have a greater impact on the cavitation jet. Li (2014) summarized the previous researches on cavitation jets, introduced the mechanism of cavitation, measurement methods, and factors affecting cavitation, discussed the rationality of commonly used cavitation numbers, and explained the reason that cavitation jets have greater rock breaking ability than ordinary jets. Sun et al. (Sun et al. 2019) studied the atomization of high-pressure fuel injector nozzles; they used high-pressure fuel injector nozzles to atomize liquid and generate spray in the combustion chamber, finding that the internal flow of the nozzle, especially the cavitation, plays an important role in promoting the breakup of the liquid jet.

Akira et al. (2014) used the Eulerian-Lagrangian approach for bubble tracking to further study cavitation in the jet of diesel engine liquid injection, excellently reproduced the morphological characteristics when the cavitation collapsed, and quantitatively predicted the length and thickness of the cavitation area. Kozlova et al. (2019) studied the self-excited oscillation of cavitation caused by the liquid flow in the double resistance pipe. Based on their study, the natural frequency of the resonant cavity mainly depends on the nature of the cavity and the conditions of outflow into the atmosphere, and the generation of different self-excited oscillation modes is directly related to the wave number formed along the length of the cavity. Giussania et al. (2020) introduced the development of a single-fluid solver, which can accurately capture the evolution of the three fluids and the approximate volume of fluid (VOF). Piscaglia et al. (2019) developed a dynamic two-phase VOF solver to study the physical characteristics of the primary jet breaking and flow transient caused by the nozzle geometry during the opening of the high-pressure ejector; they discussed the applicability constraints of the two-phase solver cavitation model in the simulation of flow inside the ejector nozzle. Peng et al. (Peng et al. 2020) added quartz sand particles to the underwater cavitation jet to enhance its cavitation strength and erosion ability and verified the enhancement through high-speed photography, cavitation noise measurement and erosion tests. The research of Yuan et al. (Yuan & Schnerr 2003) indicated
that since cavitation is highly sensitive to the imposed boundary conditions, the simulations that limit internal problems are qualitatively and quantitatively incorrect, and cannot reveal the principles behind phenomena such as hydraulic flipping and super-cavitation. Ma et al. (Ma et al. 2019) analyzed the oscillation characteristics of the self-oscillating water jets produced by a series of Helmholtz nozzles with different structures through spectrum analysis, and studied the cavitation effect by analyzing the noise power spectrum. According to their study, the self-oscillating water jets produced by Helmholtz nozzles have high-frequency pressure oscillations and strong cavitation effects compared to those produced by organ tube nozzles or cone nozzles. To evaluate the self-excited oscillation intensity, mass loss and surface morphology of the eroded sample of the water jet, Liu et al. (Liu & Ma 2020) conducted erosion experiments with inclination angles of $a = 0^\circ$, $5^\circ$, $15^\circ$ and $30^\circ$. Ahmed et al. (2020) proposed and verified a numerical framework based on interface capture to study cavitation and external jet formation; they performed numerical simulations on the development of cavitation and super-cavitation and qualitatively compared the liquid and vapor structures obtained in the experiments and simulations. Belkacem et al. (Belkacem & Huang 2017) studied the impact of different aspect ratios on nozzle jet atomization characteristics, using high-speed cameras to record spray patterns under various conditions and discussed the influence of the aspect ratio on cavitation and spray structure. Li et al. (Li et al. 2017) used mathematical software to solve the original Rayleigh-Plesset equations with different dynamic viscosities and obtained the approximate expression of bubble radius over time through curve fitting.

Helmholtz and organ pipe nozzles are widely applied to clean oil tanks, ships, etc. Their cleaning effects (including cleaning area and cleaning speed) directly impact the labor hours, work intensity, and operational expense (Knappt et al. 1970). Research found that the cleaning effect is associated with the standoff distance and the degree of cavitation inside the nozzle, and the cavitation plays a key role in cleaning object surfaces. The occurrence of cavitation mainly depends on the nozzle shape and structural parameters (Shen & Zeng 1986; Li et al. 2003). Many studies have been conducted to obtain the influence of structural parameters on the cavitation jet using methods combining theory and experiment. However, few studies have been performed to enhance the cavitation effect by improving the shape of the traditional nozzle. This paper simulates the traditional Helmholtz nozzle using FLUENT software. By varying the geometry of the nozzle chamber, the impacts of nozzle structural parameters on the cavitation effect were analyzed. A holistic experimental system was established to verify the impacts and dynamically capture the cavitation cycle of the nozzle chamber using a high-speed camera.

**MATHEMATICAL MODEL**

**Governing equation**

Due to the cavitation, the model to calculate the flow field in the nozzle often employs the multi-phase cavitation model. The continuity equation and the momentum equation of the mixed phase are established based on multiphase flow calculation model:

$$\frac{\partial \rho}{\partial t} + \nabla (\rho \mathbf{u}) = 0$$  

(1)

$$\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \nabla \tau + \int_{\delta(x)} \sigma \mathbf{n} \delta(x-x') dS$$  

(2)

where $t$ is time; $\mathbf{u}$ is the velocity; $\sigma$ is the surface tension coefficient; $\rho$ is the density of the mixture; $k$ is the interface curvature; $p$ is the pressure; $\tau$ is the viscous shear stress; $n$ is the unit normal vector to the surface $S$; $\delta(x)$ is the Dirac function.

The volume fraction of each phase satisfies:

$$\left( \frac{\partial \alpha_i}{\partial t} \right) + \nabla (\mathbf{u} \alpha_i) = 0 \quad i = 1, g$$  

(3)

$$\rho = \alpha_l \rho_l + \alpha_g \rho_g$$
$$\mu = \alpha_l \mu_l + \alpha_g \mu_g$$
$$\alpha_l + \alpha_g = 1$$  

(4)

where $\alpha_i$ is the volume fraction; $\alpha_l$ is the liquid volume fraction; $\alpha_g$ is the vapor volume fraction; $\rho_l$ is the liquid density; $\rho_g$ is the vapor density; $\mu$ is the viscosity of the multiphase flow; $\mu_l$ is the liquid viscosity; $\mu_g$ is the vapor viscosity.
Turbulence model

The simulation adopts the RNG k-ε turbulence model since it can better capture small-scale vortices at the end of the shear layer (Zhang & Chen 2018; Aishvarya et al. 2020). The RNG k-ε equation as:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right) + G_k + \rho \varepsilon \tag{5}
\]

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j} \right) + \frac{C_{\varepsilon 1} k}{\varepsilon} \left( \frac{\varepsilon}{\eta} \right) - C_{\varepsilon 2} \frac{\varepsilon^2}{k} \tag{6}
\]

where \( \mu_{eff} = \mu + \rho C_p k^2 \varepsilon \) is the corrected turbulent viscosity; \( G_k \) is the turbulent kinetic energy produced by the velocity gradient; \( \alpha_k \) and \( \alpha_\varepsilon \) are turbulent Prandtl numbers; \( C_{\varepsilon 1} \) and \( C_{\varepsilon 2} \) are constants.

Cavitation model

Cavitation jet involves mass transfer with phase change. The key consideration in establishing the cavitation model is mass transfer. The establishment of cavitation mass transfer is not a separate process, in which the cavitation model needs to be used as part of the balance equation to describe the generation and fragmentation of steam. In this study, the Zwart-Gerber-Belamri cavitation model (Zwart et al. 2004) was selected (see reference 30 for the detailed derivation process) to simulate multiphase flow or material transport in multiphase flow. The model assumes that the bubbles in the liquid have the same initial size, and the mass transfer efficiency is calculated based on the number of bubbles per unit volume.

\[
R = n \left( 4 \pi R_B^2 \rho_v \frac{dR_B}{dt} \right) \tag{7}
\]

where \( n \) is the number of bubbles per unit volume; \( R_B \) is the bubble diameter; \( \rho_v \) is the vapor density; \( R \) is the mass transfer efficiency.

Ignoring the second-order term, surface tension term, and viscosity term in the Rayleigh-Plesset equation, the following can be obtained:

\[
\frac{dR_B}{dt} = \sqrt{\frac{2(p_B - p)}{3\rho_l}} \tag{8}
\]

where \( p_B \) is the surface bubble pressure; \( p \) is the partial pressure of non-condensed gas;

Then, the mass transfer efficiency expressed by the volume fraction is obtained:

\[
R = \frac{3\alpha_p}{R_B} \sqrt{\frac{2(p_B - p)}{3\rho_l}} \tag{9}
\]

The final mass transfer efficiency of bubble evaporation and condensation is expressed as

\[
R_e = C_e \frac{3\alpha_{nuc}(1 - \alpha_v)}{R_B \rho_v} \sqrt{\frac{2(p_B - p)}{3\rho_l}} \tag{10}
\]

\[
R_c = C_c \frac{3\alpha_v}{R_B \rho_v} \sqrt{\frac{2(p_B - p)}{3\rho_l}} \tag{11}
\]

where \( \alpha_{nuc} \) is the gas core volume fraction; \( C_e \) is the evaporation constant term; \( C_c \) is the condensation constant term.

Finite element solution

In the simulations of the flow field inside the nozzle chamber, the inlet pressure and outlet pressure are set to 1,101,325 Pa and 101,325 Pa, respectively. The first phase is water at room temperature, and the second is water vapor at the saturated pressure of 3,540 Pa. The computational process adopts the transient model; the discrete grid uses the first-order upwind scheme; the numerical algorithm is based on PISO (Pressure Implicit with Splitting of Operators).

NUMERICAL SIMULATION

Physical model

Referring to the existing cavitation nozzle structure and size, a new cavitation nozzle with the contour of the cavitation chamber as the one-dimensional quadratic equation \( y = ax^2 + bx + c \) was proposed. After comprehensive consideration and orthogonal parameterization selection,
the optimal parameter values were determined. Figure 1 shows the two-dimensional schematic diagram of the nozzle. \( d_0 \) is the water inlet; \( d_1 \) is the water outlet; \( l_0 \) and \( l_1 \) are the upper and lower flow channels, respectively; \( D \) is the diameter of the cavitation chamber. The specific parameters are listed in Table 1, where the value of \( D \) changes with the equation curve parameters shown in Table 2. Numerical simulations were performed for different equation curve parameters; the best equation curve parameters were obtained to determine the final contour shape of the nozzle cavity.

The mesh independence was verified prior to the numerical simulation of the cavitation. The commercial software ICEM was used for meshing the computational area. The mesh type is the unstructured mesh because of its strong adaptability. Figure 2 shows a mesh schematic. The average velocity at the nozzle outlet was selected as the monitoring object. The average velocity changes at the nozzle outlet are presented in Table 3. The simulation of free flow was conducted during the mesh independence test. The inlet condition is set as the velocity inlet, and the water velocity is 20 m/s. The outlet condition is free flow. When the number of meshes increases from 18,251 to 431,859, the average speed stays roughly constant, and the maximum error is less than 0.16%, leading to negligible impact on the simulation results. Therefore, the number of meshes was set as 18,251 in this paper considering the computing load and other factors.

Due to the large number of equation parameters and the limited space, Figure 3 only presents a part of selected results. As shown in the figure, when the chamber diameter \( D \) is too large, no obvious cavitation occurs, and the degree of cavitation in the chamber is low. When \( a = -1/18 \) and \( b = 2.0 \), the cavitation generated by the chamber profile enclosed by the equation curve under this parameter is obvious, and the degree of cavitation is high. Therefore, the equation curve \( y = -1/18x^2 + 2x + 25 \) was selected as the contour curve outside the chamber.

**Reynolds number analysis**

There are many factors that affect cavitation, such as flow rate, viscosity, absolute pressure, surface tension, and the number of gas nuclei (microbubbles, solid particles) in the water, and so on. However, the two most important factors are pressure and flow rate, which usually define the cavitation number \( \sigma \) as follows:

\[
\sigma = \frac{p_\infty - p_v}{0.5 \rho V^2_\infty}
\]  

(12)
where $p_{\infty}$ and $V_{\infty}$ respectively represent the pressure and flow velocity at a certain steady flow state point; $\rho$ is the corresponding liquid density. The critical state where tiny holes occasionally appear for the first time in the flow field is called the cavitation incapacitation, and the corresponding cavitation number is called the incipient cavitation number. The incipient cavitation number is usually defined as 

$$\sigma_i = \frac{p_{\infty} - p_{\text{min}}}{0.5\rho V_{\infty}^2}$$ 

$p_{\text{min}}$ is equal to the vaporization pressure. When the flow rate does not change and the pressure decreases, the cavitation area expands; when the pressure is constant and the flow rate increases, the cavitation can disappear. The degree of cavitation can be evaluated by the cavitation number. No cavitation occurs when $\sigma > \sigma_i$; cavitation begins to present when $\sigma = \sigma_i$; cavitation is expanding when $\sigma < \sigma_i$, and a large number of cavitation bubbles appear. Therefore, the cavitation number can be used to measure the degree of cavitation. The cavitation number is affected by the flow rate or pressure. The influence of the flow rate on the nozzle cavitation effect is usually characterized by the Reynolds number, which is expressed as:

$$Re = \frac{Ud}{v}$$

Table 3 | Results of the mesh independence

| Mesh number | $V_{\text{av}}$(m/s) | Mesh number | $V_{\text{av}}$(m/s) | Mesh number | $V_{\text{av}}$(m/s) | Mesh number | $V_{\text{av}}$(m/s) |
|------------|------------------|------------|------------------|------------|------------------|------------|------------------|
| 4969       | 10.209           | 12454      | 10.350           | 69991      | 10.406           | 190271     | 10.414           |
| 5844       | 10.256           | 18251      | 10.400           | 90467      | 10.409           | 301212     | 10.413           |
| 7169       | 10.282           | 27530      | 10.403           | 109516     | 10.411           | 353299     | 10.415           |
| 9448       | 10.320           | 49113      | 10.407           | 134120     | 10.412           | 431860     | 10.417           |

Figure 3 | Contours of liquid phase volume fraction in different structural parameters.
where \( U \) is the flow rate; \( d \) is the inlet hydraulic diameter; \( \nu \) is the fluid kinematic viscosity. Therefore, the Reynolds number is used to describe the influence of flow velocity on nozzle cavitation while the influence of liquid viscosity on primary cavitation is considered. As illustrated in Figures 4 and 5, when the Reynolds number rises to \( Re = 4.0 \times 10^5 \) from \( Re = 2.0 \times 10^5 \), the liquid phase volume fraction in the nozzle chamber first decreases and then increases; the degree of cavitation increases first and then decreases accordingly. Based on Equations (12) and (13), as the Reynolds number increases, the cavitation number decreases, and the degree of cavitation increases; however, the actual calculation result differs from the aforementioned because the generation of primary cavitation is impacted by the boundary layer, whose development is affected by both liquid viscosity and flow velocity. When the Reynolds number is small, the boundary layer has not been separated or has separated incompletely. At this point, the influence of the Reynolds number on primary cavitation follows the cavitation incapacitation theory. When the Reynolds number reaches a certain value, the boundary layer is fully separated, resulting in a large low-pressure area prone to cavitation.

Nevertheless, due to the sufficient separation of the boundary layer, even if the Reynolds number continues to increase, its impact on jet cavitation does not increase. As shown in Figure 6, when the Reynolds number rises to \( Re = 4.0 \times 10^5 \) from \( Re = 2.0 \times 10^5 \), the diameter of the cavitation bubble in the center of the chamber does not change. This is because the formation of a stable low-pressure zone in the central area leads to a stable cavitation process, in which the bubble changes regularly. Due to the influence of backflow, the boundary layer is unstable, the diameters of the bubbles near the cavity wall are different, and the trend of diameter change increases first and then decreases with the increase of Reynolds number.

**EXPERIMENT**

**Cavitation cycle**

To explore the cleaning performance of the new cavitation nozzle, an experimental system, illustrated in Figure 7, was established to conduct experiments on different cleaning
target distances. The performance was analyzed and compared with that of Helmholtz nozzles. The maximum working pressure of the pressure-boost pump is 20 MPa; the maximum flow rate is 15 L/min. The target object is cleaned using room temperature (23 °C) water for 1 minute. Based on the selected contour curve outside the chamber, 3D printing was used to manufacture the solid model of the 3D nozzle, as shown in Figure 8. The nozzle cavitation cycle was photographed by a high-speed camera, and the cavitation cycle was also numerically simulated (the mesh independence is not repeated here). The results are shown in Figures 9 and 10. It can be seen that cavitation is easily generated at the nozzle structure contraction, where the turbulent kinetic energy is also the most intense due to the occurrence of cavitation. From 0 to 40 ms within the cavitation period, the water flow collides with the downstream wall of the cavitation nozzle and starts to flow back; cavitation in the chamber begins; the turbulent kinetic energy at the sudden structure change in the nozzle chamber is gradually increasing; the main vortex structure begins to collapse and becomes a sporadic secondary vortex structure; due to the change of the vortex structure, the energy distribution is uneven. From 40 to 70 ms within the cavitation period, cavitation in the chamber continues to occur, the secondary vortex structure is reduced, and the turbulent kinetic energy change tends to be stable, indicating that the energy exchange is stabilized in this interval. In addition, cavitation occurs in both downstream flow channel and outlet; the atomization at the outlet is obviously enhanced. From 70 to 100 ms within the cavitation period, the overall cavitation of the nozzle, the energy exchange,
and the atomization at the outlet are stable; only the turbulent kinetic energy changes slightly.

Experiment

The cavitation jet is expressed by Equation (14):

\[ p_i = \frac{p_s}{6.35} e^{2/3a} \]  \hspace{1cm} (14)

where \( p_i \) is the injection pressure of the cavitation jet; \( p_s \) is the injection pressure of the continuous jet; \( a \) is the vapor content in the jet. With \( a \approx 0.17 \sim 0.1 \) and \( p_i = (8.6 \sim 124) p_s \); therefore, the cleaning effect can be significantly improved. When the jet injects the target surface, the mass fluctuates due to changes of vapor and liquid densities. As a result, the pressure on the object surface also fluctuates, which is one of the reasons that the cavitation jet presents better cleaning effect than the continuous jet. It is necessary to consider the cleaning efficiency in design.
standoff distance since it also impacts the cleaning effect of the cavitation jet.

Figure 11 presents the cleaning effect. The figure demonstrates that as the cleaning target distance increases from $d = 100\text{mm}$ to $d = 600\text{mm}$, the cleaning area gradually increases, but the depth of the etch pit gradually decreases. The water jet plays the key role in the cleaning process when the standoff distance is too small, because cavitation bubbles are not fully formed and collapse before reaching the target surface. As the nozzle-to-target distance increases, cavitation bubbles are fully developed and able to collapse on the target surface; therefore, the cleaning is mainly performed by the micro-jet and water jet generated by the collapse of cavitation bubbles. However, when the standoff distance becomes too large, the cavitation bubbles also collapse before reaching the target although they have sufficient time to be completely formed; the cleaning effect cannot be remarkably improved in this condition. As shown in Figure 11, when the target distance is 300 mm, the nozzle has the best cleaning effect. Figures 12 and 13 compare the cleaning effect of the new nozzle with that of the Helmholtz nozzle under the same conditions. It can be seen from the figures that both the cleaning area and etch pits of the new cavitation nozzle are significantly increased. Meanwhile, the pulse pressure of the target surface is increased by approximately 4%, resulting in better cleaning performance.

**CONCLUSION**

The cleaning effect of a new cavitation nozzle was studied through the numerical simulations and experiments and compared with that of the Helmholtz nozzle. Based on the study, the following conclusions are obtained:

1. A new cavitation nozzle with the equation curve $y = ax^2 + bx + c$ as the outer contour of the cavitation chamber is proposed; the different parameters were selected and optimized by numerical simulations. The best cavitation effect occurs when $a = -1/18$, $b = 2.0$, $c = 25$, and effective cavitation does not occur when the diameter of the cavitation chamber is too large.

2. Within 100 ms of a cavitation cycle, when the water flow collides with the downstream wall surface of the nozzle to produce backflow, cavitation in the cavity begins to occur, the turbulent kinetic energy in the nozzle chamber gradually increases, and the main vortex structure collapses into sporadic secondary vortex structures, leading to uneven energy distribution. Then the pulse pressure of the target surface is increased by approximately 4%, resulting in better cleaning performance.
secondary vortex structure is reduced, and the turbulent kinetic energy change tends to be stable; cavitation occurs in the downstream flow channel and the outlet, and the atomization at the outlet is obviously enhanced.

(3) When the Reynolds number increases from $Re = 2.0 \times 10^5$ to $Re = 4.0 \times 10^5$, the liquid phase volume fraction in the nozzle chamber first decreases and then increases, resulting in the degree of cavitation first increases and then decreases accordingly. As the Reynolds number increases, the diameter of cavitation bubbles in the center of the chamber do not change, but the diameter of the bubbles near the cavity wall first increases and then decreases.

(4) The experiment shows that when the cleaning target distance increases from 100 to 600 mm, the cleaning area gradually increases, but the pit depth gradually decreases. When the cleaning target distance is 300 mm, the nozzle has the best cleaning effect. Compared to the Helmholtz nozzle, the new cavitation nozzle can significantly expand the cleaning area and increase the target surface pressure by approximately 4%.

**DATA AVAILABILITY STATEMENT**

All relevant data are available from an online repository: [https://figshare.com/articles/dataset/Numerical_Simulation_and_Experimental_Research_of_Cavitation_Nozzle_Based_on_Equation_Curve/14167424](https://figshare.com/articles/dataset/Numerical_Simulation_and_Experimental_Research_of_Cavitation_Nozzle_Based_on_Equation_Curve/14167424)

**REFERENCES**

Ahmed, A., Duret, B., Reveillon, J. & Demoulin, F. X. 2020
Numerical simulation of cavitation for liquid injection in non-condensable gas. *International Journal of Multiphase Flow* **127**, 103269.
Aishvarya, K., Ghabadian, A. & Jamshid, M. N. 2020 Assessment of cavitation models for compressible flows inside a nozzle. *Fluids* 5 (3), 1–24.

Akira, S., Barrs, B. & Akio, T. 2014 Numerical simulation of incipient cavitation flow in a nozzle of fuel injector. *Computers and Fluids* 103, 42–48.

Alehossein, H. & Qin, Z. 2007 Numerical analysis of Rayleigh-Plesset equation for cavitating water jets. *International Journal for Numerical Methods in Engineering* 72 (09), 780–807.

Belkacem, A. & Huang, Y. 2017 Investigation of the effect of cavitation in nozzles with different length to diameter ratios on atomization of a liquid jet. *Journal of Thermal Science and Engineering Applications* 9 (03), 031014.

Cai, T., Liu, B., Ma, F. & Yan, P. 2020 Influence of nozzle lip geometry on the strouhal number of self-excited waterjet. *Experimental Thermal and Fluid Science* 112, 109978.

Duret, B., Canu, R., Reveillon, J. & Demoulin, F. X. 2018 A pressure based method for vaporizing compressible two-phase flows with interface capturing approach. *International Journal of Multiphase Flow* 108, 42–50.

Fang, Z., Guo, X., Tao, X. & Li, D. 2019 Impact performance of Helmholtz self-excited oscillation waterjets used for underground mining. *Applied Science-Basel* 9 (16), 3235.

Giussania, F., Piscaglia, F., Saez-Mischliche, G. & Helie, J. A. 2020 A three-phase VOF solver for the simulation of in-nozzle cavitation effects on liquid atomization. *Journal of Computational Physics* 406, 109068.

Knapp, R. T., Dally, J. W. & Hammit, F. G. 1970 *Cavitation*. McGraw-Hill, New York.

Kozlova, I. I., Ocheret’yan’ya, S. A. & Prokof’eva, V. V. 2019 Different self-oscillation modes in flows with a ventilated cavity and their possible use in forming periodic pulsing jets. *Fluid Dynamics* 54 (03), 308–318.

Lee, H. J., Choi, H., Jin, Y.-I., Hwang, K.-y., Park, D.-c. & Min, S. 2018 Effect of nozzle inlet geometry in high temperature hydrocarbon liquid jets. *International Journal of Heat and Fluid Flow* 74, 1–14.

Li, G., Shen, Z., Zhou, C., Dengbi, Z. H. & Can, Y. 2005 An experimental study on impact pressure characteristics of self-resonant cavitating jets. *Chinese Journal of Hydrodynamics(A)* 18 (5), 570–575.

Li, Z. 2014 Criteria for jet cavitation and cavitation jet drilling. *International Journal of Rock Mechanics & Mining Sciences* 71, 204–207.

Li, G.-D., Deng, S. S. & Guan, J.-F. 2017 Numerical investigation on the orifice cavitating water jet considering the fluid viscosity's effects on bubbles' growth and collapse. *Journal of the BRazillian Society of Mechanical Sciences and Engineering* 39 (12), 4973–4983.

Liu, B. & Ma, F. 2020 Erosion characteristics and the corresponding self-resonating oscillations of cavitating jet on oblique surfaces. *Energies* 13 (10), 2563.

Liu, W., Kang, Y., Zhang, M., Yongxiang, Z. & Xiaochun, W. 2017 Frequency modulation and erosion performance of a self-resonating jet. *Applied Science-Basel* 7 (09), 932.

Lohse, D., Schmitz, B. & Versluis, M. 2001 Snapping shrimp make flashing bubbles. *Nature* 413, 477–478.

Ma, W., Cai, T. & Ma, F. 2019 Experimental research on the waterjet oscillating characteristics of Helmholtz Nozzle. *Journal of Applied Science and Engineering* 22 (01), 83–92.

Peng, C., Tian, S., Li, G. & Wei, M. 2020 Enhancement of cavitation intensity and erosion ability of submerged cavitation jet by adding micro-particles. *Ocean Engineering* 209, 107516.

Piscaglia, F., Giussania, F., Montorfanoa, A., Hlie, J. & Aithal, S. M. 2019 A multiPhase dynamic-VoF solver to model primary jet atomization and cavitation inside high-pressure fuel injectors in OpenFOAM. *Acta Astronautica* 158, 375–387.

Pivello, M. R., Villar, M. M., Serfaty, R., Roma, A. M. & Silveira, N. A. 2014 A fully adaptive front tracking method for the simulation of two phase flows. *International Journal of Multiphase Flow* 58, 72–82.

Shen, Z. & Zeng, C. 1986 Experimental study on submerged self-resonating cavitating nozzles under atmospheric pressure. *Journal of Jianghan Petroleum Institute* 11 (2), 79–88.

Sun, Y., Guan, Z. & Hooman, K. 2019 Cavitation in diesel fuel injector nozzles and its influence on atomization and spray. *Chemical Engineering and Technology* 42 (01), 6–29.

Versluis, M., Schmitz, B. & Von Der Heydt, A. 2000 How snapping shrimp snap: through cavitating bubbles. *Science* 289, 2114–2117.

Wu, Q., Wei, W., Deng, B., Jiang, P., Li, D., Zhang, M. & Fang, Z. 2019 Dynamic characteristic of the cavitation cloud of submerged helmholtz self-sustained oscillation jets from high-speed photography. *Journal of Mechanical Science and Technology* 33 (02), 621–630.

Yuan, W. & Schnerr, G. H. 2003 Numerical simulation of two-phase flow in injection nozzles: interaction of cavitation and external jet formation. *Journal of Fluids Engineering-Transactions of the ASME* 125 (06), 963–969.

Yuan, M., Li, D., Kang, Y. & Shi, H. 2020 The characteristics of self-resonating jet issuing from the Helmholtz nozzle combined with a venturi tube structure. *Journal of Applied Fluid Mechanics* 13 (03), 779–791.

Zhang, K. & Chen, S. 2018 Numerical simulation of self-excited pulsed cavitation nozzle in three-dimensional unsteady flow. *Journal of Drainage and Irrigation Machinery Engineering* 36 (4), 288–293.

Zwart, P. J., Gerber, A. G. & Belamri, T. 2004 A two-phase flow model for predicting cavitation dynamics. In Proceedings of the fifth international conference on multiphase flow. Yokohama, Japan: ICeMF 2004, 152.