Experimental and Numerical Simulation of Flow Pattern Evolution in a Flow Mixing Nozzle under a Moderate Inlet Flow Rate

Jin Zhao,* Zhi Ning,* and Ming Lv

ABSTRACT: A flow mixing nozzle is a novel two-phase flow nozzle with a simple structure and has broad application prospects in the industrial field. However, the research on the working characteristics of this nozzle is mainly focused on a low inlet flow rate. There are few research studies on the working characteristics of the moderate and high flow rates which are more concerned in the industrial field, which limits the application of the flow mixing nozzle. In this paper, the experimental and simulation studies on the internal two-phase flow pattern and spray form of the flow mixing nozzle under a moderate inlet flow rate are carried out. The results show that the change in the two-phase inlet flow rate and the tube hole distance of the nozzle will affect the gas and liquid inertia force, thus affecting the working characteristics of the nozzle. The enhancement of gas inertia force will cause the flow pattern inside the nozzle to change to an unstable radially concave cone, and the spray form will be converted to atomization; the atomization effect of the liquid is better. The enhancement of the liquid inertial force will cause the flow pattern inside the nozzle to change to a radially convex cone, and the spray form will be converted to breakup; the atomization effect of the liquid is poor. The liquid surface tension maintains the existence of the cone inside the nozzle and plays a dominant role in jet atomization when the gas and liquid inertial forces are weak. In addition, the study also found that because of the compressibility of gas, the orifice has a reinforcing effect on gas inertia force.

1. INTRODUCTION

A flow mixing nozzle is a new type of two-phase flow nozzle, and Gañán-Calvo9 indicates that this nozzle can stably generate droplets of controlled size at a small inlet flow rate. In the experimental study, it was found that reducing the tube hole distance of the nozzle has a significant effect on the internal flow pattern of this nozzle.10,11 This greatly increases the atomization efficiency of this nozzle, providing the possibility of atomizing heavy oil.

This property of the flow mixing nozzle has attracted the interest of many researchers. Some scholars pay much attention to the size and other characteristics of the droplet generated by this nozzle, and a large number of experiments and numerical simulations have been carried out. Researchers such as Acero and Vega improved the nozzle by changing the surface structure or adding a guide rod in the flow mixing nozzle and conducted experimental research.12,13 Moreover, Herrada used numerical simulation to study the two-phase flow inside and outside the flow mixing nozzle.14 The experimental and numerical simulation results show that the liquid-phase recirculation inside the nozzle has a significant effect on the stability of the nozzle operation and the characteristics of the droplets generated by jet breakup. By controlling the nozzle structure and the flow parameters of the fluid, the droplet characteristics can be controlled according to the actual needs. These studies make the flow mixing nozzle have important application prospects in the field of biopharmaceuticals.

On the other hand, some researchers pay more attention to the atomization effect of the flow mixing nozzle on the liquid. For example, Azevedo9 conducted an experimental study on the influence of the orifice shape on the atomization effect of the nozzle, and the results show that the conical orifice has the best atomization effect; Jiang10–13 experimental research on the atomization of the flow mixing nozzle showed that the nozzle can produce good atomization of heavy oil (such as glycerin) at a lower inlet flow rate and work stably; the results of Simmons,14–17 Azevedo,18–20 and other researchers21,22 show that the atomization effect of the nozzle on different fluids (soybean oil, biodiesel, etc.) is better. These studies show that...
the flow mixing nozzle also has a broad application prospect in the field of liquid atomization.

The working characteristics of the flow mixing nozzle are different under different conditions, which are closely related to the gas—liquid two-phase flow inside the nozzle. When the nozzle is used to generate tiny droplets, the gas—liquid flow inside the nozzle is relatively flat. Therefore, the research on the two-phase flow inside the nozzle at a low gas—liquid flow rate (the Reynolds number is less than 100) is relatively comprehensive. When the nozzle is used for atomizing liquid, the inlet flow rate of the nozzle is usually moderate or large, and current research only focuses on the atomization effect of the nozzle but pays no attention to the two-phase flow inside the nozzle and the interrelationship between the inside and outside of the nozzle.

The reason for the transition between different working characteristics of the flow mixing nozzle, namely, the transition of the gas—liquid flow pattern inside the nozzle, is also worth studying. The theoretical analysis and experimental research of Rosell-Llompart23 show that the gas—liquid flow of the nozzle can be divided into three modes: capillary flow focusing, turbulent flow focusing, and flow blurring. The results show that inertial force and surface tension between the two phases are the cause of the two-phase flow transition. These research results are mainly based on the gas—liquid two-phase flow outside the nozzle, but the research on the classification and formation mechanism of gas—liquid two-phase flow inside the nozzle is still inconclusive.

In a word, for the flow mixing nozzle, the study on gas—liquid two-phase flow about the nozzle with a low two-phase flow rate has been in depth. However, the study on the gas—liquid flow and its transformation mechanism inside nozzle with a moderate or high inlet flow rate is still incomplete. In addition, insufficient attention has been paid to the research on the relationship between internal flow and external spray characteristics of the nozzle, which has an important impact on the stable and efficient application of the flow mixing nozzle.

In this paper, the visualization flow mixing nozzle was designed and the flow mixing nozzle atomization jet experimental platform was built. The flow mixing nozzles in two cases, which are flow focusing \((H/D = 1)\) and flow blurring \((H/D = 0.3)\), were selected, and the atomization experiments of the moderate two-phase flow rate (the Reynolds number is about 100–2000) were carried out. At the same time, the numerical simulation of the working process of the flow mixing nozzle was carried out using the Reynolds time-average method and volume of fluid (VOF) method. The gas—liquid two-phase flow pattern, gas—liquid two-phase flow morphology change, and spray characteristics of the flow mixing nozzle were analyzed by experiment and numerical simulation. At the same time, the conversion mechanism of gas—liquid two-phase flow inside the nozzle mixing area and orifice was studied by numerical simulation. Finally, combining the experimental and numerical simulation results, we analyze the relationship between the internal and external flows of the nozzle.

2. EXPERIMENTAL AND NUMERICAL SETUP

2.1. Atomizer. A designed flow mixing nozzle was used in the experiment, and the structure and prototype of this nozzle are shown in Figure 1. After referring to the literature, the structure and size of the nozzle were determined according to the experimental needs in this paper. The nozzle is composed of the mixing chamber, nozzle body, and nozzle inner tube. In order to observe the two-phase flow pattern inside the nozzle, the mixing chamber was made of plexiglass, and the diameter of the spray hole below the mixing chamber is 3 mm. The liquid fluid enters the mixing chamber through the nozzle inner tube, and the inner diameter and outer diameter of the nozzle inner tube are 5 and 10 mm, respectively. The gas inlet is on the nozzle body, and the distance between the nozzle inner tube and the spray hole can be adjusted by the adjustment thread between the nozzle body and the mixing chamber. In the experiment, the distance between the nozzle inner tube and the spray hole was selected to be 1.5 and 5 mm, respectively. The inner diameter of the annular air cavity formed by the nozzle inner tube and nozzle body is 30 mm and is sealed by a sealing tube to prevent gas leakage.

2.2. Experimental Platform. The experimental platform is schematically shown in Figure 2. Water and air were used as the test liquid and atomizing gas, respectively. The liquid is pumped from the water storage device through the pressure valve, the flow meter, and the check valve into the nozzle by the water pump (QSB-003, maximum flow rate is 5 L/min). The gas is pumped through the pressure valve, the flow meter, and the

![Figure 1. Flow mixing nozzle used in the experiment. (a) Prototype of the flow mixing nozzle. (b) Nozzle schematic. [(1) mixing chamber; (2) adjustment thread; (3) nozzle body; (4) air inlet; (5) sealing tube; (6) inner tube; (7) spray hole; (8) annular air cavity].](https://dx.doi.org/10.1021/acsomega.0c02702)

![Figure 2. Experimental platform. (A) Flow mixing nozzle; (B) collecting bucket; (C) experimental iron platform; (D) light source; (E) pneumatic antivibration isolation system; (F) high speed camera; (G) macro lens.](https://dx.doi.org/10.1021/acsomega.0c02702)
Check valve into the nozzle by the air pump (Outstanding 500–8 L, maximum flow rate is 40 L/min). The flow rate of the gas–liquid two-phase into the nozzle is controlled by flow meters, and the check valve is used to prevent the fluid from flowing back. A high-speed camera (MotionPro Y4, the shooting frequency is 9000 FPS) with a macro lens (V-DX MACRO 2X) was used to capture the two-phase flow pattern and spray form of the nozzle. In order to improve the photo clarity, a yellow light source (continual light) and a light guide plate are placed opposite the high-speed camera.

In the experiment of this paper, the flow rate of the gas and liquid path is measured and adjusted by the float flow meter. The accuracy of the float flow meter is ±1.5%. In the gas and liquid path, we also installed a pressure valve with an accuracy of 2%, which can ensure that the nozzle works stably in each group of experiments. In each group of experimental conditions, a high-speed camera is used to extract five images to ensure the accuracy of experimental results.

Open the gas and liquid pump separately before each experiment to check for leaks in the two-phase path. All the experiment data were obtained through the process described below. In the experiment, first turn on the air pump and then turn on the water pump after the air path is stable. The flow rate of air and water is changed using a flow meter to obtain a two-phase flow pattern photograph and a spray form photograph of the nozzle under different conditions. Wait for half a minute after adjusting the flow meter each time and then extract the image after the nozzle work stably. The physical properties and flow parameters of experimental fluid are shown in Table 1.

### Table 1. Physical Properties and Flow Parameters of the Experimental Substance

| Physical parameter     | Symbol | Unit      | Range   |
|------------------------|--------|-----------|---------|
| Water viscosity        | \( \mu_w \) | Pa·s      | \( 1 \times 10^{-3} \) |
| Water density          | \( \rho_w \) | kg/m³     | 998.2   |
| Air viscosity          | \( \mu_s \) | Pa·s      | \( 1.8 \times 10^{-3} \) |
| Air density            | \( \rho_s \) | kg/m³     | 1.225   |
| Surface tension coefficient | \( \sigma \) | N/m      | 7.2 \times 10^{-2} |
| Water flow rate        | \( Q_w \) | L/h       | 10–35   |
| Air flow rate          | \( Q_a \) | L/min     | 2–15    |

### 2.3. Simulation Model and Computational Domain.

The experimental image obtained using the flow mixing nozzle spray experimental platform is mainly the change in the two-phase flow morphology, which is the macroscopic feature of two-phase interaction. At the same time, because of visualization, the nozzle mixing chamber is made of plexiglass, which makes the inlet flow rate not too large in the experiment, otherwise the experimental nozzle will be damaged. The numerical simulation method is more convenient to analyze the interaction between the two phases, thus revealing the working mechanism of the flow mixing nozzle. At the same time, for the case of large flow rate, numerical simulation is used to supplement it appropriately.

Referred to the numerical simulation method in ref 8, we have used the well-tested commercial software Fluent (laminar unsteady and VOF method, the version of Fluent is 6.3.2) to simulate the two-phase flow pattern of the flow mixing nozzle working process. Because of the relatively low velocity of the fluid inside the nozzle, the fluid can be considered as incompressible. Therefore, the incompressible continuity equation and momentum equation of the fluid inside the nozzle can be expressed as eqs 1 and 2, respectively.

\[
\nabla \cdot \mathbf{u} = 0 \tag{1}
\]

\[
\frac{\partial}{\partial t} (\rho \mathbf{u}) + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \nabla \cdot (\mathbf{T}) + \rho \mathbf{g} + \mathbf{F} \tag{2}
\]

where \( \rho \) is the fluid density; \( p \) is fluid pressure; \( \mathbf{g} \) is gravitational acceleration; \( \mathbf{T} \) is the stress tensor calculated from the fluid viscosity \( \mu \) and velocity \( \mathbf{u} = \mathbf{\nabla} \cdot (\mathbf{V} \mathbf{u} + \mathbf{V} \mathbf{u}^T) \); and \( \mathbf{F} \) is the surface tension calculated based on the CSF model, as shown in eq 3.

\[
\mathbf{F} = \sigma k \mathbf{n} \tag{3}
\]

where \( \sigma \) is the surface tension coefficient; \( k \) is the interface curvature; \( k_i = \mathbf{\nabla} \cdot (\mathbf{n} / \mathbf{l}) \); and \( \mathbf{n} \) is the gradient vector of the volume fraction perpendicular to the phase interface.

The VOF method is used to track the gas–liquid interface inside the nozzle, that is, gas–liquid coupling is treated by the phase volume function \( \alpha \). The phase volume function \( \alpha \) satisfies eq 4.

\[
\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \mathbf{u}) = 0 \tag{4}
\]

In this way, the fluid density and viscosity in eq 2 can be determined by the density and viscosity of the gas–liquid two-phase fluid, as shown in eqs 5 and 6.

\[
\begin{align*}
\rho & = \alpha \rho_1 + (1 - \alpha) \rho_g \\
\mu & = \alpha \mu_1 + (1 - \alpha) \mu_g
\end{align*} \tag{5, 6}
\]

where the subscript 1 represents the liquid phase fluid and g represents the gas phase fluid.

According to the structure of the experimental nozzle, the numerical model of the flow mixing nozzle is established using Gambit, and the mesh is divided. Because the mixing chamber of the experimental nozzle is axisymmetric, in order to reduce the calculation time, it is reasonable to use a two-dimensional calculation model. A schematic diagram of the nozzle’s simulation model is shown in Figure 3.

![Figure 3. Grid of the domain under study. (1) Air inlet; (2) Water inlet; (3) Mixing area and spray hole; (4) Nozzle central axis; (5) Nozzle external field.](https://dx.doi.org/10.1021/acsomega.9b02702)

A numerical simulation model of the flow mixing nozzle was established based on the mixing chamber and the orifice area of the experimental nozzle. In the simulation model, the nozzle mixing chamber, inner tube, and spray hole are all of the same size as the experimental nozzle, and the size of the nozzle external field is 10 times that of the mixing chamber to reduce the influence of the boundary conditions. The focus of the simulation calculation is the two-phase mixing area and the
orifice area, so the mesh of this part is encrypted, and the mesh of the other parts is appropriately increased.

The nozzle center axis uses symmetrical axis boundary conditions. The outlet of the nozzle external field adopts the pressure outlet boundary condition, and the outlet pressure is set to the normal atmospheric pressure value. For the two-dimensional simulation case, the calculation result is more accurate when the two-phase inlet boundary condition adopts the velocity inlet boundary condition. Therefore, the simulation case in this paper uses the velocity inlet boundary condition, and its value is calculated according to the inlet flow rate and the cross-sectional area of the nozzle. In the calculation, the flow equation uses the laminar flow equation; the pressure solution uses the PISO method; the momentum equation is discretized by the second-order upwind scheme.

Figure 4 shows the variation of the error between the calculated jet diameter and the actual value with an equivalent grid size at \( Q_w = 30.2 \text{ L/h}, Q_a = 4.28 \text{ L/min}, \) and \( H = 5 \text{ mm}. \)

As can be seen from Figure 4, when the equivalent grid size is below 0.02, the numerical calculation achieves the second-order convergence, which is exactly the order of convergence expected by the second-order upwind discrete scheme we choose. Therefore, in the numerical model, we finally determined that the equivalent grid size of focus areas such as the two-phase mixing area and orifice area is 0.01, while the equivalent grid size of other areas gradually increases to 0.02.

Figure 5 shows the comparison between the gas–liquid flow morphology inside the nozzle calculated by the numerical model and the experimental results under different conditions.

As can be seen from Figure 5, the gas–liquid flow morphology inside the nozzle obtained by numerical simulation is very close to the experimental results. The abovementioned results show that the numerical model of gas–liquid two-phase flow inside the flow mixing nozzle is relatively accurate and reliable. In this paper, this simulation model is used to study the two-phase interaction mechanism inside the nozzle, and the calculation results under the higher inlet flow rate are added to supplement the experimental results.

3. TWO-PHASE FLOW PARAMETERS OF THE FLOW MIXING NOZZLE

3.1. Nozzle Operating Parameters. As a new two-phase flow nozzle, the flow mixing nozzle is quite different from the traditional two-phase flow nozzle. A good atomization effect of the air blast atomizer requires an extremely high two-phase inlet flow rate.\(^{24}\) Some new nozzles such as effervescent atomization have lower requirements for two-phase inlet conditions, but the structure is relatively complicated and the work process is unstable.\(^ {25−27} \) The spray form of the flow mixing nozzle is not only affected by the two-phase flow rate but also by the nozzle structure. Figure 6 shows a schematic diagram of the mixing chamber structure of the flow mixing nozzle.

The air inertia force has a significant effect on liquid atomization. As shown in Figure 6, air and water interact in the mixing area between the inner tube and the spray hole. The size of the gas–liquid contact surface determines the amount of inertia forces in the mixing area. It can be seen from the figure that the gas–liquid contact surface area is related to the inner tube diameter \( D \) and the tube hole distance \( H \), but for the fixed nozzle, the influencing factor is mainly the tube hole distance \( H \).
Figure 7 shows the spray form of the flow mixing nozzle under different conditions taken in the experiment. Figure 7a is a single-phase jet image as a comparative reference.

As can be seen from Figure 7, the single-phase jet is stable, almost no atomization, except that the jet surface is fluctuating when the liquid inlet flow rate is relatively large. However, at the same liquid inlet flow rate, liquid atomization is obvious when using the flow mixing nozzle. When the tube hole distance is small, the atomization range around the jet is larger and the penetration distance of the jet center is smaller. Therefore, it can be considered that the influence parameters of the flow mixing nozzle working process are mainly the two-phase inlet flow rate, the tube hole distance, and the inner tube diameter. Based on the abovementioned four parameters, the flow pattern and spray form change and mechanism of the flow mixing nozzle will be studied.

3.2. Two-Phase Flow Pattern and Spray Form. When the two-phase inlet flow rate is low, the internal flow pattern of the nozzle is expressed as a liquid cone, and the spray form is expressed as the drop mode, jet mode, or breakup according to the ratio of the two-phase flow rate.28 However, the situation is different under a moderate two-phase inlet flow rate. Figure 8 shows some images of the two-phase flow pattern inside the nozzle and the spray form under different conditions taken in the experiment.

It can be seen from Figure 8 that at a moderate inlet flow rate, the internal flow pattern of the nozzle still exhibits a liquid cone, but the cone shape is different under different air inlet flow rates. As shown in Figure 8a, when the air inlet flow rate is small, the liquid cone exhibits a radial protrusion. Increasing the air inlet flow rate, the cone will change radially into the concave. If the air inlet flow rate is extremely large, the liquid cone volume is significantly reduced, and the outer liquid has begun to atomize. At the same time, as shown in Figure 8b, under the moderate two-phase inlet flow rate, the spray form outside the nozzle appears as two modes, breakup and atomization.
The radial convex and concave shapes of the cone represent two different gas–liquid interaction states. Therefore, under the moderate two-phase inlet flow rate, this paper uses these two flow patterns as benchmarks. The two-phase interaction mechanism of the flow mixing nozzle is given by analyzing the changes in the cone under different conditions. The spray morphology and its changes are affected by the two-phase flow pattern inside the nozzle. Therefore, the transformation of spray morphology has also been analyzed.

4. RESULTS AND DISCUSSION

The flow mixing nozzle has different two-phase interactions and operating characteristics when the two-phase inlet flow rate is different. In this paper, experimental and numerical simulation methods are used to study the two-phase flow pattern and spray form of the flow mixing nozzle under a moderate two-phase inlet flow rate, which provides a theoretical basis for its application in industrial fields. In this part, the experimental and simulation data are analyzed from the inside of the nozzle, the orifice area, and the spray field, and the two-phase interaction characteristics and mechanism of the flow mixing nozzle are given.

4.1. Two-Phase Flow Pattern and Change Mechanism.

The morphological change in the cone inside the nozzle macroscopically reflects the characteristics of the two-phase interaction. After preliminary preparation of the experimental and simulation data, we found that the cone angle \( \theta \) is a simple and clear parameter to represent the change in cone shape. Therefore, we describe the morphological change of the cone.
through the change in cone angle and analyze the gas–liquid interaction inside the nozzle.

The experimental image is processed using MATLAB software for grayscale processing to make the cone boundary clearer, and it is convenient to extract the cone angle value. Under each experimental condition, five images obtained by the experiment are processed to extract the cone angle using MB-ruler software; the average value is taken as the object of the research analysis to ensure the accuracy of the results.

For $H = 0.3$ and $H = 1$, the cone angle is defined as shown in Figure 9. It can be seen from the figure that when $H = 0.3$, the cone is concave, and the higher the concave degree is, the larger the cone angle is; when $H = 1$, the cone is convex, and the smaller the convex degree is, the smaller the cone angle is.

Figure 10 shows the cone angle distribution for different two-phase inlet flow parameters. In the figure, we take the ratio of the gas–liquid two-phase Reynolds number at the entrance of the nozzle mixing zone as the x-coordinate. Among them, the gas–liquid Reynolds number $Re_g$ and $Re_w$ are calculated by the gas–liquid flow rate, inner tube diameter, and tube hole distance as the characteristic parameters.

As can be seen from Figure 10, when the gas–liquid flow inside the nozzle approaches flow blurring ($H = 0.3$), the increase in gas flow rate will increase the cone angle and the concave degree of the cone will be aggravated. When the gas–liquid flow inside the nozzle is flow focusing ($H = 1$), the increase in gas flow rate will decrease the cone angle and the convex degree of the cone. The effect of the increasing liquid flow rate of the nozzle on the cone angle under different tube hole distances is completely opposite to that of the gas flow rate.

The magnitude of the Reynolds number indicates the strength of the influence of fluid inertial force; the weakening of the cone convex degree is actually the increasing of the concave degree. Therefore, Figure 10 also shows that the increase in gas inertial force will increase the concave degree of the cone inside the nozzle, while the increase in liquid inertial force will decrease the concave degree of the cone inside the nozzle. This indicates that the gas inertia force may be the reason for the change in flow pattern from flow focusing to flow blurring inside the nozzle, while the liquid inertia force hinders this change. The results of Rosell-Llompart show that the gas–liquid inertia force is the reason for the change in jet form outside the nozzle, which is mutually confirmed with the experimental results of two-phase flow inside the nozzle in this paper. In other words, the change in the gas–liquid inertia force affects the internal or external gas–liquid flow pattern of the flow mixing nozzle.

The abovementioned analysis of the experimental data by the Reynolds number reveals the change in flow morphology and the change mechanism of the gas–liquid two-phase flow pattern inside the nozzle. In the following, we will further study and analyze the gas–liquid two-phase flow inside the flow mixing nozzle using the numerical simulation method.

Figure 11 shows the flow morphology and velocity distribution diagram inside the nozzle under different gas–liquid flow parameters when the gas–liquid flow inside the nozzle approaches flow blurring ($H = 0.3$) obtained by numerical simulation. The gas–liquid flow morphology diagram is on the left, and the velocity distribution diagram is on the right.

Compared with Figure 11a,b, it can be seen that the increase in liquid flow rate makes liquid velocity increase in the mixing area, so the concave degree of the cone decreases. Moreover, compared with Figure 11a,c, it can be seen that the increase in gas flow rate makes gas velocity increase in the mixing area, so the concave degree of the cone increases. On the whole, when the internal flow of the nozzle is close to flow blurring, the liquid cone inside the nozzle will not appear as an obvious convex phenomenon.

Figure 12 shows the flow morphology and velocity distribution diagram inside the nozzle under different gas–liquid flow parameters when the gas–liquid flow inside the nozzle is flow focusing ($H = 1$), obtained by numerical simulation.

As can be seen from Figure 12 that in the flow focusing mode, the reason for the change in cone shape caused by the change in the gas–liquid flow rate is the same as that when the flow approaches flow blurring, that is, the gas–liquid flow rate affects the gas–liquid velocity inside the mixing area, resulting in the

---

Figure 11. Two-phase flow morphology and velocity distribution under $H = 0.3$. (a) $Q_w = 10.55 \text{ L/h}$ and $Q_a = 4.28 \text{ L/min}$. (b) $Q_w = 30.2 \text{ L/h}$ and $Q_a = 4.28 \text{ L/min}$. (c) $Q_w = 10.55 \text{ L/h}$ and $Q_a = 7.56 \text{ L/min}$.

Figure 12. Two-phase flow morphology and velocity distribution under $H = 1$. (a) $Q_w = 10.55 \text{ L/h}$ and $Q_a = 4.28 \text{ L/min}$. (b) $Q_w = 30.2 \text{ L/h}$ and $Q_a = 4.28 \text{ L/min}$. (c) $Q_w = 10.55 \text{ L/h}$ and $Q_a = 33.86 \text{ L/min}$.
change in cone shape. When the flow pattern is flow focusing, the cone inside the nozzle may also be concave, but the required gas flow rate is large. When approaching flow blurring, a twofold increase in gas flow rate is likely to make the cone concave, while a nearly tenfold increase in gas flow rate is needed to achieve the same effect when flow focusing.

The experimental study of Si28 shows that the increase in tube hole distance will change the internal cone shape of the nozzle from concave to convex. We used the numerical simulation method to calculate the gas–liquid flow inside the nozzle at different tube hole distances, and the gas–liquid flow morphology and velocity distribution diagram inside the nozzle are shown in Figure 13.

As can be seen from the gas–liquid flow morphology inside the nozzle in Figure 13, the numerical simulation results in this paper are consistent with the research conclusion of Si, and the increase in tube hole distance will change the internal cone shape of the nozzle from concave to convex. On the other hand, the distribution of gas–liquid velocity in the nozzle indicates that the change in tube hole distance essentially makes the gas velocity in the mixing area significantly change, so as to make the cone shape change.

The abovementioned analysis of the experimental and numerical simulation results in this paper shows that the change in gas–liquid two-phase flow morphology and flow pattern inside the flow mixing nozzle is caused by the difference of gas and liquid velocities in the mixing area. The changes in gas and liquid flow rates, tube hole distance, and other parameters will affect the gas and liquid velocities in the mixing area, causing fluctuations in the inertia force of the gas–liquid two-phase fluid, which will lead to changes in gas–liquid two-phase flow inside the flow mixing nozzle. Therefore, we can use a simple force analysis to describe the change mechanism of the gas–liquid two-phase flow pattern inside the nozzle, as shown in Figure 14.

As shown in Figure 14, the cone in the mixing area is formed under the action of gas inertia force $F_A$, liquid inertia force $F_W$, and liquid surface tension $\sigma$. The increase in gas inertia force will cause the resultant force on the cone to be inclined to the horizontal direction, which will make the cone concave. The increase in liquid inertial force will cause the resultant force on the cone to be inclined to the vertical direction, which will make the cone convex. The liquid surface tension keeps the cone stable. It can be predicted that when the tube hole distance is small or the gas velocity is large, the liquid surface tension cannot continue to maintain the cone existence, and the flow pattern inside the nozzle will change to flow blurring.

As shown in Figure 14, the cone in the mixing area is formed under the action of gas inertia force $F_A$, liquid inertia force $F_W$, and liquid surface tension $\sigma$. The increase in gas inertia force will cause the resultant force on the cone to be inclined to the horizontal direction, which will make the cone concave. The increase in liquid inertial force will cause the resultant force on the cone to be inclined to the vertical direction, which will make the cone convex. The liquid surface tension keeps the cone stable. It can be predicted that when the tube hole distance is small or the gas velocity is large, the liquid surface tension cannot continue to maintain the cone existence, and the flow pattern inside the nozzle will change to flow blurring.

Based on our experimental and numerical simulation results and other relevant research results, we believe that the inertial forces of the gas and liquid in the mixing area of the flow mixing nozzle are essentially the macroscopic manifestations of the gas–liquid two-phase dynamic pressure. Therefore, the gas inertia force and the liquid inertia force can be obtained by integrating the gas–liquid two-phase dynamic pressure on the cone surface

$$F_A = \int a \rho \dot{V}_a^2$$

$$F_W = \int w \rho \dot{V}_w^2$$

(7)

In the formula, $a$ and $w$ represent the density of the gas and liquid, respectively, and $V_a$ and $V_w$ represent the gas and liquid velocity in the nozzle mixing area, respectively. In this way, the change mechanism of the gas–liquid two-phase flow pattern inside the flow mixing nozzle under a moderate inlet flow rate can be given using the flow pattern diagram, as shown in Figure 15.

As can be seen from the Figure 15, there is a clear dividing line between the two flow patterns. Gas and liquid inertia forces

![Figure 13. Two-phase flow morphology and velocity distribution under different tube hole distances. (a) $H = 0.5$, (b) $H = 1.5$, (c) $H = 2.5$, (d) $H = 3.5$, and (e) $H = 5$.](image1)

![Figure 14. Schematic diagram of two-phase force in the mixing area.](image2)

![Figure 15. Flow pattern diagram in the nozzle mixing area under a moderate inlet flow rate.](image3)
contributed to this transformation. When liquid inertia force is small, the effect of gas inertia force is more obvious. At this time, even gas inertia force is less than liquid inertia force and the flow pattern is converted into a near flow blurring (convex cone). However, as the liquid inertia force increases, the influence of the gas inertia force begins to decrease. At this time, the gas inertia force needs to be the same as or even stronger than the liquid inertial force to cause a change in the flow pattern.

4.2. Flow Behavior in the Orifice Area. The two-phase flow pattern in the mixing area has a significant effect on the spray form, and the orifice area acts as a key area connecting the inside and outside of the nozzle, which also plays an important role. In the experiment, we found that there were certain differences in the liquid jets inside the orifice under different parameters, as shown in Figure 16.

![Figure 16](https://dx.doi.org/10.1021/acsomega.0c02702)

Figure 16. Experimental image of the liquid jet inside the orifice area. (a) \( Q_g = 2.64 \text{L/min}, Q_w = 10.55 \text{L/h}, \) and \( H = 5 \text{mm} \) and (b) \( Q_g = 2.64 \text{L/min}, Q_w = 10.55 \text{L/h}, \) and \( H = 1.5 \text{mm} \).

The plexiglass at the orifice is relatively thick, and the image clarity of gas–liquid two-phase flow inside the orifice obtained by the experiment is relatively bad. However, it can still be seen from Figure 16 that the jet diameter inside the orifice is different under different conditions. We took the jet diameter \( d \) at the orifice inlet as the characterization parameter of jet morphology in the orifice area and studied the changes in jet morphology inside the orifice with the parameters. Under each experimental condition, the five images obtained by the experiment are processed using MATLAB software to extract the jet diameter using MB-ruler software; the average value is taken as the object of the research analysis to ensure the accuracy of the results.

Figure 17 shows the relationship between the jet diameter inside the orifice area and the gas–liquid two-phase flow rate. The gas Reynolds number \( Re_g \) and liquid Reynolds number \( Re_l \) at the orifice inlet were used to characterize the two-phase flow rate in the orifice area, and the orifice diameter was used as the characteristic parameter to calculate the Reynolds number.

![Figure 17](https://dx.doi.org/10.1021/acsomega.0c02702)

Figure 17. Relationship between the jet diameter and gas–liquid flow rate. (a) \( H = 1.5 \) and (b) \( H = 5 \text{mm} \).

As can be seen from the Figure 17, with the increase in gas Reynolds number, the jet diameter in the orifice area decreases, while the effect of the liquid Reynolds number on the jet diameter is the opposite. This law is similar to the variation of the degree of cone concave in the mixing area with the gas–liquid flow rate. This shows that the gas–liquid interaction in the mixing area is further developed in the orifice, and the gas–liquid flow morphology in the mixing area has an obvious influence on the liquid jet in the orifice. When the cone in the mixing area is convex, the jet diameter inside the orifice is larger; when the cone in the mixing zone is concave, the jet diameter inside the orifice is smaller.

Compared with Figure 17a,b, it can be found that the change trend of the jet diameter with the gas–liquid flow rate is basically the same when the tube hole distance is different; when the gas–liquid flow rate parameter is close, the difference of the jet diameter with different tube hole distances is not significant. The results of Gañán-Calvo1 also show that the structural parameters of the nozzle have little influence on the jet diameter. This may be caused by the fact that the liquid jet has been compressed to the limit by the gas in the orifice area, and the further compression of the liquid jet has been very difficult.

The influence of different parameters on the liquid jet inside the orifice is not only reflected in morphology but also in some effects that are difficult to be observed experimentally. Therefore, we used the numerical simulation method to calculate the streamline diagram inside the orifice when the cone concave degree is low or high, as shown in Figure 18a,b.

As can be seen from Figure 18, the gas streamline in the orifice area is denser than the liquid streamline, indicating that the orifice enhances the influence of gas inertia force. In Figure 18a, the concave degree of the cone inside the mixing area is low, the jet diameter inside the orifice is large, only the fluctuation can be observed on the jet surface, and the gas effect on the jet is weak. In Figure 18b, the concave degree of the cone inside the mixing area is high, the jet diameter inside the orifice is small, the gas affects strongly, the instability fluctuation can be seen inside the jet, and there is also a strong vortex in the gas flow.

When the two-phase flow enters the orifice from the mixing area, the gas is more easily compressed than the liquid. This causes the gas velocity to increase further, that is, the orifice actually enhances the gas inertia force. When the liquid inertial force predominates (convex cone; large jet diameter inside the orifice), the enhancement effect of the orifice on the gas inertia force is not obvious, and the breakup of the jet mainly depends
on the surface tension force of the liquid. When the gas inertia force predominates (concave cone; small jet diameter inside the orifice), the enhanced effect of the orifice on the gas inertia force causes instability in both the inside and outside of the jet. Some unstable vortexes are also generated inside gas flow. Under the sudden expansion of the orifice outlet, the unstable vortex inside the jet and the vortex in the gas assist the jet to produce good atomization.

4.3. Spray Form. The influence of the nozzle two-phase inlet condition and nozzle structure parameters on the two-phase flow pattern inside the nozzle and the effect of the nozzle orifice area on the jet are finally reflected in the spray form of the flow mixing nozzle. The spray form is also the most important indicator for evaluating the operating characteristics of a nozzle. After the analysis of the flow mixing nozzle spray field images obtained by the experiment, it was found that the spray form of the flow mixing nozzle can be divided into breakup, peripheral atomization, strong atomization, and nonuniform atomization under the moderate two-phase inlet flow, which are shown in Figure 19.

With the increase in gas flow rate or the decrease in liquid flow rate, the spray form will change from breakup or peripheral atomization to nonuniform atomization or atomization. As shown in Figure 19, in the breakup mode, the surface of the jet fluctuates and breaks into large liquid blocks. Moreover, in the peripheral atomization mode, although the cylindrical jet can still be seen on the whole, the liquid around the jet has a high degree of breakup or atomization. During nonuniform atomization, the liquid jet has almost all broken into droplets, but the degree of breakup is very uneven, and the areas with a high or low atomization degree appear randomly in the spray field. Moreover, in the atomization mode, the atomization degree of the liquid jet is relatively high, and the droplets in the spray field are evenly distributed.

The spray cone angle close to the nozzle outlet reflects the degree of gas—liquid interaction. When the spray cone angle is larger, the gas—liquid interaction is stronger and the atomization effect of the liquid is better. We studied the change in spray form of the flow mixing nozzle through the relationship between the spray cone angle and gas—liquid two-phase flow rate. The definition of the spray cone angle is shown in Figure 20a, and the angle between the outermost envelope of the spray cone (at four times orifice diameter from the nozzle) and the vertical direction is defined as the spray cone angle.
As can be seen from Figure 21, with the increase in gas Reynolds number, the spray cone angle also increases significantly, indicating that the gas inertia force may be the reason for the change in spray form. The effect of the liquid Reynolds number on the spray cone angle is different with different tube hole distances. When the tube hole distance is small, the spray cone angle will decrease with the increase in liquid Reynolds number; when the tube hole distance is large, the spray cone angle will increase with the increase in liquid Reynolds number. This shows that the liquid inertia force also has an important influence on the change in spray form.

The study on the change in spray cone angle under different parameters shows that the gas inertia force and liquid inertia force affect the spray form of the nozzle. These two forces also affect the liquid cone shape inside the nozzle. At the same time, liquid surface tension is also an influential factor of the cone shape. The Weber number can represent the influence of liquid surface tension. Therefore, we analyzed the cone shape and spray form of the flow mixing nozzle through gas inertia force $F_W$, liquid inertia force $F_L$, and Weber number and studied the change in spray form and the relationship between gas--liquid flow inside and outside the nozzle.

Gas inertia force $F_A$ and liquid inertia force $F_L$ are defined in the previous section, as shown in eq 7. Based on the difference between the gas--liquid two-phase velocity and the tube hole distance, the relative Weber number is defined as eq 8.

$$We = \frac{\rho_w |W_s - W_a|^2 H}{\sigma} \quad (8)$$

In the equation, $\sigma$ is the liquid surface tension coefficient. The relationship between the internal flow pattern of the nozzle, the spray form, and the two-phase interaction force is shown in Figure 22.

As can be seen from Figure 22, with the increase in Weber number, the cone shape inside the nozzle remained basically unchanged, while the spray form changed from breakup to peripheral atomization and then to nonuniform atomization. According to eqs 7 and 8, when the ratio of two-phase inertia force and the tube hole distance are basically unchanged, the increase in the relative Weber number defined in this paper indicates that the gas--liquid two-phase velocity is increased in equal proportion, while the ratio of two-phase inertia force is unchanged at the same time, which indicates that the gas inertia force is unchanged, so we can think that the increase in the relative Weber number indicates the increase in liquid inertia force. This indicates that the change in liquid inertia force leads the spray form change among breakup, peripheral atomization, and nonuniform atomization. It can also be seen from Figure 22 that the gas inertia force is the reason why the cone inside the nozzle changes from the convex to the concave and is also the main factor for the appearance of the atomization mode.

The study on the spray form change and the gas--liquid flow inside and outside the nozzle indicates that the gas--liquid two-phase inertia force controls the gas liquid two-phase flow of the flow mixing nozzle under a moderate two-phase flow rate. When the liquid inertia force is dominant, the liquid cone inside the nozzle is convex, and the spray form may be the breakup mode, peripheral atomization mode, or nonuniform atomization mode. At this time, the atomization effect of the nozzle is not ideal. When the gas inertia force is dominant, the liquid cone inside the nozzle is concave and the spray form will change to the atomization mode, and the flow mixing nozzle has the best atomization effect for all kinds of liquids.

5. CONCLUSIONS

In this paper, the internal flow pattern and spray form of the flow mixing nozzle under a moderate inlet flow rate were studied by experimental and numerical simulation methods. A self-designed variable-structure flow mixing nozzle was used in the experiment. The effects of nozzle flow parameters and structural parameters on internal flow patterns and spray forms were investigated by experimental methods. The simulation method is used to study the transformation mechanism of the internal flow pattern of the nozzle, and then, the evolution mechanism of
the spray form is given. The major conclusions of this study are as follows:

(1) At a moderate two-phase inlet flow rate, the flow pattern inside the flow mixing nozzle appears as a convex cone or concave cone. The gas inertial force and the liquid inertial force are the main reasons for the change in flow pattern inside the flow mixing nozzle. The increase in gas inertia force will make the liquid cone inside the nozzle appear as a concave cone, and the increase in liquid inertia force will make the liquid cone inside the nozzle appear as a convex cone. Liquid surface tension is the main reason to maintain the stability of the cone. When the gas–liquid two-phase flow is relatively high and the inertia force is strong, the surface tension cannot maintain the existence of the cone, and the liquid cone will be broken into a turbulent vortex.

(2) The orifice has a promoting effect on the interaction between the gas and liquid. Because the orifice is much smaller than the nozzle mixing area and the gas is relatively easy to compress, the orifice has a significant effect on the gas inertial force. Vortexes generated inside or on the surface of the liquid jet by the action of gas inertia force inside the orifice will promote the atomization of the liquid outside the nozzle.

(3) At a moderate two-phase inlet flow rate, the spray form of the flow mixing nozzle appears as breakup, peripheral atomization, atomization, and nonuniform atomization. The change in spray form is caused by the relative change in liquid inertia force and gas inertia force. The increase in liquid inertia force will change the spray form from breakup to peripheral atomization and then further change to nonuniform atomization. The increase in gas inertia force will further change the spray form into the atomization mode.

(4) At a moderate two-phase inlet flow rate, there is a certain relationship between the internal flow pattern and the spray form of the flow mixing nozzle. When the liquid cone inside the nozzle is convex, the spray form may be the breakup mode, peripheral atomization mode, or nonuniform atomization mode, and the atomization effect of the nozzle is not ideal. When the liquid cone inside the nozzle is concave, the spray form will change to the atomization mode, and the flow mixing nozzle has the best atomization effect for all kinds of liquids.

【AUTHOR INFORMATION】

Corresponding Authors
Jin Zhao — School of Mechanical, Electronic and Control Engineering and Beijing Key Lab Powertrain New Energy Vehicle, Beijing Jiaotong University, Beijing 100044, People’s Republic of China; orcid.org/0000-0002-7857-7083; Email: 15121365@bjtu.edu.cn

Zhi Ning — School of Mechanical, Electronic and Control Engineering and Beijing Key Lab Powertrain New Energy Vehicle, Beijing Jiaotong University, Beijing 100044, People’s Republic of China; Email: zhining@bjtu.edu.cn

Author
Ming Lv — School of Mechanical, Electronic and Control Engineering and Beijing Key Lab Powertrain New Energy Vehicle, Beijing Jiaotong University, Beijing 100044, People’s Republic of China

Complete contact information is available at: https://pubs.acs.org/10.1021/acsomega.0c02702

【Notes】
The authors declare no competing financial interest.

【ACKNOWLEDGMENTS】

This study was funded by the National Natural Science Foundation of China (Grant nos. 51776016 and 51606006), Beijing Natural Science Foundation (Grant nos. 3172025 and 3182030), the National Key Research and Development Program (Grant no. 2017YFB1030401), the National Engineering Laboratory for Mobile Source Emission Control Technology (Grant no. NELMS2017A10), and the Talents Foundation of Beijing Jiaotong University (Grant no. 2018RC017).

【REFERENCES】

(1) Gañán-Calvo, A. M. Generation of Steady Liquid Microthreads and Micron-Sized Monodisperse Sprays in Gas Streams. Phys. Rev. Lett. 1998, 80, 285–288.

(2) Gañán-Calvo, A. M. Enhanced liquid atomization: From flow-focusing to flow-blurring. Appl. Phys. Lett. 2005, 86, 214101.

(3) Vega, E. J.; Montanero, J. M.; Herrada, M. A.; Gañán-Calvo, A. M. Global and local instability of flow focusing: The influence of the geometry. Phys. Fluids 2010, 22, 064105.

(4) Acero, A. J.; Rebollo-Munioz, N.; Montanero, J. M.; Gañán-Calvo, A. M.; Vega, E. J. A new flow focusing technique to produce very thin jets. J. Micromech. Microeng. 2013, 23, 065009.

(5) Acero, A. J.; Montanero, J. M.; Ferrera, C.; Herrada, M. A.; Gañán-Calvo, A. M. Enhancement of the stability of the flow focusing technique for low-viscosity liquids. J. Micromech. Microeng. 2012, 22, 115039.

(6) Vega, E. J.; Gañán-Calvo, A. M.; Montanero, J. M.; Cabezás, M. G.; Herrada, M. A. A novel technique for producing metallic micrometer and microdrops. Microfluid. Nanofluidics 2013, 14, 101–111.

(7) Liu, Z. M.; Yang, Y. Influence of geometry configurations on the microdroplets in flow focusing microfluidics. Chin. J. Theor. Appl. Mech. 2016, 48, 867–876.

(8) Herrada, M. A.; Gañán-Calvo, A. M.; Ojeda-Monde, A.; Bluth, B.; Riesco-Chueca, P. Liquid flow focused by a gas: jetting, dripping, and recirculation. Phys. Rev. E: Stat., Nonlinear, Soft Matter Phys. 2008, 78, 036323.

(9) Azevedo, C. G.; de Andrade, J. C.; de Souza Costa, F. Effects of nozzle exit geometry on spray characteristics of a blurry injector. Atomization Sprays 2013, 23, 193–209.

(10) Jiang, L.; Agrawal, A. K.; Taylor, R. P. Clean combustion of different liquid fuels using a novel injector. Exp. Therm. Fluid Sci. 2014, 57, 275–284.

(11) Jiang, L.; Agrawal, A. K. Combustion of straight glicerol with/without methane using a fuel-flexible, low-emissions burner. Fuel 2014, 136, 177–184.

(12) Jiang, L. L. Investigation of atomization mechanisms and flame structure of a twin-fluid injector for different liquid fuels. Ph.D. Dissertation, University of Alabama Libraries, 2014.

(13) Jiang, L. L.; Agrawal, A. K. Investigation of Glycerol Atomization in the Near-Field of a Flow-Blurring Injector using Time-Resolved PIV and High-Speed Visualization. Flow, Turbul. Combust. 2015, 94, 323.

(14) Simmons, B. M.; Agrawal, A. K. Drop Size and Velocity Measurements in Bio-Oil Sprays Produced by the Flow-Blurring Injector. ASME 2011 Turbo Expo: Turbine Technical Conference and Exposition, 2011; pp 701–710.

(15) Simmons, B. M.; Agrawal, A. K. Spray characteristics of a flow-blurring atomizer. Atomization Sprays 2010, 20, 821–835.

(16) Simmons, B. M.; Agrawal, A. K. Flow Blurring Atomization for Low-Emission Combustion of Liquid Biofuels. Combust. Sci. Technol. 2012, 184, 660–675.
(17) Simmons, B. M.; Panchasara, H. V.; Agrawal, A. K. A Comparison of Air-Blast and Flow-Blurring Injectors Using Phase Doppler Particle Analyzer Technique. ASME Turbo Expo 2009: Power for Land, Sea, and Air, 2009; pp 981–992.

(18) de Azevedo, C. G.; de Souza Costa, F.; da Silva Couto, H. Initial Study of a Blurry Injector for Biofuel. 20 Workshop em Engenharia e Tecnologia Espaciais, 2011.

(19) de Azevedo, C. G.; de Andrade, J. C.; de Souza Costa, F. Characterization of a blurry injector for burning biofuels in a compact flameless combustion chamber. 14th Brazilian Congress of Thermal Sciences and Engineering, Rio de Janeiro, Brazil, 2012.

(20) de Azevedo, C. G.; de Andrade, J. C.; de Souza Costa, F. Effects of injector tip design on the spray characteristics of soy methyl ester biodiesel in a blurry injector. Renewable Energy 2016, 85, 287–294.

(21) Modesto-López, L. B.; Gañán-Calvo, A. M. Visualization and size-measurement of droplets generated by Flow Blurring in a high-pressure environment. Aerosol Sci. Technol. 2018, 52, 198.

(22) Pongvuthithum, R.; Moran, J.; Sankui, T. A flow blurring nozzle design for combustion in a closed system. Appl. Therm. Eng. 2018, 131, 587–594.

(23) Rosell-Llompart, J.; Gañán-Calvo, A. M. Turbulence in pneumatic flow focusing and flow blurring regimes. Phys. Rev. E: Stat., Nonlinear, Soft Matter Phys. 2008, 77, 036321.

(24) Batarseh, F. Z.; Gniś, M.; Roisman, I. V.; Tropea, C. Fluctuations of a spray generated by an airlast atomizer. Exp. Fluid 2009, 46, 1081–1091.

(25) Konstantinov, D.; Marsh, R.; Bowen, P. J.; Crayford, A. Effervescent atomization for industrial energy-technology review. Atomization Sprays 2010, 20, 525–552.

(26) Sovani, S. D.; Sojka, P. E.; Lefebvre, A. H. Effervescent atomization. Prog. Energy Combust. Sci. 2001, 27, 483–521.

(27) Gadgil, H. P.; Raghunandan, B. N. Some features of spray breakup in effervescent atomizers. Exp. Fluid 2011, 50, 329–338.

(28) Si, T. Experimental and theoretical investigation on flow focusing. Ph.D. Dissertation, University of Science and Technology of China, 2009.