Research Article

Formation of Residual Bubbles in Diesel Engine Nozzle and Their Influence on Initial Jet

Xiaonan Ni ¹ and Hua Wen ²

¹Haojue R & D Center, Jiangmen 529000, China

²College of Mechanical and Electrical Engineering, Nanchang University, Nanchang 330031, China

Correspondence should be addressed to Hua Wen; nijie26@163.com

Received 28 November 2020; Revised 20 April 2021; Accepted 23 July 2021; Published 12 August 2021

Academic Editor: Parvizi Ghadimi

Copyright © 2021 Xiaonan Ni and Hua Wen. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

The method of combining experiment and numerical simulation was used to study the cavitation and gas backflow phenomena during nozzle off-flow stage and the influence of residual bubbles on the initial jet in the near field. An equal-size optical nozzle based on acrylic material is designed, and the injection process of the fuel nozzle is photographed using high-speed photography technology. Establish a cavitation mathematical model to analyze the details of internal flow and initial jet. The results show that after the needle valve starts to close, cavitation occurs in the orifice and the sac in sequence, and the amount of cavitation in the sac is large. The collapse of cloud of cavitation bubbles will cause the outside air to flow back into the nozzle. The volume of the backflow air is slightly larger than the total volume of cloud of cavitation bubbles. The study found that the initial position of the residual bubbles has a significant effect on the initial atomization shape. When the residual bubble was in the front of the orifice, the initial tip was formed at the front of jet, and then, it stretched into a thin ligament due to vortex ring motion around the jet.

1. Introduction

The better atomization quality of the diesel engine nozzle makes the combustion in the cylinder more full, which can improve the engine's emission performance. Some studies [1–3] found that the spray atomization process is not only affected by aerodynamics, but also closely related to the internal flow field of the engine nozzle. During the continuous injection process of the nozzle, the gas outside the nozzle will flow back into the nozzle. Gas backflow will cause continuous bubble column or multibubble phenomenon during the injection interval, which is different from our traditional belief that the orifice is filled with liquid fuel after the end of injection. So, what caused the external gas to flow back into the nozzle to form residual bubbles? And the influence of these bubbles on the subsequent jet morphology is worthy of our in-depth study.

The high-speed flow inside the nozzle of diesel engine often causes cavitation. In recent years, many articles have been presented focusing on the occurrence of cavitation in the nozzle. Lopez et al. [4] found that testing cavitating injection nozzles shows a strong reduction in mass flow rate when

cavitation appears (the flow is choked), while the momentum flux is reduced to a lesser extent, resulting in an increase in effective injection velocity. This is related to the vapor appearance inside the injection hole produces a decrease in the viscosity of the fluid near the wall. Based on optical testing and three-dimensional CFD simulation, Tao et al. [5] studied the cavitation phenomenon in the injector and its influence on the injector flow rate and analyzed the cavitation process and flow characteristics in the injector. Du et al. [6] studied how geometry-induced cavitation affects the reliability of injection rate estimates based on momentum flux measurements. The results show that the cylindrical nozzle orifice has a strong tendency to induce cavitation, and the convergent nozzle orifice has a strong tendency to suppress cavitation.

At present, there has been much research, both experimental and numerical, concerning the diesel engine nozzle cavitation and spray atomization [7–10]. In [11], the whole process of the internal flow of the nozzle and the near-field spray was photographed. It was found that there were large bubbles in the pressure chamber (sac) and the nozzle hole (orifice) before the start of the injection, and it was pointed out that these residual bubbles were formed by the backflow

of external gas. It is pointed out in [12] that the phenomenon of external gas backflow has an important effect on the formation of sediments and HC emissions. In [13], the phenomenon that external gas was swallowed into the nozzle during the fuel stop was discovered, and the relationship between the volume of the backflow gas and the injection pressure, orifice length, and back pressure was studied. In [14], the effects of the orifice diameter and back pressure on the external gas backflow were studied in detail. Cavitation bubbles occur during the process of the high-speed jet of fuel flowing out of the nozzle. Jiang et al.'s [15] research found that the lower the viscosity of the fuel and the higher the saturated vapor pressure, the easier it is to produce cavitation. Regarding the characteristics of cavitation bubbles, Nigmatulin et al. [16–18] focused on the effects of fluid physical parameters and molar mass on the collapse and over-compression of cavitation bubbles. Regarding the bubble deformation and movement in the liquid, Ivashnyov et al. [19–21] established a model that considers the difference in phase velocity and a model based on the two-phase interfaces, respectively, to deal with bubble bursting and vapor bubbles moving in superheated liquid. The problem of thermal growth is solved, and the simulation results are consistent with the experimental results. Brujan et al. [22, 23] conducted experiments and simulations on the behavior of laser cavitation bubbles near two vertical rigid walls and their relationship with the wall distance. In addition, many scholars have also studied jet spray morphology [24–26]. The presence of residual bubbles will cause the nozzle to form several types of liquid filaments [27] at the front of the liquid column at the beginning of the injection. In [28], five types of initial jet crushing forms were summarized through experimental observation statistics, but no theoretical analysis was made on the crushing mechanisms of various forms. Most research scholars are still focusing on the cavitation in the stable injection stage, but there is still no detailed research on the effect of residual bubbles on the initial jet breakage, especially the jet structure. In recent years, the application of multiple injection technology for diesel engines has become more and more extensive. Therefore, the flow at the opening and closing stages of the needle valve is worthy of in-depth study.

In this paper, in order to study the relationship between the cavitation phenomenon and the backflow of external air in the diesel engine nozzle at the end of injection, the transparent nozzle is made of acrylic transparent material and installed on the head of the fuel injector to visualize the fuel flow inside the nozzle. Use a camera and a high-magnification, high-resolution long-distance microscope to visualize the actual size of the transparent nozzle, obtain high-definition, high-resolution test images, and reveal the development process of fuel cavitation in the nozzle hole. The development process of the jet microstructure in the near-field area was observed microscopically, and the formation mechanism of the “needle” liquid filaments on the head of the jet mushroom was explained in combination with numerical simulation. At the same time, based on the open source software OpenFOAM platform and Rayleigh dynamic relationship, a multiphase flow model of the cavitation fuel jet breakup process was established to explore the effects of cavitation and air

on the initial breakup of the jet in the near field. The formation mechanism of the “needle”-shaped liquid filament on the head of the jet mushroom is explained.

2. Description of Experiment and Simulation Methods

In order to study the internal flow of the diesel nozzle in detail, a combination of experiment and numerical simulation is used. Figure 1 shows the principle of the visual experimental device. The experimental device mainly includes fuel injection device and image acquisition system. The fuel injection device consists of the Delphi Multec DCR1400 high-pressure common rail fuel injection system and the fuel pump test bench. The fuel injection control system uses Delphi Multec DCR1400 high-pressure common rail fuel injection system, including PSDB2000IIA fuel injection pump test bench, angle signal panel, high pressure fuel pump, high pressure common rail pipe, fuel injector, and ECU control unit. The fuel injection system can control the circulating fuel injection quantity, speed, fuel injection pressure, fuel injection advance angle, and so on. Parameters such as speed, dead center signal, and crank angle signal are transmitted to the ECU through the angle signal disk and its signal collector to achieve the detection and control of the injection frequency and injection time.

The flow test in the nozzle uses a PhantomVEO-710L high-speed camera plus a QUESTAR QM100 long-distance microscope head for zooming shots. The frame rate is set to 20,000, and the exposure time is $1\mu\text{s}$, which can effectively freeze the flow field during the flow cutoff. The window range of QUESTAR QM100 is 0.375~8.000 mm (diameter), the working distance is 150~350 mm, the resolution is $1.1\mu\text{m}$, the maximum magnification is $\times 381$ ($\times$ is multiplied), and the test uses $\times 30$ magnification. Figure 2 shows the assembly of the transparent nozzle for the experiment. In the experiment, the sac and the orifice of the ball head part of the electromagnetic injector were ground away, and the sealing cone surface of the needle valve was retained. The polymethyl methacrylate (acrylic) material was processed into a transparent structure containing the sac and the orifice structure.

The transparent injection nozzle processed in the experiment is axisymmetric straight hole structure, the diameter of the sac is 0.75 mm, the orifice diameter is 0.3 mm, and the orifice length is 1.8 mm. Considering that the material strength of the transparent injection nozzle is low, the rail pressure of the diesel injection system is set to 40 MPa, and the back pressure is 3 MPa. The test fuel is No. 0 commercial diesel oil, and its physical properties are shown in Table 1.

The numerical simulation analysis is based on the open source software OpenFOAM, adding the Kunz cavitation model [29] to the interMixingFoam solver, establishing a two-phase three-component multiphase flow solver based on the fluid volume (VOF) method and coupling the LES large eddy simulation. VOF method gives the mixed phase continuity equation and momentum equation as follows:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0, \quad (1)$$

FIGURE 1: Schematic of visualization experimental system, including fuel supply system, fuel injection system, and image acquisition system. Use a high-speed camera to photograph the flow of diesel inside the transparent nozzle.

FIGURE 2: Assembly diagram of fuel injector and transparent nozzle for test. (a) Original nozzle. (b) Polished nozzle. (c) Transparent nozzle. (d) Injector with transparent nozzle.

TABLE 1: Physical properties of diesel.

Parameter	Numerical value
Density $\rho/\text{kg}\cdot\text{m}^{-3}$	843
Dynamic viscosity $\mu/\text{Pa}\cdot\text{s}$	0.0065
Air density $\rho/\text{kg}\cdot\text{m}^{-3}$	1.21
Air dynamic density $\mu/\text{Pa}\cdot\text{s}$	1.85×10^{-5}
Diesel steam density $\rho/\text{kg}\cdot\text{m}^{-3}$	0.14
Saturated vapor pressure P_v/Pa	2000
Surface tension $\sigma/\text{kg}\cdot\text{s}^{-2}$	0.03

$$\frac{\partial \rho u}{\partial t} + \nabla \cdot (\rho u u) = -\nabla p + \nabla \tau + \int_{S(t)} \sigma \kappa n \cdot \delta(x - x') ds, \quad (2)$$

where t represents time, ρ represents mixing density, u is fluid velocity, p is pressure, and τ is viscous shear stress. The integral term of momentum equation (2) is used to express the momentum source term generated by surface tension. The surface fraction of the liquid volume is solved for the area score. σ represents the surface tension coefficient of cavitation bubbles. κ is the interface curvature between the gas and the liquid, and n is the normal unit vector pointing to the surface S of the cavitation bubbles.

The cavitation phenomenon is the process of converting the liquid phase into the vapor phase, so it is necessary to

add the phase change source term to the VOF transport equation to construct the air-liquid-vapor of three-component VOF transport equation:

$$\frac{\partial(\alpha_l \rho_l)}{\partial t} + \nabla \cdot (\alpha_l \rho_l \mathbf{u}) = \dot{m}, \quad (3)$$

$$\frac{\partial(\alpha_v \rho_v)}{\partial t} + \nabla \cdot (\alpha_v \rho_v \mathbf{u}) = -\dot{m}, \quad (4)$$

$$\frac{\partial(\alpha_g \rho_g)}{\partial t} + \nabla \cdot (\alpha_g \rho_g \mathbf{u}) = 0, \quad (5)$$

$$\alpha_l + \alpha_v + \alpha_g = 1. \quad (6)$$

In the formula, l , v , and g represent liquid phase, vapor phase, and air phase, respectively, $\dot{m}$ represents mass exchange between liquid phase and vapor phase, and α represents volume fraction. The convection term in equation (3) can be converted to the following:

$$\nabla \cdot (\alpha_l \rho_l \mathbf{u}) = \rho_l (\nabla \cdot (\alpha_l \mathbf{u})) + \alpha_l (\mathbf{u} \cdot \nabla \rho_l). \quad (7)$$

Based on the following two points, first, the injection pressure is not particularly high; second, the duration of an injection cycle is very short, so the temperature is considered unchanged. Therefore, it is assumed that the fuel flow is incompressible to simplify the numerical calculation. For incompressible fluids, the transport equations for the liquid and air phases are as follows:

$$\frac{\partial \alpha_l}{\partial t} + \nabla \cdot (\alpha_l \cdot \mathbf{u}) = \alpha_l (\nabla \cdot \mathbf{u}) + \dot{m} \left(\frac{1}{\rho_l} - \alpha_l \left(\frac{1}{\rho_l} - \frac{1}{\rho_v} \right) \right), \quad (8)$$

$$\frac{\partial \alpha_g}{\partial t} + \nabla \cdot (\alpha_g \cdot \mathbf{u}) = \alpha_g (\nabla \cdot \mathbf{u}) - \alpha_g \dot{m} \left(\frac{1}{\rho_l} - \frac{1}{\rho_v} \right). \quad (9)$$

The cavitation model uses the Kunz [29] cavitation mass transfer model. Kunz's mass transfer model for cavitation liquids is based on the research of Merkle et al. [30] and modified the original Merkle mass transfer model to meet the characteristics of the fluid near the transition. Compared with other similar models that only rely on a set of schemes to simultaneously express the characteristics of steam generation and collapse, the Kunz mass transfer model is based on two different schemes. The steam generation rate $\dot{m}^-$ is proportional to the difference between the pressure in the flow field area and the saturated vapor pressure, and the steam collapse rate $\dot{m}^+$ is based on a cubic polynomial function with respect to the volume fraction.

The governing equations of the Kunz mass transfer model are given below.

$$\dot{m}^- = \frac{C_v \rho_v \alpha_l \min [0, P_l - P_v]}{1/2 \rho_l U_\infty^2 t_\infty}, \quad (10)$$

$$\dot{m}^+ = \frac{C_c \rho_v \alpha_l^2 (1 - \alpha_v)}{t_\infty}. \quad (11)$$

In the formula, P_l is the fluid pressure, P_v is the vapor pressure, and C_v and C_c are the evaporation coefficient and the condensation coefficient, respectively, and they are based on the empirical constant of the average flow. U_∞ is the free flow velocity, $t_\infty = L/U_\infty$ is the characteristic time scale, and L is the characteristic length. Take the relevant parameters of the Kunz model as $C_v = 1000$, $C_c = 200$, $U_\infty = 20$, and $t_\infty = 0.001$.

With reference to the structure of the straight-hole nozzle in the experiment, the upper part of the sac is taken as the entrance boundary of the calculation domain. Numerical calculation errors have an important influence on the calculation results [31, 32]. Using different grid strategies and different numbers of integration steps, it is possible to obtain reliable or unreliable results. Therefore, it is necessary to verify the independence of the computing grid. The grid information and structure size of the calculation domain are shown in Figure 3. The first layer of the boundary layer grid in the nozzle hole is $1 \mu\text{m}$, and the boundary layer growth rate is 1.2. Figure 4 is verification of mesh-dependence. When the number of grids is increased to 5 million and the thickness of the first layer near the wall is $1.6 \mu\text{m}$, the cavitation volume remains basically unchanged. In this paper, the number of grids calculated is 8 million, and the thickness of the first layer near the wall is $1 \mu\text{m}$, which meets the requirements of grid independence. In addition, the cumulative error value in the calculation process is ultimately less than 1.

In order to simulate the fuel cutoff process at the end of fuel injection, the boundary condition of the nozzle inlet is set to a varying mass flow to simulate the change in the nozzle mass flow caused by the movement of the needle valve. The change of mass flow with time is shown in Figure 5. The nozzle outlet is a constant pressure boundary condition of 3 MPa.

In the calculation, the PISO method is used to couple velocity and pressure and iteratively solve the continuity equation and momentum equation in steps. The convection term in the governing equation is discretized using the Normalized Variable (NV) Gamma difference format, the diffusion term is discretized in Gauss linear format, and the time-dependent unsteady term is discretized in Euler format. The time step is adaptively adjusted by the CFL number. In the current study, the CFL number is set to 0.3 and the average time step is 1.2×10^{-9} s.

3. Fuel Injector Cavitation and Gas Backflow

The shooting results of the flow field test in the nozzle are shown in Figure 6. The black part in the figure is caused by light scattering at the air-liquid interface, which can reflect the gas-liquid two-phase distribution. The time marked in the photo is based on the moment when the fuel injector receives the fuel injection signal (0 seconds). As can be seen from Figure 6(a), at the time of $0 \mu\text{s}$, the needle valve has not been opened. At this time, there are still residual bubbles inside the nozzle. These bubbles are formed by the air flowing backwards at the end of the last injection. It is found that the initial position of the residual bubbles in the orifice is

FIGURE 3: Nozzle calculation domain grid information. The grid type is hexahedron, the number of grid cells is about 8 million, the thickness of the first layer near the wall is $1 \mu\text{m}$.

FIGURE 4: Verification of mesh-dependence. The cavitation volume decreases as the number of grids increases. After the number of grids reaches 5 million, the cavitation volume is no longer affected by the number of grids.

FIGURE 5: Inlet mass flow changes with time. In the numerical simulation, mass flow increases linearly from 0 to 0.01 ms, keep 10 g/s unchanged from 0.01 to 0.03 ms, and finally, the flow rate decreases linearly to 0 at 0.05 ms.

random. At $150 \mu\text{s}$, the opening of the needle valve caused a slight contraction of the residual bubbles in the orifice. At $200 \mu\text{s}$, the residual bubbles in the orifice are discharged from the nozzle. At $350 \mu\text{s}$, the residual bubbles in the sac and the orifice are completely discharged out of the nozzle. Because the chamfer of the orifice of the test nozzle is large and no cavitation occurs in the orifice, there is no black area in the picture. Figure 6(b) is the process of needle valve closing. With the needle valve closing, cavitation is born at $450 \mu\text{s}$ at the entrance corner of the orifice, and then, cavitation gradually develops. The development and growth of cloud of cavitation bubbles will also be accompanied by the collapse of the bubbles, the very high values of the pressure [33] occurring during the collapse of a bubble cloud. So the surface of the cavitation bubbles tends to be wrinkled. Cavitation occurs throughout the nozzle at $600 \mu\text{s}$. Figure 6(c) is the image after the needle valve is completely closed, that is, after the end of injection. The image at $650 \mu\text{s}$ shows that the cavitation vapor in the orifice begins to collapse and collapse. At 1 ms, the air surface in the orifice began to become smooth and shiny, which is the characteristic of air, indicating that the external air has flowed back into the orifice at this time, and finally at 50 ms, the bubble in the orifice is larger, and there are also residual bubbles in the sac.

Figure 7 shows the change of the gas volume fraction in the nozzle with time. After starting the flow cutoff, cavitation began to occur in the orifice. Because the inlet flow rate drops sharply during the flow cutoff, and the fuel flow in the orifice remains unchanged due to inertia, causing the fuel in the orifice to be stretched, and at the same time due to the necking effect at the entrance of the orifice, cavitation (about 0.4 ms) occurs first at this location. Since the static pressure of fuel in the sac is much higher than that of the orifice, the initial moment of cavitation caused by the flow interruption inside the sac is delayed compared to the orifice. Cavitation occurs in the sac at about 0.6 ms. With the collapse of the cloud of cavitation bubbles in the sac and the orifice, the outside air began to flow back into the nozzle until the fluid was still. On the other hand, from the data changes, it can be seen that the maximum cavitation volume is slightly smaller than the backflow air volume. This is because in addition to cavitation, the fuel will squeeze out of the nozzle, and the fuel will also flow out of the nozzle due to inertia, so the amount of backflow gas is always greater than the maximum cavitation amount.

Because the backflow bubble inside the nozzle can be stable after the injection, and the volume of the bubble is more convenient to measure, the backflow bubble volume is used for simulation and experimental comparison research. Figure 8 shows 10 images of circulating backflow bubbles. The volume of the backflow gas is calculated by measuring the diameter of the spherical bubble and the length of the bubble column. Table 2 shows the corresponding bubble volume and volume fraction, and the average backflow bubble volume is 0.18 mm^3 , accounting for 15% of the nozzle volume. Figure 9 shows the comparison between the simulation results and the test results. The volume fraction of backflow bubbles is 16%, and the simulated value is basically consistent with the experimental value.

FIGURE 6: Plots of the flow field in the nozzle during an injection cycle: (a) needle valve opening process, (b) needle valve closing process, and (c) after the needle valve is completely closed.

FIGURE 7: Gas volume fraction changes with time, including cavitation volume in orifice (curve 1), sac (curve 2), entire nozzle (curve 3), and backflow gas volume in the entire nozzle (curve 4).

4. Effect of Residual Bubble Position on Initial Jet Structure

Considering that the pressure wave at the upper end of the nozzle will cause larger deformation of the bubble, OpenFoam's own two-phase compressible solver (compressiblein-

terFoam) is used for calculation to simulate the gas-liquid two-phase flow. The nozzle inlet boundary type is pressure boundary, and the nozzle inlet pressure is set within $20 \mu\text{s}$, linearly increasing from 3 MPa to the injection pressure (40 MPa), which is used to simulate the transition process of diesel pressure from needle valve opening to stable pressure. The nozzle outlet boundary is a fixed back pressure of 3 MPa.

It can be seen from the time-varying graph of the internal flow field of the nozzle that external air will enter the nozzle at the end of the injection, that is, there will be residual air bubbles in the sac and orifice before each injection. Obviously, the existence of these residual bubbles will cause the initial jet morphology in the near field to be different. Therefore, several shots were taken on the initial jet shape in the near field, and it was found that the jet shape was not the same, which was initially considered to be the effect of the residual bubbles.

The experimental conditions are the same as the previous section. The same device is used to perform the visual experiment under the same injection pressure, and a clearer picture of the initial jet structure of the nozzle outlet after the needle valve opened is obtained. Figure 10 shows several typical initial jet shapes in the near-nozzle region. After analysis and conclusion, it is believed that the initial jet shapes in

FIGURE 8: Image of backflow bubble volume distribution inside the nozzle.

TABLE 2: Bubble volume statistics inside nozzle.

Images	1	2	3	4	5	6	7	8	9	10	Average value
Backflow volume of gas/mm ³	0.172	0.165	0.185	0.174	0.17	0.172	0.19	0.205	0.182	0.185	0.18
Gas volume fraction/%	14.3%	13.8%	15.4%	14.5%	14.2%	14.3%	15.8%	17.1%	15.2%	15.4%	15%

FIGURE 9: Volume fraction comparison of backflow bubbles inside the nozzle. The numerical simulation results (black line) show that the bubble volume fraction is finally stable at about 16%, and the average value (red line) of the experiment is 15%, with a small error.

the near field can be basically divided into four categories. The first type belongs to the traditionally recognized jet shape, that is, the jet head forms a “mushroom head” shape. The reason is that under the aerodynamic force of the jet tip, the two ends are rolled up to form a “mushroom head” shape, and it is observed that the early broken droplets are formed by the front edge of the jet falling off, as shown at $240\ \mu\text{s}$ in Figure 10(a). Figure 10(b) is the second type, and there is a long “needle” liquid filament at the front of the jet, and the length of the liquid filament gradually becomes shorter over time. The shape in Figure 10(c) is different from the previous two categories, at $200\ \mu\text{s}$, and the front surface of the jet is not smooth, with surface waves and liquid filaments. At $220\ \mu\text{s}$, a “needle” liquid filament appears in the middle of the jet. When $240\ \mu\text{s}$, the front of the jet forms a relatively thick liquid filament, and the front end of the main jet forms a “mushroom head” shape. The fourth type is to form a large

“umbrella” head jet structure at the front of the main jet, and there are a lot of liquid filaments and droplets around the “umbrella” jet head, as shown at $240\ \mu\text{s}$ in Figure 10(d). By photographing multiple groups of images, it is found that the occurrence probabilities of four different jet structures are as follows: (a) 52%, (b) 15%, (c) 4%, and (d) 29%, respectively. The low probability of type (c) indicates that the filament structure is formed only under certain conditions.

Considering the randomness of the residual bubble position, the influence of the residual bubble on the near-field jet morphology at different positions is simulated. Set the bubble diameter equal to the diameter of the orifice, and the physical properties of the remaining bubbles are consistent with the ambient air.

Figure 11 is a comparison diagram of jet patterns when residual bubbles are in the sac, the middle position of the orifice, and the front end position of the orifice under the same pressure conditions. Comparing the cloud maps of the three groups of diesel phase volume fraction distribution, it can be seen that the initial jet morphology is affected by the position of residual bubbles. In Figure 11(a), the nozzle needle at $0\ \mu\text{s}$ begins to open and the inlet pressure starts to rise. In the subsequent $20\ \mu\text{s}$, due to the influence of the sac structure, the residual bubbles were first squeezed into a column in the nozzle and then were crushed into several small bubbles. At $22\ \mu\text{s}$, when the air bubbles are discharged out of the orifice along with the fuel, the instability of the jet surface is enhanced by the aerodynamics, and then, the surface waves are generated.

In Figure 11(b), within $16\ \mu\text{s}$, the residual bubbles have undergone a compression and then expansion process. At $24\ \mu\text{s}$, under the action of bubble extrusion and aerodynamics, the fuel in front of the bubble forms a thick oil film after being ejected from the orifice, and the tip formed by the diesel behind the bubble merges with the front oil film to form a “mushroom” shape.

The jet shape of Figure 11(c) is quite different from the previous two groups. Combined with the change of the residual bubble pressure in Figure 12, it can be seen that within $8\ \mu\text{s}$, as the pressure at the upper end of the nozzle continues

FIGURE 10: Plots of the initial jet structure in the near field at injection pressure of 30 MPa. The picture shows the four jet structures taken during the fuel injection cycle: (a) traditionally recognized mushroom head morphology, (b) needle-shaped liquid filament morphology, (c) needle-shaped liquid silk pierces mushroom head morphology, and (d) umbrella head shape.

to increase, the residual bubbles in the orifice are squeezed and deformed by the upstream diesel, and the residual bubble pressure also increases. At this stage, the bubble pressure is greater than the ambient pressure and the diesel behind the bubble is pushed out of the nozzle.

At $12\ \mu\text{s}$, the residual bubble pressure returned to the ambient pressure, and the fuel in front of the bubble was pushed forward by the residual bubble. At $14\ \mu\text{s}$, under the action of aerodynamics, the fuel in front of the bubble stretched to form a thin oil film. The fuel behind the bubble not only formed a “mushroom” shape, but also had a long thin liquid filament at the front of the jet, as shown in Figure 13.

The simulation results show that there is a similar “umbrella” oil film in the anterior segment of the liquid silk, which is very similar to the jet morphology shown in the image at $220\ \mu\text{s}$ in Figure 10(c). At $18\ \mu\text{s}$, the length of the liquid filament continued to increase. Due to the thin oil film, the liquid filament pierced the oil film.

5. Formation Mechanism of Liquid Filament Shape at the Tip of Jet

The formation and development mechanism of the elongated liquid filament formed by the front end of the jet is further analyzed by the momentum distribution of the front end of the jet. Momentum distribution at $10\ \mu\text{s}$, $12\ \mu\text{s}$, $14\ \mu\text{s}$, and $16\ \mu\text{s}$ for residual bubbles at the tip of the orifice is shown in Figure 14. At $10\ \mu\text{s}$, due to the residual bubble pressure gradually recovering, the residual bubble expands and is still in a compressed state under the action of the front and back diesel fuel. At $12\ \mu\text{s}$, the fuel in front of the bubble stretches into an oil film with high-speed droplets at the front of the oil film. At $14\ \mu\text{s}$, the bubbles could not be closed and diffused into the environment, the oil film gradually expanded, and the entire bubble shape showed a “concave” shape structure. At the same time, the velocity at the edge of the diesel fuel behind the bubble is greater than the velocity on the axis. This is because the presence of the bubble causes the wall

FIGURE 11: Plots of the comparison of jet morphology when residual bubbles are in different positions: (a) bubble in the sac, (b) bubble in the middle of the orifice, and (c) bubble in front of the orifice.

FIGURE 12: Residual bubble pressure varies with time. During the opening phase of the needle valve, the pressure in the bubble gradually increases and reaches the maximum value at the 8×10^{-6} second; then, the bubble is pierced by the diesel liquid wire, and the pressure gradually decreases.

shear force to disappear, so that the velocity in the shear layer is greater than the velocity on the axis. In the subsequent development process, the velocity on the front edge of the jet was always higher than the velocity on the axis. Under the action of aerodynamics, the front edge of the jet was continuously moved closer to the axis, causing the front edge of the jet to converge and form a tip, as shown in the time diagram of $18 \mu s$.

Figure 15 is the isosurface of the second invariant of the velocity gradient tensor [34] ($Q_c = 2 \times 10^{12} s^{-2}$) and the volume fraction of the diesel phase ($\alpha = 0.6$) at different times. Q_c represents the shape of the instantaneous vortex core.

The time of $10 \mu s$ shows that the jet has not yet flowed out of the orifice, and an annular vortex has been formed at the outlet of the nozzle. This is due to the shear-induced vortex generated by the density gradient during the air injection in the orifice, which is consistent with Ghiji et al.'s [25] test results. At $12 \mu s$, fuel is ejected from the orifice, and the high velocity gradient between the jet and the air causes the annular vortex's range to continue to increase. From $14 \mu s$ and $16 \mu s$, it can be observed that there are vortex ring structures in the neck and head of the liquid filament, and the length of the liquid filament is continuously increasing during this process. This is because the axial resistance to the center position of the vortex ring is the smallest. At this time, the vortex ring acts like a necking to cause the center speed to increase, thereby promoting the elongated liquid wire. At $18 \mu s$, the vortex ring at the tip of the liquid filament began to break, and the tip of the liquid filament also broke away.

The above mechanism can be explained as follows: there is a residual bubble inside the injection hole during the intermittent injection period. The bubble is squeezed so that the pressure rises and pushes the fuel in front of the bubble to the outlet of the orifice. The diesel fuel behind the bubble causes the wall shear force to disappear due to the presence of the bubble so that the speed on the edge is higher than the speed on the axis. After the diesel fuel in front of the

FIGURE 13: Numerical simulation of jet shape at $16 \mu\text{s}$.

FIGURE 14: Image of momentum distribution at different moments.

FIGURE 15: Instantaneous isosurfaces of the jet surface and Q_c .

bubble flows out of the orifice, it will continue to thin under the action of residual bubble compression and aerodynamics to form an “umbrella” oil film until it breaks into small droplets. On the other hand, the edge of the diesel jet behind the bubble continues to move closer to the axis to form a tip, and at the same time, it is contracted by the motion of the surrounding vortex ring, causing the tip to stretch into a long thin filament.

6. Conclusion

After shooting the injection cycle of the transparent nozzle, it can be concluded that the orifice and the sac of the nozzle are cavitation in sequence, and the cavitation amount in the sac is greater than the orifice. There are residual bubbles inside the nozzle during the intermittent injection period, and the bubble shape and initial position are random. The initial jet shape of diesel has two traditional “mushroom” forms and a “mushroom” form with elongated liquid filaments at the front, but the former appears the probability is much greater than the latter.

The numerical simulation results show that during the needle valve closing stage, cavitation occurs in the orifice and the sac, and the amount of cavitation in the sac is relatively large. The collapse of the cavitation vapor inside the nozzle is the main factor causing the backflow of external air. The residual bubbles at different positions have a great influence on the jet shape at the initial jetting stage. When the residual bubbles are at the front end of the orifice, the “mushroom” shape with elongated liquid filaments will form at the front of the jet. The residual bubble in the orifice causes the wall shear force to disappear, which makes the edge velocity of the jet higher than the axis velocity. This is a prerequisite for the formation of elongated liquid filaments at the front of the jet. At the same time, the movement of the vortex ring around the liquid filament causes the length of the liquid filament to further increase.

Data Availability

The processed data required to reproduce these findings cannot be shared at this time as the data also forms part of an ongoing study.

Conflicts of Interest

The authors declare no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

Acknowledgments

The authors wish to thank Professor of H. WEN for his interest to study and discussion of the results.

References

- [1] R. Payri and F. Salvador, “Study of cavitation phenomenon using different fuels in a transparent nozzle by hydraulic characterization,” *Experimental Thermal and Fluid Science*, vol. 44, no. 1, pp. 235–244, 2013.
- [2] Z. Gao, Q. Wang, and Z. He, “Test on spray characteristics of burr nozzles in injectors,” *Journal of Internal Combustion Engine*, vol. 33, no. 6, pp. 530–536, 2015.
- [3] J. Desantes, R. Payri, and F. Salvador, “Influence of cavitation phenomenon on primary break-up and spray behavior at stationary conditions,” *Fuel*, vol. 89, no. 10, pp. 3033–3041, 2010.
- [4] J. J. Lopez, F. J. Salvador, and O. Garza, “A comprehensive study on the effect of cavitation on injection velocity in diesel nozzles,” *Energy Conversion & Management*, vol. 64, pp. 415–423, 2012.
- [5] T. Qiu, X. Song, and Y. Lei, “Cavitation process and flow characteristics inside diesel injector nozzle,” *Transactions of the Chinese Society for Agricultural Machinery*, vol. 47, no. 9, pp. 359–365, 2016.
- [6] C. Du, S. Andersson, and M. Andersson, “The effect of cavitation on the estimation of fuel injection rates based on momentum flux measurements,” *Fuel*, vol. 238, pp. 354–362, 2019.
- [7] L. Liu, X. Ma, and F. A. Magagnato, “Extended modeling of decelerating turbulent jets for diesel spray penetration after end-of-injection,” *Fuel*, vol. 199, pp. 324–331, 2017.
- [8] D. Jarrahbashi, S. Kim, and C. L. Genzale, “Simulation of combustion recession after end-of-injection at diesel engine conditions,” *Journal of Engineering for Gas Turbines and Power*, vol. 139, no. 10, 2017.
- [9] S. Moon, W. Huang, and Z. Li, “End-of-injection fuel dribble of multi-hole diesel injector: comprehensive investigation of phenomenon and discussion on control strategy,” *Applied Energy*, vol. 179, no. 1, pp. 7–16, 2016.
- [10] G. Jiang, Y. Zhang, and E. Medhat, “Visualization experiment of flow and spray morphology in different fuel nozzles,” *TCSAE*, vol. 45, no. 5, pp. 22–29, 2014.
- [11] E. G. Santos, J. Shi, M. Gavaises, C. Soterio, M. Winterbourn, and W. Bauer, “Investigation of cavitation and air entrainment during pilot injection in real-size multi-hole diesel nozzles,” *Fuel*, vol. 263, p. 116746, 2020.
- [12] T. Hayashi, M. Suzuki, and M. Ikemoto, “Effects of internal flow in a diesel nozzle on spray combustion,” *International Journal of Engine Research*, vol. 14, no. 6, pp. 646–654, 2013.
- [13] K. Saha, E. Abu-Ramadan, and X. Li, “Modified single-fluid cavitation model for pure diesel and biodiesel fuels in direct injection fuel injectors,” *Journal of Engineering for Gas Turbines and Power*, vol. 135, no. 6, 2013.
- [14] H. Wen, C. Wang, and M. Elkenawy, “The effect of back pressure on gas backflow in the process of fuel injection nozzle fuel cutoff,” *TCSAE*, vol. 10, no. 48, pp. 364–369, 2017.
- [15] G. Jiang, Y. Zhang, and H. Wen, “Study of the generated density of cavitation inside diesel nozzle using different fuels and nozzles,” *Energy Conversion & Management*, vol. 65, no. 103, pp. 208–217, 2015.
- [16] R. I. Nigmatulin, A. A. Aganin, and T. D. Yu, “Possibility of cavitation bubble supercompression in tetradecane,” *Doklady Physics*, vol. 63, no. 8, pp. 348–352, 2018.
- [17] A. A. Aganin, M. A. Il’gamov, and R. I. Nigmatulin, “Evolution of distortions of the spherical shape of a cavitation bubble in acoustic supercompression,” *Fluid Dynamics*, vol. 45, no. 1, pp. 50–61, 2010.

- [18] R. I. Nigmatulin, R. T. Lahey, and R. P. Taleyarkhan, "On thermonuclear processes in cavitation bubbles," *Physics-uspekhi*, vol. 57, no. 9, pp. 947–960, 2014.
- [19] O. I. Ivashnev, M. N. Ivashneva, and N. N. Smirnov, "Rarefaction waves in nonequilibrium-boiling fluid flows," *Fluid Dynamics*, vol. 35, no. 4, pp. 485–495, 2000.
- [20] O. E. Ivashnyov and N. N. Smirnov, "Thermal growth of a vapor bubble moving in superheated liquid," *Physics of Fluids*, vol. 3, no. 3, 2004.
- [21] O. E. Ivashnyov, M. N. Ivashneva, and N. N. Smirnov, "Slow waves of boiling under hot water depressurization," *Journal of Fluid Mechanics*, vol. 413, pp. 149–180, 2000.
- [22] E. A. Brujan, T. Noda, and A. Ishigami, "Dynamics of laser-induced cavitation bubbles near two perpendicular rigid walls," *Journal of Fluid Mechanics*, vol. 841, pp. 28–49, 2018.
- [23] E. A. Brujan, H. Takahira, and T. Ogasawara, "Planar jets in collapsing cavitation bubbles – ScienceDirect," *Experimental Thermal and Fluid Science*, vol. 101, pp. 48–61, 2019.
- [24] C. Crua, T. Shoba, and M. Heikal, *High-Speed Microscopic Imaging of the Initial Stage of Diesel Spray Formation and Primary Breakup*, Sae, Technical Papers, 2010.
- [25] M. Ghiji, L. Goldsworthy, and P. Brandner, "Analysis of diesel spray dynamics using a compressible Eulerian/VOF/LES model and microscopic shadowgraphy," *Fuel*, vol. 188, pp. 352–366, 2017.
- [26] C. Badock, R. Wirth, and A. Fath, "Investigation of cavitation in real size diesel injection nozzles," *International Journal of Heat and Fluid Flow*, vol. 20, no. 5, pp. 538–544, 1999.
- [27] N. Mitroglou, M. McLorn, and M. Gavaises, "Instantaneous and ensemble average cavitation structures in diesel micro-channel flow orifices," *Fuel*, vol. 116, pp. 736–742, 2014.
- [28] H. Ding, Z. Wang, and Y. Li, "Initial dynamic development of fuel spray analyzed by ultra high speed imaging," *Fuel*, vol. 169, pp. 99–110, 2016.
- [29] R. F. Kunz, D. A. Boger, and T. S. Chyczewski, "Multi-phase CFD analysis of natural and ventilated cavitation about submerged bodies," in *In: Proceedings of 3rd ASME/JSME Joint Fluids Engineering Conference*, ASME Paper FEDSM99-7364, 1999.
- [30] C. L. Merkle, J. Feng, and P. E. O. Buelow, *Computational Modeling of the Dynamics of Sheet Cavitation*, Third International Symposium on Cavitation, Grenoble, France, 1998.
- [31] N. N. Smirnov, V. B. Betelin, and V. F. Nikitin, "Accumulation of errors in numerical simulations of chemically reacting gas dynamics," *Acta Astronautica*, vol. 117, pp. 338–355, 2015.
- [32] N. N. Smirnov, V. B. Betelin, and R. M. Shagaliev, "Hydrogen fuel rocket engines simulation using LOGOS code," *International Journal of Hydrogen Energy*, vol. 39, no. 20, pp. 10748–10756, 2014.
- [33] E. A. Brujan, T. Ikeda, and Y. Matsumoto, "Shock wave emission from a cloud of bubbles," *Soft Matter*, vol. 8, no. 21, pp. 5777–5783, 2012.
- [34] T. Van, W. Edward, and A. Michael, "Eddies, stream, and convergence zones in turbulent flows," *Physics of Fluids*, vol. 30, no. 10, pp. 512–516, 2015.