Study of the influence laws of the flow and cavitation characteristics in an injector control valve

Rina Ren | Tiexiong Su | Fukang Ma | XiaoJun Wu | Chunlong Xu | Xin Zhao

1School of Mechanical Engineering, North University of China, Taiyuan, China
2School of Energy & Power Engineering, North University of China, Taiyuan, China
3China Northern Engine Research Institute, Tianjin, China
4College of Mechatronics Engineering, North University of China, Taiyuan, China

Correspondence
Tiexiong Su, School of Mechanical Engineering, North University of China, Taiyuan 030051, China. Email: sutiexiong@nuc.edu.cn

Abstract
Fuel cavitation can cause damage to the structure of an injector control valve, which can further affect the service life of the injector. Therefore, it is of great importance to carry out research on the mechanism of cavitation in injector control valves. In this study, the dynamic evolution of the cavitation and fuel flow characteristics during the opening of a ball valve in a high-pressure common rail injector control valve were studied. First, a transient CFD simulation of the cavitation during the opening of the control valve was performed using dynamic mesh technology to analyze the evolution and formation mechanism of the cavitation. The cavitation was influenced by the fuel flow velocity, which was affected by the sealing cone angle and the ball valve movement. Additionally, the fuel reflux could effectively inhibit the development of cavitation. By comparing with the pressure, velocity, and mass flow rate of the fuel under conditions without cavitation, it was found that the cavitation had a remarkable effect on reducing the hydraulic shock to the ball valve and sealing cone as well as stabilizing the fluctuation of the fuel velocity in the control chamber. In addition, the cavitation noticeably decreased the fuel mass flow rate (by up to 32% at 0.015 ms) and increased the average pressure in the chamber (by up to 29.1% at 0.01 ms), ultimately leading to a slower response of the control valve. The analysis of different inlet pressures of the control valve indicated that the increase in the inlet pressure caused the cavitation to extend upward along the sealing cone, and that the cavitation ring area and the cavitation strength increased.

KEYWORDS
control valve, flow and cavitation characteristics, fuel reflux, inlet pressures, numerical simulation
1 | INTRODUCTION

Diesel engines possess tremendous power performance, strong adaptability, a wide range of applications, and other significant advantages. For a long time in the future, they will still be an important source of power for construction machinery, ships, military equipment, and other fields. As the "brain" and "heart" of a high-speed and high-power diesel engine, the high-pressure common rail injection system enables a precise adjustment of the fuel injection to increase the output power of a diesel engine, improve fuel economy, and reduce the emission of nitrogen oxides and particulate matter significantly. As the core component of a high-pressure common rail injection system, a fuel injector represents the highest technology level and development difficulty for the common rail system, and it exerts an important impact on the power performance, fuel economy, and emission control of diesel engines. The strong flow recirculation and local depression inside a fuel injector provide favorable conditions for the occurrence of cavitation. Cavitation, which exerts a facilitating effect on the atomization of the fuel, mainly occurs inside injector nozzles and control valves. However, the cavitation phenomenon caused by this will cause damage to the injector structure to a certain extent; that is, fuel cavitation has a critical impact on the performance of the high-pressure common rail injection system. Therefore, the fuel flow and cavitation around the nozzle and inside the control valve are important factors for the injector performance.

The flow and cavitation phenomena around an injector nozzle have been studied by many scholars. Asi et al. analyzed the failure of a diesel engine injector nozzle and found that cracks sprouted in the area of cavitation damage and expanded in the form of fatigue cracks, resulting in the actual service life of the injector nozzle being much lower than its theoretical service life. Hassanazadeh et al. studied the effect of wall injection on the cavitation phenomenon inside a diesel engine injector based on a two-phase mixing model. The results showed that the larger the flow coefficient was, the lower the number of bubbles was when the injection angle was determined, and the cavitation number was within a certain range. Jia et al. conducted numerical simulations of the cavitation flow at a nozzle using a mixed multiphase model and a full cavitation model in order to investigate the law of cavitation flow in the injector. The results revealed that the evolution of cavitation had a significant effect on the liquid film thickness and the velocity at the nozzle outlet, which in turn caused significant changes in the spray angle and the droplet mean diameter. Cao et al. conducted an experimental study on the optical visualization of a nozzle and found that the vortex and cavitation were important factors influencing pressure fluctuations in the injection system. Lešnik et al. used a combination of numerical simulations and experiments to investigate the influence of the flow field, cavitation, and fuel characteristics inside a nozzle on the spray development and initial combustion processes. The results indicated that cavitation inside the injector had a positive influence on the spray breakup and spray decomposition processes.

As an important part of the injector control structure in the high-pressure common rail system, the control valve is the pivotal link of the injector hydraulic servo drive mechanism and a volumetric functional component that closely links the motion of the ball valve with the motion of the needle valve. The change in pressure in the control chamber is caused by the opening and closing of the ball valve. The reliability of the sealing of the two metal parts of the ball valve and the valve seat is a key factor that affects the response of the injector needle valve. Therefore, the structural characteristics of the control valve affect the injection system of the injector, which in turn affects the impact resistance property of the diesel engine. Furthermore, the flow and the cavitation of the fuel in the control valve cause certain degrees of destruction for its structural characteristics. In summary, it is of great significance to carry out a study of the flow and cavitation of the fuel in the control valve. An et al. employed numerical analysis to design a marine anti-cavitation control valve that was superior to conventional control valves. Yaghoubi et al. carried out a study on spherical control valves using numerical analysis. The results revealed that an appropriate increase in the valve internals was beneficial for reducing the intensity of cavitation in the valve and retarding the occurrence of cavitation. Experimental studies of injector cavitation damage conducted by Duan et al. illustrated that cavitation-induced cavitation occurred on the inner wall surface of a control valve during injector operation, which indicated that cavitation had an important effect on the damage to injectors. Wang et al. found that the variation in the internal structure of a control valve was an important factor influencing the degree of cavitation through a comparative study of the structural parameters of the control valve.

In general, research on injector control valves has mainly focused on the effects of the structure and the low inlet pressure on the degree of cavitation, but there has been little research on the specific development process of fuel cavitation in control valves, flow characteristics, or the effects of high inlet pressure on the degree of cavitation. In this study, research about the evolution of cavitation in an injector control valve as well as the flow characteristics of the fuel and the high inlet pressure on the degree of cavitation was carried out with CFD numerical simulation. During the study, the accuracy of the 3D model of the injector control valve was verified in terms
of both the single injection volume and the injection duration using the construction of a 1D mathematical model of the injector control valve. The research results could further improve the theory related to the cavitation effect in the injector valve and provide an important theoretical basis for the development of the common rail system toward high-power density.

2 | NUMERICAL SIMULATION

2.1 | Governing equations

In this research, the cavitation phenomenon inside a control valve was numerically simulated based on a multiphase flow model with a fluid mixture of liquid and vapor fuel as the research object. According to the laws of mass conservation, energy conservation, and momentum conservation, the continuity equation in fluid mechanics

\[ \frac{\partial}{\partial t} (\rho m \vec{v}_m) + \nabla \cdot (\rho m \vec{v}_m \vec{v}_m) = -\nabla p + \nabla \cdot \left[ \mu_m \left( \nabla \vec{v}_m + \nabla \vec{v}_m^T \right) \right] + \rho_m \ddot{\vec{F}} + \nabla \cdot \left( \sum_{k=1}^{n} \alpha_k \rho_k \ddot{\vec{v}}_{dr,k} \right). \]  

\[ (4) \]

In Eq. (4), \( \ddot{\vec{F}} \) is the fluid volume force, \( \mu_m \) is the mixture viscosity, and \( \ddot{\vec{v}}_{dr,k} \) is the slip velocity of the \( k \)th phase.\(^{20} \)

The viscosity for each phase of the fluid mixture is

\[ \mu_k = \sum_{k=1}^{n} \alpha_k \mu_k. \]  

\[ (5) \]

The slip velocity of the \( k \)th phase is

\[ \ddot{\vec{v}}_{dr,k} = \ddot{\vec{v}}_k - \ddot{\vec{v}}_m. \]  

\[ (6) \]

The turbulence kinetic energy \( k \) and the turbulence dissipation rate \( \varepsilon \) in the model can be obtained by solving the following two equations:

\[ \frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_j} (\rho u_j \varepsilon) = \frac{\partial}{\partial x_j} \left[ \mu + \frac{\mu_i}{\sigma_k} \frac{\partial \varepsilon}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k. \]  

\[ (7) \]

\[ \frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho u_j \varepsilon) = \frac{\partial}{\partial x_j} \left[ \mu + \frac{\mu_i}{\sigma_k} \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1k} \frac{\varepsilon}{k} (G_k + G_M G_b) - C_{2k} \rho \varepsilon^2 + S_k. \]  

\[ (8) \]

In Eqs. (7) and (8), \( G_k \) and \( G_b \) represent the sum of the turbulent kinetic energy due to the velocity gradient and the buoyancy. \( C_{1k}, C_{2k}, \) and \( G_M \) are the model constants. The turbulent viscosity coefficient \( \mu_i \) is

\[ \mu_i = \rho C_{\mu} \frac{k^2}{\varepsilon}. \]  

\[ (9) \]

In Eq. (9), \( C_{\mu} \) is a constant. The calculation method adopted in this research was the second-order discretization method, which had higher calculation accuracy. The standard turbulence model was used for calculation. This model was applicable to the shear flow, boundary layer flow, mixed layer flow, and jet flow, and the relevant model constants were determined by referring to the empirical values obtained from turbulent flow experiments.

The values of the model constants are as follows\(^{21} \):

\( C_{1k} = 1.44, C_{2k} = 1.92, C_{\mu} = 0.09, \sigma_k = 1.0, \sigma_\varepsilon = 1.3. \)
For the simulation calculation of the cavitation model, it was generally assumed that the fuel flow would produce a large quantity of bubbles, and the process of increasing in the numbers and in the shape of the bubbles could be observed from the simulation results. There was no velocity slip between the two phases of the fluid. The dynamic equation of the bubbles is

\[ \frac{D^3 \rho_{B}}{Dt^3} + \frac{3}{2} \left( \frac{DS_{B}}{D_t} \right)^2 = \left( \frac{P_B - P}{\rho_l} \right) - \frac{4v_i}{\rho_{B}} \rho_{B}B - \frac{2S}{\rho_l} \]

In Eq. (10), \( \rho_{B} \) is the radius of the bubble, \( S \) is the coefficient of tension on the surface of the fluid molecules, \( \rho_l \) is the density of the fluid, \( P_B \) is the sum of the pressures on the surface of the bubble, and \( P \) is the far-field pressure of the developing bubble.

If the interaction of surface tension is not considered, the above equation can be simplified as

\[ \frac{D \rho_{B}}{D_t} = \sqrt{2 \left( \frac{P_B - P}{\rho_l} \right)} \]  

Eq. (11) characterizes the interaction of the bubble development and the cavitation phenomena.

When the fluid flow is simulated using the mixing model, the mass transport equation for the vapor in the mixed fluid is

\[ \frac{\partial}{\partial t} (\alpha \rho_v) + \nabla \cdot (\alpha \rho_v \mathbf{V}_v) = R. \]  

In Eq. (12), \( \alpha \) is the volume fraction, \( \rho_v \) is the density, \( \mathbf{V}_v \) is the average motion speed, \( v \) is the viscosity coefficient, and \( R \) is the net mass source term.

The equation for the volume fraction of vapor in the cavitation model (Schnerr–Sauer model) is

\[ \frac{\partial}{\partial t} (\alpha \rho_v) + \nabla \cdot (\alpha \rho_v \mathbf{V}_v) = \frac{\rho_v \rho_l \text{Da}}{\rho} \frac{D}{Dt}. \]  

The equation for the net mass source is

\[ R = \frac{\rho_v \rho_l \text{Da}}{\rho} \frac{D}{Dt}. \]  

In the cavitation model, the number of bubbles per unit volume is closely related to the size of the vapor volume fraction in the fluid mixture, which can be expressed by the equation:

\[ a = \frac{n_b \frac{4}{3} \pi \rho_{B}^3}{1 + n_b \frac{4}{3} \pi \rho_{B}^3}. \]

Combining Eqs. (14) and (15), \( R \) is expressed as

\[ R = \frac{\rho_v \rho_l}{\rho} \alpha (1 - \alpha) \frac{3}{2} \sqrt{\frac{2}{3} \left( \frac{P_v - P}{\rho_l} \right) - \frac{2S}{\rho_l} \}} \]  

In Eqs. (16) and (17), \( K \) is the rate of fluid mass transfer in the fluid mixture, \( \rho_{B} \) is the radius of the bubble, \( P_v \) is the saturation vapor pressure at the specified temperature in the fluid mixture, \( n \) is the number of bubbles per unit volume in the fluid mixture, and \( P \) is the pressure at the center of the bubble.

When using the Schnerr–Sauer model for simulation calculations, the density of the bubbles needs to be determined. When no cavitation occurs in the fluid, the bubble density can be assumed to be constant.

### 2.2 Geometry and grid modeling of control valve

The control valve, as the pivotal link of an injector hydraulic servo drive mechanism, controls the motion of the needle valve through the motion of the ball valve, which can achieve the injection and unloading of a fuel injector. The geometric model of the injector control valve in this research is shown in Figure 1, and its geometric dimensions are shown in Table 1.

To analyze the mechanism of the formation and development of the cavitation in a control valve, a detailed understanding of the flow inside the control valve during the opening of the ball valve is required. In this research, a three-dimensional (3D) numerical model is built based
on a commercial code FLUENT was used to calculate the fuel flow during the control valve opening process and for a period of time (0.09 ms) after full opening.

In order to simulate the motion process of the ball valve, in this research, the motion trajectory of the ball valve was defined based on the mesh reconstruction methods (spring smooth method and local reconstruction method) and the user-defined function UDF that came with the simulation software. Additionally, the dynamic mesh technique was used to simulate the internal flow during the opening of the sealed ball valve. Since the topology of the model could not be changed during the grid movement, resulting in the model being unable to be started with the ball valve completely closed, the initial clearance between the ball valve and the sealing seat was set to 0.01 mm in this research. Comprehensively considering the calculation accuracy and scale of the model, the mesh at the sealing seat was locally encrypted, especially at the outlet of the throttle orifice where flow separation occurred, and the total number of mesh cells of the final injector control valve was 1866324. The mesh model is shown in Figure 2.

Since the initial model corresponded to 0.01 ms after the opening of the ball valve and the displacement was 0.01 mm, the initial value of the pressure boundary was the inlet and outlet pressure of the control valve at 0.01 ms after the ball valve started to move in the actual operation of the control valve. Specifically, the inlet pressure was 180 MPa, and the outlet pressure was 0.2 MPa. For the wall boundary, a boundary condition without slip was used. The computational step length of the model was set to $10^{-7}$ s, the total number of steps was 2000, and the maximum number of iterations was 30.

The two-phase mixture materials studied in this research were liquid diesel and vapor diesel. The density of the liquid diesel was 835 kg/m$^3$, the viscosity was 0.0024 Pa·s, and the surface tension was 0.02 N. The density of the vapor diesel was 0.029 kg/m$^3$, the viscosity was $3.1 \times 10^{-6}$ Pa·s, the initial bubble diameter was $1 \times 10^{-6}$ m, and the saturation vapor pressure was 380 Pa.

### 2.3 | Grid independence verification

In order to ensure the calculation accuracy and reduce the calculation time, the mesh independence was verified in this research. As shown in Table 2, different total numbers of grids were set up for the control valve model for the calculation, and the mass flow rate at the outflowing control-orifice and the average volume fraction of the vapor were compared. The calculation results are shown in Figures 3 and 4.

Figures 3 and 4 show that as the total number of grids increased from 0.84 to 1.86 million, the mass flow rate at the outflowing control-orifice and the average volume fraction of the vapor in the control valve showed an increasing trend. When the total number of grids increased from 1.86 to 2.82 million, the mass flow rate at the outflowing control-orifice and the average volume fraction of the vapor in the control valve were basically the same, but the calculation time increased significantly. Therefore, the total number of grids was set to 1.86 million to meet the requirements of calculation accuracy with less time.

### TABLE 1 | Structural parameters of the control valve

| Structure name          | Parameter setting |
|-------------------------|-------------------|
| Quality of ball valve   | 2.5 g             |
| Diameter of ball valve  | 1.3 mm            |
| Lift of ball valve      | 0.06 mm           |
| Ball valve seat angle   | 120 deg           |
| Aperture of OZ          | 0.25 mm           |
| Aperture of OA          | 0.27 mm           |
| Diameter of seat orifice| 0.35 mm           |

### FIGURE 2 | Computational mesh of the control valve

### TABLE 2 | Comparison of grid sizes

| Element count | Minimum length scale (mm) | Maximum length scale (mm) |
|---------------|----------------------------|---------------------------|
| 846,543       | $6.58e^{-6}$               | $1.38e^{-4}$              |
| 1,157,061     | $6.25e^{-6}$               | $1.29e^{-4}$              |
| 1,866,324     | $5.91e^{-6}$               | $1.29e^{-4}$              |
| 2,252,268     | $5.95e^{-6}$               | $1.29e^{-4}$              |
| 2,824,782     | $5.42e^{-6}$               | $1.29e^{-4}$              |
2.4 | Model verification

The calculation results needed to be verified based on experimental data. Due to the limitations of the measurements and the test equipment, the internal flow and cavitation of the control valve could not be measured directly, so a one-dimensional (1D) numerical model of the injector was built based on a commercial code AMESim was used to verify the accuracy of the 3D model. As the “core” component of the common rail system, the injector was also the most accurate and difficult part to manufacture, which directly affected the performance of the fuel injection. Hence, the 1D model of the injector had to be built with as much detail as possible.

The injector module mainly included the control valve assembly, control chamber assembly, and needle valve assembly. The main structural parameters of the injector are shown in Table 3. In the modeling process, the structure was somewhat simplified according to its working principles and the research requirements, and the control of the solenoid valve was simplified to the closure or disconnection of the solenoid valve with time control. The EMLTR02 model was selected for the solenoid valve, which could convert the input control signal into the corresponding current. The magnetic flux and the electromagnetic force of the solenoid valve were time-varying, and they were imported into the solenoid valve model in the form of a data file during the modeling. The injection pulse width of the injector was set to 0.8 ms. Since the software came with a nozzle module with a maximum of 10 nozzles, which could not meet the modeling requirements, the nozzle part was modeled separately, and the nozzle module was formed along with the cone valve parts. The simulation model of the injector is shown in Figure 5.

Figure 5 gives the 1D simulation model of the injector. The created 1D injector model was compared with the 3D model with the same boundary conditions to verify the accuracy of the 3D model. The calculated results for the injection volume and the injection duration of the 1D model were compared with the experimental data to verify the accuracy of the 1D model.

Figure 6 presents a comparison of the fuel outflowing control-orifice mass flow rates of the 1D model and the

| Structure name                  | Parameter setting | Structure name                  | Parameter setting |
|---------------------------------|-------------------|---------------------------------|-------------------|
| Number of injection orifices    | 12                | Quality of ball valve           | 2.5 g             |
| Aperture of injection orifice    | 2.5 mm            | Quality of needle valve          | 3 g               |
| Pretightening force             | 25 N/mm           | Diameter of seat orifice         | 0.35 mm           |
| Spring stiffness                | 28.08 N           | Diameter of ball valve           | 1.3 mm            |
| Needle valve stroke             | 0.35 mm           | Seal cone angle                  | 120 deg           |
| Diameter of needle valve        | 4.5 mm            | Spring stiffness of electromagnet| 70000 N/m         |
| Aperture of OZ                   | 0.25 mm           | Initial tension of spring of electromagnet | 85 N            |
| Aperture of OA                  | 0.27 mm           | Lift of ball valve               | 0.06 mm           |
| Flow coefficient of injector hole | 0.8               | Diameter of control piston      | 4.5 mm            |
3D model, as well as the 3D model without the cavitation model. The figure shows that the trends of the fuel outflowing control-orifice mass flow rate calculated with the 1D model and the 3D model were the same, and the maximum error was about 5.3%, which proved the accuracy of the 3D model. The error between the 1D model and the 3D model was due to the exclusion of the effect of the spatial structure of the control valve on the fuel flow in the 1D model. Before 0.06 ms, the mass flow rate at the OA of the 1D model increased, and the error of the mass flow rate at the OA for both models was larger because the slant iron lift was approximated by the primary function equation in the 3D calculation, so the slant iron lift was slightly different from the slant iron lift in the 1D model. After 0.06 ms, the slant iron lifts of the 1D model and the 3D model tended to be similar, and the error of the mass flow rate at the OA for both models was stabilized at about 5%.

In order to verify and validate the fluid numerical model, the common rail system was tested on the test bench. The test system consisted of a Delphi High-Pressure Common Rail fuel injection system, a Hansmann 22-kW fuel injection pump testbed, an EMI-II transient parameter measurement and analysis system, an EFS8233 common rail injector solenoid valve control device, an EFS8244 rail pressure control device, a track pressure sensor, and a high-resolution angle sensor.22,23 The test system diagram and the principle map are presented in Figures 7 and 8.

FIGURE 5 Simulation model of fuel injector

The overall simulation model of the high-pressure common rail injection system was verified using the single injection volume and the injection duration at different control pulse widths, which were measured with the high-pressure common rail injection system injection performance test bench, as shown in Figure 9(A) and (B).

The maximum relative errors between the 1D model and the experimental single injection volume and injection duration for different control pulse widths were 2.5% and 3.9%, and the validation results showed that this calculation model had good flow simulation capability.

3 | RESULTS AND DISCUSSION

3.1 | The development process of cavitation in control valves

Although the inlet of the control valve was an asymmetric structure, the fuel flowed into the area where the ball valve was located through the pilot orifice, inlet volume orifice, and throttle orifice. Hence, the asymmetry of the inlet had less effect on the pressure gradient of the fuel around the ball valve and the violent turbulent movement of the fuel at the ball valve, and the asymmetry of the cavitation at each section was not obvious, as shown in Figure 1. During the opening of the ball valve, cavitation mainly occurred in the region between the ball valve and the sealing cone. Cross-sections were created at the centers of the inlet, control chamber, and outlet, and the cavitation flow in this critical region was analyzed in detail based on the calculation results, as shown in Figure 10(A)–(D).

The cavitation, velocity streamline, and pressure cloud diagram inside the control valve at the moment of 0.001 ms are shown in Figure 10(A). When the ball valve was opened, the high-pressure fuel flowed into the control chamber through the OZ, then through the guide orifice into the OA, and finally through the throttle orifice into the gap between the ball valve and the sealing cone, forming a high-pressure jet with small radial and axial ranges. About 800 streamlines were drawn uniformly in the upstream flow path along the Y-axis.

At 0.001 ms, cavitation first occurred at the location of the ball valve tangent to the sealing cone. Cavitation occurred at this location for two reasons. As shown in the streamline, the fuel flowed close to the valve seat, and when the fuel flowed through the corner of the seat cone, the streamline suddenly turned, causing the liquid to separate and form a small vortex, which in turn formed a negative pressure zone at the corner, resulting in the occurrence of cavitation. At the same time, the high-pressure difference between the inlet and the outlet caused the fuel to form a high-pressure jet when it flowed at a high speed between
the ball valve and the sealing cone, and a velocity discontinuity occurred with the low-pressure fuel near the outlet of the control valve, resulting in turbulence between the ball valve and the sealing cone. Additionally, the turbulent fuel pressure at the gap decreased, and cavitation occurred at the cone gap when the pressure dropped below the saturated vapor pressure of the fuel.

Figure 10(B) depicts the cavitation, velocity streamline, and pressure clouds of the control valve between 0.003 and 0.019 ms during the opening. The cavitation cloud map shows that the cavitation area upstream continued to increase as the ball valve was opened. From the velocity streamline diagram, it can be found that the high-velocity area of the fluid between the ball valve and the sealing cone increased with the expansion of the vapor area. This was because the decrease in the fuel viscosity in the cavitation area led to reduced friction, which made the fuel velocity in the cavitation area increase, thus causing a decrease in the pressure and further formation of cavitation.

By observing the calculation results, it could be found that the guide orifice effectively stabilized the fuel velocity. However, the relatively small aperture of the OA and the throttle orifice made the fuel velocity increase. The flow direction of the fuel was in the upstream flow channel, and the throttle orifice angle was about 60°. When the fluid from the orifice entered the area where the ball valve was located, it had a serious impact on the ball valve, and a very serious sudden bending phenomenon occurred. As can be seen from the pressure cloud map, when the ball valve opened (0.01 ms), the fuel pressure in the flow channel below the ball valve decreased abruptly, but the fuel pressure at the bottom of the ball valve remained essentially unchanged, with a pressure transient value of about 150 MPa.

As time passed, the valve movement increased the gap between the ball valve and the sealing cone, and turbulence began to occur around the armature, with the vortex expanding between 0.01 and 0.019 ms. The reason for this was that the vortex expanded in the radial and axial directions as the high-pressure jet was carried by the fuel toward the outlet. In the control chamber, the vortex first occurred at the wall of the control chamber near the OZ, continued to expand as the ball valve opened.

As shown in Figure 10(C), the cavitation of the fluid at the armature vortex occurred during the time period from 0.02 to 0.26 ms. The figure shows that the increase in the vortex during this time period caused the cavitation area to gradually increase and merge with the fuel vapor below.

The cavitation cloud in Figure 10(D) shows that cavitation was observed in the seat orifice turning area at 0.035 ms. This was due to the reduction in the fuel viscosity in the cavitation area that reduced the friction between the fuel, thus leading to a pressure drop and an increase in the fuel velocity between the two cavitation areas, which
in turn led to cavitation in the seat orifice turning area. Finally, the two cavitation areas were connected.

As the ball valve opened, the negative pressure area between the sealing cone and the ball valve gradually increased, leading to an increase in the cavitation region. The vapor volume fraction reached a maximum at the point where the ball valve lift reached 0.037 mm (0.054 ms).

The velocity flow diagram in Figure 10(D) shows that reflux occurred at the outlet of the control valve as the ball valve lift gradually increased. The graph shows that as the fuel pressure in the cavitation region decreased, the increase in the fuel vapor volume prevented the fuel from flowing out of the control valve. Therefore, the reflux phenomenon at the outlet of the control valve became more serious.

After a period of time when reflux occurred in the valve (the ball valve lift was 0.037 mm) the volume of the fuel vapor in the control valve gradually decreased. The graph shows that as the fuel domain gradually increased. As a result, the average vapor volume fraction tended to increase more slowly, and the average vapor volume fraction decreased. However, as the fuel filled the control valve rapidly, the average value of the vapor volume fraction increased more rapidly. It could be seen that the growth rate of the average vapor volume after the ball valve was opened was significantly higher than the growth rate before the ball valve was opened, and the opening of the ball valve caused the fuel volume to increase, which had a positive effect on the increase in the vapor volume fraction.

Some conclusions could be drawn from Figures 10 and 11(A). When the lift of the ball valve rose from 0.01 to 0.02 mm, bubbles occurred near the ball valve and the sealing cone surface. The bubbles grew along the inner surface of the ball valve and formed a bubble circle, and the bubbles adhered to the surface of the ball valve to form a sheet of cavitation and expand along the surface, and the average volume fraction of the vapor continued to increase. As the ball valve lift rose from 0.02 to 0.037 mm, the cavitation gradually spread from around the ball valve to the entire upstream flow path region, and the average volume fraction of the vapor continued to increase. When the ball valve lift rose from 0.037 to 0.06 mm, the bubbles began to break and move downward due to the instability of the sheet cavitation, forming a cloud of cavitation. Finally, the bubbles kept breaking, the cloud cavitation disappeared, and the average volume fraction of the vapor decreased.

Figure 11(b) shows the mass flow rate at the control valve outlet at an inlet pressure of 180 MPa. As can be seen from Figure 11, the increase in the fuel vapor volume in the control valve prevented the fuel from flowing out, thereby causing the outflow for the control valve to
decrease. The fuel vapor volume fraction reached its maximum value \((t_1)\) when the ball valve lift was 0.037 mm (0.054 ms), and the reflux (the inflow mass flow rate of the fuel that was greater than the outflow mass flow rate) phenomenon \((t_3)\) occurred at the outlet of the control valve. After the reflux occurred at the outlet of the control valve, the average volume fraction of the fuel vapor gradually decreased to the lowest value \((t_2)\) before the sealed ball valve was fully opened for 0.018 ms, and the reflux phenomenon at the outlet increased first before decreasing until it disappeared \((t_4)\). The average volume fraction value of the fuel vapor in the control valve and the mass flow rate value at the outlet reach balanced, maintaining a relatively stable state of fluctuation. Therefore, the appearance of the fuel reflux at the outlet of the control valve was accompanied by the reduction in the cavitation.

By analyzing the calculation results of the cavitation inside the control valve, it was found that the cavitation mainly occurred at the ball valves, sealing cone, and OZ. This was consistent with the results obtained from the fatigue test in Figure 12.15 Specifically, the reason that the cavitation ring was larger than the sealing ring was that the area of the ball where strong cavitation occurred was higher than the ball valve drop seat impact area.
3.2  Flow characteristics of the control valve opening process

Studies have shown that the occurrence of cavitation can have a significant effect on the flow rate of a control valve. The flow rate of a control valve directly affects the injector unloading efficiency and the stopping injection time, which has a significant impact on the following combustion process. Therefore, it is necessary to analyze the flow characteristics of the fuel inside a control valve and calculate the no-cavitation case for the same boundary conditions to study the effect of cavitation on the fuel flow inside a control valve.

3.2.1 Analysis of the fuel pressure in the control valve

Figure 13(A) and (B) displays the control valve with and without the cavitation phenomenon for the five points a, b, c, d, and e of the wall pressure of the time history curve. As can be seen from the figure, the cavitation had an important impact on the pressure distribution of the control valve sealing cone. With cavitation, at the moment of the opening of the ball valve (0.01 ms), the fuel flow over the sealing cone surface momentarily generated very high pressure (water hammer pressure), which could reach nearly 80 MPa, but only for a very short time. Then, the pressure rapidly decreased and remained at 13 MPa (stagnation pressure) for a relatively long time. The maximum value occurred at the location of point a, that is, near the seat orifice. Without cavitation, during the control valve opening, the pressure on the sealing cone surface was negative, and at the moment the ball valve opened, the wall surface was subjected to severe hydraulic shock, with the maximum value occurring at the d-point position, where the pressure dropped sharply to −196 MPa at the moment of high-pressure jet shock. This only lasted for a very short time, and then, the pressure increased to −16 MPa.

To further analyze the hydraulic impact of fuel on the ball valve with and without cavitation in the control valve, the pressure conditions at five points on the surface of the ball valve were analyzed. Figure 14 illustrates the pressure time history curves at the five points f, g, h, i, and j on
the surface of the ball valve. The ball valve surface was subjected to hydraulic shock, as shown in Figure 14(A). The closer the position was to the bottom of the ball valve, the larger the hydraulic shock was. The maximum value is 145 MPa, so the hydraulic shock to the ball valve when there was cavitation mainly occurred at the bottom. As shown in Figure 14(B), the maximum point of the fuel impact on the ball valve at 0.01 ms (the moment the ball valve opened) was point j (the gap between the ball valve and the sealing cone), and the maximum value was −200 MPa. The figure shows that the cavitation phenomenon could effectively reduce the hydraulic impact on the sealing cone and the ball valve.

The pressure variation in the control chamber was studied based on the calculation results of cavitation and no cavitation with the same boundary conditions. During the opening of the control valve, the average pressure in the control chamber without cavitation was always smaller than the average pressure with cavitation, and the relative value reached 29.1% at 0.01 ms (the moment the ball valve opened). The relative values of the pressure in the control chamber with and without cavitation gradually increased with the increase in the vapor volume fraction, the difference between the two gradually became stable after the vapor volume fraction stabilized, and the difference between the two varied with the change in the vapor volume fraction.

Ignoring the frictional force in the direction of motion, the equation of motion of the needle valve could be expressed according to the force analysis of the injector needle valve body:

\[
m \frac{d^2h}{dt^2} = p_r A_N - p_{cc} A_N - k (h + h_0) - G - \int \frac{dh}{dt} (18)
\]
In Eq. (18), \( m \) is the mass of the needle valve component, \( p_r \) is the rail pressure, and \( p_{cc} \) is the control chamber pressure. \( A_{Ne} \) is the effective pressure-bearing area of the needle valve in the fuel tank, \( A_N \) is the effective pressure-bearing area of the needle valve in the control chamber, \( k \) is the needle valve spring elasticity coefficient, \( h \) is the lift of the needle valve, \( h_0 \) is the original shift of the needle valve spring, \( G \) is the gravity of the needle valve components, and \( f \) is the friction coefficient of the needle valve.

It could be determined from the needle valve motion equation that the control chamber pressure \( p_{cc} \) changes determined the movement of the needle valve. Hence, the adjustment of the control chamber pressure changes for the injector was used to achieve control of the movement of the needle valve. Therefore, according to the above continuity equation, it could be seen that the needle valve state depended on the pressure difference between the control chamber and the fuel tank, the higher the pressure difference was, the faster the needle valve moved.

In connection with this, as the control piston movement speed was also accelerated, the injector response speed increased. In this research, the control valve structure and the rail pressure were the same. Figure 15 shows that the cavitation phenomenon caused the pressure in the control chamber to increase. Therefore, the pressure difference between the control chamber and the fuel tank was significantly reduced compared with the absence of cavitation, and the cavitation phenomenon significantly reduced the response speed of the needle valve.

### 3.2.2 Analysis of fuel flow velocity in control valve

It can be seen from Figure 17 that when the control valve was open, the velocity of each plane (shown in Figure 16) with and without cavitation was the same because there was less cavitation. As the lift of the ball valve gradually
increased, the average velocity of the fuel without cavitation was greater than that of the fuel with cavitation, so the generation of cavitation reduced the flow rate of the fuel in the control valve. As the control valve opened, the pressure loss gradually decreased, and consequently, the relative values of the velocity of each cross-section with and without cavitation decreased. After 0.054 ms, as shown in Figure 17(A) and (C), the average velocities of the planes (A and C) at the OZ and OA with cavitation and without cavitation had the same trend.

After the ball valve reached the maximum lift, the average fuel flow velocity of plane D first increased and then stabilized, which was because cavitation occurred at the throttle orifice wall, causing the space at plane D to be occupied by fuel vapor and causing the liquid fuel flow through the space of plane D to become smaller, so the average fuel velocity increased. The cavitation could effectively reduce the velocity fluctuation in the pilot orifice is displayed in Figure 17(B).

### 3.2.3 Mass flow rate during the opening of the control valve

The mass flow rate was an important parameter for analyzing fluid flow characteristics. During the opening of the control valve, although the differential pressure decreased, the mass flow rate of the fuel at the OA increased with time, which was caused by the rapid expansion of the fuel flow through the area caused by the opening of the control valve. As can be seen in Figure 18, the mass flow rate of the OA with cavitation had a similar trend with no cavitation, but its value was smaller than that with no cavitation. Before 0.002 ms, the mass flow rates with and without cavitation were the same because the cavitation at the gap between the ball valve and the sealing cone was not strong, and the volume fraction of vapor in the gap was less than 1%. After 0.002 ms, the vapor volume fraction of the fuel increased sharply, which led to a decrease in the mass flow rate at the OA, with the largest difference at 0.015 ms compared with the mass flow rate without cavitation, with a value of 32%. The results indicated that cavitation could significantly reduce the mass flow rate at the OA when the vapor volume fraction reached 0.1% in the control valve.

### 3.3 Effect of inlet pressure on cavitation

Figure 19 shows the comparison of the average volume fraction of the vapor in the control valve at different inlet pressures. The average volume fraction of the vapor in the control valve first increased, then decreased, and finally plateaued. The growth rate of the average volume fraction of vapor and its peak value increased as the pressure of the inlet increased. However, the rate of decrease in the average volume of the vapor decreased as the pressure increased. The rate of decrease in the volume fraction of the vapor at the inlet pressures of 240 and 260 MPa was much greater than that of the other inlet pressures.

The comparison of the mass flow rate at the outlet of the control valve at different inlet pressures are presented in Figure 20. The larger the inlet pressure value of the control valve was, the shorter the time taken for the mass flow rate at the outlet to reach its maximum value was, and the earlier the reflux was generated at the outlet. As can be seen from Figures 19 and 20, when the cavitation in the control valve continued to occur and the vapor gradually filled the space, negative pressure appeared in the chamber and reflux was generated. However, with the reflux of the fuel, the growth rate of the bubbles in the control valve was smaller than the breakage rate, so the vapor in the control valve gradually decreased, and when the pressure...
inside the valve tended to stabilize, the outflow of the fuel at the outlet and the reflux rate tended to stabilize, the vapor volume fraction tended toward a stable value, and the bubble growth rate tended to balance with the breakage rate.

A comparison of the mass flow rates at the OA of the control valve for different pressures of the inlet is shown in Figure 21. The mass flow rate at the OA continued to increase as the ball valve was raised. After the ball valve reached a maximum lift of 0.06 mm, the mass flow rate at the OA reached a peak, and the higher the inlet pressure was, the faster the peak was reached. The graph clearly shows that the mass flow rate at the OA increased as the inlet pressure increased. The mass flow characteristics of the OA at an inlet pressure of 260 MPa were better than those for other cases, with the mass flow rate reaching a peak of 18.08 g/s at 0.06 mm of ball valve lift (37% higher than the inlet pressure of 100 MPa).

The curves of Figure 22 display the increase in the mass flow rate of the OA for every 10-MPa increase in the inlet pressure. From the figure, it can be determined that the increase in the mass flow rate of the OA gradually
decreased when the inlet pressure gradually increased from 100 to 260 MPa, which meant that the higher the inlet pressure was, the smaller the increase in the mass flow rate of the OA was. We could see that the mass flow rate increase in the OA was larger for the inlet pressure from 100 and 240 MPa, while the mass flow rate increase in the OA decreased significantly for the inlet pressures of 240 and 260 MPa.

Figure 23 illustrates the peaks of the average volume fraction of fuel vapor in the control valve and the mass flow rate of the OA at different inlet pressures. As the inlet pressure increased, the peaks of the average volume fraction of fuel vapor in the control valve and the mass flow rate of the OA gradually increased, which indicated that the cavitation in the valve gradually increased with the increase in the inlet pressure during the opening stage of the ball valve.

Figure 24 describes the cavitation phenomenon in the radial direction of the control valve at different inlet pressures for the cavitation steady state. Five planes (named 1, 2, 3, 4, and 5) are shown in the figure. These planes were equally spaced from each other with an average spacing of 0.3 mm to determine the radial distribution of cavitation.
Bubbles could be found in the planes, and bubble rings appeared that were attached to the inner and outer boundaries of the planes and extended toward the outlet of the control valve. As the pressure at the inlet increased, the bubbles on the surfaces of Planes 4 and 5 also gradually increased, and the volume fraction of the vapor on both surfaces gradually increased. In the radial direction, the vapor–liquid interfaces in the planes approached each other and finally intersected. The increase in the inlet pressure caused the cavitation to extend upward along the seal cone, and the cavitation ring area and the cavitation intensity increased.

Figures 25 and 26 present the average volume fraction of vapor for the five different faces and the average volume fraction of vapor for the five faces at different inlet pressures. As can be seen from the figures, after the vapor volume fraction in the control valve reached a stable value, an increase in the inlet pressure had a smaller effect on the average vapor volume fractions of Planes 1, 2, and 3 near the throttle orifice, while it had a larger effect on Planes 4 and 5, which were the two faces near the outlet of the control valve. The vapor volume fraction increased with the increase in the inlet pressure, but the average vapor volume fraction in the radial direction was the same for the inlet pressures of 240 and 260 MPa, which indicated that the inlet pressure greater than 240 MPa did not cause the increase in the vapor volume in the radial direction in the control valve.

4 CONCLUSION

In this research, CFD numerical simulation was used to study the evolution of cavitation in a fuel injector control valve, the flow characteristics of the fuel, and the effect
of the high-pressure inlet on the degree of cavitation. The main conclusions were as follows.

1. During the opening of the control valve, a high-pressure jet and cavitation were formed at the gap between the sealing cone and the ball valve. When the ball valve was opened, the upstream flow path where the ball valve was located was quickly occupied by vapor with the rise of the ball valve. Then, the reflux generated at the outlet caused the bubble at the top of the ball valve to gradually break up and disappear, and the volume of fuel vapor was reduced. When the ball valve was completely open, the pressure inside the valve was stable and the cavitation also tended to be stable, but inside the OA and the throttle orifice, the cavitation was gradually enhanced due to the existence of the low-pressure area.

2. The fuel reflux was accompanied by a reduction in the volume of the fuel vapor. After the fuel reflux occurred, the growth rate of the bubbles in the control valve was less than the breakage rate, resulting in a gradual decrease in the volume of vapor in the control valve. When the pressure inside the control valve stabilized, the outflow and reflux of the liquid at the outlet tended to stabilize, the average volume fraction of the vapor inside the control valve tended to stabilize with small fluctuations, and the bubble growth rate and breakage rate gradually tended to balance.

3. By comparing the hydraulic shock on the ball valve and the sealing cone, the pressure in the control chamber, the fuel flow rate in the control valve, and the mass flow rate in the OA with and without cavitation, it was found that cavitation had a significant effect on the reduction in the hydraulic shock for the ball valve and the sealing cone and stabilizing the fluctuation of the fuel velocity in the control chamber. However, cavitation prevented the flow rate of the fuel at the OZ and the OA, which caused a significant decrease in the fuel flow rate. The average pressure in the control chamber increased significantly compared with that without cavitation (by up to 29.1% at 0.01 ms when the ball valve opened), which eventually led to a slower response of the needle valve.

4. The effect of the high pressure of the inlet on the degree of cavitation in the results showed that the control valve inlet pressure gradually increased from 100 to 240 MPa in the process of increasing the steam volume fraction, and the time taken to reach the extreme value gradually became smaller. When the inlet pressure increased from 240 to 260 MPa, the change in the peak value of the average volume fraction in the control valve was small, the increase in the inlet pressure led to the cavitation phenomenon along the seal cone upward extension, and the cavitation ring area and the cavitation intensity increased.

**ORCID**

Rina Ren  https://orcid.org/0000-0003-1786-8589

**REFERENCES**

1. Wu B, Zi Z, Zou X, Li J, Jin S. Effect of diesel and gasoline blending fuel coordinate with in-cylinder charge conditions on efficient and clean combustion based heavy-duty diesel engine. Fuel. 2021;297:120790.

2. Lee J, Min K. Effects of needle response on spray characteristics in high-pressure injector driven by piezo actuator for common-rail injection system. J Mech Sci Technol. 2005;19(5):1194-1205.

3. Ramalingam S, Mahalakshmi NV. Influence of high-pressure fuel injection system on engine performance and combustion characteristics of Moringa oleifera biodiesel and its blends. Fuel. 2020;279(6):118461.

4. Gao Z, Li G, Li H, Xu C, Kang Y. Experiment investigation of injection parameters effects on injection stability of a high pressure common rail system. Sci Prog. 2021;104(1):003685042098362.

5. Chong Y, Li-yun F, Quan D, En-zhe S, Bing-qi T. Correlation analysis of factors influencing the electronic unit pump cycle fuel injection quantity under overall operating conditions for diesel engines. J Mech Sci Technol. 2012;26(9):2929-2939.

6. Wang L, Li G-X, Xu C-L, Xi X, Wu X-J, Sun S-P. The investigation of geometric parameters on the injection characteristic of the high pressure common-rail injector. J Eng Gas Turbines Power. 2019;141(2):022801.1-022801.12.

7. Wang X, Han Z, Su W. Numerical study of the impact on high-pressure and evaporating spray behavior of nozzle cavitation at typical diesel engine conditions. Int Commun Heat Mass Transfer. 2017;81:175-182.

8. Jiang G, Zhang Y, Wen H, Xiao G. Study of the generated density of cavitation inside diesel nozzle using different fuels and nozzles. Energy Convers Manage. 2015;103:208-217.

9. Asi O. Failure of a diesel engine injector nozzle by cavitation damage. Eng Fail Anal. 2005;13(7):1126-1133.

10. Hassanazadeh A, Saadat Bakhsh M, Dadvand A. Numerical study of the effect of wall injection on the cavitation phenomenon in diesel injector. Eng Appl Comput Fluid Mech. 2014;8(4):562-573.

11. Jia M, Xie M, Liu H, Lam W-H, Wang T. Numerical simulation of cavitation in the conical-spray nozzle for diesel premixed charge compression ignition engines. Fuel. 2011;90(8):2652-2661.

12. Cao T, He Z, Zhou H, Guan W, Zhang L, Wang Q. Experimental study on the effect of vortex cavitation in scaled-up diesel injector nozzles and spray characteristics. Exp Thermal Fluid Sci. 2020;113:110016.

13. Lešnik L, Keğl B, Bombeck G, Hočevar M, Biluš I. The influence of in-nozzle cavitation on flow characteristics and spray break-up. Fuel. 2018;222:550-560.

14. Ding Y, Liu R, Yao J, Zhang Q, Wang L. Stellite alloy mixture hardfacing via laser cladding for control valve seat sealing surfaces. Surf Coat Technol. 2017;329:97-108.

15. An YJ, Kim BJ, Shin BR. Numerical analysis of 3-D flow through LNG marine control valves for their advanced design. J Mech Sci Technol. 2008;22(10):1998-2005.
16. Yaghoubi H, Madani SAH, Alizadeh M. Numerical study on cavitation in a globe control valve with different numbers of anti-cavitation trims. *J Cent South Univ*. 2018;25(11):2677-2687.

17. Duan L, Yuan S-Q, Hu L-F, Yang W-M, Yu J-D, Xia X-L. Injection performance and cavitation analysis of an advanced 250 MPa common rail diesel injector. *Int J Heat Mass Transf*. 2016;93:388-397.

18. Wang C, Li G-X, Sun Z-Y, et al. Effects of structure parameters on flow and cavitation characteristics within control valve of fuel injector for modern diesel engine. *Energy Convers Manage*. 2016;124:104-115.

19. White FM. *Fluid Mechanics*. McGraw-Hill; 2003.

20. FLUENT Incorporation. *Fluent User’s Guide Version 6.3*. FLUENT Inc.; 2006.

21. Launder BE, Spalding DB. *Lectures in Mathematical Models of Turbulence*. Academic Press; 1972.

22. Su H, Zhang Y, Wang S, Sun S. Numerical-experimental investigation of injection characteristics of high-pressure common rail injector. *Trans Chin Soc Agric*. 2011;42(01):22-26+30.

23. Coppo M, Dongiovanni C. Experimental validation of a common-rail injector model in the whole operation field. *J Eng Gas Turbines Power*. 2007;129(2):596.

**How to cite this article:** Ren R, Su T, Ma F, Wu X, Xu C, Zhao X. Study of the influence laws of the flow and cavitation characteristics in an injector control valve. *Energy Sci Eng*. 2022;10:932–950. doi:10.1002/ese3.1074