Numerical investigation into the cavitating jet inside water poppet valves with varied valve seat structures

Cong Yuan, Lisha Zhu, Zunling Du and Shiqi Liu

ABSTRACT
Owing to the strong three-dimensionality and transient evolution, the flow dynamics of a cavitating jet inside a water poppet valve is poorly understood, leading to insufficient basis for exploring the governing mechanism of cavitation effects. In this paper, a three-dimensional simulation considering the compressibility of each constituent phase is performed, to clarify the governing mechanisms under the cavitating flow inside two water poppet valves. The cavitation structures inside the poppet valves are primarily located at three regions and triggered by different mechanisms. The vortex cavitation, mainly confined within the free shearing layer, is due to vortex dynamics, while the attached cavitation at the poppet trailing edge and within the chamfered groove arises from flow separation. The fast laminar–turbulent transition process contributes to the three-dimensionality within the free shearing layer and the rear part of the chamfered groove. The flow separation due to the chamfered groove leads to increased velocity of the central potential core, contributing to a different flow discharge performance from that of the poppet valves with a sharp seat. In addition, the periodic variation in cavitation reveals the significant interaction between the shed cavitating vortex and attached cavitation at the poppet trailing edge. In conclusion, the change in velocity distribution due to different poppet valve seat structures leads to variation in flow discharge performance, and the vortex dynamics makes sense for all three kinds of cavitation occurring inside poppet valves.

1. Introduction

Explicit recognition of the cavitation phenomenon inside various hydraulic components has achieved a general consensus for a long time. Cavitation has been confirmed as the fundamental cause of various deteriorations in performance, including flow collapse, increased vibration, elevated noise level and flow pulsation. As the main stage of a pressure relief valve, the poppet valve has to endure an extremely high pressure difference, even under normal operation. The high susceptibility to cavitation inside a poppet valve contributes to the strongest noise or vibration in hydraulic control units, causing difficulties in control precision and accuracy (Gao et al., 2006). Despite the optimum properties of mineral oil as the fluid medium in a hydraulic system, water hydraulics are attracting increasing interest, particularly in ocean engineering, such as underwater cleaning and underwater hydraulic devices. However, the poor properties of water pose a series of challenges in water hydraulic technology. Besides its weaker lubrication ability and higher corrosiveness, water exhibits a significantly higher tendency toward cavitation, compared to mineral oil. Thus, cavitation is of the most serious problems in water hydraulics. Backe (1999) pointed out that cavitation is the primary factor in the reduced lifespan of water hydraulic components. For these reasons, continuing efforts have been put into researching the cavitating jet inside a poppet valve, to quantify the flow dynamics with the occurrence of cavitation and explore the underlying mechanism of the induced cavitation effect.

To quantify various cavitation effects, a series of experimental studies was performed as the earliest investigations into the cavitation phenomenon inside a poppet valve. Oshima and Ichikawa (1985a, 1985b, 1986) provided the first visual evidence of cavitation inside a poppet valve, and revealed the overall distribution of cavitation and the induced effect on flow performance. Kumagai et al. (2016) recorded the transient evolution of cavitation bubbles and the induced vibration under different operating conditions, with the aim of exploring the periodic characteristics of the flow force. Washio et al. (2010) proposed an explanation for flow collapse of
a poppet valve, and developed a new method in flow performance prediction. Yi et al. (2015) demonstrated the intensification of vibration due to the presence of cavitation in a poppet valve. Even though both the cavitation morphology and the vapor distribution in a poppet valve have been revealed through accumulated experimental evidence, the formation process has remained poorly understood until a recent study. Yuan et al. (2019a) clarified the close correlation between cavitation development and coherent vortex evolution for oil cavitating flow inside a poppet valve, based on instantaneous experimental images together with a transient simulation.

Owing to the limitations of experimental technology, most of these preliminary attempts were confined to empirical documentation of the tendency of the flow performance due to cavitation. However, with continuous improvements in computer technology and fluid mechanics, computational fluid dynamics (CFD) has found wide application in engineering owing to its ability to resolve microscale flow structures (Mosavi et al., 2019; Ramezanizadeh et al., 2019). Ueno et al. (1993) reported enhanced suppression of cavitation in a poppet valve through structural optimization derived from flow simulation results and visual observation results. Nie et al. (2006) studied the correlation between critical cavitation number and geometric dimension of a two-stage poppet valve, and established an optimization criterion in terms of cavitation suppression. Liang et al. (2016) reported the dynamics of cavitating flow inside a water poppet valve under fluctuating inlet pressure. Han et al. (2017) provided a comprehensive study of the flow dynamics inside water poppet valves with different structure, on the basis of flow simulation.

The latest progress in CFD technology injected new momentum into the exploration of the physical process underlying the cavitating jet flow through small-sized orifices. Egerer et al. (2014) derived a homogeneous model of cavitating flow, which was used to investigate the cavitating jet through a square nozzle. The results distinguished the formation process of cavitation for two different flow conditions. With a detailed comparison between the vortex string observed in experiments and the flow streamlines of numerical simulation, Reid et al. (2014) succeeded in interpreting the origin of the cavitating lobes in a diesel fuel injector. Koukovinis et al. (2017) proposed a similar numerical method, and studied the correlation between the cavitation evolution and turbulence distribution. Previous experimental evidence (Kumagai et al., 2016) indicated that an intimate inter-relationship exists between the periodic behavior of the cavitating bubbles and the induced pressure fluctuation inside a poppet valve, but the underlying mechanisms associated with the microscopic flow dynamics have not been thoroughly clarified. Notably, an earlier study on numerical methods for a transitional cavitating jet (Yuan et al., 2019b) motivated a two-dimensional transient simulation of the cavitating jet inside an oil poppet valve, which revealed the complicated correlation between cavitation evolution and flow performance. However, owing to the much smaller viscosity of water fluid, the unsteady behavior of a water cavitating jet may display a substantial difference from that of an oil cavitating jet, and simulation of the water cavitating jet requires both an elaborate numerical treatment and significantly more computational resources to resolve the small-scale vortex cavitation structure featured with both three-dimensionality and highly transient characteristics.

The extensive incorporation of CFD methods into cavitating flow not only provided an insight into the internal flow dynamics, but also added extra impetus to the proposal of creative approaches to the cavitation-related phenomenon. Nie et al. (2006) clarified the optimal structural parameters for a two-stage water poppet valve with superior cavitation suppression capability, based on pressure distributions obtained from CFD methods. Aung and Li (2014) and Yang et al. (2015) proposed an innovative flapper shape to remove the attached cavity on the corner of the flapper edge for a typical flapper-nozzle valve, based on a corresponding simulation. Zhang et al. (2018) revealed the role of an obstacle in the inhibition of detached cavitation in a water hydrofoil, through numerical simulation. Macian et al. (2003) analyzed the flow structure of a diesel nozzle by means of CFD, and proposed the effective role of converging holes in cavitation suppression. The difference in flow performance due to variation in geometric structure had been noted in earlier experiments (Oshima et al., 2001). However, insufficient knowledge on the internal flow dynamics poses a huge barrier to the interpretation of such differences, contributing to a deficient basis for the derivation of feasible optimization in geometric structures.

Acquisition of comprehensive data of the flow field through experimental measurement is difficult for the highly unsteady and three-dimensional (3D) cavitating jet through a poppet valve, since the typical dimensions of a real-size prototype are in the order of $10^{-3}$ m or even smaller, so that standard experimental methods are generally unfeasible. The present state of the art usually incorporates numerical simulation as a complement to experimental results, to enable a more detailed view of the internal flow structure. However, the complicated vortex cavitation was inadequately treated in most of the previous numerical studies on water poppet valves, possibly owing to constraints on computational resources and numerical methods. Encouraged by the unclear mechanism pertaining to the flow dynamics of a cavitating jet
inside a water poppet valve and the progress achieved in flow dynamics inside oil poppet valves, a simulation considering both the cavitation phenomenon and vortex characteristics corresponding to an earlier experiment (Oshima et al., 2001) is carried out in the current study, to shed some light on the complicated flow dynamics inside. First, the simulation data are validated through a comparison with available experimental results, in terms of cavitation morphology and pressure distribution. Subsequently, the origin sources of the cavitation structures are clarified with incorporation of associated flow structures. The critical flow structures are employed to study the difference in discharge performance due to variation in the valve seat structure. Finally, the dynamic behavior of the cavitating flow is investigated, including the periodic behavior of cavitation and the interaction between cavitation structures originating from separated regions. These efforts complement the present knowledge on cavitation jets inside hydraulic components, by clarifying the governing mechanisms behind the cavitation behavior and the physics relevant to the differences in the valve seat structure.

2. Mathematical model

The numerical method used in this study involves the volume of fluid (VOF) algorithm with incorporation of an extra cavitation model, which is responsible for the prediction of cavitation evolution. Since two phases are present inside the poppet valve, only one single phase transport equation is required to represent the cavitating flow.

\[
\frac{\partial \alpha_l}{\partial t} + \nabla \cdot (\alpha_l U) + \nabla \cdot \left[ \alpha_l \alpha_v \left( \nabla \cdot U \right) \right] = \alpha_l \alpha_v \left( \frac{\rho_v}{\rho_v} - \frac{\rho_l}{\rho_l} \right) + \alpha_l \left( \nabla \cdot U \right) + m \left[ \frac{1}{\rho_l} - \alpha_l \left( \frac{1}{\rho_l} - \frac{1}{\rho_v} \right) \right]
\]  

(1)

where the liquid water phase is represented by the subscript \( l \), and the vapor water phase by the subscript \( v \). The third term on the left-hand side of Equation (1) consists of the relative velocity between the liquid and vapor phases indicated by \( U_r \), to achieve a thin interface between phases (Ubbink & Issa, 1999). The expression for such relative velocity is given by

\[
U_r = c_1 |U| \frac{\nabla \alpha}{|\nabla \alpha|}
\]  

(2)

The local velocity is determined by pressure, viscous shearing effect and surface tension, given by

\[
\frac{\partial \rho U}{\partial t} + \nabla \cdot (\rho U \otimes U) = -\nabla p + \nabla \cdot \tau + \sigma \kappa \nabla \alpha
\]  

(3)

The third term on the right-hand side of Equation (3) is used to model the surface tension effect, and the curvature of the interface is approximated with the phase gradient as

\[
\kappa = \nabla \cdot \left( \frac{\nabla \alpha}{|\nabla \alpha|} \right)
\]  

(4)

Establishment of the pressure equation (Demirdžić et al., 1993) first involves a discretization of the momentum equation:

\[
U_p = \frac{H(U)}{a_p} - \frac{1}{a_p} \nabla p
\]  

(5)

A divergence procedure is performed to Equation (5). In conjunction with the velocity divergence obtained from the phase transport equations, the following pressure equation is derived:

\[
- \left( \frac{\alpha_l}{\rho_l} \left( \frac{\partial \rho_l}{\partial t} + \frac{\alpha_v}{\rho_v} \left( \frac{\partial \rho_v}{\partial t} \right) \right) + m \left( \frac{1}{\rho_l} - \frac{1}{\rho_v} \right) \right)
\]

\[
= \left( \nabla \cdot \left( \frac{H(U)}{a_p} \right) - \nabla \cdot \left( \frac{1}{a_p} \nabla p \right) \right)
\]  

(6)

The compressibility of both phases in the pressure equation assumes an implicit treatment to enable the transonic calculation of the mixture region:

\[
- \left( \frac{\alpha_l}{\rho_l} \frac{\partial \psi_l}{\partial t} + U \cdot \nabla (\psi_l p) \right) + \frac{\alpha_v}{\rho_v} \frac{\partial \psi_v}{\partial t} + U \cdot \nabla (\psi_v p) + m \left( \frac{1}{\rho_l} - \frac{1}{\rho_v} \right)
\]

\[
= \left( \nabla \left( \frac{H(U)}{a_p} \right) - \nabla \cdot \left( \frac{1}{a_p} \nabla p \right) \right)
\]  

(7)

where the perfect fluid assumption is applied. Compressibility assumes a constant value:

\[
\rho = \psi
\]

Zhang et al. (2017) reported that incompressible treatment would lead to decreased accuracy in predicting the flow dynamics in a hydraulic pump. In a previous study (Yuan et al., 2019b), the simulation results in a similar study case were compared between the compressible method and the incompressible method, which confirmed that a vital flow structure associated with the Kelvin–Helmholtz could only be treated appropriately by the compressible method. The fundamental reason for this is the extra shear-induced instability beyond the incompressible method, as reported in a theoretical analysis by Furukawa and Tanaka (2006). In the case of shear-induced cavitation, the ignorance of such extra term leads to inaccurate prediction of the vortex structure, resulting in the failure in modeling the cavitation behavior.
The Schnerr–Sauer model, derived from the generalized Rayleigh–Plesset equation (Schnerr & Sauer, 2001), is applied for evaluation of the interphase mass transfer:

\[
\begin{aligned}
\dot{m}^- &= \frac{3\rho_l \rho_v}{\rho} \alpha_\gamma (r R_b) \sqrt{\min(p - p_v, 0)} \\
\dot{m}^+ &= \frac{3\rho_l \rho_v}{\rho} \alpha_\gamma (r R_b) \sqrt{\max(p_v - p, 0)}
\end{aligned}
\]  \hspace{1cm} (8)

More details of the numerical method can be found in Yuan et al. (2019b).

3. Simulation configuration

The mathematical model is implemented on the open-source platform OpenFOAM v1612. The Crank–Nicolson method with two-order accuracy is used for discretization of the temporal terms, while the divergence term utilizes the limitedLinear method with a factor of 1. The phase transport equation is solved with the MULES algorithm, and time advancement is addressed with the PIMPLE loop, a combination of PISO and SIMPLE methods. The detailed calculation flowchart for the derived governing equations is presented in Figure 1.

According to previous studies on oil cavitating jets through small-size orifices, the cavitation phenomenon may be comparable to or even outweigh the turbulence effect, largely because cavitation evolves within a much shorter time than the transition time of turbulence (Schmidt & Corradini, 2001). Since the cavitation intensity in water poppet valves is larger than that in oil poppet valves owing to the higher susceptibility of liquid water to cavitation, the current simulation assumes the direct simulation strategy, for obtain a correct representation of the cavitation phenomenon.

The geometric structure of the poppet valve is shown in Figure 2, which also includes the grid model. The computational domain covers a sector of 90° for the sharp valve seat case, and a sector of 45° for the chamfered valve seat case. The total amounts of the calculation grid are approximately 5 million for the sharp valve seat case and 4.8 million for the chamfered valve seat case. The inlet is set as a fixed total pressure including the static and dynamic pressure, and the outlet boundary employs a non-reflective constant pressure condition. The total pressure of the inlet is set constantly as 50 bar, while the outlet pressure is adjusted to change the pressure drop. The two sides are specified as a symmetry plane, while the other boundaries are set as non-slipping walls. Indication of the boundary conditions is included in Figure 2.

The experimental data are cited from a previous study with a half-cut prototype (Oshima et al., 2001). In the experimental device, a transparent plate is clutched on to the half-cut model so that the cavitation morphology of the internal jet flow can reflect the shed illumination and appear in the recorded images of the fast-speed camera. To ensure a clear recording of the cavitation morphology, the shutter speed of the camera and the frequency of the stroboscopic light are adjusted so that only one flash can be caught during a single exposure event. Meanwhile, a micro-hole drilled on the transparent plate allows measurement of the pressure via transmitters. The position of the transparent plate with respect to the half-cut model can be changed precisely with corresponding adjustment screws and displacement gauges, and consequently the drilled hole can be moved delicately on the half-cut model, enabling the measurement of time–mean pressure along the poppet surface or valve seat surface. The flow rate is detected by an electromagnetic flowmeter located downstream of the outlet port of the valve prototype. The fluid medium used in the experiment and simulation is tap water, with the properties listed in Table 1. In the experiment, the inlet pressure is kept constant at 50 bar with a varying outlet pressure, so that the adjusted pressure drop produces flow conditions with different strengths of cavitation behavior.

Figure 1. Flowchart of the numerical method.
4. Results and discussion

4.1. Validation of calculation results

4.1.1. Cavitation morphology

Figure 3 illustrates the cavitation morphology predicted by the numerical method for the case of the sharp valve seat, with a comparison to the experiment under conditions of the same pressure drop. For a pressure drop of 45 bar, the cavitation is incepted at the free shearing layer slightly downstream of the sharp valve seat corner in both the experimental image and the simulation result, rather than being attached to the valve seat corner. In the simulation result, the distance from the initial cavitation to the sharp valve seat is approximately 0.20 mm, compared to the distance of 0.20–0.25 mm in the experimental image. In the middle, between the valve seat and the poppet trailing edge, the sectional profile of the cavitation structure displays a size comparable to the valve seat opening, i.e. the potential core thickness, indicating a fast growing evolution from the inception nuclei. At the region prior to the trailing edge of the poppet, cavitation features with a dense distribution, as shown by the brightly dazzling region in the experimental image, which corresponds to the gathering of cavitating bubbles with the appearance of a bubble cloud. However, significant discrepancy is observed between the experimental image and the simulation result. Cavitation morphology in the image assumes a continuous organization in the

Table 1. Water physical fluid properties.

| Parameter                      | Value               |
|--------------------------------|---------------------|
| Liquid type                    | Water               |
| Liquid density                 | 1000 kg/m³          |
| Liquid dynamic viscosity       | 0.001 N s/m²        |
| Vapor pressure                 | 2000 Pa             |
| Vapor density                  | 0.02558 kg/m³       |
| Vapor dynamic viscosity        | 0.00126 N s/m²      |
| Temperature                    | 313 K               |
| Surface tension                | 0.0782 N/m          |

Figure 2. Geometric structure of the poppet valve and the grid model, and illustration of boundary condition specification. Dimensions are in mm.
streamwise direction. In contrast, the predicted cavitation, even in the downstream region occupied by the bubble cloud, exhibits a clear separation, producing an intermittent pattern. This discrepancy will be analyzed in subsection 4.2.1. At the trailing edge of the poppet, the bubble cloud meets the attached cavitation on the cylinder wall, and extends beyond the poppet trailing edge along the streamwise direction of the potential core. This rear part region features sparsely distributed large-size bubbles, and covers an area of 4.0 mm × 3.9 mm in the image and 4.1 mm × 3.5 mm in the simulation result.

At a pressure drop of 40 bar, cavitation is triggered at a region further downstream from the valve seat, compared to the 45 bar pressure drop case. Clear evidence of small-size transitory cavitation in the presented simulation result can be found in the middle region between the valve seat and the poppet trailing edge, mainly located at the free shearing side of the potential core. However, in the experimental image, footprints of such transitory cavitation are lacking, mainly because the short transit time and small dimensions of such cavitation could not produce sufficiently strong reflection of the shed light during the exposure time of the camera. At the poppet trailing edge, attached cavitation with comparatively larger size is present in both the simulation result and the experimental image.

Figure 3. Cavitation morphology identified by iso-surface $\alpha = 70\%$ in simulation for the sharp valve seat case. (The experimental image by Oshima et al., 2001 is included for comparison, with copywrite permission from River Publishers.)

Figure 4 illustrates the cavitation morphology predicted by the numerical method for the case of the chamfered valve seat, with a comparison to the experiment under conditions of the same pressure drop. For a pressure drop of 45 bar, according to both the experimental image and the simulation result, an extra appearance of a cavitation structure as attached cavitation on the chamfered valve seat is triggered at the inlet corner of the chamfered groove, in addition to the free shearing layer of the potential core and the trailing edge of the poppet. According to the experimental image, the cavitation at the free shearing side of the potential core downstream of the chamfered groove outlet has a continuous extension to the attached cavitation inside the chamfered groove, and the cavitation close to the outlet of the chamfered groove is mainly concentrated at the free shearing side of the potential core, similarly to the case of the sharp valve seat. This feature of cavitation morphology is also captured by the simulation morphology. At a region further downstream, prior to the poppet trailing edge, densely distributed cavitation in the form of a bubble cloud is formed with a continuous appearance in the experiment, compared to the intermittent pattern in the simulation result. This discrepancy is also mentioned in the sharp valve seat case, and will be explored in subsection 4.2.2. The bubble cloud extends beyond the poppet trailing edge, with a direction slightly biased toward the poppet.
cylinder. Akin to the sharp valve seat case, the rear part of the cavitation structure is comprised of sparsely distributed large-size bubbles, and covers an area of an area of 4.4 mm × 3.4 mm in the image and 4.6 mm × 3.2 mm in the simulation result.

At a pressure drop of 40 bar, the attached cavitation inside the chamfered groove has the clearest traces in both the experimental image and the simulation result. The cavitation at the free shearing side of the potential core identified in the experimental image is also predicted in the simulation result. In comparison, the attached cavitation at the trailing edge is negligibly small in both the experimental image and the simulation result.

According to the pressure distribution along the valve seat measured in the experiment, the case of the chamfered valve seat exhibits a sudden pressure drop at the inlet of the annular clearance, which is in agreement with the attached cavitation in the recirculating region. This tendency is captured by the numerical method with an evident deviation. For the 20 bar pressure drop, the simulation predicts a stronger negative pressure peak, but favorable agreement is achieved for the 39 bar pressure drop. Concerning the pressure distribution on the poppet surface, the simulation generally predicts a stronger decreasing trend, and the deviation in negative pressure peak becomes negligible with the experiment at a 39 bar pressure drop. However, the low-pressure region extends further downstream in the experimental data. The disparity between the simulation and the experiment in the chamfered valve seat case may be attributed to the ideal geometric structure in the simulation model, which usually assumes a trivial rounded feature in real applications and produces a significant influence on the flow through small-size orifices.

Concerning the deviation in pressure distribution on the poppet wall surface between the experiment and the simulation, the poppet valve with a chamfered valve seat at a pressure drop of 39 bar and 0.4 mm openness is selected.
Figure 5. Pressure distribution along the poppet surface and seat surface. (The experimental data from Oshima et al., 2001 are included for comparison.)
as the first study case. As shown in Figure 6, the pressure distribution tends to knit into a single line with a poppet wall thickness below 0.0008 mm. But the deviation remains remarkable. It is possible that the unavoidable presence of a minor rounding at the chamfered groove inlet, which is neglected in the simulation but reported as a suppression of attached cavitation, retards the commencement of attached cavitation. Thus, the downstream low-pressure region on the poppet wall is shifted evidently in the experiment. The second study case indicates that the pressure distribution achieves a general convergence with a poppet wall thickness of 0.0005 mm, but 0.0002 mm is specified for the meshes used for simulation to capture the small-size vortex structure.

Figure 6. Mesh independence study for two study cases with a chamfered valve seat. The dimensions of the wall layer thickness are in mm.

4.2. Governing mechanism of cavitation

4.2.1. Sharp valve seat case

Figure 7 shows a 3D view of the cavitation distribution for the sharp valve seat case under varied pressure drop, with the inclusion of the velocity magnitude contour at the corresponding time instant. According to the underlying mechanism and location, the cavitation structure inside the poppet valve with a sharp valve seat is divided into two categories, for separate examination, namely the attached cavitation at the poppet trailing edge and the cavitation prior to the trailing edge.

The most noticeable flow dynamics appearing in all four exhibited operating conditions with varied pressure drop refers to the flow separation of the potential core at the trailing edge of the poppet, which is extensively
observed in water hydrofoil cavitation and proposed as the main mechanism for the attached cavitation on the leading edge of the hydrofoil (Arndt, 2002). From the presented cavitation distribution in Figure 7, attached cavitation is persistently present beyond the trailing edge of the poppet, coincident with the flow separation region. Furthermore, the attached cavitation at the trailing edge generally has a larger size than the cavitation prior to the poppet trailing edge, particularly under the operating conditions with low pressure drops of 40 and 38 bar (Figure 7c, d). It can also be inferred that the attached cavitation is first produced with the increase in pressure drop.

According to the cavitation morphology in Figure 7, the cavitation structure prior to the poppet trailing edge is positioned persistently at a distance from the poppet surface, for each of the four operating conditions. With incorporation of the corresponding velocity magnitude contour, it can be clearly seen that vortex cavitation is dominantly limited within the free shearing layer of the potential core. Furthermore, the cavitation within the free shearing layer, even for the bubble cloud stage at a pressure drop of 45 bar, assumes an intermittent pattern, instead of a streamwise continuous organization connecting the inception nuclei to the downstream bubble cloud. Such a discontinuous feature can be more

Figure 7. Three-dimensional view of cavitation morphology and two-dimensional intersection of velocity distribution contour. (Cavitation morphology identified by iso-surface $\alpha_1 = 70\%$, openness is 0.6 mm.)
clearly identified from the overall view of the cavitation structure in Figure 8. Concerning the case with a 45 bar pressure drop (Figure 8a), the cavity at the free shearing side of the potential core is clearly identified as three stages with different organization: the cavitating ring with the appearance of a stretched thin layer upstream, the separated cavitating vortex ligament in the intermediate region, and the bubble cluster or cavitating vortex lumps downstream. In the upstream region near the valve seat, the inceptive cavity appears as a thin layer with a smooth surface. In the intermediate region, the cavity with the appearance of ligaments assumes an elliptical sectional contour, as shown in Figure 7, with a rugged surface. In the downstream region, the circumferential extension of the cavity is totally disintegrated, and cavitation is organized as a cluster of disordered bubbles. This distinctive organization may have a close relationship with the variation in the flow dynamics, which will be investigated as follows.

Figure 8 shows the vortex structure identified through the Q-criterion at the same instant, with the pressure drop in accordance with Figure 7. Comparison between the cavitation structure and the vortex structure reveals an important fact, that the cavitation within the free shearing layer prior to the poppet trailing edge is persistently located inside vortex structures. The remarkable
coincidence of the cavity structure within the large-scale vortex core demonstrates the close correlation between cavitation and vortex dynamics. First, coherent vortex structures featured with a local pressure drop in the core region are generated at the sharp corner, which obtains a sufficient level with gradual development of the vortex strength and triggers the earliest traces of cavitation. Second, the initial coherence of the vortex with the organization of a circumferential ring gradually disintegrates into several individual pieces of ligament, and assumes an elliptical shape owing to the reduced shearing effect. It should also be noted that each individual vortex ligament shares favorable but evidently decreased coherence with its circumferentially neighboring ones. And the separated elliptical cavity ligaments within the vortex exhibit less distinct circumferential order than the neighboring ones. The development of elliptical ligaments is accompanied by the formation of a steamwise vortex structure. Third, further downstream, the bubble cluster is either within the vortex core or surrounded by the ambient chaotic vortex lumps, as shown in Figure 8(a).

Furthermore, the above-mentioned variation in cavitation organization at different longitudinal positions can be explained based on the vortex dynamics and velocity distribution. The thin layer cavity in the upstream region has relevance to the strong vortex stretching owing to the high shearing effect of the free shearing layer of the potential core. The initial coherent vortex ring originating from Kelvin–Helmholtz instabilities exhibits favorable circumferential symmetry. Such two-flow dynamics contribute to the circumferential coherence and the smooth surface of the thin layer cavity. In the intermediate region, the vortex goes through an anti-cascade process through vortex pairing, and thus the growth of the cavity cross-section is expected. Moreover, surface tension force and the attenuated shearing effect of the diverging flow are responsible for the recovery toward a circular shape of the cavitating vortex cross-section. Downstream, the vortex ring is disrupted into disordered vortex lumps, generating several bubble clusters. The velocity magnitude contour provides clues for the potential core. Compared to the non-cavitating or weakly cavitating case, the potential core exhibits a clear break-off behavior with the formation of bubble clusters at high cavitation intensity with pressure drops of 45 and 42 bar. The evolution from a coherent vortex ring into cavitating vortex lumps involves a sequence of events, which will be rendered in a later discussion.

Summarizing from the analysis pertaining to the cavitation morphology with the sharp valve seat, two regions are susceptible to cavitation inception, namely the free shearing layer of the potential core and the trailing edge of the cone poppet, which are governed by vortex dynamics and flow separation, respectively. The cavitation structure within the free shearing layer is called vortex cavitation in other studies (Egerer et al., 2014), owing to its close dependence on vortex dynamics. In the experimental image of Figure 3(a), the imposition of a background cavitation structure at different azimuthal angle leads to the submergence of the intermittent pattern. Besides, the vortex cavitation, mainly located within the free shearing layer, cannot be clearly discerned.

4.2.2. Chamfered valve seat case
As shown in Figure 9, cavitation is triggered at the inlet of the chamfered groove, besides the free shearing layer and the trailing edge of the cone poppet. Similarly to the case with the sharp valve seat, the attached cavitation at the trailing edge is due to flow separation. The attached cavity inside the chamfered groove shares similarities with the inertia-induced cavitation in the spray nozzle, which is frequently associated with the vena-contracta phenomenon (Payri et al., 2012). The vena-contracta is widely confirmed as a flow structure with high susceptibility to cavitation inception, owing to the flow recirculation at the inlet corner. Consequently, cavitation inception occurs under an operating condition with a lower pressure drop, compared to the case of the sharp valve seat. Owing to the occasional detachment of the attached cavitation within the chamfered groove, the shedding vortex cavitation is transported to the downstream region beyond the chamfered groove outlet. Consequently, the vortex cavitation within the free shearing layer originates from the attached cavitation within the chamfered groove, instead of the Kelvin–Helmholtz instability as in the sharp valve seat case. Thus, the vortex cavitation has a more disperse distribution than that in the sharp valve seat case, and appears to be connected directly to the upstream attached cavitation in the experimental image as a consequence of the imposition of cavitation in the background. As demonstrated in Figure 9, the cavitation vortex structure already generated at the outlet of the chamfered groove contributes to a stronger laminar–turbulent transition process than the sharp valve seat case. In addition, the vortex structure has a larger vortex size and stronger vortex intensity than the sharp valve seat case, according to the comparison between Figures 8 and 9. The stronger transition process, as well as the larger vortex size, offers an interpretation of the more disperse distribution of vortex cavitation in the chamfered valve seat case. Figure 9(a) also reveals that vortex cavitation with small size is predicted within the wall shearing layer at a 44 bar pressure drop, which is seldom present in the sharp valve seat case at a 45 bar pressure drop. The reason for this is associated with the confinement on the potential core imposed by the chamfered groove,
which produces an elevated wall shearing intensity, particularly at the outlet region of the chamfered groove. Thus, a small-size vortex structure is formed within the wall shearing layer, with an intensity evidently larger than that in the sharp valve seat case. As shown in Figure 9(a), this vortex adjacent to the poppet wall appears as a thin layer, similar to the inception nuclei of vortex cavitation within the shearing layer of the sharp valve seat case. Since the cone poppet imposes a gradually diverging tendency on the potential core, the shearing effect within the thin wall shearing layer is kept attenuated constantly along the downstream direction. Therefore, the thin-layered vortex fails to grow into a size comparable to the counterpart on the free shearing side, explaining its infrequent presence. Apart from these minor discrepancies, cavitation governed by the vortex factor exhibits a similar pattern to the case of the sharp valve seat, including the distinct break-off of the potential core and the formation of bubble clusters.

4.3. Flow discharge performance

4.3.1. Flow rate curve

Figure 10 presents both predicted and measured flow discharge coefficients with respect to pressure drop. For the case of the sharp valve seat, the discharge coefficient displays only a slight decline with the occurrence of cavitation, while the flow choking phenomenon is quite evident in the case of the chamfered valve seat. The predicted flow rate reproduces both the flow performance tendency with cavitation and the deviation due to different valve seat structures, with an inaccuracy of roughly 6.6%. The deviation between the experiment and simulation in terms of flow performance may be ascribed to the leakage due to the slit between the half-cut prototype and the clench transparent plate, which cannot be reasonably compensated for in the calculation.

4.3.2. Difference in throttling effect due to alteration in valve seat structure

According to Figure 10, the flow performance between the sharp valve seat case and the chamfered valve seat case displays two evident differences. The first is that the flow rate of the chamfered valve seat is approximately 20% higher than that of the sharp valve seat under non-cavitating and weakly cavitating flow conditions, and the difference in flow rate is gradually decreased with the occurrence of cavitation. This issue was not addressed in prior studies (e.g. Oshima et al., 2001). Second, the flow choking phenomenon is only present in the chamfered

Figure 9. Three-dimensional view of cavitation morphology and vortex structure in the chamfered valve seat case. The cavitation morphology is identified by iso-surface $\alpha_1 = 70\%$, the vortex structure is visualized by iso-surface $Q = 1 \times 10^{10}$ and openness is 0.6 mm.
As shown in the time–average velocity magnitude contour of Figures 11 and 12, the vena-contracta phenomenon creates a recirculating region at the inlet tip on the chamfered groove, contributing to a substantial influence on the time–average velocity distribution. Figures 11 and 12 also include the streamwise velocity profile of the white line in the velocity magnitude contour. As presented in Figure 12(c), the fluid neighboring the valve seat inside the chamfered groove experiences a local acceleration at the inlet of the groove. As a consequence, the velocity profile exhibits a sharp spike. Meanwhile, the velocity profile at the poppet surface is free from such an effect, and exhibits no such spike. In contrast, the flow through the sharp valve seat experiences a sudden flow relaxation, which, together with the diverging tendency imposed by the cone poppet, leads to a local reduction in the potential core thickness. This phenomenon produces a kind of contraction of the potential core in the sectional view and a similar spike in velocity distribution. Consequently, the sectional effective area is also suppressed. According to Figure 12(c), the velocity in the central part in the chamfered valve seat case is roughly 24% higher than that in the sharp valve seat case, in addition to the larger effective cross-sectional area. According to Figure 12(f), the velocity in the central part in the chamfered valve seat case is roughly equivalent to that in the sharp valve seat case, with a comparable effective cross-sectional area. According to the flow measurement in the experiment (Oshima et al., 2001), as shown in Figure 10, the non-cavitating flow discharge coefficient in the chamfered case is approximately 25% larger than the sharp case, coincident with the difference in the central velocity magnitude. The case is similar for an openness of 0.4 mm. It is safe to conclude that flow contraction of the potential core is present in both cases. Besides, the presence of the chamfered groove produces a local flow acceleration, responsible for the larger flow discharge coefficient.

Both Figures 11(c, f) and 12(c, f) indicate that both the maximum thickness and the maximum reversal velocity of the recirculating region in the chamfered valve seat case are reduced in the cavitating flow condition. However, the recirculating region extends almost to the outlet of the chamfered groove channel in the cavitating condition, as shown in the velocity magnitude contour in Figure 12(b, e), indicating a more persistent presence of the recirculating phenomenon, responsible for choking.
the flow. Such flow physics has been widely reported previously (Yu et al., 2017), and thus no further discussion is performed in the current paper. In comparison, the flow characteristics in the sharp valve seat case are free from such a mechanism. Moreover, Figures 11(f) and 12(f) reveal that the right-hand side of the velocity profile in the sharp valve seat case shifts toward that of the chamfered case, indicating a growth in effective cross-sectional area as a result of elevated shearing strength. Consequently, the flow discharge coefficient tends to remain unaltered in the sharp valve seat case.

4.4. Flow dynamics of cavitating jet

4.4.1. Sharp valve seat case

4.4.1.1. Evolution of vortex cavitation in the sharp valve seat case. As mentioned subsection 4.2.1, the vortex cavitation at the free shearing side experiences three successive stages with different organization, which are highly correlated with the evolution of the vortex structure. Yuan et al. (2019c) clarified that the transition from wall boundary to free shearing boundary at the sharp valve seat leads to the widely known Kelvin–Helmholtz instability, which further contributes to the coherent vortex ring. As shown in Figure 13(a), the primary mechanism of cavitation inception is the pairing process of the coherent structure, which produces a sufficient pressure drop, giving rise to the dominant coherence of the initial cavitation. Furthermore, the strong shearing effect of the potential core produces an intense stretching of the cavity, which outweighs the surface tension force and contributes to the thin-layer feature of the inception cavity. Subsequently, a further merger occurs between the neighboring cavitating vortexes, which is responsible for the expansion of the cavitation inside the vortex core. Yet, the vortex ring feature can be unambiguously identified, as shown in Figure 13(b). Meanwhile, the gradual growth of the 3D vortex behaves as a perturbation to the circumferential coherence, and therefore the cavitation displays a gradual loss of circumferential coherence, contributing to the formation of cavitating vortex ligaments (Figure 13c). Further downstream, the streamwise vortex structures are elevated to a comparable level as a result of the interaction between ligaments, which disrupts the vortex ring and generates several cavitating vortex lumps, characterized by dominant three-dimensionality (Figure 13d, e). At the same time, the formation of bubble clusters is frequently accompanied by disintegration of the potential core (Figure 14). This indicates that the formation of cloud cavitation from the merger of vortex cavitation coincides with the break-off process of the potential core, which may further contribute to flow or pressure pulsation and should be investigated in the future. The rupture of the potential core into two parts also serves as a separation between the intermediate elliptical vortex ligaments and the downstream bubble cluster.

4.4.1.2. Evolution of attached cavitation in the sharp valve seat case. The attached cavitation on the trailing edge of the cone poppet appears earlier than the vortex cavitation at the free shearing side of the potential core, as shown in Figure 3. At high cavitation intensity, large-scale cavitating vortex lumps are created concurrently and transported downstream toward the trailing edge of the cone poppet. Thus, the dynamics of the attached cavitation is characterized by an interaction with the cavitating vortex lumps. Figure 15 presents the temporal variation of total vapor volume, which mainly consists of periodic large-amplitude fluctuations. To investigate the underlying mechanism for the variation in total vapor volume, the cavitation morphology at different instants within a fluctuation cycle is demonstrated in Figure 16. At T1, as shown in Figure 16(a), attached cavitation is triggered at the poppet trailing edge as a result of flow separation of the potential core. Meanwhile, the arrival of large-size cavitating vortex lumps at the edge of the cone poppet

Figure 12. Time–mean velocity magnitude contour and sectional streamwise velocity distribution. Openness = 0.6 mm.
**Figure 13.** Evolution of vortex cavitation in the sharp valve seat case with 0.6 mm openness and 45 bar pressure drop. Time interval between neighboring images is $2 \times 10^{-5}$ s. The cavitation structure is identified by the blue-colored iso-surface $\alpha_l = 90\%$, while the vortex structure is identified by the transparent white-colored iso-surface $Q = 1 \times 10^{10}$.

**Figure 14.** Break-off of potential core with evolution of cavitation. The vortical structure is identified by the transparent blue surfaces.
inevitably contributes to the growth of the attached cavitation. At T2, as shown in Figure 16(b), the cavitating vortex lumps arrive at the cone poppet edge, where attached cavitation is initiated. It should also be mentioned that the flow separation produces a recirculating vortex close to the wall surface, once the broken-off part of potential core passes the cone poppet edge. And the attached cavitation together with the corresponding recirculating vortex has merged with the lumps soon after inception, as shown in Figure 16(d), which exhibits the departure of attached cavitation from wall surface. Consequently, the self-resonant dynamics of the attached cavitation is destroyed. Meanwhile, the cavitating vortex lumps experience a substantial growth with the involvement of the attached cavitation. At T4, the vapor volume achieves the maximum magnitude with the full expanding development of the cavitating vortex lumps. Afterwards, the cavitation cloud experiences a gradual collapse. The collapse process involves a merger with the subsequently upcoming cavitating vortex lumps, which delay the collapse process (Figure 16e, f). As a consequence, the decreasing tendency of the total vapor volume slows down temporarily at the corresponding moment.

As discussed previously, the highly three-dimensional cavitating vortex lumps have strong asymmetry, and have a large range of characteristic parameters such as configuration, orientation, shape and convection velocity. Thus, the amalgamated structure from the interactive entrainment between vortex lumps and attached cavitation is also highly three-dimensional and asymmetric. Since the dynamics of attached cavitation involves an interaction with the cavitating vortex lumps, which play a dominant role in the variation of the total vapor volume, the periodic process occasionally exhibits significant stochastic behavior. Around the time instant 1 ms in Figure 15, the fluctuation has a rather small magnitude, indicating an unsuccessful interactive evolution of the attached cavitation. Occasionally, two successive processes are positioned rather close together, and the vapor volume variation curve will exhibit two neighboring fluctuations with a partial overlap. In an earlier experiment (Yi et al., 2015), the characteristic frequency of cavitation–induced vibration covered a wide spectrum, justifying the large dispersion of the flow dynamics in the simulation results.

### 4.4.2. Evolution of shed vortex cavitation in the chamfered case

In the chamfered valve seat case, the attached cavitation exerts a significant influence on the downstream vortex structure within the free shearing boundary layer and cavitation structure.

As shown in Figure 17, attached cavitation is recurrently shed downstream, which disrupts the formation of Kelvin–Helmholtz instability and produces the prevalent cavitating vortexes downstream of the chamfered groove outlet. Since the shed cavitating vortex shows high three-dimensionality and strong asymmetry, the vortex structure at the free shearing layer may exhibit negligible coherence. Frequently, the shed vortex assumes a circumferentially aligned string organization, which further connects with neighboring segments and contributes to a vortex ring, as shown in Figure 17(f). Because the triggered cavitation structure at the free shearing layer depends largely on the local vortex structure, which may vary remarkably from other processes, the vortex ring shares insignificant common configurations between each cycle.

Figure 18 presents the temporal variation of total vapor volume, which also includes pronounced fluctuation cycles, as in the sharp valve seat case. To explore the underlying mechanism of these large-amplitude fluctuations, the cavitation morphology at several instants within a fluctuation cycle is examined, as shown in Figure 19. Figure 19(a–c) illustrates the formation of the cavitating vortex ring from the shed cavitating vortex of the attached cavitation inside the chamfered groove, in the method described in Figure 17. From T3 to T5, as shown in Figure 19(c–e), a new vortex ring is generated from another shed cavitating vortex of attached cavitation, which moves toward the decaying vortex ring at the poppet trailing edge but involves no clear merger process between T5 and T6 (Figure 19e–g). The growth of the second vortex ring is accompanied by the formation of a third vortex ring, and in the following moment, the amalgamation between the second and third vortex rings produces a bubble cloud with an extremely high volume (Figure 19g). When passing the poppet trailing edge, the vortex cavitation similarly experiences a merger with the attached cavitation. At the end stage within the period,
Figure 16. Transient cavitation evolution at varied instants within a fluctuation cycle in the sharp valve seat case with 0.6 mm openness and 45 bar pressure drop. Time instants are indicated by (j). The cavitation structure is identified by the blue-colored iso-surface $\alpha_l = 90\%$, while the vortex structure is identified by the transparent white-colored iso-surface $Q = 1 \times 10^{10}$.

the cavitating vortex lumps experience a gradual collapse, accompanied by a decrease in total vapor volume. The detailed periodic behavior in the chamfered case can be referred to the appended video file. It is summarized that a sole period of variation in vapor volume involves multiple vortex rings originating from the recurrent shedding of attached cavitation within the chamfered groove. Similarly to the sharp valve seat case, the periodic fluctuation cycles exhibit evident variation, due to the large dispersion of shed detached cavitating vortexes in terms of characteristic parameters.

4.4.3. Interaction between attached cavitation at the poppet trailing edge and shed vortex cavitation

It was mentioned previously in subsection 4.1 that the cavitation covering a profound region beyond the poppet trailing edge features relatively large-size cavitating bubbles. Such cavitation morphology can be reasonably interpreted by the detailed evolution between the attached cavitation at the poppet trailing edge and the vortex cavitation shed upstream, as presented in Figure 20. The initial vortex attached to the poppet trailing edge has a small size and undergoes a slow growth,
Figure 17. Evolution of detached cavitation in the chamfered valve seat case with 0.4 mm openness and 45 bar pressure drop. Time interval between neighboring images is $3 \times 10^{-6}$ s. The cavitation structure is identified by the blue-colored iso-surface $\alpha_l = 90\%$, while the vortex structure is identified by the transparent white-colored iso-surface $Q = 1 \times 10^{10}$.

Figure 18. Temporal variation in total vapor volume.

Figure 18. Temporal variation in total vapor volume.

as shown in time instants (i)–(iii) of Figure 16. The approach of the shed vortex cavitation contributes to the detachment of the attached cavitation, so that the slow growth of the independent cavitation evolution is disrupted. Furthermore, the vortex cavitation merges with the small-size attached cavitation, producing a subsequent expansion of the detached cavitation. Without such an interaction, the shed vortex cavitation generally would experience a gradual collapse, since the vortex structure is progressively dissipated beyond the poppet trailing edge.

In summary, the presence of attached cavitation produces a secondary growth mechanism beyond the poppet trailing edge, which satisfactorily explains the presence of large-size cavitating bubbles located beyond the poppet edge appearing in the experimental images.

5. Conclusion

In the current paper, a numerical study is carried out to investigate the water cavitating jet inside poppet valves with different structures, with an in-depth examination of the flow dynamics. The following conclusions are drawn.

1. In the sharp valve seat case, cavitation is initiated first as attached cavitation at the cone poppet edge and vortex cavitation is produced at a higher pressure drop, mainly within the free shearing boundary of the potential core. The cavitation at the free shearing side of the potential core exhibits three different kinds of organization, depending on the streamwise position. Examination of the vortex structure
Figure 19. Transient cavitation evolution at varied instants within a fluctuation cycle in the chamfered valve seat case with 0.6 mm openness and 44 bar pressure drop. Time instants are indicated by $k$. The cavitation structure is identified by the blue-colored iso-surface $\alpha_l = 90\%$.

Figure 20. Interaction between the attached cavitation at the poppet trailing edge and shed vortex cavitation. The cavitation structure is visualized by iso-surface $\alpha_l = 90\%$. 
confirms that the alteration in cavity organization is primarily associated with the change in vortex dynamics.

(2) In the chamfered valve seat case, cavitation is initiated first at the inlet of the chamfered valve seat. The vortex cavitation within the free shearing layer has its origin in the upstream attached cavitation within the chamfered groove. The shed cavitating vortex from the inlet of the chamfered valve seat has a higher intensity than the Kelvin–Helmholtz instability, thus contributing to more dispersed cavitation distribution at the free shearing side of the potential core. In addition, small-size vortex cavitation within the wall shearing layer is created as a result of the higher local vortex intensity.

(3) The central velocity in the chamfered valve seat case is larger than that in the sharp valve seat case with a factor corresponding to the difference in flow discharge coefficient. The difference in flow dynamics associated with cavitation causes variation in the velocity profile shape, providing an interpretation for the different flow performance.

(4) For high cavitation intensity, the total vapor volume variation is mainly determined by the interaction between attached cavitation and vortex cavitation in the sharp valve seat case, while the shed cavitating vortex in the chamfered valve seat case produces cavitating vortex lumps through the merger of cavitating vortex rings and contributes to the primary variation in the total volume.

The current study also reveals a critical interaction between vortex and cavitation, which is not yet fully understood based on the currently obtained simulation data. Future work will be committed to the viscosity effect on the cavitating flow by comparing oil-cavitating flow and water cavitation flow, to complement the knowledge on the flow dynamics. The vortex dynamics should also be paid additional attention, by studying the changes in flow dynamics due to the occurrence of cavitation.

Disclosure statement
No potential conflict of interest was reported by the authors.

Funding
This work was supported by Guangdong Provincial Applied Science and Technology Research and Development Program [grant number 2019A151510844]; National Natural Science Foundation of China [grant number 51975511]; Department of Education of Guangdong Province [grant number 2019KQNCX175].

ORCID
Cong Yuan http://orcid.org/0000-0002-4101-9045

References
Arndt, R. E. (2002). Cavitation in vortical flows. Annual Review of Fluid Mechanics, 34(1), 143–175. https://doi.org/10.1146/annurev.fluid.34.082301.114957
Aung, N. Z., & Li, S. (2014). A numerical study of cavitation phenomenon in a flapper-nozzle pilot stage of an electrohydraulic servo-valve with an innovative flapper shape. Energy Conversion and Management, 31–39. https://doi.org/10.1016/j.enconman.2013.09.009
Backe, W. 1999. Water-or oil-hydraulics in the future. In The Sixth Scandinavian International Conference on Fluid Power, May 26–28 (pp. 51–65).
Demirdžić, I., Lilek, Ž, & Perić, M. (1993). A collocated finite volume method for predicting flows at all speeds. International Journal for Numerical Methods in Fluids, 16(12), 1029–1050. https://doi.org/10.1002/fld.1650161202
Egerer, C. P., Hickel, S., Schmidt, S. J., & Adams, N. A. (2014). Large-eddy simulation of turbulent cavitating flow in a micro channel. Physics of Fluids, 26(8), 085102. https://doi.org/10.1063/1.4891325
Furukawa, A., & Tanaka, H. (2006). Violation of the incompressibility of liquid by simple shear flow. Nature, 443(7110), 434. https://doi.org/10.1038/nature05119
Gao, H., Lin, W., & Tsukiji, T. (2006). Investigation of cavitation near the orifice of hydraulic valves. Proceedings of the Institution of Mechanical Engineers Journal of Aerospace Engineering, 220(4), 253–265. https://doi.org/10.1243/09544100JAERO26
Han, M., Liu, Y., Wu, D., Zhao, X., & Tan, H. (2017). A numerical investigation in characteristics of flow force under cavitation state inside the water hydraulic poppet valves. International Journal of Heat and Mass Transfer, 111, 1–16. https://doi.org/10.1016/j.ijheatmasstransfer.2017.03.100
Koukouvinis, P., Naseri, H., & Gavaises, M. (2017). Performance of turbulence and cavitation models in prediction of incipient and developed cavitation. International Journal of Engine Research, 18(4), 333–350. https://doi.org/10.1177/1468087416658604
Kumagai, K., Ryu, S., Ota, M., & Maeno, K. (2016). Investigation of poppet valve vibration with cavitation. International Journal of Fluid Power, 17(1), 15–24. https://doi.org/10.1080/14399776.2015.1115648
Liáng, J., Luo, X., Liu, Y., Li, X., & Shi, T. (2016). A numerical investigation in effects of inlet pressure fluctuations on the flow and cavitation characteristics inside water hydraulic poppet valves. International Journal of Heat and Mass Transfer, 103, 684–700. https://doi.org/10.1016/j.ijheatmasstransfer.2016.07.112
Macian, V., Payri, R., Margot, X., & Salvador, F. J. (2003). A CFD analysis of the influence of diesel nozzle geometry on the inception of cavitation. Atomization and Sprays, 13, 579–604. https://doi.org/10.1615/AtomizSpr.v13.i5.80
Mosavi, A., Shamshirband, S., Salwana, E., Chau, K. W., & Tah, J. H. (2019). Prediction of multi-inputs bubble column reactor using a novel hybrid model of computational fluid dynamics and machine learning. Engineering Applications of Computational Fluid Mechanics, 13(1), 482–492. https://doi.org/10.1080/19942060.2019.1613448
Nie, S., Huang, G., Li, Y., Yang, Y., & Zhu, Y. (2006). Research on low cavitation in water hydraulic two-stage throttle poppet valve. *Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering*, 220(3), 167–179. https://doi.org/10.1243/09544089PME78

Oshima, S., & Ichikawa, T. (1985a). Cavitation Phenomena and performance of Oil hydraulic poppet valve: 1st Report, mechanism of Generation of cavitation and flow performance. *Bulletin of JSME*, 28(244), 2264–2271. https://doi.org/10.1299/jsme1958.28.2264

Oshima, S., & Ichikawa, T. (1985b). Cavitation Phenomena and performance of Oil hydraulic poppet valve: 2nd Report, influence of the Chamber Length of the seat and the flow performance. *Bulletin of JSME*, 28(244), 2272–2279. https://doi.org/10.1299/jsme1958.28.2272

Oshima, S., & Ichikawa, T. (1986). Cavitation Phenomena and performance of Oil hydraulic poppet valve: 3rd Report, influence of the poppet angle and Oil temperature on the flow performance. *Bulletin of JSME*, 29(249), 743–750. https://doi.org/10.1299/jsme1958.29.743

Oshima, S., Leino, T., Linjama, M., Koskinen, K. T., & Vilenius, M. J. (2001). Effect of cavitation in water hydraulic poppet valves. *International Journal of Fluid Power*, 2(3), 5–13. https://doi.org/10.1080/14399776.2001.10781115

Payri, F., Payri, R., Salvador, F. J., & Martinezlopez, J. (2012). A contribution to the understanding of cavitation effects in diesel injector nozzles through a combined experimental and computational investigation. *Computers & Fluids*, 58, 88–101. https://doi.org/10.1016/j.compfluid.2012.01.005

Ramezanizadeh, M., Alhuyi Nazari, M., Ahmadi, M. H., & Chau, K. W. (2019). Experimental and numerical analysis of a nanofluidic thermostyphon heat exchanger. *Engineering Applications of Computational Fluid Mechanics*, 13(1), 40–47. https://doi.org/10.1080/19942060.2018.1518272

Reid, B. A., Gavaises, M., Mitroglou, N., Hargrave, G. K., Garner, C. P., Long, E. J., & Mcdavid, R. M. (2014). On the formation of string cavitation inside fuel injectors. *Experiments in Fluids*, 55(1), 1–8. https://doi.org/10.1007/s00348-013-1662-8

Schmidt, D. P., & Corradini, M. L. (2001). The internal flow of diesel fuel injector nozzles: A review. *International Journal of Engine Research*, 2(1), 1–22. https://doi.org/10.1243/146808701545316

Schmerr, G. H., & Sauer, J. (2001). *Physical and numerical modeling of unsteady cavitation dynamics*. Fourth International Conference on Multiphase Flow. May 27 - June 1.

Ubbink, O., & Issa, R. I. (1999). A method for capturing sharp fluid interfaces on arbitrary meshes. *Journal of Computational Physics*, 153(1), 26–50. https://doi.org/10.1006/jcph.1999.6276

Ueno, H., Okajima, A., & Muromiya, Y. (1993). Visualization of cavitating flow and numerical simulation of flow in a poppet valve. *Proceedings of the Jfps International Symposium on Fluid Power*, 1993, 385–390. https://doi.org/10.5739/isfp.1993.385

Washio, S., Kikui, S., & Takahashi, S. (2010). Nucleation and subsequent cavitation in a hydraulic oil poppet valve. *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science*, 224(4), 947–958. https://doi.org/10.1243/09544062MES1618

Yang, Q., Aung, N. Z., & Li, S. (2015). Confirmation on the effectiveness of rectangle-shaped flapper in reducing cavitation in flapper–nozzle pilot valve. *Energy Conversion and Management*, 98, 184–198. https://doi.org/10.1016/j.enconman.2015.03.096

Yi, D., Lu, L., Zou, J., & Fu, X. (2015). Interactions between poppet vibration and cavitation in relief valve. *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science*, 229(8), 1447–1461. https://doi.org/10.1177/0954406214544304

Yu, H., Goldsworthy, L., Brandner, P. A., & Garaniya, V. (2017). Development of a compressible multiphase cavitation approach for diesel spray modelling. *Applied Mathematical Modelling*, 45, 705–727. https://doi.org/10.1016/j.apm.2017.01.035

Yuan, C., Song, J., & Liu, M. (2019a). Investigation of flow dynamics and governing mechanism of choked flow for cavitating jet in a poppet valve. *International Journal of Heat and Mass Transfer*, 129, 113–131. https://doi.org/10.1016/j.ijheatmasstransfer.2018.09.065

Yuan, C., Song, J., & Liu, M. (2019b). Comparison of compressible and incompressible numerical methods in simulation of a cavitating jet through a poppet valve. *Engineering Applications of Computational Fluid Mechanics*, 13(1), 67–90. https://doi.org/10.1080/19942060.2018.1552202

Yuan, C., Song, J., & Liu, M. (2019c). Coherent structure of paired vortex and transition in flow pattern in cavitating jet through a poppet valve. *International Journal of Mechanical Sciences*, 152, 19–33. https://doi.org/10.1016/j.ijmecsci.2018.12.046

Zhang, L., Chen, M., & Shao, X. (2018). Inhibition of cloud cavitation on a flat hydrofoil through the placement of an obstacle. *Ocean Engineering*, 1–9. https://doi.org/10.1016/j.oceaneng.2018.01.068

Zhang, B., Ma, J., Hong, H., Yang, H., & Fang, Y. (2017). Analysis of the flow dynamics characteristics of an axial piston pump based on the computational fluid dynamics method. *Engineering Applications of Computational Fluid Mechanics*, 11(1), 86–95. https://doi.org/10.1080/19942060.2015.1091686