Comparison of compressible and incompressible numerical methods in simulation of a cavitating jet through a poppet valve

Cong Yuan\(^a\), Jinchun Song\(^a\) and Minghe Liu\(^b\)

\(^a\)School of Mechanical Engineering and Automation, Northeastern University, Shenyang, People’s Republic of China; \(^b\)School of Mechanical Engineering, Shenyang Jianzhu University, Shenyang, People’s Republic of China

**ABSTRACT**

The regime of compressible flow generally refers to the super/subsonic case. However, several remarkable cases with low Mach number could not be appropriately described with the incompressible method. It is a similar case for a cavitating jet inside a poppet valve. In order to comprehensively address the discrepancy between incompressible and compressible methods, both non-cavitating and cavitating cases are performed in experiment and calculation based on OpenFOAM. Experiment reveals a transition in flow pattern in both non-cavitating and cavitating flow. For example, for 0.7 mm openness and 30-degree poppet angle, transition happens at approximately 29 and 27 bar pressure drop for the two cases, respectively. In general, results from the compressible method exhibit better agreement with experiment regarding both flow performance and flow structure. By contrast, the incompressible method could not provide an accurate description for the transition process under the applied flow condition. A series of studies are carried out with emphasis on such discrepancy. Firstly, the deviation in flow performance is addressed based on velocity profile and turbulence level. Secondly, the disparity in flow structure is illustrated and the mechanism for cavitation inception is discussed, which combined provide an interpretation of the deficiency of the incompressible method. Thirdly, different inlet boundary conditions are applied, and the results confirm the independence of deficiency of the incompressible method for inlet fluctuation. Finally, a re-examination is proposed concerning traditional notion of compressible flow as well as the applicability of incompressible numerical method.

**1. Introduction**

Computational fluid dynamics (CFD) enables more details of flow structure to be captured. And with continuous improvement in computational power, it is therefore widely employed in diverse engineering applications (Ardabili et al., 2018; Mou, He, Zhao, & Chau, 2017). It is quite a similar case in the area of oil-hydraulics. Several scholars (Aung & Li, 2014; Aung, Yang, Chen, & Li, 2014; Li, Aung, Zhang, Cao, & Xue, 2013; Yang, Aung, & Li, 2015; Zhang & Li, 2015) have performed a series of investigation on cavitation phenomenon in a flapper-nozzle valve with a combined experimental and numerical method. This revealed the potential cavitation region as well as the evolution of cavitation structure with advance in cavitation intensity. Based on the improved understanding of the internal cavitating jet flow, they proposed a new flapper structure, which is corroborated to have enhanced cavitation-suppression ability. Amirante, Del Vescovo, and Lippolis (2006) conducted several attempts to estimate flow force for spool valves with different structures, based on analytical calculation, flow simulation, and experimental measurement. For an open center directional control valve, the maximum flow force value occurs when the recirculation flow rate vanishes, while in the first opening phase the flow force acts in the opening direction. For a proportional valve, the U-type notch has little influence on flow force, while an adequate spool profile provides a compensation effect and is thus effective in balancing flow force at a large opening. Under the cavitation condition, pressure variation from cavitation behavior generates a large fluctuation in flow force, and thus causes difficulties in control stability (Amirante, Distaso, & Tamburrano, 2014).

Han, Liu, Wu, Zhao, and Tan (2017) performed a numerical study on cavitation effect on flow force due to cavitating flow inside a poppet valve. The results revealed that flow force under the condition of cavitation is highly dependent on geometric structure, and a two-stage configuration is superior in cavitation suppression but suffers from larger flow force. Liang, Luo, Liu, Li, and Shi (2016) investigated the influence of inlet pressure fluctuation on flow and cavitation characteristics. Cavitation structure
exhibits periodic dynamics consistent with the imposed fluctuation, and cavitation intensity is dependent on both geometric structure and the characteristic frequency of imposed fluctuation.

However, most of the research on internal cavitating jet flow through a poppet valve, whether numerical or experimental, seldom involves internal flow dynamics. As a consequence, the physical process associated with cavitation is poorly understood. In order to improve knowledge about choking mechanisms, we recently undertook a comprehensive experimental and numerical study (Yuan, Song, & Liu, 2019). It revealed that cavitation inception is mainly governed by vortex factors, and demonstrated that the cavitation phenomenon generates an aggressive fragmentation of the potential core under the choking condition. In the establishment of a numerical method, great emphasis is paid to the importance of accounting for compressibility for both single phases, because negligence in compressibility leads to inaccurate results according to our numerical experience. But the problem was not discussed in depth. It might bring an illuminating insight into selection for numerical methods and thus deserves a systematic investigation.

First of all, a fundamental truth of cavitation phenomenon should be clarified: that the inter-phase mass transfer at evaporation pressure produces a highly compressible mixture (Coutier-Delgosha, Reboud, & Delannoy, 2003). It produces a critical basis for a homogeneous cavitation model, which postulates a barotropic state law for description of the highly compressible mixture (Coutier-Delgosha, Stutz, Vabre, & Legoupil, 2007). Important supporting evidence is the sudden decrease in effective sound speed to the range of 1–2 m/s in the cavitating region according to experimental measurement (Shamsborhan, Coutier-Delgosha, Caignaert, & Nour, 2010). As a matter of fact, the ability to capture high compressibility for a cavitating mixture constitutes the essence of cavitation model. Or, in other words, all cavitation models, both homogeneous ones and two-phase ones, are built to handle the most possibly supersonic flow in the cavitation region, which plays a critical role in reproducing a cavitating process. Consequently, all numerical methods with cavitation modeling ability, whether based on a compressible or an incompressible framework, fundamentally involve compressible flow, since the cavitation-induced volumetric variation leads to a similar effect as compressibility on the pressure equation in the numerical algorithm. The definition of the compressible and incompressible methods for cavitating flow mainly refers to the difference that the incompressible method only addresses the compressibility of the cavitating mixture, while the compressible also takes the compressibility of the pure liquid and vapor phase into account.

Most of two-phase methods with cavitation models (Merkle, 1998; Schnerr & Sauer, 2001; Singhal, Athavale, Li, & Jiang, 2002) were originally developed based on an incompressible framework, i.e. the compressibility of each single phase is ignored. And they are proved successful in description of cavitating flow across a hydrofoil or Venturi-tube, where cavitation behavior is mainly governed by an inertial factor (Som, Aggarwal, El-Hannouy, & Longman, 2010) and the coupling between cavitation and turbulence is negligibly weak. However, there is considerable numerical or experimental evidence that demonstrates the role of compressibility in a non-cavitating region with low Mach number.

Under a flow condition with Strouhal number about 0.13 in a hydrofoil, the cavitation structure involves strong pressure propagation in the downstream region due to previous bubble cloud collapse (Coutier-Delgosha et al., 2007). It leads to a secondary growth and detachment of sheet cavity. A similar phenomenon is observed in the case of a Venturi-tube (C. Wang, Huang, Wang, Zhang, & Ding, 2017) and submerged circular nozzle (Stanley, Barber, & Rosengarten, 2014), where the pressure propagation produces discontinuity in vapor distribution. The Mach number in the non-cavitating region is less than 0.1. However, the compressibility should be taken into account, since pressure propagation relies on density variation.

In the case of a square nozzle, several numerical investigations (Edelbauer, Strucl, & Morozov, 2016; Karrholm, Weller, & Nordin, 2007) based on the compressible method have revealed the presence of streamwise vortices at the four wall corners under different flow conditions. Under a choking condition, the streamwise aligned vertical vapor cavities are almost stationary in the LES and essentially symmetric to the central plane (Egerer, Hickel, Schmidt, & Adams, 2014). Furthermore, the close correspondence between cavitation and these vortical structures is extensively confirmed (Edelbauer et al., 2016; Karrholm et al., 2007; Mauger, Méés, Michard, Azouzi, & Valette, 2012). Egerer et al. (2014) conducted a density divergence of calculation results, and the results in the non-cavitating region clarified that small-scale density fluctuation mainly comes from the near wall plane, while the in the center plane stronger density variation is generated from a shock wave due to cavity collapse. Additionally, shear layer instabilities are observed in the central plane near the intake of the throttle. Experimental evidence of propagating shock waves and shear layer instabilities for such types of flows are given by Mauger et al. (2012). The coincidence between numerical analysis from Egerer...
et al. (2014) and the corresponding experiment (Mauger et al., 2012) indicates that not only a shock wave but also turbulent structure is associated with density fluctuation.

In the case of computation in oil-hydraulic components, the discrepancy between incompressible and compressible methods is again observed. Zhang, Ma, Hong, Yang, and Fang (2017) performed a numerical analysis of the flow dynamics characteristics of an axial piston pump. The compressible method shows better agreement with experiment than the incompressible one. Pertaining to flow fluctuation, the compressible method is able to capture the most significant flow ripples with a close amplitude and period, while the incompressible one predicts a fluctuation with roughly one-third the intensity. And they came to the conclusion that the ripple is generally ascribed to oil compressibility. Fard and Moin (2008) applied an incompressible VOF (volume of fluid) method to jet through a poppet valve under flow condition of weak cavitation intensity. The calculation result is able to reproduce the pressure pulsation due to periodic cavitation behavior. However, it could not capture the strong fluctuation within a cycle, which is probably due to ignorance in compressibility.

Jet flow through a small-sized orifice frequently involves a transitional region where coherent structure is predominant. The first aim is to make clear whether adequate description of coherent structure requires consideration of compressibility. Thus it leads to a tentative speculation. It has long been widely accepted that a flow structure can be assumed to be incompressible unless the Mach number becomes comparatively large (Williams, 1963). Most of the experimental research (Gopalan, Katz, & Knio, 1999; Hussain, 1986) with a coherent structure method on turbulence declares the studied flow condition to belong to a regime of incompressible flow, and the numerical studies frequently employed the incompressible method (Cerutti, Knio, & Katz, 2000; Xing, Li, & Frankel, 2005), largely because coherent structure is not altered except in a sub/super-sonic flow (Brown & Roshko, 1974). However, from the available evidence on the importance of compressibility at low Mach flow documented above, it is reasonable to propose a question whether compressibility has been inadequately ignored for long. If in the present case the compressible method shows improvement in capturing the evolution of coherent structure, the significance of compressibility at least in numerical calculation is further justified, and extends to the regime in a specific case with low Mach number, which was once commonly simplified as incompressible.

Another important motivation comes from the unanswered question of whether cavitation is triggered by inlet turbulence and therefore inappropriate boundary condition contributes to the failure of the incompressible method. Even though in our previous investigation, a diaphragm accumulator was placed upstream of the test valve, in order to suppress even the slight pressure fluctuation, we are quite aware that the inlet fluctuation could not be completely removed. Furthermore, it was reported that inlet fluctuation could exhibit critical influence on the evolution of jet flow (Crow & Champagne, 1971; Zaman & Hussain, 1980), and in numerical study the characteristic frequency corresponding to the preferred mode is introduced to capture the essence of induced cavitating structure. In the case of the poppet valve, it was fully corroborated that inlet fluctuation exerts profound influence on flow performance as well as cavitation structure (Yuan et al., 2019). The correlation between coherent structure and cavitation is confirmed (Gopalan et al., 1999). Regardless, it remains unclear whether transition in flow pattern is attributable to inlet fluctuation.

In order to systematically address the role of compressibility in numerical methods, the discussion is divided into two sections covering non-cavitating and cavitating flow. In the non-cavitating case for both calculation and experiment, the outlet pressure is held at a high enough level that cavitation is eliminated completely under the applied flow condition. And in the cavitating case the outlet pressure is kept low so that cavitation in jet flow exhibits a gradual development including non-cavitating, cavitation inception to bubble cloud regime. Studies are undertaken for the two cases with a combination of both numerical and experimental results. Then the individual conclusions from the two separated cases are incorporated for a further analysis. It is worth highlighting that the present study intends to demonstrate the role of compressibility in numerically capturing the flow structure inside a submerged jet through a poppet valve, instead of evaluating the effect of compressibility (variation of compressibility or Mach number) on flow structure. The following concerns run through the current study:

(1) Re-examination of the applicability of the two numerical methods.
(2) Analysis of the deficiency of the incompressible method.
(3) Evaluation of the difference between the two numerical methods.
(4) Estimation of the influence of inlet fluctuation on transitional process.

And the conclusion is not confined to the present study case. It is expected that the discussion will serve
as general guidance for numerical studies on small-sized orifices in hydraulic area.

2. Mathematical model

2.1. Compressible approach

In the present study, a single fluid model in conjunction with a VOF technique is employed to address the two-phase flow. The VOF method uses a phase transport equation as indicator function, which has the advantage of predicting a sharp interface in a volume conservative manner (Ubbink, 1997). The inter-phase mass transfer is enabled with the integration of an additional cavitation model. Derivation of the numerical approach with consideration of compressibility for both phases begins with transport equations for both liquid and vapor phases:

\begin{equation}
\frac{\partial (\rho \alpha_i)}{\partial t} + \nabla \cdot (\rho \alpha_i U_i) = \dot{m} \\
\frac{\partial (\rho \alpha_v)}{\partial t} + \nabla \cdot (\rho \alpha_v U_v) = -\dot{m}
\end{equation}

where the subscripts \(l\) and \(v\) represent liquid and vapor phases, respectively, and \(\dot{m}\) indicates the total inter-phase exchange rate due to either evaporation or condensation:

\begin{equation}
\dot{m} = \dot{m}^- + \dot{m}^+
\end{equation}

The velocity of the mixture is determined with volume averaging of each single phase:

\begin{equation}
U = \alpha_v U_v + \alpha_l U_l
\end{equation}

The advective term in the transport equation can be rearranged as

\begin{equation}
\nabla \cdot (\alpha_i \rho_i U_i) = \rho_i \nabla \cdot (\alpha_i U_i) + \alpha_i (U_i \cdot \nabla \rho_i)
\end{equation}

Substituting this into Equations (1) and (2) gives the following expression:

\begin{equation}
\frac{\partial \alpha_i}{\partial t} + \nabla \cdot (\alpha_i U_i) = -\frac{\alpha_i}{\rho_i} \left( \frac{\partial \rho_i}{\partial t} + U_i \cdot \nabla \rho_i \right) + \frac{\dot{m}}{\rho_l}
\end{equation}

For conciseness, the first and second terms are combined to give the total derivative of phase density:

\begin{equation}
\frac{\partial \alpha_i}{\partial t} + \nabla \cdot (\alpha_i U_i) = -\frac{\alpha_i}{\rho_l} \left( \frac{D \rho_l}{D t} \right) + \frac{\dot{m}}{\rho_l}
\end{equation}

\begin{equation}
\frac{\partial \alpha_v}{\partial t} + \nabla \cdot (\alpha_v U_v) = -\frac{\alpha_v}{\rho_v} \left( \frac{D \rho_v}{D t} \right) - \frac{\dot{m}}{\rho_v}
\end{equation}

The divergence of mixture velocity refers to variation of the total volume for some fluid particle, and is derived from the addition of Equations (8) and (9):

\begin{equation}
\nabla \cdot U = -\left[ \frac{\alpha_l}{\rho_l} \left( \frac{D \rho_l}{D t} \right) + \frac{\alpha_v}{\rho_v} \left( \frac{D \rho_v}{D t} \right) \right] + \dot{m} \left( \frac{1}{\rho_l} + \frac{1}{\rho_v} \right)
\end{equation}

Combining Equations (8) and (10) leads to

\begin{align*}
\frac{\partial \alpha_l}{\partial t} + \nabla \cdot (\alpha_l U) + \nabla \cdot [\alpha_l \alpha_v (U_l - U_v)] &= \alpha_l \alpha_v \left[ \frac{\dot{\rho}_l}{\rho_v} - \frac{\dot{\rho}_l}{\rho_l} \right] + \alpha_l [\nabla \cdot U] \\
&\quad + \dot{m} \left[ \frac{1}{\rho_l} - \alpha_l \left( \frac{1}{\rho_l} - \frac{1}{\rho_v} \right) \right]
\end{align*}

A further rearrangement provides a final phase transport equation, which is constructed and solved in OpenFOAM:

\begin{align*}
\frac{\partial \alpha_l}{\partial t} + \nabla \cdot (\alpha_l U) + \nabla \cdot [\alpha_l \alpha_v U_r] &= \alpha_l \alpha_v \left[ \frac{\dot{\rho}_l}{\rho_v} - \frac{\dot{\rho}_l}{\rho_l} \right] \\
&\quad + \alpha_l [\nabla \cdot U] + \dot{m} \left[ \frac{1}{\rho_l} - \alpha_l \left( \frac{1}{\rho_l} - \frac{1}{\rho_v} \right) \right]
\end{align*}

where \(U_r\) represents the relative velocity between phases:

\begin{equation}
U_r = U_l - U_v.
\end{equation}

Weller (2008) refers to the third term on the left-hand side (LHS) including the relative velocity between phases as counter transport, which is active only for mixture fluid. De Villiers, Gosman, and Weller (2004) proposed a numerical scheme to preserve the sharpness of interface, which introduces a relative velocity in the direction of phase gradient for the intermediate two-phase mixture:

\begin{equation}
U_r = c\text{Alpha}|U| \frac{\nabla \alpha}{|\nabla \alpha|}
\end{equation}

where a coefficient \(c\text{Alpha}\) is introduced to control the rate of compression for the interface (Ubbink, 1997).

The momentum equation utilizes the VOF method to model surface tension:

\begin{align*}
\frac{\partial \rho U}{\partial t} + \nabla \cdot (\rho U \otimes U) &= -\nabla p + \nabla \cdot \tau \\
&\quad + \int_{S(t)} \sigma \kappa \cdot \delta(x - x') \, ds
\end{align*}

Mixture density is determined with a volume averaging:

\begin{equation}
\rho = \rho_l \alpha_l + \rho_v \alpha_v
\end{equation}

In momentum Equation (14), surface tension is determined by the third term on the right-hand side (RHS).
based on continuum surface force (Brackbill, Kothe, & Zemach, 1992):

$$\int_{S(t)} \sigma \kappa n \cdot \delta(x - x') ds \approx \sigma \kappa \nabla \alpha$$

(16)

The surface tension term actually introduces an additional pressure gradient in the direction perpendicular to the interface:

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

(17)

In OpenFOAM, the pressure gradient is combined with the force of gravity, which is given as

$$\nabla p = \nabla p_{\text{rgh}} + \nabla (\rho \mathbf{g} \cdot \mathbf{h})$$

(18)

Usually, the force of gravity is negligibly small in a cavitating jet compared with a large pressure drop.

The Navier–Stokes equation becomes

$$\frac{\partial \rho U}{\partial t} + \nabla \cdot (\rho U \otimes U) = -\nabla p_{\text{rgh}} + \nabla \cdot \mathbf{\tau} + \sigma \kappa \nabla \alpha$$

(19)

A discretization of this is required for the derivation of pressure Poisson equation (Demirdžić, Lilek, & Perić, 2010; Patankar, 1980):

$$U_p = \frac{H(U)}{ap} - \frac{1}{ap} \nabla p$$

(20)

With a divergence operation on both sides, the above equation is transformed as

$$\nabla \cdot U = \nabla \left( \frac{H(U)}{ap} \right) - \nabla \cdot \left( \frac{1}{ap} \nabla p \right)$$

(21)

Combination with continuity Equation (21) enables the pressure equation to be solved:

$$- \left( \frac{\alpha_l}{\rho_l} \left( \frac{D\rho_l}{Dt} \right) + \frac{\alpha_v}{\rho_v} \left( \frac{D\rho_v}{Dt} \right) \right) + \dot{m} \left( \frac{1}{\rho_l} - \frac{1}{\rho_v} \right)$$

$$= \left( \nabla \left( \frac{H(U)}{ap} \right) - \nabla \cdot \left( \frac{1}{ap} \nabla p \right) \right)$$

(22)

A perfect fluid is assumed for both liquid and vapor phases, and density variation is a function of ambient pressure:

$$\rho = \psi p$$

(23)

For a cavitating jet through a small-sized orifice, the thermal effect is usually neglected, largely due to the extremely high density ratio between liquid and vapor (Asnaghi, Feymark, & Bensow, 2017; Som et al., 2010). Consequently, the compressibility is assumed to be unchanged (Franc & Michel, 2006; Goncalvès & Charrière, 2014):

$$\psi = \text{constant}$$

The substantial derivative of density is correlated to pressure variation with conjunction of perfect fluid assumption:

$$\frac{D\rho}{Dt} = \frac{\partial \psi}{\partial t} + \mathbf{U} \cdot \nabla (\psi p)$$

(24)

If we combine this with Equation (24), the derivative of density can be eliminated, and the pressure equation is written as

$$- \left( \frac{\alpha_l}{\rho_l} \left( \psi_l \frac{\partial p}{\partial t} + \mathbf{U} \cdot \nabla (\psi_l p) \right) + \frac{\alpha_v}{\rho_v} \left( \psi_v \frac{\partial p}{\partial t} + \mathbf{U} \cdot \nabla (\psi_v p) \right) \right) + \dot{m} \left( \frac{1}{\rho_l} - \frac{1}{\rho_v} \right) = \left( \nabla \left( \frac{H(U)}{ap} \right) \right) - \nabla \cdot \left( \frac{1}{ap} \nabla p \right)$$

(25)

Such fully implicit treatment of compressibility is advantageous in pressure wave prediction, and also helps to promote stability and accuracy in numerical calculation.

The cavitation model is introduced for the calculation of terms associated with inter-phase mass transfer. The current study uses the Schnerr–Sauer model, which is derived from a simplified Rayleigh–Plesset equation (Schnerr & Sauer, 2001):

$$\begin{cases}
\dot{m}^+ = \frac{3 \rho_l \rho_v}{\rho} \alpha_l \alpha_v (r R_b) \sqrt{\min(p - p_v, 0)} \\
\dot{m}^- = \frac{3 \rho_l \rho_v}{\rho} \alpha_l \alpha_v (r R_b) \sqrt{\max(p - p_v, 0)}
\end{cases}$$

(26)

### 2.2. Incompressible approach

The incompressible method could be derived from the corresponding equations of its compressible counterpart, with the elimination of the terms involved in density variation. The phase transport equation and pressure equation are given by

$$\frac{\partial \alpha_l}{\partial t} + \nabla \cdot (\alpha_l \mathbf{U}) + \nabla \cdot [\alpha_l \alpha_v \mathbf{U}_l] = \alpha_l [\nabla \cdot \mathbf{U}] + \dot{m}$$

$$\times \left[ \frac{1}{\rho_l} - \alpha_l \left( \frac{1}{\rho_l} - \frac{1}{\rho_v} \right) \right]$$

(27)

$$\dot{m} \left( \frac{1}{\rho_l} - \frac{1}{\rho_v} \right) = \left( \nabla \left( \frac{H(U)}{ap} \right) \right) - \nabla \cdot \left( \frac{1}{ap} \nabla p \right)$$

(28)
2.3. Numerical set-up

2.3.1. Solution algorithm
The solver is developed on OpenFOAM. For the phase transport equation, an explicit MULES (Multi-Dimensional Universal Limiter with Explicit Solution) algorithm is utilized due to its advantage in boundedness (Damián, 2013; Weller, 2008), and for pressure equation, a PIMPLE loop is used. An upwind-based second-order discretization scheme is applied to the convection term in the momentum equation. Laplacian derivative terms are addressed by a conservative, bounded, second-order scheme, and temporal terms by a second-order, backward discretization scheme. The time step is adjusted with a constant Courant number of 0.22–0.4, depending on flow conditions, and remains at an order of $10^{-8}$ s.

2.3.2. Calculation mesh
Because the structure of poppet valve is axis-symmetric, 2-D calculation is performed for the sake of saving time. Experimental evidence (Browand & Laufer, 1975; Hussain, 1986) shows that, in the transitional region of a circular jet, the predominant coherent structure is highly axis-symmetric and assumes the form of a vortex ring, which further justifies the applied 2-D calculation. Due to its benefits in efficiency and stability (Jasak, Weller, & Gosman, 1999), a structured grid is employed with an arrangement consistent with the flow direction. In a previous study, structured grids exhibited an improvement in capturing a sharp gradient at the nozzle inlet (Macián, Payri, Margot, & Salvador, 2003). The mesh size at annular clearance is refined so that the minimum...
length is 2μm, comparable to the Kolmogorov scale of 3.25μm, which represents the smallest turbulent scale. In the wake of the potential core, mesh size is slightly increased to between 10 and 20μm. In a region far from the cavitation flow, no refinement is involved. With these considerations, a computational grid with in total 240,000 cells is employed for calculation, as shown in Figure 1. A Mesh independence study is carried out in terms of flowrate and distribution of velocity magnitude, and the results are presented in Figure 2.

2.3.3. Boundary condition
A no-slippage boundary is imposed on solid walls with a zero-gradient boundary condition for all hydrodynamic variables. A pressure outlet with simplified non-reflective treatment is chosen for the outlet boundary condition. The total pressure is applied to inlet. Such a configuration is consistent with experiment, and is frequently applied in numerical calculation in a nozzle jet with a large pressure drop. The predicted flow rate is compared with measured data as a method of verification. It should be mentioned that no external excitation is imposed on inlet and outlet boundaries, i.e. the pressure is kept constant without any fluctuation. Even though special care is paid to suppression of pressure pulsation in experiment, the disturbance cannot be totally eliminated. Without external excitation, the free stream is generally considered to have a fluctuation with a widespread spectrum. Generally speaking, the instantaneous inlet pressure fluctuation is far below 0.5% of the pressure drop in experiment. In order to illustrate the difference between the two numerical methods, the inlet boundary fluctuation is varied, and will be specified in the captions of related calculation results.

2.3.4. Turbulence model
Within the applied computational grid, the first layer neighboring the wall is refined to ensure that the magnitude of yPlus is less than 1. Thus the incorporation of wall boundary function is not needed. The choice of turbulence model is controversial for a generic cavitating jet in internal flow (Edelbauer et al., 2016). Generally, the RANs-based turbulence model is considered inappropriate for an unsteady internal cavitating jet, because the internal flows strongly depend on specific problems (Posa, Oresta, & Lippolis, 2013). Schmidt and Corradini (2001) reported that turbulence should not be considered separately. Furthermore, for a circular jet without any external excitation, e.g. tripping or vibration, the pressurizing jet experiences a gradual development towards full turbulence (Browand & Laufer, 1975; Hussain, 1983, 1986; Hussain & Clark, 1981). In the same sense, Faeth (1991) showed that turbulence does not have time to fully develop due to the short length of the orifice. As a consequence, no turbulence model is employed and the present study produces a quasi-DNS (direct numerical simulation) calculation.

3. Experimental set-up
3.1. Experimental apparatus
Figure 3(a) illustrates the structure of the test valve, which is composed of a cone-shaped poppet and a sharp-edged
nozzle hole. The poppet angle is 30°. The inlet port is located at the bottom, and the outlet port at the top. So the fluid first goes through the circular nozzle hole, then gets across the annular clearance between nozzle hole and poppet. It produces an initial converging flow and afterwards a diverging flow along the sloping poppet surface. The current study is concerned with the quasi-steady state of cavitation performance, so the poppet displacement is fixed at 0.7 and 1.0 mm.

A schematic of the hydraulic system is given in Figure 3(b). The system is motivated by a piston pump. A pressure relief valve controls the operating pressure and a bladder accumulator is employed to suppress flow pulsation. A throttle valve regulates the inlet pressure. In order to keep a stable inlet pressure, a film-type accumulator is introduced to remove small pressure fluctuations. The other throttle valve is used to adjust the outlet pressure.

### 3.2. Measurements for operating conditions

For a small-sized orifice, pressure loss is mainly generated in the effective throttling area (Payri, Payri, Salvador, & Martínez-López, 2012). In the current flow, the effective throttling area refers to the annular clearance as shown in Figure 3(a). As a result, the pressure transmitter for inlet pressure is mounted upstream of the nozzle inlet, with the other one for outlet pressure downstream of the poppet. Flowrate is measured with a flow meter downstream of the test valve. Flowrate variation against pressure drop indicates the global flow performance, and is sometimes used to detect the critical cavitation point. In some experiments (Oshima & Ichikawa, 1985; Winklhofer, Kull, Kelz, & Morozov, 2001), cavitation extent is regulated with a constant inlet pressure and a gradually decreasing outlet pressure. The flowrate stagnation is established with intensification of the cavitation effect. In the current study, the outlet pressure is fixed at 15 bar for non-cavitating flow and 5 bar for cavitating flow, while the inlet pressure is varied to change flow conditions and produce an alteration in flow pattern. Data are acquired with the establishment of quasi-steady flow conditions. During operation, a heat exchanger keeps the temperature of the fluid at approximately 313 K. Table 1 provides details of properties of the fluid medium.

#### Table 1. Property of fluid medium involved in the current study

| Fluid property               | Value      |
|------------------------------|------------|
| Liquid type                  | mineral oil #46 |
| Density for liquid phase     | 872 (kg/m³) |
| Liquid kinematic viscosity   | 4.81 × 10⁻⁵ (m²/s) |
| Vaporization pressure        | 3000 (Pa)  |
| Density for vapor phase      | 0.1 (kg/m³) |
| Vapor kinematic viscosity    | 5.72 × 10⁻⁵ (m²/s) |
| Temperature                  | 313 (K)    |
| Coefficient for surface tension | 0.03 (N/m) |

### 3.3. Flow structure visualization

The valve house is made from transparent material. A high speed camera of IX 221 combined with an LED with power of 150 W as illuminant enables flow visualization. Light is shed from opposite side of the camera. In the cavitating region, the interface reflects most of the transmitted light, so the cavitating bubble in recorded images produces a blurred dark. An exposure time of 2 ns is applied to allow instantaneous observation of cavitation structure. For the sake of revealing periodic behavior of the cavitation structure, the present experiment applies a filming speed of up to 43,000 frame per second.

### 4. Results and discussion

#### 4.1. Non-cavitating flow

##### 4.1.1. Flow performance

For the non-cavitating flow, no inter-phase transfer is involved in either calculation or experiment, thus the applied numerical methods are actually single phase flow. Figure 4 provides a comparison for flow performance from experiment and prediction for the two numerical methods. For the non-cavitating case, flow performance generally exhibits a piecewise linear tendency with respect to pressure drop, and the transition point is approximately 29 bar for 0.7 mm openness and 26.3 bar for 1 mm openness. The linearity against pressure drop without cavitation is extensively observed in other experimental studies for both poppet valve (Washio, Kikui, & Takahashi, 2010) and circular nozzle (Winklhofer et al., 2001). However, as the pressure drop increases, the rising pressure drop tends to produce a minor deviation from the original linearity, and the pressure effect on flow performance is slightly reduced. The linearity is persistent even with inceptive cavitation. A similar trend is observed in other studies (Oshima & Ichikawa, 1985; Washio et al., 2010) concerned with cavitation, but is seldom discussed. For the flow conditions under consideration, predictions of flow performance from incompressible and compressible approaches agree well with each other at low pressure drop before the transition point, and both are able to capture the initial linearity exhibited in experimental measurement. Nonetheless, the compressible method provides better agreement for the transition in flow performance as pressure drop increases.

##### 4.1.2. Transition in flow structure from experiment

Figure 5 illustrates the transition in flow pattern as the pressure drop increases. It is worth mentioning that the
evolution process involves a gradual development of the disturbance in the shearing layer, instead of a critical transitional point similar to the concept of cavitation inception, at which the flow pattern undergoes a sudden alteration.

At a pressure drop of 17 bar, the smooth laminar potential core remains stable all the way along the poppet surface. In the experimental images, the structure of the shearing layer is clearly captured due to the vignetting of their refracted light. Higher light intensities associated with such a feature enables its distinction from ambient fluid with less shearing strength (Egerer et al., 2014). Consequently, when the jet potential core retains its persistent stability as shown in Figure 5(a), the rim of the shearing layer appears as a clear intersection line separating the main jet core and the ambient stationary fluid, similar to the laminar case in a square nozzle (Mauger et al., 2012). At a pressure drop of 26 and 31 bar, evident disturbance is observed in local section, and the potential core assumes a waving form in the downstream region as illustrated in Figure 5(b)–(c). But the disturbance is not sufficient to bring about a disintegration to the jet potential core. It apparently indicates that the generated disturbance, even with a remarkable
strength, fails to achieve its saturation state (Yule, 1978). The process is highly dependent on geometric structure. Due to the cone shape of the poppet, a diverging flow is generated, which leads to a gradual decrease in shearing intensity. On the other hand, the transitional process actually constitutes a backscatter evolution of coherent structure (Sarkar & Schlüter, 2014), whose growth relies on extraction of energy from the shearing layer. As a consequence, even though the jet flow shows evident disturbance, the shearing layer strength is remarkably lowered due to the diverging flow. And the disturbing part cannot get further development, thus the slightly waving potential core extends downstream. Such a quasi-stable status covers a definite intermediate range of pressure drop. When the pressure drop is sufficiently elevated, as in the case of Figure 5(d), the disturbance is further enhanced with promotion of shearing layer intensity. As a result, the potential core becomes disrupted. The evolution process for openness of 1.0 mm and poppet angle of 30° shows a similar tendency, as shown in Figure 6(a)–(c).

4.1.3. Comparison between experiment and calculation
The comparison is divided into three parts according to numerical methods and boundary condition.

(a) Compressible method
It could reproduce the transitional process qualitatively as shown in Figure 5(a)–(d) and Figure 6(d)–(f). At \( \Delta p = 17 \text{ bar} \), calculation results show that the laminar potential core is persistent thoroughly along the poppet. At \( \Delta p = 26 \) and 31 bar, they show the intermediate flow pattern with quasi-stability. In the downstream region, the potential core develops a waving form. However, the potential core is not destroyed. At \( \Delta p = 41 \) bar, the results capture the

Figure 6. Calculation results from compressible method illustrating flow pattern with pressure drop of 17, 26, 31, and 41 bar (Reynolds number of 170, 212, 233, and 271) for 0.7 mm openness amount. (a–d) are velocity distribution corresponding to the 4 flow conditions in Fig.5, and (e–h) are pressure distribution for the enlarged view of rectangle in (a).
evolution of the potential core from the original laminar state after the outlet to the eventual disruption.

The corresponding pressure distribution, seen in Figure 7(e)–(h) and Figure 6(g)–(i), demonstrates the extensive existence of coherent structure, organized as a paired vortex. Its formation process is discussed in section 4.2. The transition in flow pattern is associated with intensity of coherent structure. At $\Delta p = 17$ bar, the weak coherent structure is not yet sufficient to exert any evident influence on the smooth potential core. At $\Delta p = 26$ and 31 bar, its intensity is perceptibly increased, thus an evident disturbance is generated. At $\Delta p = 41$ bar, the coherent structure experiences an abrupt promotion through vortex pairing (Winant & Browand, 1974), which is to be addressed in the cavitating case. And it leads to a disruption of the smooth potential core. The evolution of the coherent structure is in general agreement with the transitional process. A transition

process is widely encountered in oil-hydraulic components. Posa et al. (2013) reported in a DNS study on spool valves that the flow pattern changes and produces the so-called “Coanda effect.” The preference for the DNS model is largely because it can produce the instantaneous structure of the flow. A similar phenomenon was confirmed in the case of a servo valve orifice (Pan, Wang, & Lu, 2011).

(d) Incompressible method without inlet fluctuation
Under the considered flow conditions, the incompressible method without inlet fluctuation shows that the persistent laminar state is not even slightly disturbed, and that the pressure distribution exhibits no sign of coherent structure, as illustrated in the pressure distributions of Figure 8(e)–(h) and Figure 6(n)–(o).

(f) Incompressible method with inlet fluctuation
It is one of our aim to clarify the influence of inlet fluctuation on the transition in flow pattern.

Figure 7. Calculation results from incompressible method illustrating flow pattern with pressure drop of 17, 26, 31, and 41 bar (Reynolds number of 170, 212, 233, and 271) for 0.7 mm openness amount. (a-d) are velocity distribution corresponding to the four flow conditions in Fig.5, and (e-h) are the corresponding pressure distribution. The inlet disturbance is of 0%.
Figure 8. Calculation results from incompressible method illustrating flow pattern with pressure drop of 17, 26, 31, and 41 bar (Reynolds number of 170, 212, 233, and 271) for 0.7 mm openness amount. (a-d) are velocity distribution corresponding to the 4 flow conditions in Fig.5, and (e-h) are the corresponding pressure distribution. The inlet disturbance is of 2%.

Figure 9(e)–(h) shows that coherent structure is generated due to the imposed strong inlet fluctuation, which is quite beyond the experimental condition. However, the potential core generally remains stable regardless of pressure drop, as shown in Figure 9(a)–(d). When $\Delta p = 26$ and 31 bar, the coherent structure shows an intensity quite comparable to the compressible method’s results, but the disturbance of the potential core is much weaker.

Three conclusions are drawn from the above analysis. First, only the compressible method captures the transitional process qualitatively consistent with experiment. Second, the incompressible method is deficient in describing the transitional flow, and the pressure distribution may provide a clue to interpretation. Third, inlet fluctuation has little to do with the deficiency of the incompressible method.

4.1.4. Analysis of deficiency of the incompressible method

For vortical structure, a pressure drop is generated in the core region (Arndt, 2002; Franc & Michel, 2006). Since density variation is correlated with pressure distribution, a vortex inevitably involves density variation. Mauger et al. (2012) provide experimental evidence for the slight density fluctuation generated by turbulence structure by multiple means, which was later further corroborated by Egerer et al. (2014). Our experiment could not provide any trace of density fluctuation due to vortical structure. However, the compressible numerical method captures the paired coherent structure, which assumes the form of a paired vortex. In place of strong coherent structure, the potential core shows evident disturbance, as shown in Figure 7(c) and (g), while in the case of the incompressible method, (a) with inlet fluctuation, even though the coherent structure is present, and has a quite similar strength as shown by the contour.
Figure 9. Experimental and numerical results for case of 1.0mm openness amount with pressure drop of 20, 23, 28bar (Reynolds number of 307, 325, 357) respectively. (a-c) illustrate experiment images. (d-i) illustrate results from compressible method. (j-o) illustrate results from incompressible method.

legend in Figure 9(c), (d), (g), and (h), it does not lead to a perceivable disturbance; and, (b) without inlet fluctuation, the coherent structure is not present, and the laminar status is persistent along the poppet surface. From these studies, a conclusion is drawn that not only the presence of coherent structure but also the correlation between vortex and density fluctuation is indispensable for prediction in disturbance in jet potential core.

4.1.5. Discrepancy in turbulence level and flow performance

Even though the predicted flow performance of both methods is close to experimental measurements at low pressure drop, only the compressible approach captures transition of flow pattern and thus the critical alteration in piecewise linear flow performance. This implies that the compressible method provides a more accurate prediction of flow dynamics at high Reynolds
Figure 10. Profiles of mean velocity at different sections downstream from valve seat. The red line is the incompressible method result, and the blue the compressible.
Figure 11. Turbulence level (velocity fluctuation) for openness amount of 0.7 mm at different sections downstream from valve seat. The red line is the incompressible result, and the blue the compressible.
number. Regarding predicted flow performance, one could observe a distinct discrepancy between the two methods except in the coincidence of linearity at low Reynolds number, as mentioned above. The discrepancy between compressible and incompressible method regarding flow performance could be interpreted from the following summaries for Figures 10 and 11.

Figure 10 illustrates the time average velocity profile at different sections for incompressible and compressible method results, respectively. At a low Reynolds number of 162, the two sets of results coincide well with each other. As Reynolds number increases, the deviation becomes more evident. Saha, Abu-Ramadan, and Li (2013) reported in a numerical study on cavitation through an injector that calculation results with variable liquid density exhibit a noticeable difference in velocity distribution. Similarly, it is reasonable to ascribe the deviation in flow performance predicted by the two numerical methods shown in Figure 4 to the difference in velocity distribution.

Figure 11 illustrates the turbulence level (variation of velocity) at different sections from annular clearance. It can be clearly observed that at a low Reynolds number of 162 the turbulence intensity is at a comparatively low level for both sets of results. However, as Reynolds number increases, the compressible method predicts a much stronger turbulence level than the incompressible method. Zhang et al. (2017) have reported that the incompressible method predicts a smaller flow ripple, and a general flow rate slightly higher than in the compressible calculation results. They proposed that with consideration of compressibility the fluid can absorb the pressure fluctuation and produces a stronger turbulence. This indicates that the correlation between pressure and density is compensated with consideration of compressibility. And thus the calculation results are improved. For the present case, the higher turbulence level predicted by the compressible method leads to stronger pressure loss, and thus contributes to a more precise flow performance.

Pan et al. (2011) and Posa et al. (2013) demonstrated that the transition in flow pattern shows a noticeable influence on flow performance. The current study provides a quite similar conclusion that the alteration in flow performance cannot be predicted without a precise description of the transition in flow pattern.

4.2. Cavitating flow

4.2.1. Flow performance

Figure 12 illustrates flow performance for the case of cavitating flow. The occurrence of cavitation produces a more prominent deviation from the original linearity, compared with the effect of transition in flow pattern for the non-cavitating case. Generally, the predicted flow performance from compressible method results complies with experiment. It not only reproduces the trend due to cavitation effect, but also is satisfactorily close to experimental measurements with a discrepancy less than 5.4%.

4.2.2. Cavitating morphology between experiment and compressible method results

Figures 13 and 14 illustrate a comparison of cavitation morphology from experiment and compressible method numerical results. In the picture as a whole, paired vortex structure is extensively involved in the flow dynamics of the cavitating jet.

(a) At small pressure drop, it is quite clearly observed that a laminar pattern is present immediately downstream of annular clearance and extends all along the poppet, which shows much in common with the non-cavitating case. Both experiment and numerical results demonstrate that cavitation nucleation is not yet developed in the smooth potential core.
The stable laminar flow pattern is accredited to the weak paired vortex structure, which is not sufficient to produce any noticeable perturbation of the potential core, as discussed in the non-cavitating case. Furthermore, in the case of the persistent laminar potential core, no cavitating bubble is present, as demonstrated in both Figure 13(a) and Figure 14(a).

(b) With a rise in pressure drop, a wavy pattern is developed from the laminar potential core in the downstream region, quite similar to the transitional process in non-cavitating flow. However, occurrence of cavitation produces a two-phase flow essentially. It is worth mentioning the paired structure of incepting cavitation nuclei in both experiment and compressible calculation results as shown.
Figure 14. Comparison between experiment and compressible calculation results in terms of cavitation morphology for 0.7 mm openness amount: (a)–(h) instantaneous cavitation morphology corresponding, respectively, to pressure drop of 25, 28, 30, 32, 36, 40, 44, and 50 bar.
in Figure 13(b)–(c) and Figure 14(c)–(d), surprisingly consistent with the organization of coherent structure in the non-cavitating case in Figure 7. It has been comprehensively corroborated in a previous study (Yuan et al., 2019) that cavitation inception is mainly governed by the development of coherent structure. In the present study, a similar conclusion could also be drawn here from three aspects. The first is that cavitation does not begin until a wavy pattern in the potential core is generated, which indicates a perceptible intensification of coherent structure as demonstrated in the non-cavitating case. The second is that cavitation coincides with the coherent structure. The third is that a vortex structure involves a pressure drop in the center region (Arndt, 2002), which is favorable for cavitation inception.

(c) As the pressure drop further increases, a cavitation bubble cluster is generated, which indicates that

Figure 15. Periodic cavitation performance for pressure drop of 46 bar, with 1.0 openness. Consecutive images are approximately $2.5 \times 10^{-5}$ s apart for experiment, and $2.5 \times 10^{-5}$ s apart for calculation.

Figure 16. Periodic cavitation performance for pressure drop of 41 bar, with 1.0 openness. Consecutive images are approximately $2.5 \times 10^{-5}$ s apart for experiment, and $2.5 \times 10^{-5}$ s apart for calculation.
Incorporated with the flow performance of Figure 12, it reveals that the flowrate gradually deviates from the second linear part under a bubble cluster regime. In addition, characteristic behaviors such as roll-up and segregation are captured in both experiment and calculation. The segregation indicates an aggressive break-up of the potential core, as shown in Figure 13(e)–(f). Furthermore, the cavitation exhibits a typical periodic behavior, as shown in Figures 15 and 16. The roll-up and segregation could be observed in detail in a cyclic process. After segregation, the downstream cavity structure is convected by the separated main flow, accompanied by a gradual dissipation.

4.2.3. Origin of paired vortex and mechanism of cavitation nucleation

The instability of the free shear layer has its origin at the sharp edge of the valve seat, where the initial wall boundary layer undergoes an abrupt transition to a free shear boundary layer, while on the poppet surface side the wall boundary layer persists across the annular clearance. Moreover, the pressure decreases more sharply at the free shear side, leading to a drastic pressure gradient. Consequently, the first instability develops first on the shearing layers side, as shown in Figure 17. As mentioned above, the weak instability at first exerts minor influence on the laminar potential core. It produces the typical Kelvin–Helmholtz instability. Initially, the instability appears as a singular negative peak in pressure distribution. In a slightly downstream region, a counterpart vortex is developed on the poppet surface side, leading to the formation of paired vortex structure. With further intensification with propagation, pressure is further decreased in the vortical core, susceptible to cavitation inception. As shown in Figures 13 and 14, cavitation frequently begins on the free shear side, instead of the wall side where separation occurs. This suggests the dominance of the vortex on the free shear side, and as well implies that the cavitation is governed mainly by vortex dynamics, instead of by the inertial effects commonly seen in a hydrofoil or nozzle injector (Coutier-Delgosha et al., 2003; Leroux, Coutier-Delgosha, & Astolfi, 2005). Figures 5 and 6 also reveal that the coherent structure is quite remarkable in the transitional region of the potential core for the non-cavitating case, where the potential core assumes a smooth laminar state.

4.2.4. Comparison between compressible and incompressible methods

Numerical calculation from the compressible method and experimental results in terms of cavitation morphology exhibit general agreement. For the laminar potential core, cavitation structure is not present. Furthermore, it is also observed that cavitation is not developed until the generation of a streamwise wave. This further demonstrates the governing mechanism of cavitation in the present cavitating jet. Figure 18 illustrates numerical results from the incompressible method. In the incompressible numerical results, no cavitation is predicted in the cases of (a) and (b), and the potential core assumes a laminar state thoroughly along poppet surface. Further inspection of the corresponding pressure distribution indicates that paired vortex structure is not present in the incompressible method results shown in Figure 18(a)–(b). Even though the paired vortex structure is predicted in case (c) by the incompressible method and cavitation appears in the core of a strong vortex, obviously the predicted cavitation structure is
not consistent with experiment. Besides, the periodic generation and gradual development process of coherent structure, as exhibited by the compressible method results in Figure 17, are not captured. The failure in cavitation prediction is again attributable to the deficiency in description of the coherent structure. As a consequence, the predicted flow performance becomes inaccurate without an appropriate description of cavitation behavior. The defect in the incompressible approach also indicates the dependence of cavitation generation on paired vortex structure. It reveals that the predominant governing mechanism for cavitating flow is vortex dynamics, instead of the commonly encountered inertial factor (Egerer et al., 2014). Negligence in compressibility effect leads to absence of a paired vortex, and eventual to deficiency in cavitation prediction.

4.3. Re-examination of compressibility/compressible flow

The inadequate description of coherent structure by the incompressible method leads to failure to capture the transition from laminar to turbulent status. As demonstrated, cavitation is correlated with vortex dynamics, and consequently the incompressible method also fails in the prediction of cavitation structure. It indicates the non-negligible role of compressibility in the present transitional flow, even though the maximum Mach number in the non-cavitating region is less than 0.1. It has been extensively accepted that compressibility does not alter the structure of jet turbulence until the mean flow becomes comparable to the speed of sound (Williams, 1963), as has been validated experimentally by Brown and Roshko (1974). Consequently, compressibility is frequently ignored in flow with Mach number. However, the present study as well as other preliminary investigations (Egerer et al., 2014; Wang et al., 2017; Zhang et al., 2017) confirm that flow with low Mach number may involve flow dynamics closely correlated with density fluctuation. Claims such as that a redefinition is required for compressible flow are obviously an overstatement, even though the traditional notion is at variance with the findings of the present research. It is, however, reasonable to invite a re-examination of the applicability of the incompressible numerical method. From the other side, the significance of compressibility deserves further consideration.

A question might be raised of whether the flow under analysis is compressible or incompressible. It should be pointed out unambiguously that the regime of compressible flow not only covers flow with high Mach number, but also includes all flow dynamics that involve a non-negligible role of compressibility. And the former obviously belongs to a specific case of the latter, because sonic flow involves flow structures like shock wave which could not be precisely predicted without consideration of compressibility. In a similar sense, the flow considered here belongs to a regime of compressible flow because its coherent structure involves an important role of compressibility.

5. Conclusion

In the current study, investigation of the influence of compressibility on numerical methods for the flow dynamics of a cavitating jet through a poppet valve has
been conducted based on a comprehensive numerical and experimental method. The following conclusions are drawn.

(1) For non-cavitating flow, experiment captures a transition from laminar to turbulent status with advance in pressure drop. The compressible method is able to capture the transition in flow pattern. Thus it predicts the transition in piecewise linear flow performance, consistent with experimental measurement. In comparison, the incompressible method fails to reproduce the transitional flow. Thus at high Reynolds number the predicted flow performance deviates noticeably from experimental results.

(2) The comparison between the compressible and incompressible approaches with inlet fluctuation demonstrates that it is imprecise to ascribe the deficiency in the incompressible method to ignorance of inlet fluctuation, even though coherent structure is generated with the imposition of inlet fluctuation.

(3) The turbulence level distribution reveals that the compressible method predicts a stronger turbulence intensity. And it leads to remarkable discrepancy in velocity distribution. As a result, the two numerical methods show noticeable difference in flow performance as pressure drop increases.

(4) For cavitating flow, the compressible results agree well with experiment in terms of both flow performance and cavitation morphology. Cavitation morphology from both compressible calculation and experiment demonstrates that coherent structure is extensively involved in the flow dynamics of the cavitating jet. This clarifies that it is a vortex factor rather than inertial factor that mainly governs the cavitation, especially at the stage of inception.

(5) The pressure distribution reveals that coherent structure in the form of a paired vortex is extensively involved in jet dynamics. The deficiency of the incompressible method in both non-cavitating and cavitating flow is attributable to failure to describe the coherent structure.

(6) The present study as well as several preliminary studies demonstrate the significance of compressibility even with low Mach number. Consequently, the categorization of compressible flow solely based on Mach number is inadequate or, to be exact, misleading.

Since a small-sized orifice frequently involves a transitional process, related numerical research should carefully consider compressibility. At present, it is too early to assert that coherent structure in a transitional region cannot be adequately predicted without the incorporation of compressibility. To further corroborate the generality of its key role, a comprehensive investigation is required to cover other cases of transitional flow, such as circular and square nozzles.

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

Funding
We are grateful for financial support from National Natural Science Foundation of China [grant number 51705342].

References
Amirante, R., Del Vescovo, G., & Lippolis, A. (2006). Evaluation of the flow forces on an open centre direction control valve by means of a computational fluid dynamic analysis. Energy Conversion and Management, 47(13-14), 1748–1760.

Amirante, R., Distaso, E., & Tamburrano, P. (2014). Experimental and numerical analysis of cavitation in hydraulic proportional directional valves. Energy Conversion and Management, 87, 208–219.

Ardabili, S. F., Najafi, B., Shamshirband, S., Bidgoli, B. M., Deo, R. C., & Chau, K. W. (2018). Computational intelligence approach for modeling hydrogen production: A review. Engineering Applications of Computational Fluid Mechanics, 12(1), 438–458.

Arndt, R. E. (2002). Cavitation in vortical flows. Annual Review of Fluid Mechanics, 34(1), 143–175.

Asnaghi, A., Feymark, A., & Bensow, R. E. (2017). Improvement of cavitation mass transfer modeling based on local flow properties. International Journal of Multiphase Flow, 93, 142–157.

Aung, N. Z., & Li, S. (2014). A numerical study of cavitation phenomenon in a flapper-nozzle pilot stage of an electro-hydraulic servo-valve with an innovative flapper shape. Energy Conversion and Management, 77, 31–39.

Aung, N. Z., Yang, Q., Chen, M., & Li, S. (2014). CFD analysis of flow forces and energy loss characteristics in a flapper-nozzle pilot valve with different null clearances. Energy Conversion and Management, 83, 284–295.

Brackbill, J. U., Kothe, D. B., & Zemach, C. (1992). A continuum method for modeling surface tension. Journal of Computational Physics, 100(2), 335–354.

Browand, F. K., & Laufer, J. (1975). The Roles of large scale structures in the initial development of circular Jets. Turbulence in Liquids, 35, 333–342.

Brown, G. L., & Roshko, A. (1974). On density effects and large structure in turbulent mixing layers. Journal of Fluid Mechanics, 64(4), 775–816.

Cerutti, S., Knio, O. M., & Katz, J. (2000). Numerical study of cavitation inception in the near field of an axisymmetric jet at high Reynolds number. Physics of Fluids, 12(10), 2444–2460.

Coutier-Delgosha, O., Reboud, J. L., & Delannoy, Y. (2003). Numerical simulation of the unsteady behaviour of cavitating flows. International Journal for Numerical Methods in Fluids, 42(5), 527–548.
Coutier-Delgosha, O., Stutz, B., Vabre, A., & Legoufipil, S. (2007). Analysis of cavitating flow structure by experimental and numerical investigations. *Journal of Fluid Mechanics*, 578, 171–222.

Crow, S. C., & Champagne, F. H. (1971). Orderly structure in jet turbulence. *Journal of Fluid Mechanics*, 48(3), 547–591.

Damian, S. M. (2013). An extended mixture model for the simultaneous treatment of short and long scale interfaces (Ph. D. Thesis), Universidad Nacional Del Litoral.

Demirdzic, I., Lilic, Z, & Peric, M. (1993). A collocated finite volume method for predicting flows at all speeds. *International Journal for Numerical Methods in Fluids*, 16(12), 1029–1050.

De Villiers, E., Gosman, A. D., & Weller, H. G. (2004). Large eddy simulation of primary diesel spray atomization (No. 2004-01-0100). SAE Technical Paper.

Edelbauer, W., Struc, J., & Morozov, A. (2016). Large Eddy simulation of cavitating throttle flow. In *Advances in Hydroinformatics* (pp. 501–517). Singapore: Springer.

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.

Faeth, G. M. (1991). Structure and atomization properties of dense turbulent sprays. Symposium (International) on Combustion (Vol. 23, No. 1, pp. 1345–1352). Elsevier.

Fard, M. P., & Moin, H. (2008). A computational study of cavitation in a hydraulic poppet valve. In *Fifth International Conference on Transport Phenomena in Multiphase Systems*.

Franc, J. P., & Michel, J. M. (2006). *Fundamentals of cavitation* (Vol. 76). Springer science & Business media.

Goncalvès, E., & Charrière, B. (2014). Modelling for isothermal cavitation with a four-equation model. *International Journal of Multiphase Flow*, 59, 54–72.

Gopalan, S., Katz, J., & Knio, O. (1999). The flow structure in the near field of jets and its effect on cavitation inception. *Journal of Fluid Mechanics*, 398, 1–43.

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 & Mass Transfer*, 111, 1–16.

Hussain, A. F. (1983). Coherent structures—reality and myth. *The Physics of Fluids*, 26(10), 2816–2850.

Hussain, A. F. (1986). Coherent structures and turbulence. *Journal of Fluid Mechanics*, 173, 303–356.

Hussain, A. K. M. F., & Clark, A. R. (1981). On the coherent structure of the axisymmetric mixing layer: A flow-visualization study. *Journal of Fluid Mechanics*, 104, 263–294.

Jasak, H., Weller, H. G., & Gosman, A. D. (1999). High resolution NVD differencing scheme for arbitrarily unstructured meshes. *International Journal for Numerical Methods in Fluids*, 31(2), 431–449.

Karholm, F. P., Weller, H., & Nordin, N. (2007). Modelling injector flow including cavitation effects for diesel applications. In *ASME/JSME 2007 5th joint fluids engineering conference* (pp. 465–474). American Society of Mechanical Engineers.

Leroux, J. B., Coutier-Delgosha, O., & Astolfi, J. A. (2005). A joint experimental and numerical study of mechanisms associated to instability of partial cavitation on two-dimensional hydrofoil. *Physics of Fluids*, 17(5), 052101.

Li, S., Aung, N. Z., Zhang, S., Cao, J., & Xue, X. (2013). Experimental and numerical investigation of cavitation phenomenon in flapper–nozzle pilot stage of an electrohydraulic servo-valve. *Computers & Fluids*, 88, 590–598.

Li, 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.

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(5&6), 579–604.

Mauger, C., Méès, L., Michard, M., Azouzi, A., & Valette, S. (2012). Shadowgraph, Schlieren and interferometry in a 2D cavitating channel flow. *Experiments in Fluids*, 53(6), 1895–1913.

Merkle, C. L. (1998). Computational modelling of the dynamics of sheet cavitation. In proceedings of the 3rd international symposium on Cavitation, Grenoble, France, 1998.

Mou, B., He, B. J., Zhao, D. X., & Chau, K. W. (2017). Numerical simulation of the effects of building dimensional variation on wind pressure distribution. *Engineering Applications of Computational Fluid Mechanics*, 11(1), 293–309.

Oshima, S., & Ichikawa, T. (1985). 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.

Pan, X., Wang, G., & Lu, Z. (2011). Flow field simulation and a flow model of servo-valve spool valve orifice. *Energy Conversion and Management*, 52(10), 3249–3256.

Patankar, S. (1980). *Numerical heat transfer and fluid flow*. New York: CRC press.

Payri, F., Payri, R., Salvador, F. J., & Martínez-López, 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.

Posa, A., Oresta, P., & Lippolis, A. (2013). Analysis of a directional hydraulic valve by a direct numerical simulation using an immersed-boundary method. *Energy Conversion and Management*, 65, 497–506.

Saha, K., Abu-Ramadan, E., & Li, X. (2013). Modified single-fluid cavitation model for pure diesel and biodiesel fuels in direct injection fuel injectors. *Journal of Engineering for Gas Turbines and Power*, 135(6), 062801.

Sarkar, A., & Schlüter, J. (2014). Large eddy simulations of turbulent mixing layers excited with two frequencies. *Flow, Turbulence and Combustion*, 92(3), 651–671.

Schmidt, D. P., & Corradini, M. L. (2001). The internal flow of a flow model of servo-valve orifice. *Journal of Fluid Mechanics*, 411(6), 062801.

Schneider, G. H., & Sauer, J. (2001). Physical and numerical modeling of unsteady cavitation dynamics. In *Fourth International Conference on Multiphase Flow* (Vol. 1). ICMF New Orleans.

Shamsborhan, H., Coutier-Delgosha, O., Caignaert, G., & Nour, F. A. (2010). Experimental determination of the speed of sound in cavitating flows. *Experiments in Fluids*, 49(6), 1359–1373.

Singhal, A. K., Athavale, M. M., Li, H., & Jiang, Y. (2002). Mathematical basis and validation of the full cavitation model. *Journal of Fluids Engineering*, 124(3), 617–624.
Som, S., Aggarwal, S. K., El-Hannouny, E. M., & Longman, D. E. (2010). Investigation of nozzle flow and cavitation characteristics in a diesel injector. *Journal of Engineering for Gas Turbines and Power*, 132(4), 042802.

Stanley, C., Barber, T., & Rosengarten, G. (2014). Re-entrant jet mechanism for periodic cavitation shedding in a cylindrical orifice. *International Journal of Heat and Fluid Flow*, 50, 169–176.

Ubbink, O. (1997). *Numerical prediction of two fluid systems with sharp interfaces* (Ph. D. Thesis), Imperial College of Science, UK.

Wang, C., Huang, B., Wang, G., Zhang, M., & Ding, N. (2017). Unsteady pressure fluctuation characteristics in the process of breakup and shedding of sheet/cloud cavitation. *International Journal of Heat and Mass Transfer*, 114, 769–785.

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.

Weller, H. G. (2008). A new approach to VOF-based interface capturing methods for incompressible and compressible flow. OpenCFD Ltd. Report TR/HGW, 4.

Williams, J. F. (1963). The noise from turbulence convected at high speed. *Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences*, 255(1061), 469–503.

Winant, C. D., & Browand, F. K. (1974). Vortex pairing: The mechanism of turbulent mixing-layer growth at moderate Reynolds number. *Journal of Fluid Mechanics*, 63(2), 237–255.

Winklhofer, E., Kull, E., Kelz, E., & Morozov, A. (2001). Comprehensive hydraulic and flow field documentation in model throttle experiments under cavitation conditions. In *Proceedings of the ILASS-Europe conference*, Zurich (pp. 574–579).

Xing, T., Li, Z., & Frankel, S. H. (2005). Numerical simulation of vortex cavitation in a three-dimensional submerged transitional jet. *Journal of Fluids Engineering*, 127(4), 714–725.

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.

Yuan, C., Song, J., & Liu, M. (2019). Investigation on flow dynamics and governing mechanism of choked flow for cavitation jet through poppet valve. *International Journal of Heat and Mass Transfer*, 129(3), 113–131.

Yule, A. J. (1978). Large-scale structure in the mixing layer of a round jet. *Journal of Fluid Mechanics*, 89(3), 413–432.

Zaman, K. B. M. Q., & Hussain, A. K. M. F. (1980). Vortex pairing in a circular jet under controlled excitation. Part 1. General jet response. *Journal of Fluid Mechanics*, 101(3), 449–491.

Zhang, S., & Li, S. (2015). Cavity shedding dynamics in a flapper–nozzle pilot stage of an electro-hydraulic servo-valve: Experiments and numerical study. *Energy Conversion and Management*, 100, 370–379.

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.