Investigation of the Flow-Induced Noise and Optimization Design of a Short Tube Throttle Valve  

K. P. Zhang¹, D. Z. Wu¹,²,†, J. Y. Wang³, Y. X. Song³, Z. B. Feng¹, Y. C. He¹ and L. H. Xuan⁴  

¹ Institute of Process Equipment, College of Energy Engineering, Zhejiang University, Hangzhou 310027, China  
² The State Key Laboratory of Fluid Power Transmission and Control, Zhejiang University, Hangzhou 310027, China  
³ School of Thermal Engineering, Shandong Jianzhu University, Jinan 250101, Shandong, China  
⁴ Zhejiang DunAn Artificial Environmental Equipment Co., Ltd., Zhuji 311835, China  

†Corresponding Author Email: wudachuan@zju.edu.cn

ABSTRACT

Compared with the globe valves and other on-off valves, cavitation and noise are more severe in the throttle control valve of the refrigerating system due to the higher saturated vapor pressure of the refrigerant. A short tube throttle valve (STTV) is an important throttle control component and cavitation noise easily occurs because of the pressure drop caused by flow throttling. To suppress the cavitation noise, this paper proposed a numerical investigation verified by experimental data into cavitation and noise characteristics inside the STTV. Three kinds of typical throttle valve structures are proposed in this study. The flow field and cavitation noise of the STTV under different refrigerant flow rates were analyzed numerically. The noise levels of the prototype and three optimized structures are comparatively assessed and analyzed. The findings indicate that the level of cavitation noise rises with an increase in flow rate through the STTV. Specifically, when the flow rate transitions from 0.014 kg/s to 0.024 kg/s, there is a corresponding increase in noise levels, moving from 102.8 dB to 122.5 dB. There is less cavitation noise upstream and the flow is stable, while the noise is mainly concentrated downstream with symmetrical distribution characteristics. The distribution seems to have small noise near the internal wall, while large noise is in the center of the downstream in the STTV, and the maximum noise is observed at the corner of the valve seat of the valve outlet. The optimized valve featuring a slope structure on the valve seat significantly reduces cavitation noise, with a maximum noise reduction of 24.94 dB compared to the prototype valve.

1. INTRODUCTION

STTV is predominantly used in air conditioning to manage the phase of refrigerant, thereby facilitating cooling and heating processes within these systems. The principles of throttling and pressure drop in STTV are also implemented by the orifice plate, similarly to how they are applied in traditional capillaries. In fact, there is only liquid refrigerant at the valve inlet, and a gas–liquid two-phase flow exists at the outlet. This is because the throttle valve produces a pressure drop during the throttling process. When the refrigerant flows into the STTV at high pressure, the local pressure in the throttle valve is reduced due to the throttling effect of the valve. When the pressure is below the saturated vapor pressure of the refrigerant, issues such as severe cavitation often arise in the throttle valve's flow, resulting in a two–phase flow state at the valve's outlet. (Park et al., 2022; Li et al., 2022). Vibration and noise occur when the local pressure is higher than the saturated vapor pressure of the refrigerant (Valdés et al., 2014; Zhang et al., 2020). Consequently, cavitation inception and collapse are the main reasons for STTV noise.

Numerous pertinent studies have been conducted due to the detrimental impacts of cavitation and noise in throttle valves (Kim & O’Neal 1994; Yang & Zhang, 2005; Zhang & Yang, 2005; Valdés et al. 2014; Xu et al., 2016; Liu et al., 2017, 2019). The intensity of cavitation and flow noise of the throttle valve is related to many factors, such as the structure and number of valves, the state parameters of the inlet and outlet, and the shape and flow rate of the fluid. Zhao et al. (2022) found that the number of throttling windows, pressure difference, valve opening and flow direction of the sleeve regulating valve would have an impact on cavitation. Liu et al. (2022) utilized a visualization system with high-speed to investigate the intention and evolution of cavitation in the internal flow channel of an injector ball.
valve. The internal flow field is obtained through numerical simulation with dynamic boundary conditions, and the formation mechanism and nomenclature in the cavitation flow of the valve are investigated. The consequences reveal that cavitation initially occurs in the clearance of the ball valve, followed by the deflector hole and exit control hole. The primary cause of cavitation is the low-pressure area created by throttling. Additionally, streamline and vortices significantly impact the distribution of cavitation. Wang et al. (2022) conducted a comprehensive investigation into the flow and cavity characteristics of tandem multistage pressure-reducing valves, utilizing both experimental and numerical simulation methods in their analysis. It is found that increasing the flow angle is beneficial for increasing the flow coefficient. The cavitation intensity achieves the minimum value when the rake angle of the flow channel is 60% and the flow chamfer is less than 6mm. Wang et al. (2022) analyzed the flow cavitation phenomenon of the control valve in the pipeline through numerical simulation and found that the structure of the control valve at the throttling stage had a significant influence on the cavitation intensity. Habibnejad et al. (2022) investigated the influence of globe valve opening and inlet flow rate on cavitation intensity and found that the cavitation intensity increases significantly with increasing inlet velocity and decreasing opening. Yuan et al. (2019) studied the cavitation–vortex interaction of the internal jet flow in the poppet valve. Cavitation continues to exist in the core of the vortex, indicating that cavitation has a strong correlation with vortex dynamics. Ye et al. (2019) found that cavitation was an important factor affecting the efficiency of the micropump through numerical simulation. It has been suggested that enhancing the number of inlet valves can effectively mitigate cavitation, enhance pump flow, and consequently, improve pump efficiency. Yang et al. (2019) proposed deploying continuous micro jets around each nozzle injection flow of the baffle-nozzle pilot valve to restrain cavitation. Liang et al. (2016) conducted a numerical analysis to investigate the relationship between the unsteady cavitation process and inlet pressure fluctuation in a water hydraulic poppet valve with high inlet pressure. The study examined the impact of inlet pressure fluctuation and valve groove on the unsteady cavitation process and flow field. Cavitation can be reduced by the presence of a valve groove. Semrau et al. (2019) investigated the cavitation noise of hydraulic systems through numerical simulation and experiment, and found that reducing discharge pressure can reduce the cavitation noise of the system. Ou et al. (2015) found that cavitation in the flow field decreases to varying degrees with increasing outlet pressure or decreasing the inlet pressure. Zhou et al. (2022) experimentally studied the flow noise characteristics of two-phase refrigerant R410A when it flows through an electronic expansion valve under different flow conditions, and found that a low refrigerant flow rate is helpful reducing the flow noise. Berestovitskiy et al. (2015) provided a basis for selecting the perforating sleeve form of a low noise regulator for the hydraulic system. Zhang et al. (2022) conducted both numerical simulations and experimental research, which revealed that the noise generated by two-phase flow increases with an increase in pressure difference between the inlet and outlet of the electronic expansion valve. To address this issue, they proposed a structure featuring a spiral guide vane downstream, thereby achieving the goal of mitigating flow noise.

Based on the existing literature, numerous significant approaches have been proposed to investigate the flow and noise of the throttle valve, including the examination of the impact of structural parameters on the flow field, the study of flow field visualization, the study of the cavitation phenomenon and the study of the combination of experiments and numerical simulation. All these studies provide effective support for revealing the fluid flow and cavitation phenomena in the throttle valve.

Unfortunately, there is insufficient evidence to demonstrate the impact of structural factors on the flow and noise characteristics of the STTV, which is insufficient to reveal either the turbulence structure or its correlation with noise induced by flow. The analysis of

<table>
<thead>
<tr>
<th>Nomenclature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \rho )</td>
<td>flow density</td>
</tr>
<tr>
<td>( k )</td>
<td>turbulence kinetic energy</td>
</tr>
<tr>
<td>( x_i \cdot x_j )</td>
<td>pressure coefficient</td>
</tr>
<tr>
<td>( u_i )</td>
<td>velocity of ( x_i )</td>
</tr>
<tr>
<td>( \mu )</td>
<td>coefficient of viscosity</td>
</tr>
<tr>
<td>( \varepsilon )</td>
<td>turbulent dissipation rate</td>
</tr>
<tr>
<td>( G_k )</td>
<td>the generation of turbulence kinetic energy due to the mean velocity gradients</td>
</tr>
<tr>
<td>( G_b )</td>
<td>the generation of turbulence kinetic energy due to buoyancy</td>
</tr>
<tr>
<td>( Y_M )</td>
<td>dilatation in compressible turbulence to the overall dissipation rate</td>
</tr>
<tr>
<td>( n )</td>
<td>bubble number densities</td>
</tr>
<tr>
<td>( P_i )</td>
<td>the saturated vapor pressure at that state</td>
</tr>
<tr>
<td>( \rho_v \cdot \rho_l )</td>
<td>vapor density and liquid density</td>
</tr>
<tr>
<td>( U )</td>
<td>turbulent velocity</td>
</tr>
<tr>
<td>( \alpha_k )</td>
<td>sound velocity</td>
</tr>
</tbody>
</table>

\( \rho_k, \rho_e \) the turbulent Prandtl numbers for \( k \) and \( \varepsilon \) respectively

\( S_k, S_\varepsilon \) user-defined source terms

\( C_{4\varepsilon}, C_{2\varepsilon} \) constants

\( \alpha \) vapor volume fraction

\( \rho^v \) vapor density

\( \nu^v \) vapor phase velocity

\( R_c \) vapor generation rate

\( R_c \) vapor condensation rate

\( \mathcal{R}_B \) bubble radius

\( P \) liquid pressure

\( F_{\text{sup}}, F_{\text{cond}} \) evaporation and condensation correction factors

\( P_A \) sound power

\( l \) turbulent characteristic length
cavitation flow in valves has mainly focused on velocity and pressure in previous studies. Consequently, the mechanism between flow and noise in STTV is not yet adequate.

In this paper, the flow-induced noise of a short tube throttle valve under different inlet flow rates is analyzed by means of numerical analysis and experimental methods. The flow-induced noise varies with the refrigerant flow rate. Based on the results of numerical analysis, three optimization schemes of the valve body structure of the STTV were proposed, and the flow-induced noise characteristics of the prototype and the optimized models were compared. The optimization model obviously reduced the flow-induced noise. This work aims to serve as a reference for the optimal design of STTV, which holds significant importance in reducing noise during the operation of air conditioning systems.

2. NUMERICAL MODEL AND METHOD

The STTV exhibits remarkable characteristics, including phase transition, vortex generation, and cavitation evolution, due to the pronounced throttling effect resulting from the abrupt contraction and expansion of the flow area. The Reynolds–averaged Navier–Stokes equation (RANS) method is employed, which effectively captures the cavitation process during phase transition due to throttling. In this section, the simplified three-dimensional model of the flow region of STTV was developed. To capture the intricate details of cavitation flow, Fluent software was employed and various models were utilized such as the cavitation model, turbulence model, multi-phase flow model, and noise model for cavitation two-phase flow based on the Reynolds-Averaged Navier Stokes (RANS) method.

2.1 Physical Model

A common STTV for air conditioners is selected in this study. The three-dimensional geometry of the valve is shown in Fig.1. The STTV primarily consists of a valve seat, pipe and filter device. The internal throttling of the STTV is predominantly generated in the region demarcated by the red dashed line in Fig. 1, which corresponds to a throttling orifice passing through the valve seat. To investigate the cavitation and noise issues caused by the short tube throttle, the computational fluid domain is generated through reverse modeling of the aforementioned model, as illustrated in Fig.2.

2.2 Mathematical Modeling

2.2.1 Turbulence Model

The interior of the STTV experiences a significant pressure difference and intricate eddy currents, which could cause the calculation to be unstable and not easily to converge, and the fluids in the STTV are characterized by high Reynolds numbers. The standard \( k-\varepsilon \) turbulence model, a two-equation turbulent model, is deemed suitable for meeting the simulation requirements. This model also boasts an optimal balance between accuracy and computational efficiency. Consequently, it has been chosen for use in this study.

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_i} \left[ \mu + \mu_t \frac{\partial k}{\partial x_i} \right] + G_k + G_p - \rho \varepsilon - Y_m + S_k \tag{1}
\]

\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_i} \left[ \mu_t \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1\varepsilon} \frac{\varepsilon^2}{k} - C_{2\varepsilon} \rho \varepsilon + S_\varepsilon \tag{2}
\]

\( \rho \) is the flow density, kg/m³; \( k \) is the turbulence kinetic energy, J; \( \varepsilon \) is the coordinate positions; \( u_i \) is the velocity of \( x_i \), m/s; \( \mu \) is the coefficient of viscosity, Pa·s; \( \varepsilon \) is the turbulent dissipation rate; \( G_k \) is the generation of turbulence kinetic energy produced by the mean velocity gradients, J; \( G_p \) is the generation of turbulence kinetic energy produced by buoyancy, J; \( Y_m \) is the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate; \( \rho_k \) and \( \rho_\varepsilon \) are the turbulent Prandtl numbers for \( k \) and \( \varepsilon \) respectively; \( S_k \) and \( S_\varepsilon \) are user-defined source terms; and \( C_{1\varepsilon}, C_{2\varepsilon} \) and \( C_{3\varepsilon} \) are constants.

The turbulent (or eddy) viscosity \( \mu_t \) is computed by combining \( k \) and \( \varepsilon \) as follows:
\[ \mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \] 

where \( C_\mu \) is a constant.

The model constants \( C_1, C_2, C_3, \rho_k \) and \( \rho_e \) are assigned the following default values:

\[ C_1 = 1.44, \quad C_2 = 1.92, \quad C_3 = 0.09, \quad \rho_k = 1.0, \quad \rho_e = 1.3. \]

These model constant values are obtained through a large number of experiments and continuous correction, which is proven to be widely used for wall–bounded and free–shear flows with good accuracy and robustness.

### 2.2.2 Multiphase Flow Model

The Eulerian model, mixture model and VOF (Volume of Fluid) model are commonly used multiphase flow models, and the Eulerian model is mainly used for bubble columns, upwelling and other more complex problems. In consideration of the simplicity of the object of this study, mixture model or VOF model will be selected. Moreover, the mixture model allows interpenetration between phases, which is more suitable for two-phase flow. Consequently, the mixture model is employed to represent the cavitation flow of the STTV.

#### 2.2.3 Cavitation Model

Cavitation is a phenomenon that arises when the static pressure of a fluid falls below its vapor pressure at a specific temperature. By comparing the Schonn–Sauer and Zwart–Gerber–Belamri cavitation models, the Schonn–Sauer model is used to simulate the cavitation phenomenon, taking into account the phase change process between the gas–liquid phases and the effect of turbulent pressure pulsation on the cavitation pressure. The corresponding mathematical control equations are as follows:

\[ \frac{\partial}{\partial t} (\alpha \rho) + \nabla \cdot (\alpha \rho \bar{V}) = R_v - R_e \]  

\[ \mathbf{R}_b = \left( \frac{\alpha}{1 - \alpha} \right)^{\frac{1}{3}} \]  

when \( P \leq P_{sv} \),

\[ R_v = F_{\text{vap}} \frac{\rho_l \rho_v}{\rho} \alpha (1 - \alpha) \frac{3}{2 \mathbf{R}_b} \sqrt{\frac{2(P_{sv} - P)}{3 \rho_v}} \]  

when \( P > P_{sv} \),

\[ R_e = F_{\text{cond}} \frac{\rho_l \rho_v}{\rho} \alpha (1 - \alpha) \frac{3}{2 \mathbf{R}_b} \sqrt{\frac{2(P - P_{sv})}{3 \rho_l}} \]

where \( \alpha \) is the vapor volume fraction; \( \rho_l \) is the vapor density, \( \text{kg/m}^3 \); \( \bar{V}_v \) is the vapor phase velocity, \( \text{m/s} \); \( R_v \) is the vapor generation rate; \( R_e \) is the vapor condensation rate; \( \mathbf{R}_b \) is the bubble radius, \( \text{m} \); \( n \) is the bubble number densities; \( P \) is the liquid pressure, \( \text{Pa} \); \( P_{sv} \) is the saturated vapor pressure at that state, \( \text{Pa} \); \( F_{\text{vap}} \) and \( F_{\text{cond}} \) are evaporation and condensation correction factors, respectively; and \( \rho_l \) and \( \rho_v \) are the vapor density and liquid density, respectively, \( \text{kg/m}^3 \).

#### 2.2.4 Noise Model

The broadband noise source model has the characteristic of higher accuracy in the case of more noise sources, and the cavitation noise is mainly distributed in the broadband range. Hence, the broadband noise model (broadband noise sources) is selected for noise analysis, and the control equations are as follows:

\[ P_A = \alpha \rho_l \left( \frac{U^3}{l} \right) \frac{U^5}{\alpha_0^5} \]

where \( P_A \) is the sound power, \( \text{W/m}^2 \); \( \alpha \) is the model constant; \( U \) is the turbulent velocity, \( \text{m/s} \); \( l \) is the turbulent characteristic length, \( \text{m} \); and \( \alpha_0 \) is the sound velocity, \( \text{m/s} \).

### 2.2.5 Mesh and Grid Independency

The STTV structure is relatively straightforward, necessitating the use of a larger grid for mesh generation. Due to the small diameter and area of the throttle region, the grids of this region are locally refined to fulfill the needs of the three-dimensional turbulent model and accurately depict the liquid–vapor interface. Boundary layer grids were generated in the near-wall region, and the range of the \( \rho^+ \) values on the wall of the flow domain was 10–30. Four distinct mesh models are generated through the use of varying mesh sizes, as depicted in Fig. 3.

To eliminate the impact of grid number on numerical outcomes, the simulation domain's grid count is independently verified. In previous simulation research on cavitation flow, it was found that the parameter of flow rate parameter is significantly influenced by the number of grids in the grid independency test (Jin et al., 2020). Considering the main purpose of this study, the grid independence test is verified when the inlet flow rate is 0.016 kg/s and the outlet pressure is 1 Mpa. The grid independence test results for the outlet flowrate are shown in Table 1.

The table clearly illustrates that the number of grids is directly proportional to the calculation result. The more grids there are, the smaller the deviation is, and the deviation of the outlet flow rate is within 0.1% when the grid number is 540,000. In consideration of the smaller total number of grids, the calculation grid of 540,000 is selected to ensure the calculation accuracy and it will not take long to compute.

#### 2.2.6 Boundary Conditions and Initial Conditions

The R134a refrigerant is used as the medium for calculation. Two monitoring points were selected as the condition of convergence which are marked by the red points in Fig. 4.

To investigate the influence of varying inlet flow rates on two-phase flow and noise, boundary conditions were applied to study flow–induced noise in the throttling region of a STTV under diverse flow conditions.
Specifically, the outlet static pressure was maintained at 1 Mpa, while the mass flow rate inlet was varied from 0.014 kg/s to 0.024 kg/s at intervals of 0.02 kg/s.

3. RESULTS AND DISCUSSION

The unsteady two-phase flow inside the STTV was investigated by numerical methodology in this section. The flow field and cavitation distribution were obtained. Furthermore, the acoustic power level distribution of cavitation with different flow rates was analyzed.

3.1 Analysis of Simulation Results

3.1.1 Analysis of the Pressure Field and Velocity Field

The distributions of flow field pressure and velocity are shown in Fig. 6 and Fig. 7, respectively, for various flow rates (0.014 to 0.024 kg/s) and an outlet pressure of 1MPa. The refrigerant flowed through the STTV with a higher pressure at the inlet and a lower pressure at the outlet throttles based on the principle of the throttle orifice plate. It can be observed from Fig. 6, which the pressure is higher at the inlet and gradually decreases after throttling because of the energy transforming from pressure potential energy to kinetic energy and the pressure at the outlet is almost 1 Mpa. It could also be obtained that there is a higher pressure drop at the throttle orifice with a larger inlet flow rate and a lower pressure drop with a smaller inlet flow rate. In other words, the flow rate can influence the pressure distribution within the valve's flow field. This phenomenon arises due to the larger inlet flow rate possessing greater potential energy. The larger flow rate would have a larger pressure drop if it kept the outlet pressure constant. When the pressure changes, the refrigerant may be accompanied by cavitation. At this point, a temperature change will occur at the orifice. However, because temperature changes only occur in the local area of the orifice, the impact on noise is not significant, so this article will not elaborate separately.
Fig. 5 Pressure field under different inlet flow rates

Fig. 6 Velocity field under different flow rates

Fig. 7 Maximum flow rate under different flow rates
3.1.2 Analysis of the Velocity Field

The refrigerant flows through the STTV at a certain flow rate near the throttle orifice, where the flow velocity increases sharply due to the reduction in the contraction section area. It can be observed from the velocity field presented in Fig. 7 that the maximum flow velocity within the flow field gradually increases with an increase in flow rate. The velocities upstream and downstream near the outlet of the valve are relatively low, and their velocity gradients can be almost entirely disregarded. However, the velocity and the velocity gradient at the throttle orifice are larger due to the pressure potential energy transforming into kinetic energy, which is accompanied by a high velocity, and there is a form of jet with a velocity gradient after the orifice outlet due to the sudden expansion of the tube diameter variation. As depicted in Fig. 8, the maximum flow velocity varies under varying flow rates. A noticeable trend is observed where the highest velocity occurs in the throttle area and progressively rises with an increase in the inlet flow rate.

The velocity in the center of the throttle hole is larger, and the velocity in the area near the wall is lower, so a larger velocity gradient is formed.

3.1.3 Gas Phase Volume Fraction Distribution

The refrigerant flows into the short pipe throttle valve in the liquid state, and flows out in gas–liquid two–phase state with throttling and pressure reduction. The gas phase volume fraction distribution of the flow field under varying inlet flow rates is depicted in Fig. 9. From the figure, it is evident that the gas phase volume fraction remains approximately constant despite varying valve flow rates. This can be attributed to the fact that the valve outlet pressure is less than the saturated vapor pressure of the refrigerant, leading to a pronounced downstream cavitation phenomenon. Although the level of cavitation in the flow field is not the same at different flow rates, the distribution states are extremely similar. The gas phase volume is gathered at the downstream area of the valve under different flow rates, which are...
distributed symmetrically. It can also be clearly seen near the throttle hole area of the valve that cavitation is generated along the wall, and there is no accumulation of cavitation at the center, due to the high-velocity flow in this area. However, the downstream flow rate is smaller, and the pressure at the outlet of the valve is lower than the saturated pressure, so the cavitation accumulates downstream, and the cavitation phenomenon is significant.

3.1.4 Acoustic Power Level Distribution

The flow of refrigerant is often accompanied by the generation of noise. Part of the noise is generated by fluid dynamic noise, which results from the collision between fluids and the wall during the fluid flow process; the other part is the two-phase flow generated by fluid cavitation, and cavity shifts in tandem with the fluid until the pressure surpasses the saturated vapor pressure of the refrigerant. The bubble collapse generates a significant amount of energy, some of which is released as radiation and noise. Figure 10 is the acoustic power level distribution of the valve under different flow rates. The noise predominantly manifests in the downstream region of the valve, exhibiting a symmetrical pattern. The peak noise level in the flow field is observed near the inlet of the throttle tube, which is demarcated by a red line. As the flow rate increases from 0.014 kg/s to 0.024 kg/s, the maximum noise value of the flow field escalates from 102.8 dB to 122.5 dB. These results reveal that a gradual rise in the noise level within the flow field as the valve inlet flow rate increases. The acoustic distribution along the Y-direction on the XY section is depicted in Fig. 11,
under conditions where \( G=0.022 \) kg/s. The noise is approximately 0 before point A at the upstream of the valve. There is a sudden increase at point B, which is produced by the flow instability caused by the change in flow direction at the corner position. In the B-C interval, that is, the throttling section, the noise at this position has been reduced, but there is another extreme value when encountering the corner at point C, which is the exit of the throttling hole. The C-D section is located downstream of the valve, where noise levels increase. However, after point D, the noise begins to decrease.

Figure 12 illustrates the noise distribution and trace diagram along the Y direction under different flow rates. It can be observed from the figure that the noise level in section A is almost zero, as confirmed by the trace diagram of the flow field. This indicates that the flow in this area is relatively stable and there is minimal energy loss. There are three noise extremum value regions in the whole flow process. There is a sudden noise increase in region 1, which is due to the shrinkage of the tube diameter during throttling (as shown in the trace diagram). As the flow velocity increases, the conservation of momentum dictates that a portion of the kinetic energy is converted from pressure potential energy. At the same time, the energy loss is radiated in the form of sound energy, so the phenomenon of a sudden increase in noise occurs. The same phenomenon in which noise suddenly increases occurs in region 2, but for a different reason. The flow rate of the refrigerant flowing out of the orifice is relatively large; when the flow direction changes, it will be subject to greater resistance, the energy loss will be greater, and the radiated sound energy will be greater, which is the reason noises suddenly increases. Section B–C has a slope increase in noise. The trace diagram reveals a highly turbulent flow in this section, with the presence of three distinct vortices. Due to the existence of the vortex, the pressure pulsation in this area is more significant, resulting in an increase in flow-induced noise. In the C–D section, the turbulence phenomenon is obviously weakened, and the noise in this area also tends to decrease.

3.2 Experimental Comparison and Analysis

To validate the accuracy of the numerical analysis, the test is performed through the noise experimental platform under the \( G=0.016 \) kg/s condition in this study. As shown in Fig.12, the experimental platform for acoustic signal acquisition was constructed to measure the running noise. The acoustic signal is obtained by the microphone with a sampling rate of 51,200 and a duration of 1 second. The comparison between the experimental and simulated results is conducted under the operating conditions of an outlet pressure of 1 MPa and an inlet flow rate of 0.016 kg/s, as illustrated in Fig. 13. It can be observed from the figure that the frequency...
spectrum distribution of noise obtained by the experiment ranges from 1245.41 Hz to 11573.4 Hz, while the range from 1204.33 Hz to 11472.6 Hz by numerical calculation. Therefore, the experimental results are consistent with those obtained by numerical calculation, which verifies the reliability of the numerical calculation.

Additionally, some test uncertainties could lead to deviations between test results and simulation values. The purity of the refrigerant will affect the test results, and impurities in the refrigerant will affect the flow and noise of the air conditioning system. The noise insulation of the test chamber is an important factor affecting the test results, and poor sealability of the test chamber can result in data that are not authentic. The accuracy and sensitivity of the test equipment is another factor that affects the test results, and the test results obtained by low frequency and accurate equipment lead to considerable error.

4. STRUCTURE OPTIMIZATION OF THE SHORT TUBE THROTTLE VALVE

4.1 Optimization Scheme

The flow velocity increases with the fluid passing through the throttle orifice due to the contraction of the cross-section, and the flow direction of part of the fluid changes abruptly, which tends to cause flow noise. The structure for the throttling region which could provide a design with a buffer area would be beneficial to solve the noise problem. Fillet and slope instead of a perpendicular scheme is an effective solution.

Based on the above factors, three kinds of optimization schemes of STTV have been proposed to relieve the noise problem in the throttling section of the valve, which are shown in Fig. 14. One consists of an STTV with a 90° fillet on the valve seat (Fig. 14(b)). The second is a 15° slope on the inlet and outlet of the valve seat (Fig. 14(c)), and the third consists of a 75° fillet.
4.2 Comparative Analysis of the Prototype and Optimization Models

To mitigate the noise generated by throttling and cavitation in the flow process of the SSTV, the fillet and slope structure on the throttle region can alleviate the dramatic change in the flow, which would introduce cavitation. Accordingly, the noise induced by the cavitation can be suppressed. Considering an inlet flow rate of 0.016 kg/s and a pressure outlet of 1 Mpa as a case study, this paper compares and analyzes the cavitation and noise of the prototype and the optimization scheme.

4.2.1 Gas Phase Volume Fraction

The volume fraction distribution of the gas phase in the flow throttle section of both the prototype and optimization models is depicted in Fig. 15. The flow direction of the fluid is indicated by the line head in the figure. At the inlet, cavitation is weak and not very obvious, as observed in the figure. However, cavitation is visible in the region of the throttling region, especially at the outlet. Cavitation in the original valve and VS–S valve is generated along the throttle orifice wall, while in the VS–F and VS–FS valves, cavitation is generated along the fillet position of the outlet.

4.2.2 Analysis of the Noise Reduction Effect

In the process of fluid flow, due to the throttling effect and the changing direction of fluid flow, resulting in the noise of fluid colliding with the tube wall and fluid colliding with each other, in addition to the bubble generated at the low-pressure area flowing out with the fluid flow, in the region of higher pressure, it will produce bubble collapse, and thus the role of throttling pressure reduction in the short tube throttling valve, mechanical noise will inevitably occur as well as fluid dynamic noise.

Figure 16 shows the prototype and the optimization models acoustic power level distribution. The noise is predominantly observed in the throttling region and

and 15° slope on the inlet and outlet of the valve seat (Fig. 14(d)). The details of the geometric parameters of the three structures are listed in Table 2. The diameters of the throttling hole of these STTVs are the same.

Fig. 15 Comparison of the gas phase volume fraction of the prototype and optimization models

Fig. 16 Acoustic power level distribution of prototype and optimization models
downstream area of the original model, as evident from the figure. The maximum noise level is observed at the corner of the valve seat outlet, with an acoustic power level of 113.75 dB. Therefore, the model was optimized by changing the inlet valve seat structure. The VS-F model is an optimization scheme that changes the inlet and outlet structure of valve seat from perpendicular to the fillet. Although the maximum noise position is also at the corner of downstream valve seat, the noise has been significantly reduced, from 113.75 dB to 108.70 dB compared with the prototype model, which shows that the noise reduction effect is obvious.

The sudden shrinkage of the tube diameter will also cause noise in the flow. Therefore, the inlet of the valve seat for the gradual reduction and expansion scheme (VS–S) is proposed, to increase the variable-speed buffer area, so that the fluid flows in a gradual manner. The same principle is applied to the outlet, so that the original jet-like flow of fluid along the slope, effectively reduces the flow noise. As shown in Fig. 16(c), the noise position is similar to that of the prototype model, noise reduction is effective by replacing the abruptly reduced valve seat with a gradual reduction surface with a slope, and the maximum acoustic power level is decreased from 113.75 dB in the prototype model to 88.81 dB. The scheme of VS–FS which combines VS–F and VS–S also reduces the acoustic power level, but the noise reduction effect is not obvious, and the maximum acoustic power level is decreased from 113.75 dB in the prototype model to 111.75 dB. The reason why the fillet and slope structure for the inlet and outlet of the throttling orifice could reduce the acoustic power level is that both of these schemes can alleviate the impact of upstream fluid and reduce the extent of sudden shrinkage of the tube diameter. Compared with the three optimization models, the slope structure on the valve seat (VS–S) is the best scheme to reduce the cavitation noise.

5. CONCLUSION

In the current study, the two-phase flow of refrigerant flowing through a short tube throttle valve is numerically simulated and validated for reliability, and the flow rate is a factor affecting the flow-induced noise. Through the optimization of the existing model, three distinct types of low-noise short tube throttle valves have been successfully designed, leading to several significant conclusions:

1. The noise generated by two-phase flow increases with an increase in the inlet refrigerant flow rate. Specifically, the maximum noise increases from B102.8 dB to 122.5 dB as the flow rate changes from 0.014 kg/s to 0.024 kg/s, provided that the inlet and outlet pressure remain constant.

2. Three optimization schemes are proposed for the flow characteristics and the location of noise generation: chamfering the valve seat rounding at 90° (VS–F); designing the valve seat with a slope of 15° (VS–S); filleting the corner of valve seat and designing the slope of the valve seat (VS–FS).

3. There is nearly a small amount of noise upstream. In the prototype model and VS–S model, cavitation occurs mainly at the position of the throttle orifice close to the wall, while in the optimized schemes VS–F model and VS–FS model, cavitation predominantly manifests at the outlet corner of the valve seat.

(4) The optimization schemes under comparison were found to be effective in reducing noise levels when compared to the original model. Specifically, the maximum noise of the flow field was reduced by 5.05, 24.94, and 2.0 dB for VS–FS, VS–S, and VS–FS, respectively. Therefore, the optimization scheme of using an inclined plane structure for the valve seat has the most significant effect on suppressing two-phase flow induced noise.

ACKNOWLEDGEMENTS

This work was supported by the National Natural Science Foundation of China (No. 51839010); the key research and development project of Zhejiang Province, China (No. 2021C03133)

CONFLICT OF INTEREST

The author is responsible for declaring any financial or non-financial interests in this part. If no conflict of interest in there, the author must state that they have no conflicts to disclose.

CREDIT AUTHORSHIP CONTRIBUTION STATEMENT


DECLARATION OF COMPETING INTEREST

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

REFERENCES


valve. *Vacuum*, 204, 111335.  
https://doi.org/10.1016/j.vacuum.2022.111335


https://doi.org/10.1016/j.flowmeasinst.2022.102259

https://doi.org/10.1016/j.ijrefrig.2022.11.011