A combined numerical-experiment investigation on the flow and acoustic field of an electronic expansion valve (EEV) is conducted in this paper. In an electronic expansion valve (EEV) under refrigeration condition, it is usually complex about internal flow. There are high pressure gradient and velocity gradient in the flow field, and it also involves the process of flashing phase change, physical property change, and heat transfer, which is difficult to simulate directly. After many explorations in this paper, the flow field of EEV under refrigeration condition is simulated by a four-step computational fluid dynamics (CFD). On this basis, the noise of EEV which is induced by the internal turbulent flow and propagates outward through the shell of EEV is simulated by a Hybrid Method. Numerical simulations of the mass flow rate (MFR) and sound pressure level (SPL) are verified by experimental data. Then, the characteristics of the internal flow field causing the external acoustic radiation are analyzed and used to create an improved design that can reduce the SPL while the MFR hardly changes.

1. Introduction

The inverter air-conditioner is mostly popular with consumers as a result of its energy-saving and emission-reduction. Then, the electronic expansion valve (EEV) (as shown in Figure 1) is the most used throttling element in the inverter air-conditioner. It plays an important role in adjusting the temperature and the flow rate of refrigerant. However, during the throttling process, the parameters of the flow field such as pressure and velocity change dramatically, resulting in strong turbulence, which is easy to produce noise and worsen the user experience. Thus, the study of the flow field and the radiated noise of EEV has been an important phase in design. Over the past decades, a great number of studies have been mainly devoted to large control valves. The research on small throttling expansion equipment in air-conditioner system is still less.

Umeda [1, 2] found, by experiments, that when the refrigerant flows into the expansion valve in a horizontal way, slug flow can be avoided, thus reducing the pressure fluctuation and noise. Kannon [3] also confirmed by experiments that when the vapor content of refrigerant increases suddenly, there will be a transient pressure wave and the noise will increase. Hirakuni [4] suggested that porous metal should be installed in front of the expansion equipment to change the flow mode from the slug flow to bubble flow, thus reducing the noise. Later, Han et al. [5, 6] also analyzed the influence of the annular flow on the acoustic performance of the expansion valve and proposed to reduce the noise by controlling the flow rate and the dryness of the refrigerant. Enrique and George [7] studied the influence of system operation on the noise of expansion equipment through experiments. When the mass flow rate and outlet pressure of the system are constant, the noise increases linearly with the inlet pressure and the outlet dryness, respectively.

Under refrigeration condition, the refrigerant passing through the EEV will cause a great pressure loss, and then the phase change will occur, forming a vapor-liquid two-phase flow. According to the study, this process is called flashing phase change [8]. Recently, the numerical methods such as CFD (computational fluid dynamics) have been widely used to study the flashing phase change. Yazdani et al. [9] used the mixture model in ANSYS FLUENT to simulate the flashing of CO2 through nozzles and ejectors and obtained that the...
boiling near the center of the nozzle is the dominant factor of phase transition. Lee et al. [10] used OPENFOAM to simulate the flashing in the nozzle based on the uniform relaxation model. As a matter of fact, in order to reveal the noise source of the EEV, it is necessary to conduct a numerical simulation of its flow field. However, due to the existence of flashing phase change, the difficulty of the simulation is also increased, which will be reflected in section 2. Meanwhile, how to decrease the sound radiation of the EEV has been a challenging question for air-conditioner manufacturers. This paper aims at answering this question.

In this paper, the flow and acoustic field of EEV are studied by CFD and CAA (computational aerodynamics acoustic) in ANSYS FLUENT and ACTRAN. It consists of numerical simulation, experimental verification, and analysis of noise mechanism. Section 2 and Section 3, respectively, detail the CFD and CAA simulations including the meshes of the computational model and the selection of algorithm. In section 4, the characteristics of the internal flow field causing the external acoustic radiation are analyzed. Section 5 describes the development of the improved scheme for EEV based on the aforementioned results. Section 6 concludes the paper.

2. CFD Simulation

The numerical calculation of the flow field in EEV under refrigeration condition with phase change is completed in ANSYS FLUENT 19.0.

2.1. The Complexity of Simulation. The opening degree of the EEV is represented by the number of pulses. One pulse corresponds to the lift of the valve needle at 0.00625 mm. When the pulse number is 0, the EEV is completely closed. When the pulse number is 500, the EEV is fully open. The object of numerical simulation is the EEV with 96 pulses in the opening degree. The lifting height of the valve needle is 0.6 mm and the flow passage at the throttle port is annular. The inner radius (i.e., the valve needle) is 0.565 mm and the outer radius (i.e., the valve throat) is 0.675 mm, between which the difference is only 0.11 mm. So it makes a large pressure gradient at this point. In addition, the diameter of the inlet pipeline is 6.4 mm, and according to the continuity condition, the flow rate remains unchanged, so the velocity increases sharply, that is, there is a large velocity gradient at the throttle port as well. Besides, in the process of flashing, the temperature drops dramatically under the influence of intense heat and mass transfer. Therefore, the effects on the physical parameters of the refrigerant, such as density, dynamic viscosity, specific heat, and thermal conductivity, cannot be ignored. Then, the change of physical properties must be considered in the numerical simulation, so that the flow variables and physical property parameters can be coupled.

2.2. Meshing of Computational Model. First of all, it is necessary to mesh appropriately the computational model of EEV to CFD simulation. In this paper, a polyhedral mesh is used, which is a kind of unstructured mesh. It combines the advantages of tetrahedral mesh and hexahedron mesh and has better geometric adaptability and less mesh number. As shown in Figure 2, the maximum mesh size of the computational domain is 0.3 mm, and the minimum mesh size is 0.02 mm in the throttle port. Besides, in order to ensure accuracy, the growth ratio between large and small meshes is set to 1.05. The number of meshes in the whole computational domain is about 900000.

2.3. Flashing Model and Boundary Conditions. VOF (Volume of Fluid) model is suitable for capturing phase interface and has advantages in predicting jet. In this paper, the jet will form, and phase transition will undergo after the refrigerant passes the throttle port, so the VOF multiphase flow model is selected. In addition, the Evaporation–Condensation model in ANSYS FLUENT, a phase transition model driven by thermal imbalance and more in line with the process of flashing, is adopted as the model of phase transition. In the process of flashing, the liquid phase releases heat by vaporization and transfers energy to the vapor phase. Meanwhile, the temperature decreases. This process involves the transfer of energy, so it is also necessary to add the solution of the energy equation in ANSYS FLUENT.
2.4. Setting of Physical Parameters. Before and after throttling, the pressure and temperature difference are large, so that the physical parameters of the refrigerant also change greatly, as shown in Table 1.

Besides, the physical parameters of liquid and vapor phase changes with the variety of temperatures, so it is necessary to set the physical parameters to improve the accuracy of the simulation. Within the temperature range of 253K to 323K, the physical parameters are fitted as the quadratic function of temperature, namely:

\[ y = a_0 + a_1T + a_2T^2. \]  

(1)

The fitting coefficient is shown in Table 2.

In the process of flashing, when it gets to the saturation temperature, the liquid refrigerant starts to vaporize. With the variety of pressure, the saturation temperature also changes. Therefore, it is necessary to fit between saturation temperature and pressure. The result is as follows:

\[ T_s = 234.88 + 5.38 \times 10^{-5}p - 9.18 \times 10^{-12}p^2. \]  

(2)

Here, \( T_s \) is the saturation temperature and \( p \) describes the pressure.

2.5. Numerical Computations. The flow is complex under refrigeration condition, and it consists of a high pressure and velocity gradient, and also involves phase transition, heat transfer, and physical parameters change in the flow field. For such a complex situation, it is very difficult to converge if all the above conditions are directly added to the calculation. In this paper, after many explorations, a four-step method is obtained, in which the above conditions are gradually added in each step.

The first step is the calculation of steady single-phase flow with constant physical parameters, which provides a good pressure field and velocity field for the calculation of multiphase flow. The Standard \( k-\varepsilon \) model is used for simulation and the second-order upwind method is used in the discretization of pressure and momentum, and the SIMPLE algorithm is utilized to solve the pressure–velocity coupled equations.

The second step is to add the multiphase flow equation and calculate the steady multiphase flow with constant physical parameters. After finishing, good temperature field and phase fraction field are obtained to prepare for the calculation of multiphase flow with variable physical parameters.

The VOF multiphase model is used for simulation. The Evaporation–Condensation model is adopted as the model of phase transition and the COUPLED algorithm and the pseudo-transient method are utilized to solve the pressure-velocity coupled equations.

The third step is to calculate the steady multiphase flow with variable physical parameters, which needs to set the parameters varying with temperature.

Finally, the transient multiphase with variable physical parameters flow is calculated. The large eddy simulation model is used for simulation, and the second-order upwind method is used in the discretization of pressure and momentum, and the SIMPLE algorithm is utilized to solve the pressure–velocity coupled equations. The time step is \( 5 \times 10^{-6} \) s.

After the four steps, the LES method for the transient simulation of multiphase flow, the VOF multiphase model, the Evaporation–Condensation phase change model, and physical parameters varying with temperature have all been added into the solution to obtain the simulation result which is the most consistent with the actual flow.

In order to verify the accuracy of computations, the mass flow rates (MFR) of EEV corresponding to the lift of the valve needle 0.6, 1.25, and 1.875 mm are presented, respectively, and compared to the experimental results in Figure 3. It is shown that the simulation model in this paper is reliable and the MFR of EEV could be predicted accurately.

3. CAA Simulation

In the process of flow, complex turbulence is often generated. Especially when the refrigerant passes through the EEV, the flow field will be more complex, resulting in producing noise. In this paper, the main noise source of the EEV is the turbulence acting on the solid wall through the fluid-solid coupling action, which causes the vibration of the solid wall, and then produces noise. This excitation is called turbulent wall pressure fluctuation (TWPF). Since TWPF comes from unsteady turbulence, it can be obtained by sampling the wall pressure of the EEV and then performing Discrete Fourier Transform (DFT). The numerical simulation of the acoustic field is carried out in ACTRAN 17.0.

3.1. Meshing of Computational Model. In the acoustic simulation, the fluid inside the pipe, the pipe, and the air outside the pipe all need to participate in the calculation, so the three parts all need to be meshed. In addition, acoustic meshes are limited by the wavelength of the acoustic field: at least 6 meshes are required for each wavelength within a linear cell. The specific calculation formula is as follows:

\[
\begin{align*}
\lambda_{\text{min}} &= \frac{c}{f_{\text{max}}} \\
\lambda_{\text{max}} &= \frac{6}{\lambda_{\text{min}}}
\end{align*}
\]  

(3)
Here, $f_{\text{max}}$ is the maximum frequency of the acoustic field, $\lambda_{\text{min}}$ is the minimum wavelength of the acoustic field, $l_{\text{max}}$ represents the maximum size of the acoustic field mesh, and $c$ is the velocity of the sound.

Because the acoustic model is complex and involves a variety of computational domains, it is difficult to complete the meshing of the whole sound field model at the same time. Therefore, the three parts of the fluid domain, pipe domain, and air domain are meshed, respectively, in this paper. Generally speaking, the quality of the mesh affects the calculation accuracy, and the number of meshes affects the calculation speed. When dividing the mesh, the hexahedral mesh usually has high quality, but the geometric adaptability is not good, while the tetrahedral mesh has good geometric adaptability, but the accuracy is relatively low, and the number of meshes generated is large.

In this paper, for the fluid domain in the pipe, in order to ensure the calculation accuracy of the sound source, a high-quality acoustic grid in the fluid domain is required. At the same time, by observing the structural model of the fluid domain, its inlet and outlet pipeline area is a regular cylinder, while the structure of the fluid region is located in the valve body, such as the valve cavity, throat, and valve needle, is more complex. Therefore, the mixed grid is used to divide the fluid domain in the pipe. That is to say, hexahedral mesh division is carried out in the fluid domain inside the two pipeline regions, and tetrahedral mesh is used in the fluid domain inside the valve body, so as to ensure the mesh quality and reduce the number of meshes at the same time. Similarly, for the pipeline solid domain, the above analysis results can also be imitated. The combination of hexahedral grid and tetrahedral grid will be used to divide the hexahedral grid of the inlet pipeline and outlet pipeline, and the tetrahedral grid will be used for the valve body in the middle. Finally, for the air domain, because its geometric model is complex, it is difficult to divide into multiple small parts, and the outer boundary grid of the air domain needs good geometric adaptability, so that it can better fit the boundary. Therefore, tetrahedral grid generation is directly carried out for the whole air domain.

All the above meshing processes are carried out in ANSYS ICEM CFD, and the completed mesh is shown in Figure 4.

### 3.2. Sampling of the Sound Source.

In acoustic calculation, the sampling frequency of the acoustic field is determined by the time step of the flow calculation. According to NYQUIST sampling theorem, the highest frequency of the acoustic field that can be restored is half of the sampling frequency. The time step of the flow calculation is $5 \times 10^{-6}$ s, and the sampling frequency is 200000 Hz. Therefore, the maximum frequency of the acoustic field that can be restored is 100000 Hz. The frequency range of audible sound is 20 Hz–20000 Hz. Then, the sampling is conducted every 5 steps to reduce the sampling frequency of the flow field to 50000 Hz. Therefore, the maximum frequency of the acoustic field can be restored is 25000 Hz. For the minimum quality acoustic grid in the fluid domain is required.

### Table 1: Physical parameters.

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Density (kg/m$^3$)</th>
<th>Dynamic viscosity (Pa·s)</th>
<th>Specific heat (J/kg/K)</th>
<th>Thermal conductivity (W/m/K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Liquid</td>
<td>Inlet</td>
<td>1016.3</td>
<td>$1.05 \times 10^{-4}$</td>
<td>1806.1</td>
</tr>
<tr>
<td></td>
<td>Outlet</td>
<td>1121.1</td>
<td>$1.48 \times 10^{-4}$</td>
<td>1588.7</td>
</tr>
<tr>
<td>Vapor</td>
<td>Outlet</td>
<td>43.1</td>
<td>$1.28 \times 10^{-5}$</td>
<td>1262.1</td>
</tr>
</tbody>
</table>

### Table 2: Fitting of the variable physical parameters.

<table>
<thead>
<tr>
<th>Physical property</th>
<th>Fitting coefficient</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (kg/m$^3$)</td>
<td>$a_0$</td>
</tr>
<tr>
<td>Dynamic viscosity (Pa·s)</td>
<td>$a_1$</td>
</tr>
<tr>
<td>Specific heat (J/kg/K)</td>
<td>$a_2$</td>
</tr>
<tr>
<td>Thermal conductivity (W/m/K)</td>
<td></td>
</tr>
<tr>
<td>Liquid</td>
<td></td>
</tr>
<tr>
<td>Dynamic viscosity (Pa·s)</td>
<td>0.00132</td>
</tr>
<tr>
<td>Specific heat (J/kg/K)</td>
<td>13328.5</td>
</tr>
<tr>
<td>Thermal conductivity (W/m/K)</td>
<td>0.305</td>
</tr>
<tr>
<td>Vapor</td>
<td></td>
</tr>
<tr>
<td>Dynamic viscosity (Pa·s)</td>
<td>$3.846 \times 10^{-5}$</td>
</tr>
<tr>
<td>Specific heat (J/kg/K)</td>
<td>21497.38</td>
</tr>
<tr>
<td>Thermal conductivity (W/m/K)</td>
<td>0.1732</td>
</tr>
</tbody>
</table>

**Figure 3:** MFR versus the lift of the valve needle.
frequency of 20 Hz, it takes 0.05 s to calculate in the flow field, that is, the required number of flow field calculation steps is 10000.

\[
f_{\text{max}} = \frac{1}{2\Delta t} = \frac{1}{2 \times 5 \times 10^{-6}} = 100000 \text{Hz},
\]

\[
f_{\text{min}} = \frac{1}{N \cdot \Delta t} = \frac{1}{10000 \times 5 \times 10^{-6}} = 20 \text{Hz}.
\]

Here, \( f \) is the frequency of the acoustic field, and the subscripts max and min represent the maximum frequency and minimum frequency, respectively. \( \Delta t \) is the time step of the flow field and \( N \) represents the number of time steps.

Therefore, on the basis of section 2, 10000 steps are calculated and the wall pressure is sampled every 5 steps. After sampling, the wall pressure is imported into ACTRAN 17.0 for DFT, thus TWPF excitation is obtained.

3.3. Numerical Model and Boundary Conditions. TWPF is loaded into the acoustic meshes, and the boundary conditions and measuring point is set to get the complete numerical model of the acoustic field. The interface between meshes in each computation domain is set as a coupling surface. The outer surface of the air domain is set as an infinite element to simulate sound propagation in infinite space outside EEV. Non-reflective boundary conditions and fixed constraints are set at the inlet and outlet of the pipeline.

3.4. Numerical Computations. According to the numerical computations, the sound pressure level (SPL) at the measuring point is 34.3dBA while the experimental result is 30.6dBA. The SPL spectrum of simulation and experiment is shown in Figure 5. It can be seen that the simulated spectrum is generally consistent with the experimental spectrum, but the total difference is slightly larger (3.7dBA). In addition, the SPL is generally higher in the range of 0~12500 Hz. After 12500 Hz, the SPL is generally lower than 0.

4. Analysis of Flow Field Characteristics

Based on the TWPF excitation generated inside EEV, the flow field characteristics are analyzed. The contour of total pressure is shown in Figure 6. It can be seen that the section at the throttle port is the smallest, which causes a huge pressure loss and minimizes the pressure here. When the refrigerant flows into the valve throat, the section gradually expands and the pressure rises. However, due to the influence of a large pressure gradient in the throttling process, the distribution of the pressure at the valve throat presents complicated. When the refrigerant flows through the throat and into the pipe, the flashing phase transition happens, which generates a flash vapor-liquid flow behind the valve, making the pressure distribution uneven.
Although the flow field at the throttle port is complex, on the one hand, due to the refrigeration condition, there must be a large pressure loss at the port to realize the expansion of refrigerant, so as to achieve the refrigeration. On the other hand, the change of the throttle port is bound to have a great impact on the MFR. Therefore, in order not to affect the working performance of the EEV, the throttle port should not be improved. Then, the flow field characteristics at this point should not be studied in-depth, but the flow field characteristics behind the valve can be mainly studied to provide a basis for improvement.

To study the relationship between the pressure distribution behind the valve and the flashing two-phase flow, the contour of velocity and liquid phase fraction is taken for analysis, as shown in Figure 7. As can be seen, a jet is formed behind the valve, and the friction is generated with the low-speed refrigerant behind the valve at the boundary of the jet, thus making the velocity distribution uneven. As the flow continues, the velocity of the jet decreases gradually due to the viscosity, and the velocity distribution in the pipe becomes slightly uniform. Meanwhile, the refrigerant mainly begins the flashing behind the valve, because there is a large space, and the vapor generated by vaporization flows downstream continuously, which will not inhibit the vaporization of the refrigerant, so that the flashing can continue. Besides, the vaporization of the refrigerant is carried out around the liquid core of the jet, which eventually makes the liquid core disappear.

Section (1)–section (9) are made at the straight pipe and elbow of the outlet pipe, see Figure 8, used to study multiphase flow behind the valve by combining the contour and the structure of the valve.

Firstly, sections (1)–(3) are analyzed. It mainly consists of two parts, the evaporation of the refrigerant and the secondary flow in the pipe. The contours are shown in Figure 9. It can be found that the liquid phase fraction is lower in most of the annular region, while the highest in the center, and there is a transition between the center and the annular region, indicating that the vaporization starts from the surface of the liquid core. As the flow moves downstream, the liquid core gradually decreases and the transition expands. In the vaporization, the liquid core transfers its
kinetic energy to the vapor phase, so that the liquid core and the surrounding vapor are at a higher flow rate. Besides, the outer contour of the liquid core is not circular, indicating that the refrigerant is not symmetrical during evaporation, which results in inconsistent vapor phase fraction and uneven pressure distribution in the pipe. Under such conditions, secondary flow is easily generated in the pipe. Secondary flow refers to the flow in the pipe that is perpendicular to the main flow direction because of the transverse pressure difference. In the flashing, the refrigerant continuously produces vapor, which makes the pressure around the liquid core increase, while the central pressure decreases, thus forming a transverse pressure difference in the section. Therefore, the secondary flow is formed.

The above is the analysis in the straight pipe. It is also complicated in the bent pipe, analyzed below. The flow in the bend also includes two aspects: the evaporation of the refrigerant and the secondary flow. As shown in Figure 10, from sections (4) to (5), the liquid core begins to contact the wall of the bent pipe. In section (4), there is still a clear liquid core. In section (5), although the liquid core exists, it begins to deform due to the obstruction of the bent pipe and begins to flow and diffuse to both sides. As the flow continues, on the one hand, because the refrigerant still vaporizes, the liquid phase continues to decrease. On the other hand, the liquid phase is constantly deformed due to the obstruction of the bent pipe, and the liquid core is finally broken and disappeared. In section (9), the liquid phase distribution on the section becomes more uniform. The velocity core always coincides with the liquid core, indicating that the liquid phase will impact the bent pipe.

The secondary flow of sections (4)–(9) is shown in Figure 11. From sections (4) to (6), the secondary flow is becoming more complex, because, in this pipe, the mainstream direction of refrigerant is changing, coupled with the impact of liquid core crushing and liquid phase vaporization, so that the pressure distribution is very uneven. The secondary flow of sections (7) and (8) is relatively weak because the change of flow direction of refrigerant has been basically completed, and the vaporization of the refrigerant is also relatively sufficient in these two sections. While at section (9), the refrigerant enters the straight pipe. The liquid phase of the refrigerant causes stratification here, that is, there is more liquid in the middle and less around. As a
result, secondary flow occurs again in this section due to the higher pressure on the outside.

Based on the above analysis, it can be concluded that there are two main sources of noise under the refrigeration condition. Firstly, the high-speed jet behind the valve cannot be fully vaporized before contacting the bent pipe, so the liquid core will impact the bent pipe and cause the complicated pressure distribution, thus causing the valve vibration and producing noise. Secondly, due to the vaporization of the liquid core, the section of the pipe generates transverse pressure difference, and secondary flow is generated, which further aggravates the non-uniformity of pressure distribution in the pipe and causes the valve vibration and noise generation.

5. Improvement Design

A noise reduction structure of separating flow is designed. A conical rotator is added at the valve throat. The principle of this structure is to separate the original single streamflow in the throat into three streams, so that when the refrigerant flows out of the valve throat, the original single strong jet will be divided into three weak jets, and there is a certain distance between each jet, so that the kinetic energy of the weak jet will be reduced quickly, and the velocity behind the valve will be more uniform. Besides, because the liquid core of a weak jet is small, it will be evaporated quickly in a short distance, thus weakening the impact on the bent pipe.

The CFD and CAA simulation of the structure is carried out and the results are shown in Table 3. It can be seen that the structure can reduce the SPL by 2.3dBA under the premise of only a 4.57% impact on the MFR. Meanwhile, the flow field before and after the improvement is compared, as shown in Figure 12. It can be seen that the velocity of the refrigerant entering the valve is greatly reduced, the jet is not obvious, and the velocity distribution behind the valve is more uniform. Compared with the contour of the liquid phase fraction, it can be found that the liquid core of the refrigerant is not obvious and the liquid distribution is dispersed, which can reduce the impact on the bent pipe.

6. Conclusions

In this paper, the flow field characteristics and acoustic field characteristics of EEV under refrigeration conditions are studied by numerical simulation. Based on the method, a noise reduction structure of separating flow is designed to optimize the flow field and the acoustic field of EEV. The conclusions are summarized:

(i) The flashing two-phase flow model is established. The mass flow rates (MFR) of EEV corresponding to the lift of the valve needle 0.6, 1.25, and 1.875 mm are compared to the experimental results. It is shown that the simulation model in this paper is reliable and the MFR of EEV could be predicted accurately.

(ii) It is found that the simulation of the noise spectrum is in good agreement with the experiment, indicating that the numerical model of the acoustic field is also reliable. Then, according to the analysis of the calculation results, it is found that the turbulent excitation is caused by the flashing multiphase flow behind the valve and the secondary flow on the section after the refrigerant throttling.

(iii) The noise reduction structure of separating flow is designed. The original single strong jet is divided into multiple weak jets, and the SPL is reduced by 2.3 dBA on the premise that the MFR is only 4.57%.

Data Availability

The data used to support the findings of this study are available from the corresponding author upon request.

Conflicts of Interest

The author(s) declare no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.
References


