ISSN: 2414-1895 DOI: 10.6919/ICJE.202107_7(7).0002

Flow Analysis of Pressure Relief Valve based on Fluent Numerical Simulation

Kang Wang

School of Mechanical Engineering, Shanghai Maritime University, Shanghai 201306, China.

18621151229@163.com

Abstract

The pressure regulator relief valve is used to control the circulation of saturated water vapor at high temperature and high pressure. Good adjustment performance and accurate flow calculation are of great significance to the project. Use Fluent Meshing to mesh the three-dimensional model of the pressure regulator relief valve, and use Fluent to simulate the valve. Through theoretical calculations, the saturated water vapor Mach number inside the valve body is greater than 0.3, so saturated water vapor is regarded as an ideal gas. The Realizable k-e turbulence model is used to verify the results of the Fluent simulation with the valve flow coefficient measured by the experiment. The results show that the error between the two results is 5%, which verifies the accuracy of the Fluent numerical simulation. Further, the simulation experiment is carried out under the given actual working pressure and temperature, and the flow rate under this condition is obtained. At the same time, comparative analysis to obtain the changes of flow under different opening degrees, temperatures and pressures, to find the maximum flow under the best working conditions, which provides a theoretical basis for the selection and design of subsequent valves.

Keywords

Fluent; Valve; Traffic Analysis; Simulation.

1. Introduction

The regulator pressure relief valve mainly maintains working requirements by controlling the flow and pressure of its fluid, and the flow rate of the regulator pressure relief valve is different under different working environments. Under the same opening degree, the flow rate will be different for different pressure difference and temperature. Through Fluent numerical simulation [2], it is possible to analyze the changes of various parameters such as the flow rate and pressure of the pressure regulator relief valve, and to detect the stability and quality of the pressure relief valve of the regulator. HE Hui-min [1] and others studied the unsteady steam flow fields of the control valve fast switched from fully open to close position were calculated and analyzed using Fluent dynamic mesh, during which dynamic curves of the valve lift, mass flow rate and downstream pressure varying with closing time presented. Based on fluid mechanics CFD software, JIN Shu-Jun [3] conducted a three-dimensional simulation study on the internal flow performance of the three-way regulating valve in the key device of the cooling system, and obtained flow field information such as the internal pressure and flow velocity of the valve. Based on the Fluent flow field simulation software, Zhang Jing [5] conducted numerical simulation and visualization research on the internal flow field of a spool valve. Under the same calculation conditions, the steady-state simulation of the three-dimensional models under different valve openings was carried out to obtain the velocity pressure, flow characteristics and the change law of the flow coefficient of the internal flow field of the spool valve. In the project,

ISSN: 2414-1895
DOI: 10.6919/ICJE.202107_7(7).0002

due to continuous changes in the environment [4], the performance of the regulator pressure relief valve will dynamically change. Through the establishment of a mathematical model of the regulator pressure relief valve, the flow coefficient is mathematically solved [6], and finally verified by experiments and simulations.

2. Basic Theory of Fluid Mechanics

2.1 Fluid Dynamics Continuity Equation

The flow of fluid inside the pressure relief valve obeys the most basic laws of physics, including the law of conservation of mass, the law of conservation of momentum and the law of conservation of energy. However, it is generally difficult to obtain a mathematical analytical solution for the actual model. With the development of computers, the numerical solution of the model has gradually become the main method of solving the model. According to the difference of the discrete principle, it is roughly divided into the finite difference method, the finite element method and the finite volume method. The finite volume method has become the current mainstream solution because of its clear physical meaning of the coefficients of the discrete equation and small amount of calculation.

According to the increase in the mass of the micro-element within a unit volume at the same time, it is equal to the net mass of the inflow, there is:

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \tag{1}$$

among them ρ is density,

t is time,u, v and w is velocity vector in x, y and z component of direction.

At the same time, any flow system must also satisfy the conservation of momentum, there is:

$$\frac{\partial(\rho u)}{\partial t}+div(\rho u\vec{u})=div(\mu gradu)-\frac{\partial p}{\partial x}+S_u \tag{2}$$

At the same time, any flow system must also satisfy the conservation of momentum, there is:

$$\frac{\partial(\rho u)}{\partial t}+div(\rho u\vec{u})=div(\mu gradu)-\frac{\partial p}{\partial x}+S_u \tag{3}$$

$$\frac{\partial(\rho v)}{\partial t}+div(\rho v\vec{u})=div(\mu gradv)-\frac{\partial p}{\partial y}+S_v \tag{4}$$

$$\frac{\partial(\rho w)}{\partial t}+div(\rho w\vec{u})=div(\mu gradw)-\frac{\partial p}{\partial z}+S_w \tag{5}$$

Among them S_u,S_v,S_z is Generalized Source Term of Momentum:

$$S_u=F_x+S_x \tag{6}$$

$$S_u=F_y+S_y \tag{7}$$

$$S_w=F_z+S_z \tag{8}$$

S_x,S_y and S_z of expression is as follows:

$$S_x=\frac{\partial}{\partial x}\left(\mu\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu\frac{\partial v}{\partial x}\right)+\frac{\partial}{\partial z}\left(\mu\frac{\partial w}{\partial x}\right)+\frac{\partial}{\partial x}(\lambda di\vec{u}) \tag{9}$$

$$S_y=\frac{\partial}{\partial x}\left(\mu\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial y}\left(\mu\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu\frac{\partial w}{\partial y}\right)+\frac{\partial}{\partial y}(\lambda di\vec{u}) \tag{10}$$

$$S_z=\frac{\partial}{\partial x}\left(\mu\frac{\partial u}{\partial z}\right)+\frac{\partial}{\partial y}\left(\mu\frac{\partial v}{\partial z}\right)+\frac{\partial}{\partial z}\left(\mu\frac{\partial w}{\partial z}\right)+\frac{\partial}{\partial z}(\lambda di\vec{u}) \tag{11}$$

The above formula has also become an $N-S$ equation.

Because the studied fluid is high-speed, high-temperature and high-pressure saturated water vapor, there will be non-negligible heat exchange during the flow process. According to the first law of thermodynamics, the rate of energy increase in the micro-element body is equal to the net heat flow into the micro-element body plus the work done on the micro-element body by physical and surface forces:

ISSN: 2414-1895 DOI: 10.6919/ICJE.202107_7(7).0002

$$\frac{\partial(\rho T)}{\partial t} + div(\rho\vec{u}T) = div\left(\frac{k}{C_p} gradT\right) + S_T \tag{12}$$

In this engineering model, the fluid dynamics continuity equation, momentum conservation equation and energy conservation equation can be used to theoretically solve the flow and change of fluid in the valve body. However, due to the complexity of the model and the huge difficulty of mathematics, the Fluent numerical simulation software is used to solve the problem.

3. Parameter Setting and Simulation

3.1 Meshing

The quality of the mesh has an important influence on the accuracy and convergence of the model solution. Fluent Meshing is used for meshing, and a mixed method of polyhedron and regular hexahedron is used to divide the mesh. The number of volumes meshes is 584912, the number of nodes is 1671045, and the mesh quality is above 0.47, as shown in Figure 1.

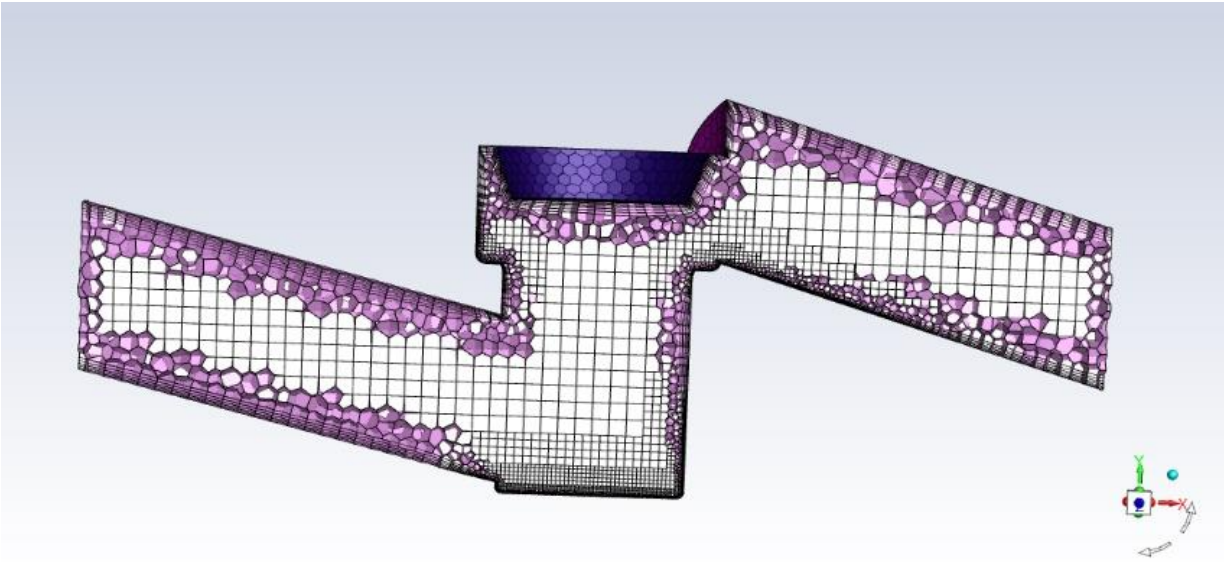

Fig. 1 Section view after meshing

3.2 Choose Turbulence Model

The flow rate of saturated water vapor in the valve body has a high Reynolds number, and the flow can be compressed. Standard k-e will produce a certain amount of distortion in strong swirling, curved wall flow or curved streamline flow. The Realizable k-e model introduces content related to rotation and curvature in the turbulent viscosity calculation formula. The production term in the e equation no longer includes the production term G_k in the K equation, and now it can better represent the energy conversion of the spectrum. The penultimate term in the e equation does not have any singularity. Even if the K value is small or zero, the denominator will not be zero, which greatly improves the standard k-e model. Using wall function method and Realizable k-e model combination, it has good calculation ability for high Reynolds number and high temperature saturated water vapor fluid, and it can also handle the fluid with insufficient development of near-wall turbulence.

3.3 Set Up Materials

Pre-experiment with the model found that due to the large pressure difference between the two ends, the flow velocity has exceeded the local speed of sound in the actual flow process, and the average Mach number has been greater than 0.3. Therefore, saturated water vapor can be considered as a compressible fluid in actual flow. Set the material property to fluid, select the material as saturated water vapor, and set the density of saturated water vapor as ideal gas.

ISSN: 2414-1895 DOI: 10.6919/ICJE.202107_7(7).0002

3.4 Set Boundary Conditions

The setting of boundary conditions directly affects the accuracy of the experimental results. In the flow field inside the pressure relief valve, the density of high-temperature and high-pressure saturated water vapor in the flow field is uncertain, and the average flow velocity of the fluid is already close to the local speed of sound. It is reasonable and correct to treat it as a compressible fluid. In the experiment, the pressure at the inlet and the inlet of the pressure relief valve were measured to be 17.23Mpa and 0.12Mpa, respectively. The inlet is set as a pressure inlet in Fluent, and the temperature is 353 degrees Celsius.

4. Analysis of Numerical Calculation Results

4.1 Flow Coefficient

The flow coefficient is the main technical index of the pressure relief valve. The value of the flow coefficient indicates the flow capacity of the pressure relief valve. The expression of the international standard flow coefficient is:

$$K_e = Q\sqrt{\frac{1}{\Delta p} \times \frac{\rho}{\rho_0}} \tag{13}$$

The experimental measurement method of K_v is that when the pressure difference between the two ends of the valve is 100KPa, the valve body flows through a volume with a density of 1000kg/m3 in one hour, and its unit is m^3/h.

4.2 Simulation Result Analysis

As can be seen from figure 2. At the right inlet of the valve, the pressure is relatively stable and even.

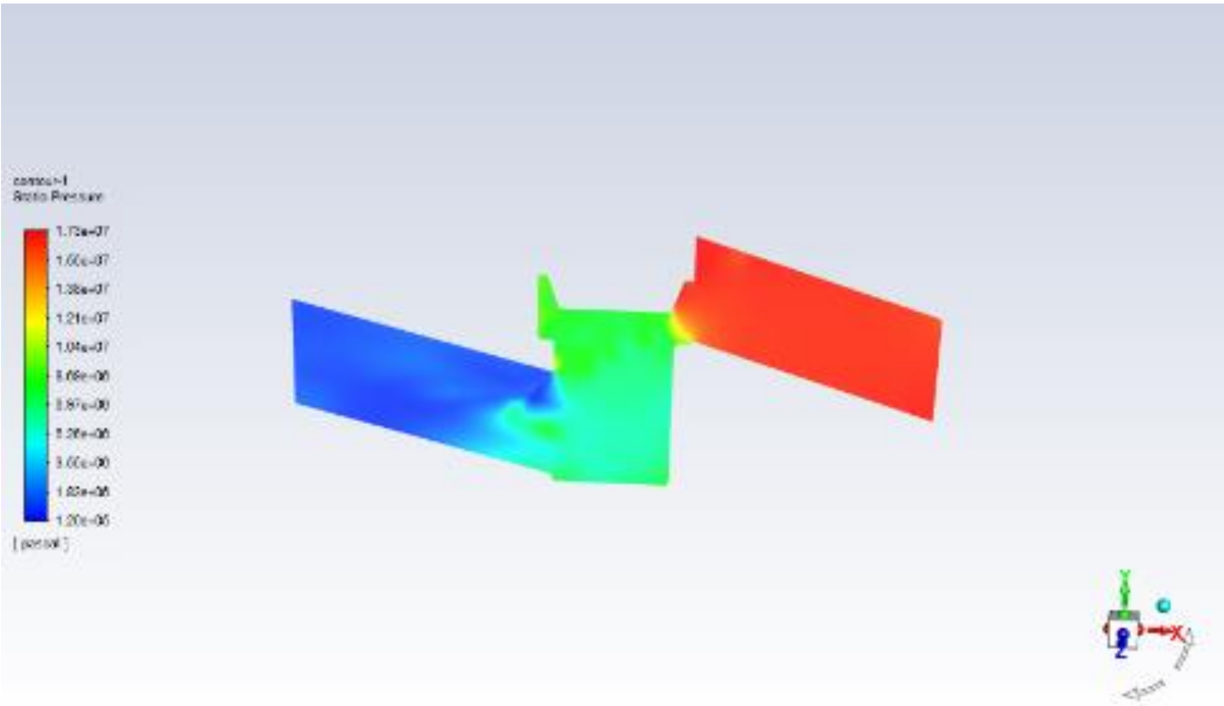

Fig. 2 Static pressure distribution graph

When the fluid flows through the valve inlet, the pressure drops rapidly. When the fluid passes through the valve outlet, the pressure drops again. It can be seen that after the fluid passes through the pressure relief valve, the pressure of the fluid is significantly reduced, indicating that the pressure relief valve can achieve the ideal pressure reduction effect.

ISSN: 2414-1895
DOI: 10.6919/ICJE.202107_7(7).0002

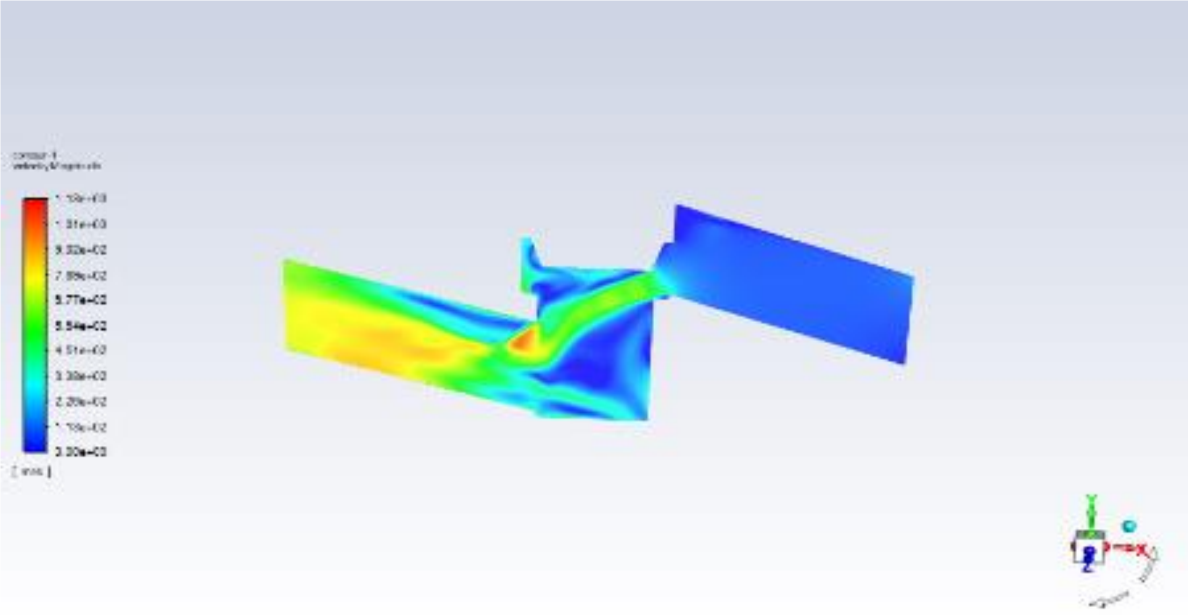

Fig. 3 Velocity distribution cloud diagram of pressure relief valve

Figure 3 shows the velocity distribution cloud diagram of the pressure relief valve, it can be seen that when saturated water vapor flows through the valve port, due to the low inlet velocity, according to the steady flow Euler equation of motion:

$$\rho V\frac{dV}{dx}=-\frac{dp}{dx} \tag{14}$$

Due to:

$$Ma=V/c \tag{15}$$

Get:

$$\frac{dA}{A}=(Ma^2-1)\frac{dV}{V} \tag{16}$$

dA and dV have different signs, and dp are the same signs, it can be obtained that when the flow cross section of the subsonic flow becomes smaller, the fluid accelerates and the pressure decreases. The average flow velocity in the valve body is 317m/s, and the highest flow velocity reaches 1062m/s.

4.3 The Change of Outlet Flow Under Different Opening Degrees

It can be seen from figure 4 below that when the opening degree and the outlet flow rate are in a linear relationship, when the opening degree increases, the outlet flow rate of the compressible saturated water vapor is proportional to the opening degree. Due to the complicated internal structure of the valve, the outlet flow rate increases steadily and linearly with the increase of the opening, which indicates that the working conditions of the valve are close at different openings and the working conditions are stable.

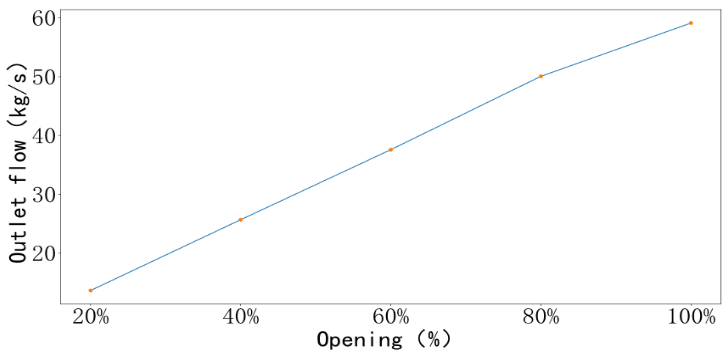

Fig. 4 Velocity distribution cloud diagram of pressure relief valve

ISSN: 2414-1895 DOI: 10.6919/ICJE.202107_7(7).0002

As shown in Figure 5, under the same conditions of inlet pressure, temperature and pressure, as the opening degree increases, the pressure drop decreases. It can be obtained that when the valve is working stably, as the opening degree increases, the pressure drop at both ends of the inlet and outlet shows a downward trend, and its change is stable, and it has a linear relationship with the opening degree.

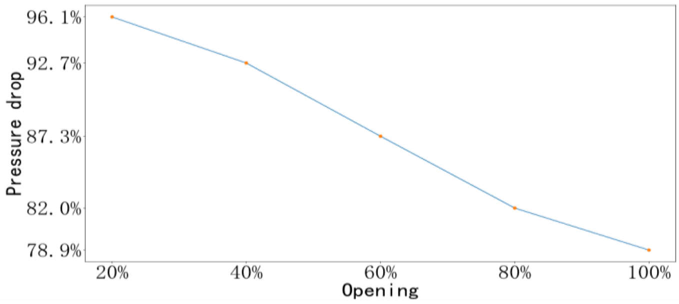

Fig. 5 Velocity distribution cloud diagram of pressure relief valve

4.4 Pressure Characteristic Curve of Different Opening Under Different Inlet Pressure

Figure 6 is a comparison diagram of inlet and outlet pressure drops at different openings under different inlet pressures. It can be seen from the figure that as the opening of the relief valve increases, the pressure drop at both ends of the relief valve decreases linearly. After changing the opening of the relief valve, the pressure drop at both ends of the relief valve does not change nonlinearly, which shows that the pressure relief capacity of the pressure relief valve is stable. When the opening is around 20%, 60%, and 100%, the difference in pressure drop between the inlet and outlet is significantly reduced under different inlet pressures. When the opening of the pressure relief valve is around 40% and 80%, the difference in pressure drop between inlet and outlet increases significantly under different inlet and outlet pressures. It can be obtained that when the pressure relief valve is at 20%, 60%, and 100% opening, the sensitivity of the inlet and outlet pressure drop to the inlet pressure is reduced. Therefore, it is a better solution to choose 20%, 60%, and 100% when the import and export pressure needs to be changed frequently.

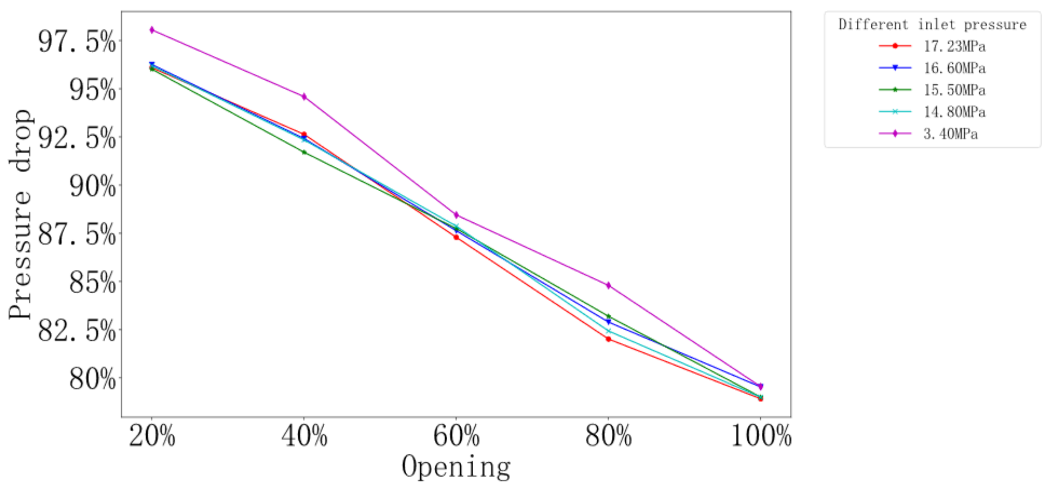

Fig. 6 Comparison chart of inlet and outlet pressure drop of different inlet and outlet pressure relief valves

4.5 Flow Characteristic Curve of Different Opening Under Different Inlet Pressure

The fluid medium is high temperature and high-pressure saturated water vapor. Due to its compressibility, different inlet pressures will have a certain impact on the outlet flow of the pressure

ISSN: 2414-1895 DOI: 10.6919/ICJE.202107_7(7).0002

relief valve. Therefore, different inlet pressures are used to explore the influence of inlet pressure on the outlet flow.

Figure 7 is a comparison diagram of the outlet flow at different openings under different inlet pressures. When the opening of the pressure relief valve remains unchanged, the greater the inlet pressure, the greater the outlet flow. And as the opening degree continues to increase, the difference between different inlet pressures is greater. It can be obtained from the figure that the smaller the inlet pressure is, the smaller the slope of the outlet flow rate of the pressure relief valve with different openings is. When the inlet pressure is 17.23MPa, the difference in the outlet flow between the maximum and minimum opening is 78.46kg/s. When the inlet pressure is 3.40MPa, the difference in the outlet flow between the maximum and minimum opening is 9.805kg/s. The adjustable outlet flow of the former is 8 times that of the latter. In practical engineering applications, when the inlet pressure is higher and a larger outlet flow adjustment range is required at the same time, the pressure relief valve can be selected to have a larger outlet flow adjustment range.

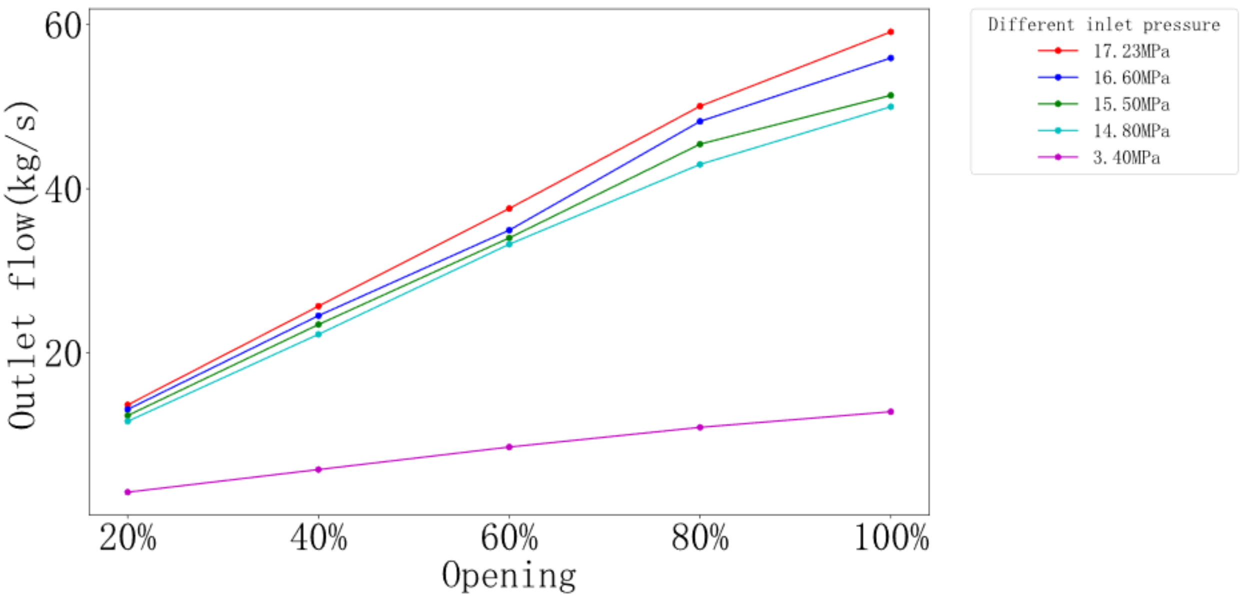

Fig. 7 Comparison chart of pressure relief valve flow under different inlet and outlet pressures

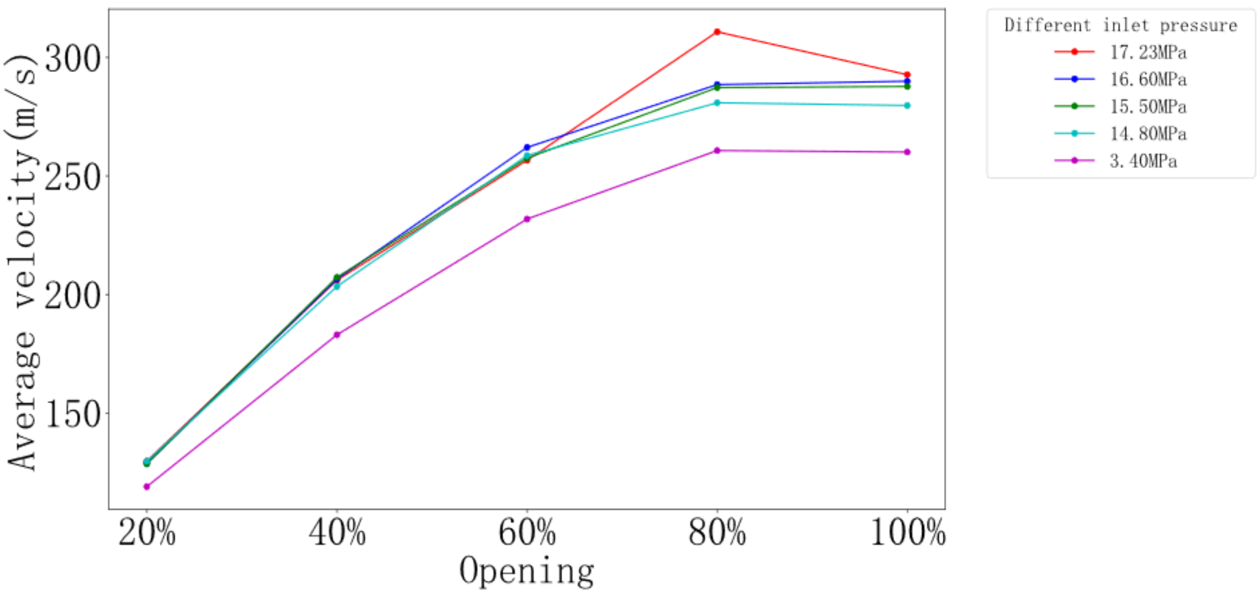

Fig. 8 Average flow rate diagram of different inlet and outlet pressure relief valves

ISSN: 2414-1895 DOI: 10.6919/ICJE.202107_7(7).0002

4.6 Flow Velocity Characteristic Curve of Different Opening Under Different Inlet Pressure

The analysis of the flow velocity distribution inside the valve can provide a certain theoretical basis for the structure and selection of the valve. When the flow velocity of the fluid in the valve is too high, the valve diameter will be relatively small, the resistance loss will be large, and the valve will be easily damaged. If the flow rate is small, the efficiency is reduced and the economy is not good. Figure 8 is a comparison chart of the average flow velocity of relief valves with different openings under the same inlet pressure. It can be found that when the opening of the pressure relief valve is less than 60%, the average internal flow velocity of the pressure relief valve continues to increase; when the opening of the pressure relief valve is greater than 60%, the increase speed of the internal average flow velocity begins to slow down; When the opening of the pressure relief valve is greater than 80%, the average flow velocity inside the pressure relief valve basically no longer increases and tends to be stable.

5. Summary

When the pressure relief valve is opened at 60%, the actual working condition measured the outlet flow to be about 145t/h, and under the same working condition setting, the fluent simulation result is 135.32t/h, the error is about 6.7%.

In the case of constant inlet pressure, as the opening of the pressure relief valve increases, the pressure drop at both ends of the inlet and outlet shows a downward trend, and its change is stable, which is in a linear relationship with the opening. When the inlet pressure changes, the greater the inlet pressure, the greater the slope of the outlet flow rate of the pressure relief valve with different opening degrees. The pressure relief valve can have a larger outlet flow adjustment range.

When the pressure relief valve is opened at 20%, 60% and 100%, the sensitivity of the inlet and outlet pressure drop to the inlet pressure is reduced. Therefore, under the frequently changing inlet and outlet pressure conditions, choose 20%, 60% and 100 % is a better solution. When the opening of the pressure relief valve is in the range of 20%-60%, the average flow rate inside the pressure relief valve increases continuously. When the opening of the pressure relief valve is greater than 80%, the average flow velocity inside the pressure relief valve basically no longer increases and tends to be stable.

References

[1] HE Hui-min, ZHANG Xiao-Yi. Numerical Simulation of Two-phase Flow Characteristics of Control Valve in AP1000 Nuclear Power Steam. Turbine Technology. Vol. 59 (2017) No. 01, p. 43-46.

[2] HE Qing-zhong, LIU Jia, DONG Xue-lian, et al. Flow field in regulating valve of water storage tank for ultra-supercritical unit boiler: characteristics and optimization. Thermal Power Generation. Vol. 45 (2016) No. 10, p. 82-88.

[3] JIN Shu-jun, LI Zhong, ZHANG Mao-sen. Research on Internal Flow Performance of Three-way Control Valve Based on CFD Technology. Valve. (2019) No. 04, p. 11-14.

[4] WEI Lang, HU Wenchao, GOU Ruirui. Numerical Simulation and Analysis of Ball Valve with Different Valve Spool Structure Based on Fluent. Machine Tool & Hydraulics. Vol. 47 (2019) No. 20, p. 148-151+186.

[5] ZHANG Jing, ZHANG Jing, ZHANG Jing. The Simulation and Analysis of a Spool Valve Based on Fluent. Machine Tool & Hydraulics. (2015) No. 04, p. 56-59.

[6] ZHANG Xiheng, WANG Ting, JIAO Yuanyang, et al. Simulation and Analysis of Flow Field Based on FLUENT Software in All-welded Ball Valve Under Different Media. Journal of Xihua University (Natural Science Edition). Vol. 36 (2017) No. 02, p. 6-9+86.