Numerical Simulation on Hydraulic Characteristics of Spray Regulator Valve

ZHOU Long-yu¹, Hao Si-ja², YANY Zhi-da²*, Hou Ying-zhe¹

¹Wuhan Second Ship Design and Research Institute, Wuhan, Hubei, 430064, China;
²College of Nuclear Science and Technology, Harbin Engineering University, Harbin, Heilongjiang, 150001, China
*Corresponding author’s e-mail: yangzhida@hrbeu.edu.cn

Abstract: The hydraulic properties of the internal fluid of the spray regulator valve are studied by numerical simulation. The velocity field, pressure field, flow coefficient and adjustable ratio of spray regulator valve at different opening are analyzed. The results show that the main factors affecting the flow field distribution and the maximum flow velocity inside the valve body are the valve opening and the inlet and outlet pressure difference. There is a high speed area in the narrow channel area around the valve spool, which will decrease as the valve opening decreases. The pressure in the fluid domain changes rapidly from the inlet pressure to the outlet pressure at the narrow channel around the valve core. The adjustable ratio of the spray regulator valve is 72.9.

1. Introduction
The spray regulating valve is one of the pressure control equipment for the reactor coolant system of a PWR nuclear power plant [1-2], and its function is to make the pressure in the pressurizer under control within a certain range [3]. At present, the pressurizer spray regulating valves of PWR nuclear power plants in service and under construction in China are imported from abroad. The pressurizer spray regulating valve of Qinshan Phase I Nuclear Power Plant adopts a sleeve type control valve structure. The "SUPERO" pneumatic diaphragm type regulating valve, manufactured by Okano, Japan, throttles the valve seat through an equal percentage sleeve profile structure. For the Daya Bay Nuclear Power Plant and the AP1000 third-generation PWR nuclear power plant under construction, the pressurizer spray regulating valve is the V-shaped ball valve manufactured by American Fisher company [4]. Because of the long-term blockade by foreign technology, the domestic research on the pressurizer spray regulating valve started late. Therefore, the pressurizer spray regulating valve has become one of the key valve technologies for nuclear power plants in China.

The spray regulating valve studied in this article uses electromagnetic force to drive the spool to adjust the flow or pressure, control the opening of the valve by controlling its input voltage, so as to adjust the flow rate. It has fast action, no leakage, small size, easy maintenance and many other advantages.

The hydraulic characteristics of the internal fluid are analyzed with CFD technology [5-8], which provides the necessary basis for the development of model machine.
2. Design Conditions and Numerical Solutions

2.1 Geometric Model and Design Conditions of Spray Regulating Valve
The electromagnetic proportional spray regulating valve model calculated in this paper is shown in Figure 1a. The valve body is shown in Figure 1b. Because the valve flow domain is symmetrical, one-half of the model is selected for calculation is enough. The computational domain is shown in Figure 2.

For the actual operating environment of the valve, the boundary conditions are:
- Design conditions: the medium is water, the inlet pressure is 15.85MPa, the inlet temperature is 262.3 °C, and the outlet pressure is 15.70MPa.
- Comparative conditions: the medium is water, the inlet pressure is 15.85MPa, the inlet temperature is 262.3 °C, and the outlet pressure is 15.75MPa.

2.2 Establishment of Coordinate System
The center of the circle of the valve body's circular water outlet is selected as the origin of the coordinate system in this paper. The establishment of the coordinate system is shown in Figure 3.
The model valve body size calculated in this paper is shown in Table 1.

| coordinate Axis | minimum position (mm) | maximum position (mm) |
|-----------------|------------------------|------------------------|
| X               | 0                      | 60                     |
| Y               | -60                    | 190                    |
| Z               | 0                      | 230                    |

2.3 Analysis Cross Section Selection
The flow fields and pressure fields in the three directions of x, y, and z are analyzed and compared in this article. For different operating conditions and different opening, x = 0, y = 0, z = 50mm, z = 100mm, and z = 130mm, z = 135mm, z = 140mm, z = 150mm these eight sections are selected for analysis. Figures 4 (a), (b), and (c) show the analysis cross sections in the three directions of x, y, and z, respectively.

![Analysis cross section](image)

3. Meshing and Solving
An unstructured grid is used in this paper. The grid generation is shown in Figure 5. The width of the runner under different valve opening are different, especially when the valve opening is very small, the narrowest runner width is only a few millimeters. Therefore, this article chose to use local encryption to encrypt the grid. This condition ensures that the calculation can be performed smoothly and improves the accuracy of the calculation.
Since the valve flow domain model under different opening is only different near the spool, in order to improve the calculation efficiency, for the flow domain model inside the valve with different opening, the wide flow domain is set with the same grid size, and different encryption sizes are set in the areas to be encrypted. The grid sensitivity analysis was performed for the valve at different opening. In order to verify the calculation independent solution at the position where the speed change rate is large, 8 monitoring points in total were selected on the two cross sections shown in Figure 6. Extract the point with the largest error among the 8 points. The grid sensitivity analysis at different opening is shown in Figure 7. After performing the sensitivity analysis, the number of grids selected at different opening is shown in Table 2.
Figure 7. Grid sensitivity analysis

The k-epsilon model is used in this paper for large opening operating conditions [9-10], and the RSM transient model is used for small openings. All walls are assumed to have no slip conditions. The discrete equations are solved using the SIMPLE algorithm with pressure-velocity coupling correction [11-12]. Residual errors of momentum, turbulent flow energy, and turbulent dissipation rate are converged to the order of $10^{-4}$, energy residual errors are converged to the order of $10^{-7}$, and the quality difference between the inlet and outlet is in the order of $10^{-4}$.

Table 2. The number of grids selected at different opening

| opening degree (%) | the number of grids (1×10^4) |
|--------------------|-----------------------------|
| 1                  | 775                         |
| 2                  | 450                         |
| 3                  | 370                         |
| 4                  | 350                         |
| 5                  | 340                         |
| 10                 | 300                         |
| 20                 | 275                         |
| 30                 | 155                         |
| 40                 | 135                         |
| 50                 | 95                          |
| 60                 | 85                          |
| 70                 | 75                          |
| 80                 | 70                          |
| 90                 | 50                          |
| 100                | 40                          |

4. Calculation Results and Analysis

4.1 Analysis of Internal Flow Under Spool Relative Stroke
Under the conditions of inlet pressure of 15.85 MPa, inlet temperature of 262.3 °C, and outlet pressure of 15.70 MPa, the flow conditions are analyzed with opening of 10% (Figure 8a, Figure 9a), 40% (Figure 8b, Figure 9b), and 70% (Figure 8c, Figure 9c), 100% (Figure 8d, Figure 9d).
From the $x = 0$ cross section fluid velocity cloud diagram (see Figure 8), it can be known that under all operating conditions, the maximum velocity in the flow field appears at the narrowest flow channel around the spool. And under the condition of constant inlet and outlet flow rate and temperature, when the valve opening is large, the maximum speed change in the flow field is small, and when the valve opening is below 30%, the maximum speed in the flow field suddenly drops. In Figure 8, a high-speed area appears at the spool under different operating conditions. The range of this area decreases with the decrease of the valve opening, and the distribution of this high-speed area has a certain symmetry. Even the upper part of the body has the influence of the water inlet, it has little effect on the symmetry of the high-speed area distribution.

Figure 8. Velocity cloud diagram $x = 0$ cross section under different opening
From the $y = 0$ cross section fluid velocity cloud diagram (see Figure 9), it can be known that because the fluid flows from the valve inlet through the narrow area of the flow channel around the spool and then enters the valve outlet, the flow velocity at the front end of the spool is relatively low, and the fluid that runs along the export channel from the spool periphery has gradually developed. At the same time, the phenomenon occurs at the valve outlet that the fluid converges toward the center of the outlet.

Figure 10 shows under the design condition, the velocity cloud diagram of the cross section under the design condition and 100% valve stroke, $z = 50\text{mm}$ (Figure 10a), $z = 100\text{mm}$ (Figure 10b), $z = 130\text{mm}$ (Figure 10c), $z = 135\text{mm}$ (Figure 10d), $z = 140\text{mm}$ (Figure 10e), $z = 150\text{mm}$ (Figure 10f).
Comparing Figure 10 (a) and Figure 10 (b), it can be seen that with the gradual development of the fluid in the valve outlet channel, the effect of gravity on the fluid is gradually obvious, and the fluid gradually gathers below the valve outlet channel. Comparing Figure 10 (c), Figure 10 (d) and Figure 10 (e), it can be seen that as the fluid flow channel gradually narrows, the range of the high flow rate area will gradually increase. Due to the influence of the boundary layer effect, the velocity of the fluid near the wall surface is low, and the velocity of the fluid is high at the center of the flow channel. And the effect of the boundary layer effect of the valve body shell is greater than the effect of the boundary layer effect of the spool component, so the area where the highest flow rate appears is offset from the center and is close to the spool component. It can be seen from Figure 10 (f) that in the tapered area of
the flow channel, the larger flow velocity is concentrated near the neck, and the remaining position will cause the flow direction to change and the flow velocity to slow down due to the large resistance.

4.2 Flow Characteristic Curve and Adjustable Ratio Analysis
The adjustable ratio of the regulating valve is the ratio between the maximum flow and the minimum flow that the regulating valve can control [13-14]. Adjustable ratio is also called adjustable range. If it is expressed by R, then

\[ R = \frac{Q_{\text{max}}}{Q_{\text{min}}} \]

Adjustable ratio can be divided into ideal adjustable ratio and actual adjustable ratio. When the pressure difference on the regulating valve is constant, it is called the ideal adjustable ratio. The ideal adjustable ratio is equal to the ratio of the maximum flow coefficient to the minimum flow coefficient. It reflects the adjustment capacity of the regulating valve and is determined by the structural design. Generally, it is always that the larger adjustable ratio, the better, but the flow coefficient will not be small due to the structural design and processing restrictions of the spool.

The regulating valve is either in series with the piping system or in parallel with the bypass valve in actual work. As the resistance of the piping system changes or the opening degree of the bypass valve is different, the adjustable ratio of the regulating valve also changes accordingly. The adjustable ratio is called the actual adjustable ratio. This article only analyzes the ideal adjustable ratio of the valve. Draw the valve flow curve according to the design conditions, and compare this curve with the valve flow characteristic curve. The valve flow rates under different spool opening are shown in Table 3.

| Spool opening degree (%) | Flow rates (kg/s) |
|-------------------------|------------------|
| 100                     | 5.830            |
| 90                      | 4.950            |
| 80                      | 4.530            |
| 70                      | 4.220            |
| 60                      | 3.680            |
| 50                      | 2.790            |
| 40                      | 2.370            |
| 30                      | 1.710            |
| 20                      | 1.060            |
| 10                      | 0.400            |
| 5                       | 0.129             |
| 4                       | 0.080             |
| 3                       | 0.018             |
| 2                       | 0.008             |
| 1                       | 0.007             |

Figure 11 is a line chart of the valve flow rates at different opening. The flow rate characteristic of the valve under the design condition is between the quick opening characteristic and the linear characteristic, which is close to the linear characteristic. When the valve opening is small, the flow is small, but the relative change is large and the sensitivity is high. A small change in stroke will cause a large change in flow, so it is easy to oscillate when the valve opening is small. At a large opening degree, the flow is large, but the relative change of the flow is small, and the sensitivity is low. The flow only changes when the stroke has a large change. Therefore, when at large opening degree, the control is sluggish and the adjustment is not timely and easy to overshoot and the transition becomes slower. It can be seen from the figure that when the valve opening is between 1% and 3%, the flow rate does not change much with the changing of the valve opening. When the flow rate is greater than 4%, the effect of the valve opening change on the flow rate is increased. Therefore, the flow rate at the valve opening of 4% is considered to be the lowest limit of the adjustable flow rate.
Based on the results of Table 3, Table 4, and Figure 11, the flow rate is 5.83 kg/s when Qmax is 100%, and the flow rate is 0.08 kg/s when Qmax is 1%. At this time, R = 72.9.

4.3 Flow of the Maximum Opening of the Spool

Compare the flow situation in the design condition with the cross-section flow situation with the same opening (100% opening degree) and x = 0 when the inlet and outlet pressure difference is 100kPa (comparative condition).

It can be seen from the comparison that the flow field distributions are very similar when the spool openings are the same, even if the pressure difference between the inlet and outlet is different under the two operating conditions. The difference is mainly reflected in the different flow velocity and flow. When the pressure difference between the inlet and outlet is large, a large flow velocity will be generated in the flow field. From the velocity vector diagram, it can be seen that as there is a flow channel sudden expansion area in the front of the spool, a vortex structure will appear around this area, and the rest of the flow field part basically has no vortex.
4.4 Minimum Adjustable Opening Fluid Flow of the Valve Core

Compare the flow situation at the design condition with the flow situation at the spool with the same opening degree (4% opening degree) and x=0 cross section when the inlet and outlet pressure difference is 100kPa. Because the flow is small at this opening degree, no comparison is made regards the velocity field. As shown in Figure 13. Figure 13 (a) shows the pressure distribution under design conditions, and Figure 13 (b) shows the pressure field distribution under comparative conditions.

It can be seen from the pressure cloud diagram in Figure 13 that the fluid in the valve body can be divided into three areas: the valve inlet and the interior of the valve body cavity, the narrow channel area around the spool, and the valve outlet area. The pressure change in the valve inlet and the interior of the valve body cavity is small, and the pressure in all positions can be regarded as the inlet pressure. The pressure in the narrow channel area around the spool changes the most violently, and the pressure will quickly drop to the inlet pressure. This area range is small, and the smaller the valve opening, the smaller the area. The pressure change in the valve outlet area is also small, and the pressure in all positions can be regarded as the outlet pressure.

5. Conclusion

1) The main influencing factor of the flow field inside the valve body is the valve opening. The main influencing factor of the highest flow velocity inside the valve body is the pressure difference between the inlet and outlet.

2) There is a high-speed area in the narrow flow channel area around the valve body, where the flow velocity in the fluid domain is greater than the flow velocity in other parts of the valve, but the range of the high-speed area will decrease as the valve opening decreases. The pressure in the fluid domain will change abruptly in the narrow flow channel around the spool, and the inlet pressure will drop rapidly to the outlet pressure.

3) A comparative analysis of the flow coefficients at the spool opening at 1%, 2%, 3%, 4%, and 5% shows that the adjustable ratio of this spray regulating valve is 72.9.

References

[1] Fu Xiaobo, Zhang Dafa. (2009) Effect Analysis of Spray Plant Fault to Pressurizer Control System. Chinese Mechanical Engineering Society, Lanzhou.

[2] M.F.Rahmat, Sy Najib Sy Salim, N.H.Sunar. (2012) Identification and Non-linear Control Strategy for Industrial Pneumatic Actuator. International Journal of the Physical Sciences, 7(17): 2565-2579.

[3] Zang X.N. (2010) Nuclear Power Plant Systems and Equipment. TSINGHUA UNIVERSITY PRESS, Beijing.
[4] Fisher CONTROL VALVE HANDBOOK Fourth Edition, (2005) Emerson Process Management.
[5] Davis J A and Stewart M. (2002) Predicting Globe Control Valve Performance—Part I: CFD Modeling. Journal of Fluids Engineering, 124(3): 772–777.
[6] Davis J A and Stewart M. (2002) Predicting Globe Control Valve Performance—Part II: Experimental Verification. Journal of Fluids Engineering, 124(3): 778–783.
[7] Roorda O. (1998) Computer Simulation Helps Reduce Pressure Loss. Water Sciences and Engineering Technology, 145: 22–24.
[8] Ueno H, Okajima A, Tanaka H and Hasegawa T. (1994) Noise Measurement and Numerical Simulation of Oil Flow in Pressure Control Valves. Trans. Jpn. Soc. Mech. Eng., Ser. A, 37: 336–341.
[9] Li Gaoming, Li Ming. (2012) Ansys 13 Flow Field Analysis Technique and Application. CHINA MACHINE PRESS, Beijing.
[10] Launder B E , and Spalding D B. (1972) Lectures in Mathematical Models of Turbulence, Academic, London.
[11] Versteeg H K , Malalasekera W. (2000) An Introduction to Computational Fluid Dynamics. World Publishing Corporation, Beijing.
[12] Palau Salvador G , Arviza J , and Frankel S H. (2004) Three-Dimensional Control Valve With Complex Geometry : CFD Modeling and Experimental Validation. 34th AIAA Fluid Dynamics Conference and Exhibit, Portland.
[13] He Yanqing. (2005) Control Valve Engineering Design and Application. Chemical Industry Press, Beijing.
[14] Lu Peiwen. (2007) Practical Technology of Control Valve. CHINA MACHINE PRESS, Beijing.