

# Study on the Regulating Performance of Sliding Regulation-Valve

Wei Hu<sup>1,\*</sup>, Xiaoyong Peng<sup>1</sup>, Yuan Zhang<sup>1</sup>, Yulan Zheng<sup>1</sup> and Fangyao Zhu<sup>1</sup>

<sup>1</sup>School of civil engineering, University of South China, Heng yang, hu nan, 421000

\*Corresponding author

**Abstract.** Using a proven reliable method of CFD to study the regulating performance of a sliding regulation valve with a conical spool and rugby body. The numerical simulation results indicate that no matter where the spool is located, the flow field always has a vortex at the center of the valve body; When the spool is at the origin, the vortex and resistance coefficient of the valve are the minimum; When the spool moves from the origin to the right (the opening of the valve becomes smaller) to reach a certain position later, vortex currents also begin to appear around the tube wall behind the orifice. In addition, the vortex increases as the throttling port decreases whereas the resistance coefficient of the valve ascends slowly with the increase of the deviation of the spool and the rise in series; This type of regulating valve has S type (slow at both ends, sensitive at the center) flow characteristics at the stroke, and is not affected by the size of Re.

**Keywords:** Control valve; CFD; Flow characteristics; Resistance coefficient.

## 1. Introduction

In the intelligent control of modern chemical plant, the control valve plays a very important role in the exchange of energy or mass between the system and the outside and the change of the system pressure, and the performance characteristics of the valve are determined by the flow mechanism inside the valve; Therefore, the study of the internal flow mechanism of the regulating valve is of great significance to the design and application of the regulating valve [1-2].

In order to quickly and accurately grasp the internal flow mechanism of valve, Davis J A J expounds the application of FLUENT software in the development of regulating valve from the point of view of two and three dimensions, providing a way to shorten the design and research cycle of valves [3]. So far, the numerical simulation method has been widely used in the research of internal flow mechanism of regulating valve at home and abroad; The numerical simulation using CFD fluid analysis software is in good agreement with the experimental results, which proved to be reliable [4-9].

In this paper, a kind of sliding valve which have a conical spool and a rugby body is studied; The inlet and outlet of the valve body are round and the sectional area is uniform; Using a proven reliable method of CFD to study the regulating performance of a sliding regulation-valve.

## 2. Governing Equations and Numerical Methods

### 2.1. Governing Equations and Numerical Methods

The sliding control valve studied has a rotating body, the air flow in the valve belongs to an axisymmetric stationary field with low velocity and incompressible turbulence; the governing equations in cylindrical coordinates are as follows:



$$\frac{\partial r U_r}{\partial r} + r \frac{\partial U_x}{\partial x} = 0 \quad (1)$$

$$U_r \frac{\partial U_r}{\partial r} + U_x \frac{\partial U_r}{\partial x} = -\frac{1}{\rho} \frac{\partial P}{\partial r} + f_r + \nu \left( \Delta U_r - \frac{U_r}{r^2} \right) \quad (2)$$

$$U_r \frac{\partial U_x}{\partial r} + U_x \frac{\partial U_x}{\partial x} = -\frac{1}{\rho} \frac{\partial P}{\partial x} + f_x + \nu \Delta U_x \quad (3)$$

$U$ (m/s) is the air velocity;  $r$ (m) and  $x$ (m) are the cylindrical coordinate radial and horizontal direction respectively;  $\rho$ (kg/m<sup>3</sup>) represents air density;  $P$ (Pa) represents the atmospheric pressure;  $f_r$ (N) represents the volume force;  $\nu$ (m<sup>2</sup>/s) represents the kinematic viscosity.

The flow of air in the valve is the fully developed turbulence, adopting the standard turbulence model of  $\kappa$ - $\epsilon$  for it is well suited for the fully developed turbulence [10~12].

The numerical method uses the SIMPLE algorithm based on the finite volume method to solve the discrete equations simultaneously; the discrete scheme is calculated by the two order upwind scheme, so as to ensure good calculation accuracy.

## 2.2. Numerical Method Validation

### 2.2.1. Background flow Physics Model.

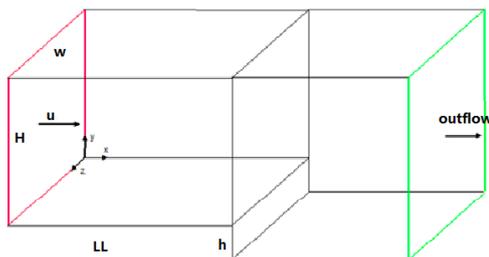
#### (1) Physics model

The three-dimensional computational domain established according to the document [13] is shown in Figure 1, and the coordinate system is shown in the diagram. The upper channel height ( $H$ ) of the step is 470mm, the height of the step ( $h$ ) is 30mm, the channel width ( $W$ ) is 300mm, and the distance from the step to the flow inlet ( $LL$ ) is 3000mm. The length of the inlet and outlet passages is designed to allow for the full development of the flow regime [14-15].

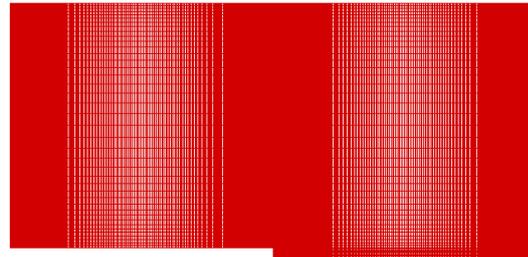
#### (2) Meshing and boundary conditions

Using hexahedral grid in the calculation area and refining it at the boundary and steps, the inflow and the outflow region are relatively sparse; figure 2 is a schematic diagram of an XY section. The number of grid is  $8 \times 105$ .

Boundary conditions: the inlet is the velocity inlet (uniform inflow) and the outlet is the outflow; the step and other walls are subjected to solid walls without sliding.



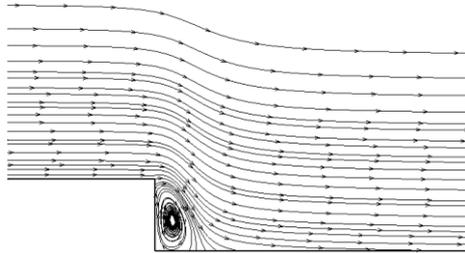
**Figure1.** Computational domain model



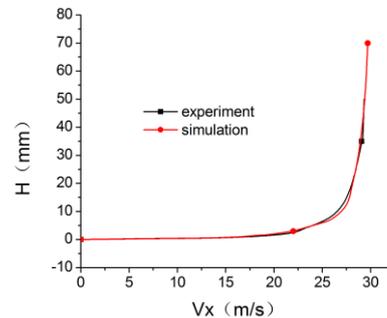
**Figure 2.** Schematic diagram of  $Z=150$ mm grid section

**2.2.2. Numerical Simulation Results and Analysis.** Selecting the cross sectional streamline diagram of  $Z=150$ mm as shown in Figure 3; The flow field has the vortex at the corner of the step, and the other regions are evenly distributed. At 5mm ahead of the step, taking  $V_x$  values in the  $Y$  direction and comparing with the experimental values of the literature [13]. As shown in Figure 4, finding that the two basically coincide; Taking the friction stress at the downstream wall of the step and fitting the curve, as shown in Figure 5, drawing that the distance between the step and the reattachment is  $X_r$ , along with the simulation value of  $X_r/h$  is 4.71, and the experimental result is 4.98; The relative error

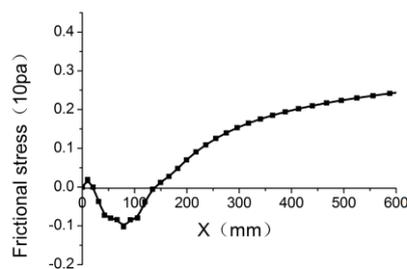
of the two is 4.8%. The simulation results are in good agreement with the literature, which shows that the calculation method used in this paper has good reliability.



**Figure 3.** cross sectional streamline diagram of  $Z=150\text{mm}$



**Figure 4.** velocity variation curve of the Y direction at 5mm ahead of the step



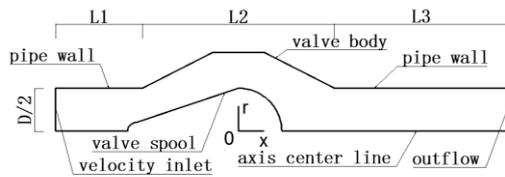
**Figure 5.** Frictional stress at the downstream wall of the step

### 3. Analysis of Regulating Performance of Sliding Regulating Valve

#### 3.1. Physical Model and Working Conditions

**3.1.1. Physical Model and Working Conditions** Regulating valve and its inlet and outlet pipe meridian plane, coordinate system origin are as shown in figure 6; The whole calculation domain is made up of pipe wall, valve body, spool, axis center line, velocity inlet, outflow and equal boundary. Among them, L1 and L3 are import and export pipelines, and the length of them are  $5D$  and  $10D$ , the radius of pipe diameter is  $D/2$ ; The L2 part is valve, which is composed of a valve body and a valve core. The length of the inlet and outlet passages is designed to take into account the full development of the flow regime. Simulating the flow distribution of the regulating valve at different valve position and different  $Re$ .

**3.1.2. Meshing and Boundary Conditions.** As shown in figure 7, the computational domain is composed of quadrilateral structured mesh and triangular unstructured mesh. Refining the throttle portion of the regulating valve, yet the inflow region and the outflow region are relatively sparse. The number of grid is  $1.3 \times 10^5$ . Boundary conditions: the inlet of computational domain is velocity inlet (uniform flow) and the outlet is outflow; the boundary conditions of the valve core surface, the valve body surface and the pipe surface are wall, the lower boundaries are Axis.



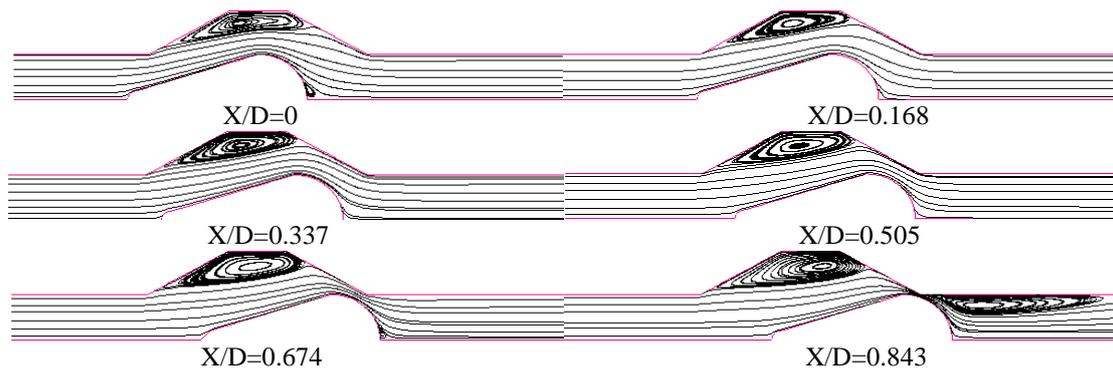
**Figure 6.** meridian surface of regulating valve and pipe



**Figure 7.** The mesh of meridian surface of regulating valve and pipeline

### 3.2. Numerical Simulation Results and Analysis

**3.2.1. Flow Field Analysis.** Fig. 8 is a streamline diagram of six different spool positions ( $X/D$ ) at the same  $Re$ . When the spool position is in origin,  $X/D=0$ , while the  $X/D$  value increases gradually with the spool moves to the right. Seen from the diagram, no matter where the spool is located, the flow distribution always has vortex at the center of the valve body. When the spool is at the origin, the vortex and resistance coefficient of the valve are the minimum; When the spool moves from the origin to the right (the opening of the valve becomes smaller) and reaches a certain position later, vortex currents also begin to appear around the tube wall behind the orifice; In the whole process, the size of the vortex increases with the deviation of the valve core.

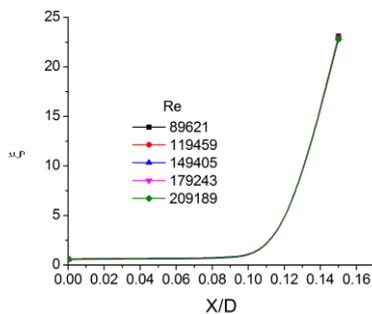


**Figure 8.** Streamlines of different spool positions

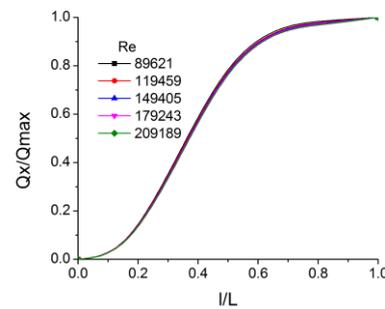
**3.2.2. Drag Coefficient Analysis.** The coefficient of resistance is an important parameter to evaluate the energy consumption of regulating valve, which can be obtained by the pressure loss formula (4) [16]. Figure 9 shows the relationship between the coefficient of resistance ( $\xi$ ) and the spool position ( $X/D$ ) at different  $Re$ . As shown by the diagram, the drag coefficient is independent of  $Re$ ; when the spool is at the origin, resistance coefficient of the valve is the minimum; Whereas the resistance coefficient of the valve ascends slowly with the increase of the deviation of the spool and then rise in series when the spool moves from the origin to the right. The analysis indicates that the resistance coefficient is increasing in series because there is a new vortex inside the valve body.

$$\xi = 2 \nabla P / \rho v^2 \quad (4)$$

**3.2.3. Characteristic Flow Curves.** The flow characteristics of valve are shown in figure 10 under different  $Re$  conditions.  $l$  represents the spool travel,  $L$  represents the maximum travel (when the valve is fully closed, the trip was 0; when the valve is fully open, the trip is maximum); This type of regulating valve has S type (slow at both ends, sensitive at the center) flow characteristics at the stroke, and is not affected by the size of  $Re$ .



**Figure 9.** drag coefficient curve under different Re



**Figure 10.** flow characteristic curve under different Re

#### 4. Conclusion

(1) No matter where the spool is located, the flow field always has vortex at the center of the valve body. When the spool is at the origin, the vortex and resistance coefficient of the valve are the minimum; When the spool moves from the origin to the right (the opening of the valve becomes smaller) and reaches a certain position later, vortex currents also begin to appear around the tube wall behind the orifice; In the whole process, the size of the vortex increases with the deviation of the valve core.

(2) The drag coefficient is independent of Re; When the spool is at the origin, resistance coefficient of the valve is the minimum; Whereas the resistance coefficient of the valve ascends slowly with the increase of the deviation of the spool and then rise in series when the spool moves from the origin to the right.

(3) This type of regulating valve has S type (slow at both ends, sensitive at the center) flow characteristics at the stroke, and is not affected by the size of Re.

#### 5. Acknowledgement

This research was financially supported by the National Science Foundation. Hunan Province, colleges and universities scientific and technological achievements industrialization project (12CY001).

#### 6. Reference

- [1] Yu-dong XIE, Yong WANG, Yan-jun LIU. Control and instruments in chemical industry, J. 2012; 39(6):1111-1114. In Chinese.
- [2] Xin XIAO. Flow Analysis and Structure Optimization on Control Valve. D. Harbin: Harbin Engineering University. In Chinese.
- [3] Davis JAJ. development of control valve design Tools utilizing computational fluid Dynamics. Ar-kanasa: Arkanasauniversity 2000.
- [4] Shi-yang LI, Peng WU, Lin-lin CAO. CFD simulation of dynamic characteristics of a solenoid valve for exhaust gas turbocharger system, J. Applied Thermal Engineering, 2016.
- [5] Edward Lisowski, Grzegorz Filo. CFD Analysis of the Characteristics of a Proportional Flow Control Valve with an Innovative Opening Shape, J. Energy Conversion and Management, 2016,
- [6] Moftah Alshaikh, William Dempster. A CFD Study on Two Phase Frozen Flow of Air/Water through a Safety Relief Valve, J. International Journal of Chemical Reactor Engineering, 2015, 13(4).
- [7] Hai-feng LI. J. China Rural Water and Hydropower, 2015, (01):164-166+171. In Chinese.
- [8] Yi-qiu GAO. J. Turbine Technology, 2011, 53(05):328-330. In Chinese.
- [9] Zhen YAN, J. Chinese Hydraulics and Pneumatics, 2014, (02):98-100. In Chinese.
- [10] Juan SHI. Numerical simulation and analysis of the 3D flow and opening closing Processes in control valve, D. shanghai: university of shanghai for Science and technology, 2005. In Chinese. In Chinese.

- [11] Wei-zheng ZHANG. Numerical simulation and optimization analysis of interior flow field within the control Valve, D. Lanzhou: Lanzhou University of Technology, 2007. In Chinese. In Chinese.
- [12] Liang-liang XU. Numerical Simulation and Structure Optimization of Flow in Control Valve Based on CFD, D. Shanghai: East China University of Science and Technology, 2016. In Chinese.
- [13] Guo-ding CHEN. Research on the Backward-facing Step Flow Control, D. Nanjing University of Aeronautics and Astronautics, 2012. In Chinese.
- [14] Theodor Nitulescu, Stefan Talu, Applications of descriptive geometry and computer aided design in engineering graphics, Cluj-Napoca, Romania, Risoprint Publishing house, 2001, ISBN973-656-102-X.
- [15] Adrian Florescu-Gligore, Stefan Talu, Dan Noveanu, Representation and visualization of geometric shapes in industrial drawing, Cluj-Napoca, Romania, U. T. Pres Publishing house, 2006, ISBN-10 973-662-230-4, ISBN-13 978-973-662-230-4.
- [16] Tianyu LONG, Zeng-ji CAI. Fluid mechanics. China construction industry press, 2004:90-91. In Chinese.