Analysis of a Control valve's Inner Flow Field Characteristics Based on CFD

Wang Liang
Beihang University
Beijing, 100191, China
ad.wang@qq.com

Abstract—**In this paper, the inner flow field characteristics of a control valve were analysed through dynamics simulation and showed by using the three-dimensional visualization. Through the analysis of simulation results, reasons were found for the energy loss, which was, then, reduced by the optimized flow path. Calculations about the optimized positions were carried out, the results of which showed an improvement of flow and a significant decrease in energy loss.**

Keywords- CFD; Control Valve; Inner Flow Field

I. INTRODUCTION

Control valve, whose main function is regulating flow velocity, is indispensable among the control equipment and has a significant influence on national economy. With the continuous improvement of automation degree, the control valve, such as the bulky valve body with poor reliability, has been widely used in a variety of industrial sectors. And with the rapid development of the computer and CFD (Computational Fluid Dynamics), the numerical simulation, whose superiority is more and more obvious, has gradually become a very important adjunct to the engineering design.

II. CALCULATION MODELS AND BOUNDARY CONDITIONS

Acquiring numerical solution is a discrete approximate calculation method. Numerical simulation, with which the feasibility and reliability were proved, were compared with theoretical calculation for the flow coefficient of the control valve. In order to make an approximation of the unknown quantities through nodes using computers, we need to proceed from a given differential equation or some basic laws of physics firstly, then to establish an approximation with unknown amount between these nodes algebraic equations secondly. Thus we can solve the equation through the parameters and the conditions by computer.

A. Calculation model

To solve problems in fluid motion, some basic equations are needed. If the flow is in a turbulent state, the system should comply with the additional turbulence transport equation. The control equation is a mathematical description of conservation laws.

The inner flow field of the control valve contains three-dimensional viscous incompressible fluid under constant flow conditions. A three-dimensional Cartesian coordinate is used in the control equation. And the N-S equations' form is as follows:

$$\begin{cases} \rho \dfrac{du}{dt} = \rho F_x - \dfrac{\partial p}{\partial x} + \mu(\dfrac{\partial^2 u}{\partial x^2} + \dfrac{\partial^2 u}{\partial y^2} + \dfrac{\partial^2 u}{\partial z^2}) \\ \rho \dfrac{dv}{dt} = \rho F_y - \dfrac{\partial p}{\partial y} + \mu(\dfrac{\partial^2 v}{\partial x^2} + \dfrac{\partial^2 v}{\partial y^2} + \dfrac{\partial^2 v}{\partial z^2}) \\ \rho \dfrac{dw}{dt} = \rho F_z - \dfrac{\partial p}{\partial z} + \mu(\dfrac{\partial^2 u}{\partial x^2} + \dfrac{\partial^2 u}{\partial y^2} + \dfrac{\partial^2 u}{\partial z^2}) \end{cases}$$

The continuity equation:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

The N-S equation is the most basic equation of fluid dynamics calculation. The analytic work of almost all of the viscous fluid is concerned with solving N-S equations.

B. The boundary conditions for the calculation

In this paper, three-dimensional incompressible N-S equation was solved. And the standard $\kappa - \varepsilon$ model was used as turbulence model. A standard wall function processing was for the area near the solid wall, while the simple algorithm on unstructured grids was for the discrete equations. The problems in velocity and pressure field were solved by using the method of hidden-iterative solution. The boundary conditions require the total pressure and outlet pressure of the import.

III. GEOMETRIC MODELING AND MESHING

A. The regulating valve's three-dimensional solid modeling

The control valve's solid modeling was completed by Solidworks 3D design software, and its Boolean operations extracted the control valve's flow channel and extend pipeline part. To facilitate a better simulation, the control valve inlet duct is lengthened. And its bodies before and after the pipe are both 190 mm in length (approximately 5 times the diameter of the pipe). Figure 1 is generated as three-dimensional geometric entities.

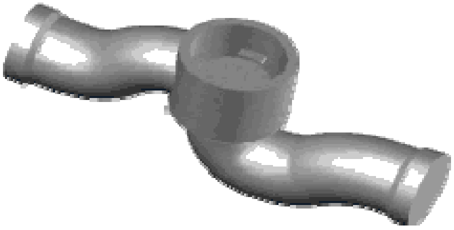

Figure 1. Sketch of control valve flow

B. Computational domain and meshing

Will import three-dimensional entities regulating valve Gambit handle numerical computation ago. Regulating valve runner overall mesh grid number is relatively large, high requirements for computer memory, calculation speed is slow. Since the geometric structure of the HTS-regulating valve, the symmetric part of the calculations to adjust the valve flow path for the computational domain, it is possible to greatly reduce the number of the mesh, conducive to improve the density of the grid, thereby improving the calculation accuracy and speed.

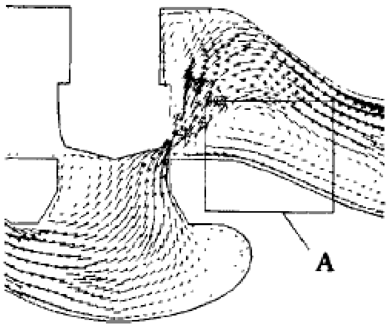

Figure 2. Velocity vectors of symmetrical surface

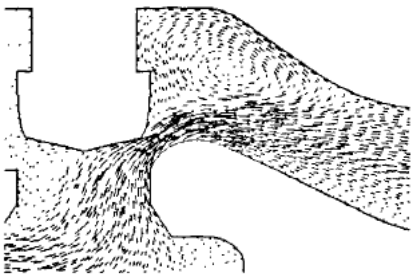

Figure 3. Velocity vectors of improved symmetrical symmetrical surface

The entire region was divided into three parts. Spool and valve seat were sealed according to pressure's relative changes around the grid. And the control valve was divided into unstructured tetrahedral mesh and straight pipe flow in the inlet and outlet of the flow channel's stabilization part of the grid. Wedge (hexahedral) grid can be used in the division, and the grid can be divided more sparsely account for the small amount of 10% (approximately) grids. The number of divided grid is about one million. Figure 3 is a valve meshing with a 100% opening valve channel.

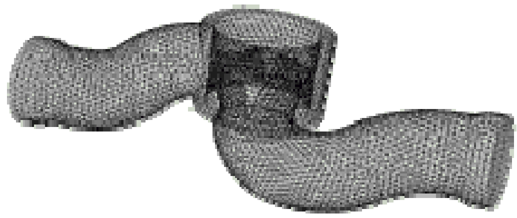

Figure 4. Grid of a full control valve

IV. RESULTS AND ANALYSIS

The calculation of the control valve was fully carried out under different working conditions.

TABLE I. PRESSURE CONDITIONS AT INLET AND OUTLET

No.	Inlet pressure (kPa)	Outlet pressure (kPa)
1	540	500
2	540	480
3	540	450
4	540	425
5	540	400

And the inner flow field's symmetry plane of the control valve was analysed to study the distribution.

The control valve in Figure 2 meets a total import pressure of 540 kPa, and an outlet pressure of 400 kPa. The pressure dropping in the flow channel is mainly used to overcome the resistance before and after the control valve.

The flow coefficient is one of the most important indicators of flow characteristics. The calculation equation is as follows:

$$K_V = Q/(9.9\sqrt{\Delta P})$$

The Q in the equation is the flow rate.

So we can see the flow characteristics through the calculation that the control valve's inlet and outlet velocity are usually evenly distributed (about 2.34 m / s). And the computational domain was taken to let the water full flow. According to the continuity equation (pVA = C), the diameters of the inlet and outlet of the pipe are equal. And the average velocities of inlet and outlet are quite high (friction is not considered). The spool flow of water gradually stagnates at the bottom of the plane, where the pressure rises to its highest point while kinetic energy turns into potential energy. And then, when the water flow is between the spool and the valve seat, the flow area decreases rapidly, so the pressure decreased rapidly too.

V. OPTIMUM CHANNEL DESIGN AND ANALYSIS

With a 100% opening valve channel, the pressure before changes, causing a very-obvious large-scale vortex at the right bottom of the valve channel. Followed by the mainstream concentrating in a tube of the upper wall in the exit channel of the control valve, lower wall makes the velocity's distribution and gradient unevenly.

The vortex that forms in the recirculation zone of the control valve causes the main loss of energy, according to the numerical simulation results. The reason for this situation is that the control valve at right side of the bottom in the flow channel cross-sectional area is too large. So we can reduce the cross-sectional area in that region as to change the wide range whirlpool into a good condition. However, considering the installation requirements of the control valve, I think, to reduce the energy loss, only the right part in the flow channel of the valve can be improved in the case of full flow.

VI. CONCLUSION

Control valve is an important part of the industrial automation application. CFD method can regulate valve internal flow field. And CAD be able to get the flow field, the pressure distribution, the flow line, and the energy loss of the visual results.

The results showed that:

(1) The smaller flow area of the flow field within the process to adjust the valve flow. The throttling effect is more obvious, and the corresponding pressure will be smaller, while the flow rate will increase. Throttle at the turbulent kinetic energy change, turbulence easy development and changes, the change of fluid flow, flow resistance is large;

(2) When constant pressure before and after the control valve, the outlet flow regulator valve and the valve opening degree is closely related. Spool linear for the flat type, regulating the flow characteristics of the valve features quick, open process to quickly increase the flow, the opening degree of 20%, the percentage of traffic to be able to achieve about 50%;

(3) Spool opening degree is kept constant, is closely related to the outlet flow regulating valve and the front and rear differential pressure, increasing the pressure drop, can be a corresponding increase in the outlet flow. However, with the increase of the pressure drop, flow incremental increase per unit amount of the pressure drop brought gradually reduced.

REFERENCES

[1] Wang Y P, Wilkinson G B, Drallmeier J A. Parametric study on the fuel film breakup of a cold start PFI engine[J]. Experiments in Fluids, 2004, 37(3) : 385-398.

[2] Kwan Chin-Tarn. A study of process and die design for ball valve forming from stainless steel tube[J]. The International Journal of Advanced Manufacturing Technology, 2005, 26 : 983-990.

[3] Piller Marzio, Nobile Enrico, Thomas J. DNS study of turbulent transport at low Prandtl numbers in a channel flow[J]. Journal of Fluid Mechanics, 2002, (458) : 419-441.

[4] Wissink J G. DNS of separating low Reynolds number flow in a turbine cascade with incoming wakes[J]. International Journal of Heat and Fluid Flow, 2003, 24(4) : 626-635.

[5] Michelassi V , Wissink J G , Rodi W . Direct numerical simulation, large eddy simulation and unsteady Reynolds·-averaged Navier-Stokes simulations of periodic unsteady flow in a low—pressure turbine cascade：A comparison[J]. Journal of Power and Energy, 2003, 217(4) : 403-412.

[6] ollet−Miet P, LaurenceD, FerzigerJ. LES and RANS of turbulent flow in tubebundles[J]. International Journal of Heat and Fluid Flow, 1999, 20(3) : 241-254.

[7] FeizA A, Ould-Rouis M, Lauriat G. Large eddy simulation of turbulent flow inarotating pipe[J]. International Journal of Heat and Fluid Flow, 2003, 24(3) : 412−420.

[8] Grigoriadis D G E, Bartzis J G, Goulas A. Efficient treatment of complex geometries for large eddy simulations of turbulent flows[J]. Computers and Fluids, 2004, 33(2) : 201−222.