Numerical Simulation of Fluid Flow inside the Valve

Qin Yang\textsuperscript{a}, Zhiguo Zhang\textsuperscript{a*}, Mingyue Liu\textsuperscript{a}, Jing Hu\textsuperscript{b}

\textsuperscript{a} School of Naval Architecture and Ocean Engineering Huazhong University of Science and Technology, Wuhan 430074, China
\textsuperscript{b} PERA GLOBAL, Wuhan, Hubei, P.R. 430071, China

Abstract

Stop valves are commonly used as fluid flow control equipments in many engineering applications. Thus it’s more and more essential to know the flow characteristic inside the valve. Due to the fast progress of the flow simulation and numerical technique, it becomes possible to observe the flows inside a valve and to estimate the performance of a valve. This paper presents the researches of the authors in modeling and simulation of the stop valves. The flow system with stop valves is complex structure and has non-linear characteristics, because the construction and the hydraulic phenomena are associated of stop valves. In this paper, three-dimensional numerical simulations were conducted to observe the flow patterns and to measure valve flow coefficient and flow fluctuations when stop valve with different flow rate and uniform incoming velocity were used in a valve system. The spectra characteristics of pressure fluctuation on the flow cross section were also presented here to investigate the wake induce of the valve part. These results not only provided people with the access of understanding the flow pattern of the valve with different flow rate, but also were made to determine the methods which could be adopted to improve the performance of the valve. Furthermore, the results of the three-dimensional analysis can be used in the design of low noise and high efficiency valve for industry.

© 2011 Published by Elsevier Ltd. Open access under CC BY-NC-ND license.

Keywords: Valves, Fluid flow, Numerical simulation, Design;

1. Introduction

Valves are common components of process industry systems. They had played an important role in a variety of different industries as a hydraulic device for fluid flow control. Different types of valves have applications of their own. Valves are often used for safety reasons in flow control systems. When used for flow control, the dynamics of the valve has to match the dynamics of the flow system. The relation between valve position and the pipe system would make the pressure drop and flow highly non-linear. This implies difficulties to predict the properties of down- or up-sized. Though there are both CFD and
CAD tools for valve design on the market, much design rests on experience and experiment. So it is very important to understand fluid flow inside the valve when engineers want to improve the performance. Some detail studies have been carried out in past. However, making detailed flow measurements within complex valve passages is very difficult and almost impossible. Therefore, it is important to imply other techniques, such as numerical methods, to compliment experimental studies and provide additional information regarding valve flow patterns. Yuan [1] et al. applied the RNG k-ε turbulence model to simulate turbulent flows of curved inlet cut-off valves and used PIV to display the flow field of it. The results show that the flow pattern of inclined inlet cut-off valve is much better than that of curved inlet cut-off valve. Salvador [2] et al. developed a model which accurately represents the flow behavior through a three-dimensional control valve, as measured by experimental validation. Ramanath [3] et al. took advantage of CFD to identify the flow-related problems inside the valve trim area and the flow visualization gives a better insight. Yuan [4] et al. applied the RNG k-ε turbulence model to simulate turbulent flows in symmetrical planes and in three dimensions of curved inlet cut-off valves and found that the RNG k-ε turbulence model can be used to simulate the separated turbulent flows with vortices. A comparison between experimental and numerical data, recently performed by Amirante et al. [5,6] for an open centre directional control valve, has confirmed the good accuracy of the CFD predictions in terms of forces evaluation. Kim [7] et al. focused on the investigation of the detailed hydrodynamic characteristics to compare analysis results by PIV and CFD, the investigation shows that numerical analysis are verified as a high accurate tool and could be used for various application in the flow fields of marine and industrial. Cho [8] et al. investigated the force balance of an unbalanced globe valve. A CFD analysis is carried out to evaluate the pressure distribution and forces acting on the top and bottom planes of the valve plug. The results of this analysis have been verified through experimentation. Song [9] conducted three-dimensional numerical simulations to observe the flow patterns and to measure valve flow coefficient and hydrodynamic torque coefficient when butterfly valve with various opening degrees. By contrast, a group of experimental data is used to compare with the data obtained by CFX simulation to investigate the validity of numerical method. Srikanth [10] et al. carried out a series numerical study in valve with moving grids, and it was found that CFD simulation with valve element mesh motion indicates that pressure history is significantly affected by the velocity of moving contact in the puffer chamber. Ba [11] et al. carried out the dynamic simulation of flow field when stop valve opening or closing based on CFD. Flow phenomenon such as whirlpool water hammer and stagnant water zones can be observed from the CFD analysis result which provides a theoretical basis for valves structure optimization. Lumír [12] deals with the simulation of pulsating flow in the pipeline for options with and without the hydraulic accumulator, when the throttle valve is at the outlet of pipeline. Time dependent flow through the pipeline is simulated in the software Matlab-SimHydraulics. Numerically simulated amplitude-frequency characteristics of pressure are verified experimentally. JIANG [13] et al. search the reason which affect the capability and lead the noise of hull valve base on the visual result of simulation. KIM [14] studied the flow characteristics of a 3/2 -way solenoid valve used in the four stroke propulsion engine of a ship by performing numerical simulation using ANSYS CFX 12.0 commercial software package. The results of the three dimensional numerical simulation will be helpful in the proper design of solenoid valve for the industrial applications.

This paper aims at a detailed CFD analysis of the 3D flow field in the chambers of a stop valve. The well known CFD package, Fluent, including the Gambit grid generator, has been used to perform all the numerical computations. Numerical method used here had been valid in previous work. RNG k-ε turbulence model has been applied to simulate turbulent flows in three dimensions of curved inlet stop valves at the conditions of different flux in order to learn the flow characteristics inside the valve. Numerical results show some important conclusions which could be used to improve the stop valve design.
2. Numerical implementation

2.1. Governing equations and Turbulence model

Steady flow of a Newtonian, incompressible fluid was assumed. The governing equations were solved using the SIMPLE algorithm developed by Patankar. The conservation of mass and momentum equations are:

\[
\frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} + \frac{\partial \bar{w}}{\partial z} = 0
\]

\[
\rho \frac{\partial \bar{v}}{\partial t} + (\bar{v} \cdot \nabla)\bar{v} = -\rho g \nabla h - \nabla p + \nabla \bar{\tau}^e
\]

where \( \nabla \bar{\tau}^e \) stands for the divergence of the stress tensor. Considering this term, the number of unknowns turns to be 13 (velocity components, pressure and stress field), which make the equation above very difficult to be solved. However, calculations are simplified by using the Navier-Stokes equations, which relate the stress field to the fluid viscosity. The Reynolds-averaged form of the Navier-Stokes equations are typically solved for engineering flows. Although the commercial code FLUENT offers a wide range of turbulence models, only RNG k-\( \varepsilon \) model was used here based on the past experience and studied result. The major parameter used to evaluate in this study is the different flow rate into the valve at pipe system inlet.

Turbulent model used here is RNG k-\( \varepsilon \). In this model, the unknown Reynolds stresses are related to the known mean strain rate via a turbulent viscosity which is calculated as:

\[
\mu_t = C_{\mu} \frac{k^2}{\varepsilon}
\]

Specification of this viscosity requires solution to two additional modeled transport equations for the turbulent kinetic energy and the dissipate rate. Default values for the model constant in the above equation were used. The RNG k-\( \varepsilon \) turbulent model was selected for subsequent studies for reasons of computational efficiency and accuracy.

2.2. Numerical method

Here, the valve is connected as a flow control valve, as is typically the case. The geometry is complex and 3D geometry was drawn using Solidworks to facilitate further changes to the geometry. The geometry model of this curved valve is shown in Fig 1. Its sectional view is shown in Fig 2.

Fig. 1. valve geometry  Fig. 2. Sectional view of the valve

The numerical grid for the chambers and the ducts inside the valve are generated by means of Gambit, a pre-processor of Fluent. The computational domain is shown in Fig 3, which provides a view of the unstructured (the complex geometry does not allow the use of structured grids) structure computational grid employed, formed by about 1223298 cells. Fig 3 also shows the computation domain in this study. Fig 4 shows the test valve for the code validation. Fig 5 shows grid structure of the valve numerical analysis model. Preliminarily, the grid size function has been used to refine the grid in the sudden change
of the area to ensure an adequate grid size where the maximum velocity and pressure gradients occur. In particular, the grid is significantly refined in the block section and in the portion of the chamber, a downstream of the metering section, where the pressure drop is converted into kinetic energy and turbulent phenomena occur.

Fig. 3. Computation Domain

Fig. 4. Test valve for validation

Fig. 5. Grid structure of the valve

2.3. Boundary conditions and numerical simulation

The method discretizes the equations in a collocated grid node with second order upwind scheme to ensure the accuracy of the simulation results. With regards to the inlet and outlet boundaries (10D from the inlet and outlet of the valve), velocity and pressure conditions were specified, respectively.

A proper estimation of turbulent phenomena has great importance to determine the valve flow features. In particular, the flow inside a hydraulic valve is characterized by the coexistence of “free shear flows”, due to the flow jet at the exit of the metering section, and “wall bounded flows”, which are strongly influenced by the wall effects. The most suitable turbulence model for this kind of problem appears to be the RNG-k-ε model. This model gives a reliable estimation of the turbulent quantities upstream and downstream of the restricted sections and is able to estimate properly both the free jet and the wall bounded region. The numerical studies were conducted for steady state to help people understand mean flow feature and unsteady flow to investigate the flow induce vibration and pressure fluctuation on the valve component. Note that such turbulence models do not describe turbulent fluctuation in detail but provide only the average effects of fluctuations on the short-time averaged flow quantities through the modeling of the Reynolds stress terms. The time step is set based on the flow condition. The pressure fluctuations were analyses based the time period and spectra characteristic to identify the fluid flow noise when the flow pass the valve channel.
3. RESULTS AND DISCUSSION

3.1. Mean Flow Feature

Fig 6 and Fig 7 show the simulation results for steady state of the pressure and the velocity field at longitudinal section when the Reynolds number is 6e5.

The pressure field Fig 6 shows that the main pressure drop is produced in the throat path, and the velocity distribution is shown in Fig 7. Jet produced by the valve at the exit of the throat can be easily distinguished. Fluid velocity in the throat is about 2.5 times its mean velocity at the inlet boundary condition. The pressure and velocity distribution trend with different velocities are completely the same. Part of fluid is hampered because of the valve piston. Direction of fluid particle can't suddenly change, so backflow appears near the throat path.

![Fig. 6. Pressure distribution at valve longitudinal section](image1)

![Fig. 7. Velocity distribution at valve longitudinal section](image2)

Fig 8 shows that there is flow recirculation inside the valve flow path. Fig 9 shows the pressure distribution at the center cross section from inlet to outlet.

![Fig.8 Streamline inside the valve](image3)

![Fig. 9 pressure distribution inside valve body along the flow direction](image4)

3.2. Unsteady flow characteristic

The unsteady flow characteristic inside the pipeline will be the major factor which would cause the pipe system vibration and flow induced the noise. In order to understand valve flow characteristics, four cross sections before, inside and after the valve are selected to monitor the pressure oscillation which would provide the pressure wake inside the pipe system.

Since it is not convenient to use the absolute value of pressure, whose range is too wide to reflect the fluid flow oscillation, the value of pressure oscillation were analyzed similar to sound pressure level to show the difference strength of pressure wake inside the valve. The sound pressure level could be calculated use:
\[ L_p = 10 \log \frac{P^2}{p_0^2} = 20 \log \frac{P}{p_0} \]  

(4)

where \( p \) stands for the pressure, \( p_0 \) stands for reference sound pressure. The using of Sound pressure level in this paper is just a convenient and obvious way to present pressure oscillation of flow, which does not mean the same as decibels in noise radiation level. So the sound pressure level based on pressure oscillation in time domain for different cross section of the pipe and valve could be used to obtain the corresponding frequency spectrum of the pressure fluctuation level through FFT transformation. These results would be very useful to analyze the vibration caused by flow fluctuation inside the pipe and valve system. The monitoring sections in valve are presented in Fig 10.

Figure 10 Monitor section position

The frequency-pressure relation is shown in fig 11. Frequency characteristic of the unsteady pressure wake from simulation shows the similar distribution and magnitude compared with the available data from the published paper. The fluid pressure wake mainly happens at low frequency band, which tell us that low frequency properties of fluctuating pressure would be the major source acts on the valve body and pipe system. The trend of the pressure fluctuation at the different Reynolds number is analyzed, the result shows that the fluctuation magnitude of pressure wake increases as Reynolds number increases.
4. CONCLUSION

This paper has provided a numerical investigation of the fluid flow inside a stop valve, including the modeling and the simulation of the stop valves. The flow system with stop valves is complex structure and has non-linear characteristics, because the construction and the hydraulic phenomena are associated of stop valves. The simulation results show that the main pressure drop is generated along the throat path. Flow jet produced by the valve exit of the throat can be easily distinguished. Fluid velocity in the throat is about 2.5 times its velocity in the inlet boundary condition. To sum up, when the fluid flows in the throat path between the piston and its seat, circulation area diminishes quickly. Due to it, the fluid pressure falls rapidly and the fluid here has the maximum velocity magnitude. With the growth of the velocity, trends of the pressure and velocity distribution inside the valve body is completely the same. Their only difference is the magnitude change. On the other hand, Part of fluid is hampered because of the valve piston. Direction of flow can’t suddenly change, so backflow appears near the throat path. Flow recirculation exists between the piston and valve body.

Moreover, the spectra characteristics of pressure wake fluctuation on the flow cross section were also presented here to investigate the wake induce the vibration of valve and pipe system. Five flow cross sections of valve pipe system are selected as monitoring faces which can present us the pressure wave of monitor section. Frequency-spectra of the wake at different monitor position show that low frequency properties of fluctuating pressure source would be the major source acts on the valve body and pipe system. The simulation results provide the useful information for continue work on the modification of the geometry inside the valves, which would improve the performance of the valve with low flow resistance and pressure oscillation. The reducing of vortex strength would be very helpful to reduce the pipe system vibrations and forces of the valve body.

References


