Journal of Natural Gas Science and Engineering 33 (2016) 108–117

Contents lists available at ScienceDirect

Journal of Natural Gas Science and Engineering

journal homepage: www.elsevier.com/locate/jngse

Influence of flashboard location on flow resistance properties and internal features of gate valve under the variable condition

Zhe Lin [a, b, c, **], Guangfei Ma [a, b, *], Baoling Cui [a, b], Yi Li [a, b], Zuchao Zhu [a, b], Nansen Tong [d]

[a] Institute of Mechatronic Engineering, Zhejiang Sci-Tech University, Hangzhou 310018, China
[b] The Zhejiang Provincial Key Laboratory of Fluid Transmission Technology, Hangzhou 310018, China
[c] Zhejiang Institute of Mechanical & Electrical Engineering co., Ltd, Hangzhou 310011, China
[d] College of Pharmaceutical Science, Zhejiang Chinese Medical University, Hangzhou 310053, China

ARTICLE INFO

Article history:
Received 26 September 2015
Received in revised form
28 February 2016
Accepted 4 May 2016
Available online 9 May 2016

Keywords:
Gate valve
Variable condition
CFD
RSM
Flow coefficient
Resistance coefficient
Pressure coefficient
Numerical simulation
Experiment

ABSTRACT

This article is intended to elaborate different inlet velocities that respectively are 13 m/s,15 m/s and 17 m/s to influence flow resistance characteristics and the internal flow characteristics of gate valve in the medium-low pressure gas transmission. This paper presents the CFD analysis of flow resistance characteristics and the internal flow characteristics of gate valve in a gate valve with different inlet velocity. Gate valves under different inlet velocity have eight relative opening degrees that respectively are 1/8, 2/8, 3/8, 4/8, 5/8, 6/8, 7/8 and 1. For every relative opening degree of gate valve structure, the systematic CFD simulations of flow resistance, pressure fluid, velocity fluid, velocity streamline and pressure coefficient characteristics. The study mainly analyses the pressure field distribution, velocity distribution, the velocity streamline distribution when the relative opening degree is 1/8, 2/8 and 1. Experimental measurements are also conducted for flow resistance characteristics and then compared with simulated flow resistance results. The CFD simulated flow resistance results show a good agreement with that of experimental flow resistance based on the Reynolds Stress Model (RSM) turbulence model provided by fluent software. Through analysis of flow resistance properties and internal flow characteristics, we concluded that it is not stable before the relative opening degree 2/8 that the valve flow resistance characteristics and internal flow characteristics, which need to be optimized; however, the valve flow resistance characteristics and flow characteristics gradually can be gradually stabilized after the relative opening degree 2/8, and does not have the characteristics of regulating valve. In the process of gas transmission, pressure energy loss mainly occurs in the valve core.

1. Introduction

Valve is an important part of the pipeline system. It can control the opening and closing of circulation medium, reversing, ensure the safety of the system, adjust the flow and pressure, etc. Gate valve is a shutter opening and closing pieces. The movement direction of the fluid through gate valve is perpendicular to the gate, which does not alter the flow direction of the fluid. Gate valve fully open when the least of drag coefficient for almost all the valves, and applicable scope of caliber, pressure and temperature range is very wide. Compared with the same caliber cut-off valve, the installation size of gate valve is small. So it is widely used in the chemical production, gas transmission air inlet and exhaust outlet location. The gate valve is suitable for the pipe system of conveying air, steam, water, solvents and other fluids, the shutdown does not require high frequent opening and closing of the occasion. However, under different conditions and at different locations, the internal flow resistance characteristics and the internal features of gate valve change research remains small.

Researchers commonly use numerical simulation and experimental methods to obtain the flow resistance characteristics and internal flow characteristics of the valve. Edvardsen et al. (2015) conducted an experimental and numerical analysis of single-phase pressure drops in a down hole shut-in valve. Wu et al. (2015) elaborate on specific computational fluid dynamics (CFD)

* Corresponding author. Institute of Mechatronic Engineering, Zhejiang Sci-Tech University, Hangzhou 310018, China.
** Corresponding author. Institute of Mechatronic Engineering, Zhejiang Sci-Tech University, Hangzhou 310018, China.
E-mail addresses: linzhe0122@zstu.edu.cn (Z. Lin), maguangfei@126.com (G. Ma).

http://dx.doi.org/10.1016/j.jngse.2016.05.025
1875-5100/

Nomenclature

C_d	flow coefficient of the gate valve
ξ	resistance coefficient of the gate valve
Cp	pressure coefficient of the gate valve
ρ	the fluid density (kg/m^3)
ν	dynamic viscosity (m^{2}s)
RSM	Reynolds Stress Model
P	Centerline pressure (pa)
length	total length (1530 mm)
height	Square tube cross section height (40 mm)
P_{in}	inlet static pressure (N/m^2)
$\triangle$P	pressure drop between inlet and outlet (bar = 10^5pa)
Q	volume flow rate (m^3/s)
K_ν	flow coefficient of the gate valve
U	magnitude velocity (m/s)
CFD	computational fluid dynamics
w_b	bulk mean velocity of the gas (m/s)
width	Square tube cross section width (20 mm)
3D	Three-dimensional

simulation methods for fitting the flow-pressure curve of a pressure control valve. Song et al. (2014) developed a numerical model to investigate the fluid and dynamic characteristics of a direct-operated safety relief valve. Dimitrov, (2013) investigated the flow-pressure coefficient of a pilot-operated pressure relief valve with theoretical analysis and verified the results with experiments. Valdés et al. (2014) performed and validated a series of CFD simulations of the capitation to flow through a ball check valve. They analyzed the characteristic flow coefficient of the valve and the hydraulic forces on the ball. Aung et al. (2014) analyzed flow forces and energy loss characteristics in a flapper–nozzle pilot valve with different null clearance by CFD software. Experimental measurements are also conducted for energy loss characteristics and then compared with simulated results. Chattopadhyay et al. (2012) investigated the flow process inside a pressure regulating valve by using a computational fluid dynamic approach. The commercial code FLUENT was found to aptly model the complicated flow processes inside the domain of interest. Lisowski and Rajda (2013) investigated the reduction of flow resistance in a hydraulic system, which is focused on a spool type directional control valve with pilot operated check valves. The system of designed flow paths is verified by CFD analysis with the use of ANSYS/FLUENT program on a three-dimensional model. Obtained results are compared with the results of the characteristics given in catalogues and coming from experimental research on the prototype. Amirante et al. (2014) evaluated the effects of cavitation upon the directional valve by means of thorough experimental and numerical investigations. Mougaes and Jagan, (2008) analyzed the ball valve internal three-dimensional flow field under different opening of the pressure drop and turbulent flow through Commercial software STAR − CD. Ye Y et al. Ye et al. (2014) clarify the effects of the groove shape on the flow characteristics of the spool valve through computational fluid dynamics (CFD) and experimental investigations. They simplified the structure of the valve, which was solved by using the RNG $k-\varepsilon$ turbulence model to simulate the pressure distributions of the flow fields inside three notches with their corresponding typical structural grooves. Lisowski et al. (2014) presented an innovative directional control valve based on the use of logic valves and a methodology followed for the design of it by using Solid Edge CAD and ANSYS/Fluent CFD software. Posa et al. (2013) carried out an analysis of the discharge coefficient and the flow force of a directional valve utilizing 2D CFD method and, in addition, the fluid-body interaction had been represented by an immersed boundary technique. Although the above studies used numerical simulation or numerical simulation and experimental research method of combining the valve, did not study for flow resistance characteristics and internal flow characteristics of the gate valve at different speeds.

The present study introduces CFD simulation methods to investigate flow resistance characteristics and internal flow characteristics of the gate valve at different speeds. The universal CFD package, Fluent (FLUENT 14.5, 2014), and Gambit grid generator has been applied to perform all the 3D quasi-static numerical computations. It is simplified that the flow channels structure of the gate valve. Flow resistance and the internal flow characteristics of gate valve are simulated by solving the Reynolds Stress Model (RSM) turbulence model at different positions of the gate and under the variable inlet velocity. At the same time, we conducted the flow resistance characteristic experiment of gate valve. The results provide effective guidance for the design of an adjustable gate valve.

2. Physical model and experiment

This article mainly studies for the pipeline structure of the gate valve a rectangular structure. Gate valve working principle is through the stem to drive disc gate valves opening and closing. In order to study the flow resistance characteristics of multi conditions and within the rectangular pipe valve flow characteristics. Fig. 1 shows an Auto CAD generated two dimensional model of the valve used. Fig. 2 shows a SolidWorks generated three dimensional flow channel model of the valve used. And according to the flow diagram, we processed the flow channel to conduct the flow resistance characteristics experiment of gate valve. Fig. 3 is the experimental field photo and circuit scheme. Pipe structure of gate valve is consisting of the upstream flow, groove, downstream of three parts. Fig. 4 is the two dimensional flow channel model of the valve. Table 2 is the value of pipeline structure. In order to fully develop the turbulent flow, the upstream pipe section of the valve is longer than 5 times of the hydraulic diameter of the pipe, the downstream pipe section of the valve is longer than 10 times the diameter of the pipe (Xue et al., 2009; Chern et al., 2013). In this paper, the hydraulic diameter of the pipeline is 26.7 mm. Therefore, there is 805 mm in length in the upstream before the valve center and 725 mm in length in the downstream after the valve center. The choice of upstream and downstream length is to achieve the fully developed turbulence flow. Opening of the gate valve is fully opened 40 mm. In order to study the variety of the opening of the change, we studied flow resistance characteristics and the internal flow characteristics of the gate valve that is the relative opening degree of 1/8, 2/8, 3/8, 4/8, 5/8, 6/8, 7/8, 1 under different inlet conditions (during the Table 1). On the basis of obtaining the physical model, the external characteristic experiment of the gate valve is carried out, and the flow coefficient and resistance coefficient are obtained. In order to ensure the repeatability and accuracy of the experiment, each relative opening is carried out 20 experiments, and the data are processed by means of the mean value.

3. Numerical model

In order to study the gate location of gate valve flow resistance properties and the influence of the internal features under the variable condition. CFD simulations using commercial software FLUENT are performed, which can obtain the mass flow rate