Modeling the working media flow in shut-off valves with displacement of the regulating body perpendicular to the flow axis

V V Soloveva¹, A S Pugachuk¹,² and A V Chernyschev¹

¹ Bauman Moscow State Technical University, (Russia, Moscow, 5/1 2nd Baumanskaya Street)
² Joint Institute for High Temperatures of the Russian Academy of Sciences (Russia, Moscow, 13/2 Izhorskaya Street)

*E-mail: solovevavica@gmail.com

Abstract. The mathematical model of the working fluid movement in the flow section of the wedge type two-disc parallel gate valve is developed. The simulation of the fluid flow through the valve cavity is carried out, as a result the flow parameters are obtained in a wide range of Reynolds numbers at the entrance to the calculated area. The dependence of the hydraulic resistance as a function of the Reynolds number for liquid and gas flow is calculated. The various positions of the shut-off body in the flow part of the valve are considered and the area of reduced pressures in which the effect of cavitation may occur during fluid flow is estimated.

1. Introduction

During the operation of pipeline systems there is a need to interrupt the supply of the working fluid in case of accidents and to connect new branches to the main line for repair and maintenance of pipes, equipment, and other purposes – this function is implemented by shut-off valves. One of the widespread elements of the flow control is gate valves. They are actively used in water supply systems, as well as in the transportation of hydrocarbons and aggressive media that do not destroy the seal and the valve body. This type of shut-off valve has several advantages: a simple design, a compact size, the corrosion resistance and a long service life. The main elements of the gate valves are the body, the locking element, the spindle and the flywheel or the electric drive.

One gate valve could be used with different working fluids. It can be either gas or liquid. The effect of the working fluid on the flow parameters (e.g. pressure drops and velocity) in the gate valve can be determined either using experiments or mathematical modeling of the process. There are several research papers on this topic. In [1] authors simulate the water flow through a two-disc parallel gate valve of the wedge type and determine the effect of the Reynolds number on the pressure drop at different degrees of opening of the gate valve. Specific to the gate valves, lots of works have been focused on various issues, e.g., the flow resistance and the temperature distributions. The authors of [2] show the results of the loss coefficient calculation in three different gate valve models in two open positions. The research was carried out with the ice suspension with a mass fraction of ice particles of 30%, 25%, 20%, 15%, 10% and 5%. A large mass fraction of ice in the mixture was found to increase the value of the loss coefficient. In the article [3] numerical studies of the static characteristics of the rotary valve at different angles of its deviation were carried out by setting different values of the
pressure drop on the valve effecting the determined mass flow rate and the valve axes. The analysis of
the flow structure is carried out: zones of acceleration and separation of the flow, vortices, etc. The
increase in the hydraulic resistance of the valve with a decrease in the angle of inclination of the valve
and the increase in the mass flow rate, as well as an increase in the torque with a decrease in the angle
of deflection of the valve and an increase in the mass flow rate, are noted. Previously, the authors also
carried out numerical studies of processes in the elements of the pneumatic systems and the
determination of hydraulic resistances of various types of shut-off and control valves [4-6].

2. Mathematical modeling
The purpose of the research is to determine the hydraulic characteristics of the two-disc gate valve and
to study the effect of the working fluid on the flow parameters.

For creating the mathematical model of the fluid flow in the gate valve the following assumptions
are made:
- Each working fluid is considered as an ideal gas;
- The working fluid flow is isothermal;
- There is no roughness of the internal surface of the gate valve;
- The working fluid is homogeneous;
- The action of mass forces is not considered;
- There is no gap between the end surfaces of the disks and the walls of the flow part.

The object of the study is the flow part of the two-disc gate valve. The calculation scheme is shown
below (fig. 1).

![Figure 1. The calculation scheme of the flow part of the two-disc gate valve.](image)

Symbols shown in the figure:
- \( D = 100 \text{ mm} \) – diameter of the flow part;
- \( b = 50 \text{ mm} \) – thickness of the regulatory body;
- \( h = \frac{5}{8} \cdot D = 62.5 \text{ mm} \) – lifting height of the regulatory body;
- \( L_1 = 2 \cdot D = 200 \text{ mm} \) – length of the flow part in front of the gate valve;
- \( L_2 = 10 \cdot D = 1000 \text{ mm} \) – length of the flow part behind the gate valve.

The mathematical model of the fluid flow in the gate valve is based on the following equations:
\[
\begin{align*}
&\frac{\partial v}{\partial t} + v \cdot \nabla v = F - \frac{1}{\rho} \nabla p + \eta \nabla \Delta v \\
&\nabla (\rho v) = 0 \\
&du + d(pV) + \frac{d\nu^2}{2} = 0 \\
&dV = \left( \frac{\partial V}{\partial T} \right)_\rho dT + \left( \frac{\partial V}{\partial p} \right)_T dp,
\end{align*}
\]

where \( v \) is the velocity of the fluid flow, \( t \) is the time, \( F \) is the external specific force, \( p \) is the pressure of the fluid flow, \( \eta \) is the kinematic coefficient of the fluid flow viscosity, \( \mu \) is the dynamic coefficient of the fluid flow viscosity, \( \rho \) is the fluid flow density; \( u \) is the fluid flow internal energy, and \( V \) is the fluid flow volume.

The finite element method is used to solve the equations of the mathematical model. The calculation grid for the used geometric model is constructed. The grid consists of \( 15 \times 10^5 \) elementary cells with the tetrahedral shape. The radius of the described sphere of the cell is 5 mm. A boundary layer is specified with a maximum thickness of 2 mm. Parameters selected in the flow simulation are outlet pressure - 1 (atm abs), inlet velocity - \( v \) (m/s), inlet temperature - 298 K for the water and air flow and 373 K for the vapor flow. It is acceptable for describing the wall boundary layer by the \( k-\varepsilon \) turbulence model. The simulation of the fluid flow in the valve cavity is carried out, therefore the flow parameters are obtained in a wide range of Reynolds numbers at the entrance to the calculated area (fig.1). The values of the pressure drop of the fluid flow in the valve are obtained, therefore with Darcy-Weisbach equation (5) the dependence of the hydraulic resistance on the Reynolds number for liquid and gaseous working media is calculated (fig. 2, 3).

\[
\xi = \frac{2\Delta p}{\rho v^2}, \quad (2)
\]

\[ \text{Figure 2. Pressure drop of the fluid flow in the gate valve as a function of the Reynolds number (for open position at 5/8).} \]
Cavitation is a negative phenomenon in hydraulics that leads to destruction of equipment. The developed mathematical model is used to estimate zones with possible occurrence of cavitation. For this purpose, the calculation of water flow with different velocity modes in the position of gate valve opening at 3/8 is carried out. The distribution area of the absolute pressure for the water flow with the Reynolds number of 500000 (average flow velocity of 5 m/s) is obtained. The analysis of this distribution shows that the low-pressure area is located near the gate valve discs. In the low pressure area, cavitation occurs when the flow rate increases. At higher Reynolds numbers, the pressure reaches the value of the saturated water vapor pressure, and cavitation occurs in this region of low-pressure (fig. 4).

The results of the calculation of pressure losses during the flow of various working media through the valve in the open position at 3/8 are shown in Figure 5.
3. Results
At an increase in the Reynolds number of the water flow, the change in the pressure drop in the valve is close to a polynomial increase. For the air and vapor flow, a smoother increase in the pressure drop is observed. The loss coefficient has maximum values for the laminar flow and decreases sharply with increasing Reynolds number for all the studied working fluids. The pressure loss for water is 3-3.7 times higher than for air, and the pressure loss for air at normal temperature is about 10% higher compared to the pressure loss for vapor at 100 °C.

Conclusions
As a result of the analysis of the data obtained by numerical modeling, it has been determined that with an increase in the Reynolds number of the water flow, the form of change in the pressure drop in the flow part of the gate valve is close to polynomial. The coefficient of hydraulic resistance has maximum values for laminar flow and decreases sharply with an increase in the Reynolds number of the flow for the studied working fluids. The pressure loss for the liquid flow is 3-4 times higher than for the gaseous ones. The pressure loss for the air is about 10% higher than for the vapor. The areas of possible occurrence of cavitation in the wedge type two-disc parallel gate valve have been determined.

References
[1] Hui Wu, Jun-ye Li, Zhi-xin Gao Flow 2019 Flow characteristics and stress analysis of a parallel gate valve MDPI 7 803
[2] Solek M, Mika L 2019 IOP Conf. Series: Earth and Environmental Science 214
[3] Boldyrev A, Karelin D, Mulyukin V 2016 IOP Conf. Series: Materials Science and Engineering 158
[4] Pugachuk A, Vorozheeva O, Chernyschev A 2019 AIP Conference Proceedings 2141 030056
[5] Pugachuk A, Fominyh N, Gavrilova Y 2020 Journal of Physics: Conference Series 1675(1) 012016
[6] Bolshakov N, Pugachuk A, Zelenov M, Chernyschev A 2020 Pipeline fittings and equipment 5(110) 20–1