CFD Analysis of the Hydraulic Characteristics of a Gate Valve

Gong Hee Lee *
Reactor System Department, Korea Institute of Nuclear Safety, Daejeon, 34142, Korea

Corresponding author: ghlee@kins.re.kr

***Keywords :** computational fluid dynamics, gate valve, hydraulic performance, in-service testing, loss coefficient

1. Introduction

Nuclear power plant (NPP) operators in Korea perform in-service testing (IST) to verify the operational readiness of safety-related valves and to monitor performance degradation during plant operation. Gate valves are among the valve types subject to IST and are commonly installed in safety injection and shutdown cooling systems for flow-path opening and closure and for system isolation.

The flow field through a partially opened gate valve is highly complex and can be affected by operating conditions, valve geometry, and the valve disc opening. Accurate characterization of key valve performance parameters, including pressure loss, flow coefficient, seat leakage, stem thrust, and loss coefficient, is essential for evaluating valve operability and ensuring that system design requirements are satisfied.

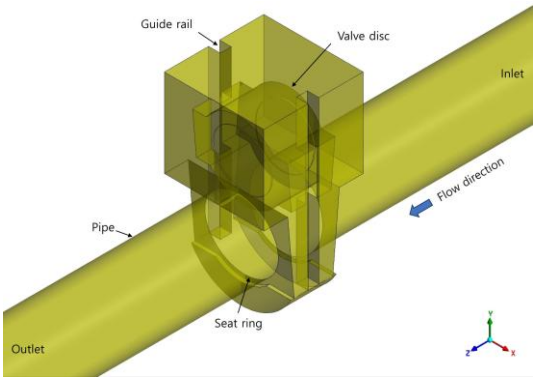
In this study, computational fluid dynamics (CFD) simulations were performed using ANSYS CFX R21.2 to investigate the hydraulic characteristics and complex flow structures around a gate valve as a function of the normalized disc stroke.

2. Analysis Model

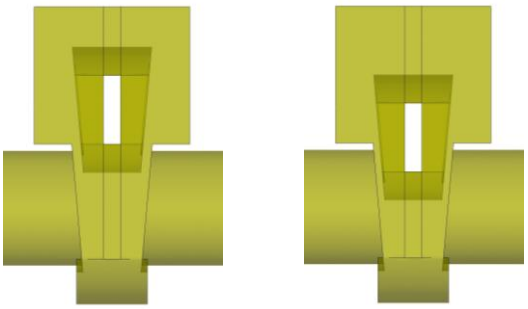
Fig. 1 shows a schematic diagram of the present analysis model. The valve geometry corresponds to the model used in the EPRI Motor-Operated Valve (MOV) Performance Prediction Program [1]. The geometric specifications of the analysis model are summarized in Table I. Most geometric parameters were normalized by the mean diameter of the seat ring, D. The downstream pipe length was set to 11D to ensure that reverse flow did not reach the outlet boundary. The thermophysical properties of water at 25°C were used in the CFD calculations.

Table I: Geometric specification of the analysis model.

Parameters	Value	Parameters	Value
Disc inner diameter	0.80	Guide span	1.10
Disc outer diameter	1.10	Disc/Body gap	0.03
Disc thickness	0.50	Overlay thickness	0.015
Seat width	0.06	Wedge angle	5°
Under disc depth	0.25	Tip angle	0°



(a) Full domain



(b) St/D = 0.92 (c) St/D = 0.67

Fig. 1. Schematic diagram of an analysis model.

3. Numerical Modeling

In this study, the flow through the gate valve was modeled as steady, incompressible, turbulent, and single-phase. The numerical schemes and boundary conditions used in the CFD analysis are summarized in Table II.

An unstructured hybrid mesh composed of tetrahedral, wedge, and pyramid elements was generated using ICEM CFD. The full three-dimensional geometry of the gate valve was modeled to capture potential flow asymmetry as the fluid passed through the partially opened valve disc. Based on the mesh sensitivity study, approximately 2.7×10^6 elements were used in the calculations as a compromise between computational accuracy and cost. Overall, the simulation results showed minor variations with mesh size; the loss coefficient defined in Eq. (1) varied by about 5%. Local mesh refinement was applied near the valve disc and pipe wall to adequately resolve the complex flow structures, including flow separation and reverse flow. This grid pattern is generally recommended for the flow

simulation around a gate valve. The numerical framework used in the present study was previously validated for a butterfly-valve flow problem [2].

Table II: Numerical methods and boundary conditions for flow analysis

Numerical methods		**Note**
Discretization accuracy for convection term	Momentum eqn.	High resolution
	Turbulence eqn.	High resolution
Turbulence model		SST k-ω
Velocity-pressure coupling		Rhie Chow (4th order)
Near wall treatment		Automatic wall treatment
Convergence criterion		$< 3\times10^{-4}$
Boundary conditions		**Note**
Inlet	Velocity (m/s)	1.0
	Turbulence	medium intensity (5%) eddy viscosity ratio (10)
Outlet		average static pressure (0 Pa)
Wall		No-slip & smooth wall

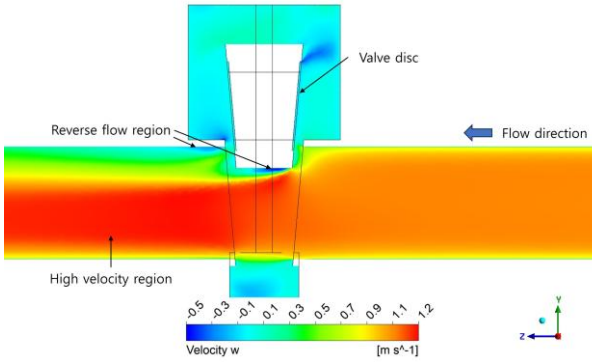
4. Results and Discussion

Fig. 2 presents the calculated axial-velocity distributions at three normalized disc stroke positions (St/D=0.92, 0.86, and 0.67). As the upstream flow passed through the restricted flow passage beneath the valve disc, the fluid was accelerated, resulting in the formation of a high-velocity jet. Downstream and lower part of the valve disc, flow separation produced a distinct region of reverse axial flow. The disc stroke strongly affected both the jet characteristics and the size/strength of downstream reverse-flow region. For example, as St/D decreased and the valve disc moved downward, the available flow area decreased, resulting in an increase in the maximum axial velocity. At the same time, the reverse flow region downstream of the disc became more pronounced and expanded further downstream.

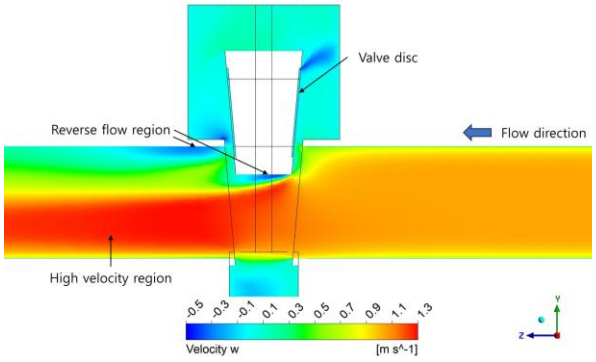
Fig. 3 shows the loss coefficient, K_L, as a function of the normalized disc stroke. The loss coefficient is defined as

$$K_L = \frac{2\Delta P}{\rho V^2} = \frac{2(P_u - P_d)}{\rho V^2} \quad (1)$$

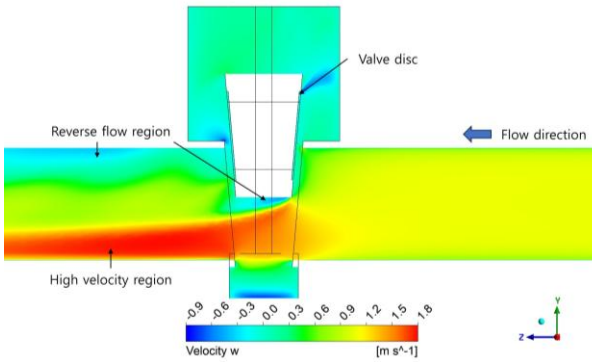
where ρ is the fluid density, V is the mean flow velocity in the pipe, P_u and P_d represent the upstream and downstream pressures measured at locations 2D upstream and 6D downstream of the valve disc, respectively. As the valve disc moved downward and St/D decreased, the loss coefficient gradually increased. This trend can be attributed to the progressively restricted flow passage, which causes substantial fluid acceleration through the valve opening and intensifies high-velocity jet formation, flow separation, reverse flow, and turbulent mixing downstream of the disc. These irreversible flow processes dissipate mechanical energy and consequently increase the pressure loss across the valve.



(a) St/D = 0.92



(b) St/D = 0.86



(c) St/D = 0.67

Fig. 2. Calculated axial velocity distributions at the pipe center plane (X=0)

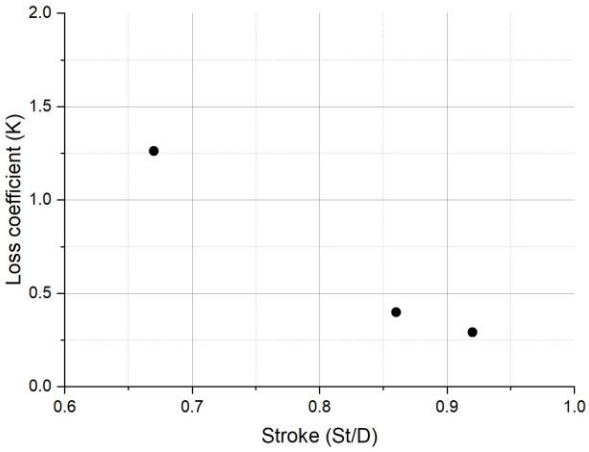


Fig. 3. Calculated loss coefficient, K_L, as a function of the normalized disc stroke, St/D.

5. Conclusions

In this study, steady-state CFD simulations were performed using ANSYS CFX R21.2 to investigate the effect of normalized disc stroke on the hydraulic characteristics of a gate valve. As St/D decreased, the restricted flow passage beneath the valve disc resulted in stronger flow acceleration and the formation of a higher-velocity jet. The downstream reverse-flow region also became more pronounced as the valve disc moved toward the closed position. In addition, the calculated loss coefficient increased with decreasing St/D.

Overall, the CFD results reproduced the characteristic flow behavior expected for a partially closed gate valve, including jet formation, flow separation, reverse flow, and increased hydraulic loss. The present results provide a basis for further investigation of the disc-stroke-dependent hydraulic performance of gate valves used in nuclear power plant applications.

DISCLAIMER

The opinions expressed in this paper are those of the author and not necessarily those of the Korea Institute of Nuclear Safety (KINS). Any information presented here should not be interpreted as official KINS policy or guidance.

ACKNOWLEDGEMENT

This work was supported by the Korea Institute of Nuclear Safety (A3FD26030, A6FD26021 & S2661124).

REFERENCES

[1] Electric Power Research Institute, EPRI MOV Performance Prediction Program, 1993, TR-103254.
[2] G.H. Lee, J.H. Bae, Numerical Study on the Effect of an Upstream Flow Disturbance on the Butterfly Valve’s Dynamic Performance, NUTHOS 12, Sep. 5-10, 2022, Hsinchu, Taiwan.