# CFD Simulation of the Hydraulic Characteristics inside a butterfly valve using ANSYS CFX

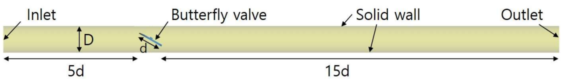
Gong Hee Lee [a*], June Ho Bae [a]
[a]*Regulatory Assessment Department, Korea Institute of Nuclear Safety, Daejeon, 34142, Korea*
[*]*Corresponding author: ghlee@kins.re.kr*

## 1. Introduction

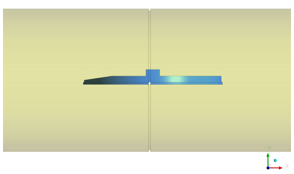
Domestic nuclear power plant (NPP) operators have conducted in-service testing (IST) to verify the safety functions of safety-related valves and to monitor the degree of vulnerability over time during reactor operation. A butterfly valve, one of the representative IST-related valve types, is most commonly used in low pressure and low temperature water systems (for example, the primary component cooling water system and in containment purge and venting systems) for various purposes, such as opening/closing the pipeline and flow rate control. It is well-known that the flow field around a butterfly valve was extremely complex and might be influence by the operating conditions, the valve disc configuration, and the valve disk angle. On the other hand, optimizing butterfly valve's performance factors, such as pressure loss, hydrodynamic torque, flow coefficient, loss coefficient, and torque coefficient, is necessary for fluid system designers to account for system requirements to properly operate the valve and prevent severe damage from occurring. In this study, the numerical simulation around a butterfly valve was conducted with commercial CFD software, ANSYS CFX R19.1 to investigate the complex flow pattern depending on the valve disk angle.

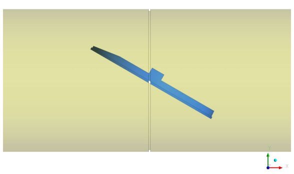
## 2. Analysis Model

Fig. 1 shows a schematic diagram of the present analysis model. The corresponding model was used in the Tennessee Valley Authority's Great Falls Hydro Plant [1]. Geometrical specification for the analysis model was summarized in Table I. For reference, the downstream length (15d) of the present analysis model was sufficiently extended to guarantee no reverse flow at the outlet boundary. As shown in Fig. 1(b), a disk angle of zero (α=0°) indicates the valve to be fully open. The water properties at 25°C were applied.




(a) Full domain





(b) α=0° (c) α=30°

Fig. 1. Schematic diagram of an analysis model.

Table I: Geometrical specification of an analysis model.

| Parameters | Unit | Magnitudes |
|---|---|---|
| Valve disk diameter (d) | m | 3.53 |
| Pipe diameter at inlet/outlet (D) | m | 3.66 |
| Upstream length | m | 17.65 (5d) |
| Downstream length | m | 52.95 (15d) |
| Valve disk angle (α) | deg. | 0, 10, 20, 30, 40, 50, 70 |

## 3. Numerical Modeling

### *3.1 Numerical Method*

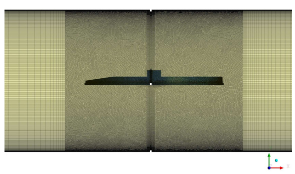
It was assumed that the flow around a butterfly valve was steady, incompressible, turbulent, and single-phase flow. High resolution scheme with the quasi-second order accuracy was used for the convection terms of momentum and turbulence equations. The solution was considered to be 'converged' when the residuals (root mean square) of variables were below $10^{-3}$ and global imbalances for the governing equations were less than 1%.

### *3.2 Turbulence Model*

The standard k-ε model among the turbulence models based on the Reynolds-averaged Navier–Stokes (RANS) equation available in ANSYS CFX was used to simulate the complex flow around a butterfly valve. This model is numerically stable and has a well-established flow regime with the good predictive performance [2].

### *3.3 Grid System and Boundary Conditions*

In this study, an unstructured hybrid (consisting of tetrahedral, wedges, and hexahedral type) grid system generated by ANSYS Advance Meshing was used (see Fig. 2). The full geometry of a butterfly valve was considered in case the flow could not maintain the symmetrical pattern while passing through the valve disk. Based on the grid sensitivity study [3], the total nodes number of about $6.5\times10^6$ was used in the calculation. To properly predict the complex flow (for example, flow separation and recirculation, etc.) around the valve disk



(a) α=0° (b) α=30°

Fig. 2. Grid system.

Table II: Grid information (α=0°).

| Items | Original | 1d ext. | 2d ext. |
|---|---|---|---|
| Total number of nodes | $6.5\times10^6$ | $7.3\times10^6$ | $8.2\times10^6$ |
| Total number of elements | $2.0\times10^7$ | $2.1\times10^7$ | $2.3\times10^7$ |

and its effect on the hydrodynamic torque, dense grid distribution near the valve disk and pipe wall were used. This grid pattern is generally recommended for the flow simulation around a butterfly valve. In this study, to examine the need to extend the dense grid distribution region around the valve disk, additional calculations were conducted for two different grids shown in Table II. '1d ext.' and '2d ext.' represent the extension of the original dense grid distribution region by one and two times the diameter (d) of the valve disk toward the downstream direction. The predicted discharge coefficient $C_d$ (defined as Equation (1)) for the three different grids was nearly the same magnitude, about 1.64. Therefore, the original dense grid distribution region near the valve disk may be considered to be reasonable.

Inlet condition was the specified constant volumetric flow rate in the range of between $Q_{in}$ = 1.4 (α=70°) and 56.6 $m^3/s$ (α=0°); depending on the valve disk angles, turbulence intensity of 5%, and eddy viscosity ratio of 10. The average static pressure of 0 Pa was used as the outlet condition. The solid walls were assumed to be smooth with zero surface roughness and a no-slip condition was applied there. To model the flow in the near-wall region, the scalable wall function was applied.

## 4. Results and Discussion

Fig. 3 shows the comparison of discharge coefficient between the experiment and simulation. The discharge coefficient $C_d$ can be defined as follows:

$$C_d = \frac{Q}{A\sqrt{2g(H_u - H_d)}} \quad (1)$$

where Q is the volumetric flow rate, $A$ is pipe cross-sectional area, $g$ is the gravitational acceleration, $H_u$ and $H_d$ is upstream (2D from the valve disk) and downstream (6D from the valve disk) head. As the valve disk angles increased, the discharge coefficient gradually decreased. The predicted $C_d$ showed good agreement with the measured data.

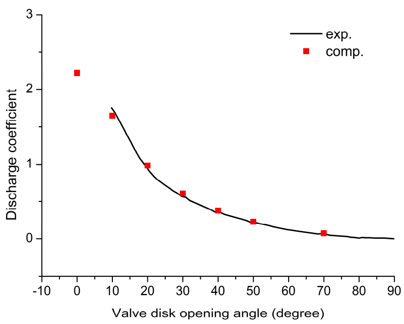
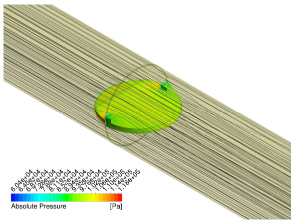
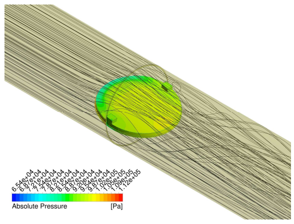



Fig. 3. Comparison of discharge coefficient between the experiment and simulation.

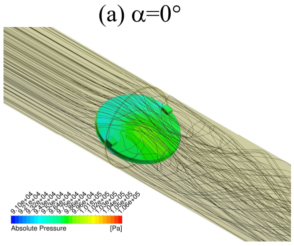
Fig. 4 shows the isometric view of absolute pressure distribution on the valve disk and streamline depending on the valve disk angles. As the valve disk angles increased, the difference between the minimum and maximum pressure on the valve disk became smaller. Also, due to the flow separation and swirl flow, the flow patterns downstream of the valve disk became more complex.



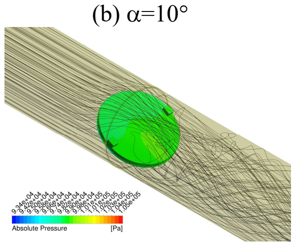
(a) α=0°



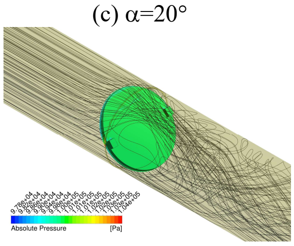
(b) α=10°



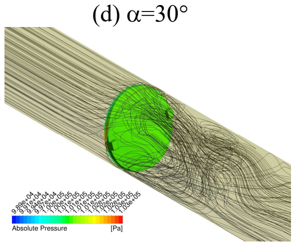
(c) α=20°



(d) α=30°



(e) α=50°



(f) α=70°

Fig. 4. Isometric view of absolute pressure distribution on the valve disk and streamline.

## 5. Conclusions

In this study, the numerical simulation around a butterfly valve was conducted with ANSYS CFX R19.1 and the complex flow pattern depending on the valve disk angle was examined in detail. The predicted $C_d$ showed good agreement with the measured data. Therefore, the present numerical modeling may be considered to be valid. On the other hand, the comparison of the simulation results between CFX and FLUENT [3] will be shown in a separate paper.

## 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.

## ACKNOWLEGEMENT

This work was supported by the Nuclear Safety Research Program through the Korea Foundation Of

Nuclear Safety (KOFONS) using the financial resource granted by the Nuclear Safety and Security Commission (NSSC) of the Republic of Korea (No. 1805007).

## REFERENCES

[1] F. Lin, G. A. Schohl, CFD Prediction and Validation of Butterfly Valve Hydrodynamic Forces, World Water & Environmental Resources Congress 2004, 2004, Salt Lake City, Utah, USA.
[2] Y. Duan, M. D. Eaton, M. J. Bluck, C. Jackson, Assessments of Different Turbulence Models in Predicting the Performance of a Butterfly Valve, ICONE26-82376, 2018, London, England.
[3] J. H. Bae, G. H. Lee, Comparative Study for the Flow Analysis Results for Butterfly Valve using Different Turbulence Models, Journal of Computational Fluids Engineering, Vol.26, 2021 (under publication).