Comparison of ω-based Turbulence Models for the Hydraulic Characteristics of a Butterfly Valve and 4-Hole Orifice Combination; Multiple Bends in the Different Plane Case

Gong Hee Lee [a*], Seon Ho Choi [a]
[a]*Reactor System Department, Korea Institute of Nuclear Safety, Daejeon, 34142, Korea*
[*]*Corresponding author: ghlee@kins.re.kr*

***Keywords :** butterfly valve, computational fluid dynamics, in-service testing, multi-hole orifice, turbulence model

1. Introduction

As the operational period of the domestic nuclear power plants increases, various types of issues arise in the field of in-service testing (IST), which verifies the current performance of safety-related components and monitors their performance changes [1]. For instance, there was a case where the butterfly valve was damaged because the straight length between the multi-hole orifice installed at the pump discharge of the essential service water system (a representative IST-related system) and the butterfly valve was very short [1].

The competitiveness of computational fluid dynamics (CFD) is growing steadily due to rapid developments in computer hardware technology [2], but the computing capacity is still a limiting factor for CFD calculations to produce completely accurate results for the prediction of the complex flow in a butterfly valve and multi-hole orifice combination. Therefore, turbulence model is required to bridge the gap between the real flow and the statistically averaged equations [2]. Turbulence model is one of the main causes of model error [3].

Therefore, in this study, the hydraulic characteristics of a butterfly valve and 4-hole orifice combination; multiple bends in the different plane case was examined using the ω-based turbulence models (SST k-ω, baseline Reynolds stress model (RSM)) available in ANSYS CFX R19.1 and the predicted results were compared.

2. Analysis Model

Fig. 1 shows a schematic diagram of the present analysis model. The model consists of the multiple 90° bends in the different plane, butterfly valve and 4-hole orifice. Orifice hole patterns were similar to those of the Moosa and Hekmat [4]. A valve disc angle of zero (α=0°) indicates the valve to be fully open [5]. Geometrical information of an analysis model was explained in Table I. The water properties at 25 °C were applied [5].

3. Numerical Modeling

The flow was assumed to be steady, incompressible, turbulent, and single-phase. The numerical methods and boundary conditions used in this study are summarized in Table II.

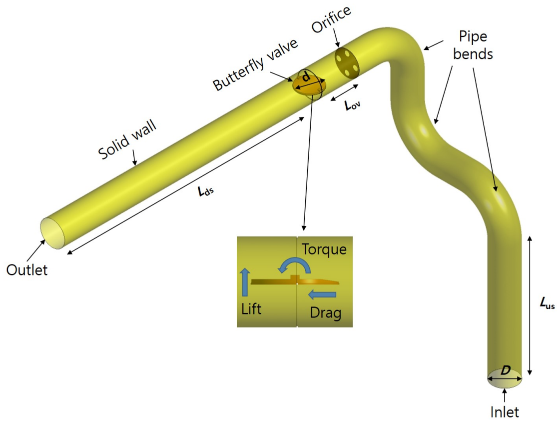


Fig. 1. Schematic diagram of an analysis model.

Table I: Geometrical specification for an analysis model

Parameters	Unit	Magnitudes
Pipe diameter (*D*)	m	3.66
Orifice hole diameter (d_0)	m	0.74
Valve disc diameter (*d*)	m	3.53
Valve disc angle (α)	Deg.	0, 10, 20, 30, 40, 50, 70
Radius of curvature	m	1.5*D*
Upstream straight length (L_{us})	m	5*D*
Downstream straight length (L_{ds})	m	10*D*
Straight length between an orifice and a butterfly valve (L_{ov})	m	0.1*D*, 1.1*D*, 2.1*D*, 3.1*D*

Table II: Numerical methods and boundary conditions.

Numerical methods		Note
Discretization accuracy for convection term	Momentum eqn.	High resolution
	Turbulence eqn.	High resolution
Turbulence model		SST k-ω, Baseline RSM
Near wall treatment		Automatic wall treatment
Convergence criteria		$< 4\times10^{-4}$
Boundary conditions		**Note**
Inlet	Flow rate	1.4～56.6 m^3/s
	Turbulence	Intensity (5%) & eddy viscosity ratio (10)
Outlet		Average static pressure (0 Pa)
Wall		no-slip & smooth wall

An unstructured hybrid (consisting of tetrahedral, hexahedral, wedge, and pyramid type) grid system generated by ICEM CFD was used. The entire geometry of a butterfly valve was considered in case the flow could not maintain the symmetrical pattern [5]. Based on the grid sensitivity study, the total elements number between about 1.09×10^7 and 1.53×10^7 depending on the straight length between an orifice and a butterfly valve (L_{ov}) was finally used in the calculation. To properly predict the complex flow (for example, flow separation and recirculation, etc.) around the valve disc/orifice and its effect on the hydraulic torque, dense grid distribution near the valve disc, orifice and pipe wall were used [5]. This grid pattern is generally recommended for the flow simulation around a butterfly valve and an orifice [5].

4. Validation of the Numerical Modeling

To validate whether the numerical modeling (refer to section 3) available in ANSYS CFX R19.1, predicts reliably and accurately the complex flow through the multi-hole orifice, the CFD simulation was performed on the 4-hole orifice installed in the straight pipe and then the predicted pressure drop between upstream (1*D*) and downstream (0.5*D*) sections of the 4-hole orifice depending on the inlet flow velocity was compared with the measured data [6]. The geometrical specification of the 4-hole orifice used for the validation is different from that in Table I. Detailed information for the experimental facility can be found in the reference [6].

Fig. 2 shows the comparison of the measured and predicted differential pressure versus inlet velocity. As the inlet velocity increased, the magnitude of the differential pressure also increased, and the predicted differential pressure with baseline RSM was consistent within a maximum deviation of 0.93 % compared to the measured data, which was smaller than 1.25 % with SST k-ω model.

On the other hand, the validation result for the same butterfly valve installed in the straight pipe can be found in the author's separate paper [5].

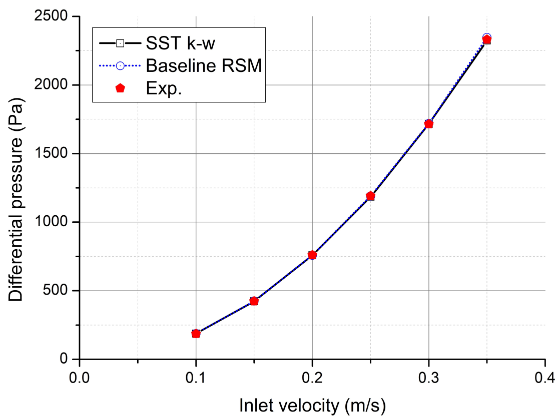
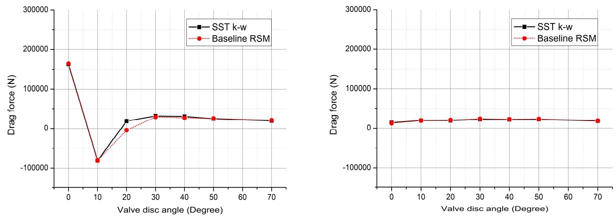


Fig. 2. Comparison of the measured and predicted differential pressure versus inlet velocity depending on the turbulence model.

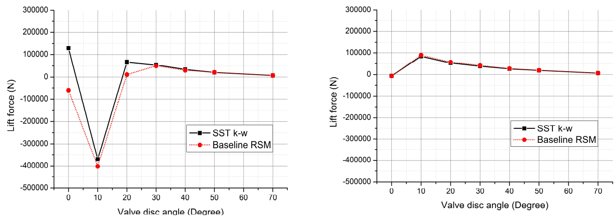
5. Results and Discussion

Fig. 3 shows the magnitude of the hydraulic forces and torque acting on the valve disc depending on the L_{ov}.

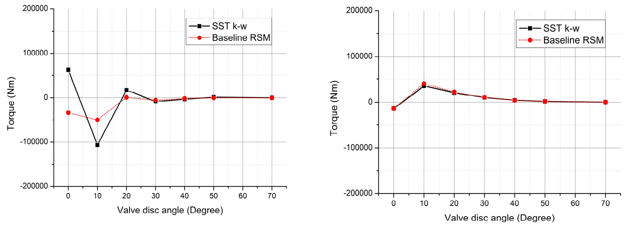
For L_{ov}=0.1*D*, the predicted hydraulic forces and torque acting on the valve disc showed the large variation (magnitude itself) and difference (depending on turbulence model) in the relatively high inlet flow rate (α=0°~20°). The above-mentioned CFD results were not found for L_{ov}=3.1*D*, because the effect of an upstream flow disturbance (caused by 4-hole orifice) became weak.



(a) Drag force



(b) Lift force

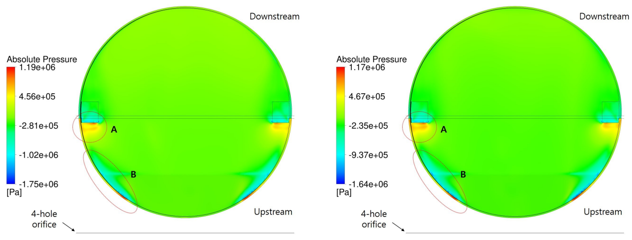


(c) Torque

Fig. 3. Hydraulic forces and torque depending on L_{ov}; left L_{ov}=0.1*D*, right; L_{ov}=3.1*D*.

For a butterfly valve, high torque fluctuation of the valve disc may result in the bearing wear and further the failure of torque train [1]. Therefore, it is necessary to have the capability to assess the proper L_{ov} for obtaining the integrity of the butterfly valve. To achieve this goal, it is required to additionally provide detailed measurement data (velocity, pressure, valve flow coefficient, valve loss coefficient etc.) to validate the CFD software for complex flow patterns that may occur in a butterfly valve and multi-hole orifice combination.

Fig. 4 shows the top view of absolute pressure distribution on the valve disc (L_{ov}=0.1*D* case). The pattern of the jet flow passing through 4-hole orifice and its collision with the valve disc was predicted differently depending on the turbulence model. As a result, the predicted absolute pressure distribution on the valve disc was different in the local region (especially as denoted A, B in Fig. 4).



(a) SST k-ω (b) Baseline RSM

Fig. 4. Top view of absolute pressure distribution on the valve disc (L_{ov}=0.1D).

6. Conclusions

In this study, the hydraulic characteristics of a butterfly valve and 4-hole orifice combination; multiple bends in the different plane case was investigated using the ω-based turbulence models (SST k-ω, baseline Reynolds stress model) available in ANSYS CFX R19.1 and the predicted results were compared. For L_{ov}=0.1D, the predicted hydraulic forces and torque acting on the valve disc showed the large variation (magnitude itself) and difference (depending on turbulence model) in the relatively high inlet flow rate (α=0°~20°). On the other hand, the above-mentioned CFD results was not found for L_{ov}=3.1D, because the effect of an upstream flow disturbance (caused by 4-hole orifice) became weak. The results obtained in this study may be applicable to the different multi-hole orifice & bends configuration and as a result, can assess the proper L_{ov} for obtaining the integrity of the butterfly valve.

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.

ACKNOWLEDGMENTS

This work was supported by the Korea Institute of Nuclear Safety (A3FD24030, A6FD24021 & S2461143).

REFERENCES

[1] G. H. Lee, CFD Study for the Effect of Upstream Orifice Flow Disturbance on the Hydraulic Characteristic of the Butterfly Valve; Multiple Bends in the Different Plane Case, ICTAM2024, August 27, 2024, Daegu, Korea.
[2] G. H. Lee, J. H. Bae, Prediction of Cavitating Flow inside a Sharp-edged Orifice using Different RANS Turbulence Models, AICFM15, September 25-27, 2019, Busan, Korea.
[3] G. H. Lee, Suggestions for the Proper Treatment of Buoyancy Force Effect on Single Phase Thermal Stratification Phenomenon, Heat Transfer Engineering, Vol.43, p.283, 2022.
[4] M. Moosa, M. H. Hekmat, Numerical Investigation of Turbulence Characteristics and Upstream Disturbance of Flow Through Standard and Multi-hole Orifice Flowmeters, Flow Measurement and Instrumentation, Vol.65, p.203, 2019.
[5] G. H. Lee, J. H. Bae, Numerical Study on the Effect of an Upstream Flow Disturbance on the Butterfly Valve's Dynamic Performance. NUTHOS 13, September 5-10, 2022, Hsinchu, Taiwan.
[6] A. Golijanek-Jędrzejczyk, A. Mrowiec, S. Kleszcz, R. Hanus, M. Zych, M. Jaszczur, A Numerical and Experimental Analysis of Multi-hole Orifice in Turbulent Flow, Measurement, Vol.193, 110910, p.1, 2022.