NUMERICAL ANALYSIS OF BUTTERFLY VALVE-PREDICTION OF FLOW COEFFICIENT AND HYDRODYNAMIC TORQUE COEFFICIENT

Xue guan Song¹, Young Chul Park² ¹Graduate student, songxguan@yahoo.com.cn ²Professior, parkyc67@dau.ac.kr CAE Lab, Department of Mechanical Engineering, Dong-A University, 840 Hadan-dong, Saha-gu, Busan 604-714 TEL: (82)051-200-6991; FAX: (82)051-200-7652 South Korea

ABSTRACT

Butterfly valves are commonly used as control equipments in applications where the pressure drops required of the valves are relatively low. As shutoff valve (on/off service) or throttling valves (for flow or pressure control), the higher performance and the better precision of butterfly valves are required. Thus it's more and more essential to know the flow characteristic around the valve. Due to the fast progress of the flow visualization and numerical technique, it becomes possible to observe the flows around a valve and to estimate the performance of a valve. In this paper, three-dimensional numerical simulations by commercial code CFX were conducted to observe the flow patterns and to measure valve flow coefficient and hydrodynamic torque coefficient when butterfly valve with various opening degrees and uniform incoming velocity were used in a piping system. By contrast, a group of experimental data is used to compare with the data obtained by CFX simulation to investigate the validity of numerical method. Researching these results did not gave only access to understand the process of the valve flows at different valve opening degrees, but also was made to determine the accuracy of the employed method. Furthermore, the results of the three-dimensional analysis can be used in the design of butterfly valve in the industry.

KEY WORDS

butterfly valve, numerical simulation, flow coefficient, hydrodynamic torque coefficient

1. Introduction

A butterfly valve (Fig. 1) is a type of flow control device that controls the flow of gas or liquid in a variety of process. It consists of a metal circular disc with its pivot axes at right angles to the direction of flow in the pipe, which when rotated on a shaft, seals against seats in the valve body. This valve offers a rotary stem movement of 90 degrees or less in a compact design.

The importance of butterfly valves has been more and more increasing in the pipe system. And there are so many reports on the characteristics, i.e. the flow coefficient, the torque coefficient, the pressure recovery factor and so on. Kerh et al. [1] performed an analysis of the butterfly valve on the basis of the experimental results. Sarpkara [2] theoretically treated the characteristics of a flat butterfly valve. Kimura and Tanaka [3] studied the pressure loss characteristics theoretically for a practical butterfly valve and so on.

With the development of the Computational Fluid Dynamics (CFD), the approach of using the technique of computational fluid dynamics has been substantially appreciated in mainstream scientific research and in industrial engineering communities. By now, the CFD simulation by commercial code has been proved its feasibility to predict the flow characteristic. There have been also many reports on valve using Computational Fluid Dynamics analysis. Huang and Kim [4] performed a three-dimensional numerical flow visualization of incompressible flows around the butterfly valve which revealed velocity field, pressure distributions by using commercial programs. Lin and Schohl [5] performed an analysis about the application of CFD commercial package in the butterfly valve field. Chern and Wang [6] employed a commercial package, $STAR-CD^{TM}$, to investigate fluid flows through a ball valve and to estimate relevant coefficient of a ball valve.

The main object of this research is to develop a model by using the commercial code ANSYS CFX 10.0, which accurately represents the flow behaviour and provide a

three-dimensional numerical simulation of water around the butterfly valve and estimate the pressure drop, flow coefficient and hydrodynamic torque coefficient. It is the first step towards improving valve design.



Fig. 1 Butterfly valve (D=1.8m)

2. Flow and hydrodynamic flow coefficient 2.1 Flow coefficient *C_V*

The flow coefficient is used to relate to the pressure loss of a valve to the discharge of the valve at a giving valve opening angle. Today, C_V is the most widely used value for valve size and pipe system. By using the C_V , a proper valve size can be accurately determined for most applications. The most common form used by valve industry is Equation (1):

$$Cv_{ISA} = \frac{Q_{gpm}}{\sqrt{\Delta P_{ISA} / S_g}} \tag{1}$$

where the pressure drop ΔP_{ISA} can be measured from static wall taps located 2 pipe diameters upstream and 6 pipe diameters downstream of the valve. And ΔP is the pressure drop in units of psi, *Qgpm* is in units of gpm, and *Sg* is the specific gravity of the fluid (*1 for water*).

In fact, this Equation ignored the affecting of friction force. According to ISA testing specifications, it is noticed that at valve of Cv/d^2 greater than 20, the affecting of the friction is significant and must be considered. In terms of practical experience and evaluation in advance, most water valves, including this object, have value of Cv/d^2 greater than 20. Hence another Equation about pressure drop with the fiction factor can be derived as Equation (2):

$$\Delta P_{net} = \Delta P_{ISA} - 0.008986 \cdot S_g \cdot f \cdot \frac{Q^2}{d^4}$$
(2)

where f is the friction factor, d is the diameter of valve in units of inches.

So, the corrected Equation of C_V can be written as Equation (3):

$$Cv_{net} = \frac{Q}{\sqrt{\Delta P_{ISA} / S_g - 0.008986 \cdot f \cdot \left(\frac{Q}{d^2}\right)^2}}$$
(3)

*Notice: Cv is a dimensional value.

2.2 Dynamic flow coefficient Ct

Hydrodynamic torque $T(\alpha)$ is the valve shaft produced by the flow passing through the valve at a given valve opening angle α . The hydrodynamic torque coefficient Ctis a factor, which is independent of the size of the valve. For a given valve and valve opening, it is easy to calculate the hydrodynamic flow torque by using Ct times the different pressure drop, Equation (4) shows the relation between Ct, T, pressure drop and valve diameter.

$$C_T = \frac{T(\alpha)}{\Delta P_{net} \cdot d^3} \tag{4}$$

3. CFD Model

3.1 Model description

The prototype size is 1.8 m in diameter and it is manufactured from cast steel with machined inside

surfaces. For getting a better result in this simulation, The CFD model of butterfly valve is created at a 1:1 scale with a rough (roughness height estimated to be about 0.5 mm) inside surface. It has a shape similar to "Discus" and its maximum thickness in the middle of the valve is 360 mm and the minimum along the flange is 20 mm. The upstream length L1 and downstream length L2 are added to provide a flow field.

3.2 Flow Pattern

The fluid, which was modeled as water, is given a uniform velocity of 3m/s at the inlet and zero reference pressure at the outlet.

Through rough calculation, the range of Reynolds Number of flow in this study is larger than 10^5 , hence the effect of the Reynolds Number is so small that it can be neglected [7].

3.3 Numerical Method

Incompressible and viscous fluid (water) flows through the butterfly valve. The flow pattern reveals that the flow studied is turbulence flow.

To deal with the turbulence modeling, the eldest approach, Reynolds-averaged Navier-Stokes Equations (RANS), is utilized. Its common form can be written as Equation (5)

$$\overline{u}_{j}\frac{\partial\overline{u}_{i}}{\partial x_{j}} = g_{j} - \frac{1}{\rho}\frac{\partial\overline{p}}{\partial x_{i}} - \frac{\partial\overline{u_{i}'u_{j}'}}{\partial x_{j}} + v\frac{\partial\overline{u_{i}}}{\partial x_{j}^{2}}$$
(5)

where u is the mean velocity and the subscript, i, $j=1\sim3$, refers to Reynolds-averaged components in three directions respectively.

 $\partial \overline{u'_i u'_i}$ is the Reynolds stresses

3.4 Turbulence model

In fact, the Reynolds Averaged *Navier-Stokes* (RANS) Equations are the models which seek to modify the original unsteady *Navier-Stokes* Equations by the introduction of averaged and fluctuating quantities.

However, the averaging procedure introduces additional unknown terms containing products of the fluctuating quantities, which act like additional stresses in the fluid. These terms, called Reynolds stresses, are difficult to determine directly and so become further unknowns. The Reynolds (turbulent) stresses need to be modeled by additional equations of known quantities in order to achieve "closure".

To solve this, many turbulence models have been created. Hereinto, three models are most commonly used, i.e. the k- ϵ model, k- ω model and Reynolds Shear Stress Model. After comparing the three models for valve opening of 55°, as a result, the author chose the k- ϵ model because the k- ϵ model does not involve the complex non-linear damping functions required for the other models and is therefore more accurate and more robust [8].

Correspondingly, the $k-\omega$ model offers no advantage and the Reynolds stress model is too expensive in computation. Table 1 shows more details about the three models.

able i Stiengths and weaknesses of unrefent models
--

strengths	weaknesses
k- ϵ and k- ω model are	k-ε overestimates
computationally cheap	turbulence
k-ω model is more	Reynolds stress is
accurate at boundary level	computationally
flows	expensive
Reynolds stress is	Reynolds stress
generally more accurate	underestimated long
	range effect

3.5 Pipe length

In terms of the research of Huang and Kim (1996), the upstream length (L1) and the downstream length (L2) should be at least 2 times and 8 times of the diameter respectively. Fig. 2 shows the various velocity vector along the pipe at length of n*d (n=-2~8). As shown in Fig. 2, the fluid upstream and downstream far from the valve is so well-proportioned, by contraries, the fluid downstream near the valve is in disorder. This phenomena also validates that both the upstream length and the downstream length should be long enough.

In this work, for the reason of accuracy and convenience, originally, L1 is set to 8 times of diameters, and L2 is set to 10.2 times of diameters. As illustrated in Fig. 2, there are no reverse flows near the outlet. Meanwhile, the error of the average velocity between 8 times diameters downstream and 10 times diameters downstream is less than 0.01%, which indicates that the length of the additional pipe can satisfy the accuracy required of the simulation.



Fig. 2 velocity vector at various lengths

3.6 Torque computation

As a kind of application of the computational fluid dynamics (CFD), CFX uses finite difference numerical procedures to solve the governing equations for fluid velocities, mass flow, pressure, temperature, turbulence parameters and other fluid properties. Numerical techniques involve the sub-division of the domain into a finite set of neighboring cells known as "control volumes" and applying the discretized governing partial differential equations over each cell. As a result, an approximation of the value of each variable at specific points throughout the domain can be obtained. In this way, one can derives a special picture of the behavior of the flow by subtracting the needed data of nodes, which attached to the surface of the valve disc. In this study, the resulting hydrodynamic torque is calculated by integrate the instantaneous parietal pressure torque [9] acting on the valve disc for rotation axis ("z" axis). Fig. 3 illustrates the way how to calculate hydrodynamic torque simply. As shown in Fig. 3, force at every node can be separated to rectangular coordinate components, and then using Equation (6) to multiply them by their corresponding arms of force, the hydrodynamic torque can be obtained.

$$T_{z}(\alpha) = \sum_{i=1}^{n} F_{y} \cdot x + \sum_{i=1}^{n} F_{x} \cdot y$$
(6)

where "i" is a limited number of nodes, which attaches to the disc and stem surface, "z" is the rotational axis.



Fig. 3 Method of torque calculation

3.7 Mesh of geometry model

Besides those remarkable factors, another important factor affecting the accuracy of the simulation is the quality of "meshing". Theoretically, the more elements in the geometry, the higher mesh quality, and the better the accuracy of the results is. Simultaneity, a longer computer calculation time will be take.

Based on the guidance of the CFX user's manual [10] and several simulations with different meshing size, the mesh with 948721 elements and 231054 nodes was used. The details about the mesh and calculation are listed in table 2.

Table 2 Various mesh sizes and calculation mormation				
Total Elements	142536	948721	1275822	
Total Nodes	34715	231054	327351	
Total CPU time (s)	1.456E4	5.826E4	1.672E5	
Discretization, total	12%	4%	2%	
Domain Imbalance	0.008 %	0.0009 %	0.0014%	

Table 2 Various mesh sizes and calculation information

4. Results and Discussion

Fig. 4 shows the computed velocity streamlines and total pressure at the inner surface of valve for the 20° opening angle. Fig. 5 illustrates the velocity contours on the middle surface at four different opening degrees. It can be found that as the valve opening degree increase,

the number and the region of vortex decrease; hence the fluid flows more smoothly.



Fig. 4 Total pressure and Streamlines by velocity



Fig. 5 Velocity contours at different opening angle

These figures demonstrate the capability of CFX to simulate the complicated flow in 3-D space. They do not

only show the flow feature, but also provide obvious evidence of the prediction's validity.

Fig. 6 shows the pressure drop and Torque, which are normalized by the maximum value of the pressure drop and Torque separately. As the velocity injected is constant and the area of the whole hatch increases, the trends showed in Fig. 6 can be understood easily. It is mentioned, even though the trends of the torque and pressure drop are similar, however the slopes at each particular position are different.



Fig. 7 compares the experimental results and the simulation result of the C_V . As the valve opening degree increases, the value of C_V vary from zero to 3.7E5. Actually in term of Equation (2), C_V is mainly dependent on the root of pressure drop, so the trend of C_V is opposite to Fig. 6. However, it must be noticed that at valve opening smaller than 20 degree, the minimum error between CFX simulation and experimental data reach to 49.27958%.



Fig. 8 shows the experimental results and the simulation result of the C_T the simulation data agree well with the experimental data. At an opening degree smaller than 60 degree, C_T creases slowly. The peak of the torque coefficient occurs at valve opening between 70 and 80 degree. Larger than 80°, the value of C_T drops to almost zero rapidly.

5. Conclusion

The results of CFX simulation generated trend which agreed with the experimental data very well. However, at some peculiar position, especially at the valve opening degree smaller than 20°, it didn't agree well. This may be due to the disadvantage of the k- ϵ turbulent model of its own. It's suggested to use another turbulent model which is good at treatment of near-wall such as k- ω model and SST turbulent model.

The simulation by CFX was very sensitive to the degree of valve opening near to the fully closed, where the flow near the valve is highly turbulent. So small subdivision is recommended near this region, and the result should be used with the comparison of the test values.

In general, the result obtained by using commercial code ANSYS CFX 10.0 agrees with the experimental result very well. However, it is recognized that all CFD-based predictions are never possible to be 100%-reliable. Hence further investigation must be performed before the computational simulation can be used directly in the industry.

Acknowledgements

The work was supported by grant No.RTI04-01-03 from the Regional Technology Innovation Program of the Ministry of Commerce, Industry and Energy (MOCIE). The Authors are very gratefully to thank them for the great help.

References

[1] Kerl, T. J, J, Lee and L. C. Wellford, "Transient fluid-structure interaction in a control valve", Journal of Fluid Engineering, 119 (1996), pp. 354-359

[2] T. Sarpkara, "Torque and cavitation characteristics of butterfly valve", ASME Journal of Applied Mechanics (1961), pp. 511-518

[3] T. Kimura and T. Tanaka, "Hydrodynamic characteristics of a butterfly valve-Prediction of pressure loss characteristic", ISA Transactions 34 (1995), pp. 319-326

[4] C.D. Huang and R.H. Kim, "Three-dimensional analysis of partially open butterfly valve flows", Transactions of the ASME, 118 (1996), pp. 562-568

[5] F. Lin and G. A. Schohl, "CFD prediction and validation of butterfly valve hydrodynamic force", World Water Congress (2004), pp.

[6] M. J. Chern and C. C. Wang, "Control of volumetric flow-Rate of ball valve using V-port", Journal of Fluid Engineering 126 (2004), pp. 471-481

[7] K. Ogawa and T. Kimura, "Hydrodynamic characteristics of a butterfly valve-prediction of torque characteristic", ISA Transactions 34 (1995), pp. 327-333

[8] B. Mohammadi and O. Pironneau, "Analysis of the K-Epsilon Turbulence Model (Research in Applied Mathematics)" John Wiley & Sons Ltd (Import) (August 1994)

[9] F. Danbon and C. Solliec, "Aerodynamic Torque of a Butterfly Valve-Influence of an Elbow on the Time-Mean and Instantaneous Aerodynamic Torque", ASME J. of Fluid Eng., 122(2000), pp. 337-344

[10] ANSYS CFX 10.0 User's Manual, ANSYS, Inc.