Middle-East Journal of Scientific Research 22 (7): 1077-1081, 2014
ISSN 1990-9233

DOI: 10.5829/idosi.mejsr.2014.22.07.21972

Prediction of Flow Coefficient and Hydrodynamic Torque Coefficient in Butterfly Valve

K. Jeyaprakash, M. Mohamed Ariffuddeen, J. Selvakumar and V. Karthik

SNS College of Technology, Coimbatore, India

Abstract: A Butterfly Valve is commonly used as control device in applications where the inlet velocity is high and the pressure drop required is relatively low. As the butterfly valves are used as shut off valves or throttling valves, higher performance and better precision are required. Thus it is very essential to know about the flow characteristic around the valve. In order to numerically analyze the incompressible fluid flow through a butterfly valve, the available three turbulence models can be used. The available models are (1) k- ε turbulence model (2) Reynolds Shear Stress turbulence model (3) k- ω turbulence model. Amongst all the available models, the k- ω turbulence model is more accurate at boundary level flows especially at the valve opening angle smaller than 20° and computationally cheap when compared to k- ε and Reynolds Shear Stress turbulence model. Hence, the present work aims to conduct the numerical simulation using k- ω turbulence model. In this proposed work, the performance factors of the valve such as flow co-efficient, hydrodynamic torque co-efficient and also the accuracy of the employed method are predicted using CFX module in ANSYS. The results are analyzed to understand the process of different fluids flow at different valve opening angle with different surface roughness of the disc of the butterfly valve. The results of the three dimensional analysis can be used in the design of butterfly valve for industrial purpose.

Key words: Turbulence model · Flow coefficient · Hydrodynamic torque coefficient · Butterfly valve and surface roughness

INTRODUCTION

Butterfly valves are commonly used in industrial applications to control the internal flow of both compressible and incompressible fluids. A Butterfly valve typically consists of a metal disc formed around a central shaft, which acts as its axis of rotation. As the valve's opening angle, θ, is increased from 0° (fully closed) to 90° (fully open), fluid is able to more readily flow past the valve. Butterfly valves must be able to withstand the stresses and forces that results from high Reynolds number flows. Characterizing a valve's performance factors, such as pressure drop, hydrodynamic torque and flow coefficient are necessary for fluid system designers to account for system requirements to properly operate the valve and prevent permanent damage from occurring to the valve.

This paper compares a 1.8 m diameter Butterfly valve's experimental performance factors to those obtained using Computational Fluid Dynamics (CFD) and to assess the feasibility of using CFD to predict performance factors of Butterfly valve. The butterfly valve of 1.8 m diameter is shown in Fig. 1.

Fig. 1: Butterfly Valve of 1.8 m Diameter

Fig. 2 shows the flowchart of the proposed method.

Organization: The paper is organized into the following sections. Section 2 is an overview of related work. The valve performance parameters are described in

Corresponding Author: K. Jeyaprakash, SNS College of Technology, Coimbatore, India. E-mail: jprakashgct@gmail.com.

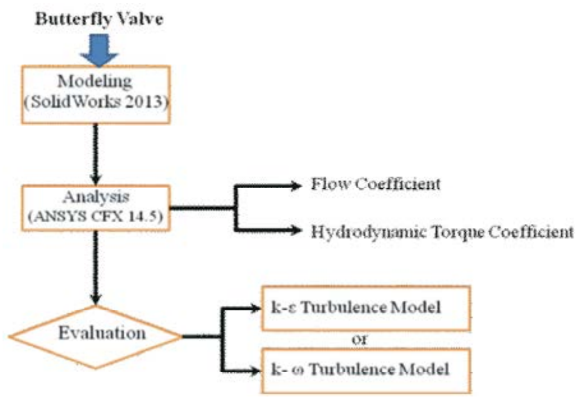

Fig. 2: Flowchart of the Proposed Scheme

section 3. The turbulence model is described in section 4. Section 5 discusses the CFD geometry model. Results & conclusion are contained in section 6&7 respectively.

Related Work: Song and Park [1] conducted a numerical simulation of a three dimensional butterfly valve operating in an incompressible fluid for various valve opening angles. The flow field was visualized and the pressure drop, flow coefficient and hydrodynamic torque coefficient values were determined.

Danbon and Solliec [2] conducted an experimental research to study the influence of an elbow located upstream of a butterfly valve on the time average and on the fluctuating aerodynamic torque. Due to the rapid development in computing power and the progress in computer visualizations, more researchers have adopted the use of numerical methods and computational fluid dynamics (CFD) techniques in their research.

Leutwyler and Dalton [3] performed a CFD study in two and three dimensions for symmetric Butterfly valves in compressible fluids at various angles and over a range of pressure ratios. The general purpose CFD code FLUENT was used with the following turbulence models: Spalart - Allmaras, k- ε and k-ω. Leutwyler [4] favored the k- ε turbulence model for its well-rounded capabilities and moderate computational costs. In addition to examining grid refinement, coefficients for lift, drag and torque were validated against experimental values.

Henderson *et al*. [5] measured torque and head loss of a symmetrical Butterfly valve installed in a hydro-electric power generating scheme for steady flow at Reynolds numbers of order 106. This was done for valve opening angles of 10 to 80 degrees in 10 degree increments.

Prema, *et al*. [6] investigated numerically the butterfly valve disc geometry and its effect on the valve performance at the fully open position when operating in an incompressible fluid flow.

Song *et al*. [7] performed a structural analysis of large Butterfly valves, in addition to validating three-dimensional experimental data of a Butterfly valve's pressure drop, flow coefficient and hydrodynamic torque coefficient using general purpose CFD code CFX. The k-ε turbulence model was selected by Song since it does not involve the complex non- linear damping functions required by other models. In the 20 degree case, differences between experimental and simulation data were found to be nearly 50%.

The aim of the present work is to develop a CFD model of a butterfly valve using the CFX module in ANSYS 14.5, which accurately represents the flow-field and provides insight of three-dimensional behavior of the flow around the valve and predicts the performance factors of the valve [8].

Flow and Hydrodynamic Torque Coefficient

Flow Coefficient: The flow coefficient is used to relate to the pressure loss of a valve to the discharge of the valve at a given 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):

$$C_v = \frac{Q}{\sqrt{\Delta P / S_g}} \tag{1}$$

where the pressure drop ΔP 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, Q is discharge of the incompressible fluid and S_g is the specific gravity of the fluid.

Hydrodynamic Torque Coefficient: Hydrodynamic torque T (α) is produced by the shaft of the butterfly valve when the fluid flows through the valve at a given valve opening angle α. The hydrodynamic torque coefficient C_T is a factor, which is independent of the size of the valve. Equation (2) shows the relation between C_T, T and valve diameter [9].

$$C_T = \frac{T(\alpha)}{\Delta P_{net} d^3} \tag{2}$$

Table 1: Comparison of Different Turbulence Models

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

Turbulence Model: 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. The transport equation of k and ω can be described in equation (3) & (4) [10].

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k-Y_k+S_k \quad (3)$$

$$\frac{\partial}{\partial t}(\rho \omega)+\frac{\partial}{\partial x_i}(\rho \omega u_i)=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_j}\right]+G_\omega-Y_\omega+S_\omega \quad (4)$$

where G_k represents the generation of turbulent kinetic energy the arises due to mean velocity gradients and G_ω is generation of ω. Y_k and Y_ω represent the dissipation of k and ω due to turbulence. Y_k and Y_ω are the turbulent Prandtl numbers for k and ω respectively.

Correspondingly, the k-ϵ model offers no advantage and the Reynolds stress model is too expensive in computation. Comparison of different turbulence model is given in Table I.

CFD Geometry Model

Geometry Description: The valve disk diameter considered for the present study is 1.8 m. A rubber seal around rim of the valve was not modeled to avoid problems associated with meshing very small volumes; the gap closed by the seal is small and was not expected to have a significant effect on the flow through the valve.

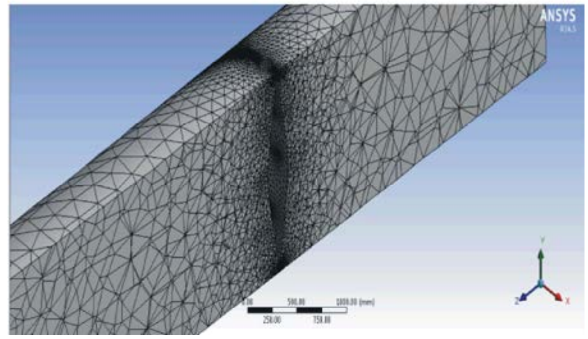

Fig. 3: Mesh at valve opening angle 20 degree

Table 2: Boundary Conditions

Surface Domain	Boundary Condition
Inlet	Velocity Inlet
Outlet	Pressure Outlet
Valve Disc	Wall
Pipe Surface	Wall

According to the Research of Song and Park [1] the upstream and downstream pipe lengths should be at least 2 and 8 disk diameters respectively [11].

Meshing of Valve: 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 [12].

Fig. 3 shows the closed view of CFD mesh on the X-Z symmetry plane for a valve angle of α = 20. Tetrahedral elements were used for meshing the model with prism layers for modeling.

Boundary Conditions: A summary of the boundary conditions used is given in Table. II., inlet velocity was set at 3 m/s with fully developed velocity profile, while the pressure at the outlet was set to zero Pascal. The valve disk and pipe surface were both set as wall with surface roughness height 0.5 mm.

RESULTS AND DISCUSSION

Fig. 4 shows the flow coefficient obtained for different turbulence models at various valve opening angles. The valve opening angle of 5°, 10°, 15° and 20° the percentage of accuracy between CFX simulation using k-ω turbulence model and experimental data has been improved to 1.76, 4.12, 0.43 and 0.39% respectively compared with k-ε model. C_V is mainly dependent on the root of pressure drop.

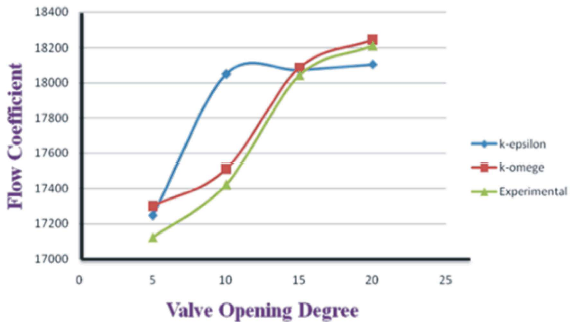

Fig. 4: C_V at various valve opening angle

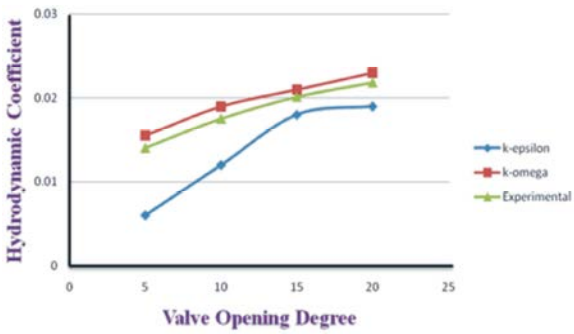

Fig. 5: C_T at various valve opening angles

Fig. 5 shows the hydrodynamic torque coefficient obtained for different turbulence models at various valve opening angles. The valve opening angle of 5°, 10°, 15° and 20° the percentage of accuracy between CFX simulation using k-ω turbulence model and experimental data has been improved to 46.43, 22.86, 5.97 and 7.34% respectively when compared to k-ε model.

The simulation data from k-ω turbulence model agree well with the experimental data.

CONCLUSION

The investigation of the flow characteristics through a butterfly valve under different opening angles and for incompressible flow regime was studied using computational fluid dynamics techniques. Computational fluid dynamics has the potential to be an useful tool in analyzing incompressible flow through butterfly valve. Numerical simulation is carried out using CFX module in ANSYS to predict the flow coefficient and hydrodynamic torque coefficient for the butterfly valve with 1.8 m diameter and the graphs are plotted using the details obtained from CFX module. The results obtained from CFX simulation revealed that the k-ω turbulence model achieve higher percentage of accuracy with experimental data. Therefore, the k-ω turbulence model is preferable than k-ε turbulence model. This result of the three dimensional analysis can be used in the design of Butterfly valve in industrial purpose. This work can be further expanded to different valve opening angles and different turbulence models.

REFERENCES

1. Song, X.G. and Y.C. Park, 2007. Numerical Analysis of Butterfly Valve-Prediction of Flow Coefficient and Hydrodynamic Torque Coefficient, Proceedings of the World Congress on Engineering and Computer Science, San Francisco, USA, pp: 759-763.
2. Danbon, F. and C. Solliec, 2000. Aerodynamic Torque of a Butterfly Valve - Influence of an Elbow on the Time-Mean and Instantaneous Aerodynamic Torque, Journal of Fluids Engineering, 122(2): 337-344.
3. Leutwyler, Z. and C. Dalton, 2006. A Computational Study of Torque and Forces Due to Compressible Flow on a Butterfly Valve Disk in Mid-stroke Position, Journal of Fluids Engineering, 128(5): 1074-1082.
4. Leutwyler, Z. and C. Dalton, 2008. A CFD Study of the Flow Field, Resultant Force and Aerodynamic Torque on a Symmetric Disk Butterfly Valve in a Compressible Fluid, Journal of Pressure Vessel Technology, 130(2): 021302- (1-10).
5. Henderson, A.D., J.E. Sargison, G.J. Walker and J. Haynes, 2007. A Numerical Study of the Flow through a Safety Butterfly Valve in a Hydro-Electric Power Scheme, Proceedings of the 16th Australasian Fluid Mechanics Conference Crown Plaza, Gold Coast, Australia, pp: 1116-1122.
6. Prema, B., S. Bahojani and N. Gopalakrishnan, 2010. Design Optimization of Butterfly Valve Using CFD, Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power, IIT Madras, Chennai, India.
7. Song, X.G. and Y.C. Park, 2009. Fluid and Structural Analysis of a Large Diameter Butterfly Valve, Journal of Advanced Manufacturing System, World Scientific Publishing Company, pp: 81-88.
8. Wang, L., X.G. Song and Y.C. Park, 2009. Dynamic Analysis of Three – Dimensional Flow in the Opening Process of a Single–Disc Butterfly Valve, Journal of Mechanical Engineering Science, pp: 329-336.
9. Song, X.G., L. Wang and Y.C. Park, 2009. Analysis and Optimization of a Butterfly Valve Disc, Journal of Mechanical Engineering Science, pp: 81-89.

10. Mc Donough, J.M., 2004. Introductory Lectures on Turbulence, University of Kentucky.
11. Mohammadi Bijan and Olivier Pironneau., Analysis of the K-Epsilon Turbulence Model.
12. Solliec, C. and F. Danbon, 1999. Aerodynamic Torque Acting on a Butterfly Valve, Comparison and Choice of a Torque Coefficient, Journal of Fluids Engineering, 121(4): 914-917.