A Numerical Investigation of the Incompressible Flow through a Butterfly Valve Using CFD

A. Dawy¹, A. Sharara², A. Hassan³

¹M.Sc. Student, Department of Mechanical Engineering, AASTMT, Alexandria, Egypt
²Assistant Professor, ³Professor, HOD, Department of Marine Engineering, AASTMT, Alexandria, Egypt

Abstract—The results of a numerical investigation of a butterfly valve under incompressible flow conditions are reported. The commercial CFD code Fluent was used for the study, and the standard K-ε turbulence model was selected. The simulations provided insight into the flow field characteristics of the valve. The valve performance factors such as the flow coefficient and hydrodynamic flow coefficient were computed for different valve opening angles. The results were compared with experimental data and good agreement was found. The results demonstrated the importance of CFD as a valuable tool in visualizing the complex structures in the flow field of the valve. The pressure drop across the valve and the valve flow coefficient were found to be directly dependent on the opening angle of the valve disk. The hydrodynamic torque coefficient curve was computed at different valve opening angles, and the angle at which maximum torque coefficient occurs was identified.

Keywords—Butterfly Valve, Control Valve, Computational Fluid Dynamics, CFD, Numerical Analysis, Turbulence.

I. INTRODUCTION

A butterfly valve (Fig. 1) is a flow control device that is widely used in a variety of piping applications including oil & gas transportation, power generation plants, petrochemical plants, water distribution, sewage and firefighting applications. They owe much of their popularity to their simple mechanical assembly, low cost, fast closing and opening times, and most importantly their low pressure drop in the fully opened position.

The three main components of a butterfly valve are the valve body, the valve disk and the shaft (Stem) which is typically connected to an actuator. The shaft acts as the center of rotation for the disk, and turns the disk in a 90 degree motion from a position parallel to the fluid flow (fully opened position) to a position where the disk is perpendicular to the fluid motion (fully closed position). For that reason the butterfly valve belongs to the quarter turn valves family.

Fig. 1. A Typical Butterfly Valve

The study of butterfly valves has evolved over the years; most of the earlier studies were based on analytical and experimental approaches. Sarpkaya [1] developed a theoretical model for predicting the cavitation and torque characteristics of a flat disk butterfly valve. Eom [2] studied experimentally the performance of two different configurations of butterfly valves: perforated disk and different diameters of solid disks that allow partial opening of the valve at closed position. The results supported the suitability of butterfly valves as good flow controllers. Morris and Dutton [3] experimentally investigated the aerodynamic torque characteristics of butterfly valves using two dimensional planar models and three dimensional prototype valves at choked and un-choked operating points, and the results revealed the significance of the flow separation and reattachment phenomena on the aerodynamic torque characteristics of butterfly valves.

Morris and Dutton [4] also investigated the operating characteristics of two similar butterfly valves mounted in series, and an experimental investigation concerning the operating characteristics of a butterfly valve downstream of a 90 degree mitered elbow [5]. Kimura et al. [6] theoretically studied the pressure loss characteristics and cavitation stages for a practical butterfly valve.
A theoretical pressure loss coefficient was formulated, and the relation between the loss coefficient and the critical cavitation factor was studied. Ogawa and Kimura [7] conducted a theoretical investigation that focused on the prediction of hydrodynamic torque characteristics of butterfly valves. An equation of torque characteristics derived from the theoretical investigation taking into account the experimental measured torque data was proposed. Danbon and Sollicie [8] 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. Among the earliest researchers to use a commercial computational fluid dynamics code to investigate the flow through a butterfly valve are Huang and Kim [9] who carried out a three-dimensional numerical analysis for incompressible fluid flows in a butterfly valve using a commercial software, the study revealed the velocity field and the pressure distribution. Lin and Schohl [10] conducted a numerical study to validate the use of the commercial CFD package FLUENT in predicting the hydrodynamic forces acting on a partially open butterfly valve. The results showed that CFD modeling can provide a reasonable estimate of the force and discharge coefficients, as a comparison was made between the CFD predicted and experimental data. Leutwyler and Dalton [11] [12] [13] conducted various numerical studies to assess the capability of CFD codes to predict butterfly valve performance coefficients, and to provide insight into the compressible flow-field in a butterfly valve. Song and Park [14] 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. Henderson et al. [15] conducted a numerical study of the flow through a safety butterfly valve in a hydroelectric power scheme to stop water supply to a downstream penstock in order to predict the hydrodynamic torque versus opening angle characteristic. The numerical results showed that a strong vortical flow pattern developed downstream from the valve. Perma, et al. [16] investigated numerically the butterfly valve disk geometry and its effects on the valve performance at the fully open position when operating in an incompressible fluid flow.

The aim of the present work is to develop a CFD model of a symmetric disk type butterfly valve using the commercial code FLUENT, that 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.

II. CFD MODEL

A. Model Geometry Description

The valve disk diameter considered for the present study is 50 mm with pipe diameter set to be 52 mm. 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. According to the Research of Huang and Kim [9] the upstream and downstream pipe lengths should be at least 2 and 8 disk diameters respectively. The upstream and downstream pipe lengths were set to be 2 and 15 disk diameters respectively.

B. Meshing

Tetrahedral elements were used for meshing the model with prism layers for modeling boundary layer as shown in Fig. 2., a grid independence study was conducted to ensure grid independency, and grids were constructed for valve openings of 0, 15, 30, 45, 60, and 75 degrees. The number of elements for each case was between 979198 and 1055453 elements.

(a) Extended View of Mesh at 45 deg.
Fig. 2. Generated Mesh

C. Boundary Conditions

A summary of the boundary conditions used is given in Table. I. Inlet velocity was set at 5 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 no-slip boundary condition.

<table>
<thead>
<tr>
<th>Surface Domain</th>
<th>Boundary Condition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>Velocity Inlet</td>
</tr>
<tr>
<td>Outlet</td>
<td>Pressure Outlet</td>
</tr>
<tr>
<td>Valve Disk</td>
<td>Wall</td>
</tr>
<tr>
<td>Pipe Surface</td>
<td>Wall</td>
</tr>
</tbody>
</table>

D. Numerical Method

Incompressible, viscous, turbulent flow was simulated through the valve by solving the Reynolds Averaged Navier-Stokes equations (RANS); its common form can be written as:

$$\rho \frac{\partial \bar{\mathbf{u}}}{\partial t} + \nabla \cdot (\rho \bar{\mathbf{u}} \bar{\mathbf{u}}) = -\nabla P + \mu \nabla^2 \bar{\mathbf{u}} + \rho \mathbf{\nabla} \cdot \mathbf{\tau}$$  \hspace{1cm} (1)

To deal with turbulence many models have been developed over the years. Two equation turbulence models have served as the foundation for much of the turbulence model research during the past two decades [20], they are the most commonly used in turbulent engineering problems due to robustness, lower computer requirements and relative accuracy. The Standard k-ε model which is a semi-empirical two equation model was chosen for this study. It relates the turbulent viscosity, the turbulent kinetic energy and the turbulent dissipation rate.

The transport equations for the standard k-ε model can be described as follow [21]:

$$\frac{\partial}{\partial x_j} \left( \rho k \frac{\partial k}{\partial x_j} \right) = \frac{\partial}{\partial x_j} \left( \mu \frac{\partial k}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_j} \frac{\partial \bar{u}_k}{\partial x_j} \right) - \rho \varepsilon \frac{\partial k}{\partial x_j} + \mu_t \left( \frac{\partial \bar{u}_k}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_k} \right) \frac{\partial \bar{u}_i}{\partial x_j} \frac{\partial \bar{u}_j}{\partial x_i}$$  \hspace{1cm} (2)

$$\frac{\partial}{\partial x_j} \left( \rho \varepsilon \frac{\partial \varepsilon}{\partial x_j} \right) = \frac{\partial}{\partial x_j} \left( \mu \frac{\partial \varepsilon}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_j} \frac{\partial \bar{u}_k}{\partial x_j} \right) - \rho C_{\mu} \frac{\varepsilon}{\varepsilon} \frac{\partial k}{\partial x_j} - \rho C_{\mu} \frac{\varepsilon}{\varepsilon}$$  \hspace{1cm} (3)

Where $\mu_t = \rho C_{\mu} \frac{k^2}{\varepsilon}$

And the empirical constants for the model are given in Table. II

<table>
<thead>
<tr>
<th>Model Constants</th>
<th>$C_{\mu}$</th>
<th>$C_{\varepsilon 1}$</th>
<th>$C_{\varepsilon 2}$</th>
<th>$\delta_k$</th>
<th>$\delta_\varepsilon$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_{\mu}$</td>
<td>0.09</td>
<td>1.47</td>
<td>1.92</td>
<td>1</td>
<td>1.3</td>
</tr>
</tbody>
</table>

III. Results & Discussions

A. Flow Field Description

A common feature that is found in all non-zero opening angles is the formation of two counter rotating swirling vortices directly downstream of the valve disk; this can be observed in the velocity streamlines illustrated in Fig. 3. This behavior was also noted by Henderson et al. [15].
At the fully opened angle the flow is fairly smooth with very small separation region visible at the trailing edge of the disk, as the valve opening angle increases the flow around the valve becomes highly turbulent and the extent of the separation increases until most of the downstream valve disk becomes separated at 45° opening as shown in Fig. 4. The separation downstream the valve and the formation of vortices can be attributed to the sudden pressure drop across the valve which increases with the increase in valve opening angle as seen in Fig. 5, and also the reduction in the area between the edges of the disk and the pipe wall with the increase in the opening angle leads to acceleration of the flow which acts as a jet leading to flow recirculation and flow separation from the disk.

In Fig. 6 a close up view of the valve disk at an opening angle of 75° is illustrated showing the vortices formed downstream the disk as a result of boundary layer separation.

**B. Velocity Profiles**

The fully developed velocity profile described earlier as the inlet boundary condition can be observed in Fig. 7. at a distance of 2 D upstream the valve disk. The downstream velocity profiles were observed at different opening angles and it was found that at a downstream pipe length of about 8 to 10 disk diameters the flow recovered to the fully developed pattern as illustrated in Fig. 8.
C. Valve Performance Factors

The main factors considered during classification and selection of butterfly valves are the valve flow coefficient, and the hydrodynamic torque coefficient. The knowledge of these two factors is essential for piping designers and valve manufacturers to ensure safe and reliable operation of the valves and the piping system. The most common equations used to describe these coefficients are [18]:

\[ C_v = 11.56Q \sqrt{\frac{S_d}{\Delta P}} \]  

\[ C_t = \frac{T_d}{D_d^4 \Delta P} \]  

The valve flow coefficient can be seen in Fig. 9, the curve starts with a maximum value at the zero opening degree and gradually decreases to a minimum value at the fully closed position, the trend of \( C_v \) is opposite to the trend of the pressure drop curve of Fig. 5, this can be easily understood by examining Eq. 4. Where \( C_v \) is inversely proportional to the root of the pressure drop across the valve. The computed hydrodynamic torque coefficient is illustrated in Fig. 10 and compared with experimental data from the literature. The comparison confirms that the numerical curve follow the trend of the experimental curves as the curves qualitatively agree with maximum torque coefficient occurring between 10° and 16°, at opening degrees greater than 16° the value of \( C_t \) starts to decrease gradually till it reaches a minimum value near fully closed position.

IV. CONCLUSIONS

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. The flow field was visualized and the performance factors of the valve were computed and compared against experimental data. Based on the results the following conclusions can be highlighted:

- CFD provides a means of gaining valuable insight into the flow field of the valve, where complex fluid structures including vortex formation and flow separation and velocity profile recovery can be observed and studied.
• Strong vortical behavior is present downstream of the valve, the extent of vortices formation and flow separation increases with the increase in the opening angle of the valve disk.

• The pressure drop across the valve & the valve flow coefficient were found to be directly dependent on the position of the valve disk, where increasing the opening angle of the valve results in an increase in the pressure drop, and consequently a drop in the flow coefficient.

• The maximum value of the hydrodynamic torque coefficient was observed at a valve opening angle of 10 to 16 deg., this was confirmed by comparing the results with experimental test data from the literature.

Nomenclature

\[
\begin{align*}
\text{CFD} & \quad \text{Computational Fluid Dynamics} \\
\overline{V} & \quad \text{Mean Velocity} \\
P & \quad \text{Pressure} \\
\rho & \quad \text{Density} \\
\mu & \quad \text{Viscosity} \\
\tau_{ij} & \quad \text{Reynolds Stress Tensor} \\
\mu_t & \quad \text{Turbulent Viscosity} \\
k & \quad \text{Turbulent Production Rate} \\
\varepsilon & \quad \text{Turbulent Dissipation Rate} \\
C_p & \quad \text{Flow Coefficient} \\
Q & \quad \text{Discharge} \\
S_p & \quad \text{Specific Gravity} \\
\Delta P & \quad \text{Pressure drop across valve} \\
C_f & \quad \text{Hydrodynamic Torque Coefficient} \\
T_d & \quad \text{Hydrodynamic Torque} \\
D_d & \quad \text{Disk Diameter}
\end{align*}
\]

REFERENCES


