Abstract: - Nowadays, numerical analysis becomes more and more popular in designing large butterfly valve. In this paper, fluid structure interface (FSI) analysis is carried out to estimate fluid field and performance of large butterfly valve in diameter of 2.2m. A three-dimensional model of butterfly valve is used to do simulation perfectly. The FSI analysis not only analyzes the fluid field at various valve openings, but also verifies whether the valve can work safely under serious conditions or not. The result shows this type valve is not safe when valve is opened at 15 degree or when valve is closed. To ensure that the valve can work safely, the improvement of this type valve is carried out. And some important dimensions of valve components are chosen to be assembled in different values. And the analyzed results of different assemblages are compared to determine the best one which can strengthen the structure and reduce the weight of valve mostly furthest.

Key-Words: - Large butterfly valve, numerical analysis, FSI analysis, improvement

1 Introduction

Large butterfly valves are normal industrial piping components, which are used for flow control or safety equipment in a variety of processes in transport system (see [1] and [2]). A butterfly valve consists of three basic components: the valve body, the valve disc, and the supporting shaft. Because of its simple mechanical assembly, and the small flow resistance of the disk in the fully open position, butterfly valves provide a relatively high flow capacity that fluid can flow fast with relatively low pressure loss through the valves (see [3] and [4]). In the past, people conduct experiments to analyze flow characteristics of butterfly valves in the laboratory, which require a number of equipments, a lot of time, funds and so on. Nowadays, due to the fast progress of computer visualization and numerical technique, it becomes possible to do analysis using simulation technique. Huang and Kim [5] performed a three-dimensional numerical analysis of incompressible flows around the butterfly valve which revealed the basic characteristics of fluid field around butterfly valves by using commercial programs. Danbon and Sollicie [6] revealed the influence of an elbow on the time-mean and instantaneous aerodynamic torque of a butterfly valve. Song and Park [7] investigated the flow coefficient and hydrodynamic torque coefficient by using numerical analysis. Leutwyler and Dalton [8] performed a computational study of butterfly valve disc to validate the ability of accurately predicting flow characteristics of butterfly valve using computational fluid dynamics. Salvador, Altozano and Valverde [9] studied the three-dimensional modeling and geometrical influence on fluid characteristics of valves.

However, researchers mostly studied fluid field which flow through the butterfly valve. There is no structure analysis which is very important in the actual application of industry engineering. Actually, the numerical analysis is not only used to study the flow characteristics of butterfly valves, but also to improve the design of butterfly valves. In this study, fluid structure interface (FSI) analysis is conducted to determine whether the large butterfly valve can work
safely or not. And the improvement of butterfly valve is conducted in this paper. The large butterfly valve is as shown in Fig. 1.

![Diagram](image1)

**Fig. 2 Approaches for analysis and improvement**

Fig. 2 illustrates the approaches to analyze and improve a large butterfly valve in this paper. Firstly, initial analysis (FSI analysis) is conducted to reveal the hydrodynamic pressure on valve surfaces and stress distribution in valve when butterfly valve is opened under various angles and a uniform inlet flow velocity of 3m/s by using commercial package CFX. Then structure analysis of valve is carried out to find out the worst situation when butterfly valve disc works at various positions.

In this paper, the structure of valve is improved by changing some selected parameters. Comparing the analyzed results of different structures, the best shape of butterfly valve is determined and recommended to valve manufacturers.

**2. Fluid Structure Interface (FSI) Analysis**

Fluid structure interaction (FSI) occurs when a fluid interacts with a solid structure, exerting pressure that may cause deformation in the structure and, thus, alter the flow of the fluid itself. The ANSYS FSI solution provides designers and analysts with the most flexible and advanced coupled structural–fluid physics analysis tool available [10]. Fluid structure interaction is required for many industry applications, such as biomedical—elastic artery modeling for stent design, aerospace—foils flutter and civil engineering—wind loading of structures.

FSI simulations can be broadly categorized as one-way coupled or two-way coupled as shown in Fig. 3. One-way coupling exists when the flow field is not greatly affected since the resulting deformation of the solid caused by fluid pressure is small. This allows CFD and FEA solutions to be run independently, with loads transferred in only one direction. The two-way coupled FSI analysis is used for some cases in which the structure deforms significantly affects the flow field. Two-way coupled FSI uses the ANSYS Multi-field solver to provide a true bidirectional FSI capability through domain interface with moving/deforming geometry.

![Diagram](image2)

**Fig. 3 Two kinds of FSI simulations**

In this paper, for the solid deformation affected the fluid field very slightly, One-way coupling FSI analysis is conducted to find out the worst situation when valve works.

**3. Initial Analysis**

**3.1 Numerical model**

The prototype of numerical model is 2.2m in diameter which is in accordance with true butterfly valve as shown in Fig. 4. And the flow fluid defined as water is given an inlet velocity of 3m/s and an opening outlet, while a static hydrodynamic pressure of 0.88Mpa is put on valve surfaces when valve is closed.

![Diagram](image3)

**Fig. 4 The numerical model**

According to the research of Huang and Kim (1996) [5], the upstream pipe length and the downstream pipe length which are required to be at least 2 times and 8 times of pipe diameter are set to 6 and 10 pipe diameters respectively in this paper. In addition, $k-\varepsilon$ turbulence model is used to estimate the flow field through large butterfly valve. To get
more accurate results, meshes with 1389947 elements and 373572 nodes are generated.

### 3.2 Fluid analysis

Three-dimensional numerical technique is used to observe the fluid field and to denote flow velocity inside valve and hydrodynamic pressure on valve surfaces when the 2.2m diameter butterfly valve is operated at various angles and under a uniform inlet flow velocity of 3m/s.

![Flow velocity contour under various openings](image1)

Fig. 5 displays the flow velocity at middle plane under four valve openings. From Fig. 5, we can see that there exist separated vortex zones which are caused by boundary layer separation [5]. As the angle increases, the vortex zone becomes smaller and smaller, and the faster velocity zone (red color zone) becomes increasing to the larger zone, but the magnitude of velocity becomes smaller.

![Total pressure on valve surfaces at different angles](image2)

Fig. 6 denotes the total pressure on valve surface at different angles. From Fig. 6, we can observe that the higher hydrodynamic pressure zone (the red color zone) on valve surfaces becomes decreasing to smaller zone as the valve opening angle increases, and the magnitude of the hydrodynamic pressure becomes smaller too. The flow also becomes more and more smoothly as the opening angle increases [5]. And the magnitude of the maximum stress in valve should become smaller since the hydrodynamic pressure is imported to structure analysis as a boundary condition in one-way coupled FSI simulation.

### 3.3 Structure analysis

The large butterfly valve studied in this paper must resist a constant static hydrodynamic pressure of 0.88MPa when it’s closed. The situation of valve working with a constant static pressure of 0.88MPa means that all the inner upstream valve surfaces are under the pressure of 0.88MPa.

To find out the worst situation when valve works, the conditions with a static hydrodynamic pressure of 0.88MPa and a fast flow velocity of 3m/s must be compared. As one-way coupled FSI simulation is conducted to import the results of fluid analysis into structure analysis as a boundary condition, the states of valve working with a fast flow velocity of 3m/s are analyzed under different openings by using commercial package ANSYSWORKVENCH.

![Stress distributions in valve: a) closed; b) 15°](image3)

Fig. 7 shows the stress distributions in valve when vale is closed and at an angle of 15°. As we see from
the Fig. 7, the higher stress mostly distributes near the contact place of valve components and the central bottom of disc. And Table 1 displays the maximum stress of valve components in different situations. From Fig.8 and Table 1, we can see that the worst situation of valve working is when valve is closed. It also can be deduced that the maximum stress of valve is very high when valve is opened at an angle less than 15°. These also demonstrate that valve can’t work safely when valve is closed and opened at a small angle. So the improvement of valve is carried out to make a good performance of valve and the new shape of valve must be validated.

Table 1 the maximum stress of valve (Unit: MPa)

<table>
<thead>
<tr>
<th>Yield stress</th>
<th>Closed</th>
<th>15°</th>
<th>30°</th>
<th>45°</th>
<th>90°</th>
</tr>
</thead>
<tbody>
<tr>
<td>disc</td>
<td>240</td>
<td>422.8</td>
<td>403.3</td>
<td>94.4</td>
<td>31.9</td>
</tr>
<tr>
<td>body</td>
<td>240</td>
<td>277.7</td>
<td>212.1</td>
<td>76.1</td>
<td>24.3</td>
</tr>
<tr>
<td>shaft</td>
<td>690</td>
<td>874.1</td>
<td>867.3</td>
<td>196.4</td>
<td>50.7</td>
</tr>
</tbody>
</table>

4. Improvement of Large Butterfly Valve Structure

To improve the structure of valve, firstly some proper parameters should be chosen. In this study, there are three important dimensions to be chosen: the diameter of SHAFT $D$, the wall thickness of DISC $T_1$ and the wall thickness of BODY $T_2$ as shown in Fig. 8. These three changeable dimensions can affect the stress distributions in valve seriously.

The diameter of shaft $D$, the disc wall thickness $T_1$ and the body wall thickness $T_2$ are changed several times to compare structure analysis results and determine which assemblage is better. Table 2 denotes the structure analysis results of different assemblages. From the Table 2, the assemblage of $D=240$mm, $T_1=40$mm, $T_2=30$mm is chosen to be the new dimensions of valve structure considering both the economy of manufacturing and the safety of valve working.

Table 2 the maximum stress of different models

<table>
<thead>
<tr>
<th></th>
<th>$D$ (mm)</th>
<th>$T_1$ (mm)</th>
<th>$T_2$ (mm)</th>
<th>Disc (MPa)</th>
<th>Body (MPa)</th>
<th>Shaft (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tensile yield stress</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>240</td>
<td>240</td>
<td>690</td>
</tr>
<tr>
<td>Initial model</td>
<td>178</td>
<td>57</td>
<td>40</td>
<td>422.8</td>
<td>277.7</td>
<td>874.1</td>
</tr>
<tr>
<td>Optimized model 1</td>
<td>220</td>
<td>57</td>
<td>40</td>
<td>186.9</td>
<td>158.5</td>
<td>553.2</td>
</tr>
<tr>
<td>Optimized model 2</td>
<td>240</td>
<td>57</td>
<td>40</td>
<td>116.4</td>
<td>90.6</td>
<td>412.1</td>
</tr>
<tr>
<td>Optimized model 3</td>
<td>240</td>
<td>57</td>
<td>30</td>
<td>123.7</td>
<td>103.3</td>
<td>436.7</td>
</tr>
<tr>
<td><strong>Optimized model 4</strong></td>
<td><strong>240</strong></td>
<td><strong>40</strong></td>
<td><strong>30</strong></td>
<td><strong>164.2</strong></td>
<td><strong>133.2</strong></td>
<td><strong>491.2</strong></td>
</tr>
</tbody>
</table>

Reduction (%) * - - - 61.2% 52.0% 43.8%

Note: *The result of optimized model 4 selected as the final model is compared with the result of initial model.
Table 3 displays the mass change of valve compared initial model with new model. The mass of new model decreases about 10.5% contrasting with initial model. The final result of improvement validates the availability of numerical analysis in designing large butterfly vale.

<table>
<thead>
<tr>
<th></th>
<th>Disc mass</th>
<th>Body mass</th>
<th>Shaft mass</th>
<th>Total mass</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial model</td>
<td>2814.9</td>
<td>3272.4</td>
<td>426.3</td>
<td>6513.6</td>
</tr>
<tr>
<td>New model</td>
<td>2143.1</td>
<td>2911.9</td>
<td>775.1</td>
<td>5830.1</td>
</tr>
<tr>
<td>Reduction (%)</td>
<td>23.9%</td>
<td>11.1%</td>
<td>-81.8%</td>
<td>10.5%</td>
</tr>
</tbody>
</table>

5. Conclusions
The validity of simulation by CFX is verified by the results of fluid analysis. And the simulation is very sensitive to the valve openings near to the fully closed, where the flow around the valve is highly turbulent. And as the valve opening angle decreases, the hydrodynamic pressure becomes bigger and bigger. It’s suggested that large butterfly valve should not be fixed at a low opening angle. FSI method used in this paper is one-way coupled. In one-way coupled FSI simulation, fluid analysis and structure analysis are conducted respectively. There is only one direction to transfer loads such as hydrodynamic pressure which is the result of fluid analysis and imported into structure analysis as a boundary condition. It’s an effective method to estimate the performance of large butterfly valve including the stress distribution in valve and the hydrodynamic pressure on valve surface when valve is opened.

The modifications of the valve structure in this paper are conducted to improve the performance of large butterfly valve. And the modifications are also validated to make sure the valve work safely. Considering the economy of manufacturing and the safety of valve working, the final structure of valve must be chosen carefully.

Today, because the materials of valve become more and more expensive, reducing the weight of valve is a very important method to cut cost down. In this paper, the improvement of large butterfly valve in diameter of 2.2m verifies that it’s effective to reduce the weight of valve and strengthen the structure of valve using numerical analysis. This method has many advantages such as less of time and funds in valve design. It will be more and more popular in the future. But necessarily, the new shape of valve must be verified by experiments in a laboratory before it’s used to be manufactured in factories.

Acknowledgment
This work was supported by Technical Center for High-Performance Valves from the Regional Innovation Center (RIC) Program of the Ministry of Knowledge Economy (MKE).

References: