ISSN: 2472-3703 DOI: 10.6911/WSRJ.202008_6(8).0005

Flow Field Analysis of Seawater Butterfly Valve for Nuclear Power based on the CFD

Jinxin Dong[1, a], Qingxiao Kong[1, b, *], Shuwei Pan[1, c] and Lilong Lin[1, d]

[1]Department of Mechanical Engineering, Wenzhou Polytechnic, Wenzhou, China.

[a]1013125842 @qq.com, [b]k_qingxiao@163.com, [c]393753643@qq.com, [d]329992589@qq.com

Abstract

Seawater butterfly valve for nuclear power requires good overflow performance and open-close characteristics, and analysis on the flow pattern of internal field was conducted. A finite element model was constructed using Fluent software, and the model was applied to predict the characteristics of the designed valve flow field for disk positions ranging from 5 degree to 90 degree. The relationship between flow coefficient and disk positions was constructed, as well as the relationship between flow coefficient and disk positions. Simulation results verified the overflowing property of seawater butterfly valve satisfies the requirement of inlet and outlet butterfly for steam condenser.

Keywords

Seawater Butterfly Valve; Flow coefficient; Flow resistance coefficient; numerical simulation.

1. INTRODUCTION

The seawater butterfly valve at the inlet and outlet of the condenser in nuclear power plant is used to connect or cut off the circulating water system during operation or maintenance. The flow rate and the flow resistance coefficient are two key design parameters for the sea butterfly valve at the inlet and outlet of the condenser, which directly affects the performance of condenser and changes the temperature rise of cooling water as well as the heat transfer coefficient of circulating water system. On the basis of satisfying the flow rate determined by the purchaser, the seawater butterfly valve should be further tested to verify whether the flow coefficient satisfies the requirement or not by experiment. To improve design efficiency and accuracy, flow field analysis should be conducted to acquire flow characteristics and flow resistance characteristics at the design stage.

Computational Fluid Dynamics (CFD) is a numerical analysis of physical problems such as fluid flow and heat transfer by using computer technology, whose essence is to establish a flow field model for physical problems. By solving the continuity equation and Navier Stokes equation, numerical results of the flow field such as velocity field can be obtained [1]. As a numerical simulation method, CFD analysis is widely used in flow field simulation of valve with the improvement of flow physical model. Liu Jian et al used FLUENT software to simulate and analyze the three dimensional flow of heavy caliber butterfly valve [2]. Xueguan Song et al used the CFX software to simulate and analyze the flow characteristics of butterfly valve [3]. Qinzhao Zhang et al obtained the flow resistance characteristics of common three eccentric butterfly valve by simulation [4].

The DN2700 seawater butterfly valve at the inlet and outlet of the condenser in the circulating water system of Sanmen Nuclear Power Unit 3/4 was taken as the research object [5]. The simulation was carried out according to the technical specifications, in which the required

ISSN: 2472-3703 DOI: 10.6911/WSRJ.202008_6(8).0005

water flow was 47800m3/h while the valve pressure loss was required less than 111mm Hg. The Flow field simulation model of nuclear seawater butterfly valve was set up based on the CFD software, and the RNG k-ε turbulence model was adopted to analyze the characteristics of fluid flow at every 5 degree ranging from 5degree to 90 degree. The relationship curve between flow coefficient, flow resistance coefficient and opening degree of valve was constructed to indicate the performance of the designed seawater butterfly valve.

2. CHARACTERISTIC PARAMETERS OF FLOW FIELD

2.1. Flow coefficient C_v

The flow coefficient is used to indicate the flow capacity of the valve, which is determined by the structure, shape, size and other factors of the valve. The larger the flow coefficient value is, the better flow capacity of the valve is, which means less pressure loss when the unit fluid passes through the valve. The formula of flow coefficient is given in the IEC 60534 industrial process control valve standard as follows [6].

$$C_v = C \times Q \times \sqrt{SG / \Delta P} \tag{1}$$

In formula (1), Q is the unit flow. SG is the specific gravity of the fluid medium. ΔP is the pressure loss. Constant C is the unit conversion coefficient, which is determined by the unit of Q and ΔP. Generally, when the unit of Q is gal/min and the unit of ΔP is psi, the value of constant C is 1.

2.2. Flow Resistance Coefficient ξ

The flow resistance coefficient is used to indicate the resistance of the valve to the fluid, which is determined by the structure, size and other factors of the valve. The larger the flow resistance coefficient value is, the higher resistance of the valve to the fluid is. The formula of Flow resistance coefficient is given as follows [7].

$$\xi = \frac{2\Delta P}{\rho V^2} \tag{2}$$

In formula (2), ΔP is the pressure loss. ρ is the density of the fluid. V is the average velocity of fluid.

3. THEORETICAL BASIS OF CFD

3.1. Control Equation of Flow Field

All flows of fluids in nature follow the law of mass conservation, Newton's second law and energy conservation. The control equations of CFD consist of the continuity equation, momentum conservation equation and energy conservation equation.

The continuity equation is expressed in equation (3).

$$\frac{\partial \rho}{\partial t} + div\left(\rho \vec{V}\right) = 0 \tag{3}$$

ISSN: 2472-3703 DOI: 10.6911/WSRJ.202008_6(8).0005

The momentum conservation equation (Navier Stokes equation) is expressed in equation (4).

$$\begin{cases} \frac{\partial(\rho u)}{\partial t} + div\left(\rho u\vec{V}\right) = div\left(\mu gradu\right) - \frac{\partial p}{\partial x} + S_u \\ \frac{\partial(\rho v)}{\partial t} + div\left(\rho v\vec{V}\right) = div\left(\mu gradv\right) - \frac{\partial p}{\partial y} + S_v \\ \frac{\partial(\rho w)}{\partial t} + div\left(\rho w\vec{V}\right) = div\left(\mu gradw\right) - \frac{\partial p}{\partial z} + S_w \end{cases} \tag{4}$$

In equation (4), ρ is the density of the fluid. t is the time. $\vec{V}$ is the velocity vector, and u、v、w is component of velocity vector on x、y、z axis respectively. p is the pressure on fluid unit. S_u、S_v、S_w are the generalized source terms.

3.2. Turbulence Model

For the flow in the pipe, when Re is greater than 2300, the flow state is turbulent. Obviously, the flow state of three eccentric butterfly valve is turbulent. For turbulence state, it is very difficult to solve the control equations directly, and the N-S equation is usually treated by time average. But after the N-S equation processed by time average, the control equations become under determined equations. So additional equations are required to make the equations closed, and the additional equation is just turbulence model.

4. NUMERICAL SIMULATION OF FLOW FIELD

4.1. Geometric Model

According to the designed structure of DN2700 seawater butterfly valve, the 3D model of large curvature arc disc was constructed as shown in Figure 1. The 3D model of butterfly valve was imported into the CFD software to build flow field analysis model. In order to ensure that the fluid in the calculation domain reaches a stable state, the length of the pipeline before the valve was taken as 6 times of the pipeline diameter as well as the length of the pipeline behind the valve is taken as 10 times of the pipeline diameter, the three-dimensional model of flow field is shown in Figure 2.

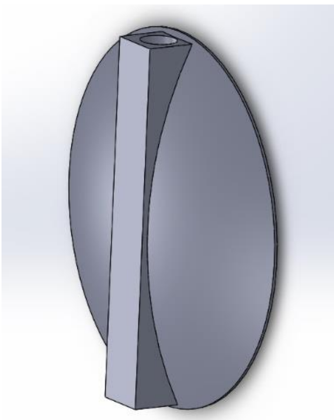

Figure 1. The 3D model of large curvature arc disc

ISSN: 2472-3703 DOI: 10.6911/WSRJ.202008_6(8).0005

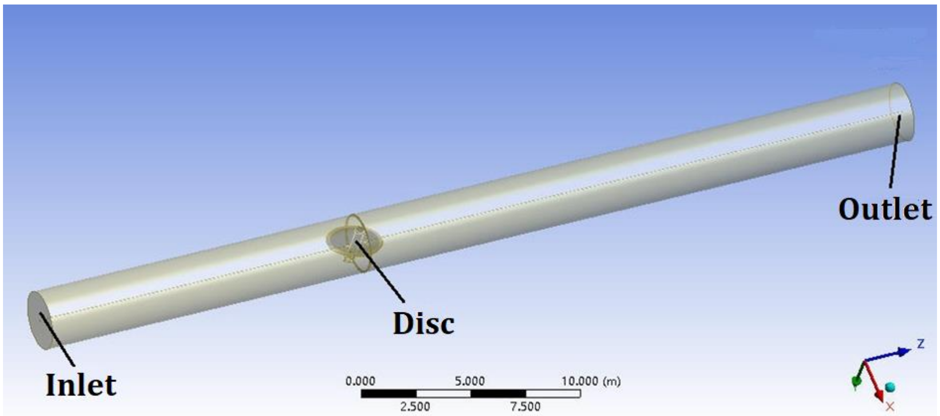

Figure 2. The three-dimensional model of flow field

4.2. Finite Element Model

The solid model was dispersed to form finite element model. Before meshing, inlet, outlet and contact surface between disc and media was selected as a set named Inlet, Outlet and FSI respectively. In meshing, the contact region between the fluid medium and the disc is the key analysis object, and the element of which was divided fine. On the premise of ensuring the accuracy of simulation analysis results, the meshing size is large in the area far away from the contact region to improve the efficiency of simulation calculation. The finite element model is shown as Figure3.

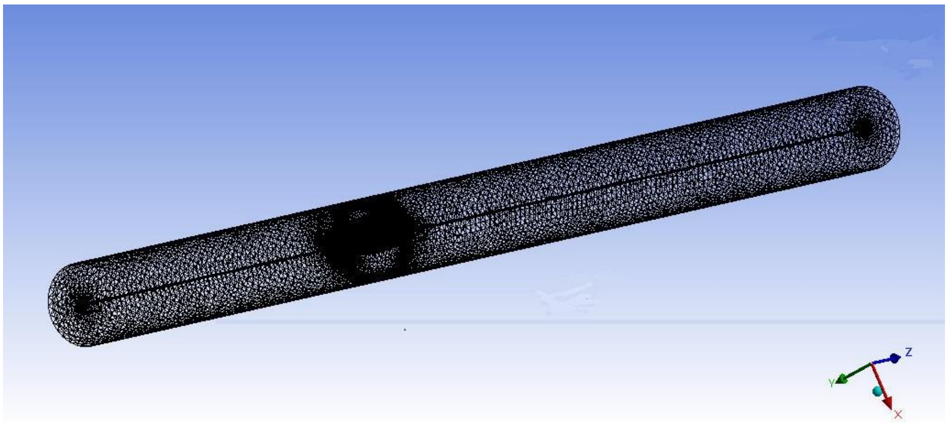

Figure 3. The finite element model

4.3. Boundary Conditions and Calculation Methods

In the boundary conditions, the inlet was set as velocity inlet while the outlet was set as pressure outlet, and the contact surface between disc and media was set as wall. According to the flow requirements of butterfly valve, the inlet velocity was set to 6m/s, and the outlet pressure was set to 0 Pa. Turbulence specifying method was set as turbulence intensity and hydraulic diameter.

In solution control, SIMPLE was selected as the solution algorithm. The spatial discrete schemes of momentum, turbulent kinetic energy and turbulent dissipation rate adopted the second upwind. The under-relaxation factors had great influences on the calculation convergence. If the value of under-relaxation factors was set too large, it would lead to non convergence of calculation results. If the value of under-relaxation factors was set too small, it would lead to slow the convergence. Therefore, the pressure was set to 0.3. Density, bulk force and turbulent viscosity were set to 1. Momentum was set to 0.7. Turbulent kinetic energy and

ISSN: 2472-3703 DOI: 10.6911/WSRJ.202008_6(8).0005

turbulent dissipation rate were set to 0.8. The simulation iterations were set to 300, and the flow rate of outlet was observed.

5. SIMULATION RESULTS AND ANALYSIS

5.1. Visual Simulation Results of Flow Field

The characteristics of fluid flow at every 5 degree ranging from 5degree to 90 degree were obtained by CFD simulation. Velocity vector diagram and pressure nephograms at three different opening state were listed and analyzed below, for original opening at 5degree, medium opening at 50degree and full opening at 90 degree, to explain the direct effect of butterfly valve opening on fluid flow.

5.1.1 Disc opening at 10 °

Figure 4 shows the fluid velocity distribution near the region of the disc at 10° opening. It can be seen from Fig.4 that when the disc is at 10° opening, the flow channel formed by the disc and the valve body surface is very small, and the flow velocity at both edges of the disc increases significantly. What’s more, there are two obvious vortices formed behind the disc.

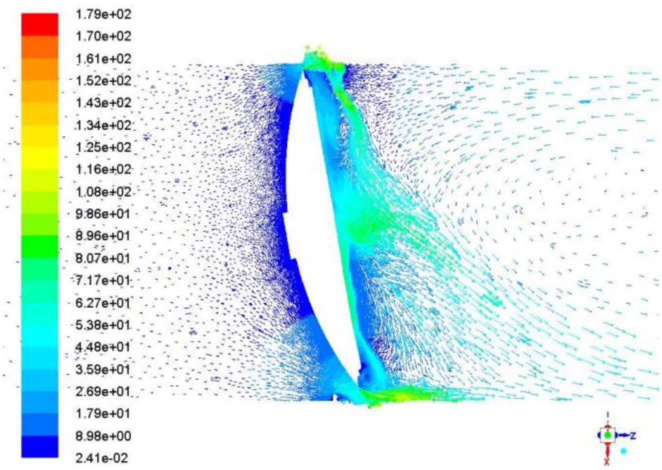

Figure 4. Velocity vector diagram at 10° opening

Figure 5 shows the pressure distribution near the region of the disc at 10° opening. The pressure in front of the valve is significantly different from the back pressure behind the valve, because only a little fluid flows into the pipeline behind the valve.

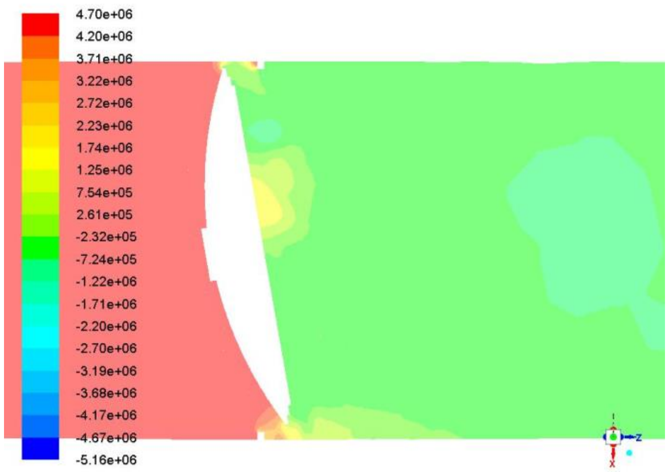

Figure 5. Pressure nephogram at 10° opening

ISSN: 2472-3703 DOI: 10.6911/WSRJ.202008_6(8).0005

5.1.2 Disc opening at 50 °

Figure 6 shows the fluid velocity distribution near the region of the disc at 50° opening. With the increase of the opening angle, the flow channel formed by the disc and the valve body surface increases, so the flow velocity at edges of the disc decreases, and vortices formed behind the disc attenuate.

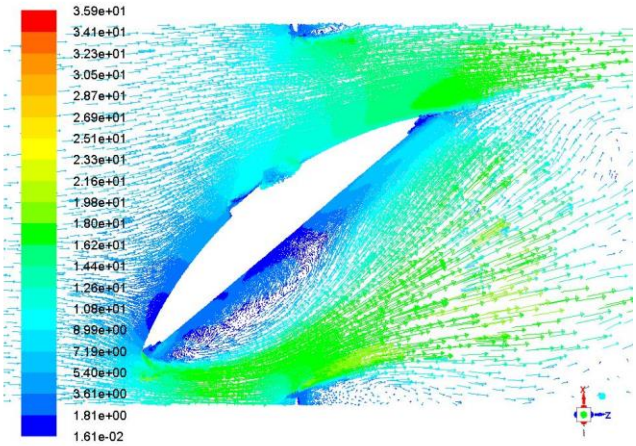

Figure 6. Velocity vector diagram at 50° opening

Figure 7 shows the pressure distribution near the region of the disc at 50° opening. With the increase of flow channel, the pressure difference before and after the valve decreases gradually, because more fluid flows into the pipeline behind the valve.

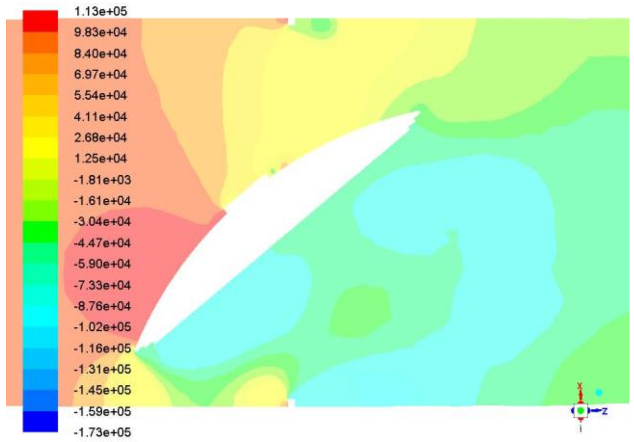

Figure 7. Pressure nephogram at 50° opening

5.1.3 Disc opening at 90 °

The disc at 90° opening means the butterfly valve is in full open position. Figure 8 shows the fluid velocity distribution near the region of the disc in the fully opening state. It can be seen from the Fig. 8 that the fluid flow is relatively stable and there is no vortex formed behind the disc. Figure 9 shows the pressure distribution in the fully opening position. At this position the pressure difference before and after the valve is the minimum.

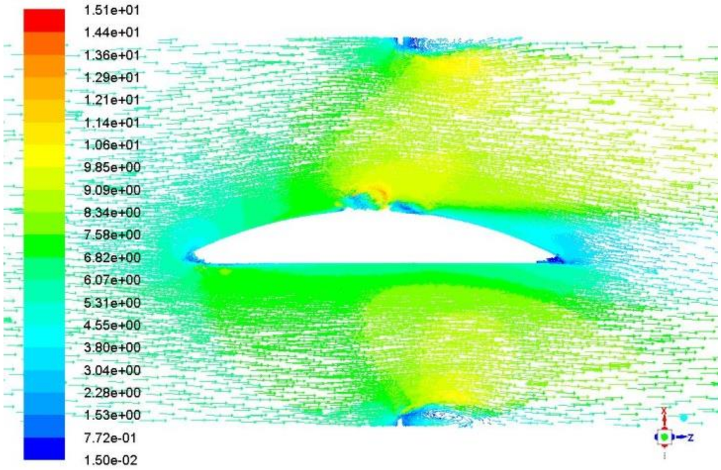

Figure 8. Velocity vector diagram at fully opening position

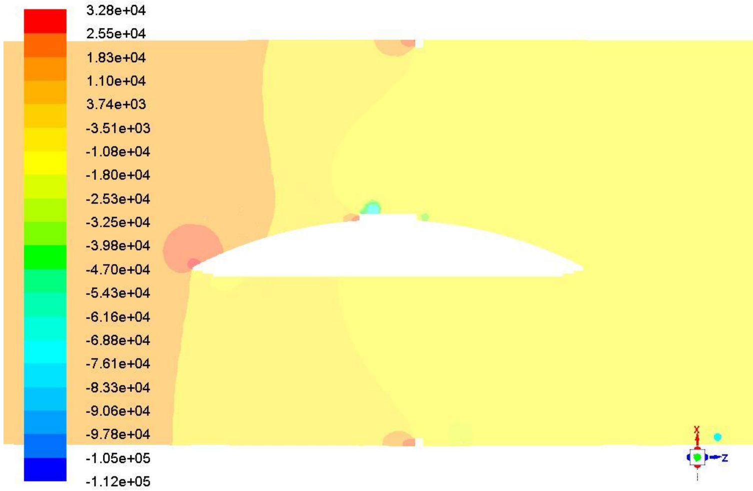

Figure 9. Pressure nephogram at fully opening position

5.2. Relationship between Pressure Difference ΔP and Opening Angle

During the fluid through the valve, there is the pressure loss of the fluid, which causes the pressure difference before and after the valve. The pressure difference before and after the valve can be applied to calculate the flow coefficient and flow resistance coefficient in formula (1) and formula (2). In the actual fluid test, the flow coefficient and flow resistance coefficient are also calculated by measuring the pressure difference before and after the valve.

In the simulation analysis of this paper, the pressure in front of the valve is obtained at 4 times of the diameter before the valve, while the pressure behind the valve is obtained at 6 times of the diameter after the valve. Pressure difference ΔP is obtained by subtracting the pressure in front of the valve by the pressure behind the valve. Table 1 shows the pressure difference Δ P at different opening. The pressure loss before and after the valve is 1.08×104pa (81mmHg). So the designed valve meets the requirements of pressure loss less than 111mmhg in the technical specification.

ISSN: 2472-3703 DOI: 10.6911/WSRJ.202008_6(8).0005

Table 1. The pressure difference ΔP at different opening

Opening/°	ΔP/Pa	Opening/°	ΔP/Pa
5	1.25×107	10	4.68×106
15	2.70×106	20	1.60×106
25	9.77×105	30	5.89×105
35	3.71×105	40	2.26×105
45	1.44×105	50	9.40×104
55	6.17×104	60	4.26×104
65	3.26×104	70	2.40×104
75	1.82×104	80	1.29×104
85	1.14×104	90	1.08×104

5.3. Flow Coefficient and Flow Resistance Coefficient

The flow coefficient C_v and flow resistance coefficient ξ can be calculated by formula (1) and formula (2) respectively by using the ΔP data listed in Table 1. The calculated results are listed in Table 2.

Table 2. Flow coefficient C_v and flow resistance coefficient ξ at different opening

Opening/°	C_v	ξ	Opening /°	C_v	ξ
5	1774.036	695.033	10	2900.925	259.9307
15	3818.365	150.0289	20	4953.223	89.1546
25	6283.229	54.2567	30	8174.746	32.7329
35	10296.68	20.6317	40	13196.54	12.5604
45	16528.65	8.0069	50	20464.54	5.2231
55	25266.6	3.4264	60	30414.29	2.3647
65	34753.2	1.8111	70	40531.73	1.3316
75	46491.54	1.0119	80	55243.33	0.7167
85	58894.36	0.6307	90	60463.56	0.5984

The flow coefficient and opening curve can be obtained by plotting the flow coefficient against the opening, as shown in Fig. 10. The flow resistance coefficient and opening curve can be obtained in the same way, as shown in Figure 11. It can be seen from Figure 10 that the value range of flow coefficient is 0-60000. As the opening of disc increases, the value of flow coefficient C_v also increases. The trend of the flow coefficient is almost the same as that given in reference [9]. However, when the opening reaches 80°, the flow coefficient fails to continue the previous trend, because the rectangular shape of the valve stem shaft hole protrudes the outer surface of the arc of the disc. And the protruded rectangular shape causes the resistance to the fluid.

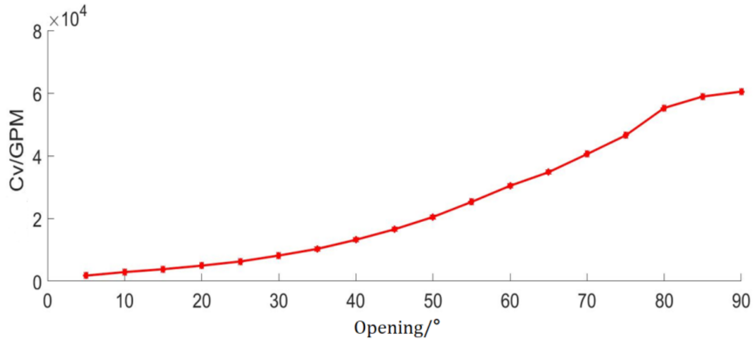

Figure 10. Flow coefficient Cv at different opening

In Figure 11, the flow resistance coefficient decreases with the increase of the opening, especially in the opening range of 5° to 25° the flow resistance coefficient decreases sharply. Flow resistance coefficient at 25° is less than one-tenth of that at 5°, and flow resistance coefficient at 50° is less than one-hundredth of that at 5°. From 50° to full opening, the flow resistance coefficient decreases slowly and the value is also very small. In the fully opening position, the flow resistance coefficient is 0.5984.

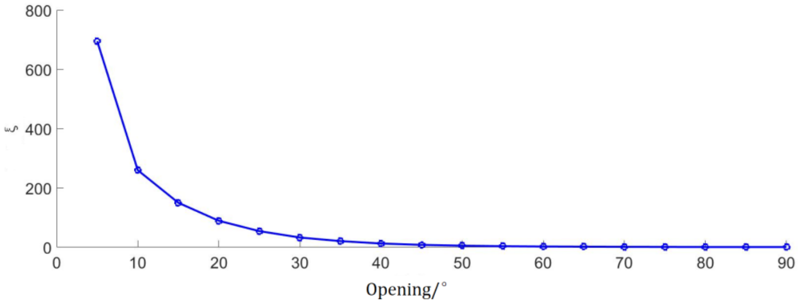

Figure 11. Flow resistance coefficient ξ at different opening

6. CONCLUSIONS

(1) The simulation model of nuclear seawater butterfly valve was constructed by CFD software, and the characteristics of fluid flow were analyzed at every 5 degree ranging from 5degree to 90 degree. The simulation results indicate that the vortex formed behind the disc attenuate with the increase of opening, and there is no obvious vortex at 65° opening.

(2) The valve pressure loss was obtained at different opening by simulation, and the pressure loss before and after the valve is 1.08×104 Pa(81mmHg) at fully opening position, and it meets the requirements of technical specifications.

(3) The relation curve between flow coefficient and opening was constructed. The results indicate that the flow coefficient increases as the opening increases. When the opening is greater than 80°, the protruded rectangular shape influences the increasing extent of the flow coefficient.

(4) The relation curve between flow resistance coefficient and opening was constructed. The results indicate that the flow resistance coefficient is 0.5984 in the fully opening position, and it meets the requirements of butterfly valve at the inlet and outlet of condenser.

ACKNOWLEDGEMENTS

This paper was supported by New Technology Application Projects for Three Major Industries of Wenzhou (WZYSDCY2018003) and Wenzhou Polytechnic Research Project (YLXM2018021).

ISSN: 2472-3703 DOI: 10.6911/WSRJ.202008_6(8).0005

REFERENCES

[1] P.F. Li, M.Y. Wang: Proficient in CFD Engineering Simulation and Case Practice (Posts and Telecom Press, China 2011), p.1-2. (In Chinese)

[2] J. Liu, F.T. Li. Numerical Simulation and Analysis on 3D Flow in Largedi ameter Butterfly Valve, Fluid Machinery, vol. 36 (2008) No. 9, p.30-32+29.

[3] X.G. Song, L. Wang, Z.H. Jiang, et al. Numerical Analysis of Flow Characteristics of Butterfly Valve prediction, Fluid Machinery, vol. 36 (2008) No. 5, p.25-29.

[4] Q.Z. Zhang, F.S. Liu, H. Wang. Study of Flow Field and Resistance Characteristics of Triple Eccentric Butterfly Valve, Fluid Machinery, vol. 41 (2013) No. 11, p.1-5+19.

[5] J.X. Dong, Q.X. Kong, T. Li. Structure Design of a Novel Butterfly Valve for Nuclear Power, Journal of Wenzhou Vocational & Technical College, vol. 18 (2018) No. 2, p.60-63.

[6] K. Ji: Research on Flow Field of Large Diameter Butterfly Valve Based on Fluent Software (M.D., Northeastern University, China 2011), p.28.

[7] Z. X. Yang, N. Yu, W. P. Mao et al. Numerical Simulation and Flow Characteristics Analysis of Center Butterfly Valve Flow Field, Chinese Hydraulics & Pneumatics, (2016) No. 1, p.95-99.

[8] W. E. Gong: Interior Flow Field Numerical Simulation and Static Analysis of Butterfly-vale and Multi-discs Valve, (M.D., Northeastern University, China 2010), p.13-14.

[9] P. W. Lu. Practical Valve Design Manual (Mechanical Industry Press, China 2012), p.27-32. (In Chinese)