Dynamic hydraulic characteristics of a prototype ball valve during closing process analysed by 3D CFD method

F F Wen, Y G Cheng and W W Meng

1 State Key Laboratory of Water Resources and Hydropower Engineering Science, Wuhan University, Wuhan, China

yghcheng@whu.edu.cn

Abstract. During the runaway process of high-head turbine-generator units due to wicket gate failure, the ball valve should be closed promptly to prevent accidents, and the dynamic hydraulic characteristics during the closing process should be studied. In this paper, the flows through a prototype ball valve are simulated by the latest 3D CFD method. First, the steady flows in different openings are calculated, and we find that the headloss coefficients agree well with experimental results. Then, the water hammer led by linear closing of the ball valve is simulated by both 3D CFD and 1D Method of Characteristic (MOC), and the reasonable agreements further validate feasibility of the 3D CFD method. After that, we investigate the evolution laws of flow patterns, pressure and forces around the valve, and find that the pressure variations before, within and behind the valve differ greatly. Besides, the unstable flow fields within the valve are essential causes of pressure and force fluctuations, bringing potential danger to the safety of ball valve operation. This research provides basic information for studying ball valve vibration and its impact on the turbine and hydraulic system.

1. Introduction

Ball valves are widely used as the main valves located before turbines in high-head hydropower and pumped-storage power stations. These ball valves are important because they are the main means to control runaway accidents after wicket gate failure [1].

The main ball valves traditionally do not play the role of adjusting flow rate. However, the situation is changing recently. Ball valves acting as assist of wicket gates for reducing waterhammer pressures [2] and stabilizing unit operations [3] were proposed. Many units in China such as Huizhou [4], Zhanghewan [5], and Heimifeng [6] have successfully applied the ball valve to regulate the load rejection process. In these practices, one found that the three-dimensional (3D) flow patterns in the valve have obvious influence on the stability of the pump-turbines [7]. Therefore, studying the hydraulic characteristics of the ball valve during closing transient process is important.

Initially, existing studies on ball valves mainly paid attention to the steady pressure-flow characteristics by model or prototype tests. Chern M. J. et al. [8] studied the resistance coefficients of ball valve for large openings by a model test, and some visual pictures of flow patterns were captured. After that, the relation curves of headloss coefficients against openings for many ball valves were obtained by numerical simulations [9-12], since computational fluid dynamics (CFD) has matured. These studies, however, simplified the ball valves into two-dimensional (2D) models and there were no model tests to verify the accuracy of the simulated results. Shirazi N. T. et al. [13] simulated the flow patterns of a ball valve in different openings and found the openings were the main influence...
factors of the flow patterns and pressure drops. These studies, however, were confined to the static hydraulic characteristics, and the dynamic characteristics during closing and opening processes, as well as the small opening characteristics were lack of researches. Cui B. L. et al. [14] studied the opening and closing processes of a ball valve, and found that the flow patterns were different even at the same opening for these two inverse transient processes. This tells us that the opening is not the only factor that influences the flow patterns and pressure fluctuations during transient processes.

In this paper, the 3D CFD simulations for both the steady-state and transient process of a prototype ball valve are reported. First, to validate the method, the headloss coefficients for 8 openings obtained by steady-state simulations are compared with the existing data. Then, the transient process induced by a linear closure of the ball valve is simulated, and the waterhammer pressure history is compared with that by the traditional 1D method. At last, the flow patterns in the ball valve and the forces on the valve during the closing process are analysed in detail.

2. Numerical models and methods

2.1. Numerical models

The prototype ball valve of a pumped-storage power station was adopted in this study. As shown in figure 1, the whole computational domain contains 4 parts: the upstream pipe (P1), the valve, the downstream pipe (P2), and a thin pipe (P3). The sudden contraction between P2 and P3 was specially imposed to replace turbine’s consumption of water head. Table 1 shows the basic parameters of these pipes. During the closing process, the valve rotates anticlockwise around the X axis. When the valve rotates 90°, the flow along the Y axis will be cut off completely.

| Geometric parameters | P1 (m) | P2 (m) | P3 (m) | Diameter of Rotating diameter of ball valve (d(m)) | Diameter of P3 (d’(m)) | Closing time (s) | Rotating speed (ω(rad/s)) |
|----------------------|--------|--------|--------|-----------------------------------------------|------------------------|-----------------|--------------------------|
| Mesh numbers (million) | P1     | P2     | P3     | Ball valve                                  | Total                  |                |                          |
|                      | 500    | 2.376  | 3.567  | 15                                           | 10                     | 0.751           | 30                       | 0.05236                |
|                      | 1.637  | 0.206  | 0.003  | 0.05                                         | 1.896                  |                |                          |                        |

Figure 1. The simplified computational domain

2.2. Mesh and monitoring planes

The domain was discretized into hexahedral mesh by ICEM CFD. The mesh is refined and improved in both quantity and quality in the three places: the valve itself, the near wall zone of all pipes, and the zone near the sudden contraction joint. Mesh independence analysis was performed by comparing five densities, and at last the case of 1.896 million mesh elements was chosen, concerning both accuracy and computational costs. With this mesh, the average Y+ valve of the whole domain for openings 70°, 40° and 20° are 135, 228 and 375, respectively, according the steady-state simulations (Section 3). Therefore, the mesh is good enough for the next simulations. Table 1 also shows the mesh elements of
different parts. To analyse the flow characteristics, five monitoring planes along the pipes and 3 points respectively on S2 and S5 were set to record the data during the simulation (figure 2).

![Figure 2. Monitoring planes](image)

![Figure 3. Mesh near the ball valve](image)

### 2.3. Boundary conditions and simulation settings

The inlet of P1 should be the reservoir boundary condition and the outlet of P3 should be the open air condition, which were specified as the Pressure-inlet 980000Pa (100m water head) and Pressure-outlet 0Pa boundaries, respectively. There are 3 pairs of interfaces in the domain, including the inlet of the valve, the outlet of the valve, and the sudden contraction joint. The left and right faces of the valve are parts of a sphere face with a radius equal to the rotating radius. The Sliding Mesh model was adopted to rotate the valve zone (figure 3). The other faces were set as the Wall boundary conditions.

The commercial software ANASY Fluent 16.0 was adopted, and 48 computational nodes of a workstation were used in simulations. The SST k-ω turbulence model was chosen. In order to calculate the water hammer, the water was set as a compressible fluid and the density was defined by UDF according to the wave speed of pipes [15]. 8s simulation was conducted first to reach a stable flow for preparing the initial condition. The ball valve closes linearly in 20s, which corresponds to the angular velocity 0.05236 rad/s of the valve. The timestep was set as 0.01s and the residual errors of all parameters are set as 1.0e-4. The maximum iteration of each timestep was set as 40, and it was found that the results can reach the convergence standard in no more than 35 iterations in every timestep.

### 3. Headloss coefficients by steady-state simulation

The headloss coefficient $\zeta$ of a ball valve is an important factor representing the flow capacity of the valve, which can be quantified as the ratio between the mechanical energy loss passing through the valve and the kinetic energy of the main flow [16]. As for the model in this paper, the local headloss coefficient $\zeta$ is defined as equation (1)

$$\zeta = \frac{H_{s2} - H_{s5}}{H_{s5}}$$  

(1)

where $H_{s2}$ and $H_{s5}$ are respectively the total water head on monitoring face S1 and S5, $H_{s5}$ is dynamic water head on S5.
The 3D CFD simulated headloss coefficients against 8 openings are shown in figure 4. The horizontal coordinate $\theta$ represents the inner angle between the central axis of valve and the pipe axis, and the angle and the opening are one-to-one corresponding. Good agreements between the CFD results and those from the empirical formula derived by Zhang Z. J. [17] and the model test by Thorley A. R. D. [18] and Hou C. S. [19] are shown. Therefore, the models, mesh, and parameters in the 3D CFD model are reliable, which lays the foundation for transient process simulations.

4. Evolution laws of pressure and force during ball valve closing process

4.1. Waterhammer pressure

Figure 5 is the waterhammer head histories during the valve closing process, showing that the 3D CFD curve matches the 1D (Impulse) curve, which is calculated by the commercial software Impulse. The maximum pressure head in 3D case (about 124.0 mH$_2$O) is 3.13% smaller than that in 1D case (about 128.0 mH$_2$O). Meanwhile when the valve closes completely, the wave fluctuating periods in 1D case (about 2.1s) and 3D case (about 2.0s) are nearly the same, which are both close to the data calculated by theoretical formula $4L/c$ (where $L$=500m as mentioned, and $c$=1000m/s is the wave speed). As to the wave fluctuating values after valve closure, the 1D case shows equal peak-to-peak values while the 3D case shows a faster attenuation. The root reason is that the reflecting factor of the valve for the 1D case is 1, and one in 3D case is smaller due to the elastic liquid model. The reflecting factor at the reservoir for the 1D case is -1, and one in 3D case is also smaller in value. Therefore, in the 3D case the energy damps when the pressure waves spread and are reflected.

Figure 4. Loss coefficients comparing with empirical and measured data

Figure 5. Waterhammer head histories at S2 plane

Figure 6. Volume flow rate histories at S2 plane
Even though the valve closes linearly, the flow rate $Q$ decreases actually non-linearly and at $t=27.7s$ the flow rate $Q=0$ m$^3$/s. Thus the period between $t=0.0s$ to $t=27.7s$ is divided into three equal stages to analyze the flow characteristics (figure 6). During the early stage the flow rate $Q$ in two cases is similar because the opening in the early stage remains large and the flow in the valve is relative smooth. During the middle stage, the flow rate in 1D case decreases more quickly than that of 3D case, which corresponds to the fact that the pressure increases more obvious in 1D case (figure.5). When the late stage comes, the flow rate drops drastically to zero. The flow rate decreases about 70.07% and 88.55% in 1D case and 3D case separately.

4.2. Pressure and force characteristics

The changing laws of the pressure at monitoring planes S2, S3, and S4 are shown in figure 7, which represents the pressure at the valve front, in the valve itself, and at the valve back, separately. It is found that the maximum pressure at the valve front is 124.0 mH$_2$O at $t=25.1s$, which is about 1.26 times of the initial water head, which shows there exists the possibility of structural damage at the valve front. Meanwhile the minimum pressure at the valve back is -4.35 mH$_2$O at $t=26.5s$, which shows there exists the possibility of cavitation and cavitation damage at the valve back. The time of these two peak values is very close, thus these two potential damages should be considered in valve design. When the valve closes completely, the pressure at the valve front damps gradually to the pressure at the pipe inlet (100 mH$_2$O), while the pressure at the valve back damps drastically to zero. As for the pressure in the valve itself, because the pressure in the valve is influenced by the pressure in the valve front, the pressure at S3 decreases slowly to 46.2 mH$_2$O compared to the pressure drop at S2.

The changing laws of the forces and moment that the valve bears are shown in figure 8. The force along the flow direction (Y axis) Fy is much larger than Fx. The maximum value of Fy is about 210.9kN at $t=19.7s$, while the maximum value of Fx is only 9% of Fy-max. When the ball valve rotates along X axis, the side wall of the valve bears the impact of the water upstream gradually, which leads the forces along Y axis increases. The main force along Z axis Fz is gravity. At the beginning Fz is about 117.0kN, which is equal to the hydrostatic gravity of the water in the valve and all the gravity is born by the side wall of the valve. However, when the valve rotates, the gravity of the water in the valve is partly born by the interface, which is not considered in the Fluent. In the late stage, the reason why all the forces show fluctuation characteristics is that when the opening of the valve becomes small the flow patterns in the valve are quite turbulent and unbalanced. When the valve closes completely, the water in all pipes as well as in the valve becomes stationary, thus Fx and Fy becomes zero. The gravity along Z axis remains the same of the initial value, the only difference is that the face of the valve that bears gravity changes from side wall to interface. As for the moment M, the changing law of moment is similar to that of Fy, and it tends to accelerate the valve closure. However, the maximum value of moment is $t=21.1s$ when $M=195.6$kN.m, which is not the same time as the maximum value of Fy. It indicates that the force on flow direction is the main but not the only factor that determines the moment that the valve bears.
Figure 7. Pressure head histories at S2, S3, and S4

Figure 8. Histories of Force in X, Y and Z directions and torque on the ball valve

5. Correlation between the flow evolution and force of ball valve

The forces of ball valve are closely related to the flow behaviour. By analysing the velocity field, the evolution laws of flow patterns in the middle cross section of the valve were obtained (figure 9). Before 70°, the flow field is mainly divided into three parts: low velocity zone near the exit, high velocity zone near the entrance and the ambient medium velocity zone. These zones are bilaterally symmetric and there are vortices generated at low velocity zone. After 60°, another high speed zone arises in the centre of the top. Its area increases with the opening decreasing, but is smaller than that of bottom high velocity zone, which is due to energy conversion. When the flow traverses the ball valve, part of kinetic energy transforms into potential energy, which leads to the decrease of kinetic energy at exit. Therefore, we can conclude that the flow patterns within ball valve keeps steady laterally at early and middle stage, which validates the previous analysis of force in X direction.

When the ball valve closes to 30°, the internal flow field falls into instability. This phenomenon predominantly stems from the intrinsic characteristics of the fluid. At this stage, the rate of flow is high, which means the liquid has high Reynolds number. The energy conversion between the entrance and exit is rapid and violent, and many disturbances exist in the flow field. However, the viscosity of water is much less than inertia, unable to suppress the effects of these disturbances, causing the instability. The symmetry of the top vortex structure is broken and deforms or even disappears. The low velocity zone expands and extends to the two sides of the high speed zone, which not only increases the transverse velocity gradient at the bottom, but also contributes to new vertex structure, exacerbates the flow field. Obviously, in this process, the force of the ball valve in X direction constantly fluctuates due to the unsteadiness until the ball valve completely closed, which also validates the previous analysis.
6. Flow patterns around the ball valve

By comparing flow patterns obtained by CFD simulation with the steady flow provided by model test (figure 10), it indicates that the flow patterns during ball valve dynamical closing has similarity with the steady flow at corresponding openings, especially for the recirculation area and backflow length behind the valve. Therefore, it can be considered that the flow field obtained by numerical simulation is accurate.

The evolution process of flow field under eight opening are shown in figure 11. Apparently, the velocity at entrance and exit of the valve increases as opening decreases. Meanwhile, the eddy flow emerges at the top and bottom of the valve. This is because, on the one hand, the water in valve cannot discharge completely out. Part of water strikes the bottom wall of the valve and reflects back, other part follows the main stream until approaching the exit of the valve, where the flow strikes the top wall again and also reflects back. The backflow is carried by the main flow once more until discharges out of the valve. Such reduplicative relative motion activates the vortex structure in flow field.

There is still a vortex structure behind the valve. When the ball valve closes to a certain degree, the outflow of valve is equivalent to jet flow due to the mutational bottom, which is largely dominated by gravity and tends to move downwards. After the flow strikes the bottom of P2, it bounces off, and flows up. Soon afterwards, part of rebounded water flows back, motivating vortices, and the other part is merged by the main stream. With the opening of ball valve getting smaller, the backflow length increases instead, which demonstrates that the energy dissipation gets more intense.

Chern M. J.[8] and Shirazi N. T.[13] respectively investigated the steady flow under large openings by experimental and numerical simulations, and found that there were mainly three vortices in flow field, two within the ball valve, one behind the ball valve, which is the same as the results of this paper. But there was no more research about small openings. In fact, the flow patterns at small opening degrees are much more severe than that at large degrees, and has greater influence on the hydraulic system, which needs to be further studied.

After the opening decreases to 30°, new vortex structure emerges at the top of P1 before the ball valve, and becomes larger with time. This is also resulted from the backflow. When the opening arrives to 20°, another new vortex structure appears at the bottom of P1, and its recirculation length increases with time. When the valve is nearly closed, the bottom flow of P1 is congested high, and is pushed back due to the blockage of the forward valve entrance, producing the vortex structure. Therefore, there are totally five vortex structures during ball valve closing, not as simple as the previous researchers found.
Figure 10. Comparisons between the simulated flow patterns and the test results

Figure 11. Velocity streamlines at different openings.

7. Conclusions
In this paper, the dynamic hydraulic characteristics of a ball valve during its linearly closing process are analysed based on 3D CFD simulations. The evolution laws of pressure, forces, flow patterns and the relationship among them are provided in detail. The conclusions are as follows:

- The 3D CFD can accurately simulate the dynamic hydraulic characteristics of a ball valve.
- As the ball valve closes, the pressure before valve increase firstly then decrease, and the pressure behind the valve is in opposite trend. But the pressure within the valve decreases continuously to a fixed value until the valve closes completely, in which the decreasing rate is slower than that behind the valve.
- The valve bear is mainly on flow direction, which also dominates the moment of valve. The transverse force is much smaller, and the longitudinal force is mainly gravity. The extrem
unsteadiness of flow field at late period is the vital factor for force fluctuation, which has huge impact on instability of hydraulic system.

• During ball valve closing, energy transformation and dissipation is always existing, and turns to be increasingly severe until the valve completely closes, which is the immanent reason for flow instability. The flow patterns behind the valve are much more turbulent than those before the valve, and directly influence the stability of volute inflow.

• Since inlet valve throttling is quite effective to control the instability of hydraulic system in the engineering, it’s mechanism is still to be cleared. Besides, the specific ways of control and the extent of influence on pump turbine stability are also to be detailed, which need to be done in the future based on this paper.

Acknowledgement
This work was supported by the National Natural Science Foundation of China (Grant No. 51579187).

References
[1] Chern M. J., Wang C. C., Ma C. H. Performance test and flow visualization of ball valve[J]. Experimental Thermal & Fluid Science, 2007, 31(6):505-512.
[2] Liu H. P., Chen F., and Ma B. Numerical simulation of flow field in the valve based on the moving mesh and the UDF[J]. Turbine Technology. 2008, 50(2):106-108.
[3] Zhang Y. X., Li Z. J., and Zhang X. Flow field simulation analysis of the ball valve pipe under different opening[J]. Fluid Machinery. 2016(10):55-58.
[4] Yang G. Q., and Li Z. P. Simulation and research on internal flow fields of ball valve based on fluent software[J]. Mechanical Science and Technology for Aerospace Engineering, 2014, 33(12).
[5] Zhang S. C., Zhang Y. L. Fang Z. M., and Ke Y. L. 3D flow numerical simulation of ball valve based on CFD[J]. Journal of Drainage and Irrigation Machinery Engineering, 2013, 31(2):157-161.
[6] Shirazi N. T., Azizyan G. R., Akbari G. H. CFD analysis of the ball valve performance in presence of cavitation[J]. Life Science Journal, 2013, 9(4):1460-1467.
[7] Cui B. L., Lin Z., Zhu Z. C., et al. Influence of opening and closing process of ball valve on external performance and internal flow characteristics[J]. Experimental Thermal & Fluid Science, 2017, 80:193-202.
[8] Zhang X. X, Cheng Y. G., et al. Simulating of the load rejection transient process of a Francis turbine by using a 1D-3D coupling approach, Journal of Hydrodynamics, 2014, 26(5):715-724.
[9] Liu J. J., and Zhang B. H. Fluid mechanics[M]. Beijing: Peking University Press, 2006.
[10] Zhang Z. J Transient Flows in a Pipe System with Pump Shut-Down and the Simultaneous Closing of a Spherical Valve[C]// IOP Conference Series: Earth and Environmental Science. IOP Conference Series: Earth and Environmental Science, 2016:052001.
[11] Thorley A. R. D. Fluid Transients in Pipeline Systems [M]. John Wiley and Sons Ltd, 2004.
[12] Hou C. S., and Cheng Y. G. Optimized closing procedures of wicket gate and ball valve for high head reversible pump-turbine unit[J]. Engineering Journal of Wuhan University, 2005, 38(3):59-62.
[13] Wu J. B., Piao C. G. The pump turbine inlet spherical valve technology development of HEC [J]. Large Electric Machine and Hydraulic Turbine, 2014(3):77-80.
[14] Zhou D. Y., Ren T. Z., Deng L., and Zhou P. CFD-based flow characteristics research of main valve in pumped-storage power station[J]. Water Power, 2015(3):53-55.
[15] Dörrfler, P. K., Engineer, A. J., Pendse R. N., et al. Stable operation achieved on a single-stage reversible pump-turbine showing instability at no-load[C]// Iahr Section Hydr. Machinery and Cavitation, Symposium. 1998.
[16] Lin K, Jiang Y. J. Test and research on spherical valve dynamic closing of Huizhou
Pumped Storage Power Station[J]. HUNAN ELECTRIC POWER, 2011, 31(4):23-26.

[17] Tang Y. J., Zhang F., Deng L., and Zhou P. Spherical valve dynamic closing test analysis of Zhanghewan pumped storage power station[J]. Water Resources and Power, 2016(11):165-167.

[18] Wang J., Cao C. Y., He Z. H., and Jiang J. C. Ball valve dynamic closing test analysis of Heimiifeng pumped storage power station[J]. Mechanical & Electrical Technique of Hydropower Station, 2016, 39(B12):50-53.

[19] You J. F. Cavitation characteristics and transient processes of a pump-turbine analyzed by three-dimensional simulation[D]. Wuhan University, 2017.

[20] Chern M. J., Wang C. C., Ma C. H. Performance test and flow visualization of ball valve[J]. Experimental Thermal & Fluid Science, 2007, 31(6):505-512.

[21] Liu H. P., Chen F., and Ma B. Numerical simulation of flow field in the valve based on the moving mesh and the UDF[J]. Turbine Technology. 2008, 50(2):106-108.

[22] Zhang Y. X., Li Z. J., and Zhang X. Flow field simulation analysis of the ball valve pipe under different opening[J]. Fluid Machinery. 2016(10):55-58.

[23] Yang G. Q., and Li Z. P. Simulation and research on internal flow fields of ball valve based on Fluent software[J]. Mechanical Science and Technology for Aerospace Engineering, 2014, 33(12).

[24] Zhang S. C., Zhang Y. L. Fang Z. M., and Ke Y. L. 3D flow numerical simulation of ball valve based on CFD[J]. Journal of Drainage and Irrigation Machinery Engineering, 2013, 31(2):157-161.

[25] Shirazi N. T., Azizyan G. R., Akbari G. H. CFD analysis of the ball valve performance in presence of cavitation[J]. Life Science Journal, 2013, 9(4):1460-1467.

[26] Cui B. L., Lin Z., Zhu Z. C., et al. Influence of opening and closing process of ball valve on external performance and internal flow characteristics[J]. Experimental Thermal & Fluid Science, 2017, 80:193-202.

[27] Zhang X. X, Cheng Y. G., et al. Simulating of the load rejection transient process of a Francis turbine by using a 1D-3D coupling approach, Journal of Hydrodynamics, 2014,26(5):715-724.

[28] Liu J. J., and Zhang B. H. Fluid mechanics[M]. Beijing: Peking University Press, 2006.

[29] Zhang Z. J Transient Flows in a Pipe System with Pump Shut-Down and the Simultaneous Closing of a Spherical Valve[C]// IOP Conference Series: Earth and Environmental Science. IOP Conference Series: Earth and Environmental Science, 2016:052001.

[30] Thorley A. R. D. Fluid Transients in Pipeline Systems [M]. John Wiley and Sons Ltd, 2004.

[31] Hou C. S., and Cheng Y. G. Optimized closing procedures of wicket gate and ball valve for high head reversible pump-turbine unit[J]. Engineering Journal of Wuhan University, 2005, 38(3):59-62.