Analysis on flow induced vibration of main steam bypass valve in a nuclear power plant

Junfeng Liu*, Xudong Chen, Chuangao Han, Xiaolong Ma, Fangjie Wu
Xi’an Thermal Power Research Institute Co., Ltd, Xi’an, China

*Corresponding author e-mail: liujunfeng@tpri.com.cn

Abstract. Based on the fact that the main steam bypass system of a nuclear power plant vibrates violently during operation, the failure of couplings and other parts of the bypass valve occurs many times, which seriously affects the safe operation of the unit. According to bypass discharge system, and the bypass valve related drawings bypass valve fluid dynamics model is established, on the opening of the bypass valve interior flow field in CFD numerical simulation analysis, combining with the field vibration test results, for the opening of the bypass valve interior flow field in CFD numerical simulation analysis and vibration mechanism research, found that the structure of the piping system vibration and vortex shedding in the valve to inspire the chattering of the valve stem and even the axis of the main causes of fracture and valve is the high degrees of vortex-induced resonance is the main reason of the turbulent flow hood.

Key words: the bypass valve; flow field dynamics simulation; fluid-induced vibration; vortex-induced resonance.

1. Introduction
The main steam bypass system of a nuclear power plant mainly discharges the excess main steam into the condenser under the starting and stopping working conditions and abnormal transient conditions (turbine unit trip, load rejection and load step) to avoid the over-temperature and overpressure of the main steam pipeline during the operation of the unit. Among them, the main steam bypass valve is an important equipment of the bypass discharge system, and the main steam flow is regulated by controlling its opening change [1-6]. In this paper, under the transient operation condition of a nuclear power plant bypass valve, the valve and its pipeline vibrate frequently, which finally leads to the fracture of limit indicator, coupling and other parts. In this paper, the vibration characteristics of bypass valve under different opening degrees are tested in the field, and the mechanism of flow induced vibration of bypass valve and its preventive measures are revealed based on the fluid dynamics simulation model.

2. Vibration test
In order to analyze the source and characteristics of the vibration of the bypass valve, a triaxial acceleration sensor was arranged on the inlet pipe (measuring point P1V1), valve body (P1V2) and coupling (P1V3) respectively for synchronous vibration measurement. The layout of each measuring point is shown in FIG. 1.
The test process is divided into several steps: (1) Quick opening test: the valve is fully open within 5s from the closed state, and then the opening is adjusted by 10% reduction; (2) Slow opening test: 0% ~ 100% slow opening operation was performed at a range of 10%.

3. Hydrodynamic analysis
Computational fluid Dynamics (CFD) can be applied to various discrete mathematical methods, numerical experiments, computer simulation and analytical research on various problems of convection physics to solve various practical problems. It has been widely used in engineering and scientific research because of its advantages of simulating complex processes, visualization and high resolution accuracy. Its basic principle is to numerically solve the differential equation controlling the fluid flow and obtain the discrete distribution of the fluid flow field in the continuous region, so as to approximately simulate the fluid flow situation. Scholars at home and abroad have applied CFD to the thermal hydraulic numerical simulation and calculation of nuclear power plants and obtained a large number of research results [7-10].

3.1. Calculation model
According to the design materials and field measurements, the two-dimensional fluid dynamics model of the valve is established. The two-dimensional fluid dynamics model established is shown in Figure 2. The entire calculation area of the model is composed of the valve area and the external pipeline. The steam flows from the inlet to the valve and flows into the condenser through the regulating action of the valve core. In the modeling process, the outer outlet pipe length is greater than 5 times of the pipe diameter, and the valve cage and turbulence cover have been reasonably equivalent according to the flow area. It is found that the influence of pilot valve on bypass valve is negligible. On the one hand, in the analysis, the pilot valve is in the fully open state, the cross section area of the flow passage in the valve is similar to the cross section area of the pipeline, and the fluid flow state is basically unchanged. The pilot valve, on the other hand, is a gate valve, and the fluid flows out of the pilot valve and through a section of pipeline adjustment into the control valve. Therefore, in the fully open state, the influence of pilot valve on control valve can be ignored.
3.2. Initial condition setting
The two-dimensional calculation model of bypass valve was introduced into FLUENT, and the pressure correction method was used for numerical solution in the unstructured grid. The inlet was set as the pressure inlet boundary, and the average pressure was 9.9mpa. The outlet was set as the pressure outlet boundary, and the outlet pressure was set as 8.9mpa according to the velocity value obtained by three-dimensional calculation. Simulation results under different operating conditions are obtained by changing different opening degrees.

4. The simulation results

4.1. Velocity distribution at different valve openings
The variation characteristics of flow rate and spool pressure with opening are shown in Figure 3 and Figure 4.

![Fig. 3 Relation curve of valve inlet opening degree.](image)

![Fig. 4 Relation curve between spool velocity and pressure and opening degree.](image)

It can be seen from FIG. 3 and 4 that the inlet flow velocity of bypass valve is 49.32m/s under the maximum opening, and the regulation of flow is close to linear. Spool pressure fluctuates with the increase of opening, and has a minimum value at 30% opening, and the overall trend rises.

4.2. Pressure distribution at different valve openings
Select the bypass valve opening of 60% and 100% as simulation test nodes, and get the bypass valve pressure cloud diagram under different opening degrees as shown in Figure 5 below.
As can be seen from Figure 5, the position of the valve cage and spool is the area where the pressure changes greatly, the pressure inside and outside the valve cage changes greatly under each opening, and the pressure in the inlet and outlet pipelines changes little. With the increase of the opening, the flow area of the valve increases, and the pressure inside the valve cage and turbulent hood increases.

4.3. Velocity profile at different valve openings
Select the bypass valve opening of 60% and 100% as simulation test nodes, and get the bypass valve velocity cloud diagram under different opening degrees as shown in Figure 6 below.

As can be seen from Figure 6, the position of the valve cage and spool is the area where the speed changes greatly, the speed on both sides of the valve cage changes greatly at each opening, and the speed in the inlet and outlet pipelines changes little. As the opening becomes larger, the valve flow resistance decreases, the valve outlet velocity increases, and the inhomogeneity of the valve inlet velocity increases.

The valve cage position, near the lower opening flow rate is relatively high, more vulnerable to erosion damage. At a 20% opening, the fluid on both sides of the spool is wall-mounted, and the fluid on both sides of the spool meets at the turbulence dome. As the opening becomes larger, the bypass valve flow area increases, the fluid intersection position at the lower part of the valve core gradually rises, and the high-speed intersection collision of fluid at the lower part of the valve core is enhanced. It can be seen that with the increase of the opening, the increase of fluid turbulence on both sides of the valve core and the high speed intersection collision are the important causes of valve vibration.

4.4. Analysis of flow induced vibration mechanism of bypass valve
Through the analysis of the above calculation results, the main mechanism of flow induced vibration of bypass valve is obtained as follows.

(a) the opening of the 60%                 (b)  the opening of the 100%
Fig. 5 Pressure cloud diagram of bypass valve at each opening

(a) the opening of the 60%                 (b)  the opening of the 100%
Fig. 6 Bypass valve velocity cloud diagram at each opening

4.3. Velocity profile at different valve openings
Select the bypass valve opening of 60% and 100% as simulation test nodes, and get the bypass valve velocity cloud diagram under different opening degrees as shown in Figure 6 below.

As can be seen from Figure 6, the position of the valve cage and spool is the area where the speed changes greatly, the speed on both sides of the valve cage changes greatly at each opening, and the speed in the inlet and outlet pipelines changes little. As the opening becomes larger, the valve flow resistance decreases, the valve outlet velocity increases, and the inhomogeneity of the valve inlet velocity increases.

The valve cage position, near the lower opening flow rate is relatively high, more vulnerable to erosion damage. At a 20% opening, the fluid on both sides of the spool is wall-mounted, and the fluid on both sides of the spool meets at the turbulence dome. As the opening becomes larger, the bypass valve flow area increases, the fluid intersection position at the lower part of the valve core gradually rises, and the high-speed intersection collision of fluid at the lower part of the valve core is enhanced. It can be seen that with the increase of the opening, the increase of fluid turbulence on both sides of the valve core and the high speed intersection collision are the important causes of valve vibration.

4.4. Analysis of flow induced vibration mechanism of bypass valve
Through the analysis of the above calculation results, the main mechanism of flow induced vibration of bypass valve is obtained as follows.

(a) the opening of the 60%                 (b)  the opening of the 100%
Fig. 5 Pressure cloud diagram of bypass valve at each opening

(a) the opening of the 60%                 (b)  the opening of the 100%
Fig. 6 Bypass valve velocity cloud diagram at each opening

As can be seen from Figure 6, the position of the valve cage and spool is the area where the speed changes greatly, the speed on both sides of the valve cage changes greatly at each opening, and the speed in the inlet and outlet pipelines changes little. As the opening becomes larger, the valve flow resistance decreases, the valve outlet velocity increases, and the inhomogeneity of the valve inlet velocity increases.

The valve cage position, near the lower opening flow rate is relatively high, more vulnerable to erosion damage. At a 20% opening, the fluid on both sides of the spool is wall-mounted, and the fluid on both sides of the spool meets at the turbulence dome. As the opening becomes larger, the bypass valve flow area increases, the fluid intersection position at the lower part of the valve core gradually rises, and the high-speed intersection collision of fluid at the lower part of the valve core is enhanced. It can be seen that with the increase of the opening, the increase of fluid turbulence on both sides of the valve core and the high speed intersection collision are the important causes of valve vibration.

4.4. Analysis of flow induced vibration mechanism of bypass valve
Through the analysis of the above calculation results, the main mechanism of flow induced vibration of bypass valve is obtained as follows.
1) There is an obvious boundary layer separation phenomenon in the flow field of the valve, the main flow form is turbulence, and the fluid load acting on the inner wall of the valve body has a significant random broadband characteristic.

2) According to the test results, low-frequency vibration around 3.2Hz exists in the pipeline, valve body and coupling in front of the valve; In the high frequency part, the main vibration frequency of the three positions is different, which has obvious local resonance characteristics.

3) The first natural frequency of the pipeline is about 3.5Hz, and the modal shape is bending vibration, and the maximum deformation is the position of the valve body, which is close to the low-frequency vibration test frequency of 3.2Hz; the fourth natural frequency is about 22.9Hz. The modal vibration mode is mainly the axial direction vibration, and the maximum deformation is the front elbow position of the pilot valve, which is close to the low-frequency vibration test frequency of 25.5Hz. It is shown that the frequency is caused by the excitation of the first and fourth modes of the pipeline. 130 ~ 280Hz is the main natural frequency range of the valve stem and coupling, which is close to the excitation frequency of the vortex shedding of the valve cage, thus causing significant vibration of the valve stem and coupling. 400 ~ 550Hz is the circumaxial shell mode frequency of the pipeline in front of the valve. Under the action of fluctuating pressure and turbulent fluid excitation force, this local mode is excited. The maximum deformation position of corresponding mode mode mode is mainly concentrated in the pipeline in front of the valve, which is consistent with the vibration test results. In the modal calculation, the first two natural frequencies of the turbulence cover are 2089Hz, and the boundary layer separation and vortex desorption frequencies are also near this frequency, and they appear prominently in the vibration test data. It is also confirmed that vortex induced resonance is the main cause of the fatigue failure of the turbulence cover. In addition, the frequencies of 634Hz and 1100 ~ 1300Hz in the test data should be caused by the excitation of the natural frequency of the shell parts in the valve.

4) When the valve has a small opening, the flow field in the valve cage has an obvious vortex phenomenon, and its frequency is about 100 ~ 300Hz. Boundary layer separation and turbulence induced forced vibration of the valve body structure or part of the structure, and the vibration of the valve body was mainly buffeting. When the valve opening increases to about 40% or more, the fluid vortex phenomenon in the turbulent hood is significantly enhanced, and the alternating fluid load generated by the alternating Karman vortex shedding is coupled with the fundamental frequency (2089Hz) of the turbulence hood to generate vorticity resonance.

5) Vortex induced resonance is the main cause of turbulence dome failure. At a large opening, that is, more than 40% of the opening, the boundary layer separation and vortex shedding in the flow field at the turbulent cover position are significantly enhanced, and the periodic shedding of the Carmen vortex causes the structural resonance of the turbulence cover. In the case of resonance, the fluid excitation force is the same as the fundamental frequency lock, the phase is the same, the turbulence mask amplitude is larger, and the fatigue failure occurs.

6) Under the influence of saturated steam, liquid droplets or liquid mist may exist in pipelines and valves. Liquid droplets or liquid mist contained in saturated steam bound by high speed steam will have an impact on the turbulence mask, which is also an important inducement of turbulence mask damage. However, compared with vortex-induced resonance, this factor is secondary. Because the valve cage which is also likely to be subjected to the liquid shock is basically intact, and the turbulence cap whose stiffness is not significantly different from it is completely destroyed, indicating that it is difficult to destroy the turbulence cap simply due to the liquid shock, even if the lower turbulence cap is subjected to a larger impact. In addition, physical observation shows that the lower part of the turbulence cap is all damaged, rather than the position of the main inlet of the downstream pipeline with the largest liquid impact.

5. Conclusion
The problem of pipeline vibration of bypass valve is caused by the joint action of the pipeline structure vibration and the vibration caused by the flow in the valve. The fracture of the valve stem and shaft
coupling is caused by the combined action of the structural vibration of the pipeline system and the chattering induced by the vortex shedding in the valve. Turbulent hood failure is due to fatigue failure due to vorticity resonance combined with liquid impact at high valve opening.

The significant turbulence and boundary layer separation characteristics in the valve and nearby pipelines excite the high frequency local modes in the pipeline and the low order modes of the valve internal parts, which are the main frequencies in the vibration test.

References

[1] Feng Q, Yao GN. Hardware Modification and Control logic Optimization of steam turbine exhaust valve in Unit 3 of Tianwan Nuclear Power Plant, Industrial & Science Tribune, 2019, 18(8): 45-46.

[2] Liu JF, Ma XL, Han CG and Dong L. The Optimization for Bypass Control Mode in Start-up and Shutdown System of HTR-PM, Turbine Technology, 2019, 61(6): 413-416, 464.

[3] Li YJ. Causes Analysis and Transformation Suggestions of Turbine Bypass Steam Valve Internal Leakage, Power Station Auxiliary Equipment, 2019, 40(4): 35-38.

[4] Guo F, Guo XY, Zhang B and Zhang ZL. Cause Analysis and Renovation of High-pressure Bypass Valve Leakage for a MW Subcritical Unit, Zhejiang Electric Power, 2017, 36(5): 45-47, 61.

[5] Tang B, Lu YA, Lu YW and Cai BG. Restoration of Main Valve Seat of HP Bypass in a Coal-Fired USC Power Unit, Electric Power, 2015, 48(7): 68-71.

[6] Hao HR, Yang XY, Wang J. Influence of Bypass Valve Regulation on Transient Characteristics of Closed Brayton Cycle Coupled with HTR, Atomic Energy Science and Technology, 2016, 48(4): 612-620.

[7] Sui ZG, Yang J, Yang Y, Dong SH and Xu LJ. Simulation of Reflux in U-Tube of Steam Generator Based on CFD Method, Nuclear Power Engineering, 2019, 40(2): 10-15.

[8] He CH, Lu XN, Wang Q. Research on flow characteristics of nuclear main pump based on CFD simulation, Pump Technology, 2016, (5): 25-28.

[9] Wang Q, Li TB, Lu XN. Flow characteristics of CAP1400 core main pump based on CFD simulation, General Machinery, 2016, (10): 52-55.

[10] Ji Y, Li JZ, Hao Y, Li YG and Wang C. Structure Optimization and CFD Simulation of Passive Autocatalytic Recombiners in Nuclear Power Plants, Energy Conservation Technology, 2014, 32(3): 205-209.