Evaluation of self-induced oscillation of the flow control valve by fluid structure interaction analysis

Shinji Fukao¹, Kensuke Futahasi¹, Ryuichi Umehara¹ and Takeo Baba¹

¹Research & Innovation Center, Mitsubishi Heavy Industries, Ltd.
2-1-1 Shinshama Arai-cho Takasago, Hyogo, Japan

E-mail: Shinji_fukao@mhi.co.jp

Abstract. In the flow control valve used in the upper or the lower stream of a turbomachinery, when opening is small, the self-induced oscillation resulting from a clearance gap flow may occur. In this study, the mechanism of self-induced oscillation which occurs on the small valve opening condition was examined. And a concept to control the self-induced oscillation was built. The air model test of the flow control valve was carried out, and it was confirmed that self-vibration occurs under specific conditions. Moreover, stability evaluation using CFD was carried out, and it checked that the calculation results become the almost same tendency as a test results. The area where the instability force is generated on a valve was grasped from the simulation results, and it checked that stability was changed with flow field differences. Based on these results, stabilization of the flow control valve was attained by improving clearance gap geometry of the valve sheet area. It checked that the intended flow field was obtained in the geometry in CFD analysis, and it confirmed that fluid additive damping became positive in CFD.

1. Introduction

Flow control valve used in the upper or the lower stream of a turbomachinery to adjust the flow rate. The flow control valve acts very important role of adjusting output of power and shut-off. On the other hand, vibration problems often occur with the flow control valve. Suzuki et al.[1] reported that large fluid force is generated in the valve as forced vibration due to changing flow separation point under certain conditions. Moreover, there are many reports which a self-induced oscillation generates in a flow control valve on the small valve opening.[2],[3],[4] The self-induced oscillation causes fatal damage to apparatus. For evaluation of the self-induced oscillation, stability analysis using CFD is often used in many fields. It is used in estimation of the blade flutter [5],[6],[7] and the aircraft wing flutter [8] in addition to the flow control valve oscillation. Although the products are different, they are all aimed at evaluating self-induced oscillation due to fluid negative damping. In this study, the self-induced oscillation of the flow control valve was evaluated using CFD analysis, and the results were compared to the experiment. We clarified the mechanism of the self-induced oscillation and devised a concept to suppress it.
2. CFD analysis

For the evaluation of self-induced oscillation by simulation, fluid-structure interaction analysis what is called two-way coupled analysis is sometimes used, but one-way coupled analysis is often used when dealing with simple modes or small vibration linear problems. Since the calculation time of one-way coupled analysis much shorter than two-way coupled, it is useful as a design evaluation method. In this study, the self-induced oscillation evaluation of the flow control valve was carried out by one-way coupled analysis with CFD. A valve that generates self-induced oscillation by one-way coupled analysis was designed to elucidate the mechanism although it is different from the shape used in the actual machine. The valve shape is shown in figure 1. It has a simple shape. The flow direction is from the bottom to the side.

The evaluation method of self-induced oscillation using CFD is described. A forced vibration displacement is applied to the valve body using a specific vibration mode as an input condition in CFD calculation. The reproduction of the vibration mode used the mesh morphing method. As the vibration mode, the first bending mode was set as the input condition. The vibration mode is shown in figure 2. The direction of vibration was perpendicular to the outlet pipe. If the excitation amplitude is too large, the nonlinearity becomes strong and the stability cannot be evaluated by this method. On the other hand, even if the excitation amplitude is too small, the coherence will be low and it cannot be evaluated. For this reason, based on the conventional experience, the excitation amplitude was set to 1/10 of the gap width. The excitation frequency was 40 Hz, which is the first bending eigenvalue calculated from the valve body shape and material.

The solver ANSYS-CFX 19.0 was used. The k-ω SST turbulent model was applied. Figure 3 shows the computational area and computational grid. Total pressure and total temperature were set in the inlet boundary and averaged static pressure was set in the outlet boundary. Inlet total pressure was changed according to pressure ratio conditions. Inlet total temperature was 20℃, turbulence intensity was 5%, outlet averaged static pressure is 1 atm. Ideal air was used as fluid properties. The number of grids consisted of about 2 million elements, element type was hexahedral. Transient calculation was carried out.
The stability of the valve is evaluated by calculating the aerodynamic work on the valve body by the unsteady CFD analysis. When the aerodynamic work is positive, energy is stored in the valve, and thus the fluid additive damping is negative, that is, self-induced oscillation occurs. The aerodynamic work evaluation formula is shown below:

$$W = \int_t^{t+T} \int_A p\mathbf{n} \cdot \mathbf{v} dA dt$$

(1)

A: Surface area [m$^2$], n: Normal vector [-], p: Static pressure [Pa], t: Time [s], T: Vibration cycle [s], v: Mesh velocity [m/s], W: Aerodynamic work [J], \(\omega\): Angular velocity [rad/s]

3. Experimental model

The valve shape designed to generate self-excited vibration, which was designed by CFD analysis, was conducted. In order to measure the mechanical damping, nonlinear damping due to backlash was eliminated, and the mechanical damping ratio was reduced. The electromagnetic damper structure was applied to control the mechanical damping. The test apparatus is shown in figure 4 and figure 5. The tapping test was carried out to confirm the vibration characteristics of the valve element. The damping was adjusted by the electromagnetic damper structure. Without air flow condition, an accelerometer was installed on the main valve, and the main valve was hit with a hammer with a load meter to measure the acceleration waveform. The natural frequency, generalized stiffness, generalized mass, and damping of the valve body were estimated from the transfer functions of acceleration and excitation force.
The random vibration response and damping of the valve when a fluid was flowing were measured in an air model test. Assuming that the external force due to the flow is a steady Gaussian process, the natural frequency and damping were estimated from the spectrum of the displacement response obtained by random wave measurement under the condition without excitation.

4. Result and discussion
4.1 Comparison between CFD and experiment
CFD analysis was performed under several conditions with the valve opening condition and pressure ratio condition. In this valve, there were pressure ratio conditions at which self-induced oscillation occurs when the valve opening was small. The definition of the pressure ratio is defined as the mass flow averaged total pressure ratio at the inlet “Pin” and at the outlet boundary “Pout”. The definition of the valve opening is defined by the ratio between the valve lift “L” and the diameter of the throat “D”. As the pressure ratio in the experiment, the average value of the wall static pressure of the upstream and downstream pipe of the valve is used. The static pressure on the pipe is measured by pressure taps installed at a pitch of 90 degrees in the circumferential direction.

Figure 6 shows comparison of the experiment results and the CFD results for the fluid additive damping. The fluid addition damping coefficient in the experiment was obtained by taking the difference between the damping ratio when the air was not flowing and the damping ratio when the air was flowing. Fluid additive damping is a relative value, with the experimental result of L/D=0.005 and Pout/Pin=0.2 taken as 1.0. Under the condition of L / D = 0.005, the fluid additive damping is negative at Pout / Pin = 0.6 and 0.8 in the experiment. As Pout / Pin increases, so does the damping coefficient. The CFD results show qualitatively similar trends, although the quantitative values are different. The condition of L / D = 0.03 shows the same tendency between the experiment and CFD. At L / D = 0.05, negative damping is observed at Pout / Pin = 0.4, but positive damping is observed at 0.2 in the experiment. On the other hand, in the CFD results, the damping is positive at Pout / Pin = 0.4, and the damping is negative at Pout / Pin = 0.2. Although there is no measurement result, the CFD result at Pout / Pin = 0.1 shows that the damping tends to increase. The tendency that the damping has
minimum value at a certain pressure ratio agrees, but the absolute value of the pressure ratio is different. It is considered that there are problems in the accuracy of the flow field prediction to the pressure ratio condition.

![Graph](image)

**Figure 6** Comparison of the experiment results and the CFD results for the fluid additive damping

4.2 Generation mechanism of fluid negative damping

Figures 7 to 9 show the Mach number distribution of the central cross section and the damping coefficient distribution per unit area on the valve body in order to grasp the state of the internal flow and the region where the unstable force occurs. In the damping distribution, minus region is unstable and plus region is a stable.

Under the condition of \( L / D = 0.005 \) in figure 7, a large negative damping occurs near the exit of the parallel gap where the gap is narrowest. Although the gap is parallel, the area is increasing, so that the velocity is decelerated. In general, negative damping tends to occur under a decelerating flow. On the downstream side, the jet flows along the casing wall side, and separation flow occurs in the enlarged flow path. Near the reattachment point of the separation, small negative damping region is generated again. On the other hand, large positive damping acts on the first half of the parallel gap on \( P_{\text{out}} / P_{\text{in}} = 0.4 \) and 0.6, and the entire valve becomes stable. The flow field changes at \( P_{\text{out}} / P_{\text{in}} = 0.2 \). The jet flows along the valve body, and negative damping occurs in the deceleration part after the separation. However, positive damping region is generated immediately downstream, and the entire valve has positive damping.
Figure 7: Mach number distribution of the central cross section and the damping coefficient distribution per unit area on the valve body (L/D=0.005).

Figure 8 shows the result under the condition of L / D = 0.03. The similar tendency is confirmed under the condition of L / D = 0.03. The negative damping region is confirmed at the region similar to L / D = 0.005 at Pout / Pin = 0.8. The tendency for large positive damping to occur at the entrance of the parallel gap is also similar to L / D = 0.005 under the condition of Pout / Pin = 0.6. At Pout / Pin = 0.4 or less, the flow field changes. Separation of the jet attached to the casing side is eliminated. At Pout / Pin = 0.4, a flow field similar to Pout / Pin = 0.2 with L / D = 0.005 is confirmed. At Pout / Pin = 0.2, the velocity distribution in the jet increased and decreased. The negative damping region on the valve is generated as related to the velocity increase and decrease, and as a result, fluid additive damping is reduced.
Figure 8 Mach number distribution of the central cross section and the damping coefficient distribution per unit area on the valve body (L/D=0.03)

Figure 9 shows the result under the condition of L / D = 0.05. Regarding L / D = 0.05, the condition of Pout / Pin = 0.8, 0.6, 0.4 shows the same tendency of damping distribution as other valve opening. Under the condition of Pout / Pin = 0.2, the supersonic occurs downstream of the gap, and the velocity increases and decreases even more than the condition of Pout / Pin = 0.2 with L / D = 0.03, and the damping distribution change has occurred. As a result, the entire valve damping is negative. The tendency for the damping to decrease under specific pressure ratio conditions is captured. It is considered that the CFD prediction is different from the experiment for the flow field where the velocity increases and decreases in the jet with respect of the pressure ratio condition. One of the reasons is that the natural frequency differs between the experiment and CFD. The natural frequency is unified at 40 Hz for CFD, but the measurement results is around 40 Hz due to the influence of fluid additive mass and fluid additive rigidity. In addition, a problem of prediction accuracy of a supersonic flow having a shock wave may be considered. Alternatively, in the CFD, no eccentricity is set, but in an experiment, it is conceivable that the measurement is carried out in an eccentric state due to a steady fluid force. Therefore, it is possible that the difference in the boundary conditions may have an effect.
From the above, it is confirmed that the largest negative damping occurs in the gap flow path in the present valve shape. This is considered to be caused by the deceleration flow. In addition to the above, separation and reattachment of the jet, and increase and deceleration of the jet velocity due to the supersonic flow are also considered to be some of the factors of instability. Therefore, in order to stabilize the valve body vibration, it is important to suppress the decelerated flow in the gap. In addition, since separation, reattachment, and fluctuation of the jet may cause a secondary instability, it is preferable that the jet generated in the gap does not follow the valve body.

4.3 Examination of improved valve for vibration suppression

The stable valve shape was devised based on the above-mentioned self-induced oscillation generation mechanism. Figure 10 shows the improved valve shape and Mach number distribution in the cross section under the condition of \(L/D = 0.005\). By changing the shape of the valve body and the casing, the shape of the gap flow path was changed so that the velocity was reduced in the gap, and the jet did not follow the valve body even when the valve opening was small.

Figure 11 shows comparison of the fluid additive damping between the original shape and the improved shape obtained from the CFD results. It can be seen that the improved valve results have positive damping within the pressure ratio condition range studied.
It is confirmed from the aerodynamic work distribution shown in Figure 12 that the negative damping region in the gap is improved. On the other hand, under the condition of $P_{out} / P_{in} = 0.2$, the damping coefficient is smaller than that of the original valve. It is considered that the jet flow did not follow the valve body and the influence was reduced, but the velocity in the jet flow increased and decreased, and the damping decreased accordingly. In the damping distribution shown in Figure 13, it is confirmed that the negative damping region is locally generated only at the gap exit, and the overall stability is improved. It was confirmed that increasing the flow velocity in the gap and preventing the jet flow downstream of the gap from following the valve body were effective as countermeasures.
5. Conclusion

In this study, self-induced oscillation of the valve was evaluated by unsteady CFD. By comparing with experimental results, it was confirmed that the fluid additive damping can be qualitatively predicted by CFD. It was shown that this method was useful as design method for the valve. In addition, by analyzing the CFD results, the fluid additional damping acting on the valve body was grasped, and the effects of flow pattern on the fluid damping distribution were captured. It was confirmed that negative damping occurs in the deceleration flow region in the gap, and control of the flow field downstream of the gap is also important since the separation flow and reattachment flow of the jet from the gap have secondary effect on stabilization. Based on the results, the shape of the improved valve was designed, and the effect of the countermeasure was confirmed by CFD.

References
[1] F. Suzuki, S. Nishida, S. Fukao and M. Tsuruta, Verification of combined main steam valve pressure distribution and vibration characteristics with downscale model test on air condition, Proceedings of ASME Turbo Expo GT2019-90406, 2019
[2] T. Eguchi, Study on self-excited vibration of governing valves for large steam turbines, Proceedings of ISROMAC, 1998
[3] Li-Fei ZENG, Research on the coupling mechanism between alternating flow pattern and valve stem system of steam turbine control valve, Proceedings ASME Turbo Expo GT2014-26988, 2014
[4] Stefan Wallat, A Test rig concept to study fluid structure interactions in a steam turbine valve, Proceedingsof ASME Turbo Expo GT2018-75094, 2018
[5] A. Takeishi, T. Watanabe, T. Himeno and C. Inoue, Multimode flutter analysis of transonic fan using fsi simulation, Proceedings of ASME Turbo Expo GT2014-26702, 2014
[6] R. Srivastava and G.L. Stefko, Flutter analysis of a transonic fan, NASA/TM 2002-211818, 2002
[7] M. Aotsuka, N. Tsuchiya, Y. Horiguchi, O. Nozaki and K. Yamamoto, Numerical simulation of transonic fan flutter with 3D N-S CFD code, Proceedings of ASME Turbo Expo GT2008-50573, 2008
[8] T. Yumitori, K. Ishikawa, K. Takenaka and T. Azuma, Development of flutter analysis tool using next-Generation CFD (Computational Fluid Dynamics) algorithms, JAXA Special Publication JAXA-SP-16-008E, 2016