Transactions of the Korean Nuclear Society Spring Meeting
Gyeongju, Korea, May 29-30, 2008

Numerical Approach for Performance Evaluation of Fluidic Device in APR1400

Sang-Gyu Lim, Suk-Ho Lee and Han-Gon Kim*
Korea Hydro & Nuclear Power Co., Ltd (KHNP), 25-1 Jang-Dong, Yuseong-Gu, Taejon, Korea
**Corresponding author: kimhangon@khnp.co.kr*

1. Introduction

APR1400 is an Advanced Pressurized Water Reactor with 3983MWt power and adopts direct vessel injection system. Fluidic Device (FD) is adopted to regulate the safety injection flow rate in a Safety Injection System (SIT) of APR1400. In case of loss of coolant accident, the injection water flows into primary system in two steps; Initial high flow rate for certain period of time and subsequent low flow rate. By adopting two-step control of the discharge flow rate, FD can ensure the effective use of water in SIT.

A full-scale FD (VAPER: Valve Performance Evaluation Rig) has been tested to obtain a required flow characteristic with full pressure and height of prototype test performed by Korea Atomic Energy Research Institute (KAERI).

Based on the experimental results using VAPER facility, this paper shows the prediction result for the performance characteristics of FD using CFX and FLUENT – which are Computational Fluid Dynamics (CFD) - and compares them with the experiment result. [1,2] This paper also carries out sensitivity analysis through numerically analytical method to improve performance and credibility of FD.

2. Numerical Simulation of Fluidic Device

The experimental result shows 16 in high-flow phase and 160 in low-flow phase on the average for the pressure drop coefficient (K-factor).[3] Table 1 shows the representative results of reputation test for FD performance. Based on this result, this paper analyzes FD performance quantitatively, compares it with the experiment result, and conducts a sensitivity analysis on several conditions that were not applied to the experiment. For the 3-Dimension numerical analysis, ANSYS CFX 11.0 and FLUENT 6.3 are used.

Table. 1 Representative results of reputation test

Test ID	Peak Flow (kg/s)	K-factor	
		Low flow	High flow
Test-II(b)-C-H-1	1040	156.3	16.2
Test-II(b)-C-H-2	1010	158.6	15.6

2.1 Numerical model

The features of fluid flowing into FD are assumed to be three-dimensional, incompressible, steady-state, and turbulent. A standard k-e model that is commonly used in general engineering problems [4] and a Renormalized Group (RNG) k-e model that can simulate turbulence flow under abrupt change in the cross-sectional area are adopted as the turbulence model for numerical simulation.[5] Due to isothermal condition in SIT, heat transfer is not considered.

2.2 Method and Boundary conditions

Unlike the experimental facility composed of SIT and FD, the analysis adopted a simplified modeling of FD for efficiency. Modeling was conducted in ANSYS workbench, and mesh was generated using ICEM CFD, which is transformed into analysis mesh for CFX and FLUENT use. FD performance was classified into high-flow condition and low-flow condition for a steady-state analysis.

Actually, the inlet boundary condition (B.C.) that affects FD is pressure B.C. that combines pressure of compressed air in the upper part of FD and hydrostatic pressure of SIT. The outlet B.C. is an atmospheric pressure. Preliminary analysis shows that, considering the assumption of incompressible flow, the initial pressure B.C. give rise a divergence at high pressure B.C. Moreover, the code manual admits that the analysis is unlikely to yield a robust solution if the B.C of the inlet and outlet are pressure, and suggests that the solution is only partial.

Therefore, for high-flow condition in the control and supply port, a boundary condition was established with an assumption that the mass flow is the same, using experiment value that converts SIT water level change into mass flow rate. In case of low-flow condition, both mass flow and pressure inlet B.C. were used to compare with each result.

3. Results and Discussions

3.1 High Flow Phase

According to the analysis in the high-flow condition, CFX converge fast. Meanwhile, FLUENT didn't, and we conducted a post-processing while maintaining flux amount on a stable level. As shown in Figure 1, K-factor calculated for each code accurately compare with the experiment results. In a vortex chamber, as was expected in the experiment, vortex did not occur due to a conflux of flow from the control and supply port; the flow exited through a discharging nozzle instead.

3.2 Low Flow Phase

CFX did not converge in the low-flow condition, while FLUENT converged rapidly. In regard to CFX, the extent of convergence is lower for pressure B.C., and thus the assumption was made that the flow is injected through the control port only; this resulted in expedite convergence under all conditions. A vector plot shows that a swirling flow is generated in the vortex chamber. As shown in Figure 2, however, K-factor is under-predicted compare to the experiment results.

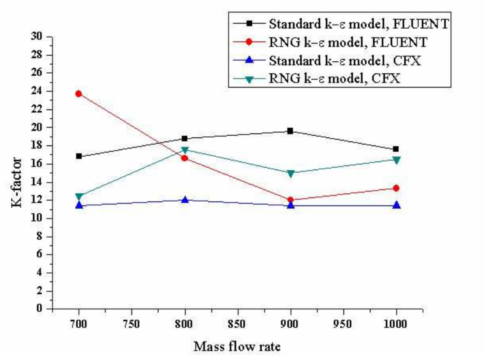

Fig. 1. K-factor of high flow phase

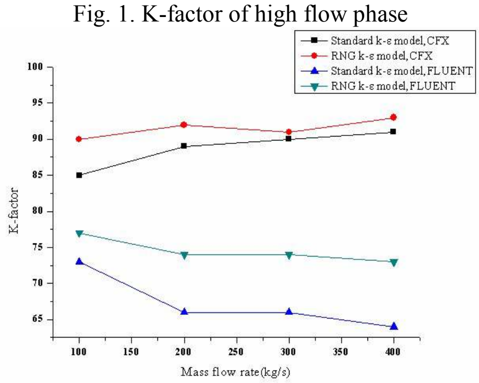

Fig. 2. K-factor of low flow phase

3.3 Sensitivity Analysis

With a high-flow condition, both CFX and FLUENT predicted a similar result to that of the experiment. In comparison, low-flow condition showed different result from the experiment, and thus a sensitivity analysis was conducted by modifying design variables that are deemed to affect a K-factor.

Firstly, in case of internal flow, wall roughness influences pressure drop. Accordingly, we conducted analysis with roughness of 0.0002m and 0.0004m by modifying the initial assumption for a smooth wall. However, K-factor of FD decreased opposed to the general expectation that wall roughness height is likely to increase pressure drop. This is attributed to the fact that, under low-flow condition that incurs strong rotational flow, the wall roughness of the vortex chamber bended the flow vector toward a discharging nozzle, resulting in the low K-factor.

Secondly, design variables that affect K-factor under low-flow condition included the angle of a vortex chamber and control nozzle and impact of back flow to the supply port. We calculated K-factor with a modified design which the vortex chamber and control nozzle are tilted at 10 degrees. As shown Figure in 3, Calculation of K-factor without a supply port demonstrated that the value increased 250%. In the tilted nozzle case, K-factor is increased approximately 8%.

The numerical analysis identified some of the main factors that determine K-factor in the low-flow condition: generation of back flow to the supply port, length of the control nozzle, and angle of vortex chamber and control nozzle. Additional sensitivity analysis is required.

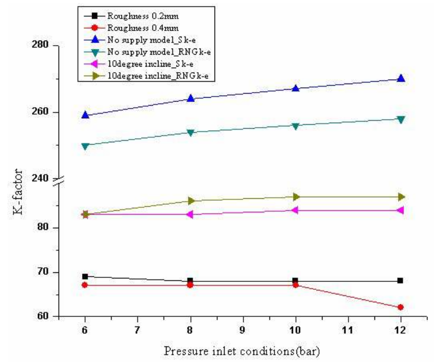

Fig. 3. Sensitivity analysis results

5. Conclusion and Further Study

In this paper, we have examined the performance characteristics of FD through the three-dimensional flow analysis using CFX and FLUENT codes.

A. In the high-flow condition, the estimate of K-factor was relatively accurate. As was expected, the flow from the control and supply port resulted in the conflux that went through a discharging nozzle.

B. In the low-flow condition, occurrence of swirling flow inside the vortex chamber was well predicted. However, the result was under-predicted for about 40% compared to the experiment result.

C. According to the sensitivity analysis, wall roughness inside the vortex chamber bends the vector of swirling flow toward a discharging nozzle, resulting in the low K-factor value.

D. K-factor increased for about 8% after giving 10-degree tilt to the control nozzle, and the value increased 250% after eliminating the supply nozzle.

E. An additional sensitivity analysis using more specific modeling and the main design variables that affect K-factor will be performed.

REFERENCES

[1] ANSYS, CFX 11.0 User Guide, ANSYS SYSTEM, 2006.

[2] FLUENT Inc., FLUENT 6.3 User Guide, FLUENT Inc., 2005.

[3] C. H. Song, Performance Verification Test for APR1400 Fluidic Device, A03NJ02, 2005.

[4] J. C. Jo, CFD Analysis of Unsteady Flow Field in the RWT for Prediction of Air Ingression into the ECC Supply Line During SBLOCA at KSNP, KINS/RR-408, 2006.

[5] K. C. Kim, Flow Characteristic and Optimal Design for RDT Sparger, KSME-B, Vol.23/11, pp. 1390~1398, 1999