Transactions of the Korean Nuclear Society Autumn Meeting
PyeongChang, Korea, October 30-31, 2008

Analysis of Unsteady Flow and Design Modification of Fluidic Device using CFD code

Sang-Gyu Lim*, Han-Gon Kim
Korea Hydro & Nuclear Power Co., Ltd (KHNP), 25-1 Jang-Dong, Yuseong-Gu, Taejon, Korea

1. Introduction

Fluidic Device (FD) located in the bottom of the safety injection tank (SIT) is adopted to use effectively safety injection water during LOCA. FD can provide high flow rate into the reactor at early stage of LOCA, and then maintains the low flow rate for long term cooling at the reflood phase. This flow switching, due to the difference of flow resistance in each of operation modes, enables to regulate the flow rate of safety injection without any operator's action and particular moving parts.

In the previous paper [1], the K-factor of FD, which is a main performance parameter of FD, was computed for performance evaluation by using CFX based on the results of full scale test (VAPER). The past study had the key point to select the suitable turbulence model under steady state condition. Also, a series of sensitivity analysis were carried out to enhance the K-factor by modifying the design criteria that are deemed to affect the K-factor.

Based on the previous results under steady state conditions, this paper analyzes the effect of unsteady flow such as secondary flow and separation in control port nozzle using transient simulation of CFX. Also, this paper predicts the K-factor in unsteady inlet boundary condition.

The results of VAPER test showed that additional discharging time of about 20 sec is required to satisfy the design performance requirement (200 sec.) of FD.

To enlarge the total discharging time of prototype of SIT, a new design is proposed, which modified the direction of control port to use the dead volume of bottom of SIT. The analysis of a new design carried out under same boundary condition of prototype.

2. Numerical Simulation

This section describes numerical models and boundary conditions which include layout of analysis and turbulence models. For the 3-Dimension numerical analysis, ANSYS WORKBENCH, ICEM-CFD and CFX (ver. 11.0) are used.

2.1 Numerical Model

This numerical simulation assumes the characteristics of fluid in FD are incompressible, transient, isothermal, single fluid and turbulent flow. According to the previous study, the standard k-e model in two-equation model is more suitable than the other turbulence models to predict the K-factor in FD. However, it is known that the standard k-e model may not well predict a secondary flow and a separation of the internal flow [2]. On the other hand, the Scale-Adaptive Simulation (SAS) model based on Shear Stress Transport (SST) model well predicts the flow separation, and is used for prediction of swirling flow. To compare with SAS model, Reynolds Stress Model (RSM) is also used.

Two geometry models are generated using tetra element mesh by ICEM-CFD. One is the same layout of VAPER test facility [3]. The other is modified design for the direction of control port which can use the additional water in the dead volume of SIT to enlarge the discharging time of SIT as shown in fig. 1 (b).

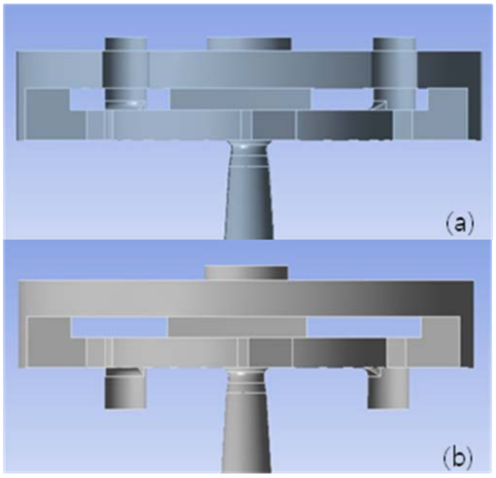

Fig.1. Slice view of the two analysis model;
(a) Prototype, (b) Modified design

2.2 Boundary Condition

The purpose of this study is to analyze the K-factor due to transient effects that occurred by swirling flow at the end of control port nozzle and by the changes of inlet boundary condition versus time rate change. Hence, as shown in fig. 2, the mass flow rates of VAPER test results are assumed as the inlet boundary conditions.

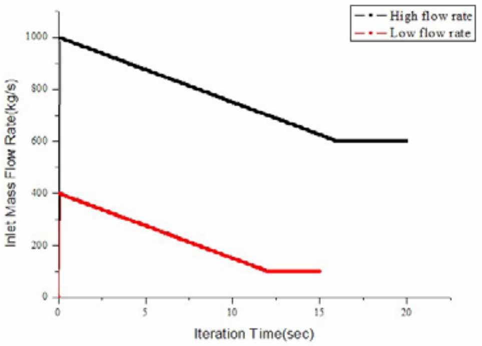

Fig. 2. Inlet mass flow rate per iteration time

3. Results and Discussions

3.1 Prototype model

Under a high flow condition, all of turbulence models well predict the K-factor compared to VAPER results. In case of the standard k-e model, oscillation of the K-factor occurred between 10 sec and 20 sec. In high flow phase, two different direction of fluid flowing into the vortex chamber are merged inside the vortex chamber and flows out to the outlet. Strong cultivated swirling flow arises in the discharge nozzle due to the difference of fluid velocity between wall and center of fluid.

In low flow phase, the water flows through the control port only. In this situation, flowing water bump into the edge of supply port nozzle and then secondary flow are developed in the vortex chamber. Also, strong swirling flow is generated in the discharging nozzle similar to high flow case. These phenomena increasing the amplitude of oscillation as shown in fig.3 can be observed in the result of VAPER test.

As a result, the standard k-e model is more suitable for prediction of K-factor than SAS and RSM model. SAS and RSM model underpredicts the K-factor about 19% compared to the standard k-e model.

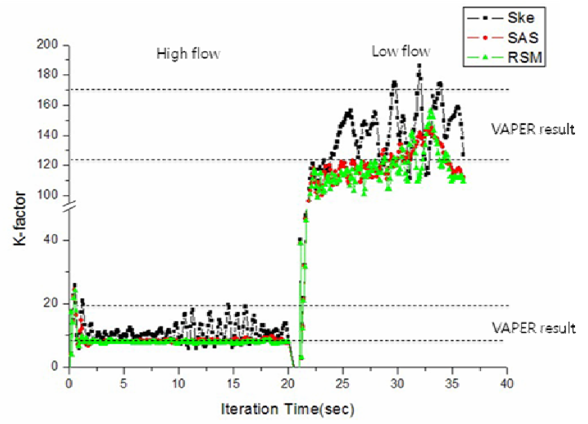

Fig. 3. Analysis results of K-factor (Prototype)

3.2 Modified design model

To increase the total discharging time, FD design criteria is modified to use the dead volume of SIT under the bottom of FD. According to modification, the direction of control port is designed oppositely. Therefore, the analysis of the modified design is necessary for verification of the K-factor.

Similar to the analysis of prototype, oscillation of the K-factor is also observed in case of the standard k-e model under high flow phase. In low flow condition, K-factor is higher than that of prototype model about 12 % by using standard k-e model. The results of SAS and RSM model are similar to prototype model case, but underpredicted the K-factor about 29% in comparison with the standard k-e model.

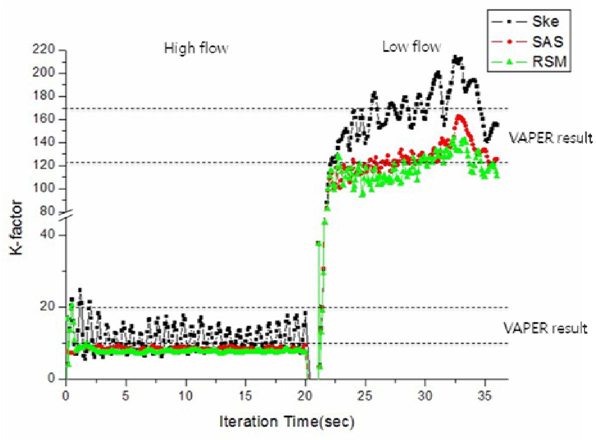

Fig. 4. Analysis results of K-factor (Modified model)

4. Conclusions

This paper examined the transient characteristic of fluid in FD for the unsteady boundary condition using CFX.

A. The standard k-e model well predicts the K-factor in both high and low flow condition compared to SAS and RSM model.

B. Oscillation of the K-factor arises due to a swirling flow which formed at the end of supply nozzle and the discharging nozzle.

C. In case of prototype model SAS and RSM model underpredicts the K-factor for about 19% at high flow phase, compared to the standard k-e model.

D. In case of modified design, K-factor is higher than that of prototype model about 12 % at a low flow condition. The results of SAS and RSM model underpredict the K-factor about 29% compared to standard k-e model.

E. Consequently, a new design is more reliable than a prototype to enlarge the total discharging time of SIT due to the increased K-factor.

REFERENCES

[1] Sang-Gyu Lim, Suk-Ho Lee, Han-Gon Kim, Numerical Approach for Performance Evaluation of Fluidic Device in APR1400, Transactions of the Korean Nuclear Society Spring Meeting, 2008

[2] ANSYS, CFX 11.0 User Guide, ANSYS SYSTEM, 2006

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