Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 22, 2009

CFD Analysis of Flow Characteristics in a Fluidic Device for Design Optimization and RELAP5/MOD3.3 Code Validations with LBLOCA Using the New Pressure Loss Models

Sang-Gyu Lim, Seok-Ho Lee, and Han-Gon Kim*
Advanced Plant Development Office, Nuclear Engineering & Technology Institute, Korea Hydro & Nuclear Power Company, Ltd., Kumbyungro, Yuseong-gu, Daejeon 305-343, Republic of Korea
**Corresponding author: kimhangon@khnp.co.kr*

1. Introduction

Full-scale experiments (VAPER) were done by the Korea Atomic Energy Research Institute (KAERI) to obtain required flow characteristics of a Fluidic Device (FD) (Chu et al., 2008). The test results satisfied the performance requirements of the APR1400 plant design: the pressure loss coefficient (K-factor) and total discharging time of the ECC water. However, a design modification using a full scale test facility is undesirable when determining and updating optimal design because it is time-consuming and very expensive.

In this paper, therefore, a methodology to evaluate the performance of the FD using commercial CFD code is presented. On the basis of experimental data, a benchmark analysis is carried out to obtain an applicable methodology for validating flow characteristics in the FD and to predict the performance requirements. A series of sensitivity analyses are performed for design optimization of the FD by modifying the dimension of the FD. According to the results of sensitivity analysis, several optimal design models are proposed. Then, CFD calculation is carried out for predictions of the flow characteristics in the optimal FD design models. In accordance with the calculated pressure loss data of the optimal design analysis, a system analysis using RELAP5, validated the APR1400 plant system behaviors affected by new pressure loss models of the FD.

2. Benchmark and Sensitivity analysis

2.1 Numerical models according to best practice guidelines

The numerical and mathematical models are described according to the best practice guidelines (Menter, 2002). A high-resolution discretisation scheme was employed to discretize the convective term. The Shear Stress Transport (SST) model and Baseline-Second Momentum Closure (BSL-SMC) model were used, allowing for a more accurate near-wall treatment (ANSYS, 2006).

Table I: Geometric Models Description

Analysis	Geometrical type	I.D.
Benchmark analysis	Same scale with VAPER test	B
Sensitivity analysis	Increased control nozzle angle (5^{o}, 10^{o})	S-A series

The geometric models used in the benchmark and the sensitivity analysis, are summarized as shown in Table I.

2.2 Results and Discussions

In the high flow case of the benchmark result, as shown in Fig. 1(a), the pressure loss is underestimated compared to the average pressure loss of the VAPER test. Although the SST model overestimates the turbulence mixing and the decay of the core vortex, it is an acceptable result considering the uncertainty of the experiments. Fluctuation of the pressure loss originates by the growing and decaying of the swirl flow. The main streams of the merged flow detach from the vortex center. This phenomenon brings about a weakening of the swirl flow, which leads to lowering the pressure loss at the discharge nozzle. On the other hand, the main streams of the merged flow are reattached to the vortex center, which increases the pressure loss with a strong swirl flow. In case of the S analysis, the pressure loss of the S-A10 case is underestimated by about 1/2 of the other results. As pressure loss decreases in the high flow mode, SIT can rapidly deliver the ECC water into the lower plenum, and this helps to remove the decay and stored heat of the reactor core. However, the duration of the high flow mode can be shorter than in the prototype case. Therefore, plant system analysis is necessary to validate the effect of the shortened high flow mode using the RELAP code.

In the low flow case of benchmark results, the numerical results are reliable when taking into the fluctuation range of the VAPER test, as presented in Fig. 1(b). The fluctuation in the low flow case likely originated from the growing and the decaying of the swirl flow, which is related to the leading edge separation of the main stream. Under the low flow mode, the flowing water collides with the end edge of the supply port nozzle, which generates a leading edge separation flow in the vortex chamber. This phenomenon changes the direction of the main stream from a tangential direction to an oblique direction. This can lead to a strong swirl flow and can increase the pressure loss coefficient. For this reason, it was realized that geometric modification is necessary to stabilize the fluctuation and to increase the pressure loss. In the low flow case results of the sensitivity analysis, the pressure loss is enhanced with an increase

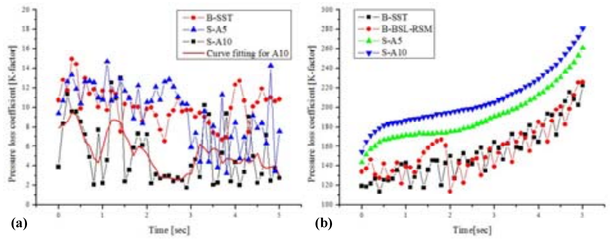

Fig.1. Pressure loss coefficient of the FD in the B and S-A Analysis: (a) high flow mode and (b) low flow mode

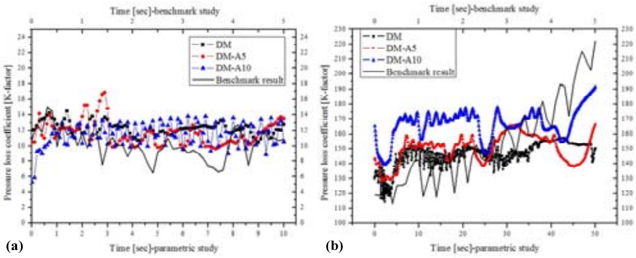

Fig.2. Pressure loss coefficient of the FD in the DM analysis: (a) high flow mode and (b) low flow mode

in the angle of the control nozzle and the fluctuation of pressure loss is diminished. This is because the increase of the angle eliminates the leading edge separation at the end of the supply nozzle, and facilitates the formation of the strong swirl flow in the vortex chamber.

3. Optimal Design Analysis

3.1 Numerical models

To extend the total discharging duration, new models are proposed in which the direction of control port is modified so as to use the dead volume of the ECC water in the SIT bottom. According to the previous analysis results, the changed control nozzle angle models enhance the pressure loss of the FD in the low flow mode. In this analysis, therefore, three different types of models can predict the pressure loss for the optimal design of the FD.

3.2 Results and Discussions

The pressure loss characteristics of the DM (Design Modification) cases are enhanced by increasing the angle of the nozzles, as the flowing water from control port is delivered directly into the vortex chamber and the leading edge separation is dissipated. This phenomenon creates a well-formed swirl flow in the vortex chamber. It can be concluded that the parametric effects are simulated well, as expected, from the benchmark and the sensitivity analysis. The enhancement of the pressure loss can extend the total discharge time duration in the LBLOCA reflood phase.

4. RELAP5/MOD3.3 Code Validations

In this section, RELAP5 code validations are performed to evaluate the effect of new pressure loss models in the FD, which were addressed in the previous CFD calculation. The calculated minimum and maximum values of the pressure loss coefficient are applied in the APR1400 standard input deck. The results of separable effect analyses are presented as below.

As shown in Fig. 3, the quenching time of the fuel rod is pre-predicted by about 25 sec compared to the base case calculation when the pressure loss coefficient of the FD in the high flow mode is 5. On the other hand, the quenching time is delayed about 10 sec when the pressure loss coefficient is 25. In the low flow case of the sensitivity analysis, total discharging time duration of the FD is extended by about 20sec when the pressure loss coefficient of the FD is about 190.

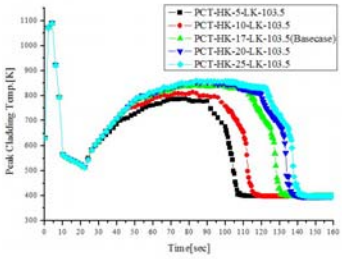

Fig.3. Peak cladding temperature

According to the RELAP5 analysis, the quenching time of the fuel rod is not affected by increasing the pressure loss in the low flow mode. The quenching time of the fuel rod is governed by the excess mass flow rate in the early period of the high flow mode.

5. Conclusions

The pressure loss coefficient of the capabilities of different turbulence models was presented in comparison with the VAPER test results and the three-dimensional flow characteristics were estimated using the quantitative variables and a non-dimensional value stemming from the effect of the swirl flow in the vortex chamber. On the basis of the benchmark results and the newly proposed design, an optimal design analysis was carried out by comparing the pressure loss behaviors in each case. As a result, increasing the control port angle resulted in an enhancement of the pressure loss as well as deterioration of the leading edge separation in the low flow mode.

REFERENCES

[1] Chu, I.C. et al., Development of Passive Flow Controlling Safety Injection Tank for APR1400, Nuclear Engineering and Design, Vol. 238, 200-206, 2008.

[2] Menter, F.R., CFD Best Practice Guidelines for CFD Code Validation for Reactor Safety Applications, Evaluation of Computational Fluid Dynamic Methods for Reactor Safety Analysis (ECORA), European Commission, 5th URATOM FRAMEWORK PROGRAMME, 2002

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