Detached-Eddy Simulation of a Fluidic Device for a Prediction of Pressure Loss Characteristics in a Low Flow Mode

Sang-Gyu Lim*, Suk-Ho Lee, 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: sglim@khnp.co.kr*

1. Introduction

The Advanced Power Reactor 1400(APR1400) adopts a passive flow controller in Safety Injection Tanks (SITs) as one of Advanced Design Features (ADFs). This device, called a 'Fluidic Device (FD)', controls the flow rate of safety injection water in a passive manner. A flow control mechanism varies the flow resistance in the vortex chamber corresponding to the SIT water level hence the flow rate can be adjusted by the specific flow resistance in a specific flow regime.

A full-scale test was performed and the test results met the design requirement of APR1400 [1]. To enhance the performance of the FD more effectively, a series of CFD analysis were implemented and remedy of design modification was proposed on the basis of a series of CFD analysis [2].

The results of CFD analysis showed that total discharge time of the fluidic device is to be increased by enhancing the K-factor in consequence of changing the control nozzle angle. However, a tendency of a pressure loss was under-estimated as a limitation of turbulence models such as Reynolds Averaged Navier-Stokes (RANS) models compared to the experimental data.

This paper shows that pressure loss characteristics of the FD can be predicted using a Detached-Eddy Simulation (DES) turbulence model, which can provide valuable flow characteristics far exceeding RANS simulations.

2. Numerical Models and Boundary Conditions

This section describes several numerical models that reduce the numerical errors of a CFD analysis and boundary conditions to simulate accurate physical characteristics of the vortex flow in the FD. Selection of the numerical models complies with the best-practice guidelines by Mentor [3].

2.1 Numerical models

Discretization errors can be reduced by using a higher-order discretization scheme and smaller time step sizes. A High-Resolution (HR) advection scheme and a second-order backward Euler transient scheme are used in this study. The time step sizes are set by 0.01sec and the total analysis time is set to 5sec. Finer meshes are used in the geometry of the vortex chamber, which can reduce the discretization errors and grid-induced errors.

According to the previous studies [2], to stimulate a strong curvature flow in the vortex chamber, turbulent flow modeling more realistic than the RANS turbulent models might be necessary. In an attempt to improve the predictive capabilities of turbulence models in highly separated regions, Spalart [4] suggested a hybrid method that combines RANS formulations with a Large Eddy Simulations (LES) model. This concept is based on the idea of dealing with the boundary layer by the RANS model and switching to the LES model in the detached regions. In order to perform an unsteady simulation using the DES turbulence model, an initialization is carried out by a steady-state calculation using the RANS model such as a Shear-Stress Transport (SST) model.

2.2 Boundary Conditions

As the main goal of this study is to extend the total discharging time of the SITs without increasing of the SIT capacity, the pressure loss in the low flow mode should be increased. Thus, analysis focuses on the low flow mode during the total discharge period.

To validate the FD pressure loss characteristics in the low flow mode, the measured data obtained by Chu [1] is used as the inlet and outlet boundary conditions. According to the experimental results, the flow rate of the low flow period was observed as 400 kg/s-100 kg/s. As presented in Table 1, the transient simulations are performed as a null transient, which the inlet flow condition is set by 400 kg/s. A no-slip wall condition and an automatic near-wall treatment are used for the wall boundary layer. The main boundary conditions are summarized at Table 1.

3. Computational Results

Pressure loss coefficients in the vortex chamber are compared among the DES model cases, RANS model cases and experimental data.

3.1 Pressure loss characteristics

As shown in Figure 1, the result of the DES case and fine mesh case (case3) tends to be close to the experimental data.

Table I: Boundary Conditions, grid information and numerical models

<table>
<tr><th>Cases</th><th>Mesh no. [Million]</th><th>Inlet [kg/s]</th><th>Outlet [atm]</th><th>First Layer Height</th><th>No. of layer</th><th>Turbulence model</th><th>Remark</th><th>Calculated Average K-factor</th></tr>
<tr><td>Case1</td><td>1.8</td><td rowspan="5">400</td><td rowspan="5">1</td><td rowspan="3">0.25</td><td rowspan="3">7</td><td rowspan="3">DES</td><td rowspan="3"></td><td>133.1</td></tr>
<tr><td>Case2</td><td>2.7</td><td>139.0</td></tr>
<tr><td>Case3</td><td>3.5</td><td>148.3</td></tr>
<tr><td>Case4</td><td>0.35</td><td rowspan="2">0.2</td><td rowspan="2">5</td><td rowspan="2">SST</td><td rowspan="2">Previous analysis results</td><td>127.6</td></tr>
<tr><td>Case5</td><td>0.30</td><td>133.3</td></tr>
</table>

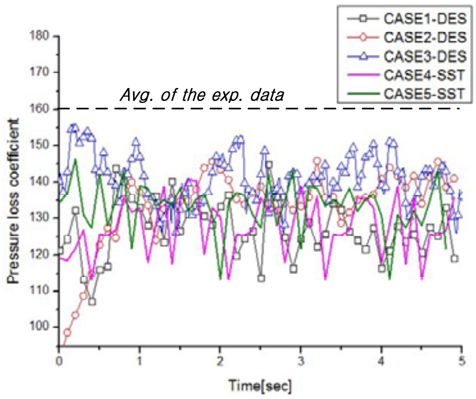

Fig. 1. Comparison of the pressure loss characteristics in the low flow period in terms of the turbulence model effects

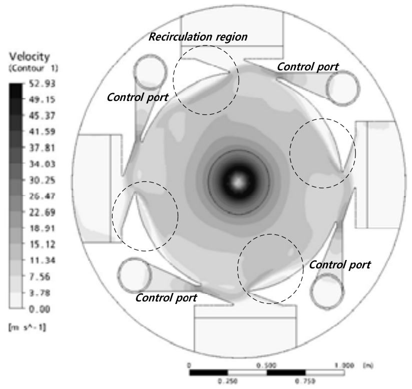

Fig. 2. Velocity contour in the vortex chamber at 2sec and observation of recirculation regions by a leading edge separation

This confirms that the DES turbulence model is more suitable for simulating the vortex characteristics in the chamber. The separated flow of a boundary layer occurs at the leading edge of the vortex chamber and inlet of the discharge nozzle thus the pressure loss depends on a prediction of the separation effect and the strong streamline curvature in the vortex core. The DES model can capture the unsteady flow structure of the separated shear layer by resolution of the developing turbulent structure. However, the SST model has an important weakness which is insensitive to the streamline curvature and rotating flow. In particular, the SST model can lead to an over-prediction of turbulent mixing and to a strong decay of the core vortex for swirling flows.

3.2 Leading edge separation effect

In the low flow period, the water of the SIT is delivered only through the control port. The nozzles of the control ports are designed so that the directions of the flow are tangential to the vortex chamber and this provides the strong swirling flow in the vortex chamber.

According to previous analysis results, a leading edge separation arises by an interaction between the main stream of the discharge flow and end edges of the supply ports. Figure 2 reveals that the leading edge separation is observed in the DES simulation and the recirculation region is more developed than in the SST simulation results. This is because velocity profiles in the vicinity of the recirculation region are found from calculation to be more accurate in terms of the prediction of separation points in the boundary layer.

3. Conclusions

A prediction of the strong curvature flow characteristics in a FD using a DES turbulence model is more feasible than using SST models. Although a DES model provides more details of the swirling flow structure than a SST model, a single fluid simulation is still limited in predicting accurate pressure loss characteristics in the FD. This limitation can be overcome by using a two-fluid, free-surface simulation.

REFERENCES

[1] Chu, I.C., Song, C.H., Cho, B.H. and Park, J.K., Development of Passive Flow Controlling Safety Injection Tank for APR1400, Nuclear Engineering and Design 238, p. 200–206, 2008.
[2] Lim, S.G., Lee, S.H. and Kim, H.G., Benchmark and Parametric Study of a Passive Flow Controller (fluidic device) for the development of optimal designs using a CFD Code, , Nuclear Engineering and Design 240, p. 1139–1150, 2010.
[3] 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 Progamme, 2002.
[4] Spalart, P.R., Jou, W.H., Strelets, M. and Allmaras, S.R., Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach, 1st AFOSR Int. Conf. on DNS/LES, Aug.4-8, 1997, Ruston, LA.