Transactions of the Korean Nuclear Society Autumn Meeting
Jeju, Korea, October 21-22, 2010

Sensitivity Study for CFD Analysis on Debris Transport to ECCS Sump for CANDU Type Plant in Korea

Byung Il KWON [a*], Jong Uk KIM [a], Jae Seon CHO [a], Tea Keun PARK [b], Sang Won LEE [c], Hyeong Taek KIM [c]
[a]*FNC Technology Co., LTD.*
Seoul National University Bldg. #135 San 56-1 Shilim-dong, Gwanak-gu, Seoul, 151-742 Korea
[b]*Korea Power Engineering Co., Inc.*
[c]*Korea Hydro & Nuclear Power Co., LTD.*
[*]*Corresponding author: colossus@fnctech.com*

1. Introduction

Once containment recirculation pumps are activated and emergency core cooling (ECC) flow is supplied from the recirculation sump during loss of coolant accident (LOCA), various insulations and coatings on a pipe, equipments and structures damaged by LOCA break jet as well as additional debris sources are transported to recirculation sump screen by the break flow and containment spray flow drainage. This debris may result in loss of net pressure suction head (NPSH) of the recirculation pumps, and have a threat to long term cooling and containment heat removal capacity. In this case, flow patterns of containment pool are important to confirm behaviors of debris transport for predicting various flow paths to the recirculation sump screen[1,2]. In this paper, models using commercial computational fluid dynamics (CFD) software CFX are developed for containment pool simulation during recirculation mode. The specific plant used for this analysis is CANDU type plant, in Korea.

2. 3-D CAD Model and Meshes

Geometry modeling consists of two stages. First stage is three dimensional geometry modeling for containment pool based on general arrangement of containment structure using computer aided design (CAD) software. The bottom floor where the recirculation sump is located is at Elevation-93.9 m. Fig. 1 shows containment CAD model for CFD analysis.

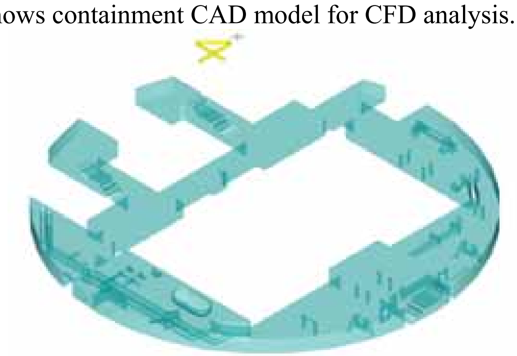

Fig. 1. 3-Dimensional CAD drawing for bottom floor

Next stage is mesh generation based on containment structure geometry model. Commercial mesh generator ANSYS ICEM CFD was used as a mesh generator. Tetrahedral meshes were adopted and clustered around some areas considering geometry shapes. A total of 1.3 million tetrahedral meshes were generated as shown in Fig. 2.

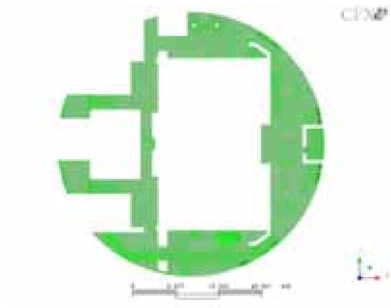

Fig. 2. Computational Mesh Generation

3. Specification of Boundary Conditions

Boundary conditions were assumed for double-ended hot leg break in the beginning of safety injection and in recirculation mode[3]. And blowout panels and stairs were assumed as pathways of water to sump. Assumed boundary conditions are summarized in Table I.

Table I: Summary of Boundary Conditions

Description	Boundary Type	Assumed Value*
Stair A	Inlet (mass flow)	168.2kg/sec
Stair B	Inlet (mass flow)	168.2kg/sec
Stair C	Inlet (mass flow)	168.2kg/sec
Blowout Panel A	Inlet (mass flow)	45.6kg/sec
Blowout Panel B	Inlet (mass flow)	114.6kg/sec
Blowout Panel C	Inlet (mass flow)	32.8kg/sec
Blowout Panel D	Inlet (mass flow)	32.8kg/sec
Blowout Panel E	Inlet (mass flow)	50.9kg/sec
Blowout Panel F	Inlet (mass flow)	32.8kg/sec
Blowout Panel G	Inlet (mass flow)	112.3kg/sec
Sump suction(1,2)	Outlet (static pressure)	0 bar
Solid wall	Wall	No slip
Water Surface	Wall	Free slip

* It was assumed that each flow from the blowout panels was determined according to its flow path area fraction.

4. CFD Simulation

Commercial CFD software CFX was used to simulate three dimensional containment pool flow behaviors. For the flow field calculation of ECC recirculation mode, the steady state or quasi steady state is analyzed. Thus, the simulated volume was considered to be completely full of water. Pool water surface was modeled as slip wall, and the other surface of solid structure as no-slip wall. Reference turbulent model selected is Renormalization Group k-epsilon (RNG k-ε) model, which is known to be adequate for complex geometry. Root mean square (RMS) residuals of the mass, momentum, k-epsilon turbulence were monitored to check convergence history during iterations.

5. Simulation Results

In the preliminary study, three cases for debris transport were simulated such as the pathway of water to sump were only blowout panels, only stairs and blowout panels/stairs[4]. And several sensitivity analyses are introduced in this study; (1) outlet pressure, (2) single failure and (3) severe case.

1. Outlet Pressure

At long term cooling mode of CANDU type plant, containment pressure would be reduced almost by atmospheric pressure. But in the beginning of recirculation mode, containment pressure would be about 3 bar(g). As shown in the Fig.3, in static CFD analysis, outlet pressure would not a big influence on the flow field.

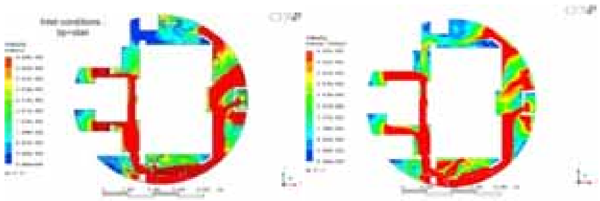

Fig. 3. Flow Fields for Outlet Pressure (0 and 3 bar)

2. Single Failure

If one ECC system train fails when LOCA occurs, all debris will be transported to another ECC sump screen and it is more likely to serious problem of pump head loss[5]. Therefore, single failure of the ECC pump should be considered.

In the Fig.3, the amount of large size debris transported to sump screen is calculated as ~42%. And as shown in the Fig.4, the amount of large size debris transport to sump screen of case 2 is less than those of case 1, ~20%. But because fine/particulate debris and latent debris can be transported to sump screen directly and one train of ECC sump screen is not usable, head loss of ECC pump by debris will be larger than the case of two trains of ECC system operable.

3. Severe Case

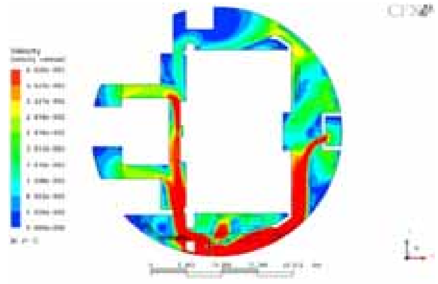

Fig. 4. Flow Fields for Single Failure

When LOCA, blowout panels in upper floor will break by pressure difference(~1 psid) between upper floor and bottom floor. Once one blowout panel breaks first, pressure difference between upper floor and bottom floor will be diminished and other blowout panels cannot break. This is the severe case, because flow velocity of bottom floor will be very high and more debris will be transported to sump screen than case 2. In the Fig.5, the amount of large size debris transported to sump screen is calculated as ~45%.

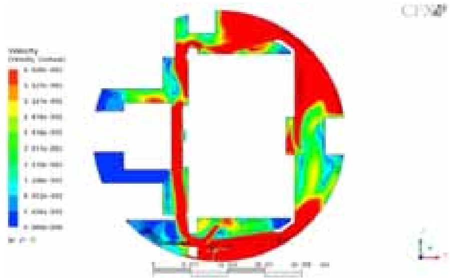

Fig. 5. Flow Fields for Severe Case

6. Conclusion

CFX program is used in analyzing CFD (Computational Fluid Dynamics) for containment pool simulation during recirculation mode of CANDU plant. The flow fields are analyzed for various cases, and the ratios of debris transported to sump screen are calculated respectively. As a result, the severe case in the aspect of ECCS sump performance is assessed.

REFERENCES

[1] Regulatory Guide 1.82, Revision 3, "Water Sources for Long Term Recirculation Cooling Following a Loss-Of-Coolant Accident Sump Performance Evaluation Methodology," U.S. Nuclear Regulatory Commission, November 2003.
[2] NEI 04-07, "PWR Sump Performance Evaluation Methodology," May 2004.
[3] KHNP, "Wolsong Unit 1 Final Safety Analysis Report," Korea Hydro & Nuclear Power.
[4] KWON, B. I., et al., "Preliminary CFD Analysis on Debris Transport to ECCS Sump in Recirculation Mode for CANDU Plant, in Korea," Korean Nuclear Society Spring Meeting, May 2010.
[5] "Safety Evaluation by The Nuclear Reactor Regulation Related to NRC Generic Letter 2004-02," U.S. Nuclear Regulatory Commission.