Preliminary CFD Analysis on Debris Transport to ECCS Sump in Recirculation Mode 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.*
25-1 Jang-dong, Yuseong-gu, Daejon, 305-343 Korea
[*]*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, preliminary 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. Description and Results

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.

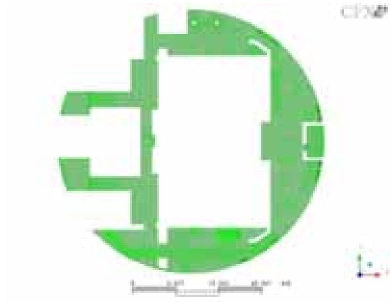

Fig. 2. Computational Mesh Generation

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 pa
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

Debris transport is determined by velocity field and turbulent kinetic energy (TKE) field on 1 cm elevation plane from the floor. Three cases for debris transport are simulated such as the pathway of water to sump is (1) only blowout panels, (2) only stairs and (3) blowout panels and stairs. The results are shown in Fig. 3 and Fig. 4. Tumbling velocity of specific debris is used to judge whether the debris may start to move, and settling velocity (eventually transformed to minimum TKE) is used to judge whether the debris may keep suspended[4]. Streamline shown as Fig. 5 is also used to determine transport fraction of specific debris. If the velocity along a particular streamline became smaller than a debris threshold velocity, the debris would not migrate to the sump screen. By using streamline analysis at potential debris entry locations, a basis for determining whether the debris would transport to the sump screen could be developed. As shown in the Fig.3, the amount of debris transport to sump screen of case 3 is less than those of case 1 or 2. The fire protection doors of upstairs are not opened usually on normal operation. So when LOCA, opening the fire protection doors of upstairs into bottom floor is helpful to mitigate the loss of NPSH's of pumps by reducing the debris transported on the sump screen.

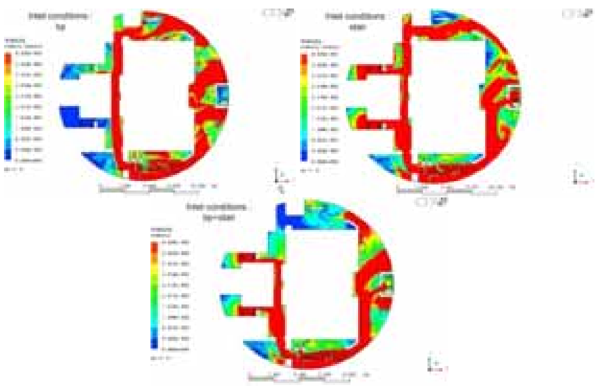

Fig. 3. Velocity Contour

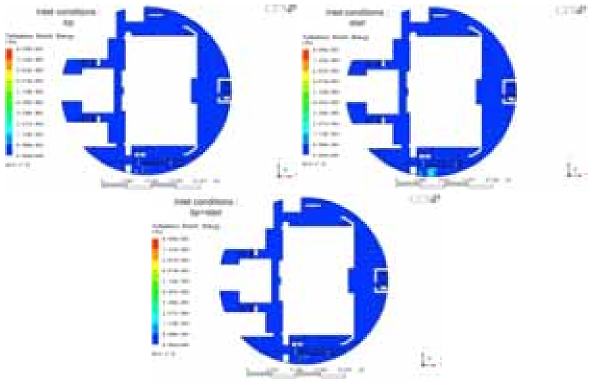

Fig. 4. Turbulence Kinetic Energy Contour

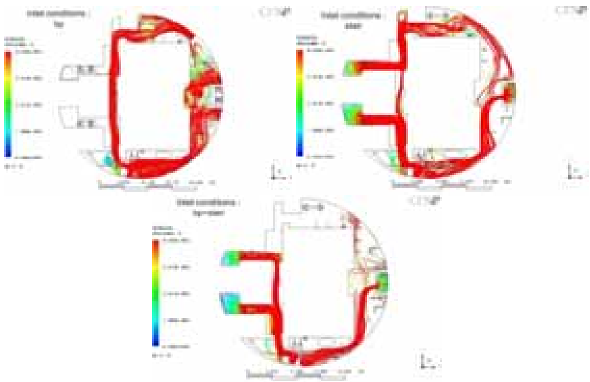

Fig. 5. Streamline from Break to Recirculation Sump

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, TKE's and streamlines are analyzed for three cases, and the ratios of debris transported to sump screen are calculated respectively. As a result, when LOCA, it is concluded that opening the fire protection doors into bottom floor is helpful in the aspect of ECCS sump performance.

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 4 Final Safety Analysis Report," Korea Hydro & Nuclear Power.
[4] "Safety Evaluation by The Nuclear Reactor Regulation Related to NRC Generic Letter 2004-02," U.S. Nuclear Regulatory Commission.