Preliminary CFD Analysis on Debris Transport to ECCS Sump in Recirculation Mode for Kori Unit 3

Su Won LEE [a*], Kyung Jin LEE [a], Soon Joon HONG [a], Sung Bok LEE [b], Hyeong Taek KIM [b]
[a]*FNC Technology Co., LTD.*
Seoul National University Bldg. #135 San 56-1 Shilim-dong, Gwanak-gu, Seoul, 151-742 Korea
[b]*Korea Hydro & Nuclear Power Co., LTD.*
25-1 Jang-dong, Yuseong-gu, Daejon, 305-343 Korea
**Corresponding author: swlee@fnctech.com*

1. Introduction

Once containment recirculation pumps are activated and emergency core cooling (ECC) flow is drawn from the recirculation sump during loss of coolant accident (LOCA), various insulations and coatings on a pipe, equipment 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 of 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 for containment pool simulation during recirculation mode using commercial computational fluid dynamics (CFD) software, CFX are made. The specific plant used for this analysis is Kori Unit 3, three-loop Westinghouse plant, in Korea.

2. Description and Results

Geometry modeling consists of two stages. One is three dimensional 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-100 ft. Recirculation sump is located near the steam generator room A. Large obstacles on the bottom floor are a pressurizer relief tank and a reactor coolant drain tank. Obstructions whose diameter is larger than 6 inch are also considered. Fig. 1 shows containment CAD model for CFD analysis.

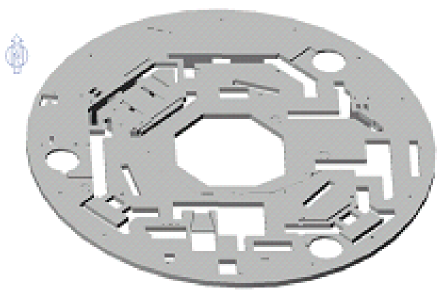

Fig. 1. 3D CAD drawing for bottom floor

The other is mesh generation based on containment structure CAD model. Commercial mesh generator ANSYS ICEM CFD was used to mesh generator. Tetrahedral meshes were adopted and clustered around some areas considering geometry shapes. A total of 3.5 million tetrahedral meshes were generated as shown in Fig. 2.

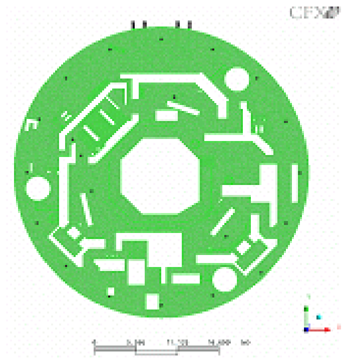

Fig. 2. Computational Mesh Generation

3. Specification of Boundary Conditions

Boundary conditions were assumed for double-ended pump suction break in the beginning of safety injection and containment spray in recirculation mode[3]. Assumed boundary conditions are summarized in Table I.

Table I: Summary of Boundary Conditions

Description	Boundary Type	Assumed Value*
Break	Inlet (mass flow)	442.9kg/sec**
Grating	Inlet (mass flow)	175.6kg/sec
Refueling pool	Inlet (mass flow)	89.1kg/sec
Steam generator A	Inlet (mass flow)	33.0kg/sec
Steam generator B	Inlet (mass flow)	28.1kg/sec
Steam generator C	Inlet (mass flow)	29.2kg/sec
Pressurizer	Inlet (mass flow)	9.0kg/sec
Sump suction(1,2,3,4)	Outlet (static pressure)	0 pa
Solid wall	Wall	No slip
Water Surface	Wall	Free slip

* It was assumed that each flow from the spray was determined according to its flow path area fraction.
** Design flows of all RHR and SI pumps were considered.

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 good in complex geometry. Root mean square (RMS) residuals of the mass, momentum, k-epsilon turbulence were monitored to check convergence history 1000 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. 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 method for determining whether the debris will transport to the sump screen could be developed.

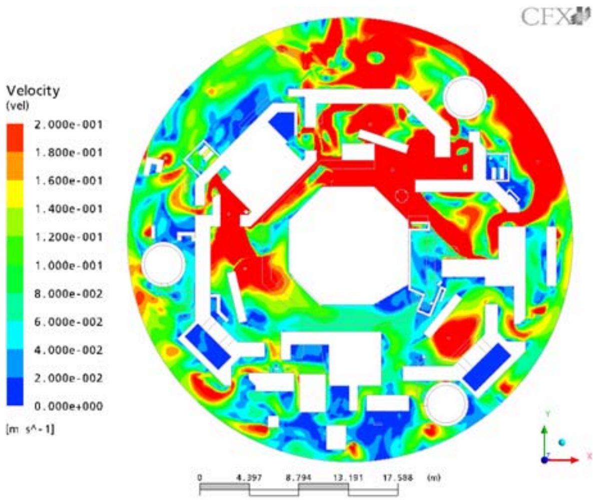

Fig. 3. Velocity Contour

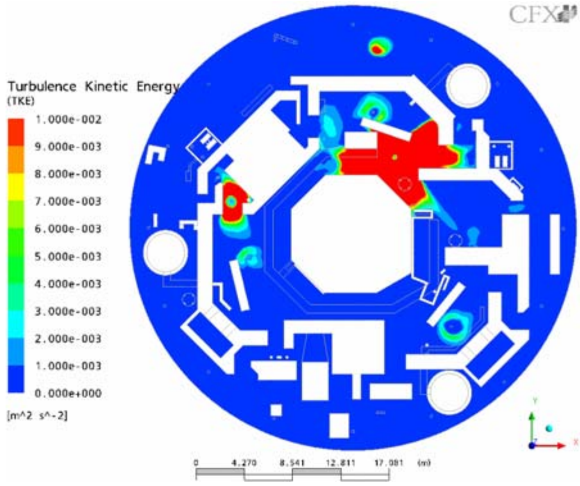

Fig. 4. Turbulence Kinetic Energy Contour

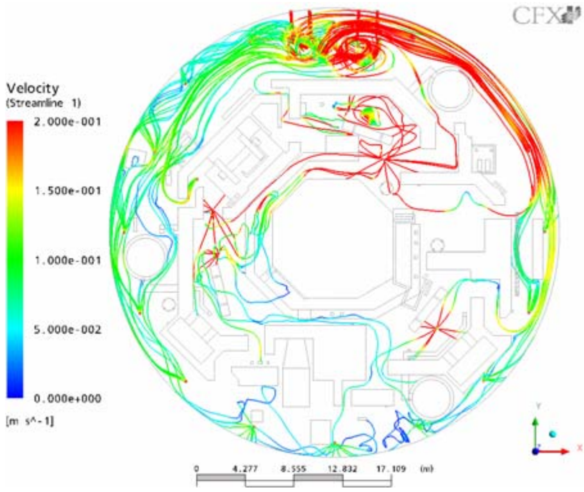

Fig. 5. Streamline from Break to Recirculation Sump

6. Conclusion

In this paper, we prepared feasibility study for debris transport in recirculation mode from geometry modeling to evaluation of transport factor using three dimensional CFD analysis. This study provides a useful method of the CFD analysis for the debris transport in recirculation mode.

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, “Kori Unit 3 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.