Transactions of the Korean Nuclear Society Spring Meeting
Chuncheon, Korea, May 25-26 2006

Feasibility Study for Containment Pool Using Three Dimensional Computational Fluid Dynamics

Lee, Doo-Yong, Hong, Soon-Joon, Park, Byung-Gi and Lee, Byung-Chul
FNC Technology Co. SNU Research Park Innovation Center 516, San4-2,Bongchun-7 dong, Kwanak-Gu, Seoul, KOREA dylee@fnctech.com

Kim, Hyeong-Taek and Park, Jong-Woon
KHNP Nuclear Environmental Technology Institute, Munji-Dong, 103-16, Yusung-Gu, Taejeon , KOREA

1. Introduction

Once containment recirculation pumps are activated and ECCS flow is drawn from the recirculation sump during LOCA, various insulations and coatings on pipe 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 NPSH loss of 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, we have made preliminary models for feasibility of containment pool simulation during recirculation mode using commercial Computational Fluid Dynamics(CFD) software, CFX.

2. Description and Results

2.1 Geometry Modeling

Geometry modeling consists of two stages. One is three dimensional structure modeling for containment pool based on general arrangement of containment structure using CAD software. AutoCAD was used to simulate containment structure in this paper. Generated containment structure model which has diameter 33.49m(109.88ft) includes spray flow paths like gratings, stairwells and other spray flow drainage sources and inner structures such as reactor compartment, steam generator compartment, RCP compartment and other obstacles which obstruct flow. It also includes break inflow boundary and containment recirculation sump structure. Figure 1 shows containment structure model using CAD software. 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 adapted to CAD model and clustered around some areas considering geometry shapes. A total of 0.93 million tetrahedral meshes were generated as shown Figure 2.

2.2 Specification of Boundary Conditions

Boundary conditions were assumed for double-ended pump suction break in the beginning of safety injection recirculation and containment spray recirculation mode. Assumed boundary conditions are summarized in Table 1.

Table 1. Summary of Boundary Conditions

Description	Boundary Type	Assumed Value
break	Inlet (mass flow)	222.6kg/sec
grating	Inlet (mass flow)	61.7kg/sec
stairwell	Inlet (mass flow)	10.9kg/sec
upper spray drainage	Inlet (mass flow)	7.9kg/sec
lower spray drainage	Inlet (mass flow)	7.9kg/sec
sump	Outlet (pressure)	-
solid wall	no slip	-

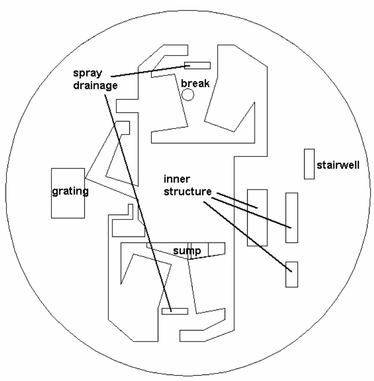

Figure 1. Containment Structure CAD Model

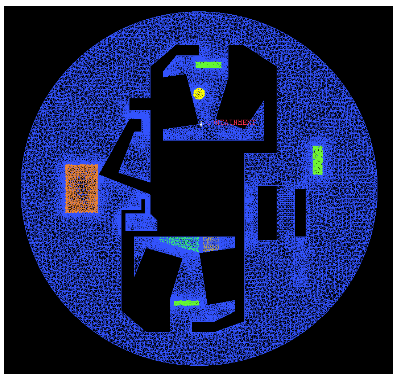

Figure 2. Computational Mesh Generation

2.3 CFD Simulation

Commercial CFD software CFX was used to simulate three dimensional containment pool flow behaviors. For safety injection and containment flow recirculation phases, the steady state or quasi steady state is analyzed, because of the balance between break/spray inlet flow and containment recirculation sump intake. Thus, the simulated volume was considered to be completely full of water. The k-epsilon turbulence model was used in conjunction with turbulence kinetic energy and turbulence eddy dissipation. RMS residuals of the mass, momentum, k-epsilon turbulence were monitored to check convergence history during 1000 iterations.

2.3 Simulation Results

The simulation results under assumed boundary conditions typically show that break flow with vertical motion sweeps out debris near upper compartment, and debris is transported circumferentially to the recirculation sump along containment boundary shown in Figure 3 to 5. The debris migration threshold velocity such as 0.085m/s for the Reflective Metal Insulation(RMI) and 0.037m/s for the fiber flock were used to determine the fraction of the pool flow area in excess of the debris transport threshold velocity[3]. Convergence history as RMS residuals of the mass and momentum increases initially and converges into ~$1.0e^{-3}$ orders, and RMS residuals of the k-epsilon turbulence shows similar behaviors and converges into ~$1.0e^{-4}$ orders. Total running time after 1000 iterations in the Pentium 4 CPU 2.4GHz with 2G RAM machine was about 30 hours.

3. Conclusion

In this paper, we prepared three dimensional computational fluid dynamic models for containment pool analysis in case of recirculation phase during postulated Large Break Loss Of Coolant Accident(LBLOCA) to study feasibility of large scale simulation. The simulation using commercial CFD software, CFX showed physically reasonable results. Thus, we could confirm feasibility for large scale containment pool simulation. And we expect that these results can be used as a base study for more accurate analysis of the specific plant.

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] “Safety Evaluation by The Nuclear Reactor Regulation Related to NRC Generic Letter 2004-02,” U.S. Nuclear Regulatory Commission.

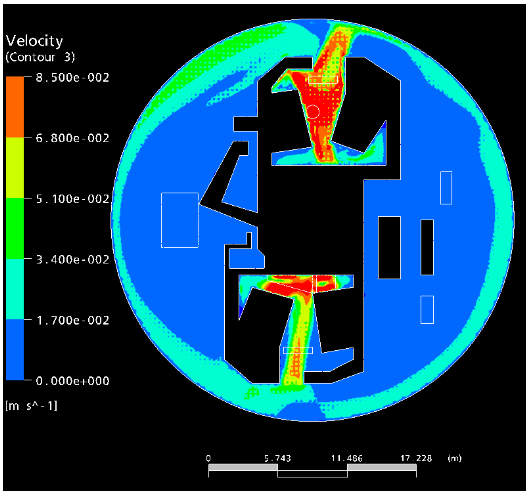

Figure 3. Velocity Contour (speeds greater than or equal to the RMI threshold are colored red.)

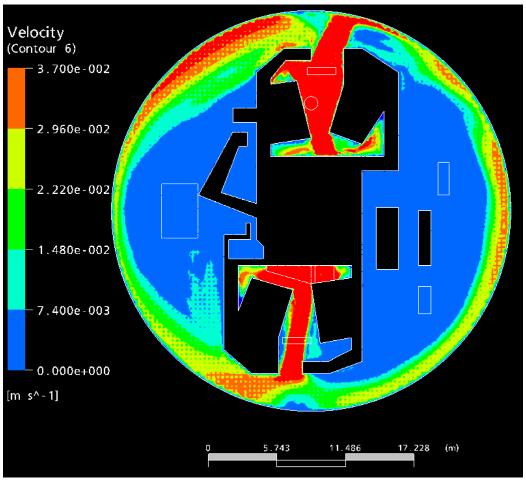

Figure 4. Velocity Contour (speeds greater than or equal to the fiber threshold are colored red.)

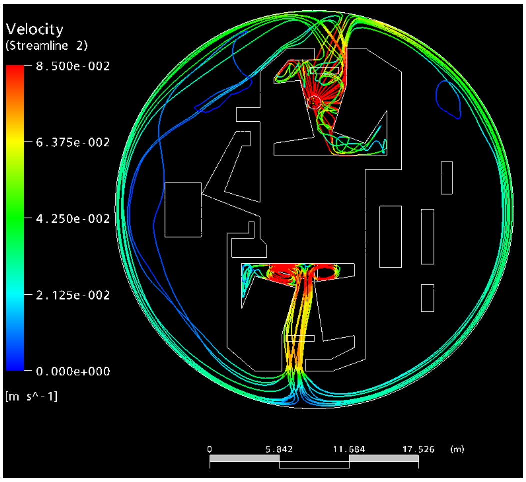

Figure 5. Streamline from Break to Recirculation Sump