Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 10-11, 2007

Three Dimensional CFD Analysis on Containment Pool Recirculation in ECCS Sump Performance Evaluation

Jonguk Kim[a], Soon Joon Hong[a], Byung Chul Lee[a],and Jong Woon Park[b]

[a]*FNC Tech. Co. Ltd., SNU Research Innovation Center 422,*
San 4-2, Bongchun 7-Dong, Kwanak-Gu, Seoul, 151-818, S. Korea
skj1@fnctech.com
[b]*Nuclear Engineering & TEChnology Institute, Korea Hydro & Nuclear Power Co. Ltd.,*
Yuseong Post Box 149, Daejeon, 305-600, S. Korea

1. Introduction

Once containment recirculation pumps are activated and ECC flow is drawn from the recirculation sump during LOCA, various insulations and coatings on pipe, equipment and structures damaged by loss-of-coolant accident break jet as well as additional debris sources are transported to recirculation sump screen by the break flow and containment spray flow drainage[1].

Once the containment recirculation pumps are activated and the ECCS flow is drawn from the recirculation sump, detailed flow patterns in the containment pool can be obtained using state-of the-art 3D computational fluid dynamics. The flow is quasi-steady or steady state at this stage of the ECC operation. Thus, fully three-dimensional flow patterns can be obtained, which in turn can be used to predict the various flow paths to the recirculation sump(s), including detailed 3D velocities for the debris transport analysis. The flow velocities and turbulent kinetic energy can be compared to the debris-specific settling velocities, incipient and bulk transport velocities to determine the percentage of debris expected to transport to the sump screen[2].

In this paper, we have made preliminary models for feasibility of containment pool transport simulation for the Westinghouse 2-loops plant during recirculation mode using commercial CFD software, CFX[3].

2. Geometry Modeling and Conditions

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, and the other is mesh generation based on the containment structure CAD model.

2.1 A Geometry Modeling

The bottom floor where the recirculation sump of the Westinghouse 2-loops plant is located is at Elevation EL-6' floor. Large structures are pressurizer relief tank (PRT) and reactor coolant drain tank (RCDT). Obstructions larger than 6 inches are identified through the containment walkdown and considered in the CAD drawing.

The minimum water level at recirculation mode was assumed 1 meter high from the bottom floor. Thus, the considered domain for 3-dimensional CAD drawing is from the bottom floor to water surface of 1 meter high. Actual CAD drawing is shown in Figure 1.

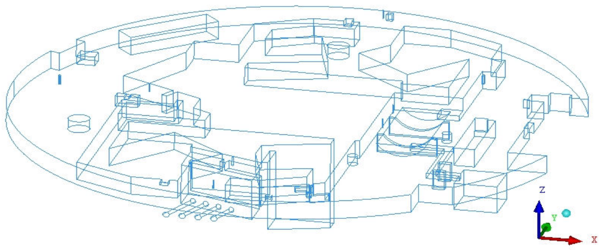

Fig. 1 3-dimensional CAD drawing for bottom floor

2.2 Mesh Generation

ICEM[4] program was utilized to generate mesh. Finer mesh was generated near the internal obstructions and near the passage where higher water velocity is expected. Large and hollow space where lower velocity is expected is filled with relatively large mesh. Tetrahedral mesh was selected. Tetrahedral mesh has some merits in modeling the complex geometry like bottom floor of containment. The number of tetrahedral mesh is about 3 millions, and the theoretically largest size is 5 cm. However, mesh of 5 cm size is extremely rare. Figure 2 shows the generated mesh for entire domain.

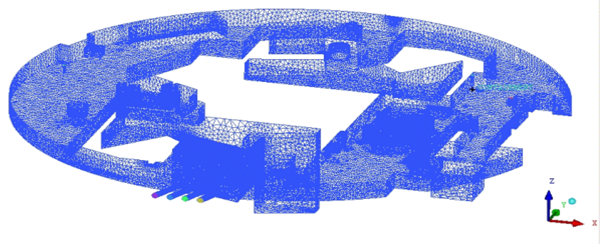

Fig. 2 3-dimensional view for an entire surface mesh

2.3 Initial and Boundary Conditions

Double ended pump suction leg break in SG room A was postulated and the location of break is near the center of SG room A. Reference time in transient of break is just after recirculation actuation signal (RAS) is generated.

Pool water surface are 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.

3. Simulation Results

In the determination of specific debris transport, 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. Streamline is also used to determine transport fraction of specific debris.

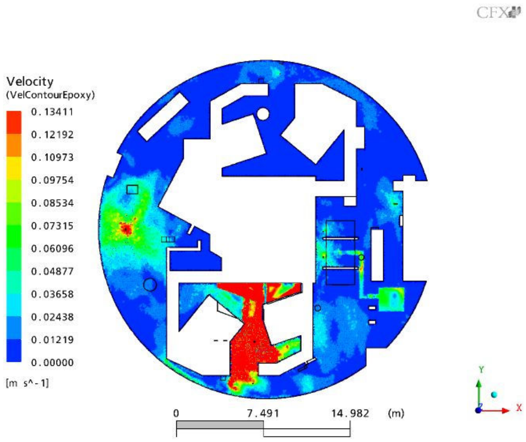

Fig. 3 Velocity contour field

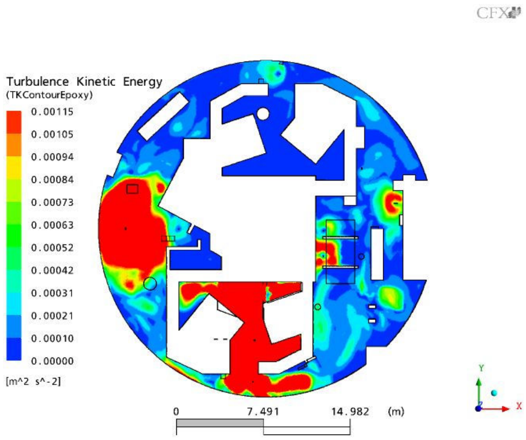

Fig. 4 TKE contour field

The water from the grating and the stairway forms principal flow along the containment wall, and the water from break in SG room A is directed to the sump. The velocity contour is shown in Figure 3, and the TKE contour in Figure 4. Red zones in Figures. 3 and 4 are those where the velocity and TEK are larger than tumbling velocity and minimum TKE for 5-mil epoxy, respectively. Tumbling velocity and minimum TKE for 5-mil epoxy are 0.4ft/sec and 0.009ft^2/sec^2. Near the inlet flow from the grating high velocity/high TKE region is formed. In the other place relatively low velocity and TKE are formed. The fastest region is in SG room A. In this room the largest inflow from the break and all the outflows to the pump suctions are located. So, a direct flow from the break to the pump suctions is formed.

Figure 5 shows the streamline. Some of water inflow from the grating in left side of Figure 5 flows to the sump counterclockwise along the containment wall. The water flows clockwise and reaches a pit in the right side of Figure 5, and finally it is trapped in that pit. Thus, it can be deduced that this water hardly reaches the sump, and resultantly the debris along that streamline rarely migrates to sump.

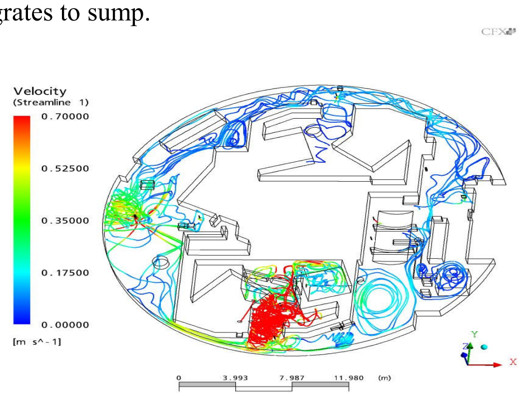

Fig. 5 Stream lines

Sensitivity studies were performed for mesh type (hexagonal mesh), outlet condition(free outlet), and turbulence model(SST k-ε model), respectively. The results showed very similar results to above.

4. Conclusions

This paper provides full scope feasibility study for debris transport in recirculation mode of Westinghouse 2-loops plant containment by geometry modeling to evaluation of transport fraction using CFD analysis. At first, the analysis domain was determined using minimum water level. And then inner obstructions and spray flow path were identified. In the modeling of spray flow from upper floor the effect of gravitation was also reflected. Three millions meshes in tetrahedral shape is sufficient to produce reasonable CFD results, regardless of the calculation options.

This study provides a useful method of the CFD analysis for the debris transport in recirculation mode

REFERENCES

[1] NEI, "Pressurized Water Reactor Sump Performance Evaluation Methodology", NEI 04-07, 2004

[2] USNRC, "Water Sources for Long-Term Recirculation Cooling Following a Loss-of-Coolant Accident", Regulation Guide 1.82 Rev. 3, 2003

[3] CFX, Code Manual

[4] CFX, ICEM Manual