Sump Pool Flow Simulation during Fill-up Phase of LOCA Using on CFD for OPR1000 Plant

Kyung Sik Choi [a], Jong Pil Park [a], Ji Hwan Joeng [a*], Manwoong Kim [b]

[a] *School of Mechanical Engineering, Pusan National University, Jangjeong-dong, Geumjeong-gu, Busan, Korea*
[b] *Korea Institute of Nuclear Safety, Guseong-dong, Yuseong-gu, Daejeon, Korea*
[*] *Corresponding author: jihwan@pusan.ac.kr*

1. INTRODUCTION

During LOCA (Loss of Coolant Accident) in design bases accident (DBA), emergency core coolant supplements form a recirculation sump and cooled core and containment. When the double ended guillotine Break (DEGB) at the hot leg near steam generator, due to the jet impingement discharge flow, the debris could be potentially generated at pipe or wall nearby steam generator and be transported to the recirculation sump. Therefore, the debris, such as insulations and paint chips, could be accumulated and be clogged in the recirculation sump screen. If debris is blocked the sump strainer, the pressure drop is increased at the screen so as to increase the pressure loss of ECCS (Emergency Core Cooling System) pump NPSH (Net positive suction head). It is potentially influenced to decrease the long-term cooling capability of the recirculation sump. The recirculation sump screen clogging accident has happened in BWR of USA and Sweden. Considering the important to safety, US NRC (Nuclear Safety Commission) has issued the recirculation sump blockage as GSI-191(Generic Safety Issue-191). Moreover, US NRC published Regulatory Guide 1.82 Rev.3 incorporated the R&D findings and experiences in 2003. NEI (Nuclear Energy Institute) introduced the methodology procedure to solve this safety issue in the NEI 04-07 report. In the meanwhile, US NRC also published individually the regulatory guidelines as a SER (Safety Evaluation Report) report for PWR plant. However, the current available technical information including the reports is applicable to the generic PWR plants not the plant specific plant. Therefore, the additional research reflecting characteristics of plant specific plant is necessary to develop the methodology and technical guides on the recirculation sump clogging issue. The objective of this study is addressed to explore the characteristics of sump pool flow during LOCA by using CFD for the OPR1000 plant

2. POOL FLOW SIMULATION

The containment of OPR1000 plant was similarly modeled using 3D CAD for the dry containment with 142ft 12in in diameter and 219ft in height. The structures influenced on the coolant fluid flow of the containment floor such as the recirculation sump, elevator pit, RCP (reactor coolant pump) lube oil collection tank mount, normal sump at the location of 86ft were considered.

2.1 Initial and boundary conditions

In this study, to simulate the sump pool flow, the pipe break location was selected the hot leg DEGB in the vicinity of the steam generator so that maximum thermal insulating material debris (Nukon) and the worst debris collection during LOCA are taken into consideration in accordance with the NEI report 04-07 and US NRC SER. The initial and boundary conditions are showed in Table 1 and the initiation of recirculation pump is assumed after RAS (Recirculation Action Signal). Beside, the submergence water level of containment sump is assumed at 3.13ft from the sump floor. A mass flow rate from both HPSI (high pressure safety injection) and LPSI (low pressure safety injection) assumed 6355gpm. In the CFD analysis, it is considered that one pump among two recirculation pumps is only available. As to the numerical model, RNG k-ε turbulent model and VOF(Volume Of Fluid) model are applied.

Table I: Initial & Boundary Condition for CFD Analysis

Initial condition & Boundary condition	OPR1000 plant
Temperature	140 °F
Mass flow rate	6,355 gpm (HPSI+LPSI)
Minimum water level for flooding	3.13 ft

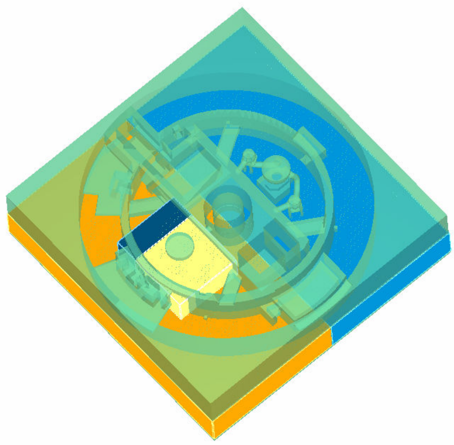


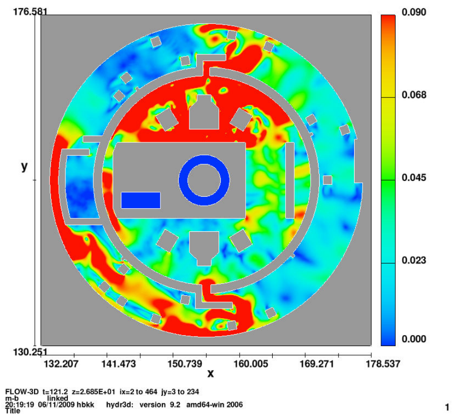
Fig. 1 Containment floor block and mesh

2.2 Numerical Model

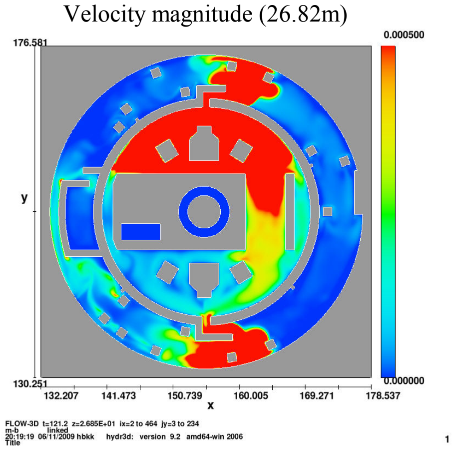
The CFD analyses were performed with the FLOW-3D Ver9.2 computer program which is a general-purpose software package applicable to the open channel fluid flow simulation in adequate. To conduct the CFD analysis, the structure is modeled into 4 blocks using the 3D CAD as shown in Fig.1. The physical model with 0.3ft in diameter at 4 blocks was meshed with the total of about 3 million meshes.

2.3 Result and Discussion

Fig.3 showed the velocity magnitude and turbulence kinetic energy (TKE) until 120sec using CFD simulation. The velocity prediction results are similar to those of NUREG/CR-6772 assumed as 0.28ft tumbling velocity with 2in x 2in size debris (stainless-steal reflective metallic insulation). Comparing between the tumbling velocity and TKE, it shows that the high TKE is presented at the high tumbling velocity zone. But the high TKE is also appeared at low tumbling velocity zone in the vicinity of the structures such as RCP Lube oil collection tank mount and the edge of sodium storage facilities. This implies that the secondary flow induced by the structures affects on an increase of the TKE.



Velocity magnitude (26.82m)



Turbulence kinetic energy (26.85m)

Fig. 3 VM and TKE at 120sec on OPR1000 Plant

Fig.4 shows the characteristic contours of the tumbling velocity and TKE at the location of y (150in diameter) and z (26.85in diameter). In the vicinity of the structures, TKE is higher than tumbling velocity due to the influence of the containment geometry and structure. In other word, due to the generation of vortex flow near structures, although the tumbling velocity decreases, the turbulent intensity is increased.

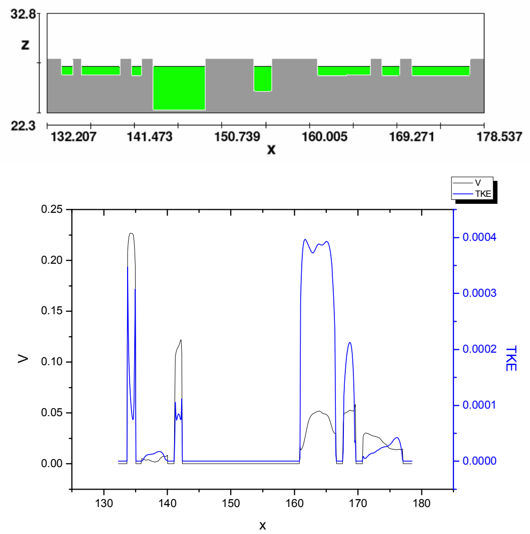


Fig. 4 VM and TKE at 120sec on OPR1000 Plant(Y=150m)

3. CONCLUSIONS

The sump pool flow simulation during fill-up phase of LOCA using on CFD for OPR1000 plant has conducted. As the result, it concludes that the high tumbling velocity and TKE are identified during 120sec and the structures inside sump are influenced on the debris tumbling velocity as well as TKE. The results of this study will be applied as a reference data to design the local experimental facility. At present, the construction of the facility is progressed about 80% and the experiment will be completed on end of October. Thereafter the validation between the experiment and CFD studies will be carried out.

REFERENCES

[1] USNRC, Regulatory Guide 1.82, Revision 3, “Water Sources for Long-term Recirculation Cooling Following a Loss-of-Coolant Accident”, Washington D.C., November 2003.

[2] NEI, “Pressurized Water Reactor Sump Performance Evaluation Methodology”, NEI0 4-07, May 2004.

[3] USNRC, NUREG/CR-6772 “GSI-191: Separate-Effects Characterization of debris Transport in Water”, Washington D.C., August 2002.