Numerical analysis of containment pool flow and transport of insulation debris

Tae Hyub Hong*, Sang Won Lee and Hyeong Taek Kim
Nuclear Engineering & Technology Institute, Korea Hydro & Nuclear Power Co., Ltd., 25-1 Jang-dong, Yuseong-gu, Daejeon, Republic of Korea
**Corresponding author: hth@khnp.co.kr*

1. Introduction

In the event of a loss-of-coolant accident (LOCA) at a nuclear power plant, insulation debris can be released near the break. Some of this debris can be transported in the containment water pool to the vicinity of the sump and increase the pressure drop across the sump screen, at which point the emergency core cooling system (ECCS) can fail to re-circulate coolant to the reactor core. In the present study, a CFD (Computational Fluid Dynamics) analysis of the coolant flow in the containment pool is carried out and debris transportation is evaluated using the Lagrangian particle tracking method.

2. Numerical Method

3.1 CFD Simulation

The commercial CFD code FLUENT is used to calculate the steady state flow of the containment pool of KSNP. The minimum anticipated depth of the containment pool is assumed to be 3.13 ft (0.95 m); the fluid domain of the containment pool is shown in Fig.1. A hot leg double ended guillotine break is assumed for the accident scenario. In the long term recirculation cooling mode, the LPSI pump (1,235 gpm) and the HPSI pump (5,120 gpm) are engaged and the total flow rate is assumed to be 6,355 gpm. This flow rate is applied at the break location for the inlet boundary condition. The sump is set in a pressure outlet (p = 0 kpa) condition. The top surface, representing the free surface, is assumed to be a slip boundary and no-slip is applied at walls except for the top surface. The number of grids is about 1,000,000, comprised of tetrahedron cells.

Reynolds Averaged Navier-Stokes equations and Continuity equations are solved and the RNG k-e model is used for turbulence modeling because these methods treat swirling flows rather better than the standard k-e model does. For the discretization of the convection term, a second order upwind scheme is adopted. It is assumed that the fluid domain is at rest for the initial set of conditions. The transient calculation is done until steady state solutions are achieved.

3.2 Debris Transportation Simulation

Debris transportation is simulated using the DPM (Discrete-Phase Model) of FLUENT. Debris is assumed to be comprised of spherical particles and the motions of particles are governed by the following force balance equation, defined as:

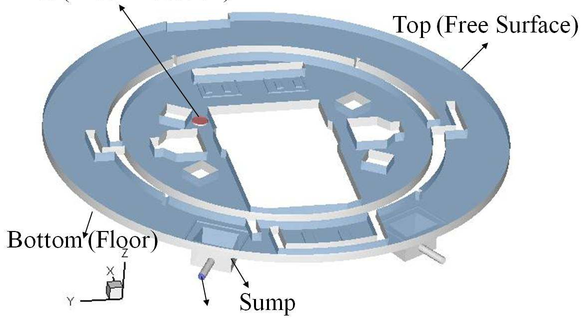


Fig.1 Geometry of the fluid domain

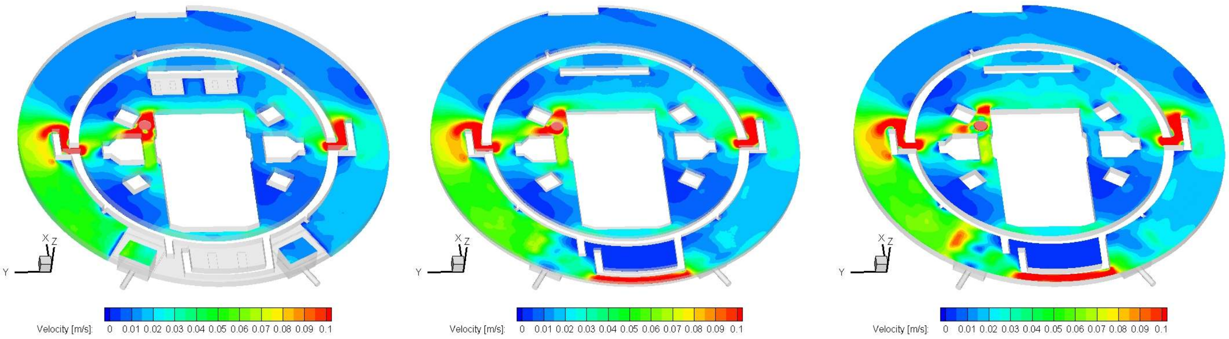
$$\frac{du_p}{dt} = \frac{3}{4d_p}\frac{\rho}{\rho_p}C_D\left|u_p - u\right|(u - u_p) + \frac{g_x(\rho_p - \rho)}{\rho_p} + F_x \qquad (1)$$

Where u and u_p are the coolant velocity and the particle velocity, respectively, ρ is the density of coolant, and ρ_p is the density of the particles. d_p is the particle diameter and F_x is an additional acceleration (force/unit particle mass). The density of the dry NUKON fiber is 2,800 kg/m^3. Actually, wetted insulation debris mostly consists of water [1]; the density of this insulation debris is slightly larger than the density of water. Therefore, it is assumed that the NUKON particles have virtual mass (= 1010 kg/m^3) and virtual particle velocity in the water. The 1 inch particle size is considered.

About 12,000 particles are injected into the fluid domain after the steady state solution is achieved. The injected particles are equally spaced with 1 m in the tangential direction of the top surface and 0.25 m in the normal direction of the top surface.

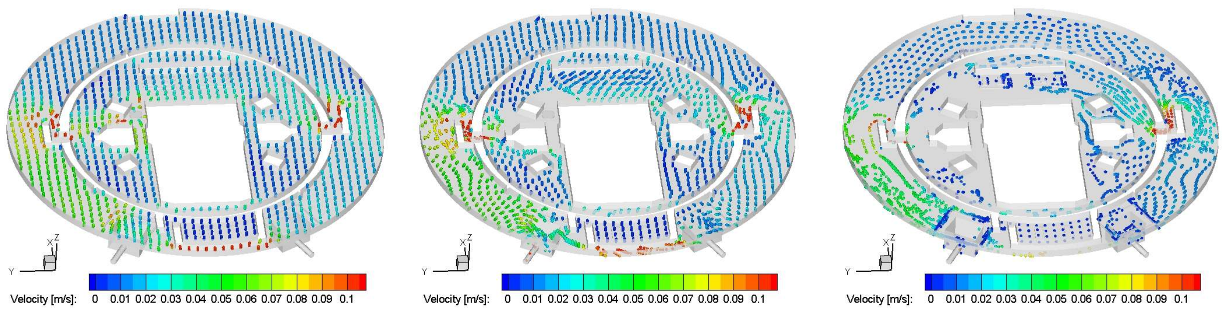
3. Results and Discussion

The velocity contours at the three heights of the level (0.01 m, 0.5 m and 1.0 m) are shown in Fig 2(a) ~ (c). A flow from the break location to the outer annulus



(a) height = 0.01 m **(b) height = 0.5 m** **(c) height = 0.954 m (Free surface)**

Fig. 2 Velocity magnitude Contour



(a) t = 10 s **(b) t = 100 s** **(c) t = 500 s**

Fig. 3 Motion of debris particles

zone is observed. Relatively high velocity is observed in the vicinity of the break location and the entrance way to inner circular region. The coolant passed from the entrance way flows to the sump along the outer annulus wall. The maximum magnitude of velocity at the free surface (4.20 m/s) is the highest among the three planes because the viscous effect due to the floor surface is the smallest at the free surface.

According to the Safety Evaluation Report for NEI 04-07 Appendix III [2], the transport fraction of debris transported to the sump screen and available to accumulate on the screen can be evaluated by calculating the fractional areas, which are a percentage of the area in excess of the threshold velocity. The fractional areas calculated at the height of 0.01 m (Fig. 2(a)) and at the free surface (Fig. 2(c)) are 16.54 % and 21.93 %, respectively. Also, in the present study, it is proposed that the transport fraction can be evaluated based on the fractional volume. The fractional volume represents the ratio of the sum of the finite volumes of which velocity exceeded the threshold velocity to the total volume of the fluid domain. The calculated transport fraction based on the fractional volume is 19.91 %.

Fig. 3 shows debris behavior with time. The evaluation of the transport fraction of the debris is based on the results of particle tracking. After enough time has elapsed, particles passing through the sump (outlet) and vanishing from the fluid domain are counted. The number of vanished particles divided by the number of initial injected particles is the transport fraction, which has a value of 8.4 %. A large number of particles sank to the containment floor. The transport fraction calculated using the particle tracking method is much lower than that based on the threshold velocity. That is, the method using the threshold velocity does not consider the gravity effect or the buoyancy effect. Therefore, the method is a conservative one.

4. Conclusions

In the present study, the three dimensional single phase flow of a containment pool is simulated and the evaluation of transport fraction of insulation debris is carried out using FLUENT. The method of evaluation of the transport fraction based on the threshold velocity is more conservative than the method using the particle tracking method. To achieve better evaluation results for debris transport, it will be necessary to perform an experimental study to investigate the characteristics of debris motion in water and the drag coefficients of insulation debris of various types, shapes and sizes.

REFERENCES

[1] Krepper. E., et al., “CFD-modeling of insulation debris transport phenomena in water flow”, Nuclear Engineering and Design, 2009

[2] “Safety Evaluation Report for NEI04-07 Appendix III, ANSI/ANS Jet Model,” Dec 06, 2004.