Computational Studies on the Performance of Flow Distributor in Tank

Soo Jai Shin*, Young In Kim, Seungyeob Ryu, Youngmin Bae
Korea Atomic Energy Research Institute, Daedeok-daero 989-111, Yuseong-gu, Daejeon 305-353, Republic of Korea
**Corresponding author: shinsoojai@kaeri.re.kr*

1. Introduction

Flow distributors such as spargers are generally observed in several nuclear power plants. These devices are installed in the core make-up tank (CMT), in-containment refueling water storage tank (IRWST) and so on. CMT is full of borated water and provides makeup and boration to the reactor coolant system (RCS) for early stage of loss of coolant accident (LOCA) and non-LOCA. The top and bottom of CMT are connected to the RCS through the pressure balance line (PBL) and the safety injection line (SIL), respectively. Each PBL is normally open to maintain pressure of the CMT at RCS, and this arrangement enables the CMT to inject water to the RCS by gravity when the isolation valves of SIL are open. During CMT injection into the Reactor, the condensation and thermal stratification are observed in CMT and the rapid condensation disturbed the injection operation. To reduce condensation phenomena in the tank, CMT was equipped with the flow distributor.

Cho et al. [1] experimentally investigated the performance of flow distributor with varying the multi hole pattern and pitch-to-hole diameter ratios (*p/d*). They reported that the effect of the hole pattern such as staggered and parallel type is relatively smaller than that of *p/d*, and the dominant frequency of pressure oscillation increases with *p/d*. Tuunanen et al. [2] experimentally studied on the performance of CMT of advanced light water reactors in small break LOCA conditions, and they demonstrated the importance of flow distributor in the CMT to limit rapid condensation.

The optimal design of the flow distributor is very important to ensure structural integrity of the reactor system and their safe operation during some transient or accident conditions. In the present study, we numerically investigated the performance of flow distributor in tank with different shape factor such as the total number of the holes, the pitch-to-hole diameter ratios (p/d), the diameter of the hole and the area ratios. These data will contribute to the design the flow distributor.

2. Computational method

Simulations have been performed for the steady, incompressible, three dimensional Newtonian fluids. Reynolds Averaged Navier-Stokes (RANS) equations are used to model the flow distributor in Tank. The computations have been carried out using a segregated and double precision solver in Fluent 13.0. Pressure-velocity coupling is achieved via semi-implicit methods for a pressure-linked equation (SIMPLE) algorithm. A second-order upwind method has been employed for the discretization of the momentum and turbulent transport equations. The standard wall function is adopted for the near-wall treatment.

Figure 1 illustrates a schematic of the flow distributor which is installed in the upper part of the tank. The diameter of the inlet pipe (PBL) is D, and the diameter of holes in the flow distributor is d. The pattern of the holes is staggered pattern as shown in Fig.1.

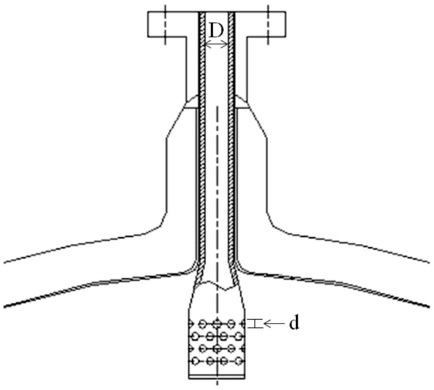

Fig. 1. Schematic of the flow distributor

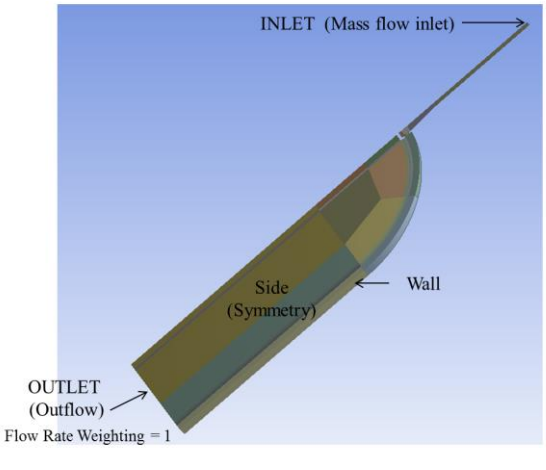

Fig. 2. Schematic of the computational domain.

Figure 2 shows a schematic of the computational domain, which consists of the pipe, flow distributor and upper part of the tank. The mass flow inlet boundary condition was used at the inlet of the pipe, and the outflow boundary condition was used at the outlet of the tank. No slip boundary condition was used at the wall of the pipe, flow distributor and tank. Symmetry boundary condition was used at the sides of computational domain.

3. Results and discussion

We simulated the model of the flow distributor in tank with varying the number of the holes (N), the ratio between the diameter of the hole and diameter of the inlet pipe (d/D), the pitch-to-hole diameter ratios (p/d), and the area ratio. The area ratio is defined as the ratio between the area of the inlet pipe and the total area of the holes in the flow distributor. Table I shows the parameters of three different flow distributors. The total number of the holes in distributor is 40, 40 and 60 and, the number of holes per row x number of rows in distributor is 10 x 4, 10 x 4 and 10 x 6 for type A, B and C, respectively. The Type B of the distributor is shown in Fig. 1.

Table I: Parameters of three types of flow distributors

	number of holes	*d/D*	*p/d*	area ratio
type A	40 (10x4)	0.15	2.4	0.88
type B	40 (10x4)	0.31	1.7	3.96
type C	60 (10x6)	0.31	1.7	5.95

Figure 3 shows the contours of fluid velocity around the distributor. The diameter of the holes for type B is two times larger than that for type A. In Fig. 3(a) and 3(b), the velocity difference between upper hole and lower hole for type A is smaller than that for type B. In other words, as the diameter of the hole is decreased, the velocity difference between upper and lower holes also decreases.

The inlet mass flow rate is same for all simulations. Because the area ratio for type A is smaller than that of type B and C as shown in Table I, the velocity of the holes for type A is faster than that for other types. Although the area ratio for type B is smaller than that for type C, the velocity of the holes for type B and C is similar in Fig. 3(b) and 3(c). For type C in Fig. 3(c), the velocities of two upper holes in six holes are very small and almost mass flow is ejected through four lower holes. In in Fig. 3 (b) and (c) (for *d/D*=0.31), it is observed that the velocity of the holes is not in inverse proportional to the area ratio, as the number of the row is increased from 4 to 6.

Figure 4 demonstrates the contours of fluid velocity in *xz* plane of the computational domain. Because the velocity of the holes for type A is larger than type B and C, the velocity magnitude of the tank for type A is generally larger than other types.

4. Conclusion

In the present study, the model of the flow distributor in tank is simulated using the commercial CFD software, Fluent 13.0 with varying the different shape factor of the flow distributor such as the total number of the holes, the diameter of the holes and the area ratio. As the diameter of the hole is smaller, the velocity difference between holes, which is located at upper position and lower position of the flow distributor, also decreases. For larger area ratio, the velocity of the holes is slower. When the diameter of the hole is large enough for the velocity difference between holes to be large, however, the velocity of the holes is not in inverse proportional to the area ratio.

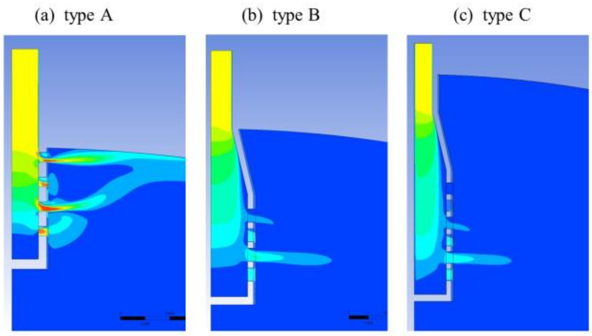

Fig. 3. Fluid velocity contours around the distributor

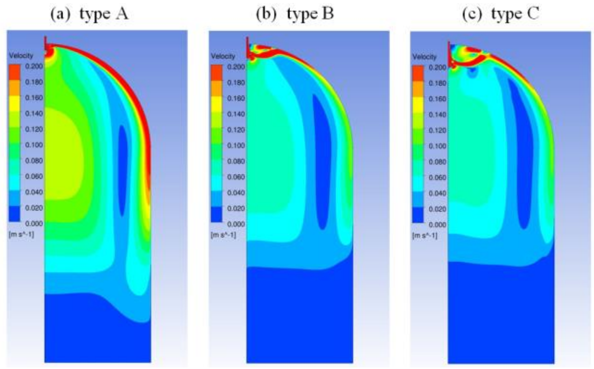

Fig. 4. Fluid velocity contours in computational domain.

ACKNOWLEDGEMENTS

This work was supported by the National Research Foundation of Korea (NRF) funded by the Korea government (MSIP) (No. NRF-2012M2A8A4025974).

REFERENCES

[1] S. Cho, S.-Y. Chun, W.-P. Baek, Effect of multiple holes on the performance of sparger during direct contact condensation of steam, Experimental Thermal and Fluid Science Vol. 28, pp. 629-638, 2004.

[2] J. Tuunanen, J. Vihavainen, F. D’Auria, G. Kimber, Assessment of passive safety injection systems of ALWRs, technical research centre of finland, 1999.