Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 29-30, 2014

CFD analysis of Siphon Break in a Research Reactor

Hong Beom Park*, Kyoungwoo Seo, Seong Hoon Kim, Dae Young Chi
KAERI, 111 Daedeok-Daero 989 beon-Gil, Yuseong-gu, Daejeon
**Corresponding author: hbpark@kaeri.re.kr*

1. Introduction

In open pool type research reactor, reactor core is cooled by natural circulation after the primary cooling pump is turned off and the pool water is used as the ultimate heat sink. The pool water also behaves as a shielding barrier for many kinds of radio-nuclides from the reactor core and the spent fuel. Pool water is essential for nuclear safety. So guaranteeing the pool water inventory to be higher than the required minimum level is one of the most important tasks of a research reactor design. The lowest pool penetration of cooling pipes should be located above the reactor core against a cooling pipe break. However, system components outside the pool can be installed below the core level due to the component purpose such as the acceptance of a Net Positive Suction Head(NPSH) of a pump for downward core flow research reactor. So the pool water can be drained below the core through siphon effect and the core can't be cooled through natural circulation when a postulated pipe break occurs below the reactor core position. Therefore siphon breaker should be installed to limit the pool water drain.

In this study, 3D numerical simulations are performed to be applicable to the siphon breaker design for a research reactor because undershooting(height between the end of siphon break line and the final pool water level) is expected for a large pipe break(Fig. 1). ANSYS CFD is used to solve the Navier-Stokes equation with the turbulent model and two-phase model.

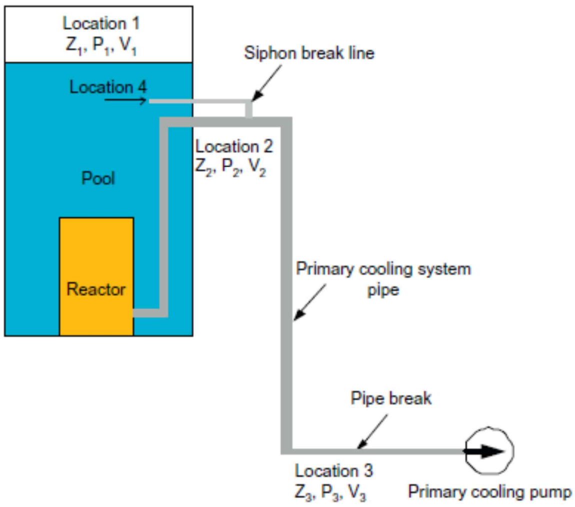

Fig. 1. Schematic diagram of the siphon break phenomena

2. Methods and Results

Numerical simulation using the commercially available CFD code, ANSYS CFD, which solved the Navier-Stokes, turbulent model, and two-phase model for various fluid dynamics was used to aid in the understanding of the basic mechanism of siphon break phenomena in a large pipe installed at an actual research reactor.

2.1 Numerical Model

In order to simulate two-phase phenomena, two-phase model is used. The ANSYS CFD employs homogeneous model and inhomogeneous model. Since the homogeneous model assumes that both phases are moving with same velocity. It solves the bulk transport equation.

$$\frac{\partial}{\partial t}(\rho\varphi) + \nabla \cdot (\rho U \varphi - \Gamma \nabla \varphi) = S$$

For siphon break, each fluid can have own flow field and fluids interact through interphase transfer terms. The inhomogeneous model gives solution for each separate phase.

$$\frac{\partial}{\partial t}(r_\alpha \rho_\alpha) + \nabla \cdot (r_\alpha \rho_\alpha U_\alpha) = S_{MS\alpha} + \sum_{\beta=1}^{N_p} \Gamma_{\alpha\beta}$$

$$\frac{\partial}{\partial t}(r_\alpha \rho_\alpha U_\alpha) + \nabla \cdot (r_\alpha(\rho_\alpha U_\alpha \otimes U_\alpha))$$
$$= -r_\alpha \nabla p_\alpha + \nabla \cdot (r_\alpha \mu_\alpha (\nabla U_\alpha + (\nabla U_\alpha)^T)) + S_{M\alpha} + \sum_{\beta=1}^{N_p} (\Gamma^+_{\alpha\beta} U_\beta$$
$$- \Gamma^+_{\beta\alpha} U_\alpha) + M_\alpha$$

$$\sum_{\alpha=1}^{N_p} r_\alpha = 1$$

Where r, S_{MS} and S_M describe the void fraction, mass and momentum sources of phase.

2.2 Mesh and boundary condition

Fig.2 shows mesh of siphon break simulation. The number of mesh is above 1 million. The meshes are composed of hexagonal type and tetrahedral type. The pool water surface is modeled by the standard free surface model and the opening type boundary condition. The fluid flows out through bottom pipe. Simulation starts from the initial condition of the stagnant flow of the pool water and is progressed by the head of the pool water.

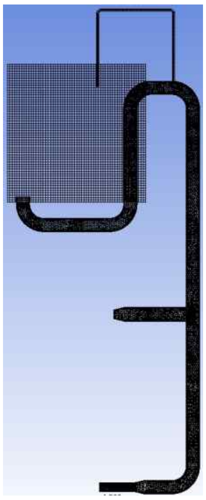

Fig. 2. Mesh of siphon break

2.3 Results

Fig. 3 shows water volume fraction after siphon break. SST turbulence model is used. Height of final pool water level is under the end of siphon break line. Undershooting means difference of height between the end of siphon break line and pool water level. Undershooting height is different with various models. The undershooting height of homogeneous model is about 2 times higher than that of the inhomogeneous model. The result for free surface option of two-phase model is in table 1. When using standard free surface and no free surface, there is no difference to undershooting height. It can be adapted to continuum surface force surface tension model and no surface tension model. It shows two-phase phenomena for siphon break is not associated with free surface. Table 2 shows comparison of undershooting height with different surface tension coefficient. It also shows free surface is not associated with siphon break phenomena.

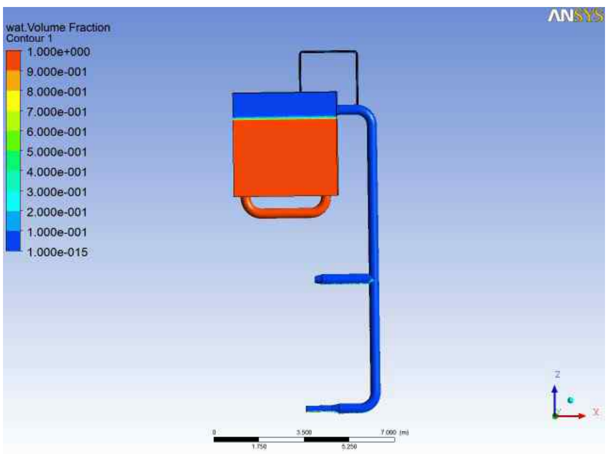

Fig. 3. Water volume fraction after siphon break

Free surface option	Surface tension model	Turbulence model	Two phase model	Undershooting height ratio
None	None	SST	Inhomogeneous	1(criterion)
Standard	None	SST	Inhomogeneous	1
Standard	Continuum surface force	SST	Inhomogeneous	1

Table 1: Comparison of undershooting height with different free surface option

Surface tension coeff. (ratio)	Turbulence model	Two phase model	Undershooting height ratio
0.5	SST	Inhomogeneous	1
1 (criterion)	SST	Inhomogeneous	1 (criterion)
1.5	SST	Inhomogeneous	1

Table 2: Comparison of undershooting height with different surface tension coefficient

Table 3 is comparison of undershooting height with different interface length scale. Interface length scale is used in mixture model and affects interfacial area per unit volume in ANSYS. The result shows undershooting height is affected by interface length scale. So it shows mixture model can have major effect to siphon break phenomena.

Interface length scale (ratio)	Turbulence model	Two phase model	Undershooting height ratio
3	SST	Inhomogeneous	2
1.5	SST	Inhomogeneous	1.6
1(criterion)	SST	Inhomogeneous	1(criterion)
0.8	SST	Inhomogeneous	0.9

Table 3: Comparison of undershooting height with different interface length scale

3. Conclusions

Siphon breaker was designed to satisfy the minimum pool water level requirement during pipe break in a research reactor, and it is necessary to analyze siphon break phenomena.

The results employing the homogenous model and inhomogeneous model with the SST turbulence model were compared. Homogeneous model has higher undershooting height than inhomogeneous model. And results with various free surface option show siphon break phenomena is not associated with free surface.

Based on a numerical simulation, it was evaluated that a siphon break is dependent on the air-water flow mixture and interface length scale.

REFERENCES

[1] K.Seo, S.H.Kang, J.M.Kim, K.Y.Lee, N.Jung, D.Y.Chi, J.Yoon and M.H.Kim, Experimental and numerical study for a siphon breaker design of a research reactor, Annals of Nuclear Energy, 50, 94-102, 2012