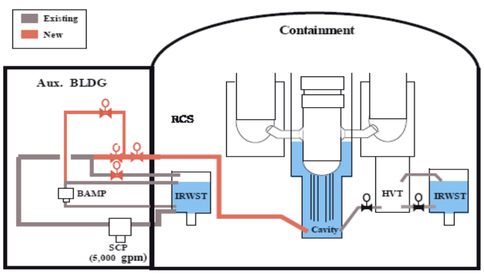
Numerical analysis of boiling and two-phase natural circulation flow in IVR-ERVC system

Tae-Hyub Hong [a], Choengryul Choi [b*], Hyeong-Taek Kim[a]
[a] *Central Research Institute of KHNP, Yuseong, Daejeon, 305-343, Republic of Korea*
[b]*ELSOLTEC Inc., Suwon, Gyeonggi, 443-851, Republic of Korea*
[*]*Corresponding author: crchoi@elsoltec.com*

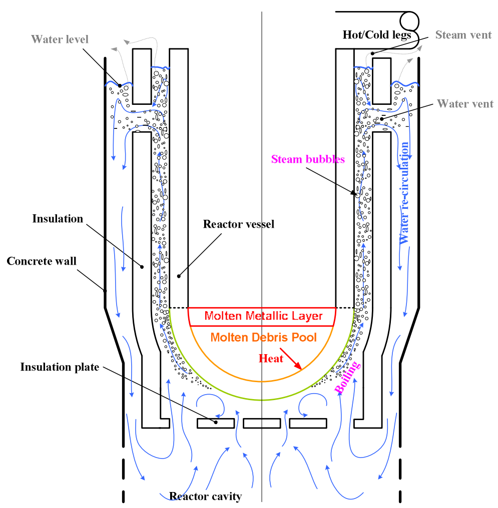
1. Introduction

The hypothetical scenario of a severe accident with core meltdown and formation of a melt pool in the lower plenum of the reactor pressure vessel (RPV) can result in the failure of the RPV and the discharging of the melt to the containment. One accident management strategy could be the stabilization of the RPV by cooling the outside vessel wall with water. This is the concept of In-Vessel Retention External Reactor Vessel Cooling (IVR-ERVC) (Fig. 1).

The purpose of this study is to offer detailed information on thermal hydrodynamic phenomena in APR1400 for IVR-ERVC. The thermal hydrodynamic phenomena are simulated using a computational fluid dynamics (CFD) technique with phase change heat/mass transfer approach.



(a) Emergency core cooling system



(b) General flow pattern

Fig. 1 Schematic diagram of IVR-ERVC.

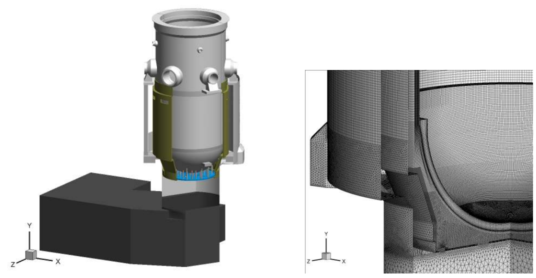


Fig. 2 3D Configuration for the numerical analysis.

2. Numerical Methods

The thermal hydrodynamic phenomena in APR1400 for IVR-ERVC are simulated using a computational fluid dynamics (CFD) technique. The generation of steam bubbles on the outer surfaces of the reactor vessel is modeled using a phase change heat/mass transfer approach. Also, two-phase (gas-liquid) flow of the generated steam bubbles and cooling water is simulated using an explicit VOF (Volume Of Fluid) model.

A three-dimensional geometric model for numerical simulation based on the current design drawings is generated as shown in Fig. 2. For a numerical calculation based on finite volume formation, a three-dimensional mesh system is generated with about 7,000,000 cells which consisted of hexahedral and tetrahetral cells.

Several conservation equations are numerically solved to obtain flow and temperature fields. A volume fraction equation is solved to track the interfaces between two phases (gas-liquid; steam bubbles - cooling water) [1,2]. The momentum equation is solved to obtain velocity and pressure fields. The energy equation is solved to obtain temperature distribution. To include turbulent effects in the flow domain, the standard k-ε model is adopted, and thus the turbulent kinetic equation and turbulent kinetic energy dissipation equation are solved together. The generation of steam bubbles by boiling on the outer surfaces of the reactor vessel is modeled using a phase change heat/mass transfer approach via UDF (user defined function).

2.1 Mass transfer

A mass transfer model concerning the process of evaporating is used via UDF. In this study, we assume that the phase change is assumed to occur at a constant pressure and at a quasi thermo-equilibrium state, and the mass transfer is mainly dependent of the saturate temperature. Applied phase change process is described as

$$S = -\gamma_l \alpha_l \rho_l \left|(T_l - T_{sat})\right| / T_{sat} \qquad T_l \geq T_{sat}$$
$$= \gamma_v \alpha_v \rho_v \left|(T_{sat} - T_v)\right| / T_{sat} \qquad T_l < T_{sat}$$

In this study, the interfacial temperature is assumed at the saturation temperature.

2.2 Heat transfer

The heat transfer is simply determined from the mass rate of boiling, in addition to the conduction and convection. As long as the mass transfer is obtained, the heat transfer could be directly determined as

$$Q = -h_{LH} S$$

Transient numerical calculation is implemented in a commercial computational software with the generated mesh system and calculation conditions. Pressure-based approach is used for flow analysis. The velocity field is obtained from the momentum equation, and pressure field is extracted by solving a pressure correction equation which is obtained by manipulating continuity and momentum equations. This approach solves the governing equations for the conservation of mass and momentum, and for energy and other scalars such as turbulence. For transient simulation, the governing equations are discretized in both space and time with the second order upwind scheme and the second order implicit scheme, respectively. For the pressure-velocity coupling, the Pressure-Implicit with Splitting of Operators (PISO) pressure-velocity coupling scheme is used. The PRESTO! (PREssure STaggering Option) scheme interpolates the pressure values at the faces using momentum equation coefficients. To ensure sharp interface treatment between two phases, the explicit VOF scheme and Geo-Reconstruct scheme are used [3,4,5].

3. Results

The calculation domain is initially filled up with cooling water whose initial temperature is 363K, and the flow is initially assumed to settle down. The heat flux curve shown in Fig. 3 is applied to the inner surface of the reactor vessel as heat boundary conditions.

Steam bubble generation and behavior are shown in Fig. 4. The temperature distribution on the outer surface of the reactor vessel, the temperature distribution of the cooling water and velocity vectors are also found in Fig. 4. These numerical simulation shows that boiling actively occurs on the outer surface of the lower hemispherical vessel. Also, the circulation of the cooling water is induced by the generated steam bubbles.

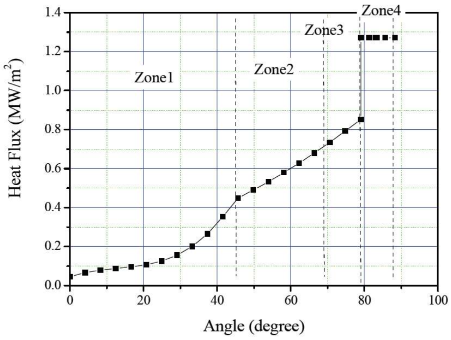


Fig. 3 Heat flux on the reactor inner wall.

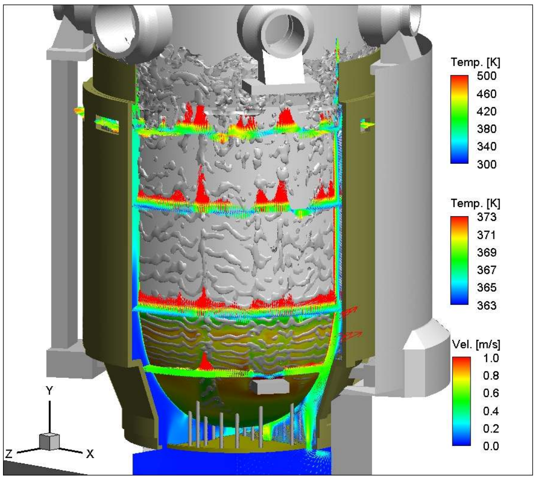


Fig. 4 Bubble behavior, velocity vectors and temperature distributions on the outer surface of the vessel and cooling water (T_{water} = 90°C).

4. Conclusions

The thermal hydrodynamic phenomena for IVR-ERVC in APR1400 have been successfully simulated using a computational fluid dynamics (CFD) technique. The applicability of phase change heat/mass transfer model has been proved through this study.

REFERENCES

[1] T. B. Anderson and R. Jackson, A Fluid Mechanical Description of Fluidized Beds, I & EC Fundam., 6, 527-534, 1967.
[2] R. M. Bowen, Theory of Mixtures, In A. C. Eringen, editor Continuum Physics, Academic Press, New York, 1-127, 1976.
[3] D. L. Youngs, Time-Dependent Multi-Material Flow with Large Fluid Distortion. Numerical Methods for Fluid Dynamics, K. W. Morton and M. J. Baines, editors Academic Press., 1982.
[4] C. W. Hirt and B. D, Nichols, Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries, J. Comput. Phys., 39, 201-225, 1981.
[5] J. U. Brackbill, D. B. Kothe, and C. Zemach, A Continuum Method for Modeling Surface Tension, J. Comput. Phy., 100, 335-354, 1992.