Comparison of the Phase-Change Models of Enthalpy-Porosity Methodology for Mushy Zone Problems in LIVE L7V test

Seokwon Whang[a], Kukhee Lim[b*], Hyun Sun Park[a]
[a]*Nuclear Research Institute for Future Technology and Policy, Seoul National Univ., Seoul, Republic of Korea*
[b]*Department of Nuclear Safety Research, Korea Institute of Nuclear Safety, Daejeon, Republic of Korea*
[*]*Corresponding author: limkh@kins.re.kr*

1. Introduction

To evaluate in-vessel retention (IVR) strategy which is the severe accident management strategy of nuclear power plants, the thermal load of the corium relocated to the reactor pressure vessel (RPV) should be investigated. Various phenomena, such as natural convective flow, phase-change of the core melt, conjugate heat transfer between melt and vessel wall, heat transfer of the external vessel, etc. can be affected to the thermal load on the RPV. Among the corium heat transfer experiments, the object of LIVE experiment is to include most important phenomena of corium pool. In addition, one of the goals of LIVE experiment is to investigate the phase-change phenomenon in mushy zone by using non-eutectic binary material KNO_3-$NaNO_3$.

In previous research [1], a numerical platform was developed to simulate phase-change problem for the pure material by using Enthalpy-porosity method. Although the numerical platform was validated with LIVE experiment, the phase change temperature was arbitrary determined between solidus and liquidus temperature. In this paper, a phase-change model of Enthalpy-porosity method is improved to consider the mushy zone problems. Following the previous research, the various phase-change model for the mushy zone problem are implemented in the OpenFOAM code, and compared with the LIVE experiment.

2. Numerical method

2.1 Enthalpy-porosity methodology for phase change problem

Enthalpy-porosity methodology (EPM) is one of the methodologies to simulate phase change phenomenon [2]. One of the advantages of EPM is that the simulation can be done in a fixed-grid system by including source term in momentum and energy equation. In the EPM, a liquid fraction (g_l) is calculated according to its temperature, and the computational domain can be classified a liquid region (g_l = 1) and a solid region (g_l = 0). In the momentum equation in Eq. (2) and the temperature equation in Eq. (3), source terms ($A\vec{u}$ and Sh) are contained, respectively. As previously mentioned, the liquid fraction (g_l) can be calculated based on the field temperature during the simulation in Eq. (4) and Eq. (5). Firstly, the method can be divided into two categories depending on the material i.e. pure and mixture material (EPM and EPM$^+$). While the liquid fraction is zero or one depending on the field temperature in Eq. (4), the liquid fraction of mixture material varies between solidus and liquidus temperature in Eq. (5). Linear variation was assumed in this study. The source terms $A\vec{u}$ and Sh can be modelled as function of liquid fraction based on modelling assumptions. The models used in this study are summarized in Table I.

$$\nabla \cdot \vec{u} = 0 \tag{1}$$

$$\frac{\partial \vec{u}}{\partial t} + \nabla \cdot (\vec{u}\vec{u}) = -\nabla p_m + \nabla \tau_{ij} + A\vec{u} \tag{2}$$

$$\frac{\partial c_p T}{\partial t} + \vec{u} \cdot \nabla (c_p T) = \nabla \cdot (\frac{\kappa}{\rho} \nabla T) + S_h \tag{3}$$

$$g_l = \begin{cases} 0 & T < T_m \\ 1 & T > T_m \end{cases} \tag{4}$$

$$g_l = \begin{cases} 0 & T > T_l \\ \left(\frac{T - T_s}{T_l - T_s}\right) & T_l > T > T_s \\ 1 & T < T_s \end{cases} \tag{5}$$

$$A = -C(1 - g_l) \tag{6}$$

$$A = -C\frac{(1 - g_l)^2}{g_l^{\,3} + b} \tag{7}$$

$$S_h = -L\frac{\partial g_l}{\partial t} \tag{8}$$

$$S_h = -L(\frac{\partial g_l}{\partial t} + \vec{u} \cdot \nabla g_l) \tag{9}$$

Table I: Phase change models of Enthalpy-porosity method

Model	g_l	A	Sh
EPM(T_{liq})	Eq. (4)	Eq. (6)	Eq. (8)
EPM(T_{sol})	Eq. (4)	Eq. (6)	Eq. (8)
EPM_a^+	Eq. (5)	Eq. (6)	Eq. (8)
EPM_b^+	Eq. (5)	Eq. (6)	Eq. (9)
EPM_c^+	Eq. (5)	Eq. (7)	Eq. (8)
EPM_d^+	Eq. (5)	Eq. (7)	Eq. (9)

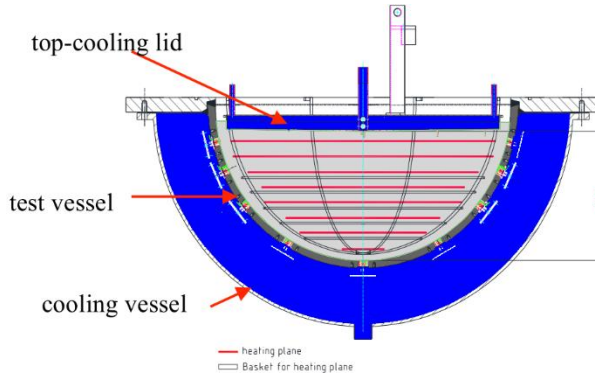

Fig. 1. LIVE test facility with top-cooling lid [3]

2.2 LIVE test [3]

The LIVE experiment was conducted to investigate the thermal behavior of core melt during IVR. Since the salt (a non-eutectic binary mixture of 20 mol% $NaNO_3$-80 mol% KNO_3 composition) was used as a simulant material for specific cases, the solidification along the vessel wall can be simulated. Fig. 1 shows the LIVE facilities, which consists of a 3D hemisphere (R = 0.5m), a side vessel wall (thickness = 0.025m), and an upper lid. The height of the upper lid can be adjusted according to the conditions, and the height of the lid is fixed to 0.42m in this study. Inside the hemisphere, there is a spiral heater, which simulates decay heat. A sufficient amount of water is supplied outside the side wall surface and the upper lid to maintain constant cooling conditions.

LIVE-L7V is selected in this study which is lateral and upper cooling conditions, and the working fluid is non-eutectic binary material. The phase-change along the cooling boundary condition is expected. Table II shows the main thermal-physical properties of salt used in the L7V. The vessel wall is SS316Ti and has a thermal conductivity of 14.6 W/m-K, heat capacity of 500 J/kg-K, and a density of 7,870 kg/m^3.

2.3 Numerical conditions for LIVE simulation

Fig. 2 shows the grid system used in the simulation. The grid was largely divided into a working fluid region and a vessel wall region, and in particular, the grid for the fluid region was clustered near the wall to capture the near wall behavior. After performing the grid test [1], the optimal number of grids both conditions is selected about 568,000. The time step was determined as 0.1 seconds, which satisfied the Courant and Diffusion number in the fluid and solid regions sufficiently small. In this study, the OpenFOAM was used, and the '`chtMultiRegionFoam`' which is a standard solver for fluid flow and solid heat conduction, along with conjugate heat transfer between regions etc. was modified to include EPM.

Table II: Thermal-physical properties of 20 mol% $NaNO_3$-80 mol% KNO_3 [2]

Parameters	Values
Density	1868 kg/m^3
Dynamic viscosity	1.81×10^{-3} kg/m-s
Thermal conductivity (liq.)	0.439 W/m-K
Thermal conductivity (sol.)	0.6 W/m-K
Specific heat capacity (liq.)	1331 J/kg-K
Specific heat capacity (sol.)	1060 J/kg-K
Liquidus temperature	557 K
Solidus temperature	439 K
Latent heat of fusion (L)	161,956 J/kg

Fig. 2. Grid used for the simulation.

3. Results

The simulation was performed using various EPM models as listed in the Table I. In here, EPM and EPM$^+$ represent the EPM for pure and mixture material, respectively. In the figure, the square symbol represents the experimental data; dashed and dotted line represent EPM with T_{liq} and T_{sol}, respectively. Finally, the solid line represents EPM$^+$ result.

Fig. 3 shows the melt temperature at 0.175 m from the center line. Results with EPM vary greatly depending on the phase change temperature. The results with EPM with T_{liq} and EPM$^+$ show similar tendency. As mentioned previous study [1], it is expected that the crust will be formed below the height of 0.05 m. Except that regions, the results with EPM with T_{liq} and EPM$^+$ are well matched with experimental data. Fig. 4 shows the heat flux profile along the vessel wall. Except the EPM with T_{sol}, all the data shows similar trend from the bottom to the middle part of the vessel wall (polar angle < 50°). While the maximum heat flux can be seen near the top surface, the predicted heat flux by using EPM$^+$s decreases near the top. EPM_b^+ slightly underestimated the heat flux of upper part compared to other EPM$^+$. Finally, crust thickness along the inner vessel wall is compared in Fig. 5. The line of the EPM$^+$ represents a liquid fraction of 0.9. The length between the liquid fraction 0 and 0.8 at the bottom of the pool is about 0.015m. All the EPM$^+$ data located between EPM with T_{liq} and T_{sol}.

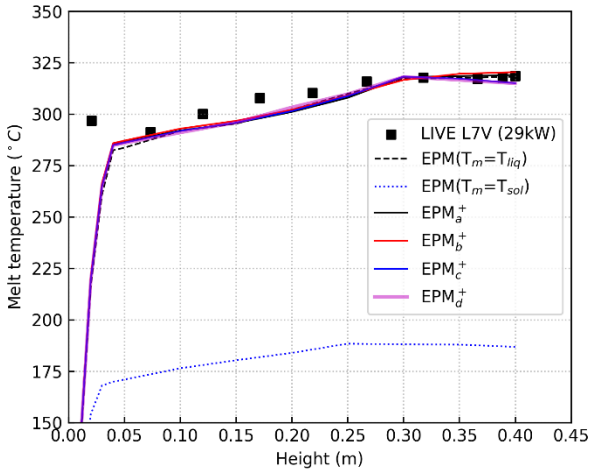

Fig. 3. Melt temperature measured at 0.175m from the center line

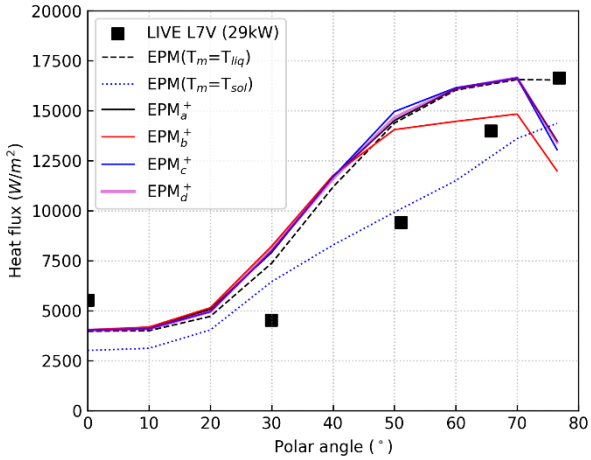

Fig. 4. Heat flux profile along the vessel wall

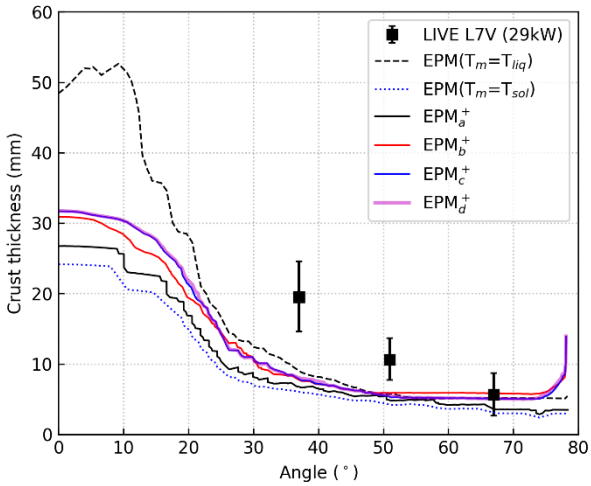

Fig. 5 Crust thickness along the inner vessel wall

4. Conclusion

The complex phenomena of in-vessel corium should be investigated to evaluate the thermal load to the RPV during the severe accident. In this paper, the numerical platform was improved to simulate the phase change heat transfer of mixture material, and the model was validated with the representative experiment, LIVE L7V. Current result with the EPM for mixture material shows good agreement with experimental data.

Before applying this method to the reactor case, further validation such as L7V with different power level and L7TC which has different thermal boundary condition is needed. Furthermore, high turbulence is expected in the reactor scale, the turbulence model study with EPM is also needed.

ACKNOWLEDGEMENTS

This work was supported by the Nuclear Safety Research Program through the Korea Foundation of Nuclear Safety (KoFONS) using the financial resource granted by the Nuclear Safety and Security Commission (NSSC) of the Republic of Korea (Grant No. 2106033-0222-CG100).

REFERENCES

[1] Whang, S., Lim, K., Park. H. S., 2022, Implementation of Enthalpy-Porosity Methodology in OpenFOAM for Validation of LIVE L7V and L7W tests, Korean Nuclear Society Spring Meeting.

[2] Brent, A.D., Voller, V.R., Reid, K.J., 1988. Enthalpy-porosity technique for modeling convection-diffusion phase change: Application to the melting of a pure metal. Numer. Heat Transf. 13, 297–318.

[3] Miassoedov A, Gaus-Liu X, Cron T, Fluhrer B, 2014. LIVE experiments on melt pool heat transfer in the reactor pressure vessel lower head, HEFAT2014.

[4] Gau, C., Viskanta, R., 1986. Melting and Solidification of a pure metal on a vertical wall. J. Heat Transfer 174-181.