CFD simulation of Boiling Two-Phase Flows with Different Thermophysical Properties for Core-Catcher Cooling System

Sangmin Kim [a]*, Keunsang Choi[a] and Jeahoon Jung[a]
aKorea Atomic Energy Research Institute, 111, Daeduk-daero 989, Daejeon, Korea
Corresponding author: ksm0226@kaeri.re.kr

1. Introduction

PECS (Passive Ex-vessel corium retaining and Cooling System) installed inside EURO-APR1400 is a system that cools the ex-vessel corium by using core-catcher equipment in the event of a nuclear accident [1]. The water filled inside the containment is naturally circulated due to the high temperature of the corium, which continuously cools the core-catcher. During cooling, a boiling phenomenon occurs on the wall of the core-catcher, and the cooling water takes the form of a two-phase flow including water vapor. Therefore, in order to understand the cooling performance of the core-catcher using CFD, it is important to predict the two-phase flow including boiling.

The density, dynamic viscosity, and specific heat capacity of water vary from 5% to 90% with temperature. (Fig. 1) However, in general CFD solvers, these thermophysical properties are treated as constants for convenience of calculation.
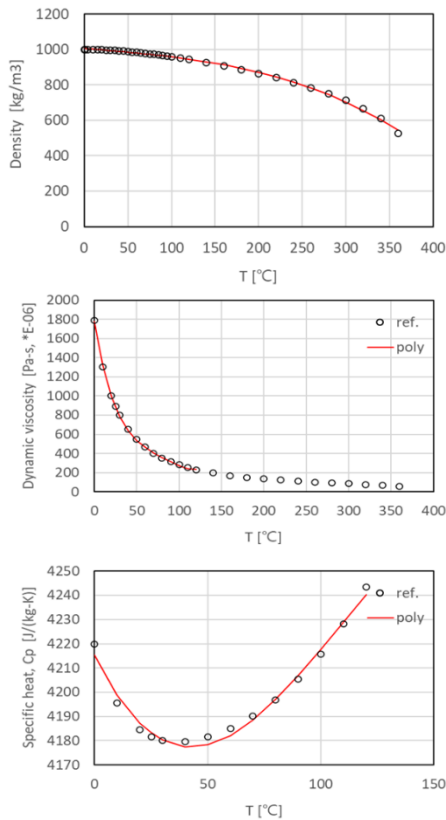
In this study, the difference in two-phase flow was compared when the thermophysical properties of water were treated as constants and when applied in the form of polynomials according to temperature. For this purpose, a solver code was developed so that the liquid phase properties can be entered as polynomials.

The CFD calculation benchmarked the CE-PECS (Cooling Experiment-PECS) T8-4 experiment [2] performed by KAERI based on OpenFOAM [3].

2. CE-PECS test

Fig. 2 shows a schematic diagram of the CE-PECS experiment. In CE-PECS, Heat flux from the molten corium to the core catcher was realized by the electrical heating block. Water is filled to a height of 7.56 m from the floor of the test facility, and the initial temperature of the cooling water is 89 ℃. A total of 7 heating blocks are located on the upper surface of the core-catcher, and a cooling channel is located at the bottom. The heat flux transferred from each heating block to the core catcher is shown in Table 1. And the detailed location of the heating block is shown in Figure 3.
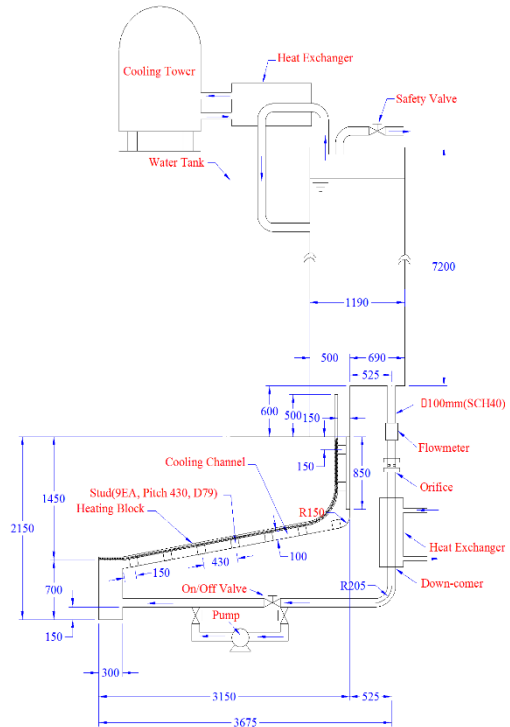

Fig. 1. Thermophysical properties.

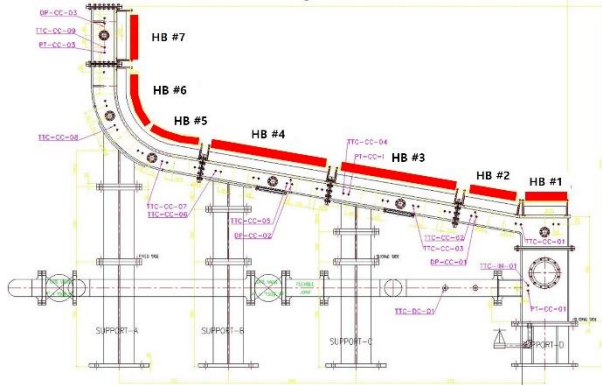Fig. 2. Schematic of the CE-PECS test facility.

Fig. 3. Location of HBs (Heating Blocks)

Table I: Heat fluxes in CE-PECS T8-4 experiment.

Average heat flux in heating block #1	60.9 kW/m²
Average heat flux in heating block #2	46.4 kW/m²
Average heat flux in heating block #3	89.1 kW/m²
Average heat flux in heating block #4	98.3 kW/m²
Average heat flux in heating block #5	150.3 kW/m²
Average heat flux in heating block #6	220.9 kW/m²
Average heat flux in heating block #7	261.0 kW/m²

3. CFD analysis

3.1 Governing equations

A solver for CFD simulation was developed based on chtMultiRegionTwoPhaseEulerFoam in OpenFOAM. The solver handles all of conjugate heat transfer, boiling-condensation phenomenon, and two-phase flow. The governing transport equations used for the two-phase flow is:

$$\frac{\partial \alpha_\varphi \rho_\varphi Y_i}{\partial t} + \nabla \cdot (\alpha_\varphi \rho_\varphi \bar{\mathbf{U}}_\varphi Y_i) - \nabla \frac{\alpha_\varphi \mu_\varphi}{Sc_{i\varphi}} \nabla(Y_i) = \frac{dm_i}{dt} \quad (1)$$

Dynamic viscosity(μ) effects on this equation. Since this solver is for compressible flow, density(ρ) is handled in all governing equations such as energy conservation equation and momentum conservation equation in addition to the above equation. And specific heat capacity(C_p) is used in the temperature calculation.

3.2 Thermophysical models

Based on the input of thermodynamic properties of liquid water, it was divided into a constant case and a polynomial case. The thermophysical values in the polynomial case are calculated as a function of temperature. The values used for CFD analysis are summarized in Table 2 below. The properties of water vapor and air were set the same in both cases.

Table II: Thermophysical values for liquid water.

	Constant case	Polynomial case
C_p [J/(kg K)]	4195	4900 − 2.94*T − 2.34*10⁻³*T² + 1.43*10⁻⁵*T³
μ [kg/(m s)]	3.645*10⁻⁴	0.34238 − 3.8343*10⁻³*T + 1.6173*10⁻⁵*T² − 3.0383*10⁻⁸*T³ + 2.1423*10⁻¹¹*T⁴
ρ [kg/m³]	1000	1250 − 1.91*T + 5.51*10⁻³*T² − 6.73*10⁻⁶*T³

3.3 Mesh

The mesh was created in the form of left-right symmetry modeling the shape of the CE-PECS equipment using the SALOME program. It consists of about 50,000 hexahedral cells and is divided into three parts: stud, flow region, and catcher.
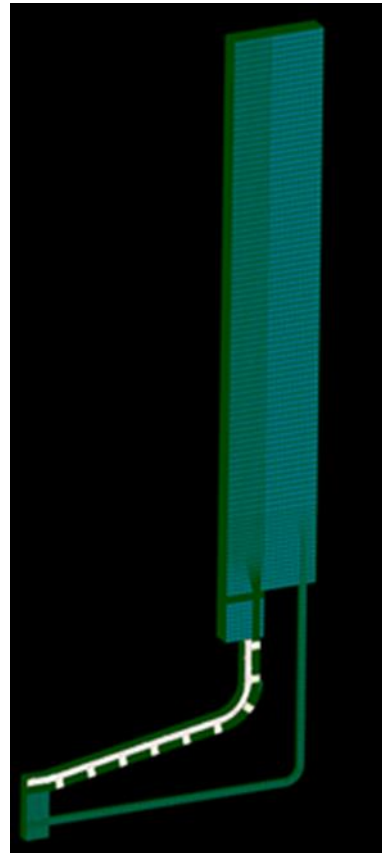

Fig. 4. Mesh system: green region – flow region, white region – catcher and studs)

3.4 Shaping Amplifier Model

Figure 5 shows the location of the heating block and the realization of heat on the catcher surface. This heat is transferred to the cooling water in the cooling channel to generate a natural circulation flow.
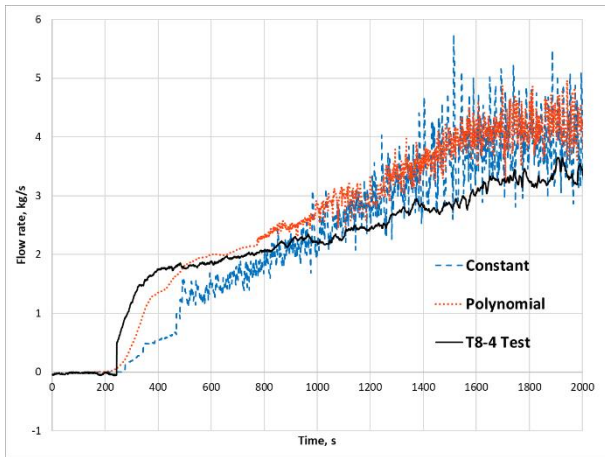

Fig. 5. Heats from heating blocks

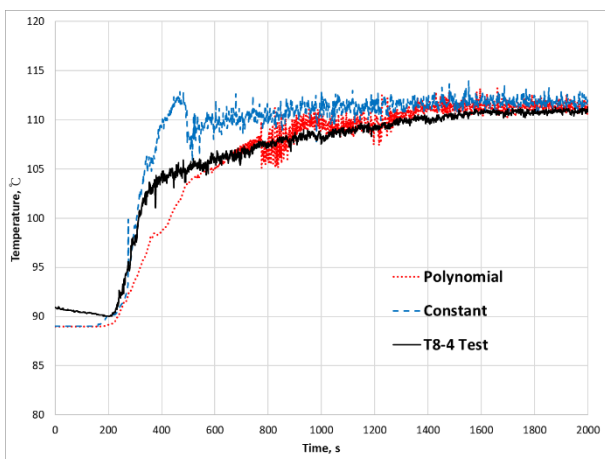Fig. 6. Flow rate in cooling channel.

+

Fig. 7. Temperature at the exit of cooling channel

Figure 6 is a graph of flow rate change according to time inside the cooling channel after heating starts.

At about 400 s after the start of the experiment, the temperature of the cooling water reached the saturation temperature, and boiling started on the wall of the core-catcher. Before the flow due to the rise of water vapor generated by boiling occurs, the flow occurs due to the difference in the density of water.

In the constant case, the flow increased slowly. This is because there is no change in the density of water according to the temperature, and the dynamic viscosity is also constant, so the flow does not become smooth even when the temperature increases. The maximum flow rate was predicted similarly to the polynomial case, but the amplitude was quite large, resulting in unstable results.

Figure 7 is a graph of temperature change with time at the outlet of the cooling channel.

The temperature range of the cooling water in the experiment is 89°C to 110°C. In this section, the specific heat capacity increases with the temperature, and as the temperature of the cooling water increases, more heat is required to raise the temperature. In the case of the constant case in which C_p is fixed, the heat required to raise the temperature is lower, so the temperature of the coolant rises faster and more steeply than in the polynomial case.

3. Conclusions

The CFD code has been improved so that the thermophysical properties of the fluid can be treated as a polynomial type that changes with temperature. As a result of benchmarking the CE-PECS experiment using this code, it was shown that the accuracy was significantly improved compared to the previous case where only constant properties could be input. The code developed in this study can be used as a tool to evaluate the cooling stability of the PECS system in case of a severe accident.

4. Acknowledgment

REFERENCES

[1] J. H. Song, et al. A core catcher design for advanced light water reactors. Proceedings of International Conference on Advanced Nuclear Power Plants, 2011.
[2] K. S. Ha, et al. KAERI/TR-5813/2014, Korea Atomic Energy Research Institute, 2014.
[3] OpenFOAM Web site, https://openfoam.org.