Transactions of the Korean Nuclear Society Virtual Autumn Meeting
October 21-22, 2021

Development of Two-Phase Flow CFD Code to Improve Core-Catcher Cooling Performance

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

1. Introduction

The European APR1400 has a core catcher installed for PECS (Passive Ex-vessel corium retaining and Cooling System) in the event of a nuclear accident [1]. When high-temperature corium falls on the surface of the core catcher, it is cooled through the natural circulation of the coolant inside the core catcher. The CE-PECS (Cooling Experiment-PECS) experiment [2] conducted by KAERI simulates the natural circulation of such cooling system. A schematic diagram of the CE-PECS experiment can be seen in Figure 1. Heat flux from the core melt to the core catcher was realized by the electrical heating block.

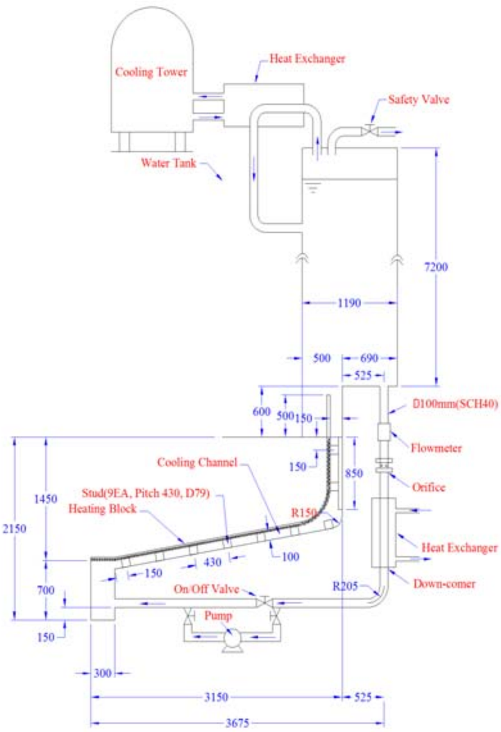

Fig. 1. Schematic of the CE-PECS test facility.

The code used in this study was developed to simulate the flow inside the core catcher due to the heat flux of corium based on the OpenFOAM CFD tool [3]. A dedicated code for calculating the cooling water circulation and cooling of the core catcher has not yet been developed, and so far has been done using the RELAP5 tool [4].

To verify the code, benchmark simulations of CE-PECS T8-4 experiments were performed.

2. CE-PECS test

The experimental device was manufactured with a width of 0.3 m, a length of about 3 m, a height of about 10 m, and a cooling path gap of 10 cm.

Table 1 below shows the input conditions of the CE-PECS T8-4 experiment.

Table I: CE-PECS T8-4 experiment condition.

Coolant level	7.56 m
Initial coolant temperature	89 ℃
Total heating time	2830 s
Total power	123.3 kW (75%)
Average heat flux in heating block #1	60.9 kW/m^2
Average heat flux in heating block #2	46.4 kW/m^2
Average heat flux in heating block #3	89.1 kW/m^2
Average heat flux in heating block #4	98.3 kW/m^2
Average heat flux in heating block #5	150.3 kW/m^2
Average heat flux in heating block #6	220.9 kW/m^2
Average heat flux in heating block #7	261.0 kW/m^2

The exact location of each HB (heating block) is shown in Figure 2.

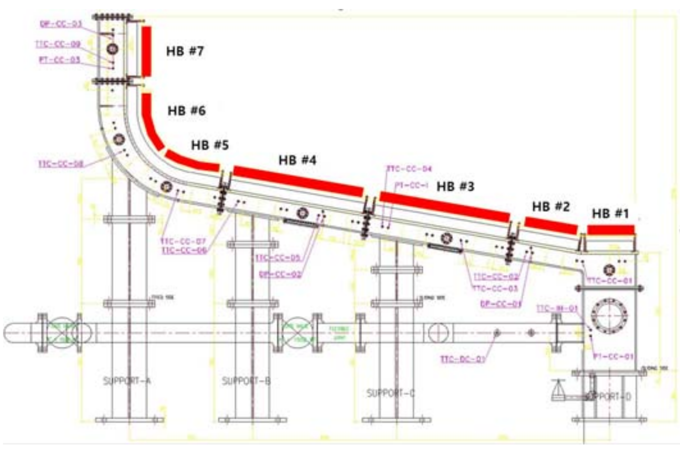

Fig. 2. Location of HBs (Heating Blocks)

3. Methods and Results

A solver for simulating was developed based on chtMultiRegionTwoPhaseEulerFoam in OpenFOAM. In the core catcher analysis boiling at the wall of the cooling channel, heat transfer of core-catcher, natural convection, water vapor condensation and two-phase flow should be considered. For the flow phenomena listed above, the following models were used in this solver. Subcooled and saturated nucleate boiling model, solid-fluid conjugate heat transfer model, convective natural circulation model, pool condensation model and two-phase bubbly-slug flow model.

Figure 3 shows the shape of the grid used in this study. Simulation was performed using this mesh and the conditions in Table 1.

It consists of about 50,000 fully hexahedral cells and is divided into three parts: flow, catcher and stud.

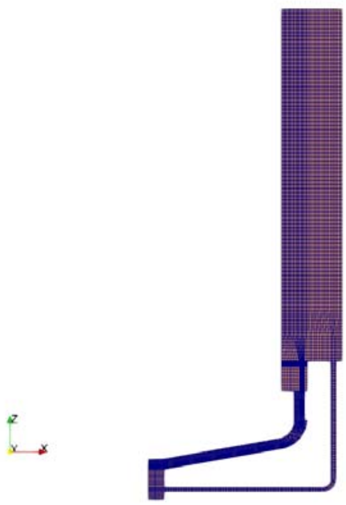

Fig. 3. Mesh used in CFD simulation

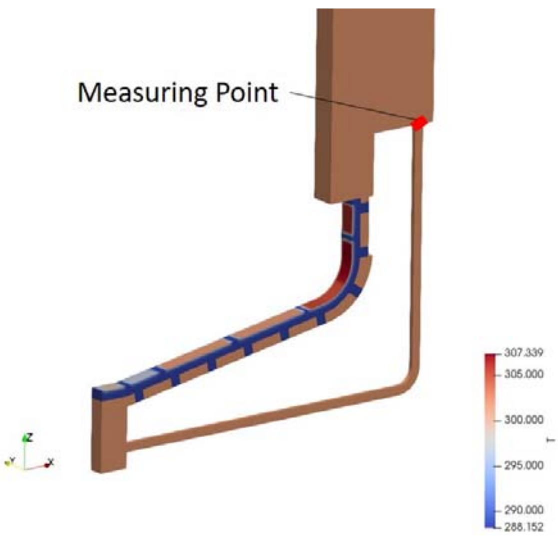

Fig. 4. Heat from heating blocks

Figure 4 shows that the core catcher is being heated by the heat flux of the heating block. Since it is the part in contact with the coolant, it is maintained below the boiling temperature.

In both experiments and simulations, changes in flow rate and temperature were measured and compared at the 'Measuring Point' in Figure 4.

Figure 5 is a graph of the temperature of the coolant that changes as the heating block is heated. As a result of the simulation, the temperature increase trend was exactly the same, but a difference of up to 4 degrees was recorded.

Figure 6 is a graph measuring the flow rate change over time. In the simulation, the reverse flow occurred up to 200 s, but the flow rate change trend after that was accurately predicted. The maximum flow rate was about 3.5 kg/s in the experiment and 3.75 kg/s in the simulation, showing an error of about 7%.

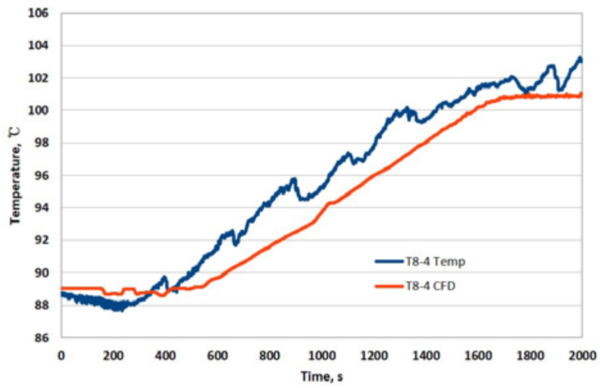

Fig. 5. Temperature comparison result of experiment and CFD

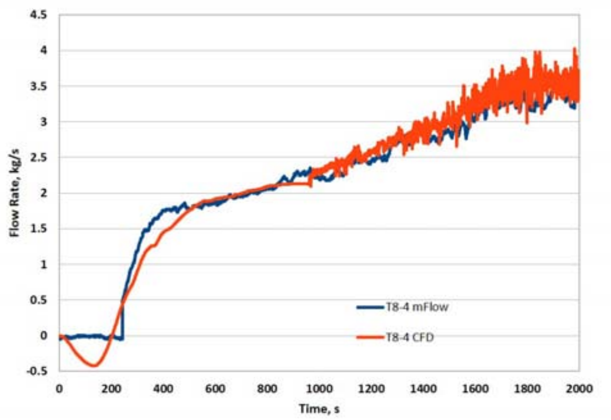

Fig. 6. Mass flow rate comparison result of experiment and CFD

3. Conclusions

A code was developed to simulate the flow circulation inside the core catcher installed in the European APR 1400.

As a result of benchmarking and simulating the CE-PECS T8-4 experiment, the flow rate and temperature rise trends were well predicted within a maximum error of 7%.

T8-4 was an experiment using only 75% of the possible power input for the heating block. In the future, we plan to benchmark experiments using 100% and 125% power.

We expect to find ways to improve the cooling performance of the core catcher by using this code and to predict possible variables..

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.
[4] R. J. Park, et al. Detailed evaluation of two phase natural circulation flow in the cooling channel of the ex-vessel core catcher for EU-APR1400. Nuclear Engineering and Design, 2016.