Numerical Simulation on Natural Convection Cooling of a FM Target

Jong-Pil Park [a*], Suki Park [a]
[a]*Research reactor safety analysis group, Korea Atomic Energy Research Institute, Daejeon, Korea*
[*]*Corresponding author:pjp3381@kaeri.re.kr*

1. Introduction

The KIJANG research reactor (KJRR) is an open-pool type reactor and has 6 irradiation holes in the reactor core in order to produce fission-molly (FM). The irradiated FM target is unloaded from the irradiation hole during normal operation, and then cooled down in the reactor pool for a certain period of time. Therefore, it is necessary to identify the minimum decay time needed to cool down FM target sufficiently by natural convection. In the present work, numerical simulations are performed to predict cooling capability of a FM target cooled by natural convection using commercial computational fluid dynamics (CFD) code, CFX.

2. Methods

Two types of simulation including steady state (S) and transient simulation (T) are performed. The steady state simulation is carried out to assure a tendency of the natural convection such as buoyancy driven flow field of reactor pool water flowing through the FM assembly and to provide a certain condition for transient simulation.

2.1 Numerical Model

The FM target is composed of 8 FM target plates and upper and lower comb. The FM target is loaded or unloaded into/from the reactor core as a FM assembly, which combines element of FM target and FM holder, during normal operation. To evaluate cooling capability of a FM target, the half model of the FM assembly is used in the present work and the combs are not considered as shown in Fig. 1. The 9 million computational meshes are generated in the fluid and solid domains for 3-dimensional conjugate heat transfer analysis. The y+ values are maintained at a level between 0.005 and 1 for all simulation cases.

2.2 Numerical Method

The flow field of reactor pool water flowing through a FM assembly is solved using Reynolds averaged Navier-Stokes (RANS) equation for incompressible flow. The k-w SST model is selected to solve the turbulent flow and automatic wall function [1] is used to treat the flow within the turbulent boundary layer in the present simulation. The set of governing equation is solved using finite volume method (FVM). To simulate buoyancy driven flow, the density difference model with IAPWS water property table [1] is used. The initial conditions for the present simulation are summarized in Table 1. At first, the small decay power is selected as an input to avoid nucleate boiling in the present work (S1). Then, the decay power is gradually increased until the maximum fuel surface temperature reaches saturation temperature (S2~S5). The normalized decay power of a FM target and the selected decay power for steady state and transient simulation are shown in Fig. 2.

Table 1: Initial conditions

	Pool temp. [°C]	Pool press. [kPa]	Decay power [% full power]
S1	36.0	180	4.1
S2	36.0	180	4.6
S3	36.0	180	5.1
S4	36.0	180	5.6
S5	36.0	180	6.1
T1	36.0	180	5.1
T2	36.0	180	5.6

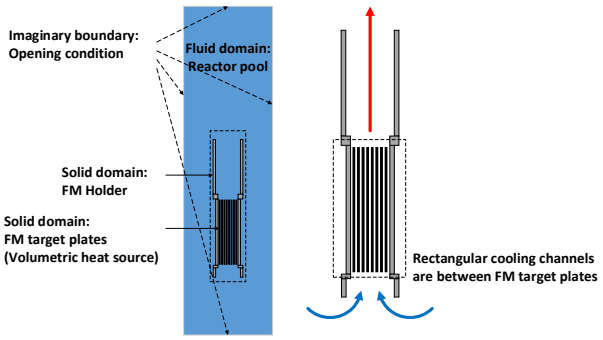


Fig. 1. Schematic for computational domain.

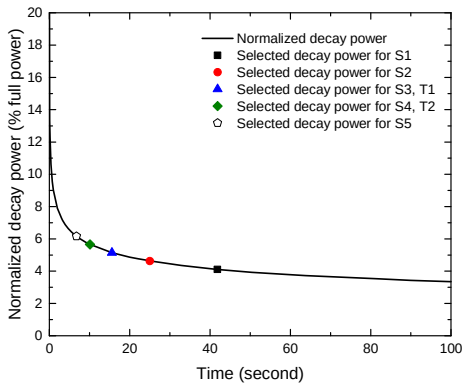


Fig. 2. Normalized decay power of a FM assembly.

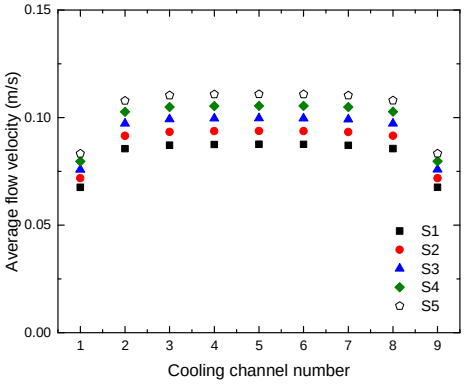
3. Results

3.1 Steady State Simulation

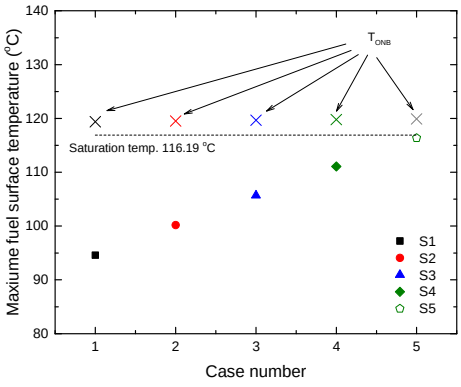
The buoyancy driven flow velocity in the cooling channels of the FM assembly and the maximum fuel surface temperature are shown in Fig. 3. Here, the cooling channel means the rectangular flow path between FM target plates within the FM assembly as shown in Fig 1. The buoyancy driven flow velocity and maximum fuel surface temperature increases with increasing the decay power. The analysis results show that nucleate boiling does not occur in the present simulation because the maximum fuel surface temperature is lower than the saturation temperature as well as the temperature of onset of nucleate boiling, which is calculated by Bergles and Rohsenow' ONB correlation as follows [2]:

$$\left(T_w - T_{sat}\right)_{ONB} = 0.556\left[\frac{q''_w}{1082P^{1.156}}\right]^{0.463P^{0.0234}} \quad . \quad (1)$$

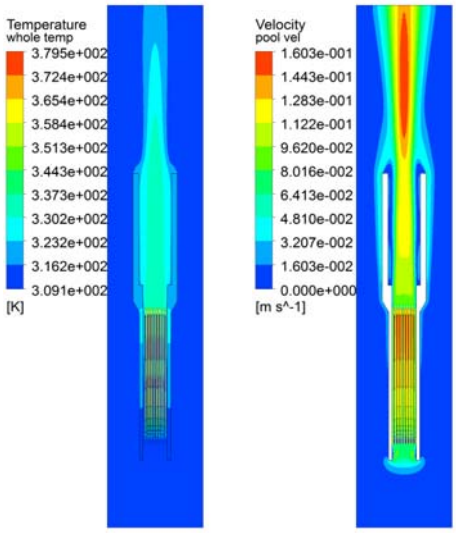
Here, P is in bars, T is in Kelvins, and q''_w is in W/m^2. The overall temperature and velocity distribution of the reactor pool including the FM assembly is shown in Fig. 4.



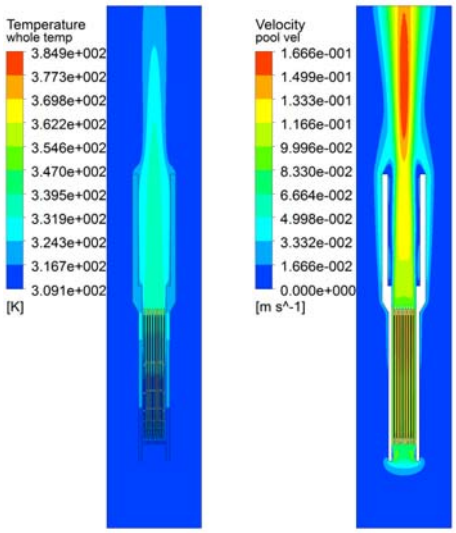
(a) Flow velocity in the cooling channel



(b) Maximum fuel surface temperature
Fig. 3. Steady state simulation result.



(a) S3 (Decay time 15.60 seconds, 5.1% full power)



(a) S4 (Decay time 10.10 seconds, 5.6% full power)
Fig.4. Temperature and velocity distribution for steady state.

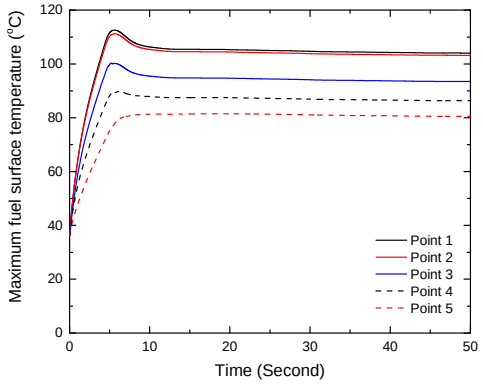
3.2 Transient Simulation

Based on the steady state calculations, the transient simulations corresponding to S3 and S4 were calculated to consider peak fuel surface temperature during the transient cooling in evaluating cooling capability of the FM target. The total simulation time and time step are 50 seconds and 0.01 seconds, respectively. Fig 5 shows the fuel surface temperature variations during the simulation time. Here, the points mean the specific locations on the FM plates, which are expected to reach peak temperature during the transient cooling. In the present work, 120 points are selected based on steady state simulation. The point 1 shown in Fig. 5 is that the local point on FM plates has maximum peak fuel surface temperature among them. The fuel surface temperatures sharply increase at the start of simulation, and then reach peak temperature at about 5.5 seconds after starting simulation for all points. After the peak point, fuel surface temperatures decrease slowly. The peak fuel surface temperatures for T1 and T2 are about

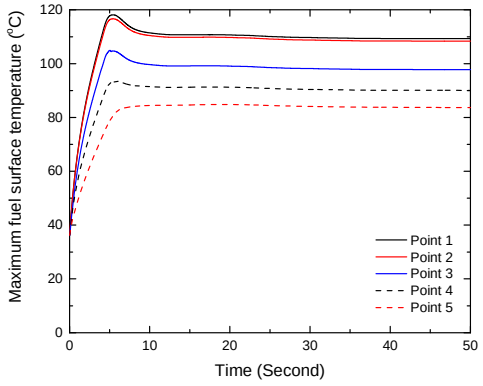
7°C higher than the maximum fuel surface temperatures corresponding to the steady state simulation (S3, S4). The transient simulation results are summarized in Table 2. For transient simulation, the maximum peak fuel surface temperature is also lower than the T_{ONB}, but ONB margin (T_{ONB}-T_w) is lower than 3°C for T2, which is the design limit for ONB margin of KJRR. Fig 6 shows the overall temperature and velocity distribution at peak time.

4. Conclusions

The present study is carried out using CFD code to investigate cooling capability of a FM target cooled by natural convection. The steady state simulation as well as transient simulation is performed in the present work. Based on the transient simulation (T1), the minimum decay time that the maximum fuel temperature does not reach the design limit temperature (T_{ONB}-3°C) is around 15.60 seconds. In other words, the maximum fuel surface temperature does not reach the design limit temperature if the FM assembly begins to be cooled down by natural convention in the reactor pool about 15.60 seconds after the FM assembly is withdrawn from the reactor core.



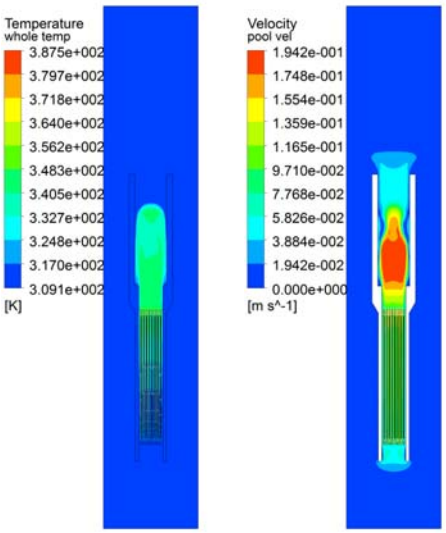
(a) T1: 5.1% full power



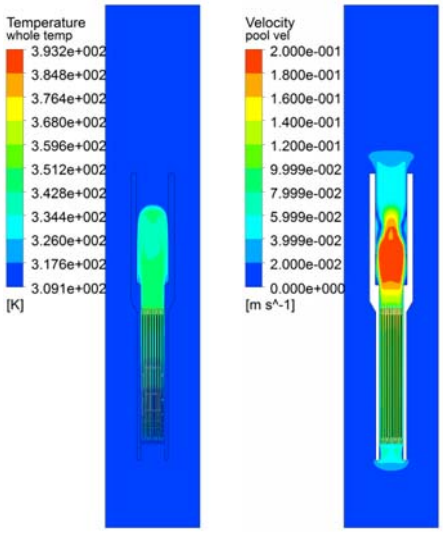
(b) T2: 5.6% full power
Fig. 5. Fuel surface temperature variations.

Table 2: Transient simulation results

Case	Peak temp. [°C]	T_{sat} [°C]	T_{ONB} [2] [°C]
T1	112.57	116.91	119.66
T2	118.15	116.91	119.79



(a) T1 (15.60+5.65 seconds after unloading FM assembly)



(b) T2 (10.10+5.45 seconds after unloading FM assembly)
Fig. 6. Temperature and flow field at peak time.

Acknowledgement

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP: Ministry of Science, ICT and Future Planning) (NRF-2012M2C1A1026916).

REFERENCES

[1] ANSYS CFX-Solver modeling guide, ANSYS Inc., 2001.
[2] A. E. Bergles, W. M. Rehsenow, The determination of forced surface boiling heat transfer, Int. J. Heat Mass Transfer, 86, p. 365, 1964.