Numerical Study for Complete Flow Blockages of Plate-type Fuel Assembly

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

1. Introduction

A plate-type fuel assembly is widely used in the research reactors in order to enhance power density. This type fuel assembly consists of a number of fuel plates and supporting plates. A number of cooling channels of fuel assembly are between the fuel plates. Due to cooling channels consisting of parallel fuel plates and side plates, the cooling channels are isolated from each other so that the cross flow between the channels are completely restricted. When complete blockage at the inlet of the cooling channel occurs, therefore, the coolant flow through the blocked cooling channel will be completely interrupted. Accordingly, the blocked cooling channel loses its own cooling capability. This event may cause initiation of nucleate boiling and flow instability (FI) in the first unblocked cooling channel. If flow instability occurs in the first unblocked cooling channel, the instability can propagate to another cooling channel (the second unblocked cooling channel) and lead to that even the fuel plate facing the unblocked cooling channel is also damaged during the accident. This safety issue is one of the great concerns in safety analysis of research reactors.

The preliminary study has been performed to assure possibility of a CFD application for this safety issue [1]. Based on the this study, in the present work, numerical simulation for complete flow blockage of a plate-type fuel assembly was performed to evaluate thermal-hydraulic phenomena during flow blockage accident and to look for a certain condition under which FI in the first unblocked cooling channel occurs due to the flow blockage.

2. Methods

Two types of simulation were performed using the commercial CFD code, CFX 16.1. The steady state simulation without flow blockage was carried out to provide the initial condition for the transient simulation. The transient simulations were performed to evaluate thermal-hydraulic phenomena in the unblocked cooling channels of the plate-type fuel assembly after complete flow blockages occur.

2.1 Numerical model

The selected plate-type fuel assembly is composed of 21 fuel plates and 2 supporting plates. The quarter model of the plate-type fuel assembly was used in the present work as shown in Fig. 1. The 2 million computational meshes are generated in the fluid and solid domains for 3-dimensional conjugate heat transfer analysis.

2.2 Numerical Method

Two-fluid model based on Eulerian multiphase flow was used with conventional wall boiling scheme [2], because the boiling in unblocked channels is expected due to the enhanced heat by the fuel plate of blocked channels. In addition to this, Kocamustafaogullari's bubble departure diameter model [3], Hibiki and Ishii's active nucleate site density model [4], and Kocamustafaogullari and Ishii's bubble departure frequency model [5] were implemented in the CFX code to calculate heat partitioning on the wall at low pressure condition using user defined function (CEL function). The mean bubble diameter model suggested by Hibiki et al. [6], which was develop to predict bubble size under low pressure boiling flow, was also implemented in the code.

The blockage was modeled as thin porous block of 1 mm thick, which were located at upstream of channel inlet. For steady state simulation, resistance coefficients of these porous blocks were applied as a unity. While, for transient simulation, the resistance coefficients were applied as very high values in order to model blockage at the inlet of channels due to the foreign object.

2.3 Initial and boundary conditions

The initial and boundary conditions for the present simulation are summarized in Table 1. In case of the steady state simulation, the inlet boundary condition is set constant mass flow rate at the entrance of fuel assembly and the outlet boundary condition is modeled as a relative pressure of 0 Pa at the end of downstream region. In case of transient simulations, while, the inlet boundary condition with time dependent mass flow rate is specified at the entrance of fuel assembly to simulate inlet flow reduction of computational domain due to the flow blockages. The entrance region and downstream region are sufficiently long to provide a fully developed flow at the inlet and outlet in order to obtain an appropriate converged solution. The symmetry boundary condition is applied on two vertical side planes of computational domain as shown in Fig. 1. The volumetric uniform heat source of the fuel meat is taken into consideration as shown in Table 1.

Table 1: Initial conditions

Initial temperature [°C]	36
Initial pressure [kPa]	202.63
Inlet mass flow rate [kg/s] : Steady state	4.09
Volumetric heat source [W/m^3]	6.415e9

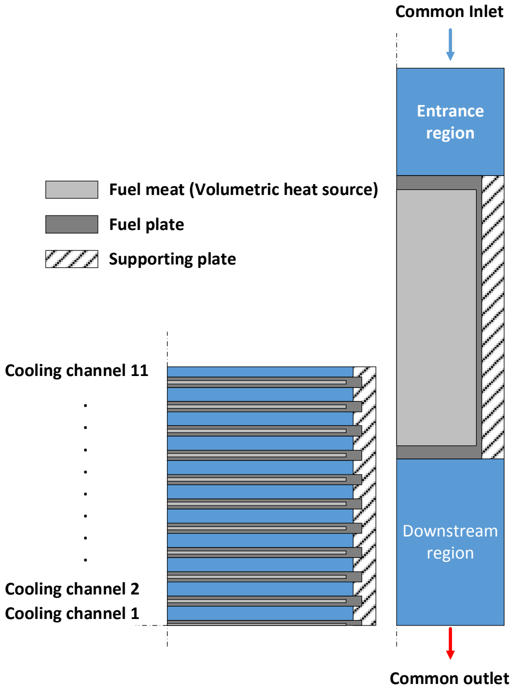

Fig. 1. Schematic for computational domain

3. Results

4.1 Steady State Simulation

The result for steady state shows that the nucleate boiling does not take place though wall boiling model is applied on steady state simulation. It is because the water temperature of the computational meshes near the wall does not exceed boiling activated temperature.

4.2 Transient Simulation

In order to simulate thermal-hydraulic phenomena in the unblocked cooling channels, the transient simulation was performed based on the steady state calculations. The total simulation time and time step are 10 seconds and 0.005 seconds, respectively.

After blockage occur, the fuel plate facing blocked cooling channel is cooled by one side. The temperature of fuel plate between blocked cooling channel increases sharply and reaches melting temperature (570 °C) as shown in Fig. 2. The heat produced from this plate transfer through side plates to the first unblocked cooling channel.

Fig 3 shows the mass flow rate of coolant at the outlet of each cooling channel for the complete blockage of two cooling channels (Cooling channel 1). After blockage occurs, the coolant flow begins to be redistributed in unblocked cooling channels during the short time. After then the same flow rate as steady state flow rate is maintained. Fig 4 shows the temperature variations at the outlet of each cooling channel. The temperature of the first unblocked cooling channel decreases due to the increase of the mass flow rate at first. After then the temperature gradually increases from 73.35 °C (steady state) to 94.60 °C since total heat produced by the fuel plate facing blocked cooling channel is transferred to the first unblocked cooling channel (Cooling channel 2). For the two cooling channel blockage, nucleate boiling in the first unblocked cooling channel takes place owing to the heat produced from the fuel plate facing blocked cooling channel as well as the heat transferred through the supporting plates from the fuel plate between the blocked cooling channels. Hence, void fraction at outlet of the first unblocked cooling channels increases continuously as shown in Fig. 5. Fig. 6 shows the coolant temperature and void fraction at the outlet of each cooling channel after 10 seconds of clogging.

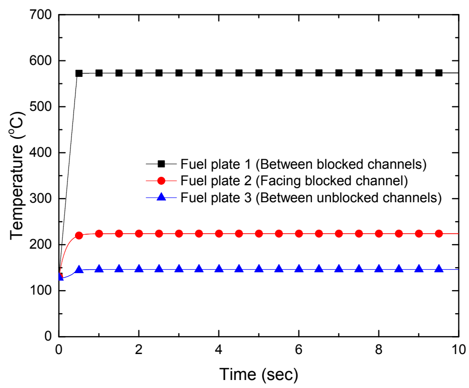

Fig. 2. Temperature variation of the fuel plate

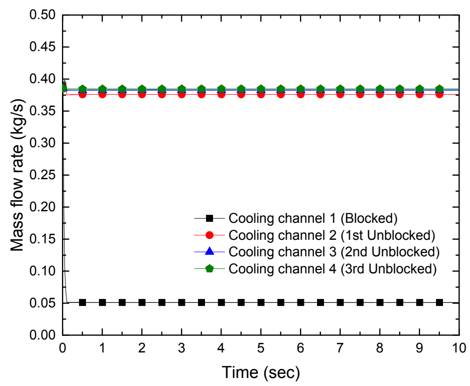

Fig. 3. Variation of coolant mass flow rate at outlet of cooling channels

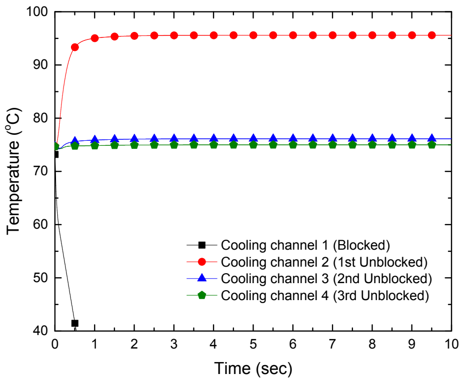

Fig. 4. Variation of coolant temperature at outlet of cooling channels

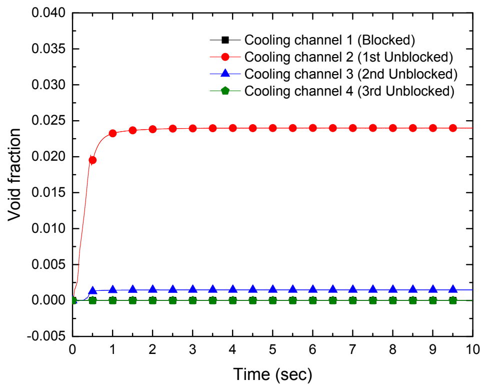

Fig. 5. Variation of void fraction at outlet of cooling channels

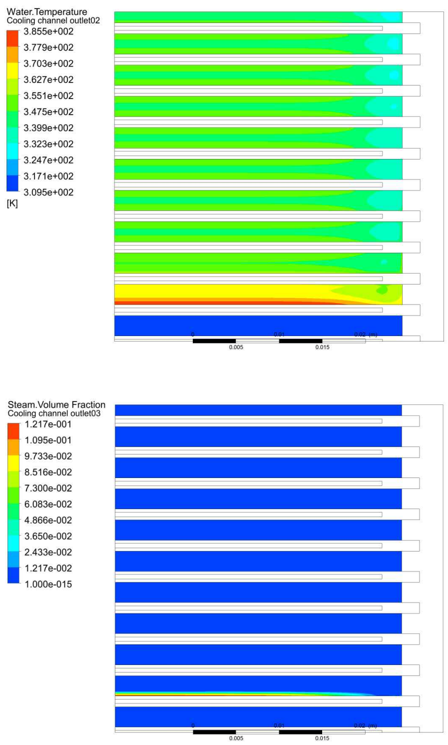

Fig. 6. Coolant temperature (upper) and void fraction (lower) at outlet of each cooling channel (Transient, Time=10 sec).

4. Conclusions

The present study reports a 3-dimensional CFD simulation using two-phase model for complete flow blockage of two cooling channels of a plate-type fuel assembly. The results of present study show that it is possible to occur nucleate boiling in the unblocked channel of a plated-type fuel assembly due to the added heat caused by flow blockage. Furthermore, it is expected that stronger boiling in the first unblocked cooling channel will be occurred and it may lead to FI if the number of blocked cooling channels increase based on the present result. Therefore, the simulations for a number of cooling channels blockage (4~12) are working at the moment.

Acknowledgement

This work was supported by the Korea government (MSIT: Ministry of Science and ICT).

REFERENCES

[1] J. P. Park, S. K. Park, Preliminary analysis for flow blockage of plate fuel using a commercial CFD code, Transactions of the KNS autumn meeting, Gyeongju, Korea, October 27-28, 2016.

[2] N. Kurul, M. Z. Podowski, On the modeling of multidimensional effects in boiling channels, ANS Proc. 27th National Heat Transfer Conference, Minneapolis, MN, July 28-31, 1991.

[3] G. Kocamustafaogullari, Pressure dependence of bubble departure diameter of water, Int. Comm. Heat Mass Transfer, Vol. 10, pp. 501-509, 1983.

[4] T. Hibiki, M. Ishii, Active nucleate site density in boiling system, International Journal of Heat and Mass Transfer, Vol. 46, pp. 2587-2601, 2003.

[5] G. Kocamustafaogullari, M. Ishii, Foundation of the interfacial area transport equation and its closure relations, International Journal of Heat and Mass Transfer, Vol. 38, pp. 481-493, 1995.

[6] T. Hibiki et al. Interfacial area concentration in boiling bubbly flow systems, Chemical Engineering Science, 61, pp. 7979-7990, 2006.