Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 27-28, 2016

Preliminary Analysis for Flow Blockage of Plate Fuel using a Commercial CFD Code

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

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, supporting plates, and narrow rectangular cooling channels between the fuel plates. Due to the narrow rectangular channels consisting of parallel plate fuels and side plates, the cooling channels are isolated from each other so that the cross flow between the channels are completely restricted. When blockage at the inlet of the channel occurs, therefore, the coolant flow through the blocked channel will be greatly reduced or completely interrupted. Accordingly, the blocked channel loses its own cooling capability. This event may cause initiation of nucleate boiling and two-phase flow instability (FI) in the blocked channel. If two-phase flow instability occurs in the blocked channel, the instability can propagate to adjacent channels and lead to that even the fuel plate facing the unblocked channel is also damaged during the accident. This safety issue is one of the great concerns in safety analysis of research reactors.

In recent years, CFD analyses of flow blockage of a research reactor have been performed by many researchers [1-3]. However, most of these researches have been performed at a limited range of conditions that coolant flow has been maintained single-phase forced convection even for blockage. In order to utilize CFD codes in flow blockage analysis under more extreme condition, it is necessary to assess the capability of CFD code and to develop appropriate CFD methodology for the two-phase boiling flow and flow instability caused by flow blockage. Therefore, in the present work, preliminary analysis for flow blockage of a plate-type fuel assembly was performed under certain conditions in order to assure possibility of a CFD application for damage propagation due to two-phase flow instability.

2. Methods

Two types of simulation including steady state and transient simulation were performed using the commercial CFD code, CFX 16.1. The steady state simulation was carried out to provide the initial condition for the transient simulation. This condition is that flow blockage does not occur (base case). The transient simulation was performed to evaluate boiling phenomena in the unblocked channel adjacent to the blocked channel. The boiling is expected due to the enhanced heat by the fuel plate of blocked channel.

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 [4]. In addition to this, Kocamustafaogullari’s bubble departure diameter model [5], Hibiki and Ishii’s active nucleate site density model [6], and Cole’s bubble departure frequency model [7] were implemented in the CFX code to calculate heat partitioning on the wall at low pressure condition using user defined function (CEL function). 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.

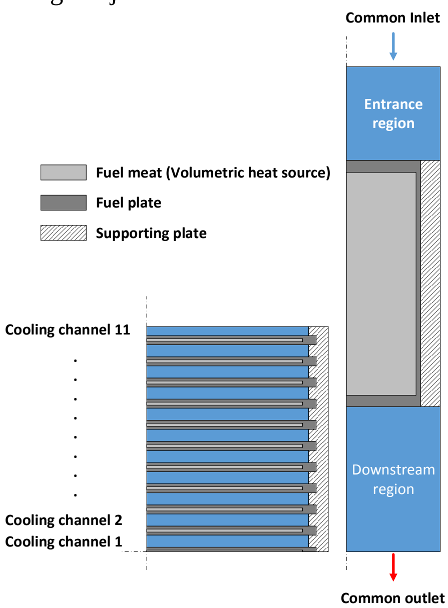


Fig. 1. Schematic for computational domain.

2.3 Initial and boundary conditions

The initial and boundary conditions for the present simulation are summarized in Table 1. 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. 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]	196.91
Inlet mass flow rate [kg/s]	4.9
Volumetric heat source [W/m^3]	6.415e9

4. Results

4.1 Steady State Simulation

The coolant velocity and temperature at the outlet of each cooling channel are shown in Fig. 2. 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 ($T_{act}=T_{sat}+0.05$ K) in this condition.

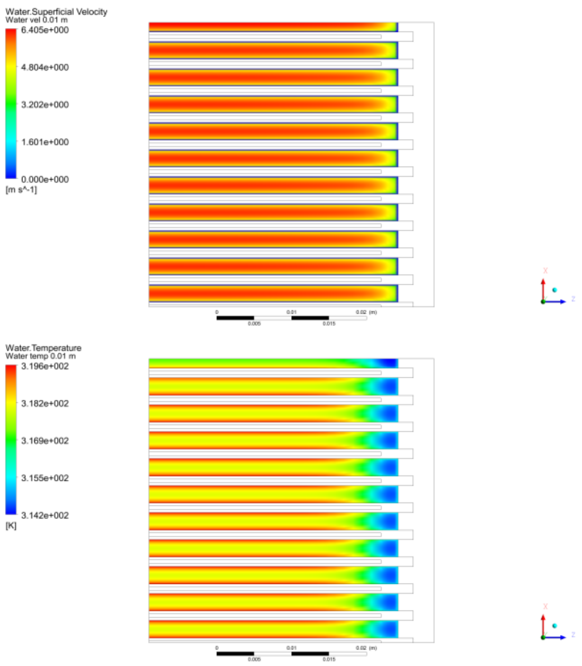


Fig.2. Coolant velocity (upper) and temperature (lower) at outlet of each cooling channel (Steady state)

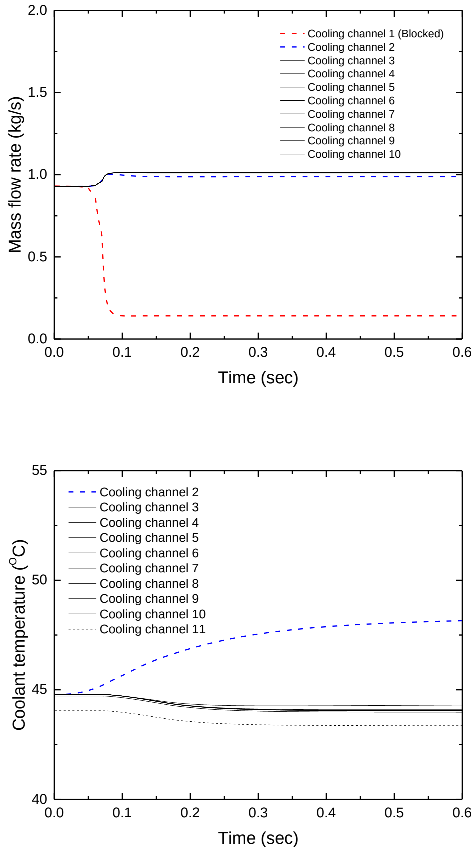


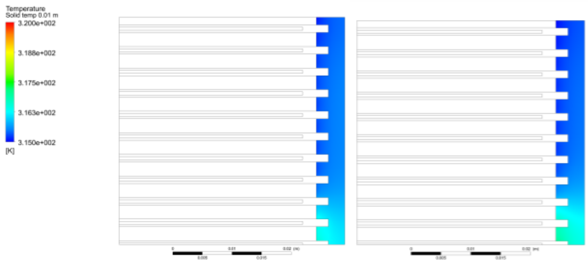
Fig.3. Variation of coolant mass flow rate (upper) and temperature (lower) at outlet of cooling channel

4.2 Transient Simulation

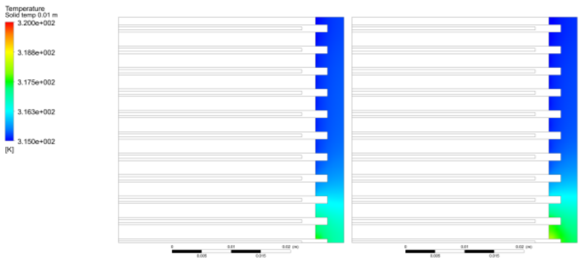
In the present work, a complete blockage of two channels was considered. In order to simulate boiling phenomena in the unblocked channels, the transient simulation was performed based on the steady state calculations. The total simulation time and time step are 5 seconds and 0.002 seconds, respectively.

The results in this paper were written based on the simulation in progress since the simulations have not been finished yet. The simulation for two channels blockage is working at the moment.

Fig 3 shows the mass flow rate of coolant and the temperature at the outlet of each channel for the complete blockage of two channels (Cooling channel 1). After blockage occurs, the coolant flow begins to be redistributed in unblocked channels (Cooling channel 2~11). This leads to new temperature distributions in the unblocked channels. The temperatures of the unblocked channels decrease with the increase in the mass flow rate except the first unblocked channel (Cooling channel 2). The temperature of the first unblocked channel decreases due to the increase of the mass flow rate at first. After then the temperature gradually increases since total heat produced by the fuel plate facing blocked channel is transferred to the first unblocked channel. The effect of added heat is predicted very small in this case as shown in Fig. 4. For the two channel blockage, even though all heat produced by the fuel plate facing blocked channel is cooled by one side, nucleate boiling in the adjacent unblocked channel does not take place against expectations. Fig 5 shows the local temperature of the fuel meat. The peak temperature of the fuel meat is also predicted below its melting temperature according to the results of simulation in progress.



(a) 0.3 seconds (b) 0.4 seconds



(c) 0.5 seconds (d) 0.6 seconds

Fig.4. Temperature of the supporting plate with time.

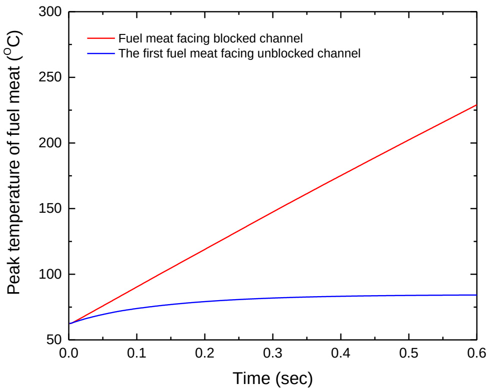


Fig.5. Temperature variations of the fuel meat.

5. Conclusions

The present study reports a 3-dimensional CFD simulation using two-phase model for flow blockage of two channels of a plate-type fuel assembly. The present result shows limited information since the simulation is working at the moment. However, the present study indicates that it is possible to predict a damage propagation of a plate-type fuel assembly due to the flow instability caused by flow blockage using a commercial CFD code.

Acknowledgement

This work was supported by the Korea government (MSIP: Ministry of Science, ICT and Future Planning).

REFERENCES

[1] Amgad Salama, CFD analysis of fast loss of flow accident in typical MTR reactor undergoing partial and full blockage: The average channel scenario, Progress in Nuclear Energy, Vol. 60, pp. 1-13, 2012.
[2] W. Fan et al., CFD study on inlet flow blockage accidents in rectangular fuel assembly, Nuclear Engineering and Design, Vol. 292, PP. 177-186, 2015.
[3] W. Fan et al., A new CFD modeling method for flow blockage accident investigations, Nuclear Engineering and Design, Vol. 303, pp. 31-41, 2016.
[4] 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.
[5] G. Kocamustafaogullari, Pressure dependence of bubble departure diameter of water, Int. Comm. Heat Mass Transfer, Vol. 10, pp. 501-509, 1983.
[6] T. Hibiki, M. Ishii, Active nucleate site density in boiling system, International Journal of Heat and Mass Transfer, Vol. 46, pp. 2587-2601, 2003.
[7] R. Cole, A photographic study of pool boiling in the CHF, AIChEJ, Vol. 6, pp. 533-542, 1960.