Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 26-27, 2017

Numerical Analysis on OFI in Narrow Rectangular Channels

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

1. Introduction

MTR type fuel assemblies (plate-type fuel assembly) have been widely used in the research reactors to enhance power density. This type fuel assembly generally consists of a number of narrow rectangular cooling channels and the cooling channels are completely isolated from each other. Hence, if complete flow blockage occurs in any one of the cooling channels or in multiple cooling channels, it can cause initiation of onset of flow instability (OFI) in an adjacent cooling channel and therefore the instability can propagate to the other cooling channels. This phenomenon is called flow instability propagation, one of severe concerns in safety analysis of research reactors. Therefore, it is necessary to precisely assess the OFI and its propagation caused by the flow blockage in order to improve the safety of a research reactor. In this study, 3-D computational fluid dynamics (CFD) analyses were performed to assess the applicability of CFD code to OFI in narrow rectangular channels prior to precise assessing the OFI and flow instability propagation in the plate-type fuel assembly during the flow blockage accident.

2. Methods and Results

2.1 OFI in a uniformly heated channel

Whittle and Forgan [1] conducted a series of experiments of two-phase pressure drop in the uniformly heated narrow rectangular channels near atmospheric pressure and obtained OFI data, which represent the minimum points of pressure drop curves (demand curves). Detailed information of the test section used in the experiments is summarized in Table 1.

Two-types of simulation including steady state and transient simulation were performed using a commercial CFD code, CFX 16.1, where two-fluid model based on Eulerian multiphase flow are used with a conventional wall boiling scheme. In addition, various local boiling models, such as Kocamustafaogullari's bubble departure model [2], Hibiki and Ishii's nucleation site density model [3], were implemented in the code to calculate heat partitioning on the heated wall at low pressure condition using user defined function (CFX Expression Language, CEL function). The mean bubble diameter model suggested by Hibiki et al. [4], which was develop to predict bubble size under low pressure boiling flow, was also implemented in the code. The rectangular-shaped computational domain consists of a heated region (both side heating) and two unheated regions located in upstream and downstream of the computational domain as shown in Fig. 1. The inlet boundary condition is set mass flow rate at the entrance of the computational domain and the outlet boundary condition is modeled as a relative pressure of 0 Pa at the end of computational domain.

The calculated two-phase pressure drops are compared with the experimental data in Fig. 2. It is found that the CFD code with the models implemented in this study can predict the trend that the two-phase pressure drop increases after the OFI although the mass flow rate decreases. For pre-OFI regions of all the simulation cases, the calculated pressure drops show a good agreement with the experimental data with error of ±13%. For post-OFI regions, however, the maximum difference between the calculation and the measurement is approximately -26%. The predicted-to-measured ratios (P/M) of the mass flow rates at OFI are shown in Fig. 3. The CFD code slightly under-predicts the measured mass flow rate by around -13% except for low heat flux condition (TS No.3, 0.66 MW/m^2≈ -17%). These results show that it is possible to predict OFI in a narrow rectangular channel under low pressure using the CFD code with the models implemented in this study.

Table 1: Details of test section [1]

TS No.	Gap (mm)	Width (mm)	Heated length (mm)
1	3.23	25.4	609.6
2	2.44	25.4	406.4
3	2.03	25.4	406.4

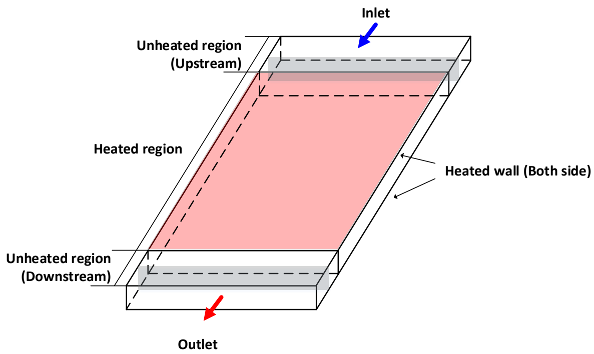

Fig. 1. Schematic for computational domain

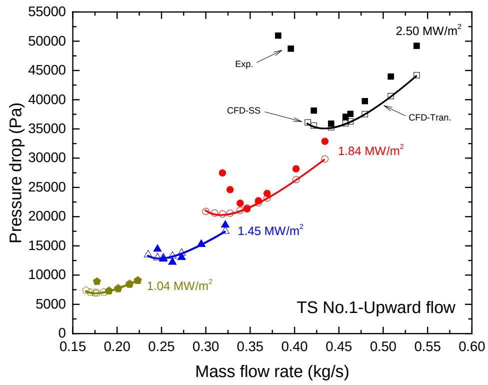

Fig. 2. Comparison of the calculations with the experiments

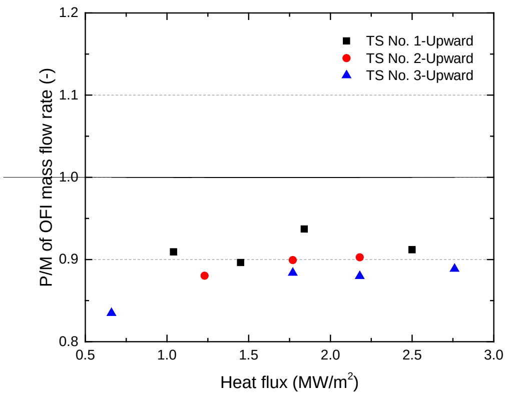

Fig. 3. Prediction of OFI mass flow rate

2.2 OFI in a non-uniformly heated channel

It is also necessary to assess the applicability of CFD code to OFI in a non-uniformly heated narrow rectangular channel because the heat generation of fuel plates adjacent to blocked channel is not uniform during the flow blockage accident. Based on the CFD analyses for OFI experiments conducted by Whittle and Forgan, additional CFD analyses were performed for the prediction of OFI in a non-uniformly heated narrow rectangular channel, TS No. 2. This test section has a similar gap size to the cooling channel of the plate-type fuel assembly of KIJANG research reactor (2.35 mm). In case of CFD analysis under non-uniform heating condition, computational model (geometry and mesh), numerical methods and initial/boundary conditions are the same with the CFD analysis for OFI experiment conducted by Whittle and Forgan except for axial power distribution. In this study, two power distributions with both sides heated as shown in Fig. 4 were assumed to assess the effects of axial power distribution on OFI.

Fig. 5 shows the differential pressures with mass flow rates in the various axial power distributions. The calculated pressure drop for pre-OFI region and the mass flow rates at OFI are nearly identical regardless of the axial power distributions. From the CFD analysis, it is indicated that the axial power distributions have no effects on the OFI in the narrow rectangular channel.

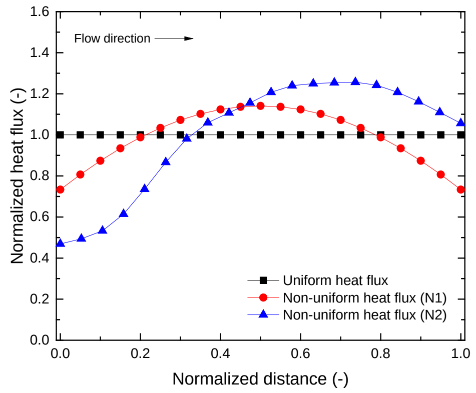

Fig. 4. Non-dimensional axial power distributions used in the CFD analysis

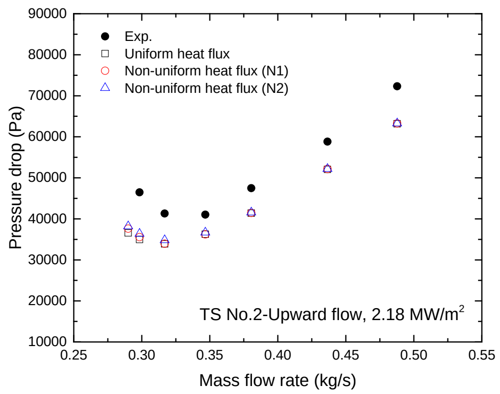

Fig. 5. Pressure drop curves with mass flow rates in various axial power distributions

3. Conclusions

From the comparison with the experimental data of Whittle and Forgan, it is found that the CFD code with the models implemented in this study can predict two-phase pressure drop characteristics and OFI in narrow rectangular channels reasonably well. In addition, it is indicated that the effects of axial power distribution on OFI are negligible in the present investigation, even though the results of CFD calculation are not validated from experimental data. The CFD technology for an assessment of the flow instability propagation due to a flow blockage accident in a plate-type fuel assembly will be developed based on the present CFD analysis results.

Acknowledgement

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

REFERENCES

[1] R. H. Whittle, R. Forgan, A correlation for the minima in the pressure drop versus flow-rate curve for sub-cooled water flowing in narrow heated channels, Nuclear Engineering and Design, Vol. 6, pp. 89-99, 1967.
[2] G. Kocamustafaogullari, Pressure dependence of bubble departure diameter of water, Int. Comm. Heat Mass Transfer, Vol. 10, pp. 501-509, 1983.
[3] T. Hibiki, M. Ishii, Active nucleate site density in boiling system, International Journal of Heat and Mass Transfer, Vol. 46, pp. 2587-2601, 2003.
[4] T. Hibiki et al. Interfacial area concentration in boiling bubbly flow systems, Chemical Engineering Science, 61, pp. 7979-7990, 2006.