Applicability of Two-fluid Model in Predicting Onset of Flow Instability in a Narrow Rectangular Cooling Channel under Low Pressure

Jong-Pil Park [a*], Hyeonil Kim [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

Plate-type fuel assemblies have been widely used in the research reactors to enhance power density. This type fuel assembly consists of a number of fuel plates, supporting plates, cooling channels between the fuel plates, common inlet and outlet plenums. The cooling channels are completely isolated from each other so that the cross flow between the channels does not occur. Hence, if onset of flow instability (OFI) takes place in any one of cooling channels during the flow blockage accident in a plate-type fuel assembly, it can develop CHF and cause OFI in an adjacent cooling channel. This phenomenon is called flow instability propagation, one of the greatest concerns in safety analysis of research reactors. Therefore, it is necessary to precisely assess the two-phase flow instability in a narrow rectangular cooling channel and its propagation caused by the flow blockage in order to improve the safety of a research reactor. In the present work, CFD analysis was performed to assess the capability of the CFD code in predicting OFI in a narrow rectangular cooling channel under low pressure.

2. Methods

2.1 Experiments

Whittle and Forgan [1] conducted a series of experiments of two-phase pressure drop in the narrow rectangular channels as well as a round tube near atmospheric pressure and obtained OFI data, which represents the minimum point of the channel demand curve (pressure drop curve) as shown in Fig. 1. Detailed information of five different types of test section used in experiments is summarized in Table 1. In the present work, two different set of the experiments with rectangular test section, No.1 and No.3, were selected for the assessment of the applicability of the CFD code in predicting OFI.

2.2 Numerical Method

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 [2]. In addition, various local boiling models, such as Kocamustafaogullari's bubble departure model [3], Hibiki and Ishii's nucleation site density model [4], 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 rectangular-shaped computational domain consists of heated region (both side heating) and two unheated regions located in upstream and downstream of the computational domain as shown in Fig 2. 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 experimental conditions for the present simulation are summarized in Table 2.

Table 1: Details of test section [1]

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
4	1.40	2.54	533.4
5	Round tube dia. = 6.45 (mm)		609.6

Table 2: Experimental conditions

Case	Test section	Inlet temp. (℃)	Outlet press. (kPa)	Heat flux (kW/m2)
C1	No. 1	55	117	1040
C2		45	117	820
C3		60	117	1100
C4	No.3	65	117	1100
C5		55	117	980
C6		55	117	660

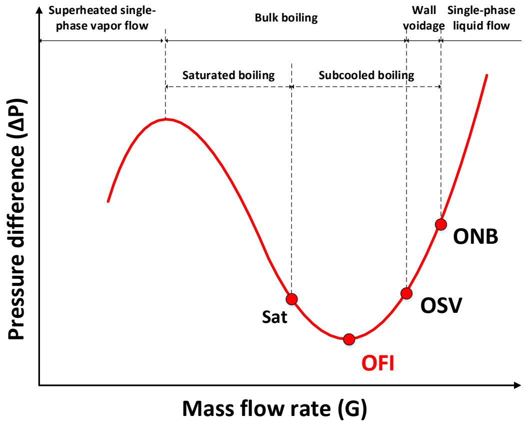

Fig. 1. Schematic of channel demand curve for an imposed heat flux condition

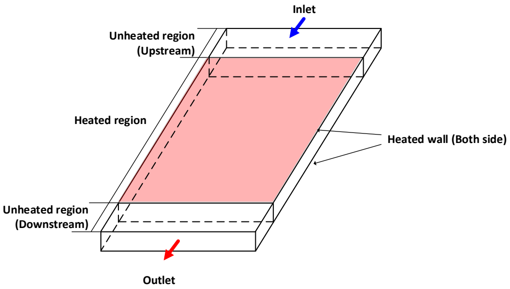

Fig. 2. Schematic for computational domain

3. Results

The calculated two-phase pressure drops are compared in Fig. 3 through Fig. 6 with the experimental data. It is found that the CFD code with the models implemented in this study can predict the tendency of two-phase pressure drop that the pressure drop increases after the initiation of OFI although the mass flow rate decreases. For pre-OFI region of all the simulation cases, the CFD result shows a good agreement with the experimental data (P/M≈ 0.9). In case of C1 used in test section No. 1 with relatively large gap size, however, CFD code gives worst performance with a maximum 20% difference for post-OFI region (Fig. 3). The calculated mass flow at OFI point is presented in Table 3. The CFD code slightly under-predicts the measured mass flow rate by about 10% except for C6 case (C6 case≈ 14%). These results show that it is possible to predict OFI under low pressure in a narrow rectangular channel using a commercial CFD code with the models implemented in this study.

Table 3: Comparison of the mass flow at OFI point

Case	Calculated mass flow rate (kg/s)	Measured mass flow rate (kg/s)	P/M
C1	0.182	0.194	0.94
C2	0.143	0.154	0.93
C3	0.215	0.231	0.93
C4	0.159	0.172	0.92
C5	0.111	0.123	0.90
C6	0.073	0.085	0.86

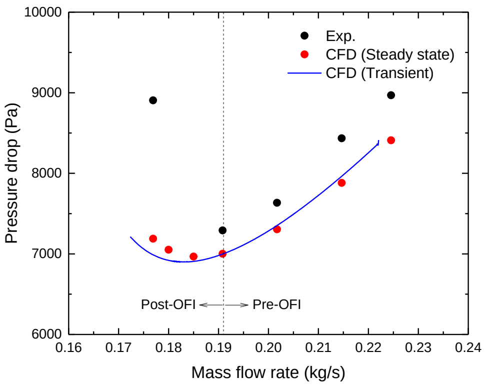

Fig. 3. Comparison of the experiments with CFD result (C1)

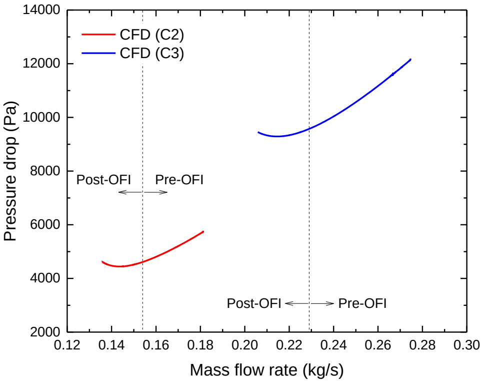

Fig. 4. Comparison of the experiments with CFD result (C2, C3)

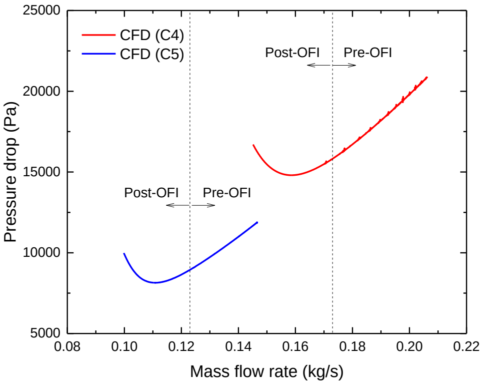

Fig. 5. Comparison of the experiments with CFD result (C4, C5)

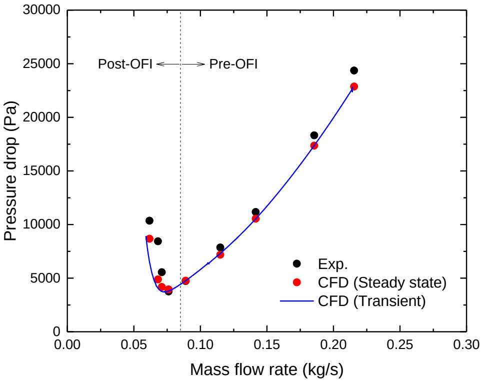

Fig. 6. Comparison of the experiments with CFD result (C6)

Acknowledgement

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

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] 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.