Numerical Study of Thermal-Hydraulics in CANDU-typed Moderator

Jae Ryong Lee [a*], Han Young Yoon [a], Hyoung Tae Kim [a], Jae Jun Jeong [b]
[a]*Korea Atomic Energy Research Institute,1045 Daeduk-daero, Daejeon, Korea, 305-353*
[b]*Pusan National University, 30, Jangjeon-dong, Keumjeoung-gu, Busan, Korea, 609-735*
[*]*Corresponding author: jrlee@kaeri.re.kr*

1. Introduction

The moderator plays an important role to sustain a robustness of the fuel channel during a hypothetical loss of coolant accident (LOCA) in the Canada Deuterium Uranium (CANDU) reactor vessel. Since the pressure tube stain to contact its surrounding Calandria tube (PT/CT contact) could lead a CT dryout, it is important to estimate a local subcooling of moderator inside the Calandria vessel. However, it is possible to measure the local temperature only in the inlet/outlet region. Thus, it is needed to estimate the local subcooling of moderator at arbitrary region inside the Calandria vessel for the accident such as LOCA.

In order to measure the local temperature quantitatively, numerous experimental and numerical researches have been investigated [1,2]. Over all the numerical studies, the subcooling in the moderator and local velocity and temperature fields have been the only purpose rather than the boiling and corresponding two-phase phenomena for the LOCA accident. In this study, the CUPID code [3] is used to validate the existing mixed and natural convection regime as well as two-phase phenomena feasibility.

2. Numerical Methodology

2.1 Governing equation

A component scale thermal hydraulic analysis code, CUPID (Component Unstructured Program for Interfacial Dynamics), is being developed for the analyses of components of a nuclear reactor, such as reactor vessel, steam generator, containment, etc. It adopts three-dimensional, transient, two-phase and three-field model, and includes various physical models and correlations of the interfacial mass, momentum and energy transfer for the closure relations of the two-fluid model.

To simulate the two-phase flow in the fuel assembly region, a porous media approach was adopted. Since it is nearly impossible to model the fluid and structural regions of the reactor core exactly, the porous media approach can be an alternative tool to simulate thermal hydraulics effectively. Moreover, it has been known to give a reasonable solution for a two-phase flow analysis in a complex geometry. Generally, the porosity and permeability are introduced to represent the porous media approach. Porosity is a measure of the void space in an arbitrary medium

In addition to defining the porosity and permeability, the pressure drop in the porous media should be modeled appropriately. Hadaller et al. [1] derived the empirical frictional pressure loss coefficient in tube bundle geometry which is applicable to the CANDU moderator. The CUPID implemented this correlation implicitly.

2.2 STERN Experiment

The CUPID code with empirical pressure drop model is validated against the STERN 2D experiments. Fig 1 shows the test section for the simulation. The heat-generating tube bundle is located in the core of the thin "slice-typed" circular region to represent the Calandria vessel. The tube bundle is consisted of 440 inconel heater with a size of 0.033mm diameter and 0.0715mm pitch among tubes. The coolant inlet nozzle is 6mm thin planar jet and located at 50mm away from the horizontally inner wall. Though the nozzle is designed to vary the width from 6mm to 18mm and its angle up to 15^o from the vertical direction toward the center, it is fixed as 6mm width at vertical direction in this study. The outlet nozzle is 15mm width at the bottom of the test section.

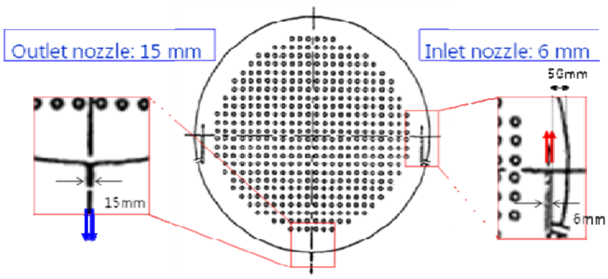


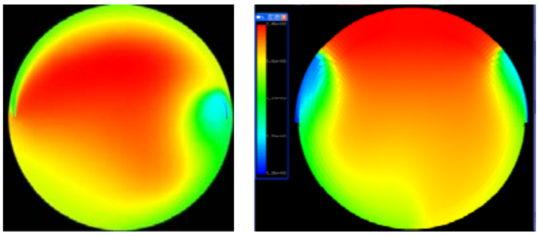
Fig 1 Schematics for validation

3. Result and Discussion

3.1 Single-phase behavior

For the validation, the nominal case [2] is selected, which represent the normal operation in real CANDU reactor. The total power into the heater is 100kW and the inlet mass flow rate is 2.4kg/s. Fig 2 shows the contour of the liquid temperature for both the nominal case and less coolant injection case. At the beginning of the simulation, there is no heat generation in order to stabilize the convection flow. At this stage, the flow pattern shows exact y-axis symmetry. As the heat starts being supplied into the porous zone, the buoyant force is generated. Consequently, at arbitrary state, the y-axis symmetry is collapsed by the instability between the advection from the impinging jet against the buoyant force from the heat generation. The stagnation point formed by the both impinging jet from the coolant nozzles is tilted at any direction, which can be determined by any small numerical perturbation due to the flow instability. The snapshot at Fig 2 (a) is when the flow goes to the final steady state, called as the mixed regime. The vertical profile of the liquid temperature at centerline (x=0) is plotted at Fig 3, in which the result of the CUPID calculation is good agreement with other numerical calculation as well as experimental measurement.

For the case of less coolant injection, the coolant from the nozzle does not overcome the buoyant force from the porous zone. The impinging jet is circulated near its discharged nozzle. Meanwhile, the heated fluid is retained in the upper region. Thus, the flow pattern is called as the natural convection regime as shown in the Fig 2(b).



(a) mixed regime (b) natural convection regiome

Fig 2 contours of the liquid temperature

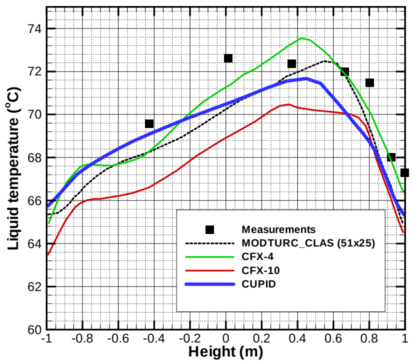
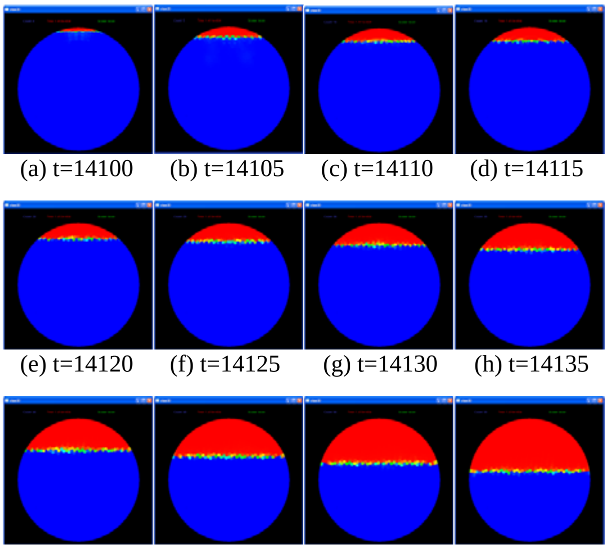


Fig 3 Profile of the liquid temperature (x=0) in the nominal case

3.2 Two-phase behavior

Since the CUPID has an advantage of being capable of two-phase calculation, the artificial transient LOCA scenario for less mass flow rate is qualitatively simulated. Fig 4 shows the snapshots of the void fraction for two-phase flow, in which the inlet velocity is 0.2 m/s with invariant thermal power of the core.



(a) t=14100 (b) t=14105 (c) t=14110 (d) t=14115

(e) t=14120 (f) t=14125 (g) t=14130 (h) t=14135

(i) t=14140 (j) t=14145 (k) t=14150 (l) t=14155

Fig 4 snapshots of the void fraction for two-phase flow; red:superheatd vapor region; blue: subcooled liquid region

4. Conclusions

The CUPID code was validated against nominal test condition of the STERN 2D experiment and the results showed good agreement with both the experiments and the previous researchers' results. And, the two-phase phenomenon for an artificial severe accident scenario was qualitatively well calculated.

For further study, a couple of activities should be investigated such as

- Heat transfer model in porous zone
- Effect of turbulent model
- Quantification of the two-phase flow simulation

Acknowledgement

This work was supported by the Nuclear Research & Development Program of the NRF (National Research Foundation) grant funded by the MEST (Ministry of Education, Science and Technology) of the Korean government.

REFERENCES

[1] G.I, Hadaller, et al., Frictional Pressure Drop for Staggered and In Line Tube Bank with Large Pitch to Diameter Ratio," Proceedings *of 17th CNS Conference*, Federiction, New Brunswick, Canada, June 9-12, 1996

[2] C. Yoon, et al., "Development and Validation of the 3-D Computational Fluid Dynamics Model for CANDU-6 Moderator Temperature Predictions", *Nuclear Technology*, vol.148, pp.259-267, 2004

[3] H.Y. Yoon, et al., "CUPID CODE MANUAL VOLUME I: Mathematical Models and Solution Methods", *KAERI/TR-4403/2011*, KAERI, 2011