Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 12-13, 2016

Subchannel Scale Thermal-Hydraulic Analysis of Rod Bundle Geometry under Single-phase Adiabatic Conditions Using CUPID

Seok-Jong Yoon [a], Goon-Cherl Park [a], Hyoung Kyu Cho [a*]
[a]*Department of Nuclear Engineering, Seoul National University 1 Gwanak-ro, Gwanak-gu, Seoul 151-744*
[*]Corresponding author: *chohk@snu.ac.kr*

1. Introduction

High-fidelity and multi-physics simulation with coupled T/H (Thermal-Hydraulics) code and neutronics code for a light water reactor core has become a subject of special interest in the nuclear reactor safety analysis. Considering the limitation in computational power, subchannel scale analysis would be desired for a practical simulation of the full core pin-by-pin analysis. Recently, in the CASL (Consortium for Advanced Simulation of Light water reactors) project, subchannel analysis code COBRA-TF has been used to simulate high-precision full core T/H analysis with coupled multiple codes.

In Korea, subchannel analysis code, MATRA has been developed by KAERI (Korea Atomic Energy Research Institute). MATRA has been used for reactor core T/H design and DNBR (Departure from Nucleate Boiling Ratio) calculation [1]. Also, the code has been successfully coupled with neutronics code and fuel analysis code. However, since major concern of the code is not the accident simulation, some features of the code are not optimized for the accident conditions, such as the homogeneous model for two-phase flow and spatial marching method for numerical scheme.

For this reason, in the present study, application of CUPID for the subchannel scale T/H analysis in rod bundle geometry was conducted. CUPID is a component scale T/H analysis code which adopts three-dimensional two-fluid three-field model developed by KAERI [2]. CUPID code has MPI-domain decomposition and it is expected to realize full core simulation. In this paper, key subchannel models were implemented into CUPID and the code was validated with four tests under single phase conditions.

2. Implementation of subchannel models to CUPID

2.1 Form loss, wall friction, grid spacer Model

Cross flow model is one of the important subchannel models to analyze single phase condition. Due to pressure difference at each subchannels, cross flow can occur. Subchannel models were added to CUPID with consideration of flow direction. For axial direction, wall friction and grid spacer model formulated with form loss were included. These models were added to axial momentum conservation equation as follows,

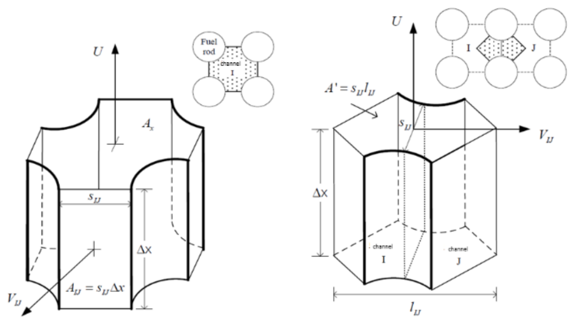
$$\Delta P = -\frac{1}{2}\left(\frac{f}{d_{hy}} + K\right)\left(\frac{G^2}{\rho}\right), \tag{1}$$

where, f and K are the wall friction factor and the form loss coefficient for a grid spacer, respectively.

To consider the change of fuel gap by the rod arrangement, form loss model was added to transverse momentum equation as follows [1].

$$\Delta P_L = -\frac{K_G}{2}\left(\frac{W_{IJ}\left|W_{IJ}\right|}{l_{IJ}\rho_{IJ}s_{IJ}}\right), \tag{2}$$

where, W_{IJ} is the mass flow which flows subchannel I to J and l_{IJ} is the length between the center of two subchannels and s_{IJ} is the gap size between two rods. K_G means lateral form loss coefficient and the default value is 0.5. Fig. 1 [1] shows control volumes for calculating axial and lateral governing equations.



(a) Axial direction (b) Lateral direction

Fig. 1. Control volumes for subchannel governing equation

2.2 Turbulent mixing Model

Turbulent mixing model contributes for momentum and energy redistribution by exchanging flow mass because of velocity difference at adjacent subchannels. For single phase flow, EM (Equal Mass exchange) model was applied to axial momentum equation as presented in Equation (3).

$$\Delta P = -\sum w'_{IJ}\left(U_I - U_J\right),$$
$$w'_{IJ} = \beta \times s_{IJ} \times \bar{G}, \tag{3}$$

where, w'_{IJ} is the amount of mixing flow which flows subchannel I to J and β is turbulent mixing coefficient which is determined by user's input [1]. In CUPID, the above-mentioned four subchannel models were implemented and the code was validated against the experiment data and the MATRA simulation results.

3. Validation results of CUPID

3.1 CNEN 4×4 mixing test

The CNEN 4×4 test was performed at Studsvik Laboratory for verifying mixing effect between subchannels [3]. In this experiment, the velocity was measured at the outlet of corner, side, center subchannels under various inlet velocity conditions. In the code calculation, simplified square geometry with 1,250 (5×5×50) cells was used. One grid spacer was located at the middle elevation of the test section.

Fig. 2 shows three dimensional calculation result of CUPID. In the figure, the liquid is concentrated to the center as flows upward. This occurs due to the cross-flow model which attributes low flow resistance at center subchannel compared with other subchannels. With grid spacer model, CUPID could calculate pressure drop along axial direction and the results were well agreed with that from MATRA as shown in Fig. 3.

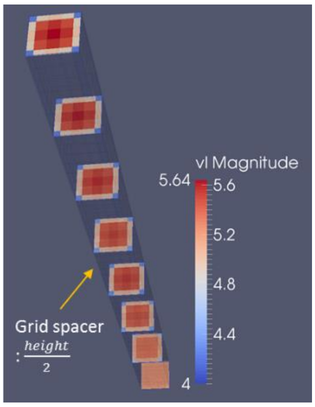


Fig. 2. Velocity contour using Paraview 4.1.0

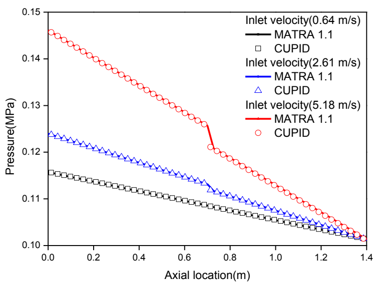
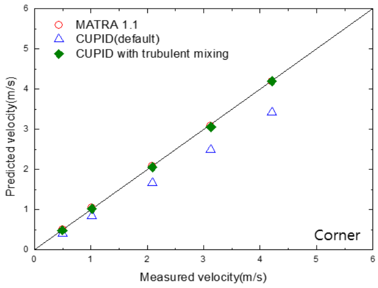
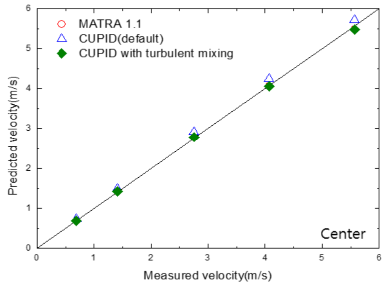


Fig. 3. Pressure drop along axial direction

Fig. 4 indicates the outlet velocity at corner and center subchannel and it clearly shows the effect of turbulent mixing model. With turbulent mixing model, CUPID could capture the experimental result with error below 2.6%.



(a) Outlet velocity along axial direction (corner)



(b) Outlet velocity along axial direction (center)

Fig. 4. Velocity at corner and center subchannel along axial direction

3.2 PNL 7×7 flow blockage test

The PNL 7×7 test investigated the flow pattern near the sleeve blockage. The sleeve blockage could be occurred as a result of swelling or ballooning on the fuel rod during design basis accidents [4]. The number of cells used for the CUPID calculation was 1,600 (8×8×25) and three grid spacers were located at the test section.

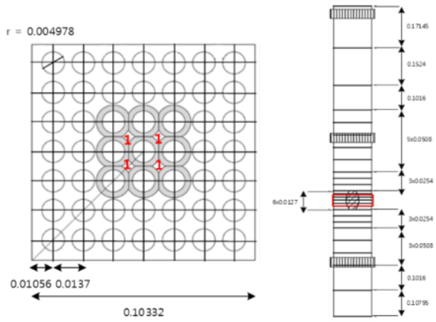


Fig. 5. Cross section and axial view of PNL 7x7 test section

As shown in Fig. 5, the sleeve blockage was located at the center of nine rods and the area was reduced to 70% at four marked center subchannels between two grid spacers. Change of the subchannel geometry near nine rods was considered in calculation. The bypass flow in front of the blockage is described as shown in Fig. 6 and Fig. 7. Calculation results indicate that CUPID could reproduce the jet effect at blockage and flow recovery by turbulent mixing after passing the blockage.

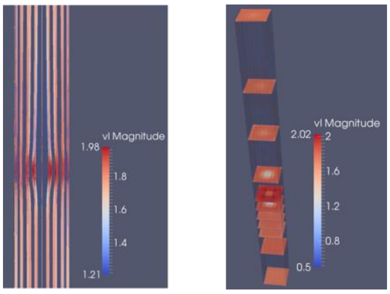


Fig. 6. Stream line and velocity contour along axial direction

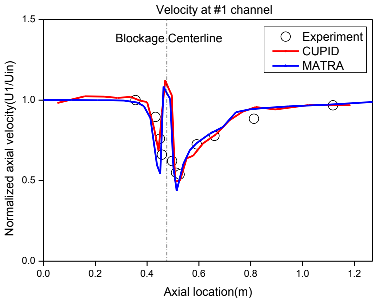


Fig. 7. Comparison results of the velocity at 70% blockage

Further calculation was carried out with larger blockage ratio up to 99%. As the ratio increases, the amount of liquid which flows into the subchannel decreases and velocity reaches almost zero as shown in Fig. 8.

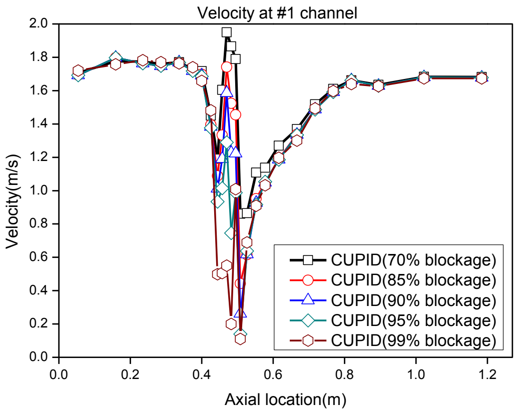


Fig. 8. Velocity with increasing of blockage ratio

3.3 CE 15×15 inlet jetting test

The objective of test was for verifying the influence of non-uniform inlet velocity to flow distribution in a rod bundle [5]. The number of cells used at calculation was 8,192 (16×16×32) and one grid spacer was located at the middle of the test section. The MATRA and CUPID calculation results show reasonably good agreement as presented in Fig. 10 and the maximum error is 8.2% at center line and 9% at tangent line.

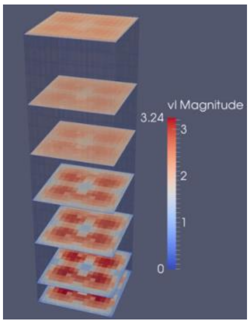
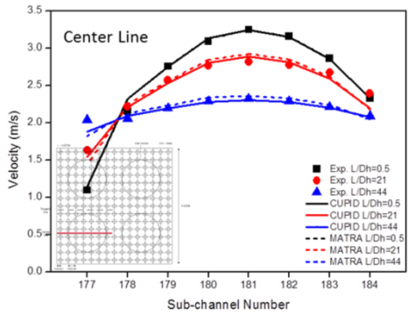
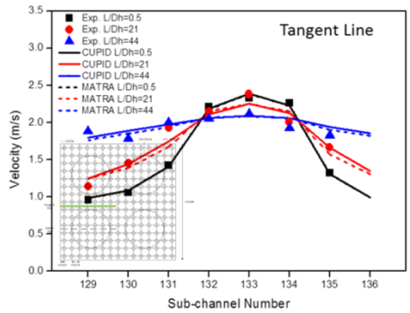


Fig. 9. Velocity contour with non-uniform inlet velocity



(a) Velocity along center line



(b) Velocity along tangent line

Fig. 10. Velocity distribution at center and tangent line along axial direction

3.4 WH 14×14 blockage test

The test section of WH 14×14 test consisted of two open 14×14 array fuel assemblies. The test investigated the flow redistribution between two open fuel assemblies caused by partial or complete blockage at the entrance of one assembly [6]. Inlet mass flows were different at two fuel assemblies to simulate a partially or completely blocked in one of the fuel assemblies. The number of grids used at calculation was 16,530 (15×29×38).

In the case of the partial blockage, flow redistribution between two fuel assemblies and velocity distribution along axial direction were calculated as shown in Fig. 11 and Fig. 12. Mass flows in each assembly are gradually flattened as the liquid flows upward. The velocity peak is shown on the middle line between two subchannels where the flow area is enlarged. These calculation results are comparable with those of MATRA and experiment.

Additionally, the case of fully blocked was simulated to confirm the blockage modelling capability of CUPID and a qualitatively reasonable result could be obtained including the flow recirculation near the completely blocked inlet nozzle as indicated in Fig. 13.

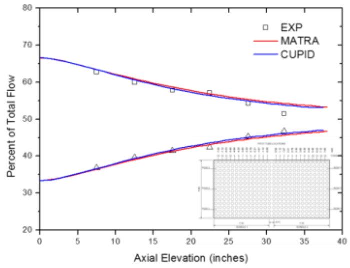


Fig. 11. Flow redistribution between two open rod bundles

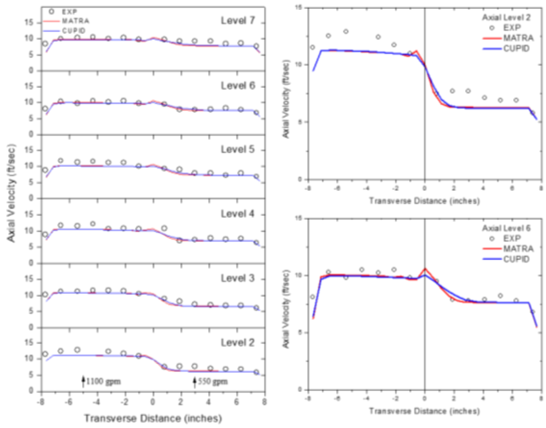


Fig. 12. Velocity distribution along axial direction at partial blockage case

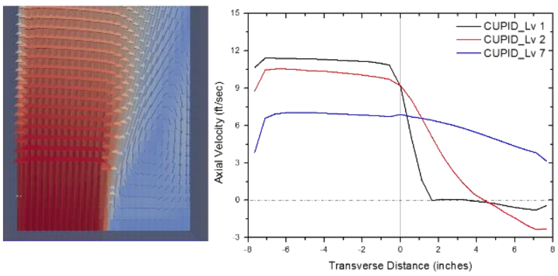


Fig. 13. Reverse flow occurrence at complete blockage case

4. Conclusions

In this paper, the validation results of the CUPID code for subchannel scale rod bundle analysis at single phase adiabatic conditions were presented. At first, the physical models required for a subchannel scale analysis were implemented to CUPID. Afterwards, four validation calculations were conducted with comparison of the calculation result of MATRA and the experimental data. From these validation results, subchannel scale simulation capability of CUPID at single phase conditions was confirmed. In addition, CUPID showed its capability handling a reverse flow at full blockage of a subchannel.

In the future, the scope of validation tests will be extended to diabetic and two phase flow conditions and required models will be implemented into CUPID.

ACKNOWLEDGEMENT

This work was supported by National R&D program through the Ministry of Education of the Republic of Korea and National Research Foundation of Korea (NRF). (No. NRF-2015M2A8A4021769)

REFERENCES

[1] KAERI, Validation of a Subchannel Analysis Code MATRA Version 1.1, KAERI/TR-5581/2014.

[2] H.Y. Yoon, H.K. Cho, J.R. Lee, I.K. Park, J.J. Jeong, Multi-Scale Thermal-Hydraulic Analysis of PWRs using the CUPID Code, Nuclear Energy and Technology, Vol.44, pp.831-846, 2012.

[3] V. Marinelli, L. Pastori, B. Kjellen, Experimental Investigation on Mass Velocity Distribution and Velocity Profiles in an LWR Rod Bundle, Trans. ANS 15, pp.413-415, 1972.

[4] J. M. Creer, D. S. Rowe, J. M. Bates, A. M. Sutey, Effects of Sleeve Blockages on Axial Velocity and Intensity of Turbulence in an Unheated7x7 Rod Bundle, BNWL-1965, 1976.

[5] R. C. Marshall, R. P. Letendre, Influence of Inlet Geometry on Flow in the Entrance Region of a Nuclear Reactor Rod Bundle, ASME, 68-WA/HT-34, pp.1-8, 1969.

[6] H. Chelemer, P. T. Chu, L. E. Hochreiter, THINC-IV: an Improved Program for Thermal-Hydraulic Analysis of Rod Bundle Cores, WCAP-7956, Westinghouse Electric Corp., 1973.