Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 18-19, 2017

Numerical Analysis on Single & Two-phase Turbulent Flow in Subchannel of 2x2 Rod Bundle Using the CUPID code

Jong Hyuk Lee [a*], Jae Ryong Lee[a], Ik kyu Park [a], J. Kim [b]
[a]*Thermal Hydraulics Safety Research Division, Korea Atomic Energy Research Institute, 989-111 Daedeok-daero, Yuseong-gu, Daejeon, 305-353, Ref. of Korea.*
[b]*Dept. of Mechanical System and Design Engineering, Seoul National Univ. of Science and Technology, 232, Gongneung-ro, Nowan-gu, Seoul, Ref. of Korea.*
[*]*Corresponding author: leejonghyuk@kaeri.re.kr*

1. Introduction

It is important to understand the flow feature in a subchannel of nuclear rod bundle in the design process of a nuclear reactor considering the safety margin. However, it is not easy to precisely expect the flow phenomena in the subchannel due to the effect of turbulent mixing. And, because of the heat from the reactor core the flow is usually not a single- phase in the subchannel. When the flow is two-phase, it is more difficult to expect and understand what it happened. In order to analyze the turbulent flow in a subchannel of rod bundle, turbulent model should be validated in single and two-phase flow.

In this paper, we conducted the validation of turbulent flow in the subchannel of a 2x2 rod bundle using the CUPID code[1-3]. The governing equations of the CUPID code, based on two-fluid, three-field model, are similar to those of the time-averaged two-fluid model derived by Ishii and Hibiki[4]. The standard k-ε turbulence model was adopted to validate the turbulent flow in single and two phase flow.

2. Model & Results

2.1 Turbulent Model

There are additional unknown terms in the averaging procedure for a momentum equation. The terms, called 'Reynolds (turbulent) stresses", contains the products of the fluctuating quantities and acts like additional stresses in a fluid. It is difficult to determine the value directly. The turbulent eddy viscosity model is adopted for the turbulent flow analysis of single and two phase flow fields in the CUPID code.

The turbulence energy (*k*) and the turbulence length scale (*ε*, viscous dissipation rate) are determined from the differential transport equation in standard *k-ε* turbulence model. The effective viscosity of a continuous liquid phase is the sum of the laminar viscosity, turbulence viscosity, and bubble effect.

$$\mu_{l,eff} = \mu_l + \mu_{l,T} + \mu_{b,T} \tag{1}$$

where,

$$\mu_{l,T} = C_\mu \rho_l \frac{k_l^2}{\varepsilon_l} \tag{2}$$

$$\mu_{b,T} = C_{\mu,g} d_b \rho_l \alpha_g \left| \underline{u}_g - \underline{u}_l \right| \tag{3}$$

The *k-ε* transport equations for continuous liquid phase can be shown as follows:

$$\begin{aligned} &\frac{\partial(\alpha_l \rho_l k_l)}{\partial t} + \nabla \cdot (\alpha_l \rho_l k_l \underline{u}_l) \\ &= \nabla \cdot \left[\alpha_l \left(\mu + \frac{\mu_T}{\sigma_k} \right)_l \nabla k \right] + \alpha_l P_k - \alpha_l \rho_l \varepsilon \end{aligned} \tag{4}$$

$$\begin{aligned} \frac{\partial(\alpha_l \rho_l \varepsilon_l)}{\partial t} + \nabla \cdot (\alpha_l \rho_l \varepsilon_l \underline{u}_l) = \nabla \cdot \left[\alpha_l \left(\mu + \frac{\mu_t}{\sigma_k} \right)_l \nabla \varepsilon \right] \\ + \frac{\alpha_l \varepsilon}{k} \left(C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho_l \varepsilon \right) \end{aligned} \tag{5}$$

where $C_\mu, \sigma_\varepsilon, \sigma_k, C_{\varepsilon 1}, \text{and}\, C_{\varepsilon 2}$ are recommended to have a value of 0.09, 1.3, 1.0, 1.44, and 1.92 in the standard $k-\varepsilon$ model, respectively.

The wall-function in the CUPID code is an extension of the method of Launder and Spalding[5]. In the log-law region, the near wall tangential velocity is related to the wall-shear-stress by means of a logarithmic relation. In the wall-function approach, the viscosity affected sublayer region is bridged by employing empirical formulas to provide near-wall boundary conditions for the mean flow and turbulence transport equations. The logarithmic relation for the near wall velocity is given by.

$$u^+ = \frac{U_t}{u_\tau} = \frac{1}{\kappa} \ln(Cy^+) = \frac{1}{\kappa} \ln\left(C \frac{\rho \Delta y u_\tau}{\mu} \right) \tag{6}$$

where u^+, u_τ, U_t, y^+, κ, and C are the dimensionless velocity, friction velocity, near wall tangential velocity known velocity tangent to the wall at a distance of Δy from the wall, dimensionless distance from the wall, von Karman constant, and constant depending on the wall roughness.

A wall-function simulation normally requires that y^+ of the first cell outside the walls is in the log-layer, which starts at about $y^+ = 20$, and depending on the Reynolds number, extends up to say $y^+ = 200$.

2.2 Modeling

The experimental study[6] was selected to analyze and validate the single and two phase turbulent flow phenomena in the rod bundle. They have conducted in a rectangular channel including 0.01 m diameter of four rods, as an experiment for measuring the velocities for each phase in 2x2 rod bundle.

Fig. 1 shows the geometry and grid used for 2x2 rod bundle tests. The SALOME open-source code, which can provide both structured and unstructured mesh, was used in the process of the modeling and the grid generation.

In order to consider the wall-function condition, it consists of the structured grid at the wall and the unstructured grid in the other regions. If the first distance from the wall is 1 mm, the minimum value of y+ is approximately 27 and 52 for each Re condition, respectively. In this case, the number of nodes and cells is 73,067 and 112,800, respectively.

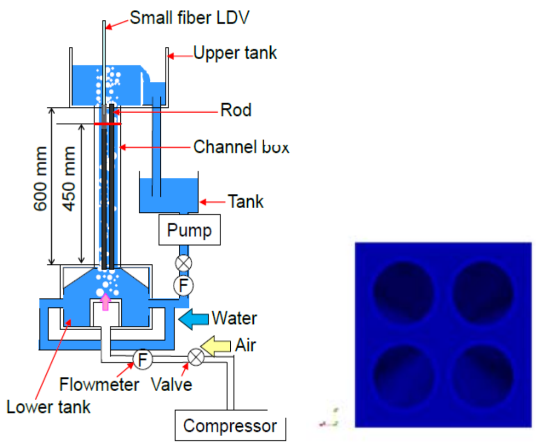


Fig. 1.Geometry and grid for 2x2 rod bundles [6].

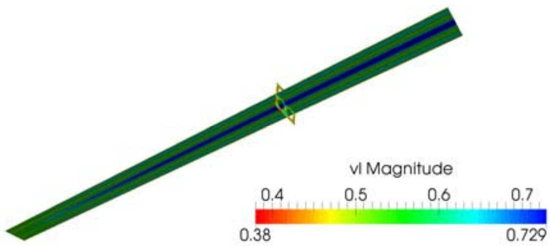
2.3 Results

The boundary conditions for the simulations are summarized in Table I. The artificial liquid velocity was 0.5 and 1.0 m/s and the gas velocity was 0 and 0.08 m/s.

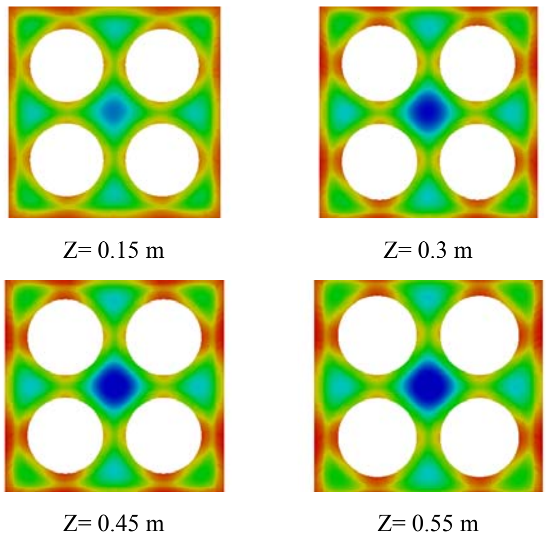
Fig. 2 shows the velocity distributions of cross section to the axial direction and at an each height (z=0.15, 0.3, 0.45, 0.55 m) for the 1S case. From the figure, we can check that the velocity was developed through the flow channel. Because of applying no slip condition at the wall, the flow velocity at the center of the subchannel was the fastest comparing of the ones at the inter-rods or near the wall. Because there are only four rods in the flow channel without a flow mixer like a spacer grid, the flow represents a symmetry. Based on this result, it is reasonable to compare the results considering the symmetric distribution, although the CUPID have simulated all 2x2 rod bundles.

Table I: Boundary conditions

Case	J_L [m/s]	J_G [m/s]	Re
1S	0.5	0	4850
2S	1.0	0	9700
1B	0.5	0.08	4850
2B	1.0	0.08	9700



(a) Velocity distribution in axial direction



(b) Velocity distribution for each height

Fig. 2. Two-dimensional sectional velocity distribution (1S case).

In the experiment [6], the velocities were measured at 0.45 m distance from the inlet according to the azimuth angle ranged from 0 to 45 degree. Fig. 3 represents the liquid velocity distribution at 0.45 m height and 0 and 45 degree of azimuth angle for each boundary condition. At 45 degree of azimuth angle, the calculated velocities were a good agreement with the experimental data.

However, the results at 0 degree of azimuth angle shows a little lower than the ones for the experiments.

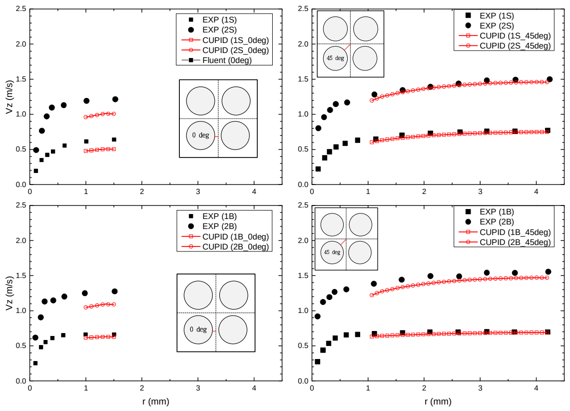


Fig. 3. Liquid velocity distribution at 0 degree & 45 degree of azimuth angle for each BCs.

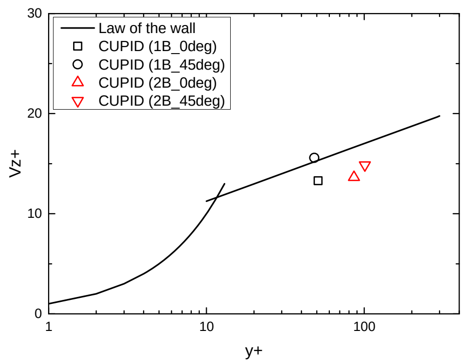


Fig. 4. Dimensionless axial velocity for 1B & 2B case.

In order to clarify the reason for the discrepancy between the CUPID code and experimental results, we also checked the relationships between the dimensionless axial velocity, Vz+, normalized by the frictional velocity $u_\tau = \sqrt{\tau_w / \rho}$ and the dimensionless distance y+ from the wall as shown in fig. 4. The axial dimensionless velocity near the wall at 45 degree of azimuth angle is closer than 0 degree case from the law of the wall. Therefore, the velocities for 45 degree cases are more similar to the experimental results. The difference between 0 and 45 degree cases is caused by the velocity, which can be determined by the wall-function at the wall.

3. Conclusions

It is important to understand the flow phenomena in a rod bundle in terms of reactor design considering the safety. In this study, the CUPID code was used to simulate the flow phenomena in the 2x2 rod bundle. The simulation of 2x2 rod bundle test under single and two phase flow condition were conducted to validate the turbulence model. The liquid velocities at 0.45 m along the flow path with 0 and 45 degree of azimuth angle were compared with experimental data. The results for 45 degree of azimuth angle were a good agreement. However, the ones for 0 degree case has a little discrepancy. Based on the comparison of dimensionless axial velocity, the differences might be caused by the velocity near wall, which can be determined by wall-function. If the velocity near the wall is improved in the CUPID code, the flow in a rod bundle can be expected more precisely.

ACKNOWLEGEMENTS

This work was supported by the National Research Foundation of Korea (NRF) and the Korea Foundation of Nuclear Safety (KoFONS) grant funded by the Korean government (MSIP & NSSC)(Nuclear Research and Development Program: 2017M2A8A4015005, Nuclear Safety Research Center Program: 1305011).

REFERENCES

[1] J. J. Jeong, H. Y. Yoon, I. K. Park, H. K. Cho, and J. Kim, A semi-implicit numerical scheme for transient two-phase flows on unstructured grids, Nuclear Engineering and Design, 238, pp. 3403–3412, 2008.

[2] J. J. Jeong, H. Y. Yoon, I. K. Park, and H. K. Cho, The CUPID code development and assessment strategy, Nuclear Engineering and Technology, 42(6), pp.636–655, 2010.

[3] H. Y. Yoon, H. K. Cho, J. R. Lee, I. K. Park, and J. J. Jeong, Multi-Scale Thermal-Hydraulic Analysis of PWRs using the CUPID Code, Nuclear Engineering and Technology, 44(8), pp.831–846, 2012.

[4] M. Ishii and T. Hibiki, Thermo-Fluid Dynamics of Two-Phase Flow, Springer, 2006.

[5] B.E. Launder and D.B. Spalding, The Numerical Computation of Turbulent Flows, Computer Methods in Applied Mechanistic and Engineering, Vol.3, No.2, pp.269-289, 1974.

[6] S. Hosokawa, Y. Ogawa, and A. Tomiyama, Liquid velocity distribution in turbulent bubbly flow in a 2x2 rod bundle, NTHAS9, Buyeo, Korea, Nov. 16-19, 2014.