Numerical simulation of two-phase flow in 4x4 simulated bundle

Ayako ONO*, Susumu YAMASHITA*, Takayuki SUZUKI* and Hiroyuki YOSHIDA*

*Japan Atomic Energy Agency
4-2 Shirakata, Tokai-mura, Naka-gun, Ibaraki 319-1195, Japan
E-mail: ono.ayako@jaea.go.jp

Abstract
An evaluation methodology of a thermal-hydraulics based on a mechanism in light water reactors (LWRs) is needed from a viewpoint of the safety analysis during normal operation and unanticipated transient such as under a severe accident. Currently, the evaluation of safety for the nuclear reactor has been implemented by a best estimate (BE) code and subchannel analysis code. These analysis codes contain models and empirical correlations. Therefore, the full-scale mock-up tests are needed to evaluate the reliability and validation of code. And the model and empirical correlation are allowed to be applied only in the range where the experiments were implemented. The large mock-up tests are once again needed in order to consider the new geometry and boundary conditions when the design of components is changed. Hence, the 3D detailed numerical simulation by the mechanistically based method is expected to be applied for the preliminary analysis to improve the design of fuel assemblies and evaluate the safety. This 3D detailed numerical simulation can reduce the large mock-up tests. The detailed numerical simulation method can provide much information relating to the two-phase flow such as the bubble size, its velocity, and detailed void distribution which, for example, are needed to predict the critical heat flux based on the mechanism. Moreover, JAEA is implementing the development of the nuclear-thermal-coupling code by using a detailed two-phase flow analysis code based on the VOF method like a JUPITER code. In this study, the numerical simulation of two-phase flow in the 4x4 bundle was examined by numerical simulation code JUPITER in order to examine the possibility of the JUPITER code for the large scale two-phase flow analysis. The simulation results are verified by the previous experimental data of two-phase flow.

Keywords: Two-phase flow, Light water reactor, Fuel bundle, VOF, Surface tracking method

1. Introduction

The evaluation methodology of a thermal-hydraulics based on a mechanism in light water reactors (LWRs) is needed from a viewpoint of the safety analysis during normal operation and unanticipated transient such as under a severe accident. Currently, the safety evaluation has been conducted based on the experimental data of full-scale mock-up tests to predict the critical heat flux (CHF) and the pressure loss coefficient, and so on. The various subchannel codes have been mainly used for the evaluation of thermal-hydraulics in the fuel bundles. However, the subchannel codes contain the empirical equations and need the information relating to the flow regime to choose the appropriate correlations. Therefore, it is difficult to apply the subchannel code for the transient unexpected event. The best estimate (BE) codes have been applied for the abnormal transient event and accident where the damaging of the core would not happen. They adopt the 1-D model verified by the scale model tests. However, these models and empirical equations in the subchannel codes and best estimate codes were developed in the limited experimental range. Therefore, the conditions for which these codes can apply are limited in the range where the experiments were implemented.

The recent dramatic development of supercomputers enables us to simulate the large scale phenomena directly with the detailed simulation method based on the mechanism. The preliminary detailed analysis by the computational fluid dynamics (CFD) is expected to investigate the thermal-hydraulics in the reactor and aid development of the BE code and subchannel code.
JAEA has been developing some detailed thermal-hydraulic codes including the JUPITER (JAEA Utility Program for Interdisciplinary Thermal-Hydraulics Engineering and Research) code which was developed to simulate the melting behavior of the core based on the VOF method. The JUPITER code is one of the candidates to apply for the detailed 3-D nuclear-thermal-coupling code which will be developed by JAEA in the future R&D plan. Therefore, in this study, the possibility of applying the JUPITER code for the two-phase flow in the fuel bundle is examined. The two-phase flow in the fuel bundle has been investigated by some researchers. Mizutani et al. (Mizutani, 2007) implemented the two-phase flow experiments in the 4x4 simulated fuel bundle by using the air and water. They observed the two-phase flow structure by using a high-speed camera. And they proposed the new flow regime map in the fuel bundle which is different from the one in the single pipe such as Mishima’s flow map (Mishima, 1984). In their flow map, there is no “Slug flow regime” since they did not observe any cap bubbles. Ren et al. (Ren, 2018) implemented the water-air experiments in a 5x5 simulated fuel bundle to connect the characteristics of the signal from their void meter and the flow regime which they observed by a high-speed camera. They measured the void fraction in the subchannel by using their impedance void meter. They analyzed the time sequence of the signal from the void meter and classified the flow regime based on the probability density distributions of the void signals. This method is a considerably reasonable method to define the flow regime from the simulation result without any subjective view. Finally, they showed the relation between the void fraction in the subchannel and the superficial gas velocity under the constant liquid velocity. This data is so useful to validate our simulation result by comparing the data between the experiment and simulation. In this study, the experimental results by Ren et al. will be used to validate our simulation results.

2. Numerical simulation by JUPITER
2.1 Numerical model

The JUPITER code (Yamashita, 2017) has been developed as a CFD code based on the thermal-hydraulic equations and multiphase simulation model. In the JUPITER, the equations of continuity and Navier-Stokes, and energy are solved as follows:

\[ \nabla \cdot \mathbf{u} = 0 \tag{1} \]

\[ \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\frac{1}{\rho} \nabla p + \frac{1}{\rho} \nabla \cdot (\mu [\nabla \mathbf{u} + (\nabla \mathbf{u})^T] + g + \mathbf{F} \tag{2} \]

\[ \frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T = -\frac{1}{\rho C_v} \nabla \cdot (\lambda \nabla T) + \frac{Q}{\rho C_v} \phi \tag{3} \]

In the JUPITER, the interface between the liquid and gas is described by advection equations of the VOF function (Hirt, (1981)), \( \phi_l \):

\[ \frac{\partial \phi_l}{\partial t} + \nabla \cdot (\mathbf{u} \phi_l) = \phi_l \nabla \cdot \mathbf{u} \tag{4} \]

The advanced surface capturing methods, THINC (Tangent of Hyperbola for Interface Capturing) (Xiao, 2005) or THINC/WLIC (Yokoi, 2007), are applied for the JUPITER. The VOF fraction is calculated by Eq. (4) where \( \phi_l \) is discretized as follows:

\[ \phi_l^{n+1} = \phi_l^n - \frac{F_l^{n+1/2} - F_l^{n-1/2}}{\Delta x} - \frac{F_l^{n+1/2} - F_l^{n-1/2}}{\Delta y} - \frac{F_l^{n+1/2} - F_l^{n-1/2}}{\Delta z} \tag{5} \]

Here, \( F_l \) is defined as follows:

\[ F_l = \phi_l \mathbf{u} \Delta t \tag{6} \]

The computational cost can be reduced and the interface between phases is captured sharply and efficiently by the above schemes. In addition to them, the immersed boundary method (IBM) (Kim, 2001) is adopted to reduce the calculation cost and effort for creating the elaborated numerical grid. This method can deal with complicated shapes in a Cartesian
grid system, which can apply for any shape of a structure. The interface between a solid body and fluid is defined by the level-set function, \( \psi \), which is defined at a center of a mesh (Sussman, 1994). The level-set function is defined as the signed distance function from the interface as follows:

\[
\psi(x) > 0 \quad \text{Fluid} \\
\psi(x) = 0 \quad \text{Fluid – body interface} \\
\psi(x) < 0 \quad \text{Body}
\] (7)

The CSF (Continuum Surface Force) model is adopted as a surface tension model (Brackbill, 1992). In this model, the surface tension is assumed as a body force and expressed by the delta function \( \delta(\psi) \) as follows:

\[
F_s = -\sigma \delta(\psi) \nabla \psi
\] (8)

The surface tension is assumed to distribute uniformly and the wettability is not considered in the JUPITER code.

The fractional-step method is adopted as the time-integration method for Eqs. (1)-(4). The velocity field is corrected by the pseudo-pressure, in which the continuity equation is satisfied at each computational time step. The third-order TVD (Total Variation Diminishing) Runge-Kutta method (Gottlieb, 1998) is applied for the time integration of convection and diffusion terms in Eqs. (2) and (4). The improved new Poisson solver is developed with hybrid MPI-OpenMP parallelization, and the hybrid parallelization for the whole code to deal with the large-scale simulation. The new solver is based on the conjugate gradient (CG) method with block Jacobi preconditioning (Saad, 2003). The parallel speed is increased up to ~200k core on the K-computer by the advanced numerical scheme such as the interface capturing scheme and the new hybrid parallel Poisson solver, which enables to simulate the thermal-hydraulics in the whole core. The JUPITER code has been verified for the single-phase flow, multi-phase flow which the single bubble rises in the pool, the solid boundary in the flow, and the phase change (Yamashita, 2017).

JAEA is implementing the two-phase flow experiment by using a 4x4 simulated fuel bundle under up to 2.6 MPa. The test fluid is water and vapor under the adiabatic condition. The test section has the wire mesh sensor to measure the void fraction in the subchannel and the measurement of the void distributions is undergoing. Therefore, the 4x4 simulated fuel bundle is chosen as a calculation domain since the numerical simulation results will be used as a preliminary analysis to draw up the experimental plans. And, the simulation results will be validated the simulation result by the data from JAEA experiments in the future. Figure 1 shows the computational domain in this simulation study. In the 56 x 56 x 1500 mm\(^3\) computational domain, there are the squarely arranged 16 simulated fuel pins. The diameter of the pin is 10 mm and the pitch between the fuel pins is 13 mm. The hydraulic diameter is 10.35 mm. At the bottom of the calculation domain, there are 65 squared shape orifices and their cross-section is 2x2 mm\(^2\). The air is injected from these orifices.

![Fig.1](image-url) The calculation domain for the JUPITER in this study. This domain is created based on the dimensions of JAEA experimental apparatus to measure the subchannel void fraction by the wire mesh sensor under the high-pressure two-phase flow experiments.
since the phase change is not taken into account in this simulation. Though the experimental test section has the spacer grids, they are not modeled in this study. The size of the mesh is 1x1x1 mm³ and the total mesh number is 4,704,000. The boundary condition is outflow at the top surface and the pressure is kept constant value at the outlet. The liquid velocity is given at the bottom surface of the domain as a constant value.

Table 1 shows the calculation conditions in this study. The simulation is implemented by using the water and air under the atmospheric conditions. In Tab.1, each flow regime is shown based on the flow regime map by Mizutani et al. (Mizutani, 2007) and Ren et al. (Ren, 2018). In this study, the flow regimes of calculation results from Case-1 to -3 are compared with a flow regime map by Mizutani et al. and Ren et al. The validation of calculation results is confirmed whether the calculation would reproduce each flow regime or not.

Table 1 Conditions for the calculations which are set by considering the experiments by Ren et al. (Ren, 2018). The flow regimes are defined based on the flow–regime maps by Mizutani et al. (Mizutani, 2007) and Ren et al. (Ren, 2018).

| Case Name                  | Case-1 | Case-2 | Case-3          |
|----------------------------|--------|--------|-----------------|
| Superficial liquid velocity, $j_l$ [m/s] | 0.100  | 0.100 | 0.100           |
| Superficial gas velocity, $j_g$ [m/s]    | 0.034  | 0.157 | 0.781           |
| Flow regime by Mizutani et al. (Mizutani, 2007) | Bubbly | Churn | Churn           |
| Flow regime by Ren et al. (Ren, 2018)     | Bubbly | Cap-bubbly | Cap-turbulent |

Ren et al. (Ren, 2018) implemented the air-water flow experiments to measure the void fraction in a 5x5 simulated fuel bundle under the atmospheric condition and room temperature. The test section was made of transparent polymethyl methacrylate. The size of the test section was 66.1 x 66.1 x 1500 mm³. The pin diameter was 9.5 mm and the pin pitch was 12.6 mm. The impedance void meter named as SCIVM was set at $L/D=76.9$ from the inlet. The air bubbles were produced at the inlet of the test section. In Tab.2, the experimental results by Ren et al. (Ren, 2018) are summarized. They implemented the observation of the flow structure by the high-speed camera, simultaneously, the void fraction was measured by the void meter. Finally, they showed the criteria to define each flow regime from the signal from the void meter. Table 3 shows the experimental conditions in which Ren et al. provided the result of observation by high-speed camera, the signals from SCIVM, and their probability distributions.

In Tab.3, the calculations in this study are implemented in three hatched experimental conditions. Although it is found that the flow regime maps are different between two-phase flow in the single pipe and the pin bundle as shown by Mizutani et al., the dependency of the number of the pins on the flow regime map is not pointed out. The two-phase flow condition in the chosen cases in Tab.3 does not seem to be observed the large bubbles which occupy over more than 3 subchannels according to the observation. This means that the effect of the distance between the sidewall and bubbles is considered to be considerably small in comparison between Ren’s data and the author’s simulation result. Therefore, the data of the flow regime provided by Ren et al. are compared with the simulation result in this study. In this study, the data from the calculation are analyzed based on the method of the flow regime classification proposed by Ren et al.

2.2 Result and discussion

Figure 2 (a)-(c) shows the two-phase flow structures at a moment in Case-1, -2 and -3. The left side in each case is the whole calculation domain and the right side in each case is the enlarged image from $L/D = 57.58$ to 96.23. In Case-1, the cap bubbles and discrete bubbles are observed. The shape of the cap bubble is the mushroom shape which the lateral width is larger than the vertical width. In Case-2, the various sizes of bubbles are confirmed such as the discrete bubble, cap shape bubble, and the deformed large bubble. The deformed large bubbles occupy in the whole area of subchannel or across some subchannels. In Case-3, since most of the gas flow the core part of each subchannel due to its large flow rate, the liquid area is formed on the fuel pins. The part looked blue color is the bare part which is not covered with the liquid.

The calculation domain in the horizontal cross-section is divided into 25 subchannels as shown in Fig.3. In this study, it is defined that the subchannels are classified into 3 regions based on their locations and hydraulic diameters,
Table 2 Summarized experimental results by Ren et al. (Ren, 2018). They implemented the water-air experiments and measured void fraction in subchannel from the impedance void meter (SCIVM). They connected the result from the visual observation and the signal from SCIVM. In this table, the summarize the results from visual observation and measurement by SCIVM and the schematics of probability distributions which express the characteristic points in their results.

| Flow regime       | Analysis of the signal from void meter and observation by high speed camera                                                                 | Schematics of probability distributions |
|-------------------|--------------------------------------------------------------------------------------------------------------------------------------------|----------------------------------------|
| Bubbly flow       | <Observation by high speed camera>                                                                                                                                                                       |
|                   | • Aggregations of small bubbles                                                                                                                                                                         |
|                   | • Single distorier bubbles                                                                                                                                                                               |
|                   | <Void fluctuation in center subchannel>                                                                                                                                                                  |
|                   | • Small peaks of void fraction around 0.1 were detected.                                                                                                                                                 |
|                   | • Some large peaks less than 0.3 were detected.                                                                                                                                                         |
| Cap bubbly flow   | <Observation by high speed camera>                                                                                                                                                                       |
|                   | • Cap bubbles                                                                                                                                                                                           |
|                   | • Dense distorted bubbles                                                                                                                                                                                |
|                   | <Void fluctuation in center subchannel>                                                                                                                                                                 |
|                   | • Larger peak is close to 0.9.                                                                                                                                                                           |
|                   | • Continuous small fluctuations between large peaks                                                                                                                                                    |
| Cap turbulent flow| <Observation by high speed camera>                                                                                                                                                                       |
|                   | • Large slug bubbles                                                                                                                                                                                    |
|                   | • Cap bubbles                                                                                                                                                                                           |
|                   | • Interval necks of bubbles                                                                                                                                                                              |
|                   | • Small bubbles in the liquid slug                                                                                                                                                                      |
|                   | <Void fluctuation in center subchannel>                                                                                                                                                                 |
|                   | • Low fluctuation around 0                                                                                                                                                                               |
|                   | • Trapezoid shape signal                                                                                                                                                                                 |
|                   | • Void fraction covers values from 0 to 1.                                                                                                                                                               |
| Churn turbulent flow| <Observation by high speed camera>                                                                                                                                                                       |
|                   | • Large bubbles                                                                                                                                                                                         |
|                   | • Liquid slugs                                                                                                                                                                                          |
|                   | <Void fluctuation in center subchannel>                                                                                                                                                                 |
|                   | • Averaged void fractions is almost larger than 0.6.                                                                                                                                                     |
|                   | • Wide peak corresponding to large distorted bubbles                                                                                                                                                     |
|                   | • Downward peak corresponding to short liquid slug                                                                                                                                                       |
| Annular flow      | <Observation by high speed camera>                                                                                                                                                                       |
|                   | • Liquid film                                                                                                                                                                                           |
|                   | • Disturbed waves of the liquid film                                                                                                                                                                     |
|                   | <Void fluctuation in center subchannel>                                                                                                                                                                 |
|                   | • Void fluctuation is almost above 0.85                                                                                                                                                                 |
|                   | • Downward peaks corresponding to disturbed waves of liquid film                                                                                                                                          |
|                   | • Base of void fluctuation corresponding to thickness of liquid film substrate                                                                                                                             |
Table 3 Experimental conditions provided in Ren’s study. In these experimental conditions, the result of observation by high-speed camera, the signal from SCIVM, and the probability distributions of void fraction.

| Flow regime    | Superficial liquid velocity [m/s] | Superficial gas velocity [m/s] |
|----------------|-----------------------------------|-------------------------------|
| Bubbly flow    | 0.066                             | 0.034                         |
|                | 0.10                              | 0.034                         |
|                | 0.10                              | 0.057                         |
| Cap bubbly flow| 0.066                             | 0.126                         |
|                | 0.1                               | 0.125                         |
|                | 0.1                               | 0.157                         |
| Cap turbulent flow| 0.065                          | 0.78                          |
|                | 0.10                              | 0.78                          |
|                | 0.10                              | 1.02                          |
| Churn turbulent flow| 0.067                        | 4.87                          |
|                | 0.10                              | 4.81                          |
|                | 0.10                              | 6.30                          |
| Annular flow   | 0.067                             | 10.47                         |
|                | 0.10                              | 10.25                         |
|                | 0.15                              | 9.93                          |

which are the inner subchannel, side subchannel, and corner subchannel. Especially, the No.13 subchannel is called as the center subchannel. It is noted that this definition is slightly different from them in Ren’s study since the numerical domain in this study is 4x4 fuel bundles. The void fluctuation in the center subchannel, No.13 is calculated as shown in Fig.4. The instantaneous space-averaged void fraction in the center subchannel at a certain height is calculated from the calculation result of the bubble distribution at a moment, \( t = t_0 \). The time sequences of the instantaneous space-averaged void fraction in the center subchannel in each case are shown in Fig.5 (a)-(c). In Fig.5 (a), some large peaks are observed in Case-1, of which maximum void fraction reaches around 0.6. These may correspond to the cap bubbles in Fig.2 (a). Most of the cap bubble in Case-1 is smaller than the area of subchannel since the maximum peak of the void fraction does not reach 1.0. According to the bubbly flow in Tab.2, Ren et al. also detected some peaks of void fraction and their maximum value was 0.3. The size of the cap bubble in the simulation seems to be larger than that in the experiments though the number of the cap bubbles is not so large.

In Fig.5 (b), the number of peaks of which maximum values are from 0.6 to 1.0, which is larger than that in Case-1. This means the size of coalesced bubbles became larger and tend to occupy the whole area of the subchannel. According to the cap bubbly flow in Tab.2, the maximum values of peaks reached around 0.9 and this value almost agrees with the calculation results. Therefore, it is found that the size of bubbles in the calculation result agrees well with the experimental results. The small peaks are also observed and these peaks appear in a row. This may correspond to some discrete bubbles in the liquid slug. These small continuous peaks were also confirmed in the experiment by Ren et al. In Fig.5 (c), the void fraction keeps a large value above 0.3. This means the void is always existing in the center subchannel and there is no liquid slug. As shown in Fig.2 (c), the core part of the subchannel is occupied by the air and the liquid is constantly existing near the rods. There are various types of peaks that are from a shorter duration to a longer duration. This seems to correspond to changing the amount of liquid near the rods. Though Ren et al. observed the low fluctuation of signal from the void meter, which was corresponding to the small bubbles in the liquid slug, the calculation result in Case-3 does not reproduce the existence of the liquid slug. A few of the downward peaks are observed, which seem to correspond to the liquid remained between the pins as shown in Fig.2 (c). The calculation does not seem to reproduce the flow structure of the cap turbulent flow described in Tab.2.

Figure 6 (a)-(c) show the probability distributions of void fraction in the center area subchannels (See Fig.3) in each case. In Fig.6 (a)-(c), the open circle is the calculation result, and the broken red line is the experimental results by Ren et al. (2018). In Fig.6 (a), the sharp peak is observed below 0.01 of void fraction in the calculation result. On the other hand, the probability of void fraction from 0.01 to 0.04 in the calculation results is less than that of the experimental results. This means the number of the bubbles corresponding to the void fraction from 0.01 to 0.04 is smaller than that from the experiment by Ren et al. Since the JUPITER treats the incompressible fluid, the rising bubble volume in the experiments will become larger than that in the simulations depending on the height in the bubbly flow under the
Fig.2  Calculation results in Case-1, -2, and -3. The left figure in each case is whole image and the right figure in each case is the enlarged image centering around $L/D = 76.9$ where Ren et al. measured the void fraction.

Fig.3  Definition of term relating to the location of subchannel in this study. The subchannels are classified into 3 which are the corner subchannel, side subchannel, and inner subchannel. Especially, the subchannel, No.13, is named as the center subchannel in this study.
atmospheric condition. Therefore, all of the bubble sizes at the $L/D=76.9$ may be underestimated. However, the qualitative configuration of distribution from the calculation result agrees with the distribution of the bubbly flow. Therefore, the calculation result should be classified as the bubbly flow and the calculation could reproduce this flow regime. In Fig.6 (b), the probability in the higher void fraction region is larger compared with the Case-1. In Case-2, some types of bubbles are observed as shown in Fig.2 (b) and Fig.5 (b). The existence of the deformed bubbles increases the probability of a larger void fraction in the calculation result. This characteristic is also observed in the experimental result of the cap bubbly flow. In Case-3, the peak of probability from 0.4 to 0.9 is observed in the calculation result. Comparing with the

Fig.4 Method to calculate the void fraction in the center subchannel. In this study, the fluctuation of void fraction is focused as a characterized parameter to identify the flow regime.

Fig. 5 Fluctuation of void fraction in the center subchannel at $L/D = 76.9$. The method to calculate the void fraction is shown in Fig.4. Since the duration of peak seems to correspond to the bubble size, the flow regime is conjectured by the configuration of peaks.
experimental data of the cap turbulent flow as shown by the red broken line, the configuration of probability distribution does not agree well. The configuration of the probability distribution from the calculation is close to that in the churn turbulent flow in Tab.2. Therefore, the calculation cannot reproduce the flow regime in the higher gas velocity region.

Fig.6 Probability distribution of void fraction in inner area subchannel. According to the data base by Ren et al. in Tab.2, the flow regime calculation results can be identified from the probability distributions which represents the flow regime.

Fig.7 Distribution of void fraction in the flow direction from $L/D=0$ to 30. The left figure in each case is the distribution of time averaged void fraction for 6 seconds. The right figure in each case is the instantaneous two-phase flow structure by calculation.
To summarize the calculation results, there are two inconsistencies between the calculation result and the experimental result by Ren et al. One is the size of the cap bubble in the bubbly flow, another is the inconsistency of the flow regime in a higher gas velocity region. Figure 7 shows the distribution of void fraction near inlet ($L/D = 0$–$30$) and instantaneous image of calculated two-phase flow structure. The calculated void fraction is spatially averaged over the horizontal cross-section at each height and temporally averaged for 6 seconds. In the Case-1, there is the issue of inconsistency relating to the bubble size. One reason may be the neglect of the compressibility of air as mentioned before. Here, another reason will be discussed. In the two-phase flow image of Fig. 7 (a), the injected gas becomes longer jet shape. This is also found from the low void fraction near the inlet. The bubbles are created by the necking of this gas jet. The size of the bubble is determined by the force balance to the interface between the gas and liquid at this time. In the calculation, the two types of bubbles are created such as the small discrete bubbles and the cap bubbles. Therefore, it is found that the characteristics of bubbly flow are governed by the method of injecting gas phase at the inlet and the initial bubble size should be adjusted to the experimental result by changing the gas velocity from orifices and the number of orifices. This is the main reason why the bubble sizes in the calculation in Case-1 is inconsistent with them in the experimental result. In the Case-3, there is the mismatching of flow regime between the calculation result and experimental result by Ren et al. In the distribution of the void fraction, the void fraction increases around $L/D = 0.2$ and starts to decrease in the flow direction. Therefore, the rapid coalescence occurs near the inlet and create the large gas core since the velocity of gas from orifices is much larger than that of the liquid. This would promote the shift to the churn turbulent flow regime and cause mismatching the flow regime. In this way, the method to inject the gas phase should be considered carefully to reproduce the experimental conditions. Except for these issues relating to the utilization method of the JUPITER code, how the user should give the inlet conditions and initial conditions, it is confirmed the possibility of direct simulation for the two-phase flow by the JUPITER code in the wide range.

3. Conclusion

In this study, the numerical simulation of two-phase flow in the 4x4 bundle was examined by numerical simulation code JUPITER to examine the possibility of the JUPITER code for the large scale two-phase flow analysis. The simulation results were verified by the previous experimental data of two-phase flow.

(1) The JUPITER code has a higher possibility to simulate the two-phase flow in the 4x4 simulated fuel bundle in the wide range of the superficial gas velocity on the mechanistically based method.

(2) In the Case-1, where is in the bubbly flow region according to Ren’s study, the size of the cap bubble in the simulation result is not agreed with them in the experimental result. Since the force balance around the bubble strongly affects the shape and size of bubbles, the inlet velocity from the orifice and the number of the orifice should be set carefully considering the bubble size in the experiment result. Besides, the neglecting of compressibility would be one of the reasons why the bubble size was not reproduced.

(3) In the Case-3, where is in the cap-turbulent flow region according to Ren’s study, there is the mismatching of flow regime between the calculation result and experimental result by Ren et al. This is caused by the coalescence of gas near the inlet due to much larger gas velocity comparing with the surrounding liquid velocity. In the calculation of the higher superficial gas velocity region, the gas velocity from orifices and number orifice should be taken into account.

References

Brackbill, J.U., Kothe, D.B., and Zemach, C., “A continuum method for modeling surface tension”, J. Comput. Phys. Vol.100, (1992), pp.335-354.

Gottlieb, S. and Shu, C., “Total variation diminishing Runge-Kutta schemes”, Math. Comp., No.67,(1998), pp.73-85.

Hirt, C.W., and Nichols, B.D., Volume of fluid (VOF) method for the dynamics of free boundaries, J. Compt. Phys., Vol.39, (1981), pp.201-225.

Kim, J., Kim, D., and Choi, H., “An immersed boundary finite-volume method for simulations of flow in complex geometries”, J. Comput. Phys. No.171, (2001), pp.132-150.

Mishima, K. and Ishii, M., “Flow regime transition criteria for upward two-phase flow in vertical tubes”, Int. J. Heat
Acknowledgement

This research was conducted using the supercomputer SGI ICE X in the Japan Atomic Energy Agency.

Nomenclature

\[ A_{bi} \]  \quad \text{Area of the bubble in subchannel} \quad [\text{m}^2]

\[ A_{cs} \]  \quad \text{Area of the center subchannel}

\[ F \]  \quad \text{external force} \quad [\text{N}]

\[ F_s \]  \quad \text{surface tension force} \quad [\text{N}]

\[ F_{li\pm 1/2}, F_{lj\pm 1/2}, F_{lk\pm 1/2} \]  \quad \text{VOF flux on } i, j, k\text{-th control volume surface for } x, y, z \text{ direction} \quad [\text{m}]

\[ g \]  \quad \text{acceleration due to gravity} \quad [\text{m/s}^2]

\[ j_g \]  \quad \text{superficial gas velocity} \quad [\text{m/s}]

\[ j_l \]  \quad \text{superficial liquid velocity} \quad [\text{m/s}]

\[ L \]  \quad \text{Length of bundle} \quad [\text{m}]

\[ Q \]  \quad \text{heat source} \quad [\text{W/m}^3]

\[ t \]  \quad \text{time} \quad [\text{s}]

\[ T \]  \quad \text{temperature} \quad [\text{K}]

\[ u \]  \quad \text{velocity vector} \quad [\text{m/s}]

\[ x \]  \quad \text{position vector} \quad [\text{m}]

\[ \alpha \]  \quad \text{void fraction} \quad [-]

\[ \Delta x \]  \quad \text{grid width for } x\text{-direction} \quad [\text{m}]

\[ \Delta y \]  \quad \text{grid width for } y\text{-direction} \quad [\text{m}]

\[ \Delta z \]  \quad \text{grid width for } z\text{-direction} \quad [\text{m}]

\[ \kappa \]  \quad \text{curvature} \quad [\text{m}^{-1}]

\[ \mu \]  \quad \text{viscosity} \quad [\text{Pa s}]

\[ \rho \]  \quad \text{density} \quad [\text{kg/m}^3]

\[ \sigma \]  \quad \text{surface tension coefficient} \quad [\text{N/m}]

\[ \phi \]  \quad \text{volume of fluid function} \quad [-]

\[ \psi \]  \quad \text{level-set function} \quad [\text{m}]

Subscripts

\[ i \]  \quad \text{i\text{-th grid point} }
$j$  $j$-th grid point

$k$  $k$-th grid point

$l$  $l$-th component or liquid

$g$  gas