

日本原子力研究開発機構機関リポジトリ  
Japan Atomic Energy Agency Institutional Repository

Title	Development of numerical evaluation method for fluid dynamics effects on jet breakup phenomena in BWR lower plenum
Author(s)	Takayuki Suzuki, Hiroyuki Yoshida, and Fumihisa Nagase
Citation	Journal of Nuclear Science and Technology, 51(7-8), pp.968-976
Text Version	Author Accepted Manuscript
URL	<a href="http://jolissrch-inter.tokai-sc.jaea.go.jp/search/servlet/search?5044701">http://jolissrch-inter.tokai-sc.jaea.go.jp/search/servlet/search?5044701</a>
DOI	<a href="http://dx.doi.org/10.1080/00223131.2014.924447">http://dx.doi.org/10.1080/00223131.2014.924447</a>
Right	This is an Accepted Manuscript of an article published by Taylor & Francis in Journal of Nuclear Science and Technology on 19/06/2014, available online: <a href="http://www.tandfonline.com/10.1080/00223131.2014.924447">http://www.tandfonline.com/10.1080/00223131.2014.924447</a>

## **Development of numerical evaluation method for fluid dynamics effects on jet breakup phenomena in BWR lower plenum**

Takayuki Suzuki<sup>a\*</sup>, Hiroyuki Yoshida<sup>a</sup> and Fumihisa Nagase<sup>a</sup>

<sup>a</sup>*Japan Atomic Energy Agency, 2-4, Shirane, Shirakata, Tokai-mura, Naka-gun, Ibaraki 319-1195, Japan;*

*(Received )*

The jet breakup phenomena of the molten cores during a severe accident are affected by some complicated structures, such as control rod guide tubes, instrument guide tubes and core support plate, in the lower plenum of the boiling water reactors (BWRs). A multi-phase computational fluid dynamics approach combined with experiments is considered to be the best way to estimate the jet breakup phenomena in the BWR lower plenum, and a numerical analysis method has been developed based on the interface tracking method code TPFIT (Two Phase Flow simulation code with Interface Tracking). The developing analysis method was applied to single/multi-channel experiments for verification and validation in this study. Furthermore, results from the numerical analysis were compared to the experimental results obtained using the multi-phase flow visualization technique using a high speed camera and the particle image velocimetry (PIV) method. As a consequence, it is found that the simulation method developed in this study can qualitatively simulate the jet breakup phenomena in the complicated structure.

***Keywords; BWR; lower plenum; jet breakup; molten core; severe accident, two-phase flow; numerical analysis; interface tracking method; TPFIT***

---

\*Corresponding author E-mail:suzuki.takayuki74@jaea.go.jp

## 1. Introduction

To evaluate the accident at the Fukushima Daiichi nuclear power plant (NPP), it is necessary to obtain more information on various phenomena during a severe accident in the reactor pressure vessel (RPV) and primary containment vessel of BWRs. Assuming that the core disruptive accident (CDA) occurred in the Fukushima Daiichi NPP, molten core would have relocated into the coolant of the lower plenum region and finally to the lower head of the RPV. The distance from the reactor core plate to the RPV lower head in the BWR is longer than that in the pressurized water reactors (PWRs). It owns possibility that the molten core has already been broken up and fragmented before piling up on the lower head of the RPV [1-2]. In addition, there are many complicated structures in the BWR lower plenum, such as control rod guide tubes (CRGTs), control rod drive housings, and instrument guide tubes in comparison with the PWRs. And, the molten core behavior, namely jet breakup behavior, is considered to be affected by these complicated structures and these effects on the molten core behavior should be evaluated. The information of the molten core behavior is important to evaluate cooling of debris bed (fragmented, resolidified and piled molten core).

However, it is difficult to evaluate these effects on molten core jet behavior only by experiments, and numerical approach combined with experiments is considered to be the best way for the evaluation. In the jet breakup phenomena, interfaces between molten core jet and coolant change dynamically and complicatedly. The shapes of these interfaces interact with the jet behavior, and an evaluation of the interface behavior is necessary in this numerical approach. To evaluate the interface behavior, it is thought that the multi-fluid computational approach is a possible way to achieve this requirement. Moreover, experimental database including the information of the interface shapes are required to validate the results of the multi-computational approach. Therefore, a numerical analysis method based on the multi-fluid computational fluid dynamics approach should be established and validated by experiments.

In this study, a numerical analysis method based on the multi-fluid computational fluid dynamics approach is developed to predict the flow characteristics of molten core jet, the

changes of interface shapes, the jet breakup length and the size of the fragmented molten core, including the effects of the complicated structures in the lower plenum.

The simulation code is based on the interface tracking method code TPFIT [3], whose detail information is provided in Section 2. The jet breakup behavior in the complicated structures under an isothermal condition was selected as a target of numerical simulation, and the main attention was focused on the detailed interface changing behavior. The experimental data from Saito, et al. [4] was used to validate the simulation results.

## 2. Numerical Simulation Method for Jet Breakup Behavior Based on TPFIT

### 2.1. Outline of TPFIT

The TPFIT was developed by Japan Atomic Energy Agency (JAEA) to simulate detailed two-phase flow behaviors in nuclear systems [3]. Governing equations used in the TPFIT consist of averaged (mixed) mass, momentum and energy conservation equations for compressible fluid and transport equations for the mass of both phases as shown in the follows.

Mass:

$$\frac{D\rho}{Dt} = -\rho \frac{\partial u_i}{\partial x_i} \quad (1)$$

where  $u$  denotes the velocity component,  $x$  denotes the coordinate. Density  $\rho$  is calculated by the following equation using the density and the volume fraction  $f$  of the gas and liquid phases.

$$\rho = \rho_l f_l + \rho_g f_g, \quad f_g = 1 - f_l \quad (2)$$

Momentum:

$$\frac{Du_i}{Dt} = -\frac{1}{\rho} \cdot \frac{\partial p}{\partial x_i} + \frac{1}{\rho} \cdot \frac{\partial \tau_{ij}}{\partial x_j} + g_i + \sigma_i \quad (3)$$

where  $p$  denotes the static pressure,  $\tau$  denotes the shear stress,  $g$  and  $\sigma$  are the accelerations subjected by gravity and surface tension respectively. The surface tension force is evaluated using the continuum surface (CSF) model [5].

Energy:

$$\frac{De}{Dt} = -\frac{p}{\rho} \cdot \frac{\partial u_i}{\partial x_i} + \frac{1}{\rho} \cdot \frac{\partial}{\partial x_i} \left( \lambda \frac{\partial T}{\partial x_i} \right) \quad (4)$$

where  $e$  denotes the internal energy,  $\lambda$  denotes the thermal conductivity,  $T$  denotes the temperature. In the evaluation of the volume fraction, as to improve the precision of the analysis, the mass of both gas and liquid phases are evaluated.

Mass of both phases:

$$\frac{D\rho_m f_m}{Dt} = -\rho_m f_m \frac{\partial u_i}{\partial x_i} \quad (5)$$

where subscript  $m$  denotes the gas or liquid phase. The mass and the volume fraction of liquid or gas are evaluated using the advanced interface tracking method developed by Yoshida, et al. [3]. The cubic-interpolated pseudo particle (CIP) method [6], is applied to the convection terms of other equations. The Incomplete Lower Upper Conjugate Gradient Squared (ILUCGS) method [7], is applied to solve the Poisson equation of pressure.

## **2.2. Outline of Numerical Simulation Method for Jet Breakup Behavior**

As mentioned above, the TPFIT is developed for two-phase flow in the reactor cores and can treat interface behavior between gas and water directly. The TPFIT code has been successfully applied to simulate some flow patterns of two-phase flow, for example, a water jet flow [8]. However, to perform the numerical simulation of the jet breakup behavior in the

reactor core, two liquid components, water and molten core must be treated. Then, we modified the TPFIT to treat two liquid components. Here, average density of the working fluid was defined as follows.

$$\rho = \rho_1 f_1 + \rho_2 f_2, \quad f_2 = 1 - f_1 \quad (6)$$

where subscripts 1 and 2 denote the heavy liquid component (molten core) and the light liquid component (water) respectively.

### 3. Single-Channel Experiment Analysis

#### 3.1. Numerical Conditions of Single-Channel Experiment Analysis

Saito et al.'s, experiments were conducted under the isothermal condition [4]. In these experiments, two different test sections were observed using the multi-phase flow visualization technique. One is a “single-channel experiment” and the other is a “multi-channel experiment”. The single-channel experiment was performed to obtain the basic information of the jet behavior in the complicated structures like the CRGT and check the applicability of the multi-phase visualization technique in such kind flow. Detailed validation data was obtained in the multi-channel experiment, in which structural components in the lower plenum in the BWR were modeled in high precision. In this section, as to check the basic applicability of the above developed TPFIT code, a numerical simulation of the single-channel case in the experiment was performed.

**Figure 1** shows the computational domain of the numerical simulation of the single-channel experiment analysis [4]. The width ( $x$ -direction), depth ( $y$ -direction) and height ( $z$ -direction) were set to be 129.2, 129.2 and 160.0 mm respectively. The computational domain was initially filled with water at room temperature (300 K) and atmospheric pressure (0.1 MPa). An outflow boundary condition was used in the top surface of the computational domain except the nozzle area. At the inlet of the nozzle, the pressure and temperature were set as constant,

and a uniform velocity profile was used as the inlet velocity. On the side and bottom walls, so called “non-slip wall” conditions were applied.

In the numerical domain, there were four cylindrical structures (with 32 mm of outside diameters), which simulated the modeled CRGT in the single-channel experiment. A circular shape flow channel (both the inside diameter and the length equals 10 mm) was set at the center of the top surface to simulate the injection nozzle used in the single-channel experiment. In the numerical simulation, the molten core (Fluorinert (FC-3283)) and water were used as two components of the working liquid (the density of the Fluorinert is larger than water’s). **Table 1** shows the detailed fluid properties of the Fluorinert and the water. The surface tension coefficient at the interface between Fluorinert and water was 0.043 N/m [4]. The inlet velocity was set to 0.8 m/s, same with that in the single channel experiment. The total number of numerical grids was 12,800,000 (=200×200×320). The grid sizes in the  $x$  and  $y$ -direction in the center part ( $-34.6 \text{ mm} < x < 34.6 \text{ mm}$ ,  $-34.6 \text{ mm} < y < 34.6 \text{ mm}$ ) and in the peripheral part of the numerical domain was 0.5 and 1.0 mm respectively. The grid size in the  $z$ -direction was 0.5 mm.

< Figure 1 >

< Table 1 >

### ***3.2. Numerical Results of Single-Channel Experiment Analysis***

As an example of the single-channel analysis, the jet interface in the  $x$ - $z$  plane at different time is shown in **Figure 2**, where Figure 2 (a) is the simulation results and Figure 2 (b) is the experimental results. In Figure 2 (a), an interface was defined as an isosurface of the volume fraction of water ( $f_2$ ) and  $f_2$  is equals to 0.5. In the experimental results, Fluorinert was injected into the water through the circular nozzle, and formed the Fluorinert jet. The jet interface between Fluorinert and water became instable gradually. This instability of the interface was more obvious at the lower part of the test section. Some small Fluorinert the droplet were seen around the instable interface, it was conjectured that the fragmentation of the

Fluorinert jet flow occurred around the instable interface.

In Figure 2 (a), fluctuation of core of the jet was also observed. This tendency was almost same with that in experiment shown in Figure 2 (b). However, the fluctuation width of the jet in the simulation was larger than that measured in experiment. In addition, the interface structure was also larger in the simulation. Moreover, in the experimental results, the injected Fluorinert arrived at the bottom of the test section at  $t=0.175$  s (it means that the arrived time:  $t_a=0.175$  s). In contrast with measured results,  $t_a$  was as long as 0.35 s in the numerical simulation and the tip velocity of the jet was remarkably smaller than the measured results. To solve these problems, further modification of numerical method and numerical conditions were performed in the next section.

< Figure 2 >

### ***3.3. Modification of Numerical Method to Improve Prediction Accuracy***

As proposed in the previous section, the tip velocity of the jet was underestimated by the TPFIT code. It is considered that the tip velocity of the jet has large effects on the jet shape, the fragmented Fluorinert droplet size and the distribution of the fragmented molten core. Hence, this underestimation of the tip velocity of the jet must be improved in this study.

As mentioned above, a larger interface structure formed in the simulation. This tendency was strongly related to the underestimation of the arrival time,  $t_a$ , in the single-channel analysis. In the critical Weber number theory, large interface structures appear in the case of weak shear stress. Therefore, we considered that one of the main causes of the underestimation of the tip velocity of the jet is the overestimation of shear stress in the TPFIT code.

In the current version of the TPFIT, one velocity model is used. In the one velocity model, different fluids own same velocity in one numerical cell (see **Figure 3** (a)). Therefore, in the one velocity model, the velocity difference between water and Fluorinert is ignored in the calculation. As a result, one velocity model may lead to the underestimation of the shear stress between water and Fluorinert. Figure 3 (b) shows velocities at the numerical cell



boundaries in the actual situation. The velocity evaluated by the one velocity model is the average velocity of the water and Fluorinert. Therefore, in the case of Figure 3 (a), transferred volume of water between numerical cells is overestimated and that of Fluorinert is underestimated. It is thought that it caused a decrease of the tip velocity. Consequently, we guess that one velocity model reason of the underestimation of the tip velocity of the jet in the above simulation.

To solve these problems, we improved the TPFIT to calculate the velocity of water and Fluorinert separately instead of using one velocity model. The governing equation for momentum used in the modified TPFIT as follows,

$$\frac{\partial u_{n,i}}{\partial t} + u_{n,j} \frac{\partial u_{n,i}}{\partial x_j} = -\frac{1}{\rho_n} \cdot \frac{\partial p}{\partial x_i} + \frac{1}{\rho_n} \cdot \frac{\partial \tau_{ij}}{\partial x_j} + g_i + \sigma_i \quad (7)$$

where subscript  $n$  denotes flow materials, water or Fluorinert. If an interface is not included in a considered numerical cell  $(i, j, k)$ , shear stress tensor was evaluated as follows,

$$\tau_{xy,i,j,k} = \mu_n \left( \frac{u_{n,i,j+i,k} - u_{n,i,j,k}}{\Delta x} + \frac{v_{n,i+1,j,k} - v_{n,i,j,k}}{\Delta y} \right) \quad (8)$$

In the above equation, for example, an  $x$ - $y$  component of a shear stress tensor was shown. If an interface exists in the objective numerical cell, shear stress tensor was evaluated using Equation (9),

$$\tau_{xy,i,j,k} = \mu_{12} \frac{u_{1,i,j+i,k} - u_{2,i,j,k}}{\Delta x} + \sum_{n=1}^2 \mu_n \frac{v_{n,i+1,j,k} - v_{n,i,j,k}}{\Delta y} \quad (9)$$

For simplicity, it is assumed that located an interface in the cell and perpendiculars to the  $y$ -axis.

Here, fluid 1 and 2 are separately in a plus side and a minus side of  $y$ -axis.  $\Delta x$  and  $\Delta y$  are the size of objective grid size in the  $x$  and  $y$  direction respectively. In the equation,  $\mu_{12}$  is the effective viscosity in the considered numerical cell,

$$\mu_{12} = \frac{\mu_1 \cdot \mu_2}{\mu_1 f_1 + \mu_2 (1 - f_1)} \quad (10)$$

where  $f_1$  is a volume fraction of fluid 1 in the considered numerical cell. And  $\mu_1$  and  $\mu_2$  are the viscosity of fluid 1 and 2 respectively. By use of Equation (7), we can treat the velocity of water and Fluorinert separately, and the problems related to the one velocity model can be avoided.

According to the existing knowledge related to the jet, the shape of the inlet nozzle and the flow conditions of the nozzle have large effects on the behavior of the jet. Then, we considered that the other main causes were inappropriate modeling of the inlet nozzle. In the experience, flow area was suddenly decreased near the nozzle outlet (see **Figure 4**). Then, velocity profile at the outlet of the nozzle was almost uniform. In the numerical simulation, resolution in the nozzle was not sufficient to simulate this situation, and uniform velocity profile at the nozzle outlet was not obtained in the numerical simulation. Therefore, this part must be improved and the uniform velocity distribution at the outlet of the nozzle must be got in the simulation.

< Figure 3 >

< Figure 4 >

### ***3.4. Numerical Results of Modified Method for Single-Channel Experiment Analysis***

A numerical simulation of the single-channel experiment analysis using the modified TPFIT codes was performed again to check the applicability of the modified TPFIT. And, to solve problems related to the inlet nozzle, a simplified nozzle model (uniform inlet velocity) was used. The analysis domain and the boundary conditions of this simulation were the same

as that of numerical simulation performed in Section 3.1 except the boundary condition of the nozzle outlet. The objective of this simulation was to validate the improvement of the modified TPFIT and the boundary condition of the nozzle outlet.

A simulated jet images in  $x$ - $z$  plane were shown in **Figure 5**. In Figure 5, the measured results were also shown (same as Figure 2). Interface structure predicted using the modified TPFIT was smaller than that predicted by the original TPFIT. The simulated arrival time was around 0.175 s and almost the same as the measured results. **Figure 6** shows simulated and measured tip position of the jet. As explained in Section 3.2, the simulated tip position without modification was underestimated. In contrast with this result, the simulated tip position with modification was almost the same as the measured data. As a result, it was confirmed that the underestimation of the tip velocity of the jet was improved by using the modified TPFIT and the boundary condition of the nozzle outlet.

< Figure 5 >

< Figure 6 >

## **4. Multi-Channel Experiment Analysis by Improved TPFIT**

### ***4.1. Numerical Conditions of Multi-Channel Experiment Analysis***

We performed the “multi-channel experiment” analysis that simulate the BWR lower plenum by using improved TPFIT. **Figure 7** shows the computational domain. It simulated the center part of the multi-channel experiment performed by Saito, et al [4]. The width ( $x$ -direction), depth ( $y$ -direction) and height ( $z$ -direction) was 181.2, 64.4 and 505.0 mm respectively. The computational domain was initially filled with water at room temperature (300.0 K) and atmospheric pressure (0.1 MPa). The outlet boundary condition was applied at the top surface of the computational domain. At the  $x$ -direction and  $y$ -direction surfaces, outlet boundary and non-slip wall conditions were applied respectively. At the outlet of the nozzle, the pressure and the temperature were fixed as constant based on results of Section 3.

In this numerical domain, there were twelve cylindrical structures (outer diameter of

each cylindrical structure is 27 mm in the upper part and 15 mm in the lower part of the cylindrical structure) which simulated the control rod guide tube in the multi-channel experiment. Here, a flow nozzle (which has a circular shape and corresponds to the diameter of a nozzle of 7.0 mm) was set at the center of the top surface to simulate the inlet nozzle used in the multi-channel experiment. As same as the numerical simulation performed in section 3, Fluorinert (FC-3283) and water were used as two components of the working fluid. The inlet velocity (i.e., velocity of the nozzle outlet) was adjusted based on the experimental information and set to 2.1 m/s. The velocity profile at the inlet is uniform. The total number of numerical grids was 46,080,000 ( $[x, y, z] = 360 \times 128 \times 1000$ ). Grid sizes of each direction were about 0.5 mm.

< Figure 7 >

#### ***4.2. Numerical Results of Multi-Channel Experiment Analysis***

**Figure 8** shows simulated and measured tip positions of the jet. In the results, the simulated tip positions by using the modified TPFIT were almost the same as the measured data. Time series of visualized jet images in  $x$ - $z$  plane were shown in **Figure 9**. In these images, as same as those in Figure 2 and Figure 5, the interface was defined as an isosurface that the volume fraction of water is equal to 0.5. In the measured results (see Figure 9 (b)), interfacial instability occurred as same as the single-channel experiment analysis. Furthermore, fragmentation of Fluorinert was observed in both simulated and measured results. Fragmented Fluorinert moved into the gap between the structures. Afterwards, fragmented Fluorinert moved to adjacent channels. These tendencies observed in the measured results were also reproduced by the numerical simulation (see Figure 9 (a)).

To evaluate performance of the modified TPFIT, predicted interface shapes at an early stage of the injection were compared with the measured results in **Figure 10**. Same with the measured results, so called the mushroom shape of the interface was observed. Interface shape predicted by the numerical simulation was almost the same as the measured results.

Comparison of numerical results and experimental results of the velocity distribution around the interface are shown in **Figure 11**. The measured result of Figure 11 (b) was obtained by PIV method. In the measured results, vortexes around convex shape interfaces of Fluorinert were observed and similar velocity distributions were reproduced in the numerical results. These vortexes around convex shape interfaces have an important role on fragmentation of Fluorinert. Therefore, it was concluded that the modified TPFIT developed in this study can be successfully used to predict the jet breakup phenomena in the complicated structure under the isothermal condition qualitatively.

< Figure 8 >

< Figure 9 >

< Figure 10 >

< Figure 11 >

## 5. Conclusion

We developed a numerical simulation method for the jet breakup behavior in complicated structures in the lower plenum of the BWR based on a detailed two-phase flow analysis code TPFIT. To validate the applicability of the numerical simulation method, the method was applied to simulate the single-channel experiment with modeled complicated structures using the Saito et al.'s experimental results as validation data. From the comparison results, between numerical data and experimental data it was found that the numerical results are mostly in agreement with the measured results qualitatively. However, the predicted tip velocity of the injected simulant fluid was smaller than that of the measured results.

To solve the underestimation of the tip velocity in numerical simulation, we improved the calculation method of the velocity of the both fluids. By use of the modified TPFIT, it was confirmed that the underestimation of the tip velocity was improved. Moreover, we performed the multi-channel experiment analysis that simulated BWR lower plenum using the modified TPFIT. Furthermore, we compared the interface shape of the jet and the velocity distribution

around the interface with the experimental results. As a result, the analytical method developed in this study can effectively be used to simulate the jet breakup phenomena in the complicated structure qualitatively.

### **Nomenclature**

$e$	Internal energy [J/kg]
$f$	Volume fraction of fluid [-]
$g$	Gravitational acceleration [ $\text{m/s}^2$ ]
$p$	Static pressure [Pa]
$q$	Internal heat value per unit volume [ $\text{W/m}^3$ ]
$t$	Time [s]
$T$	Temperature [K]
$u$	Velocity components [m/s]
$x, y, z$	Coordinate [m]

### Greek letters

$\mu$	Viscosity [Pa s]
$\rho$	Density [ $\text{kg/m}^3$ ]
$\sigma$	Surface tension [N/m]
$\tau$	Shear stress [Pa]
$\lambda$	Thermal conductivity [W/K]

### *Subscripts*

$g$	gas phase
$i, j, k$	$i, j$ and $k$ direction
$l$	liquid phase
$m$	gas or liquid phase

$n$  fluid 1 or 2

### **Acknowledgements**

A part of the present study was conducted as a part of a program sponsored and organized by the Agency for Natural and Energy, Ministry of Economy, Trade and Industry. This research was conducted using a supercomputer of Japan Atomic Energy Agency.

### **Reference**

- [1] Imai S, Sato K, Hamazaki R, Henry RE. Experimental study on in-vessel cooling mechanisms. In: Proceedings of 7th International Conference on Nuclear Engineering; 1999 April 19-23; Tokyo, Japan.
- [2] Spencer BW, Wang SK, Blomquist CA, McUmbler LM, Schneider JP. Fragmentation and quench behavior of corium melt streams in water. (IL): Argonne National Laboratory; 1994. Technical Report (NUREG/CR-6133)
- [3] Yoshida H, et al. Investigation of water-vapor two-phase flow characteristics in a tight-lattice core by large-scale numerical simulation, (1); development of a direct analysis procedure on two-phase flow with an advanced interface tracking method. Nucl Eng Technol. 2004;3(3):233-241
- [4] Saito R, Abe Y, Yoshida H. Experimental study on breakup and fragmentation behavior of molten material jet in complicated structure of BWR lower plenum. Journal of Nuclear Science and Technology. 2014;51(1), 64-76
- [5] Brackbill JU, et al. A continuum method for modeling surface-tension. Journal of computational physics. 1992;100, 335-354
- [6] Yabe T, Aoki T. A universal solver for hyperbolic equations by cubic-polynomial interpolation 1; One-dimensional solver. Computer physics Communications. 1991;66,219-232
- [7] Oguni T. Matrix calculation software. Maruzen. 1991;336

- [8] Yoshida H, Nagayoshi T, Zhang W, Takase K. Numerical Simulation of Fluid Mixing Phenomena in Boiling Water Reactor Core Using Advanced Interface-Tracking Method. Transactions of the Japan Society of Mechanical Engineers Series B. 2008;742:1278-1286

### **Figure captions**

- Figure 1. Computational domain of single-channel analysis
- Figure 2. Comparison of analytical results and experimental results (single-channel)
- Figure 3. Definition of velocity at numerical cell boundary
- Figure 4. Shape of inlet nozzle in experiment
- Figure 5. Comparison of improved numerical results and experimental results (single-channel)
- Figure 6. Tip position of jet from nozzle (single-channel)
- Figure 7. Computational domain of multi-channel experiment analysis
- Figure 8. Tip position of jet from nozzle (multi-channel)
- Figure 9. Comparison of numerical results and experimental results (multi-channel)
- Figure 10. Detail of jet shape at an early stage of injection
- Figure 11. Comparison of numerical results and experimental results of the velocity distribution around the interface

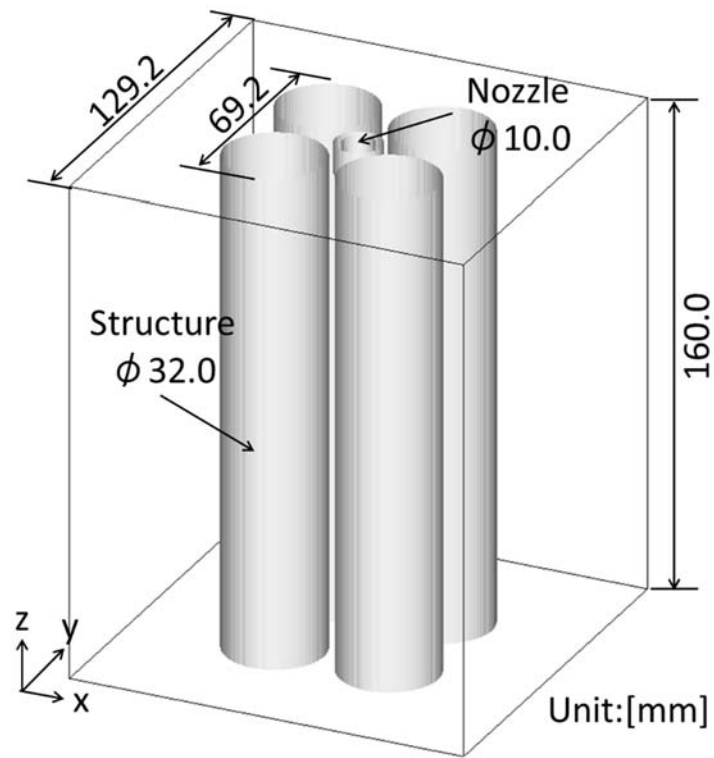
### **Table captions**

- Table 1. Fluid properties of Fluorinert and water

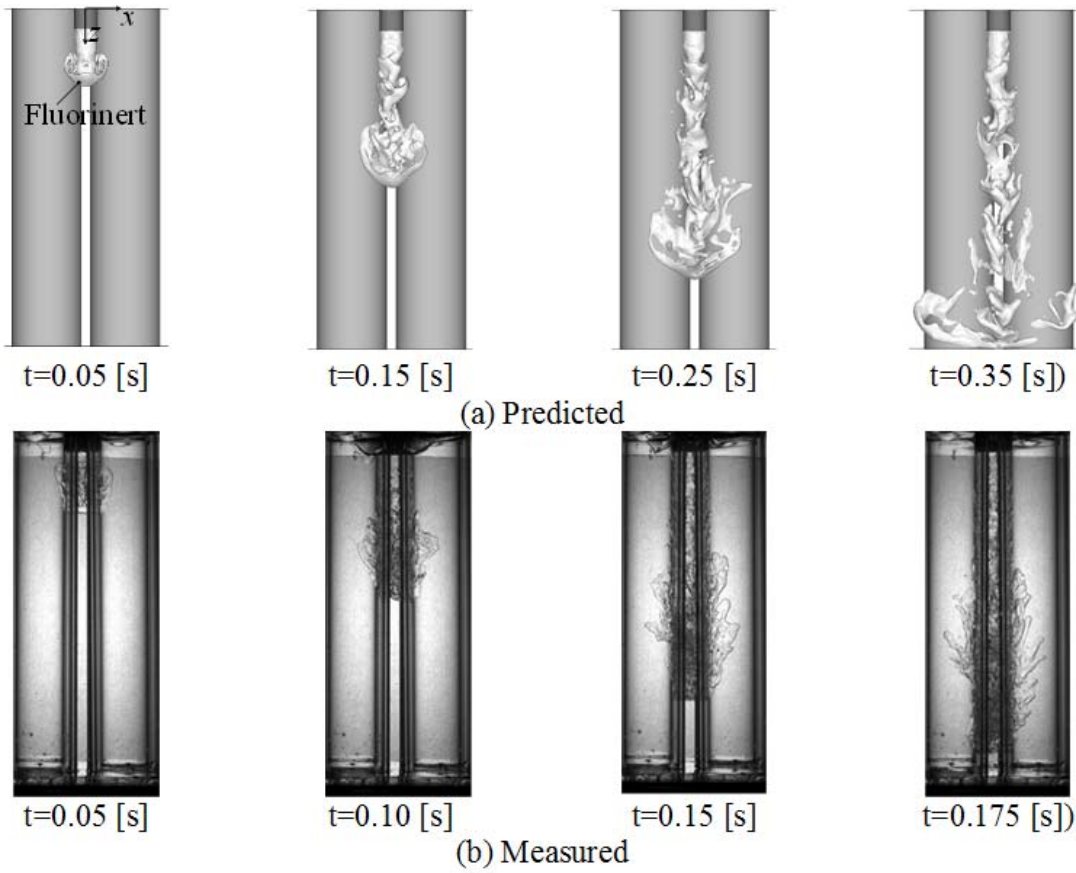


Table 1. Fluid properties of Fluorinert and water

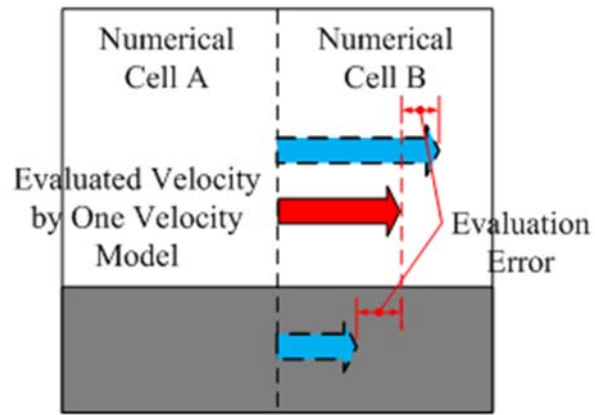
	Fluorinert(FC-3283)	Water
Density [kg/m <sup>3</sup> ]	1830	998
Surface tension [N/m]	0.043(underwater)[4]	
Dynamic viscosity [m <sup>2</sup> /s]	0.82×10 <sup>-6</sup>	1.00×10 <sup>-6</sup>
Viscosity [Pa·s]	1.50×10 <sup>-3</sup>	1.00×10 <sup>-3</sup>



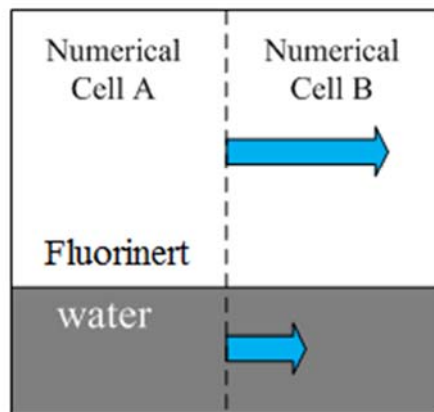
**Figure 1.** Computational domain of single-channel experiment analysis



**Figure 2.** Comparison of analytical results and experimental results (single-channel)



(a) One velocity model

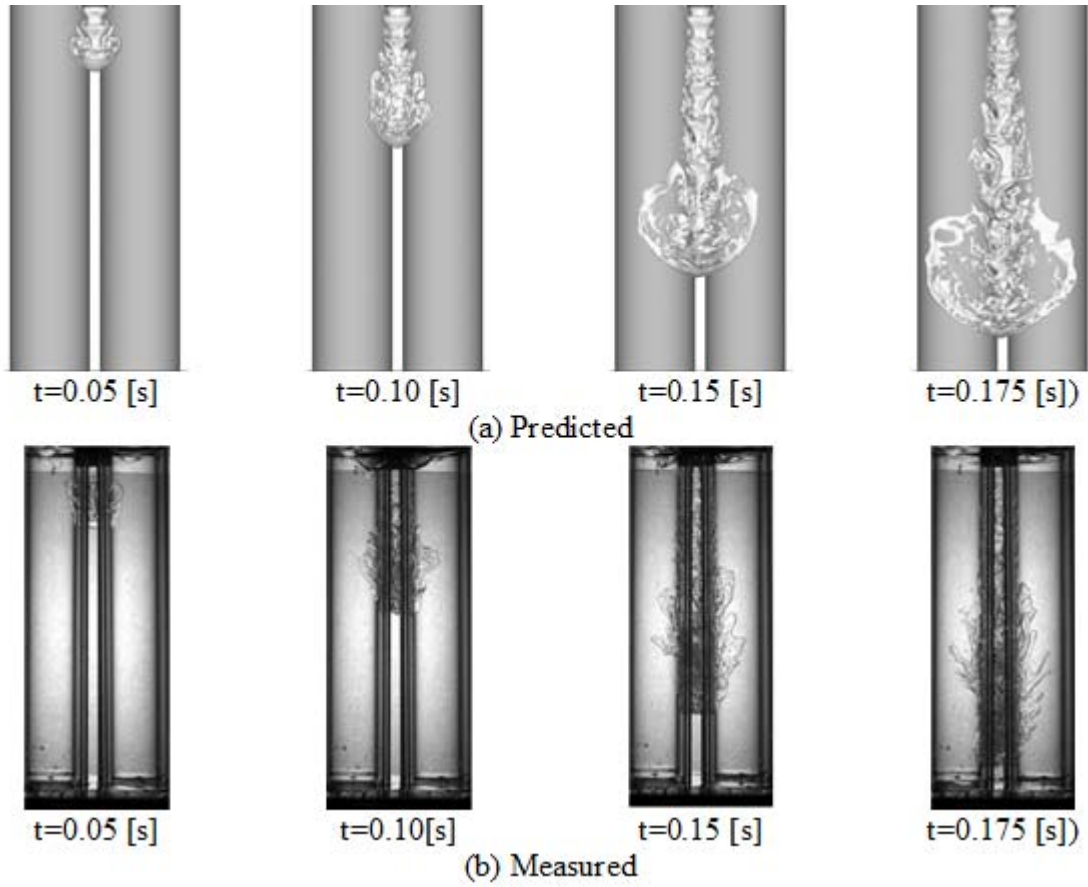


(b) Actual velocity

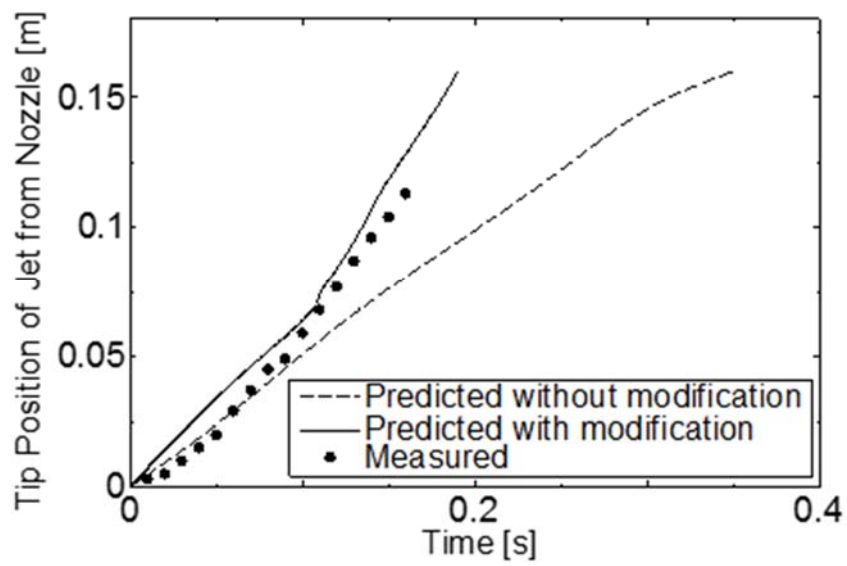
**Figure 3.** Definition of velocity at numerical cell boundary



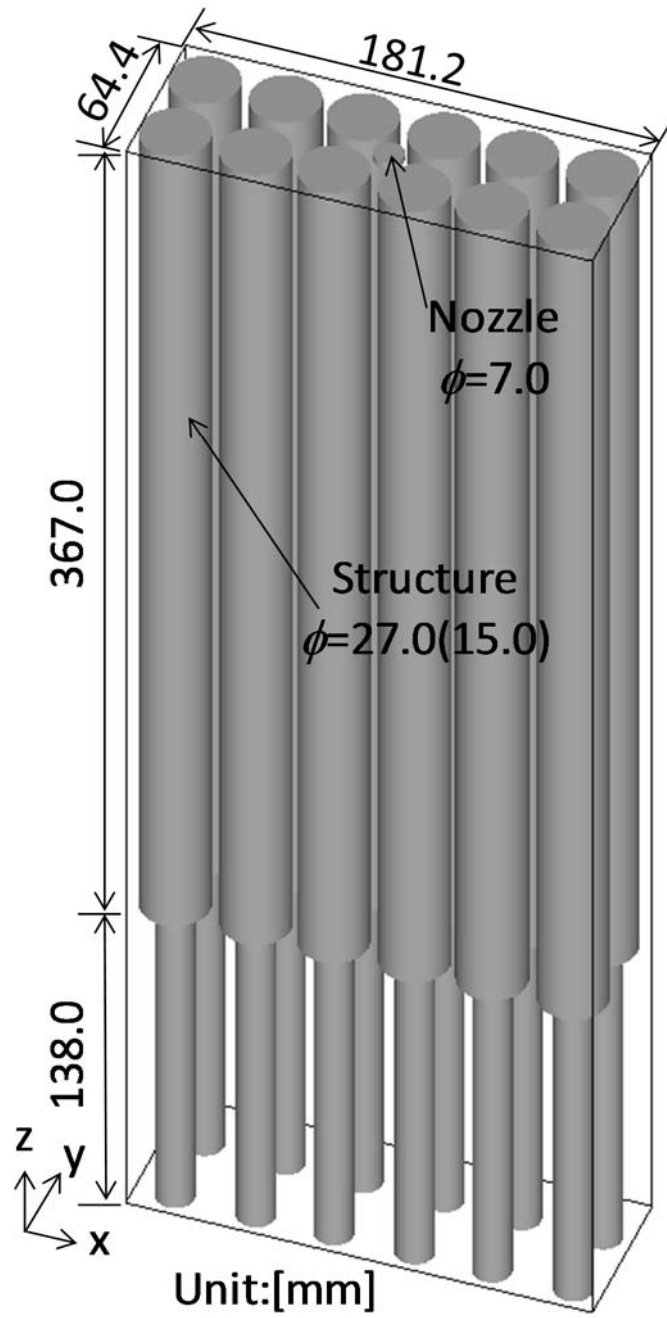
**Figure 4.** Shape of inlet nozzle in experiment



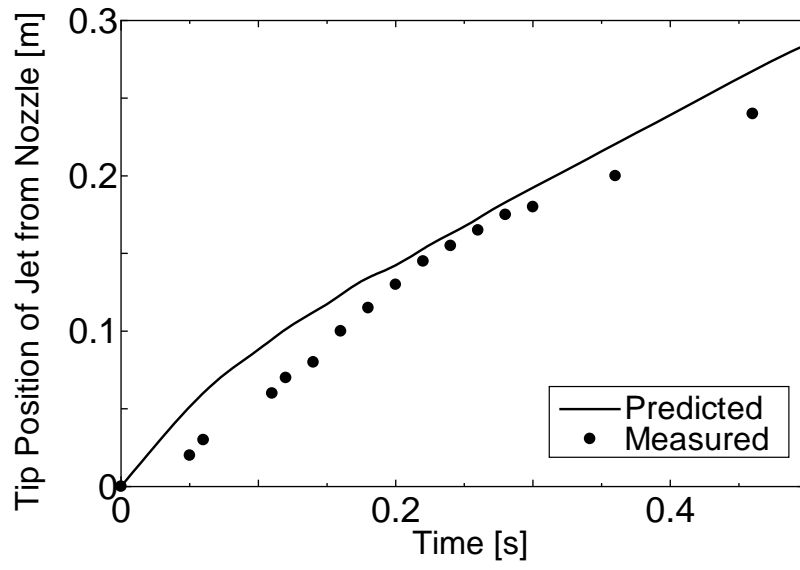
**Figure 5.** Comparison of improved numerical results and experimental results (single-channel)



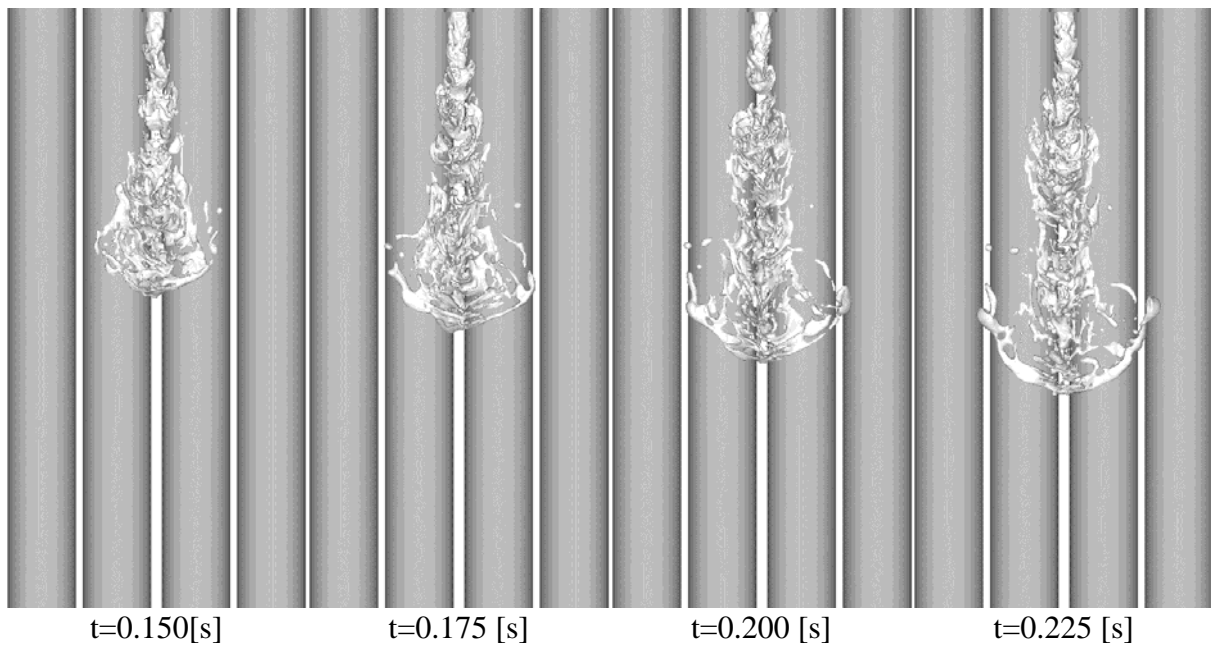
**Figure 6.** Tip position of jet from nozzle (single-channel)



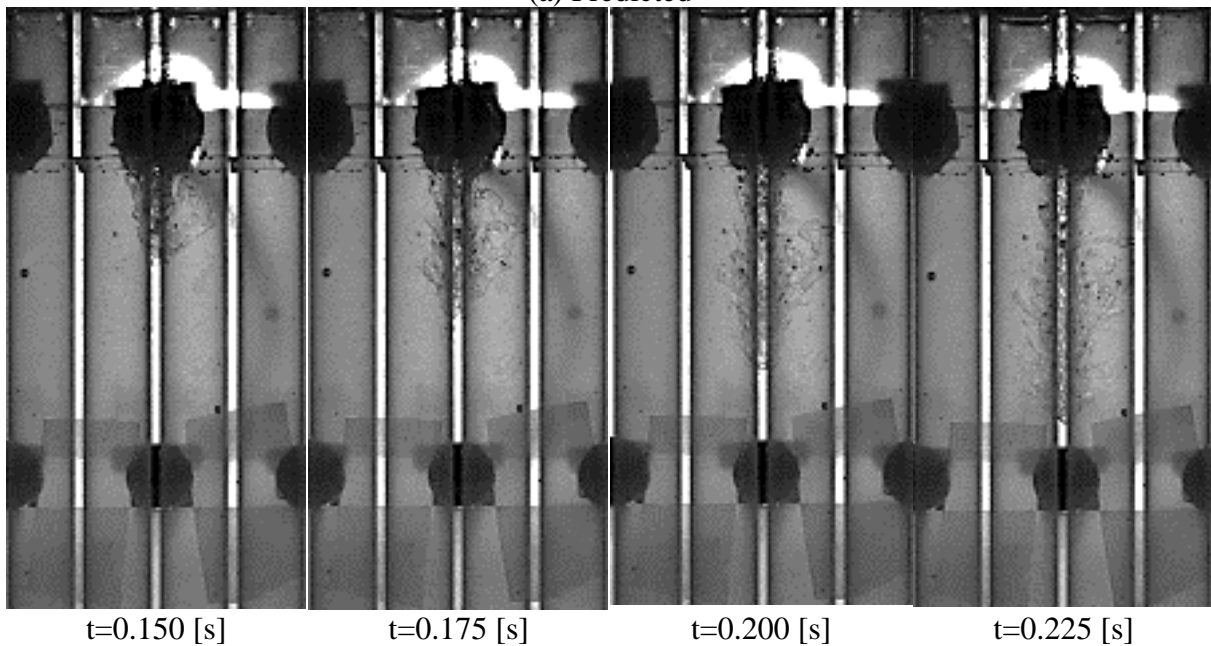
**Figure 7.** Computational domain of multi-channel experiment analysis



**Figure 8.** Tip position of jet from nozzle (multi-channel)

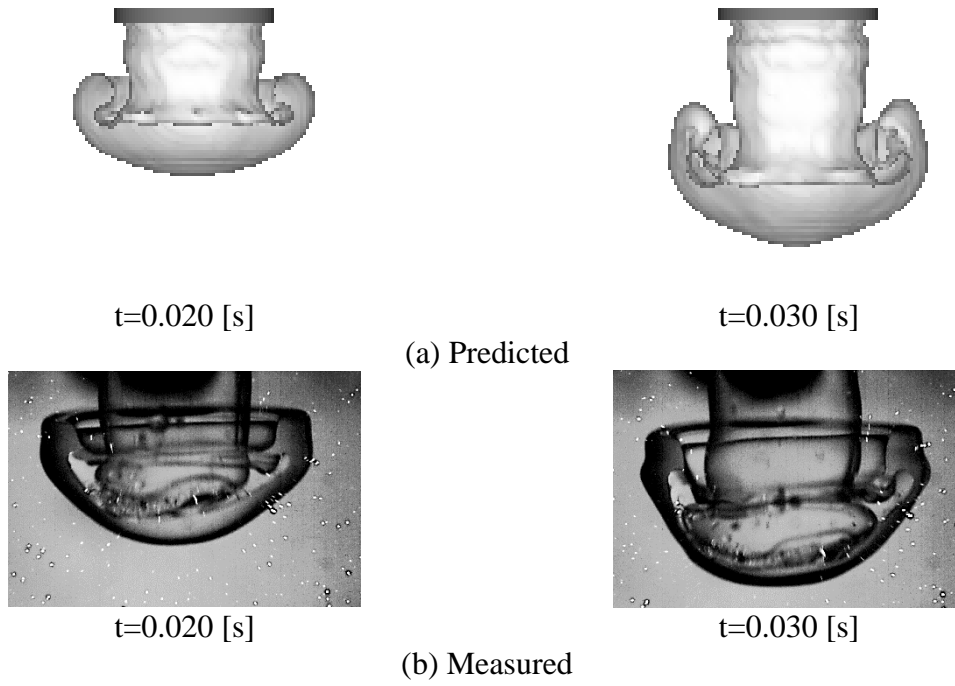


(a) Predicted

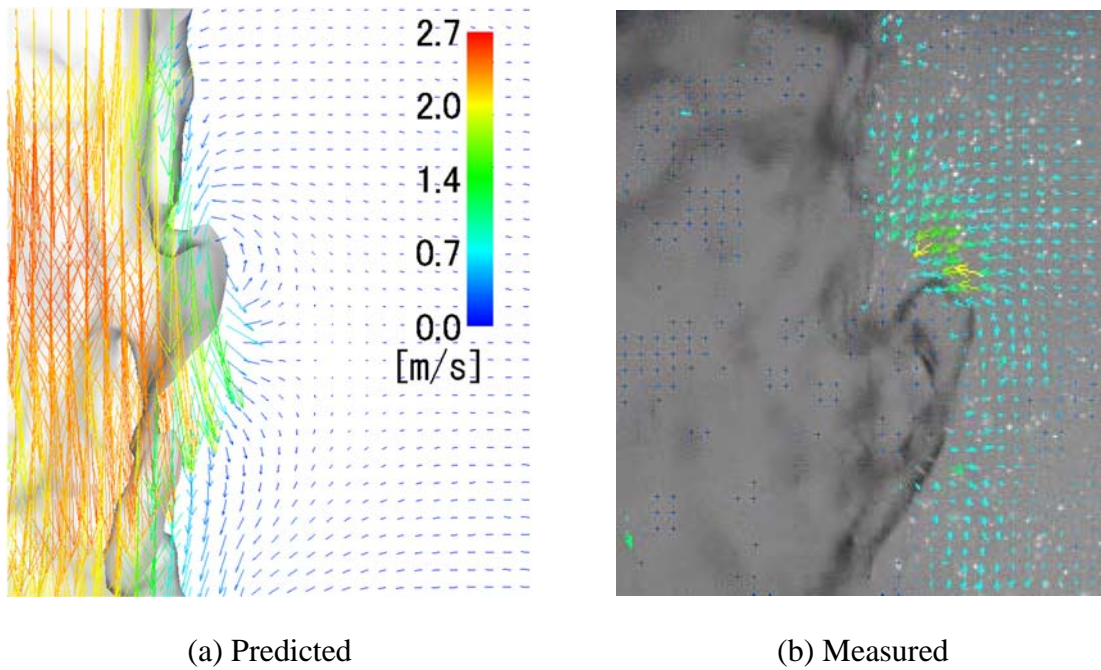


(b) Measured

**Figure 9.** Comparison of numerical results and experimental results (multi-channel)



**Figure 10.** Detail of jet shape at an early stage of injection



**Figure 11.** Comparison of numerical results and experimental results of the velocity distribution around the interface

T. Suzuki

Development of numerical evaluation method for fluid dynamics effects on jet breakup phenomena in BWR lower plenum