We present a coupled interface capturing and multi-fluid model (CIM) method for computing fuel injection behavior from a nozzle internal flow to a liquid jet seamlessly. This method is a kind of CFD algorithm for multi-phase and multi-scale flow simulation. In a computational cell, a relatively large-scale gas/liquid interface captured by the computational mesh is handled by the PLIC-VOF method, and small-scale dispersed gas bubbles and liquid droplets are handled by the four-fluid model. The MF-ICE method is employed for coupling of pressure and velocity, where the compressibility of the gas phase is taken into account. A cavitation model and the CSF model are implemented into the four-fluid model and the VOF four-fluid model. The MF-ICE method is employed for coupling of pressure and velocity, where the compressibility of the gas phase is handled by the PLIC-VOF method, and small-scale dispersed gas bubbles and liquid droplets are handled by the multi-scale flow simulation. In a computational cell, a relatively large-scale gas/liquid interface captured by the computational mesh is handled by the PLIC-VOF method, and small-scale dispersed gas bubbles and liquid droplets are handled by the multi-scale flow simulation. In the cavitation number; in this work the cavitation number is defined by the ratio of the pressure difference between the ambient pressure and the saturated vapor pressure. The nozzle internal flow grows from a cavitating bubble flow to a hydraulic flip [1] when the cavitation number increases to some degree. The range of the bubble size in the nozzle orifice is from several microns to tens of microns. When the nozzle internal flow becomes the hydraulic flip, the scale of the gas/liquid interface becomes much larger than the bubble flow. In the disintegration process from a liquid column to spray, multi-scale gas/liquid interfaces also exist together whose scales are between several microns and the nozzle orifice diameter. Moreover the nozzle internal flow affects the disintegration of the liquid jet and characteristics of the subsequent spray. Therefore a multi-scale CFD modeling is necessary for computing a nozzle internal flow and a liquid jet seamlessly.

Most multi-phase CFD models are classified into two types according to the ratio of the interface size scale to the computational mesh size scale. A method with the computational mesh size which is larger than the interface size scale is regarded as a macroscopic scale calculation. The averaging techniques (the Eulerian multi-fluid model [8], the homogeneous equilibrium model (HEM) [9,10], the drift flux model [11] etc.) and the lagrangian particle tracking methods (the bubble tracking method [12], the discrete droplet model (DDM) [13] etc.) belong to this group. Each method has a limit of interface size to be properly handled with against its mesh size. Therefore, when using those method the appropriate model has to be selected depending on the specific size scale of the gas/liquid interface on their request. In order to solve this limitation, hybrid approaches have been proposed by Černe et al. [14] and Han and Alajbegovic [15] (switching between the VOF method and the two-fluid model at each cell) and also by Tomiyama et al. [16] (combination of the VOF method, the multi-fluid model and the bubble tracking method).

In this study, we present another new hybrid method; a coupled interface capturing and multi-fluid model (CIM) method to compute the coexisting large- and small-scale interfaces in a computational cell simultaneously. In the following sections, the computational method of the CIM is described, and results of typical multiphase flow problems are validated using this method, then the conclusions are summarized.

2. COMPUTATIONAL METHOD
2.1 Distribution of Liquid and Gas in a Cell
In a computational cell, the distribution of the liquid phase and gas phase is classified into continuous phase and dispersed phase. Therefore, four types of fluids are required for modeling as shown in Fig. 1; Fluid 1 for continuous phase of liquid, Fluid 2 for dispersed phase of gas, Fluid 3 for continuous phase of gas, and Fluid 4 for dispersed phase of gas. The gas/liquid interface between Fluid 1 and Fluid 3 is the large-scale interface to be captured by the PLIC-VOF method [17].

\[ \frac{\partial}{\partial t} \left( \theta_1 \rho_1 u_{1,j} \right) + \frac{\partial}{\partial x_j} \left( \theta_1 \rho_1 u_{1,j} u_{1,j} \right) = \Gamma_{1,j} \]

where \( \Gamma_{1,j} \) is the cell volume at \( j \)-th time-step, \( \theta_1 \) is the face-centered value and \( \hat{\theta}_1 \) denotes the velocity after momentum exchange. The momentum equation for phase \( k \):

\[ \frac{\partial}{\partial t} \left( \theta_k \rho_k u_{k,j} \right) + \frac{\partial}{\partial x_j} \left( \theta_k \rho_k u_{k,j} u_{k,j} \right) = -\theta_k \frac{\partial p}{\partial x_j} + \tau_{\theta\theta} \frac{\partial u_{k,j}}{\partial x_j} + \Gamma_{k,j} + M_{k,j} + E_{k,j} \]

The momentum exchange term \( M_{k,j} \) takes into account drag force and turbulent dispersion force. The momentum exchange term, for example, between phase 1 and phase 2:

\[ M_{12,j} = C_D \frac{1}{8} \rho_1 A^2 \frac{j}{2} \left[ \left| u_{1,j} - u_{2,j} \right| - \frac{1}{3} \theta_1 \right] \]

\[ + C_D \rho_1 k_{\theta} \frac{\partial \theta_1}{\partial x_j} = -M_{21,j} \]

where \( C_D \) is the drag coefficient, \( A^2 \) is the interfacial area density and \( C_{TD} \) is the turbulent dispersion coefficient. The drag force formulation is based on the drag law of a single sphere [20]. The turbulent dispersion force accounts for the bubble dispersion due to turbulent mixing processes. The bubble induced turbulent viscosity [21], which is caused by the momentum interaction at the interface, is added to the viscosity term.

The energy equation for phase \( k \):

\[ \frac{\partial}{\partial t} \left( \theta_k \rho_k e_{k,j} \right) + \frac{\partial}{\partial x_j} \left( \theta_k \rho_k u_{k,j} e_{k,j} \right) = -\theta_k P \frac{\partial u_{k,j}}{\partial x_j} \]

\[ + \frac{\partial}{\partial x_j} \left( \theta_k \lambda_k \frac{\partial T_k}{\partial x_j} \right) + \theta_k \Phi_k + e_1 \Gamma_{1,j} + E_{1,j} \]

where the energy exchange term \( E_{1,j} \) formulation between continuous phase and dispersed phase is based on the heat exchange of a single sphere.

It is assumed that the pressure field is common to all phases. A cell-centered ICE method [22], which is extended ICE method to treat multi-phase flow field (MF-ICE), is used for coupling of the pressure and the phase velocity, where the compressibility of the gas phase is taken into account. Semi-discretized pressure equation [22]:

\[ V^* \sum_{i=1}^{4} \frac{\theta_i v_i}{c_i^2} - \Delta t \nabla \cdot \left( \sum_{i=1}^{4} \langle \theta_i \rangle^v \nabla \rho_i \right) \Delta x \]

\[ = \sum_{i=1}^{4} v_i \Delta m_i - \Delta t \nabla \cdot \left( \sum_{i=1}^{4} \langle \theta_i \rangle^v \right) \nabla \rho_i \]

where \( V^* \) is the cell volume at \( n \)-th time-step, \( \nabla \) is the specific volume, \( c \) is the speed of sound, \( \Delta t \) is the time increment and \( m \) is the mass in a cell volume. The operator \( \nabla \cdot \) represents the cell-centered gradient, \( \nabla \rho_i \) represents the face-centered gradient, the quantity in \( \langle \theta_i \rangle^v \) is an upwinded value, the quantity with superscript \( f \) denotes the face-centered value and \( \hat{u} \) denotes the velocity after momentum exchange.

2.2 Multi-fluid Model
An Eulerian multi-fluid model [18, 19] is employed for the governing equations in the CIM method where four fluids are necessary as mentioned above. In the multi-fluid approach, the model equations are obtained through the ensemble averaging process [8]. Mass, momentum, energy conservation equations, and turbulence kinetic energy and its dissipation rate equations are solved separately for each phase. The governing equations for phase \( k \) are characterized by the volume fraction \( \theta_k \) and additional inter-phase exchange terms, which take into account the interaction interface between two phases.

The mass conservation equation for phase \( k \):

\[ \frac{\partial}{\partial t} \left( \theta_k \rho_k \right) + \frac{\partial}{\partial x_j} \left( \theta_k \rho_k u_{k,j} \right) = 0 \]

The energy equation for phase \( k \):

\[ \frac{\partial}{\partial t} \left( \theta_k \rho_k c_{\theta,k} \right) + \frac{\partial}{\partial x_j} \left( \theta_k \rho_k c_{\theta,k} u_{k,j} \right) = -\theta_k \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \theta_k \lambda_k \frac{\partial \theta_k}{\partial x_j} \right) + \theta_k \Phi_k + E_{\theta,k} \]

The momentum exchange term \( E_{\theta,k} \) takes into account drag force and turbulent dispersion force. The momentum equation for phase \( k \):

\[ \frac{\partial}{\partial t} \left( \theta_k \rho_k u_{k,j} \right) + \frac{\partial}{\partial x_j} \left( \theta_k \rho_k u_{k,j} u_{k,j} \right) = -\theta_k \frac{\partial p}{\partial x_j} + \tau_{\theta\theta} \frac{\partial u_{k,j}}{\partial x_j} + \Gamma_{k,j} + M_{k,j} + E_{k,j} \]

The momentum exchange term \( M_{k,j} \) takes into account drag force and turbulent dispersion force. The momentum exchange term, for example, between phase 1 and phase 2:

\[ M_{12,j} = C_D \frac{1}{8} \rho_1 A^2 \frac{j}{2} \left[ \left| u_{1,j} - u_{2,j} \right| - \frac{1}{3} \theta_1 \right] \]

\[ + C_D \rho_1 k_{\theta} \frac{\partial \theta_1}{\partial x_j} = -M_{21,j} \]

where \( C_D \) is the drag coefficient, \( A^2 \) is the interfacial area density and \( C_{TD} \) is the turbulent dispersion coefficient. The drag force formulation is based on the drag law of a single sphere [20]. The turbulent dispersion force accounts for the bubble dispersion due to turbulent mixing processes. The bubble induced turbulent viscosity [21], which is caused by the momentum interaction at the interface, is added to the viscosity term.

The energy equation for phase \( k \):

\[ \frac{\partial}{\partial t} \left( \theta_k \rho_k c_{\theta,k} \right) + \frac{\partial}{\partial x_j} \left( \theta_k \rho_k c_{\theta,k} u_{k,j} \right) = -\theta_k \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \theta_k \lambda_k \frac{\partial \theta_k}{\partial x_j} \right) + \theta_k \Phi_k + E_{\theta,k} \]

where the energy exchange term \( E_{\theta,k} \) formulation between continuous phase and dispersed phase is based on the heat exchange of a single sphere.

It is assumed that the pressure field is common to all phases. A cell-centered ICE method [22], which is extended ICE method to treat multi-phase flow field (MF-ICE), is used for coupling of the pressure and the phase velocity, where the compressibility of the gas phase is taken into account. Semi-discretized pressure equation [22]:

\[ V^* \sum_{i=1}^{4} \frac{\theta_i v_i}{c_i^2} - \Delta t \nabla \cdot \left( \sum_{i=1}^{4} \langle \theta_i \rangle^v \nabla \rho_i \right) \Delta x \]

\[ = \sum_{i=1}^{4} v_i \Delta m_i - \Delta t \nabla \cdot \left( \sum_{i=1}^{4} \langle \theta_i \rangle^v \right) \nabla \rho_i \]

where \( V^* \) is the cell volume at \( n \)-th time-step, \( \nabla \) is the specific volume, \( c \) is the speed of sound, \( \Delta t \) is the time increment and \( m \) is the mass in a cell volume. The operator \( \nabla \cdot \) represents the cell-centered gradient, \( \nabla \rho_i \) represents the face-centered gradient, the quantity in \( \langle \theta_i \rangle^v \) is an upwinded value, the quantity with superscript \( f \) denotes the face-centered value and \( \hat{u} \) denotes the velocity after momentum exchange.

2.3 Reconstructing the Interface
The large-scale gas/liquid interface between Fluid 1 and Fluid 3 is immiscible, that is neither diffused nor dispersed (even for ensemble averaged interface), so that the PLIC-VOF method is adopted. We introduce the volume fraction \( \alpha \) defined by the sum of the volume fractions of Fluid 1 and Fluid 2 for the VOF method:

\[ \alpha = \theta_1 + \theta_2 \]

Following Rider and Kothe's method [17], the interfaces are reconstructed every time step by geometrical calculation. The interface in a cell is approximated by a linear segment as shown in Fig. 2. In order to determine the unique location of the segment, an interface normal vector \( \mathbf{n} \) and a distance \( d \) from the cell-center is necessary. In this study the interface normal vector is determined from the gradient of the volume fraction.
The distance $d$ is constrained by volume conservation and uniquely determined when the interface normal vector $\mathbf{n}_\rho$ and the volume fraction $\alpha$ have been given, such that the volume fraction of the polygon, which is the part of the cell truncated by the segment, is equal to the volume fraction $\alpha$. The shape of the polygon, which is formed by the interface segment truncating the cell, changes between triangle, quadrangle and pentagon according to the distance. Therefore, to determine the distance value, an iterative method is suitable rather than a direct computation. We employed the Brent’s iterative method [23], in which the solution is usually converged within several iterations.

The distance $d$ is uniquely determined as just described above whereas the normal vector can be arbitrarily determined. Moreover the normal vector affects the accuracy of the constructed shape of the interface. A number of algorithms proposed for computing the normal vector were examined by Rider and Kothe [17] and also by Scardovelli and Zaleski [24]. They assessed reconstruction errors of those algorithms by applying them to known interface geometries. According to them, the spatial accuracy of the reconstructed interface shape by the normal vector calculated by Eq. (7) converges in approximately first-order as the grid is refined. Several algorithms with correction of the normal vector, such as a least square fit and an error minimization method, showed approximately second-order spatial accuracy. In this study we employed the first-order accurate interface reconstruction because the reconstruction cost of the second-order techniques are much more expensive especially in irregular mesh.

\[
\rho = \frac{-\nabla \alpha}{|\nabla \alpha|} \quad (7)
\]

Fig. 2 Interface approximation by PLIC reconstruction

2.4 Advection Algorithm

In order to prevent numerical diffusion of the phase volume fraction and to capture the interface sharply, face flux volume at each cell face is calculated geometrically for all phases. The advection of x-direction and y-direction is calculated all at once by using an unsplit multi-dimensional method [17]. In addition the geometrical face flux evaluation is carried out not only for the continuous phases but for the dispersed phases, because the distribution of the dispersed phase is strongly connected with the corresponding continuous phase (i.e. Fluid 2 is distributed within Fluid 1, and Fluid 4 is distributed within Fluid 3). A schematic diagram of the advection calculation method is illustrated in Fig. 3. An uniform flow $\mathbf{u}_\rho$ to upper right direction is assumed as a simple example case in the figure. The distribution of each fluid (Fluid 1 to 4) has been already determined by the interface reconstruction algorithm before advection calculation.

The geometrical advection calculation for each phase requires the following steps:

1. A polygon of the flux volume passing through the cell face swept by the phase velocity $\mathbf{u}_\rho$ during the time step is constructed.
2. The flux polygon is clipped by the large-scale interface (between Fluid 1 and Fluid 3) by using a polygon clipping technique. Therefore the flux polygon is divided into two parts (“Fluid 1+2 side” and “Fluid 3+4 side”).
3. The clipped flux polygons are clipped again by the surrounding cells.
4. Consider that the surrounding cells are translated by $-\mathbf{u}_\rho \cdot \Delta t$. The clipped flux polygons are also clipped by these shifted cells.
5. The net flux volume of each phase in each clipped flux polygon is evaluated according to the distribution of the continuous phase and the dispersed phase. In the figure, the polygon of the flux volume passing through the cell face $f$ (between cell $i$ and $j$) swept by $\mathbf{u}_\rho$ (a common velocity $\mathbf{u}_\rho$ is assumed for the brief explanation) is illustrated as a translucent parallelogram. The flux polygon is divided into four regions, numbered in the figure, being clipped by the interface and the surrounding cells:

1. Region 1 is a part of the flux polygon of the continuous gas side (“Fluid 3+4 side”, above the interface) in cell $j$, and it is swept into a cell (not illustrated in the figure) above cell $i$ at next time step.
2. Region 2 is a part of the flux polygon of the continuous gas side in cell $j$, and it is swept into the cell $i$ at next time step.
3. Region 3 is a part of the flux polygon of the continuous liquid side (“Fluid 1+2 side”, below the interface) in cell $j$, and it is swept into the cell $i$ at next time step.
4. Region 4 is a part of the flux polygon in cell $k$ filled

Fig. 3 Face volume flux by geometrical calculation and the unsplit multi-dimensional method
with the continuous liquid and the dispersed gas, and it is swept into the cell \( i \) at next time step.

Note that Region 4 is overlapped with the flux volume polygon between cell \( j \) and cell \( k \). In order to prevent the overlapped region being counted twice, correction of the volume flux polygon is needed. The present method uses the most simple way, which counts Region 4 not in the flux polygon of the face \( f \) but in the flux of the face between cell \( j \) and cell \( k \).

### 2.5 Surface Tension Model

The continuous surface force (CSF) model [25] is used to take into account the surface tension force as a body force. In this study, the surface tension force of the interface between the continuous phases is considered so that the volume fraction \( \alpha (=\theta_1+\theta_2) \) is used for calculating the surface tension force.

The body force obtained by the CSF model:

\[
f_c = \frac{1}{2} \kappa \alpha \kappa \alpha dV
\]

where \( \sigma \) is the surface tension coefficient and \( \kappa \) is the interface curvature obtained by the following equation:

\[
\kappa = -\nabla \cdot \hat{\rho} = -\nabla \left( \frac{V \alpha}{|\nabla \alpha|} \right)
\]

The surface tension force of the interfaces between the continuous phase and the dispersed phase is not considered by the CSF model. Therefore it must be taken into account via submodels for treating the dispersion phase.

### 2.6 Models for phase change

Table 1 shows the relation of the mass exchange between two phases. At present, a cavitation model [26] for the bubble flow between Fluid 1 and Fluid 2 (arrow A) and simple transition models between continuous phase and dispersed phase (D,E) have been implemented. The implementation of the phase change models for the rest (B,C) and the detailed breakup and coalescence models is the future work.

In the cavitation model [26], the rate of the mass exchange caused by the evaporation and the condensation is calculated using Antoine’s equation.

\[
\Gamma_{21} = \rho_2 \frac{N^*}{C_g} \frac{dR}{dt} = -\Gamma_{12}
\]

where \( N^* \) is the bubble number density. The factor \( C_g \) allows treating a delay for onset of re-condensation of the vapor produced.

As for the transition models between continuous phase and dispersed phase, when the large-scale gas/liquid interface shape has transformed into small structure which cannot be captured by the VOF method without any numerical diffusion, the continuous phase is converted into the dispersed phase through the mass exchange term. A dispersion function proposed by Černe et al. [14] is applied to judge the large-scale interface to be converted or not. On the other hand, when the dispersed particles have gathered and have become more than a certain volume fraction, the dispersed phase is converted into the continuous phase.

### 3. RESULTS

#### 3.1 Oscillating 2D Liquid Column

In order to validate the implementation of the CSF model and the VOF method, calculation of an initially square liquid ethanol column oscillating in zero gravity driven by surface tension was performed under the same condition as Blackbill et al. [25]. Figure 4 shows the calculated motion of the liquid column. The cross section of the initial square column is 3.5cm×3.5cm on a 30×30 grid with \( \Delta x=\Delta y=0.25cm \). The liquid density \( \rho \) is 797.88 kg/m³ and surface tension coefficient \( \sigma \) is 0.0236 N/m. The liquid viscosity is not taken into account. Although there were only the continuous liquid phase and the continuous gas phase in the field, the calculation was performed by using the four-fluid model with zero volume fraction of the dispersed phases.

The calculated liquid motion was corresponding to the result of Blackbill et al. [25] well. The oscillation period of the first cycle is approximately 0.44s. The corner shape of the liquid square becomes round as it cycles because of the nature of the PLIC-VOF that cannot handle the corner shape which needs two line segments within a cell and numerical diffusion of the momentum. As the corner shape becomes round, the oscillation period of the fourth cycle is reduced to approximately 0.42s, which is roughly corresponding to the fourth mode oscillation period (0.416s) of the cylindrical liquid column due to Lamb’s theory.

#### 3.2 Dam Breaking Problem

The collapse of the water column presented by Martin and Moyce [27] which is often used in the validation of the
modeling of the free surface flows was calculated. In addition an ability of the present method to handle the coexisting large- and small-scale interfaces simultaneously in subsequent breaking return wave is also illustrated. The initial state of the water column is a rectangle with the base length $L=1.25\text{ins.} \approx 2.86\text{cm}$ and the height $2L$ as shown in Fig. 5a. The computational domain is $4.5\text{ins.} \times 3.375\text{ins.} \approx 11.43\text{cm} \times 8.57\text{cm}$. Non-slip boundary conditions are applied to all boundaries. The calculations were carried out with two different grid resolutions: $80 \times 60$ grid and $160 \times 120$ grid.

Calculated motion of the collapsing water column on the $160 \times 120$ grid is shown in Fig. 5. Right after the calculation start, the water column against the left side wall starts to fall due to the gravitational acceleration. Then the water flows on the floor of the water tank to the rightward (Fig. 5b and 5c). After that the water climbs the right side wall (Fig. 5d). The water against the right side wall falls again (Fig. 5e) and a return wave flows and breaks against the left side wall (Fig. 5f).

Figure 6 shows the non-dimensional position of the leading edge of the collapsing water column versus the non-dimensional time. Numerical predictions for the $80 \times 60$ grid and the $160 \times 120$ grid are shown. The predicted leading edge position corresponds well with the experimental data presented by Martin and Moyce [27]. The calculated results show that the predicted leading edge become closer to the experimental data when the grid resolution increases.

The surface of the return wave becomes complex shape. As the return wave surges over the water on the floor toward the left, the air bubbles are trapped inside the water as shown in Fig. 7. After that the water crashes against the left side wall and splashes the water droplets. The original VOF method is not able to capture these air bubbles and water droplets properly because the spurious interface reconstruction and advection in such region cause the numerical diffusion of the phase volume. In the present method these small-scale particles are handled by the four-fluid model as the dispersed phase distributed within
the corresponding continuous phase. Therefore the present method can handle the gas/liquid distribution with no volume dissipation and diffusion.

### 3.3 Liquid Jet from a Cylindrical Nozzle

Calculations of liquid jets from nozzle internal flow were performed. The configuration of the nozzle and two injection conditions, which are shown in Table 2, were chosen from the experiment of Tamaki et al. [28]. A water jet is injected from a simplified straight cylindrical nozzle, whose length-to-diameter ratio is L/D=4, into atmospheric pressure air by certain pressure differences. The calculations were done on the cylindrical coordinates system. Fig. 8 shows the computational domain and mesh. Diameter of the upstream of the nozzle orifice was set to 16mm in order to prevent the upstream flow from having an effect on the nozzle internal flow according to Tamaki et al. [28].

Fig. 9 shows the relation of discharge coefficient of the cylindrical nozzle to cavitation number: $CN = (P_f - P_a) / (P_a - P_v)$ where $P_f$ is the injection pressure, $P_a$ is the ambient pressure and $P_v$ is the saturated vapor pressure. The liquid jet from nozzle internal flow are shown in Fig. 10. In the calculation result, the volume fraction distribution in a axisymmetrical cross section is displayed. The black part indicates the gas phase in the nozzle, and the liquid phase in the liquid jet.

In the measurement data, the discharge coefficient sharply decreases from 0.81 to 0.65 between $CN=1.4$ and $CN=2.2$. The flow regime changes together with the transition of the discharge coefficient. The nozzle internal flow with $1<CN<1.4$ becomes a cavitating bubble flow, whereas it becomes a hydraulic flip state when $CN>2.2$. The disintegration of the downstream liquid jet is enhanced when the cavitation bubbles within the nozzle orifice are distributed up to the vicinity of the exit. On the other hand, the liquid jet with hydraulic flip does not disintegrate at all.

As for the calculation results, the tendency by the increase of the pressure difference is corresponding to the measurement data. The discharge coefficient at $CN=1.2$ (Case 1) and $CN=30.6$ (Case 2) is 0.78 and 0.58, respectively. In Case 1, cavitation occurs at the inlet edge of the nozzle orifice and flows downstream. In Case 2, flow separation occurs at the inlet edge of the nozzle orifice and

### Table 2 Nozzle configuration and injection condition

<table>
<thead>
<tr>
<th></th>
<th>Case 1</th>
<th>Case 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nozzle Diameter / Length</td>
<td>2mm / 8mm</td>
<td></td>
</tr>
<tr>
<td>Injection Pressure [MPa]</td>
<td>0.24</td>
<td>3.1</td>
</tr>
<tr>
<td>Ambient Pressure [MPa]</td>
<td></td>
<td>0.1</td>
</tr>
<tr>
<td>Cavitation Number CN</td>
<td>1.4</td>
<td>30.6</td>
</tr>
<tr>
<td>State of nozzle internal flow</td>
<td>cavitating bubble flow</td>
<td>hydraulic flip</td>
</tr>
</tbody>
</table>
the flow regime becomes a hydraulic flip state. The liquid jet of Case 2 is not disintegrated at all whereas the disintegration of Case 1 is enhanced very much due to the cavitation bubble. In the region containing the cavitation bubbles, which flow together with vortices, local pressure becomes low. When the cavitation region reaches to the nozzle exit, the ambient air is drawn toward the inside of the liquid jet and after that the surface of the liquid jet becomes perturbed. The instability of the perturbed surface grows downstream. Eventually disintegration of the surface transforms the continuous liquid into dispersed phase.

While the hydraulic flip is well predicted in Case 2, the cavitation bubble distribution and amount in Case 1 looks different from the measurement. The difference in the amount of the generated cavitation bubbles is mainly attributed to the influence of the cavitation model. On the other hand, the disintegration process of the liquid jet is not fully simulated because of the 2D axisymmetric calculation. The 3D calculation with detailed breakup model and coalescence model for the liquid droplets is the future work.

4. CONCLUSIONS

The CIM method, which is a multi-scale method comprising an interface capturing method and a multi-fluid model for multi-phase flow simulation, has been presented. The main features of the present method are the following:

- The continuous phase and the dispersed phase of gas and liquid (a total of four phases) are taken into account in one cell simultaneously.
- The grid-resolved large-scale gas/liquid interface is reconstructed every time step by applying the PLIC-VOF method.
- The sub-grid small-scale gas/liquid interface is treated by the Eulerian four-fluid model.
- Geometrical advection calculation is carried out for all phases, in which the phase flux volume is calculated from the polygon of the cell face flux clipped by the large-scale interface and phase volume distribution in it.
- The compressibility of gas phase is also taken into account by using the Cell-Centered Multi-Phase ICE method.

The following results were obtained using the CIM method:

- The large-scale interface was captured sharply. The behavior of the oscillation of the liquid column driven by surface tension and the dam breaking flow driven by gravity was simulated quantitatively.
- In case of the liquid jet from a cylindrical nozzle, the decrease in the discharge coefficient was quantitatively predicted together with the transition of the flow regime from a cavitating bubble flow to a hydraulic flip as the pressure difference increases.
- The disintegration process due to the nozzle internal cavitation was shown. When locally low pressure cavitation region reached to the nozzle exit, the free surface of the liquid jet was perturbed. The instability of the perturbed surface grew downstream. Eventually the liquid surface was disintegrated and transformed into the dispersed phase.

5. NOMENCLATURE

- $A^w$: interfacial area density [1/m]
- $C_D$: drag coefficient
- $C_E$: Egler pressure coefficient
- $CN$: Cavitation number
- $C_R$: condensation reduction factor
- $C_{TD}$: turbulent dispersion coefficient
- $c$: speed of sound [m/s]
- $E$: energy source [J/(m$^3$·s)]
- $e$: specific energy [J/kg]
- $F_v$: body force [N/m$^3$]
- $k$: turbulent kinetic energy [m$^2$/s$^2$]
- $M$: momentum source [N/m$^3$]
- $N^w$: number density [1/m$^3$]
- $\vec{n}$: interface normal vector
- $\rho$: pressure [Pa]
- $p_E$: Egler pressure [Pa]
- $p_{sat}$: saturated vapor pressure [Pa]
- $R$: radius [m]
- $T$: temperature [K]
- $t$: time [s]
- $u$: velocity [m/s]
- $v$: specific volume [m$^3$/kg]
- $\alpha$: volume fraction in the VOF method
- $\varepsilon$: turbulent kinetic energy dissipation rate [m$^2$/s$^3$]
- $\Phi$: energy dissipation [J/(m$^3$·s)]
- $\Gamma$: mass source [kg/(m$^3$·s)]
- $\kappa$: interface curvature [1/m]
- $\lambda$: thermal conductivity [W/(m·K)]
- $\theta$: volume fraction in the four-fluid model
- $\rho$: density [kg/m$^3$]
- $\sigma$: surface tension coefficient [N/m]
- $\tau$: stress [Pa]

subscripts
- $k, l$: phase index
- 1: continuous liquid phase
- 2: dispersed gas phase
- 3: continuous gas phase
- 4: dispersed liquid phase

6. REFERENCES