IOP Conf. Series: Materials Science and Engineering **407** (2018) 012100 doi:10.1088/1757-899X/407/1/012100

Simulation on Relocation of Non-Compressed Fluid Flow using Moving Particle Semi-Implicit (MPS) Method

M Ilham[1], Y Yulianto[1] and A P A Mustari[2]

[1]Magister Program of Physics Department,
Faculty of Mathematics and Natural Science, Bandung Institute of Technology
Jalan Ganesha 10, Bandung 40132, Indonesia
[2]Nuclear Physics and Biophysics Research Division, Physics Department,
Faculty of Mathematics and Natural Science, Bandung Institute of Technology
Jalan Ganesha 10, Bandung 40132, Indonesia

pramutadi@fi.itb.ac.id

Abstract. The purpose of this research is to use two-dimensional MPS method to simulate the fluid relocation process from a certain height using water and its interaction with the solid surface. The phenomenon of fluid flow can be applied to the case of nuclear reactors. This research was conducted to see the fluid flow motion and its interaction on a solid surface, which was simulated by MPS method. This research performed several variations of parameters with the motion of falling water with various obstacles. The variation conditions were conducted to see the relationship between the parameters and the mechanism of falling liquid and its splash. As for simulation, validation is done by doing an experiment using the simple instruments. The obtained results of simulations with MPS method show acceptable agreement with the experiment.

1. Introduction

Nuclear is one of the alternative energy that can overcome the lack of electricity in the world. To meet the energy needed, America and Japan have used nuclear technology long time ago to support their conventional electricity sources. However, several accidents in nuclear power plant such as the Chernobyl, the TMI-2, and the Fukushima inhibit the public acceptance especially to the developed country, that is about to start their nuclear energy program. Thus, better understanding on how the accident occurs and progress is utterly essential.

The Fukushima BWR reactor accident became an essential phenomenon in analyzing how the movement or relocation of a molten fuel fluid in the core. The melting process of fuel and control rod causing system failure on the reactor core [1-6]. This melting then results in the relocation of the molten fuel to the lower plenum. In order to have better understanding computational simulation should be conducted. The computational simulation will reduce the cost in comparison to experiment, even though experiment is still important for some situation.

One of the methods that can be used to simulate the phenomenon of fluid flow for relocation is Moving Particle Semi-implicit (MPS) method. The MPS method was introduced by Koshizuka and Oka [7]. This method can be used to analyze non-compressed fluid free surface flow, without using a grid. MPS method is good enough in simulating and analyzing the movement of molten fuel in the reactor

IOP Conf. Series: Materials Science and Engineering **407** (2018) 012100 doi:10.1088/1757-899X/407/1/012100

core. Several studies had been conducted related using MPS method in relation to the severe accident. The improved MPS, i.e. MPS_LER, had been used predict the penetration rate of uranium fuel melting on stainless steels at high temperature [1, 2]. Simulation on melting penetration on CAFÉ experiments at 1200^{O}C is analyzed using two-dimensional MPS [4]. There have been many studies related to fluid motion in case of reactor core melting using the MPS method. However, most of the research is analyzing the melting interaction between core melt and solid material, simulation on the relocation of the molten is still limited. The relocation mechanism may affect melting process in the lower plenum component, such due to fluid splashing.

Thus, the purpose of this research is to use two-dimensional MPS method to simulate the fluid relocation process from a certain height using water and its interaction with the solid surface. The results of this MPS simulation will be validated by simple experiment.

2. Physical model and numerical method

2.1. General equation

The general equation for the non-compressed fluid used is the Navier-Stokes equation. The Navier-Stokes equation can explain the movement of a fluid such as liquids and gases. These equations suggest that the momentum of the fluid particles depends on the internal viscosity force and the external pressure viscosity force acting on the fluid [7].

As for the equations for mass conservation:

$$\frac{d\rho}{dt} = 0 \tag{1}$$

And for the law of conservation of momentum:

$$\frac{Du}{Dt} = -\frac{1}{\rho}P + v\nabla^2 u + f \tag{2}$$

Where ρ is the density, t is the time, $\vec{\boldsymbol{u}}$ is velocity, ∇ is the gradient, P is the pressure, $\boldsymbol{v}$ is the kinematic viscosity, and $\vec{\boldsymbol{f}}$ is the force of gravity.

2.2. Moving Particle Semi-implicit (MPS) method

The MPS method is a particle-based method for analyzing non-compressed fluid (Koshizuka and Oka, 1996). The particle interaction is defined by its position against the reference particles. The schematic diagram for the nearest neighbor of particle i is shown in Figure 1. Using the concept of weight function based on distance r, the movement of particles can be calculated from the interaction forces between the two closest particles.

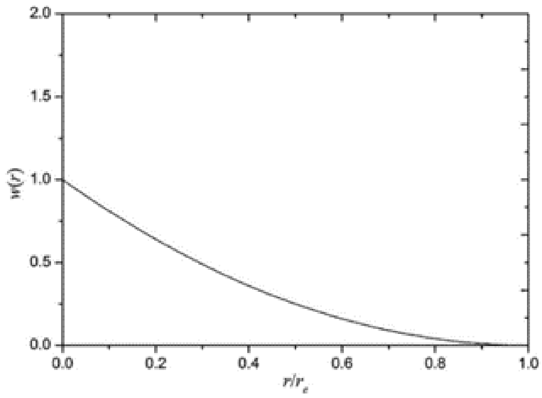

Figure 1. Weight function [7].

The density of particles, which is proportional to the fluid density, in the particle i position can be approximated by

$$n_i = \sum_{j \neq i} w(|r_j - r_i|) \tag{3}$$

Where r_i and r_j are the position vectors of particles i and j. Gradient, Divergences and Laplacian models can be discretized using:

$$< \nabla\phi >_{ij} = \frac{(\phi_j - \phi_i)(r_j - r_i)}{|r_j - r_i|^2} \tag{4}$$

$$< \nabla\phi >_i = \frac{d}{n^0} \sum_{j \neq i} w(|r_j - r_i|) \frac{(\phi_j - \phi_i)(r_j - r_i)}{|r_j - r_i|^2} \tag{5}$$

$$< \nabla^2\phi >_i = \frac{2d}{\lambda n^0} \sum_{j \neq i} w(|r_j - r_i|)\,(\phi_j - \phi_i) \tag{6}$$

Where d is the number of dimensions, n^0 is the density of the number of particles suitable for the non-compressed system, and λ is the selected parameter for the obtained result to be proportional to the analytic solution. The value of λ can be approximated by using the formula:

$$\lambda = \frac{\sum_{j \neq i} w(|r_j - r_i|)\,|r_j - r_i|^2}{\sum_{j \neq i} w(|r_j - r_i|)} \cong \frac{\int_v w(r) r^2 dv}{\int_v w(r) dv} \tag{7}$$

The density of the particles is made constant for the internal particles to maintain the non-compressed conditions when the condition is decreased for particles located on the free surface as shown in Figure 2.

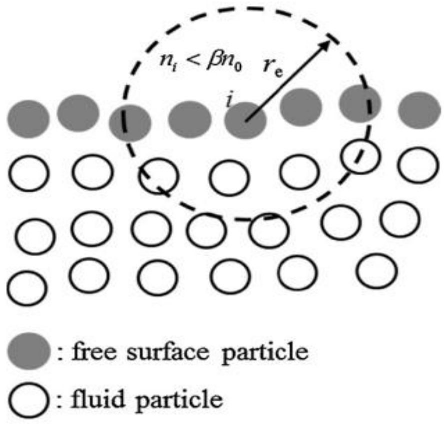

Figure 2. Boundary condition of free surface.

2.3. Experiment and simulation

2.3.1. Experiment. This research uses water as a fluid particle with the case of water falling is with an obstacle. Box-shaped containers with length, width and height are 165 mm x 30mm x 40mm, while bottle as the cylindrical fluid source, shrink at the bottom and have no lid base. The sized of straight iron pipes are 7.5 cm, 5 cm, 4 cm. Its inner diameters are 10mm, 7mm, and 3mm respectively. The density of water is 1000 kg/m^3 and its kinematic viscosity is 1,004 mm^2/s.

Figure 3. Simple experimental instruments and materials.

2.3.2. Simulation. A total number of particles in each simulation is more or less 8000 particles. The simulation is in two dimensions.

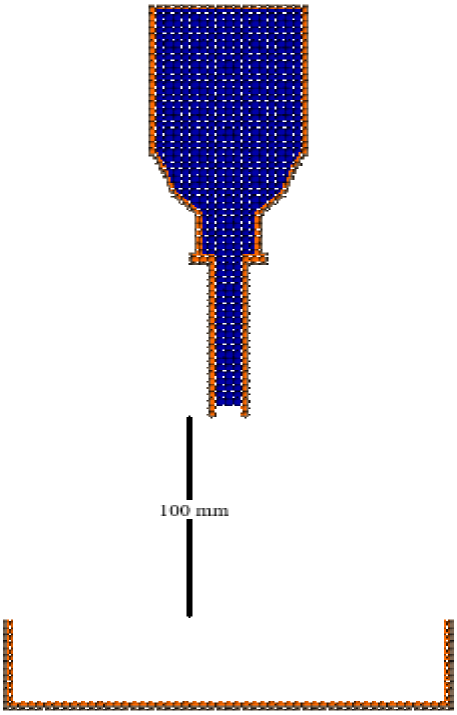

Figure 4. Simulation design for fluid free fall motion with obstacle, 10 cm from surface with straight pipe.

IOP Conf. Series: Materials Science and Engineering **407** (2018) 012100 doi:10.1088/1757-899X/407/1/012100

3. Results

3.1. Effect of straight pipe diameter size

3.1.1. Fluid free fall motion with obstacle (Straight pipe diameter 10 mm with water)

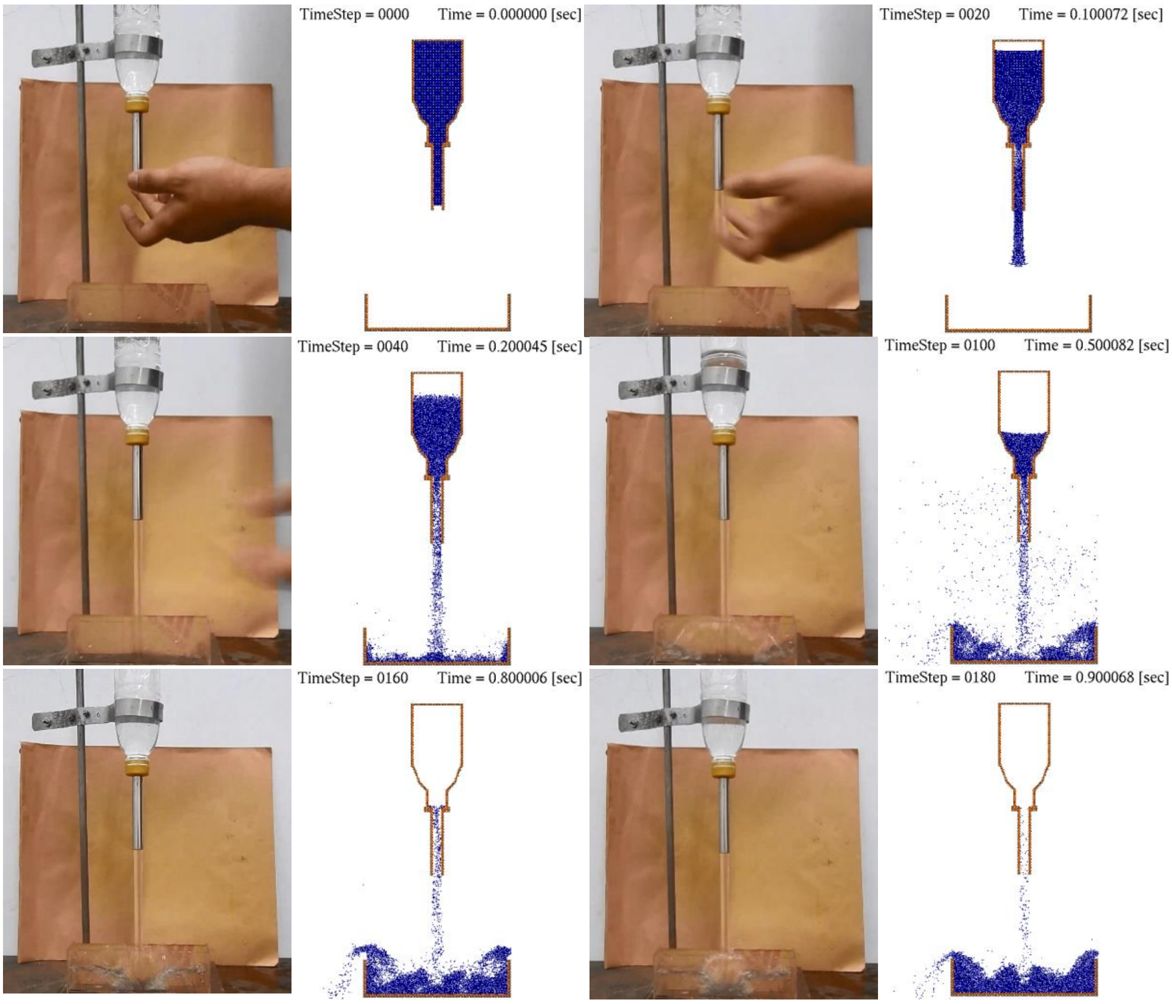

Figure 5. Comparison of simulation results with experimental of falling motion with obstacle straight pipe diameter 10 mm.

Figure 5 shows a case of falling water motion with an obstacle with a pipe of 10 mm diameter. The results are similar to the cases of falling water without obstacle, but it appears that with added obstacles, it gives fewer water splashes compared with the case without obstacle. At the time 0.1s and 0.2s, the motion of water is very similar to the experiment; water spread in the center and toward the container walls. At 0.5s and 0.8s, there’s splashed water coming out of the container, but slightly visible in the experimental results, although the shape of the water when it touches the container is quite similar between these two. This simulation lasts up to 1 second.

3.1.2. Fluid free fall motion with obstacle (Straight pipe diameter 7 mm with water)

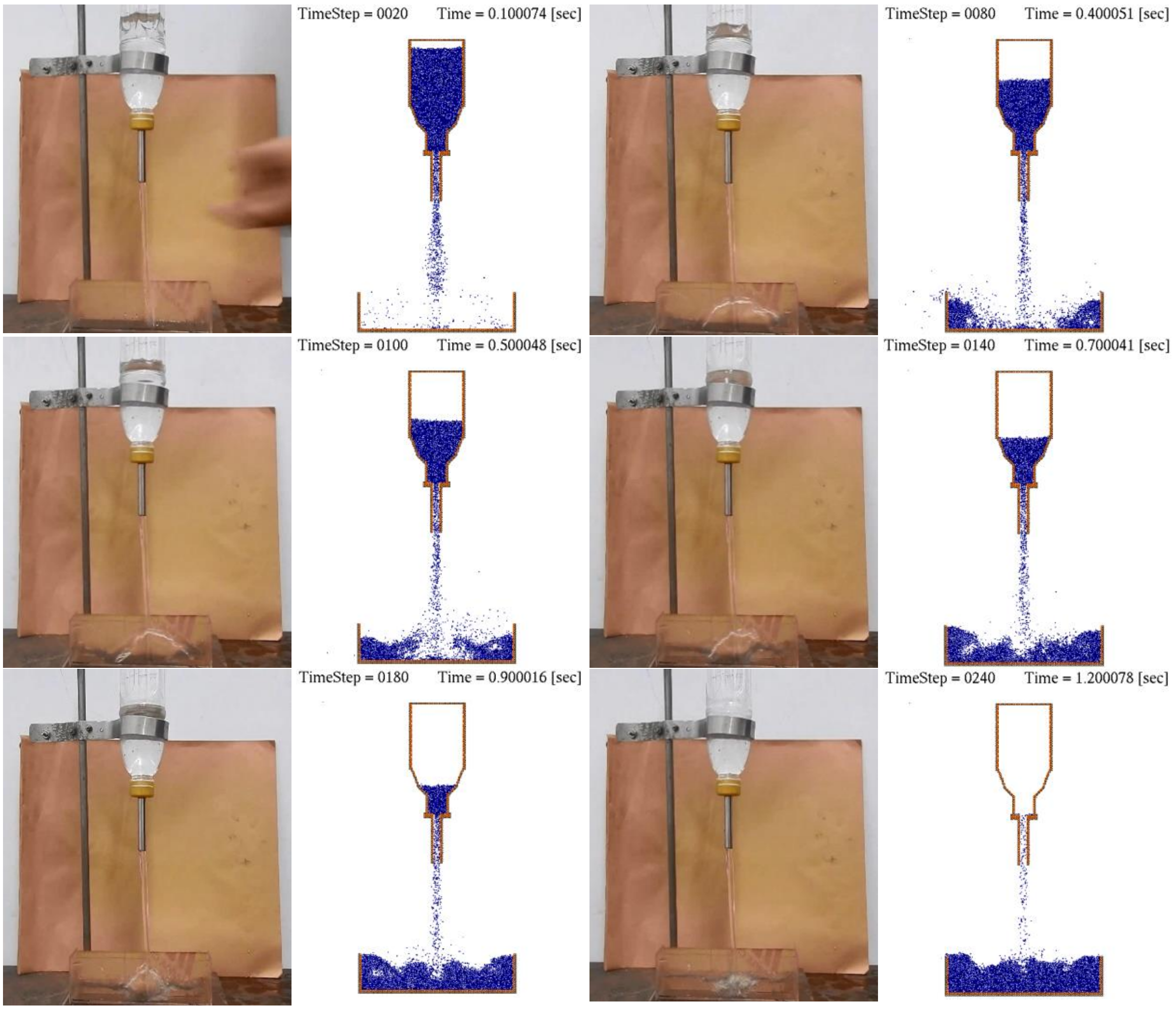

Figure 6. Comparison of simulation results with experimental of falling motion with obstacle I pipe diameter 7 mm.

Figure 6 in case of falling water motion with obstacle straight pipe with diameter 7 mm. When the pipe diameter is reduced, the splash of water in the simulation also decreases where the shape and motion of it became smooth as it touches the container, these results also indicated by the experiment. Visible to the experiment, there is a reversed bowl splash which is also shown by the simulation at 0.5s. The shape of water ripples in containers at 0.9s and 1s are also generated by simulations. This simulation lasts up to 1.5 seconds.

IOP Conf. Series: Materials Science and Engineering **407** (2018) 012100 doi:10.1088/1757-899X/407/1/012100

3.1.3. Fluid free fall motion with obstacle (Straight pipe diameter 3 mm with water)

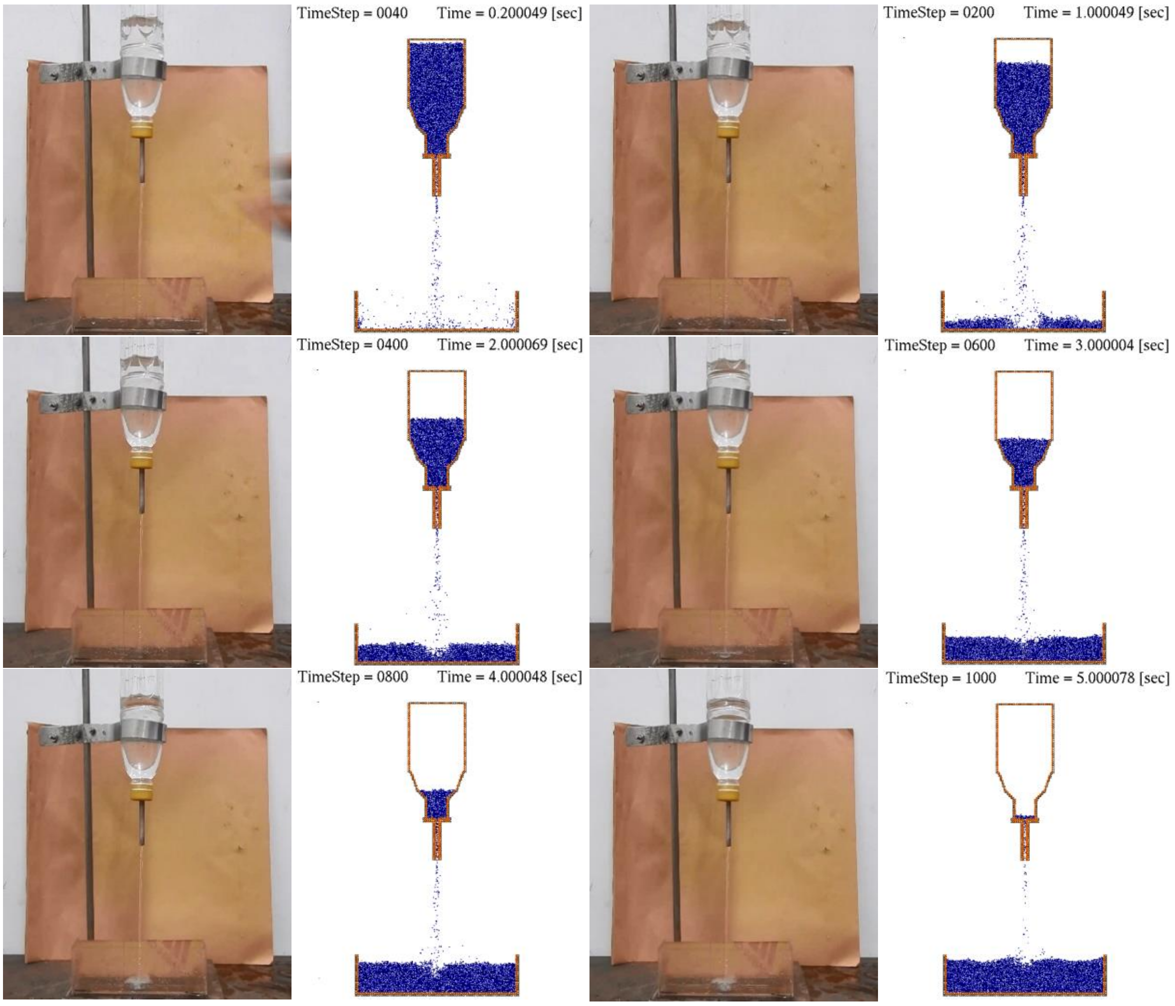

Figure 7. Comparison of simulation results with experimental of falling motion with obstacle I pipe diameter 3 mm.

Figure 7 shows the motion of falling water with obstacle straight pipe 3 mm in diameter. When the pipe diameter is reduced to 3mm, the splash on the simulation is not generated. Falling water that fills the container is very slowly in the experiment, also generated by the simulation where the water fills the container with a stable shape. This simulation lasts up to 6 seconds.

4. Discussion

The result for the case of falling water motion with the straight pipe, when the diameter of the pipe decrease from figure 5 to figure 7, the motion of water falling on the container also changes, less the water splash out of the container until no splashes at all. There is a reversed bowl in Figure 6 in the experiment; this is because of the bottle's mouth in the experiment, close enough to the front wall of the container to form the splash. There is a time difference in the length of time the falling water on the bottle until it runs out. Diameter of larger pipe (10 mm) has time difference that is quite small, but for the smaller pipe (3 mm), there is a significant difference. The reason why there is such a time difference between experiment and simulation, most likely due to turbulence phenomena occurred in the bottleneck near straight pipe of the experiment, while current MPS used for this simulation have not been improved to tackle such a case.

5. Conclusion

In this research, a two dimensional MPS method is used in simulating the fluid relocation process of certain stages for water and oil fluids and their interaction with solid surfaces. The results of the simulation will be verified with the results obtained in a simple experiment.

The results obtained show the simulation of fluid fall motion gives a shape that is close enough to the experimental results. With straight pipe obstacle, makes the splash of fluids decrease and the motion of it's become smoother. Decreasing the diameter of the pipes will effects the time of the bottle runs out of the water. The simulation results in these two dimensions indicate that the MPS method has the ability to analyze the fluid fall action behavior that can later be used in large-scale simulations such as reactor core melt.

Acknowledgments

This study is fully funded by Ministry of Research, Technology and Higher Education 2017, Indonesia.

References

[1] Pramutadi A, Syeilendra and Irwanto D 2017 "Prediction of Melting Penetration of Armco Iron by Liquid Uranium Using MPS_LER," *J. Phys.: Conf. Ser.* 799 01201.
[2] Pramutadi A, Yamaji A and Irwanto D 2016 "Melting Penetration Simulation of Fe-U System at High Temperature Using MPS_LER," J. *Phys.: Conf. Ser.* 739 012016.
[3] Pramutadi A and Oka Y 2014 "Molten Uranium Eutectic Interaction On Iron-alloy by MPS method," *Nuclear Engineering and Design* **278** 387–394.
[4] Pramutadi A, Iksal and Sumiati 2012 "MPS simulation on melt penetration of CAFÉ experiment at 1200°C," *Indonesian Journal of Physics* **23** (1).
[5] Pramutadi A and Takahashi M 2011 "Study on corrosion of welded steel for LBE-cooled fast reactors," *Progress in Nuclear Energy* **53** 1073-1077.
[6] Pramutadi A, Oka Y, Furuya M, Takeo W and Chen R 2015 "3D simulation of eutectic interaction of Pb-Sn system using Moving Particle Semi-implicit (MPS) method," *Annals of Nuclear Energy* **81** 26-33.
[7] Koshizuka S and Oka Y 1996 "Moving particle semi-implicit method for fragmentation of incompressible fluid," *Nuclear Science and Engineering* **123** 421-434.