Simulation of Liquid Jet Breakup Process by Three-Dimensional Incompressible SPH Method

T. Takashima, T. Ito, M. Shigeta, S. Izawa and Y. Fukunishi
Corresponding author: takashima@fluid.mech.tohoku.ac.jp

Tohoku University, Japan.

Abstract: A liquid jet breakup process is simulated using Smoothed Particle Hydrodynamics (SPH) method extended for incompressible flows. The method includes the gas-liquid interface model to evaluate the surface tension. The results agree well with the experiment. Two different breakup processes are observed under a same condition.

Keywords: Smoothed Particle Hydrodynamics, Surface Tension, Liquid Jet Breakup.

1 Introduction

A breakup process of a liquid jet issuing from a nozzle into an ambient gas has been studied last several decades[1]. The phenomenon under low speed conditions can be observed in daily life, such as droplets dripping from a faucet. Under high speed conditions, the liquid jet becomes a spray which is used in various industrial situations such as ink-jet printings and fuel injections in automobile engines. In this study, the behavior of liquid jet from a faucet is simulated by a Smoothed Particle Hydrodynamics (SPH) method extended for incompressible flows. The results are compared with the experiment that used a high-speed camera.

2 Numerical Method and Experimental Condition

SPH method is originally developed to simulate compressible fluid motions, since in the governing equations, the acceleration of a particle is based on the density gradient of the particle distributions. In this study, the density non-uniformity is smoothed every time step after the particles move according to their inertia. The position of each particle is adjusted so that the incompressible fluid motion is simulated[2].

The surface tension is modeled by the Laplace pressure and the force is applied to the fluid particles at the surface.

Experiment is also carried out and a liquid jet ejected from a tap of 3 mm in diameter is observed. The averaged water velocity $\bar{u}$ is changed from 0.2 m/s to 0.85 m/s and the behavior of the liquid jet is recorded using a high-speed camera. The computational conditions are set to coincide with the experiment, where the flow inside a tap is assumed to be a Poiseuille flow.

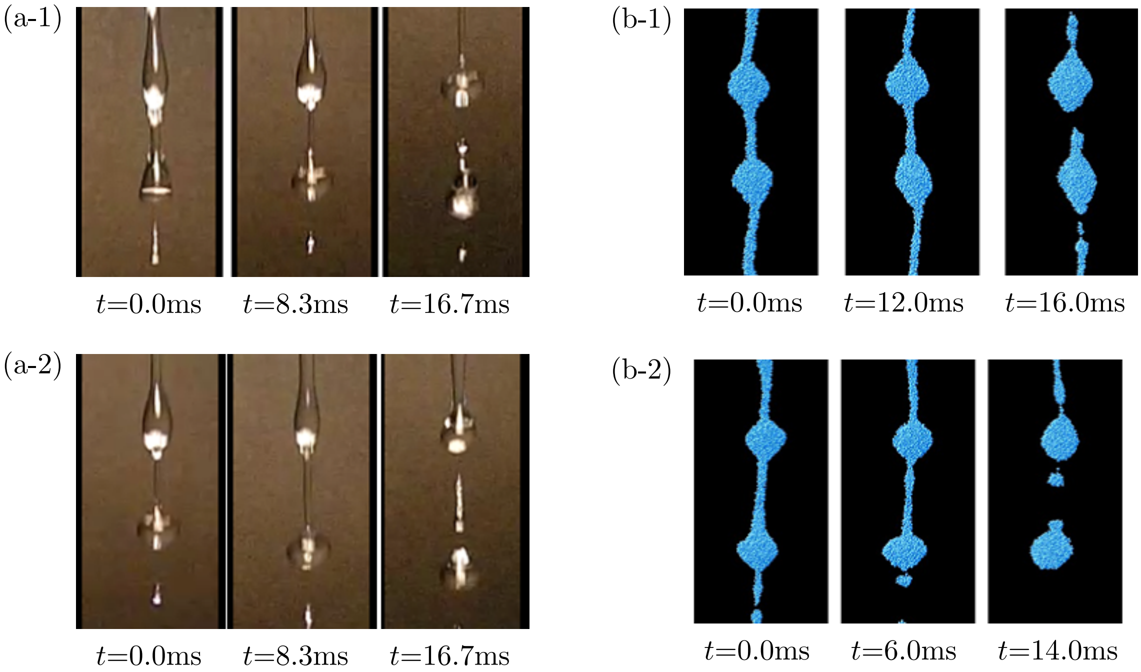

Figure 1: Droplets formation at $\bar{u}$ = 0.68 m/s, (a) experiment and (b) simulation.

3 Results and Discussion

Figure 1 shows the experimental and numerical results at the location where droplets are formed. The time t is defined as the time relative to the moment shown in the left figure. As shown in the Fig.1(a-1) and (b-1), a part of a liquid jet becomes thinner as time advances. Finally it breaks up at the neck of the column and forms a droplet. However, in some cases, the middle liquid column is further elongated before breaking up, as shown in Fig.1(a-2) and (b-2). In such cases, the liquid jet breaks up into three droplets. Although experimental and numerical results are similar to each other, the length from a tap to the breakup point is shorter in the simulations than the experiments. This is probably due to the small vibrations of particles existing only in simulations.

4 Conclusion

The breakup process of a liquid jet from a tap nozzle was calculated by SPH method. The numerical result exhibited a breakup behavior that agreed well with the experiment.

References

[1] J. Eggers and E. Villermaux. Physics of liquid jets. *Rep. Prog. Phys.*, 71:036601, 2008.

[2] M. Shigeta, T. Watanabe, S. Izawa and Y. Fukunishi. Incompressible SPH Simulation of Double-Diffusive Convection Phenomena. *Int. J. Emerging Multidisciplinary Fluid Sci.*, 1:1-18, 2009.