Simulation of liquid jet breakup by the level set method

Pan Y., Suga K.

Toyota Central R & D Labs., Inc., Nagakute, Aichi 480-1192, Japan

Direct numerical simulations of water jets into air are presented. The level set method is applied to capture the interface between the liquid and the surrounding gas during the evolution of the jet. The jet exit conditions are limited to a water flow without cavitation, the jet exit Reynolds numbers of 1000-15000, the jet exit Weber numbers of 44-4994 and the liquid/gas density ratio of 816. The dynamic features of liquid/gas jet flows are well captured by the present simulations. The Rayleigh instability and breakup are observed at low Reynolds numbers. At high Reynolds numbers, our dynamic simulations clearly present the transition from laminar instability to turbulent breakup and spray formation. The effects of the initial fluctuations at the jet exit and the effect of surface tension are also investigated.

1. Introduction

The present study is motivated by our desire to understand the behavior of a liquid jet at near an injector-exit region and the mechanism of its breakup. It is a part of our effort to study the pressure-atomized fuel injection/spray that provides the fine atomization needed for rapid mixing of liquid and gas phases during practical combustion processes. Most early studies were on the breakup length and the distributions of the size and velocity for the droplets after breakup, see, for example [1]. In spite of a great amount of investigations in this research area, the mechanism of turbulent jet breakup and spray formation are still not well understood. People are interested in the behaviour of liquid jets and the mechanism of jet break-up at the very initial stage right after the injector exit. In such a region, a liquid jet undergoes topological transitions associated with very small length and time scales, which are difficult to observe and characterize with current experiment techniques. It also challenges the continuum-based numerical simulations by posing singularities caused by small length and time scales accompanying transitions.

In the present paper, we present the dynamic simulations of a liquid stream injected into a still gas. The level set method (Chang *et al*. [2]) is used as the interface tracking methodology. Assuming a non-cavitation containing liquid, one can characterize a jet by its Reynolds number (Re), Weber number (We) at the exit and the density ratio of liquid to gas (ρ_L/ρ_G). In most practical processes, liquid jets are featured by high Reynolds numbers and large density ratios. For instance, the Re and ρ_L/ρ_G of a pressure-atomized fuel injection are typically in the order of 10^5 and 10^3, respectively. Currently, however, most numerical studies in the literature are still limited to problems of relatively low Re and ρ_L/ρ_G. Here we attempted the simulations of water jet into air with ρ_L/ρ_G =816 and Re numbers ranging from laminar to turbulence regime. In the numerical aspect, we incorporate the scheme of the level set method by Sussman *et al*. [3] and Chang *et al*. [2] into a Navier-Stokes solver for an incompressible single phase flow based on a finite-volume method by Kashiwa and Rauenzahn [4]; Kashiwa *et al*. [5]. The surface tension force is treated as a body force by adopting the continuum surface force (CSF) model by

Brackbill *et al.* [6].

2. Numerical models

We consider the fluid motion of a liquid injected into a space, which is initially filled with a gas of different density and viscosity. Two liquids are immiscible to each other. The interface between the two liquids remains throughout the motion and a surface tension exists at the interface. The flow motion is governed by the Navier–Stokes equation,

$$\rho\left(\frac{\partial \mathbf{u}}{\partial t}+\mathbf{u}\cdot\nabla\mathbf{u}\right)=-\nabla p+\nabla\cdot(2\mu\mathbf{S})+\sigma\kappa(\theta)\nabla\theta\delta(\theta)+\rho\mathbf{g} \tag{1}$$

The third term on the right-hand-side of the above equation represents a model, called the continuum surface force (CSF) model proposed by Brackbill *et al*. [10], for approximating the surface tension forces. In this model, the effect of surface tension can be expressed in terms of a singular source which is defined by an indicative function, here the level set function, θ.

The level set method (Chang *et al.* [5]) is used to capture the interface between two fluids of different densities and viscosities. A level set function, say, θ, is a distance function about the interface. It has positive values outside the interface and negative inside the interface. At the interface θ keeps a value of zero. The magnitude of the level set function at any location represents the distance from this location to the interface. The level set function, like any passive scalar variables, moves with the fluid, and it follows,

$$\frac{\partial\theta}{\partial t}+\mathbf{u}\cdot\nabla\theta=0 \tag{2}$$

which moves the zero level of θ exactly as the actual interface. The density and viscosity are calculated through out the computational domain depending on the value of θ, by

$$\rho=\begin{cases}\rho_i & \\ (\rho_i+\rho_o)/2 & \\ \rho_o & \end{cases} \quad \text{and} \quad \mu=\begin{cases}\mu_i & \theta<0\\ (\mu_i+\mu_o)/2 & \text{for}\ \ \theta=0\\ \mu_o & \theta>0\end{cases} \tag{3}$$

The Dirac function δ is regularized by

$$\delta_\varepsilon(x)=\begin{cases}\frac{1}{2}(1+\cos(\pi x/\varepsilon))/\varepsilon & \text{if } |x|<\varepsilon,\\ 0 & \text{otherwise},\end{cases} \tag{4}$$

and the corresponding regularized Heaviside function H_ε is defined as,

$$H_\varepsilon(x)=\begin{cases}0 & \text{if } x<-\varepsilon,\\ (x+\varepsilon)/(2\varepsilon)+\sin(\pi x/\varepsilon)/(2\pi) & \text{if } |x|\le\varepsilon,\\ 1 & \text{if } x>\varepsilon,\end{cases} \tag{5}$$

Using the regularized Heaviside function H_ε, we can define the corresponding regularized density function ρ and the regularized viscosity μ as,

$$\rho_\varepsilon(x)=\rho_i+(\rho_o-\rho_i)H_\varepsilon(\theta(x)) \tag{6}$$

$$\mu_\varepsilon(x)=\mu_i+(\mu_o-\mu_i)H_\varepsilon(\theta(x)) \tag{7}$$

In our computations, we use $\varepsilon=\Delta x$. We denote ε as the prescribed "thickness" of the interface. The nature of the level set method being a signed normal distance function from the interface has essentially to be kept all the time throughout the simulation. A procedure of re-initialization and re-normalization (Sussman *et al.* [7]) is therefore performed at every time step during simulation to pertain such property. This is achieved by solving the following equation to a steady state,

$$\frac{\partial \theta}{\partial t} = \mathrm{sgn}\left(\theta_0\right)\left(1 - |\nabla \theta|\right) \tag{8}$$

with an initial condition,

$$\theta(\mathbf{x}, 0) = \theta_0(\mathbf{x}) \tag{9}$$

where $\theta_0(\mathbf{x})$ is the level function before the re-normalization. By the above procedure we build a distance function $\theta(\mathbf{x})$ whose zero set is the same as $\theta_0(\mathbf{x})$.

3. Results and Discussions

Our numerical experiments simulate the water injection into air in a two-dimensional domain. The diameter of the jet at inlet, D, is fixed at 400 μm. This value is quite close to the size of fuel injectors in most DI engines for automobiles. The key material properties are ρ_{water}=1 g/cm^3, ρ_{air}=1.226×10^{-3} g/cm^3, μ_{water}=1.137×10^{-2} g/(cm s), μ_{air}=1.78×10^{-4} g/(cm s), g=981 cm/s^2 and σ=72.8 dyes/cm. Re and We are based on the liquid properties at the exit. A uniform Cartesian mesh is used with Δx=Δy=10μm(=D/40), i.e., the initial jet is resolved by 40 computational cells. Depending on the Re number based on D and the jet speed U_0, the dimensions of the computational domain are 11-22D and 80-130D in spanwise and streamwise direction, respectively.

3.1. Reyleigh Breakup

Our first case is a laminar jet at Re=10^3. By using a surface tension coefficient of 12.8 dyes/cm, the Weber number is 252. A parabolic profile of velocity distribution is imposed constantly at the jet exit. Figure 1(a) shows a series of time-resolved images of the jet. One can clearly observe the wave formation and evolution caused by disturbances. The interesting point here is that no disturbances of any kind are artificially introduced at the jet exit in this case. It seems the disturbances are created naturally from the simulation. And the wavelength of such disturbances is larger than the minimal wave number to be damped out by the surface tension. Notice that the jet develops into a multimode wave motion after the primary wave at early stage. The simulated jet shape and motion are therefore very similar to the experimentally observed evidences. According to Raleigh's analysis [7], for laminar jet the breakup length, L, follows the form,

$$L/D = We^{0.5}(1+3Oh)\ln\left(\frac{D}{2\delta_0}\right) \tag{10}$$

in which δ_0 is the initial disturbance. It is difficult to determine quantitatively the value of δ_0. The most recent empirical correlation is provided by Sallam *et al.* [1] as,

$$L/D = 5.0We^{0.5}, \qquad \text{for} \qquad \text{We}<400 \tag{11}$$

This gives L/D=79.4 for the present case. The computational domain in streamwise for the present case happened to be 80D. The jet's disintegration may hence marginally occur at the boundary of the computational box. (Further discussions on the breakup length are currently underway.) The effect of surface tension is examined by increasing the surface tension coefficient 4 times while keeping all the other parameters unchanged. The calculated jet (not shown in the figure) is a steady and straight liquid column indicating the damping out of all the disturbances by the surface tension.

3.2. Effect of initial disturbances

One of the major causes of the primary breakup of a liquid jet is turbulence. The turbulence carried by the liquid from inside the injector is unknown in the present simulations. In order to

model the fluctuations caused by turbulence at the jet exit, we introduce a disturbance on the diameter of the jet, D, at the exit as a time-dependent boundary condition,

$$D = D_0\left[1+\varepsilon_1 \sin\left(2\pi t / T_0\right)+\varepsilon_2 \cos\left(\pi t / T_0\right)+\varepsilon_3 \sin\left(\pi t / 2T_0\right)\right] \tag{12}$$

in which the period, T_0, is determined by the most favourable wavelength, λ_0, for drop formation,

$$\lambda_0 = \sqrt{2}\pi d\left(1+\frac{3\mu_{water}}{\sqrt{\rho_{water}\sigma d}}\right)^{0.5} \qquad \text{and} \qquad T_0 = \lambda_0 / U_0 \tag{13}$$

(a) without initial disturbance (b) with initial disturbance

Fig. 1 Water jet into air at Re=1000; D=400μm; ρ_L/ρ_G=816; U_0=2.84m/s**.** (a) We=252; (b) We=44.

Accordingly, the fluctuations on the streamwise and spanwise velocity at the exit are also introduced. Figure 1(b) shows the effect of initial fluctuation on the jet behavior. In this simulation the magnitudes of the fluctuation modes, ε_1, ε_2 and ε_3, are set to 0.025, 0.02 and 0.0175, respectively. This condition yields a liquid jet whose surface becomes distorted in the cross-stream direction; the distortion is irregular (i.e., not axisymmetric); and the liquid jet breakup process is associated with liquid disturbances having characteristic dimensions comparable to the diameter of the liquid jet itself. Notice that the Re number is kept at 10^3, same as the undisturbed laminar jet Fig.1(a). At this Re number there is essentially no turbulence. The simulated jet is very similar to the jet at a Reynolds number of 13690 observed as the turbulent breakup regime by experiment [1]. Our results shows the effects of initial disturbances on the jet's breakup mode and the possibility of modeling the content of turbulence at the jet exit via artificially adding fluctuations on the boundary conditions at the jet exit.

3.3. Breakup at high speed

The jets in turbulent regime are simulated at $Re=5\times10^3$, 10^4 and 1.5×10^4, respectively. (Since the present simulations are 2-dimensional, strictly speaking, the results cannot contain full turbulence features, but could include some of those in the laminar to turbulent transitional regime.) For the case of $Re=5\times10^3$ and 1.5×10^4, no disturbances are added at the jet exit. For the case of $Re=10^4$, a strong fluctuation is imposed at the jet exit. Figure 2 and 3 record time series of injection process. The whole breakup process is clearly observed from these images. The naturally occurred instability induces a distortion of the jet and in turn results in a sudden breakup. It is emphasized here that for these two cases, the boundary conditions at inlet, i.e. jet diameter and velocity, are constant without disturbances. Noticing the short time duration of these records, the injections are still at initial stage. For the case of $Re=10^4$, a three-modes disturbance, Eq.12, is imposed at the jet exit with $\varepsilon_1=0.03$, and $\varepsilon_2 = \varepsilon_3=0.026$. Figure 4 shows three images arbitrarily chosen at different time instant during a later stage of the simulation. It seems the injection is quite close to a fully developed one. Certain structures in the spray after breakup are observed and are most likely correlated with the driving modes at inlet.

4. Conclusions

This investigation attempted to numerically simulate the process of breakup and spray formation of when a liquid jet is injected into a gas. The major contributions of the study are as follows:

1. A Navier-Stokes solver for incompressible flows coupled with the level set method and the CSF model of surface tension is able to simulate the process of liquid injection into gas. Such a simulation requires, in general, sufficiently high mesh resolution.
2. At a low jet Reynolds number, a Rayleigh-like breakup naturally develops from a smoothly injected jet. Its dynamic feature and breakup length agree well with the experimental knowledge.
3. The turbulent primary breakup and the spray formation are well captured at a high Reynolds number.
4. The breakup modes and the characteristics of the spray are closely correlated with the disturbance introduced at the jet exit. This brings up an important issue on how to set the boundary conditions for velocity fluctuations at the inlet of computational domain. This issue is also closely related to the mechanism of the primary breakup particularly at a high Reynolds number and a large density ratio.

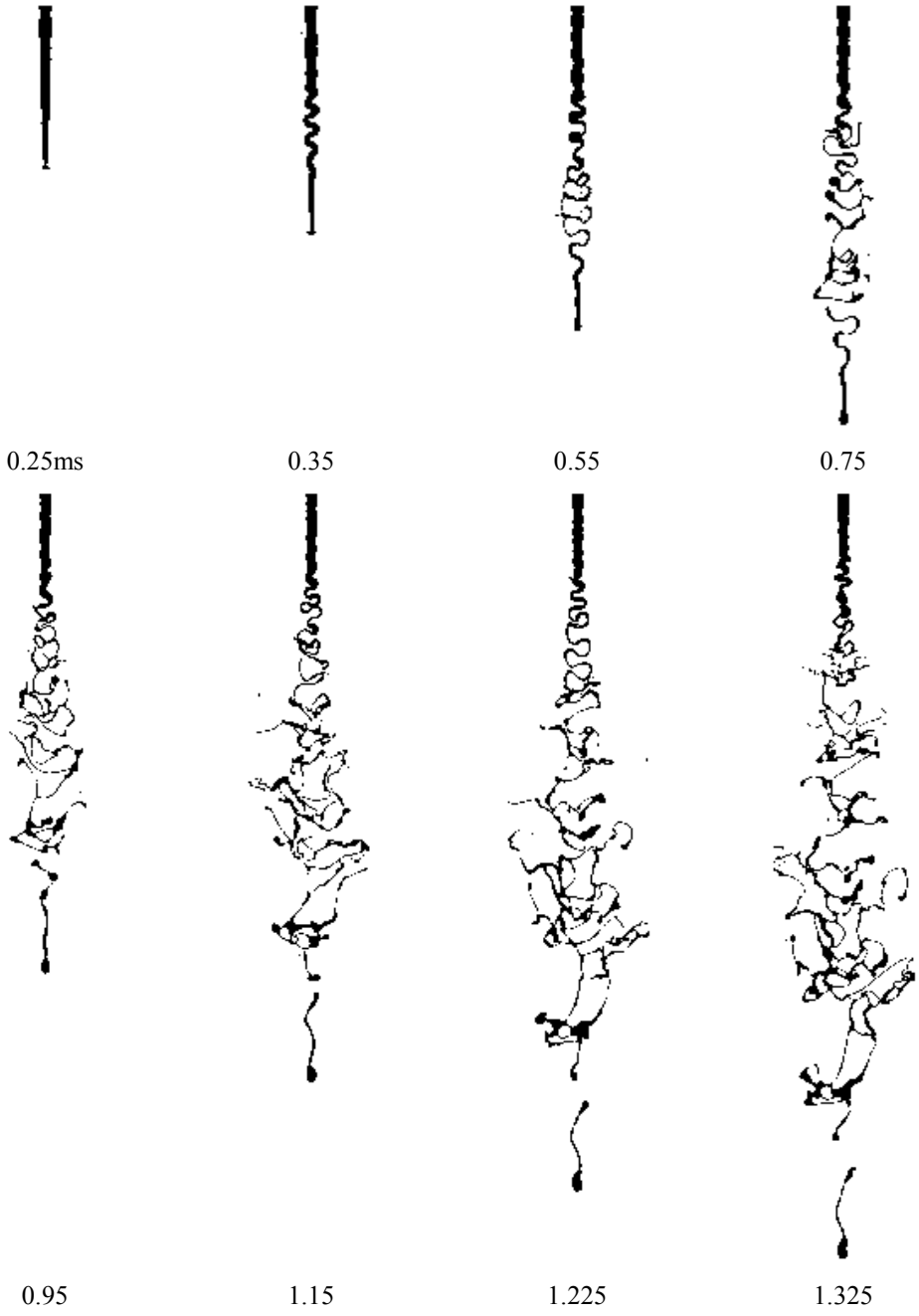

Fig. 2 Water jet into air at Re=5000; We=555 D=400μm; ρ_L/ρ_G=816; U_0=14.21m/s**.**

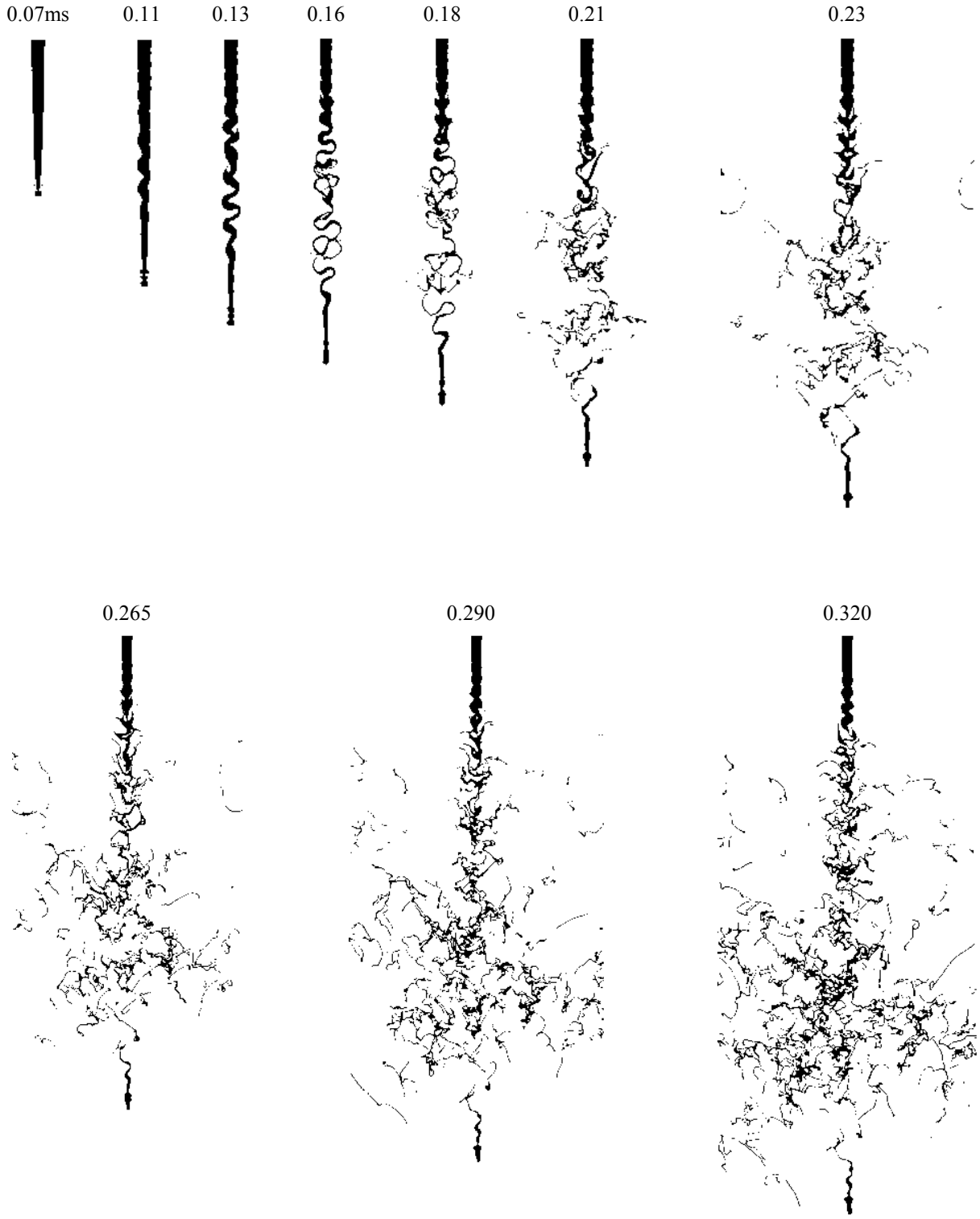

Fig. 3 Water jet into air at Re=15000; We=4994; D=400μm; ρ_L/ρ_G=816; U_0=42.64m/s.

Fig. 4 Water jet into air at Re=10^4; We=2219; D=400μm; ρ_L/ρ_G=816; U_0=28.42m/s.

5. References

[1] Sallam, K. A., Dai, Z., Faeth, G. M., 2002, "Liquid breakup at the surface of turbulent round liquid jets in still gases," *Int. J. Multiphase Flow*, 28, 427-449.

[2] Chang, Y. C., Hou, T. Y., Merriman, B., Osher, S., 1996, "A level set formulation of Eulerian interface capturing methods for incompressible fluid flows," *J. Comp. Phys*. 124, 449-464.

[3] Sussman, M., Smereka, P., Osher, S., 1994, "A level set approach for computing solutions to incompressible two-phase flow," *J. Comp. Phys*. 114, 146-464.

[4] Kashiwa, B. A., Rauenzahn, R. M., 1994, "A multi-material formulation," ASME-FED 185, 149-157.

[5] Kashiwa, B. A., Padial, N. T., Rauenzahn, R. M., VanderHeyden, W. B., 1994, "A cell-centered ICE method for multiphase flow simulations," ASME-FED 185, 159-167.

[6] Brackbill, J. U., Kothe, D. B., Zemach, C., 1992, "A continuum method for modelling surface tension," *J. Comp. Phys.* 100, 335-354.

[7] Lefebvre, A. H., *Atomization and Spray*, Taylor & Francis, 1989.