**Abstract**

The flow of a liquid through a nozzle and resulting jet in a stagnant gas is investigated numerically. The flow is considered to be laminar and axisymmetric. Creation and growth of surface waves due to Kelvin-Helmholtz and capillary instabilities leading to breakup of jet are captured. The disturbance in jet is caused by perturbing the mass flux of liquid through nozzle by five percent. Effects of the geometry such as length-to-diameter ratio and curvature of corners of the nozzle on the growth of interfacial waves and breakup distance are studied. Numerical simulation is performed using a finite-volume method. A boundary-fitted orthogonal grid has been used. Two-phase flow and surface tension are modeled using a level-set formulation.
1 Introduction

Breakup of liquid jets has been under extensive study for years. Rayleigh [1, 2] considered the linear instability of a column of liquid due to surface tension effects. Bogy [3] employed a higher order perturbation method to study the viscous jet instabilities and was able to fit the solution by Rayleigh for viscous jet instability.

Mehring and Sirignano studied the nonlinear capillary waves on annular liquid sheets [4] and planar liquid sheets [5] by a reduced dimension method. They showed that breakup occurs faster in swirling annular sheets than nonswirling ones.

Funada and Joseph [6] did a Viscous potential flow analysis of capillary instability. They also considered the effects of viscoelasticity on the capillary instability of a column of liquid [7].

Chaudhary and Redekopp [8] and Chaudhary and Maxworthy [9, 10] looked for conditions of liquid jet with no satellite droplet by imposing higher harmonics perturbation to the first harmonic.

Many numerical studies are also performed on the breakup of liquid jets. Most of the studies consider the flow outside of the nozzle and not much attention is paid to the internal flow of nozzles and its effects on the breakup of emerging jet.

Ashgriz and Mashayek [11] performed temporal analysis of capillary jet breakup numerically. They measured the growth rate of instabilities for different wavelengths and flow parameters and also observed that almost always one or more satellite droplet will be created for each primary drop.

Homma et al. [12] studied laminar axisymmetric breakup of a liquid jet in another immiscible liquid and also considered the mass and/or heat transfer.

Recently Tamaki et al. [13, 14] and Hiroyasu [15] showed that the cavitation inside the nozzle could lead to enhancement of atomization of liquid jet. It is proposed that this enhancement is a result of the disturbance caused by collapse of the cavitation bubbles. In an experiment He and Ruiz [16] observed that when cavitation is present in an orifice flow, the turbulence intensity is much higher that the flow without cavitation but with same Reynolds number.

In this study we will look at the effects of nozzle geometry and nozzle internal flow on the breakup of liquid jet.

Governing Equations and Numerical Method

In this study, we consider flow of a liquid through an orifice and the resulting exiting jet into a stagnant gas. The physical problem and the computational domain and grid are shown in figure 1. Governing equations for an unsteady, incompressible viscous flow are the Navier-Stokes equations:

\[ \rho \left( \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p + \nabla \cdot (2\mu \mathbf{D}) + \sigma \kappa \delta(d) \mathbf{n} \tag{1} \]

\[ \mathbf{D} = \frac{1}{2} \left[ (\nabla \mathbf{u}) + (\nabla \mathbf{u})^T \right] \tag{2} \]

\[ \nabla \cdot \mathbf{u} = 0 \tag{3} \]

where \( \mathbf{u} \) is the velocity, \( \rho \) and \( \mu \) are the fluid density and viscosity, respectively, which could be properties of either liquid or gas phase. \( \mathbf{D} \) is the strain rate tensor. The last term represents the surface tension as a force concentrated on the interface. \( \sigma \) is the surface tension coefficient, \( \kappa \) is the curvature of the interface, \( \delta \) is the Dirac delta function. \( d \) represents the distance from the interface and \( n \) corresponds to the unit normal vector at the interface. The flow is characterized by the density ratio of gas to liquid, viscosity ratio, and the nondimensional parameters, Reynolds number (Re) and Weber number (We), which are defined as follows:

\[ Re = \frac{\rho \mu U D}{\rho \mu q}, \quad We = \frac{\rho \mu U^2 D}{\sigma} \tag{4} \]

Figure 1. Geometry of the axisymmetric orifice and the boundary fitted grid. (flow from left to right)

Here, \( D \) is the orifice diameter and \( U \) is the average velocity of the flow through the orifice.

The numerical solution of the incompressible unsteady Navier-Stokes equations is performed using the finite-volume method on a staggered grid. The convective term is discretized using the Quadratic Upwind Interpolation for Convective Kinematics (QUICK) (by Hayase [17]). The Semi-Implicit Method for Pressure-Linked Equation (SIMPLE), developed by Patankar [18], is used to solve the pressure-velocity coupling. The time integration is accomplished using the second-order Crank-Nicolson scheme.
2 Interface Tracking and Level-Set Formulation

We are considering incompressible flow of two immiscible fluids. The interface between these fluids moves with the local velocity of flow field. To track the motion of the interface, the level-set method is used which has been developed by Osher and coworkers (e.g., [19] and [20]). The level-set function, denoted by $\theta$, is defined as a signed distance function. It has positive values on one side of the interface (gas phase), and negative values on the other side (liquid phase). The magnitude of the level-set at each point in the computational field is equal to the shortest distance from that point to the interface.

The level-set function is being convected by the flow as a passive-scalar variable:

$$\frac{\partial \theta'}{\partial \tau} + u \cdot \nabla \theta = 0$$

It is obvious that, if the initial distribution of the level-set is a signed distance function, after a finite time of being convected by a nonuniform velocity field, it will not remain a distance function. Therefore, we need to re-initialize the level-set function so it will be a distance function (with property of $|\nabla \theta| = 1$) without changing the zero level-set (value at the interface).

Suppose $\theta_0(x)$ is the level-set distribution after some time step and is not exactly a distance function. This can be reinitialized to a distance function by solving the following partial differential equation [19]:

$$\frac{\partial \theta'}{\partial \tau} = \text{sign}(\theta_0)(1 - |\nabla \theta'|)$$

with initial conditions:

$$\theta'(x, 0) = \theta_0(x)$$

where

$$\text{sign}(\theta) = \begin{cases} 
-1 & \text{if } \theta < 0 \\
0 & \text{if } \theta = 0 \\
1 & \text{if } \theta > 0 
\end{cases}$$

and $\tau$ is a pseudo time. The steady solution of equation (6) is the distance function with property $|\nabla \theta| = 1$ and since $\text{sign}(0)=0$, then $\theta'$ has the same zero level-set as $\theta_0$.

Now using the level-set definition, the fluid properties can be defined as:

$$\rho = \rho_{liq} + (\rho_{gas} - \rho_{liq})H_{\varepsilon}(\theta)$$

$$\mu = \mu_{liq} + (\mu_{gas} - \mu_{liq})H_{\varepsilon}(\theta)$$

where $H_{\varepsilon}$ is a modified Heaviside function that has a smooth jump:

$$H_{\varepsilon} = \begin{cases} 
0 & \theta < -\varepsilon \\
(\theta + \varepsilon)/(2\varepsilon) + \sin(\pi\theta/\varepsilon)/(2\pi) & |\theta| \leq \varepsilon \\
1 & \theta > \varepsilon 
\end{cases}$$

where $\varepsilon$ represents the thickness of the interface and has the value of 1.5$h$ where $h$ is the cell size. This Heaviside function corresponds to a delta function that can be used to evaluate the force caused by surface tension:

$$\delta_{\varepsilon} = \begin{cases} 
[1 + \cos(\pi\theta/\varepsilon)]/(2\varepsilon) & |\theta| \leq \varepsilon \\
0 & \text{otherwise} 
\end{cases}$$

The last term in the momentum equation (1) includes the normal unity vector and the curvature of the interface which can be calculated as follows:

$$n = \frac{\nabla \theta}{|\nabla \theta|}, \quad \kappa = -\nabla \cdot n$$

Results and Discussion

In this study we considered the breakup of a liquid jet emerging from a nozzle with curved inlet. Figure 1 shows the geometry of the axisymmetric orifice with the boundary fitted orthogonal grid. No slip condition is applied on the nozzle walls. On the downstream external face of the nozzle a full slip boundary condition is applied on the top boundary and a zero Lagrangian derivative is applied to the exit side of the domain. On the upstream boundary, the mass flux of the liquid is specified. The flow is perturbed by adding a harmonic fluctuation to the mass flux so the average velocity in the nozzle will be

$$U' = U(1 + \epsilon \sin \omega t)$$

where $\epsilon$ is the amount of perturbation and in this study has a fixed value of $\epsilon = 0.05$. The frequency of perturbation is selected in such a way that the wavelength is equal to maximum unstable wave based on linear theory of capillary instability. Therefore

$$T = \frac{2\pi}{\omega} = 4.5\frac{D}{U}$$

Upstream corner is at a distance of $5D$ from the orifice and downstream domain extends to $50D$ downstream of the orifice with a size of $2D$ in the spanwise direction. The grid for inside of the nozzle comes from a potential function and stream function of a potential flow in the same geometry and it is matched with a Cartesian grid downstream of the nozzle. The number of grids in spanwise direction
is 40 and in streamwise direction is 1000. Calculation is performed on a finer grid as well showing the independency of the results to the size of the grid. Calculation is performed for nozzles with different length-to-diameter ratios and different curvature at the upstream corner.

![Figure 2](image1.png)

**Figure 2.** Breakup of liquid jet emerging from a nozzle with $L/D = 1$ and $r/D = 0.04$ at $Re = 500$ and $We = 50$ (a) three instants with $T/3$ time separation, (b) whole jet with aspect ratio of one corresponding to the third part of (a).

Figure 2 shows snapshots of breakup of the liquid jet into droplets, at $Re = 500, We = 50$. The length-to-diameter ratio ($L/D$) is one and the ratio of radius of curvature of the corner to diameter ($r/D$) is 0.04. Here, breakup occurs at $x = 58$. Drops on the downstream of the jet are not shown. A satellite droplet is created during breakup in this case.

For higher Reynolds number figure 3 shows snapshots of breakup of the liquid jet into droplets, at $Re = 2000, We = 50$. Geometry parameters are kept unchanged. Here, breakup occurs earlier (at $x = 45$) because of less viscosity of liquid. An increase in the Weber number causes a longer breakup distance since the growth rates are smaller for lower values of surface tension. Figure 4 shows the same calculation but Weber number has been increased to 200. In this case breakup occurs at $x = 80$. Also in this case the interface is more wavy because of lower surface tension, more deformation on the interface is possible.

The effects of nozzle geometry on the breakup of liquid jet are also studied. Flow through nozzles with different length-to-diameter ratios and different radii of curvature of upstream corner are simulated. For example, figure 5 shows the flow through nozzle with $L/D = 4$ and $r/D = 0.04$ at $Re = 2000, We = 50$. It can be seen that the breakup point has been moved further downstream.

The effects of radius of curvature of the upstream corner of the orifice is investigated by simulating a flow through a nozzle with larger $r/D$ ratio. In figure 6, breakup of a jet though a nozzle with $r/D = 1$ and $L/D = 1$ can be seen and figure 7 shows the results of a nozzle with $r/D = 1$ and $L/D = 4$. Again, the breakup of the liquid jet in the nozzle with shorter length occurs earlier compared to the longer nozzle. The velocity profile in the nozzle varies for different geometries causing differences in the shape of the jet and breakup properties. Figure 8 shows the velocity profile at the nozzle exit at two instants when the flux has the maximum value and when it has the minimum value. Longer nozzles lead to a more stable jet since the profile develops in the nozzle and changes become more uniform. How-
ever in a short nozzle the changes in the orifice due to oscillation of flux, reach downstream jet without suppression and causes a sharper wave form in the jet interface.

3 Conclusion

The axisymmetric laminar flow of a liquid through an orifice and breakup of the resulting jet is simulated for different geometries of the nozzle. Effects of Reynolds number, Weber number and geometrical parameters such as length-to-diameter ratio and curvature of corners are studied. It is observed that longer nozzles create a more stable jet with longer breakup distances.

Nomenclature

- $D$: orifice diameter
- $L$: orifice length
- $p$: pressure
- $r$: radius of curvature of the orifice inlet
- $Re$: the Reynolds number
- $t$: time
- $T$: periodic time
- $u$: velocity vector
- $U$: average jet velocity
- $\Delta U$: amplitude of velocity modulation
- $We$: the Weber number
- $\epsilon$: amplitude of velocity oscillation
- $\lambda$: wavelength
- $\mu$: viscosity
- $\rho$: density
- $\sigma$: surface tension
- $\theta$: level-set function

Subscripts

- gas: gas phase
- liq: liquid phase

References


Figure 6. Breakup of liquid jet emerging from a nozzle with $L/D = 1, r/D = 1$ at $Re = 2000, We = 50$


Figure 7. Breakup of liquid jet emerging from a nozzle with $L/D = 4, r/D = 1$ at $Re = 2000, We = 50$


Figure 8. $U$-velocity profiles at nozzle exit at the maximum and minimum flux instants. left: $L/D = 1$, right: $L/D = 4$