Numerical Study of Laminar Annular Two-Phase Flow in Effervescent Atomizers

C.K. Mohapatra and M.A. Jog*
Department of Mechanical Engineering
University of Cincinnati
Cincinnati, OH 45221-0072 USA

Abstract
Computational simulations of two-phase laminar flow in effervescent atomizers have been carried out. The flow is considered in the annular flow regime. A non-uniform, structured computational grid of 48000 cells is used for an axisymmetric effervescent atomizer consisting of a mixing chamber, convergence section and an orifice section. Volume of Fluid (VOF) model is used to investigate the water-air two-phase flow. The flow is considered compressible and the formulation is axisymmetric. The Gas-to-Liquid Ratio or GLR is varied from 0.005 to 0.07, with a liquid flow rate variation from 0.14 l/min to 0.27 l/min. For each case, the liquid sheet thickness and velocity at the orifice exit are obtained from the numerical solutions. At a constant liquid flow rate, the liquid sheet thickness varies inversely with gas-to-liquid ratio (GLR). Moreover the decrease in the sheet thickness is sharper at lower values of GLR and the variation becomes more gradual as GLR increases. Furthermore, an increase in liquid flow rate results in an increase in the sheet thickness. With an increase in the orifice diameter, non-dimensionalized sheet thickness increases. Sheet thickness varies marginally with changes in orifice length, angle of the convergence section, and liquid surface tension. Based upon the computational study, an empirical correlation to predict the sheet thickness as a function of the exit Reynolds number and GLR is proposed.

*Corresponding Author: Milind.Jog@uc.edu
Introduction

Atomizer is a device which disintegrates bulk liquid mass into small liquid spray droplets in a gaseous atmosphere. Effervescent atomizer is a special kind of atomizer which works on the principle of bubbling small amount of gas into the liquid phase inside the atomizer. This principle was first proposed by Lefebvre and co-workers\cite{1}, \cite{2}, \cite{3}, and \cite{4} in late 1980s. Subsequently, a significant amount of experimental work has been carried out on effervescent atomizer performance.

In an effervescent atomizer air is injected into the liquid at a slightly higher pressure just before mixing chamber. Then according to the flow rates of atomizing gas and liquid, the two phase flow inside the mixing chamber evolves into different flow regimes such as bubbly, slug or annular. Afterwards both phases pass through a nozzle and exit through an orifice to gaseous ambiance and form small spray droplets. The presence of air bubbles helps in reducing liquid sheet thickness and aids in sheet breakup, thereby providing an edge over other atomizers.

Over the last two decades many researchers have investigated the effect of flow and geometrical parameters on the atomizer performance characteristics. Spray characteristic for any atomizer is generally defined by Sauter mean diameter (SMD), spray velocity distribution, spray cone angle, and jet momentum rate. Liquid flow rate, GLR, injection pressure are the most important flow operation parameters, which govern the spray performance characteristics\cite{5}. Additionally, liquid properties such as density, viscosity, and surface tension and geometrical parameters such as mixing chamber diameter, nozzle angle, orifice diameter, length to diameter ratio (L/D ratio) and shape are also some of the parameters that affect spray quality. Fig. 1 summarizes all the dependent and independent parameters of effervescent atomizer.

Several researchers\cite{1}, \cite{3}, \cite{4}, \cite{6}, and \cite{7} have reported that SMD decreases non-linearly with GLR. At low GLR with increase in GLR SMD decreases rapidly, but after certain value of GLR(around 0.03) the reduction in SMD becomes gradual with increase in GLR. Whitlow et al. \cite{6} have plotted the variation of SMD with GLR at different injection pressure. Their data has been graphed in Fig. 2.

Effect of liquid physical properties like viscosity and surface tension on spray mean diameter has been studied by a few researchers. Buckner and Sojka\cite{8}, and \cite{9} found no variation in SMD with fluids of different viscosity. However, Lund et al.\cite{10} and Sutherland et al.\cite{11} found the SMD to vary with viscosity, but the variation was minimal. Similarly the observation for surface tension effect is also somewhat contradictory. Lund et al.\cite{10} found the SMD to decrease with increasing surface tension, whereas Sutherland et al. \cite{11} found the variation to be within the experimental uncertainty. This contradiction in observation can be attributed to difference in setup of the atomizer investigated in their respective experiments. Sutherland et al. \cite{11} had a ligament control insert in their atomizer, whereas Lund et al.\cite{10} did not have any kind of insert.

The influence of atomizer’s geometrical parameters such as orifice diameter, L/D ratio, nozzle angle and mixing chamber diameter on spray performance has been widely explored. In\cite{1}, \cite{3}, \cite{8} and \cite{9} little influence of orifice diameter on SMD has been noted. However, Wang et al.\cite{4} observed smaller SMD with smaller orifice diameter and higher SMD with higher orifice diameter. The effect of nozzle angle and orifice L/D ratio was studied by Chin and Lefebvre \cite{12}. They observed no effect of nozzle angle on SMD as long as it was less than 120°. They also found a little effect of L/D ratio on the SMD in bubbly flow regime.

The internal flow dynamics, which drives the spray characteristic has not been explored in the literature because of the difficulty in observing the internal characteristics experimentally. Lin et al.\cite{13} developed a correlation for sheet thickness based upon liquid and gas flow rates after studying the internal flow characteristics of a rectangular shaped effervescent atomizer with L/D ratio of 20. But, the expression did not include important geometrical parameters and Reynolds number.

Unlike experimental work, there has been only
a limited number of computational studies for effervescent atomizers. Ramamurthi et al.\cite{14} first developed a 2-D model for effervescent atomizer to study different flow regimes, but their computational results lacked the agreement with their experiments. Esfarjani et al.\cite{15} developed a 3-D model for rectangular shaped effervescent atomizer with a L/D ratio of 20 for plasma spray and found the sheet thickness to vary non linearly with GLR. They also found no effect of nano-particle concentration on the sheet thickness.

Atomizers employed in automotive and aerospace industries for fuel injection operate in high pressure, high temperature environment. In these cases investigating the spray performance become difficult because of the elevated ambient pressure and temperature conditions in the combustor. To overcome the above difficulties it is essential to investigate internal flow physics of the atomizer, which is difficult to measure experimentally. Hence we have chosen to perform a numerical investigation of the internal flow characteristics of effervescent atomizer.

**Numerical Methodology**

It is required to have an accurate liquid-gas interface tracking while modeling the two phase flow inside the effervescent atomizer, as the interface is very sensitive to pressure fluctuation. The Volume of Fluid (VOF) method with finite volume formulation has been used to solve the internal two phase flow. To simplify the problem; the flow domain has been considered from the mixing chamber, when the flow has been developed into an annular flow. A cylindrical geometry consisting of mixing chamber, nozzle and orifice has been modeled. For further simplification a structured grid with different spacing at different region for a 2-D axisymmetric flow domain has been created using blockmesh. Fig. 3 represents the schematic of the flow domain with the dimensions and the zoomed view of the grids in mixing chamber, nozzle and orifice region.

*Figure 2.* Experimental Variation of SMD with GLR. Data taken from Whitlow et al.\cite{6}.

*Figure 3.* Schematic representation of flow domain and grids.

For OpenFOAM a wedge shaped grid of $5^\circ$ wedge angle with single cell spanning in Z-direction has been created. The grid spacing in the mixing chamber is $50\mu m \times 50\mu m$. In the nozzle section the grid spacing varies from $50\mu m \times 50\mu m$ to $18.75\mu m \times 50\mu m$ and in the orifice region it is fixed at $18.75\mu m \times 18.75\mu m$. Y-axis has been chosen as the axis of the atomizer.
**Governing Equation**

CompressibleInterFOAM; the VOF solver of OpenFOAM has been used for all the simulations. Interface fluctuation because of the inlet pressure fluctuation has compelled us to choose a compressible multiphase solver for our study. Water has been treated as incompressible fluid, whereas air has been treated as perfect gas. In this method a single set of equation for continuity, momentum and energy are solved, which are mentioned below.

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U) = 0 \tag{1}
\]

\[
\frac{\partial (\rho U)}{\partial t} + \nabla \cdot (\rho U U) = -\nabla p + \nabla \cdot (\mu \nabla U) + S_U \tag{2}
\]

\[
\frac{\partial (\rho C_pT)}{\partial t} + \nabla \cdot (\rho U C_pT) = \nabla \cdot (k \nabla T) + S_T \tag{3}
\]

In the above equation \( \rho, \mu, U, T \) represents density, coefficient of viscosity, velocity field vector and temperature field shared by both the fluids respectively. \( S_U \) and \( S_T \) in the momentum and energy equation represent the source terms in corresponding equation. The multiphase solver of OpenFOAM employs a special parameter denoted as \( \alpha \), which represents the phase fraction and carries a value ranging from 0 to 1. \( \alpha = 0 \) represents one phase whereas \( \alpha = 1 \) represents the other phase and \( 0 < \alpha < 1 \) represents the interface. The solver considers the two immiscible fluid as one effective fluid through out the domain and calculates the physical properties based upon phase fraction and individual fluid properties shown in (4), (5).

\[
\rho = \rho_1 \alpha + \rho_g (1 - \alpha) \tag{4}
\]

\[
\mu = \mu_1 \alpha + \mu_g (1 - \alpha) \tag{5}
\]

To avoid unwanted smearing of the interface the CompressibleInterFOAM solver uses extra compression term and a source term based upon the pressure correction equation of previous time step, in the phase fraction transport equation. The phase transport equation is mentioned below.

\[
\frac{\partial \alpha}{\partial t} + \nabla \cdot (U \alpha) + \nabla \cdot (U_c \alpha (1 - \alpha)) = -\frac{\alpha D_p}{\rho_l} \frac{Dp_l}{Dt} \tag{6}
\]

In (6) \( U_c \) is the relative velocity vector, which is defined in (7).

\[
U_c = U_l - U_g \tag{7}
\]

\[
U = \alpha U_l + (1 - \alpha) U_g \tag{8}
\]

All the above governing equations are subjected to certain boundary conditions mentioned in Table 1.

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Boundary Condition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air Inlet</td>
<td>( u_a = \text{constant}, T = \text{constant} )</td>
</tr>
<tr>
<td>Water Inlet</td>
<td>( u_w = \text{constant}, T = \text{constant} )</td>
</tr>
<tr>
<td>Atomizer Wall</td>
<td>( u = 0, T = \text{constant} )</td>
</tr>
<tr>
<td>Axis</td>
<td>Symmetry</td>
</tr>
<tr>
<td>Outlet</td>
<td>( p = \text{constant} )</td>
</tr>
</tbody>
</table>

**Table 1. Boundary Conditions.**

**Verification and Validation**

A grid independent study has been performed to verify the accuracy of the CFD methods. Three different sizes of grid have been chosen. Grid type-I has a grid variation from 100 \( \mu m \times 100 \mu m \) to 37\( \mu m \times 37 \mu m \). However, Grid type-II and type-III have variations from 50 \( \mu m \times 50 \mu m \) to 18.75 \( \mu m \times 18.75 \mu m \) and from 25 \( \mu m \times 25 \mu m \) to 9.375 \( \mu m \times 9.375 \mu m \) respectively. For the grid independent study the base atomizer geometry of 45° nozzle angle, 3mm orifice diameter(\( D_o \)) with a L/D ratio of 0.5 has been simulated for a water flow rate (\( Q_w \)) of 0.23 l/min. To attain different GLR the air flow rate(\( Q_a \)) has been varied from 0.9 l/min to 13.5 l/min. The variation of sheet thickness with GLR for different grid type have been compared and have been plotted in the Fig. 4. From the figure it can be inferred our simulation is free of any kind of numerical error. Between type-I and type-II grid maximum difference is 3%, while the difference between type-II and type-III is less than 2%.

For validation of our CFD analysis, computational results have been compared with existing experimental results. We have used the formulation developed by Lund et al. [10] to calculate SMD \( \text{SMD}_n \) based upon ligament diameter. For an annular flow the sheet thickness can be treated as ligament diameter. However, the formulation represented in (9) calculates the approximate SMD value after primary atomization.

\[
\text{SMD}_n = \left[ \frac{3}{2} \sqrt{2\pi} D^2 \left( 1 + \frac{3\mu}{\sqrt{\rho_l \sigma} D} \right)^{0.5} \right]^\frac{1}{2} \tag{9}
\]
In the above formulation (9) $D_L$, $\mu_l$, $\rho_l$ and $\sigma$ represents ligament diameter, liquid viscosity, liquid density and surface tension respectively.

The $SMD_n$ values at different GLR have been non-dimensionalized by the $SMD_n$ calculated at GLR = 0.005. Similarly experimental SMD values by Whitlow et al. [6] have been non-dimensionalized by the SMD value measured at GLR=0.005. The comparison of the variation of non-dimensionalized $SMD_n$ and the non-dimensionalized experimental SMD value with different GLR have been plotted in the Fig. 5. From the Fig. 5 it is evident that our numerical results follow the same trend as experimental findings. Numerically calculated SMD value have been found to be slightly higher than the corresponding experimental counterpart. Experimentally the SMD is measured after secondary atomization, but the formulation (9) is based upon primary atomization, which might explain the difference between the two.

Results and Discussion

We have studied the variation of sheet thickness with GLR ranging from 0.005 to 0.07 for liquid flow rate ranging from 0.14 l/min to 0.27 l/min and air flow rate varying from 0.75 l/min to 15.8 l/min. The variation of sheet thickness with GLR has been found to be nonlinear. For lower GLR sheet thickness decreases rapidly but the reduction becomes gradual at higher GLR. We believe this variation of sheet thickness is driving the SMD variation. We have also observed a reduction in sheet thickness with increase in liquid flow rate at a constant GLR. This happens because to maintain a constant GLR air flow rate needs to be increased for an increment in liquid flow rate. Fig. 6 summarizes the variation of non-dimensionalized sheet thickness with liquid flow rate and GLR. The sheet thickness has been non-dimensionalized by the orifice diameter ($D_o$).
till 90°. However, sheet thickness has been found to
decrease slightly for nozzle angle 120°. This phe-
nomena can be attributed to the unstable interface
at the throat for nozzle angle 120°. The variation
has been summarized in Fig. 7.

Fig. 9 represents the effect of L/D ratio of orif-
ce on sheet thickness. We have investigated L/D
ratio of 0.5, 1 and 2 keeping the orifice diameter
constant at 3mm. From the Fig. 9 it is clear that
the change in L/D ratio from 0.5 to 1 does not have
any influence on the sheet thickness. However, for
L/D ratio of 2 there is a reduction in sheet thickness.
We have observed for longer orifice i.e. L/D ratio of
2 the multiphase flow inside the orifice fluctuates a
lot. This might be because of pressure loss in longer
orifice.

We have also investigated the effect of sur-
face tension on sheet thickness. For our study we
have chosen water-air and Sodium Dodecyl Sulphate
(SDS) solution-air. Though both the liquid have
same viscosity, but they have different surface ten-
sion. Water-air has a surface tension of 0.072N/m,
whereas SDS-air has 0.038N/m. The variation of
sheet thickness with surface tension has been repre-
sented in Fig. 10. We have not found any effect of
surface tension on the sheet thickness.

From the above analysis we can infer, sheet
thickness is clearly a function of GLR, liquid flow
rate and orifice diameter. Superficial Reynolds
number(Re_s) which is defined in (10) is a function
of liquid mass flow rate and orifice diameter. Hence
based upon all our results, we have performed a re-
gression analysis using MATLAB and have proposed
a correlation involving non dimensional sheet thick-
ness, GLR and superficial Reynolds number.
Figure 10. Variation of sheet thickness with GLR and surface tension

\[ Re_s = \frac{4\dot{m}}{\pi D_o \mu_{avg}} \]  

(10)

In (10) \( \dot{m} \) represents the total mass flow rate through the orifice and \( \mu_{avg} \) is the average viscosity, calculated based upon void fraction (\( \gamma \)).

\[ \mu_{avg} = \gamma \mu_w + (1 - \gamma) \mu_a \]  

(11)

\[ \gamma = \frac{m_w}{m_w + m_a} \]  

(12)

Based on the computational results, a correlation has been proposed for the sheet thickness in (13). The correlation has a \( R^2 \) value of 0.97. The correlation (13) is valid for annular flow regime with a GLR range of 0.005-0.07, Superficial Reynolds number ranging between 1120-3020. The nozzle angle need to be less than 120° and the L/D ratio less than or equal to 1.

\[ \frac{t}{D_o} = 0.52GLR^{-0.38}Re_s^{-0.32} \]  

(13)

Conclusions

We have investigated the effect of different input flow parameters, atomizer geometrical parameters and liquid properties on the sheet thickness. Based upon our CFD simulations we have found GLR, Liquid flow rate, orifice diameter are the prime parameters which govern the annular sheet thickness at the exit. We have observed the liquid sheet thickness to decrease non-linearly with increase in GLR. We have also witnessed a reduction sheet thickness for an increment in liquid flow rate for a constant GLR. Sheet thickness has also been found to increase with increase in orifice diameter at a constant GLR. Based on the numerical results, a correlation for sheet non-dimensionalized sheet thickness as a function of GLR and superficial Reynolds number at the exit plane has been proposed.

Nomenclature

- \( D \) Diameter
- \( k \) Thermal conductivity
- \( m \) Mass flow rate
- \( p \) Pressure
- \( S \) Source term
- \( t \) Sheet thickness
- \( T \) Temperature
- \( u \) Axial velocity
- \( U \) Velocity vector
- \( Re \) Reynolds Number
- \( \dot{m} \) Mass flow rate through orifice
- \( \rho \) Density
- \( \mu \) Viscosity
- \( \alpha \) Phase fraction
- \( \sigma \) Surface tension
- \( \gamma \) Void fraction

Subscripts

- \( g \) gas
- \( l \) liquid
- \( w \) water
- \( a \) air
- \( o \) orifice
- \( L \) ligament
- \( n \) numerically calculated
- \( s \) superficial
- \( avg \) average

References


