Dynamics and Stability of Impinging Jets

Xiaodong Chen*, Dongjun Ma and Vigor Yang
School of Aerospace Engineering
Georgia Institute of Technology
Atlanta, GA 30332, USA

Abstract

High fidelity numerical simulations are performed to study the dynamics and stability of impinging jets over a broad range of operating conditions. An improved volume-of-fluid (VOF) method augmented with adaptive mesh refinement (AMR) is used to simulate the formation and breakup of the liquid sheet formed by two impinging jets. The behaviors in various Reynolds and Weber number regimes are studied systematically. The predicted liquid sheet topology, atomization, and droplet size distribution agree well with experimental measurements. Several different patterns of sheet and rim configurations are obtained, including liquid chain, closed rim, fish-bone, disintegrating sheet, disintegrating rim and impact wave. Characteristics of stable liquid sheets are analyzed to shed light into the underlying physics. The instability mechanisms of sheets and rims are studied based on the concepts of absolute and convective instabilities. New knowledge is acquired about the onset of sheet and rim instabilities. In addition, stationary asymmetrical waves are observed and compared with existing theories. The effects of the velocity profile and vorticity distribution on the development of the impact waves are investigated in detail. The shear layers near the liquid sheet surfaces result in wave motions. The subsequent interaction between the two liquid surfaces causes strong resonance at well characterized frequencies. Within the parameter region of practical interest, the ratio of the wavelength to the jet diameter remains independent of the jet velocity, liquid viscosity and surface tension.

*Corresponding author: xiaodong.chen@aerospace.gatech.edu
1. Introduction

Collision between two cylindrical jets is one of the generic configurations for the generation of liquid sheets, the dynamics and stability of which have attracted a great deal of attention due to their relevance to the spray atomization and combustion in liquid propellant engines. [1–4] The impingement of liquid jets is a very efficient method for atomization and mixing whereby the dynamic head of the propellant is used to destabilize an opposing liquid propellant stream. This in turn results in fragmenting the liquid into ligaments and droplets. [5] The oblique collision of two cylindrical laminar jets at low jet velocities leads to the liquid flowing outward from the impingement point, producing a leaf-shaped expanding sheet which lies in a plane perpendicular to the plane containing the two liquid jets. A rich variety of flow structures, from single oscillating jet obtained at low flow rates, to the violent disintegration of flat sheets obtained at higher flow rates, have been observed depending on the Weber and Reynolds numbers of the jets. In the specific usage of liquid propellant rocket engine, the liquid sheet shows full-developed violent breakup with quickly growing waves. The liquid sheet destabilizes, breaks and eventually disintegrates into ligaments or droplets under the influence of surface tension, viscous, inertial, and aerodynamic forces. The impinging jets can provide rapid mixing and atomization. This unstable hydrodynamic wave is usually called impact wave. [6] Impact wave dominates the breakup and atomization process in most rocket combustors using impinging jet injectors. [7]

Different from another capillary wave, impact wave shows a nonlinear behavior that cannot be described linear stability analysis.

There are numerous experimental and theoretical studies on the mechanism of impinging jets atomization. Taylor [8] produced several definitive papers on the formation, breakup and disintegration of flat sheets by the collision of two equal coaxial water jets. Dombrowski and co-workers [6] conducted extensive experiments to analyze the factors influencing the breakup of sheets and made significant contributions to the understanding of wave motions of high velocity liquid sheets. Huang [9] examined the breakup mechanism of axisymmetric liquid sheets, and analyzed the speed of antisymmetric waves propagating on the sheets. Lin [10] has provided a detailed theoretical review of the literature on the sheet instability and introduced the concept of absolute instability for sheet breakup. Some analytical studies have resulted in the development of models for thickness [11–13], shape [11–13] and velocity [13] of the liquid sheet, as well as the size of the droplets detaching from the rim of the liquid sheet [4]. Extensive experimental studies have also been focused on the thickness [8, 14], shape [8, 12, 13, 15], and velocity [15–17] of the liquid sheet and on the droplet shape and size [13, 18]. Various flow patterns have been investigated by experiments [4, 19, 20] in a wide range of Reynolds and Weber numbers.

Dombrowski et al. [6] studied the factors influencing the break-down of sheets formed by the impingement of two liquid jets. It was shown that disintegration generally results from the formation of unstable waves of aerodynamic or hydrodynamic origin. While the characteristics of the former waves were fairly well understood, little is known about the latter. The results of their study indicated that hydrodynamic (or “impact”) waves are generated when the Weber number of each jet is above a critical value, and that their formation is independent of the Reynolds number. It was found that for the turbulent jets, the critical value of the Weber number (ρDU sinθ/σ, here ρ is the liquid density, D the jet diameter, U the mean jet velocity, θ the half-angle of impingement, and σ is the surface tension), above which impact waves are produced lies between 66 and 165. This may be compared with the range of 84 to 126 calculated from the results of Heidman et al. [19] Taylor [8] has studied the impingement of laminar jets with flat velocity profiles and found that aerodynamic waves only were produced with 0.227 cm diameter jets impinging at an angle of 60° at a head of 155 cm water. The jets were produced from sharp-edged orifices and caused to impinge downstream of the vena-contrata. The jet diameters at impingement were not recorded and it is thus not possible to accurately calculate the appropriate Weber number. However, approximate measurements made from Taylor's photographs indicate that undisturbed sheets were produced with a Weber number of 134 at an orifice Reynolds number of 6900. Drop sizes have been measured and are shown to be critically dependent upon the mechanism of disintegration.

Anderson et al. [7] conducted an experiment to characterize the formation and effect of impact waves on the atomization process. The model flow consisted of opposed turbulent water jets at atmospheric conditions. The impact waves were formed with a characteristic wavelength of about one jet diameter. A computational study of the flow structure around the stagnation point were carried out and showed that the effects of impingement extend about one jet diameter upstream and that maximum gradients and incipient disruption of the surface occur at a normalized radius of 1.2, where an inflection in the jet flow from predominately axial to predominantly radial occurs. Using these observations and measurements, and existing correlations for breakup length drop size, a three-step phenomenological model of atomization (impact wave formation and propagation, sheet breakup into ligaments, and ligament disintegration into droplets) was developed.
Ibrahim[21] employed a second-order non-linear perturbation analysis to predict the characteristics of the spray produced by atomization of an attenuating liquid sheet formed by the impingement of two liquid jets of equal diameters and momenta. The evolution of harmonic instability waves that lead to sheet distortion and fragmentation was modeled. The onset of atomization occurs when the uneven surface modulations of the thinning sheet bring its upper and lower interfaces in contact. It was found that the sheet is torn into ligaments at each half wavelength. The instability of the ligaments causes their eventual disintegration into drops. The results indicated that sheet breakup length, time, and resultant drop size decrease as Weber number is increased. The theoretical predictions of the present non-linear model were in good agreement with available experimental data and empirical correlations for sheet breakup length and drop size. However, the results offered no assurance that aerodynamic-based models can be used to render reliable predictions of turbulent impinging-jet atomization at high Weber.[22]

Choo et al.[16] analyzed the velocity characteristics of liquid elements (ligaments for high-speed jets and droplets for low-speed jets) formed by two impinging jets using a double-pulse image capturing technique. The magnitude of the maximum velocities of ligaments and droplets observed around the axis of the spray was close to the corresponding jet velocity, which is consistent with the previous assumption that the velocities of liquid elements are equal to the jet velocity. The shedding angles of liquid elements increased linearly but at a slightly lower value than the azimuthal angle, which implies that the direction of movement of the liquid elements is almost disposed radially about the impingement point.

Ashgriz et al.[23] conducted an experimental to provide some understanding of mixing processes and the mechanism of the mixing in impinging jet atomizers. Both miscible and immiscible liquid jets were tested. The volume fraction profiles were used to characterize the mixing processes and determine the quality of mixing. Results showed that two distinct processes control mixing: 1) the ability of the two jets to redirect each other on impact and before atomization (preatomization process) and 2) the turbulent dispersion in the spray region (postatomization process). The preatomization process can result in two types of atomizations: reflective and transverse atomization. In the reflective atomization, the jets basically bounce off of each other, causing the fluid of each jet to remain on the same side of the jet. In the transverse atomization, the direction of the momentum does not change completely, and the fluid of each jet flows to the side of the other jet. On the other hand, the turbulent dispersion in the postatomization region improves the mixing.

Arienti et al.[24] presented two simulations of like-on-like jet impingement at low and high injection velocity of relevance to liquid rocket engine combustion using a multiphase computational fluid dynamics code utilizing a coupled level-set/volume-of-fluid method to simulate liquid fuel atomization. The coupled approach combined the mass conservation properties of the volume-of-fluid (VOF) method with the accurate surface reconstruction properties of the level-set (LS) method, and includes surface tension as a volume force calculated with second-order accuracy. Extensions to the model included coupling to a Lagrangian dispersed phase model for postbreakup tracking of droplets and multiple level-sets for tracking of surfaces and droplets of multiple species. However the mechanism of impact wave was not well addressed.

In the present paper, high fidelity numerical simulations have been performed to study the atomization patterns and breakup characteristics of turbulent impinging jets. A three-dimensional Volume-Of-Fluid (VOF) method with adaptive mesh refinement (AMR) based on octree meshes is used to simulate the primary atomization. The high fidelity numerical setup allows one to vary the dominant physical parameters individually so that numerical simulations can be performed as “numerical experiments”. This method is useful to identify the parameters that most strongly influence primary breakup phenomena. Two benchmarks for low velocity and high velocity impinging jets are investigated as the standard numerical assessment of verification and validation. Basing on experimental results, dynamic process of impact wave is estimated which seems to be arbitrary and physical details are not investigated because of the limitations by experimental tools. Numerical simulations give a method to look into the dynamic of flow field. This paper focus on apply an accuracy and low-cost method to impinging jets simulation and explore the physical detail of impinging jet atomization.

2. Numerical Method

Atomization process involves large density ratios of gas and liquid phases, high surface-tension and low viscosity at practical length scales. When impact wave tends to unstable and furthermore most of these flows tend to generate complex and evolving interface geometries on spatial scales ranging over several orders of magnitude. The numerical method[25] combining an adaptive quad/octree spatial discretisation, geometrical Volume-Of-Fluid interface representation, balanced-force continuum-surface-force surface tension formulation and height-function curvature estimation. The method is shown to recover exact equilibrium (to machine accuracy) between surface tension and pressure gradient in the case of a stationary droplet, irrespective of viscosity and spatial resolution.
2.1 Theoretical formulation

The three-dimensional incompressible, variable-density, conservative equations with surface tension can be written as:

\[ \rho(\partial_t \mathbf{u} + \mathbf{u} \cdot \nabla \mathbf{u}) = -\nabla p + \nabla \cdot (2 \mu \mathbf{D}) + \sigma \kappa \delta_n \]  
(1)

\[ \partial_t \rho + \nabla \cdot (\rho \mathbf{u}) = 0 \]  
(2)

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

where \( \mathbf{u} = (u,v,w) \) is the fluid velocity, \( \rho = \rho(x,t) \) is the fluid density, \( \mu = \mu(x,t) \) is the dynamic viscosity, and \( \mathbf{D} \) is the deformation tensor defined as \( \mathbf{D} = \frac{1}{2}(\mathbf{\partial u} + \mathbf{\partial v}) \). The Dirac delta function \( \delta_n \) expresses the fact that the surface tension term is concentrated on the interface; \( \sigma \) is the surface tension coefficient, \( \kappa \) and \( n \) the curvature and normal to the interface, respectively.

A volume-of-fluid (VOF) function \( c = c(x,t) \) is introduced to trace the multi-fluid interface. It is defined as the volume fraction of a given fluid in each cell of the computational mesh, and the density and viscosity can be defined as

\[ \rho(\bar{c}) = \bar{c} \rho_1 + (1 - \bar{c}) \rho_2 \]  
(4)

\[ \mu(\bar{c}) = \bar{c} \mu_1 + (1 - \bar{c}) \mu_2 \]  
(5)

with \( \rho_1, \rho_2 \) and \( \mu_1, \mu_2 \) the densities and viscosities of the first and second fluids, respectively. Field \( \bar{c} \) is either identical to \( c \) or is constructed by applying a smoothing spatial filter to \( c \). Better results were obtained if a smoothed field was used to define the viscosity, for cases with high liquid/gas density ratio. When spatial filtering is used, field \( \bar{c} \) is constructed by averaging the eight (in 3D) corner values of \( c \) obtained by bilinear interpolation from the cell-centered values. When spatial filtering is used, the properties associated with the interface are thus “smeared” over three discretisation cells[25].

According to mass continuity, the advection equation for the density can then be written as an equivalent advection equation for the volume fraction:

\[ \partial_t \overline{c} + \nabla \cdot (\overline{c} \mathbf{u}) = 0 \]  
(6)

2.2 Volume-of-fluid advection scheme

In order to solve the advection equation for the volume fraction, a piecewise-linear geometrical VOF scheme[26] is used based on the quad/octree spatial discretization. Interface reconstruction algorithm is performed using a piecewise-planar interface representation in each cell. Once interface reconstruction has been performed, direction split geometrical fluxes can be computed easily on regular Cartesian grids. The resulting advection scheme preserves sharp interfaces and has been shown to be close to second-order accurate for practical applications.

The accurate estimation of the surface tension term \( (\sigma \kappa \delta_n)_{n=1/2} \) in the discretized momentum equation has proven to be one of the most difficult aspects of the application of VOF methods to surface-tension-driven flows. The original Continuum-Surface-Force (CSF) approach[26] is known to suffer from problematic parasitic currents when applied to the case of a stationary droplet in theoretical equilibrium. The combination of a balanced-force surface tension discretization and a Height-Function curvature estimation can be used to solve this problem[25].

2.3 Simulation setup

The like doublets impinging jet with two streams of the same propellant impinge on each other is the common type of impinging jet injector. In this paper, this simplest combination is used. Figure 1 shows the simulation setup with schematic diagram showing low speed impingement. Two equal-diameter liquid jets impinge to form thin liquid sheet. The angle between the centers of the jets, \( 2\alpha \) is called impingement angle. Two important non-dimensional parameters are Weber number, \( \mu u_0 d_j / \sigma \), and Reynolds number, \( \mu u_0 d_j / \mu \), where \( \rho \) is the liquid density, \( d_j \) the jet diameter, \( u_0 \) the mean jet velocity, \( \sigma \) is the surface tension, and \( \mu \) is viscosity of liquid.

\[ \text{Figure 1. Schematic diagram of doublet impinging jets system.} \]

2.4 Refinement criterion

In order to resolve the multi-scale phenomenon of sheet breakup and atomization, the overall scheme allows for space and time-varying grid resolution. The adaptive mesh refinement and coarsening technique is used based on the quad/octree spatial discretization. To simplify the implementation the sizes of neighboring cells cannot vary by more than a factor of two. One of
the advantages of the octree discretization is that mesh refinement or coarsening are cheap and can be performed at every time-step if necessary, which has a minor impact on overall performance. Interpolation of quantities on refined or coarsened cells is also relatively simple on regular Cartesian mesh and is done conservatively both for momentum and volume fraction[25]. Several refinement criteria can be used simultaneously depending on the problem. Since AMR using multiple levels mesh, cares are requested to the refinement criteria in order to ensure that the result is accuracy and close to physics.

In present paper, three refinement criteria are applied to the region of interesting. There are gradient-based, value-based and thickness-based refinements. An important task during simulation is to ensure that the interface carrying surface tension energy is adequately resolved. Gradient-based is used to ensure a fine grid on the interface to avoid overall refinement of the simulation domain. Similarly, a fine mesh is required to avoid excessive dissipation of kinetic energy due to numerical viscosity. According to the value of volume fraction of liquid, the interior of the liquid phase is refined. As knowing that sheet thickness changes temporally and spatially, it can be expensive to refine all the interface grid according to the minimum grid requirement. Ideally, a thickness-based refinement should be robust to solve the multi-scale problem. An experimental refinement method adapted from digital topologic theory[27] is extended and implemented in this paper to ensure that there are at least two grids in the thickness direction dynamically at every time-step. It should be noticed that although the thickness-based method can save computational cost, it is still expensive to apply it to all the simulation since time-step can be much small when the minimum cell size is small.

3. Grid Independent Study

3.1 Stable liquid sheet

Since the sheet shape, rim size and liquid velocity field can be quantified and represented experimentally[13], a case with low impinging velocity and resulting stable liquid sheet is simulated as the first benchmark. The liquid is glycerin-water solutions. The non-dimensional parameters are Reynolds number 40.4 and Weber number 58.8 with impinging angle of 89°. The parameters are selected such that the flow pattern is a closed flat sheet. A stable sheet exists due to the equilibrium between the surface tension force, centrifugal force and inertia on the edges of the sheet. Since the stable shape shows a Weber number similarity[9, 28, 29], the experimental data normalized by $dW_e$ for the same impact angle[13] can be used to validate the simulation results.

As a benchmark case, only the first two refinement criteria are used for simplify. Table 1. Lists all the verification cases with different grid refinement levels. The basic grid level is 5 for all the cases, and the small gas phase structures will be resolved under this resolution. Among the four cases, the interfacial resolution is set to be one level higher than that in liquid phase. Since the simulation domain is only refined locally and adaptively, both the computer memory and computation time are substantially reduced compared to the uniform-mesh method with the same effective resolution. For example, with maximum refinement level of 9, only about 1,077,182 cells are used, while a total of $(2^9)^3$= 134,217,728 cells are required for uniform meshes for the same resolution. The grid number ratio is about 0.008. It is a significant effect of adaptive mesh refinement. If with the proposed thickness-based refinement method, the saving can be even more significant.

Figure 2 shows the different front views of numerical verifications for low-velocity impinging jets and their comparison with the rim curve from experimental result[13], where realistic ray-tracing rendering has been used to visualization of the interface. With maximum refinement level of 7, the simulation fails to resolve full liquid sheet. The grid size on the interface is too large to capture the thin film. The liquid sheet cannot sustain the equilibrium and break into ligaments. For other cases, the obvious convergent results are obtained along with the increasing of adaptive refinement grid levels.

3.2 Full-developed atomization

An experimental case[30] is selected to be a validation case for full-developed pattern. The standard case is water impinging jets under $We=2860$, $Re=11748$ with $dW_e = 635\mu m$. Three cases with different grid levels in liquid phase and on interface are carried on to discuss the independent grid level. The base level of simulation domain is 5. The refinement of liquid phase is one level lower than that of the interface as shown in Table 2. The grid sizes for the three cases are 80, 40, and 20 $\mu m$ with 7.68, 15.36, and 30.72 cells in one diameter respectively. Since droplets with diameters smaller than three of minimum grid size cannot be well resolved, they are removed from the simulation domain to reduce the numerical cost. Simulation results under different resolution are shown in Figure 3. A Ray-tracing technique is used to get a realistic view of the complex atomization process. When using low resolution as shown in Figure 3(a), the liquid sheet breakup at the boundaries. Since the small droplets are removed, there are only a few droplets at the downstream of impingement. In additional, simulation of low resolution fails to predict the formation of impact wave. With higher resolution as shown in Figure 3(b), simulation can resolve the large amplitude wave which is so-called impact wave. The breakup length is similar to the low resolution one. At the downstream of the impingement, there is not
obvious long ligament. With high resolution, the simulation resolves a more complex impact wave and the formation of liquid ligaments and droplets as shown in Figure 3(c).

<table>
<thead>
<tr>
<th>Case No.</th>
<th>level in liquid phase</th>
<th>level on interface</th>
<th>Basic level</th>
</tr>
</thead>
<tbody>
<tr>
<td>V1</td>
<td>6</td>
<td>7</td>
<td>5</td>
</tr>
<tr>
<td>V2</td>
<td>7</td>
<td>8</td>
<td>5</td>
</tr>
<tr>
<td>V3</td>
<td>8</td>
<td>9</td>
<td>5</td>
</tr>
<tr>
<td>V4</td>
<td>9</td>
<td>10</td>
<td>5</td>
</tr>
</tbody>
</table>

Table 1. Low-velocity verification cases with different adaptive refinement grids levels.

<table>
<thead>
<tr>
<th>Case No.</th>
<th>level in liquid phase</th>
<th>level on interface</th>
<th>Basic level</th>
</tr>
</thead>
<tbody>
<tr>
<td>C1</td>
<td>6</td>
<td>7</td>
<td>5</td>
</tr>
<tr>
<td>C2</td>
<td>7</td>
<td>8</td>
<td>5</td>
</tr>
<tr>
<td>C3</td>
<td>8</td>
<td>9</td>
<td>5</td>
</tr>
</tbody>
</table>

Table 2. High-velocity validation cases with different adaptive refinement grids levels.

Figure 2. Front view of numerical verification for low-velocity impinging jets, where volume rendering has been used to visualization of the isosurface of interface.

Figure 3. Grid independence study results of numerical validation for high-velocity impinging jets. a) minimum $\Delta x = 80 \, \mu m$ (7 levels of refinement), b) $\Delta x = 40 \, \mu m$ (8 levels of refinement) and c) $\Delta x = 20 \, \mu m$ (9 levels of refinement).

Figure 4(a) shows the spatial distribution of the droplet of the finest simulation. It is clear that the droplet size distribution varies in a large range. From the data visualization process, direct insight can be achieved to the atomization process. The Probability Density Function (PDF) of droplet size after atomization of impact wave under different grid resolution is shown in Figure 4(b) comparing with experiment data[30]. Since the droplets with diameter smaller than three times of minimum grid size are rejected in the statistics, the PDF curves do not start at zero. As resolution increasing, the PDF curves converge to the experimental data. The peak diameter under the low resolution is around $180 \, \mu m$, while the experiment data has a flat peak from 100 to $156 \, \mu m$. The peak diameter under the middle and high resolution is almost the same, say $108 \, \mu m$. The different between the middle and high resolution is that the latter has a flat distribution which is closer to the experiment data, while the whole curve is closer to the experimental one. This indicates that the high resolution is enough the resolve the atomization process.

4. Observations

4.1 Various patterns

All the conditions of simulation cases in the present study are shown in Figure 5 on the We-Re regime diagram from experiment[31]. All the patterns obtained from simulations are shown in Figure 6. Governing by the two non-dimensional parameters, Re and We, the liquid rim and sheet formed by impinging jets show different flow structures. Figure 6(a) shows a clear fluid chains pattern. Both liquid sheet and rim are formed in the first chain. After that, the rims from two sides collide at the tip point to form another chain with both sheet and rim. Since the liquid viscosity dissipation reduces the inertial energy of the combined liquid stream, after the second chain, no liquid sheet is formed. The colliding jets coalesce into a single jet that oscillates under inertial-capillary interaction. Eventually, the breakup happens at the end of the chains under Rayleigh-Plateau instability.

As the jet velocity increasing, larger liquid sheet is formed as shown in Figure 6 (b). Since the liquid sheet
and rim formed after the first liquid chain is much smaller compared to the first one, this pattern is usually called closed rim. When the jet inlet condition is not symmetric with respect to the impact plane, the liquid rim can be unstable. Figure 6 (c) shows a pattern with fishbone instability. This pattern is proved to be consistent with a Rayleigh-Plateau mechanism associated with centripetal forcing [4].

With higher jet velocity, the liquid sheet can be unstable as shown in Figure 6 (c). This pattern can be called disintegrating sheet or open rim. Small holes are formed on the disturbed thin liquid sheet in the downstream. The increasing of diameter of holes can eventually reach the rim and form two horizontal liquid ligaments. The lower ligament can then breakup under capillary effect.

The rim formed by impinging jets is unstable under higher velocity as shown in Figure 6 (d). Waves are also shown on the liquid sheet but decayed at downstream. This instability is caused under the mechanism of Kelvin-Helmholtz instability. The liquid sheet can also become more unstable and with violent flapping as shown in Figure 6(e). This pattern is called impact wave. It is shown that the waves are close to each other. The size of the liquid sheet is much larger and with more complex wave on the interface. The parameter of this case is similar to that of the high-speed benchmark case.

Figure 4. Spatial distribution of droplets and PDF results under different resolutions.

Figure 5. Location and simulation results in We, Re co-ordinates.

4.2 Sheet thickness distribution

Figure 7 shows the velocity contours of the low-velocity benchmark case. The inlet velocity profile is parabolic. The velocity profile is changed when approaching to the impact plane. The stagnation zone is clear shown in Figure 7 (a). The isolines of the velocity show concentric ellipse pattern. In each radial direction, the velocity decreases with the distance to the stagnation point and reach minimum value in the liquid rim. The velocity of the rim at the rear point is lower than other position. This can be also observed from Figure 7 (b). After the two jets interacted at the impact plane, the liquid sheet is formed. It is shown in Figure 7 (b), the velocity distribution of liquid sheet is not uniform. The variation is from about 1.15 to about 1 of \( \frac{u_s}{u_j} \).
The result of the low velocity benchmark case is also used to compare the thickness distribution from theoretical analysis. Figure 8 shows schematic diagrams of the jet impact system[13]. The radial flow in the sheet can be represented by a source with an origin located at the stagnation point S as shown in Figure 8(a). The jet streamlines are initially parallel and one of them, the separation streamline, reaches the stagnation point S. Mass, energy and momentum fluxes through an angular element of the jet taken in the plane parallel to the sheet and relative to the separation streamline are equal to the fluxes through a vertical section of the sheet at a distance r from S. The section of the jet parallel to the sheet plane is shown on Figure 8(b). This is an ellipse with a major axis equal to $2r_j / \sin \alpha$ and a minor axis equal to $2r_j$, where $r_j$ is the radius of the incident jets. The separation streamline crosses this ellipse at the separation point $P$ which is taken as the origin of the cylindrical coordinate system $(q, \theta)$. The distance between $P$ and the centre of the jet is equal to $b$.

Derivations[17] show that the sheet velocity $u_s$ and the thickness parameter $K$ can be expressed as

$$u_s = (G/F)^{\frac{1}{2}}$$

$$K = hr = 2 \sin \alpha F^{\frac{3}{2}} / G^{\frac{1}{2}}$$

where

$$F = \int_0^\theta qu dq$$

$$G = \int_0^\theta q u dq.$$  

The variable $b$, which determines the position of the stagnation point in the liquid sheet, can be obtained from momentum conservation. Hasson and Peck[11] showed that a simple relationship exists as

$$b = \beta \frac{1}{\tan \alpha}$$

where the proportional constant $\beta$ is 1.0 for uniform flow and 0.68 for Poiseuille profile[17].

It is found from present simulation that the elastic non-dissipative impact assumption is not true for real case. The streamline near the stagnation point is affected by the pressure gradient. The measured values of $b$ for different imping angle are shown in Figure 9(a) together with the theoretical curves for uniform and parabolic velocity profiles. The measured data is larger than that of uniform velocity profile. The curve fitting shows that the value of $\beta$ is 1.24. This means point $S$ is closer to rear point than predicted. If projecting the data along the streamline to cross-section of an incident jet parallel to the sheet plane, the data lays close to the curve for uniform velocity profile.

Figure 7. Velocity contours. (a) Cross-section of jet and stagnation area; (b) Cross-section of liquid sheet.
Parameter. It is clear that data from present simulations lays between the theoretical curves of uniform and parabolic velocity profile. This result is reasonable because the velocity profile near the impact point is between uniform and parabolic. On the other hand, the measured data from experiments [17] seems to underestimate the thickness parameter.

Overall, even without good prediction of the location of the stagnation point, the theory can provide good estimation of the thickness distribution as well as the velocity distribution.

Figure 8. (a) Coordinates of the jet impact system; (b) Cross-section of an incident jet parallel to the sheet plane.

Figure 9 (b) compares thickness parameter between the simulated, experimental and predicted thickness parameter. It is clear that data from present simulations lays between the theoretical curves of uniform and parabolic velocity profile. This result is reasonable because the velocity profile near the impact point is between uniform and parabolic. On the other hand, the measured data from experiments [17] seems to underestimate the thickness parameter.

Overall, even without good prediction of the location of the stagnation point, the theory can provide good estimation of the thickness distribution as well as the velocity distribution.

Figure 9. (a) Comparison of stagnation point location between the simulated and predicted stagnation location; (b) Comparison of thickness parameter between the simulated, experimental and predicted thickness parameter.

5. Sheet and Rim Instabilities

5.1 Theory of absolute and convective instability of liquid sheet

A good understanding of the breakup phenomena of liquid sheets and jets requires a sound basic scientific knowledge of the dynamics of flows involving interfaces between different fluids [10]. Linear stability analysis can be used to predict the onset of jet and sheet instability. The disturbance consisting of all Fourier components is allowed to grow both spatially and temporally in the sheet or jet flows. However, the detailed process leading to the eventual breakup requires nonlinear theories to describe [10].

The methodology of convective and absolute stability analyses can be found to find that the inviscid
liquid sheet of $\text{We} > 1$ in the presence of inviscid ambient gas is convectively unstable. The disturbance grows in time as it travels downstream. When the Weber number is reduced to less than one, the sheet becomes absolute unstable. The disturbance then grows in time and spreads in both the upstream and downstream directions.\[10\] Depending on the relative location of the regions of $\text{We} > 1$ and $\text{We} < 1$, one would expect different physical consequences to the entire flow.\[10\] When including viscosity of liquid and gas, the transition Weber number depends on the Reynolds number with the rest of the flow parameters fixed. The transition Weber number approaches one from above, as $\text{Re} \to 0$.\[10\] It is also shown that the surface tension is responsible for the absolute instability, while it actually reduces the amplification rate of disturbance in the regime of convective instability.\[10\] Applying energy budget analysis showed that the surface tension is responsible for absolute instability, while gas pressure is responsible for the convective instability of both the varicose and sinuous modes.\[32\]

5.2 Absolute instability of liquid sheet

It was shown\[11\] that if assuming a uniform velocity profile and an elastic non-dissipative impact, the sheet velocity remains equal to the jet velocity and the thickness distribution can be given by

$$
\frac{hr}{d} = \frac{\sin^3 \alpha}{(1 + \cos \theta \cos \alpha)^2}
$$ \[10\]

At $\theta = 0$, the initial position the rim can be expressed as

$$
\bar{r}_0 = \text{We} \frac{\sin^3 \alpha}{8(1 + \cos \alpha)^2}
$$ \[11\]

where $\text{We} = \rho d \mu u^2 / \sigma$.

The Weber number based on the sheet thickness and velocity can be expressed as $\text{We} = \rho \mu u^2 / \sigma$. Simple derivation shows that $\text{We}$ near the rim at $\theta = 0$ can be approximately expressed as

$$
\text{We}_c^0 = (u_j / u_j)^2
$$ \[12\]

It is of interest to know that when $u_x, u_j < 1$, $\text{We}_c^0 < 1$. That means the liquid sheet at this point is absolute unstable. In the classic theory, $u_j$ is assumed to be equal to $u_j$. From above derivation, for $u_x, u_j = 1$, $\text{We}_c^0 = 1$. This means the liquid sheet is absolute stable. However, in practical, $u_j$ is less than $u_j$ in the rear top region. For cases with impinging angle of $78^\circ$, $u_x, u_j$ is about 0.56. This means $\text{We}_c$ can be less than unit before reaching the rim. The evolution of the absolutely unstable disturbance and that of the present disturbance have an important common character: both of them remain non-vanishing for all time and all $z$. Note that the absolutely unstable disturbances are modified by nonlinear effects before they become unbounded and may become non-linearly stable.\[32\]

In order to prove above theoretic prediction, three cases with the same configuration but different velocity value are carried out to provide direct insight to parameter requirement to the absolute instability of liquid sheet near the rear point. The interfaces at the cross-sections are obtained as shown in Figure 10. For case with lowest velocity, liquid sheet is not formed near the rear point as shown in Figure 10(a). The flow around the rear point is stable. With higher velocity, liquid sheet is formed with stable thick sheet as shown in Figure 10(b). When the velocity increases further, the liquid sheet becomes longer and thinner near the rim. It is obvious that the liquid sheet becomes unstable when reaching to certain thickness value as shown in Figure 10(c). Detail measurement from simulation data shows that the local $\text{We}$ number at the initial unstable location is about 1. This confirms the unstable of the liquid sheet is caused by the natural of absolute instability.

![Figure 10. Cross-section of around the rear point under different velocity value.](image)

The rim shapes under different velocity values are also shown in Figure 11. Different from the two cases with stable liquid rim, the unstable motion of rear point of liquid sheet disturbs the shape of liquid rim. It should be also noticed that there are a stationary sinuous wave near the liquid rim. The origin of the stationary wave coincides with the liquid rim near the rear point.

![Figure 11. Rim shapes under different velocity values.](image)
Figure 11. Effect of velocity value to the liquid rim and sheet.

5.3 Stationary antisymmetrical wave

On the liquid sheet shown in Figure 6(c) and Figure 11(c), there is a stationary wave pattern from the top. This is an antisymmetrical wave of the same type as the cardioids observed by Taylor[8]. The form of the antisymmetrical waves which can remain at rest can be described as[8]

\[
\sin^2 \phi = \frac{2\sigma}{\rho u^2 h} \quad (13)
\]

With the geometric relationships of Equ. (7) and

\[
\tan \phi = \frac{rd\theta}{dr} \quad (14)
\]

together with simple derivation, the differential relationship of \( r \) and \( \theta \) can obtain. By applying the initial point \((r_0, 0)\), the shape of the antisymmetrical wave can be obtained in polar coordination. In Taylor’s experiments[8], parallel relation between the antisymmetrical wave and the disintegrated edges of the sheet was used to determine the shape of the leaf edge with the help of measurement.

Figure 12. Shape of antisymmetrical wave.

Figure 12 shows the wave locations obtained from simulation compared with theoretical predictions. Similar to experiment observation[13], there are divination apart from the theoretical prediction. The reason is found to be the deviation of the initial location. When introducing a parameter upon We number to include the real factor and match the rear point location, good agreement is obtained as shown in Figure 12. However, the shape obtained from simulation appears to be slimmer than from prediction. This may cause by the assumption that sheet velocity is equal to that of liquid jet.

From Eq. (13), the We number based on the local velocity perpendicular to the antisymmetrical wave is equal to 1. This means if a rim is bounded at the location outside the critical antisymmetrical wave, the rim and sheet can be absolute unstable since the local We number can easily reach to 1. This coincides with Taylor’s observation[8] that the edges of the sheet lie approximately parallel to the stationary wave.

5.4 Convective instability of liquid rim

Figure 13 shows the evolution of a disturbance in liquid rim. As noticed in previous section, the disturbance can be naturally introduced by the absolute instability of the liquid sheet at the rear point. The deformation of the liquid rim leads to the formation of a ligament which will breakup into several droplets. After getting unstable, the diameter of deformation increases as it moving along the liquid rim. It is clear that the
center of the deformation appears to apart from the main rim shape. This is caused by centripetal effect. This detail observation confirms that the breakup of liquid rim formed by impinging jet is caused by capillary instability associated with centripetal forcing along the liquid rim.

6. Impact Wave Dynamics

6.1 Onset of impact wave

In order to show the formation process of impact wave, snapshot from simulation result is clipped to show the cross section as shown in Figure 14. The formation of impact wave can be divided into four parts, impingement of two liquid jets, formation of impact wave, breakup of liquid sheet into ligament, and breakup of ligament into droplet. The large amplitude of the impact wave will disturb the sheet to breakup. Figure 15 shows the breakup sequence of a point on the liquid sheet. Under the effect of flapping sheet, breakup of liquid sheet happens near the boundary. As the breakup spreads to the downstream, long ligaments are formed.

Figure 14. Clipped view of impact wave.

Figure 16 shows the velocity profiles of different axial locations on the cross section. The inlet velocity profile is 1/7 power law velocity profile. Location 2 is near the stagnation point. The velocity is the lowest at the center. Further downstream, the velocity profile at the center increases and eventually shows a parabolic velocity profile at location 4.

Sander et al.[33] performed direct numerical simulations based on the volume-of-fluid method in order to identify the influence of the inflow velocity conditions on the sensitivity of primary breakup phenomena. A liquid sheet ejected into a gaseous environment at moderate Reynolds numbers ranging from Re=3000 to 7000 was considered. The focus of their study was directed to the identification of those parameters that most strongly enhance primary breakup phenomena. These key parameters were the flow quantities such as the range of the inflow velocity and the inherent character of the mean velocity profile as well as the corresponding dimensionless groups and turbulence quantities of the nozzle flow. Their results showed that in addition to these well-known quantities the kinetic energy flux, which depends on the character of the mean velocity profile generated by the nozzle geometry, has a drastic influence on the instabilities appearing. Impinging jets interact at the impact point and form a liquid sheet with a parabolic velocity profile.
locity profile are simulated under Re = 9240, We = 3556. The results in Figure 17 show that with parabolic velocity profile unstable sinuous wave is formed, while the other case only irregular waves are formed on both sizes of liquid sheet. The similarity of the sinuous wave with the impact wave can be used to get the mechanism of formation of impact wave. Figure 17 also shows the vorticity field of the two cases. It is obvious that both cases have two vorticity layers with reverse direction. But the former one has larger intensity of vorticity and closer distance between two vorticity layers.

The vorticity is stronger than other part of the impact wave. High vorticity at the wave crest and trough attends to tear the wave to ligament. It is the shear force that deforms the liquid sheet and causes breakup. Since the wavelength changes a little along the downstream direction, the characteristic wave length, $\lambda$, takes that of the first visible cycle of the impact wave. The velocity of the liquid sheet, $U_\lambda$, takes the speed of the wave crest and trough. In order to consider the effect of velocity to the dynamics of impact wave, the cross sections of two cases with velocity ratio 3:2 are shown in Figure 18 (b) and (c). Since the thickness of liquid sheet is independent with inlet velocity, the shape of liquid sheet is almost the same before formation of impact wave. The ratio of wavelength and jet diameter, $\lambda/D$ is 1.16 and 1.17 for two cases respectively. Measurement of the wavelengths shows that it is also independent with inlet velocity. Wave propagation frequency can be expressed as $f = U_\lambda/\lambda$. Then, the Strouhal number, $St = fD/U_\lambda = D/\lambda \approx 0.86$.

**Figure 16.** Evolution of velocity profile near the impact point.

**Figure 17.** Effect of velocity profile to the stability of liquid sheet.

**6.2 Effect of inlet velocity value**

Figure 18 (a) shows the interface and vorticity field of the cross section. It is clear that vorticity of the two sides in the liquid sheets are reverse. The breakup of liquid sheet happens at the wave crest and trough where

**Figure 18.** Effect of inlet velocity to wavelength. Conditions: (a) and (c): water, $D = 0.33$ mm, $U = 18.7$ m/s, $2\theta = 60^\circ$, Re=6160, We=1580; b) water, $D = 0.33$ mm , $U = 28$ m/s, $2\theta = 60^\circ$, Re=9240, We=3556.

**7. Conclusions**

High fidelity numerical simulations is performed to study the fluid dynamics of liquid sheets formed by two impinging jets. A fully three-dimensional volume-of-fluid (VOF) method with adaptive mesh refinement (AMR) based on octree meshes is used to simulate the formation and breakup of the liquid sheet. Two benchmarks for low velocity and high velocity impinging jets are built up as the standard numerical assessment of verification and validation. Grid refinement is applied to the steady and unsteady flow-field for assessing the
grid resolution and solution convergence. The verification results of low velocity impinging jets show that the shape of rim and sheet agrees well with the theoretical formulation. The validation results of high velocity impinging jets show that the characteristics of the atomization process, i.e. the sauter mean diameter agrees well with the available experiments. Various patterns of liquid sheet and rim formed by impinging jets are obtained. Detail flow-field is studied based on existing theoretical prediction. New insights are obtained to the flow-field near the stagnation point. The onset of instability of the liquid sheet and rim are analyzed based on concept of absolute and convective instabilities. It is also shown that the impact wave is caused by the interfacial shear stress which forms the surface waves on the two sides of liquid sheet. The interaction of waves on the two sides forces the liquid sheet to resonate at natural frequency. The ratio of wavelength and jet diameter is independent to jet velocity, liquid viscosity and surface tension. All the studies above allow us to get the schematic diagram for various instabilities existing in impinging jets atomization as shown in Figure 19.

Figure 19. Schematic diagram of instabilities caused by impinging jets.

Acknowledgments

This work was sponsored by the U.S. Army Research Office under the Multi-University Research Initiative under contract No. W911NF-08-1-0124. The support and encouragement provided by Dr. Ralph Anthenien are gratefully acknowledged. Special thanks is due to Dr. Stéphane Popinet for allowing us to use his work on the VOF and AMR algorithms.

References

2. Lefebvre, A.H., Atomization and sprays. 1989: CRC.
14. Shen, Y.B. and D. Poulikakos, Thickness variation of a liquid sheet formed by two impinging jets using holographic


