ILASS–Europe 2017, 28th Conference on Liquid Atomization and Spray Systems, 6-8 September 2017, Valencia, Spain

http://dx.doi.org/10.4995/ILASS2017.2017.4865

Numerical Simulation of Like and Unlike Impinging Jets

Hamed Dolatkhahi[1], Ghader Oliaee[1,] Azadeh Kebriaee*[1]

[1]Aerospace Engineering Department, Tehran, Iran

Abstract

In the present study, using the open source OpenFOAM code, a numerical simulation is performed taking the adaptive mesh refinement (AMR) technique during solution. Formation of liquid sheet after the impact of two identical cylindrical jets at various conditions is studied. Since the flow pattern depends upon the Reynolds and Weber numbers, numerical tests are conducted at a variety of flow velocities and Reynolds numbers to demonstrate the effect of these parameters on the sheet formation. It is then concluded that at various conditions, different instabilities occur in the flow; hence, different sheet formations a flow patterns happen.

In this study, impact of two dissimilar cylindrical fluid jets is successfully simulated for the first time in literature. Actually, water and oil jets are taken into account and their impact behavior is studied. In the presence of the surrounding air, an unstable sheet will form after impact due to the high injection speed of the jets. As depicted in the results, since the inertia and other physical characteristics of the two fluids are dissimilar, different phases are more intensely diffused.

Keywords: *Two phase flow, Injector, Impingement jet, numerical solution, Dynamic mesh.*

Introduction

Atomization is defined as a process of producing a large number of droplet from a liquid spot [1]. In the atomization process, liquid jet or a sheet of liquid are broken apart due to high kinetic energy content, exposure to high velocity air/gas or the mechanical energy transferred to the system in the form of rotational movements of vibrations. Due to random characteristic of atomization, the produced spray consists a large spectrum of various size of diameters. More recently, on account for a wide range of spray utility, liquid atomization has been turned out to be a fascinating subject.

The spray produced in the Impingement jet injectors has a variety of industrial applications. Impingement injectors are among the most common fuel injectors in turbine engines. Such injectors are also utilized to control droplets in combustion, recreational fountains and also to provide uniform painting/coating [2]. Impingement jet injectors are widely used in liquid propellant and solid propellant motors due to their simplicity in manufacturing as well as high efficiency and excellent atomization characteristics. They have also been recently in the center of attention due to their low cost and high efficiency. Combustion performance of liquid rocket engines highly depend on mixture uniformity which is created by impingement jet injector. Impingement jet injector is an appropriate method to control droplet size as well as spray distribution which are highly applicable in industry. This injector is used in liquid rocket jet where reactants are produced by impinged fuel and oxidizer jets [3].

In such injectors, impact of two cylindrical jets leads to liquid sheets. Overall shape and thickness of the sheet is dependent upon the impacting angle between two jets, diameter of the jets, velocity and physical characteristics of the jets [4]. A variety of impingement jet injectors exist, which are used for different fluids (liquid, gas or gel) and are based on different parameters such as motor conditions, cooling of the walls in the combustor and its length, mixture ratio and the operating pressure [5].

Although studies on Impingement jet injectors date back to a long time ago, simulations of such injectors are so scarce in literature due to complexities inherent in multiscale two-phase flows and the computational difficulties in such simulations. In a first simulation, Inoue et al. [6] presented a numerical study to provide characteristics of the impingement jet injectors. Their results were then compared with experimental findings and analytical solutions. Despite the shape of liquid sheet being periodic, their simulation was incapable of providing accurate measures for ligaments and droplets during collapse due to coarse mesh and inadequate mesh studies.

In 2013 Arienty et al. [7] investigated the effect of mesh refinement on the calculations of the jet atomization. In this research, interfacial surface is analyzed by an appropriate combination of VOF and Level Set, using impingement jet atomization by means of CLSVOF formulation. Doing this so, collapse of the produced sheet by means of two jets with a high velocities at various levels of a refined mesh was investigated and compared with each other.

Dong-Jun Ma et al. [8] have also used numerical simulations to study the behavior of impingement injectors and atomization patterns. In this simulation, they used refined adaptive mesh to study initial atomization. They also studied the non-Newtonian impingement jets and obtained two distinct flow patterns. In the most recent study and simulation, X. Chen et al. [9] investigated the flow using VOF method along with the refined adaptive mesh. They also provided the details of the flow behavior in a vast range of Reynolds and Weber numbers and compared their findings with the experimental data.

In the present study, the open source solver (Open-FOAM) is utilized to provide a numerical simulation for the flow of Impingement jet injectors in a variety of conditions. For the models under study, refined dynamic meshes are utilized which can significantly minimize the computational cost and provide a reliable solution for two phase flows with different liquid flow sizes after collapse (comprising droplets and ligaments). Moreover, in this study, impact of two jets with distinct physical properties is simulated for the first time.

Flow Physics and Governing Equations

Studies show that when two jets collide, a sheet is formed perpendicular to the colliding plane, as shown in figure (1).

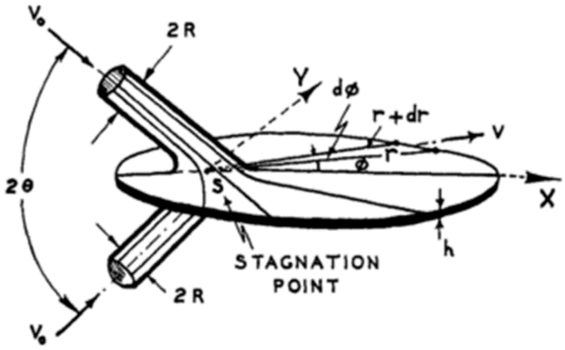


Figure 1. Schematic collapse of two impingement jets [3].

If the two jets are concentric, the resulting sheet will be circular otherwise the sheet will have an oval or leaf shape. To express the spray characteristics properly, a variety of parameters such as collision angle, jet velocity, jet radius, radial position and sheet thickness must be taken into account. After formation of the sheet, waves generate over the sheet and develop until the sheet is transformed into droplets. The resulting sheet of fluid has an oval cross section and its principal axis lies in the plane of impingement. The size of droplets in an impingement jet injectors is directly proportional to the sheet's thickness. The sheet thickness on the other hand is inversely dependent upon the radial distance to the collision point and also some other miscellaneous parameters of the flow.

Actually development the flow in the atomization can be divided to four categories as follows: jet impingement, sheet oscillation, ligament formation, and droplet formation. Impingement waves grow due to hydrodynamic instability, and hence form disturbances with high amplitude. As a result, sheet is broken up. After impingement of two jets, a sheet is produced which is oriented from the impingement point, where the oscillated sheet breaks up and becomes instable. This is because, the waves are created on this sheet and spread on them radially. As a result, capillary instabilities are formed and converted in to the droplet by means of Rayleigh instabilities [10].

Injectors have usually simple configurations but the two phase flow and the contact of distinct phases in the injector is highly complicated. Flow is usually transient with some parts being turbulent and rotational. Considering a generic control volume comprising two sub-control volumes with a common interface, the governing equations and stress tensor for each incompressible phase are described as Eqs. (1-3)

$$\frac{D(\rho_i u_i)}{Dt} = \nabla.\Pi_i + \rho_i g \qquad (1)$$

$$\nabla.u_i = 0 \qquad (2)$$

$$\Pi_i = -p_i I + \mu_i(\nabla u_i + (\nabla u_i)^T) \qquad (3)$$

Where the subscripts *i* denotes each of the first or second phases of the fluid. Also, ρ and μ represent the density and viscosity, respectively. For the interface of liquid-gas Γ which is observable in the atomization process, impermeability condition holds and there is no mass transfer between the two phases, hence the continuity equation yields

$$u_1 = u_2 \; for \;\; x \in \Gamma \qquad (4)$$

Variations of the normal stress over the interface is balanced with surface tensions. Taking the coefficient of surface tension to be constant, Laplace-Young boundary condition for conservation of momentum over the interface is defined as

$$\Pi_1 - \Pi_2 = \sigma k n \;\; for \;\; x \in \Gamma \qquad (5)$$

Where *k* is the curvature radius of the interface and *n* is the unit vector perpendicular to the interface as well as σ is surface tension coefficient. To provide a numerical simulation of the flow model, interface of fluid and gas should be determined accurately. The motion of interface is described with Eq. (6) where scalars are defined differently according to following the interface.

$$\frac{\partial C}{\partial t} + u.\nabla C = 0 \qquad (6)$$

In the VOF method, C is the volume fraction. Moreover, physical quantities such as density, viscosity, normal function and curvature are defined for the VOF method as Eqs. (7-10)

$$\rho = \rho_2 + (\rho_1 - \rho_2)C \qquad (7)$$

$$\mu = \mu_2 + (\mu_1 - \mu_2)C \quad (8)$$
$$n = \Delta C/|\Delta C| \quad (9)$$
$$k = -\nabla . n \quad (10)$$

Solution Method

The main purpose of this numerical solution is to simulate the flow of two liquid jets impingement under different conditions. The simulation is carried out in OpenFOAM. In OpenFOAM, there are different solvers for simulation of multiphase flows in which one of the most well-known solvers is the InterDyMFoam used to simulate two phase, isothermal and incompressible based on VOF method. Energy equation is not utilized in this solver.

In simulations of colliding jets, the two liquid jets collide and a thin sheet is formed in the vicinity of the two jets, Figure (2).

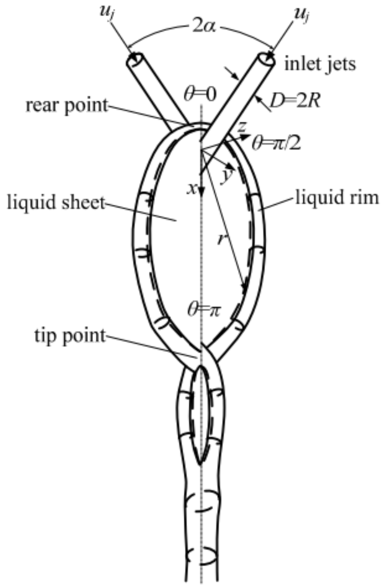
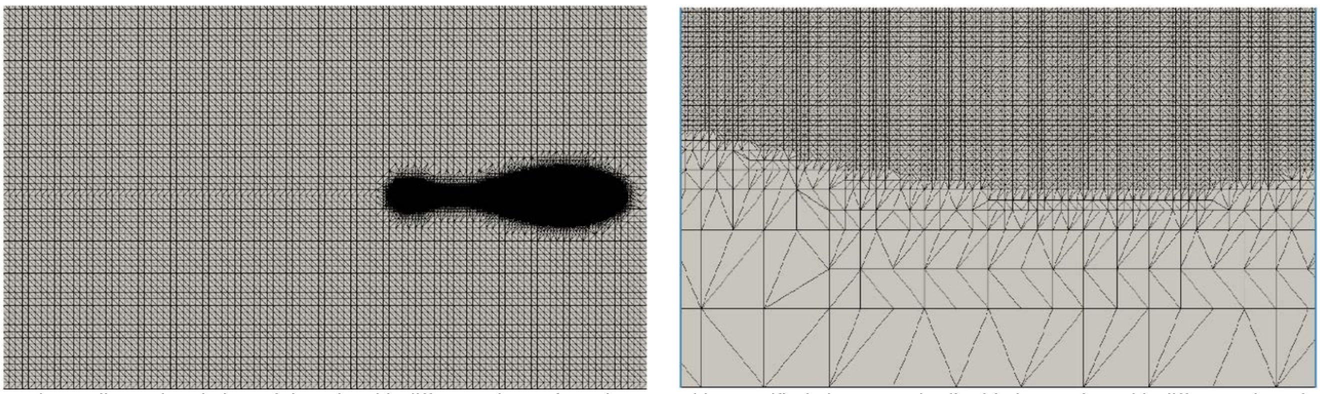


Figure 2. Schematic collision of two jets [10]

The domain for the present study is a cube purely air-filled domain as the initial condition. In order to eliminate the boundary condition effects, the volume is considered with the dimension $50d \times 30d \times 25d$, where d is jet radius. The fluid used in this study is Newtonian at standard temperature and pressure conditions. Gravity force is neglected due to negligible implications.

In accordance to atomization characteristics and two phase flow simulations where there are various length scale, mesh refinement modeling is carried out. All domain is solved with coarse size of mesh. In addition, the implemented method for refining the mesh is carried out at different stages. In this simulation the main areas which requires refinement mesh are as follows: 1- impingement area with high curvature 2- area of liquid sheet formation 3- area of rim at surrounding of the liquid sheet 4- areas with small size of droplet. Meanwhile, the size of cells at the rest of areas (where there is no liquid and are located at a specific distance from the impingement area), is constant. The schematic of the solution domain is displayed at Figure (3).



a) two dimensional view of domain with different sizes of mesh, adapted in the region of liquid sheet formation

b) magnified view near the liquid sheet edge with different size of grid mesh

Figure 3. Schematic of the solution domain

According to the physics and conditions of the problem, two jets are considered on the upper edge as the input with an velocity boundary condition. The nozzle has a circular geometry and the liquid jet is pumped into the domain with a volume fraction of unity during the solution. In other boundaries, far field conditions hold which are zero pressure gradient, zero velocity gradient and zero volume fraction. If the flow direction is inward, the tangential component would take a constant value.

Results

As a result, the sheet breaks up into ligaments and these ligaments further will be breaks into droplets. In Figure 4, verification of an example model with the analytical and numerical ones presented in Refs. [11-12] are provided. As can be seen, the length and shape of the formed sheet is in a very good agreement with experimental results.

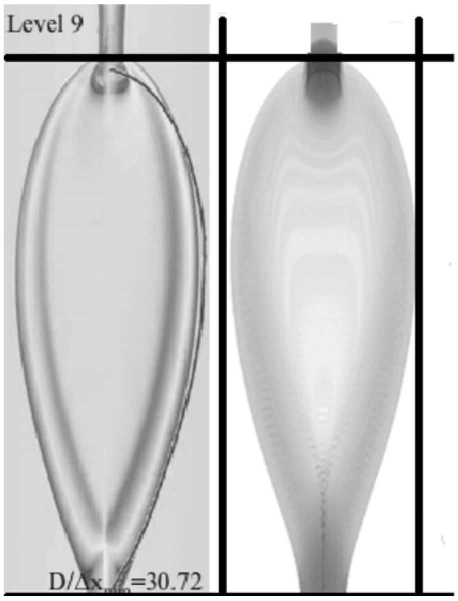


Figure 4. Validation of one model with numerical and analytical simulations.

Experimental and simulation conditions for collision of the jets at low velocities are taken as the simulations performed by Bremond and Villermaux [11]. The injected liquid is water-glycerin with a diameter and jet velocity of 400 micro meters and 3.3 meters per second, respectively. Collision angle of the two jets is also taken to be 89 degrees. According to the experimental conditions, Reynolds and Weber numbers of the flow are 40.4 and 58.8, respectively. Based on the physics of the flow, a stable sheet is formed according to the balance of surface tension forces, centrifugal force and inertial force at the edge of the sheet. For such conditions, when the two jets with low velocities collide, they will finally form a single jet after formation of the sheet, if surface tension force and inertial force are balanced.

$t = 10ms$ $t = 8ms$ $t = 6ms$ $t = 4ms$ $t = 2ms$

Figure 5. Numerical solution of collision of the two jets for the $2\alpha = 89^o$, $We = 58.8$, $\mathrm{Re} = 40.4$, $u_j = 3.3\frac{m}{s}$

Figure 6. Close-up view of the collision of two jets with condition $2\alpha = 89^o$,$We = 58.8$, $\mathrm{Re} = 40.4$, $u_j = 3.3\frac{m}{s}$.

At the peak point of the sheet and after a unit jet formation, the prescribed jet is ultimately break into a large droplets, where this breakup is consistent with capillary instability. The results of the simulation is illustrated in Figures 5 and 6 for development of liquid sheet bounded by a rim.

In low velocities and high viscosities, the fluid can be followed by segregated form of the sheet by means of two impingement jets. After impingement, whole fluid is bounded by an edge surrounding the sheet. Injected fluid flows along of the surrounding edge of the sheet. After that, two fluids are impinged until a jet or a secondary liquid film will be formed at the lower peak point of the sheet. Increasing of the velocity in this case will cause small grains at the surrounding edges, caused by small disturbances, and thus the sheet is disintegrated. In another simulation with an enhanced flow rate and Reynolds number, results are depicted in Figure 7. In this simulation the nozzle diameter is $D = 400\mu m$ and collision angle of the two jets is 60 degrees. The jet velocity under this condition is 18.5 m/s and the Weber and Reynolds numbers are 2987 and 11724, respectively.

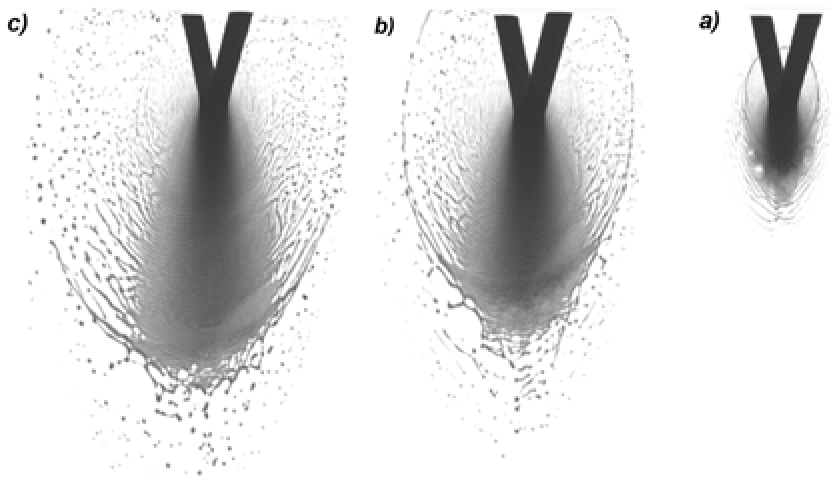


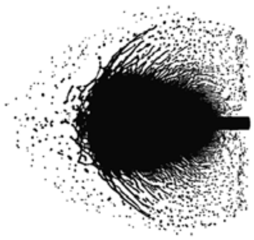
Figure 7. Numerical solution to collision of two jets for the $2\alpha = 60^o$, $We = 2987$, $\mathrm{Re} = 11724$, $u_j = 18.5\frac{m}{s}$, a) 0.2 msecond b) 0.3 msecond, c) 0.46 msecond

Large amplitude hydrodynamic instabilities which is typically known as the collision wave, can be observed on the sheets of Fig.7. These waves have high frequency which are dominant in sheet collapse for high velocities and high collision angles. In addition, advancing the sheet through the time (0.2 millisecond – 0.46 millisecond) and forming the final shape of the sheet can be followed in Fig. 7.a – Fig. 7.c.

One of the most significant advantages of the present study is the refined mesh algorithm. To make it more clear, we may compare the number of computational cells for the present algorithm which is 5408549 with the traditional and uniform mesh (with the smallest cell being defined according to the smallest cell in the dynamic simulation) which is 491520000 (almost 91 times larger).

(a) $\mathrm{Re} = 294$, $We = 152$

(b) $\mathrm{Re} = 3536$, $We = 353.5$



(c) $\mathrm{Re} = 11724$, $We = 11724$

Figure 8. Sheet formed as a result of a 60 degree collision with different Reynolds and Weber numbers

In Figure 8, three different formation modes of the sheet are depicted for a variety of Reynolds numbers. As the jet velocity increases, if the film of liquid grows sufficiently, some random holes would appear in the sheet. For further increments of the liquid velocity, gas inertia may lead to formation of some waves over the surface of the sheet and creates a rugged surface. These perturbations also create asymmetric waves with increasing radial amplitudes. In general, the overall flow pattern is dependent upon the Weber and Reynolds numbers. As mentioned earlier, increasing the jet velocity or the Reynolds number make liquid sheet unstable. In this case, by increasing of the velocity, the properties of the sheet will highly depend on the jet velocity and liquid properties. Increasing velocity also disappear the rim of the sheet, and hence collapse is seen at near lower peak point. The main reason for this instability is the direct interaction of the sheet with gaseous media surrounding the sheet. In this way, the rapid growth of the wave on the sheet would be occurred, and when the wave is reached to its critical value the collapse will be happened. As it is pointed out in the beginning of the simulation, two liquid jets at low Weber and Reynolds numbers will form leaf like sheet bounded by a rim. Increasing of the velocity may create small satellite droplets at the surrounding of the edges caused by small disturbances which may result in the sheet breakup. Formation of the droplets can be formulated following by the Rayleigh of capillary instabilities. In this case, a liquid droplet will be formed which its size will be increased as the velocity increases.

In order to simulate heterogeneous fluid impacts, the multiphaseInterFoam solver which is a typical in OpenFOAM, is utilized. This solver employs the same equations as the InterFoam solver and uses the same sets of codes to simulate incompressible and isothermal two phase flows by means of VOF method. Energy equation is not included in this solver. The only distinction of the multiphaseInterFoam solver with InterFoam is that the former is not restricted for two phase flows and can be used for multiphase flow simulations as well.

For this solver all phases are considered to be incompressible. Moreover, a fluid must be taken as the base fluid which is usually the air for the most applications. The rest of present phases with a phase ratio between zero to one are compared with the base fluid. Also, according to the number of fluids used, a separate surface stress must be defined between any two phases of the flow.

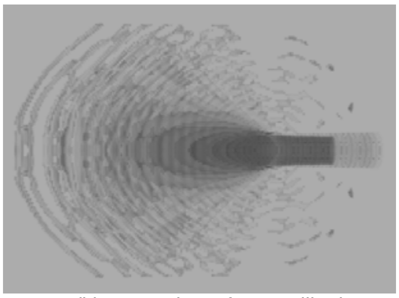
Iterative loops are commonly used in this solver to determine the physical quantities such as density and viscosity. The loop for determination of the surface tension is different. The surface tension is indeed calculated between two phases and is defined as Eq. 11

$$F_s = \sigma \left(\nabla . \left(\frac{\nabla \alpha}{|\nabla \alpha|} \right) \right) (\nabla \alpha) \tag{11}$$

Where $\nabla \alpha$ is the normal vector to the interface of the two phases and σ is the surface stress.

In this simulation, two oil and water liquid jets which have different properties are impinged with each other. In this case, by considering of air, a multi-phase flow will be formed which an instable sheet would be formed due to high velocity of the injected jets. As it will be shown in the results section, since the inertial and properties of two fluid are not equal, advancing each fluid on the surface of the other is different. It means the extension of each fluid in the gaseous medium is different and balancing between inertia force, viscosity and surface tension is a crucial role in the shape of unlike impinging jets.

(a) Side view of two unlike jets



(b) Top view of two unlike jets

Figure 9. Unstable sheet formed by the collision of two jets of water and oil

Conclusion

In this study, the behavior and physics of formation of a sheet as a result of collision of two cylindrical jets in the laboratory condition is investigated. According to the flow physics and injection conditions, collision of the two jets can cause a variety of instabilities in the sheet. To this aim, using the open sourced OPENFOAM code, numerical simulation is performed with adaptive refined mesh to characterize the formation of the flow after collision of two cylindrical jets of liquid. Results of the present study are also verified with analytical, experimental and other numerical simulations. In the numerical simulations performed, different experimental conditions are studied for a variety of Reynolds and Weber numbers. It is found that in different conditions due to the variety of instability conditions that can occur, the shape of the sheet and flow is different.

Dynamic mesh refinement makes it possible to start the solution with coarse meshes and refine the mesh as required in the critical and sensitive regions. This will finally reduce the computational costs and make it possible to provide solutions of two-phase flows, i.e. sprays with different dimensions, which require very tiny mesh sizes.

In addition, simulation of two unlike impinging jets was done by a refined mesh for water and petroleum. The shape of the sheet and breakup mechanism of sheet in unlike fluid are generally similar to like fluid and penetration of each liquid in to another can be observed in the simulations.

References

[1] A. H. Lefebvre, *Atomization and Spray*. 1989.

[2] J. C. P. Huang, “The Break-Up of Axisymmetric Liquid Sheets,” *J. Fluid Mech.*, Vol. 43, No. 2, P. 305, Aug. 1970.

[3] D. Hasson And R. E. Peck, “Thickness Distribution in A Sheet Formed by Impinging Jets,” *Aiche J.*, Vol. 10, No. 5, Pp. 752–754, Sep. 1964.

[4] G. Taylor, “Formation of Thin Flat Sheets of Water,” *Proc. R. Soc. Lond. A. Math. Phys. Sci.*, Vol. 259, No. 1296, Pp. 1–17, 1960.

[5] N. Ashgriz, Ed., *Handbook of Atomization and Sprays*. Boston, Ma: Springer Us, 2011.

[6] C. Inoue, T. Watanabe, and T. Himeno, “Study on Atomization Process of Liquid Sheet Formed by Impinging Jets,” In *44th Aiaa/Asme/Sae/Asee Joint Propulsion Conference &Amp; Exhibit*, 2008.

[7] M. Arienti, X. Li, M. C. Soteriou, C. A. Eckett, M. Sussman, And R. J. Jensen, “Coupled Level-Set/Volume-of-Fluid Method for Simulation of Injector Atomization,” *J. Propuls. Power*, Vol. 29, No. 1, Pp. 147–157, Jan. 2013.

[8] D.-J. Ma, X.-D. Chen, P. Khare, And V. Yang, “Atomization Patterns and Breakup Characteristics of Liquid

Sheets Formed by Two Impinging Jets," In *49th Aiaa Aerospace Sciences Meeting Including The New Horizons Forum And Aerospace Exposition*, 2011.
[9] X. Chen And V. Yang, "Thickness-Based Adaptive Mesh Refinement Methods for Multi-Phase Flow Simulations with Thin Regions," *J. Comput. Phys.*, Vol. 269, Pp. 22–39, Jul. 2014.
[10] X. Li, M. Arienti, M. Soteriou, And M. Sussman, "Towards an Efficient, High-Fidelity Methodology for Liquid Jet Atomization Computations," In *48th Aiaa Aerospace Sciences Meeting Including The New Horizons Forum and Aerospace Exposition*, 2010.
[11] N. Bremond And E. Villermaux, "Atomization by Jet Impact," *J. Fluid Mech.*, Vol. 549, No. 1, P. 273, Feb. 2006.
[12] X. Chen, D. Ma, V. Yang, And S. Popinet, "High-Fidelity Simulations of Impinging Jet Atomization," *At. Sprays*, vol. 23, no. 12, pp. 1079–1101, 2013.