Analysis of a Liquid Jet in Supersonic Crossflow using Large-Eddy Simulation

M.B. Kuhn* and O. Desjardins
Sibley School of Mechanical and Aerospace Engineering
Cornell University
Ithaca, NY 14853 USA

Abstract
In understanding the detailed physics of scramjet engine cold-start, simulations can provide useful insights for design. To better grasp the complexity of this design problem, we perform large-eddy simulations of a liquid jet in a supersonic gas crossflow and compare our findings with experiments. These simulations employ a fractional-step method that includes a hybrid advection step, which couples semi-Lagrangian transport with an implicit low-dissipation scheme, and an implicit pressure projection step based on a Helmholtz equation. We explore this relevant flow from qualitative and quantitative perspectives, analyzing liquid plume trajectory, droplet size distribution, and frequency of instabilities. From this analysis, we draw preliminary conclusions about the effects of inflow conditions, evaluating the significance of turbulence in the liquid jet. Additionally, we characterize the spray by measuring spatial distributions of droplet statistics.

*Corresponding Author: mbk225@cornell.edu
Introduction

One of the many challenges in scramjet design and development is fuel delivery. The configuration of a fuel injection system has direct consequences on the nature of the fuel spray, which impacts the process of evaporation and the resulting fuel vapor distribution, determining the likelihood of fuel ignition and affecting the performance of the engine [1]. In the context of a scramjet engine, further study is needed to better understand the characteristics of liquid break-up in highly compressible flow conditions, even in the straightforward case of liquid fuel injected directly into the supersonic crossflow. Improving the understanding of liquid break-up in this setting requires leveraging both experiments and computational simulations, but the state of the art in computational modeling has been limited in supporting such simulations of compressible liquid atomization.

In the literature, studies of liquid atomization in a transonic environment are few. The simulations of Zakrzewski et al. [2] and Im et al. [3] showcased some of the relevant features yet some of the shock waves observed in experiments were absent. Another study, by Majidi and Afshari, used a ghost fluid method combined with exact and linear Riemann solvers on an axisymmetric domain and captured much of the relevant physics associated with early-stage instabilities of the jet [4]. However, this study considered only the initial instability of the jet, neglecting important processes like later-stage ligament formation and droplet breakup. Li et al. used a Eulerian-Lagrangian framework to simulate the atomization of a liquid jet in supersonic crossflow (LJSC) [5], and the results aligned well with experiments. This study, though an important progression in simulating this flow and an astute use of modern simulation capabilities, suffers from limited predictive capabilities since much of the two-phase flow relies on a combination of empirical and analytical models. Garrick et al. successfully simulate a liquid jet in supersonic crossflow at varying Weber numbers, using a diffuse representation of the liquid-gas interface with a modification of the THINC (Tangent of Hyperbola for INterface Capturing) scheme to sharpen the interface [6]. Though this study was successful, it focused only on the large-scale features of the flow and on the instability of the liquid surface, and much remains to be understood in properly characterizing the atomization process involved with this flow.

Previously, we have demonstrated the capability of our flow solver for this transonic multi-phase flow problem [7]. In this work, we quantitatively analyze the atomization of a liquid jet in crossflow via our simulations. We investigate the nature of the flow field, measure properties of the droplet distribution in the wake of the jet, and make comparisons with experimental data [8].

Numerical Methods

The key components of our numerical strategy are semi-Lagrangian convection step that provides accurate interface transport and convective stability [9], a second-order volume-of-fluid (VOF) method for capturing the liquid-gas interface with interface reconstruction based on PLIC [10], and an implicit pressure projection that eliminates the restraint of the acoustic CFL [11]. In addition to this “backbone” of the code, we use a volume fraction source term from [12], and a kinetic energy exchange term from [13], which are only active in cells with more than one phase. We have found this special treatment of multiphase cells to be crucial to stability, especially in LJSC simulations. Finally, to reduce the unphysical kinetic energy dissipation introduced by the semi-Lagrangian convection, we set up the solver in a hybrid framework, where we implicitly solve a centered scheme inspired by [14] in smooth single-phase regions, and we use the semi-Lagrangian transport of momentum in cells near an interface or near a shock, which we detect using a dilatation-based shock sensor from [15].

Results and Discussion

Simulation description

We simulate LJSC to mimic the experimental setup of [16]. Specifically, we consider the case where the liquid mass flowrate is \( m_L = 18.2 \text{ g/s} \), the injector is flush with the wall, and the jet is pure water, not having been pre-mixed with air. The Mach 2 supersonic crossflow is calculated from a steady-state characterization of the wind tunnel in Research Cell 19 of the Air Force Research Laboratory, Wright-Patterson Air Force Base. We set this constant profile as a Dirichlet condition at the beginning of the domain. For the liquid jet, we simulated an incompressible, turbulent pipe simulation separately and recorded the data while the turbulence was fully developed, and we use this as a time-varying Dirichlet condition. We also perform simulations with a constant profile for the liquid inflow, for the sake of comparison. Surrounding the jet, the remainder of the bottom face of the computational domain is made to be a wall. The boundaries in the spanwise directions are considered periodic, and the remaining boundaries are treated with convective boundary conditions.

To model the fluids in this simulation we use
the stiffened-gas equation of state, where the relationship between pressure, \( p \), and specific internal energy, \( \rho e \), is

\[
p = (\gamma - 1) (\rho e) - \gamma p^\infty.
\]  

(1)

For the equation of state of each phase, we use the material properties listed in Table 1.

<table>
<thead>
<tr>
<th></th>
<th>( \gamma )</th>
<th>( p^\infty )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gas</td>
<td>1.4</td>
<td>0</td>
</tr>
<tr>
<td>Liquid</td>
<td>4.4</td>
<td>( 6 \times 10^8 ) Pa</td>
</tr>
</tbody>
</table>

Table 1. Material properties used for each phase in the second and third test cases. The expression for the equation of state is shown by (1).

In this study, the computational mesh is refined to 18 cells across the diameter of the jet, which is 1.02 mm, and it extends 18.6 diameters past the jet. Above and outside of the wake of the jet, we reduce the mesh resolution for the sake of computational efficiency. In the future, we plan to simulate this flow at 2 to 3 times the current resolution, such that the mesh size can be on the order of the smallest scales of the flow.

We begin recording statistics after the liquid jet has reached a steady penetration height into the domain. The initial transient is complete after 16 flow-through times, and then statistics are sampled over 29 flow-through times. Data from the simulation with a constant liquid inflow condition, shown for comparison, are sampled over 21 flow-through times.

Finally, in this simulation, we ignore heat conduction and keep the surface tension and viscosity constant. To account for turbulent dissipation not representable by the resolution of the mesh, we dynamically calculate an eddy viscosity along Lagrangian pathlines [17].

Characterizing the flow

As the liquid is injected into the domain, a bow shock forms in front of the windward side of the jet. As the strong crossflow deflects the jet and exposes it to shear, surface instabilities form on this side of the jet, introducing disturbances that propagate and grow downstream and contribute to the atomization process. The presence of the jet induces a low-pressure wake behind it, creating a recirculation region there. These flow features are shown in the simulation snapshot included in Figure 1. In seeking a quantitative understanding of this flow, we begin by illustrating the detail of basic aspects of the flow, including the streamlines surrounding the jet, the penetration height of the spray, and the behavior of surface instabilities.

To look more closely at the flow surrounding the liquid jet, we plot streamlines constructed from the mean velocity, as shown in Figure 2. At the \( x \)-location of the jet, corresponding to the first picture, the flow is diverted around the jet, which has a smaller velocity around it due to the presence of the bow shock upstream, and narrow vortices form near the wall. At an \( x \)-location 2.5 diameters downstream, a pair of vortices has formed in the wake of the jet, constituting a recirculation region, and the size of the shock profile has grown. The presence of the volume fraction isosurface shows the deflection of the jet, indicating that the atomization process has not fully dispersed the liquid phase at this location.

Next, we investigate the penetration height of the spray. In [16], the penetration height is measured using a “90% transmittance rule”, applied to an averaged shadowgraph image. To mimic this methodology, we average a 2D projection of the volume fraction field to make a numerical approximation of a shadowgraph. We calculated the 0.1 isocontour of this average volume fraction field, using the presence of the liquid phase as an estimation for opacity. In Figure 3, we compare penetration curves from simulations with an experimental correlation. Lin et al. developed this correlation for pure liquid jets in supersonic crossflow [8], relating the penetration height, \( h_0 \), to the liquid-to-gas momentum ratio, \( q_0 \).
the atomization process of the liquid is the formation of instabilities on the liquid surface. To measure the frequency of these instabilities, we created a temporal signal at the center plane by calculating the normal displacement of the liquid surface from the mean liquid profile, averaged over the simulation. We calculated multiple temporal signals along the arc length of the mean jet surface. From each temporal signal, we determined the dominant frequency at each location. In Figure 4, we show the dominant frequency as a function of arc length and compare results between simulations with different inflow conditions.

Comparing the two different simulations reveals that the behavior of the surface instability is radically different in each. The simulation with a turbulent liquid inflow has a uniform dominant frequency, with exceptions near the endpoints. For the simulation with a constant liquid inflow, however, the dominant frequency varies considerably along the surface of the jet. The mean dominant frequency of the constant inflow case is lower than that of the turbulent inflow case, which is as expected. For comparison, Arienti and Soteriou’s study of a low-Mach liquid jet in crossflow reports dominant frequencies in the range of 3.8 to 5.6 kHz [18], while the mean dominant frequencies in our study are 11.2 and 7.9 kHz.
Characterizing the spray

In addition to understanding some of the key features of the flow, we also quantitatively measure the resulting spray. Figure 5 shows the probability density function of the droplet size. Both distributions peak at the smallest value of diameter, 0.7 µm. However, this result is well below the mesh resolution of 56 µm. Atomization taking place below the size of the mesh is not resolved or physical; it is basically only noise that results from the numerical treatment. Thus, data below the mesh size should not be considered accurate. Above this limit, the distribution peaks at a droplet size near 100 µm.

In Figure 6, we show time-averaged information from the droplet distribution, measured at 18 diameters downstream from the jet. The $U$ velocity distribution bears the distinctive horseshoe shape that is characteristic of this flow, with a velocity deficit behind the jet. The fastest droplets move at the speed of the crossflow. The $V$ and $W$ velocity distributions together indicate the presence of a recirculating region behind the flow, where droplets move in a counterclockwise pattern on the left side of the centerline ($z = 0$), as shown, and move clockwise on the right side. The number density indicates that the droplets are concentrated in packets, appearing like bands around the deflected jet. This close to the jet, the Sauter Mean Diameter remains high, with the largest SMD measurements being on the order of 200 µm. The highest values of SMD are near the jet, but there are also high values displaced laterally away from the jet and near the wall. The maximum volume flux occurs slightly away from the centerline.

Lin et al. report phase Doppler particle analyzer (PDPA) results in [8], showing data recorded 200 diameters downstream from the jet. Because the sampling location is so different, it is difficult to compare with, but the $U$ velocity is remarkably similar and the number density is concentrated in band-like regions around the deflected jet.

Conclusions and Future Work

In conclusion, we have quantitatively characterized a liquid jet in supersonic crossflow. Although the simulations referenced in this work are relatively coarse, we are currently performing mesh refinement. This study showcases the capabilities of our compressible multiphase flow solver, and suggests that numerical simulations are on the cusp of providing valuable insight about this relevant flow scenario.

Acknowledgments

This work was supported by Taitech, Inc. and the Air Force Research Laboratories. Computing resources through AFRL DSRC on Thunder are also gratefully acknowledged.
References


Figure 6. 2D pseudocolor plots created from droplet data sampled at 18 diameters downstream of the jet. The plots depict the droplet velocity in the streamwise ($x$) direction, the droplet velocity in the vertical ($y$) direction, droplet velocity in the spanwise ($z$) direction, the number density, the Sauter Mean Diameter, and the volume flux in the $x$-direction.