High-Fidelity Simulations of Fuel Injection and Atomization of a Hybrid Air-Blast Atomizer

P. C. Ma, L. Esclapez, S. E. Carbajal, and M. Ihme
Stanford University, Stanford, CA 94305

T. Buschhagen, S. V. Naik, J. P. Gore, and R. P. Lucht
Purdue University, West Lafayette, IN 47907

This paper presents high-fidelity simulations of fuel injection and atomization in a hybrid air-blast atomizer for realistic gas turbine engines. In particular, the fuel injection and atomization characteristics under lean-blowout conditions are considered which corresponds to a low liquid Weber number. A Volume-of-Fluid approach is combined with a Lagrangian particle framework to accurately simulate the fuel injection and atomization for the pilot pressure-swirl atomizer. The simulation results are compared to the experimental measurements from PDPA techniques. Good agreement is found for spray angle, droplet size and droplet velocities. However, the simulation predicts an early breakup compared to the experiments due to the stringent mesh resolution requirement under low liquid Weber number conditions.

I. Introduction

Fuel injection and atomization are of direct importance to the design of injector systems in aviation gas turbine engines. Primary and secondary breakup processes have significant influence on the drop-size distribution, fuel deposition, and flame stabilization, thereby directly affecting fuel conversion, combustion stability, and emission formation, etc. However, main issues in improving injector systems are the lack of predictive modeling capabilities for the reliable characterization of primary and secondary breakup mechanisms. Although several correlations have been developed for describing the atomization process, fundamental understanding is needed for the accurate prediction of the spray distribution, and the implementation of advanced injection strategies.

Due to the restricted access of experimental measurements to the primary atomization region near injectors, data collected from experiments is often limited to downstream regions of the nozzle where fuel droplet statistics have already reached steady state and the atomization processes have already taken place. Thanks to the recent development in numerical methods and computational resources, numerical simulations utilizing the large-eddy simulation (LES) methods are alternative approaches to the experimental investigations for the studies of understanding complicated breakup processes.

The objective of this study is to use a LES approach in conjunction with an unstructured Volume-of-Fluid (VoF)/Lagrangian-spray (LSP) framework to conduct high-fidelity simulations of breakup and atomization processes in a realistic gas turbine hybrid air blast atomizer. Conditions corresponding to lean blowout is considered in this study to assess the capabilities of the current model under very low liquid Weber number. Simulation results for pilot injection with POSF 10264 (A-1) fuel are presented. Fuel droplet statistics are compared to available experimental data measured utilizing phase doppler particle analyzer (PDPA) systems.

*Graduate Research Assistant, Department of Mechanical Engineering, AIAA Student Member.
†Post-Doctoral Fellow, Center for Turbulent Research.
‡Graduate Research Assistant, Department of Mechanical Engineering.
§Assistant Professor, Department of Mechanical Engineering.
*Graduate Research Assistant, School of Aeronautics & Astronautics, AIAA Student Member.
∥Senior Research Scientist, School of Mechanical Engineering, AIAA Associate Fellow.
∗∗Professor, School of Mechanical Engineering, AIAA Fellow.
II. Experimental Setup

Measurements with a Parker-Hannifin hybrid air-blast injector were performed at Purdue University for different aviation fuels and operating conditions under the FAA National Jet Fuel Combustion Program. The PDPA system is utilized for the measurement of droplet size and droplet velocities. An Argon ion laser (488 nm and 514.5 nm) was used with a 400 nm transmitter focal length and a 310 nm receiver focal length. Extinction tomographic measurements using an optical patternator were used to measure the liquid surface area per unit volume, and details about the measurement techniques can be found in Lim et al.9, 10 The experimental setup is shown in Fig. 1.

The injector considered in this study is a hybrid air blast atomizer7, 8 developed by Parker-Hannifin. The hybrid atomizer consists of a low flow number (FN = 4) pressure swirl pilot nozzle and a circuit of five main injectors at a flow number of 15. The pressure swirl atomizer has wider stability region while the main air-blast atomizer provides improved combustion efficiency. The hybrid air blast atomizer takes advantage of both the pressure swirl injector at low fuel-flow rates and air blast atomization at high fuel flows. The atomizer has a 90 degree spray angle.

The target conditions considered in this study is of 100 % pilot injection with the PSOF 10264 (A-1) fuel at ambient conditions. The fuel pressure drop is 1.7 atm with a mass flow rate of 2.5 g/s, and the ambient conditions are 1 atm and 15.6°C. This fuel pressure drop and mass flow rate correspond to the lean blowout conditions where the fuel spray has a very low liquid Weber number. Figure 2 shows the dynamic viscosity and the surface tension of different fuels as a function of temperature. Data is obtained through experimental measurements from the FAA project. Table 1 gives the fuel properties and injection conditions corresponding to the current study.

III. Numerical Solver

In this study, a VoF-method coupled with a LSP-framework5, 6 is adopted. The incompressible Navier-Stokes equations for immiscible, two-phase flows are solved to describe the flow field. Density and viscosity are assumed to be constant within each phase, and can be expressed as a function of the volume fraction φ:

\[ \rho = \phi \rho_1 + (1 - \phi) \rho_2, \]
\[ \mu = \phi \mu_1 + (1 - \phi) \mu_2, \]
Figure 2. Properties of different fuels as a function of temperature. The POSF 10264 (A-1) fuel injected at 15.6°C is considered in this study, which is marked with black dots.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Test condition</td>
<td>100% pilot</td>
</tr>
<tr>
<td>Fuel</td>
<td>POSF 10264 (A-1)</td>
</tr>
<tr>
<td>Fuel flow rate</td>
<td>2.5 g/s</td>
</tr>
<tr>
<td>Pilot pressure drop</td>
<td>1.7 atm</td>
</tr>
<tr>
<td>Fuel temperature</td>
<td>15.6°C</td>
</tr>
<tr>
<td>Fuel density</td>
<td>779.4 kg/m³</td>
</tr>
<tr>
<td>Fuel viscosity</td>
<td>1.64e-3 Pa·s</td>
</tr>
<tr>
<td>Fuel surface tension</td>
<td>0.024 N/m</td>
</tr>
<tr>
<td>Ambient pressure</td>
<td>1 atm</td>
</tr>
<tr>
<td>Ambient temperature</td>
<td>15.6°C</td>
</tr>
<tr>
<td>Air density</td>
<td>1.18 kg/m³</td>
</tr>
<tr>
<td>Air viscosity</td>
<td>1.86e-5 Pa·s</td>
</tr>
</tbody>
</table>

Table 1. Fuel properties and injection conditions.

where the subscripts denote different fluid phases. The Piecewise-Linear Interface Calculation (PLIC) scheme is adopted, which has advantages in conserving the mass and constructing monotone advection schemes. The overall VoF-scheme is geometric and unsplit, enforcing exact mass conservation on unstructured grids.

The VoF-method is coupled to the LSP-framework to describe the secondary breakup dynamics, which cannot be fully resolved using available computational resources. The Lagrangian particle method is applicable to droplets with small local Weber numbers in the subsequent breakup and atomization processes. In this manner, the primary breakup and the subsequent atomization can be modeled efficiently. The subgrid stresses in the LES-approach were described using a Vreman model.

For the LSP-method, the liquid droplet motion is simulated using the Basset-Boussinesq-Oseen (BBO) equation with shear force, Basset force and added mass neglected. The further breakup of Lagrangian particles into smaller drops is modeled by a stochastic breakup model. Details about the numerical implementation can be found in Kim et al.⁶
IV. Geometry and Mesh Generation

The computational domain is shown in Fig. 3 with zoomed view of the near nozzle region. To provide an accurate description of the primary break-up dynamics and cone-angle, a portion of the injector nozzle is included. The geometry used in the simulation is the same as the geometry provided by Parker-Hannifin. The injector nozzle has three feeding slots which serve as inlet boundary conditions in the simulation. According to the mass flow rate in the experiments as shown in Table 1, the inlet velocity at each feeding slot is set to 12.4 m/s. A swirl chamber is located upstream of the discharge orifice. The discharge orifice of the pilot injector has a nozzle diameter of \( D = 1.31 \) mm, and the feeding slots have diameters of about 0.33 mm. The computational domain is described by a cylinder with \( 300D \) diameter and \( 150D \) height. The geometry of the air-box and the center-cap used to install the injector are also included in the computation domain to have a better description of the recirculation region outside the injector. No-slip boundary conditions are applied to the walls inside the pilot injector, the walls of the air-box, and the upper face of the cylinder. Convective outflow boundary conditions are applied at all other walls of the cylinder.

As shown in Table 1, the current working condition has a very low fuel mass flow rate and fuel pressure drop which corresponds to lean blowout conditions considered in the experiment. In the following, a scaling analysis will be conducted to show the stringent mesh resolution requirement under such conditions and the consequences on the following mesh generation processes.

The liquid sheet thickness at the nozzle exit, \( t_f \), can be estimated using the correlation given by Rizk and Lefebvre:

\[
 t_f = 3.66 \left( \frac{\dot{m} D \mu_f}{\rho_f \Delta P} \right)^{0.25} \sim \dot{m}^{-0.25}, \tag{3}
\]

where, \( \dot{m} \) is the fuel mass flow rate, \( D \) is the nozzle diameter, \( \mu_f \) is the fuel dynamic viscosity, \( \rho_f \) is the density of the fuel, and \( \Delta P \) is the fuel pressure drop. Note the flow number, defined as \( FN = \dot{m} / \sqrt{\Delta P} \), is a constant. This formula gives about 0.2 mm liquid sheet thickness inside the discharge orifice and this can be well resolved since the total volume inside the injector is limited compared to the open domain outside the injector. From Eq. (3), it can be seen that \( t_f \) is a weak function of \( \dot{m} \).

With liquid sheet thickness \( t_f \) and mass flow rate \( \dot{m} \), the liquid velocity at the nozzle exit can be estimated based on mass conservation:

\[
 U = \frac{\dot{m}}{\rho_f \pi t_f (D - t_f) \cos(\theta/2)} \sim \dot{m}^{1.25}, \tag{4}
\]
where \( \theta \) is the spray angle. The liquid Weber number can be defined as

\[
We_l = \frac{\rho_f U^2 t_f}{\sigma} \sim \dot{m}^{2.25},
\]

(5)

where \( \sigma \) is the surface tension of the fuel. As can be seen from Eq. (5), the liquid Weber number is a strong function of the mass flow rate, and the current lean blowout condition with very low mass flow rate yields a also very low liquid Weber number which is about 400.

The liquid sheet length or the breakup length of liquid spray, \( L_b \), has been shown to increase rapidly with decreasing liquid Weber number.\[\text{11}\] The liquid sheet thickness right before breakup can be estimated using mass conservation:

\[
t_{f,b} = \frac{t_f(D - t_f)\cos(\theta/2)}{D + 2L_b \tan(\theta/2)}.
\]

(6)

Since \( t_f \) is a weak function of \( \dot{m} \) and \( L_b \) increases with decreasing \( \dot{m} \), \( t_{f,b} \) also decrease rapidly when \( \dot{m} \) is small. For the conditions considered in this study, \( t_f \) is about 0.2 mm, \( L_b \) is about 10 mm from the experiment, and the spray angle is 90 degrees. This gives us a liquid sheet thickness of about 4 \( \mu m \). The droplet diameter after the atomization processes can be estimated using the correlation given by Lefebvre\[\text{11}\]

\[
\text{SMD} = 2.25 \sigma^{0.25} \rho_f^{0.25} \dot{m}^{0.25} \Delta P^{-0.5} \rho_a^{-0.25} \sim \dot{m}^{-0.75},
\]

(7)

where SMD is the Sauter mean diameter, and \( \rho_a \) is the density of air. Under the current conditions, the correlation gives an SMD of 90 \( \mu m \).

To sum up the scaling analysis, the lean blowout condition considered in this study is characterized by a very low mass flow rate, which results in a considerably low liquid Weber number. The low Weber number means longer liquid sheet and hence thinner liquid sheet thickness, both of which impose challenging mesh resolution requirement in numerical simulations. The droplet diameter is also affected by the mass flow rate but this does not give any restrictions on the mesh resolution requirement.

Based on the scaling analysis, the mesh utilized in the current study is generated subsequently. A fully structured skeleton mesh with about 0.2 million hexahedron cells was generated and locally adapted in the
near-nozzle region following the spray and droplet trajectory. A relatively coarse mesh was initially used to determine the region needed to be refined. Due to the stringent resolution requirement under the lean blowout condition, the final mesh utilized in the current study is a 120 degree section of the cylinder with periodic boundary conditions in the azimuthal direction. The final mesh consists of about 50 million cells, most of which are hex cells. Inside the nozzle, the minimum mesh spacing is 0.01D (13 µm). The truncated cone with 90° spreading angle from the discharge orifice to 2D downstream has a maximum cell size of 0.05D and follows the trajectory of the spray all the way to 15 mm downstream the nozzle exit, the minimum mesh spacing is 0.02D (26 µm). The mesh transitions from the fine region to the coarse region. Figure 4 displays the mesh at the cut-plane of z = 0 (origin at center of nozzle exit) and the zoomed view in the near nozzle region.

V. Results and Discussions

Simulation results for 100% pilot with POSF 10264 (A-1) fuel corresponding to the conditions in Table 1 are presented in this section. The simulation was first performed without LSP model. Once the flow-field is statistically stationary with regard to the VoF-field, the LSP approach is then activated for the modeling of the subsequent breakup and atomization processes.

Figure 5 displays representative simulation results for two different view angles. The iso-surface of VoF $\phi = 0.5$ is shown in green for displaying the liquid sheet and the liquid ligaments. The Lagrangian particles are shown in blue sphere symbol and the size of the particles are scaled by the droplet diameters. It can be seen that the liquid fuel is discharged from the nozzle exit in the form of a conical hollow liquid sheet due to the swirling provided by the atomizer. As the liquid sheet expands, perforations are formed and the liquid sheet deforms into liquid ligaments. At this low Weber numbers, the main mechanism for the breaking up of the liquid fuel spray is the hydrodynamic instability within the liquid sheet rather than the external aerodynamic forces. This is also found in the current study that there is little interaction between the liquid sheet and the surrounding air and the turbulence level is low in the velocity field. Finally, the liquid ligaments further break up and disintegrate into fuel droplets which are represented as Lagrangian particles in the simulation. Due to the lean blowout conditions considered here, after the primary breakup the fuel droplets do not go through further secondary breakup because the shear force between the fuel droplet and air is so small that the local droplet Weber number is on the order of unity. The numerical simulation qualitatively captures essential features of the breakup and atomization processes for the pilot pressure-swirl atomizer under the conditions considered, demonstrating the capabilities of the current modeling techniques.

Figure 6 shows simulation results and the experimental results next to each other for comparison. The instantaneous experimental image was taken from a high-speed camera. The simulation results show the iso-surface of $\phi = 0.5$ augmented with Lagrangian particles scaled by droplet size, both of which are colored green. From Fig. 6 it can be seen that the spray angle predicted by the numerical simulation is about 90 degrees which is in good agreement with experiment. However, the breakup length predicted by the
Figure 6. Comparison between experiment (left) and numerical simulation (right). The image on the left was taken by a high-speed camera in the experiment. Simulation results for iso-surface $\phi = 0.5$ and Lagrangian particles scaled by droplet size are both colored in green shown on the right.

Simulation is shorter than the experiment as can be seen in Fig. 6. The early breakup in the simulation is believed to be due to the stringent mesh resolution requirement needed under the current low Weber number conditions. As discussed in the scaling analysis, the liquid sheet thickness right before breakup can be as thin as $4 \mu m$ while the minimum resolution of the computational mesh is about $26 \mu m$. To resolve the thin liquid sheet, typically at least 4-6 grid points are needed. Moreover, the low Weber number condition also yields a long liquid sheet which needs to be resolved.

Figure 7 shows the fuel droplet statistics collected from the simulation at the measurement plane which is 20 mm below the nozzle exit. Figure 7(a) shows the droplet size distribution at the measurement plane along with the log-normal and Rosin-Rammler fitting to the simulation results. It can be seen at the measurement plane, the droplet distribution corresponds more likely to a log-normal distribution and has a peak slightly below 100 $\mu m$. Figures 7(b) to 7(d) show the simulation results of the droplet SMD and droplet velocities at the measurement plane in comparison with the experimental measurements. The simulation results are averaged in both time and the azimuthal direction and then binned at different radial distances to collect statistics. Due to calibration issues in the experiment, the SMD measurements are expected to be 20 $\mu m$ smaller. In Fig. 7(b) the corrected experimental results are shown in solid symbols along with raw data in open symbols. The correlation for SMD, given by Eq. (7), is also shown in Fig. 7(b) for comparison. In the center of hollow cone (radial distance $\leq 5$ mm), no droplets were found in the simulation due to the shorter runtime and therefore zero values are plotted in all three figures, while in the experiment, at these radial distances droplets were collected by the PDPA system. As can be seen in Fig. 7(b) the simulation results are in good agreement with the corrected experimental data. Both the experiment and simulation have higher SMD compared to the value predicted by the correlation. Results for the droplet axial and radial velocities show that the simulation results are in good agreement with the experimental measurements.

VI. Conclusions

In this study, an unstructured VoF-method was adopted and coupled with an LSP-framework to conduct high-fidelity simulations of the primary breakup and atomization processes of a hybrid air-blast atomizer under lean blowout conditions. Full pilot operation is considered with the POSF 10264 (A-1) fuel. Simulation
Figure 7. Statistics of fuel droplet at measurement plane $x = 20$ mm downstream the nozzle exit. Log-normal and Rosin-Rammler distributions are used to fit the simulation results of the droplet diameters. Simulation results at the measurement plane are averaged in both time and azimuthal direction and then binned at different radial locations. One std is shown for the error bar of the simulation results. Correlation of SMD given by Eq. (7) is also shown for comparison.

results are compared measurements collected using a PDPA system.

The current numerical schemes captured essential features of the fuel breakup and atomization processes in comparison with experiments. Due to the stringent mesh resolution requirement imposed by the low liquid Weber number corresponding to the lean blowout conditions, the thin liquid sheet cannot be fully resolved with the currently available computational resources and therefore an early breakup was observed in the simulation results compared to the experiment. The droplet size and droplet velocities predicted by the numerical simulation were compared to the experimental measurements at 20 mm below the nozzle exit and good agreement was observed, demonstrating the predictive capabilities of the current numerical techniques.

Ongoing work includes considering swirling effects on the spray breakup and atomization processes where the secondary breakup may take place due to higher shear experienced by the fuel droplets. Fuel effects are also considered by studying the atomization characteristics of different types of fuels.
Acknowledgments

Financial support through the FAA National Jet Fuel Combustion Program is gratefully acknowledged. We would like to thank the FAA-program for providing the injector geometry. Helpful discussions with Dr. Dokyun Kim at Cascade Technologies are appreciated.

References