Impact of Operating Conditions on the Spray in a High-Shear Nozzle/Swirler Injector Investigated using High-Fidelity Simulations

Xiaoyi Li*
United Technologies Research Center
East Hartford, CT 06108 USA

Abstract
Practical aero-engine fuel injection systems are highly complicated, combining complex fuel atomizing nozzle and air swirling elements to achieve good fuel-air mixing as well as long residence time in order to enhance both combustion efficiency and stability. Detailed understanding of the multiscale, multiphase flow processes (liquid jet atomization, swirling liquid transport, film formation/atomization) occurring in a realistic injector has been limited due to the complex geometries and the challenges encountered in experimental measurements, especially under the high temperature-pressure operating environment in aero-engine combustors. Recent advances in numerical methods and increases in computational power have enabled the first-principle high-fidelity simulation of such processes to achieve a comprehensive physics-based understanding by probing into the near-field two-phase details. In the past, we have conducted a detailed simulation of a high-shear nozzle/swirler injector at ambient conditions and validated the results against experimental measurements. In this work, we extend the same computational approach to study the impact of operating conditions on the spray physics in such a complex device. Numerical algorithms to handle liquid evaporation at elevated temperature have been developed. The previously validated ambient condition case was used as the baseline for comparison with other cases with higher temperature, pressure, and liquid fuel flow rate. The high fuel loading cases are compared with low fuel loading ones to help understanding the impact of filming on injector performance. The spray details regarding jet trajectory, atomization degree, film thickness, and jet-to-film volume flux ratio were analyzed under the ambient as well as the elevated conditions. It was observed that high temperature/pressure condition has little impact on the gaseous flow upstream of fuel injection, but significantly modifies in the near-field liquid spray inside the swirler in various aspects. The condition-induced changes also depend on the dominant liquid atomization mechanisms (jet-in-swirling-flow atomization or film-edge breakup), which are determined by the fuel loading conditions.

*lxy2@utrc.utc.com
1. Introduction

The atomization of liquid fuel and its subsequent evaporation and mixing with the air streams are critical to the performance of combustors in aerospace propulsion applications. The spatiotemporal distribution of the resulting fuel-air mixture controls the efficiency, stability, noise and emission of the combustion process. Due to the high energy density of the liquid fuel, small changes at the introduction of the liquid in the combustor have a massively amplified impact on the downstream combustion process. Therefore, highly complicated fuel injection devices are designed to optimally control the fuel spray quality over a wide range of operating regimes. Strong demand exists for an in-depth physical understanding of liquid fuel atomization process in such devices in order to further improve the fuel-injector design.

A typical example of a fuel injector for aerospace propulsion devices is shown in Fig. 1 [1]. In such injectors, the swirler is used to generate/amplify the aerodynamic shear forces that are used to atomize the liquid fuel, increase fuel residence time, and create recirculation zones downstream to provide flame stabilization. Inside the inner swirler, the liquid fuel is injected from a nozzle with several orifices normal to the wall of the swirler passage. Liquid breakup occurs mostly due to the interactions between the liquid fuel and the swirling air. Depending on the engine operating conditions, liquid may reach the opposite wall of the swirler passage and create a film structure, which is further atomized at the swirler exit by the shear generated between the inner and outer swirling air streams. Combustion of the swirling fuel-air mixture occurs downstream beyond the swirler exit.

The complex injector geometries and harsh operating conditions pose significant challenges for understanding and designing such fuel injectors through experimental analyses. The obstruction presented by the injector geometry restricts the high-speed spray imaging (shadowgraph/holograph etc.) to be conducted at downstream locations beyond the injector exit. Other spray measurements also have to be made significantly downstream of the injector where the physical impact of the spray is moderate (relevant to intrusive measurements such as Patternation) and the spray is less dense (relevant to non-intrusive measurements such as PDA). The challenge may be partially addressed using advanced X-ray diagnostics [2, 3]. Yet for all these approaches, measurements have to be taken in relatively low temperature, low pressure, non-combusting (often ambient) environments. Liquid properties (such as density, viscosity and surface tension) that affect the breakup processes are highly dependent upon operating conditions [4]. With such constraints, these measurements cannot be directly used to assess the impact of the injector design on the combustor operation. It becomes quite difficult to make informed design decisions, since correct conceptualization of complex two-phase flow process inside the injector may be lacking. The primary use of such measurements is in the validation of models that can then be used to predict the sprays inside the injector at realistic conditions.

Meanwhile, current engineering simulations of aero-engine combustors mostly rely on empiricism-based phenomenological spray models [5-12], which were developed/calibrated by data obtained mostly in canonical configurations [13-16]. Such phenomenological models have been used to capture individual sub-processes including jet primary breakup, secondary breakup, droplet collisions, and spray/wall interactions. The cost of executing these models is low and their ability to reproduce the experimental data within the regime of calibration tends to be good. Clearly, one of the drawbacks is that experimental data must always be available a priori for calibration. Unfortunately, such spray measurements were mostly acquired in the canonical configurations at ambient conditions, and had quite low spatiotemporal resolution [13, 14]. Extrapolating the model to high temperature-pressure engine conditions poses high risks due to the lack of a physics-basis for these models. Therefore, the constraints in the calibration data severely limit the predictive ability of these models when applied to realistic systems.

In the recent past, first-principle based high-fidelity simulations utilizing direct interface tracking/capturing methods have been demonstrated to be able to predict atomization and fuel-air mixing processes in a number of canonical configurations [17-23] including straight liquid jet into quiescent gas and liquid jet in cross flow. While the broad range of spatial and temporal scales that emerge in the flow poses significant challenges on the computational cost, the expensive execution of such high fidelity simulations has been enabled by recent advances in computer software and hardware, particularly for modern high-performance-computing platforms. Still, the simulations have been mostly executed under simple canonical settings. The extension to more complex realistic injector systems for aerospace applications has been rare except in the recent work by Li and co-workers [24, 25]. Methodology enhancements have been developed to deal with the additional physics that is present and to treat the complex injector geometry. In particular, they have used a hybrid Eulerian-Lagrangian method that combines the Coupled Level-Set and Volume Of Fluid (CLSVOF) [26] method to track/capture liquid-gas interfaces, and a Lagrangian droplet transformation/tracking approach to capture the smallest spherical droplets [27]. The framework applies Adaptive Mesh Refinement (AMR) [28, 29] to cluster grid points around the liquid-gas interface. An embedded boundary (EB) methodology [30, 31] has been developed for the treatment of the injector geometry. The simulation results for a high-
shear nozzle/swirler injector have been validated by experimental measurements at the ambient condition [25]. Recently, they also developed numerical algorithms to capture liquid evaporation at high temperatures [32, 33], further enabling the methodology to simulate sprays in elevated operating conditions. The simulation results can offer valuable physical insight of the sprays in real injector geometries at true engine conditions, which is extremely challenging to obtain using other approaches. Therefore, the simulation approach can be leveraged to help evaluating/verifying injector design concepts and reducing the number of design cycles. Furthermore, such simulations at engine operating conditions can provide valuable data to develop/calibrate reduced-order spray models, which can be applied over a wide range of conditions in engine design.

As a first step, in this paper, we focus on using the developed computational framework to investigate the impact of operating conditions on the sprays physics in the same injector geometry as previously demonstrated. Development of reduced-order spray models based on such simulation analysis will be the focus of future work. Here, the previously validated ambient condition case will be used as the baseline for comparison with other cases with higher temperature, pressure, and liquid fuel flow rate. High fuel loading cases will be simulated and compared with low fuel loading ones to help understanding the impact of film on injector performance. The analysis will focus on the physics of the near-field two-phase flow inside the swirler where experimental measurements are difficult to obtain, because such physics drives spray dynamics downstream of the swirler.

The outline of this paper is as follows: The formulation and description of the main elements of the computational framework are highlighted first. The swirl injector geometry, operating conditions and computational set-up are described next. Qualitative comparisons of the gaseous flow and spray are followed by the quantitative analysis of the jet and film structures inside the swirler. Main conclusions are summarized at the end.

2. Computational Approach

2.1 Formulation and Numerical Methods

The incompressible non-evaporating two-phase flow of liquid and gas can be represented by a single-fluid formulation. The governing equations are
\[ \nabla \cdot \mathbf{u} = 0 \]
\[ \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = \frac{1}{\rho} \nabla \cdot (\rho \mathbf{I} + 2 \mu \mathbf{D}) - \frac{\kappa \nabla \Phi}{\rho} \]
where \( \rho \) is the pressure, \( \rho \) the density, \( \mu \) the viscosity, \( \mathbf{I} \) the identity tensor, \( \mathbf{D} \) the strain rate tensor, \( \kappa \) the surface tension, \( \Phi \) the local curvature, and \( H(\Phi) \) the Heaviside function defined as
\[ H(\Phi) = \begin{cases} 1 & \Phi \geq 0 \quad \text{(liquid)} \\ 0 & \Phi < 0 \quad \text{(gas)} \end{cases} \]

Here \( \Phi \) is a function that identifies the interface location. The density and viscosity are defined as
\[ \rho = \rho_l H(\Phi) + \rho_g [1 - H(\Phi)] \]
\[ \mu = \mu_l H(\Phi) + \mu_g [1 - H(\Phi)] \]
where the subscript \( l \) and \( g \) denote liquid and gas phase, respectively. The motion of the interface follows
\[ \frac{\partial \Phi}{\partial t} + \mathbf{u} \cdot \nabla \Phi = 0 \]

When two-phase evaporation needs to be represented, the flow can be formulated by introducing the energy and species conservation equations within each individual phase and imposing the evaporating condition (e.g. the Clausius-Clapeyron equation) at the interface. More details on the evaporating flow formulation can be found in Ref. [33].

Since the numerical methods adopted in this paper have been comprehensively described in our previous work [20, 21, 24, 32-34], only a brief overview is provided here for the completeness of the paper. Our computational approach uses the Coupled Level-Set and Volume Of Fluid (CLSVOF) method [35] to capture the liquid-gas interface. The method capitalizes on the advantages of both the accurate geometric interface representation in the level set method and the volume-preserving properties in the volume of fluid method. The Eulerian CLSVOF interface tracking method is implemented under the framework of a block structured Adaptive Mesh Refinement (AMR) [28, 35], and also coupled with a Lagrangian droplet transformation and tracking approach to capture the smallest spherical droplets with significant cost-saving benefits. The Eulerian to Lagrangian transformation follows the algorithms similar to the implementation by Herrmann [27] and a number of criteria (e.g. size, sphericity and diluteness criteria) are required to be met before the transformation occurs [20, 36]. The transformed droplets were tracked using standard droplet drag and evaporation models [32]. The flow solver features a two-fluid advection approach [35, 37] to avoid artificial smearing of the velocity field across the interface, which causes poorly resolved gas velocity gradients leading to solver divergence at high density ratios. The pressure projection equation is solved using a Multi-Grid Preconditioned Conjugate Gradient method (MGPCG). The method is augmented by a ghost fluid (GF) treatment for pressure and density jump conditions to achieve a stable and fast pressure solution. The Eulerian surface evaporation algorithms are also based on the GF approach to accurately capture the jump in heat fluxes, and therefore accurately compute the rate of evaporation. Such a suite of sharp interface treatments mitigate the problem of solver divergence that typically occurs at high density ratios.
To avoid the difficulty of creating body-fitted meshes for simulating two-phase flows in complex injector geometries, we use an embedded boundary method. Specifically, the technique relies on defining an additional level set signed distance function $\psi$ to represent the solid body. This level set function is defined as positive outside the body and negative inside the body. A first order implementation of wall boundary conditions is employed. The velocity is forced to be zero whenever the local $\psi$ value is below zero. In the current work, an isothermal and no-penetration boundary is also imposed by setting the temperature to be the solid temperature, and the mass diffusivity to be zero when the local $\psi$ value is below zero. A 90 degree contact angle is imposed for the liquid-gas interface when interacting with the solid boundary. More comprehensive discussions on the embedded boundary approach used here can be found in Ref. [30].

2.2 Computational Setup

In our previous work [24], the computational approach was validated against experimental measurements at ambient conditions. In this work, we inherit the same computational configuration and use the validated ambient case as the baseline for investigating the impact of operating conditions.

The details of the swirler/nozzle geometry used in this investigation are shown in Fig. 1. A pressurized fuel line was connected to the fuel injector with six identical nozzles (Fig. 1c) to inject the same amount of liquid from each nozzle. The angle between nozzle injection and the injector bar axis is 60 degrees. Pressurized air is allowed to travel through the 12 inner and the 12 outer passages of the swirler (Fig. 1b) to generate swirling air flow that interacts with the fuel jets. The swirling angles for the inner and outer passages are 50 and 71 degrees, respectively, and the flow split between them is 4.62 to 1.

Based on this injector geometry, we set up four cases with varying temperature ($T$), pressure ($P$), and fuel flow rate ($\dot{Q}$). The conditions and flow parameters are listed in Table 3. Case 1 is the baseline case at the ambient condition ($T_i=300K$, $P=10^4Pa$). Based on previous results for this case, the liquid jets do not penetrate far enough to form liquid films at the inner swirler wall. In Case 2, the fuel flow rate is doubled and the liquid jet penetrates farther, more likely reaching the swirler wall to create filiming structures. Case 3 and 4 correspond to Case 1 and 2 respectively, but at a much higher gaseous temperature ($T_g=811K$) and operating pressure ($P=5 \times 10^6Pa$). For all the cases, the liquid phase is introduced at ambient temperature ($T_i=300K$) and reaches a maximum liquid temperature at the liquid-air interface. The solid wall is maintained at the same temperature $T_g$ as the gas phase. In this work, we assume the existence of sub-critical two-phase interfaces at all conditions. Here, we also make the low Mach number assumption, and assume the impact of fluid dynamic pressure on the change of air density is negligible. The two-phase thermodynamic and transport properties are computed based on the operating pressure and local temperature conditions. The $P-T$ dependence of the liquid properties are represented by empirical correlations extracted from Ref. [38]. The gaseous phase behavior is assumed to follow the ideal gas law and the $P-T$ dependence of the gas (mixture of air and fuel-vapor) properties is represented by the correlations in Ref. [39, 40]. The computed properties of the fluids (Jet-A liquid and air) at inlet conditions are listed in Table 1 and 2.

Liquid jet atomization in crossflow has commonly been characterized in terms of the momentum flux ratio $q=p_i U_i^2/p_g U_g^2$ and gas Weber number $We=\rho_p U_g^2 d_0/a$. For the case of liquid jet in a swirling crossflow, if assuming a similarity with plain crossflow atomization, we can compute the corresponding $q$ and $We$ as listed in Table 3. Here, $U_1=2Q_i/(3\pi \rho_d d_0^2)$ is the fuel injection velocity, which is computed from the total fuel flow rate $Q_i$ (for six fuel orifices) and orifice diameter $d_0=0.64 mm$. The gas flow velocity in the inner passage $U_{g,i}=Q_g/(\rho_{g} A_{i})$ can be estimated based on the total inner-swirler air flow rate $Q_g$ and the estimated swirler inner annular channel area of $A_{i,n}=432 mm^2$. Case 3 and 4 are scaled from Case 1 and 2, respectively, by maintaining the same $q$, with the expectation that these cases at different $P-T$ conditions have similar jet penetration and trajectory. As the density ratios are determined by the $P-T$ conditions, the velocity ratio $r_1=U_1/U_{g,i}$ can be computed using the constant $q$ scaling. The inflow conditions can be completely determined if $U_1$ or $U_{g,i}$ values are further specified. Here we choose to maintain the same gaseous velocity $U_{g,i}$ for all the cases. Apart from $q$ and $We$, the other three independent non-dimensional parameters that control atomization physics are the Reynolds number $Re=\rho_p U_g d_0 \mu_g$, the density ratio $r_{d}=\rho_d/\rho_g$ and the viscosity ratio $r_{\mu}=\mu_{\mu,\mu}$ as shown in Table 3. The $Re$ for all the cases are quite large (>10⁶), indicating that the impact of viscosity is small. As shown in a recent study of a canonical liquid jet in a crossflow [23], varying density ratio leads to a significant impact on the atomization physics.

As in previous work [24], the injector simulation was performed in a rectangular domain of size $3cm \times 6cm \times 6cm$ as shown in Fig. 2. The longitudinal $x$-direction is aligned with the axis of the injector. The swirler and injector geometry is used to initialize a solid level set field and create an embedded boundary for the simulation. An embedded wall of 0.75cm thick (horizontal plate in Fig. 2) as in the experimental setup separates the domain into an upstream and a downstream section. Fig. 2 also shows the main types of boundary conditions employed. A no-penetration slip wall boundary is implemented for the four transverse faces in the upstream sec-
tion. This allows an easy calculation of inflow gas velocity at the inlet boundary by matching with the flow rate. The fuel is forced into the nozzle at a flow rate of $Q_f$. For the downstream section, out flow open boundary condition is imposed to allow convection out of the domain. We define a based grid of $192 \times 384 \times 384$ and apply 3 levels of refinement for the liquid-gas interface near the nozzle, which leads to a refined grid size of $\Delta x_{ref} = 19.5 \mu m$ ($dx/\Delta x_{refine} = 32$). The simulations were performed for 10000 steps ($1ms$ in physical time) before approximately reaching stationary state and an additional 10000 steps ($1ms = 7.5$ flow-through time, flow-through time estimated based on velocity of $71m/s$ and a distance of $1cm$ between jet orifice and injector exit) for producing results for analysis. The cost of each simulation case was 30 days with 2000 processors (CRAY 2.6GHz) running in parallel.

2.3 Calculation of the Degree of Atomization

A common way to quantify the degree of liquid atomization is to measure or extract the size of droplets after liquid breakup. As in our previous simulation work of liquid jet in crossflow [21], the droplet size information becomes readily available after the droplet structures are transformed from the Eulerian to the Lagrangian representation. The transformation approach using pre-defined size (smaller than AMR coarse grid) and sphericity criteria works well for sampling droplets at planes sufficiently downstream of the primary atomization. The liquid is atomized to a large degree that all the droplets can meet the criteria and be transformed into the Lagrangian representation before reaching the plane. However, to quantify the degree of atomization for near-field (e.g. inside the swirler) liquid structures, the transformation approach cannot be applied because of the existence of large and highly deformed ligaments/blobs, which under the same transformation criteria remain in the Eulerian representation. In fact, in the current simulation, the Lagrangian transformation is turned off inside the swirler for the reason that we would like to capture the process of droplets impacting on the wall and forming liquid films under the Eulerian representation.

In this work, we employ an alternative approach to characterize the averaged degree of atomization in the more complex scenario. We further post-process the simulation results under Eulerian liquid representations by computing an averaged liquid volume to area ratio, and therefore the effective size (Sauter Mean Diameter or SMD) of the liquid structures. The liquid surface is first discretized using a set of surface elements. The surface area and volume are then computed by numerical surface integration $A = \sum \Delta S_i$ and $V = \sum (\mathbf{x}_i \cdot \mathbf{n}_i) \Delta S_i / 3$ (based on divergence theorem). These volume and area computations can be performed over any subset domain of interest. For example, the liquid film can be separately characterized by restricting the processing of liquid within a threshold distance ($d_{threshold} = 0.5mm$ in this work) from the swirler wall. Finally, the time-averaged effective size (SMD) of structures can be extracted as

$$D_{eff} = 6 \sum_k V_i^k \left/ \sum_k A_i^k \right.$$  \hspace{1cm} (10)

where the $k$ summation is over different time samples.

3. Results and Discussions

3.1 Qualitative gaseous flow comparison

Fig. 3 and 4 displays the details of the simulated flow field inside the swirler using $x-z$ plane cutting through the centerline of the swirler. The air is first forced to flow through the inner and outer swirler vane at different angles, and then pushed in the axial direction towards the swirler exit. The low pressure at the centerline axis causes the local recirculation of air in the axial direction (Fig. 3). At the swirler exit, the axial (Fig. 3) and azimuthal (Fig. 4) velocity differences between the air in the inner and outer swirler passages generate a strong local shear, potentially helping liquid films break up into finer droplets. For the 4 cases shown in Table 3, the inlet air velocity is maintained to be the same. As shown in Fig. 3 and 4, despite the drastic change in air properties (see Table 3) between the ambient and high T&P cases, the unsteady gas flow structures do not seem to be impacted much, especially in regions upstream of liquid injection orifice. Comparison of the mean velocities also confirms the same observation that the gaseous flow field is little impacted by the change of conditions. For the low fuel loading (Liq. 1X) cases in Fig. 3 and 4, the interaction of liquid jets with the gaseous flow seems to be similar between the ambient and high T&P conditions. As the fuel flow rate increases (Liq. 2X), the blockage effects of the liquid jets cause observable changes from the low fuel loading case in the flow field downstream of the liquid injection orifices. The flow fields also show variations between the ambient and high T&P cases at the high fuel loading condition, which might be due to the change in sprays as will be discussed next.

3.2 Qualitative spray comparison

The details of the atomization process for all six jets and the formation/breakup process of the liquid film are shown in Fig. 5, with instantaneous and averaged results compared for all four cases. The zoomed-in details of jet atomization and interaction with the filming wall are shown in Fig. 6. The detailed breakup processes including surface wave development, surface stripping, ligament formation, column breakup, film formation and breakup are captured by the simulation. Compared to the baseline ambient cases, increasing T&P does not seem to have first-order impact on the
overall penetration of the jets, confirming that the penetration is mostly controlled by the momentum flux ratio $q$, which is the same for the corresponding cases at those two conditions. Minor change of liquid trajectories can be observed between Fig. 6(a) and 6(c), 6(b) and 6(d). Also slightly larger amplitude surface waves are observed for the high T&P case compared to the ambient case. As shown in Table 3, with the constant $q$ and $U_{eq}$ scaling, the liquid flow rate $Q_l$ and velocity $U_l$ are increased for the higher T&P cases, causing more liquid volume accumulated inside the swirler and attached to the wall as shown in Fig. 5 and 6.

Compared to the low fuel loading (Liq. 1X) cases, the higher fuel flow rate in the Liq. 2X cases causes the liquid jets to penetrate more into the swirling flow, and large portion of the liquid reaches the inner swirler wall and forms liquid films. The structures of extracted wall film liquid for the four cases are shown in Fig. 7, with a zoom-in-view (black-rectangle region in Fig. 7) shown in Fig. 8. For the low fuel loading (Liq. 1X) cases, film liquid is absent at the ambient condition and shows up only in spotty locations at the high T&P condition. For the high fuel loading (Liq. 2X) cases, significantly larger volume of liquid is attached to the swirler wall to create film structures. The liquid film at the ambient condition not only has less volume due to the lower fuel flow rate, but also displays a different surface pattern compared to the high T&P case (compare Fig. 8(b) and 8(d)).

3.3. Quantitative comparisons inside the swirler: jet trajectories and liquid atomization degree

The mean trajectories of all six liquid jets are compared for the four cases in Fig. 9, in terms of the mean radial locations varying in the axial direction (Fig. 9(a)(b)) and in the azimuthal direction (Fig. 9(c)(d)). It is apparent that doubling the fuel flow rate leads to approximately double of the jet penetration. Despite the constant $q$ and $U_{eq}$ scaling followed, the high T&P cases consistently show slightly less bending and higher penetration than the ambient cases at both fuel flow rates. This is also consistent with the visual observations in Fig. 6. Such changes in penetration may be attributed to the changes in liquid breakup processes due to operating condition changes. Note that for high T&P cases, the Weber number $W_{eq}$, which primarily controls liquid breakup, is much higher than for the ambient cases as shown in Table 3. The changes of penetration may have an impact on downstream combustion by altering the downstream trajectories of liquid droplets from jet atomization or affecting the degree of filming that further changes the downstream droplet size and trajectories.

The averaged degree of atomization for the liquid inside the swirler (i.e. before the swirler exit) is computed and compared for the four cases in Fig. 10. The liquid volume for the jet and film inside the swirler is first computed and compared in Fig. 10(a) and (b) respectively. Under the constant $q$ and $U_{eq}$ scaling, the high T&P cases has a higher fuel flow rate, therefore leading to more liquid volume accumulation inside the swirler. The increases in liquid volume mostly manifest in the jet volume for the low fuel loading cases (Fig. 10(a)) and in the film volume for the high fuel loading cases (Fig. 10(b)). The film volume for the low fuel loading cases (Liq. 1X) is order of magnitude smaller than for the high fuel loading cases (Liq. 2X). Therefore we can neglect the liquid contributing to the filming part for the Liq. 1X cases in the discussions below. The liquid area shown in Fig. 10(c) and (d) follows the similar trend that the area is larger in the high T&P cases than in the ambient cases for both the jets and the film. The degree of atomization is quantified in Fig. 10(e) and 10(f) by computing the liquid volume-to-area ratio. The ratio is smaller in the high T&P cases compared to the corresponding ambient cases. And the trend holds for both the liquid jets and the film structures. The results indicate that at the high T&P condition, the liquid has a larger degree of atomization and produces structures of smaller overall effective size. One possible reason for the enhanced atomization at the high T&P condition is the larger $W_{eq}$ as shown in Table 3. The increased aerodynamic shear relative to the surface tension leads to more intensive liquid breakup for the liquid jet and stronger instability development and interface fragmentation for film liquid. It is also worth noting that for the liquid jets, increasing the fuel flow rate at the same T&P condition also leads to more intensive atomization (smaller effective size) (see Fig. 10(e)), probably because of the increased exposure of the liquid column to the shear in the swirling flow.

3.4. Quantitative comparisons at the swirler exit: film thickness and jet-to-film volume flux ratio

The averaged film thickness for the four cases are compared in Fig. 11(a). For the low fuel loading (Liq. 1X) cases, the averaged film thickness is smaller at the high T&P condition. However, the total film volume is very small and contribution of films to the downstream spray is negligible. For the high fuel loading (Liq. 2X) cases, the trend reverses and thicker film is formed at the high T&P condition. A plausible explanation is that the thicker film is caused by the larger volume of liquid on the filming wall at the high T&P condition due to a higher fuel flow rate under the constant $q$ and $U_{eq}$ scaling. Since the size of the droplets resulting from film breakup is largely proportional to the film thickness at the edge, larger droplets may be generated from film breakup at the high T&P condition. This is opposite to the trend for the size of droplets due to jet atomization as discussed above.

To further determine the dominant role of different atomization mechanisms at different fuel loading conditions, we separately compute the fuel volume fluxes at
the swirler exit from the jet and the film, and compare their ratio in Fig. 11(b). The ratio is above 1 for the low fuel loading cases and below 1 for the high fuel loading cases, indicating a dominant role of liquid-jet-in-swirling-flow atomization for the former and film-edge breakup for the latter. Based on above discussions, the effective droplet size downstream of the swirler may be smaller at the high T&P condition compared to the ambient condition for the low fuel loading cases (jet-in-swirling flow atomization dominant) and the trend may reverse for the high fuel loading cases (film-edge breakup dominant).

3.4. Comparisons of spray evaporation at high T&P

The change of dominant atomization mechanisms with the fuel loading condition may have a direct impact on the dominant evaporation mechanisms as well. In Fig. 12, the fuel mass fraction distribution inside the swirler at the x-z plane cutting through the centerline of the swirler is compared between the two high T&P cases at different fuel loading. The evaporation process mostly occurs near the liquid surfaces. The fuel vapor is predominantly distributed in the center region of the swirler. Because of the formation of film in the high fuel loading case, intensive liquid evaporation occurs near the hot swirler wall, which causes a significant impact on the downstream fuel vapor distribution.

5. Summary and Conclusions

In summary, a previously validated high-fidelity spray simulation approach has been extended to study the impact of operating conditions on the spray physics in a complex high-shear nozzle/swirler injector. The validated ambient condition case was used as the baseline for comparison with other cases with higher temperature, pressure, and liquid fuel flow rate.

It has been found that under the constant momentum flux ratio and constant gaseous velocity scaling, changes in T&P do not have a significant impact on the unsteady gas flow structures, especially in regions upstream of liquid jet orifice. The operating condition change has a minor impact on the overall penetration of the jets, with slightly less bending of the jets observed at the high T&P condition. A larger degree of overall atomization of the liquid inside the swirler (smaller effective size) was observed at the high T&P condition based on the computed volume-to-area ratio. Such changes in jet trajectories and atomization degree at different T&P are most likely due to the changes in the aerodynamic Weber number. On the other hand, for the cases where the filming process dominates, thicker film is formed at the high T&P condition, due to the increased fuel flow rate and larger volume of liquid on the filming wall. Therefore, the size of the droplets resulting from film-edge breakup may be larger, as opposed to the smaller effective size of droplets generated by jet-in-swirling-flow atomization.

The dominant atomization mechanism varies with the fuel loading conditions. At low fuel flow rate, the liquid breakup is dominated by the jet-in-swirling-flow atomization mechanism. Increase in the fuel flow rate causes the liquid jets to penetrate more into the swirling flow, and pushes a larger portion of the liquid to the swirler wall to form liquid films. At high fuel flow rate, the dominant atomization mechanism transitions to the film-edge breakup mechanism. The evaporation of liquid is also impacted by the change of the atomization mechanism.

Acknowledgements

The author would like to acknowledge the funding support from United Technologies Corp. Valuable technical discussions with Marios Soteriou and Scott Liljenberg at United Technologies Research Center are gratefully acknowledged.

References

9. Madabhushi, R.K., A model for numerical simulation of breakup of a liquid jet in


### Table 1. Liquid inlet conditions and corresponding properties of Jet-A.

<table>
<thead>
<tr>
<th>Cases</th>
<th>P (Pa)</th>
<th>T_l (K)</th>
<th>(\rho_l) (kg/m(^3))</th>
<th>(\mu_l) (kg/m·s)</th>
<th>(\sigma) (N/m)</th>
<th>k_l (W/m·K)</th>
<th>c_{p,l} (J/kg·K)</th>
<th>L (J/kg)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 - 4</td>
<td>(10^5)</td>
<td>300</td>
<td>798.5</td>
<td>0.00143</td>
<td>0.0228</td>
<td>0.124</td>
<td>1948.2</td>
<td>3.41e5</td>
</tr>
</tbody>
</table>

### Table 2. Gas inlet conditions and corresponding properties of air.

<table>
<thead>
<tr>
<th>Cases</th>
<th>P (Pa)</th>
<th>T_g (K)</th>
<th>(\rho_g) (kg/m(^3))</th>
<th>(\mu_g) (kg/m·s)</th>
<th>k_g (W/m·K)</th>
<th>c_{p,g} (J/kg·K)</th>
<th>D_g (m(^2)/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 &amp; 2</td>
<td>(10^5)</td>
<td>300</td>
<td>1.17</td>
<td>1.8e-5</td>
<td>0.0252</td>
<td>1017.8</td>
<td>2.1e-5</td>
</tr>
<tr>
<td>3 &amp; 4</td>
<td>(5\times10^6)</td>
<td>811</td>
<td>21.5</td>
<td>3.6e-5</td>
<td>0.0565</td>
<td>1100.9</td>
<td>2.4e-6</td>
</tr>
</tbody>
</table>

### Table 3. Flow conditions and non-dimensional parameters.

<table>
<thead>
<tr>
<th>Cases</th>
<th>P (Pa)</th>
<th>T_g (K)</th>
<th>T_l (K)</th>
<th>Q_g (kg/s)</th>
<th>Q_{g,i} (kg/s)</th>
<th>U_{g,i} (m/s)</th>
<th>Q_l (kg/s)</th>
<th>U_l (m/s)</th>
<th>q</th>
<th>We_g</th>
<th>Re_g</th>
<th>(r_p)</th>
<th>(r_p)</th>
<th>(r_U)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>(10^5)</td>
<td>300</td>
<td>300</td>
<td>0.044</td>
<td>0.036</td>
<td>71.3</td>
<td>0.0125</td>
<td>8.11</td>
<td>8.8</td>
<td>167</td>
<td>2969</td>
<td>680</td>
<td>79</td>
<td>0.114</td>
</tr>
<tr>
<td>2</td>
<td>(10^5)</td>
<td>300</td>
<td>300</td>
<td>0.044</td>
<td>0.036</td>
<td>71.3</td>
<td>0.0250</td>
<td>16.22</td>
<td>35.2</td>
<td>167</td>
<td>2969</td>
<td>680</td>
<td>79</td>
<td>0.228</td>
</tr>
<tr>
<td>3</td>
<td>(5\times10^6)</td>
<td>811</td>
<td>300</td>
<td>0.805</td>
<td>0.662</td>
<td>71.3</td>
<td>0.0535</td>
<td>34.7</td>
<td>8.8</td>
<td>3060</td>
<td>26964</td>
<td>37</td>
<td>39</td>
<td>0.487</td>
</tr>
<tr>
<td>4</td>
<td>(5\times10^6)</td>
<td>811</td>
<td>300</td>
<td>0.805</td>
<td>0.662</td>
<td>71.3</td>
<td>0.1070</td>
<td>69.4</td>
<td>35.2</td>
<td>3060</td>
<td>26964</td>
<td>37</td>
<td>39</td>
<td>0.974</td>
</tr>
</tbody>
</table>
Figure 1. Geometry of a high-shear nozzle/swirler combination: (a) Swirler geometry details, (b) schematic of the swirler/nozzle combination and flow configurations, (c) Nozzle geometry details.

Figure 2. Sketch of simulation domain and boundary conditions.
Figure 3. Simulated axial velocity field inside the swirler using x-z plane cutting through the centerline of the swirler. (a)-(d) Instantaneous field, (e)-(h) Mean field.
Figure 4. Simulated azimuthal velocity field inside the swirler using x-z plane cutting through the centerline of the swirler. (a)-(d) Instantaneous field, (e)-(h) Mean field.
Figure 5. Simulated liquid spray structures inside the swirler rendered using the ray-tracing technique. (a)-(d) Instantaneous field, (e)-(h) Mean field.
Figure 6. Zoomed-in view of the simulated liquid spray structures inside the swirler rendered using the ray-tracing technique.
Figure 7. Simulated film structures at the swirler wall rendered using the ray-tracing technique. (a)-(d) Instantaneous field, (e)-(h) Mean field. The films are extracted by separating out the liquid surfaces within a swirler wall distance of $d_{\text{threshold}}=0.5\text{mm}$. 
Figure 8. Zoomed-in view of the simulated film structures at the swirler wall rendered using the ray-tracing technique.
Figure 9. The time-averaged trajectories of liquid jets represented by the mean radial locations varying in the axial direction in (a) and (b) and in the azimuthal direction in (c) and (d). The trajectories are computed by averaging the radial locations of liquid surfaces within each axial bin. The liquid surface from all six jets are used for averaging.
Figure 10. The time-averaged volume [(a) and (b)], area [(c) and (d)] and volume-to-area ratio [(e) and (f)] for the liquid jets and film inside the swirler (i.e. before the swirler exit).
Figure 11. The time-averaged film thickness (a) and jet-to-film volume flux ratio (b) at the swirler exit. The averaged film thickness is computed based on the local solid level set distance function at the film surface. The volume flux is computed based on the local liquid volume and velocity at the swirler exit.
Figure 12. The fuel mass fraction distribution inside the swirler at the x-z plane cutting through the centerline of the swirler for the high T&P cases at different fuel loading conditions.