

A Comparison of Experimental and Modeled Velocity in Gasoline Direct-Injection Sprays with Plume Interaction and Collapse

Panos Sphicas
 Imperial College London

Lyle M Pickett, Scott Skeen, and Jonathan Frank
 Sandia National Laboratories

Tommaso Lucchini, David Sinoir, and Gianluca D'Errico
 Politecnico di Milano

Kaushik Saha and Sibendu Som
 Argonne National Laboratory

ABSTRACT

Modeling plume interaction and collapse for direct-injection gasoline sprays is important because of its impact on fuel-air mixing and engine performance. Nevertheless, the aerodynamic interaction between plumes and the complicated two-phase coupling of the evaporating spray has shown to be notoriously difficult to predict. With the availability of high-speed (100 kHz) Particle Image Velocimetry (PIV) experimental data, we compare velocity field predictions between plumes to observe the full temporal evolution leading up to plume merging and complete spray collapse. The target "Spray G" operating conditions of the Engine Combustion Network (ECN) is the focus of the work, including parametric variations in ambient gas temperature. We apply both LES and RANS spray models in different CFD platforms, outlining features of the spray that are most critical to model in order to predict the correct aerodynamics and fuel-air mixing.

CITATION: Sphicas, P., Pickett, L., Skeen, S., Frank, J. et al., "A Comparison of Experimental and Modeled Velocity in Gasoline Direct-Injection Sprays with Plume Interaction and Collapse," *SAE Int. J. Fuels Lubr.* 10(1):2017, doi:10.4271/2017-01-0837.

INTRODUCTION

Control of fuel-air mixing is an essential component of gasoline engine design. A trend towards use of Gasoline Direct-Injection (GDI) has been established, with improvements in cold-start performance, knock tendency, and overall efficiency obtained. Further improvements and shortened design cycles may be enabled through the use of computational fluid dynamics (CFD) simulations that are able to model, and optimize, these fuel-air mixing processes. To date there have been significant advances in the potential to more accurately model the engine flow field as well as the spray pattern, but a careful analysis of various CFD approaches is needed to make further progress.

One of the significant challenges towards understanding GDI fuel injector systems is that the injector holes are aligned close to one another, promoting significant interaction between neighboring plumes as well as with all of the plumes as a collective. The plumes may merge together, creating "spray collapse", which ultimately inhibits mixing. In addition to injector geometry effects such as hole-direction design and spacing, factors affecting spray collapse include charge-gas operating conditions as well as flash-boiling

conditions with heated fuel [1, 2, 3, 4, 5, 6, 7, 8, 9]. An added complexity affecting plume interaction is that GDI injector holes typically have a complicated geometry with short-length, stepped holes. The structure of the emerging plumes is difficult to characterize experimentally, but there are indications of irregular, asymmetric plumes with larger spreading angles [10,12,13,14]. There is also evidence that the nozzle tip outer wall geometry can affect plume interaction [15].

Producing a CFD tool that adequately simulates the dispersion of individual plumes and the subsequent plume interaction is a formidable task. Even with simplified geometries and isolated plumes, the atomization of liquid, mixing and evaporation is by no means a "solved" problem despite in-depth research on the topic. Progress is hindered by lack of quantitative experimental data inside the nozzle and in dense regions of the spray, and by the numerical complexity of resolving the interaction of both liquid and gas phases in the same computation. While efforts to add more physics of the internal nozzle flow and near-field breakup process continue, current computations designed to predict fuel-air mixing within the engine cylinder often rely upon simplified spray model "constants"

while necessarily neglecting unknowns about the internal flow and near-nozzle breakup. Liquid particles/droplets are typically treated in a Lagrangian framework, requiring models for momentum exchange between the liquid and the gas-phase Eulerian field. The net result of the uncertainties and model simplifications is that “tuning” of spray model constants based upon experiments is typically required, but knowing in advance how adjustment of these parameters actually affects the spray characteristics for a particular CFD implementation remains a challenge [4].

In principle, dedicating more computational resources to describe the spray mixing process should result in more accurate, predictive simulations. For example, Large-Eddy Simulation (LES) with high grid resolution through adaptive mesh refinement should resolve flow structures and plume interactions that may be missed if using coarse-mesh Reynolds-averaged Navier–Stokes (RANS) simulations. However, the need for spray model tuning described above means that a large number of preliminary simulations may be required. When also considering the fact that multiple LES simulations are needed to obtain an ensemble average, the computational cost or time may be prohibitive even when using less-expensive, engineering-level LES. Ideally, tuning of spray models or setup could be performed using less expensive (RANS or other) simulations, provided that there was an understanding about how to translate RANS spray model settings to more detailed, expensive simulations. In addition, adequate accuracy may be achieved for certain applications using RANS models alone. In this sense, we see a strong need for both RANS and LES approaches, particularly when applied to sprays where there is so much admitted uncertainty [4]. Essentially, this philosophy is consistent with the engineering practice to evaluate tradeoffs while applying the most efficient tools for the task at hand.

Progressing towards the goal of more predictive/useful RANS and LES spray simulations for multi-hole injectors requires a concerted effort with respect to CFD modeling practices, as well as careful validation against quantitative data. Recognizing the need for collaborative, multi-institution research on this topic, the Engine Combustion Network (ECN) [16] has chosen a specific multi-hole gasoline fuel injector as an experimental and modeling target known as Spray G. Many macroscopic properties as well as internal geometry and rate of injection have already been characterized experimentally for Spray G, including evidence of plume interaction and collapse [3,6,7,10]. In addition, a number of CFD simulations have been made for Spray G, including detailed modeling of the internal nozzle flow [12,13,14].

Recently, Sphicas et al. [3] temporally resolved the gas-velocity field surrounding the liquid plumes at standard Spray G operating conditions as well as variants in temperature and pressure. High-speed (100 kHz) PIV measurements were applied to provide full temporal evolution of plume-interaction and potential collapse, resolved for individual injection events. Charge-gas entrainment by the plumes, was found to create a central recirculation region, which promotes plume interaction and eventual collapse. Increasing ambient temperature or density, with enhanced vaporization and momentum exchange, was found to accelerate plume interaction [3]. This dataset,

combined with other ECN experiments, is unique with respect to past characterization and quantification of multi-hole gasoline sprays.

With the availability of this velocity dataset, there is a new opportunity to evaluate CFD modeling approaches using different platforms and different levels of fidelity (i.e. RANS vs. LES), with a special emphasis on understanding the mechanics of plume interaction and spray collapse. The approach taken in this study was to compare RANS simulations from Politecnico di Milano using Open-FOAM code [17] and LES simulations from Argonne National Laboratory using CONVERGE code [18–19] against experimental Spray G data. In each case, the simulations did not necessarily represent a highly tuned result—the goal of the effort is not simply to create the “best match” to experiment. Rather, the goal is to understand how simulations with different input conditions and different modeling approaches respond with respect to key features affecting plume interaction. Integrated with the experimental velocity data available over limited spatial positions, there is also an opportunity to use the simulation results to assist in developing a conceptual understanding of the process.

ECN STANDARDS FOR EXPERIMENTS AND SIMULATIONS

The ECN has identified experimental standards for gasoline Spray G, such that all experiments and models are consistent, thereby presenting the opportunity to combine datasets and to leverage the research of all involved. Table 1 presents the fundamental standards set for both the experimental and modeling work. The condition is intended to simulate injection during the compression stroke but is non-reacting (no spark ignition). Because the condition is non-reacting, nitrogen, or other inert gas combinations are permitted. For simplicity, the ambient environment is typically modeled as pure nitrogen. The injector features an 8-hole nozzle. A series of injectors and injector drivers were donated for this effort by Delphi corporation.

Table 1. Spray G standard operating conditions

Fuel	Iso octane
Injection Pressure	20 MPa
Fuel Temperature	90° C (363 K)
Ambient Temperature	300° C (573 K)
Ambient Density	3.5 kg/m ³
Ambient Pressure	5.97 bar (N ₂)
Ambient Oxygen	0%, inert gas
Injected Quantity	10 mg
Injection Duration	780 μs
Number of Nozzle Holes	8
Nominal Hole Diameter	165 μm
Full Outer Angle	80°

While the primary focus of this paper is to understand the standard Spray G condition given in Table 1, for this paper we also emphasize a high ambient temperature condition (800 K) while leaving all other conditions the same. Our past work [3] showed a strong, but somewhat unexplained, effect of ambient temperature on plume interaction, thereby motivating further consideration and model evaluation.



Figure 1 presents the measured rate of injection using a tube-style rate meter at the designated injection pressure and temperature, back pressure, and injection duration to produce 10 mg total injected quantity. The measurements were performed by Scott Parrish at GM R&D, and repeated at CMT-Motores Termicos [20]. The measurements specifically correspond to injector serial number AV67-028, which is the fuel injector used at Sandia for the velocity experiments. (Note that multiple injectors with the same specification are in circulation for this ECN research). As shown in the figure and Table 1, an electronic duration of 680 μs produces a 780 μs hydraulic injection. This is the injection profile used for all experimental and modeling work in this paper.

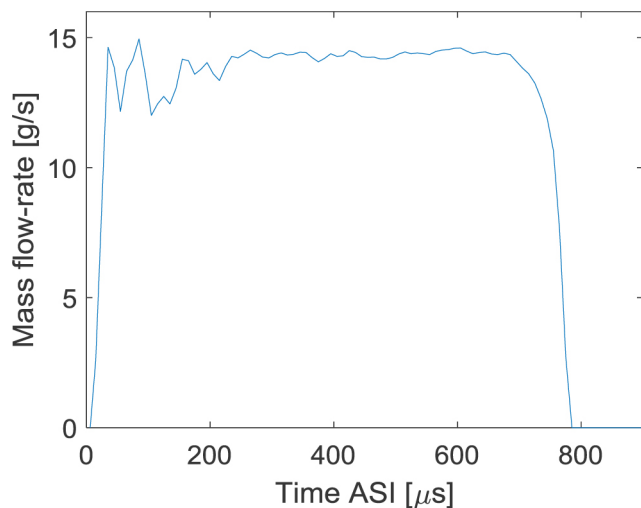


Figure 1. Measured rate of injection with 680 μs electronic pulse width for injector AV67-028.

The specified internal geometry of the injector nozzle is shown in Figure 2 and the measured three-dimensional internal geometry is also available [16]. The intersection of the hole axis with the injector centerline is known as “virtual point of plume origin”. The center of curvature of the injector tip, noted with a blue cross on the figure, is different to the virtual point of the plume origin. The counter-bores are a typical geometric feature of modern DI gasoline injectors. Note that the ECN convention for Spray G is to define the axial dimension with respect to the tip of the fuel injector [21], which means that the hole and counterbore are at slightly negative axial positions.

One may expect that the plume spreading angle or cone angle, as well as the “plume direction”, shown schematically in Figure 2, are critical parameters affecting plume interaction. Understanding the plume direction requires knowledge of the flow dynamics as fluid passes around the opening needle (full lift approximately 50 μm) and through the small hole inlet, counter-bore, and so forth. Without knowledge of these details, a modeling assumption may be to use the “drill angle”, which for Spray G is 37° with respect to the injector axis. An alternative routine would be to use measured patternation data for the fuel injector. For example, patternation data for Spray G AV67-028 was presented in Ref. [7]. Measurements were made 50 mm from the injector using cold fuel at 1 atm, which are quite different conditions than the Spray G target conditions. Based upon

the average plume pattern at 50 mm, the measured plume direction was approximately 33°. These patternation data suggest a deflection towards the centerline, as shown in Figure 2, but it must be recognized that aerodynamic interactions between plumes occur even at 1 atm conditions and a plume pattern at 50 mm from the injector may be the result of these interactions, rather than a plume direction at the nozzle exit. Another consideration is that plume direction may change with time during an injection.

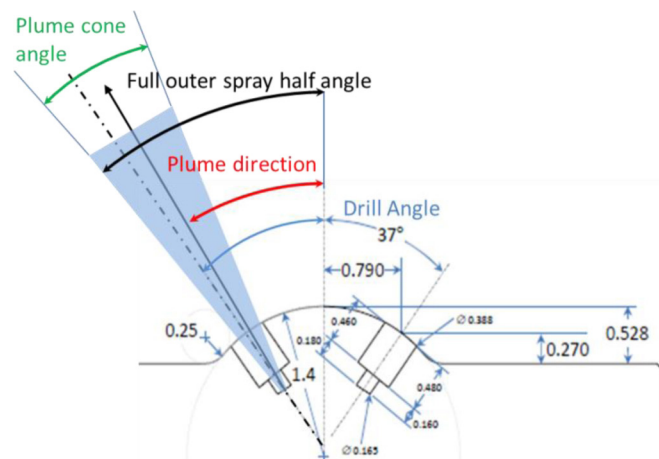


Figure 2. ECN Spray G geometry and definition of plume direction and plume cone angle. The full outer angle listed in Table 1 includes the entire spray envelope—drill angle plus plume cone angle. Dimensions given in mm.

The complexities in understanding plume direction and cone angle, as well as other three-dimensional structures within the emerging plumes, have motivated internal flow modeling and coupled Eulerian-Eulerian calculations downstream of the nozzle tip for this same injector [12,13,14]. Although unverified by experiment inside the nozzle, the modeling results universally show a plume direction shift from the drill angle towards the centerline, consistent with patternation data. In addition, x-ray radiography measurements at only 2 mm from the injector tip [10] show an initial mean plume direction of 36° which reduces to about 34° during the main period of injection. Consequently, there is consistency in the idea that the plume direction should be modified to be less than the drill angle.

MODEL DESCRIPTION

Information about the LES modeling approach applied by Argonne National Lab (ANL) and the RANS modeling approach applied by Politecnico di Milano (PoliMi) is found in this section, referring to Tables 2 and 3. In each case, certain assumptions about the plume direction and plume cone angle are made, but in addition, parametric variation is intentionally made to discover sensitivities in these parameters. In addition, ANL has applied internal/external flow modeling in coupled Eulerian-Eulerian simulations as well as “one-way” simulations where the internal flow solution is imparted as a Lagrangian input directly at the face of the injector tip [14]. The obvious goal is that the internal flow solution will impart more realistic three-dimensional inputs, rather than prescribing assumed plume direction and cone angle. Further detail about its implementation will be discussed below.



Table 2. Computational models, sub models, and plume orientations

	ANL	Polimi
CFD Code	Converge	OpenFOAM + Lib-ICE
Turbulence Model	LES – dynamic structure	Standard k-ε
Injection Model	Lagrangian/Blob	Lagrangian/Huh
Primary Break-up Model	Kelvin-Helmholtz (KH)	Huh-Gosman
Secondary Break-up	Rayleigh-Taylor (RT)	Kelvin Helmholtz (KH) + Rayleigh Taylor (RT)
Vaporization	Frössling	Spalding Number-based (mass-based)
Heat Transfer	O'Rourke	Ranz-Marshall
Collision	NTC	None
Droplet Drag	Dynamic	Dynamic without spherical correction
Initial gas turbulence	TKE=1.0; EPS=100.0	$u' = 1$ m/s; $Li = 2$ mm
Plume direction	35°	34°, 37°
Plume cone angle	25°- 40°, “one-way”	Calculated from Huh-Gosman as 9°

Politecnico di Milano RANS Implementation

PoliMi performed simulations using the open-source code OpenFOAM together with specific in-house libraries for spray and combustion modeling (Lib-ICE). Simulations were carried out with the RANS approach and the standard k-ε model was used for turbulence. The spray is modeled using the Eulerian-Lagrangian approach and supports multi-component fuels. Specific sub-models are available to describe fuel atomization, breakup, heat transfer, evaporation, collision and wall impingement. In this work, atomization of the liquid fuel jet is predicted by the Huh-Gosman model [22]. Primary parcels (blobs) are injected into the computational mesh with the same nozzle hole diameter and their velocity is a function of the injected mass flow rate profile. This means that the cross section area of the hole is utilized at 100% and effects of internal nozzle flow are not considered, such as cavitation and jet velocity distribution. This can represent a strong assumption in case of complex nozzle geometries and the possibility to couple nozzle flow and Lagrangian spray simulations will be matter of future works. Both Kelvin-Helmholtz and turbulence induced breakup on the jet surface are taken into account by the model, describing the diameter reduction of the injected parcels as follows:

$$\frac{dD}{dt} = -C_5 \frac{L_a}{\tau_a}$$

where C_5 is the main model constant, L_a and τ_a are the characteristic atomization length and time scales. The L_a is proportional to the turbulent length scale of the liquid jet, while atomization time-scale τ_a is a function of both turbulent and aerodynamic time scales. Their values are related to the geometry of the injection system, turbulence decay within the jet and Kelvin-Helmholtz instabilities. Further details about the Huh-Gosman model are available in [22]. At the nozzle exit, droplets are uniformly injected inside a cone whose angle α is predicted according to the expression:

$$\tan\left(\frac{\alpha}{2}\right) = \frac{L_a}{U}$$

where U is the droplet velocity at the nozzle exit, directly estimated from nozzle diameter, fuel density and injection rate profile. Initial turbulent length and time scales are computed from a simplified zero-dimensional nozzle flow model. Atomization process ceases as soon as primary parcel diameter, Ohnesorge or Weber numbers fall below a specified value. Secondary droplets are created from primary blobs when the amount of stripped mass is higher than a specific threshold value. Their diameter is computed from a PDF distribution whose properties and diameter range depends on the expected turbulence spectrum at the time the new droplet is created [22]. The secondary stripped droplets experience both Kelvin-Helmholtz (*wave*) and Rayleigh-Taylor (*catastrophic*) instabilities. Both breakup mechanisms operate in competition. Simulations are performed in a cylindrical domain with a Cartesian mesh structure and an initial grid size of 4 mm. To reduce the grid dependency, without significantly increasing the computational time, Adaptive Local Mesh Refinement was used. The cells, where fuel-air mixing takes place, are refined up to 1 mm size, which is common for that used in CFD simulations of direct-injection engines. Limited, second-order schemes were used for the discretization of convection terms in momentum, species and energy equations [23]. Initial values of k and ε were computed by assuming for the quiescent gas a turbulence of 1 m/s and a corresponding integral length scale of 2 mm. Such values are commonly used in spray simulations at constant volume conditions. [Appendix A](#) contains more specific constants used for these simulations.

Argonne National Laboratory LES Implementation

Argonne used the commercial software CONVERGE [24] to perform simulations using a dynamic structure LES model [18,25], as shown in [Table 2](#). Simulations employed the “blob” injection method of Reitz and Diwakar [26] in which “Lagrangian parcels” of liquid with a characteristic size equal to the effective nozzle diameter were injected into the computational domain. Kelvin-Helmholtz (KH) and Rayleigh-Taylor (RT) models were used to simulate primary and secondary breakup respectively. A dynamic drag model was used for estimating the momentum exchange between the fluid-phase (Eulerian field) and the liquid parcel (Lagrangian particle). The No Time Counter (NTC) collision method of Schmidt and Rutland [27] was used in the present simulations. NTC uses a stochastic sub-sampling of the parcels within each cell. NTC computational cost varies linearly with the parcel count, unlike O'Rourke's method where the cost varies by square of the parcel count. Contrary to the O'Rourke method the NTC approach captures the effect of collision through a repeated sampling method leading to a more reasonable solution. [Appendix A](#) contains more specific constants used in the Argonne simulations as well.

Fuel spray inputs based on a traditional Rate-of-Injection approach (ROI) as well as the “one-way” approach [14] were used. In the ROI approach, experimentally measured rate of injection is used to initiate the spray calculations. The plume cone angle is a vital input for this approach as it prescribes the conical domain within which the parcel injection will occur. Parcels are uniformly distributed within this cone-angle. The orifice diameter (165 μm) is used as the effective diameter for parcel injection. For the one-way case *a priori* internal nozzle flow simulation has been performed [14] and a map file consisting of essential flow variables at the combustion chamber was created to provide input for the Lagrangian simulations. The map file

thus provides the information for parcel mass and injection velocity. Information about the effective diameter of liquid parcels to inject is also embedded.

The mesh is generated automatically during run-time based on base mesh size (1 mm), fixed embedding (a priori mesh refinement in regions of interest and probable high gradients of some variables) and AMR (mesh refinement during the run-time depending on the gradients of key variables), with a minimum grid size of 0.125 mm. For this study, fixed embedding was used in the near-nozzle regions and AMR was used for capturing velocity gradients. A cylindrical computational domain with diameter of 108 mm and length of 108 mm was used. The tip of the injector was set at the center of one of the planar faces of the cylindrical domain. Additional tests by moving the injector tip 1 mm away from the planar surface did not yield noticeable difference in terms of liquid penetration. Since LES was performed at Argonne, multi-realization averaging was used to provide ensemble-averaged results, following the recommendations of Ameen et al. [28]. Random number seed was used to perturb the fuel spray, which provided reasonable shot-to-shot variability in diesel sprays [29].

Table 3. Computational grid characteristics

	ANL, LES	Polimi, RANS
Domain Dimensions	108 x 108 mm	169 x 248 mm
Base Cell Size	1 mm	4 mm
Min Cell Size	0.125 mm	1 mm
Nozzle tip included	No, Yes for one-way	No
Radial position of droplet parcel	0.79 mm	0.5 mm
Adaptive or Static Refinement	Both	Adaptive
Cell Type	Cartesian cut-cell	Cartesian
Total/Maximum Cell Count	14 million @ 1ms; 20 million @ 2ms	115,000
Number of spray parcels	200k per plume	20k per plume
CPU cost	5 realizations 36 processors 10 hours for 1 ms	1 realization 3 processors 40 minutes for 2 ms

A comparison of the computational cost for the LES and RANS simulations (Table 3) shows that the LES approach is three orders of magnitude more expensive than the RANS simulation. Consequently, it is difficult to perform many parametric variations. The plume direction was fixed (35°) for the LES simulations, or directly calculated from internal nozzle flow (one-way) calculations, while the RANS simulations varied the plume direction from the drill angle (37°) to 34°. For plume direction, the differences between LES and RANS simulations were therefore more subtle. However, Argonne varied the plume cone angle substantially, from 25° to 40°, in part based upon experience with RANS simulations comparing Eulerian-Eulerian and one-way Lagrangian simulations against the traditional ROI approach [14]. On the other hand, PoliMi chose not to intentionally vary the plume cone angle, but instead relied upon spray constants from past experience, finding a plume cone angle of 9°. While this seems a substantial difference, as mentioned in the

Introduction, the specific implementation of the spray model may have dependencies upon a particular computational platform. We will compare these differences by examining distribution of fuel-air across a plume for different input conditions.

EXPERIMENTAL SETUP

The experimental setup has been previously described in detail at [3]. A custom-built laser with two cavities, each capable of emitting a pulse-burst of 500 pairs of pulses at 532 nm at a frequency of 100 kHz, was used to illuminate the 1 μ m Zirconium particles at 532 nm. A Photron SA-X2 high speed CMOS camera equipped with a Nikon 200mm f5.6 lens, a 0.3 neutral density filter and a 532 nm (10 nm FWHM) band pass filter was used to collect the particle. The frame rate was externally set to 200 kHz with a 3.98 μ s exposure. The image resolution was 256×152 pixels at 36.8 μ m/pixel. Processing the 2.5 μ s spaced data was possible, but not needed in practice for the range of velocities experienced. Instead, the images from the two cavities were processed separately to produce the velocity vector field. The velocity vector fields from the two cavities were then ensemble averaged together. Utilizing the time-resolved data, a sliding average sum of correlations algorithm that includes neighboring ± 3 frames was used for PIV processing, providing temporal observation of developing turbulent structures with filtering to remove erroneous correlations.

The injector was oriented as shown in Figure 3. A dimple on the face of the injector and the electrical connector are aligned to plume 1 and the rest of the plumes are numbered sequentially, clockwise from a viewing angle looking at the face of the injector. As per ECN convention, the camera is located nearest to plume 1. More detail on the Spray G plume orientation, numbering, and geometry standards can be found on the ECN website [16]. The laser sheet was directed between the plume centers to avoid strong scatter from liquid droplets, which can interfere with the seeded gas-phase PIV measurements. The viewing plane was located in a difficult diagnostic location with very high velocity gradients and droplet interference potential, but in a location close to the tip important to understand plume interaction. This “between-plume” plane, which is at a rotation of 22.5° about the z-axis will be extensively compared to results from LES and RANS simulations for this study. But, in addition, information focused on plume centers will be extracted from simulations to understand and interpret data between plumes.

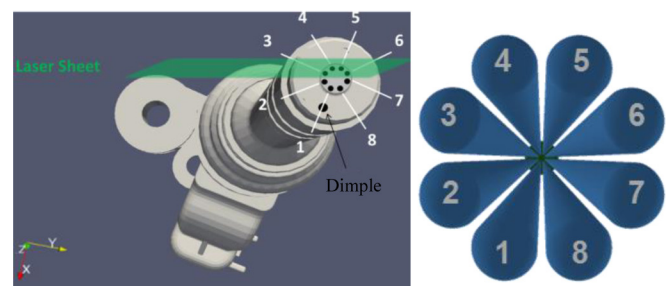


Figure 3. Illustration of the laser sheet alignment in regard to the gasoline Spray G. Note the reference x-y-z convention with z as the axial coordinate.

Figure 4 is a scaled illustration of the viewing areas observed in this study. The solid green rectangle represents the central field of view, which is viewing the central region in between the plumes and the dashed green rectangle represents the offset plane of view, which is viewing the area between neighboring plumes. The solid blue and red lines represent the plume centerlines as observed for the typical Spray G conditions. The dashed blue and red lines represent the plume edges under the same conditions. The red lines correspond to the outer plumes, while the blue lines correspond to projections of the inner plumes. The first number corresponds to the plumes closer to the camera and the second number to the plume away from the camera. Plumes 2, 1, 8 and 7 are located towards the camera in Figure 3. While the planar imaging region is located in the center of the inter-plume area, the position of neighboring out-of-plane plumes is of interest because of potential interference to the planar measurement. In addition, plumes may merge together, with faster moving droplets present in the measurement plane. Knowing the orientation of the plumes is therefore useful for understanding that the interference comes from out-of-plane sources that could affect the measurement.

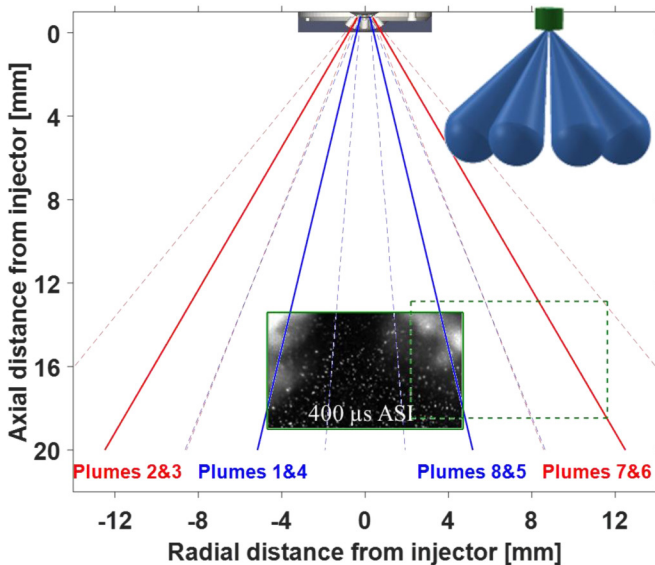


Figure 4. Scaled illustration of the PIV viewing area between the plumes of the gasoline spray injector, in “centerline” and “offset view” positions. Plume centers and edges on projection are indicated by solid and dashed lines respectively.

RESULTS

Experimental Velocity

A demonstration of the instantaneous images acquired with the aforementioned experimental setup is shown in Figure 5. The instantaneous velocity vectors obtained using the experimental setup described in the last section are overlaid on the raw PIV images. Note that overlapping interrogation windows of 16×16 pixels were used for processing, but fewer vectors are displayed for clarity. The vector length and color of the vectors correspond to the velocity magnitude. Figure 5 was acquired $510 \mu\text{s}$ after start of injection (ASI), but high-speed movies of the entire injection event are available online [30]. At this particular timing, a well-established, almost uniform, upward recirculation motion is shown. On the upper left and right corners of the particle image higher illumination intensity zones

appear. These zones originate from the indirect illumination of the out-of-laser-plane plumes 1&4 and 8&5 as illustrated in Figure 4. Scattered light out of the PIV measurement plane, which occurs as scatter from other plumes or vessel walls, illuminates the liquid plumes in front of the viewing plane. This out-of-plane plume illumination is recorded by the camera and could interfere with processing of particles on the plane of the laser. As detailed in [3], a Proper Orthogonal Decomposition use of a sliding sum of cross-correlations to interpret multiple PIV realizations in time, effectively isolates and processes PIV for the actual plane of interest. An indication of success is that the velocity vectors in the midst of the stronger interference have an upward velocity while the out-of-plane plumes have a downward velocity.

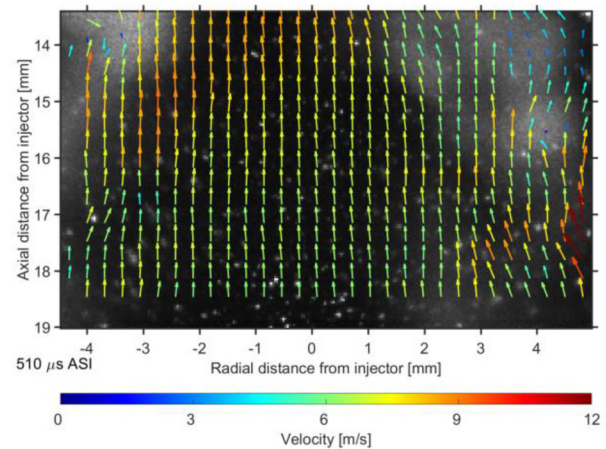


Figure 5. Instantaneous snapshot (at $500 \mu\text{s}$ ASI) of particle image and vectors generated for Spray G (573K) conditions given in Table 1.

A useful way to summarize the changes in velocity field throughout time is to consider the velocity at a relevant measurement position. Experimental results for the ensemble-averaged axial air velocity on the centerline at the 15mm plane are presented in Figure 6 and Figure 7 as a function of time. The ensemble-average data is labeled directly on the plot, and includes uncertainty bars based upon the standard deviation and number of repeats [3].

To begin, we focus on the blue line representing the standard 573 K condition. As typical for other operating conditions, there are distinct phases of flow at the centerline indicative of plume interaction and responses to injector timing. The initial phase begins as injection pulls ambient air upward towards the injector (negative axial velocity), even before the arrival of the spray at that axial position. For the 573K case, plumes arrive at the 15 mm viewing angle at about $200 \mu\text{s}$ ASI, but the recirculation motion initiates almost immediately after the beginning of injection. The recirculation increases in intensity up to about $350 \mu\text{s}$, at which time, the initially observed higher velocities on the centerline, reduces in intensity and the velocity radial profile turns almost homogeneous as shown in Figure 5. From 700 to $1100 \mu\text{s}$, the recirculation decelerates. As will be discussed later, the change in recirculation velocity begins well before the end of injection ($780 \mu\text{s}$) and even sooner than the 15-mm position will “feel” the end of injection. During this period of recirculation deceleration, the high-speed movie shows that initially the vectors on the upper and outer most edges of viewing window