

Multi-hole Injectors for DISI Engines: Nozzle Hole Configuration Influence on Spray Formation

Petter Dahlander and Ronny Lindgren

Chalmers University of Technology

Copyright © 2008 SAE International

ABSTRACT

High-pressure multi-hole injectors are one candidate injector type for closed-spaced direct injection (DI) gasoline engines. In such a system, the spark plug must be located close to the spray and, during stratified operation, the spray is ignited very soon after the fuel droplets have been vaporized. Thus there are very high demands on the sprays used in such a system. An additional challenge is the positioning of the spark plug relative to the spray; both consistent ignitability and the absence of liquid fuel droplets must be achieved. Many injector parameters influence spray formation; for example, hole diameter, length to hole diameter ratio, nozzle hole configuration etc.

This paper investigates the spray formation and spray induced air movement associated with rotational symmetrical and asymmetrical nozzle hole configurations. Four different nozzles with different hole configurations and umbrella angle were investigated both experimentally and numerically in a heated/pressurized spray chamber. Their influence on spray formation, spray induced air motion, cross-flow velocity, fuel/air ratio, turbulence and cycle-to-cycle variations were studied. It was found that rotational symmetrical configurations produce non-coherent isolated clouds of fuel. If an asymmetrical configuration is used instead (holes positioned along a horseshoe-shaped arc) then, by choosing the injector configuration carefully, it is easier to obtain a coherent fuel cloud; this also facilitates better control over the conditions at the spark plug, for example the fuel/air ratio, cross-flow velocity and turbulence. Furthermore, asymmetrical nozzles benefit from smaller fuel gradients and enhanced mixing between spray plumes as a result of shorter spray plume distance and improved spray-induced air motion. All the nozzles tested produced partially premixed vapor clouds, with cycle-to-cycle variations. These variations may be an important issue for ignition stability in a closed-spaced combustion system.

INTRODUCTION

A spray-guided combustion system for gasoline Direct Injection engines offers great potential in reducing CO₂ emissions [1,2]. For the stratified operation of a Direct Injection Stratified Charge (DISC) engine, the fuel is injected during the compression stroke instead of during the intake stroke. Due to the short time between the end of the injection (EOI) and ignition in stratified operation, the spray characteristics and mixture formation are of great importance. The fuel spray needs to be atomized, vaporized, and mixed with air so that a mixture within the flammability limits is present near the spark plug at the time of ignition. All this must happen in a matter of milliseconds. When using multi-hole injectors, high fuel pressures are required to accomplish this. There are many parameters that can be adjusted on a multi-hole injector and which greatly influence the spray formation, air entrainment, cross-flow velocity and the fuel/air ratio near the spark plug. These parameters include hole diameter, hole length/hole diameter ratio (l/d), number of holes, the angle between the holes (resulting in different spray umbrella angles) and the configuration of the holes on the nozzle tip. There are a number of ways to configure the holes on the nozzles such as having more than six holes per nozzle [3], guiding the fuel in a specific direction (offset), each hole having different properties, such as their l/d etc. Another parameter related to the spray formation is fuel pressure; this was studied in [4].

This paper focuses on evaluating different configurations of nozzle holes to see how this influences the fuel distribution, air entrainment, cross-flow velocity and turbulence. Both symmetrical and asymmetrical nozzle hole configurations are evaluated. This was achieved using both simulations and experimental methods, which included Laser-Induced Fluorescence (LIF), Mie-scattering, High Speed Video, Phase Doppler Anemometry (PDA) and Computational Fluid Dynamics (CFD).

EXPERIMENTAL AND NUMERICAL METHODS

EXPERIMENTAL METHODS

The measurements were performed in a heated and pressurized spray chamber. This provides a continuous flow of heated, pressurized air. The velocity of the air in the measurement section of the chamber is relatively low (around 0.1 m/s). Therefore, relative to the velocity of the injected spray (up to 150 m/s), the environment can be regarded as quiescent. For the LIF and Mie measurements described in this paper, a pulsed Spectraphysics Nd:YAG laser was used (operating at a wavelength of 266 nm) in conjunction with three cylindrical lenses, to form a laser sheet approximately 40 mm high and less than 1 mm thick. A laser pulse energy of 50 mJ was fired through the fuel spray 10 mm downstream of the nozzle. (Due to limitations in the experimental equipment, it was not possible to acquire images further than $z=10$ mm downstream.) The LIF/Mie images were captured using a 16-bit LaVision Dynamight intensified CCD camera. The fuel used was Iso-octane with 10 % tracer (3-pentanone). A narrow band filter, centered on 266 nm, was used for the Mie signal and a filter centered on 407 nm was used for the LIF signal. The images were acquired using LaVision DaVis 6.2 software and were further analyzed using an in-house MATLAB® code. Only one image could be captured during each injection, so each image originates from a different injection event. The measurement setup is presented schematically in Figure 1.

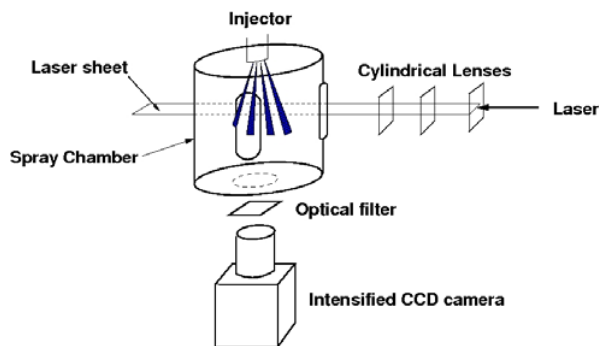


Figure 1. Schematics of the experimental setup.

The injector duration was 2.0 ms for all cases examined and the ambient conditions in the spray chamber were 1.0 MPa/600 K in all cases. Under these conditions there is a high degree of vaporization. The fuel is fully vaporized tenths of a millisecond after the end of the liquid fuel injection. The fuel was injected into the spray chamber at a rate of 0.33 Hz. The fuel pressure was 25 MPa for all measurements. The injectors available for testing were four different custom made diesel-type nozzles with $l/d=10$. This l/d ratio is higher than that of current production GDI injectors. A high l/d ratio is known to cause narrower spray plumes than a low l/d ratio and therefore all results presented here may not be

applicable to low l/d ratio nozzles. However the influence of l/d ratio is mostly significant for low fuel pressures [5].

NUMERICAL METHODS

Numerical simulations were performed to generate data which are otherwise difficult to measure. These include air entrainment, cross-flow velocities, turbulent kinetic energy and local fuel/air ratios between the spray plumes. All numerical simulations were performed using AVL FIRE™ version 8.4. These simulations require the input of experimental data.

Experimental data used for CFD calibration

Visualizations and PDA measurements of the spray were used as the input for the numerical simulations. For these types of nozzles, the degree of atomization is mostly dependent on the fuel pressure and the diameter of the nozzle holes. Since the hole diameter for all tested nozzles are the same it is assumed that the atomization quality for all nozzles are very similar. Therefore, Phase Doppler Anemometry was performed on Nozzle 1 only. This nozzle has the smallest umbrella angle making it the easiest one to measure when the limited width of the spray chamber's quartz glass is taken into consideration. In order to capture high quality data, measurements were taken 20 mm downstream of the nozzle. This is fairly close to the nozzle but since the measurements were carried out in a heated/pressurized environment it was possible to capture quality data. The validation rate varied between 30 to 50 samples per injection. Careful adjustments were carried out in order to capture data precisely in the mass center of the spray plume. The results are summarized in Figure 2, in which count- and volume-based diameter histograms are shown together with cumulative volume.

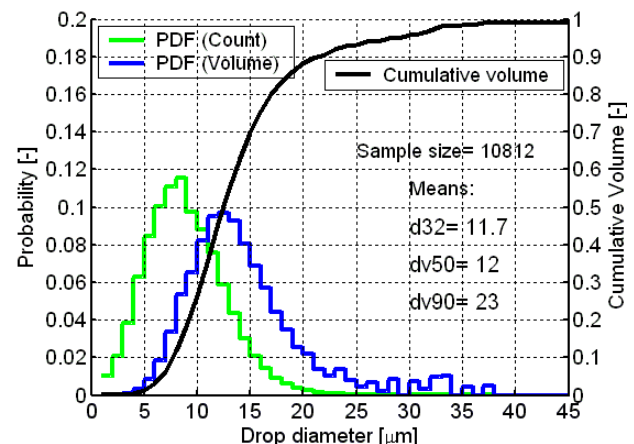


Figure 2. Histograms (PDFs) of diameter samples with cumulative volume. The sample is from the spray plume core (mass center) of one of the spray plumes of Nozzle 1. Bin size 1 micron. 1 MPa/600 K.

Direct imaging (AVL Visioscope), repeated 10 times, was used to acquire liquid phase images. From these images a liquid penetration curve was calculated and this curve, together with the PDA data, was used to calibrate the input parameters for the CFD simulations. Due to a high degree of vaporization and a high ambient density, the liquid penetration curve is limited to about 20-22 mm (Figure 3).

Set-up of the simulated sprays

The spray break-up and atomization were modeled with the Kelvin-Helmholtz/Rayleigh-Taylor Hybrid model [6] available in AVL FIRE™ version 8 [7]. The approach adopted in this work was to find a parameter setting for the break-up model such that the calculated parameters, for example spray velocity and drop sizes at a downstream location, matched the corresponding measurements. Drop collisions and coalescence were not modeled since better agreement with experimental data was obtained without modeling these phenomena. The models available in the FIRE™ software were used to simulate evaporation, drag, and turbulent dispersion.

In Figure 3, the axial liquid penetration length of the simulated spray is compared to the liquid penetration from the experimental data. (The axial liquid penetration is defined here as the downstream distance from the injector orifice to the liquid spray front.) In the CFD-code the position of the liquid spray front was defined as the location where 99% of the mass (liquid phase) is located above this position. The results show that the penetration lengths, and thus the spray front velocities, of the two sprays are well correlated. Figure 3 also shows that the simulated spray penetrates somewhat further than the experimental spray. This is probably due to a few larger parcels that penetrate further than the mean spray. It is also difficult to detect the liquid spray front experimentally when maximum penetration is reached in a vaporizing environment.

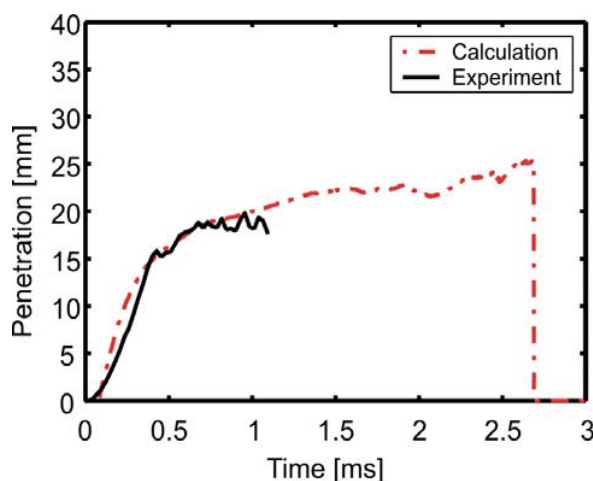


Figure 3. Result after CFD tuning. Axial liquid penetration lengths.

In Figure 4a, the measured time-resolved Sauter mean diameters of the drops captured 20 mm downstream of the nozzle tip in the free stream spray are compared with the simulated spray at the same distance from the nozzle tip. As can be seen, the simulation captures the experimental Sauter mean diameters quite well over the entire injection duration. These results show that, in the early injection phase, the simulated spray tip contains drops with larger diameters than those in the real spray. This is probably because the large drops penetrate more quickly than the smaller ones, so there are rather more large drops in the spray tip. However, there is good agreement between the simulated and experimental data for the main injection phase of the spray. In Figure 4b the simulated and experimental axial velocities of the free stream spray are compared. The figure shows that the simulated mean velocity is somewhat lower than the real mean spray velocity. The largest difference is in the tail of the spray and this is probably due to the slightly under predicted Sauter mean diameter of the spray. The smaller drops carry less momentum and have a lower velocity.

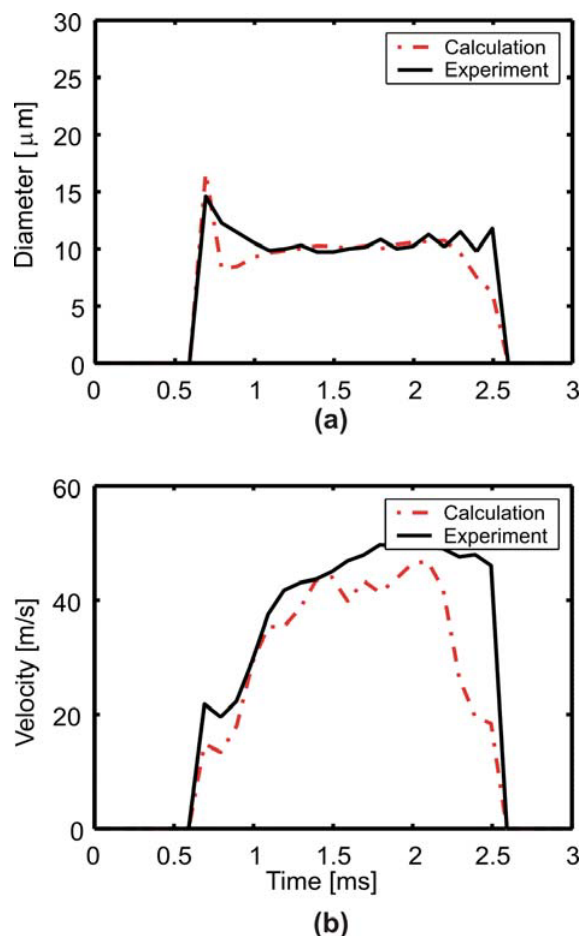


Figure 4. Result after CFD tuning. Data from 20 mm downstream of the nozzle. a) Time-resolved Sauter mean diameters and (b) free mean stream velocity in the axial direction.

NOZZLE CONFIGURATIONS

Four different configurations of the nozzle holes were evaluated, both experimentally and numerically. All nozzles have six holes located as shown in Figure 5.

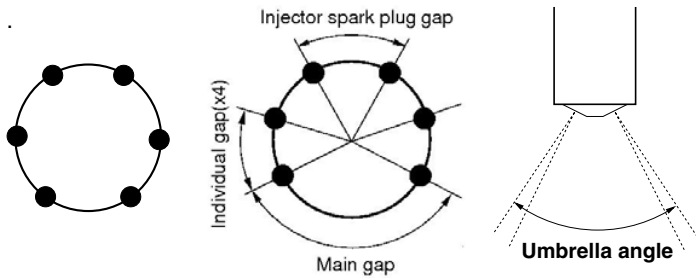


Figure 5. Nozzles with definition of parameters. Symmetrical configuration (left), asymmetrical or “horseshoe” configuration with the main gap, individual gap(x4), injector spark gap (middle) and umbrella angle (right).

The different nozzle hole configurations are summarized in Table 1. Note that for Nozzle 4, the individual gap does not equal the injector spark gap.

Table 1. The nozzle hole configurations investigated

	1	2	3	4
Config.	Symm.	Symm.	Asymm.	Asymm.
Hole diameter (microns)	101	101	101	101
l/d	10	10	10	10
Main gap	60	60	160	120
Individual gap	60(x6)	60(x6)	40(x4)	45(x4)
Injector spark gap	60	60	40	60
Umbrella angle	70	85	85	85

SPRAY-INDUCED AIR MOTION

High fuel pressure is needed to atomize the fuel into sufficiently small droplets so that they vaporize fast enough. One side effect of the high fuel pressure is that the velocity of the droplets is very high and, therefore, the injected liquid will penetrate with great force (high momentum). This high velocity will result in stable liquid spray plumes; these are relatively insensitive to an engine’s in-cylinder air motion, at least with respect to late injections when the in-cylinder density is high. This

is exemplified in Figure 6, which shows a photographic sequence of the liquid spray from Nozzle 1. The photos were acquired using a high-speed camera in a heated spray chamber. It can be concluded that the liquid is injected with such tremendous momentum that the mean direction of the droplets equals the direction of the drilled nozzle holes. Due to the droplets’ high momentum, air is pushed away very rapidly and, therefore, air must refill areas where it has been pushed away due to pressure differences. A spray-induced air motion develops, which explains how fresh air is being entrained into the spray. Figure 6 (0.7 ms, side view) also illustrates that when maximum penetration is reached it becomes difficult to experimentally detect the liquid spray front due to vaporization.

As will be seen, the spray-induced air motion differs between the different nozzle configurations. This motion has an influence on how the vaporized fuel moves in the near-nozzle region but, more importantly, it influences the cross-flow velocity and turbulence in the nozzle’s symmetry plane where a spark plug can be placed.

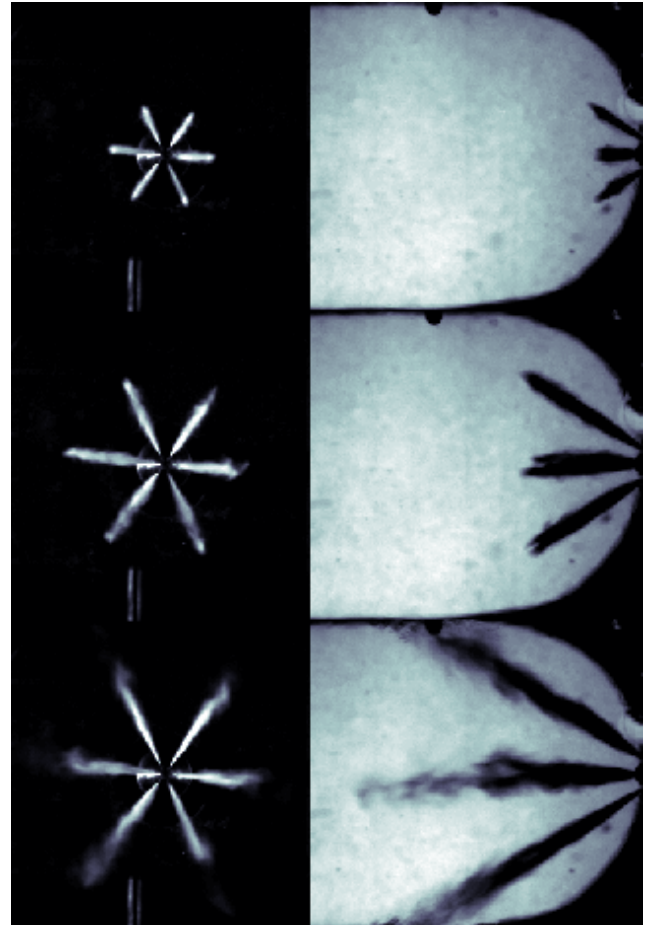


Figure 6. High-speed photograph sequence from Nozzle 1 with underside view (left) and side view (right). Liquid phase. Timing 0.2 ms, 0.3 ms and 0.7 ms ASOI.

AIR MOTION CAUSED BY A SINGLE SPRAY PLUME (VERTICAL PLANE)

Figures 7 to 9 show vertical cross-sections through the spray plume for the simulated induced velocity field of the air/vaporized fuel at three different timings: 0.3 ms (early in the injection), 0.9 ms and 1.8 ms (late in the injection). The simulated sprays have been carefully tuned to experimental data with regards to spray formation, penetration and atomization but the simulated air motion has not been verified experimentally, so the following result is based on CFD only.

The characteristics of the flow field are mainly governed by single jet theory and do not have much to do with the interaction between plumes, unless the plumes are located very close together. Thus, the following flow characteristics are valid for all tested nozzle configurations. Starting with Figure 7, which illustrates an early point during injection, air is starting to be pushed away by the fuel droplets.

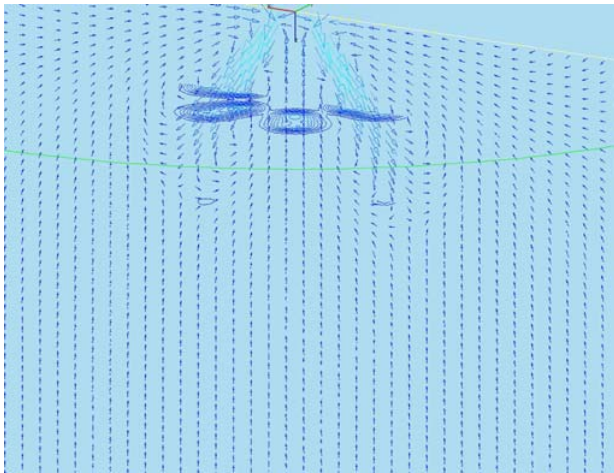


Figure 7. Velocity field of air/vaporized fuel and iso-lines of equivalence ratio at 7.5 mm downstream; early in the injection process, 0.3 ms ASOI. Nozzle 1.

A vortex is being created at the spray front. Air is starting to be entrained from the side near the nozzle pointing towards nozzle centre. Air is also entrained from inside the spray plume in an upwards direction towards the nozzle; it moves at a lower velocity than the air entrained from the side. This air entraining motion is governed by the counter-rotating vortex created at the fuel liquid spray front. The velocities are still low. In Figure 8 the centre of the vortex has moved downstream since the fuel liquid has penetrated further.

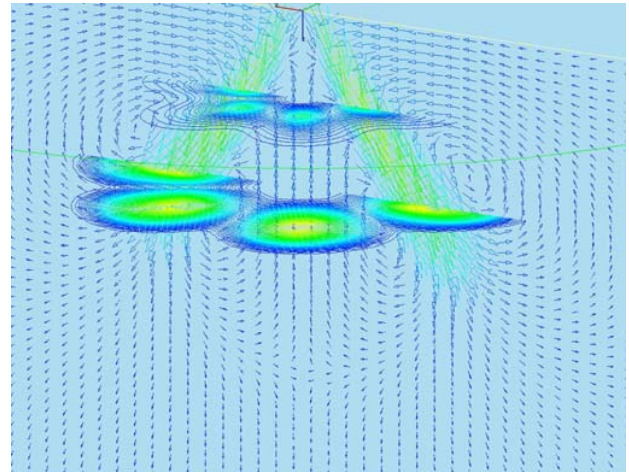


Figure 8. Velocity field of air/vaporized fuel and Iso-lines of equivalence ratio at 7.5 and 15 mm downstream, 0.9 ms ASOI. Nozzle 1.

Note that there is always an inwards-pointing velocity field above the position of the vortex. The velocities are now higher. In Figure 9, the velocity field is similar to Figure 8 but the vortex has moved further downstream.

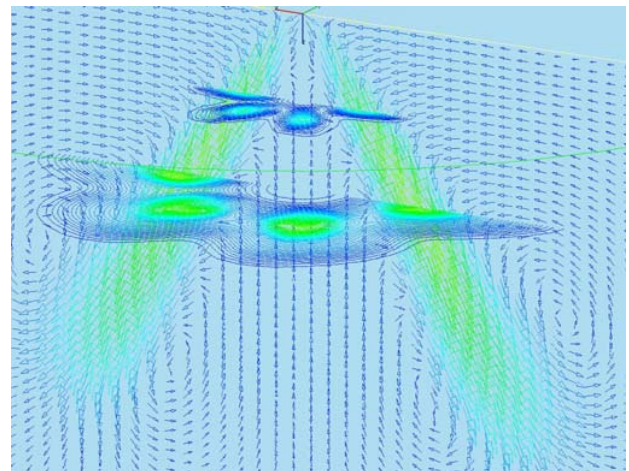


Figure 9. Velocity field of air/vaporized fuel and Iso-lines of equivalence ratio at 7.5 mm downstream, 1.8 ms ASOI. Nozzle 1.

In conclusion, above the vortex centre there is no mechanism to transport the vapor in any other direction than towards the nozzle centre. Fuel vapor exposed to this velocity field will inevitably be influenced and move towards it.