

Characterization of the Near-Field Spray and Internal Flow of Single-Hole and Multi-Hole Sac Nozzles using Phase Contrast X-Ray Imaging and CFD

2011-01-0681

Published
04/12/2011

Ming-Chia Lai, Yi Zheng and Xing-Bin Xie
Wayne State Univ.

Seoksu Moon and Zunping Liu
Argonne National Laboratory

Jian Gao
Univ. of Wisconsin

Xusheng Zhang, Kamel Fezzaa and Jin Wang
Argonne National Laboratory

Junmei Shi
Continental Automotive GmbH

Copyright © 2011 SAE International

doi:[10.4271/2011-01-0681](https://doi.org/10.4271/2011-01-0681)

ABSTRACT

It is well known that the internal flow field and nozzle geometry affected the spray behavior, but without high-speed microscopic visualization, it is difficult to characterize the spray structure in details. Single-hole Diesel injectors have been used in fundamental spray research, while most direct-injection engines use multi-hole nozzle to tailor to the combustion chamber geometry. Recent engine trends also use smaller orifice and higher injection pressure. This paper discussed the quasi-steady near-nozzle Diesel spray structures of an axisymmetric single-hole nozzle and a symmetric two-hole nozzle configuration, with a nominal nozzle size of 130 μm , and an attempt to correlate the observed structure to the internal flow structure using computational fluid dynamic (CFD) simulation. The test conditions include variation of injection pressure from 30 to 100 MPa, using both Diesel and Biodiesel fuels, under atmospheric condition. The imaging technique utilizes a 150 pico-second synchrotron-based Ultrafast Phase-contrast X-ray in order to freeze-capture the

fast moving jet and achieve excellent spatial resolution in order to compare the wavy jet structure. Both nozzle holes originate from a sac of identical geometry, but different flow structure inside the nozzle cause significant difference in the observed flow structure near the nozzle exit. The ultrafast images revealed unique surface and internal morphology of the fuel sprays that can be identified. The two-hole nozzle produces much more unstable jet structure under same injection conditions. The early wavelength developed in the jet is measured to be on the order of 30~80 μm , with the frequency range of 5 to 10 MHz, depending on the injection conditions up to 60 MPa injection pressure when the wavelength analysis is still feasible. The differences between the nozzle configurations are investigated using CFD simulation. The results show that the three-dimensional fluid flow entering the two-hole nozzle generates stronger streamline curvature and stream-wise counter-rotational vortices which are by default absent in the axisymmetric single-hole nozzle. It also produces thicker shear layer and higher turbulence. The interactions of downwash entrance

flow with turbulence potentially enhance the instability and produce wider spray cone angles.

INTRODUCTION

Near-nozzle fuel jet development dominated by internal nozzle flow and related fluid-dynamic instability governs the primary break-up process of the injected fuel. The momentum of the fuel spray and spray atomization control the mixing of the injected fuel with surrounding oxidant gas, which is crucial to achieve highly-efficient and clean combustion for direct injection (DI) diesel and gasoline engines. Recent trend of the diesel engine is to use smaller and more orifices, and higher injection pressure. The advantages of smaller nozzle holes have been reported [e.g., 1], such as lower particulate matter (PM) and carbon monoxide (CO) emission and possibly better fuel economy. To place multiple orifices in an injector nozzle, the orifices are located off-axis to the nozzle axis and symmetrically aligned around the nozzle axis, which is most preferred DI diesel nozzle configuration. The gasoline direct injection engines also converge to the multi-hole injector geometry because of its robustness, flexibility and cost-performance. The gasoline DI multi-hole injectors sometimes use both symmetric and asymmetric hole patterns, with or without a center hole.

Visible-light imaging techniques have been dominantly used to visualize the internal and near-nozzle fuel flow and led to remarkable progress in analysis of near-nozzle spray geometry [2,3,4,5,6,7,8]. However, these techniques have their limitation in the visualization of near-nozzle jet morphology especially for dense and high-speed fuel sprays. To overcome this, single-shot ultrafast x-ray phase-contrast imaging technique has been introduced recently for dense high-speed fuel sprays [9,10,11,12]. X-ray phase-contrast imaging technique records phase variations of the emerging radiation from liquid/gas interfaces rather than absorption-induced intensity variations when the x-ray beam passes through an object [12]. Near-nozzle fuel jet morphology during high-pressure Diesel injection by axially symmetric single-hole nozzles has been unveiled from recent studies using X-Ray. However, such study has not been extended to more realistic and practical multi-hole injectors until now. The two-hole nozzle is a simplified version of multi-hole nozzle, where two orifices are symmetrically aligned around the nozzle axis. The jet morphology of the two-hole nozzle was studied with the high intensity and high-brilliance x-ray beam available at the Advanced Photon Source (APS) in Argonne National Laboratory with a sub-ns (150 ps) temporal duration and 1 μm spatial resolution.

This off-axis location of the orifices makes the near-nozzle spray to be strongly affected by the vortex flow inside the nozzle sac other than axial laminar-like flow. The structure of the vortex flow and cavitation inside the multi-hole Diesel

nozzles has been unveiled from quite a few simulation studies [2,3,4, 13, 14] and large scale-up hydraulic (water analog) model experiments. These results showed that vortical flow is formed inside the nozzle orifice and it strongly affects the near-nozzle spray development [2, 3, 14]. The CFD simulations using more complicated Eulerian and Lagrangian cavitation models have been shown to correlate the rotational flow structure observed at the scale-up hydraulic nozzle model [2, 3, 14]. Specifically, there seems to the existence of counter-vortex flows inside the scaled-up hydraulic model as visualized by the cavitation bubbles. Depending on the source of locations, these cavitation streaks are termed “geometric cavitation” from flow separation, and “streak cavitation,” which connects between the orifices via the sac.

Some previous study focused on the near-nozzle jet morphology of high-pressure Diesel spray injected by a single-hole nozzle which center of hole is located on the nozzle axis [8,9,10]. Previous test results revealed the laminar-like jet features formed very near the nozzle exit, which becomes unstable mostly due to turbulence and cavitation inside the nozzle and aerodynamic effect much later on outside the nozzle. In the case of smooth orifice inlet, where vena contracta and cavitation effects are less significant, mono-dispersed planar wave is formed on the jet surface, while totally turbulent jet morphology rather than wavy structure was observed for sharp orifice inlet. However, such a study had not extended to more realistic and practical multi-hole nozzles which jets are strongly affected by the vortex flow inside the nozzle.

The aim of this study is to investigate the quasi-steady near-nozzle jet morphology of the high-speed fuel spray injected from small-orifice nozzles, comparing an axisymmetric straight single-hole (SHN) geometry to that of a symmetric two-hole nozzle (THN). The two-hole nozzle is a simplified version of multi-hole nozzle. The near-nozzle jet morphology of the two-hole nozzle was obtained by the single-shot ultrafast x-ray phase-contrast imaging technique, implemented by the high-intensity and high-brilliance X ray beam available at the Advanced Photon Source (APS) in Argonne National Laboratory. Two CFD techniques are used: the well-validated Reynolds-Averaged Navier-Stokes (RANS) with standard $k-\epsilon$ turbulence model is used mainly to show the mean flow features and the effects of number of holes; finally, the Unsteady Reynolds-Averaged Navier-Stokes (URANS) with shear stress transport (SST) turbulence model is also shown for comparison.

EXPERIMENTAL SETUP AND CONDITIONS

Experimental Setup

Experiments are performed using a phase-contrast X-ray imaging setup at XOR 7ID beamline in APS. The x-ray beam

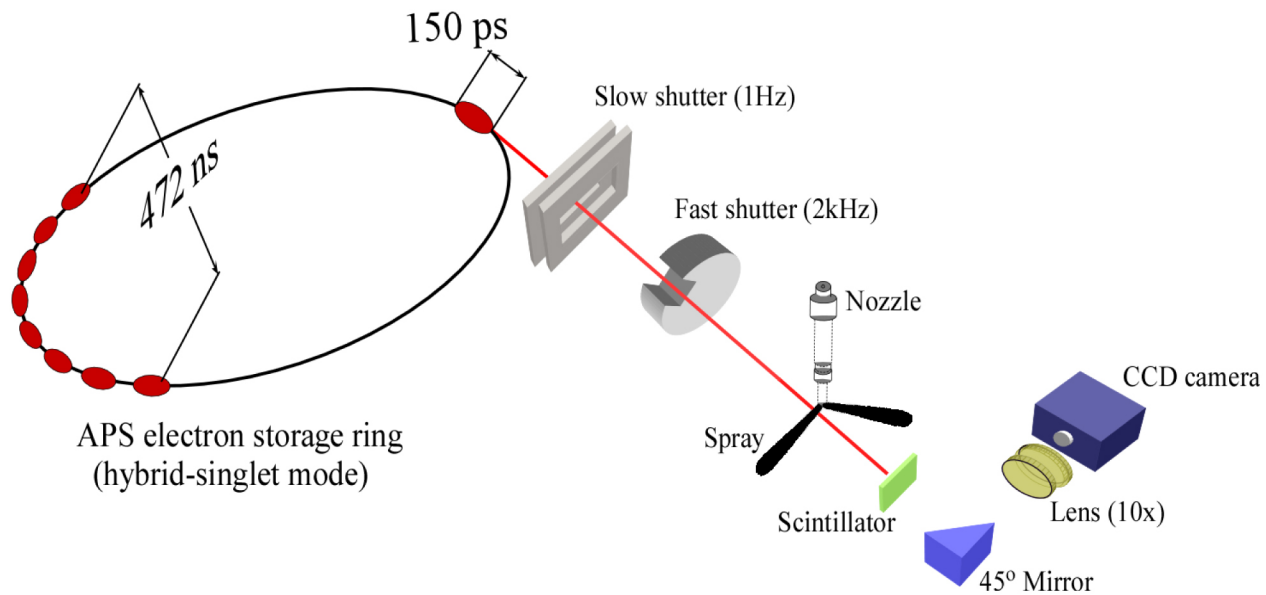


Figure 1. Experimental setup for phase-contrast x-ray imaging

is generated from an insertion device (undulator) in the APS electron storage ring. The special beam pattern (hybrid singlet mode) shown in Fig. 1 was used in this experiment. This pattern contains a single electron bunch (150 ps duration and carrying 16mA current) isolated from the remaining electron train bunch (472 ns long, 96 mA) by symmetrical 1.594 μ s gaps. To reduce the heat power, the x-ray beam was gated by two mechanical shutters: the slow shutter operating at 1Hz frequency with 10 ms opening duration and the fast shutter operating at 2 kHz frequency with 9 μ s opening duration [14]. Synchronized operation of these two shutters cuts-off more than 99% of the beam heat power.

After transmitted to the spray, the x-ray beam generates the phase-contrasted image on the scintillator crystal (LYSO:Ce), which converts the transmitted X-ray beam into visible light (432 nm). This image is reflected by a 45° mirror and then captured by a charge coupled device (CCD) camera (Sensicam, 1376×1040 pixels, from Cooke). The camera was gated at the timing when only the singlet electron bunch passes through the shutters. The remaining electron bunches were cut off by closing the camera gate at 1.5 μ s after the gate opening (exposure time of 1.5 μ s). The field of view of the camera was 1.75×1.32 mm² when a 10x objective lens (NA=0.14) is used.

The fuel was injected into a spray chamber using a common-rail injection system, composed of fuel tank, motor, high-pressure pump, pressure valve and common-rail. The pressure inside the common rail was controlled via feedback control of measured pressure inside the common-rail and bleeding fuel flow rate of the pressure valve. The spray chamber has two Kapton windows which allow the x-ray beam to pass through this without decrease in intensity.

Table 1 shows the experimental conditions. A two-hole nozzle (THN) with the nozzle angle between two-orifices of 135° was used as the test nozzle. It is a tapered hole nozzle, with the diameter at the inlet 130 μ m and 110 μ m at the outlet. A reference nozzle, single-hole nozzle (SHN) with the orifice diameter (Do) of 0.135mm, was also used in this study for the comparison with two-hole nozzle. Figure 2 shows the internal structure of SFN and TFN visualized with higher energy x-ray beam and longer exposure time [15]. These two nozzles have same sac diameter and similar level of hydro-grinding, but has slightly different nozzle hole diameter. The difference in the hole exit diameter is small (around 4%). The injection pressure range applied varied from 30MPa to 100MPa. Relatively longer injection pulse duration (3.0 ms) than that of real Diesel engines was applied to investigate the near-nozzle flow characteristics in the quasi-steady injection stage. The real injection duration, determined by the spray images, was around 4.0 ms in this injection pulse duration. Experiments were performed under room pressure and temperature conditions. Both Diesel and Biodiesel [16] were used as the test fuels, as well as a 50%-50% blend in some cases. The hole on the right-hand side shown in Fig. 2 is used for the experimental results presented in this paper.

CFD Simulation has been shown to correlate the injector design to spray performances [4, 5, 17]. To aid the interpretation of the results, steady-state CFD simulation is carried out at full needle lift for both the single-hole and two-hole nozzles using Fluent. As a first step, the Reynolds-Averaged Navier-Stokes (RANS) with standard k- ϵ turbulence model, and Eulerian full-cavitation model, which is based on the assumption of a continuous mixture of liquid and vapor bubbles having the same velocity, is used. To simplify, the axisymmetric, two-dimensional geometry

Table 1. Experimental Conditions

Nozzle	Single-hole nozzle: $D_0=0.135\text{mm}$ Two-hole nozzle: $D_0=0.130\text{mm}$, $\theta=135^\circ$
Injection Pressure, P_{inj} (MPa)	30, 40, 50, 70, 100
Injection Pulse Duration, T_p (ms)	1.0 or 3.0 (quasi-steady condition)
Ambient Conditions	N_2 with Room Conditions
Fuel	Biodiesel: $v: 5.02\text{mm}^2/\text{s}$, $\rho: 888\text{kg/m}^3$ $\sigma: 0.033\text{N/m}$ Diesel: $v: 3.18\text{mm}^2/\text{s}$, $\rho: 856\text{kg/m}^3$ $\sigma: 0.028\text{N/m}$

meshing with a total of 30,000 cells is used for single-hole nozzle, and symmetric geometry with 550,000 cells hybrid mesh is used for two-hole nozzle to begin with, as shown in Fig. 3a. These symmetry assumptions rule out needle eccentricity, surface roughness, and any initial swirl component along the injector axis. The working fluid is a mixture of diesel fuel and vapor. And non-condensable gas is considered. The computed pressure contours using the geometrical dimension measurement taken from the phase contrast image are shown in Fig. 3b. The diagnostics and resolution of cavitation bubbles, which can be sub-micron in size, inside real-size nozzle is much more difficult for visible light techniques and even the current X-ray techniques [26,27], especially for sac-type nozzles. The simultaneous matching of Reynolds, Cavitation, and Weber numbers of the scale-up model to real-size nozzle also makes it difficult to assess and the effect of scaling on cavitation models from scale-up model to real size nozzle. However, the hydro-ground hole inlets of both SHN and THN, and the converging nozzle configuration of the TFN are known to suppress cavitation and increase discharge coefficients. CFD simulations based on RANS and URANS turbulence models were applied to study the vortex structure and cavitation in injection holes. According to a comparative study of URANS approach with SST turbulence model and Scale Adaptive Simulation (SAS) [25], the current numerical approach, although may be overdamped with numerical dissipation, should be sufficient to compare the mean flow structure of the ideal nozzles tested. Finally, the full nozzle geometry results for both the RANS with standard $k-\varepsilon$ turbulence model and the URANS with SST turbulence model were compared.

RESULTS AND DISCUSSION

NEAR-NOZZLE JET MORPHOLOGY

Phase-contrast X-ray spray imaging has been shown for many SHN nozzle similar to the one used in this study [e.g., 9], although the hole size used previously was slightly larger (175 as compared to the current 130 microns). Therefore, this paper will focus more on the THN results and used the SHN as comparison. It has been shown that SHN has very stable jet, especially for nozzle with hydro-rounded inlet using more viscous fuel. It is quite easy to patch spray images from adjacent locations downstream taken at the same time to show the spatial structure coherence, but this is not the case for THN, where the transient is more dynamic and longer in duration. It is much more difficult for THN to reach quasi-steady state, especially for short injection duration or when the needle is not fully open. Fig. 4 show the typical examples of the very dynamic transient spray structures taken at 5 adjacent downstream positions at the same time of 1.0ms the start of injection (ASOI), for THN at 40, 70, and 100MPa injection pressures with a 1ms injection pulse duration. Fig. 4a, with the injection pressure condition of 40MPa, has very unstable and widely disperse sprays, primarily because the needle is not fully lifted at this time, and a transitional instability cause the spray to vary wildly. This results in the larger spray angles observed in microscopic visualization during valve opening and closing transients at the start and end of injection [8]. As the pressure increases, the needle valve is lifted faster and a more stationary turbulent spray becomes more established, although still much more dynamic and wider than the SHN shown in previous reports.

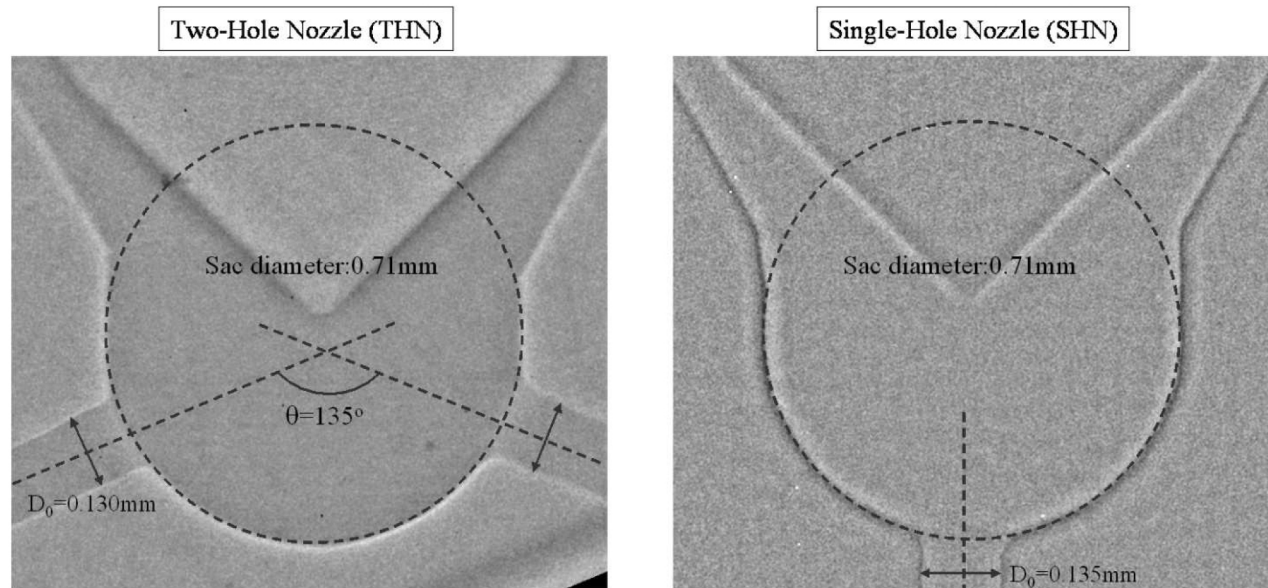


Figure 2. Internal nozzle structure of two-hole (left) and single-hole (right) nozzles captured using phase-contrast imaging technique (detailed methodology can be found in [18])

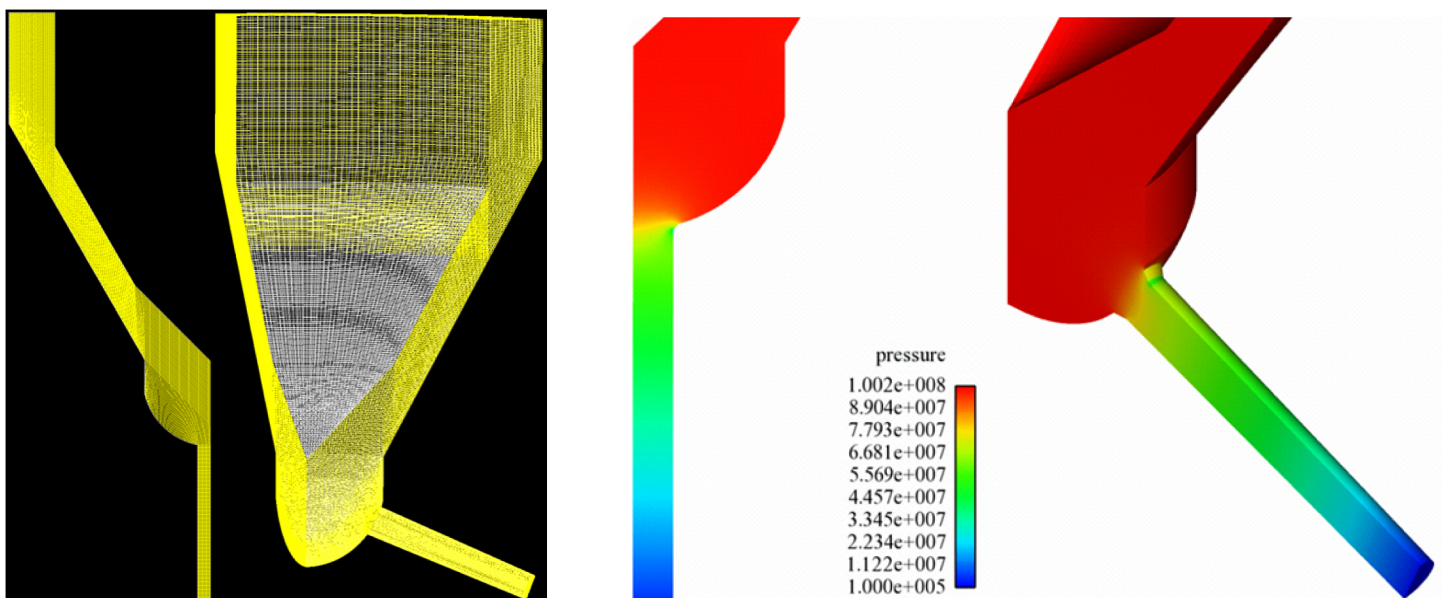
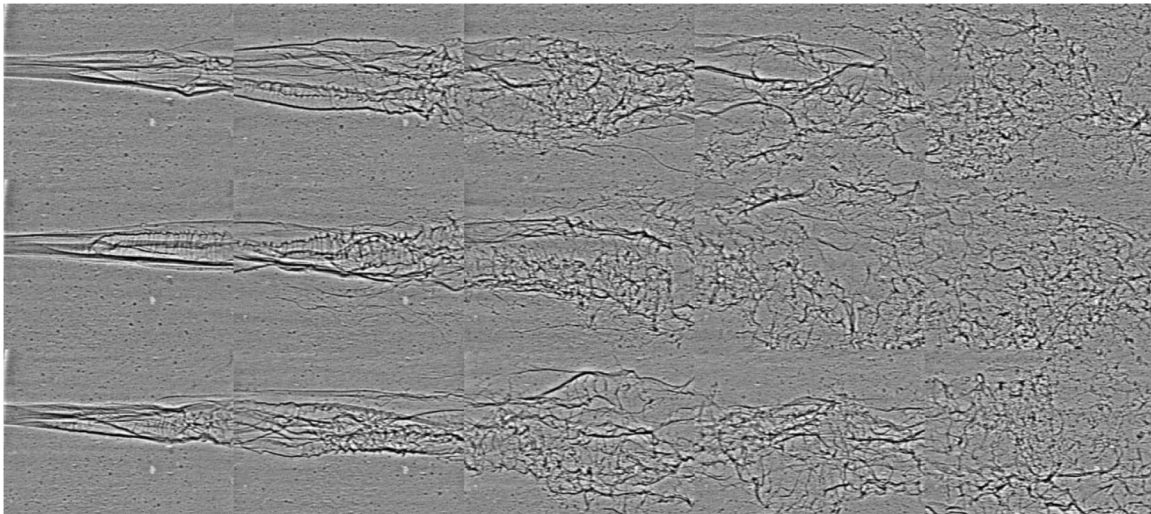


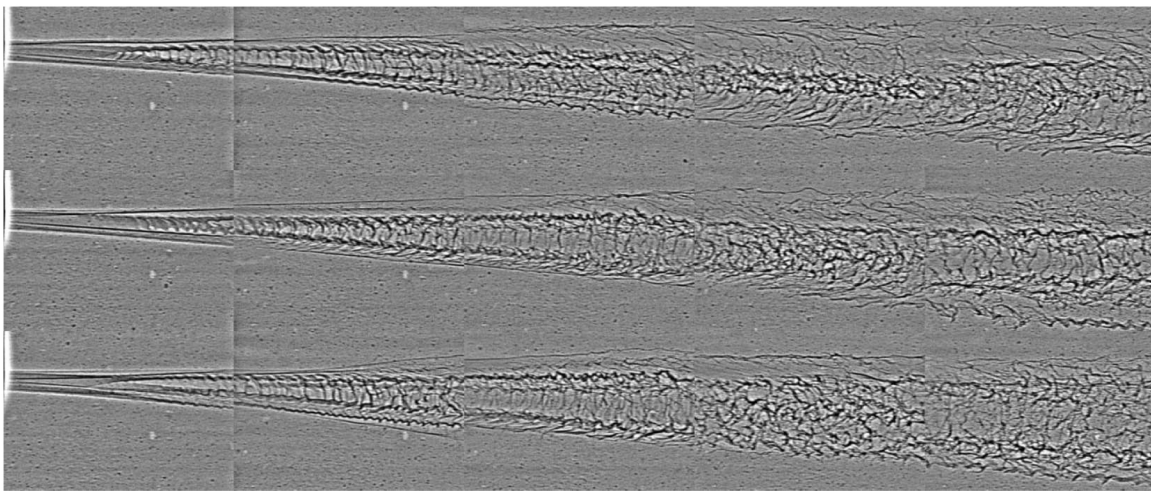
Figure 3. (a) The Computation grids and (b) the computed mean pressure contours, using Fig. 2 geometry for single-hole (left) and two-hole (right) nozzles. Inj. Pressure: 100 MPa.

In order to provide a longer aspect ratio of the spray spatial development and retain the highest spatial resolution of the current imager used, the images shown consist of only two tiles, taken at adjacent streamwise locations near the nozzle exit at the same time after the start of injection (ASOI) for a longer injection pulse duration of 3ms. They are put together to focus the quasi-steady spray structure taken at various injection pressure and fuel conditions, at 2.0ms ASOI when the needle is fully open. The dramatic difference between the

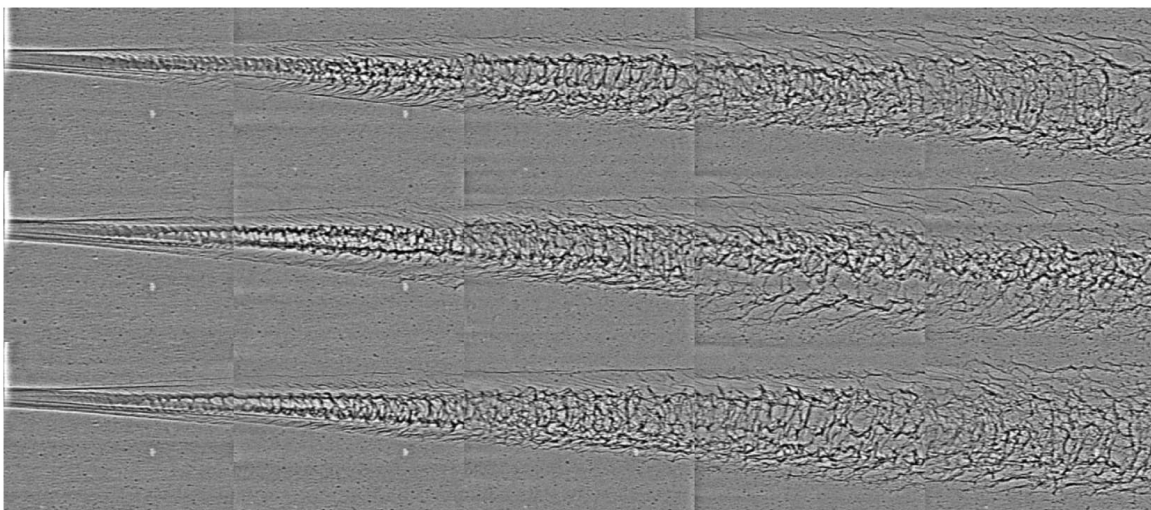
SHN and THN is shown in Fig. 5 at the low injection pressure of 30MPa, using more viscous Biodiesel fuels between the SHN and the THN cases. Very distinct wavy structure was clearly observed near the THN hole exit and the lateral jet boundary began to spread, as early as 1 mm downstream. In contrast, the jet coming out of SHN is very stable, almost laminar-like, and spread much less slowly as far as 3mm downstream, although there seems to be some disturbances propagating downstream without developing



(a)



(b)



(c)

Figure 4. Typical THN spray structures taken at 5 adjacent downstream positions at 1.0ms ASOI, at (a) 40MPa, (b) 70MPa, and (c) 100MPa injection pressures, with a 1ms injection pulse duration.