

Measurement of Diesel Spray Formation and Combustion upon Different Nozzle Geometry using Hybrid Imaging Technique

Anqi Zhang

Michigan Technological University

Alessandro Montanaro and Luigi Allocca

Istituto Motori CNR

Jeffrey Naber and Seong-Young Lee

Michigan Technological University

ABSTRACT

High pressure diesel sprays were visualized under vaporizing and combusting conditions in a constant-volume combustion vessel. Near-simultaneous visualization of vapor and liquid phase fuel distribution were acquired using a hybrid shadowgraph/Mie-scattering imaging setup. This imaging technique used two pulsed LED's operating in an alternative manner to provide proper light sources for both shadowgraph and Mie scattering. In addition, combustion cases under the same ambient conditions were visualized through high-speed combustion luminosity measurement. Two single-hole diesel injectors with same nozzle diameters (100 μ m) but different k-factors (k0 and k1.5) were tested in this study. Detailed analysis based on spray penetration rate curves, rate of injection measurements, combustion indicators and 1D model comparison have been performed. It is concluded that the nozzle geometry is causing the velocity and liquid flow area to vary at the nozzle outlet, which has an impact on air entrainment into the spray and results different combustion behavior.

CITATION: Zhang, A., Montanaro, A., Allocca, L., Naber, J. et al., "Measurement of Diesel Spray Formation and Combustion upon Different Nozzle Geometry using Hybrid Imaging Technique," *SAE Int. J. Engines* 7(2):2014, doi:10.4271/2014-01-1410.

INTRODUCTION

The combustion event occurring inside a diesel engine is initiated by an auto ignited diffusion flame. Violent chemical reactions usually take place within the boundary sections of the fuel jet, where favorable equivalence ratios for combustion are formed. Due to this nature of the combustion process, the mixing process of the fuel and air has a significant impact on the combustion quality of the fuel, thus affecting the energy conversion efficiency and the emission performance of the engine.

Understanding of diesel spray characteristics has been widely investigated [1, 2, 3]. Two major approaches, experiments and modeling, have been extensively adopted to understand detailed characteristics of diesel sprays, including the penetration rates, spray dispersion, vaporization, and fuel distribution. Volumes of experimental data obtained from various test setups lay a foundation for this research area. Test conditions cover a wide range of fuel injection pressures, nozzle geometries, ambient densities, ambient temperatures, and ambient gas compositions. Constant volume pressurized chambers capable of maintaining diesel-engine-like

environments are commonly used for diesel spray investigation [3, 4, 5]. A number of visualization techniques have been developed and applied to this research area. Schlieren/shadowgraph imaging is widely used for visualization of transparent media with temperature or density gradient. The relatively simple optical setup provides sensitivity to the gradient of index of refraction caused by the difference in density. Such techniques are suitable for observing diesel spray formation where significant differences between densities of liquid fuel, vaporized fuel, and ambient gas exist. With the help of current technology, high speed cameras (usually tens of μ s per frame), satisfactory temporal resolution can be achieved (common injection duration on the order of ms). The disadvantage of schlieren/shadowgraph imaging is that the sensitivity of the optics greatly affects the measurements, such as spray cone angle [2] and liquid penetration length in vaporizing spray. Mie scattering is another commonly used technique which takes advantage of elastic scattering of light with broken-up liquid particles of typical diameters comparable to wavelength of incident light. Only liquid parts of the spray will yield a signal, making Mie scattering an ideal method for liquid spray visualization [6, 7]. Rayleigh scattering, on the other hand, is used to visualize

gaseous fuel molecule distribution. Rayleigh scattering images acquired by Idicheria and Pickett [8] showed the varying gaseous fuel concentration distribution within the cross-sectional area of the jet plume. However the liquid part of the plume has to be avoided from the incident laser sheet to reduce interference on the Rayleigh image.

Modeling of the fuel spray jet is another powerful approach. Simplified one-dimensional spray mixing models based on different assumptions have successfully predicted the jet penetration behavior under a wide range of operating conditions [2, 3, 9, 10]. A spray model with uniform radial profile along the penetration direction was derived for non-vaporizing sprays by Naber and Siebers [9]. An extension of the 1D model was proposed by Musculus and Kattke [10] by considering more realistic radial velocity profile. This model assumes a higher center-line fluid velocity and fuel volume fraction while keeping the two values zero at the spray cone boundaries, which forms a bell-shaped curve (however, different from a Gaussian curve as the two ends have zero value). Mixing results output from the model was compared with quantitative Rayleigh scattering measurement and achieved good agreement [2]. Similarly, another 1D analytical spray model, also found in Ref [3], further advances the research by taking into consideration both the vaporizing and reacting cases [11, 12]. On the other hand, computational fluid dynamic (CFD) simulation is also increasingly effective with the help of advanced computational power and massive experimental data. Som and Aggarwal [13] presented the KHACT model to simulate liquid breakup in CFD codes, which is able to handle turbulence and cavitation effect of the high-pressure discharged liquid flow, and closer agreement with experimental results was obtained.

The effect of nozzle geometry on spray characteristics has been recognized as it considerably affects the fuel velocity at the outlet of the injector. Characterizing nozzle geometry has been performed by multiple research groups [9, 14, 15]. Several descriptive coefficients are adopted to define the nozzle geometry. A brief background is given as follows.

Steady-state mass flow rate ($\dot{m}_f$) and momentum flux ($\dot{M}_f$) delivered by an injector nozzle can be expressed as:

$$\dot{m}_f = \int_{A_o} \rho_f \cdot u dA \quad (1)$$

$$\dot{M}_f = \int_{A_o} \rho_f \cdot u^2 dA \quad (2)$$

which is the integral of fuel density (ρ_f) and velocity (u) over the outlet area (A_o). Given that cavitation inside the nozzle tunnel

has been widely observed for high-pressure diesel fuel injection [15], the distribution of fuel velocity profile in the vicinity of the injector nozzle is not uniform. Simplification has been applied by considering effective fuel velocity (u_{eff}) and effective nozzle outlet area (A_{eff}). Mass flow rate and momentum flux are able to be kept the same by adopting the two equivalent constants in the following manner:

$$\dot{m}_f = \rho_f \cdot u_{eff} \cdot A_{eff} \quad (3)$$

$$\dot{M}_f = \rho_f \cdot u_{eff}^2 \cdot A_{eff} \quad (4)$$

Ratios between effective values and theoretical values reflect the nozzle geometry effect. Three commonly used coefficients are the velocity coefficient C_v , the area contraction coefficient C_a , and the nozzle discharge coefficient C_d . Definitions of these coefficients are listed below. u_{th} stands for the theoretical fuel outlet velocity.

$$C_v = \frac{u_{eff}}{u_{th}}, \quad \text{where } u_{th} = \sqrt{\frac{2(P_f - P_a)}{\rho_f}} \quad (5)$$

$$C_a = \frac{A_{eff}}{A_o} \quad (6)$$

$$C_d = C_v \cdot C_a \quad (7)$$

Examinations of these coefficients with respect to different injection pressures and nozzle geometries helped to identify the cavitation effects inside the nozzle. Cylindrical nozzles more easily suffer from cavitation effects than conical nozzles [15]. More accurate mass flow and momentum measurement techniques were developed to evaluate cavitation near the nozzle [16]. The effect of nozzle geometry on spray penetration, vaporization, and mixing was studied with the help of a validated CFD model [17], while experimental comparison for the same topic is rare at this time.

In the current study, an experimental procedure has been carried out to evaluate the effect of nozzle geometry on high-pressure diesel spray formation characteristics under diesel engine conditions, such as penetration and disperse angle. In addition, comparison of experimental data with a 1D model provides a closer insight of detailed phenomenon within the vicinity of the tiny injector nozzle. Furthermore, results from combustion cases, including ignition and combustion properties, are extracted to further understand differences in fuel-air mixing behavior resulted from nozzle geometry effect.

EXPERIMENTAL PROCEDURE

Fuel Injection System

Two high-pressure diesel injectors are used in this study. Both injectors are solenoid controlled and feature a single-hole axial-disposed nozzle. Nozzle geometry is different for these injectors in terms of convergence, which is also denoted as the k-factor, a dimensionless factor defined as:

$$k = \frac{D_{in} - D_{out}}{10 \mu m} \quad (8)$$

The orifice is a cylindrical or conical tunnel connecting the sac and the external. D_{in} and d_{out} (in μm) stand for the orifice diameters at the sac end and the nozzle outlet end, respectively. Both injectors share the same outlet diameter (100 μm) and tunnel length (1 mm). The k0 injector has cylindrical nozzle tunnel while the k1.5 injector has a conical nozzle that is convergent towards the outlet. Pneumatic driven fuel pump is used to provide near-steady injection pressure at the back of the injector.

Rate of injection (ROI) measurements have been performed for both injectors in a previous publication [5]. The fuel injection rate profiles showed typical “top-hat” shape curve with fluctuations over the steady-state region. A two-step electrical driver signal of 1 ms has been applied and the actual injection duration is about 2 ms. The averaged steady-state mass flow rate is reported in Table 1.

Table 1. Steady-state mass flow rate under different injection pressures with 1 ms driver current input (unit:g/s).

	$\dot{m}_f$, k0	$\dot{m}_f$, k1.5
$P_{inj} = 70 \text{ MPa}$	2.54	2.62
$P_{inj} = 120 \text{ MPa}$	3.63	3.70
$P_{inj} = 180 \text{ MPa}$	4.63	4.58

Optical Setup

Spray visualization is achieved in an optically accessible constant volume pressure vessel in this study, known as the combustion vessel (CV) at Michigan Tech. This visualization facility features a cubical internal chamber of 1-L internal volume and is capable of withstanding high pressure (up to 345 bar) and high temperature (up to 2000 K). The six surfaces of the cub are made as highly configurable windows. In current setup, the injector is mounted in one of the side windows and the top window is equipped with spark plugs and mixing fan, which are used in a pre-burn procedure discussed in later chapters to generate diesel engine conditions. Rest of the side ports are sapphire windows for optical access. Three corners of the CV have been configured as the intake valve, exhaust valve and the pressure transducer. CV pressure history is recorded by a high-speed data acquisition system with 100Hz sampling rate. (The Michigan Tech. CV facility was selected in Engine Combustion Network (ECN) as one of the facilities to

provide “an open forum for international collaboration among experimental and computational researchers in engine combustion”. <http://www.sandia.gov/ecn/index.php>.)

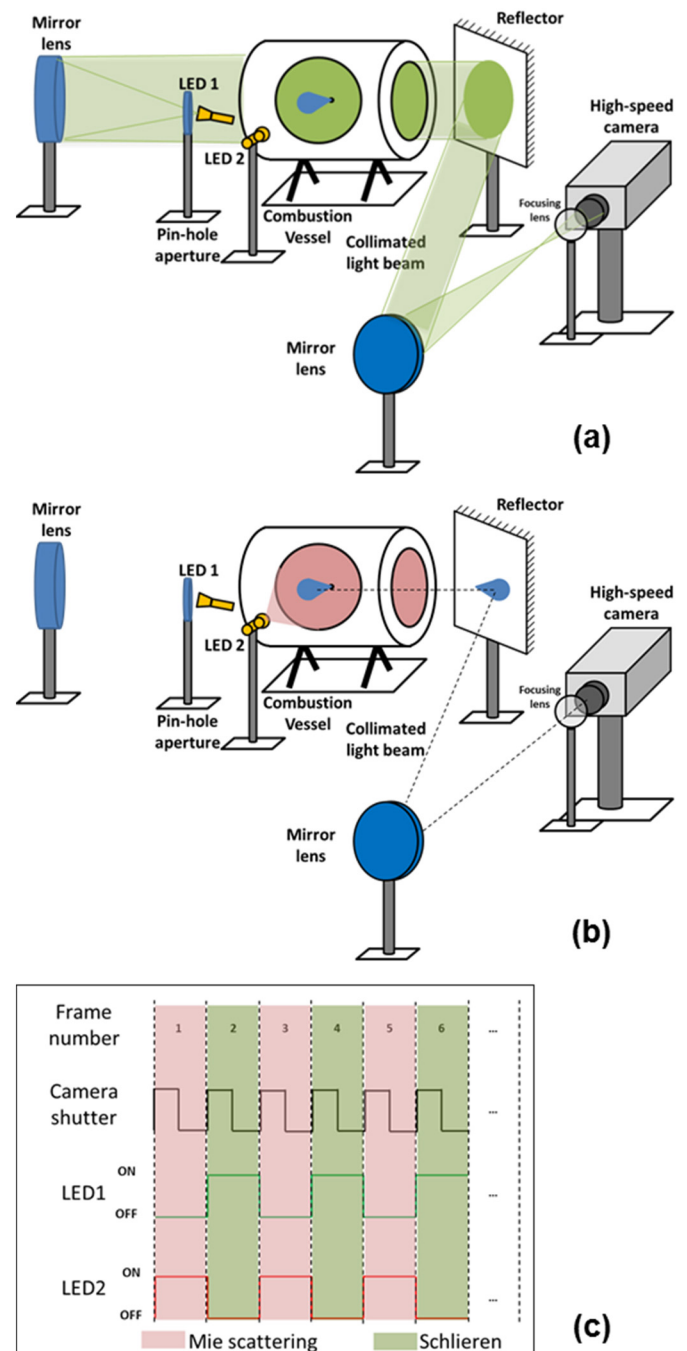


Figure 1. Optical setup of hybrid imaging system: (a) shadowgraph imaging; (b) Mie scattering imaging; (c) operation time table for camera and light sources

A hybrid imaging system similar to Parish and Zink's work [7] has been set up to observe non-combusting spray jets. This visualization setup is based on a modified Z-type shadowgraph system. As shown in Figure 1(a), a high-intensity pulsed LED (LED1) with a pin-hole aperture serves as a point light source and generates an extended parallel light beam with the help of two concave mirrors ($f_1 = 750 \text{ mm}$, 152 mm diameter and $f\#1/5$)