

Experimental and Numerical Analyses of Liquid and Spray Penetration under Heavy-Duty Diesel Engine Conditions

Noud Maes, Nico Dam, and Bart Somers
Eindhoven University of Technology

Tommaso Lucchini and Gianluca D'Errico
Politecnico di Milano

Gilles Hardy
FPT Industrial

ABSTRACT

The modeling of fuel sprays under well-characterized conditions relevant for heavy-duty Diesel engine applications, allows for detailed analyses of individual phenomena aimed at improving emission formation and fuel consumption. However, the complexity of a reacting fuel spray under heavy-duty conditions currently prohibits direct simulation. Using a systematic approach, we extrapolate available spray models to the desired conditions without inclusion of chemical reactions. For validation, experimental techniques are utilized to characterize inert sprays of n-dodecane in a high-pressure, high-temperature (900 K) constant volume vessel with full optical access. The liquid fuel spray is studied using high-speed diffused back-illumination for conditions with different densities (22.8 and 40 kg/m³) and injection pressures (150, 80 and 160 MPa), using a 0.205-mm orifice diameter nozzle. High-speed Schlieren imaging is used to analyze the influence of these boundary conditions on the spray penetration. Simulations of the fuel spray are performed using a dedicated computational mesh with refinements at the known location of the jet to capture the smallest scales of interest. Using a blob injection model refined with a primary atomization and secondary breakup model, correct trends and good agreement are achieved for both liquid and spray penetration. The capability of capturing the trends at largely varying boundary conditions with a single computational approach provides a solid base for future work.

CITATION: Maes, N., Dam, N., Somers, B., Lucchini, T. et al., "Experimental and Numerical Analyses of Liquid and Spray Penetration under Heavy-Duty Diesel Engine Conditions," *SAE Int. J. Fuels Lubr.* 9(1):2016, doi:10.4271/2016-01-0861.

INTRODUCTION

Over the past decades, heavy-duty Diesel engines have proven to be the primary choice of both manufacturers and customers in the most common applications in the transport and agriculture sectors. This preference is among others due to the large operating range with high power and load capability, while maintaining high efficiency. Nevertheless, the flexibility and fuel efficiency of these engines come at the cost of NO_x and/or soot, which has been the focus of many studies in the past [1, 3, 4]. The trade-off between soot and NO_x emissions in Diesel engines is quite well understood and has therefore been the primary subject of modeling efforts as well [5,6]. Most studies identify fuel-injection parameters as instruments in reducing emissions to acceptable levels, often involving empirical parameter studies. Yet, ever more stringent emission legislations and a desired improvement of fuel efficiency necessitate detailed understanding of the fundamental processes encountered in modern direct-injected Diesel engines. True understanding of these fundamental processes holds the potential of breaching currently accepted trade-offs by avoiding or restricting emission sources and heat losses.

The increasing capability of numerically describing fluid dynamics and turbulent combustion under high-temperature and high-pressure circumstances in a detailed manner, creates possibilities to design future engines based on a high-fidelity computational optimization of relevant parameters. To develop such numerical tools, there is a demand for detailed experiments with well-characterized boundary conditions, which allow direct comparison to assess the model performance. Such detailed experimental results are often obtained using dedicated combustion vessels [7, 8, 9]. Combustion vessels are capable of generating relevant, yet simpler boundary conditions while allowing full optical access to investigate a spray with multiple diagnostic techniques in great detail over a long period in time.

In an attempt to combine the strength of experimental and numerical efforts, the Engine Combustion Network (ECN) established a link between different research groups with specific defined test conditions [10]. Using different experimental setups with nominally identical injection equipment, similar results are achieved after due validation of the boundary conditions of the involved combustion vessels within the network [8,9,11,12]. On the numerical side,

different spray and mixing approaches have been considered and used in combination with different combustion chemistry mechanisms, computational grids and turbulent-chemistry interaction models [13,14].

In this work, a similar close coupling between experimental and numerical efforts is used to investigate and characterize inert fuel sprays under heavy-duty Diesel engine conditions. Heavy-duty conditions with densities in excess of 30 kg/m³ at temperatures relevant for internal combustion engines have only been sparsely investigated in the past. Significant contributions date back to a period between 1996 and 2001, when scaling laws for liquid length, spray penetration and flame lift-off length were determined [15, 16, 17, 18], and more recently in a study to characterize the fluctuations in the liquid penetration of evaporating sprays [19]. The highest densities in the investigated test matrix mostly exceeded actual applications at that time. Nowadays engines start to operate in such range. Achieving relevant conditions for modern heavy-duty Diesel engines in an optically accessible setup requires a robust design [7]. In addition to this, the quality of results obtained by optical diagnostic techniques generally decreases with increased density due to severe beam steering and quenching.

We first consider the well-known ECN Spray A boundary conditions to investigate the influence of the difference in orifice size by increasing the diameter to a value which is more typical for heavy-duty injectors. Like advocated in recent studies [20, 21, 22, 23, 24, 25, 26], Diffused Back-Illumination (DBI) is applied in this study to determine the liquid penetration. High-speed Schlieren imaging, which effectively records changes in refractive index is used to determine the spray penetration.

The influence of variation in ambient density and injection pressure on the liquid and jet penetration, obtained by both experimental and numerical work are shown in this paper. Boundary conditions with increased density were implemented and validated using a fast pressure transducer and fine-wire thermocouples as proposed in previous work [8,9,15, 16, 17, 18,27]. Subsequently, we present and compare the results obtained at the new boundary conditions, establishing a solid base for future work where reacting fuel sprays will be investigated.

RESEARCH APPROACH AND BOUNDARY CONDITIONS

Both experimental and numerical approaches are used in this work to characterize inert, evaporating fuel sprays under heavy-duty Diesel engine conditions. The following sections provide detailed information about the experimental equipment, optical diagnostic techniques, numerical setup, and boundary conditions which have been investigated within the scope of this study.

Experimental Setup

Combustion Vessel and Injection Equipment

Experiments were conducted in a high-pressure and high-temperature constant-volume vessel with optical access. The experimental setup is based on a pre-burn type of combustion vessel which achieves precomputed target conditions after the combustion of a reacting gas mixture [7,28]. Using this approach, a wide range in operating conditions covering typical Diesel combustion can be achieved and studied. The vessel has a cubical geometry with a volume of nearly 1300 cc where a single orifice common-rail injector is mounted in the center of a metal side port. Full optical access can be provided on all other faces of the cubical volume. The vessel can be equipped with either quartz or sapphire windows, but only the latter were used throughout the entire campaign. When optical access is not required, metal blanks are generally used to reduce reflections. Diagonal ports of the vessel are equipped with two spark-plugs, an rpm-controlled mixing fan (2000 rpm) which is in operation during the entire experiment, a pressure transducer, and intake and exhaust ports.

Optical Diagnostic Techniques

High-Speed Diffused Back-Illumination Imaging

The liquid penetration was imaged using DBI, which has been proposed as a standard technique for vaporizing diesel sprays by Manin and co-workers [20]. In their investigation, they advocate the technique as a superior method for determining the liquid length (fuel penetration) over the traditional Mie-scattering method. DBI utilizes the extinction of light by the liquid fuel and gives more consistent results between institutes since the images are normalized using a reference image. In this work, the broadband light from a 100-W LED illuminated an engineered diffuser with a 50° square pattern. The light was subsequently directed towards the spray vessel using a Fresnel lens with a focal length of 152 mm as illustrated in the upper panel of Figure 1. A high-speed CMOS camera (Photron SA-Z) with a Sigma 105-mm f/2.8 EX lens was used to collect the transmitted light with a frame rate of 160 kHz and a projected pixel size of 77.5 µm/pixel. The LED was not pulsed, since the exposure time of the camera was sufficiently short (2.5 µs) to prevent motion blur of the liquid fuel.

High-Speed Schlieren Imaging

The jet penetration of inert fuel sprays was imaged using a bright-field Schlieren imaging setup with two bi-convex lenses of 1000-mm focal length. The setup, illustrated in the bottom panel of Figure 1, uses the same camera, lens and light source as the DBI setup. However, the projected pixel size in this case was increased to 0.22 mm/pixel due to the optical arrangement. Previous spray related studies show largely different arrangements to capture the Schlieren effects, mostly based on experience, desired sensitivity and experimental conditions [22,23,29,30]. In this work, a 3-mm circular aperture was placed close to the LED to limit the size of the light source. At the detection side, a 4-mm circular aperture with a sharp edge was placed after the focal point to block the refracted light rays. The sizes and exact locations of the apertures were empirically adjusted to optimize the visual quality of the detected images.

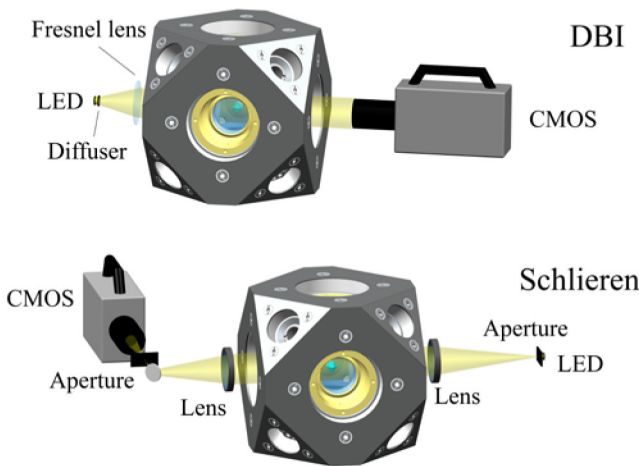


Figure 1. Schematic diagrams of the combustion vessel and the two optical diagnostic techniques used in this work.

Numerical Setup

The Eulerian-Lagrangian approach was employed to model the fuel spray, which is composed of a discrete number of computational *parcels*, each one formed by an arbitrary number of droplets with the same properties. Each parcel evolves in the computational mesh according to the mass, momentum and energy exchange with the continuous gas phase which is treated in an Eulerian way. Additional phenomenological sub-models are then required to describe the various physical processes taking place on the sub-grid length scales: atomization, secondary breakup, drag, evaporation, heat transfer, collision and turbulent dispersion. Fuel-air mixing simulations were carried out by using the Lib-ICE code, which is a set of libraries and solvers for IC engine modeling based on the OpenFOAM® technology. Over the years it was successfully applied to simulation of spray and combustion in direct-injection engines [31, 32, 33]. To describe atomization and secondary breakup, different combinations of sub-models were proposed over the years [34,35], all of them providing reasonably good results both at evaporating and non-evaporating conditions when applied to nozzles with diameters typical of passenger car engines. In Heavy Duty engines, nozzle sizes are much larger and expected liquid penetration lengths are larger. Within this context, a proper choice of sub-models, mesh size and turbulence model is of great importance for a proper prediction of the spray evolution, in particular:

- Spray sub-models (atomization and breakup): they regulate the size of the parcels once they leave the nozzle, depending on aerodynamic and turbulent instabilities. Droplet size and velocity then affect the transfer of mass and momentum from the liquid to the gas phase.
- Mesh size: due to larger spray penetration, Lagrangian and Eulerian phases will interact over a larger portion of the computational mesh. This aspect can affect the so-called *grid-dependency* phenomenon.
- Turbulence model: when the standard *k-ε* model is employed, vapor penetration is generally underestimated. The proposed corrections to model constants generally work well for gas jets or when spray penetration is small. However, the resulting increase of axial turbulent viscosity can negatively affect the

predicted relative velocity when large nozzle configurations are simulated.

Following our previous work [32,34,35], separate models were applied to predict atomization and secondary breakup processes. This is expected to better reproduce the morphology and the evolution of sprays emerging from large nozzles.

Concerning atomization, the Huh-Gosman model was used [36,37]. The liquid jet is represented by primary parcels, which are introduced within the computational mesh with a diameter equal to that of the nozzle. Their velocity is a function of the injected mass flow rate profile. 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:

$$\frac{dD_p}{dt} = -C_{\varepsilon 5} \frac{L_a}{\tau_a} \quad (1)$$

where $C_{\varepsilon 5}$ is the main model constant, L_a and τ_a are the characteristic atomization length and time scales. L_a is proportional to the turbulent length scale of the liquid jet L_t :

$$L_a = C_{\varepsilon 1} L_t \quad (2)$$

while the atomization time-scale is a function of both turbulent and aerodynamic time scales, τ_t and τ_w as follows:

$$\tau_a = C_{\varepsilon 3} \tau_t + C_{\varepsilon 4} \tau_w \quad (3)$$

τ_w is expressed according to the Kelvin-Helmholtz instability theory for an inviscid liquid on an infinite plane:

$$\tau_w = \left[\frac{\rho_l \rho_g}{(\rho_l + \rho_g)^2} \left(\frac{u_{rel}}{L_w} \right)^2 - \frac{\sigma_l}{(\rho_l + \rho_g) L_w^3} \right]^{-0.5} \quad (4)$$

where ρ_l and ρ_g are respectively the liquid and gas densities, σ_l is the surface tension, and u_{rel} the liquid-gas relative velocity. L_w is the wavelength of the perturbations on the jet surface, which is assumed to be related to L_a according to:

$$L_a = C_{\varepsilon 2} L_w \quad (5)$$

The spray cone angle is computed assuming that the spray diverges with a radial velocity L_a/τ_a . Hence the cone angle is computed as:

$$\tan\left(\frac{\alpha}{2}\right) = \frac{L_a/\tau_a}{U_{inj}} \quad (6)$$

Where U_{inj} is the instantaneous jet velocity computed from the injected mass flow rate profile. Once the liquid jet leaves the nozzle its internal turbulence decays with time due to the absence of shear

stresses. Assuming a homogeneous isotropic turbulence inside the nozzle and applying a zero-dimensional version of the k - ϵ model, the turbulence length and time scales evolve in time according to:

$$L_t(t) = L_t^0 \left(1 + C_{a1} \frac{t}{\tau_{t0}} \right)^{C_{a2}} \quad (7)$$

$$\tau_t(t) = \tau_t^0 \left(1 + C_{a1} \frac{t}{\tau_{t0}} \right) \quad (8)$$

where L_t^0 and τ_t^0 are turbulent length and time-scales at the nozzle exit which are estimated in this work from a simplified nozzle flow model [36,37].

$$L_t^0 = C_\mu \frac{k_0^{1.5}}{\epsilon_0} \quad (9)$$

$$\tau_t^0 = C_\mu \frac{k_0}{\epsilon_0} \quad (10)$$

Where $C_\mu = 0.09$ is a constant given in the k - ϵ model. In the above equations, k_0 and ϵ_0 are estimated as follows:

$$k_0 = \frac{U_{inj}^2}{8L/D} \left[\frac{1}{C_d^2} - K_c - (1 - s^2) \right] \quad (11)$$

$$\epsilon_0 = K_\epsilon \frac{U_{inj}^3}{2L} \left[\frac{1}{C_d^2} - K_c - (1 - s^2) \right] \quad (12)$$

Where C_d is the discharge coefficient, K_ϵ is a model constant, K_c and s are the form loss coefficient and the area ratio at the contraction corner, which depends on the nozzle design. L and D are the nozzle hole length and diameter, respectively. C_{a1} and C_{a2} are additional model constants describing the turbulence evolution on the liquid jet once it leaves the nozzle. As a consequence of the parent droplet diameter reduction, new droplets are created. The total amount of mass stripped from each parent droplet is computed as:

$$m_s = \rho_l n_p \frac{\pi}{6} (D_{old}^3 - D_{new}^3) \quad (13)$$

where n_p is the total number of parent droplets constituting the liquid core. New parcels representing these stripped droplets are created when the total stripped mass is higher than a specific threshold, set to be equal to 0.1 times the original parent parcel mass, following [36]. Secondary droplet diameter is computed from a PDF distribution which was created with the following assumptions:

- The number of droplets of size x is inversely proportional to the atomization time-scale τ_a ;
- The number of droplets of size x is proportional to the jet-todroplet surface area-ratio D_d^2/x ;

- The number of droplets of size x is proportional to the is proportional to the dimensionless turbulent energy spectrum [38];
- The number of size in the range $(x, x + dx)$ is proportional to the unit size interval, dx/x .

The maximum diameter of the droplet size distribution is assumed to be the parent parcel one while the minimum diameter is computed as:

$$d_{min} = 2\pi \frac{(\rho_l + \rho_g)\sigma_L}{\rho_l \rho_g u^2} \quad (14)$$

Secondary droplets inherit velocity from their parents, and the atomization process is supposed to cease as soon as one of the following conditions is satisfied: parent droplet diameter lower than the atomization length scale, Weber number (We) lower than 40 or Ohnesorge number (Oh) greater than 2 [39]. Parent droplets are not subject to drag, evaporation and heat transfer.

To better understand the effects of the atomization process on the spray evolution, two different approaches were used to initialize the velocity direction of the parent and secondary droplets. In the first one, the direction of the primary parcels is randomly chosen between 0 and initial spray angle α , computed at the nozzle exit; secondary droplets then inherit velocity from their parents when they are stripped. In the second approach, primary parcels only have the axial component. At the time the stripping process takes place, secondary droplets are deflected with a radial velocity $v_r = L_d/\tau_a$, which takes into account both turbulence at the nozzle exit and its progressive decay when traveling downstream. This is expected to better predict the spray morphology resulting from atomization.

The PDF distribution resulting from the previous assumptions mainly relates the size of the secondary droplets to the size of the eddies generated by the turbulence on the jet surface, with computed diameters ranging between 0.01 to 0.2 times the original jet diameter. The Huh-Gosman model capability to account for liquid jet turbulence on the atomization process makes it very suitable for coupled Lagrangian/nozzle-flow simulations whose predictive capabilities for spray modeling are expected to be very high [40, 41, 42].

The model originally proposed by Pilch and Erdman was applied in this work to predict the secondary breakup process [43]. According to their approach, there is a maximum stable diameter D_s below which breakup does not take place. The value of D_s is affected in two ways: (i) the decrease of droplet Weber number because of the new smaller droplet diameter and (ii) the decrease in relative velocity between the droplets and the flow-field, due to the changes in droplet acceleration (as a results of the decreased droplet diameter). The droplet breakup occurs if the decrease in Weber number is greater than a critical value We_c , accounting for the viscous effects parameterized by the Ohnesorge number as follows:

$$We_c = 6(1 + 1.0770h^{1.6})$$

The Pilch-Erdman model distinguishes between five breakup regimes: each one of them is characterized by dimensionless total breakup time $\bar{\tau}_b$ as follows as illustrated in [Table 1](#).

Table 1. Dimensionless breakup time values for different Weber number ranges as suggested by the Pilch-Erdman breakup model.

Type	Breakup time $\bar{\tau}_b$	Weber number range
Vibrational breakup	$6(2We - 12)^{-0.25}$	$6 < We \leq 9$
Bag breakup	$2.45(2We - 12)^{0.25}$	$9 < We \leq 22$
Bag-and-stamen breakup	$14.1(2We - 12)^{-0.25}$	$22 < We \leq 75$
Sheet stripping	$0.766(2We - 12)^{0.25}$	$175 < We \leq 1335$
Wave crest stripping	5.5	$1335 < We$

The dimensionless breakup time $\bar{\tau}_b$ depends on several parameters: liquid and gas densities (ρ_l and ρ_g), droplet diameter D_d and the magnitude of the liquid-gas relative velocity u_{rel} :

$$\bar{\tau}_b = \tau_b \frac{u_{rel}}{D_d} \left(\frac{\rho_g}{\rho_l} \right)^{1/2} \quad (16)$$

The stable diameter is evaluated from:

$$D_s = 2We_c \frac{\sigma_l}{\rho_g u_{rel}^2 \left(1 - \frac{V_d}{u_{rel}} \right)^{-2}} \quad (17)$$

where the droplet volume V_d is given by:

$$V_d = u_{rel} \left(\frac{\rho_g}{\rho_l} \right)^{1/2} (B_1 \bar{\tau}_b + B_2 \bar{\tau}_b^2) \quad (18)$$

Coefficients B_1 and B_2 have to be tuned to fit experimental data. The change in the diameter of the secondary diameter droplets becomes:

$$\frac{dD_d}{dt} = \frac{D_d - D_s}{\tau_b} \quad (19)$$

For what concerns the other sub-models employed, heat transfer was predicted using the Ranz-Marshall correlation, evaporation rate is computed from Spalding number. Collision is neglected since it does not significantly affect the spray behavior in evaporating sprays.

Boundary Conditions

The boundary conditions in this work were carefully selected such that both a connection to the existing ECN database and expansion to heavy-duty applications are possible. Furthermore, the effect of single parameters can be studied and compared to literature using the

proposed approach. As a single component reference fuel, ndodecane (C12H26) is used, because of its physical properties and availability of detailed reaction mechanisms. Furthermore, ndodecane is fluorescence-free and compatible with high-pressure fuel pumping systems. Similar to previous work, ECN Spray A conditions are applied with an ambient temperature of 900 K, an ambient density of 22.8 kg/m³ and a fuel injection pressure of 150 MPa [9,22]. In the present work, however, the injector has an orifice diameter size of 0.205 mm, compared to the 0.09-mm ECN standard. The used single-hole injector has an orifice which is nominally identical to those in heavy-duty nozzles with multiple holes which are currently in production. The injector has a converging hole (k-factor 1.5) and was subjected to hydro-erosive grinding to suppress cavitation as well as possible [44]. Preliminary simulations with a multiphase flow solver including cavitation modeling were performed to investigate the flow in nozzles which are similar to the ones used in this work. Using the approach suggested in [45] and considering operating conditions close to the studied cases, no cavitation was found. Furthermore, in this work, emphasis is laid upon numerically reproducing the liquid and vapor distribution obtained from experiments. Cavitation effects, if any, are obviated by tuning the models to fit experiments, which warrants realistic mixture fracture distributions for combustion studies. The abbreviation ANR refers to the non-reacting injections at ECN Spray A conditions with the large orifice injector. The heavy-duty cases feature similar boundary conditions but with a density of 40 kg/m³ and using two different injection pressures (C1 and C2). Variations in injected fuel mass were matched with realistic operating points for heavy-duty applications. Due to a mechanical failure of the fuel injector, it had to be replaced during the experimental campaign with a nominally identical piece. Because of this replacement, the ndodecane in the fuel system was also refreshed, which revealed that the old fuel had changed color over time. Liquid and spray penetration measurements were compared to the old injector for one of the boundary conditions, in order to check consistency with the preceding experiments. To distinguish this validation condition from the results with the original injector, it will be referred to as C1v. A summary of the conditions applied in this study are listed in [Table 2](#), where injector number 2 refers to the replacement piece.

Table 2. Details of the inert boundary conditions which have been investigated using an injector with an orifice diameter of 0.205 mm. Experiments with the second injector at the condition called "FPT C1" are referred to as C1v throughout the paper, which indicates the validation of the C1 case. The Spray A condition is often indicated with ANR, to indicate the non-reacting environment.

	Spray A (ANR)	FPT C1	FPT C2
Injector numbers	1	1, 2 (C1v)	2
Ambient temperature [K]	900	900	900
Ambient density [kg/m ³]	22.8	40	40
Approximate pressure [MPa]	6	10.5	10.5
Fuel injection pressure [MPa]	150	80	160
Injection duration [ms]	4.8	5.2	4.9
Injected fuel mass [mg]	60.8	46.8	64.8

RESULTS AND DISCUSSION

First, the methods for image processing and evaluating experimental parameters is discussed. Characteristic parameters are compared with available literature and influences on the results are evaluated. Then the numerical models are assessed and the determination of parameters which can be related to experiments are defined and explained. Finally, penetration and morphology of liquid- and evaporated fuel sprays obtained from experiments and calculations are compared against each other, and improvements and interests for future studies are identified.

Experimental Liquid Penetration

Following the work of Manin, we will interpret the DBI images in terms of extinction, noting that both absorption and (large-angle) scattering with contribute to this. To obtain a quantitative value of the optical thickness τ^* , the images I_i acquired using the high-speed DBI setup (index i labels the frames) are processed using the background image I_{bg} before the start of injection according to

$$\tau^* = -\ln\left(\frac{I_i}{I_{bg}}\right). \quad (20)$$

A time sequence shortly after the Start Of Injection (SOI) is presented in the left panel of Figure 2. This figure illustrates how the penetration of liquid fuel can be identified by a sharp gradient in the detected extinction. This is the case until the liquid jet approaches its maximum length at around 0.15 ms after SOI for this condition (C1). As soon as this happens, large fluctuations in liquid penetration are observed due to the detachment of ligaments, as shown in the right panel of this figure. Research dedicated to these fluctuations at similar boundary conditions, but with slightly lower ambient density (34.8 kg/m³) and injector orifice diameter (0.145 mm), reported fluctuations on the order of 10% with respect to the mean liquid length, using the Mie-scattering technique [19]. The main reason for these fluctuations was identified as the spray-induced turbulent eddies at the periphery of the sprays, which was substantiated using large-eddy simulations. Fluctuations in the order of 10% can easily be distinguished in the image sequence of Figure 2. Also, this sequence shows that the detaching ligaments indeed seem to have a size in the same order of magnitude as the large scale eddies observed at the periphery of the main liquid core. Two of such detaching ligaments are indicated with brackets in Figure 2.

Despite large fluctuations at the end of a non-interrupted liquid core, the spray is often referred to as quasi-steady, due to a relatively constant mean length, traditionally called the Liquid Length (LL). To quantify this LL, results are generally time-averaged during the quasi-steady phase. The resulting average optical thickness along the spray axis is seen to decline steeply at the end of the liquid jet, as illustrated in Figure 3. As proposed by Manin and co-workers, the interception of a linear fit through this steep decline with the abscissa (where the optical thickness is zero), is defined as the quasi-steady LL. Using such an approach results in a robust parameter which can be used to validate models. For the determination of the transient liquid penetration a fixed optical thickness threshold is generally

used, since the fluctuations in instantaneous images prevent the use of a fitting procedure. However, the derived length is then depending on the optical depth of the liquid fuel spray and sensitive to the errors induced by optical setup and technique, as discussed in previous studies [20,25,26,46].

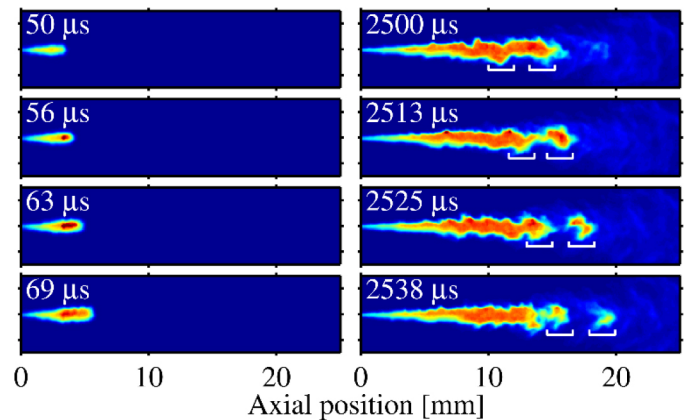


Figure 2. Two time sequences in a false color-scale from a single FPT C1 experiment: $\rho = 40 \text{ kg/m}^3$, $T = 900 \text{ K}$, injection pressure = 80 MPa. The right sequence shows an example of the fluctuations encountered during the quasi-steady period of the injection. The white brackets indicate two large eddies which eventually detach.

Qualitatively, all different optical thickness profiles in Figure 3 show a similar behavior. However, this figure reveals how the results obtained with fresh fuel and a new injector result in a reduction of the detected optical thickness value by nearly 50%. Maximum values in literature, obtained at Spray A boundary conditions with an injector having an orifice diameter of 0.09 mm, range between 2.5 and 3.0 [20, 21, 22]. With more than twice the injector orifice diameter in the current work, and therefore more than twice the path length through the liquid volume, one should expect higher values for the optical thickness. This is not the case for the new injector and fuel supply. This decrease is most probably due to the increased transparency of the new fuel. Note that the fitting procedure used to determine the LL is relatively insensitive to small variations in the absolute optical thickness.

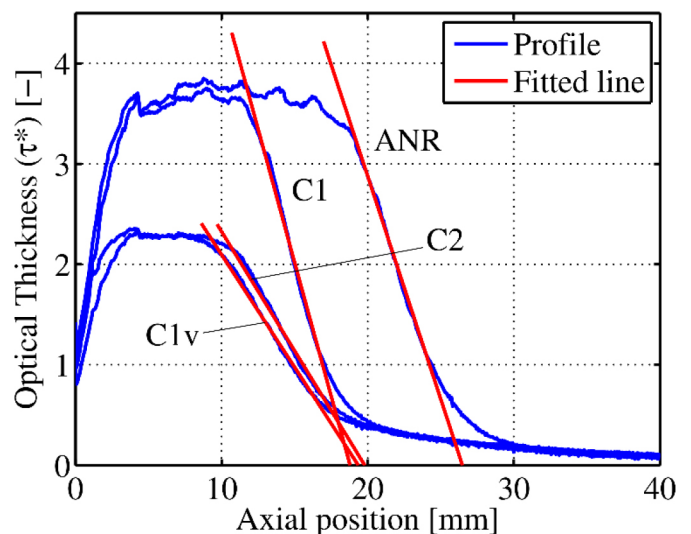


Figure 3. Average profiles of the optical thickness along the injector spray axis for the conditions listed in Table 2. The fitted lines indicate the procedure to determine the quasi-steady liquid length.