

Experimental Characterization of High-Pressure Impinging Sprays for CFD Modeling of GDI Engines

2011-01-0685

Published
04/12/2011

Alessandro Montanaro and Luigi Allocca
Istituto Motori CNR

Daniele Ettorre and Tommaso Lucchini
Politecnico di Milano

Federico Brusiani and Giulio Cazzoli
Università degli Studi di Bologna

Copyright © 2011 SAE International

doi:10.4271/2011-01-0685

ABSTRACT

Today, Direct-Injection systems are widely used on Spark-Ignition engines in combination with turbo-charging to reduce the fuel-consumption and the knock risks. In particular, the spread of Gasoline Direct Injection (GDI) systems is mainly related to the use of new generations of multi-hole, high-pressure injectors whose characteristics are quite different with respect to the hollow-cone, low-pressure injectors adopted in the last decade.

This paper presents the results of an experimental campaign conducted on the spray produced by a GDI six-holes injector into a constant volume vessel with optical access. The vessel was filled with air at atmospheric pressure. Different operating conditions were considered for an injection pressure ranging from 3 to 20 MPa. For each operating condition, spray images were acquired by a CCD camera and then post processed to evaluate the spray penetration and cone angles. A flat plate was placed inside the vessel to investigate the spray-wall impingement and liquid-film evolution.

The experimental database was used to obtain a CFD methodology defined to simulate the GDI spray behavior. This methodology was based on a classic Eulerian-Lagrangian approach and it was implemented into the Lib-ICE code, developed into the OpenFOAM platform. In

particular, specific sub-models were implemented to correctly reproduce the fuel atomization and the droplet-wall interaction.

INTRODUCTION

From 2014, EURO 6 emission standards will impose a contemporary control of the fuel consumption and pollutant emissions for IC engines. To fulfill such requirements, different solutions are investigated such as new combustion modes, together with a significant improvement of the existing technologies. In the field of Si-engines, attention is mainly focused on Direct Injection (GDI) because it allows to improve the overall engine thermal efficiency and the engine response during load variations. A significant reduction of the knock tendency is also ensured and this is a relevant aspect when turbocharging is used [1, 2]. GDI technology allows the reduction of HC and CO emissions especially at cold start (warm-up period) and transient modes when the engine works at lean-mixture conditions. To take advantage of all the theoretical benefits derived from GDI applications, it is necessary to work with a large stratification region in the engine map and this requires a strong control of the air/fuel mixing process.

The most widely used GDI injector configuration is the multi-hole mini-sac injector which is used to deliver fuel during the intake and compression strokes, with pressures ranging from 1 to 20 MPa [3]. The fuel/air mixture

distribution is the result of complex interactions between the emerging fuel spray, in-cylinder flow field and the combustion chamber geometry [4]. Hence, experimental and numerical investigations are needed to evaluate how the spray evolution is influenced by:

- *Nozzle design and spray targeting* [5]. Optimization of the nozzle design allows an improvement of the transition from the rail pressure to the momentum of the jets outgoing from the injector holes;
- *Injection pressure* [6];
- *Interaction between spray and cylinder/piston walls* [7]. As well known, generally the wall wetting has to be avoided inside a GDI engine because it has a strong influence on the mixture formation and emission quantities, therefore on the engine cycle-to-cycle variation.

In particular, multi-dimensional models are widely applied to real engine geometries to define suitable injector configurations and strategies for a wide range of operating conditions. The well-known Eulerian-Lagrangian approach is used to model the fuel spray, which is composed by a discrete set of computational parcels and their evolution depends on the exchange of mass, momentum and energy with the continuous gas phase. Additional sub-models are also required to take into account the liquid jet atomization, droplet evaporation, heat transfer, breakup, collision and wall impingement [8]. However, the application of CFD models to current GDI sprays still requires modeling efforts for a correct prediction of the following phenomena:

- liquid fuel atomization,
- interaction between sprays emerging from different injector holes,
- wall impingement.

The fuel atomization process is strongly affected by the injection pressure, the nozzle hole design and the formation of cavitating bubbles inside the nozzle hole. When cavitation takes place, reduction of the droplet diameter and increase in the spray cone angle should be correctly estimated. Furthermore, the nozzle hole positioning can be responsible for a strong interaction among the different fuel jets. In this case, the fuel-mixing process is strongly influenced by droplet collision and coalescence. Finally, the fuel might impinge on the piston surface and the effects of the resulting liquid film evolution have to be predicted. The spatial and temporal characteristics of the fuel spray, in terms of atomization and distribution in the chamber, the spray structure and the spray/gas and spray/wall interactions, play an important role because of the short time available for the fuel-air mixing process and can significantly influence mixture formation, combustion and pollutant emissions. All that has contributed to an increasing requirement of measurement techniques/procedures for the analysis of the

spray and computational approach via powerful predictive codes. Optical techniques appear to satisfy the required temporal and spatial evolution, in non-intrusive mode operation, gathering images of free evolving and wall impacting sprays and extracting parameters of interest by digital processing procedures.

In this work, experiments and calculations were performed to understand the behavior of a gasoline spray emerging from a six-hole injector mounted on the top of an optically accessible vessel. This investigation was performed to provide a reliable CFD methodology to simulate these jets. Different injection pressures were considered, ranging from 3 to 20 MPa and the injected mass flow rate was measured for each operating conditions. A CCD camera was used to acquire the spray images that were then digitally processed to measure the spray penetration and cone angles. Impinging conditions were also reproduced in the experiments by introducing a plate in the vessel orthogonally oriented with respect to a reference jet.

The proposed CFD approach was implemented by the authors into the Lib-ICE code [9, 10], which is a set of libraries and applications for engine modeling developed under the OpenFOAM® technology [11]. Specific modules are available for the simulation in in-cylinder processes (Diesel and HCCI combustion, motion of piston and valves, complex chemistry with adaptive tabulation, gas and particulate emissions, spray break-up and spray-wall interaction). In previous works, Lib-ICE was applied to simulate the fuel-air mixing process in Diesel and PFI engines [12, 13]. A simplified 1-D model of the nozzle flow was used to estimate the influence of cavitation on the liquid jet diameter and velocity while the primary and secondary liquid jet break-up processes were modeled according to the Kelvin-Helmoltz theory. A specific spray sub-model describes the interaction between the fuel spray and the cylinder walls under different impinging conditions and the evolution of the resulting liquid film is also taken into account by solving its mass, momentum and energy equations on the mesh boundary.

CFD calculations were carried out both to validate the proposed approach and also to provide new suggestions for future experimental investigations. Even if a good agreement between computed and experimental data was found in terms of spray penetration and droplet distribution, further investigations are necessary to better evaluate the effects of the interaction among the different sprays.

EXPERIMENTAL APPARATUS

To investigate the behavior of a high-pressure GDI injector under different working conditions, a specific test-rig with optical access was built and it is illustrated in [Figure 1](#). The GDI injector considered in the present work was the mini-sac six-holes Bosch HDEV 5.1 with solenoid actuation. This

injector is characterized by a hole diameter of 0.193 mm and a range of application between 5.13 and 17.1 g/s of delivered fuel for an injection pressure of 10 MPa. On a plane perpendicular to the injector axis, the six jet directions give a spray footprint characterized by a hollow-ellipsoid shape.

To avoid interference and contamination with the surrounding ambient, the fuel was discharged from the injector into a quiescent vessel (not shown in Figure) filled with air kept at atmospheric pressure (0.1 MPa) and ambient temperature. The high pressure for the injections was generated by a hydro-pneumatic pump, instead of the classic rotating pump, activated by pressured gas (air) ranging from 0.07 MPa to 0.7 MPa. It produced in output a linear injection pressure of the fuel ranging from 2.5 to 25 MPa. To dump the pressure oscillations, due to the fuel delivering and its recharging, a tank of 1.0 dm³ was located between the injection pump exit and the GDI injector (see Figure 1) The tank, such as the injector holder and the connecting adaptor, were filled with gasoline and kept at injection-like pressure. To collect the injection pressure during the experimental tests, a piezo-resistive pressure transducer was placed along the connection between pump and the tank. Moreover, the injection pressure oscillations were monitored by placing a piezo-quartz transducer just before the injector connection. The injector points downstream in Figure 1; it was driven by a Programmable Electronic Control Unit (PECU) able to manage multi-injection strategies with the needle opening time defined through the injector energizing period. The injection flow rate was measured by an AVL Meter operating on the Bosch principle [14,15]. During the experimental tests the injection pressure range from 3 MPa to 20 MPa was considered and it corresponded to a delivered fuel quantity ranging between 10 and 100 mg/stroke.

Figure 2 shows the relation between the solenoid energizing current profile and the corresponding fuel injection rate measured by the AVL Meter. In this case, for an injection pressure of 10 MPa and an energizing time of 1450 μ s a delay of 400 μ s was observed between the electronic Start of Injection (SOI) instant and the first appearance of gasoline droplets into the vessel.

During the experiments, pictures of the emerging spray were taken using a synchronized CCD camera characterized by 1376×1040 pixels, 12 bit resolution, and 0.5 μ s shutter time. A wet seal spherical holder enabled the tilting of the injector in an angular range of $\pm 15^\circ$, with respect to its axis. It allowed the perpendicular alignment between the direction of one spray and the CCD optical axis, therefore the parallax error was reduced. The spray pictures were taken at different instants starting from SOI and they were post-processed by software to extract the main parameters characterizing the dynamic of the spray: cone angle and liquid penetration. In particular, these quantities were measured only on one of the

six jets emerging from the injector. At this scope the jet characterized by the smallest interactions with the other jets was chosen. More details about image post-processing procedure can be found in [16].

Finally, a stainless steel flat plate was introduced into the vessel to reproduce the spray-wall impingement. The plate was located at 40mm from the nozzle tip thanks to a X-Y- Φ micrometric apparatus. The plate axis was parallel the axis of the less interacting jet, fully framed by the CCD camera. The average roughness of the impact-wall was 1.077 μ m, measured by the Stylus Profilometer, Model Surtronic 3 by Rank Taylor Hobson. The plate was at same ambient temperature of the gas filling the vessel. The authors decided to perform also spray impact analysis because, as well known, for GDI engines the wall film liquid formation on the combustion chamber walls during the compression strokes could be a very significant problem because it has a strong influence on the air/fuel mixing process, therefore on the combustion performance and pollutant formation. For this reason a correct prediction of wall film (generation and evolution) is of great importance for GDI engine simulations. For all the experimental tests, a standard (commercial) gasoline was used as fuel.

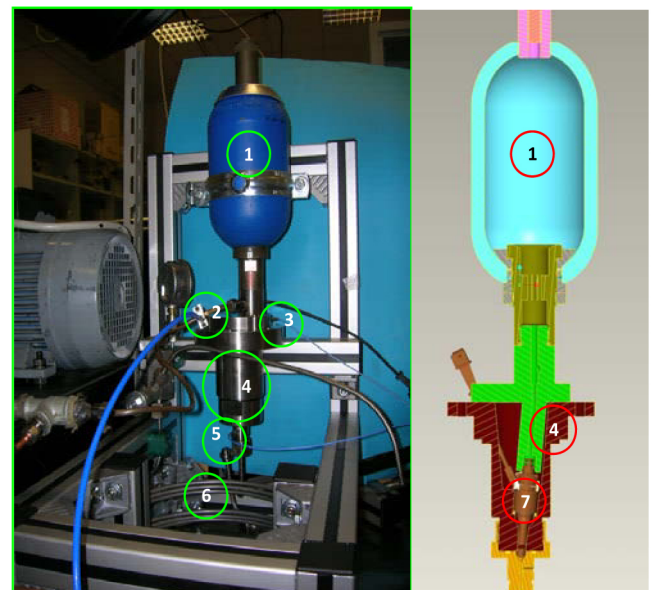


Figure 1. Injection set-up: 1) Fuel reservoir; 2) Piezoresistive pressure transducer; 3) Quartz pressure transducer; 4) Injector adapter; 5) Gauge meter quartz pressure transducer; 6) Gauge meter pipeline; 7) Injector.

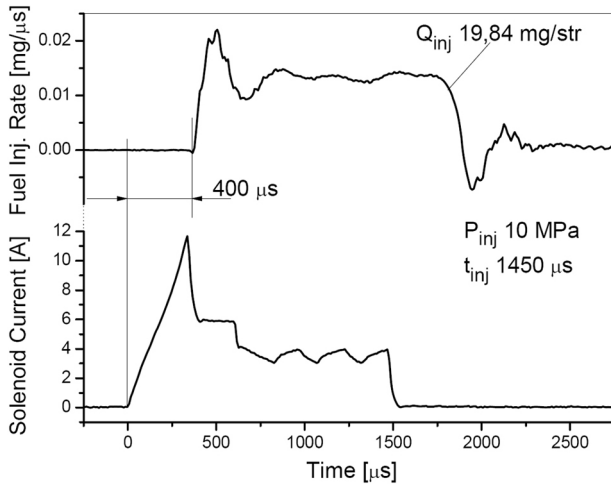


Figure 2. Solenoid energizing current and fuel injection rate.

NUMERICAL MODELS

As well known, the modeling requirements that have to be satisfied to perform a spray simulation are quite high because during the injection the jets outgoing the injector nozzles are simultaneously subjected to many phenomena: primary atomization, secondary break-up, collision/coalescence and wall-impingement. In the following sections, the spray sub-models implemented in the Lib-ICE code are presented.

BREAKUP MODEL

When the Lagrangian approach is used to simulate the spray, the liquid emerging from the nozzle is represented as a train of “blob” (parent) parcels whose diameter is progressively reduced by the growth of aerodynamic instabilities generated on the liquid surface.

To define the droplet velocity components at nozzle exits, the Huh-Gosman [18] theory was considered. A quasi-dimensional turbulence model was used to compute the turbulent kinetic energy (k_{d0}) and dissipation rate (ε_{d0}) of the emerging liquid jet, which were initialized according to the flow conditions inside the nozzle:

$$k_{d0} = \frac{U_D^2}{8L_{nozz}/D_{nozz}} \left[\frac{1}{C_d^2} - K_c - (1-s^2) \right] \quad (1)$$

$$\varepsilon_{d0} = K_\varepsilon \frac{U_D^3}{2L_{nozz}} \left[\frac{1}{C_d^2} - K_c - (1-s^2) \right] \quad (2)$$

where C_d is the discharge coefficient and $K_c - K_\varepsilon$ are two model constants respectively set to 0.47 and 0.23. L_{nozz} and D_{nozz} are the geometric length and diameter of the injector

nozzle. U_d is the droplet velocity at nozzle exit and s the area ratio at the contraction corner.

From k_{d0} and ε_{d0} it was possible to estimate the turbulent length and time scales L_{t0} and τ_{t0} at the time the blob leaves the nozzle:

$$L_{t0} = C_\mu \frac{k_{d0}^{1.5}}{\varepsilon_{d0}}, \tau_{t0} = C_\mu \frac{k_{d0}}{\varepsilon_{d0}} \quad (3)$$

C_μ is a constant whose value is equal to 0.09. The spray cone angle, α , is then estimated as:

$$\tan\left(\frac{\alpha}{2}\right) = \frac{C_1 \cdot L_{t0}}{C_3 \cdot \tau_{t0}} \quad (4)$$

Where the values of the model constants C_1 and C_3 were set to 2.0 and 1.2 respectively [18].

Since the injection pressure in GDI engines usually ranges from 5 MPa to 20 MPa, the resulting droplet Weber number at the nozzle exit is generally lower than 1000, which was fixed as threshold between turbulence-induced and aerodynamic-induced breakup [12, 19]. For this reason, the Kelvin-Helmoltz instability was chosen to be the dominant mechanism governing both the primary and secondary droplet break-up [20]. The diameter reduction of the injected blobs was computed as:

$$\frac{dD_d}{dt} = -\frac{D_d - D_{new}}{\tau_{BU}} \quad (5)$$

where D_d is the current droplet diameter, D_{new} is the diameter of the child drops and τ_{BU} is the break-up time. These values were computed according to the most unstable growth rate Ω and wavelength Λ :

$$D_{new} = 1.22 \cdot \Lambda \quad (6)$$

$$\tau_{BU} = 3.788 \cdot B_1 \cdot \frac{D_d}{2 \cdot \Lambda \cdot \Omega} \quad (7)$$

where B_1 is a tuning constant, which was set to 6 in this work. The reader is referred to [20, 21] for further details about the Kelvin-Helmoltz breakup model and the calculation of the wavelength and the growth rate.

As well known, high pressure GDI injectors are generally affected by cavitation. In a previous work, the authors studied the cavitation phenomena into a six-hole GDI injector very similar to that considered in the present analysis [22]. Figure 3 shows an example of the liquid/vapor distribution obtained at maximum lift for an injection pressure of 10 MPa. Because of cavitation development inside the injector holes, it was necessary to correct the initial droplet diameter D_d and, therefore, their initial velocity U_d . To do that, a quasi dimensional model, based on potential flow assumptions was adopted.

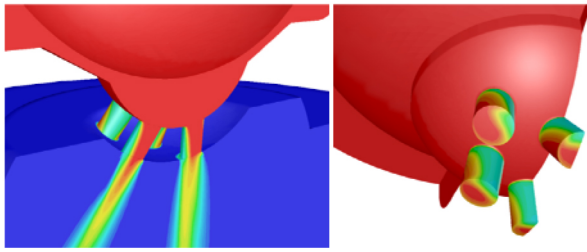


Figure 3. Liquid-Vapor distribution inside a GDI six-holes injector (Red color = 100% Liquid phase, Blue color = 100% gas phase). The computational mesh includes both for the injector geometry and the ambient gas volume.

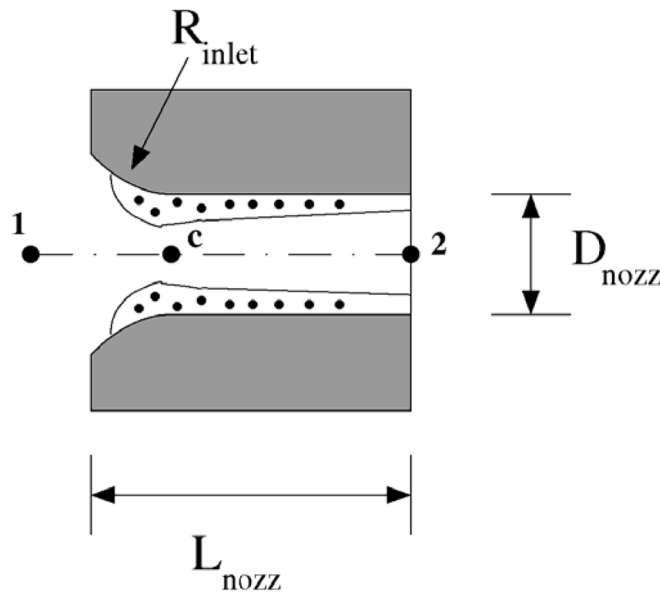


Figure 4. 1-D nozzle flow model used to correct the droplet diameter and velocity in case of cavitation. D_{nozz} : geometrical nozzle hole diameter; L_{nozz} : nozzle hole length; R_{inlet} : nozzle inlet radius.

Figure 4 illustrates that under cavitating conditions the jet diameter at the nozzle exit is lower than the effective nozzle diameter D_{nozz} . Assuming a flat velocity profile and applying

the continuity equation, the velocity in the smallest flow area (point c in Figure 3) can be estimated from the theoretical velocity U_{theo} and the area contraction coefficient C_c :

$$U_c = \frac{U_{theo}}{C_c} \quad (8)$$

where U_{theo} was computed from the injected mass flow rate, liquid density ρ_l and the nozzle diameter as follows:

$$U_{theo} = \frac{\dot{m}_{fuel}(t)}{\rho_l \pi \frac{D_{nozz}^2}{4}} \quad (9)$$

The Nurick's expression [17] was used to estimate the area contraction coefficient C_c as a function of the nozzle inlet radius (R_{inlet}) and nozzle diameter:

$$C_c = \frac{1.0}{\sqrt{\left(\frac{1}{0.61}\right)^2 - 11.4 \frac{R_{inlet}}{D_{nozz}}}} \quad (10)$$

In case of cavitation, the potential flow theory allows the application of the Bernoulli equation from point 1 to point c without any losses:

$$p_c = p_1 - \rho_l \frac{U_c^2}{2} \quad (11)$$

where p_1 was estimated to be the injection pressure for the considered operating point. If p_c was lower than the fuel vapor pressure p_v at the injection temperature, it was assumed that the flow must be fully cavitating and a new inlet pressure and discharge coefficient C_d were calculated by:

$$p_1 = p_v - \rho_l \frac{U_c^2}{2} \quad (12)$$

$$C_d = C_c \sqrt{K} = C_c \cdot \sqrt{\frac{p_1 - p_v}{p_1 - p_2}} \quad (13)$$

where p_2 was the ambient pressure. When cavitation occurs, the effective conditions at the nozzle exit were now calculated as:

$$U_d = U_c - \frac{p_2 - p_v}{\rho_L U_{theo}} \quad (14)$$

$$d_d = \pi \cdot \frac{D_{nozz}^2}{4} \cdot \frac{U_{theo}}{U_{eff}} \quad (15)$$

and they were used to initialize the droplet diameter and velocity. When no cavitation occurred, such values were set to U_{theo} and D_{nozz} , respectively.

COLLISION MODEL

In GDI engines, the spray angle and patterns can promote a strong interaction between the droplets emerging from different injectors holes. When this happens, a competition between drop break-up and coalescence phenomena determines the drop size, velocity and trajectory. To model collision in this work, the Nordin approach was adopted [8]. It states that collision between two parcels occurs only if, in the same integration step, their trajectories intersect and the intersection point is reached at the same time. This condition is fulfilled only when two parcels are travelling towards each other and their relative distance is greater than their size. The model accounts for both stretching separation and coalescence [23]. The Nordin model presents a reduced grid-dependency and this aspect is rather important when spray is simulated in unstructured and deforming meshes [8].

WALL-IMPINGEMENT MODEL

The interaction between the spray parcels and the liquid film is modeled with the approach proposed by Stanton et al. [24]. Generally, the collision between a drop and a wet surface may result in four different regimes: sticking, spreading, rebounding and splashing (see Figure 5) [25].

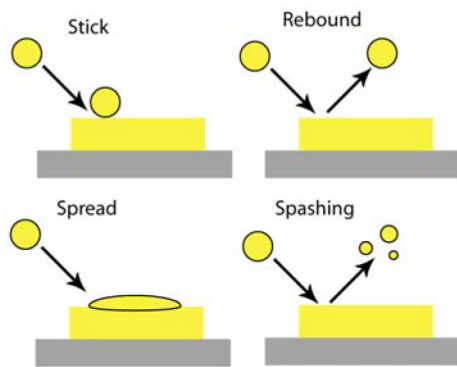


Figure 5. Regimes about collision of a droplet on a wet wall

The impingement regimes are identified by the Weber number We defined as:

$$We = \frac{\rho \cdot (\mathbf{V}_P \cdot \mathbf{n}_w) \cdot d_0}{\sigma} \quad (16)$$

Where $\mathbf{V}_P$ is the relative velocity between the droplet and the wall, $\mathbf{n}_w$ is the face normal of the impinging wall, d_0 the droplet diameter and σ is the surface tension. The following transition criteria were used:

- $We < 5$: stick
- $5 < We < 10$: rebound
- $10 < We < We_s$: spread
- $We > We_s$: splash

We_s represents the splashing threshold and is given by [26, 27]:

$$We_s = (18)^2 d_0 \left(\frac{\rho}{\sigma} \right)^{1/3} \left(\frac{\mu}{\rho} \right)^{1/4} f^{3/4} \quad (17)$$

where ρ , σ , μ , d_0 and f are the drop density, surface tension, dynamic viscosity, incident drop diameter, and frequency of drops impinging on the film. The calculation of the impact frequency for the first drop is ambiguous and it was set as:

$$f = (\mathbf{V}_P \cdot \mathbf{n}_w) / d_0 \quad (18)$$

For all the other droplets, the impact frequency is calculated in each boundary face as:

$$f = N_{impact} / t_{impact} \quad (19)$$

where N_{impact} is the total number of droplets impinging on the face and t_{impact} is the time elapsed from the first impact on the wall boundary face. In this way, the impact frequency is updated at each time-step. The value of N_{impact} is updated for every impinging parcel as follows:

$$N_{impact,new} = N_{impact,old} + N_d \quad (20)$$

where N_d is the total number of droplets contained in the impinging parcel.

The splashing process is modeled by introducing three new parcels in the mesh for each impinging droplet. A detailed