

Full-Cycle CFD Modeling of Air/Fuel Mixing Process in an Optically Accessible GDI Engine

Tommaso Lucchini, Marco Fiocco, and Angelo Onorati
Politecnico di Milano

Alessandro Montanaro, Luigi Allocca, Paolo Sementa, Bianca Maria Vaglieco, and
Francesco Catapano
Istituto Motori CNR

ABSTRACT

This paper is focused on the development and application of a CFD methodology that can be applied to predict the fuel-air mixing process in stratified charge, sparkignition engines. The Eulerian-Lagrangian approach was used to model the spray evolution together with a liquid film model that properly takes into account its effects on the fuel-air mixing process into account. However, numerical simulation of stratified combustion in SI engines is a very challenging task for CFD modeling, due to the complex interaction of different physical phenomena involving turbulent, reacting and multiphase flows evolving inside a moving geometry. Hence, for a proper assessment of the different sub-models involved a detailed set of experimental optical data is required.

To this end, a large experimental database was built by the authors. In particular, the spray morphology was characterized in detail inside a constant volume vessel, where images were acquired by a CCD camera and then post-processed to evaluate the spray penetration and cone-angles. Furthermore, experiments were carried out in an optically accessible combustion chamber reproducing a real 4-stroke, 4-cylinder, high performance GDI engine. The cylinder head was instrumented by using an endoscopic system coupled to high spatial and temporal resolution cameras in order to allow the visualization of the fuel injection and the combustion process. The complete set of spray models was tuned with experiments carried out at constant-volume conditions, then full-cycle simulations were performed for the optical engine. Four different operating points were simulated accounting for different injection pressures and charge stratification levels. Validation was carried out by comparing computed and experimental data of spray and liquid film evolutions. To further verify the computed results, computed equivalence ratio distributions at spark-timing were correlated with optical images of flame propagation.

CITATION: Lucchini, T., Fiocco, M., Onorati, A., Montanaro, A. et al., "Full-Cycle CFD Modeling of Air/Fuel Mixing Process in an Optically Accessible GDI Engine," *SAE Int. J. Engines* 6(3):2013, doi:10.4271/2013-24-0024.

INTRODUCTION

Direct-injection technology applied to SI engines ensures advantages in terms of improved thermal efficiency, better engine response during transients, reduced knock tendency and pollutant emissions during cold start. Further reduction of fuel consumption can be ensured by operation under stratified charge mode, since the engine power output can be regulated only by variation of the injected fuel mass [1]. Currently, most GDI engines available on the market employ multi-hole, mini-sac injectors delivering fuel inside the cylinder and attention is focused on understanding the relevant engine

parameters affecting fuel-air mixture quality, since it is directly related to both combustion efficiency and pollutant formation. To this end, both experimental and numerical studies revealed that the most important ones are injection pressure, strategy, liquid film formation, in-cylinder charge motions and combustion mode [2, 3, 4, 5]. However, investigation of fuel-air mixing and combustion processes requires optical engines or multi-dimensional numerical models due to the high complexity of the involved physical phenomena.

Various experimental studies were carried out in optically accessible closed vessels, model combustion chambers, and rapid compression machines and recently in optical GDI engines in order to study the fuel injection and the combustion process [6]. In particular, combined optical techniques were used in order to analyze the fuel spray distribution and evaporation, mixture preparation and combustion [7]. Few works have been carried out in an optical 4-cylinder high performance GDI engine using data of the ultra violet-visible imaging and spectroscopy acquired directly in the combustion chamber during the combustion process [8, 9]. This optical engine, built in CNR-Istituto Motori, is provided with a variable geometry turbocharger and it is capable to reach 2500 rpm, a torque of 320 Nm at 1400 rpm and a power of 85 kW at 2500 rpm. Optical diagnostics in transparent engines can provide valuable information and quantification of processes of in-cylinder flows and fuel spray using high speed and Schlieren movies and flame propagation and pollutant formation with high spatial and temporal resolution imaging or spectroscopic technique in the UV-Visible range like the two color pyrometry.

However, high costs related to maintenance and production limit the applications of optical engines only to diagnostic purposes, in order to observe and understand in detail the main physical and chemical processes taking place during fuel-air mixing and combustion processes. As a possible alternative to optical investigations, multi-dimensional models can be employed: they allow to study a large variety of configurations in a limited amount of time and with affordable costs [10, 11, 12]. Nevertheless, fuel-air mixing and combustion simulations involve many interacting phenomena that should be taken into account and still represent very challenging tasks for CFD modeling. In particular, reliable and detailed experimental data sets are necessary for a proper assessment of the sub-models that are necessary to describe gas exchange, spray evolution (including liquid film dynamics) and flame propagation in a non-homogeneous charge.

This work is focused on the development and validation of a CFD methodology that can be applied to predict fuel-air mixing in GDI engines operating under the stratified charge mode. A complete set of models was implemented to properly describe the evolution of liquid sprays emerging from multi-hole atomizers and interacting with cylinder walls [13, 14, 15]. The complete dynamics of the liquid film was also described in detail, taking effects of film evaporation into account. The proposed CFD approach has been developed within the Lib-ICE code, based on the OpenFOAM® technology [16] and extensively applied in past works to model in-cylinder flows, fuel-air mixing and combustion in engines [17, 18, 19, 20]. The choice of using the OpenFOAM technology was mainly justified by the flexibility that an open-source and object-oriented platform offers in terms of implementation and verification of new

models dedicated to different purposes (*i.e.* mesh management, spray, combustion, turbulence, ...). To validate the proposed approach, a detailed set of experiments was built. Assessment of the spray models was carried out by comparing experimental and computed data of spray penetration and morphology. Such data were derived from post-processing of spray evolution images that were acquired by a CCD camera for different injection events. To ensure consistency between simulations and experiments, for each tested condition the instantaneous injected mass flow rate was measured and used as boundary condition for CFD simulations. Once the spray model was properly tuned, simulations of the fuel-air mixing process were carried out, accounting for different injection strategies both in terms of start of injection and injection pressure. To verify if the proposed set-up was correct, computed evolution of spray and deposited liquid film were compared with experimental images at different times after start of injection. Finally, computed fuel vapor distribution inside the cylinder at spark-timing was correlated with experimental images of flame propagation. This made it possible to properly identify the main sources of soot formation for the considered injection strategies.

EXPERIMENTAL APPARATUS

Spray Characterization

In the present paper a GDI mini-sac seven-hole Bosch injector with solenoid actuation was used. This injector is characterized by a hole diameter of 0.179 mm, length to diameter ratio 1.5 and with static flow rates of 13.7 g/s at the injection pressure of 100 bar. The experimental characterization involved both measurements of the delivered instantaneous mass flow rate and of spray images acquisition in an optically accessible vessel containing nitrogen under controlled conditions of temperature and pressure (298 K and 1.0 bar, respectively). Commercial gasoline fuel was used ($\rho=740 \text{ kg/m}^3$) as delivered by a hydro-pneumatic injection system without rotating organs. A reservoir tank of 1.0 dm³, located before the electro-injector, assures the absorption of the pressure oscillations produced from the needle opening and the gas recharge. Further details of the apparatus are reported in [21]. The injection and acquisition systems were managed by a programmable electronic control unit (PECU). This is an open system able to reproduce the injector energizing currents for the desired strategy in terms of number of injection pulses, durations, rise and dwell times. The gasoline mass flow rate was measured by means of an AVL meter working on the Bosch tube principle [22, 23]. The spray visualization was made by injecting the fuel in a high pressure quiescent vessel, optically accessible, containing nitrogen at the pressure of 1.0 bar and the temperature of 300 K. The spray images, enlightened by powerful flashes, were collected at different instants from the start of injection (SOI) by means of a synchronized CCD camera, 1376×1040 pixels, 12 bit resolution, 0.5 μs shutter

time. An overview of the experimental scheme for the image acquisition is reported in [21, 24]. The images were acquired using a 50 mm focal lens reaching a calibration factor of 10 pixel/mm. The captured images were processed off-line by means of a proper software able to extract the parameters characterizing the spray dynamics. A set of 5 images was collected for each injection condition for a statistical analysis of the cycle-to-cycle dispersion. The images processing analysis was carried out in different steps: image acquisition and background subtraction, filtering, fuel spray edges determination, tip penetration and cone angle measurements. Background subtraction and median filter procedures were adopted during the image acquisition to remove impulse noise and stray light as to obtain sharp spray edges. This is determined selecting an intensity threshold level for separating the fuel region from the background ambient gas. The spray tip penetration was measured to characterize the spray at fixed conditions. The main hypothesis is that the seven jets all behave in a similar way, so that the measurements may be carried out on a single jet that is less interfering with the others. This condition could not be true for times greater than 1.0 ms, when the jets swelling produces a non-negligible interaction between them which influences both droplets penetration and size. Fig. 1 shows the fuel injection rate profiles with the corresponding injected fuel for the investigated strategies. The area under each profile is the total amount of the injected fuel per stroke. The profiles depend on the injection pressure and duration. In fact the higher the injection pressure, the higher the stationary phase level while the greater the injection duration and the longer the corresponding injection profile. It is interesting to note that all the profiles are quite similar in the overlapping parts in terms of rise time and slope. In the early stage of the injection, the dynamics of the exiting fuel is independent from the injection pressure.

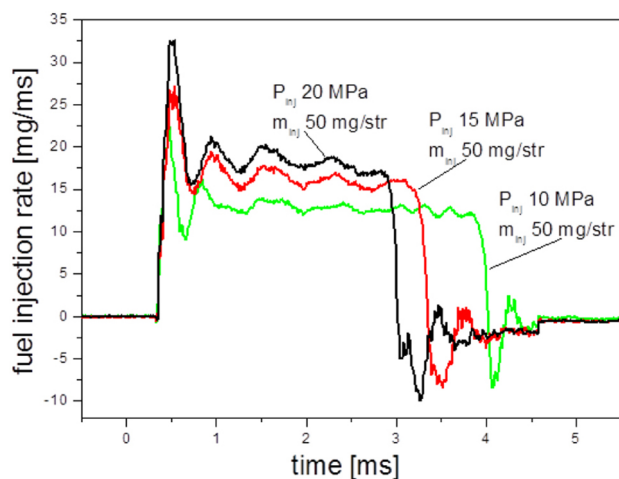


Figure 1. Measured injection rate profiles for different injection pressures.

Optical GDI Engine

An optical, inline 4-cylinder, four stroke spark ignition direct injection engine with a displacement of 1742 cm³, turbocharged, high performance and capable of about 85 kW of maximum power at 2500 rpm was used. It features a seven holes nozzle on the injector tip and a variable valve timing in order to optimize intake and exhaust lift for each regime of operation. The optical engine was equipped with four optically accessible elongated pistons. The head had four valves and a centrally located spark plug. The engine piston has the same geometry as the commercial piston and it was transparent through a sapphire window placed in the bowl region. Details of engine layout are provided in Fig. 2.

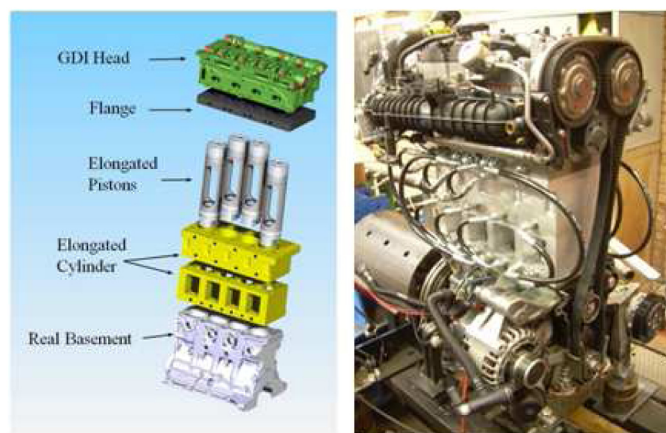


Figure 2. Exploded view and photo of the optical accessible engine on the test bench.

To reduce the window contaminations by lubricating oil, an elongated piston arrangement was used together with unlubricated Teflon-bronze composite piston rings in the optical section. Further details about the commercial engine are reported in Tab. 1.

Table 1. Optical engine geometrical specifications.

Swept volume	1742 cm ³
Swept volume/cyl	435.5 cm ³
Bore	83 mm
Stroke	80.5 mm
Turbine	Exhaust gas turbocharger
Max boost pressure	2.5 bar
Vol. comp. ratio	9.5:1
Max Torque	320.4 kW at 1400 rpm
Max Power	147.1 kW at 5500 rpm

An optical sapphire window (diameter of 5 mm) was flush installed in the engine head in the fourth cylinder. It allowed viewing of the combustion chamber by an endoscopic probe. With this configuration, the endoscope field of view turned out to be exactly centered on the combustion chamber and

was perpendicular to the axis of the cylinder, thus perpendicular to the plane of tumble motion. Using an endoscope with a viewing angle of 30° , it was therefore possible to classify an area that included the spark and the injection of fuel. Imaging and polychromatic chemiluminescence measurements from ultraviolet (UV) to visible were performed by means of the optical experimental set-up shown in Fig. 3. The injection phase was detected through the optical access on the head side and an endoscopic probe coupled with an high spatial and temporal intensified charge coupled device (ICCD) camera. The camera had high sensitivity both in the UV and visible range. In order to characterize the injection phase an intense strobe lamp was used to enlighten the combustion chamber through the transparent piston.

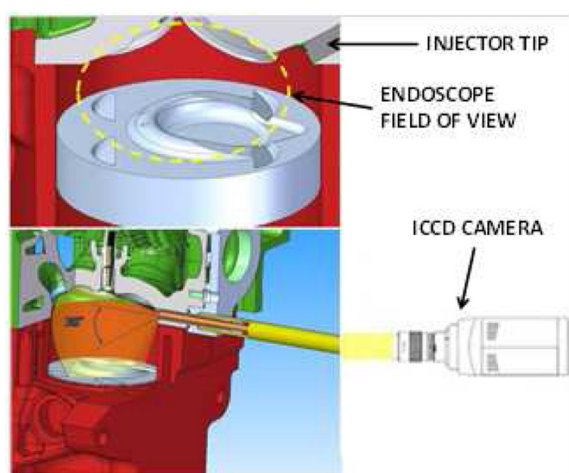


Figure 3. Sketch of the experimental setup for optical investigation and detail of the combustion chamber and ICCD for UV-Visible measurements.

NUMERICAL MODELS

The following sections illustrate the computational models developed by the authors within the Lib-ICE code to simulate gas exchange and fuel-air mixing in gasoline, direct-injection engines. In particular, a detailed description of moving mesh algorithms, spray and liquid film models is provided. All the simulations were carried out using the RANS approach with the standard $k-\epsilon$ model for turbulence and properties of iso-octane were used to describe the gasoline fuel both in the liquid and vapor phases.

Meshing Procedure

The proposed approach for CFD simulation of IC engines is illustrated in Fig. 4. A multiple number of meshes is employed, so that each mesh is valid in a certain crank angle interval during which the grid points are moved and its topology can be eventually changed. An automatic mesh motion technique, based on the Laplace equation, is used to accommodate the motion of internal grid points according to

the prescribed boundary motion [25, 26]. Polyhedral grids are supported, and this aspect is quite important when GDI engines have to be simulated, since their complex geometry usually requires a combination of hexahedrons and tetrahedrons in regions where high deformations take place. The proposed approach for mesh motion has been widely discussed in previous work and the reader is referred to [17, 18, 20] for further information.

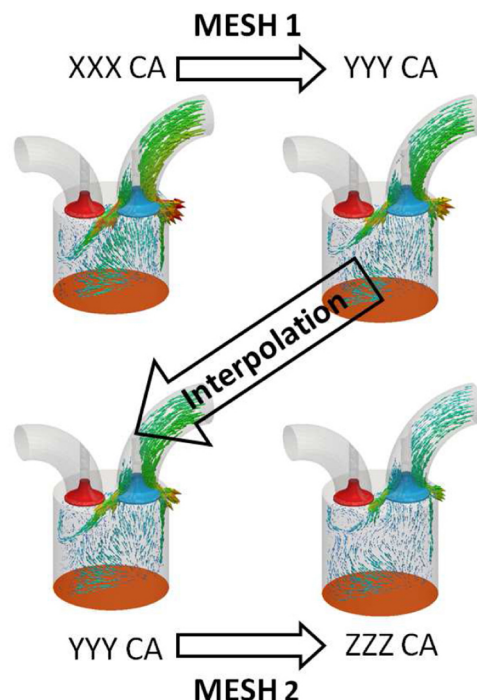


Figure 4. Summary of the strategy used for internal combustion engine simulation with moving meshes. Each mesh is used for a user-specified interval.

Generally combustion chamber optimization and design requires to test a wide variety of geometry configurations, however the strong complexity of the GDI geometries requires a very large amount of user time for grid generation. To significantly reduce the computational time required to create the engine grids, a fully-automatic approach based on the snappyHexMesh utility available in OpenFOAM was developed. At target user-specified crank-angles, the surface patches (piston and valves) of a CAD file representing the combustion chamber are moved to the required positions. Then, snappyHexMesh modifies the topology of a background hexahedral block-structured grid to fit the cylinder geometry. Possibility to insert boundary layers or suitable refinement surfaces is also allowed, making it possible to account all the details of the combustion chamber including also the full geometry of the spark-plug. Figs. 5(a)-(b) display typical layout of the grids used in this work around the valve and spark-plug regions.

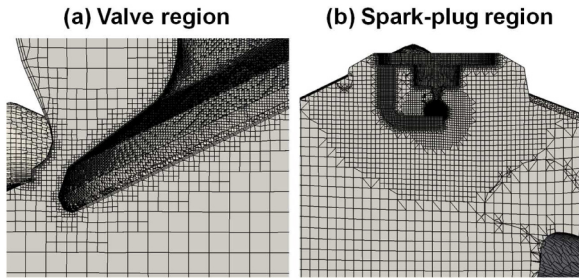


Figure 5. Examples of grid structures in valve (a) and spark-plug (b) regions achieved by using the snappyHexMesh utility available in the OpenFOAM code.

Spray Model

The spray is modeled using the Eulerian-Lagrangian approach and supports multi-component fuels. Specific sub-models are available to describe fuel atomization, breakup, heat transfer, evaporation, collision and wall impingement. In this work, atomization of the liquid fuel jet is predicted by the Huh-Gosman model [13]. Primary parcels (blobs) are injected into the computational mesh with the same nozzle diameter and 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:

$$\frac{dD}{dt} = -C_5 \cdot \frac{L_a}{D_a} \quad (1)$$

where C_5 is the main model constant, L_a and D_a are the characteristic atomization length and time scales. L_a is proportional to the turbulent length scale on the liquid jet L_t , while atomization time-scale is a function of both turbulent and aerodynamic time scales. Their values are related to the geometry of the injection system, turbulence decay within the jet and Kelvin-Helmholtz instabilities. Further details about the Huh-Gosman model are available in [13]. At the nozzle exit, the spray angle α is predicted according to the expression:

$$\tan\left(\frac{\alpha}{2}\right) = \frac{L_a/D_a}{U} \quad (2)$$

where U is the droplet velocity at the nozzle exit. Secondary droplets are created from primary blobs when the amount of stripped mass is higher than a specific threshold value. Their diameter is computed from a PDF distribution whose properties and diameter range are computed according to [13]. The secondary stripped droplets experience both Kelvin-Helmholtz (*wave*) and Rayleigh-Taylor (*catastrophic*) breakup mechanisms operating in competition [14]. In the first case, small child droplets with size D_{KH} are stripped

from the parent one, whose diameter reduction is controlled by a breakup time τ_{KH} as follows:

$$\frac{dD}{dt} = -\frac{D - D_{KH}}{\tau_{KH}} \quad (3)$$

Values of D_{KH} and τ_{KH} are function of the most unstable growth rate (Ω_{KH}) and wave number (Λ_{KH}) [14]. Catastrophic breakup occurs when the droplet life-time becomes equal to τ_{RT} , which is a function of fuel surface tension, relative velocity, droplet size, liquid and gas densities. Two model constants, B_1 and c_{RT} are incorporated into τ_{KH} and τ_{RT} and both of them need to be adjusted to match experimental data.

The interaction between the spray and the liquid film is described with the approach originally proposed by Bai and Gosman [15]. Generally, the collision between a drop and a wet surface may result in three different regimes: adhesion, rebounding and splashing [27]. The most important parameters characterizing the impinging regimes are combined together into two dimensionless parameters: Weber and Laplace impact numbers. The first represents the ratio of the droplet's kinetic energy (v_n : velocity component normal to the surface, ρ_l : liquid density, d : droplet diameter) and its surface energy:

$$We = \frac{\rho_l v_n^2 d}{\sigma} \quad (4)$$

the Laplace number measures the relative importance of surface tension and viscous forces acting on the liquid, whose dynamic viscosity is μ_l :

$$La = \frac{\rho_l \sigma d}{\mu_l^2} \quad (5)$$

For a dry wall, the transition criterion between adhesion and splash is given as [15]:

$$Adhesion \Rightarrow Splash \quad We_{crit} = A \cdot La^{-0.18} \quad (6)$$

where the coefficient A depends on the surface roughness ($A = 1320$ in this work). In the case of an already wetted wall, the transition Weber numbers are:

$$Rebound \Rightarrow Spread \quad We_{crit} \approx 5.0 \quad (7)$$

$$Spread \Rightarrow Splash \quad We_{crit} \approx 1320 \cdot La^{-0.18} \quad (8)$$

In the adhesion regime, the incoming droplets are assumed to coalesce to form a local liquid film. In the rebound regime, the droplet bounces off the wall and does not break up, but loses a small part of its kinetic energy by