

A 3D CFD Simulation of GDI Sprays Accounting for Heat Transfer Effects on Wallfilm Formation

Daniele Piazzullo
Università di Roma "Tor Vergata"

Michela Costa, Luigi Allocca, Alessandro Montanaro, and Vittorio ROCCO
Istituto Motori CNR

ABSTRACT

During gasoline direct injection (GDI) in spark ignition engines, droplets may hit piston or liner surfaces and be rebounded or deposit in the liquid phase as wallfilm. This may determine slower secondary atomization and local enrichments of the mixture, hence be the reason of increased unburned hydrocarbons and particulate matter emissions at the exhaust.

Complex phenomena indeed characterize the in-cylinder turbulent multi-phase system, where heat transfer involves the gaseous mixture (made of air and gasoline vapor), the liquid phase (droplets not yet evaporated and wallfilm) and the solid walls. A reliable 3D CFD modelling of the in-cylinder processes, therefore, necessarily requires also the correct simulation of the cooling effect due to the subtraction of the latent heat of vaporization of gasoline needed for secondary evaporation in the zone where droplets hit the wall. The related conductive heat transfer within the solid is to be taken into account.

In this work, a preliminarily validated spray model is specifically implemented by solving the strongly coupled heat and mass transfer problem describing the liquid and vapor phases thermo-fluidynamics after impact and the wall change of temperature. The discussion is made considering a different boundary condition with respect to standard simulations. Sprays are assumed from two different injectors in order to verify the wallfilm simulation model: the impact over heated walls of the ECN "Spray G" is first discussed, by comparing numerical results with experimental measurements deriving from a combined use of the schlieren and Mie-scattering techniques, then the footprint on the wall of the spray delivered from a 6-hole Bosch injector is related with infrared thermography and LIF measurements taken from the literature.

CITATION: Piazzullo, D., Costa, M., Allocca, L., Montanaro, A. et al., "A 3D CFD Simulation of GDI Sprays Accounting for Heat Transfer Effects on Wallfilm Formation," *SAE Int. J. Engines* 10(4):2017, doi:10.4271/2017-24-0041.

INTRODUCTION

The control of the mixture formation process taking place within the engine combustion chamber of direct injection internal combustion engines (ICEs) remains the preferred route towards the reduction of pollutants formation. In particular, the diminution of particulate matter (PM) emissions is one of the current objectives of the development of gasoline direct injection (GDI) engines. EU6 emissions standard for this engines limits vehicles PM to 6e11 particles/km [1, 2]. PM is mainly composed by primary and secondary organic aerosols (POA and SOA) and by organic chemicals as soot. Its formation mechanisms strictly depend upon local mixture enrichment and possible wallfilm formation, this last deriving from the often unavoidable liquid fuel impingement on surfaces during injection. GDI has recognized advantages over port fuel injection (PFI) in terms of fuel economy, greater control of the air-to-fuel ratio and possibility to adopt higher compression ratios [3], but also exhibit severe drawbacks in terms of spray-wall piston impact and undesired in-chamber mixture inhomogeneity, especially under some load or

speed conditions. Wallfilm may form as liquid droplets accumulate locally over surfaces. The heat transfer from the wall to the liquid film strongly affects secondary evaporation [4, 5]. The consequent air-to-fuel ratio inhomogeneities, coupled with moderate combustion temperatures, determine the occurrence of diffusive flames where carbonaceous compounds nucleation may occur [6]. So-called pool fires from ignition of wallfilms, indeed, play a central role in soot production, as it is quite well assessed that most of the nucleated material formed from locally rich zones undergoes oxidation due to high in-cylinder temperatures, while soot formed from pool fires originate late and can persist in the engine cycle [7, 8, 9].

Studies concerning the phenomena of spray-wall impingement performed in different operative conditions can be found in the literature. Yu et al. [10] experimentally investigated the spray-wall impingement using n-butanol/diesel blended fuels under different injection strategies. In a subsequent work [11] these results were extended to an analysis of wallfilm characteristics, employing methyl

ether (DME), n-butanol, diesel and gasoline fuels. Results showed that impingement momentum and fuel viscosity are major factors affecting wallfilm formation. The relevance of the temperature drop of the wall surfaces due to gasoline impingement has been proved as a fundamental variable influencing the chain of events leading from secondary evaporation up to soot formation. Schulz et al. [12] demonstrated the deep influence of small variations of the wall temperature in a GDI engine on the wall heat flux, thus conditioning the possible deposition of gasoline on surfaces. For the wallfilm, heat exchange is considered between the gaseous flow and the film, and the film and the wall. Jiao et al. [13] showed the different amounts of soot mass produced at the exhaust valve opening (EVO) considering two different liner and piston surface temperatures. PM emissions were one orders of magnitude higher for lower temperatures.

Nowdays, computational fluid dynamics (CFD) is getting increasingly important to accelerate the time to market of high-efficiency clean power units for transportation [14], due also to the recent increase in engine complexity and in the number of involved sub-systems [15]. Therefore, CFD approaches reveal crucial to overcome the intrinsic limits of experimental techniques.

Since, as said before, it is mandatory to take into account heat transfer phenomenon at walls at spray impact, numerical simulations need a good prediction of these events. Deng et al. [16] developed a spray wall heat transfer model, implemented in the KIVA-3V code, to account for the post-impingement movement and film formation of spray droplets, validated on experimental data of diesel spray impingement. Andreassi et al. [17] developed a new splash numerical model modifying literature empirical correlations on the basis of experimental results on the impingement of a diesel spray onto a flat surface at different operating conditions. Zhao et al. [18] provided detailed data on spray-wall impingement physics in order to support the spray-wall interaction model development from both an experimental and a numerical point of view. Spray-wall impingement with non-vaporizing spray characterization was carried out in a high pressure-temperature constant-volume combustion vessel, through a simultaneous implementation of the Mie scattering technique for the liquid spray and the schlieren for the liquid and vapor zone. Experimentally validated CFD simulations were then used to investigate the local spray characteristics in the vicinity of the wall with a particular focus on Sauter Mean Diameter (SMD) and Reynolds and Weber numbers.

In the present work, the in-cylinder thermo-fluiddynamic processes involving spray-wall interaction are studied with reference to sprays generated by two different injectors. A previously developed 3D CFD model, validated under free spray conditions for different injection devices and conditions, is applied to gain a detailed insight into the dynamics and impact over walls of both the "Spray G" of the Engine Combustion Network (ECN) circuit as delivered from a 8-hole Delphi solenoid actuated injector and of a spray from a 6-hole Bosch HDEV series multi-hole injector. An experimental characterisation of the first is made by authors through combined schlieren and Mie-scattering techniques while data from LIF measurements from the literature are taken for verification in the second spray case. A correct

3D CFD simulation of the local heat transfer between gasoline and wall is made possible through the implementation of a proper sub-model of the unsteady phenomena occurring during impingement. A step further is done by studying the thermal characteristics of the flow at the wall and by analyzing the effects of cooling on wallfilm formation.

EXPERIMENTAL CHARACTERIZATION OF THE ECN "SPRAY G"

The experimental part of the work relevant to the ECN "Spray G" was carried out at exemplified engine conditions in a quiescent vessel, controlled in pressure and temperature, housing a heated plate to simulate the piston head or the cylinder wall. These conditions permitted to isolate the fundamental processes of the impingement and the relative investigations by using non-interfering optical tools. The bowl was equipped with three wide optical accesses, two in-line and one at right angle, for enlightening and light collection. The pressure of the gas (N_2) in the vessel ranged between 0.02 and 0.35 MPa while the temperature varied among 298 and 573 K. The plate for the impingement was located at a distance of 19.5 mm from the nozzle tip inside the heating device of the vessel and assumed analogue temperature of the gas. A more complete description of the experimental setup is reported in [19]. The eight-hole "Spray-G" injector by the ECN circuit [20] was chosen to inject iso-octane at pressures of 5.0, 10.0 and 20.0 MPa while the impinging wall was set orthogonally to the injector axis. Tests were carried out supplying 10 mg of fuel per shot with 680 μ s of coil energization time, this last maintained fixed for different values of injection pressure. Depending on the thermodynamic conditions realized inside the vessel and on the plate, liquid and vapor phases of the iso-octane were present during the impact. Moreover, their percentages varied in time and space during the impact evolution. Two techniques in a quasi-simultaneous mode were applied to detect both the phases: Mie-scattering for the liquid status and schlieren for identifying the vapor/liquid mixture zones. The optical setup consists of a Z-shaped structure using a 455nm / 450 mW LED for schlieren images and a cool-white emitter, LED 5500 lm on a 40 mm², for Mie-scattering ones. The pictures were acquired by a high speed camera equipped with a 90 mm objective, $f=1:2.8$. The complete setup and was deeply described in [21]. The resulting frame speed of the camera was 12,500 fr/s involving a time resolution of 80 μ s. A homemade algorithm for image-processing procedure was used to treat the batch and outline the contours of the images [22]. Each spray image was corrected referring to the background levels and Gamma factor corrections (2.3 for schlieren, 0.7 for elastic scattering), Gaussian smoothing (11.4) and morphology filtering completed the image adjustments. A more complete description of the procedure is reported in [21].

The effects of the wall/ambient temperature on the morphology of the fluid are discussed for both the liquid and vapor phases. Figure 1 reports on the top a schlieren sequence of the liquid/vapor mixture of the iso-octane injected at a pressure of 10.0 MPa and impinging on the flat plate kept at the temperature of 473 K. The figure represents half of the impacting spray, hence the injector is on the left of each single image with its axis in orthogonal direction to the plate. The



backpressure of the gas (N_2) was atmospheric and its temperature was the same of the plate. The sequence was acquired at the frequency of 25,000 fr/s resulting the time resolution of 40 μ s. The pictures reported in Figure 1 depict a mixture of vapor and liquid phases of iso-octane flushing along the wall and rebounding on it. These two aspects give up to the width and thickness parameters that characterize the impact and we collect by the processing procedure. The slipping/rebounding fluid shows a double structure: denser and impenetrable to the schlieren light beam close to the wall, indicative of predominance of liquid, lighter and almost transparent upper the plate, symptomatic of vapor generated by the heating with the wall.

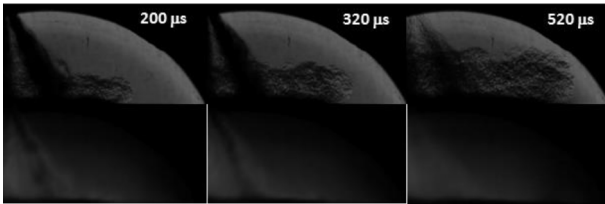


Figure 1. Impact sequences of schlieren (top) and Mie-scattering (bottom) for $P_{inj} = 10.0$ MPa, $T_w = 473$ K at atmospheric backpressure. The time refers to the impact instant.

The bottom part of the Figure 1 reports the sequence of the Mie-scattering operating in forward and in the quasi-simultaneous mode with the schlieren as described in [21]. The dense core of the jets results in a black area indicative of the total extinction of the incident beam while the surrounding droplets, stripped from the primary jet during the interaction with the gas, produce a huge scattered intensity of light. The liquid part of the fluid slips away the impact zone and penetrates the gas along the time. The liquid part, evidenced through the Mie-scattering technique, needs a quite sensitive processing procedure with a complex algorithm to set the parameters. The final width/thickness values are extracted by the experienced process as deeply described in [21].

NUMERICAL MODEL

CFD simulations of the dynamics of the gasoline spray are developed within the AVL Fire™ environment. The turbulent flow field is described through a Reynolds Averaged Navier Stokes (RANS) approach, with the k - ζ - f as closure turbulence model [23]. The classical discrete droplet method (DDM) is chosen to describe the spray evolution. The governing equations here not reported for the sake of brevity [24]. The free spray model assessment can be found in a previous work by authors [25], where it is shown that the breakup phenomenon follows the sub-model by Huh-Gosman [26] whose constant C_1 (affecting the break-up time) is adjusted through a proper tuning procedure [27], turbulent dispersion on the droplets dynamics are simulated through the sub-model by O'Rourke [28], coalescence through the sub-model by Nordin [29], evaporation through the multi-component sub-model [30].

Two different spray cases are considered within the present work. The results of the comparison between the 3D numerical simulation and the experimental campaign relevant to the Spray G before impact

are summarized in Figure 2, where the measured and calculated penetration lengths are reported as a function of time from the start of injection (SOI). The computational mesh for the spray validation is a cylinder whose diameter and height are 100x100 mm, with a total number of cells equal to 720000. The grid is reported in Figure 3. The free spray dynamics of a heated gasoline at 363 K is well reproduced by the 3D CFD tool considering an injection pressure of 20 MPa in a vessel under standard air conditions.

As regards the spray impact simulation, the sub-model by Kuhnke [31] is chosen to reproduce the different dynamic impact regimes. The model distinguishes four regimes of spray-wall interaction, defined as a function of the parameters K and T^* :

$$K = \frac{(\rho_d d_d)^{3/4} u_{d,\perp}^{5/4}}{\sigma_d^{1/2} \mu_d^{1/4}} \quad (1)$$

$$T^* = \frac{T_{wall}}{T_{sat}} \quad (2)$$

where ρ_d is the droplet density, d_d is the droplet diameter, $u_{d,\perp}$ is the droplet velocity normal to the surface, σ_d is the liquid surface tension, μ_d is the dynamic viscosity, T_{wall} is the wall temperature and T_{sat} is the boiling temperature. Further details about the four regimes of spray-wall interaction described by this sub-model may be found in ref. [32]. Heat transfer at the wall is simulated through the sub-model called Thin-Wall [33]. The wall boundary is specified with a multilayer structure of homogeneous solid materials, and the heat conduction is solved within the Thin-Wall thickness, and is coupled with the fluid energy equation (3):

$$\frac{\partial T}{\partial t} - \frac{\lambda}{\rho c_p} \frac{\partial^2 T}{\partial t^2} = 0 \quad (3)$$

where λ is the thermal conductivity, c_p is the specific heat at constant pressure, ρ is the density, T is the temperature value and t is the time.

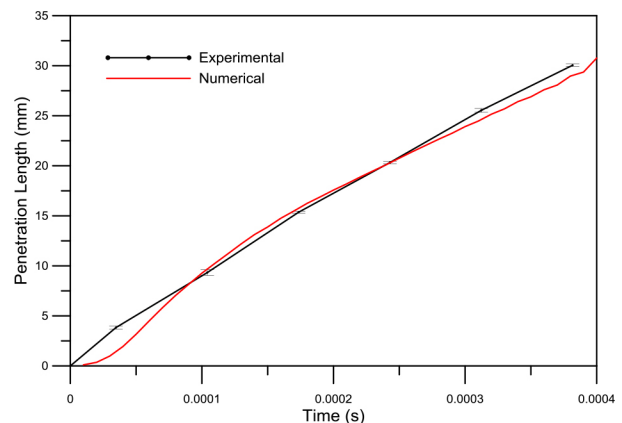


Figure 2. Comparison between calculated and measured penetration length of the ECN "Spray G" for and injection pressure of 20 MPa