

Numerical Investigation of Two-Phase Flow Evolution of In- and Near-Nozzle Regions of a Gasoline Direct Injection Engine During Needle Transients

Kaushik Saha and Sibendu Som
Argonne National Laboratory

Michele Battistoni
Universita degli Studi di Perugia

Yanheng Li, Eric Pomraning, and P. K. Senecal
Convergent Science Inc.

ABSTRACT

This work involves modeling internal and near-nozzle flows of a gasoline direct injection (GDI) nozzle. The Engine Combustion Network (ECN) Spray G condition has been considered for these simulations using the nominal geometry of the Spray G injector. First, best practices for numerical simulation of the two-phase flow evolution inside and the near-nozzle regions of the Spray G injector are presented for the peak needle lift. The mass flow rate prediction for peak needle lift was in reasonable agreement with experimental data available in the ECN database. Liquid plume targeting angle and liquid penetration estimates showed promising agreement with experimental observations. The capability to assess the influence of different thermodynamic conditions on the two-phase flow nature was established by predicting non-flashing and flashing phenomena. Mild vapor formation inside the holes of the injector was predicted for flashing and non-flashing conditions, which suggests that there was sufficient depressurization of liquid fuel in a GDI nozzle. Outlet domain size and alignment of spray plumes with a grid were observed to be critically important to avoid numerical errors. Finally, to represent the real-world scenario of a moving needle in a GDI nozzle, simulations were carried out at different fixed needle lifts to provide insight to flow development inside a GDI nozzle, from opening to closing of the needle. Predictions indicated that flow patterns, and thus vapor formation characteristics, are considerably affected at very low needle lifts, for both flashing and non-flashing scenarios.

CITATION: Saha, K., Som, S., Battistoni, M., Li, Y. et al., "Numerical Investigation of Two-Phase Flow Evolution of In- and Near-Nozzle Regions of a Gasoline Direct Injection Engine During Needle Transients," *SAE Int. J. Engines* 9(2):2016, doi:10.4271/2016-01-0870.

INTRODUCTION

The automotive industry is under constant pressure to keep up with the increasing demand of passenger vehicles and public transport. The industry also has to abide by ever-stricter emission regulations, as delineated by emission regulation agencies. Recently, gasoline direct injection (GDI) engines (in which the fuel is directly injected into the combustion chamber) have been preferred over the previous trend of port-fueled-injection (PFI) systems, in which fuel is injected upstream of intake valves and mixed with incoming flow. This is because GDI systems provide better fuel economy and can operate at higher compression ratios and ultra-lean to slightly rich fuel/air ratios, resulting in cleaner and more efficient combustion. Considerable efforts are being made by the industry and the academia to ameliorate the drawbacks of current GDI engines, such as relatively high particulate matter (PM) emissions.

Comprehensive analysis of fuel spray atomization and fuel/air mixing is warranted for developing an in-depth understanding of GDI engine operations, because combustion efficiency is

predominantly influenced by the spray formation of the liquid fuel. In this context, flash boiling of GDI sprays becomes a pertinent area of research. Flash boiling is a thermodynamic phenomenon where liquid phase being subjected to superheated conditions rapidly undergoes bulk transformation to the vapor phase, generating a wide, and typically short, mist-like spray of small droplets. Flash boiling, similar to spray atomization, can be either beneficial or detrimental to GDI engine performance, depending on the spray penetration and width, engine bore, and piston shape, among other factors. Flash boiling causes rapid vaporization of fuel droplets. Typically, spray penetration reduces due to flashing, but when multiple flashing plumes merge, momentum increases, leading to an increase in penetration. Therefore, final spray characteristics and chamber geometry govern the chances of spray-wall interaction and, hence, the unburned hydrocarbon emissions [1, 2]. Thus, flashing adds another level of uncertainty for precise control of GDI combustion.

Flashing is a non-equilibrium phenomenon, unlike cavitation in diesel injectors, where phase change occurs almost instantaneously. The typical fuel temperatures of GDI systems are high (~ 360 K) and in-cylinder chamber pressure under part-load (throttling) conditions is sub-atmospheric (~ 40 kPa) [2]. Therefore, there is a high chance that the injected fuel will experience superheated conditions and undergo flash boiling. Flash boiling typically occurs in the near-nozzle region. Vapor formation in the near-nozzle region occurs not only due to flashing, but also due to evaporation caused by high ambient temperatures. Flash boiling depends on chamber pressure and fuel temperature, while evaporation is affected by both ambient pressure and ambient temperature. As a result, in a hot environment the fuel spray will only undergo evaporation as a phase change phenomenon and will not flash if the chamber pressure is higher than the saturation pressure corresponding to the fuel temperature.

Several experimental as well as a few numerical endeavors have been reported in the literature. The experimental findings will be reviewed followed by discussion of numerical simulation results. In their planar laser-induced fluorescence (PLIF) and Mie-scattering images, VanDerWege and Hochgreb [3] observed a pressure-swirl injector spray transitioning from a hollow cone to a solid cone as the superheating temperature of the liquid fuel increased. The authors inferred from image intensity comparisons that the droplet diameter reduction due to flash boiling is on the order of 40%. Based on bubble point calculation, they also suggested that superheating by roughly 20 K is required for the flash boiling to impart noticeable effects on spray formation. Their experimental results demonstrated the potential advantages of using flash boiling to enhance spray atomization. Zeng et al. [4, 5] experimentally studied the spray from a multi-hole injector using Mie-scattering and laser induced exciplex fluorescence (LIEF). The authors quantified the macroscopic spray structure's penetration length, plume width, and normalized plume distance. The authors also characterized different macroscopic spray behaviors based on the ratio of ambient pressure to saturation pressure. A set of empirical expressions were proposed to predict the macroscopic structure of the spray under the effects of flash boiling. Weber and Leick [2] carried out high-speed Mie-scattering to characterize the global spray and investigate the near-nozzle spray structures. Droplet velocities were also measured using shadow particle image velocimetry. They observed that individual spray plume lengths decrease and initial cone angles increase under flashing conditions. They also noticed that during intense flash boiling two opposing spray plumes might interact with each other.

Few numerical studies on flash boiling for GDI injectors are available in the literature. They can be broadly classified into two types: (a) bubble dynamics-based models, and (b) empirical coefficient-based thermodynamic rate equation models. Blinkov et al. [6] assumed a mono-disperse bubble population and their model consisted of five transport equations that included vapor formation, interface heat transfer, and nucleation. Blinkov et al. employed their model to analyze flashing and choking water flows and inferred that bulk nucleation becomes increasingly significant as the volume-to-surface ratio of the geometry increases. Chang and Lee [7] modified Blinkov's approach by considering a poly-disperse bubble population in a converging water nozzle and concluded from their primary analysis that capturing the bubble size distribution is

important for multiple and distributed sources of bubble nucleation. Kawano et al. [8] incorporated bubble growth, bubble perturbation, and breakup to model flashing and non-flashing sprays. A similar approach was adopted by Zeng and Lee [9] to study n-heptane sprays, subjected to different degrees of superheat; they concluded that more flashing can be correlated with increases in superheat, smaller drop sizes, and wider sprays. In their work, Zeng and Lee studied the effect of two competing mechanisms of secondary breakup-bubble growth and aerodynamic force-in a hollow cone spray issuing out of a pintle injector. Zuo et al. [10] analyzed the external flashing gasoline sprays downstream of the nozzle exit in the case of a pressure-swirl atomizer. They studied the vaporization of droplets caused by both flashing and high-temperature convective conditions. Lu et al. [11] resorted to Large Eddy Simulation (LES) modeling of flashing sprays; they included vaporization and condensation sub-models in their Eulerian-Lagrangian Spray Atomization (ELSA) framework and presented some preliminary results of LES simulations without full implementation of ELSA approach, stating that the full implementation of their approach was still under progress. Henry and Fauske [12] utilized empirical coefficients to model non-equilibrium vapor generation as a fraction of equilibrium vapor generation and validated their model parameters with experimental data for steam-water, cryogenic fluids, and alkali-metal liquids. Another type of flash boiling model based on empirical coefficients is the homogeneous relaxation model (HRM) [13, 14, 15, 16, 17], which represents the phase transition with one equation that estimates the time scale of the phase change. The time scale provides an estimate of the deviation from thermal equilibrium. HRM lies between the two extremes of thermodynamic two-phase models: the homogeneous equilibrium model (HEM) and the homogeneous frozen model (HFM) [14, 18]. In the case of HEM, the two phases are assumed to be mixed perfectly homogeneously and heat transfer is assumed to occur continuously. In real-world scenario of two-phase flows, such as bubbly flows, instantaneous heat transfer is not feasible. At the other extreme, HFM assumes zero heat transfer, or assumes the heat transfer time scale to be infinitely long. HRM falls between these two practical two-phase flow scenarios. The vapor-liquid equilibrium properties must be known in order to implement HRM and have been estimated for pure as well as blended fuels, for GDI applications [16, 19]. Recently, Moulai et al. [20] and Saha et al. [21] carried out a numerical study of iso-octane flash boiling in a multi-hole counter-bore GDI fuel injector (Spray G) with conditions specified in the Engine Combustion Network (ECN) [22]. The authors examined cases where the outlet chamber provides superheated and non-evaporative conditions, as well as non-flashing and evaporative ambience. Both studies were able to capture the essence of flashing and non-flashing phenomena, and their quantitative predictions of vapor concentration matched flashing conditions. Backflow of ambient gas (100% N_2) into the counter-bore regions was inferred in the numerical analysis of these papers.

The current study has two main goals: (a) to conduct a rigorous assessment of the flash-boiling model and best practices and (b) to mimic the effects of needle movement through simulations of fixed needle lifts at different needle positions from full lift to a nearly closed needle. CONVERGE, a commercial computational fluid dynamics (CFD) software was utilized for

analyses. The relevant details about the CONVERGE solver are available in the literature, such as Richards et al. [23] and Saha et al. [21]. The first part of the study will evaluate the performance of the mixture multiphase model (Eulerian framework) by comparing it with the available data in ECN [22] and by outlining the best practices, such as outlet chamber size, grid resolution, and grid alignment. The second part of this paper will explore the needle lift effects on both flashing and non-flashing scenarios. Iso-octane was used as the representative fuel for the current study, in accordance with ECN standards [22].

MODEL FORMULATION

This study considered an eight-hole counter-bore GDI fuel injector, denoted as Spray G nozzle in the ECN [22]. The approximate nozzle geometry information was obtained through ECN. The internal geometry of the Spray G nozzle is not symmetric since it has eight holes and five dimples, which results in uneven flow passages. Therefore, for realistic predictions full nozzle geometry was simulated in this study. Details of the cases studied in this work are summarized in Table 1. Based on prior work on Spray G cases (Moulai et al. [20] and Saha et al. [21]), it is evident that the Spray G condition is supposed to be non-flashing, since the chamber pressure is higher than the saturation pressure that corresponds to iso-octane (liquid) fuel temperature. However, the chamber temperature is higher than the fuel temperature, so the liquid jets coming out of the eight holes are expected to evaporate due to convective heating from the ambient temperature. In the case of Spray G2, flashing is expected because the chamber pressure is lower than the corresponding saturation pressure of the fuel. For Spray G2, the chamber is considered at room temperature; therefore, evaporation is not expected. Spray G Submerged is similar to Spray G, except for the chamber, which contains liquid iso-octane at 293 K. The experimental rate of injection (ROI) is available through ECN [22] for Spray G Submerged, which provides a yardstick for model prediction assessment.

Nozzle Geometry and Computational Domain

It is very important to have a clear idea about the orientation of the eight counter-bored holes i.e., the stepped holes with respect to the dimples and the computational domain to be used for the current numerical study. Figure 1 shows the location of the eight stepped holes with respect to the five dimples. The upstream asymmetry in nozzle geometry is evident from this figure. The information about the orientation of the holes is vital for cases where hole-to-hole variations become conspicuous inside the nozzle and/or in the near-nozzle regions. The nominal diameters of the holes and the counter-bores are 165 μm and 388 μm , respectively. The length-to-diameter ratio of the nozzle holes and counter-bores are around 1 and 1.2, respectively. The peak, or full needle lift, is at 45 μm . A hemispherical outlet domain is needed to ensure the near-nozzle flow features are unraveled accurately. It is difficult to have correct *a priori* estimation of the extent of the outlet domain. Therefore, 3 mm, 9-mm-, 18-mm-, and 36-mm-diameter outlet domains were tested in this study.

Table 1. Different variations of Spray G explored in the current study

Parameters	Spray G	Spray G Submerged	Spray G2	Spray G Cool
Injection Pressure (MPa)	20	20	19.5	20
Chamber Pressure (kPa)	600	600	53	600
Chamber Temperature (K)	573	293	293	293
Fuel Temperature (K)	363	363	363	363
Chamber Fluid	Nitrogen	Iso-octane	Nitrogen	Nitrogen

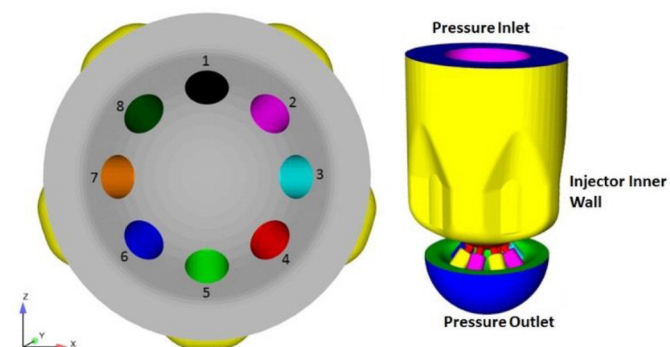


Figure 1. Layout of the eight stepped holes of the Spray G nozzle (bottom view) and a sample computation domain considered in this study (3-mm-diameter chamber).

Figure 2 illustrates a vertical cut-plane showing a mesh with a 9 mm outlet and 140 mm as a base grid size. Adaptive mesh refinement (AMR) has not been utilized to ensure efficient load balancing while executing parallel computing with millions of cells in the computational domain. Based on nozzle flow simulations in previous studies by our group [24, 25] it is evident that fixed embeddings need to be used near the walls, inside the holes, in the near-nozzle regions, and inside the chamber to capture sharp gradients in species, velocity, and temperature. Simulations have been carried out with 180- μm , 140- μm , and 120- μm as base grid sizes (maximum grid size), with minimum grid sizes of 22.5 μm , 17.5 μm , and 15 μm , respectively.

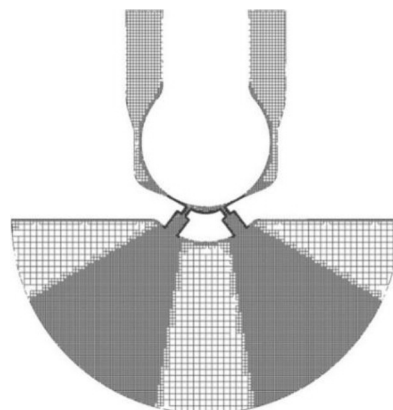


Figure 2. Vertical cut-plane showing the mesh with a 9-mm-diameter outlet domain.

Governing Equations

The simulation carried out in Eulerian framework for flash boiling requires solving the set of governing equations of mass, momentum, species, and energy conservation; additional scalar equations for turbulence; and constitutive equations for liquid and vapor phases to account for variable density due to pressure changes. The governing equations adopted in this study are available in CONVERGE v2.2 manual [23], in the previous published work on flash boiling [21], and in work our group has done on diesel injectors [24, 25, 26, 27]. The set of Favre-averaged Unsteady Reynolds Averaged (U-RANS) governing equations were solved for conserving mass, momentum, species, and energy. The constituent species are liquid iso-octane (C_8H_{18}), vapor C_8H_{18} , N_2 , and O_2 . Standard $k-\epsilon$ equations were also solved to consider the essence of turbulence.

In order to estimate the source/sink term for vapor C_8H_{18} , a phase change submodel formulation is required. This is taken care of by the homogeneous relaxation model (HRM).

Homogeneous Relaxation Model

The source term S_m in the species conservation equation is evaluated through the HRM formulation. The rate of change of local vapor quality (i.e., $\frac{Dx}{Dt}$) provides the estimate of S_m . HRM makes it possible to express $\frac{Dx}{Dt}$ as:

$$\frac{Dx}{Dt} = \frac{\bar{x} - x}{\theta} \quad (1)$$

The timescale θ is calculated as:

$$\theta = \theta_0 \alpha^{-0.54} \psi^{-1.76} \quad (2)$$

where

$$\theta_0 = 3.84 \times 10^{-7}; \psi = \frac{|p_{sat} - p|}{p_{crit} - p_{sat}}$$

The values of the parameters specified above are known to be effective from an engineering perspective for flow problems with pressures higher than 10 bar [14, 16]. The injection pressure for the GDI investigated in this study is considered to be 20 MPa (200 bar). HRM has only been proven to be effective in the literature for flashing and cavitation, and not for evaporation. In the current framework, incorporating an effective evaporation submodel for a Eulerian approach is beyond the scope of the current work. Therefore HRM is used to capture any kind of liquid-vapor phase change, such as flash-boiling, cavitation, or evaporation. It is vital to understand the relation between S_m and $\frac{Dx}{Dt}$. Battistoni et al. [28] elaborated on this relation in their work on cavitation. If x^0 is the computed current local vapor quality of a computational cell and x^1 is the corresponding value for the following time step, with a time-step size of Δt , then x^1 can be computed as $x^1 = \bar{x} - (\bar{x} - x^0)e^{-\Delta t/\theta}$. Therefore,

$$S_m = \frac{(x^1 - x^0)(m_{vap} + m_{liq})}{V\Delta t} \quad (3)$$

where V is the volume of the computational cell, and m_{vap} and m_{liq} are the mass of the vapor and liquid phases, respectively, in that computational cell. This formulation for a single component was implemented for the current study.

Boundary and Initial Conditions

Pressure is identified at the inlet and outlet of the nozzle, and a no-slip boundary condition is assumed at the walls. No-slip conditions are also assumed at the interface between liquid and vapor for the mixture multiphase approach. Turbulence boundary conditions are specified by the turbulent kinetic energy and turbulence dissipation rate at the inlet and the outlet. A liquid fuel cannot be 100% pure. Therefore it is reasonable to consider the presence of very small amounts (10^{-5} mass fraction) of non-condensable gases [21], which act like nucleation sites for the initiation of cavitation. The presence of non-condensable gases has been taken into account in some of the modeling for internal nozzle for fuel injectors. The volume of the gaseous phase obtained from the solution includes both the vapor and non-condensable gases. Initially, no vapor is present in the computational domain. However, non-condensable gases are present in the liquid fuel in very small mass fractions ($N_2 = 10^{-5}$; $O_2 = 10^{-5}$). The outlet domain is initialized with chamber pressure, chamber temperature, and only nitrogen. The eight holes and the counter-bores are initialized with chamber pressure, fuel temperature, and 99.998% liquid fuel and trace amounts of non-condensable gases, as described previously. The needle sac and the region upstream of the sac up to the pressure inlet are initialized with injection pressure, fuel temperature, and 99.998% liquid fuel and trace amounts of non-condensable gases. Such zone-wise initialization assists in achieving reasonable numerical stability in a two-phase compressible flow, as shown by previous nozzle flow simulation studies that used this code [24]. Simulations are transient, and all calculations were run until both the inflow and the outflow stabilized.

Solution Procedure

The governing equations were discretized using the finite volume approach. The Rhie-Chow algorithm was adopted to collocate all the transport variables at the cell center. The pressure implicit with splitting of operator (PISO) algorithm is used for pressure-velocity coupling, since it is expected to work well for unsteady compressible flow problems [23]. For the momentum equation second-order central differencing was employed, and for other transport equations a first-order upwind scheme was applied. A successive over-relaxation (SOR) algorithm was observed to offer better numerical stability. Volume averaging was used to estimate cell-representative viscosity and thermal conductivity. More details about averaging methods are available in our previous publication. The flash boiling problem