

Numerical Simulation of Hollow-Cone Sprays Interacting with Uniform Crossflow for Gasoline Direct Injection Engines

2011-24-0007

Published
09/11/2011Jian Gao
Univ of WisconsinMario F. Trujillo
Univ. of WisconsinSuraj Deshpande
Univ of Wisconsin

Copyright © 2011 SAE International

doi:10.4271/2011-24-0007

ABSTRACT

The interaction of fuel sprays with in-cylinder air flow is crucially important for the mixture preparation and subsequent combustion processes in gasoline direct injection (GDI) engines. In the present work, the experimentally validated computational fluid dynamics (CFD) simulations are performed to study the dynamics and physical insight of hollow-cone sprays interacting with a uniform crossflow. The basis of the model is the standard Reynolds-averaged Navier-Stokes (RANS) approach coupled to the Lagrangian treatment for statistical groups (parcels) representing the physical droplet population. The most physically suitable hybrid breakup models depicting the liquid sheet atomization and droplet breakup processes based on the linear instability analysis and Taylor analogy theory (LISA-TAB) are used. Detailed comparisons are made between the experiments and computations in terms of spray structure, local droplet diameter and velocity distributions. The computational results reveal the important features of the hollow-cone fuel spray in crossflow: the spray axis deflection by the crossflow is identified; the small droplets in the spray are displaced by the crossflow and transported to the downwind side; the vortical structure of the spray on the impacted side is significantly suppressed; there is a blockage effect of the spray on the crossflow in the near-nozzle field, and the counter-rotating vortex pair (CVP) owing to the interaction between the crossflow and the fuel spray is clearly captured. Furthermore,

test cases with different crossflow velocities and at elevated ambient temperature are carried out to study the effects on spray characteristics and mixture formation. With higher crossflow velocity, the results show that the development of CVPs become slowly along the flow field, the secondary breakup is enhanced, and therefore a more uniform spray with smaller mean droplet diameter is produced. Additionally, the cumulative fuel vapor mass increases and the low temperature field within the normal spray cone shrinks, which means that the fuel/air mixing, fuel vaporization, and air utilization could be improved in the combustion chamber of GDI engines.

INTRODUCTION

The gasoline direct injection (GDI) engine is considered to be a promising alternative to the conventional port fuel injection (PFI) gasoline engine. The potential advantages of GDI engine reflect in their enhanced fuel economy, higher power output, better transient response, and lower cold-start hydrocarbon (HC) emission levels [1, 2, 3]. Generally, the GDI engine combustion systems could be classified as wall-, flow-, or spray-guided concepts. Regardless of whichever concept is utilized, the fuel injection system is a key component to be matched with the specific in-cylinder flow field to provide the desired air-fuel mixture over the entire operating range of the engine [4, 5]. First of all, the fuel injector for GDI engines has evolved from side-oriented swirling type injectors (first generation) to multi-hole

injectors (second generation), and recently developed outwards opening piezo-actuated injectors (third generation) [6, 7, 8, 9]. Except for the second type, the injector for GDI engines features a hollow-cone spray with or without the “pre-spray” slug. Furthermore, in-cylinder airflow is usually organized in GDI engines, mainly including swirl movement-induced by valve heads and phased opening of the inlet valves-and tumble movement-generated by intake ports and augmented by pent roofs [10]. Both of them cause the interaction of fuel sprays with crossflow, which is crucially important for the mixture preparation and subsequent combustion processes.

Abundant research on liquid jet/column in crossflow has been done for characterizing the jet structure, dynamics, mixing, and primary breakup mechanisms experimentally and theoretically. Some representative works include Smith and Mungal [11], Sallam et al. [12], Linne et al. [13], Brown et al. [14], Ng et al. [15], etc. The key characteristics for liquid jets in crossflow are the penetration or breakup length and liquid column trajectory [16]. The dimensionless parameters that govern the jet behavior are the liquid Reynolds number (Re_l), jet momentum ratio (J), and crossflow Weber number (We_g).

Concerning the interaction between fuel spray and crossflow, very few studies have dealt with this topic, and most previous studies for investigating the spray characteristics were focused on free sprays using a constant-volume chamber (i.e. a spray injected into a quiescent ambience). It was reported that the maximum in-cylinder average velocity could be about 15 m/s at the motoring condition in a typical GDI engine when the fuel is injected late in the compression stroke at low load [5, 17]. This value can be even higher under firing condition. Apparently in such conditions, a spray injected can no longer be treated as a free spray. Fortunately, some experimental data concerning the fuel spray in crossflow has become available by means of laser and optical diagnostic technologies, such as the works for diesel spray by Desantes et al. [18, 19], and the works for gasoline spray by Choi et al. [5] and Nouri et al. [10]. Spray axis deflection, droplet size distribution, and air entrainment are considered to be important aspects for the spray in crossflow configuration. Parallel to these investigations, Ghosh and Hunt [20] have developed analytical models for spray jets in crossflow. They concluded that different patterns of air flow and droplet dispersion occur at different distances from the nozzle depending on the ratio of the air speed in the induced air jet to the relative cross-wind speed.

Although these works have provided useful information for understanding the spray behaviors in crossflow, the physics controlling the fuel-air mixture formation has yet to be thoroughly investigated. The work presented in this paper is therefore a study of interaction between hollow-cone fuel sprays and a uniform crossflow using numerical simulations.

The objectives are to analyze the dynamics and physical insight of hollow-cone gasoline sprays interacting with crossflow based on the simulation results, and to further investigate the effects of crossflow velocity and elevated ambient temperature on spray/mixture properties, as well as to discuss the implications for fuel/air mixing and combustion in GDI engines.

NUMERICAL APPROACH

The numerical study of hollow-cone spray in crossflow was performed using an adapted version of the dieselFoam solver of OpenFOAM CFD code [21]. The dispersed (liquid) phase is solved using Lagrangian approach and the continuous (gas) phase uses Eulerian approach with two-way coupling with the particle field. For computational efficiency the particles are tracked in groups called “parcels”. In this work, turbulence was treated with the conventional RANS κ - ϵ model, which is one of the most popular approaches for the complex dispersed two phase flows in the engineering industry [22]. The equations governing the flow and detailed description of the solver are given in [23].

Among the spray sub-models, the break-up model is most important to predict droplet velocity and size distribution accurately. Physically, after the spray is injected through the nozzle, the droplet is detached from the intact liquid core of the fuel spray or liquid sheet, and then the droplet is disintegrated into smaller droplets. Strategies for directly simulating these processes are currently under development as evidenced, for instance, in the work of Menard et al. [24]. Nevertheless, for realistic ranges in both Weber and Reynolds number, these simulations are still significantly under-resolved to yield acceptable values for droplet sizes and the numerical costs are prohibitive (excessive large grids) [22]; hence, to account for the multi-mode atomization behavior, the hybrid break-up model is adopted in this study.

Generally, the hybrid break-up model is divided into three stages: liquid sheet formation, sheet break-up (primary break-up), and atomization (secondary break-up). A number of hybrid schemes have been proposed and described in the literature. To model the process of primary break-up, the WAVE (surface wave instability or Kelvin-Helmholtz instability) model [25] and LISA (Linearized Instability Sheet Atomization) [26] model are commonly utilized. Widely used secondary break-up models include TAB (Taylor Analogy Break-up) model [27], E-TAB (Enhanced-TAB) model [28], DDB (Droplet Deformation Break-up) model [29], RT (Rayleigh-Taylor instability) model [30], and I-TAB (Improved-TAB) model [31]. According to the investigations in [32, 33, 34], for the conical (hollow-cone) spray, the performance of the hybrid models is test case dependent. The hybrid LISA-TAB model gives relatively better predictions for the atmospheric pressure test cases, while the KH-RT model performs better at above-atmospheric pressure test

cases. Taking into consideration that all the computational conditions are at atmospheric back pressure, the LISA-TAB hybrid break-up model is therefore employed in this study.

Liquid Sheet Formation

It is assumed that a conical liquid sheet with length L_s and thickness h_0 is formed at the nozzle exit of the swirl injector. The h_0 is related to the mass flowrate $\dot{m}$ measured experimentally and the axial component of velocity at the nozzle exit u by conservation of mass:

$$\dot{m} = \pi \rho_l u h_0 (d_0 - h_0) \quad (1)$$

$$u = V \cos \theta = k_v \sqrt{\frac{2\Delta p}{\rho_l}} \cos \theta \quad (2)$$

where V is the total sheet velocity and is determined from injection pressure Δp , ρ_l is the liquid density, k_v is the velocity coefficient, and θ is the spray angle. Thus, the whole liquid sheet characteristics' parameters at the nozzle exit can be calculated.

Primary Break-up Model (LISA model)

The LISA model assumes that the droplet undergoes no break-up, no drag until it reaches the break-up length, which is given by [26],

$$L_s = V\tau = \frac{V}{\Omega_s} \ln\left(\frac{\eta_b}{\eta_0}\right) \quad (3)$$

where τ is the break-up time, the value of $\ln(\eta_b/\eta_0)$ is proposed as 12 based on the work of Dombrowski and Johns [35], and Ω_s , the maximum growth rate, is determined from the dispersion equation.

Ligaments are assumed to form from the sheet fragments at the point of breakup and their diameter d_L can be obtained from a mass balance,

$$d_L = \sqrt{\frac{8h}{K_s}} \quad (4)$$

where K_s , the wave number corresponding to the maximum growth rate Ω_s , is determined from the dispersion equation based on the linear stability [26].

Furthermore, the unstable ligaments will break into droplets when the unstable wave amplitude is equal to the ligament radius, and one drop will be formed per wavelength [35]. To calculate the most unstable wave number for a ligament, Weber's result for the break-up of a cylinder, viscous liquid column was used [26]. The diameter of a drop d_D is expressed as,

$$d_D = \frac{3\pi d_L^2}{K_L} \quad (5)$$

where K_L is the most unstable wavelength on the ligament and is given by,

$$K_L d_L = \left[\frac{1}{2} + \frac{3\mu_l}{2(\rho_l \sigma d_L)^{1/2}} \right]^{1/2} \quad (6)$$

where μ_l is the liquid dynamic viscosity, and σ_l is the gas-liquid surface tension coefficient.

Secondary Break-up Model (TAB model)

Later on, the droplets will undergo the secondary break-up and turn into small droplets. The TAB model is used for simulating this break-up process. The TAB model is based on the analogy between an oscillating and distorting droplet and a spring-mass system, and the droplet break-up is due to an increase in the amplitude of the droplet oscillation. The governing equation is,

$$\ddot{y} = \frac{C_F \rho_g u^2}{C_b \rho_l r^2} - C_k \frac{\sigma}{\rho_l r^3} y - C_d \frac{\mu_l}{\rho_l r^2} \dot{y} \quad (7)$$

where $y = x/C_b r$, x is the displacement of the equator of the droplet from its equilibrium position, and r is the radius of droplet. The break-up occurs if y is larger than one. After break-up, the Sauter mean diameter (SMD) of product droplets is obtained from the law of energy conservation as,

$$SMD = \frac{d_D}{1 + 2C_k C_b^2 / 3 + (4C_b^2 - C_v^2 C_b^2)(\rho_l r^3 / 6\sigma)\dot{y}^2} \quad (8)$$

The dimensionless constants in the TAB model are $C_b=0.5$, $C_d=5.0$, $C_k=8.0$, $C_F=1/3$, $C_v=1$.

Table 1. Test conditions.

Nozzle	
Hole diameter, D	0.44 mm
Length-to-diameter ratio, L/D	2.5
Ambient gas	Air
Temperature, T_a	312, 500 K
Pressure, p_a	0.1 MPa (atmospheric)
Crossflow velocity, U_a	6, 11, 16 m/s
Fuel	Isooctane (C_8H_{18})
Injection system	
Injection pressure, p_{inj}	3.0, 8.0 MPa
Injection quantity, Q_{inj}	24.2, 39.8 mg
Injection duration, t_{inj}	3.2 ms

TEST CONDITIONS AND COMPUTATIONAL DETAILS

The test conditions in this study are summarized in Table 1. Two test cases at different injection pressures (3.0 and 8.0MPa) were selected to validate the models on hollow-cone spray predictions in crossflow. Both cases used the isooctane (C_8H_{18}) as the test fuel. The injection duration was fixed at 3.2 ms corresponding to the fuel delivery of 24.2 and 39.8 mg. The crossflow velocity was 11 m/s, and the ambient pressure and temperature were set to 0.1 MPa and 312 K based on the experimental conditions [10]. On the basis of the model validation, test cases with different crossflow velocities of 6 and 16 m/s and at an elevated ambient temperature of 500K were carried out to study the effects on spray characteristics and mixture formation. For all cases, the injection rate was assumed to be constant within the injection duration. The average initial spray cone angle was set to be 45° for the main spray based on the spray images [10]. The corresponding spray cone thickness (angle) was selected to be 5° by trial-and-error to optimize predictions of local droplet AMD and velocity.

A uniform Cartesian hexagonal three-dimensional mesh was used in the calculation. The sizes of the calculation domains were equal to 500 mm in length and $100 \times 100 \text{ mm}^2$ in plane area. The sprays were assumed to be injected at a location 4 mm below the top face of the mesh. Grid-size and parcel number sensitivity were first tested, and the mesh with cell sizes of $2.0 \times 2.0 \times 2.0 \text{ mm}^3$ and 50,000 parcels were employed in all computations. The hydrodynamic time step was automatically adjusted with the upper limit that the maximum Courant number would not exceed 0.4 for the numerical convergence. In addition, the Sac spray and droplet collision and coalescence were not considered for all computations in this study.

RESULTS AND DISCUSSION

Model Validations

Prior to the model validations, two consistency issues on droplet collision and drop drag calculations were addressed. These are considered to be source of inaccuracies associated with the point particle approach, as indicated by Salewski and Fuchs [39]. First, the inter-particle spacing (IPS) was calculated in order to identify the importance of droplet-droplet interactions in the region of interest. A similar method with [40] was used in the present study. The results of representative cases show that the IPS in the majority of the spray region is large enough (>10) despite a region near the nozzle where the IPS is as small as ~ 2 . Accordingly, droplet-droplet interaction can be ignored, and the collision model was deactivated in all computations of this study. Secondly, if the droplet Weber number is large (>10), the droplet deformation has to be considered and it will introduce certain inaccuracies if no correction on drag coefficient is taken into account. The results of calculated Weber number indicate that the largest Weber ~ 20 occurs only within about 1mm of injection. The inaccuracies related to droplet deformation are thus restricted to a very small region in the vicinity of the injector.

Two sets of experimental measurements [10] for hollow-cone fuel sprays in crossflow were selected for model validation purposes. Basic experimental conditions are summarized in Table 1. The spray structure, penetration length, local droplet AMD (D_{10}) and velocity distributions are considered in the comparisons.

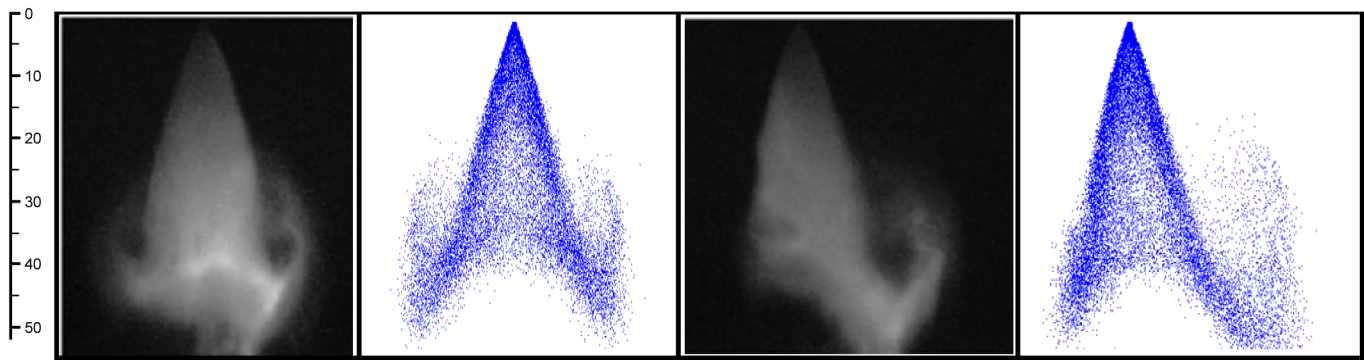


Figure 1. Comparison of experimental and computed spray images at 1.5 ms after start of injection at injection pressure of 8.0 MPa ($Q_{inj}=39.8$ mg, $p_a=0.1$ MPa, $T_a=312$ K). From left column to right column: experimental and computed spray images without crossflow; experimental and computed spray images with crossflow ($U_a=11$ m/s).

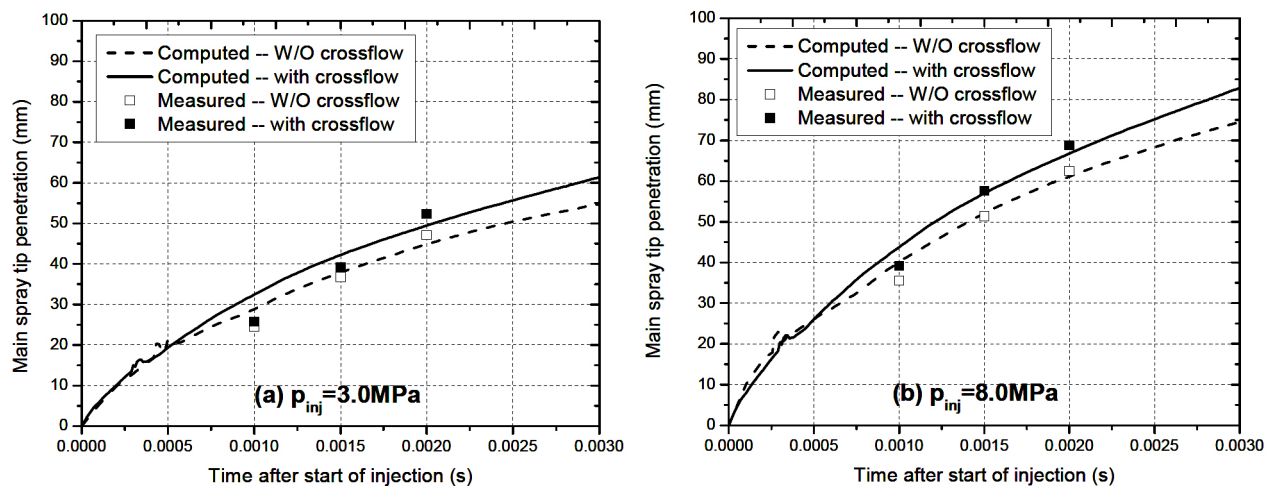


Figure 2. Comparison of computed (lines) and experimentally measured (symbols) main spray penetrations at two injection pressures (a) $p_{inj}=3.0$ MPa, $Q_{inj}=24.2$ mg, $p_a=0.1$ MPa, $T_a=312$ K, $U_a=11$ m/s; (b) $p_{inj}=8.0$ MPa, $Q_{inj}=39.8$ mg, $p_a=0.1$ MPa, $T_a=312$ K, $U_a=11$ m/s.

Figure 1 shows the experimental and computational spray images at 1.5 ms after start of injection at injection pressure of 8.0 MPa with and without crossflow. Note that only the droplets within the 1mm thickness slice in the mid-plane are displaced in the computational images for a consistent comparison with the laser measurements [10]. The agreement between the experiment and computation in the spray development is reasonably good except that the Sac spray was not considered in the computation. Without crossflow (free spray), as shown in Fig. 1, an axisymmetrical cone-shaped spray is formed. The vortex clouds are clearly observed around the periphery at the leading edge of the main spray. Previous experimental studies [36, 37] showed that the rotating vortical structure is induced by the gas motion due to the local pressure difference and the increased resistance of the spray to the ambient air with the increase of spray width. The entraining flows interact with the spray droplets and tend to carry the small droplets to form the vortex clouds.

Referring to the computed spray images, the formation of two separated vortices can be explained as follows: there exist two recirculation zones owing to the spray motion as depicted in the gas velocity vectors fields [38]. The peripheral droplets break up quickly due to the larger slip velocity. The small droplet produced only has a small amount momentum, and is surpassed by the succeeding droplets and left behind by the spray front; meanwhile, they easily follow the gas vortex flow. With crossflow, the axisymmetrical cone-shaped spray tends to collapse. The asymmetry of the spray increases with time as the droplets lose their momentum. The smaller droplets are displaced and deflected in the crossflow direction. As a consequence, the vortex cloud in the impacted side vanishes and is hard to be distinguished from the main part of the spray. On the contrary, a stronger vortical structure can be seen in the leeside.