ABSTRACT

The spray plume geometry and fuel atomization characteristics of the gasoline direct-injection (GDI) multi-hole injectors are of paramount importance with respect to the GDI engine homogenous-charge combustion and emission characteristics. A major component of the GDI combustion system development is the optimization of the spray-targeting and mixture preparation. Significant R&D efforts are directed towards optimization of the nozzle design and the manufacturing process in order to achieve optimum multi-plume spray characteristics.

The Volume-of-Fluid Large-Eddy Simulation (VOF-LES) of the injector internal flow and near-field primary atomization has been receiving attention as a tool to enable analysis of the influence of nozzle design on the key spray parameters and reduce reliance on hardware trial-and-tests for multi-objective spray optimizations. In combination with current state-of-the-art computational fluid dynamic (CFD) methods for simulation of spray atomization and transport processes, they afford a notable capability to expedite the injector valve-group and spray targeting optimization in order to reduce the time and costly hardware iterations.

The present publication reports studies of a GDI multi-hole injector valve-group and the corresponding spray plumes, with the aid of the current state-of-the-art CFD methods. The LES method is applied to study the structure and primary breakup of a single plume from a GDI multi-hole injector. A complementary study comprising the injector internal flow with the conventional Reynolds-averaged CFD method, coupled to the simulation of the spray atomization and transport processes using the stochastic Discrete Droplet method, has been performed. The correlation of simulated spray geometry with experimental data is encouraging.

INTRODUCTION

The advance of GDI engine technology has imposed stringent requirements on the injector with respect to fuel metering accuracy, spray atomization and robustness against deposit buildup in the harsh combustion environment. The GDI multi-hole injector is required to deliver the exact desired spray plume trajectory, plume cone angle, optimum spray atomization (for minimum spray penetration), in addition to uniform hole-to-hole volumetric flow rate. These requirements, in conjunction with the need for adaptation of the injector static flow and spray plume pattern to suit specific engine geometry and in-cylinder charge motion, has rendered the injector a key component of the GDI system. In turn, it has rendered the valve group a crucial part of the injector, as both the static flow rate and the spray plume pattern are determined by the valve group, and the selection of geometrical parameters such as nozzle-hole diameter, necessary to achieve the required flow rate, simultaneously affect both the spray plume geometry and atomization.

The optimization of the multi-hole GDI injector spray plume pattern in GDI combustion systems is a crucial step of the combustion system development to meet the fuel consumption and emission objectives [1, 2]. The goal is to achieve optimum in-cylinder mixture uniformity, for the broadest engine load-speed operating conditions, with minimum spray-wall interactions. The definition of the overall best spray pattern with respect to fuel economy and emission is accomplished through a process of (a) defining candidate spray plume patterns (i.e. spray targeting) based on geometric constraints (i.e. avoidance of spray interaction with intake valves, liner, etc.) and allowances for potential spray plume deflections, due to intake charge motion interactions [3], (b) CFD simulations of in-cylinder mixture preparation in order to analyze the in-cylinder charge-spray interactions, mixture preparation quality and wall wetting for a selection
of engine load-speed conditions, and (c) injector hard-ware build and engine combustion test over a broad engine load-speed map. Hence, an accurate and general GDI spray model, capable of reflecting the influence of nozzle geometry on the spray structure and atomization, can significantly reduce the reliance on injector build-and-test iterations and expedite the combustion system development process.

Since it is not possible to decouple the metering function from the spray generation function, it is crucial to gain an understanding of the relations between the geometrical parameters of the GDI multi-hole valve group and the flow rate, spray trajectory and atomization characteristics \[4\] in order to be able to define a valve group that delivers optimized injector spray pattern and atomization to achieve the optimum fuel consumption and emissions. The capability to analyze the influence of nozzle hole design parameters on the spray plume characteristics with a computational method is of significant value in the GDI injector valve-group design optimization. It affords major reduction of the hardware iteration loop for spray targeting optimization, and enables prediction of the spray characteristics, a priori, for in-cylinder CFD simulations of mixture preparation.

The recent studies of the GDI conical and planar sprays have shown strong dependence of the spray structure and atomization on details of the injector valve group design \[5, 6\] that necessitate simulation of the injector internal flow in order to correctly predict the spray primary breakup structure. The experimental evidence indicates similar dependence of the spray plume structure on the geometric features of the GDI multi-hole nozzles, in addition to the expected influence of the pintle-sack geometry on the flow upstream of the nozzle holes. The implication is that accurate prediction of the spray macro-structure and atomization requires simulation of the injector internal flow in order to adequately describe the influence of nozzle geometry on the nozzle-exit liquid jet hydrodynamic conditions.

This paper describes investigations of the GDI multi-hole injector internal flow and spray simulations, with the aid of the multi-fluid VOF-LES and the CFD standard RANS computational methods. The VOF-LES method offers the largest capability for analysis of the GDI primary jet breakup and influence of the nozzle design parameters, but its industrial application is constrained by the substantial computational requirements. The Reynolds-averaged Navier-Stokes method is the CFD standard method adopted in all commercial CFD codes and is broadly employed for Eulerian multiphase analysis of the injector internal flow \[7, 8\]. This provides the nozzle-exit flow conditions required for adequate definition of the spray initial conditions within the frame-work of the Lagrangian “discrete droplet model” \[9\] for stochastic spray atomization and transport processes. The spray simulation results are compared with the experimental data, in order to ascertain the predictive accuracy of the computational method.

**INJECTOR NOZZLE-FLOW ANALYSIS**

**LARGE-EDDY SIMULATION METHOD**

**Conservation Equations of Multi-Phase System**

The mathematical model comprises the transport equations for conservation of mass and momentum of a two-phase flow system that is composed of two immiscible, compressible or incompressible, Newtonian fluids, and includes the interface surface tension. Accordingly, the single set of conservation equations that describe the flow of the two-phase mixture are \[5\]:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0
\]

(1)

\[
\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \cdot (\rho \mathbf{U} \otimes \mathbf{U}) = -\nabla P + \nabla \cdot \tau + \int_{S(t)} \sigma \kappa n' \delta(x - x') dS
\]

(2)

Where, \( \mathbf{U} \) is the velocity, \( \rho \) is the density, \( \sigma \) is the surface tension coefficient, \( \tau \) is the stress tensor, \( \kappa \) is the curvature of the liquid surface, \( n \) represents a unit vector normal to the liquid surface, and the operators \( \nabla \cdot () \) and \( \nabla () \) signify the gradient and the divergence operations, respectively. The integral term on the right-hand-side of equation (2) represents the momentum source due to surface tension: it acts only at the interface (indicated by the Dirac function \( \delta () \)) over the entire liquid surface \( S(t) \).

**The VOF-Based Interface Tracking Method**

The principle idea of the “Volume-of-Fluid” approach is that a two- (or indeed multi-) phase system can be represented as a mixture of phases in which the phase-fraction distribution includes sharp, yet resolvable, transitions between the phases. Accordingly, the interface in a two-phase flow system is computed with the aid of the transport equation for the liquid volume fraction as the indicator function to locate the interface. The transport equation for the phase fraction \( \alpha \) is:

\[
\frac{\partial \rho \alpha}{\partial t} + \nabla \cdot (\rho \mathbf{U} \alpha) = 0
\]

(3)
According to the definition of \( \alpha \), the mixture thermo-physical properties are calculated as:

\[
\rho = \alpha \rho_f + (1 - \alpha) \rho_g \\
\mu = \alpha \mu_f + (1 - \alpha) \mu_g
\]

The VOF interface capturing/tracking method is a simple and flexible approach to simulation of multi-phase systems with free surfaces, especially for circumstances that the surface tension effects are not dominant. The main problems of VOF method relate to the accuracy of the numerical approaches used to ensure that the interface remains sharp, yet not unduly affected by numerical dispersion, conservation and mesh alignment bias.

**Large-Eddy-Simulation Method**

The LES-VOF equations are derived from equations (1), (2), (3), through a process of local volume averaging of phase-weighted properties, that entails decomposition of the dependent variables into resolvable and sub-grid scales of turbulent fluctuations and application of a filter that removes the sub-grid scales fluctuations from direct simulation. The filtering process, in conjunction with the non-linear terms in equation (2), produce additional terms, involving correlation of the variable fluctuations at sub-grid scales that require closure through mathematical models. The most notable of these terms is the sub-grid-scale (SGS) stress that represents the effect of unresolved scales of turbulence on the momentum transport process and its viscous dissipation. The Sub-Grid-Scale stress is defined as:

\[
\tau_{sgs} = \overline{UU} - \overline{\overline{U}}
\]

The closure of the sub-grid-scale stresses is affected through a sub-grid-scale eddy-viscosity model that adopts a one-equation transport model for the sub-grid turbulent kinetic energy \( \kappa \), in conjunction with the Smagorinski's dissipation model. A detailed description of the adopted sub-grid scale models and the associated literature can be found in Ref. 5.

**The Numerical Solution Method**

The OpenFOAM finite-volume CFD code [10] is used for solution of the LES-VOF system of conservation equations. OpenFOAM incorporates special treatment of the phase-fraction transport equation in order to preserve a sharp liquid-gas interface, and employs temporal and spatial discretization schemes that are second order accurate but preserve the proper limits on physically-bounded flow variables.

**Simulation Domain and Mesh**

The simulation domain, boundary conditions and the computational mesh are presented in Figure 1. The computational domain is comprised of a segment of the injector valve-group geometry containing one nozzle hole and its immediate downstream ambient. The structured computational mesh consists of a 550k mesh of the injector valve group (seat, nozzle hole, counter-bore) and a 300k mesh of the ambient environment. The mesh distribution provides mesh resolution of O (3-5 \( \mu \)m) within the valve group and resolution of O(5-20 \( \mu \)m) in the ambient. As shown in Figure 1(b) the highest mesh resolution is in the vicinity of nozzle hole domain, with the mesh expansion (in the stream-wise and lateral directions) in the ambient domain. A systematic mesh refinement study was not performed, but increase of mesh resolution to 1.4M has not altered the significant fluid dynamic features of the jet breakup. The computational time-step is dynamically controlled during calculations, according to the Courant-Friedrichs-Lewy [11] stability criterion.

The flow boundary conditions are:

- Inlet: Uniform velocity profile, with a prescribed magnitude corresponding to the fuel system pressure.
- Circumferential symmetry planes: symmetric flow boundary conditions
- Ambient: uniform pressure boundary condition for all ambient boundaries

The pintle motion is not included in the simulation. Instationary simulation of the fuel injection is performed on the fixed geometry mesh, without representation of the pintle motion. To this effect, the simulation initial condition is that the valve-group - from the inlet to the pintle sealing band - is filled with liquid at rest (the remaining volume of the computational domain is air at ambient condition). At the inlet boundary, the liquid phase velocity is accelerated to its final value (corresponding to the nozzle hole static flow) over a short time interval (of order 2-3 \( \mu \)s) and then is kept constant.

The liquid phase physical properties in the simulation are those of n-Heptane, but evaporation is not considered.

**Simulation Results**

Figure 2 presents the LES simulations of a single plume of a GDI multi-hole injector for the fuel system pressure of 20MPa. The results show a non-uniform liquid flow at entrance of the nozzle hole. The streamlines highlight the influence of the pintle presence on the flow entrance condition to the nozzle hole, showing a predominant flow direction from the injector seat periphery (i.e. upstream of the
nozzle hole) and insubstantial flow contribution from the injector sac region.

The flow at the nozzle entrance and within the nozzle resembles a waterfall, with liquid phase separation at the nozzle entrance and a segregated two-phase flow within the nozzle hole, due to the small nozzle 1/d typical of the GDI multi-hole injectors. The LES results in Figure 2 show initiation of Kelvin-Helmholtz instabilities within the nozzle and most pronounced growth of the instabilities in the counter-bore domain. The simulation result indicates initiation of the liquid jet breakup in the downstream vicinity of the counter-bore exit plane.

The plots of instantaneous distributions of VOF and the velocity magnitude (at t = 25μs) on a display-plane through the centre of the nozzle hole are presented in Figure 3. These illustrate: (a) the flow separation at the flow-approach part of the nozzle entrance edge, (b) segregated two-phase flow in the nozzle, (c) initiation and growth of Kelvin-Helmholtz instabilities on the liquid jet free surfaces, especially within the counter-bore domain, (d) primary jet breakup in the vicinity of the counter-bore exit, (e) significant acceleration, accompanied by velocity perturbations of the liquid jet, at the entrance to the nozzle and downstream. Interestingly, the results show a deviation of the liquid jet trajectory from the nozzle-hole axis angle for this nozzle geometry, layout and fuel pressure. The simulation indicates two potential causes of the plume deviation angle, namely the lateral jet inertia due to flow separation at the nozzle entrance and the pattern of the jet primary breakup downstream of the nozzle hole. The phenomenon is in accord with the experimental data.
reported in the literature [4] (and the measurement data in Figure 10) and provides evidence of the spray plume deviation angle as a “nozzle” fluid dynamic phenomena dependent on the valve-group geometric parameters.

It worth noting that the “bubbles” shown in the VOF plot in Figure 3 (in the sac volume and nozzle entrance) is not due to cavitation, but due to the slow scavenging of the air in this region (at the start of simulation) by the liquid flow.

REYNOLDS-AVERAGED SIMULATION METHOD

The Mathematical Method for a Multi-Phase System

The commercial CFD code AVL-FIRE [12] is employed for the simulation of the multi-fluid (liquid, vapor, ambient air) flow within the injector and its near-field ambient environment. The computational method adopted in AVL-FIRE involves solution of independent, coupled sets of Reynolds-averaged conservation equations of a multi-fluid system. The set of conservation equations, for every fluid-stream, comprises of the Reynolds-averaged equations of the conservation of mass (of the components), momentum (the Navier-Stokes equations) and the total enthalpy [2]. In addition, transport equations for the k-epsilon turbulence model are solved to account for the stochastic effects of turbulence on the mass, momentum and enthalpy transport processes. The simultaneous solution of the three equation systems enables determination of the field-distributions of the liquid, the fuel vapor and the air volume fractions.

Commonly, this CFD method is referred to as Reynolds Averaged Navier Stokes (RANS) method, and is a standard CFD methodology for turbulent flow applications in industry. The main draw-back of the method, inherent in the Reynolds averaging methodology, is that the simulation results represent statistical averages of the flow variables, and therefore stochastical variations (such is shot-to-shot spray atomization variation) are assimilated in the simulation results. In addition, owing to the limitations of the turbulence models for complex flows (e.g. separated or rotational flows) the predictive accuracy of the method needs to be verified for the specific flow of interest.

Here, the results of isothermal three-fluid (liquid, vapor, ambient air) simulations of the injector internal and the nozzle downstream ambient domain are performed.
Computational Domain and Mesh

Figure 4 presents the solution domain and the computational mesh for the three-fluid simulation of the injector internal flow and the nozzle near-field down-stream ambient. The computational mesh is comprised of 900K cells - with spatial resolution of 10μm - in the injector domain and 300K cells in the ambient. The computational mesh spatial distribution is non-uniform: the highest mesh resolution is in the nozzle hole and counter-bore domains, with gradual successive mesh coarsening in the downstream ambient, as depicted in Figure 4. The “ambient domain” boundaries are defined at sufficient distance from the valve-group, to minimize potential influence of their prescribed boundary condition on the liquid phase solutions.

Initial and Boundary Conditions

The methodology for simulation of the flow and cavitation requires prescription of the liquid and the gaseous-phase(s) thermo-physical conditions for the initial conditions, and at the boundaries of the solution domain. With respect the gaseous phases, this entails additional prescription of the fluids’ physical conditions (c.f. bubble number density [12]) at the start of calculations and at the boundaries.

The initial conditions for all calculations are the injector internal flow domain filled with liquid, at rest, with nominal-zero (1. e-6) volume-fraction(s) of the gaseous phase(s). In addition, the ambient domain contains air at rest (with relevant thermodynamic conditions). The flow boundary conditions at inlet are prescribed static pressure, with nominal-zero vapor and air volume fractions. The boundary condition at the ambient domain surface is prescribed static pressure.

Single-hole Multi-Fluid Flow Simulation

Figure 5 presents the RANS multi-fluid simulation results for the single-hole geometry of Figure 1 on a display-plane through the axis of the nozzle hole, which corresponds to the display-plane for presentation of VOF-LES results in Figure 3. The results pertain to a fuel pressure of 20MPa.

The contour plot of the statistical average VOF distribution at \( = 25 \mu s \text{ ASOI} \), in Figure 5, is similar to the instantaneous results of VOF-LES simulation in Figure 3. It depicts the flow separation at the nozzle entrance and a two-phase (liquid - air) flow in the nozzle and the counter-bore domains. The notable difference is the absence of prediction of the jet primary breakup process. This is due to the significant inherent physical and numerical momentum diffusion of the methodology employed and lack of an accurate diffusion-preventive liquid-air interface tracking method. The contour plots of velocity show flow acceleration at the entrance and through the length of the nozzle, concurrent with the reduction of the flow area indicated by the VOF contour plot. However, the diffusive effect of turbulent viscosity (and numerical discretization scheme) is substantial to smear the significant velocity gradients within the liquid jet, and at its interface with the ambient air, predicted by the VOF-LES simulations in Figure 3. In addition, the RANS simulation does not predict the jet - nozzle axis deviation angle for this nozzle layout / geometry.

In spite of these differences, the RANS and VOF-LES method predict closely similar valve-group hydrodynamic flow parameters (nozzle static flow rate, pressure loss, discharge coefficient) in close agreement with the experimental data, providing indirect support that the predicted mean liquid-phase flow pattern and velocity distribution are correct. This suggests that the RANS simulation of the injector internal flow may be sufficient for
describing the conditions of the issuing liquid jet for simulation of the ensuing spray.

**Complete Injector Multi-Fluid Flow Simulation**

*Figure 6* presents the RANS multi-fluid simulation of the complete injector internal flow, for a fuel pressure of 10 MPa. This fuel pressure is the condition of the experimental spray data that are used for assessment of the computational method in the succeeding section. The liquid phase in the simulation is n-Heptane, with the cavitation enabled.

As evident in *Figure 6*, the injector valve-group has a 6-hole symmetric pattern. The contour plots of VOF and velocity distribution at the nozzle and counter-bore exit planes illustrate the extent of hole-to-hole variations of the liquid jet mass flow rate and the stream-wise velocity profile, hence the hole-to-hole variations of the liquid jet momentum flux. The transformation and lateral spread of the VOF distributions between the nozzle exit and at the counter-bore exit planes are indicative of the extent of the liquid jet - ambient air interactions within the counter-bore domain.

**COUPLED NOZZLE FLOW - SPRAY SIMULATION**

**METHODOLOGY AND COMPUTATIONAL MESH**

With the objective of a simulation methodology potentially capable of accurate prediction of the spray plume structure of a GDI multi-hole injector, the VOF, mean velocity and turbulence distributions at the injector nozzle exits, obtained from the injector internal flow simulation, are utilised for prescription of the spray initial conditions in the Lagrangian “discrete droplet method” (DDM) for stochastic spray simulation [9].

The spray simulation method incorporates adapted WAVE spray atomisation models [12, 13] for the primary and secondary atomisation, including mechanism of the droplet shedding process. The initial droplet conditions for individual plumes are calculated from their respective liquid-phase condition (the liquid core geometry, velocity and turbulence intensity) at the injector counter-bore exit plane, from the RANS injector internal flow simulations. The VOF-LES simulation of the valve-group is utilised to estimate the initial “blob” size distribution.
The spray simulation replicates the experimental set-up for the “patternator” measurement of the spray foot-print. Figure 7 depicts the spray simulation domain, which contains a portion of the ambient (partitioned by a cylindrical surface) that surrounds the injector nozzles and the spray, and extends beyond the patternator plate (included in order to account for effect of its presence on the spray).

**SIMULATION RESULTS AND COMPARISON WITH DATA**

Figure 8 presents the simulation of the spray plume geometry, atomization, spray impingement on the patternator plate and formation of the liquid film footprints. In the present simulation, the droplet-wall interactions are disabled, thus all droplets that impinge the patternator plate will adhere to it and form liquid foot-prints associated with the plumes.
The simulation results pertain to n-Heptane fuel and injection conditions of fuel system pressure 10 MPa, a pulse-width of 1.5 ms and a spray simulation time of 2.5 ms.

The trajectory and cone angle of individual spray plumes are predicted using the corresponding nozzle exit velocity profile and turbulence intensity data, from the injector internal flow simulation, and the common WAVE primary and secondary atomization models. Thus they present a measure of the predictive capability of the simulation method. The results in Figure 9 present a comparison of the predicted spray plume geometry with the optical imaging data, from two orthogonal view directions. The spray cone angles are defined according to the SAE J2715 [14] from the spray imaging data and superimposed on the simulation results. There is good agreement, with respect to the plume cone angles (especially up to 40 mm from nozzle) and the spray total angle. There is indication of smaller lateral dispersion of spray plume fronts (penetration > 40 mm) in the simulation: the potential sources of this discrepancy are the atomization model, induced air entrainment (prohibiting dispersion) or vaporization of the droplets within the periphery of the plumes (comparison of the injected and collected fuel on the patternator, in both experiments and simulation, indicate significant spray vaporization).

Figure 10 presents a comparison of the experimental spray plume foot-prints, measured with the patternator at Z=50 mm downstream of the injector tip, with the simulations. The experimental spray plume foot-prints present accumulation of many injections repetitions (∼ 500), so assimilate the shot-to-shot variations of the spray plume trajectories. In addition to the foot-prints, the locations of intersection of the corresponding nozzle-hole axes trajectories with the patternator plane are marked. Figure 11 provides the corresponding quantitative comparison of the measured and predicted spray plume-centre trajectory angles.

Several aspects of the results are notable:

- Extent and pattern of deviations of the measured plume foot-print centres from the corresponding nozzle-hole axes

---

**Figure 7. Computational mesh (1/4 of the complete domain) for the Lagrangian spray simulations**

**Figure 8. Simulated spray plume geometry and interaction with the patternator plate**
trajectory intersection locations, illustrating the influence of the valve-group geometry (nozzle layout, angle, etc.)

- Good to moderate agreement between the measured and predicted location and geometry of the plume foot-print centres, depending on the lay-out of the nozzle holes. The discrepancy in the foot-print size is in part due to under-prediction of the plume front dispersion (indicated in Figure 9) and in part due to the shot-to-shot variation of the plume trajectories in the experiment (causing enlargement of the foot-print).

- Good agreement of the measured and predicted hole-to-hole variation of the injected mass (as indicated by the liquid film height), taking into consideration the manufacturing tolerances (evidenced by absence of the x-coordinate symmetry of the plume foot-prints in the experimental data)

- The simulated spray plume trajectories are aligned with the nozzle-hole axes trajectories. Hence, the largest discrepancy between the predicted and measured foot-print centre locations is for the plumes with largest deviation of the plume centre from the associated nozzle-hole axis trajectory location. Therefore, although the spray-aerodynamic and gravitational-field interactions could play a role, the expected primary cause of this discrepancy is the deviation of the liquid jet trajectory at the counter-bore exit from the nozzle-hole axis.

Overall, the results provide evidence of the predictive accuracy of the methodology for simulation of the spray geometry, hole-to-hole injected mass distribution and the individual plume trajectory and cone angle. The main shortcoming of the methodology stems from the evident inability of the RANS method to reproduce the deviation angle between the issuing liquid jet and the nozzle hole axis, depending on the valve-group geometric parameters. Further investigation of this topic is in progress.

CONCLUSIONS

The VOF-LES results highlight the specific flow characteristics of the GDI multi-hole valve group. The influence of pintle presence on the non-uniform flow separation at entrance to the nozzle is notable. The results show significant velocity gradients (hence turbulence and vorticity generation) caused the flow separation at the nozzle entrance and within the core of the liquid jet in the nozzle. This is responsible for the pressure loss and low discharge coefficient of the nozzle hole, but it promotes the liquid jet breakup. The VOF-LES simulations show initiation and growth of the Kelvin-Helmholtz instabilities, caused by the liquid jet - air interactions within the counter-bore space, and the ensuing jet primary breakup in close downstream vicinity of the injector.
The results highlight the potential capability of the VOF-LES method for analysis of the hydrodynamics of liquid jet breakup and the influence of the GDI multi-hole valve-group design details on the spray breakup / atomization process. The capability of VOF-LES to predict the deviation angle between the nozzle-hole axis and the issuing liquid jet trajectory is notable. The RANS counter-part simulation demonstrates that it predicts the nozzle flow separation and the significant hydrodynamic variables (pressure drop, nozzle discharge coefficient, etc) closely similar to the VOF-LES method and the experimental data. The notable shortcoming of the RANS method is the evident inability to predict the jet trajectory deviation from the nozzle hole-axis.

A corollary of the combined LES / RANS simulation activity is that the discrepancy between measured and predicted spray

![Figure 10. Comparison of the predicted and measured spray plume patternator foot-prints (location, geometry and liquid-film heights)](image1)

![Figure 11. Quantitative comparison of the predicted and measured plume foot-print centre of gravity)](image2)
plum foot-print geometry is likely associated with the inability of the RANS simulation of the injector internal flow to predict the deviation angle between the nozzle hole axis and the issuing liquid jet.

The coupled Eulerian-Lagrangian method for spray simulation has shown potential for correlation of the GDI multi-hole valve-group design and the important features of the spray plume geometry and atomization. The goal is to enable analysis of the spray targeting - from the GDI multi-hole valve-group geometry to the in-cylinder mixture formation - through CFD simulation, prior to injector hardware build and dynamometer combustion tests. Further evaluation and refinement of the components of the methodology are under progress.

REFERENCES


ACKNOWLEDGMENTS

The authors would like to thank N. Mastro and L. Markle of Delphi Technical Center Rochester for support of the experimental investigations and valuable exchanges.