The Spray Characteristics Investigation of GDI Injector using One-way Coupling CFD Approach

Xingyu Xue

Delphi Technologies

Abstract: Injector sac-nozzle flow and spray characteristics play a crucial role in search of GDI engine combustion and emission processes. Significant R&D effects are directed towards understanding of nozzle flow dynamics and issuing spray atomization process. One-way coupling CFD approach allows Eulerian internal nozzle flow simulation to be effectively coupled with Lagrangian spray simulation, served as an effective tool to enable injector seat-nozzle optimization and understand the micro-scope spray behaviors that would otherwise be difficult to observe from experimental techniques. This paper presents studies of internal nozzle flow and spray behavior of a Delphi GDI multi-hole injector using one-way coupling CFD approach. The results illustrate this coupled approach offers a good quantitative agreement with experimental observation. The spray characteristics such as the plume trajectory, cone angle, fuel atomization and penetration are directly impacted by the turbulence level and vortex structures from the internal sac-nozzle flow. The spray hole orientation angle has a strong impact on the vortex-driven atomization mechanism, and therefore influence the plume atomization performance, targeting and nozzle mass flow rate. The fuel spray plumes are deflected primarily due to the aerodynamic recirculation created by high velocity jets into the ambient. The plume deflection tends to be stronger with increased injection pressure. The use of this sophisticated modeling approach paves the ways to understand the fundamental physics, correlate the injector seat-nozzle design to key spray characteristics and reduce reliance on hardware trial-and-tests for spray optimization.

Key words:GDI, CFD Simulation, Converge, In-Nozzle Flow, Spray Atomization, One-way Coupling

1.Introduction

The increasingly stringent regulations on engine fuel consumption and polluting emissions reduction drive more and more advanced internal combustion engines development. In particular, recent regulations such as EURO 6c and China Stage 6 (CN6) concerning gasoline engines set a new emissions cycles with expanded test conditions. The particulate number limit for gasoline fueled vehicles is introduced at $6*10^{11}$ #/km. In addition to the tests performed on the engine test bench, a further functional approval (In-Service Conformity (ISC)) during vehicle operation with extended altitude and ambient temperature is required. Gasoline direct injection (GDI) engines are attributed for improved fuel economy and engine performance over port fuel injection (PFI) systems [1]. The GDI engines introduce the fuel spray directly into the engine cylinder, promote charge cooling thanks to heat absorption by fuel evaporation, lead to a better volumetric efficiency with possibility of higher compression ratio and efficiency. The GDI engines could enable both stratified-charge and homogeneous modes of operation, which is not possible with port injection [2].

The shape of the spray that forms in the GDI combustion chamber and the resultant atomization process are of uttermost importance to ensure the proper preparation of a homogeneous or stratified blend, reduce and control the surface wetting of the combustion chamber. The spray characteristics including accurate fuel metering, desired spray plume trajectory, plume cone/dispersion angle, optimum spray penetration are stringently imposed for the engine being designed. These requirements, in conjunction with the need for adaptation of the injector static flow and spray plume pattern to suit the specific engine in-cylinder charge motion, require significant effort on injector seat and spray nozzle design [3][4].

While experimental diagnostics on spray characteristics has been achieved extensively. These available experimental techniques generally fail to capture the subtle flow transience under gas-liquid interface scales, and not able to characterize the field turbulence and the complex vortex structure in the sac and nozzle region. A better understanding of the fundamental physics with the micro-scopic behavior of spray evolution can be revealed using advanced computational fluid dynamics (CFD) approach. Lagrangian spray simulation has been broadly adopted to investigate the spray atomization process [5][6]. The spray simulations in the engineering environment today are typically based on Discrete Droplet Method (DDM) [7]. The involved multi-phase flow phenomena, the momentum, heat and mass transfer require the numerical solution in a Lagrangian framework. The discrete parcels which are collections of drops are introduced to the domain at the injector with initial conditions of position, size, velocity and temperature. The parcels undergo several physical processes. The primary and secondary breakup, drop drag, collision and coalescence, turbulent dispersion and evaporation are all captured using a comprehensive set of models. The standard blob injection model can be initialized using rate of injection measurement (ROI). Two popular instability phenomenon as Kelvin-Helmholtz (KH) and Rayleigh – Taylor (RT) instabilities [8] are widely adopted to model the spray breakup behavior. The breakup constants are proposed and tuned against experiments.

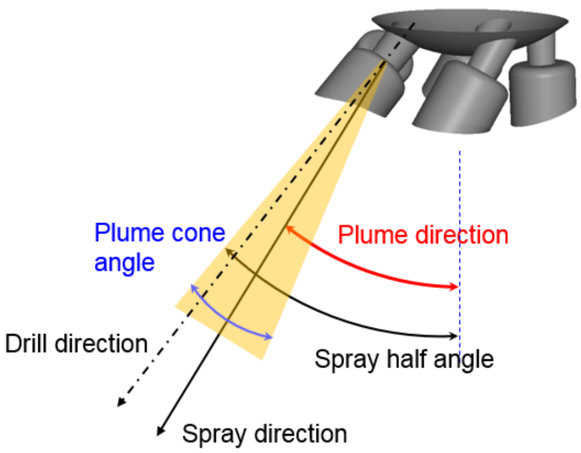


Figure 1. Schematic representations of the GDI spray

The typical drawback of Lagrangion spray simulation is the uncertainty of initial conditions of the introduced parcels as point sources. Several key spray parameters are illustrated schematically in Figure 1. The spray direction has been numerically and experimentally observed to be biased from the drill direction. The plume cone angle is directly linked to the nozzle flow dynamics. The droplets, emerging from a nozzle as a spray, are a gas/liquid mixture. The standard Lagrangion spray simulation studies [6][9] commonly tune these parameters to meet the experimental observations. The hole to hole variation, however, cannot be captured. These simulation activities therefore lose the generality without consideration of in-nozzle flow impact on ensuing spray evolution process.

An effective method treating the parcels exiting the nozzle physically is expected to offer a more realistic spray atomization prediction. The mass, momentum and enthalpy of Lagrangian parcels could be inherently initialized from the frame of the Eulerian gas flow field. The fully coupled in-nozzle flow and spray simulation in name of one-way coupling is evidently virtual for GDI injector research [10][11]. This approach allows the solution at nozzle exit from Eulerian internal nozzle flow simulation to be applied as the input for ensuing Lagrangian spray simulation. The detailed understanding on how the nozzle flow dynamics influences the spray atomization can be revealed, enabling the transfer of individual engine requirements on the spray to specific sac-nozzle design. This paper presents a detailed case study using high-fidelity one-way coupling method. The investigation is based on a 5-hole Delphi real-application injector seat. In the following sections, the internal & near nozzle flow simulation study is first implemented to provide the nozzle-exit flow conditions required for spray initial conditions. The Lagrangian spray simulation is performed for a spray issuing into a quiescent environment. The simulation results are compared with the experimental data to ascertain the predictive accuracy. Afterwards the investigation on how the nozzle flow dynamics and vortex structure influence the spray formation process is presented. The spray characteristics are discussed with different injection pressures and ambient conditions.

2.Internal & Near Nozzle Flow Simulation

In-depth analysis of internal & near nozzle flow simulation could provide insight into the nozzle flow dynamics, vortex structures and its influence on ensuing spray evolution. The internal nozzle flow involves complex multi-phase and multi-scale fluid dynamic phenomena, including turbulence, cavitation and their interactions. Accurate capture of these phenomena could provide detailed, high fidelity nozzle exit boundaries for the Lagrangian spray simulation.

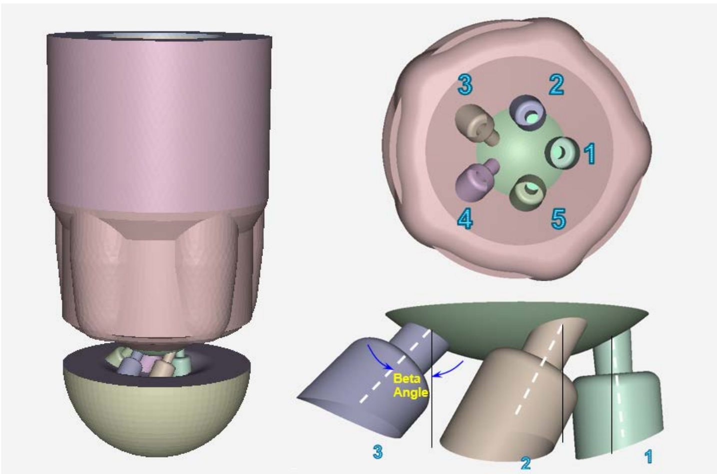


Figure 2. Layout and computation domain of Delphi 5-hole injector valve-group assembly

A 5-hole Delphi real-application injector for sided mounting GDI engine application is used for this study. Figure 2 presents the seat layout and three-dimensional computational domain which comprises the injector valve-group flow domain and its immediate near-field ambient to capture the near-nozzle flow features. Five symmetric stepped holes are purposefully designed with different orientations. This allows optimization of the spray for good fuel-air mixing while avoiding impingement of liquid fuel on solid surfaces such as the intake valves, piston top or cylinder wall. The beta skew (bend) angle is defined as the angle between the nozzle drill direction and injector axial direction.

Table 1 tabulates the simulation conditions explored in this study. Test fluid – n-heptane at 90C temperature is injected with 100bar, 200bar and 350bar fuel pressure, respectively. 350bar represents the highest GDI fuel pressure level since Delphi Multec 14 injector was introduced as the first to the market. The next generation 600bar capable Delphi GDI system is on progressing with the purpose to further reduce the fuel consumption and engine out emissions. The ambient conditions are simulated as 1bar 20C, 2bar 70C and 3bar 70C respectively. The ambient conditions explored in this study fairly represent the normal operation conditions of turbo-charged homogenous GDI gasoline engine in the market. The extreme conditions including spray flash boiling scenario with extraordinary low ambient pressure and some advanced engine applications (such as stratified spray-guided, gasoline compression ignition) with very late injection surrounding high ambient pressure are not covered in this study.

Table 1.Simulation conditions investigated for this study

Fuel	n-heptane
Fuel Temperature, C	90
Injection Pressure, bar	100, 200, 350
Ambient Pressure, bar	1, 2, 3
Ambient Temperature, C	20, 70, 70

The commercial CFD code, Converge®, is used for this analysis. The internal & near nozzle flow simulation is carried out in the Eulerian framework. Reynolds Averaged Navier-Stokes (RANS) with standard k-ε turbulence model is applied to predict the chaotic and unstable internal fluid flow behavior. The cavitation is activated and treated by using Volume of Fluid (VOF) approach [12], which tracks and locates the free-surface of a particular phase in each cell. The Homogeneous Relaxation Model (HRM) that represents the phase transition by estimating the time scale of phase change has been coupled with VOF approach. The computational mesh is of the order 2 million cells and affords a spatial resolution in the range 16 μm (within the injector seat-nozzle and ambient domain) to 256 μm (within the ball upstream domain). All critical areas including sac, nozzle and near nozzle regions are fine meshed to better capture the resolution of energy containing turbulent motions. Fixed embedding is applied near the wall to capture the boundary layer phenomena with sharp gradients of velocity. The transient needle lift is applied to simulate the injection process from start of fueling until the needle becomes stable at the nominal stroke position. Table 2 summarizes the sub-models to be used for nozzle flow simulation and Figure 3 presents the computational domain meshing with vertical cut plane.

Table 2.Summary of nozzle flow modeling methodology

Code	Converge
Framework	Eulerian
Cavitation	Yes
Turbulence	Standard k-ε
Ambient Properties	Ideal Gas
Liquid/Gas Interface	VOF
Cell Type	Cut-cell Cartesian

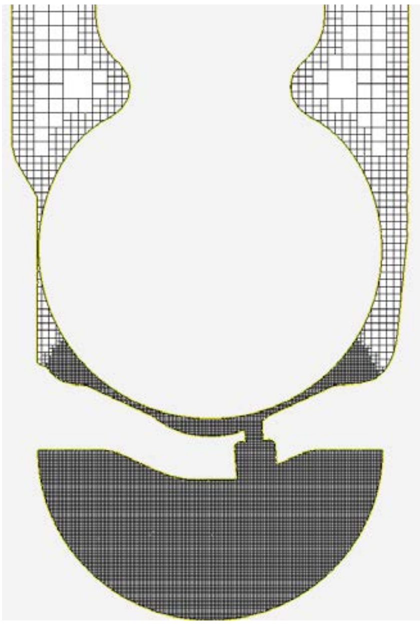


Figure 3. Computational domain meshing with 16um minimum grid.

2.1 Effect of Grid Resolution

The sensitivity of grid resolution on Eulerian simulation is performed with minimum grid at 32 um, 16 um and 8 um respectively. The mass flow rate comparison is shown in Figure 4. The mass flow rate increases following the needle movement, experience a small step decrement at ~ 0.115 ms due to the needle bounce at nominal stoke position. The mass flow rate tends to be stable after 0.12 ms. Finer grid resolution slightly reduces the mass flow. The discrepancy between 16um and 8um is negligible. It is worth noting that grid resolution down-selection is a balance between computational accuracy and efficiency. Figure 5 examines CPU time hours per 0.1 ms using 32 cores and total cell count with different minimum grid sizes. As a result, 16um mini grid size with approximately 2 million cells is a reasonable choice in consideration of both model accuracy and the computational cost.

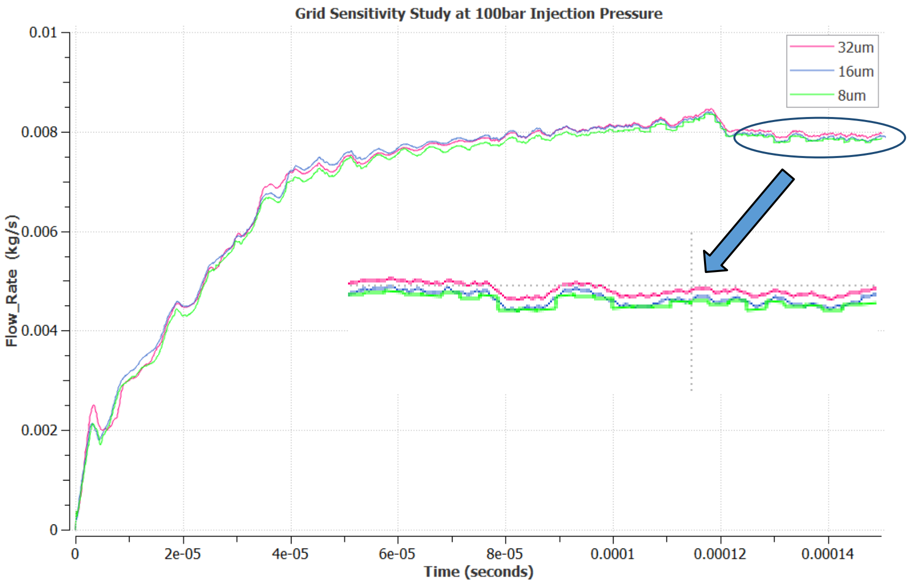


Figure 4. Mass flow rate as a function of time with different grid resolutions (100bar injection pressure)

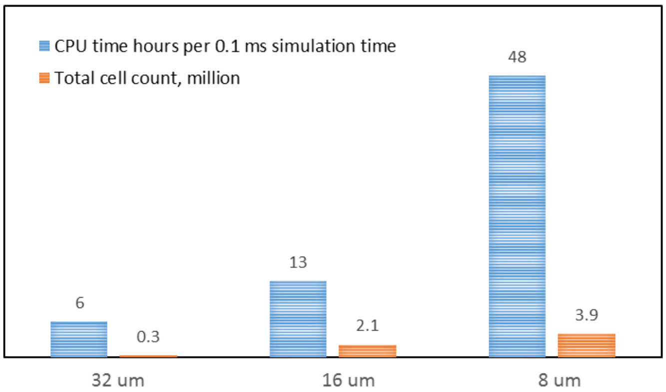
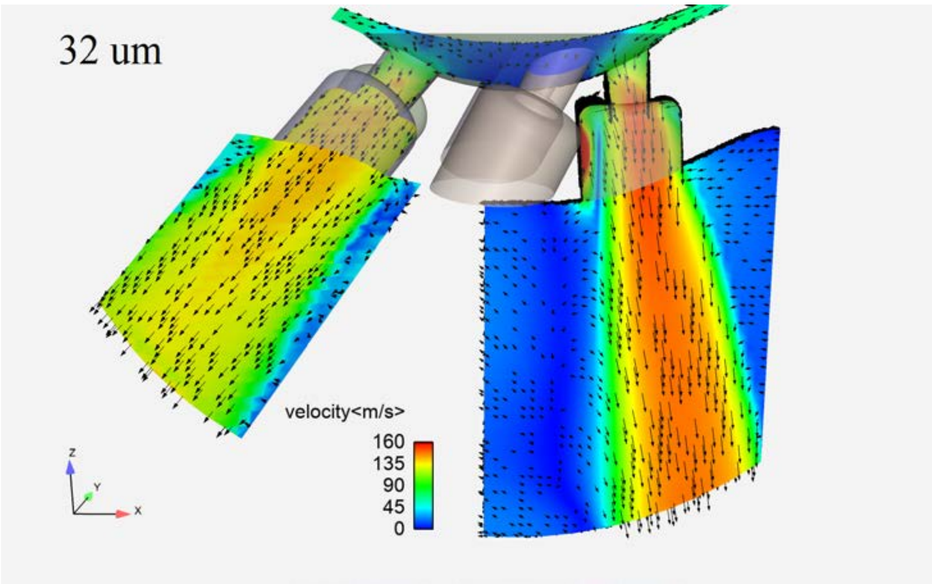


Figure 5. CPU time and total cell count as a function of grid size using 32 cores (100bar injection pressure)

A comparison for the internal & near nozzle spray at 0.15ms after start of fueling (aSOF) with different grid resolutions and the corresponding velocity vector, served as an indicator for spray and chamber gas interaction, are presented in Figure 6. The hole 1 and hole 3 velocity clip cuts are presented. As expected, the spray is biased from drill angle which is consistent to other researchers' findings [13][14]. Backflow of chamber gas into the counter-bore is found at hole 1 with low beta angle, illustrating the partial hydraulic flip phenomenon, which will influence the ensuing spray. It is evident that 32 um minimum grid is not enough to capture the spray features in the near-nozzle region. The grid size within the injector seat-nozzle and ambient domain should be at least 16um to precisely guide the plume targeting and cone angle which are critical to Lagrangian calculation of GDI spray.



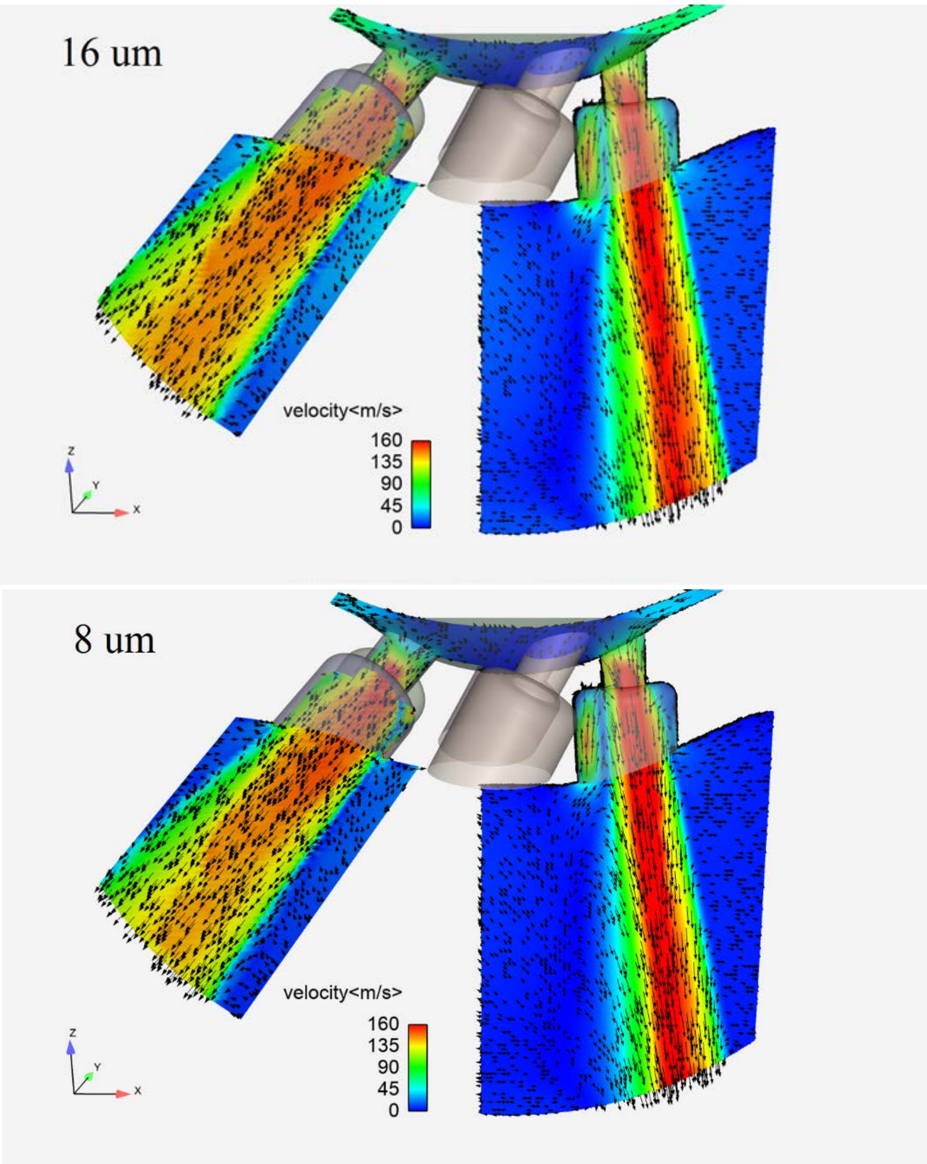


Figure 6. Spray velocity visualization from the Eulerian simulation with different grid resolutions at 0.15 ms aSOF (100bar injection pressure)

2.2Nozzle Flow Characteristics

The characteristics of nozzle flow dynamics, vortex structure and cavitation level in 1bar and 20C ambient condition are investigated in this subsection. The contour plots of liquid volume fraction at nozzle exits are presented in Figure 7. In this case, the hole to hole variation is clearly observed. The apparent two phase flow at the nozzle exits result from vapor cavities in the liquid as a consequence of rapid changes of pressure, associated with the ingestion of ambient air into the nozzle due to partial hydraulic-flip phenomena. The vapor distribution also indicates the occurrence of strong swirling vortex structures. The individual hole orientation has a significant impact on the liquid volume fraction. Higher beta angle hole presents elevated vapor volume fraction. This phenomenon involves the local deposition of energy since strong generated vortex attached with high beta angle causes higher phase change into gas. The impact of injection pressure on nozzle cavitation is not as significant as hole orientation.

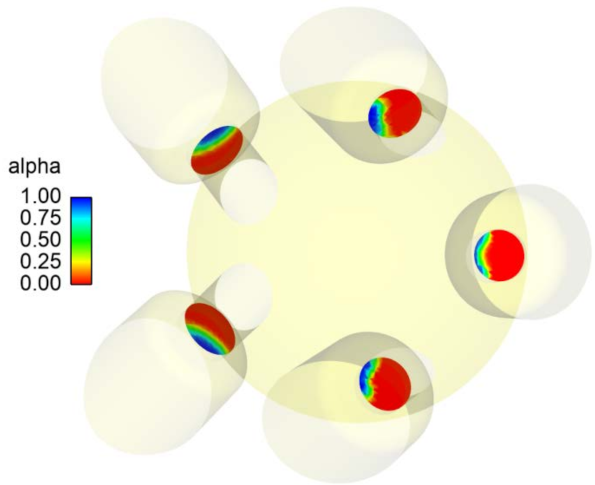


Figure 7. Contour plots of liquid volume fraction at nozzle exits at 0.15 ms aSOF (100bar fuel pressure)

Table 3 summarizes the individual hole discharge coefficient C_d at different injection pressures. The equation of C_d calculation is presented as below：

$$C_d = \frac{\dot{m}}{\rho_{liq} A_{minD} \gamma_{ber}}, \qquad \gamma_{ber} = \sqrt{(2\frac{\Delta P}{\rho_{liq}})}$$

where ΔP is the pressure difference between sac and ambient chamber. Consistent with the observed liquid volume fraction at nozzle exits, higher beta angle leads to lower discharge coefficient for each injection pressure level. A suitable adjustment of beta angle and nozzle hole size is therefore necessary to optimize the spray targeting and mass distribution for engine combustion chamber development.

Table 3. Calculated individual hole C_d at different injection pressures

Nozzle	Beta Angle	100bar P_{inj}	200bar P_{inj}	350bar P_{inj}
1	4.3°	0.733	0.744	0.746
2	31.8°	0.678	0.686	0.689
3	50°	0.652	0.655	0.658
4	50°	0.652	0.655	0.658
5	31.8°	0.675	0.683	0.687

The turbulence level and vortex structure inside the sac region are vital for spray formation, especially the primary breakup mechanism. The streamline and liquid volume fraction contour plots are presented in Figure 8. Conspicuous vortex structures are formed upstream each nozzle inlet region. The vortex-induced atomization mechanism is illustrated. The vortex tends to be increased with higher beta angle, producing stronger atomization level (as indicated by the streamline separation) in hole 3 and 4. Besides the spray atomization, the vortex is a significant driving mechanism for spray targeting. Varying injection pressure impacts the vortex structure and their intricate interactions, leading to the spray targeting variation downstream of the nozzle exit. Since the vortex-controlled atomization mechanisms are of utter importance for spray optimization, the sac volume and nozzle geometry can be purposefully adapted to build up the desired vortex structure for spray optimization, improve the mixture preparation with reduced surface wetting for engine applications.

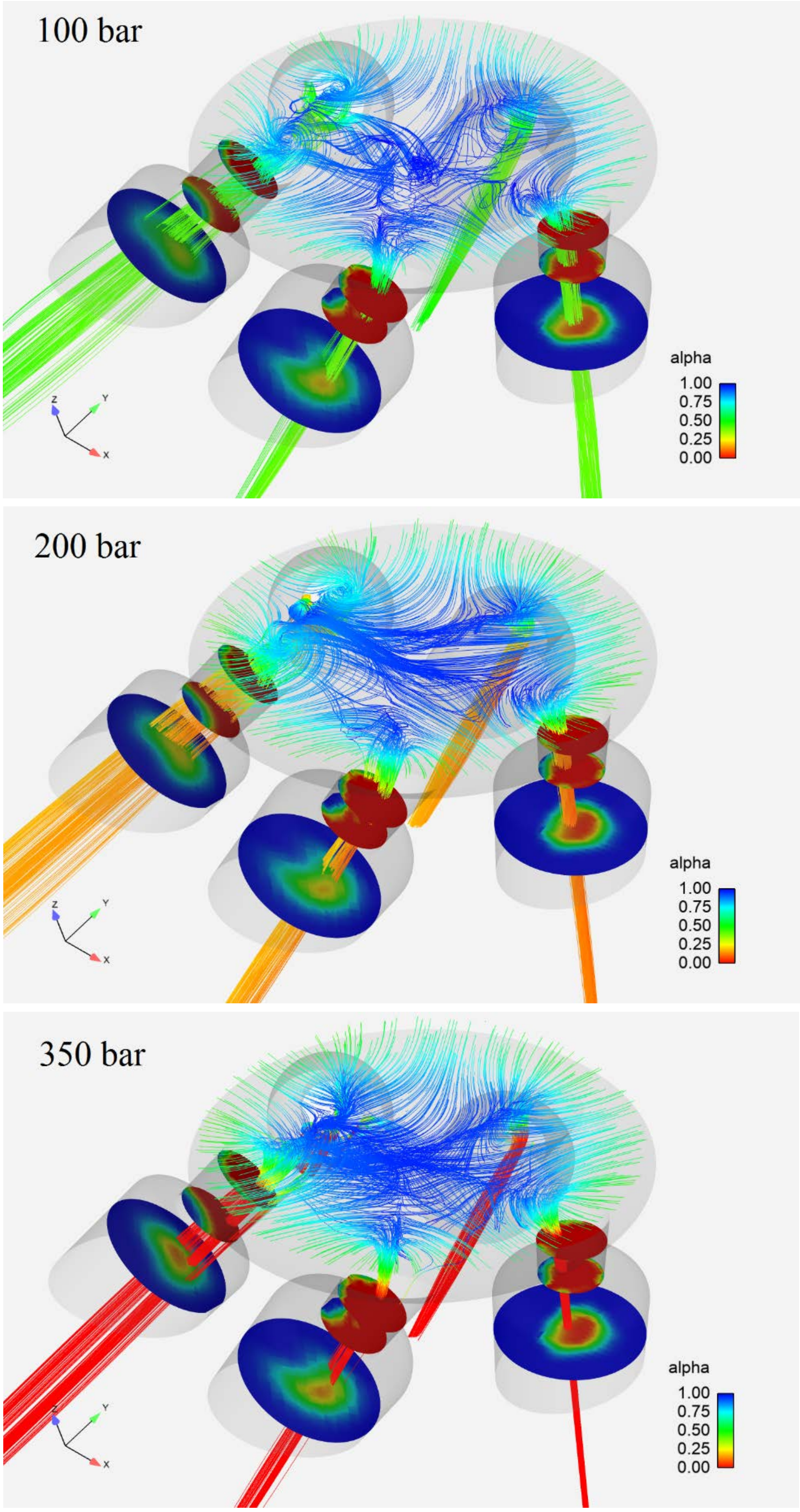


Figure 8. Streamline and liquid volume fraction over injector sac and nozzle for different injection pressures at 0.15 ms aSOF

3.Spray Simulation using One-way Coupling

One-way coupling approach is to strategically provide the detailed nozzle exit information from Eulerian simulation to Lagrangian spray simulation. This approach allows accurate capture of hole-to-hole variation, spray trajectory and the spray dispersion (spray cone angle). In this section, the spray validation against experimental data is first illustrated. The spray characteristics analysis results are presented. Finally the effect of injection pressure and ambient conditions on spray atomization are discussed.

3.1Spray Validation

The Lagrangian spray simulation is carried out by creating a spray box with injector placed at the top face of the spray chamber. Velocity and temperature adaptive mesh refinement (AMR) is activated to reconstruct the mesh according to local gradients of velocity and temperature during injection [15]. The fixed embedding is imposed around the injector.

The significance of turbulence modeling has been emphasized for both diesel and gasoline spray simulations [10][16]. The standard k-ε turbulence model is adopted in this study. $C_{\varepsilon 1}$ is recommended as 1.35 for GDI spray prediction by Saha and Quan [10]. The spray jet breakup and droplet collisions are predicted by the KH-RT instabilities generated by the shear, aerodynamic forces and density difference between two fluids [17]. Kelvin-Helmholtz model uses a liquid jet stability analysis to model the atomization process of relatively large injected parcels. Rayleigh-Taylor model describes breakup according to the Rayleigh-Taylor instabilities on the surface of each drop. Four important breakup constants are identified from KH-RT modeling. Large body of spray simulation work has been dedicated towards the study of the sensitivity of each parameter to derive the optimum combination. Detailed information can be found from reference [6][18]. The state of art model constants from other researchers and the settings used in this study are tabulated in Table 4. It is observed that the finalized model constants from each research lean towards consensus. The RT model size constant, which determines the size of spray droplets from RT breakup, has been identified as the most influential parameter on spray penetration and morphology prediction. The discrepancy on this parameter may result from the uncertainty of initial conditions of parcels as point source.

Table 4. Summary of breakup constants for spray simulation

	Kancherla (SAE 2016-28-0007	Braga (SAE 2017-36-0360)	Current Settings
RT Model Size Constant	0.25	0.5	0.6
RT Model Time Constant	1	1	1
KH Model Time Constant	7	7	5
Breakup Length Constant	0	0	0

The spray axial penetration comparison, together with the corresponding instantaneous spray morphology are presented in Figure 9 and Figure 10. It is worth noting that the axial penetration is defined as the distance between injector tip to the leading edge of spray from 2-D image plane, referring to the injector axis. The comparison results demonstrate the one-way coupling approach has a good quantitative agreement with the experimental observation. With the precise information on plume targeting, spray cone angle and gas liquid mixture from Eulerian simulation, the spray simulation is able to well capture the effect of injection pressure on spray atomization characteristics, showing close agreement with the measurement. The proper tendency can be clearly identified.

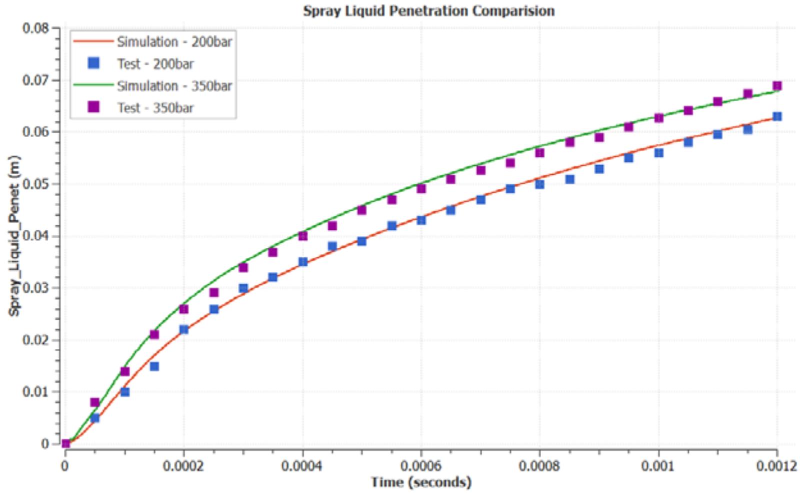


Figure 9. Spray axial penetration comparison between test and simulation at 1bar, 20C ambient condition.

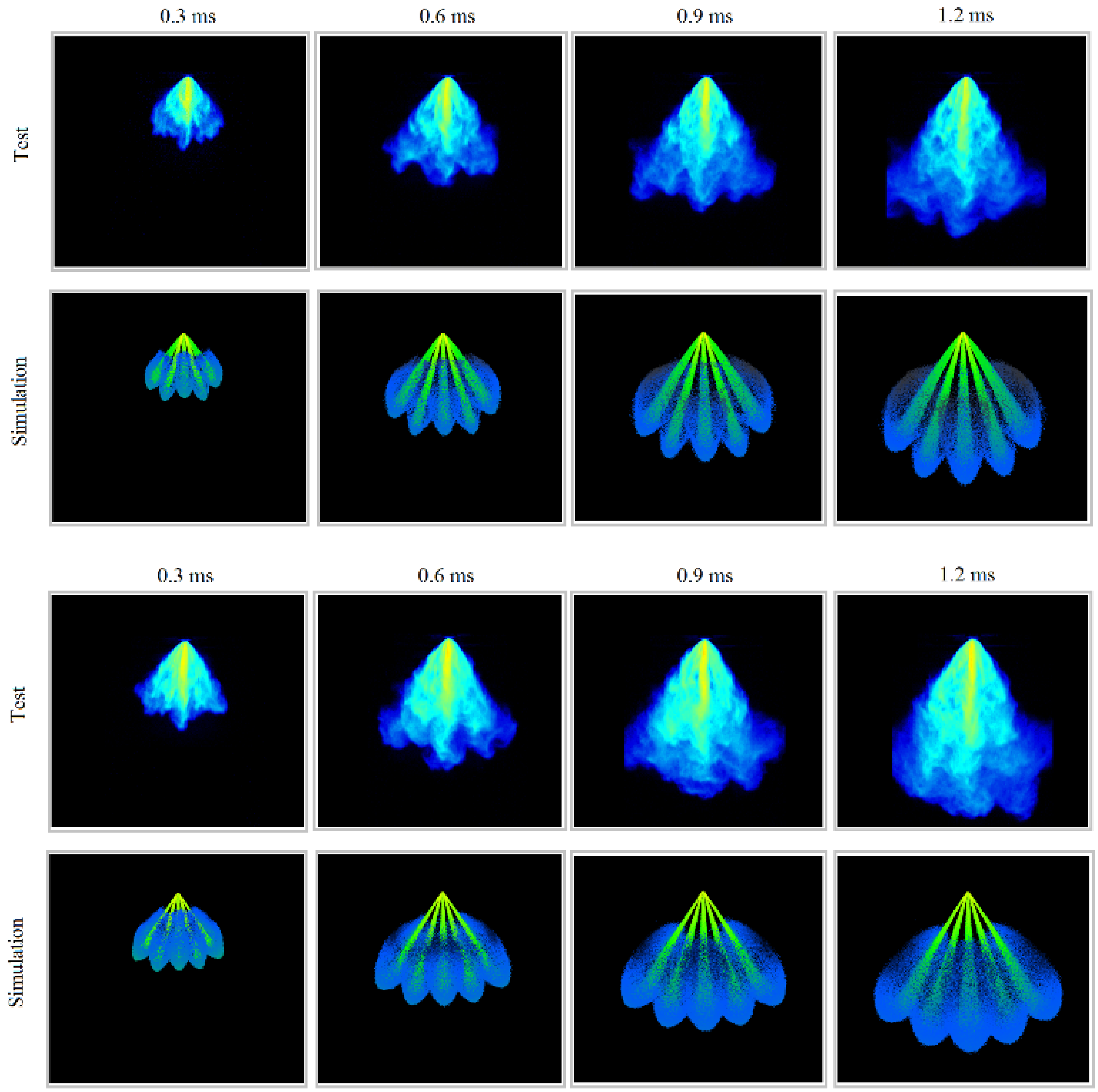


Figure 10. Spray morphology comparison between test and simulation at 1bar, 20C ambient condition (top: 200bar fuel pressure, bottom: 350bar fuel pressure).

3.2Analysis of Spray Characteristics

The aim of one-way coupling approach is to improve spray modeling prediction capability, correlate the spray characteristics with the in-nozzle flow dynamics. The detailed spray morphology with two different views is shown in Figure 11. It is evident that spray dispersion /cone angle, which is physically determined by the vortex-triggered nozzle flow turbulence associated with primary breakup and ligament formation, is vital for the spray atomization process. Table 5 summarizes the individual spray cone angle, which was an important tuning factor for standard blob injection model, is now based on imposed boundary from Eulerian simulation. Consistent with the streamline and liquid volume fraction visualization in Figure 8, hole 3 and 4 present higher dispersion angle with better atomization, the plume penetration is correspondingly mitigated. The results re-emphasize the importance of vortex driven atomization mechanism development for GDI injector seat development.

Besides the biased spray stream leaving injector nozzle due to the in-nozzle flow dynamics, the spray trajectory is deflected by the aerodynamic recirculation created by high velocity jets into the ambient. The shearing forces acting on the surface of the fuel stream and wave interaction are pronounced to generate a backward flow mechanism, which significantly effects the spray atomization and spray pattern. As shown in Figure 11, the jet 3 and 4 are deflected towards inside and the jet 1 is deflected towards left from the side view.

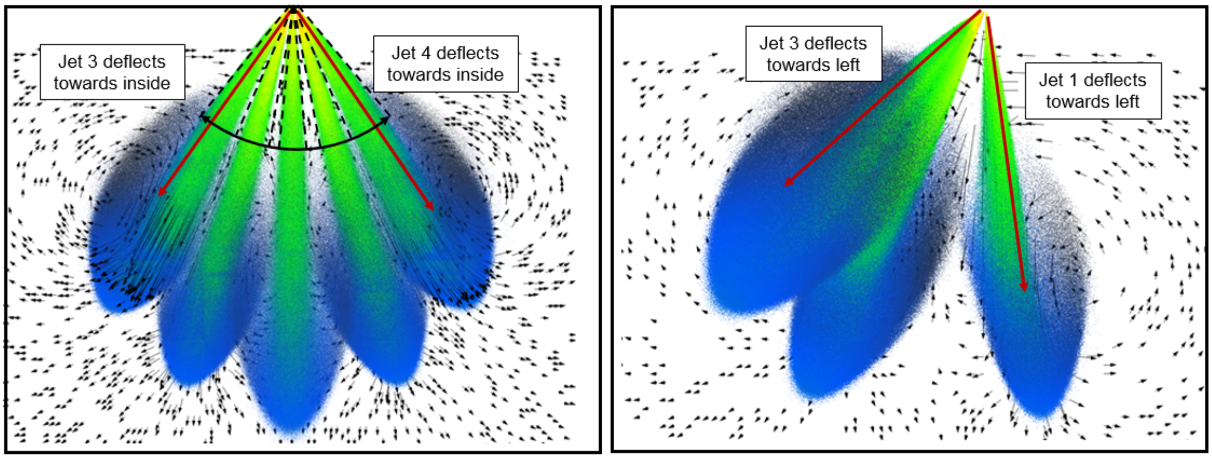


Figure 11. Spray morphology at 1.5 ms aSOF (200bar fuel pressure, 1bar and 20C ambient condition).

Table 5. Individual spray cone angle (200bar fuel pressure, 1bar and 20C ambient condition).

Nozzle	Beta Angle	Spray Cone Angle
1	4.3	10.8
2	31.8	12.6
3	50	17.8
4	50	17.7
5	31.8	12.7

It is relatively straight forward to see that the spray pattern is highly impacted by the in-nozzle flow dynamics and aerodynamic recirculation phenomenon. The state of art methodology for spray pattern assessment is to use a high-resolution spray patternator [19]. In this study, the patternator at Z = 40 mm downstream of the injector tip is applied to collect the liquid drops from many consecutive fuel spray pulses, assimilate the shot-to-shot variations of the spray plume trajectories. A user defined function (UDF) of virtual patternator model was developed and embedded to validate the spray pattern data. The UDF is designed to calculate the accumulated liquid volume as the droplets impinge a user defined location, leading to a formation of the liquid footprints. Figure 12 presents a comparison of the experimental spray pattern with the simulation. In addition to the foot-prints, the locations of centroid of each plume and seat geometry target along the hole drill axes are displayed as a Cartesian coordinates. The patternator tests were carried out using several injector samples and the pattern discrepancy stems from the manufactory tolerance and measurement errors. As expected, the spray pattern is deviated from design target, showing a good agreement with experimental data and physical observation from spay morphology. Since, at present, the GDI injector seat design draws an significant empirical knowledge in order to meet the multi-objective spray requirements such as spray pattern and penetration. The one-way coupling CFD approach could provide the engineering guidance to reduce the reliance on hardware trial-and-tests for spray optimization.

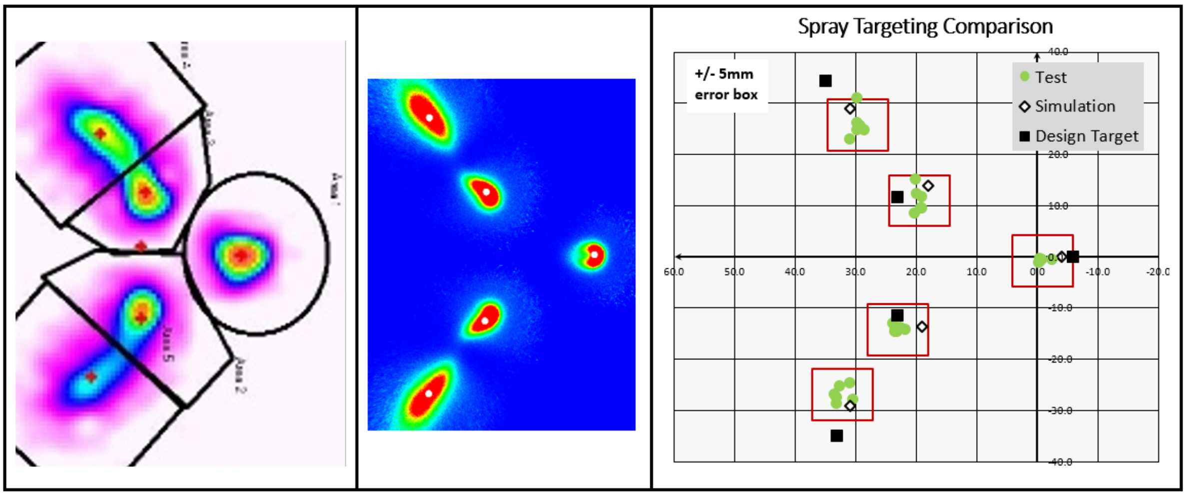


Figure 12. Spray plume pattern comparison between simulation and measurement (40 mm away from injector, 200bar injection pressure, 1bar and 20C ambient condition).

3.3Sensitivity of Injection Pressure and Ambient Conditions

In the section, the effects of injection pressure and ambient condition on spray trajectory, atomization and penetration are characterized. Figure 13 shows the spray axial penetration and saunter mean diameter (SMD) with different ambient conditions. The most noticeable trends are the decrease in penetration and SMD with an increase in ambient pressure and temperature. These trends have been observed by many others [20].

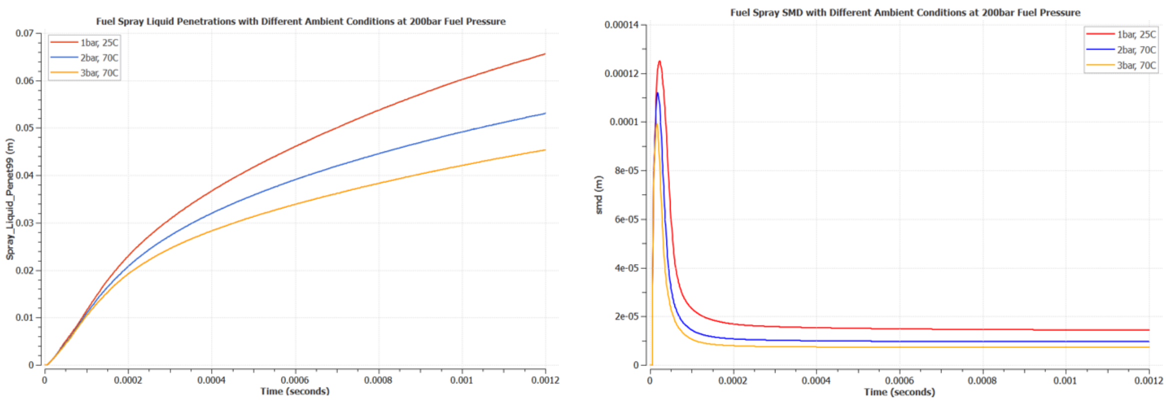


Figure 13. Fuel spray axial penetration and SMD at different ambient conditions (200bar fuel pressure).

Support for and explanation of the observed penetration and SMD behaviors can be investigated from spray morphology in Figure 14. For this purpose, the pressure contour plot and velocity vectors are demonstrated as well. Potential factors contributing to the reduced penetration and SMD are the evaporation of ballistic drops at the leading edge and redistribution of their momentum through mixing. The increased ambient pressure and temperature enhance the spray dispersion and evaporation, leading to more entrained air in the spray. Larger entrained mass slower the penetration velocity and enhance the droplet breakup mechanism.

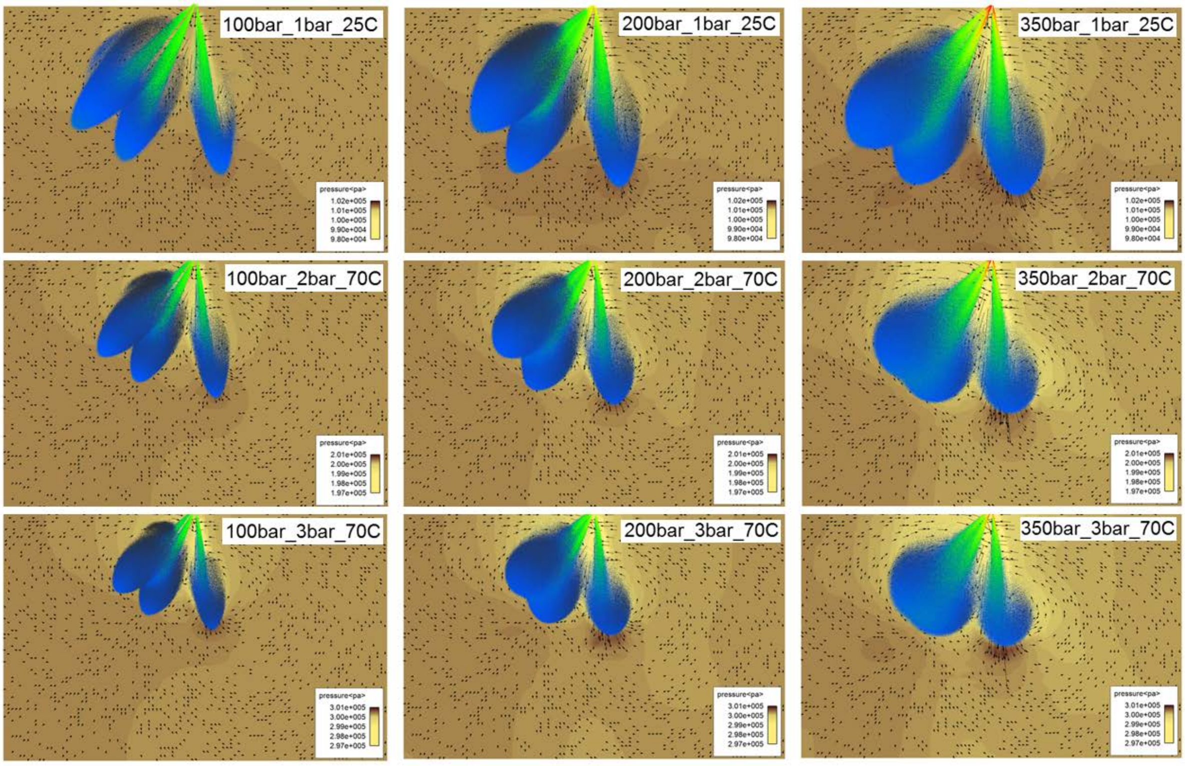


Figure 14. Spray morphology at different conditions

As explained above, aerodynamic recirculation is generated due to the entrainment of high velocity jets. The domination factor is the backward pressure gradient shown in Figure 14. A volumetric deflection occurs as the ambient gas is entrained into the spray. The overall magnitude of fuel jet deflection is more pronounced with increased injection pressure. The spray momentum increases proportionally to the available fuel pressure increase, sharpens the pressure gradient and results in increased aerodynamic recirculation structure and more air entrainment.

Figure 15 presents the spray clip cut at 30 mm downstream of injector tip for each operating condition. The Cartesian coordinates of plume centroids are shown in Figure 16. These spray clips visualize the spray droplet breakup, jet to jet interaction, and pattern information. It is observed that higher ambient conditions or higher injection pressure are advantageous to spray jet breakup and promote fuel atomization performance with better air-fuel mixing. Considerable plume-plume interaction with elevated droplet collision is presented for higher fuel pressure or ambient conditions cases. The insightemphasizes the high density and high injection pressure benefits for next generation of advanced, high efficiency GDI gasoline engines.

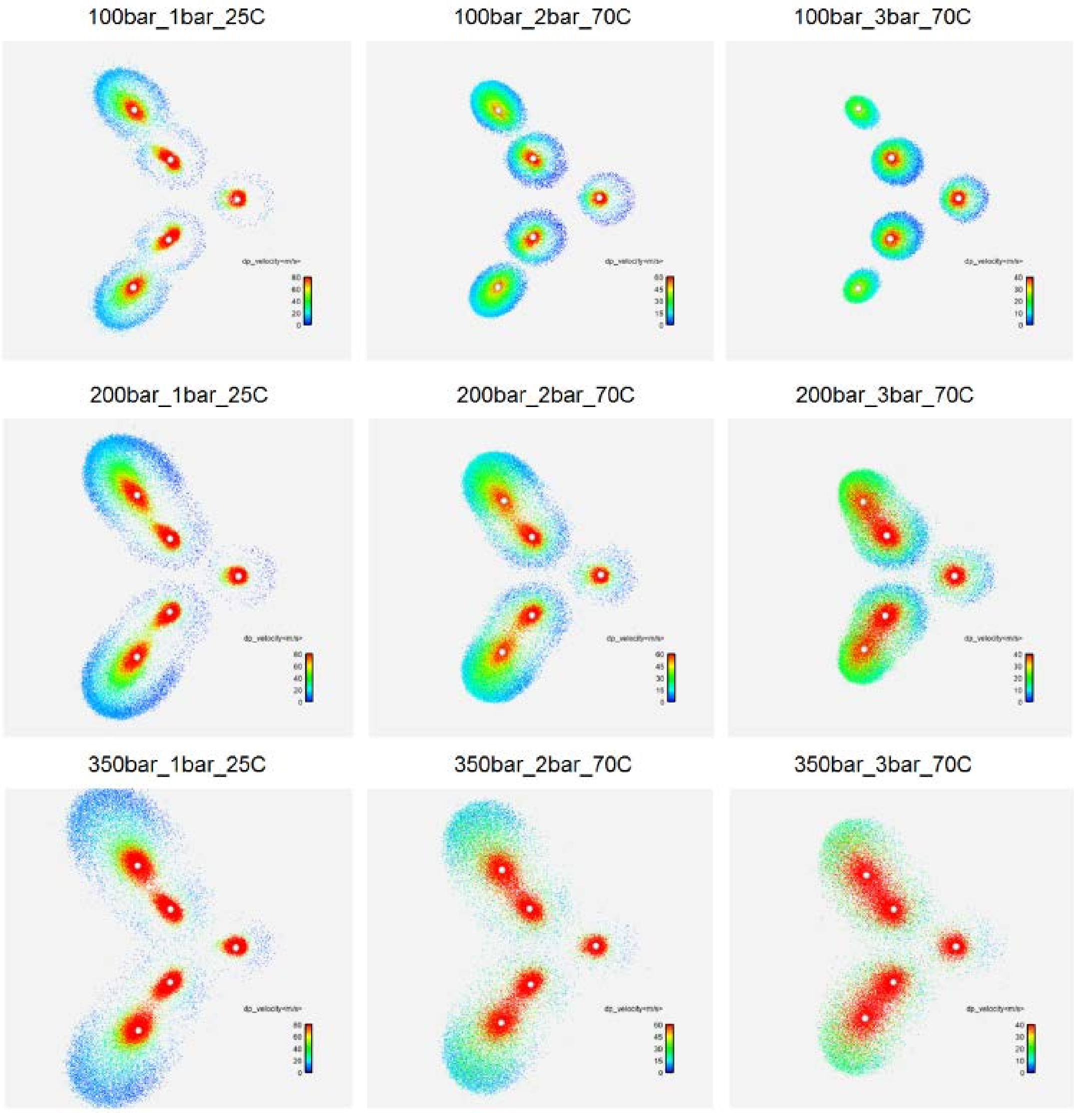


Figure 15. Spray clip cut at 30 mm downstream of injector tip.

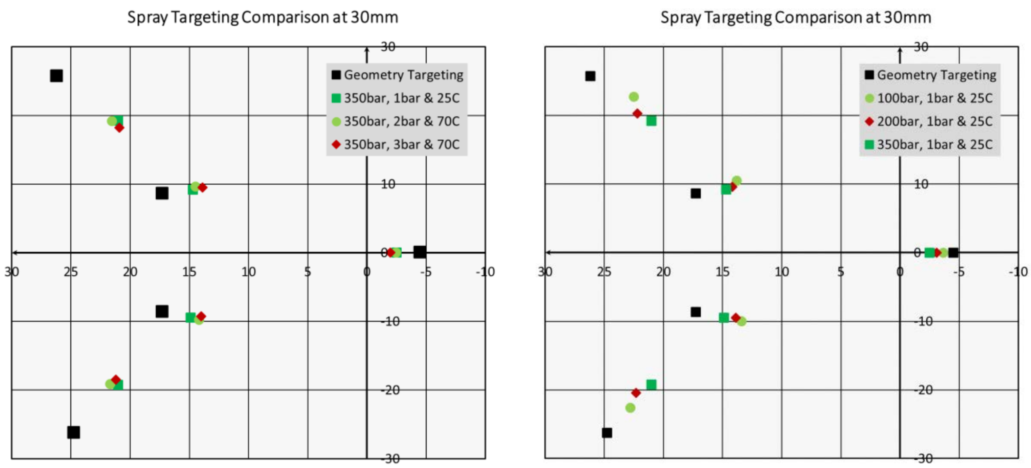


Figure 16. Spray targeting at different ambient conditions (left) and different fuel pressures (right).

Conclusion

Coupled Eulerian internal nozzle flow and Lagrangian spray simulation could bring numerous benefits to accurate prediction of GDI spray behavior. With the advantage of providing high-fidelity boundary condition for spray simulation, it can be served as an effective tool to enable transfer of the individual engine requirements on the spray to the injector sac-nozzle design. In this paper, a comprehensive assessment of one-way coupling CFD approach was presented. A Delphi 5-hole real-application GDI injector was selected to understand and quantify the potential of this coupled approach. This study uncovers several important insights of correlation between internal flow dynamics and spray atomization process.

- The turbulence level and vortex structure inside the sac region are vital for the spray atomization and plume targeting exiting the injector nozzle. Purposeful adaption of seat design to enhance vortex-controlled atomization mechanism is of utter importance for spray optimization.

- Nozzle hole orientation impacts the vortex structure, the current findings show the evidence of stronger vortex level in the upstream of higher beta angle injector. The vortex structure variation resulting for the spray hole orientation angle impacts the plume atomization performance, targeting and hole mass flow rate.

- The spray trajectory exiting the injector seat is deflected by the aerodynamic recirculation created by high velocity jets into the ambient. The backward pressure gradient is pronounced as the potential factor along with the shearing forces acting on the spray surface and wave interaction. Higher fuel pressure sharpens the pressure gradient and shearing force, resulting in increased spray deflection.

- The vortex-induced spray targeting bias and aerodynamic recirculation are the primary factors for the observed spray trajectory, spray angle and pattern. The one-way coupling approach significantly help reduce the reliance on hardware trial-and-tests to meet the spray characteristics requirement imposed by the engine performance target.

- The spray flash boiling phenomenon under extraordinary low ambient pressure and the spray behavior under extremely high ambient density condition (i.e. advanced injection strategy with very late injection) are out of scope in this study

Reference

1. Lutterman, C., and Mahrle, W., “BMW High Precision Fuel Injection in Conjunction with Twin-Turbo Technology: a Combination for Maximum Dynamic and High Fuel Efficiency, ” SAE paper 2007-01-1560.
2. Hemdal, S., “Characterization of stratified fuel distribution and charge mixing in a DISI engine using Rayleigh scattering.” Combustion and Flame, Volume 193, July 2018, Pages 218-228.
3. Hellmann, R.; Jochmann, P.; Stapf, KG. et al., “Towards design optimization of high-pressure gasoline injectors using Genetic Algorithm coupled with Computational Fluid Dynamics (CFD).” En llass Europe, 28th European conference on Liquid Atomization and Spray Systems. Editorial Universitat Politenica de Valencia. 880-887. Doi: 10.4995/ILASS2017.2017.4586.

4. Wang, X., Zhan, Z., Yu, X., Hu, T. et al., “Experimental Study on Injector Spray Pattern Optimization for a Turbocharged GDI Engine Combustion System,” SAE Technical Paper 2014-01-1439, 2014, doi: 10.4271/2014-01-1439.
5. Li, X., Zhang Y., Xu M. et al., “ Investigation on Wall Wetting for GDI Engine,” Chinese Internal Combustion Engine Engineering Vol. 33, Oct. 2012, No.5, doi: 1000-0925(2012) 05-0017-07.
6. Kancherla, R., Rathinam, B., Douailler, B., Naithani, U. et al., “Spray Modelling for GDI Application：Two Different Approach,” SAE Technical Paper 2016-28-0007, 2016, doi:10.4271/2016-28-0007.
7. Kong S. C., Senecal P. K. and Reitz R. D., “Developments in Spray Modeling in Diesel and Direct-Injection Gasoline Engines,” Oil and Gas Science and Technology, vol. 54, no. 2, pp. 197-204, 1999.
8. Beale, J. C., “Modeling Fuel Injection using the Kelvin-Helmholtz/Rayleigh-Taylor Hybrid Atomization Model in KIVA-3V,” MS Thesis, Delt. Of Mechanical Engineering, University of Wisconsin-Madison, 1999.
9. Braga, R., Vaz, M., de Moraes Martins, C., Hindi, G. et al., "3D Numerical Characterization of a Multi-Holes Injector in a Quiescent Vessel and Its Application in a Single-Cylinder Research Engine Using Ethanol," SAE Technical Paper 2017-36-0360, 2017.
10. Saha, K., Quan, S., Battistoni, M., Som, S. et al., "Coupled Eulerian Internal Nozzle Flow and Lagrangian Spray Simulations for GDI Systems," SAE Technical Paper 2017-01-0834, 2017, doi:10.4271/2017-01-0834.
11. Hoyas, S., Gil, A., Margot, X., Khuong, D., “Evaluation of the Eulerian-Lagrangian Spray Atomization (ELSA) model in spray simulations: 2D cases.” Mathematical and Computer Modelling. 10.1016/j.mcm.2011.11.006.
12. Hirt, C.W.; Nichols, B.D. (1981). "Volume of fluid (VOF) method for the dynamics of free boundaries". Journal of Computational Physics. 39 (1): 201–225. Bibcode:1981JCoPh..39..201H. doi:10.1016/0021-9991(81)90145-5.
13. Befrui, B., Corbinelli, G., D'Onofrio, M., and Varble, D., "GDI Multi-Hole Injector Internal Flow and Spray Analysis," SAE Technical Paper 2011-01-1211, 2011.
14. Saha K., Som S., Battistoni M. et al., “Numerical simulation of internal and near-nozzle flow of a gasoline direct injection fuel injector” J. Phys.: Conf. Ser. 656 012100, 2015.
15. Richards, K. J., Senecal, P. K., and Pomraning, E., Converge v2.3 Manual, Converge Science, Inc., Madison, WI 2016.
16. Torelli R., D’Errico G., Lucchini T., Ikonomou V. and McDavid R. M., "A spherical volume interaction DDM approach for diesel spray modeling," *Atomization and Sprays*, vol. 25, no. 4, pp. 335-374, 2015.
17. Reitz, R. D. (1987). Modeling Atomization Processes in High-Pressure Vaporizing Sprays. Atomization and Sprays, 3, 309–337.
18. Braga, R., Vaz, M., de Moraes Martins, C., Hindi, G. et al., "3D Numerical Characterization of a Multi-Holes Injector in a Quiescent Vessel and Its Application in a Single-Cylinder Research Engine Using Ethanol," SAE Technical Paper 2017-36-0360, 2017, https://doi.org/10.4271/2017-36-0360.
19. Das, S., Hung, D. L. S., Markle, L. E., and VanBrocklin, P. G., A New Approach for Linking Experimental Data to Spray Modeling for an Outwardly Opening Direct Injection Gasoline (DI-G) injector, Presented at the Eighth International Conference on Liquid Atomization and Spray Systems, Pasadena, CA, USA, July 2000.

20. Naber, J. and Siebers, D., "Effects of Gas Density and Vaporization on Penetration and Dispersion of Diesel Sprays," SAE Technical Paper 960034, 1996.