Jet-to-Jet Collision Studies of a Novel High-Pressure Two-hole Injector under Gasoline Engine Conditions

Ahmed Abdul Moiz, Le (Emma) Zhao, Seong-Young Lee*, Jeffrey D. Naber
Department of Mechanical Engineering – Engineering Mechanics
Michigan Technological University
Houghton, MI 49931 USA

Sam Barros, William Atkinson
Nostrum Energy LLC

Abstract
Collision of liquid jets near the nozzle orifice is obtained in a novel two-hole colliding-jet style direct-injection (DI) injector. This jet-to-jet collision produces liquid breakup that is fundamentally different from how traditional fuel injectors operate, opening avenues for applications in internal combustion engines to achieve improved atomization. In this work, a non-reacting spray study is performed using a 2-hole colliding jet injector with gasoline fuel under temperature-pressure (T-P) conditions corresponding to 30°, 60°, and 90° BTDC of a spark-ignition gasoline engine. The engine-like conditions were generated in a constant-volume high pressure-temperature pre-burn type combustion vessel. Also, Computational Fluid Dynamics (CFD) work has been performed using a Eulerian-Lagrangian modelling approach, after experimental validation in the CONVERGE-CFD code. Experimental work of the colliding jets has been performed through analysis of penetration lengths, and overall spray structure while using this data for CFD code validation, wherever possible. The Eulerian-Lagrangian model predicts the spray characteristics reasonably well. The experiments show overall better spray characteristics for the conditions of 60° BTDC and this finding is supported by the CFD work as well.
Introduction

Liquid fuel injection is a vital part of any working process involving eventual burning of the fuel for power generation purposes. This fuel injection is finally delivered into the power generation zone, where the fuel is subjected to a variety of processes starting from breakdown of the injected liquid fuel into droplets (atomization), evaporation of the atomized droplets and finally combustion of the vaporized fuel. Of primary significance is the spray injection of transportation fuels in internal combustion (IC) engines.

Spray injection in IC engines is predominantly performed through the use of multi-hole nozzles in the present engine industry. The multi-hole nozzles introduce the fine spray into the engine chamber resulting in a high spray dispersion due to the arrangement of the nozzles. Some work has been conducted in the past with a novel spray mechanism for the introduction of fuel into engines. This novel mechanism concerns colliding type sprays, with sprays emerging from a multi-hole nozzle, whose geometrical arrangement allows the sprays to collide with each other after exiting the nozzle orifice. In the past, Ghasemi et al. studied the effect of incidence angle and nozzle separation distance on the collision of two merging sprays. They concluded that increasing nozzle separation distance leads to an increase in penetration length and Sauter Mean Diameter (SMD), however, reducing the spray cone angle results in reduction of spray tip penetration and SMD. Ko et al. worked on developing the O’Rourke model by adding conservation equations before and after collision. They validated the models and concluded that the velocities of droplets have a great degree of dependence on the collision angle, and the droplet sizes depend on the collision distance.

The concept motivating the use of colliding sprays is to cause an earlier spray breakup, both temporally and spatially, than the conventional spray injections. Also, this collision would result in a higher degree of atomization (hence vaporization) resulting from the impact of droplet collisions. The spray breakup progression could be an important phenomenon that affects the ignition and combustion processes.

Prior work was done by the authors to establish a reliable spray-spray collision model, extensively validated, which is used in the present work too. Eulerian-Lagrangian method was implemented to achieve the simulation based on gas-liquid phase phenomenon. Also, the study utilized RNG k-ε turbulence to capture the averaged turbulence. KH-RT breakup model and O’Rourke collision model are preferred for modeling breakup and collision process, respectively. Furthermore, through the comparison between single-hole conventional spray and two-hole colliding sprays, the influence of collision on the spray was investigated. Validations with the experimental result on 90° bent two-hole nozzles was performed and exploration about the influence of collision angles on the characteristics of two-hole colliding sprays was undertaken, so that the efficiency for vaporization rate can be determined within three different spray cases.

Very little, if any, research is being conducted in the area of high-pressure spray-spray collision for usage in combustion industry. In the present work, the collision of the sprays at Gasoline Direct Engine (GDI) engine-like injection pressures and ambient temperature-pressure conditions is undertaken. This work is performed considering three before top dead center (BTDC) conditions viz., 90°, 60° and 30° BTDC at which spray injection was planned to be implemented in the real engine. The 90°, 60°, 30° BTDC conditions correspond to compression ratios of 1.9, 3.3 and 7.5 respectively.

Experimental Procedure

The experimental study was conducted in a 1.1 L constant volume combustion chamber capable of bearing high temperatures and pressures through a process of pre-burn. A high-pressure and high-temperature ambient environment, replicating the thermodynamic condition of an IC engine, is obtained by burning a pre-calculated composition of premixed hydrocarbon mixture. This mixture is introduced at pre-calculated pressure and then ignited by a spark while the burned products are continuously mixed by means of a rotating fan inside the combustion chamber. The burned products of this combustion reaction are the target species which would be otherwise present in the IC engine at this temperature and pressure. This combustion causes the chamber temperature and pressure to rise until target thermodynamic condition is reached; at this point the fuel is injected. A detailed description of the combustion chamber can be found in Ref. .

Figure 1. Illustration of colliding jets

A two-hole colliding jet injector assembly is mounted on one side of the vessel orthogonal to the windows. The orientation of the two holes (nozzles) is
such that both of them are in the horizontal line of sight as shown to the left of Figure 1 (In-plane view). For the high-speed visualization of spray, modified Z-type schlieren imaging system is used as shown in Figure 2.

![Image](image.png)

**Figure 2. Optical setup for the schlieren imaging**

A high-intensity and pulsed LED, together with a pin-hole aperture is utilized as the light source. A collimated beam, generated by the first schlieren mirror (f = 750 mm, 152 mm diameter with f-stop of 5), is directed passing through the optical vessel. The first mirror is placed where its focal point coincides with the light source. Deflected by a 90° reflector, the beam converges by the second schlieren mirror and finally captured by a high-speed camera through a negative bi-convex focusing lens (f = 200 mm). The position of the high-speed camera is adjustable at some distance from the second mirror’s focal point to capture the focus plane (plane passing through the injector tip). A knife edge is placed at a vertical position at the focal point to filter refracted rays. The camera is set at 16,000 fps and exposure time of 8.2 μs. The various other details of the experimental work related to the engine relevant settings are shown in Table 1.

**Table 1. Experimental test conditions**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Injector type</td>
<td>2-hole colliding jet</td>
</tr>
<tr>
<td>Impinging angle</td>
<td>90°</td>
</tr>
<tr>
<td>Hole diameter</td>
<td>239 μm</td>
</tr>
<tr>
<td>Chamber pressures for 30°, 60°, 90° BTDC</td>
<td>37.4 bar, 12.4 bar, 5.7 bar resp.</td>
</tr>
<tr>
<td>Chamber temperature for 30°, 60°, 90° BTDC</td>
<td>653 K, 490 K, 402 K resp.</td>
</tr>
<tr>
<td>Injection temperature</td>
<td>300 K</td>
</tr>
<tr>
<td>Injection pressure</td>
<td>172 bar</td>
</tr>
<tr>
<td>Injection duration</td>
<td>3 ms</td>
</tr>
</tbody>
</table>

**Image Processing**

The obtained image from the high-speed camera is processed in the MATLAB image processing toolbox to trace the boundaries of the spray geometry and to arrive at spray penetration. The process of arriving at a proper spray boundary is started by subtracting the spray containing image with the background, applying a medium filter to the subtracted image, adjusting the threshold for the filtered image and converting to gray-scale with the applied threshold. Later, the maximum area of the spray is found along with area opening operation which enables the removal of all connected components having fewer than specified amount of pixels, producing another binary image. Finally, image segmentation is done by using a morphological operator (dilation with diamond structural element) to output a uniformly filled spray boundary. The spray penetration is computed using ‘extrema’ functions.

**Simulation Approach**

Since the fuel flow from the injector, as it moves downstream, is a gas-liquid two-phase phenomenon, this study employs the Eulerian-Lagrangian method to solve it throughout the computational domain. In the present case, Eulerian method is responsible for continuous fluid model, i.e. surrounding gas. On the other hand, Lagrangian method deals with the discrete one, namely, water. Most turbulence models utilize a certain form of Reynolds stress approach. However, during the process of averaging the constitutive equations, it tends to lose fidelity. Therefore, it is desirable to find a better way to systematically eliminate the smallest scales of turbulence such that the remaining scales of turbulence can be distinguished under currently available computer capability. This is the theory of renormalization group (renormalization group, RNG) which is the turbulence model employed in this work. Furthermore, this study uses a KH-RT breakup length model in which constants are adjusted to improve the liquid penetration trend. The KH-RT model combines the effects of Kelvin-Helmholtz waves driven by aerodynamic forces, with Rayleigh-Taylor theory instabilities that are caused by acceleration of drops emitted into free stream.
The present study utilizes the O’Rourke collision model, which only includes two collision outcomes: coalescence and bounce. More post collision outcomes to compensate for the deficiency of the O’Rourke collision model were thus introduced. Therefore, the final collision outcome may include stretching separation, reflexive separation due to bounce, and coalescence. Numerical simulations are carried out using CONVERGE 2.2 by modelling gasoline surrogate spray injection into a constant volume chamber. Based on Ref., 70% iso-octane and 30% n-heptane was chosen as a surrogate for gasoline. Also, for the CFD, the inter-nozzle distance was 7 mm and the nozzle orifice diameters were 239 µm, corresponding to the original experimental geometry.

In this study, the base-grid size is set to be 3 mm. However, Adaptive Mesh Refinement (AMR) steps in to automatically refine the grid based on fluctuating boundary conditions (“velocity” in this paper with a sub-grid scale of 0.5 m/s). In addition, fixed grid refinement is used to refine the grid only in specific locations where a finer resolution is significant to the accuracy and precision of the solution. In the present study, a fixed grid refined area adjacent to the nozzle outlet was added to resolve the complex flow behavior when simulating sprays. Figure 3 shows the grid generation at start of injection (ASOI) timing of 1.5 ms for the 90° BTDC case.

It is seen from Figure 3 that Adaptive Mesh Refinement (AMR) refines the mesh. Also, CONVERGE maintains a 2 to 1 grid connectivity, thus progressively moving from the base grid to the refined one. There is another mesh refinement criteria of ‘collision mesh’ used in the simulations. In a simulation without collision mesh, parcels collide only with parcels in the same grid cell. With no collision model switched on, there may be artifacts in the spray, due to the injected parcels not colliding across cell walls. The modeling parameters considered in the study are summarized in Table 2.

CFD work used the grid convergence procedure identified in Ref. The time constant ($B_1$) for the optimization of spray break-up, the injected parcel count for Lagrangian liquid particles and the grid-size were all swept to obtain a grid converged solution. The final grid utilized a 3 mm base grid, with an AMR of level 3, which gave a fine grid zine of 375 µm, a parcel count of 100,000 per nozzle and a $B_1$ value of 11.67.

<table>
<thead>
<tr>
<th>Modeling Parameters</th>
<th>CONVERGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Method</td>
<td>Eulerian-Lagrangian</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>RNG k-ε model</td>
</tr>
<tr>
<td>Breakup model</td>
<td>KH-RT breakup length model</td>
</tr>
<tr>
<td>Collision model</td>
<td>O’Rourke model</td>
</tr>
<tr>
<td>Collision outcomes</td>
<td>Post-collision (bouncing, stretching separation, reflexive separation, or coalescence)</td>
</tr>
<tr>
<td>Dimensionality and type of grid</td>
<td>3D, structured with AMR</td>
</tr>
<tr>
<td>Grid size</td>
<td>Base grid size: 3 mm Finer grid: 375 µm</td>
</tr>
<tr>
<td>Time step</td>
<td>Variable based on spray, vaporization</td>
</tr>
</tbody>
</table>

Table 2. Modeling tools adopted in CONVERGE 2.2

Results and Discussions:

Thermodynamic analysis:

After the fuel gets injected into the hot constant volume chamber, vaporization is bound to happen. Figure 4 depicts the saturation line below which the mass remains in the vapor phase.

Figure 3. Mesh at 90 BTDC condition at 1.5 ms ASOI

Figure 4: Saturation vapor mass comparison of experimental and CFD for the three test cases

The injection mass is in the order of 50 mg for the injection conditions. For the conditions of 90°, 60°, 30° BTDC (for their respective ambient temperatures of 402K, 490K, 653K and density conditions of 4.97 kg/m^3, 8.77 kg/m^3, 19.92 kg/m^3) the injected masses thus are below the saturation line, meaning the fuel would completely vaporize at steady state. This is just a verification of the chamber’s ability to vaporize the fuel at infinite time (steady state) or when the spray is allowed to stand in the conditions for very long time durations.
Experimental and CFD correlations:

Figure 5 shows the comparison of the vapor penetration from the experiment and CFD. For the CFD, the boundary of vapor at 5% of mass fraction of the fuel is considered as the vapor extent (penetration). From Figure 5, it can be seen that the CFD vapor penetration is in good agreement with the experiment. There is some discrepancy in the initial start of injection times, with the CFD over-predicting the vapor penetration. This discrepancy decreases as injection is subjected to higher temperature and pressure scenarios of 60° and 30° BTDC. This over-prediction could be attributed to the fact that the present vapor length calculations in the models utilized by the CFD solver were developed for single component fuels. As mentioned in the CFD formulation section, the present CFD work utilized a two-component surrogate. Also, a general observation from Figure 3 is that the penetration decreases as the ambient pressure and temperature increases from 90° to 30° BTDC.

Figure 5. Vapor penetration comparison of experiment and CFD for the three test cases

A sample of liquid and vapor boundary detection is shown in Figure 6. The liquid and vapor boundaries are detected as in the process described in the image processing section. The extent of the vapor and liquid boundaries are identified with the vertical line, signifying the extent.

Figure 6. 2D line-of-sight vapor fraction comparison of experiment for the three test cases

Figure 7 shows a representative vapor fraction on the basis of line-of-sight integration of light intensity. It is assumed that darker light corresponds to dense liquid region and brighter (or less dark) light corresponds to vapor region. The boundary detected during the start of the injection timings is slightly erroneous due to the boundary detection getting weaker in the spray initiation zones, this is the reason for higher values of vapor fraction in the near-spray commencement times of ~ 0.5 ms ASOI; later on, the boundary detection is proper with visually satisfactory outcomes.

Figure 7. Vapor-liquid boundary detection

From Figure 7, it can be noted that 30° BTDC condition shows a higher level of vaporization as expected and 90° BTDC give the lowest vaporization. Also, 60° BTDC case is closely comparable to the 30° BTDC case in terms of vaporization and lie within the standard deviations of each other (as seen by the error bars). Since, the 60° BTDC case shows better penetration and close vaporization when compared to 30° BTDC case, it
can be concluded that the 60° BTDC would serve a better cause of extent of fuel delivery and vaporization; and thus would be preferred than the 30° BTDC (and 90° BTDC) scenario.

CFD work with vapor mass fraction comparisons has also been performed. Vapor fraction in the CFD work is defined in the similar way as in experiment. Here, the vapor mass being formed per injected mass is considered as vapor fraction.

Figure 8. 3D CFD vapor fraction for the three test cases

Figure 8 shows the vapor fraction of the sprays at the three engine-like conditions analyzed. The results do not quantitatively match the experimental 2D line-of-sight measurement but show a qualitative similarity in the trends of vaporization. This is due to the assumption that line-of-sight measurements of light intensity variations for creating liquid and vapor boundaries is quantitatively sufficient, which is not true. But it does show qualitatively true trends. Again, from the CFD it can concluded that 60° BTDC case of injection is preferable since it results in more vapor being formed per injected mass and is close to the 30° BTDC vapor fraction at end of injection times. Thus, CFD suggests that since the 60° BTDC spray penetrates more than the 30° BTDC case and results in same amount of vapor fraction at end of injection times, it is thus preferred to the 30° BTDC injection.

Figures 9 show the comparison of spray structure of the three engine-like ambient conditions from experiment and CFD. The experimental vapor lengths are marked with a red dashed line over the CFD images. The CFD images in the Figure 9 show the droplet distribution with scattered black dots where as the vapor distribution is shown as an iso-surface of the vapor at 5% of mass fraction of the fuel.

Figure 9: Spray structure of experiment (left column) to CFD (right column) for top - 90°, middle - 60° and bottom - 30° BTDC conditions for ASOI times of 0.5 ms, 1 ms, 1.5 ms and 2 ms from top to bottom at each image set
Conclusions
A novel colliding jet injector has been tested in a constant-volume combustion chamber under non-reaction spray conditions. The vaporization characteristics of the colliding jet injector has been studied under three conditions of prospective injection times in an engine viz., 90°, 60°, 30° BTDC. CFD work with new collision mesh equipped Lagrangian colliding spray model has been performed and validated with the experiments. The results from both the experiment and CFD work conclude that 60° BTDC is a better injection time for the injection to take place due to higher extent of vaporized fuel delivery.

Nomenclature
ASOI  After Start Of Injection
BTDC  Before Top Dead Center
DI    Direct Injection
CFD   Computational Fluid Dynamics

References

Acknowledgments
The authors acknowledge the State of Michigan and Nostrum Energy for financial support for this study. The authors also give special thanks to Convergent Science for their support on simulation code.