Three-dimensional Simulations of the Transient Internal Flow in a Diesel Injector: Effects of Needle Movement

Q. Xue¹, S. Som¹, M. Battistoni¹,², D.E. Longman¹, H. Zhao², P. K. Senecal², and E. Pomraning²
¹Energy System Division, Argonne National Laboratory, Argonne, IL-60439, USA
²Convergent Science Inc., Middleton, WI-53562, USA
³University of Perugia, 06125, Perugia, Italy

Abstract
In this work the influence of needle movement and “off-axis” motion on the three-dimensional transient flow development in a Diesel injector is studied under realistic engine operating conditions. The transient flow characteristics, such as turbulence and cavitation at the injector exit are important for accurately modeling the spray atomization and fuel-air mixture formation, and thus the combustion in the engine cylinder. Hence, it is critical to accurately predict the high-speed internal nozzle flow characterized by high-pressure gradients and two-phase flow phenomena in such small-scale devices. In this study, the two-phase cavitating flow is described by a mixture-based model in the commercial code CONVERGE. The needle movement profiles in “in-” and “off-” axial directions are measured using an X-ray phase-contrast technique at Argonne National Laboratory. These profiles are imposed as moving boundary conditions in the simulations. Model validation and grid-convergence studies are performed first. Single-hole Spray A (using n-dodecane fuel) from the Engine Combustion Network (ECN) is studied. Simulations are also performed on a five-hole nozzle by imposing the needle movement profiles. The influence of three-dimensional transient needle movement on the in-nozzle flow and cavitation development is investigated for single and multi-hole nozzles. The internal flow characteristics are compared between the needle off-axis and in-axis conditions, and the effects are discussed. The quantitative distributions of turbulence and flow parameters at the nozzle exit are also studied. The main conclusion from this study is that the needle off-axis motion may not be important in single orifice injectors, however, it can influence the flow in the sac region in multi-hole injectors thus causing differences in mass flow rates between different holes. This study highlights the importance of needle movement effects on flow development in single and multi-hole nozzles.
Introduction

Cavitation and turbulence inside a Diesel injector play a critical role in primary spray breakup and development processes, which in turn influences the combustion and emission characteristics of an engine [1,2]. The study of cavitation in realistic injectors is challenging, both theoretically and experimentally, since the associated two-phase flow field is turbulent and highly complex, characterized by large pressure gradients and small orifice geometries.

If the duration of injection is long, it is expected that the injector needle would open completely, followed by a quasi-steady period of constant lift and a ramping down of the needle to the closing position. However, the Diesel engine community is moving towards multiple injections of short durations to aid the low temperature combustion operation concepts. Hence the needle may not open completely during the injection process. Transient effects play a more dominant role under these conditions. Essentially the injection system is expected to provide a precise amount of fuel for a desired duration and locations of interest inside the combustion chamber at high injection pressures.

Transient simulations with fixed needle lifts have been performed by several authors [3,4,5,6,7,8,9]. While such simulations are insightful for model validation, they are not truly representative of the real physics of Diesel injection. Modern Diesel technology demands capturing transient effects occurring due to the motion of the needle. Transient nozzle flow simulations have also been performed in the past decade by several researchers including Mulemane et al. [10], Battistoni et al. [11,12], Som et al. [13], Payri et al. [14], Palmieri and co-workers [15,16] etc. These simulations employed different fluid and cavitation models together with various thermal equilibrium assumptions. The transient simulations were performed with a moving needle. The motion of the needle is along the needle axis and usually obtained from one-dimensional system simulations inside the injector. Hence, the needle motion is only 1D in nature. Flow inside production Diesel injectors is expected to be turbulent and three dimensional, especially inside the sac region. Under these conditions a 1D needle profile is not truly representative.

Hole-to-hole variations are expected to exist due to geometrical differences which can be caused due to manufacturing tolerances of the nozzle orifices. Another source of hole-to-hole mass flow rate variation is due to the non-uniform motion of the needle along the seat and inside the sac region. Recently experiments by Kastengren et al. [17] used phase-contrast imaging to show that the needle vibrates/wobbles as it opens and closes. The reason for this wobble is attributed to the needle being a cantilever beam which has a natural frequency of oscillation. The effect of needle wobble on flow development in the needle seat, sac regions, and the ensuing flow through each hole has not been quantified either experimentally or through computational studies.

Hence, the main objective of this paper is to capture the influence of needle motion on in-nozzle flow development. Specifically, the influence of needle on-axis and off-axis motion are quantified in the current study. The effect of needle on-axis motion has been investigated by researchers in the past, as mentioned earlier. However, to the best of our knowledge the effect of needle off-axis motion (or so called needle “wobble”) has not been studied in the past. This forms the major motivation for this study.

In order to gain systematic understanding of the effect of needle wobble, first simulations are performed on a single-hole injector. The needle wobble is expected to have a relatively small effect in the single hole injectors. The multi-hole injectors are expected to be influenced to a greater extent by the wobble since potentially the off-axis motion can restrict the flow into some orifices more than the others. The single hole injector studied here is the Spray A injector obtained from the Engine Combustion Network (ECN) [18,19]. The multi-hole injector is a 5-hole on which experiments were performed by Payri and co-workers [20,21].

The paper is organized in the following way. First, a brief outline of the governing equations and cavitation model is provided. Then the simulation set-up is described in detail together with the imposed boundary and initial conditions. Validation of the cavitation and in-nozzle flow set-up is presented thereafter. The final section discusses the results with the single and multi-hole nozzles followed by the main conclusions and some planned future studies.

Governing Equations

In this section, the Navier-Stokes and volume of fluid (VOF) equations will be introduced. A detailed description of the numerical methods used in the current software to solve these equations will also be given in this section. Further details are available in a recent submission by Zhao et al. [22].

In a single-fluid approach, the multi-phase mixture model is governed by one set of conservation equations for mass, momentum, and energy, with the addition of a turbulence closure model for Reynolds Averaged Navier-Stokes (RANS) equations. Mass and momentum equations are given below

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

(1)
\[
\frac{\partial \rho \mathbf{v}}{\partial t} + \nabla \cdot \rho \mathbf{v} \mathbf{v} = -\nabla p + \nabla \cdot \mathbf{\tau} + \rho \mathbf{f}
\]
where \(\rho\) and \(\mathbf{v}\) are the mixture density and velocity; \(p\) is the pressure; \(\mathbf{\tau}\) is the mixture stress-strain tensor due to molecular and turbulent viscosity. Analogous formalisms apply for energy conservation and turbulence closure models [22]. Equations are here omitted for brevity.

In the present study, the multi-phase system comprises of a liquid phase (1), a vapor phase (2) and non-condensable gases (3). The sum of vapor and non-condensable gases will be referred to with the subscript \(g\). In a mixture model the concept of pseudo-density is used. The mixture density is computed with the following equation:

\[
\rho = \alpha_0 \rho_1 + \alpha_2 \rho_2 + \alpha_3 \rho_3 = \alpha_g \rho_g + (1 - \alpha_g) \rho_l.
\]

In the present implementation, the void fraction is not transported directly, but the species are first solved using the species transport equation and then the void fraction \(\alpha_g\) is calculated:

\[
\frac{\partial \rho_i Y_i}{\partial t} + \nabla \cdot \rho_i Y_i \mathbf{v} = \nabla \cdot (\rho_i \mathbf{D}_i \nabla Y_i) + S_i,
\]

\[
\alpha_g = \frac{\rho_g}{\sum \rho_i / \rho_i}.
\]

The liquid phase is treated as incompressible and the gas phase is treated as compressible. A VOF method is used to model the two-phase flow. In the VOF method, a function \(\alpha\) is used to represent the void fraction and is defined as follows:

\(\alpha = 0\) : the cell is filled with pure liquid
\(\alpha = 1\) : the cell is filled with pure gas
\(0 < \alpha < 1\) : the cell is filled with both liquid and gas species

**Cavitation Model**

A cavitation model is implemented in the current work. This model has been implemented and validated with 2D Winklhofer nozzle simulations by Schmidt et al. [23]. The cavitation model is based on the flash boiling hypothesis developed by Schmidt and co-workers [24,25] with rapid heat transfer between the liquid and vapor phase. The vaporization process in cavitation is very similar to that of flash-boiling, with a few important differences. Cavitation represents the vapor formed through a constant temperature system experiencing a drop in pressure, whereas, flash-boiling represents the same system, with a lower pressure drop and elevated temperatures. A Homogenous Relaxation Model (HRM) was used to predict the mass exchange between the liquid and vapor. This model describes the rate at which the instantaneous mass fraction of vapor in a two-phase mixture will approach its equilibrium value.

\[
\frac{dx}{dt} = \frac{x - \bar{x}}{\Theta}
\]

where \(\Theta \approx 3.84 \times 10^{-7} \text{s} \), \(a = -0.54 \), \(b = -1.76\)

The non-dimensional pressure ratio is given by:

\[
\psi = \frac{P_{\text{crit}} - P}{P_{\text{sat}} - P_{\text{crit}}}
\]

The influence of needle motion is investigated on both single-hole and multi-hole injectors by performing 3D simulations using the RNG k-\(\epsilon\) model. This section outlines the boundary and initial conditions for both injectors followed by some geometrical and meshing details.

**Simulation Set-up**

The single-hole ECN Spray A tapered nozzle is shown in Figure 1. The geometrical details are obtained from Kastengren et al. [17] and also shown in Table 1. Figure 1 also shows that the orifice is not
centrally located with respect to the injector sac region. The transient needle lift (along y-axis in Figure 3), needle wobble (along x, z-axes in Figure 3), and inlet pressure profiles are shown in Figure 2 and are obtained from Ref. [17]. The base grid size for the simulation is set at 160 µm. Since the simulations are performed with an outlet chamber, it was important to use a relatively large base grid size to keep the peak cell counts low. The simulations were stopped at 1.6 ms to save wall-clock time.

Different minimum grid sizes were simulated using various levels of adaptive and fixed embedding. More information about these meshing strategies are available from the CONVERGE manual [27] and the authors’ previous publications [28,29,30]; hence are not described here. The Cartesian cut-cell method allows for the needle off-axis motion described in Figs. 2 and 5 to be accounted for in the simulations. Figure 3 shows a view of the grid generated at 0.5 ms along with a zoomed view of the sac and orifice inlet regions. The peak cell count at peak needle lift position is about 400,000 with a minimum cell size of 10 µm.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nozzle exit diameter (µm)</td>
<td>89</td>
</tr>
<tr>
<td>Nominal injection pressure (bar)</td>
<td>1500</td>
</tr>
<tr>
<td>Back pressure (bar)</td>
<td>60</td>
</tr>
<tr>
<td>Fuel type</td>
<td>n-dodecane</td>
</tr>
<tr>
<td>Peak needle lift (µm)</td>
<td>~ 470</td>
</tr>
<tr>
<td>Peak needle wobble (µm)</td>
<td>~ 41</td>
</tr>
<tr>
<td>Injection duration (ms)</td>
<td>~ 1.5</td>
</tr>
</tbody>
</table>

Table 1. Spray A nozzle details

Figure 3. A view of the grid generated at 0.5 ms. Zoomed view of the sac and orifice inlet regions are also shown.

The multi-hole injector simulated is shown in Figure 4. The 5-hole micro-sac injector geometry is obtained from refs. [20,24] and tabulated in Table 2. It should be noted that each of the holes has the nominally same dimensions. Hence, even though there might be differences in actual geometry between different holes, the simulations do not account for these artifacts since the geometry used in simulations is a nominal one. The transient needle-lift (along z-axis in
Figure 4), needle wobble (along x-axis in Figure 4), are shown in Figure 5 and are obtained from Ref. [20] and [17] respectively. The base grid size for the simulation is set at 320 µm. Since the simulations are performed with an outlet chamber it was important to use a relatively large base grid size to keep the peak cell counts low. Different minimum grid sizes were simulated using various levels of adaptive and fixed embedding. Figure 6 shows a view of the grid generated at 0.5 ms along with a zoomed view of the sac and orifice inlet regions. The peak cell counts at 0.85 ms are about 300,000 and 1.3 million with the minimum cell sizes of 20 and 10 µm respectively.

![Figure 4. Geometry of five-hole micro-sac nozzle](image)

![Figure 5. Needle lift and needle off-axis motion profiles vs. time imposed for the five-hole injector simulations](image)

### Table 2. Five-hole injector nozzle details from Payri et al. [20]

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Orifice Inlet diameter (µm)</td>
<td>150</td>
</tr>
<tr>
<td>Nozzle exit diameter (µm)</td>
<td>130</td>
</tr>
<tr>
<td>Nozzle Orifice Length (mm)</td>
<td>1</td>
</tr>
<tr>
<td>Nominal injection pressure (bar)</td>
<td>780</td>
</tr>
<tr>
<td>Back pressure (bar)</td>
<td>20</td>
</tr>
<tr>
<td>Fuel type</td>
<td>Diesel</td>
</tr>
<tr>
<td>Peak needle lift (µm)</td>
<td>~225</td>
</tr>
<tr>
<td>Peak needle wobble (µm)</td>
<td>~28</td>
</tr>
<tr>
<td>Injection duration (ms)</td>
<td>~1.8</td>
</tr>
</tbody>
</table>

![Figure 6. (a) A view of the grid generated with all the five holes and the outlet region. (b) Zoomed view of the grid generated inside the sac and orifice inlet regions. The minimum needle-lift is also shown.](image)

**Model Validation and Grid-Convergence Study**

This section presents validation of the in-nozzle flow and the cavitation modeling approach implemented in the computational fluid dynamics (CFD) software. The authors have previously published validation against Winklhofer et al. [31] data and x-ray radiography data [32] in the following references by Zhao et al. [22] and Battistoni et al. [33] respectively. In this paper some additional validation is presented for the Spray A single-hole and five-hole nozzles.

Figure 7 presents the measured [18,19] and calculated discharge coefficient ($C_d$) vs. time. The theoretical mass flow rate calculated using Bernoulli’s equation is 0.00296 kg/s whereas the quasi-steady value from simulation is 0.00261 kg/s. This results in
a quasi-steady $C_d$ value of 0.88 which is very close to the experimental measured value of 0.9. Clearly, the Spray A nozzle simulation can capture the experimental mass flow rate trends.

Figure 7. Calculated vs. measured discharge coefficient for Spray A nozzle.

Figure 8 presents the velocity contours for the Spray A nozzle at different minimum mesh resolution of 7.5 µm, 10 µm and 20 µm along a cut-plane through the center of the injector. Since the nozzle hole is tapered and rounded at the inlet edge, no cavitation is observed for this nozzle. Figure 9 shows the mass flow rate at these mesh resolutions. Although the mass flow rates between the 20 µm and 10 µm resolutions are only marginally different, we conclude that grid convergence is achieved at 10 µm mesh resolution. Further reducing the resolution does not change the results significantly.

Figure 8. Velocity contours for the Spray A nozzle at different grid resolutions.

Figure 10. Transient mass flow rate comparison against experimental data from Payri et al. [20] using two different minimum grid sizes of 10 and 20 µm.

Figure 9. Mass flow rate vs. time for three different minimum grid sizes.

Figure 10 shows the comparison of mass flow rate with experimental measurements [20] for the five-hole injector nozzle. Since the simulation starts with a minimum lift (of 20 µm), the mass flow rate first demonstrates a short transient period and then begins to increase. When the needle lift reaches about 40% of the maximum lift, the mass rate stays fairly constant. At the minimum needle lift, the mass flow rate will never reach zero as is the case with experiments. It is observed that the flow is highly transient when the needle is at low lifts. In general, simulation results with both minimum grid sizes of 10 and 20 µm are able to capture the steady portion of the mass flow rates quite well however, there are some apparent differences in the transient portions. Please note that the simulations with the 10 µm resolution are still running. This issue is being further investigated. In general the simulation results match quite well against the experimental data. The wall-clock times for the 20 and 10 µm minimum cell size cases are 194 and 1008 core hours respectively on 32 cores. It is apparent that reducing the cell size by half increases the wall-clock time by a factor of 5. Keeping in mind the accuracy and wall-clock time, the authors
decided to perform parametric studies looking at the effect of needle off-axis motion using the minimum grid size of 20 µm.

Results and Discussion

Following validation and grid-convergence studies with the single-hole and five-hole nozzles, this section focuses on the influence of needle motion on in-nozzle flow development. First results with the single-hole Spray A nozzle are presented.

Figure 11 plots the mass flow rate vs. time with a minimum grid resolution of 7.5µm, with and without the wobble. It is clear that the wobble has no influence in the overall mass flow rates for this single-hole injector.

Figure 12 plots the turbulent kinetic energy and velocity contours along a cut-plane through the middle of the injector at three different times. The minimum grid sizes for these simulations were 7.5 µm. At 0.2 ms, which is a low needle lift position without any off-axis motion, it is not surprising that the wobble and no-wobble simulations look identical. It should be noted that although the needle lift is low, since the pressure difference between inlet (1500 bar nominal) and outlet (60 bar) is very high, high injection velocities are reached very early during the simulation. At 0.5 ms, the needle is less than half open and is characterized by high wobble. Under these conditions, no apparent influence of needle off-axis motion is observed in both turbulent kinetic energy and velocity contours. At 1 ms, the needle is fully open together with significant wobble. It is perhaps not surprising that there is no influence of needle off-axis motion since the needle is too far away from the orifice inlet to influence the flow development inside the orifice. As noted earlier, even though the flow is characterized by high pressure gradients, no cavitation was observed since the nozzle orifice is tapered and rounded. Some transient effects due to needle movement are present but not pronounced. This is also observed in Figs. 9 and 11 since the mass flow rates reach steady state very quickly. The legends in Fig. 12 also show that both peak velocities and turbulent kinetic energies increase as the needle moves up, however the profiles look quite similar at all times. The overarching observation from the study of the single-hole nozzle is that the wobble has marginal influence on flow development.

In this section the influence of needle transience on a multi-hole injector is also analyzed. To the best of our knowledge, the influence of needle off-axis motion has only been investigated by assuming an eccentric needle movement profile for a valve-covered orifice (VCO) nozzle by Spathopoulou et al [34] or for a SAC-type nozzle by Chiavola and Palmieri [16]. In this study the influence of a measured 3D
needle movement profile in a micro-sac injector is studied. Figure 13 plots the mass flow rates for cases simulated with and without the needle off-axis motion with a minimum cell size of 20 µm. In the cases where needle off-axis-motion (wobble) are calculated, the simulations are performed with two different wobble profiles i.e., eccentricity along +x-axis and along –x-axis (cf. Fig. 14b). The off-axis motion along +x-axis points exactly towards hole no. 1, while the off-axis motion along –x-axis is directed in between two holes 3 and 5. The results indicate that regardless of the wobbling directions, there are two time intervals where the total mass flow is negatively affected with respect to the baseline case. These effects are evidently phased with the first and third peak of the off-axis movement, as can be seen comparing Figure 5 and Figure 13, when the needle lift is still low enough to affect the sac flow by the eccentricity. To explain the causes of such results it is necessary to analyze in detail the flow field in the sac and in each orifice. The following figures analyze the flow characteristics for the needle wobble along the +x direction.

**Figure 13.** Comparison of mass flow rates vs. time between the cases with and without needle off-axis motion.

Figure 14(a) plots the mass flow rate through each orifice for the case with needle off-axis motion along the +x direction. The orientation of the holes simulated is shown in Fig. 14(b). From Fig. 5, it should be noted that the peak needle-off-axis motion occurs during low lifts i.e., at 0.4 ms and 1.5 ms (approximately) and at ~0.9 ms which coincides with the peak needle lift of ~225 µm. Hence it is not surprising that the largest hole-to-hole differences are observed at low needle lift positions i.e., ~0.4 ms and ~1.5 ms. At high needle-lift position of 0.9 ms the needle lift is high, hence it does not influence the flow development in the sac and orifice entry region. Hence between 0.55 ms to 1.35 ms the mass flow rate predicted through each orifice is steady and on top of each other. This time period corresponds with a needle lift of ~100 µm and higher. The analysis above shows that if the needle is lifted more than 100 µm, the off-axis motion does not influence the flow development. It should also be noted that holes 2 and 5 are mostly affected by the needle off-axis motion and predict lower mass flow rates compared to the other orifices. This result might be counter intuitive since the off-axis motion was in +x direction (cf. Fig. 14b) which should constrain the flow entering hole 1 and thus decrease the mass flow rate through this hole.

**Figure 14.** (a) Figure 14 plots the mass flow rate through each orifice for the case with needle off-axis motion along the +x direction, (b) Hole orientation for the simulations.
The trend of lower mass flow rates with holes 2 and 5 is explained by plotting the velocity contours and streamlines in the sac and orifice entry region. Figures 15 and 16 plot velocity contours along a cut-plane through the center of holes 1, 2, and 3. Holes 4 and 5 are not plotted since they are symmetric to holes 3 and 2 respectively, about the x-axis going through center of hole 1. Figure 15 is plotted to focus on the orifice entry region for holes 1, 2, and 3. Comparing the contours at 1 and 1.35 ms which correspond with positions of high needle-lift (cf. Fig. 5) and quasi-steady mass flow rate (cf. Fig. 14a), the velocity contours are similar for all the holes at both the orifice inlet and exit regions. Since the flow-field inside the orifice is solved with the incompressible assumption, it is not surprising that the mass flow rates in Fig. 14a are similar for all the holes at 1 and 1.35 ms. It should also be noted that the velocity contours are symmetric (cf. Fig. 15) which shows that the flow is perhaps entering all the 3 holes in a symmetric fashion from the sac and needle seat regions. Velocities in the sac region are observed to be quite low at these high needle-lifts.

Low needle-lift and high needle-off axis motion is observed at 0.4 ms and 1.6 ms. At these times there are significant differences in the velocity contours.

Figure 15. Zoomed view of the velocity contours through holes 1, 2, and 3 at 0.4, 1.0, 1.35, and 1.6 ms along a central cut-plane through each hole for a case with needle wobble. The sac and needle seat regions are also shown.
Highest and lowest velocities are observed for holes 3 and 2 respectively at these times at both nozzle exit and entry regions. The mass flow rate also follows this trend. It is important to point out from Fig. 14a that hole 3 has a marginally higher mass flow rate compared to hole 1, which can also be explained based on the velocity contours. At low lifts the flow does not enter the orifice (cf. Fig. 15) in a symmetric fashion through sac and seat regions. Also at these low lifts the sac region is characterized by higher velocities compared to the higher lift cases. Fig. 16 further shows that for hole 1 the higher velocities are towards the lower orifice region. As the needle moves in the +x direction, the flow through the top of the orifice decreases while through the bottom of the orifice increases due to the flow entering from the sac region in hole 1. The fact that the needle is at different positions with respect to the seat region for each orifice is also clear by comparing images for hole 1 and hole 3 at 1.6 ms. The velocity distributions in the orifice and sac regions can be understood further by plotting the streamlines.

Figure 17 plots the streamlines for all the holes at 0.4, 1.0, and 1.6 ms. The profiles at 1.35 ms were omitted since they are similar to 1.0 ms. The needle and orifice inlet and outlet outlines are also shown. At all these times complicated flow structures are observed in the seat and sac regions. At high needle-lift positions i.e., at 1.0 ms, the streamlines entering all the orifices are straight i.e., there is no swirling of the flow which results in lowering of the velocities. As shown in Figs. 14a and 15, this is the cause of similar mass flow rates between all the orifices in the quasi-steady region.

Figure 18 plots the zoomed view of the streamlines entering holes 1, 2, and 3 at 0.4 ms. It is clear from this figure that there are more streamlines entering hole 1 and 3 compared to hole 2; which is resulting in lower mass flow rates with hole 2. In fact, the fuel enters hole 2 in a swirling fashion which results in higher friction and dissipation resulting in lower mass flow rate through this hole. Since hole 2 and hole 5 are symmetric about the x-axis, it is not surprising that the mass flow rate through hole 5 is also low and in fact similar to hole 2 (cf. Fig. 14a).

It should also be noted that under the conditions investigated (cf. Table 2), the injector does not cavitate. The simulation strategy demonstrates that the in-nozzle flow model developed can capture the influence of needle transience on flow development. The CFD approach can also capture finer details such as flow dynamics in the needle-seat and sac regions which determine the mass flow rate through each orifice.
Summary and Conclusions
The VOF implemented in CONVERGE is used to predict the influence of needle transience on in-nozzle flow development. The modeling methodology falls under the mixture model approach together with thermal equilibrium assumption. Due to the unique grid generation algorithm available with the CFD software, the influence of needle on-axis and off-axis motion could be investigated. First validation of the in-nozzle flow model is performed. Two test cases are chosen from literature namely, the Spray A single-hole nozzle and a five-hole nozzle. Mass flow rate and discharge coefficient validation are first shown together with grid-convergence studies. It was concluded that a resolution between 10-20 µm may suffice in resolving the flow inside the nozzle orifice, depending on the orifice size. This resolution is suggested keeping in mind both accuracy and wall-clock times.

State-of-the-art experimental facilities at Argonne National Laboratory using phase contrast imaging have facilitated needle-lift and off-axis measurements with high temporal frequency. This data is used as boundary conditions for simulations. For sin-
gle-hole injectors the needle off-axis motion did not influence the flow development appreciably. Needle transient effects were also not pronounced since the injection pressure was high.

Multi-hole injectors were significantly influenced by the needle off-axis motion. The mass flow rate trends between holes could be explained by plotting the streamlines. This clearly demonstrates that in a production multi-hole injector, the mass injected from each hole may be different due to in-nozzle flow dynamics. The modeling approach developed in this study will allow nozzle flow and engine designers to better predict hole-to-hole variations.

Future studies will focus on the Spray H injector (from ECN) which is known to cavitate. Also higher injection pressures with the five hole injector will be simulated and the influence of needle transience will be investigated.

Acknowledgements

The submitted manuscript has been created by UChicago Argonne, LLC, Operator of Argonne National Laboratory (“Argonne”). Argonne, a U.S. Department of Energy Office of Science laboratory, is operated under Contract No. DE-AC02-06CH11357. The U.S. Government retains for itself, and others acting on its behalf, a paid-up nonexclusive, irrevocable worldwide license in said article to reproduce, prepare derivative works, distribute copies to the public, and perform publicly and display publicly, by or on behalf of the Government. This research was funded by DOE’s Office of Vehicle Technologies, Office of Energy Efficiency and Renewable Energy under Contract No. DE-AC02-06CH11357. The authors wish to thank Gurpreet Singh, program manager at DOE, for his support.

We gratefully acknowledge the computing resources provided on "Fusion," a 320-node computing cluster operated by the Laboratory Computing Resource Center at Argonne National Laboratory.

The authors would also like to acknowledge Professor David P. Schmidt and Sukanta Rakshit from University of Massachusetts at Amherst for sharing the Spray A geometry information.

The authors would also like to acknowledge Mr. Kaushik Saha from University of Waterloo, Canada for helping with the Spray A set-up during his internship at Argonne National Laboratory.

References