

Effect of Off-Axis Needle Motion on Internal Nozzle and Near Exit Flow in a Multi-Hole Diesel Injector

Michele Battistoni
Universita degli Studi di Perugia

Qingluan Xue and Sibendu Som
Argonne National Laboratory

Eric Pomraning
Convergent Science Inc.

ABSTRACT

The internal structure of Diesel fuel injectors is known to have a significant impact on the nozzle flow and the resulting spray emerging from each hole. In this paper the three-dimensional transient flow structures inside a Diesel injector is studied under nominal (in-axis) and realistic (including off-axis lateral motion) operating conditions of the needle. Numerical simulations are performed in the commercial CFD code CONVERGE, using a two-phase flow representation based on a mixture model with Volume of Fluid (VOF) method. Moving boundaries are easily handled in the code, which uses a cut-cell Cartesian method for grid generation at run time. First, a grid sensitivity study has been performed and mesh requirements are discussed. Then the results of moving needle calculations are discussed. Realistic radial perturbations (wobbles) of the needle motion have been applied to analyze their impact on the nozzle flow characteristics. Needle radial motions are based on high-speed X-ray phase-contrast imaging collected at Argonne National Laboratory. Different types of wobbles are presented and the results are compared to the nominal in-axis motion behavior. Hole-to-hole differences are discussed and quantified. Complex flow structures are observed in the sac region of the nozzle and an explanation for the different flow structures emerging from each hole is provided. From this work it has been observed that in presence of needle wobble, at low and medium lifts hole-to-hole differences are observed. Also, some holes exhibits swirling flow which affects the near nozzle jet structure, the liquid mass distribution and the mass flow rate, resulting in potential spray modifications.

CITATION: Battistoni, M., Xue, Q., Som, S., and Pomraning, E., "Effect of Off-Axis Needle Motion on Internal Nozzle and Near Exit Flow in a Multi-Hole Diesel Injector," *SAE Int. J. Fuels Lubr.* 7(1):2014, doi:10.4271/2014-01-1426.

INTRODUCTION

Diesel injector nozzles have been extensively studied for quite a long time [1, 2], as sprays are recognized to have a strong impact on engine performance and emissions. The improvement of nozzle design has been one of the major research areas. Flow characteristics developed inside an injector determine many aspects of the resulting jets and of the following combustion process [3]. The ultimate purpose of a spray is indeed to increase the liquid surface area exposed to the mass, momentum, and heat transfer phenomena. Many of these features are related to the fluid dynamics that takes place inside the injector.

From a modeling standpoint, the study of the nozzle flow involves the ability to predict transient multi-phase flows due to the likelihood of cavitation [4]. Also the subsequent liquid jet

atomization and breakup are other extremely challenging areas where great efforts have been spent [5]. Breakup initialization is fundamentally three dimensional and strongly coupled with the upstream liquid turbulence generated in the nozzle. Quantifying the link between injector design and spray structure is one of the most challenging aspects of spray research [6]. Several factors can influence the process and the results. First, small sizes on the order of 100 μm hole diameter make manufacturability and reproducibility of nozzles challenging, therefore the nominal geometry is usually different from the actual one. Also, real geometrical data are seldom available, due to the high costs of the measurement techniques [7]. Second, the temporal and spatial scales at which relevant fluid dynamics processes occur are extremely small. Modeling studies in literature have examined the internal flow of diesel nozzles using different approaches. These studies have ranged from three dimensional Reynolds Averaged Navier-

Stokes (RANS) or Unsteady-RANS simulations of fixed or transient needle lift cases [8, 9, 10, 11, 12], to Large Eddy Simulations (LES) of the nozzle flow [13].

While simulations of nozzle flow are particularly challenging, it is even more difficult to incorporate them in complex models of spray and combustion. Multi-Fluid and Volume-of-Fluid (VOF) models are the most common options for nozzle two-phase flow simulations, and these method have the ability to be applied also in the near exit region linking the internal and the external flow [14, 15, 16, 17, 18]. For the internal nozzle flow, these approaches require models to account for turbulence and cavitation at a sub-grid scale level. For the external region, in addition, other models are required to predict liquid jet atomization, like the Eulerian-Lagrangian Spray Atomization Model (ELSA). In fact, it is only very recently that Direct Numerical Simulation (DNS) of high density ratio two-phase flows have been made possible using massively parallel processing [19]. Here no models are required, but calculations are still extremely expensive because of the grid requirements and are not suitable for engineering applications.

Among various aspects of nozzle internal flow that can influence spray dynamics, there is eccentric motion of the injector needle valve. Especially in valve-covered orifice (VCO) nozzles, such motions have been tied to asymmetries in the sprays from multihole nozzles [20, 21]. If the injection duration is long, the effect of lateral motion can have a limited impact on orifice flow. However, the Diesel engine development is moving towards multiple short injections to aid the low temperature combustion operation concepts. Hence, at low needle valve lifts with highly transient operations, off-axis motion (or wobble) may play a more dominant role, not only in VCO but also in mini-SAC type injectors [6, 22].

Transient nozzle flow simulations have been performed in the past decade by several researchers. The transient simulations were performed with an in-axis motion of the needle which was usually obtained from one-dimensional system simulations inside the injector [23]. Under more realistic circumstances, needle can undergo a complex 3D motion including lateral oscillations. Few studies however focused on understanding the fluid dynamics modifications under these conditions. Spathopoulou et al. [21] investigated needle eccentricity by assuming a side movement profile for a VCO nozzle and found significant differences between the flow structures developing in individual holes. Chiavola and Palmieri [20] modeled a SAC nozzle with an assumed needle eccentricity and they also found that eccentric needle displacement is responsible for asymmetric flow generation in the sac volume.

Powell et al. [6] observed significant off-axis motion of the needle in a single-hole research nozzle. They found that side motion measured by means of phase enhanced x-ray phase-contrast imaging correlates with fuel density fluctuations in the near nozzle region, determined using x-ray radiography imaging technique. They also performed CFD simulations of

the nozzle and observed alterations in the sac flow that were not reflected in relevant changes in the nozzle exit flow behavior. Xue et al. [22], recently, modeled both a single-hole and a multi-hole injector under the presence of needle wobble. They also found that for the single-hole orifice, orifice flow was not significantly affected. However, for the multi-hole case several hole-to-hole variations and mass unbalances were identified which were attributed to the needle wobble.

None of these studies so far have been able to decisively quantify the effect of needle wobble on flow development in the needle-seat gap, the sac region, and the ensuing flow through each hole, either experimentally or numerically. 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" motions are quantified in the current study. This forms the major motivation for this work.

Simulations are performed on a five-hole Diesel injector. Multi-hole injectors are expected to be influenced largely by the needle wobbles since potentially the off-axis motion can restrict the flow into some orifices more than others. The paper is organized in the following way. First, an 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. A grid sensitivity study is carried out to find the optimal mesh requirements for grid-independent results. Validation of the in-nozzle flow set-up is presented thereafter for the baseline case, without needle lateral motion. Then the needle oscillations or wobbles are applied in two different ways and effects on the internal nozzle flow are discussed. The final section discusses the results at the orifice exit and in the near exit region of the jet. A summary of the main findings and some planned future studies conclude the paper.

THEORETICAL BASIS

In this section, the Navier-Stokes equations for a homogeneous multi-phase mixture will be introduced along with a description of the Homogeneous Relaxation Model based (HRM) nonequilibrium cavitation model. These models have been recently implemented in the CFD code CONVERGE [26, 27, 28]. The authors have previously published validation of this implementation against Winklhofer et al. [29] data and Argonne x-ray radiography data [30] in the following references by Zhao et al. [28] and Battistoni et al. [31] respectively. In this paper some additional validation is presented for a five-hole nozzle.

Mixture Model

In a single-fluid approach, the homogeneous 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 RANS equations.

homogeneous mixture model is based on the assumption that the velocity, temperature, and pressure between the phases are equal. Mass and momentum equations are given below

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \vec{v} = 0 \quad (1)$$

$$\frac{\partial \rho \vec{v}}{\partial t} + \nabla \cdot \rho \vec{v} \vec{v} = -\nabla p + \nabla \cdot \vec{\tau} + \rho \vec{f} \quad (2)$$

where ρ and $\vec{v}$ are the mixture density and velocity; p is the pressure; $\vec{\tau}$ is the mixture stress-strain tensor due to molecular and turbulent viscosity. Analogous formalisms apply for energy conservation and turbulence closure models [26, 27, 28] and equations are omitted here for brevity.

In the present study, the multi-phase system is comprised of

1. a liquid phase (subscript 1), incompressible;
2. a vapor phase (subscript 2), compressible;
3. non-condensable gases (subscript 3); compressible.

The sum of vapor and non-condensable gases will be referred to with the subscript g . The concept of pseudo-density is used and the mixture density ρ is computed with the following equation:

$$\rho = \alpha_1 \rho_1 + \alpha_2 \rho_2 + \alpha_3 \rho_3 = \alpha_g \rho_g + (1 - \alpha_g) \rho_l \quad (3)$$

The volume and mass fractions are related through

$$\rho_i \alpha_i = Y_i \rho \quad (4)$$

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 α_g is calculated:

$$\frac{\partial \rho Y_i}{\partial t} + \nabla \cdot \rho Y_i \vec{v} = \nabla \cdot (\rho D_i \nabla Y_i) + S_i \quad (5)$$

$$\alpha_g = \frac{Y_g / \rho_g}{\sum Y_i / \rho_i} \quad (6)$$

In the present implementation, since the two gas phases are treated as compressible their density is determined based on the equation of state for an ideal gas:

$$\rho_i = p / R_i T \quad (7)$$

A VOF method is used to model the two-phase flow. A function α is used to represent the void fraction and it is defined as follows:

- $\alpha_g = 0$: the cell is filled with pure liquid
- $\alpha_g = 1$: the cell is filled with pure gas
- $0 < \alpha_g < 1$: the cell is filled with both liquid and gas species

Cavitation Model

In this code the mass exchange between the liquid and vapor is based on a HRM as developed by Bilicki and Kestin [32], and recently proposed by Schmidt et al. [33, 34, 35, 36] for describing non-equilibrium flash boiling processes. Both cavitation and flash-boiling are evaporation phenomena driven by a pressure drop. Cavitation is pressure-driven vaporization occurring at low temperatures in which the vapor density is so small that the latent heat flow does not affect the phenomenon. As a result, the time scale of heat transfer is much faster than the time scale of bulk motion, therefore the latter is basically controlled by the inertia of the liquid. On the contrary, a flash-boiling process occurs at elevated temperatures and in this case vapor density is much higher, therefore the liquid must provide more energy per unit volume of vapor. The process takes a non-negligible time scale and the dynamics is controlled by finite-rate heat transfer rather than by inertia [37]. In this framework the HRM approach assumes a first order rate equation for the evolution of the instantaneous mass fraction of vapor Y_v towards its equilibrium value $\bar{Y}_v$ over a given time-scale Θ . The model as proposed by Bilicki and Kestin [32] is given below

$$\frac{dY_v}{dt} = \frac{\bar{Y}_v - Y_v}{\Theta} \quad (8)$$

The equilibrium vapor quality $\bar{Y}_v$ is a function of the thermodynamic properties (enthalpy) at the local pressure, i.e.

$$\bar{Y}_v = (h - h_l) / (h_v - h_l) \quad (9)$$

The time-scale Θ is evaluated using an empirical fit proposed by Downar- Zapolski [38]

$$\Theta = 3.84 \cdot 10^{-7} \alpha^{-0.54} \phi^{-1.76} \quad [s] \quad (10)$$

where

$$\phi = \frac{P_{sat} - P}{P_{crit} - P_{sat}} \quad (11)$$

and it was determined in experiments of flashing flows of water in pipes with upstream pressures of in excess of 10 MPa.

SIMULATION METHODOLOGY AND NEEDLE MOTION

The main focus of this work is the numerical modeling of needle off-axis motion effects on nozzle flow and near exit jet characteristics, covering an actual transient injection event. The influence of needle motion including lateral oscillations is investigated on a micro-sac multi-hole injector by performing 3D simulations using the Renormalization Group (RNG) $k-\epsilon$ turbulence model. This section outlines the boundary and initial conditions for the injector model followed by some geometrical and meshing information.

Injector Nozzle Description

The simulated multi-hole injector is shown in [Figure 1](#). The 5-hole micro-sac injector geometry is obtained from the work of Payri et al. [39] and Benajes et al. [40]. The injector unit is a typical common-rail Diesel injector, with hole exit diameter of $130\text{ }\mu\text{m}$, ks-factor of 1.5, and hydro-grinded inlet edge with $R = 30\text{ }\mu\text{m}$. More nozzle details are displayed in [Table 1](#). It is worth pointing out that, in the model, each of the holes has the same nominal dimensions. Hence, even though there might be differences in the actual geometry between different holes, the simulations do not account for these effects since the geometry used in the simulations is a nominal one.

The fuel is a Diesel surrogate which was modeled using a density of 848 kg/m^3 , a viscosity of 0.0027 Ns/m^2 and a saturation pressure of 1000 Pa . For other relevant properties ndodecane has been considered. Operating conditions chosen for the study are shown in [Table 2](#). The injection pressure is 780 bar and the back pressure is 20 bar . Fuel and outlet chamber were set at 300 K . A small amount of air (non-condensable gas) is also added to the liquid fuel in the model. Following previous works by the Authors [31], the value adopted is $2\text{E-}5$ by mass, which represents a typical concentration of dissolved gases found in a liquid fuel in normal conditions. Pressure and temperature were assigned at the inlet surface as boundary condition. The injector domain and orifices were initially filled with liquid set at the same injection pressure and temperature. The outlet chamber was filled with pure air at the back pressure and temperature levels. The outlet chamber volume is large enough to be modeled as a closed domain. Lastly, a special note concerning inlet pressure assignment is required. A constant pressure value has been applied on the upper portion of the domain, even if it is fairly close to the needle valve region. The influence of the upper portion of the injector is neglected in this study. Further studies will be needed to address if this issue actually matters, and to quantify, in case, its impact on the nozzle flow results.

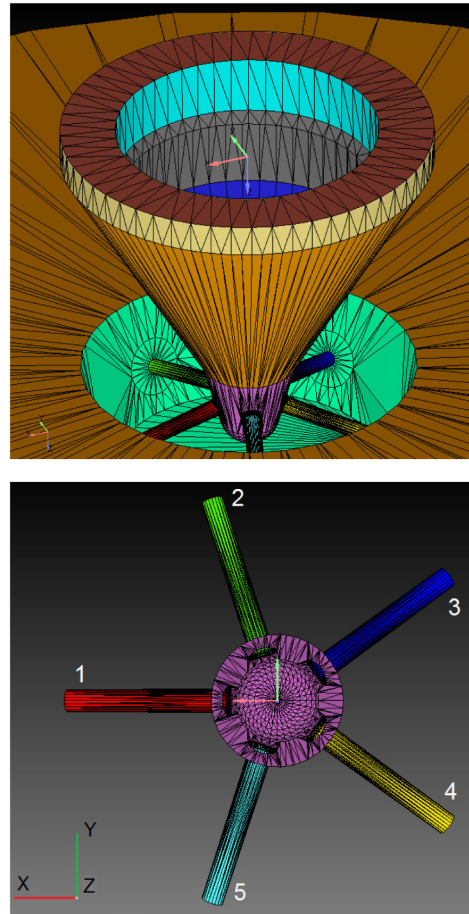


Figure 1. Geometry of the five-hole micro-sac nozzle. General view (a). Top-view of sac and holes showing hole numbering (b).

Table 1. Nozzle details (from Payri et al. [39]).

Hole Exit Diameter [μm]	130
Hole Inlet Diameter [μm]	145
Ks parameter	1.5
Inlet edge rounding radius [μm]	30
Hole Length [mm]	1
Number of holes	5
Sac + holes Volume [mm^3]	0.188
Hole inclination angle [$^\circ$]	75

Table 2. Injection condition.

Injection pressure [bar]	780
Back pressure [bar]	20
Fuel type	Diesel
Peak needle lift [μm]	~ 225
Injection duration [ms]	~ 2.0

Needle Lift and Off-Axis Motion

Recent advancements in the diagnostics techniques like high-speed x-ray phase-contrast imaging [7, 41], have shed light on many details and have provided quantitative data useful to assess needle dynamics. These studies have shown that needle motion usually has the following features:

- Lateral oscillations can be more pronounced in SAC than in VCO injectors, since in the latter needle guides are generally located closer to the tip [41].
- Needles undergo some degree of lateral motion, which starts when the needle first rises from its seat and continues for the duration of the injection event [41].
- Lateral needle oscillations do not correlate with injector line pressure fluctuations, but they are purely mechanical in nature, as suggested by Manin et al. in [42];
- Separate analysis of x and y motion components (assuming the needle is lifting along z) reveals that these lateral oscillations are quite often in-phase, therefore the lateral movement occurs in a plane [7, 41]. However, some injectors show out-of-phase x and y motions that correspond to a complex helical tip trajectory.
- Oscillations occur mainly on one side and the tip does not move back to the opposite side crossing the centerline.

In summary, these oscillations have been identified as cantilever-type vibrations of the needle with respect to the needle guide, which is located upstream of the tip. They are likely due to an elastic relaxation from an initial off-axis constraint of the needle on the seat when it is closed [7, 41].

Based on these observations, a micro-sac injector has been chosen and a specific needle wobble profile has been considered in the present study as it is displayed in Figure 2. Curves show the needle in-axis motion and the off-axis in-plane motion that have been adopted. Lift is along z axis and data shown refer to the magnitude (according to the CAD model in Figure 1 the actual direction is -z). The eccentricity curve shown is the actual magnitude of the radial perturbation applied to the needle motion, and it occurs in a plane (no helical motions have been considered). Lift curve is obtained from the work by Payri et al. [39] and the data have been measured for the specific injector and the selected operating condition. Eccentricity instead has not been measured directly on the injector, but has been obtained from ref. [41] by adapting measurement taken at the Advanced Photon Source (APS) at Argonne National Laboratory on similar injectors. By using the same amplitude of 28 μm as in [41], the period of the oscillations has been just slightly adapted to match the injection duration. As a result, three oscillation peaks have been applied over a needle opening of approximately 2 ms.

In order to carry out a comprehensive numerical analysis, three needle motion conditions have been simulated, as listed in Table 3. The first case is the baseline case, with pure in-axis motion along z. In the second case, the needle oscillates exactly towards one of the five holes (hole # 1 specifically, see

also Figure 1); the underlying idea is to study the effect of a lateral needle motion that would probably constrict the flow through one specific hole. The third case represents the opposite situation where the needle oscillates in a direction in between two holes (holes # 3 and # 4 specifically, see Figure 1).

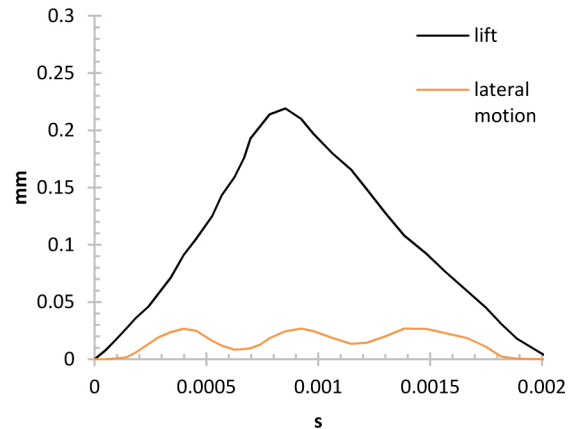


Figure 2. Needle lift and needle off-axis motion profiles vs. time used for the injector simulations (see Table 3 for case definition).

Table 3. Needle motion cases.

#	Case name	Lift	Lateral motion
1	Baseline	Lift profile shown in Figure 2	None
2	+x wobbles	Lift profile shown in Figure 2	Magnitude: in-plane eccentricity shown in Figure 2, Direction: applied along +x axis (towards hole # 1)
3	-x wobbles	Lift profile shown in Figure 2	Magnitude: in-plane eccentricity shown in Figure 2, Direction: applied along -x axis (in between two holes, i.e. hole # 3 and # 4)

Grid

A close up view of the computational grid used in the simulations is shown in Figure 3. The mesh consists of the hollow cylinder around the needle shaft, the needle seat region, the nozzle sac, five holes and a large outlet chamber. The Cartesian cut-cell method allows for the needle off-axis motion described in Figure 2 to be accounted for in the simulations in a straightforward manner. Figure 3 shows a view of the grid generated at 0.39 ms, when needle eccentricity is also visible. Needle movement is accomplished by moving the needle surface at solver run-time. The cut-cell grid is generated consequently at each time-step. To prevent adoption of extremely small cells at near-zero needle lift, a minimal gap has been used in the study (18 μm in the present cases, which correspond to about 8% of the maximum lift). More information about these meshing strategies are available from the CONVERGE manual [27] and other authors' previous publications [22, 28].