

Numerical Modelling of the In-Nozzle Flow of a Diesel Injector with Moving Needle during and after the End of a Full Injection Event

Nikolaos Papadopoulos and Pavlos Aleiferis
University College London

ABSTRACT

The design of a Diesel injector is a key factor in achieving higher engine efficiency. The injector's fuel atomisation characteristics are also critical for minimising toxic emissions such as unburnt Hydrocarbons (HC). However, when developing injection systems, the small dimensions of the nozzle render optical experimental investigations very challenging under realistic engine conditions. Therefore, Computational Fluid Dynamics (CFD) can be used instead. For the present work, transient, Volume Of Fluid (VOF), multiphase simulations of the flow inside and immediately downstream of a real-size multi-hole nozzle were performed, during and after the injection event with a small air chamber coupled to the injector downstream of the nozzle exit. A Reynolds Averaged Navier-Stokes (RANS) approach was used to account for turbulence. Grid dependency studies were performed with 200k-1.5M cells. Both $k-\epsilon$ and $k-\omega$ SST models were considered in the validation process, with the $k-\omega$ SST found to predict better the injector's flow rate. The cavitation models of Schnerr-Sauer and the Zwart-Gerber-Belamri were employed for validation against optical data of cavitation in a simplified nozzle geometry obtained from the literature. The Schnerr-Sauer model was in better agreement with the experiments, hence this model was subsequently employed for the real injector simulations. The motion of the injector needle was modeled by a dynamic grid methodology. An injection pressure of 400 bar was applied at the inlet of the injector. Two outlet pressures were examined, 60 bar and 1 bar. The results showed that the flow was far from steady-state during the injection event and that hysteresis existed between the needle opening and closing phases. This indicated the importance of transient simulations, contrary to widely-used steady state simulations at fixed needle lifts. The two outlet pressures resulted in very different final states of the flow-field in the nozzle. Specifically, the nozzle ended up either full of liquid fuel at the end of injection or full of air after most of the fuel had been ejected into the chamber downstream. These predictions highlighted phenomena that can increase HC emissions due to fuel leakage, as well as processes that may be linked to different formation mechanisms of nozzle deposits.

CITATION: Papadopoulos, N. and Aleiferis, P., "Numerical Modelling of the In-Nozzle Flow of a Diesel Injector with Moving Needle during and after the End of a Full Injection Event," *SAE Int. J. Engines* 8(5):2015, doi:10.4271/2015-24-2472.

INTRODUCTION

Background

New regulations that constantly call for lower exhaust emissions, as well as international obligations to focus on sustainability, demand higher engine efficiency. Diesel engines are a source of air pollutants, with unburnt Hydrocarbons (HC) being particularly toxic. Unburnt HC mainly form due to poor air-fuel mixing and combustion. Under mixing can be caused by fuel that ends up on the cylinder and piston walls from spray impingement or from fuel that enters the chamber late in the combustion process with low velocity. A source of the latter form can be fuel which is coming out from the nozzle hole or nozzle sac volume after the end of injection and does not mix with air sufficiently [1]. Furthermore, after the end of combustion, increased temperatures may cause fuel that is left in the sac to evaporate and move towards the chamber through the orifice. The lighter compounds evaporate first, leaving back the heavier ones, which can create deposits inside the nozzle. Those deposits can harm the injector and reduce their life time as well as the engine's performance and efficiency [2].

The injector can be designed with a smaller sac volume to reduce HC emissions. In Valve Covered Orifice (VCO) injectors the needle closes the nozzle's inlet so that fuel from the sac cannot enter the chamber. Such designs can minimize emissions [3] [4]. However, the presence of a sac is important to equalize to the pressure of the fuel at the nozzle inlets. A sacless injector typically produces poor quality sprays that can be different from each orifice of the same injector [5] [6]. The impact can be observed via different metrics, such as spray penetration, spray cone angle and rate of injection.

At real engine conditions, to promote atomization, Diesel fuel is injected with very high pressures, typically up to 2000 bar - with values expected to increase even further in the future. It has been shown that under these conditions, cavitation appears inside the nozzle [7] [8], and vapour bubbles are formed. In addition, 'hydraulic flip' that is linked to in-nozzle flow separation at the orifice inlet that never reattaches upstream the nozzle exit, might occur and affect spray formation [6].

In the case of consecutive injections, bubbles of vapour have been noticed to exist inside the nozzle before the start of injection [9] [10]. These have a random pattern of distribution, with various sizes and at various locations and are thought to be created by cavitation in the last stages of the previous injection event. Those bubbles, given enough time, tend to coalesce and form one big vapour area that fills most of the sac's volume. Also, air is entrained inside the nozzle orifice from the outlet [10].

Apart from experimentation, in-nozzle flow insights have been obtained by Computational Fluid Dynamics (CFD) due to challenges associated with the faithful manufacturing of real-size optical nozzles with moving needles that could provide satisfying quantitative information. This has been done for both Diesel and gasoline injector geometries. The existence of vapour bubbles has been noticed in [11] and attributed to the inertia of the flow while the needle was still open; this caused the pressure inside the nozzle to drop abruptly when the needle closed, promoting cavitation. In [12] it was noticed that during the injection event, air entered the orifices through flow recirculation at the nozzle exit.

While most of the published studies of injection simulations employed computational domains that extended only up to the nozzle exit, some researchers have included a small part of the combustion chamber as well [13] [14] [15]. This has been done in order to impose a boundary condition to the injector flow that could allow capturing the existence of hydraulic flip and also part of the ensuing spray. In such cases, most often, the Volume Of Fluid (VOF) model is used [12] [13] [16], which provides liquid-gas interface tracking. In combination with Large Eddy Simulation (LES) on a sufficiently dense grid, this method may also be able to predict the primary breakup [15] [17]. Also, other gases can be included in the calculation, such as air [12] [16].

Despite the transient flow and motion of the needle, it is common practice to simulate the flow as steady state at different fixed needle lifts [18] [19] [20] [21], or only at full needle lift [22]. Some work has been done on the effect of the needle's motion on the flow. For example [23] used a single-hole injector and simulated a 90° sector of the real geometry due to periodicity. In contrast, [24] [25] used multi-hole injectors but, again, due to periodicity, only one orifice was modelled. Moving needle simulations that include a part of the injection chamber have also been performed. In [26] the injection chamber was modelled as full of liquid fuel, whilst in [27] an injection chamber full of air was used for the simulation of a pressure-swirl injector and in [28] a similar setup was used, in combination with VOF, to simulate the injection from a single-hole injector. It has been shown that the flow during the opening and closing stages of the needle's motion is transient, and hysteresis effects can take place [23] [25] [26]. However, in [29] no transient phenomena were noticed, potentially due to the high inlet pressure, but there was a significant effect from the needle off-axis motion.

The flow at the end of injection has also been given some attention, due to its effect on performance and emissions. [30] performed X-ray radiography experiments to investigate the phenomena involved. [31] performed RANS simulations with a mixture multiphase model and compared their predictions to experimental results. Cavitation was noticed to occur after the closing of the needle as well as nozzle backfilling with ambient gas; there was also fuel dribble in the area near the nozzle exit.

Present Contribution

Despite a significant amount of background studies on in-nozzle flows, there is need for more information on simulations of a vertical multi-hole injector with a moving needle that has also incorporated a part of the combustion chamber filled with air as downstream boundary condition; this is to study aspects of in-nozzle phenomena both during the injection event and past the end of it. The work presented here includes transient Reynolds-Averaged Navier-Stokes (RANS) simulations of the full injection process as emerging through the geometry of a real-size multi-hole Diesel injector. The VOF multiphase model was employed. The focus was primarily on understanding the in-nozzle phenomena and not on simulating the spray formation process past the nozzle exit. The main objectives of the current work can be summarised as follows:

- To investigate the effect of the needle motion on the major characteristics of the in-nozzle flow, including the formation of vortical flow structures and cavitation during the injection event.
- To investigate the predictive effect of the presence of an air chamber downstream of the nozzle exit on the in-nozzle flow, particularly with respect to nozzle backfilling phenomena that can occur at the end of injection after needle closure and lead to simultaneous presence of liquid fuel, fuel vapour and air inside the nozzle.
- To study the effect of different 'back' pressure conditions (*i.e.* in-cylinder air pressure) on key characteristics of the in-nozzle flow, both during injection and after needle closure.

METHODOLOGY

Mathematical Formulation

Within the objectives of the current work, a commercial CFD code was employed that solves numerically the governing equations of fluid motion on a discretized computational domain by the finite volume methodology (Ansys Fluent) [32]. The flow under consideration was turbulent; a RANS formulation was employed for faithful approximation of the average quantities of the flow field and under no circumstances prediction of the jet breakup was sought after. Within this framework of study, two different turbulence modelling approaches were tested, the k - ϵ approach [33] [34] and the k - ω SST [35]. This was done because the geometry of study was associated with high pressure gradients and flow separation and reattachment effects that the k - ω SST is believed to handle more accurately than the k - ϵ .

Multiphase Flow and Cavitation Models

When two or more phases exist in a simulation, a multiphase formulation must be used to account for those. The intention of the present simulations was not to capture the interface of the bubbles within the nozzle, neither to obtain a sharp prediction of the liquid spray interface in the air chamber during the injection process, as this would require extremely dense grids and practically unrealistic running times. Instead, the main intention was to capture the liquid-air interface within the nozzle after needle closure and during backflow nozzle-filling events, *e.g.* to understand the formation of in-nozzle liquid film phenomena amongst other. Therefore, use of a VOF method was considered necessary [32]. With this methodology, when cavitation occurs and vapour appears inside the liquid continuum, this does not happen in the form of bubbles but as an average quantity inside the cell, similar to the mixture model behavior [32].

Specifically, in the Diesel injector under study, Diesel liquid, Diesel vapour and air were all considered present. VOF solves a continuity equation for the volume fraction of $n-1$ phases, with n being the total number of phases present in the simulation, as follows:

$$\frac{\partial}{\partial t}(\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q \mathbf{U}_q) = S_{\alpha_q} + \sum_{p=1}^n (\dot{m}_{pq} - \dot{m}_{qp}) \quad (1)$$

where α_q is the volume fraction of phase q , ρ_q is the density of phase q and $\dot{m}_{pq}$ is the mass transfer from phase q to phase p . S_{α_q} represents any source of phase q that might exist. The phase that is not being solved for will be calculated based on the constraint that the sum of all volume fractions in a cell must be equal to 1. The discretization scheme that was used for the solution of this equation was the Modified HRIC [32] [36].

The presence of cavitation was modelled by a mass transfer mechanism that converts the mass of a specified liquid to a specified gas (vapour). This happens when certain criteria are met, typically in this case, the condition is the local pressure. When the pressure drops below the vapour pressure, liquid is converted to vapour, while when the pressure rises again, vapour turns back to liquid. A sensitivity study was carried out between two cavitation models, the Schnerr-Sauer [37] and the Zwart-Gerber-Belamri [38]. Within the Schnerr-Sauer model formulation the vapour source term is:

$$R = \frac{\rho_v \rho_l}{\rho} \frac{d\alpha}{dt} \quad (2)$$

where:

$$\alpha = \frac{n_b \frac{4}{3} \pi R_b^3}{1 + n_b \frac{4}{3} \pi R_b^3} \quad (3)$$

with n_b being the number of bubbles in the volume of liquid (typically set as a constant of the order 10^{13}) and R_b is the bubble radius calculated by the Rayleigh-Plesset equation. In the Zwart-Gerber-Belamri model the vapour source term is calculated by:

$$R = n \left(4\pi R_b^2 \rho_v \frac{dR_b}{dt} \right) \quad (4)$$

where n is the bubble number density.

Within the objectives of the current simulations for the specific injector geometry under study, the liquid density was assumed constant and the fuel vapour and air densities were calculated by the ideal gas equation of state. As will be detailed later, the inlet and outlet boundaries were set to a temperature of 300 K. This was done partly because the experimental flow rate data that were available for validation had been obtained at this temperature. Another reason was that at higher temperatures evaporation and/or boiling could take place. These are complicated mass transfer phenomena that need appropriate sub-modelling features in the context of a faithful multi-phase calculation at high temperature. Implementation of the necessary submodels is currently work in progress by the current authors and investigation of the effects of such phenomena on the in-nozzle flow will be reported in a future publication. Table 1 summarises the properties of the liquid Diesel phase, vapour Diesel and air.

Table 1. Fluid properties used in the simulations.

Liquid Density	809.38 kg/m ³
Liquid Viscosity	1.967×10 ⁻³ kg/m·s
Vapor Pressure	1000 Pa
Surface tension	0.02 N/m
Vapour Viscosity	1.34×10 ⁻⁵ kg/m·s
Air Viscosity	1.79×10 ⁻⁵ kg/m·s

Injector Geometry

The geometry of a real, vertical multi-hole injector with sac volume was used. The total length of the geometry in the direction of the axis of the needle is 7 mm. There are 7 orifices with a length of 0.6 mm and diameter of approximately 0.12 mm. The inlet curvature radius is 0.02 mm, resulting in an r/D ratio equal to 1/6, and the inclination angle is 84° . The periodicity of the geometry allowed the use of only one sector with one orifice, in order to save computational time. The geometry is illustrated in [Figure 1](#).

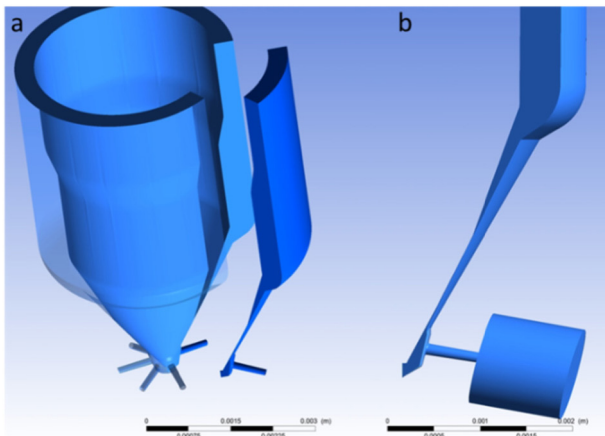


Figure 1. a) Geometry of the multi-hole injector. b) Sector of a single hole with air chamber attached.

In order to allow simulations of the fluid motion after the end of injection an extra volume (of length equal to 10 nozzle diameters), considered to be a part of the combustion chamber, was attached to the nozzle exit ([Figure 1](#)). A similar method has been used in [14] [26] as an approach to providing more accurate pressure boundary conditions, but the interior of the chamber was set to be full of liquid. The studies of [13] [16] [15] [27] have used a chamber full of air as initial condition in order to investigate the flow in the vicinity of the nozzle outlet and partial hydraulic flip.

Boundary and Initial Conditions

The boundary and initial conditions of the simulations are summarized as follows:

- A pressure inlet, where the total pressure was constant and set to 400 bar for all simulations. The liquid volume fraction at the inlet was 1.
- Static pressure at the outlet, which consisted of the circumferential side and the bottom side of the added air volume. Two different values of pressure were used and compared, 60 bar and 1 bar absolute pressure.
- The injector body, including the orifice and the top side of the attached volume were set as no-slip walls.

- The sides of the geometry were periodic boundaries. Within the objectives of the current RANS work, such an approach can provide a satisfying description of the flow field in the nozzle hole. However, in practice this approach may not be always realistic [39], so for more detailed simulations, *e.g.* when adopting LES, use of such periodic conditions would need to be reconsidered via sensitivity analysis.
- All the walls were considered adiabatic. The inlet and outlet boundaries were set to a temperature of 300 K.
- At the beginning of the simulation, the flow field was considered to be at rest. The nozzle was full of liquid fuel, while the injection chamber was full of air.

Mesh and Needle Motion

For the simulations performed, a moving mesh strategy was used by adopting the layering technique [32]. Specifically, the computational domain consisted of two regions. One was the non-moving region, which was actually the geometry when the needle was at its fully closed position. The moving region was made out of the volume that was created as the needle moved. When the needle moved, the layer of cells in touch with the needle changed size in order to account for this motion. When the size of those cells exceeded certain limits, the cells either divided into smaller ones, thus creating a new layer, or combined with the neighboring layer. The needle was a moving no-slip wall boundary. Practically, for all simulations, the needle never actually touched the needle seat. This is because the code cannot handle two different wall boundaries when these touch. In order to seal the high-pressure zone from the sac and nozzle area at needle closure, a wall interface was enabled. Specifically, there was an interface that connected the needle seat area with the needle throughout the whole simulation. While the needle was in motion, this interface was considered “interior”, so it had no effect on the flow. However, just before the start of the needle's motion, or when the needle descended to $0.5 \mu\text{m}$ lift, this interface was switched to “wall” this then resulted in two disconnected domains [31], interrupting the flow passing through and sealing the high-pressure from the low-pressure zone. During the period that this interface was defined as wall, it consisted of 4 cells in thickness.

The needle lift curve is illustrated in [Figure 2](#). This defined how the needle moved with respect to time after the start of injection. Compared to the maximum lift of $256 \mu\text{m}$, $0.5 \mu\text{m}$ was considered negligible. The injection duration was 1.86 ms long, during which the needle was ascending for 0.88 ms and descending for 0.98 ms.

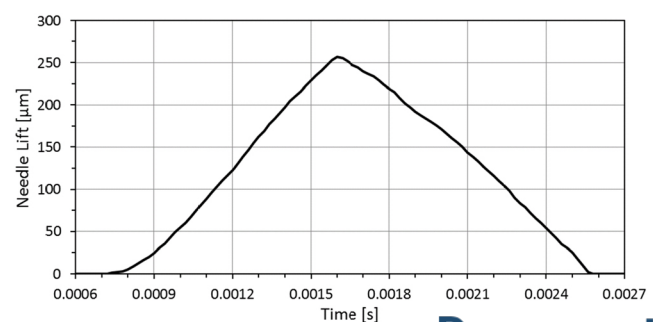


Figure 2. Needle lift curve.

A block-structured mesh was used for the discretization air domain. Aim of the mesh generation procedure was a fine boundary layer resolution, both inside the orifice and at all the wall boundaries upstream of it. The achieved wall y^+ was always less than 1 in the region of the sac and nozzle orifice. Details of the mesh are shown in Figure 3. When the needle was closed, the mesh consisted of 611,478 cells, 391,878 of which were part of the nozzle, while the rest were part of the chamber. At full lift there were 1,130,094 cells in total. 50 cells were typically placed along the nozzle diameter. At the core of the orifice the cell size was $\sim 4 \mu\text{m}$, while the first cell at the wall was $\sim 0.15 \mu\text{m}$. The average size at the sac area was also $\sim 4 \mu\text{m}$.

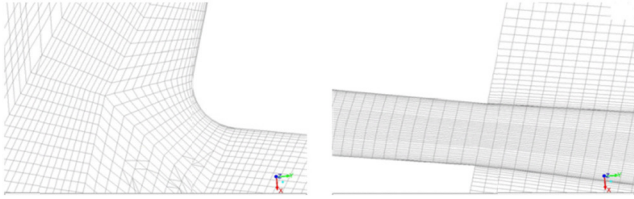


Figure 3. Details of the mesh at the orifice inlet and outlet areas.

For grid sensitivity analysis, three more meshes were created and tested. A coarse one, with 120,000 cells at minimum needle lift (excluding the extra air volume), a medium one with 250,000 cells and a very fine mesh with 1,500,000 cells. The predicted volumetric flow rate throughout the whole injection event was very close with all four meshes, with a maximum difference of 0.5% between the coarse and very fine mesh. Figure 4 compares the velocity profile of the four meshes along a diameter at the orifice inlet (X co-ordinate), where flow separation and recirculation was found to occur. The pressure boundary conditions were 400 bar inlet, 60 bar outlet with the $k-\omega$ SST model. Second-order discretisation was employed for this exercise, as well as for all the results to be presented thereafter.

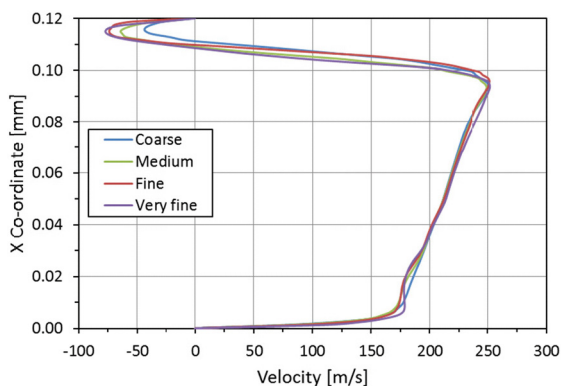


Figure 4. Velocity profile at orifice inlet.

The length of the recirculation zone was calculated for all meshes and the result is shown in Figure 5. The difference between the fine and the medium mesh was of the order of 4.6%, while for the fine and the very fine it was 2.1%. The very fine mesh was not considered practical within the limitations of the available computational power,

as it was much more computationally demanding without providing respectively higher detail. For this reason, the rest of the simulations in this paper were performed with the fine mesh.

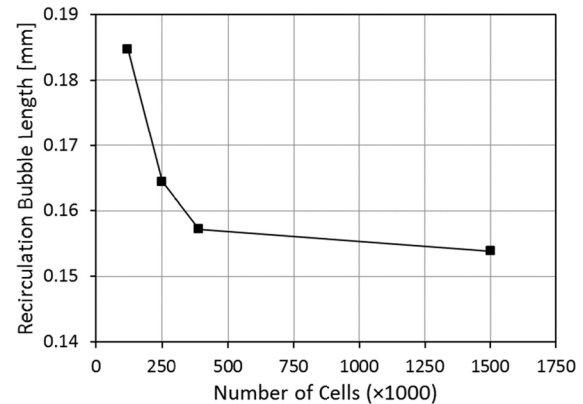


Figure 5. Length of recirculation zone.

VALIDATION

Validating in-nozzle flow simulations can be a quite challenging process, especially under realistic injector conditions. This challenge is primarily associated with the lack of available data, particular of optical nature and of quantitative type from faithful real-size optical injector nozzles. Two different validation processes of the suggested methodology were finally selected and performed within the bounds of the current work. The first one aimed at studying the flow rate during the injection process and comparing this against measured flow rate data to study the effect of turbulence model. The second one was based on a published study of cavitation in an optical nozzle obtained from the literature. This aimed at comparing the cavitation pattern under different conditions with the two cavitation models.

Injector Flow Rate

The volumetric flow rate at the injector's inlet during a whole injection cycle is illustrated in Figure 6. This figure compares the experimental data (obtained by volume flow rate measurements at the injector line upstream of the injector) to the computational results for inlet pressure of 400 bar and outlet pressure of 60 bar with the $k-\epsilon$ and $k-\omega$ SST models. This combination of inlet-outlet pressures was selected because these were the only conditions at which measured flow rate data were available for this particular injector geometry, therefore only these conditions could enable direct comparison between simulations and experiments. Three different timings, termed 1st, 2nd and 3rd time instance, are illustrated in Figure 6. These are the timings at which contours of in-nozzle liquid volume fraction and velocity vectors are shown and discussed later in the results section. They correspond to needle lifts of 25 μm during the needle opening period, maximum lift of $\sim 250 \mu\text{m}$ and 25 μm during the needle closing period (see Figure 2).

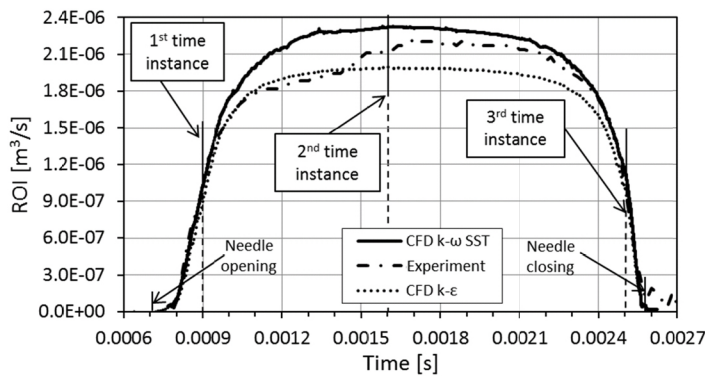


Figure 6. Volumetric flow rate at the inlet of the injector throughout injection.

A satisfying agreement with the experimental data is noticed at the very early stages of needle opening and at the latest stages of closing with the $k-\omega$ SST model. During the period that the flow rate is independent of the needle lift, however, a difference between the experimental and computational $k-\omega$ SST line is observed. This is particularly noticeable at the first half where the needle lift increases. In contrast, the $k-\varepsilon$ model seems to predict better the flow rate for most of the early stages of injection but then under-predicts the rate past maximum needle lift.

This behaviour was considered carefully and it is believed that that reasoning behind these observations can be associated directly with the fixed inlet pressure boundary condition of 400 bar that was used in the simulations. In practice, this fixed value is likely to vary from the actual pressure at the inlet of the injector during the early stages of injection. When the needle starts lifting, a pressure drop is typically noticed on the injector's fuel line, which recovers later in the injection process. Therefore the measured flowrate exhibits an asymmetric profile when comparing pre- and post- maximum needle lift periods, clearly indicating a slower rate on the early period. Unfortunately, the line pressure of the experiment was not available. Therefore, data from the literature were used to study potential effects. Specifically, the line pressure from [10] was used to draw conclusions because the respective flow rate was very similar to the one of the current study. The line pressure of [10] dropped abruptly at the beginning of the injection by ~13% of the nominal value and did not recover completely till the end of injection. In the spirit of this observation, a test simulation with a pressure inlet of 360 bar, *i.e.* 10% lower than the nominal of 400 bar, was performed. The resulting flow rate was found to match the flow rate of the experiment in the first half of the injection event (shown in Figure 6) with tolerance of ~5%. A similar type of behaviour has been noticed in [40] and [41]. Moreover, the computational work of [12] also used a constant pressure inlet and the resulting flow rate was similar to the one shown in Figure 6. With such pressure fluctuations in mind, the difference between the experimental and computational volume flow rate, can

be considered to be within 3% throughout the whole duration of the injection, based on the second half of the simulation, where the needle was descending. More to the point, the $k-\varepsilon$ turbulence model that appeared to be in better agreement with the experimental results in the first part of the simulation in Figure 6 would produce a much lower volume flow rate with the use of a correct varying inlet pressure at the early stages. Close up study of the in-nozzle flow field predictions of the $k-\varepsilon$ model also revealed much different recirculation zones than those predicted by the $k-\omega$ SST model which could be directly linked to the different discharge coefficient implied by Figure 6 for $k-\varepsilon$. For example, the flow exhibited a tendency to remain attached to the wall both at the orifice inlet and at the nozzle outlet where the liquid jet emerged. Therefore, finally the $k-\omega$ SST was considered more appropriate for this application and it was employed throughout the rest of the simulations. For completeness it is mentioned that the predicted flow rate curve of the the $k-\omega$ SST model was independent of cavitation model; both the Schnerr-Sauer and the Zwart-Gerber-Belamri models led to identical flow rates as this case was associated with almost no presence of cavitation (as will be discussed in detail later).

Optical Nozzle Study

The experimental work of [42] has been used by some authors to validate simulated cavitating flows. However, conflicting observations have been reported. For example, it was found in [43] that the Schnerr-Sauer model underestimated the size of the cavitation zone but it matched better the velocity profiles than the Zwart-Gerber-Belamri. Therefore, it was decided to identify a different optical geometry with sufficiently rich data that could be used for validation. After extensive survey, the experimental optical geometry of [44] was finally selected. This was done because [44] offered all necessary dimensions to faithfully model their optical nozzle, as well as a multitude of experimental data at different Reynolds numbers and fluid temperatures to compare with. Temperature effects may not be directly relevant to the Diesel injector simulations presented in the current paper, but as mentioned earlier, studies at different temperatures form part of our work in progress, hence [44] was considered useful to validate the methodology over a range of fluid temperatures too.

The geometry of [44] was a quasi-2D nozzle of 4 mm width, so a two-dimensional mesh was created. The computational domain extended 10 times the width upstream of the nozzle inlet and downstream of the nozzle exit. The mesh consisted of 15,825 cells with 50 cells employed across the nozzle's width. By varying the inlet velocity and the fluid's temperature (and consequently viscosity, vapour pressure, *etc.*), the effect of Reynolds number and temperature on the length of the cavitation zone was investigated and compared to the experimental results. Figure 7(i) illustrates contours of the predicted liquid volume fraction distribution produced by the