



City Research Online

City, University of London Institutional Repository

Citation: Gomez Santos, E., Shi, J., Gavaises, M., Soteriou, C., Winterbourn, M. & Bauer, W. (2019). Investigation of cavitation and air entrainment during pilot injection in real-size multi-hole diesel nozzles. *Fuel*, 263, 116746. doi: 10.1016/j.fuel.2019.116746

This is the accepted version of the paper.

This version of the publication may differ from the final published version.

Permanent repository link: <https://openaccess.city.ac.uk/id/eprint/23265/>

Link to published version: <https://doi.org/10.1016/j.fuel.2019.116746>

Copyright: City Research Online aims to make research outputs of City, University of London available to a wider audience. Copyright and Moral Rights remain with the author(s) and/or copyright holders. URLs from City Research Online may be freely distributed and linked to.

Reuse: Copies of full items can be used for personal research or study, educational, or not-for-profit purposes without prior permission or charge. Provided that the authors, title and full bibliographic details are credited, a hyperlink and/or URL is given for the original metadata page and the content is not changed in any way.

Investigation of cavitation and air entrainment during pilot injection in real-size multi-hole diesel nozzles

Eduardo Gomez Santos^{a,c}, Junmei Shi^{a,*}, Manolis Gavaises^c,
Celia Soteriou^b, Mark Winterbourn^b, Wolfgang Bauer^d

^a*Delphi Technologies, Avenue de Luxembourg, 4940 Bascharage, Luxembourg*

^b*Delphi Technologies, Courteney Rd, Gillingham ME8 0RU, UK*

^c*City, University of London, Northampton Square, London EC1V 0HB, UK*

^d*ANSYS Germany GmbH, Staudenfeldweg 20, 83624 Otterfing, Germany*

Abstract

This paper investigates the complex multiphase flow developing inside the micro-orifices of diesel injector nozzles during pilot injection. High speed micro-visualisations of a transparent serial production nozzle tip replica are used to record the multiphase flow inside the flow orifices as well as near-nozzle spray development. The physical processes taking place are explained with the aid of a three-phase (liquid, vapour and air) homogeneous mixture model utilized in the context of Large Eddy Simulations. Phase-change due to cavitation is considered with a model based on the Rayleigh-Plesset equation, while compressibility of all the phases is accounted for. Numerical simulations shed light on the interaction between the vortex flow and cavitation formation that take place simultaneously with air entrainment from the surrounding environment into the injector's sac volume during the injection.

*Corresponding author

Email address: junmei.shi@delphi.com (Junmei Shi)

tion and the dwell time between successive injections. The experimentally observed flow phenomena are well captured by the simulation model. In particular the compression of pre-existing air bubbles inside the injector's sac volume during the injector opening, cavitation vapor condensation and air suction after the needle closure are well reproduced.

Keywords: LES, Multiphase flow, Cavitation, Fuel Injection, Pilot injection, Air entrainment

1. Introduction

New European Real Driving Emission (RDE) driving cycle legislations require significant research efforts to develop emission compliant and efficient passenger car engines [1]. In this context, the so-called digital injection schemes, used to split the fuel injection into multiple small injections with close separation among them, are widely applied in modern diesel engines in order to obtain simultaneous reductions in noise and emissions without compromising engine performance and fuel consumption [2, 3]. Although the nozzle flow for static needle lift conditions has been extensively investigated (see selectively [4, 5, 6, 7]), not much work is available for the flow development during the dynamic operation of the injector, which plays a key influence on emissions [8, 9].

The digital injection schemes are often operated with fast injector needle opening and closing and with very small separation between injections; with typical dwell time of the order of $50\mu s$. This results in highly transient flow

Nomenclature

α_{air}	air volume fraction	[-]	D	injection hole diameter	[m]
α_{liq}	liquid fuel volume fraction	[-]	E	total energy	[J/kg]
α_{nuc}	nuclei content	[-]	F_{vap}, F_{cond}	empirical constants	$[m^{-1}]$
α_{vap}	vapor fuel volume fraction	[-]	p	pressure field	[Pa]
\mathbf{v}	velocity field	[m/s]	R	gas constant	[J/kg/K]
λ_g	Taylor length scale	[m]	R_b	bubble radius	[m]
μ	viscosity	[Pa s]	R_e, R_c	evaporation/condensation rate	$[kg/m^3/s]$
μ_t	turbulent viscosity	[Pa s]	Re	Reynolds number	[-]
ρ	density	$[kg/m^3]$	T	temperature	[K]
ρ_{vap}, ρ_{air}	vapour/air density	$[kg/m^3]$	y^+	non-dimensional wall distance	[-]
σ	viscous stress tensor	[Pa]			
τ_t	turbulent stresses	[Pa]			

16 and formation of cavitation inside the injection nozzle. In addition, modern
 17 diesel engines are operated under high injection pressure ($> 2500bar$) and
 18 utilise injectors with small injection hole diameters ($90 - 120\mu m$); these con-
 19 ditions pose significant difficulties in measuring and/or optically visualising
 20 the processes occurring in both the injector nozzle and within the high tem-
 21 perature combustion chamber. The majority of transparent real-size nozzle
 22 investigations featuring simplified single-hole geometries that generally con-
 23 firm the presence of geometric-induced cavitation [10, 11, 12]. The work
 24 of [13, 14, 15], and the relevant early modelling work [16] were the first to

25 substitute one of the holes of a production nozzle with a quartz window of
 26 identical geometric characteristics and was an experimental breakthrough
 27 that provided valuable information on flow and cavitation structures inside
 28 such micro-channels under realistic operating conditions; further studies were
 29 reported in [17]. A step forward was realised in [18], where a 3-hole, real-
 30 size, fully transparent nozzle allowed for unobstructed optical access inside
 31 the sac volume. Vortex cavitation is dramatically enhanced by vapour or air
 32 already present inside the nozzle volume [19]. Moreover, [20] showed that
 33 the structure of a vortex core is significantly affected by entrained vapour
 34 bubbles. Similarly, [21] demonstrated possible fragmentation of the vortex
 35 core so as to increase the vorticity at the core centre. Finally, the strong in-
 36 teraction observed between vortex properties and bubble dynamics[22], the
 37 coupling of radial and axial growth of bubbles trapped in vortices [23] and
 38 the interaction between shear (or normal strain) flow and bubble volume
 39 change [24] form a tremendously complex flow field inside an injector noz-
 40 zle, where dynamic changes in the behaviour of vortices and vapour bubbles
 41 strongly affect the emerging fuel spray. Highly transient flow phenomena
 42 caused by the fast needle response times, give rise to formation of vortical
 43 structures and therefore, to string cavitation [25]. Transient effects have also
 44 been correlated to increased probability of surface erosion damage, which
 45 is attributed to both, geometric and string cavitation [26]. Cavitation in
 46 simplified nozzle replicas has been visualized even at pressures as high as
 47 2000bar, as shown in [27, 28]. Remarkably, in very recent studies, sonolu-

48 minescence from cavitation collapse observed in a simplified nozzle replica
 49 has been observed for the first time[29] and a neutron imaging technique has
 50 been developed overcoming the disadvantages of using materials transparent
 51 to visible light[30]. All aforementioned studies report data from one or just
 52 a few injection events. The group of the authors has reported in [31, 32, 33]
 53 for the first time averaged images of cavitation developing in a real-size 6-
 54 hole transparent tip nozzle for single and pilot-main split injections up to
 55 400bar. Data from these investigations are further reported here and utilized
 56 for validation of the newly developed model. Only the very recent work of
 57 [34] has extended the range of operating conditions (injection pressures up
 58 to 1000bar and back pressures up to 30bar) and geometrical features stud-
 59 ied (hydro erosively ground inlet orifice) for long injections. These studies
 60 provide qualitative data on cavitation and air-entrainment inside the fuel
 61 injector during the opening and closing of the injector’s needle valve. A
 62 drawback of the images is that one cannot distinguish between cavitation
 63 and air, as they both appear as an indistinguishable black shadow in the
 64 obtained images.

65 Given the limited quantitative information around the flow structure in-
 66 side diesel injectors, fuel injection equipment manufacturers require robust
 67 predictive Computational Fluid Dynamics (CFD) tools, in order to under-
 68 stand the physical mechanisms taking place during injection. From a physical
 69 viewpoint, modelling of such flow conditions requires the fluid compressibil-
 70 ity [35], mass transfer (cavitation, flash boiling, evaporation etc.) and heat

transfer [36, 37, 38] to be taken into account, which increase the complexity as well as the computational cost of the simulations. Additionally, the fluid dynamics processes occur at high Reynolds number and therefore accounting for the effect of turbulence structures and vortex dynamics, is key in explaining how the injected fuel spray is formed [39, 40, 41, 42]; this can only be resolved using very fine computational grids and scale resolving simulations, such as Large Eddy Simulation (LES).

Recent LES including dynamic needle movement for the in-nozzle flow includes the work of Battistoni et al. [43] who simulated the start and end of injection for a single hole nozzle using the cut cell cartesian method for modelling the boundary movement and a homogeneous relaxation model for cavitation phenomena. The work concludes that URANS predictions for the residual liquid back flow occur without fragmentation, while in LES liquid breaks up generating complex three dimensional structures. The URANS approach predicted at the end of the injection an annular void region stemming from the needle seat, which then re-condenses as the pressure is recovered. This was not observed in LES, where regions of low pressure are produced even in areas detached from the needle seat. The predicted near spray region was also different as no ligaments were formed in URANS; instead diffusion disperses the liquid in the surrounding air even if integral values like sac pressure and liquid volume fraction were not greatly affected. Ligament formation and gas ingestion into the nozzle at the end of injection are predicted, as observed experimentally in Phase Contrast X-ray images (for additional

94 Phase Contrast X-ray studies see for example [9, 44]). The start-of-injection
 95 simulation shows how gas is ejected first, and liquid fuel starts being injected
 96 with a delay. The main result of these analyses is that if the sac volume is
 97 initially filled with gas, the liquid exit is delayed several tens of μs after the
 98 start of needle movement, which is in good agreement with the experimen-
 99 tal evidence. This delay is of the order of $100\mu s$, and it is compatible with
 100 the duration of the first slow rising part of the needle movement. Orley et
 101 al. [45] used the cut cell cartesian method to simulate with implicit LES, a
 102 barotropic homogeneous equilibrium model for cavitation and a fully com-
 103 pressible 3-phase flow model a complete 9-hole diesel injector. The focus of
 104 the work was on the vortical development of the flow and the assessment
 105 of erosion sensitive areas during the operation of the injector. After the in-
 106 jector closing, strong collapse events of vapor structures in the needle seat
 107 and the sac hole cause the formation of violent shock waves. The authors
 108 highlighted that a fully compressible description of the flow is essential to
 109 capture such phenomena. It was also concluded that despite steady needle
 110 simulations capturing the main flow features reasonably well, vapor creation
 111 during the closing phase of the needle valve requires information on the pre-
 112 viously developed flow; thus, reliable prediction of erosion-sensitive areas
 113 due to collapse events during and after the closing of the needle can only
 114 be predicted accurately by including the unsteady needle motion. Finally,
 115 the work of Koukouvinis et al. [35] used a 2-phase dynamic needle approach
 116 based on a combination of layering and stretching algorithms together with

117 a Rayleigh-Plesset based cavitation model with increased mass transfer, to
118 compute the opening phase of two different injector designs; the findings
119 have correlated the pressure peaks in the domain with areas that suffer from
120 erosion. Whichever the chosen modelling approach may have been, previous
121 studies have lacked validation [45], had indirect validation [35] or were not of
122 direct relevance to modern applications [43], since a single hole nozzle lacks
123 the complex sac recirculation flow present in modern diesel injectors.

124 On the broader perspective, reduction of exhaust gas and in the same
125 time noise emissions from engines, relies on multiple injection strategies,
126 such as digital rate shaping (DRS) [46, 47, 48, 49], which allow the use of a
127 variety of options for pilot, main, and post-(main) injection events in order
128 to provide a degree of control over the timing and phasing of the ignition
129 delay and heat release events, as reported in [50]. Recent investigations from
130 the group of the authors suggest that when the dwell-period is shortened,
131 there is significant reduction in soot while exhaust-out NO_x is controlled by
132 EGR. Similarly, the CN-soot trade-off can be decoupled by reducing pilot-
133 main dwell time, adding a greater number of pilots and increasing rail pres-
134 sure without compromising fuel consumption [51]. The use of such complex
135 strategies described relies on the ability of the fuel injection equipment to
136 accurately meter extremely small quantities of fuel per event (which may be
137 of the order of $1mg$ of fuel being injected in a period of less than $0.25ms$)
138 over the engine lifetime [49]. During these short metering events the injector
139 will not reach full lift and will be operating within the transient part of the

140 rate curve. To meet these demands, it is extremely important to avoid the
 141 accumulation of excessive carbonaceous deposits on, and within the fuel in-
 142 jector. Nozzle hole deposits can reduce the effective flow area of the fuel or
 143 cause it to be mis-directed. These effects give rise to poorer atomisation and
 144 mixing, excessive spray penetration, and increased risk of fuel impacting on
 145 the combustion chamber surfaces, with the potential to adversely affect emis-
 146 sions. The impact of deposit formation within nozzle holes and their effect
 147 on engine performance are well summarised in [52], concluding that residual
 148 fuel remaining within the injector nozzle's sac and holes are thought to be
 149 instrumental in the process [9]. With increasing number of pilot injections
 150 with short dwell time, the residual fuel in the nozzle sac after needle closure
 151 can be critical for the HC and soot emissions. However, experimentation of
 152 the detailed flow dynamics inside the injector at such conditions is practi-
 153 cally impossible; currently there is no study reporting quantitative data on
 154 the flow development during the injection events for such processes. The
 155 experimental data reported in [31, 32, 33, 34] clearly indicate that the flow
 156 and cavitation development inside the injector is different in every injection
 157 cycle, and differ significantly from the experimentally derived time-averaged
 158 field, as shown in [32]. An alternative to shed light to those processes, is the
 159 use of computational fluid dynamics. The current work, to the best of the
 160 authors knowledge, presents for the first time application of a 3-phase LES
 161 to the flow in a diesel injector for a pilot injection event, including cavitation
 162 and compressibility of all phases; simulations have utilised the optically mea-

163 sured needle valve movement from fully transparent real size 6-hole nozzle
164 tips [31, 32, 33], as reported by the group of the authors. Moreover, the
165 high-speed shadowgraph images from those studies serve as validation of the
166 developed model; these include the location/timing of cavitation initiation,
167 its further extent and eventual collapse and the air entering into the injection
168 holes and sac volume of the nozzle tip.

169 The need to employ LES derives from the necessity to predict the flow
170 formation of individual injection cycles, as opposed to cycle-averaged flow
171 distribution. The complexity of the flow is not only linked to the formation
172 of cavitation, but also to the residual air present inside the injector; this has
173 been considered in the present work by initialising the residual air distri-
174 bution inside the injector’s sac volume and injection holes from the images
175 recorded for individual injection events. Moreover, inclusion of compress-
176 ibility effects for all phases is deemed as necessary for resolving the complex
177 liquid, cavitation formation and development and air compression/expansion
178 inside the injector, as it is shown in the reported results.

179 The present paper is structured in the following way: first an overview
180 on the experimental results is given for a diesel pilot injection visualization
181 of a transparent nozzle tip. Then the numerical methodology employed is
182 described in detail, followed by the comparison of the CFD results with the
183 transparent nozzle visualisations for which good agreement is obtained and
184 interpretation of the observed phenomena is provided.

185 2. Experimentally observed multiphase phenomena

186 As already mentioned, the 3-phase simulation methodology has been val-
187 idated against high speed visualisations of a transparent Delphi Technologies
188 Diesel 6-hole nozzle tip manufactured by City, University of London. The
189 metallic injector nozzle tip was substituted with a transparent acrylic tip.
190 The design is a standard serial production geometry, i.e not just a multi-hole
191 nozzle, but a fully operational, serial production type. The detailed results
192 and findings of that experimental campaign as well as the setup details were
193 reported in [31, 32, 33] and will not be repeated here. The 6-hole transparent
194 tip has holes with no taper (zero conicity) and a nominal diameter (D) of
195 $160\mu m$. The electrical pulse activation width for a pilot injection was $0.5ms$.
196 High speed cameras recorded the events at a frame rate of $30000fps$. An
197 example of a pilot injection for a rail pressure of 300bar into atmospheric
198 conditions can be found in Figure 1. Given the image acquisition rate, the
199 pilot injection including all major events after closing lasts for 24 frames.
200 As discussed in [32, 33] air trapped in the sac after the end of the injection
201 aggregates forming bubbles in the sac and occupying part of the hole. Prior
202 to $233.33\mu s$ after the electrical trigger, no change is observed and therefore
203 images are not shown. Then the trapped bubble shows slight expansion due
204 to the initial volume created by the needle as it starts lifting ($300\mu s$ after
205 the trigger) and subsequent compression ($400\mu s$ after the trigger) highlight-
206 ing the need to model air compressibility. This is followed by void coming
207 from the seat passage and its advection into the hole ($500\mu s$ after the trigger).

208 Then, due to flow acceleration at the hole entrance, void structures are seen in
 209 the hole during the opening phase ($600\mu s$ after the trigger). During the nee-
 210 dle closing phase, vapour increases substantially in the hole and void coming
 211 from the seat reappears ($633.33\mu s$ after trigger). At the end of the injection,
 212 the sac gets full with bubbles and the spray greatly weakens ($733.33\mu s$ after
 213 the trigger), followed by what seems to be air suction ($766.66\mu s$ after the
 214 trigger). Finally, a bubbly mixture is observed floating in the sac as well as
 215 an oscillatory movement of the air in the hole ($1000\mu s$ after the trigger). An
 216 important input for nozzle flow moving needle simulations is the needle lift
 217 profile which was extracted from the images [31, 32, 33].

218 **3. Modelling approach**

219 The simulations are computed using the commercial CFD code ANSYS
 220 Fluent [53]. The nozzle flow is solved using a homogeneous, three-phase
 221 mixture model (liquid fuel, vapour fuel and air) where all phases share the
 222 same velocity, pressure and temperature. The code is supplemented with
 223 user defined functions (UDFs) for implementation of the thermo-hydraulic
 224 properties of diesel and the needle movement.

225 *3.1. Multiphase model*

226 The properties appearing in the transport equations are determined by
 227 the presence of the component phases in each control volume. Defining α_{liq} ,
 228 α_{vap} , α_{air} as the volume fraction of liquid fuel, air and vapour fuel in a cell,

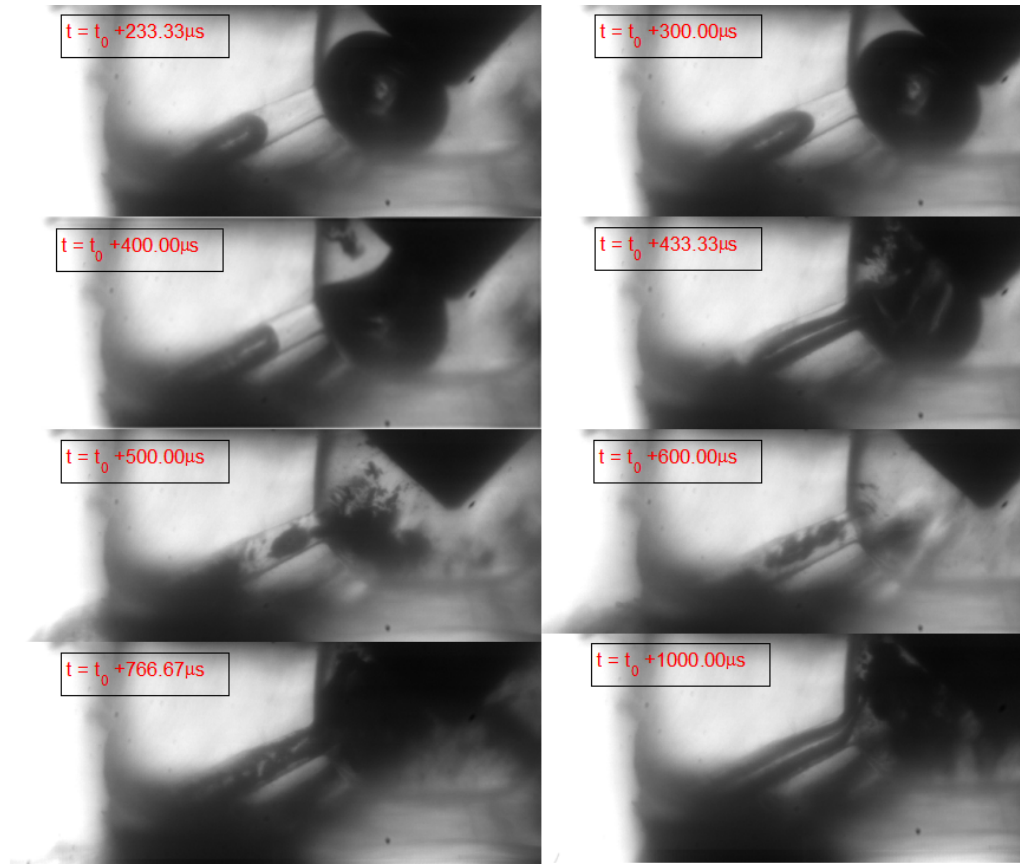


Figure 1: Experimental results. Time sequence of a pilot injection transparent nozzle tip visualisation.

229 respectively, the density in each cell is given by: $\rho = \alpha_{liq}\rho_{liq} + \alpha_{vap}\rho_{vap} +$
 230 $\alpha_{air}\rho_{air}$.

231 All other transport properties (viscosity and thermal conductivity) are
 232 computed in this manner despite the fact that for homogeneous mixtures it
 233 is not clear how one should average each phase's effect, whether based on
 234 mass, volume or area (which would require knowledge of interfacial surface-
 235 area density). Although in the case of bubbly flows some theoretical deriva-
 236 tions attributed to Einstein do exist [54], viscosity in general depends non
 237 linearly on the void fraction and in order to achieve accurate pressure drop
 238 calculations the mixture viscosity has to be empirically corrected by fitting
 239 coefficients to match experimental data sets [55]. For a review on the avail-
 240 able correlations for the transport properties the interested reader is referred
 241 to [56]. Obviously, the volume constraint $\alpha_{liq} + \alpha_{air} + \alpha_{vap} = 1$, in each cell
 242 must be respected. The solved equations consist of the continuity, momen-
 243 tum and energy of the mixture, and the mass conservation equations for the
 244 vapor and the air:

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

$$\frac{\partial \rho \mathbf{v}}{\partial t} + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) = -\nabla p + \nabla \cdot \sigma \quad (2)$$

$$\frac{\partial \rho E}{\partial t} + \nabla \cdot (\mathbf{v}(\rho E + p)) = \nabla \cdot (k_{eff} \nabla T) + \sigma \cdot \mathbf{v} \quad (3)$$

$$\frac{\partial \alpha_{vap} \rho_{vap}}{\partial t} + \nabla \cdot (\alpha_{vap} \rho_{vap} \mathbf{v}) = R_e - R_c \quad (4)$$

$$\frac{\partial \alpha_{air} \rho_{air}}{\partial t} + \nabla \cdot (\alpha_{air} \rho_{air} \mathbf{v}) = 0 \quad (5)$$

245 The source terms R_e and R_c represent the mass transfer between liquid
 246 and vapour phase due to cavitation. The effective viscous stress tensor is
 247 defined as $\sigma = \tau + \tau_t = \mu(\nabla v + (\nabla v)^T) + \tau_t$,

248 where μ is the viscosity of the mixture and τ_t are the turbulent stresses
 249 defined per the turbulence model being used. The energy is computed as the
 250 mass average for each phase and the internal energy of each phase is based
 251 on the local thermodynamic conditions of that phase [37].

252 The source terms appearing in the vapour volume fraction transport equa-
 253 tion ($R_e - R_c$) represent the mass transfer between fuel liquid and vapour
 254 phases due to cavitation bubble expansion and collapse respectively. The
 255 calculation of these values is based on the Rayleigh-Plesset equation describ-
 256 ing bubble expansion and collapse [57], and its magnitude is based on the
 257 Zwart-Gerber-Belamri cavitation model [58] which reads as:

$$R_e = F_{vap} \frac{(3\alpha_{nuc}(1 - \alpha_{vap})\rho_{vap})}{R_b} \sqrt{\frac{2 \max((p_{vap} - p), 0)}{3 \rho_{liq}}} \quad (6)$$

$$R_c = F_{cond} \frac{(3\alpha_{vap}\rho_{vap})}{R_b} \sqrt{\frac{2 \max((p - p_{vap}), 0)}{3 \rho_{liq}}} \quad (7)$$

258 F_{vap} and F_{cond} are empirical calibration coefficients, α_{nuc} is the volume
 259 fraction associated with the nuclei contained in the liquid and R_b the assumed
 260 bubble radius and p_{vap} is the vapour pressure. According to [58], values of
 261 $R_b = 10^{-6}m$, $\alpha_{nuc} = 5 \times 10^{-4}$, $F_{vap} = 50$, $F_{cond} = 0.01$ give reasonable re-
 262 sults in a wide range of flows. Nevertheless, as discussed in [59] the mass
 263 transfer magnitude for these values could be insufficient creating areas of
 264 unrealistic liquid tension and not reproducing correctly the Rayleigh-Plesset
 265 bubble collapse, the suggested solution is to increase the empirical calibra-
 266 tion coefficients several orders of magnitude to approximate the model to
 267 a Homogeneous Equilibrium Model (HEM). However, within this work the
 268 original coefficients published in [58] were used.

269 3.2. Turbulence model

270 The target when using LES is to capture the large scales that are depen-
 271 dent of the physical domain simulated while modelling the sub-grid turbulent
 272 scales. This is achieved by filtering of the Navier-Stokes equations using a
 273 spatial low-pass filter determined by the cell size of the computational domain
 274 used. This operation leaves the flow equations unchanged, but transforms
 275 the equations into equations for the filtered magnitudes [60]. During this
 276 operation terms in the equations appear representing the sub grid scale con-
 277 tributions to the equations of motions and have to be modelled. The closure
 278 of the model requires calculating a suitable sub grid turbulent dissipation
 279 (viscosity) μ_t . For such purpose, the Wall-Adapting Local Eddy-Viscosity

(WALE) model is chosen [61]. This model is capable of correctly reproducing the correct turbulence wall behaviour ($\mu_t \sim o(y^3)$) and becomes 0 at $y = 0$, being y the normal distance to the wall. Another advantage is that it returns a zero turbulent viscosity for laminar shear flows which allows the correct treatment of laminar zones in the domain, this is necessary for modelling the start of injection when flow velocities are low.

3.3. Fluid properties

High injection pressures and low lifts cause high injection velocities and transient heating effects making an incompressible approach unjustifiable [36, 37, 35]. Even if for the transparent nozzle tip testing conditions the pressure is lower than engine conditions, the diesel liquid phase is modelled as a compressible liquid based on the measurements made for the calibration oil Normafluid ISO4113. This is the usual fuel for testing and calibrating diesel fuel injection systems in both laboratories and at an industrial level. All diesel properties that follow are taken from [62, 63], where details of the measurement methodology, range of validity, method for fitting the coefficients and their values can be found (see Figure 2 for plots of the density and viscosity values for different pressures and temperatures). These properties were implemented into ANSYS Fluent following the available User-Defined-Real-Gas-Model (UDRGM) functionality as in [37]. As mentioned in the experimental results section, air compressibility effects are observed during the sac filling event and therefore the air density is modelled as an ideal gas

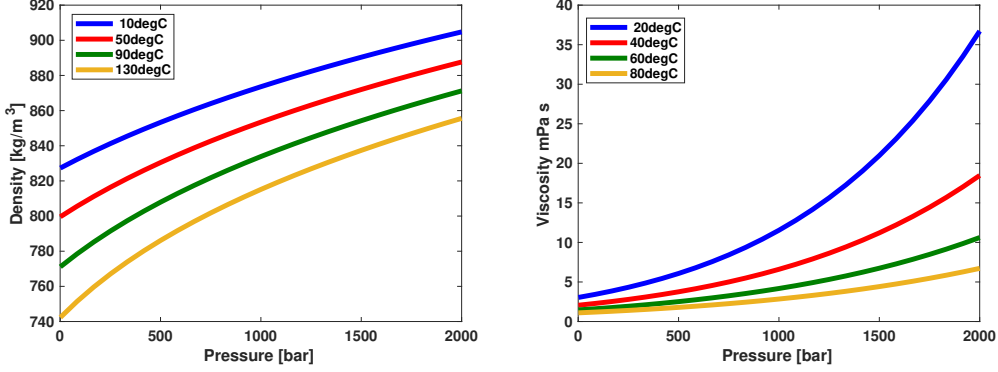


Figure 2: Diesel fuel properties implemented. Density (left) and viscosity (right) diesel fuel properties used.

302 with equation of state $p = \rho RT$.

303 3.4. Moving mesh methodology. Mesh generation and boundary conditions.

304 Modelling the dynamic movement of the needle is inherently difficult. At
 305 low lifts the cells in the seat are squeezed into very small gaps deteriorating
 306 their quality, which can have an impact on the robustness and accuracy of the
 307 simulation. Moreover, the contact between walls is not trivial to model since
 308 the continuity of the mesh is broken. Recent advances have been reported
 309 in [50] where the immersed boundary method has allowed simulations to
 310 be performed even at zero needle lift; however, this method has not been
 311 adopted here and as a compromise, the closed needle is modelled using the
 312 seat surface as a wall when the needle lift is below $1\mu m$.

313 The approach followed is based on an interpolation approach between two
 314 topologically identical meshes (key-grids) with the same number of cells and
 315 was already employed by the authors in [64]. The initial mesh has a $1\mu m$

316 lift and the high lift mesh is based on the maximum lift reached for the pilot
 317 injection $36\mu m$. Based on the node position of this two meshes any interme-
 318 diate lift is achieved by linear interpolation between the node position of the
 319 two key-grids. Another difficulty associated is the loss of resolution in the
 320 seat passage as the needle reaches high lifts, this requires interpolating the
 321 results into another pair of key-grids such as in [37]. For the pilot injection
 322 cases considered here, this was not needed due to the relatively low lift at-
 323 tained ($36\mu m$). Moreover, in order to save computational resources, just a
 324 60° sector is model (one hole) based on the nominal (target) geometry. Figure
 325 3 (left) shows the computational domain, consisting of different surfaces; the
 326 hole, housing, needle, seat inlet and side surfaces. Additionally, a 2mm long
 327 conical discharge volume is added in order to move away the outlet boundary
 328 condition from the areas of interest. The computational mesh used for the
 329 LES flow simulation is a fully hexahedral mesh.

330 The LES settings are adapted from the basis of the previous successful
 331 studies on diesel [39, 40, 41, 42] and gasoline [64, 65] direct injection and
 332 primary breakup simulations. In order to choose the appropriate filter/mesh
 333 size for the LES, the Taylor micro-scales (λ_g) have been estimated. This
 334 length scale is the intermediate length scale at which fluid viscosity signif-
 335 icantly affects the dynamics of turbulent eddies in the flow [66]. For the
 336 flow inside the transparent tip, the Reynolds number based on the nozzle
 337 hole diameter, outlet pressure and inlet temperature can be estimated to be
 338 $Re = \frac{(\rho V D)}{\mu} \sim 13000$. The Taylor micro-scales can then be approximated by

[60]: $\lambda_g = \sqrt{\frac{10D}{Re}} = 4.4\mu m$. However, in order to resolve the smallest eddies
 close to the wall, the non-dimensional wall distance based on the friction
 velocity has to be of the order of 1 ($y^+ \sim 1$) [60]. Therefore, additional
 refinement close to the walls is needed. An estimate of this value based on
 the turbulent boundary layer theory yields a cell wall distance of $\sim 0.2\mu m$
 . In order to reach a value of $\sim 5\mu m$ in the bulk flow without increasing
 excessively the number of cells, a cell growth ratio of 1.1 was applied in the
 wall. Under these constraints, a $\sim 5M$ element mesh was produced, with a
 volume change between neighbouring cells below 3, minimum cell angle of
 27° and $3D$ determinant (normalized triple product of the vectors starting
 from each cell node) above 0.6 for both key-grids. Special care was taken to
 refine the needle seat area in the stream-wise direction in order not to exceed
 for low lifts aspect ratios of 100 in the direction of the bulk flow. Figure 3
 (right) depicts the two meshes needed for the interpolation method, and a
 front view of the mesh showing the additional refinement in the seat area.
 A pressure boundary conditions was applied to the inlet of the domain. The
 pressure at the injector entrance in the high-pressure pipe was taken from
 the experimentally recorded values for every individual injection event. Dur-
 ing the opening phase, pressure decreases at the injector entrance due to the
 increasing flow through it. At the end of the injection an over pressure is
 observed due to the water hammer effect after needle closing. The pressure
 at the entrance of the injector was provided in [31]. A temperature of $300K$
 was chosen for the flow entering the domain and an air mass fraction value

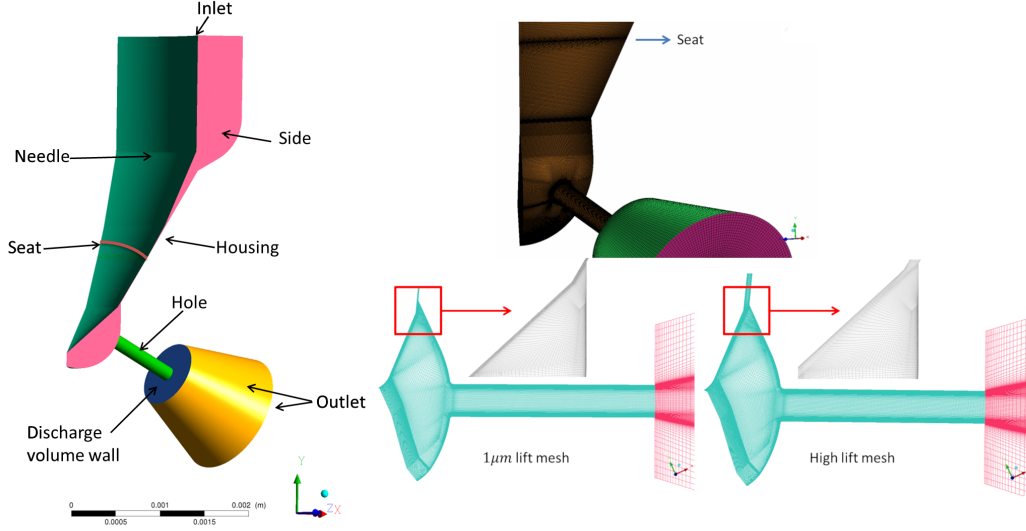


Figure 3: Geometrical model and mesh. Domain simulated and boundary conditions (left). Mesh showing seat refinement (right-top) and mesh cross section for both high and low lift meshes (right-bottom).

362 of 2×10^{-5} was imposed to take into account the possible dissolved air since
 363 it is a typical value for fuel or water exposed to ambient pressure [67]. The
 364 non-slip boundary conditions was applied to the non-moving wall (housing,
 365 hole, discharge volume wall, and, seat surface below $0.1\mu m$) as well as to
 366 the needle according to the motion profile resulting from the needle lift pro-
 367 file extracted from the images [31]. Periodic boundary condition have been
 368 applied to the side surfaces. Finally, a fixed pressure outlet was applied to
 369 the outlet surfaces, with pressure $1bar$ and $300K$ and air volume fraction
 370 prescribed as 1 in the case of back-flow.

371 The experimental images of the transparent nozzle show trapped air bub-
 372 bles inside the injector before the start of injection. The mechanism behind

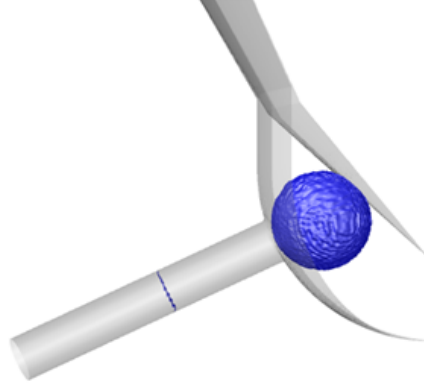


Figure 4: Initial simulation instant. Iso-surface of 0.5 liquid volume fraction and a mid-plane for the initial instant.

the appearance of this bubble is not straight forward to derive from the experimental images. Regardless, the LES nozzle flow simulation is initialised in qualitatively similar way; half of the hole is filled with air and an air spherical bubble is included in the sac (see Figure 4).

The computational domain above the seat surface is initialised at the pressure corresponding to that instant. Below the needle seat, the simulation is initialised at a pressure of $1bar$. All the domain is initialised at a temperature of $300K$ and with zero velocity. For the closing phase the movement of the needle is stopped when it reaches $1\mu m$ however the seat surface is not switched from interior to wall until the needle lift profile reaches $0.1\mu m$.

The solver used is segregated and pressure-based. The pressure-velocity coupling is achieved using the SIMPLEC algorithm [68]. Density is interpolated using a second order upwind scheme [69] while for the momentum a bounded central differencing scheme based on the normalized variable dia-

387 gram (NVD) approach together with the convection boundedness criterion
 388 (CBC) [70] was used. The bounded central differencing scheme is a com-
 389 posite NVD-scheme that consists of a pure central differencing, a blended
 390 scheme of the central differencing and the second-order upwind scheme, and
 391 the first-order upwind scheme. The first-order scheme is used only when the
 392 CBC is violated. This scheme has small numerical dissipation and sufficient
 393 numerical stability for industrial LES simulations [71]. Among the volume
 394 fraction interpolation schemes available in ANSYS Fluent when using the
 395 mixture model, the quadratic upstream interpolation for convective kinetics
 396 (QUICK) scheme is selected in order to reduce the smearing of sharp volume
 397 fraction gradients and capture high density ratios [72]. Pressure interpola-
 398 tion follows the body force weighted scheme [53] and the temperature the
 399 first order upwind scheme. Finally the calculation of gradients was done
 400 using the Least Squares Cell-Based method.

401 The used solver is pressure-based and therefore the simulation stability
 402 is not limited by the acoustic wave propagation time scale. However, tem-
 403 poral resolution for LES requires minimum diffusion for the advection of the
 404 turbulent eddies. Therefore, an adaptive time step method is employed to en-
 405 sure the advection CFL number stays below 1 throughout the computational
 406 domain.

407 3.5. LES mesh quality evaluation

408 The instantaneous fields of the LES quality metric of by Celik et al.[73]
 409 and y^+ for a representative moment at the highest lift ($t = 0.608ms$) are
 410 shown in Figure 5. Based on the y^+ the boundary layer resolution can be
 411 assessed; this value only exceeded 1 in areas above the seat and gradually
 412 transitions to values well under 1 ensuring a good wall shear resolution for
 413 the small eddies near the walls. Following [60] a good LES requires the
 414 modelled turbulent kinetic energy (k_{sgs}) to be less than 20 of the total tur-
 415 bulent energy ($k_{sgs} + k_{res}$), that is $\frac{k_{sgs}}{k_{sgs} + k_{res}} < 0.2$. However, as mentioned
 416 in [43] knowledge of k_{res} in the case of a moving needle injection can only
 417 be gained by repeating the simulation multiple times which could not be af-
 418 farded computationally. Although they are point indicative measures which
 419 are not particularly accurate for anisotropic turbulence, another option is to
 420 use metrics based on the turbulence resolution length scale such as the LSR
 421 metric; see for example [74] and its application by Battistoni et al. [43] to a
 422 moving needle injection, or the similar metric by Celik et al.[73]:

$$LESIQ_\nu = \frac{1}{1 + 0.05(\frac{\mu + \mu_t}{\mu})^{0.53}} \quad (8)$$

423 where μ_t is the sub-grid scale viscosity introduced by the WALE model.
 424 This is a number between 0 and 1 for which the constants are calibrated such
 425 that the index is perceived similar to the ratio of resolved to total turbulent
 426 kinetic energy i.e. the higher the value the better the resolution is (0.8

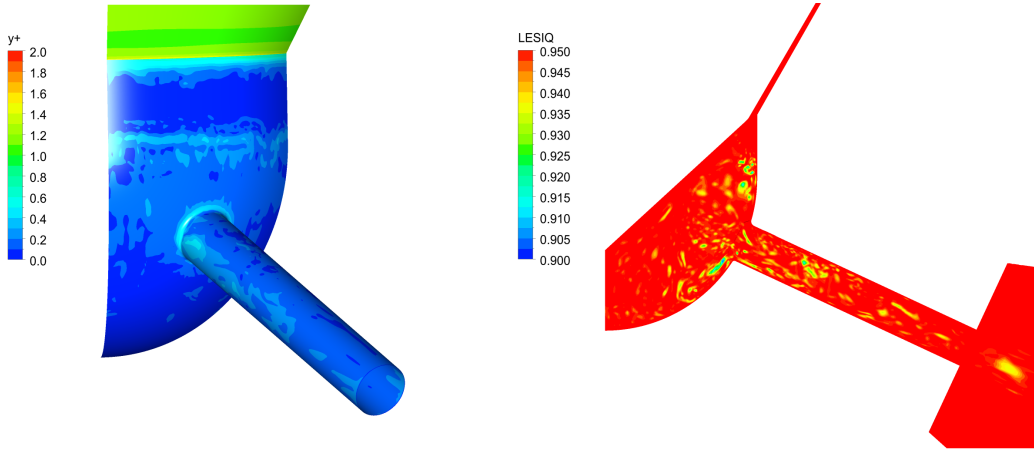


Figure 5: Mesh resolution evaluation. y^+ contours on the nozzle wall (left) and the LES quality metric of [73] (right) for highest needle lift during the pilot injection.

or above). Although [73] suggests to include as well the artificial viscosity introduced by the numerical methods, it is beyond the scope of this work to estimate such contribution. As seen in Figure 5 the value of $LESIQ_\nu$ for the same representative time instant is mostly over 0.95 throughout the domain and having a minimum values of 0.9 in the separation region that occurs at the entrance of the sac, confirming the suitability of the mesh.

4. Results and discussion

The evolution of the volume fraction inside the nozzle for the different phases is shown in Figure 6. Additionally, the imposed needle lift extracted from the image sequence shown in Figure 1 is shown as well. The simulation is started at the physical time $0.4874ms$ coincident with a lift of $1\mu m$ for the imposed profile. During the opening phase it follows from this plot that initially there is air present inside the nozzle. This air is evacuated out of

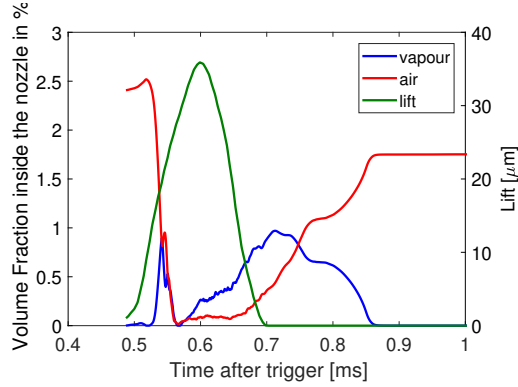


Figure 6: Integral results. Volume of vapour and air inside the nozzle and needle lift against time.

the nozzle while cavitation is generated showing a peak between $0.5ms$ and $0.6ms$, while it decreases afterwards. As the injection transitions towards the closing phase the amount of vapour increases, showing a peak just after the needle closes, while the amount of air continually increases by a process of air suction as it will be shown in the following section.

A comparison between the transparent nozzle tip images and the simulation results at the start of the injection is shown in Figure 7. In particular, a snapshot of the predicted liquid volume iso-surface of 50 at $t = 0.532ms$ is shown. At the early stages of the injection the simulation reproduces the compression of the air bubble inside the sac volume. The compression is caused by the pressure build up in the sac, justifying the inclusion of the compressibility of the air. This is quickly followed by cavitation originating at the needle seat passage, due to flow separation and shear in this area.

Sample simulation results and the transparent nozzle tip images for the needle opening phase are shown in Figure 8. The CFD results indicate that

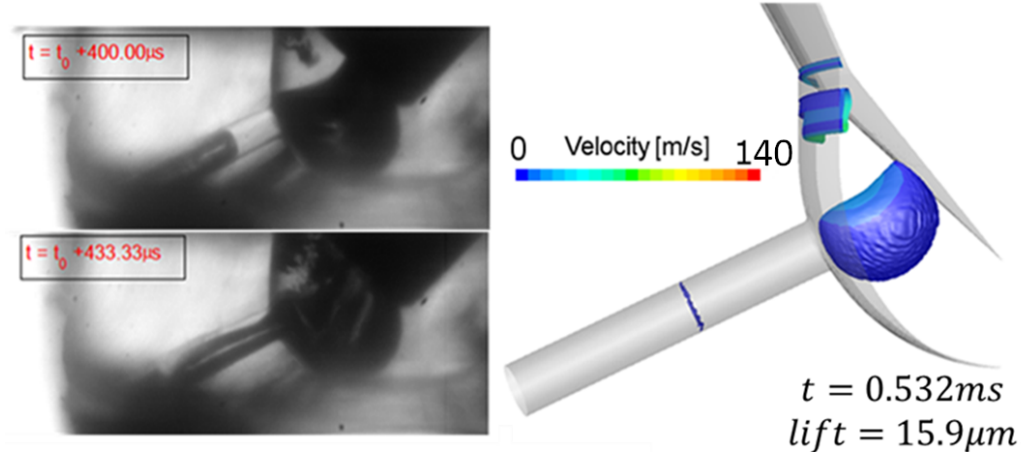


Figure 7: Start of injection results. Experimental visualisations (left), 50% liquid volume fraction iso-surface coloured by velocity magnitude (right).

455 cavitation produced at the sac entrance is transported directly into the in-
 456 jection hole. Simultaneously, the air bubble is further compressed and is
 457 pushed to recirculate parallel to the needle in the direction of the needle mo-
 458 tion. Similarly to the experimental images, the air bubble is seen breaking
 459 down and mixing with any remaining cavitation into a fine bubbly mixture
 460 which is then advected into the hole.

461 As the needle lift increases and the flow further develops, the simulation
 462 indicates that air disappears from the sac volume, as seen in Figure 9. This
 463 is attributed to a combination of two effects. Firstly, the sac pressure build
 464 up causes the air to be compressed, reducing its volume fraction. Secondly,
 465 as the air is trapped within the recirculation zone developing inside the sac
 466 volume, it enters into the injection hole, where it expands due to the local
 467 pressure drop at its entrance. This contributes to the void areas observed

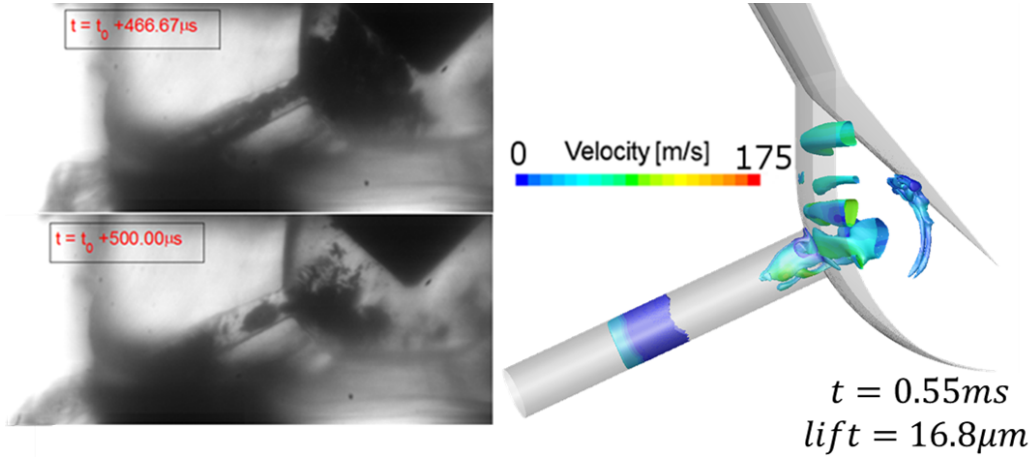


Figure 8: Needle opening phase results. Experimental visualisations (left), 50% liquid volume fraction iso-surface coloured by velocity magnitude (right).

and suggests that the void observed experimentally is a combination of air and fuel vapour. In addition, part of the void visible in the simulation can be attributed to geometrical cavitation developed at the hole inlet upper lip, which can be also seen from the experimental images.

The only two experimental frames available for the needle closing phase together with the simulation results are shown in Figure 10 (top). As the needle valve moves into the closing phase, the amount of void in the hole increases. This is in agreement with the simulation results from Figure 6, where volume content as a percentage of the injector volume of both air and vapour are plotted against time; it follows that these quantities increase during the needle closing phase. This void in the simulation has two sources, one from the unstable vortical flow developing inside the sac volume and entering into the injection hole and another due to formation of geometric-induced cavi-

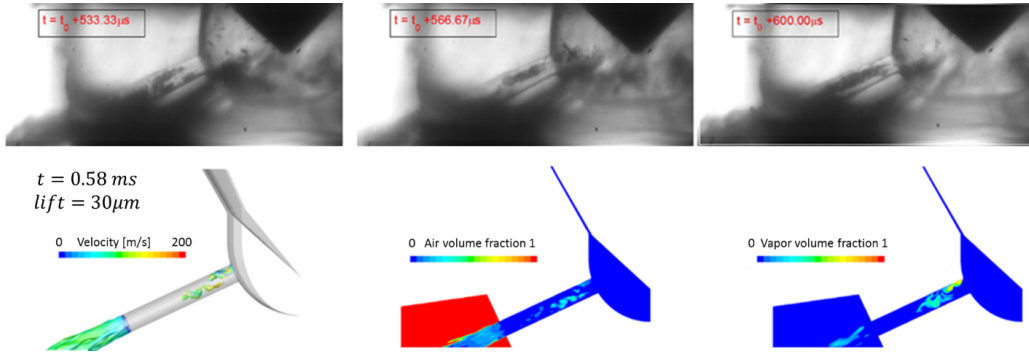


Figure 9: Results as flow further develops during the opening phase. Experimental visualisations for three time instances (top), 50% liquid volume fraction iso-surface coloured by velocity magnitude (bottom-left), air volume fraction contours (bottom-centre) and vapour volume fraction contours (bottom-right).

481 tation at the hole inlet corner. Regarding the experimental results at very
 482 low lifts ($lift = 6\mu m$), a bubbly mixture appears in the sac; bubbles having
 483 sizes similar to the hole diameter appear in the hole. The simulation model
 484 predicts high velocities in the hole; however, since the flow coming from the
 485 seat is throttled a void structure appears in front of the hole. The bubbly
 486 mixture in the sac volume correlates to the void structure created in front of
 487 the hole, which is predicted to be composed of a mixture of fuel vapour and
 488 expanded air. On the other hand, the visualised bubbles computed inside
 489 the injection hole correlate to the big amount of cavitation computed in the
 490 hole.

491 A time sequence of the pressure field is presented in Figure 11. Before
 492 the needle valve closes, the predicted sac volume pressure is still higher than
 493 the ambient pressure ($t = 0.674ms$), but immediately after the needle valve
 494 closing ($t = 0.698ms$), a pressure wave is generated that travels towards the

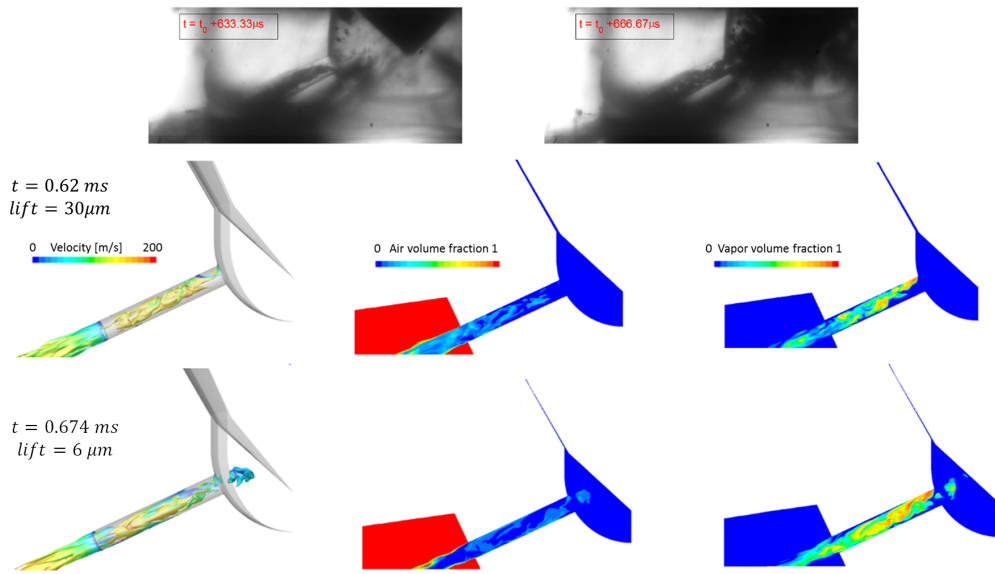


Figure 10: Needle closing results. Experimental visualisations for two time instants (top). Simulation results (center and bottom). For the simulation results 50% liquid volume fraction iso-surface coloured by velocity magnitude (left), air volume fraction contours (center) and vapour volume fraction contours (right) are presented.

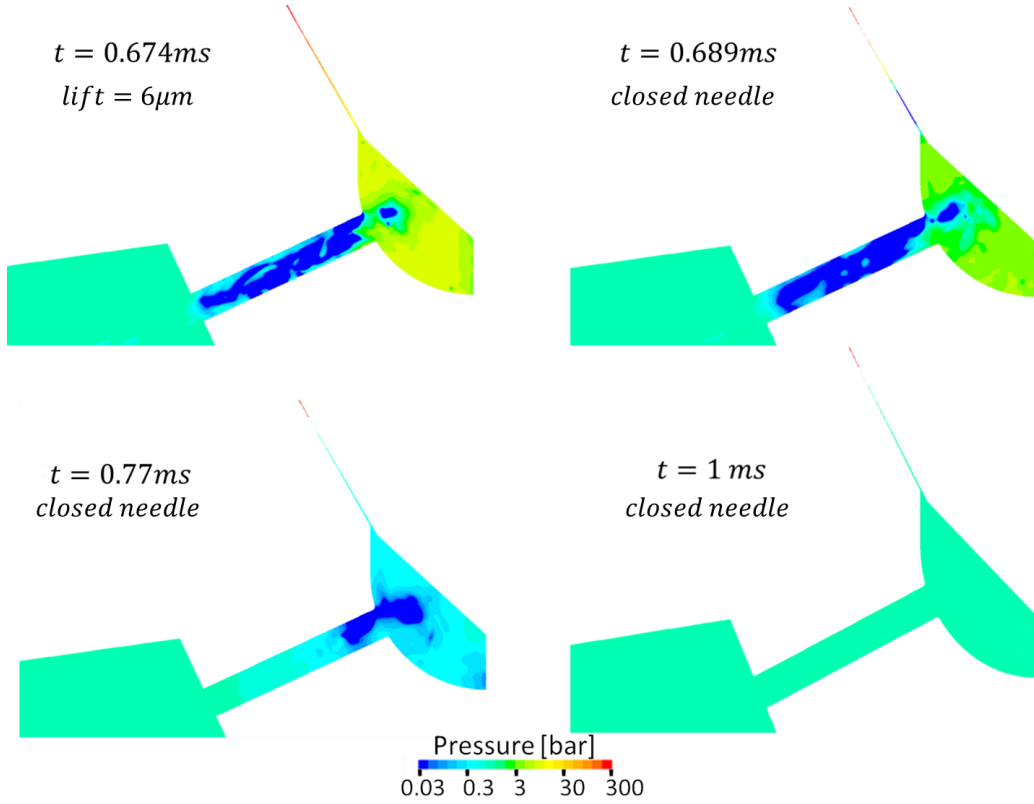


Figure 11: Pressure field time sequence. Notice that logarithmic scale has been used.

495 sac volume; this leaves the sac volume pressure below the ambient pressure
 496 ($t = 0.77ms$). In agreement with Figure 6, where air volume fraction inside
 497 the nozzle is seen to increase after needle closing, this induces the spray to
 498 weaken and air to be sucked back from the ambient into the nozzle until the
 499 sac pressure is balanced with the exterior pressure ($t = 1ms$).

500 Evidence is also provided in Figure 12, which shows a time sequence
 501 of air and vapour volume fraction fields. It clearly depicts the weakening
 502 flow momentum in the injection hole ($t = 0.698ms$) leading to air suction

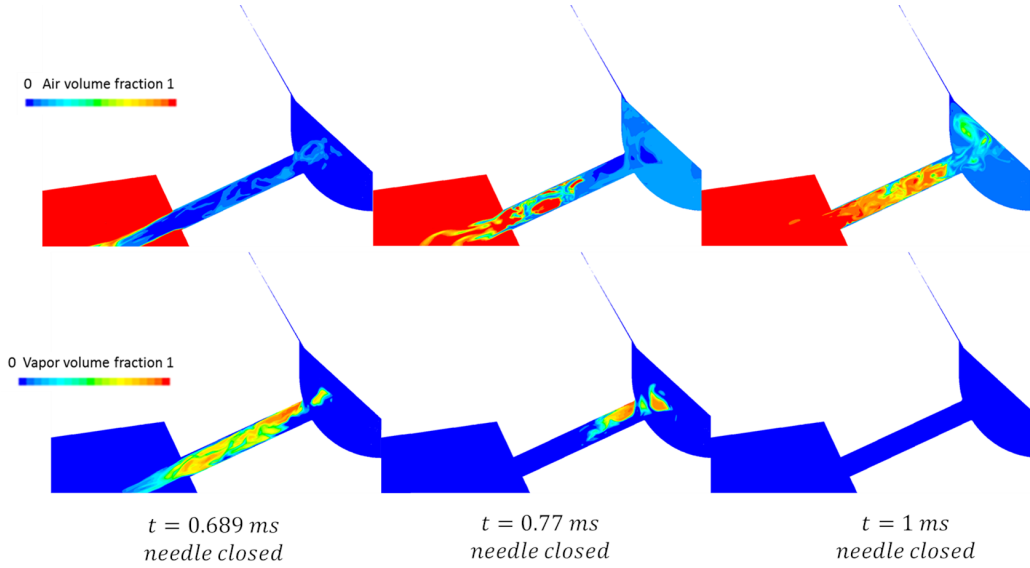


Figure 12: After needle closing results. Time sequence for air (top) and vapour (bottom) volume fraction fields.

503 ($t = 77ms$). Finally, due to the pressure balancing with the ambient pressure,
 504 vapour completely disappears ($t = 1ms$), indicating that shortly after the
 505 needle closing only liquid and air remain inside the sac volume.

506 5. Conclusions

507 This paper presents an investigation of cavitation and air interaction dur-
 508 ing a diesel pilot injection of a standard serial production six-hole geometry.
 509 The focus was to understand the complex interaction between the needle mo-
 510 tion, cavitation formation and development, and gas suction. The strategy
 511 followed has been to use high speed visualisations of a transparent nozzle
 512 tip to record the multiphase phenomena and to use CFD to explain the
 513 physics behind the observations. The CFD methodology includes LES tur-

bulence modelling, the needle valve movement, cavitation effects through a Rayleigh-Plesset based cavitation model, and the compressibility of both air and fuel. Starting from a flow field initialised according to the experimental observations (with an air bubble in the sac and a big portion of the hole filled with air), the main flow features observed are replicated by the simulations. In particular the following phenomena experimentally noticed have been explained and reproduced:

- The compression of the initial air bubble due to sac pressure build up. The inclusion of air compressibility in the simulation can be very relevant even for modest injection pressures in order to replicate the air compression in the sac at the start of the injection as well as the air expansion in the injection hole and sac.
- The appearance of cavitation stemming from the sac entry at the start of the injection, due to flow separation and shear.
- The sac flow recirculation in the sac and flow patterns inside the hole. One part of the void observed in the simulation can be attributed to cavitation both geometrical (developed at the hole inlet upper lip) and vortical (due to complex flow structure coming from the sac). Furthermore, the initial air inside the nozzle expands in the hole contributing to the void areas observed. This shows that the void observed experimentally is a combination of both air and fuel vapour.
- An increase of void inside the hole and in the sac during the needle

536 valve closing. The underlying reason being the flow throttling, since
537 liquid momentum is still high but flow passage very restricted.

538 • The air suction after the needle closing. The closure of the valve creates
539 an expansion wave that leaves the sac pressure below the ambient. This
540 induces vapour creation and air expansion in the sac and consequently
541 air is sucked from the ambient into the nozzle. When the pressure in
542 the sac is recovered, all vapour collapses. Therefore, it is shown that
543 the remaining foam at the end of the injection consists of a liquid and
544 air mixture.

545 **Acknowledgements**

546 The European Union Horizon-2020 Research and Innovation Program
547 funding to Eduardo Gomez Santos (No 675676), the ANSYS HPC research
548 license grant to Delphi Technologies, and the CPU time granted by Gcompute
549 are highly appreciated.

550 **References**

551 [1] European-Comission, “Commission regulation (eu) 2017/1151 of 1 june
552 2017,” July 2017. [https://eur-lex.europa.eu/legal-content/EN/](https://eur-lex.europa.eu/legal-content/EN/TXT/?uri=CELEX:02017R1151-20190101)
553 [TXT/?uri=CELEX:02017R1151-20190101](https://eur-lex.europa.eu/legal-content/EN/TXT/?uri=CELEX:02017R1151-20190101). Last accessed 18th November
554 2019.

- 555 [2] S. Shundoh, M. Komori, K. Tsujimura, and S. Kobayashi, “Nox re-
556 duction from diesel combustion using pilot injection with high pres-
557 sure fuel injection,” in *SAE Technical Paper*, SAE International, 1992.
558 <https://doi.org/10.4271/920461>.
- 559 [3] D. A. Pierpont, D. T. Montgomery, and R. D. Reitz, “Reducing par-
560 ticulate and nox using multiple injections and egr in a d.i. diesel,” in
561 *SAE Technical Paper*, SAE International, 1995. [https://doi.org/10.](https://doi.org/10.4271/950217)
562 [4271/950217](https://doi.org/10.4271/950217).
- 563 [4] H. Afzal, C. Arcoumanis, M. Gavaises, and N. Kampanis, “Internal flow
564 in diesel injector nozzles modelling and experiments,” in *Fuel injection*
565 *systems. Proceedings*, 1999. [https://doi.org/10.1016/j.fuel.2013.](https://doi.org/10.1016/j.fuel.2013.08.060)
566 [08.060](https://doi.org/10.1016/j.fuel.2013.08.060).
- 567 [5] E. Giannadakis, M. Gavaises, and C. Arcoumanis, “Modelling of cavi-
568 tation in diesel injector nozzles,” *Journal of Fluid Mechanics*, vol. 616,
569 pp. 153–193, 2008. <https://doi.org/10.1017/S0022112008003777>.
- 570 [6] I. H. Sezal, S. J. Schmidt, G. H. Schnerr, M. Thalhamer, and M. Förster,
571 “Shock and wave dynamics in cavitating compressible liquid flows in
572 injection nozzles,” *Shock Waves*, vol. 19, pp. 49–58, Apr 2009. [https:](https://doi.org/10.1007/s00193-008-0185-3)
573 [//doi.org/10.1007/s00193-008-0185-3](https://doi.org/10.1007/s00193-008-0185-3).
- 574 [7] F. Salvador, J.-V. Romero, M.-D. Rosello, and J. Martinez-Lopez, “Val-
575 idation of a code for modeling cavitation phenomena in diesel injector

- 576 nozzles,” *Mathematical and Computer Modelling*, vol. 52, no. 7, pp. 1123
577 – 1132, 2010. <https://doi.org/10.1016/j.mcm.2010.02.027>.
- 578 [8] M. Ikemoto, K. Shimode, K. Omae, and N.Toda, “Diesel spray and com-
579 bustion development using nozzle flow visualization , spray and combus-
580 tion analyses,” in *Proceedings of International Congress : SIA Power-
581 train -Rouen*, 2016.
- 582 [9] S. Moon, W. Huang, Z. Li, and J. Wang, “End-of-injection fuel dribble of
583 multi-hole diesel injector: Comprehensive investigation of phenomenon
584 and discussion on control strategy,” *Applied Energy*, vol. 179, pp. 7 – 16,
585 2016. <https://doi.org/10.1016/j.apenergy.2016.06.116>.
- 586 [10] H. Kyu Suh and C. Sik Lee, “Effect of cavitation in nozzle orifice on the
587 diesel fuel atomization characteristics,” *International Journal of Heat
588 and Fluid Flow*, vol. 29, pp. 1001–1009, 08 2008. [https://doi.org/10.
589 1016/j.ijheatfluidflow.2008.03.014](https://doi.org/10.1016/j.ijheatfluidflow.2008.03.014).
- 590 [11] R. Miranda, H. Chaves, U. Martin, and F. Obermeier, “Cavitation
591 in a transparent real size vco injection nozzle,” in *Proceedings of the
592 9th International conference on liquid atomisation and spray systems,
593 ICLASS, Sorrento*, 2003.
- 594 [12] I. Gilles-Birth, M. Rechs, U. Spicher, and S. Bernhardt, “Experimen-
595 tal investigation of the in-nozzle flow of valve covered orifice nozzles for

- 596 gasoline direct injection,” in *Proceedings of the 7th International sym-*
 597 *posium on internal combustion diagnostics, Baden-Baden*, 2006.
- 598 [13] C. Arcoumanis, H. Flora, M. Gavaises, and M. Badami, “Cavitation
 599 in real-size multi-hole diesel injector nozzles,” in *SAE 2000 World*
 600 *Congress*, SAE International, mar 2000. [https://doi.org/10.4271/](https://doi.org/10.4271/2000-01-1249)
 601 [2000-01-1249](https://doi.org/10.4271/2000-01-1249).
- 602 [14] C. Arcoumanis, M. Gavaises, H. Flora, and H. Roth, “Visualisation of
 603 cavitation in diesel engine injectors,” *Mecanique & Industries*, vol. 2,
 604 no. 5, pp. 375 – 381, 2001. [http://www.sciencedirect.com/science/](http://www.sciencedirect.com/science/article/pii/S1296213901011198)
 605 [article/pii/S1296213901011198](http://www.sciencedirect.com/science/article/pii/S1296213901011198).
- 606 [15] H. Roth, E. Giannadakis, M. Gavaises, C. Arcoumanis, K. Omae,
 607 I. Sakata, M. Nakamura, and H. Yanagihara, “Effect of multi-injection
 608 strategy on cavitation development in diesel injector nozzle holes,” in
 609 *SAE 2005 World Congress & Exhibition*, SAE International, apr 2005.
 610 <https://doi.org/10.4271/2005-01-1237>.
- 611 [16] C. Arcoumanis, M. Gavaises, E. Abdul-Wahab, and V. Moser, “Mod-
 612 eling of advanced high-pressure fuel injection systems for passenger car
 613 diesel engines,” in *International Congress & Exposition*, SAE Interna-
 614 tional, mar 1999. <https://doi.org/10.4271/1999-01-0910>.
- 615 [17] N. Mitroglou and M. Gavaises, “Cavitation inside real-size fully trans-
 616 parent fuel injector nozzles and its effect on near-nozzle spray forma-

- tion,” in *DIPSI workshop on droplet impact phenomena and spray investigations*, University of Bergamo, Italy, 2011.
- [18] H. Chaves, R. Miranda, and R. Knake, “Particle image velocimetry measurements of the cavitating flow in a real size transparent vco nozzle,” in *Proceedings of the 22nd European conference on liquid atomization and spray systems, ILASS, Como*, 2008. <https://doi.org/10.1017/jfm.2013.32>.
- [19] A. Andriotis, M. Gavaises, and C. Arcoumanis, “Vortex flow and cavitation in diesel injector nozzles,” *Journal of Fluid Mechanics*, vol. 610, pp. 195–215, 2008. <https://doi.org/10.1017/S0022112008002668>.
- [20] G. Sridhar and J. Katz, “Effect of entrained bubbles on the structure of vortex rings,” *Journal of Fluid Mechanics*, vol. 397, pp. 171–202, 1999. <https://doi.org/10.1017/S0022112099006187>.
- [21] A. J. Cihonski, J. R. Finn, and S. V. Apte, “Volume displacement effects during bubble entrainment in a travelling vortex ring,” *Journal of Fluid Mechanics*, vol. 721, pp. 225–267, 2013.
- [22] J. Choi and S. L. Ceccio, “Dynamics and noise emission of vortex cavitation bubbles,” *Journal of Fluid Mechanics*, vol. 575, pp. 1–26, 2007. <https://doi.org/10.1017/S0022112006003776>.
- [23] J. Choi, C.-T. Hsiao, G. Chahine, and S. Ceccio, “Growth, oscillation and collapse of vortex cavitation bubbles,” *Journal of Fluid Me-*

- 638 *chanics*, vol. 624, pp. 255–279, 2009. [https://doi.org/10.1017/](https://doi.org/10.1017/S0022112008005430)
639 [S0022112008005430](https://doi.org/10.1017/S0022112008005430).
- 640 [24] S. Dabiri, W. A. Sirignano, and D. D. Joseph, “Interaction between a
641 cavitation bubble and shear flow,” *Journal of Fluid Mechanics*, vol. 651,
642 pp. 93–116, 2010. <https://doi.org/10.1017/S0022112009994058>.
- 643 [25] M. Blessing, G. Konig, C. Kruger, U. Michels, and V. Schwarz, “Analysis
644 of flow and cavitation phenomena in diesel injection nozzles and its
645 effects on spray and mixture formation,” in *SAE 2003 World Congress &*
646 *Exhibition*, SAE International, mar 2003. [https://doi.org/10.4271/](https://doi.org/10.4271/2003-01-1358)
647 [2003-01-1358](https://doi.org/10.4271/2003-01-1358).
- 648 [26] M. Gavaises, D. Papoulias, A. Andriotis, E. Giannadakis, and
649 A. Theodorakakos, “Link between cavitation development and erosion
650 damage in diesel injector nozzles,” in *SAE World Congress & Ex-*
651 *hibition*, SAE International, apr 2007. [https://doi.org/10.4271/](https://doi.org/10.4271/2007-01-0246)
652 [2007-01-0246](https://doi.org/10.4271/2007-01-0246).
- 653 [27] B. Reid, G. Hargrave, C. P. Garner, and G. Wigley, “An investigation
654 of string cavitation in a true-scale fuel injector flow geometry at high
655 pressure,” *Physics of Fluids*, vol. 22, 03 2010. [https://doi.org/10.](https://doi.org/10.1063/1.3372174)
656 [1063/1.3372174](https://doi.org/10.1063/1.3372174).
- 657 [28] M. Gavaises, B. Reid, N. Mitroglou, G. Hargrave, C. Garner, E. Long,
658 and R. McDavid, “On the formation of string cavitation inside fuel in-

- jectors,” *Experiments in Fluids*, vol. 55, 01 2014. <https://doi.org/10.1007/s00348-013-1662-8>.
- [29] R. Lockett and A. Bonifacio, “Hydrodynamic luminescence in a model diesel injector return valve,” *International Journal of Engine Research*, 07 2019. <https://doi.org/10.1177/1468087419870421>.
- [30] L. Thimm, P. Trtik, H. Hansen, S. Jollet, and F. Dinkelacker, “Experimental cavitation and spray measurement in real size diesel injection nozzles with high resolution neutron imaging,” in *Proceedings of the 29th Conference on Liquid Atomization and Spray Systems, ILASS Europe, Paris*, 2019.
- [31] M. McLorn, *Fundamental Behaviour of Valves Used in Diesel Fuel Injection Equipment*. PhD thesis, (Unpublished), City, University of London, 2013.
- [32] N. Mitroglou, M. McLorn, M. Gavaises, C. Soteriou, and M. Winterbourne, “Instantaneous and ensemble average cavitation structures in diesel micro-channel flow orifices,” *Fuel*, vol. 116, pp. 736 – 742, 2014. <https://doi.org/10.1016/j.fuel.2013.08.060>.
- [33] M. Winterbourn, C. Soteriou, N. Mitroglou, M. Gavaises, and C. Dav-eau, “Visualising injection events in a fully operational diesel injector with a multi-hole transparent tip,” in *Thiesel, Valencia*, 2014.

- 679 [34] R. P. Fitzgerald, G. D. Vecchia, J. E. Peraza, and G. C. Martin, “Fea-
680 tures of internal flow and spray for a multi-hole transparent diesel fuel
681 injector tip,” in *29th Conference on Liquid Atomization and Spray Sys-*
682 *tems, ILASS, Paris*, 2019.
- 683 [35] P. Koukouvinis, M. Gavaises, J. Li, and L. Wang, “Large eddy simulation
684 of diesel injector including cavitation effects and correlation to erosion
685 damage,” *Fuel*, vol. 175, pp. 26 – 39, 2016. [https://doi.org/10.1016/](https://doi.org/10.1016/j.fuel.2016.02.037)
686 [j.fuel.2016.02.037](https://doi.org/10.1016/j.fuel.2016.02.037).
- 687 [36] A. Theodorakakos, G. Strotos, N. Mitroglou, C. Atkin, and M. Gavaises,
688 “Friction-induced heating in nozzle hole micro-channels under extreme
689 fuel pressurisation,” *Fuel*, vol. 123, pp. 143 – 150, 2014. [https://doi.](https://doi.org/10.1016/j.fuel.2014.01.050)
690 [org/10.1016/j.fuel.2014.01.050](https://doi.org/10.1016/j.fuel.2014.01.050).
- 691 [37] G. Strotos, P. Koukouvinis, A. Theodorakakos, M. Gavaises, and
692 G. Bergeles, “Transient heating effects in high pressure diesel injector
693 nozzles,” *International Journal of Heat and Fluid Flow*, vol. 51, pp. 257
694 – 267, 2015. [https://doi.org/10.1016/j.ijheatfluidflow.2014.](https://doi.org/10.1016/j.ijheatfluidflow.2014.10.010)
695 [10.010](https://doi.org/10.1016/j.ijheatfluidflow.2014.10.010).
- 696 [38] J. Shi, N. Guerrassi, G. Dober, K. Karimi, and Y. Meslem, “Complex
697 physics modelling of diesel injector nozzle flow and spray supported by
698 new experiments,” in *Thiesel, Valencia*, 2014.
- 699 [39] J. Shi, P. Aguado Lopez, G. Dober, N. Guerrassi, W. Bauer, and M. Lai,

- 700 “Using les and x-ray imaging to understand the influence of injection
701 hole geometry on diesel spray formation,” in *Thiesel, Valencia*, 2016.
- 702 [40] J. Shi, P. Aguado Lopez, E. Gomez Santos, N. Guerrassi, G. Dober,
703 W. Bauer, M. Lai, and J. Wang, “Evidence of vortex driven primary
704 breakup in high pressure fuel injection,” in *Proceedings of the 28th Con-*
705 *ference on Liquid Atomization and Spray Systems, ILASS Europe, Va-*
706 *lencia*, 2017. <http://dx.doi.org/10.4995/ILASS2017.2017.5707>.
- 707 [41] J. Shi, P. Aguado Lopez, N. Guerrassi, and G. Dober, “Understanding
708 high-pressure injection primary breakup by using large eddy simulation
709 and x-ray spray imaging,” *MTZ worldwide*, vol. 78, pp. 50–57, 05 2017.
710 <https://doi.org/10.1007/s38313-017-0039-4>.
- 711 [42] J. Shi, P. Aguado Lopez, E. Gomez Santos, N. Guerrasi, W. Bauer,
712 M.-C. Lai, and J. Wang, “High pressure diesel spray development: the
713 effect of nozzle geometry and flow vortex dynamics,” in *Proceedings of*
714 *the 14th International Conference on Liquid Atomization and Spray Sys-*
715 *tems, ICLASS, Chicago*, 2018.
- 716 [43] M. Battistoni, Q. Xue, and S. Som, “Large-eddy simulation (les) of
717 spray transients: Start and end of injection phenomena,” *Oil Gas Sci.*
718 *Technol. - Rev. IFP Energies nouvelles*, vol. 71, no. 1, p. 4, 2016. <https://doi.org/10.2516/ogst/2015024>.
719
- 720 [44] S. Bornschlegel, C. Conrad, A. Durst, J. Wang, and M. Wensing, “Multi-

- 721 hole gasoline direct injection:in-nozzle flow and primary breakup in-
 722 vestigated in transparent nozzlesand with x-ray,” *International Jour-*
 723 *nal of Engine Research*, vol. 19, no. 1, pp. 67–77, 2018. [https:](https://doi.org/10.1177/1468087417746860)
 724 [//doi.org/10.1177/1468087417746860](https://doi.org/10.1177/1468087417746860).
- 725 [45] F. Orley, S. Hickel, S. J. Schmidt, and N. A. Adams, “Large-eddy sim-
 726 ulation of turbulent, cavitating fuel flow inside a 9-hole diesel injec-
 727 tor including needle movement,” *International Journal of Engine Re-*
 728 *search*, vol. 18, no. 3, pp. 195–211, 2017. [https://doi.org/10.1177/](https://doi.org/10.1177/1468087416643901)
 729 [1468087416643901](https://doi.org/10.1177/1468087416643901).
- 730 [46] A. E. Catania, S. d’Ambrosio, R. Finesso, and E. Spessa, “Effects of
 731 rail pressure, pilot scheduling and egr rate on combustion and emissions
 732 in conventional and pcci diesel engines,” *SAE Int. J. Engines*, vol. 3,
 733 pp. 773–787, 04 2010. <https://doi.org/10.4271/2010-01-1109>.
- 734 [47] S. Busch, K. Zha, P. C. Miles, A. Warey, F. Pesce, R. Peterson, and
 735 A. Vassallo, “Experimental and numerical investigations of close-coupled
 736 pilot injections to reduce combustion noise in a small-bore diesel engine,”
 737 *SAE Int. J. Engines*, vol. 8, pp. 660–678, 04 2015. [https://doi.org/](https://doi.org/10.4271/2015-01-0796)
 738 [10.4271/2015-01-0796](https://doi.org/10.4271/2015-01-0796).
- 739 [48] S. Busch, K. Zha, A. Warey, F. Pesce, and R. Peterson, “On the reduc-
 740 tion of combustion noise by a close-coupled pilot injection in a small-bore
 741 direct-injection diesel engine,” *Journal of Engineering for Gas Turbines*
 742 *and Power*, vol. 138, 04 2016. <https://doi.org/10.1115/1.4032864>.

- [49] G. Dober, C. Garsi, N. Guerrassi, H. Ismail, and J. Shi, “Investigations of the spray structure of large and small diesel injections using spray momentum measurements and their link to injector performance,” in *In Proceedings of SIA Powertrain Conference*, 06 2016.
- [50] M. Gold, R. Pearson, J. Turner, D. Sykes, V. Stetsyuk, G. de Sercey, C. Crua, F. Koukouvinis, M. Gavaises, and M. Mithun, “Simulation and measurement of transient fluid phenomena within diesel injection,” *SAE Int. J. Adv. & Curr. Prac. in Mobility*, vol. 1, pp. 291–305, 01 2019. <https://doi.org/10.4271/2019-01-0066>.
- [51] H. Ismail, G. Dober, J. Shi, K. Karimi, and N. Guerrassi, “Delphi technologies internal report.” 2016.
- [52] G. Kalghatgi, *Fuel/Engine Interactions*. SAE International, 2013.
- [53] *ANSYS Fluent Theory Guide*, 2018.
- [54] D. Beattie and P. Whalley, “A simple two-phase frictional pressure drop calculation method,” *International Journal of Multiphase Flow*, vol. 8, no. 1, pp. 83 – 87, 1982. [https://doi.org/10.1016/0301-9322\(82\)90009-X](https://doi.org/10.1016/0301-9322(82)90009-X).
- [55] N. Aung and T. Yuwono, “Evaluation of mixture viscosity models in the prediction of two-phase flow pressure drops,” *ASEAN Journal on Science and Technology for Development*, vol. 29, no. 2, 2012. <https://doi.org/10.29037/ajstd.58>.

- [56] M. Awad and Y. Muzychka, “Effective property models for homogeneous two-phase flows,” *Experimental Thermal and Fluid Science*, vol. 33, no. 1, pp. 106 – 113, 2008. <https://doi.org/10.1016/j.expthermflusci.2008.07.006>.
- [57] C. Brennen, *Cavitation and bubble dynamics*. Oxford: Oxford University Press, 1995.
- [58] P. J. Zwart, A. G. Gerber, and T. Belamri, “A two-phase flow model for predicting cavitation dynamics,” in *Proceedings of the International Conference on Multiphase Flow, ICMF, Yokohama, 2004*.
- [59] P. Koukouvinis, H. Naseri, and M. Gavaises, “Performance of turbulence and cavitation models in prediction of incipient and developed cavitation,” *International Journal of Engine Research*, vol. 18, no. 4, pp. 333–350, 2017. <https://doi.org/10.1177/1468087416658604>.
- [60] S. B. Pope, *Turbulent Flows*. Cambridge University Press, 2000.
- [61] F. Nicoud and F. Ducros, “Subgrid-scale stress modelling based on the square of the velocity gradient tensor,” *Flow, Turbulence and Combustion*, vol. 62, pp. 183–200, Sep 1999. <https://doi.org/10.1023/A:1009995426001>.
- [62] E. H. I. Ndiaye, J.-P. Bazile, D. Nasri, C. Boned, and J. L. Daridon, “High pressure thermophysical characterization of fuel used for testing

- 784 and calibrating diesel injection systems,” *Fuel*, vol. 98, pp. 288 – 294,
785 2012. <https://doi.org/10.1016/j.fuel.2012.04.005>.
- 786 [63] M. Chorazewski, F. Dergal, T. Sawaya, I. Mokbel, J.-P. E. Grolier, and
787 J. Jose, “Thermophysical properties of normafluid (iso 4113) over wide
788 pressure and temperature ranges,” *Fuel*, vol. 105, pp. 440 – 450, 2013.
789 <https://doi.org/10.1016/j.fuel.2012.05.059>.
- 790 [64] E. Gomez Santos, J. Shi, W. Bauer, and M. Gavaises, “Modelling and
791 prediction of cavitation erosion in gasoline direct injection injectors oper-
792 ated with e100 fuel using a barotropic equation of state,” in *Proceedings of the IMechE Fuel Systems Conference, London*, 2018.
- 794 [65] J. Shi, E. Gomez Santos, G. Hoffmann, and G. Dober, “Large eddy
795 simulation as an effective tool for gdi nozzle development,” *MTZ world-
796 wide*, vol. 79, pp. 58–63, Oct. 2018. [https://doi.org/10.1007/
797 s38313-018-0089-2](https://doi.org/10.1007/s38313-018-0089-2).
- 798 [66] H. Tennekes and J. Lumley, *A First Course in Turbulence*. MIT Press,
799 1972.
- 800 [67] M. Battistoni, S. Som, and D. Longman, “Comparison of mixture and
801 multi-fluid models for in-nozzle cavitation prediction,” in *Proceedings of
802 ASME Internal Combustion Engine Division Fall Technical Conference,
803 ICEF, Dearborn*, 10 2013.

- [68] J. P. V. Doormaal and G. D. Raithby, “Enhancements of the simple method for predicting incompressible fluid flows,” *Numerical Heat Transfer*, vol. 7, no. 2, pp. 147–163, 1984. <https://doi.org/10.1080/01495728408961817>.
- [69] J. Ferziger and M. Peric, *Computational Methods for Fluid Dynamics*. Springer Berlin Heidelberg, 2012.
- [70] B. Leonard, “The ultimate conservative difference scheme applied to unsteady one-dimensional advection,” *Computer Methods in Applied Mechanics and Engineering*, vol. 88, no. 1, pp. 17 – 74, 1991. [https://doi.org/10.1016/0045-7825\(91\)90232-U](https://doi.org/10.1016/0045-7825(91)90232-U).
- [71] F. R. Menter, “Best practice: scale-resolving simulations in ansys cfd,” in *ANSYS Technical report*, 2015.
- [72] F. Moukalled, L. Mangani, and M. Darwish, *The Finite Volume Method in Computational Fluid Dynamics. An Advanced Introduction with OpenFOAM and Matlab*. Springer International Publishing, 1 ed., 2015. <https://doi.org/10.1007/978-3-319-16874-6>. Comparison of the QUICK and FOU schemes in pure advection of a step profile for a scalar ϕ in an oblique velocity field - pages 400 to 403.
- [73] I. B. Celik, Z. N. Cehreli, and I. Yavuz, “Index of Resolution Quality for Large Eddy Simulations,” *Journal of Fluids Engineering*, vol. 127, pp. 949–958, 09 2005. <https://doi.org/10.1115/1.1990201>.

825 [74] F. Brusiani and G. M. Bianchi, “Les simulation of ice non-reactive flows
826 in fixed grids,” in *SAE Technical Paper*, SAE International, 04 2008.
827 <https://doi.org/10.4271/2008-01-0959>.