

Numerical Simulations of the Effect of Cold Fuel Temperature on In-Nozzle Flow and Cavitation Using a Model Injector Geometry

Stavros Bontitsopoulos, Arash Hamzehloo¹, Pavlos G. Aleiferis

Department of Mechanical Engineering, Imperial College London, Exhibition Road, London SW7 2AZ, UK

Roger F. Cracknell

Shell Global Solutions (UK), London, SE1 7NA, UK

Abstract

In the present study, Large Eddy Simulations (LES) have been performed with a 3D model of a step nozzle injector, using *n*-pentane as the injected fluid, a representative of the high-volatility components in gasoline. The influence of fuel temperature and injection pressure were investigated in conditions that shed light on engine cold-start, a phenomenon prevalent in a number of combustion applications, albeit not extensively studied. The test cases provide an impression of the in-nozzle phase change and the near-nozzle spray structure across different cavitation regimes. Results for the 20°C fuel temperature case (supercavitating regime) depict the formation of a continuous cavitation region that extends to the nozzle outlet. Collapse-induced pressure wave dynamics near the outlet cause a transient entrainment of air from the discharge chamber towards the nozzle. The extent of air entrainment appears noticeably reduced on the simulations with colder fuel (−10°C, incipient cavitation) due to the shorter length of the continuous cavitation region, which impedes the backwards air motion. A comparison of the examined cases suggests that in addition to the in-nozzle flow, the injection temperature also has a noticeable effect on the macroscopic structure of the emanating fuel spray. On the colder injection, a symmetric jet is formed in the near-nozzle region, in contrast to the supercavitating cases that demonstrate a concave surface on the side of the step nozzle edge (a byproduct of intense air entrainment) and an increased spray opening angle. The above observations attest that the size and intensity of the cavitation features tend to become suppressed as the temperature of the injected fuel decreases. This has a binary effect: the injection of cold fuel may adversely influence the near-nozzle jet formation, while at the same time contributing to enhanced fuel delivery towards the combustion chamber due to the reduced flow obstruction from the entrained air and the diminished in-nozzle vapour structures.

Introduction

Research into optimizing Direct-Injection Spark-Ignition (DISI) engines remains nowadays more relevant than ever, despite the automotive industry's long term plans for gradual electrification of the passenger car fleet. Increasing emphasis is placed on the intricate sub-processes that control the DISI behaviour – from adjusting injection strategies to achieve enhanced droplet breakup and effective mixture preparation within the combustion chamber, to examining aspects like charge cooling and active control of the in-cylinder flow.

Compliance with regulation and minimization of pollutant emissions are both important factors, as internal combustion engines are expected to continue playing a role in the mobility future, with the engine efficiency optimized and fuel flexibility increased.

Cold Engine Startup

The transient engine cold-start phase is at the same time gaining more widespread attention as legislative bodies introduce policies that take it under consideration. In the context of Real-World driving emissions tests, cold-start is defined as the state that a vehicle is in, after it has been left idle for a minimum interval of 6 hours. After such preconditioning, the engine is expected to have incurred a full cool-down, with the temperature of its components (including the oil, coolant and exhaust after-treatment system) substantially lower than their normal operating temperature [1].

Since 2000, with the introduction of the 3rd European emission standards (Euro 3) for fuel consumption and emissions levels, the EU has made it a requirement that dynamic drive-cycle measurements should begin from an engine temperature of 20 to 30°C – reflecting the conditions of ambient air [2, 3]. This rule became stricter in 2002 with the introduction of the state of −7°C as the accepted cold-start temperature in the New European Driving Cycle for gasoline engines [4, 5], while diesel engines have been certified at temperatures as low as −20°C [6]. Once the engine is fired, the cold-start state is thought to end after the coolant reaches 70°C for the first time, and in any case, not later than 5 minutes after initial cranking [7].

Cold-start is especially relevant for conventional systems (not equipped with thermal encapsulation or block-heater arrangements), in which the first few cycles after initial engine cranking are accompanied by the injection of low temperature fuel inside the cylinder. This cold fuel exhibits a breakup behaviour that is noticeably different compared to a fully warmed-up operation. At such low temperatures, variations in the respective cavitation and evaporation mechanisms, as well as the absence of the frequently assistive flash-boiling process of the more volatile fuel compounds [8], are introducing disturbances to the primary atomization mechanism that can significantly alter the spray structure. This altered spray structure is a result of variations in the phase change process that occurs in low-pressure areas of the the nozzle injection holes [9, 10].

The importance of these discrepancies can come into focus by looking at the cold spray's impact on the engine's overall combustion

¹Currently at the Department of Aeronautics, Imperial College London

efficiency and pollutants release. As the fuel thermophysical properties vary with temperature, the formation of the injected sprays and the droplet breakup process are affected, leading to longer penetration lengths and an increased incidence of impingement on internal cylinder surfaces at cold conditions [5, 11, 12]. The fuel/air mixing process is also altered in a way that induces the formation of pockets where a locally rich mixture resides. These pockets can disrupt the spark ignition process and act as a source of unburnt hydrocarbon and particulate matter emissions [13]. Particularly in DISI systems, the engine cold-cranking behaviour is characterized by poor mixture preparation, a higher propensity for misfires and increased pollutant emissions.

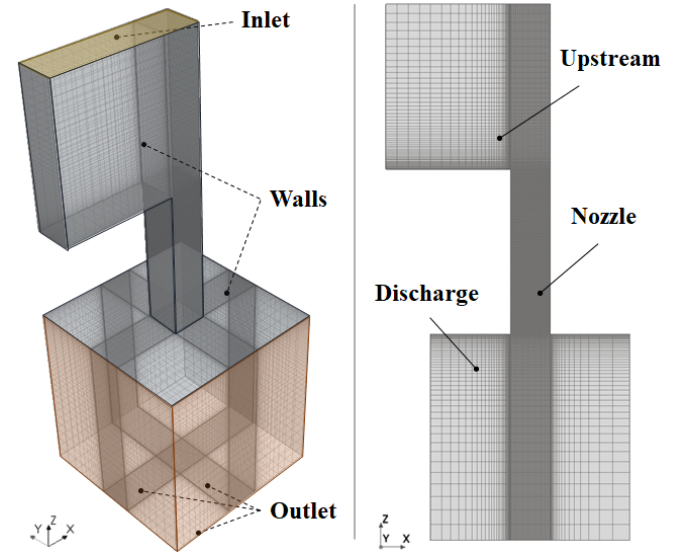
There is therefore a need to study the phenomenology of fuel sprays at cold-start conditions and characterise the underlying physical mechanisms. A deeper fundamental understanding of the effects of cold start, supported by relevant data can help mitigate its adverse effects on engine operation, thus serving industrial and consumer economical interests, along with international environmental policies.

Present Contribution

The focus of this study is on investigating the kinetic and morphological properties of cold fuel sprays at the in-nozzle and near-nozzle region. Observations made from the work presented here can be translated into techniques for optimizing the spray behaviour during the initial cooled-down engine transients. These techniques can serve the need of the automotive industry to drive DISI compliance with the ever more stringent emissions and fuel economy regulations, while also promoting the current understanding of cold spray morphology.

This study aspires to contribute to those goals through numerical simulations of cold fuel injections from a simple 3D step-nozzle geometry. This model has been used, so as to focus on the underlying physical mechanisms at play and to decouple the analysis from geometry-specific effects of real DISI injectors that vary depending on the manufacturer and engine combustion chamber geometry, such as the different number of holes and orientations. Despite the simplified geometric setup however, the operating conditions studied here, as reflected by the Reynolds, cavitation and Weber numbers, are representative of real injectors. This work provides:

- a description of the in-nozzle transient phase change in a range of injection temperatures (20°C to -10°C), which is indicative of real-world engine cranking conditions, and for varying injection pressures (0.22 to 0.28 MPa). The analysis is focused on the variation of the cavitation regime as the thermodynamic conditions of the fuel are modified, including the effect on the length of the continuous cavitation region, and its dynamic shedding characteristics.
- an observation of the near-nozzle jet formation for the aforementioned test cases. This includes an assessment of the flow field in the region directly downstream of the nozzle outlet, an estimation of the flow discharge rates as well as an inspection of the jet opening angle. The analysis is coupled with the description of the flow field within the nozzle, to comment on the mechanisms that exert an influence on the near-nozzle spray formation.
- an examination of the phenomenon of transient air entrainment from the discharge region upstream into the nozzle. As cavitation structures shed and implode near the nozzle outlet they produce pressure gradients that facilitate the backwards motion of air. The relative effect of injection temperature on this mechanism is assessed and potential implications for real-world engine performance are discussed.



Domain dimensions ($x \times y \times z$)	Upstream $8 \times 1.94 \times 8$ mm Nozzle $1.94 (= W_n) \times 1.94 \times 8$ Discharge $9.7 (= 5 W_n) \times 9.7 \times 9.7$
Cells	~ 8.6 M hexahedral
BCs	Inlet: Dirichlet p_{inj} Walls: Impermeable, No-slip Outlet: Dirichlet $p_{\infty} = 0.1$ MPa

Figure 1: Schematic and specifications of the computational grid, created using the Ansys ICEM meshing tool.

Computational Framework

The emerging cavitation structures in the nozzle and their influence on the ensuing spray formation are difficult to capture using conventional experimental techniques, due to the small scale of the associated phenomena. A number of studies in the open literature [14, 15, 16] therefore rely on numerical modelling of reference injector geometries and correlate their computationally-derived data against experimental results.

In the present work, the setup has been based on the step-nozzle geometry used by [17] that produces asymmetric cavitation modes, (geometry shown in Fig. 1). Experimental measurements are available for this nozzle, including Laser Doppler Velocimetry (LDV) measurements, as well as estimates for the Reynolds and cavitation numbers. According to [17], simulations based on in-nozzle velocities below a critical value correspond to an incipient cavitation regime, in which the number of produced vapor nuclei is constrained and the continuous cavitation region reattaches within the nozzle's length. This is in contrast to higher-velocity nozzle flows, where the cavitation region extends to the outlet surface, thus increasing the overall nozzle void fraction and affecting significantly the near-nozzle spray formation. At even higher velocities, a hydraulic flip is formed, where the effect of the cavitating flow on the jet's structure is suppressed. The present study explores conditions that pertain to the former two regimes.

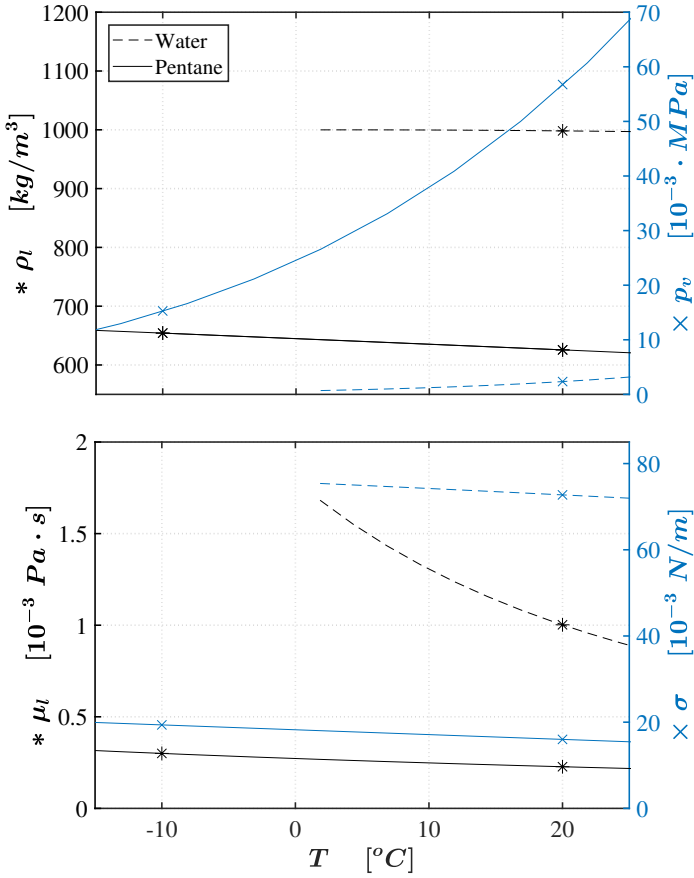


Figure 2: Variation of reference thermophysical properties of water and *n*-pentane with Temperature. Test case conditions denoted with markers. Data derived from [18].

The simulations were performed with the commercial CFD software STAR-CCM+. The modelling framework was based on Large Eddy Simulations of incompressible flow, solving for continuity and momentum equations in 3D [19]. The grid consisted of approximately 8.6M hexaedral cells, as shown on Fig. 1. Refinement was applied along the nozzle walls, around the step contraction, as well as downstream of the nozzle outlet, in the region where the jet is formed. The minimum cell size near the walls of the nozzle has been set equal to 1 μm , and the maximum at the center of the channel 18.2 μm . This refinement yielded an average y^+ of ~ 0.8 on the first cell. In the near-wall region ($y^+ \leq 10$) cell aspect ratios were kept in the range of 10 to 100, with the rest of the grid being of the order 1. The boundary layer flow was resolved through a full y^+ treatment and a WALE (Wall-Adapting Local-Eddy Viscosity) subgrid scale model [20] ($\kappa = 0.41$ and $C_w = 0.544$). The discretization was considered fine enough to model the turbulent field at lengths scales of the order of the Taylor microscale.

The mesh quality has been assessed through the following quality index proposed by [21]:

$$LES-IQ_v = \frac{1}{1 + 0.05 \left(\frac{\langle \mu_{t,eff} \rangle}{\mu} \right)^{0.53}}$$

with $\mu_{t,eff}$ representing the sum of the turbulent and molecular viscosities. The $LES-IQ_v$ metric evaluates the grid resolution through a ratio of viscosities, which is equivalent to the ratio of modelled turbulent kinetic energy through the subgrid scale wall function and

Table 1: Test matrix with examined conditions and values for reference thermo-physical parameters used [18]

	T_{inj} [°C]	p_{inj} [MPa]	ρ_l [kg/m³]	μ_l [Pa · s]	p_v [MPa]
Water	20	0.22	$1.0 \cdot 10^3$	$1.0 \cdot 10^{-3}$	$2.3 \cdot 10^{-3}$
<i>n</i> -Pentane	20	0.22	$6.2 \cdot 10^2$	$2.3 \cdot 10^{-4}$	$5.7 \cdot 10^{-2}$
	20	0.28	$6.2 \cdot 10^2$	$2.3 \cdot 10^{-4}$	$5.7 \cdot 10^{-2}$
	-10	0.22	$6.5 \cdot 10^2$	$3.0 \cdot 10^{-4}$	$1.5 \cdot 10^{-2}$

the resolved kinetic energy. [22] suggests that a minimum of 80% of the kinetic energy should be resolved in LES simulations when a near-wall model is used. This dictates that the ratio of $\langle \mu_{t,eff} \rangle / \mu$ should be less than 25. Ensemble averaging of the validation case in the current study suggests that $LES-IQ_v$ is greater than 90% throughout the domain, with minimum values appearing near the cavitation region reattachment point, and the ratio of viscosities is always less than 5. The quality metric has been computed without taking into account the contribution from artificial viscosity arising from the use of numerical methods, however it is thought to provide an approximate quality estimate, in accordance with other literature studies [23].

The flow domain has been initialized through a precursor Reynolds-Averaged Navier Stokes (RANS) simulation stage, using a k- Ω SST closure model [24]. Large Eddy Simulations then followed, using a 2nd-order implicit temporal marching scheme with a timestep of $0.5 \cdot 10^{-6}$ s and a 2nd-order bounded central differencing scheme in space, the specifications and boundedness criteria of which can be found in [25] and [26]. The convective Courant–Friedrichs–Lewy number remained well below 1 across the in-nozzle domain, with a spatially-averaged value of ~ 0.5 . The total physical runtime of each simulation corresponded to an average of 30 nozzle flow-through times t_c (defined as the ratio of the mean in-nozzle velocity magnitude $\langle V_n \rangle$ and the nozzle length L_n), with statistical averaging applied on the last $\sim 20t_c$, in which the turbulent flow and the cavitation domains developed fully and acquired quasi-steady state conditions.

The influence of both injection temperature and injection pressure have been investigated, with the temperature varying between 20 and -10 °C, in order to shed light on the effect that an engine cold-start has on the in-nozzle cavitation and the macroscopic spray structure. To account for the temperature variation of the fuel’s thermophysical properties within the examined range, the changing viscosity, density and saturation pressure values have been inputted into the software (Fig. 2 and Table 1). Simulations were performed with *n*-pentane, chosen due to its relevance for spark ignition engine applications as a representative of the high-volatility isomers of similar components contained in gasoline, as well as due to the high variation that *n*-pentane exhibits with regards to its evaporation at different temperature points. In addition to *n*-pentane, water simulations have also been carried out for benchmarking purposes against [17]’s published data.

Cavitation Modelling

The injector flow has been modelled as an Eulerian multiphase mixture comprising the liquid fuel as well as a multicomponent gas containing fuel in vapor form and air. A Volume of Fluid (VOF) approach has been used to differentiate the liquid and gaseous phases through a volume fraction transport equation [27], along with a High-Resolution Interface Capturing (HRIC) convection scheme [28, 29].

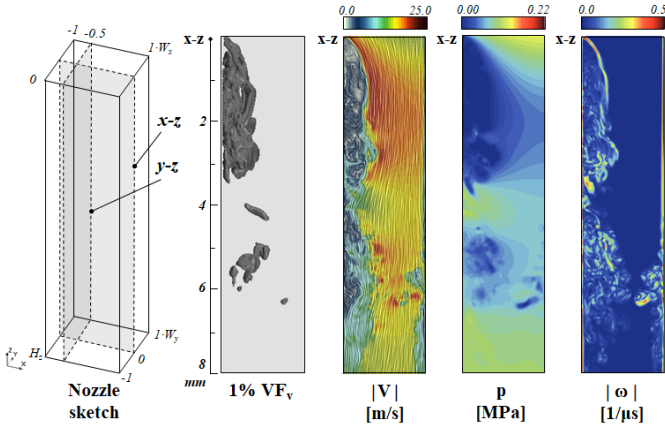


Figure 3: Validation case (water injection) – Instantaneous isocontours of flow and thermophysical properties at $t_c = 20.65$.

Energy equations have been neglected in the present analysis, with the cavitating flow treated as isothermal. This assumption is widely used in the literature [14, 30], with studies suggesting that in configurations where the flow temperature varies, thermal effects are at least two orders of magnitude slower compared to cavitation time scales [31].

In order to capture the dynamic growth, shedding and collapse of cavitating structures, the simulations rely on a homogeneous seed-based framework [32]. Seeding models work by providing a uniform distribution of spherical nuclei of equal initial radius in the fluid, which serve as surfaces for cavitation bubbles to grow once the appropriate conditions are met. Such zero-dimensional approaches are commonly used in industrial studies, although they may not resolve all physical phenomena that are present in the simulation. For the present setup, the number density of seeds in the liquid phase and initial seed diameter have been defined respectively equal to: $1.5 \cdot 10^{14}$ and $0.1 \mu m$. These values were chosen with reference to the parametric studies of [33] and [31].

The Schnerr-Sauer model [34] has been used to model cavitation through a simplified form of the Rayleigh-Plesset equation [35], in which the influence of bubble growth acceleration due to flow inertia, as well as the viscous and surface tension effects are neglected. Research studies suggest that for most common applications such simplifications are acceptable without considerable implications for the overall physical accuracy of the model [36].

To gain confidence into the computational setup, a number of preliminary analyses were performed using a reduced-order, 2D grid. These included a mesh dependency investigation with meshes bearing 25%, 50% and 200% of the total amount of the reference grid cells. Results suggested that the chosen configuration resolves the measured parameters of interest within 6.2% tolerance for the average in-nozzle void fraction (in contrast to 30% for the coarser grids), and 15.5% for the characteristic cavitation region lengths (compared to 53.1%).

Sensitivity studies were also carried out into different boundary and flow modelling conditions examining: *i*) the size of the upstream flow region (comparing a grid with a streamwise length equal to L_n with a one of $3 \cdot L_n$, on which a correction has been applied on the inlet boundary to account for pressure losses through the Darcy-Weisbach equation, and indicating that the shorter grid can capture the nozzle inlet velocity profiles and massflow rates within a tolerance of 3.6%), *ii*) the refinement at the upstream left wall (showing the presence of a recirculating flow on a refined grid that exerts no meaningful influence on the nozzle inlet velocity) and *iii*) the chosen turbulence

Table 2: Validation case (water injection) – case properties

	$\langle V_n \rangle^{20t_c}$ [m/s]	Ca [–]	Re_n [–]	We_n [–]
20 °C Sou et al. [17]	12.8	1.19	27,700	4,360
20 °C This study	12.2	1.32	23,500	3,940

model (with a comparison between WALE and Smagorinsky showing a similar capacity to capture the in-nozzle flow with a distribution that varies by 5.7% on average for the axial velocity and 9% for the turbulent velocity profiles. Those background studies helped provide further insight into the setup and physical modelling used, however they are not presented here in a more detailed graphical form in the interest of brevity.

Validation Case

Figure 3 illustrates the numerically-resolved cavitating flow field that develops during an injection with water at $T_{inj} = 20^\circ C$. This test case has been used for benchmarking the current modelling setup against Sou et al.’s [17] reference experiment. Water injected at $p_{inj} = 0.22$ MPa has been observed to exhibit a behaviour that corresponds to an “incipient cavitation” regime. At the inlet of the nozzle, a sheet cavity appears anchored at the sharp edge, giving rise to a continuous cavitation region that remains attached on the wall. The length of this continuous region varies dynamically, although characteristically for an incipient regime, reattachment happens at all times within the nozzle’s length, well upstream of the nozzle outlet.

At the inner core of this region pockets of fluid evaporate due to the low local pressure. Along the outer border, the large shear leads to a transient entrainment of vapor structures, which take the form of clouds and nuclei. These low inertia structures are entrained by the surrounding flow, leading to a transport of pockets with low local pressure to the undersaturated downstream nozzle region, where they subsequently collapse, causing the emergence of pressure waves. The genesis and disintegration of these structures, as well as the influence exerted from the surrounding flow has been described in the experimental study of [37] and the LES-based computational investigation by [38].

For the examined case, the illustrations for the flow field exhibit similar cavitation patterns with [17], exhibiting a comparable length and thickness of the continuous cavitation region. The spatial distribution of $|V|$, p and $|\omega|$ also align well with [17]’s theoretical descriptions and the corresponding plots included in their paper. Reference non-dimensional parameters including the mean in-nozzle velocity, cavitation and Reynolds numbers also stand in good agreement.

$$Ca = \frac{p_\infty - p_v}{0.5 \rho_l \langle V_n \rangle^2}, \quad Re_n = \frac{\langle V_n \rangle \cdot W_n}{\nu_l},$$

$$We_n = \frac{\rho_l \langle V_n \rangle^2 W_n}{\sigma}$$

From [17]’s reference experiment $\langle V_n \rangle = 12.8$ m/s, $Ca = 1.19$, $Re_n = 27,700$ and $We_n = 4,360$ while from the present study’s computational data $\langle V_n \rangle^{20t_c} = 12.2$ m/s, $Ca = 1.32$, $Re_n = 23,500$ and $We_n = 3,940$.

Figure 4 provides a comparison of velocity profiles obtained at reference points inside the nozzle. These are compared against [17]’s

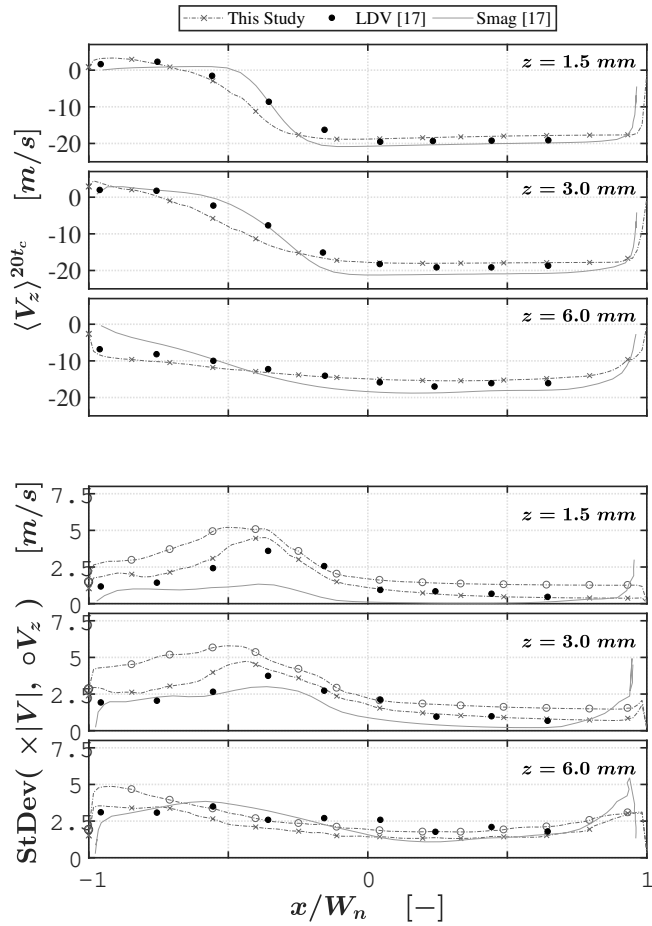


Figure 4: Validation case (water injection) – TOP: Mean streamwise velocity profiles V_z , BOTTOM: Flow turbulence velocity.

experimental LDV data, derived from an optically-accessible injector and a numerical approximation from the same reference study, which is based on LES and a Smagorinsky turbulence model. As evidenced by the graphs, the data for the mean streamwise velocity V_z match well with the experimental measurements. The plots indicate that the reattachment of the flow occurs at a position between $z = 3$ and 6 mm. In this case, the numerical results tend to underestimate the width of the separated flow region, as in the studies of [39, 40]. As noted in those papers, the average interface velocity estimated by the potential flow theory as $V_{int} = \langle V_n \rangle \cdot \sqrt{1 + Ca} = 19.3$ m/s provides a decent approximation for the computationally derived value.

A good agreement exists throughout the channel length for the magnitude of the velocity fluctuations, with the current simulations predicting well the location and the amplitude of the peak. Relative agreement also exists for the turbulent velocity fluctuations in the streamwise direction outside the continuous cavitation region. Within this region, although qualitative agreement can be seen, values are higher (almost double) particularly at the $z = 1.5$ and $z = 3.0$ mm positions. This deviation is also present in other numerical studies from the open literature [15, 17, 39].

Fuel Injection

Pentane belongs to the more volatile components of gasoline, exhibiting a vapor pressure which is generally an order of magnitude

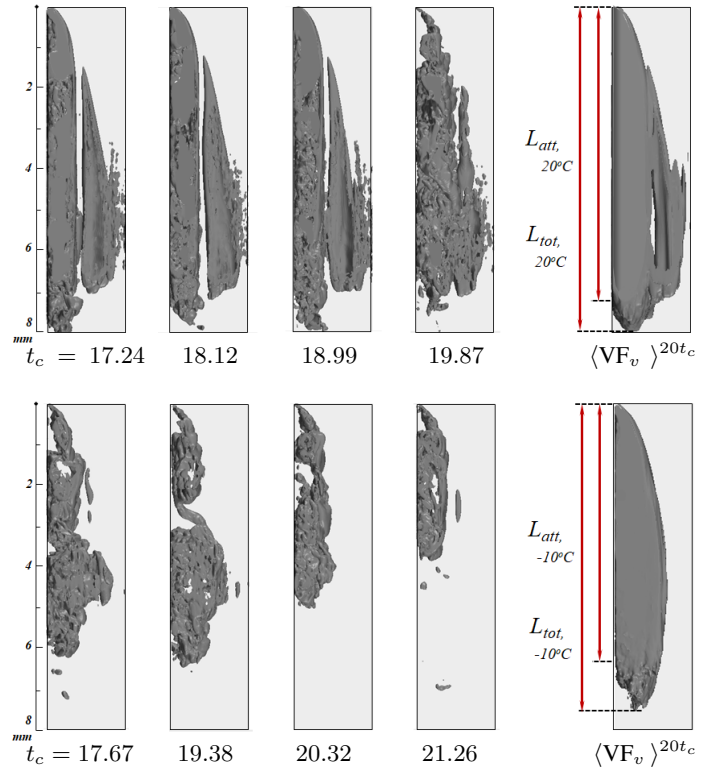


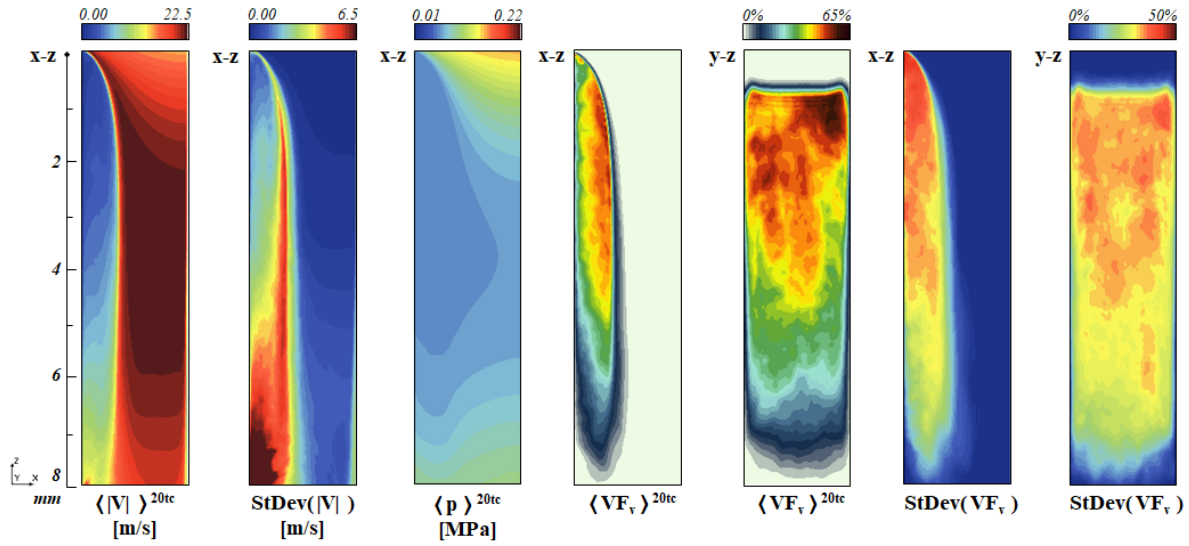
Figure 5: Pentane injection – Transient in-nozzle cavitation and mean behaviour averaged over $20 \cdot t_c$. Isosurfaces of 1% VF_v for TOP: 20°C , BOTTOM: -10°C .

higher than water within the temperature range tested. This has a direct consequence to the cavitating flow that develops in the nozzle.

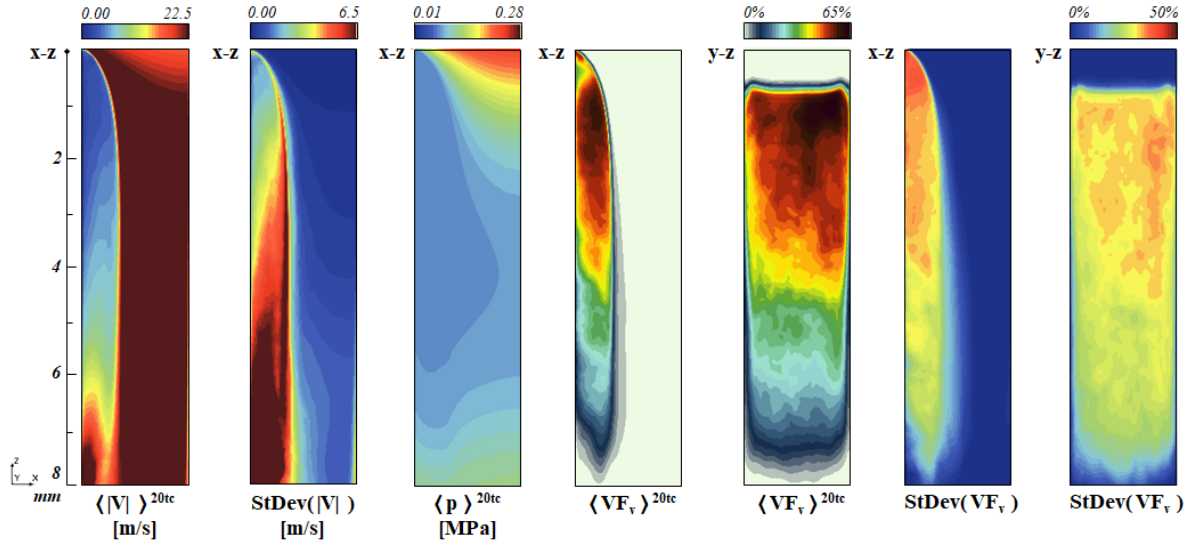
Maintaining otherwise constant boundary conditions as in the validation case, the injection of 20°C pentane leads to a formation of a “super cavitation” regime, as illustrated in the instantaneous and the time-averaged snapshots in Fig. 5. This is evident by the continuous recirculation region, which spans the entire length of the channel and reaches the nozzle outlet, with less prominent detached regions appearing also on the front and back sidewall. Vapor fills the majority of the injector’s cross-sectional area, leading to low discharge coefficients and thus hindering the delivery of fuel to the downstream chamber. Due to the significant disturbance of the in-nozzle flow, a deformation is expected for the jet emanating from the square orifice, leading to an enhancement of the primary atomization process [41]. The cavitation number in this test case is equal to $Ca = 0.70$ (with lower values signifying a stronger cavitation field), while the Reynolds and Weber numbers are respectively equal to $Re_n = 75,000$ and $We_n = 14,900$. Despite the larger size of the reference geometry used in this study, these non-dimensional numbers are in accordance with experimentally observed values found in real injector nozzles for DISI engines. [42] and [43] report Reynolds numbers in the range of $\mathcal{O}(10^3)$ to $\mathcal{O}(10^4)$, for injector body temperatures between 7 and 185°C , an injection pressure of 15 MPa and a range of examined fuels. The equivalent Cavitation number varies between $\mathcal{O}(10^{-1})$ and $\mathcal{O}(10^0)$ (adjusted from the reference paper to account for the discharge pressure used here), while the Weber number is of the order of $\mathcal{O}(10^4)$.

The sensitivity of the flow to different boundary conditions has been tested by altering the model’s injection pressure and temperature. An increase of p_{inj} from 0.22 MPa to 0.28 MPa yields a marginal

• $T_{inj} = 20\text{ }^{\circ}\text{C}$, $p_{in} = 0.22\text{ MPa}$



• $T_{inj} = 20\text{ }^{\circ}\text{C}$, $p_{in} = 0.28\text{ MPa}$



• $T_{inj} = -10\text{ }^{\circ}\text{C}$, $p_{in} = 0.22\text{ MPa}$

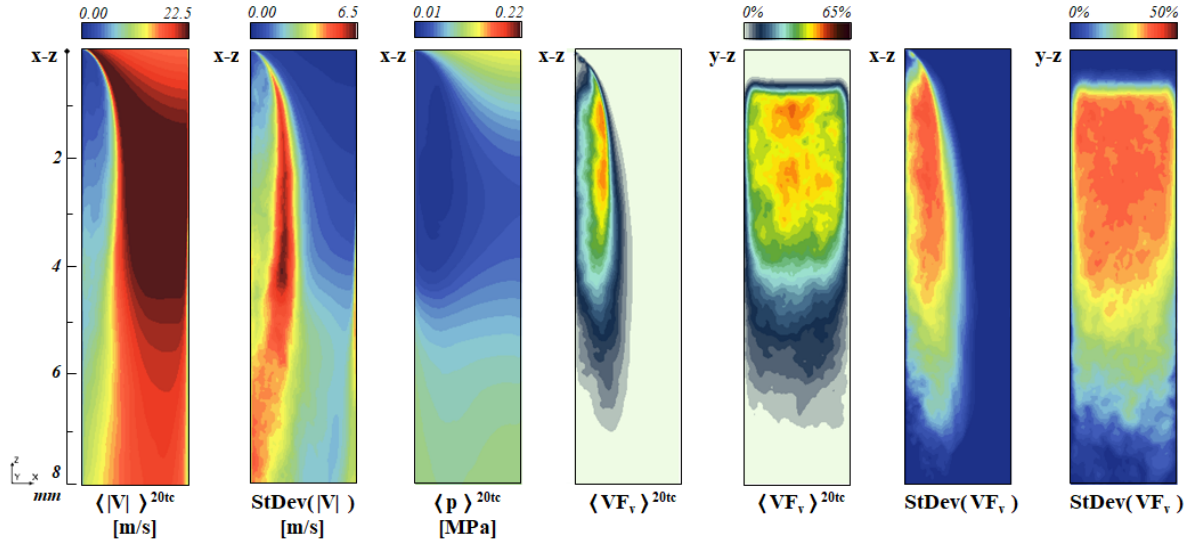


Figure 6: Pentane injection – Iso-contours of temporally averaged flow and cavitation properties.

Table 3: Pentane injection – case properties

	$\langle V_n \rangle^{20t_c}$ [m/s]	Ca [–]	Re_n [–]	We_n [–]
$T_{inj} = 20^\circ C$ $p_{inj} = 0.22 MPa$	14.0	0.70	75,000	14,900
$T_{inj} = 20^\circ C$ $p_{inj} = 0.28 MPa$	16.5	0.51	87,900	20,600
$T_{inj} = -10^\circ C$ $p_{inj} = 0.22 MPa$	15.0	1.15	63,500	14,800

enhancement in the formation of vapor, with the time-averaged volume fraction of vapor ($\langle VF_v \rangle$) increasing from 10.5% to 10.9%. This increase is apparent in Fig. 6 with $\langle VF_v \rangle^{n,20t_c}$ attaining high local maxima (62.8%) and covering an extended spatial area of the nozzle. The upstream nozzle region in particular demonstrates a higher degree of saturation, with sharp VF_v gradients appearing across the outer boundary of the cavitation region. The flow field still classifies as a supercavitating regime, with vapor covering a significant portion of the nozzle area. Due to the increase of injection pressure, the cavitation number Ca reduces to 0.51, while the increase of the average in-nozzle velocity $\langle V_n \rangle^{20t_c}$ from 14.0 to 16.5 m/s yields a higher Reynolds number $Re = 87,900$.

Decreasing the injected fuel's temperature from $20^\circ C$ to $-10^\circ C$ leads to a reversal of the flow to a less potent “incipient” cavitation regime ($Ca = 1.15$, $Re = 63,500$), as evidenced by Fig. 5 and 6, with the time-averaged velocity and pressure fields at $-10^\circ C$ resembling the behaviour observed in the injection with water. Although the increase of p_{inj} does not bring about a substantial alteration on the length of the cavitation region as the channel is already oversaturated, the decrease in temperature and the associated change in the cavitation dynamics does induce a more noticeable difference, with the overall length of the region $\langle L_{tot} \rangle^{20t_c}$ reducing from 8.0 mm (100% of the total nozzle length) to 7.5 mm (93.8%). The attached portion of the cavity $\langle L_{att} \rangle^{20t_c}$ also exhibits a notable reduction from 7.6 mm (94.8%) to 6.4 mm (79.4%). At the same time, the time-averaged value of the integral volume fraction of vapor in the channel is reduced by 44% from the original 10.5% to 5.9%.

The temporal traces of the in-nozzle void fraction have been analysed, to extract information about the dynamic variation of the cavitation phenomenon. For the two supercavitating injections with $T_{inj} = 20^\circ C$, long periods of vapor accumulation and eventual collapse are observed (Fig. 7). Multiple secondary peaks appear on the signal; this is due to the induced flow disturbance which is affected by interference from the downstream discharge region. The incipient cavitation case at $-10^\circ C$ on the other hand retains a more harmonic variation of VF_v , with a clear distinctive peak frequency.

A spectral analysis of the $\langle VF_v \rangle^n$ time series provides estimates for the characteristic frequency of each case. For the three injections, a dominant peak can be distinguished that attests to the periodic growth and shrinking of the recirculation region itself and the shedding of cavitation structures. A characteristic frequency of 1200 Hz appears for the $-10^\circ C$ signal, with the frequency reducing to $\sim 530 Hz$ for the two $20^\circ C$ cases and a second harmonic appearing at $\sim 1130 Hz$ for $p_{inj} = 0.28 MPa$. A Strouhal number can be defined based on the average length of the attached cavitation region as such

$$St = \frac{f_{cav} \cdot \langle L_{att} \rangle^{20t_c}}{\langle V_n \rangle^{20t_c}}$$

The values of St follow a similar trend as described above, by ranging between 0.29 and 0.51 for the 20 and $-10^\circ C$ cases respectively.

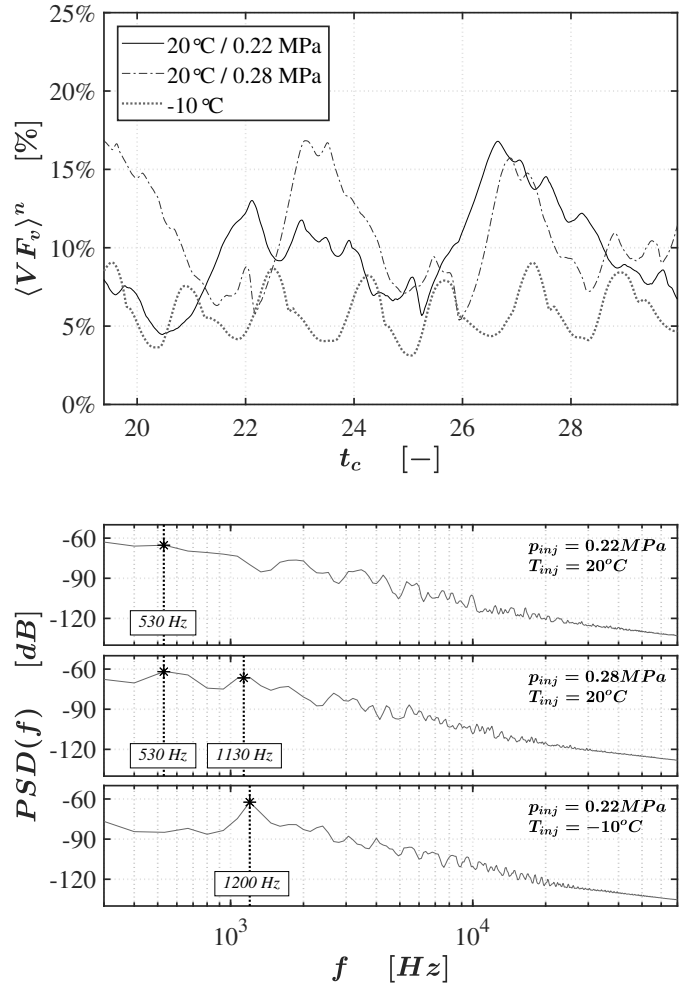


Figure 7: Pentane injection – TOP: Temporal variation of the integral in-nozzle vapor content $\langle VF_v \rangle^n$, BOTTOM: Power spectral densities of $\langle VF_v \rangle^n$

These values are in good agreement with numerical studies on similar nozzle geometries [39] and experimental estimates from cylindrical orifices [44], with the latter study reporting values of St in the range 0.3 – 0.5.

The data provide evidence that the intensity of the cavitation mechanism is reduced as the temperature of the fuel decreases. The observed reduction in the characteristic frequency is in line with [39] who comment that a decrease can be expected with a reduced cavitation number (from 1100 Hz on their $Ca = 1.19$ case with increased injection pressure to 750 Hz for $Ca = 0.84$ with lower p_{inj}).

Near-nozzle Jet Formation

The dynamic cavitating phenomena that develop inside the nozzle play a significant role in the formation of the discharged fuel spray. There is a number of processes, from the near-nozzle collapse of the detached cavitation structures, to the fuel spray atomization process, the turbulent mixing mechanism and the subsequent combustion that takes place in the discharge chamber, which are all intricately interconnected. Investigations from the open literature as well as the present study include both the near-nozzle region on their analysis, in an attempt to observe and comment upon these complex interactions.

In the current models, certain limitations exist that were taken into account when assessing the macroscopic jet formation process. The discharge chamber is dimensionally constrained compared to work from other authors; the box containing the chamber has a sidelength of $5 \cdot W_n$, whereas generally, larger outlet regions are used in the open literature (up to $160 \cdot W_n$ in [14], and $75 \cdot W_n$ in [39]) in order to minimize the influence of outlet boundary conditions from the surrounding surfaces onto the jet.

The mesh refinement inside the nozzle is a second factor that can introduce discrepancies. According to the inviscid flow of [45], the pressure recorded at the region of collapse is inversely proportional to the size of the cells at the collapse center. A scaling law to account for the inhomogeneous resolution of the mesh has been proposed by [45]. At the in-nozzle area, the mesh in the current study is roughly two times finer than the one used by [39] and approximately four times more refined compared to [17] and [14]. However, accounting for the above limitations, collapse events are not addressed in this study, and the analysis of spray structure is constrained to the near-nozzle area.

Fuel Discharge Coefficient

The density of the injected fluid, as well as its evaporation characteristics influence the amount of fuel that is delivered to the downstream chamber. The pentane injections indicate an increase in massflow as T_{inj} reduces; this increase is up to 11.8% from 33.0 g/s at 20°C, 0.22 MPa, to 37.0 g/s at -10°C, while the increased injection pressure to 0.28 MPa also yields a higher discharge amount of 38.8 g/s.

For the cold injection case that corresponds to an incipient cavitation regime, the recorded temporal trace of $\dot{m}$ has been observed to follow a periodicity similar to that of VF_v . As [39] argues, the maximum vapor content can be linked to a time-delayed maximum in the outlet massflow; this time delay is of the order of the nozzle flow-through time. In contrast to -10°C, the two 20°C cases that correspond to super-cavitating regimes are accompanied by a more random variation in the temporal trace of $\dot{m}$. A dynamic tracing of vapor structure disintegration events [38, 39] has shown more frequent collapses and potent pressure waves in cases of injections with lower cavitation numbers. Pressure-wave dynamics resulting in a reduction of the amount of discharge fuel in submerged test nozzle environments have been observed both numerically [46], as well as experimentally [47]. The collapses, which take place nearer the nozzle outlet surface, also cause an upstream entrainment of free air from the discharge chamber to the in-nozzle area; this further reduces the fuel massflow at the outlet plane.

With respect to the flow discharge coefficient

$$C_D = \frac{\dot{m}}{A_n \sqrt{2\rho(p_{inj} - p_\infty)}}$$

the variation is more subtle, with the values ranging between $C_D = 0.72$ for 20°C and 0.78 for -10°C, while the $p_{inj} = 0.28$ MPa case produces a C_D of 0.68. The current simulations follow the expected trend of increasing C_D as the cavitation number becomes higher. The results compare favourably with the 1D model theory described by [48] and experimental data aggregated from the open literature for injectors with sharp nozzle orifices.

Air Entrainment

The implosion of shedded cavitation structures inside the nozzle affects the atomization process of the emerging jet on multiple scales. This temporal and spatial occurrence of vapor structure implosions in the nozzle outlet region, their intensity, as well as their influence on

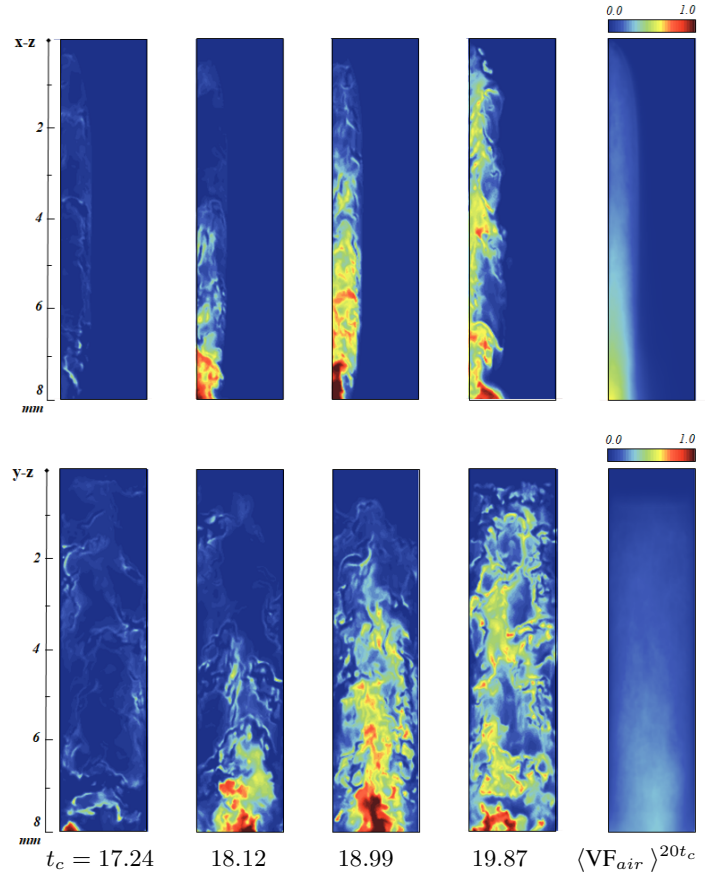


Figure 8: Pentane injection – Transient evolution of air entrainment in the nozzle for the 20 °C, 0.22 MPa case. Isocontours of VF_{air} TOP: x-z plane, BOTTOM: y-z plane.

the air entrainment mechanism have been previously described by [14, 37] and [39]. Figures 8, 9 and 10 of this study illustrate the transient entrainment of free air from the discharge chamber towards the injector, caused by collapse-induced pressure wave dynamics in the area right upstream of the outlet. The air entrainment is visualized through of isosurfaces of the 10% VF_{air} component, overlayed on images of the 1% vapor content.

As air from the discharge chamber is entrained inside the nozzle, it stays confined inside the continuous cavitation region and fills up a considerable portion of it, while mixing with vapor. The trapped air is primarily observed inside this continuous cavity; entrainment towards the two secondary recirculation regions on the front and back walls of the nozzle is more constrained on the other hand, as those regions reattach further upstream from the outflow surface. The area dominated by the liquid phase is also free of air pockets, since the increased fuel velocity hinders the backflow air motion.

The phenomenon is more pronounced in the supercavitating cases with pentane at 20 °C. For the $p_{inj} = 0.22$ MPa case, the temporal trace of VF_{air} over $20 \cdot t_c$ displays a periodic oscillation around a mean value of 5.7%, indicating the presence of regular cycles with air penetrating inside the nozzle and then eventually getting discharged, while a residue value of 2% VF_{air} remains present throughout the cycle. The higher injection pressure (0.28 MPa) is accompanied by further enhancement of the phenomenon with a mean VF_{air} over $20 \cdot t_c$ of the order of 11.5%. Figure 8 illustrates the air entrainment behaviour on the nozzle midplane and a cross-sectional surface inside

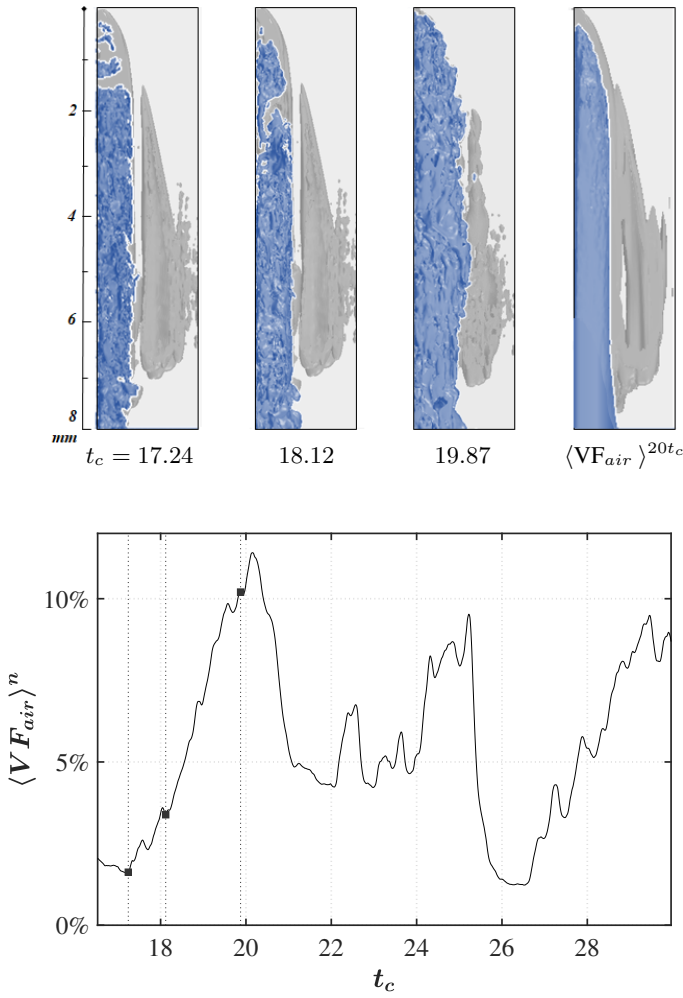


Figure 9: Pentane injection 20°C , 0.22 MPa – transient air entrainment into the nozzle. Instantaneous isosurfaces of 10% VF_{air} superimposed on 1% VF_v and average behaviour over $20 \cdot t_c$.

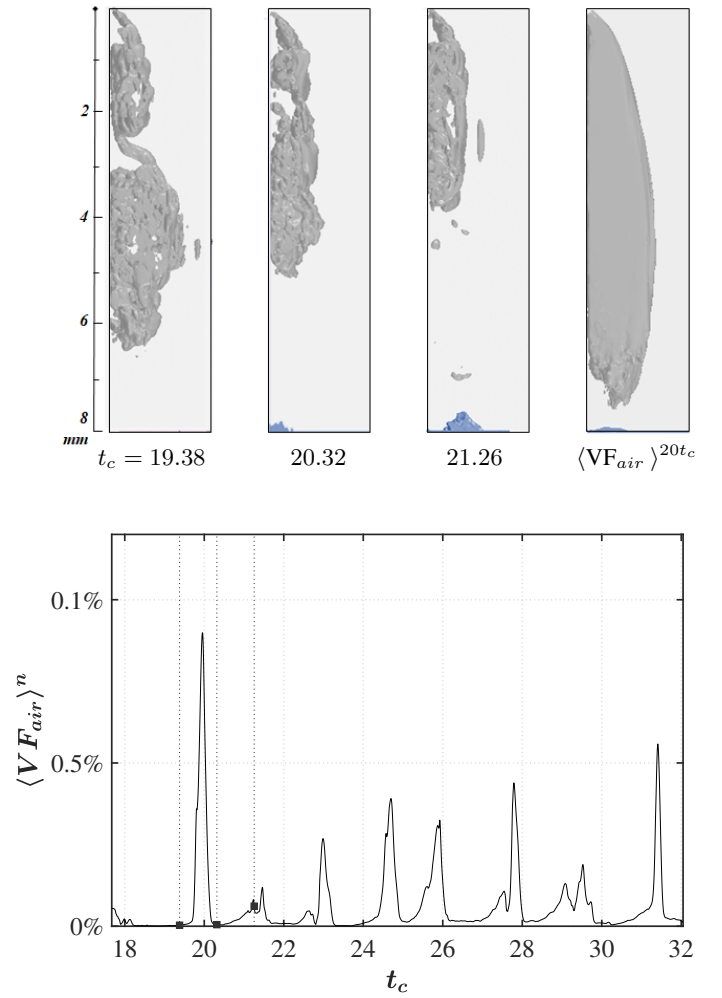


Figure 10: Pentane injection -10°C , 0.22 MPa – transient air entrainment into the nozzle. Instantaneous isosurfaces of 10% VF_{air} superimposed on 1% VF_v and average behaviour over $20 \cdot t_c$.

the continuous cavitation region. The pressure gradients are strong enough to entrain air upstream in the nozzle, close to the sharp edge. This air entrainment mechanism could lead to the formation of deposits inside the nozzle, when the phenomenon is replicated in real-world engine applications.

Although still apparent in the cold fuel injection with $T_{inj} = -10^\circ\text{C}$, the extend of air entrainment is noticeably reduced. The overall VF_{air} in the nozzle is an order of magnitude lower than in the case with the 20°C , 0.22 MPa . Air ingestion and discharge happens sporadically and at timescales of ~ 0.25 to 0.4 ms , with the temporal VF_{air} plot exhibiting sharp peaks at every entrainment event. The difference in the behaviour of the two examined cases can be attributed to the shortened length of the recirculation region and the lower potency of the cavitation phenomenon, which generally impedes the backflow of air.

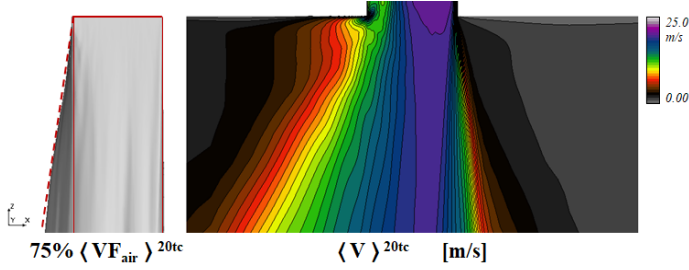
Near-nozzle Jet Angle

Figure 11 provides a visual impression of the effect that cavitation has on the emerging liquid jet. To facilitate the discussion and with reference to [39], 75% isosurfaces of the VF_{air} have been drawn, next to iso-contours of the velocity magnitude inside the discharge region. The temporally-averaged data suggest that the jet appearance on the

side of the continuous cavitation region is noticeably affected by the aforementioned collapse and entrainment mechanisms. An increase of the jet opening angle is observed from 5.4° in the incipient cavitation case (-10°C), to 8.2° on the strongly cavitating simulations (20°C). The effect is also noticeable at the opposite side of the jet, with the -10°C simulation producing a more symmetric spray that spreads equally from both directions, in contrast to 20°C . Experimental studies report values that range between 0.5° and 14° [49], which are in line with the results from the current computational models.

The entrainment of air from the discharge chamber to the inside of the nozzle is also apparent in Fig. 11. The images show that at the near-nozzle region, the surface of the spray is concave on the 20°C case (a byproduct of the air entrainment), whereas it appears to be convex on the latter, -10°C simulation. The transient suction of air introduces disturbances to the overall structure of the jet, leading to the emergence of a noticeable bulge on the spray's volume. As this bulge is convected downstream due to the jet's inertia, it bursts away, leading to higher vorticity; a phenomenon documented by [14, 50]. Particularly for the super-cavitating case where the cavitation region extends all the way to the nozzle outlet plane, the ligaments and droplets produced from this detachment are known to contribute to a further increase of the liquid atomization rates.

• $T_{inj} = 20\text{ }^{\circ}\text{C}$



• $T_{inj} = -10\text{ }^{\circ}\text{C}$

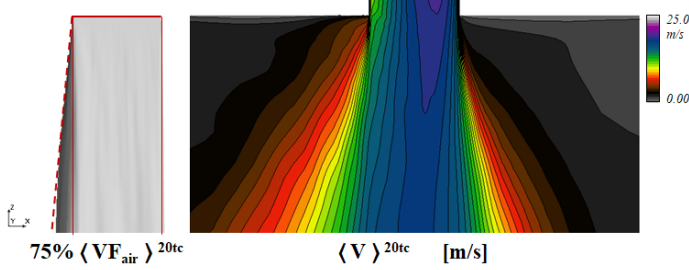


Figure 11: Pentane injection – Fuel jet visualized by the isosurface of the 75% Volume Fraction of air and velocity magnitude inside the discharge region, both averaged over $20 \cdot t_c$.

The velocity field presented next to the time-averaged jet images further supports the above observations. [41] mentions that high velocity fluctuations in the lateral direction induce an upsurge in flow momentum towards the fuel-air interface that creates distortions on the jet surface, a more intense breakup of droplets and a widening in the overall spray angle. Mean radial velocity is noticeably higher on the cavitation side of the highly asymmetric jet in the $20\text{ }^{\circ}\text{C}$ case, where the flow appears to turn around the sharp corner just downstream of the nozzle outlet. The effect of the wake of the continuous cavitation region is also visible in both contour plots, with jet flow normalizing after around $\sim 1 \cdot W_n$ for the $-10\text{ }^{\circ}\text{C}$ and further downstream for $20\text{ }^{\circ}\text{C}$. This is directly related to the length of the recirculation region; in the former case the flow reattaches at 79% of the nozzle length and thus has more space to redevelop, in contrast to the latter case, where the cavitation region stretches to the nozzle outlet area.

The results from this section indicate that development of in-nozzle cavitation introduces a number of both beneficial and adverse effects to the delivery of fuel inside the chamber; the most advantageous being the enhancement of the jet atomization process, which is a key mechanism for achieving sufficient mixing and combustion efficiency [51, 52, 53]. On the other hand, cavitation leads to a reduction of the effective cross-sectional area at the nozzle outlet [54], with the vapor content that is being produced inside the nozzle filling up a significant portion of its volume, thus hindering the efficient delivery of liquid fuel to the chamber [55].

Conclusions

In the present study, numerical simulations of *n*-pentane injections through a reference 3D step nozzle geometry were conducted. The focus has been on modeling injections in a range of temperatures that is relevant to an engine's cold start, with the examined cases spanning across different regimes, from an "incipient (developing) cavitation" ($Ca = 1.15$), to a "super cavitating" flow ($Ca = 0.51$). The

simulations shed light on the effect that cold injections have on the size and the intensity of the emerging cavitation structures.

A comparison of a benchmarking case against reference data from the literature shows that the results have the capacity to replicate the flow structure that is observed inside the nozzle, and that the in-nozzle flow dynamics compare favourably to experimental measurements. This indicates the applicability of the computational model used.

The main observations of this study are summarized below:

- In cold pentane injections, a decrease in temperature yielded a shift in the cavitation regime from "supercavitation" to "incipient cavitation". As T_{inj} decreased from 20 to $-10\text{ }^{\circ}\text{C}$ the time-averaged void fraction $\langle VF_v \rangle^{20tc}$ reduced by 44%, the total length of the cavitation region decreased by 6.2%, while the attached length reduced by 16.2%.
- The dynamic phenomena that develop in the nozzle were shown to vary accordingly, with a power spectral analysis showing an increase in the characteristic frequency with decreasing T_{inj} . This suggested that lower temperatures led to a reduction in both the size and the intensity of the in-nozzle cavitation features.
- Injection pressure showed a less noticeable effect, with a marginally increased production of vapor observed at elevated $p_{inj} = 0.28\text{ MPa}$ (10.9%) from the original 10.5% at 0.22 MPa, and the cavitation region at the upstream nozzle area demonstrating a higher degree of saturation.
- As T_{inj} reduced, an increase in nozzle outlet massflow was observed (up to 11.8% within the examined temperature range). The respective discharge coefficients varied between 0.72 and 0.78. The reduced fuel delivery in the $20\text{ }^{\circ}\text{C}$ cases was attributed to the high overall void fraction and the decreased fuel density, which hindered the delivery of fuel to the downstream chamber.
- An upstream entrainment of air was observed through the nozzle outlet surface. The phenomenon was more pronounced in the supercavitating cases at $20\text{ }^{\circ}\text{C}$, and an enhancement was observed with increased p_{inj} . The mechanism of air entrainment warrants further examination, as it can lead to the accumulation of in-nozzle deposits when this is replicated in real-world internal combustion engine injectors.
- The shedding and subsequent collapse of vapor nuclei caused an upsurge in flow momentum towards the fuel-air interface. The influence on the discharged jet formation became apparent on the side of the continuous cavitation region via an increase of the jet opening angle from 5.4° in the incipient cavitation case ($-10\text{ }^{\circ}\text{C}$), to 8.2° on the strongly cavitating simulations ($20\text{ }^{\circ}\text{C}$). The incipient cavitation simulation also produced a more symmetric jet, which spread equally from both directions.

Research is currently underway to extend the scope of the cold-start study with additional temperature points, beyond the test range reported in this publication. Further investigations have also been planned to assess the influence of temperature on the cavitation behaviour of fuels with different volatility characteristics, including other components of gasoline. The study is also planned to be transferred to real injectors geometries, dimensionally-relevant for modern engine applications, so that more detailed information can be collected relating to potentially influencing parameters, such as the effect of needle motion, the cylindrical shape of real-world injector orifices and the impact from neighbouring nozzles.

References

1. M. Weiss, E. Paffumi, M. Clairotte, Y. Drossinos, T. Vlachos, P. Bonnel, and B. Giechaskiel, "Including cold-start emissions in the real-driving emissions (RDE) test procedure," *Scientific and technical research report, Publications Office of the European Union*, 2017.
2. A. Roberts, R. Brooks, and P. Shipway, "Internal combustion engine cold-start efficiency: A review of the problem, causes and potential solutions," *Energy Conversion and Management*, vol. 82, pp. 327–350, 2014.
3. Bosch, "European Union Emission Standards for Cars and Light Trucks." <http://www.dieselnet.com/standards/eu/ld.php>, 2015.
4. ICCT, "The International Council for Clean Transportation - a technical summary of Euro 6/VI vehicle emission standards," 2016.
5. P. Efthymiou, M. H. Davy, C. P. Garner, G. K. Hargrave, J. E. Rimmer, and D. Richardson, "Insights into cold-start DISI combustion in an optical engine operating at -7°C ," *SAE Technical Paper: 2013-01-1309*, 2013.
6. DECOST, "Final report summary - DECOST (Diesel engine cold start and transient improvement)." https://cordis.europa.eu/result/rcn/171466_en.html, 2015.
7. EC, "European Commission - Regulation 2016/427." Official Journal of the European Union L 82, pp. 1-98, 2016.
8. C. Price, A. Hamzehloo, P. Aleiferis, and D. Richardson, "Numerical modelling of droplet breakup for flash-boiling fuel spray predictions," *International Journal of Multiphase Flow*, vol. 125, p. 103183, 2020.
9. D. Kolokotronis, Y. Hardalupas, A. M. K. P. Taylor, P. Aleiferis, A. Arioka, and M. Saito, "Experimental investigation of cavitation in gasoline injectors," *International Powertrains, Fuels Lubricants Meeting*, 2010.
10. P. Aleiferis, Y. Hardalupas, D. Kolokotronis, A. Taylor, A. Arioka, and M. Saito, "Experimental investigation of the internal flow field of a model gasoline injector using micro-particle image velocimetry," *Powertrain Fluid Systems Conference and Exhibition*, 2006.
11. P. Efthymiou, M. Davy, C. Garner, G. Hargrave, J. Rimmer, D. Richardson, and J. Harris, "An optical investigation of a cold-start DISI engine startup strategy," in *Internal Combustion Engines: Performance, Fuel Economy and Emissions*, pp. 33 – 52, Woodhead Publishing, 2013.
12. P. Dahlander, A. Gutkowski, and I. Denbratt, "Visualization of fuel sprays for stratified cold starts in gasoline direct injection engines," *ILASS08-AI29*, 2008.
13. C. Wang, H. Xu, J. M. Herreros, J. Wang, and R. Cracknell, "Impact of fuel and injection system on particle emissions from a GDI engine," *Applied Energy*, vol. 132, pp. 178–191, 2014.
14. F. Örley, T. Trummer, S. Hickel, M. Mihatsch, S. Schmidt, and N. Adams, "Large-eddy simulation of cavitating nozzle flow and primary jet break-up," *Physics of Fluids*, vol. 27, no. 8, p. 086101, 2015.
15. H. Naseri, P. Koukouvinis, I. Malgarinos, and M. Gavaises, "On viscoelastic cavitating flows: A numerical study," *Physics of Fluids*, vol. 30, no. 3, p. 033102, 2018.
16. M.-G. Mithun, P. Koukouvinis, and M. Gavaises, "Numerical simulation of cavitation and atomization using a fully compressible three-phase model," *Physical Review Fluids*, vol. 3, no. 6, p. 064304, 2018.
17. A. Sou, B. Biçer, and A. Tomiyama, "Numerical simulation of incipient cavitation flow in a nozzle of fuel injector," *Computers & Fluids*, vol. 103, pp. 42–48, 2014.
18. US Secretary of Commerce, "NIST reference fluid thermodynamic and transport properties database (REFPROP) version 9.1f." <https://www.nist.gov/srd/refprop>, version 9.1.
19. H. K. Versteeg and W. Malalasekera, *An introduction to computational fluid dynamics: the finite volume method*. Pearson education, 2007.
20. F. Nicoud and F. Ducros, "Subgrid-scale stress modelling based on the square of the velocity gradient tensor," *Flow, turbulence and Combustion*, vol. 62, no. 3, pp. 183–200, 1999.
21. 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.
22. S. B. Pope, *Turbulent flows*. Cambridge: Cambridge University Press, 2001.
23. E. G. Santos, J. Shi, M. Gavaises, C. Soteriou, M. Winterbourn, and W. Bauer, "Investigation of cavitation and air entrainment during pilot injection in real-size multi-hole diesel nozzles," *Fuel*, vol. 263, p. 116746, 2020.
24. F. R. Menter, "Two-equation eddy-viscosity turbulence models for engineering applications," *AIAA journal*, vol. 32, no. 8, pp. 1598–1605, 1994.
25. 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.
26. M. Darwish and F. Moukalled, "Normalized variable and space formulation methodology for high-resolution schemes," *Numerical Heat Transfer*, vol. 26, no. 1, pp. 79–96, 1994.
27. C. W. Hirt and B. D. Nichols, "Volume of fluid (VOF) method for the dynamics of free boundaries," *Journal of computational physics*, vol. 39, no. 1, pp. 201–225, 1981.
28. S. Muzaferija, "Computation of free surface flows using interface-tracking and interface-capturing methods," *Nonlinear water-wave interaction. Computational Mechanics*, Southampton, 1998.
29. M. Markiewicz, *Nonlinear water wave interaction*, vol. 24. Computational Mechanics, 1999.
30. M. Altimira and L. Fuchs, "Numerical investigation of throttle flow under cavitating conditions," *International Journal of Multiphase Flow*, vol. 75, pp. 124–136, 2015.
31. A. Anderlini, M. Salvetti, A. Agresta, and L. Matteucci, "Stochastic sensitivity analysis of numerical simulations of high-pressure injectors to cavitation modeling parameters," in *ASME 2017 Fluids Engineering Division Summer Meeting*, American Society of Mechanical Engineers Digital Collection, 2017.
32. J. Sauer and G. H. Schnerr, "Unsteady cavitating flow — a new cavitation model based on a modified front capturing method and bubble dynamics," in *Proceedings of 2000 ASME Fluid Engineering Summer Conference*, vol. 251, pp. 1073–1079, 2000.
33. W. Yuan, J. Sauer, and G. H. Schnerr, "Modeling and computation of unsteady cavitation flows in injection nozzles," *Mécanique & industries*, vol. 2, no. 5, pp. 383–394, 2001.
34. G. H. Schnerr and J. Sauer, "Physical and numerical modeling of unsteady cavitation dynamics," in *Fourth international conference on multiphase flow*, vol. 1, ICMF New Orleans, 2001.
35. L. R. OMFRS, "On the pressure developed in a liquid during the collapse of a spherical cavity," *Philosophical Magazine, Series*, vol. 6, pp. 94–98, 1917.
36. C. E. Brennen, *Cavitation and bubble dynamics*. Cambridge University Press, 2014.
37. C. Mauger, L. Méès, M. Michard, A. Azouzi, and S. Valette, "Shadowgraph, schlieren and interferometry in a 2D cavitating channel flow," *Experiments in fluids*, vol. 53, no. 6, pp. 1895–1913, 2012.
38. C. P. Egerer, S. Hickel, S. J. Schmidt, and N. A. Adams, "Large-eddy simulation of turbulent cavitating flow in a micro channel," *Physics of Fluids*, vol. 26, no. 8, p. 085102, 2014.
39. T. Trummer, D. Rahn, S. Schmidt, and N. A. Adams, "Large eddy simulations of cavitating flow in a step nozzle with injection into gas," *Atomization and Sprays*, vol. 28, no. 10, pp. 931–955, 2018.
40. 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.

41. A. Sou, S. Hosokawa, and A. Tomiyama, "Effects of cavitation in a nozzle on liquid jet atomization," *International journal of heat and mass transfer*, vol. 50, no. 17-18, pp. 3575–3582, 2007.
42. P. Aleiferis, J. Serras-Pereira, A. Augoye, T. Davies, R. Cracknell, and D. Richardson, "Effect of fuel temperature on in-nozzle cavitation and spray formation of liquid hydrocarbons and alcohols from a real-size optical injector for direct-injection spark-ignition engines," *International Journal of Heat and Mass Transfer*, vol. 53, no. 21-22, pp. 4588–4606, 2010.
43. A. J. Butcher, P. G. Aleiferis, and D. Richardson, "Development of a real-size optical injector nozzle for studies of cavitation, spray formation and flash-boiling at conditions relevant to direct-injection spark-ignition engines," *International Journal of Engine Research*, vol. 14, no. 6, pp. 557–577, 2013.
44. K. Sato and Y. Saito, "Unstable cavitation behavior in a circular-cylindrical orifice flow," *JSME International Journal Series B Fluids and Thermal Engineering*, vol. 45, no. 3, pp. 638–645, 2002.
45. M. S. Mihatsch, S. J. Schmidt, and N. A. Adams, "Cavitation erosion prediction based on analysis of flow dynamics and impact load spectra," *Physics of Fluids*, vol. 27, no. 10, p. 103302, 2015.
46. B. Beban, S. Schmidt, and N. Adams, "Numerical study of submerged cavitating throttle flows," *Atomization and Sprays*, vol. 27, no. 8, 2017.
47. D. Duke, A. Swantek, A. Kastengren, K. Fezzaa, and C. Powell, "Recent developments in X-ray diagnostics for cavitation," *SAE International Journal of Fuels and Lubricants*, vol. 8, no. 1, pp. 135–146, 2015.
48. D. P. Schmidt and M. Corradini, "The internal flow of diesel fuel injector nozzles: a review," *International Journal of Engine Research*, vol. 2, no. 1, pp. 1–22, 2001.
49. C. Stanley, T. Barber, B. Milton, and G. Rosengarten, "Periodic cavitation shedding in a cylindrical orifice," *Experiments in fluids*, vol. 51, no. 5, pp. 1189–1200, 2011.
50. W. Edelbauer, "Numerical simulation of cavitating injector flow and liquid spray break-up by combination of Eulerian–Eulerian and volume-of-fluid methods," *Computers & Fluids*, vol. 144, pp. 19–33, 2017.
51. W. H. Nurick, "Orifice cavitation and its effect on spray mixing," *Journal of Fluids Engineering*, vol. 98, pp. 681–687, 12 1976.
52. W. Bergwerk, "Flow pattern in diesel nozzle spray holes," *Proceedings of the Institution of Mechanical Engineers*, vol. 173, no. 1, pp. 655–660, 1959.
53. R. Reitz and F. Bracco, "Mechanism of atomization of a liquid jet," *Physics of Fluids*, vol. 25, no. 10, pp. 1730–1742, 1982.
54. R. Payri, S. Molina, F. Salvador, and J. Gimeno, "A study of the relation between nozzle geometry, internal flow and sprays characteristics in diesel fuel injection systems," *KSME International Journal*, vol. 18, no. 7, pp. 1222–1235, 2004.
55. G. M. Bianchi, G. Cantore, and S. Fontanesi, "Turbulence modelling in CFD simulation of ICE intake flows: The discharge coefficient prediction," *SAE 2002 World Congress Exhibition*, pp. 1901–1918, 2002.

Nomenclature

Latin symbols

<i>A</i>	Area	<i>St</i>	Strouhal number
<i>Ca</i>	Cavitation number	<i>t</i>	Time
<i>f</i>	Frequency	<i>T</i>	Temperature
<i>L</i>	Length	<i>V</i>	Velocity
<i>m</i>	Massflow	<i>W</i>	Width
<i>p</i>	Absolute pressure	<i>We</i>	Weber number
<i>Re</i>	Reynolds number		

Greek symbols

μ	Dynamic Viscosity	σ	Surface Tension
ρ	Density	ω	Vorticity

Subscripts

att	Attached region	n	Nozzle
c	Characteristic scale	t,eff	Turbulent effective
cav	Cavitation	tot	Total region
inj	Injection	v	Vapor phase
int	Interface	z	Streamwise
l	Liquid phase	∞	Discharge (farfield)

Acronyms

DISI	Direct-Injection Spark-Ignition	StDev	Standard Deviation
LDV	Laser Doppler Velocimetry	VF	Volume Fraction
LES	Large Eddy Simulations	VOF	Volume of Fluid
PSD	Power Spectral Density	WALE	Wall-Adapting Local-Eddy Viscosity
RANS	Reynolds-Averaged Navier-Stokes		

Contact Information

Author for correspondence: Prof. Pavlos G. Aleiferis
(p.aleiferis@imperial.ac.uk)

Acknowledgments

This work has been supported by the iCase project "Fuel sprays for IC engines" (award ref. no. 1815607), funded by the EPSRC, Shell Global Solutions UK & Imperial College's President PhD Scholarship scheme. Dr. Hamzehloo was supported by the EPSRC grant EP/M009424/1 ("Ultra Efficient Engines and Fuels"). The authors gracefully acknowledge the use of Imperial College London's High Performance Computing Facilities.