



City Research Online

City, University of London Institutional Repository

Citation: Wang, C., Adams, M., Luo, T., Jin, T., Luo, F. & Gavaises, M. (2021). Hole-to-hole variations in coupled flow and spray simulation of a double-layer multi-holes diesel nozzle. *International Journal of Engine Research*, 22(10), pp. 3233-3246. doi: 10.1177/1468087420963986

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/25324/>

Link to published version: <https://doi.org/10.1177/1468087420963986>

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.

City Research Online:

<http://openaccess.city.ac.uk/>

publications@city.ac.uk

39 usage for transportation² in the next two decades. Currently, diesel engines are responsible for ~30wt%
40 of soot and ~17% of man-made CO₂ emissions.³ Despite the immense reduction achieved (>90%
41 relative to 2000 levels), soot is one of the deadliest forms of air pollution: such particles inhaled at
42 city centres, are linked to serious health effects, including premature death, heart attacks and strokes,
43 as well as acute bronchitis and aggravated asthma among children.

44 Due to diesel fuel's strong impact on soot and NO_x emissions, strict combustion emission
45 regulations are imposed to diesel engines. One way of reducing emissions is to improve the injection
46 and atomization characteristics of diesel fuel during engine operation. Fuel injectors are one of the
47 major components of combustion engines as they control fuel delivery, atomisation, mixing and to
48 a large extent the combustion process. Atomisation, in particular, is known to be influenced by the
49 in-nozzle flow. Numerous studies have addressed experimentally and numerically the formation
50 and development of turbulence and cavitation inside fuel injectors for various nozzle designs and
51 their effect on atomization.⁴⁻¹⁰ Despite considerable improvement in instrumentation technology,
52 experimentation of the internal nozzle flow and spray breakup is challenging. Most of the relevant
53 studies focus on scaled-up or simplified designs of real-size nozzles.¹¹ In order to control the
54 duration of fuel injection in a reasonable range and obtain good spray atomization in heavy-duty
55 high-pressure common rail diesel engines where large amounts of fuel per cycle are injected, the
56 number of nozzle orifices are increased while they accommodate smaller hole diameters. However,
57 as the number of orifices increase, the forming spray jets are easily interacting, and thus, limiting
58 the fuel distribution. As a result, the space available for combustion in the engine are not fully
59 utilized, thereby compromising the combustion quality. In addition, modern diesel engines are
60 operated under high injection pressure (> 2500bar) with injectors having small injection hole diameters
61 of 90–120µm; these conditions pose significant difficulties in measuring and/or optically visualising
62 the processes occurring in both the injector nozzle and within the high pressure/temperature
63 combustion chamber. The majority of transparent real-size nozzle investigations have been performed
64 in simplified single-hole geometries that generally confirm the presence of geometric-induced
65 cavitation.¹²⁻¹⁵ Still, quantification of the liquid volume fraction and differentiation between the
66 vapour and gaseous cavitation is an open question. On the contrary, numerical simulations can
67 provide insight regarding the flow dynamics at a resolution that cannot be obtained with today's
68 experimental techniques. Some of the most recent work summarizing the relevant modelling
69 approaches can be found in ¹⁶⁻¹⁸.

70 Using customized test rigs featuring transparent nozzles, injection rate measurements, cavitation
71 and spray visualization techniques, researchers were able to investigate the internal flow and the
72 subsequent spray development from various viewing angles. The Bosch measuring method¹⁹ and
73 the EFS mono-injection flow meter have been used to measure the injection rates regardless of the
74 orifices numbers. This compromises the accuracy of the results since the influence of the individual
75 nozzle holes on injection rate and spray formation are different (due to different hydraulic
76 conditions). Although cavitation visualization provides detailed information regarding the
77 cavitating flow within nozzles, it has only be used at injection pressures up to 1000bar with actual
78 injector geometries, because the materials used to manufacture such nozzles cannot withstand higher
79 pressures. However, both macroscopic and microscopic spray characteristics have been obtained
80 using methods such as X-ray imaging techniques ^{10,20}, Particle image velocimetry, Phase Doppler
81 Anemometry and chemiluminescence apparatus.²¹⁻²³

82

83 Given the limited quantitative information around the flow structure inside diesel injectors, fuel
84 injection equipment manufacturers require robust predictive Computational Fluid Dynamics (CFD)
85 tools, in order to understand the physical mechanisms taking place during injection. From a physical
86 viewpoint, modelling of such flow conditions requires the fluid compressibility²⁴, mass transfer
87 (cavitation, flash boiling, evaporation^{25,26}) and heat transfer²⁷⁻²⁹ to be taken into account, which
88 increase the complexity as well as the computational cost of the simulations. Additionally, the fluid
89 dynamics processes occur at high Reynolds number and therefore accounting for the effect of turbulent
90 structures and vortex dynamics, is key in explaining how the injected fuel spray is formed³⁰⁻³⁴; this can
91 only be resolved using very fine computational grids and scale resolving simulations, such as Large
92 Eddy Simulation (LES), as initially presented in³⁵ for nozzle flows. Many different models have
93 been developed for modelling cavitation; widely utilised approaches include the heterogeneous
94 ‘multi-fluid’ model, the homogeneous ‘mixture’ model and the ‘single-fluid’ model. The multi-fluid
95 approach can model non-equilibrium conditions between the phases i.e. each phase can have a
96 different temperature, pressure and velocity.^{36,37} The interaction between the phases is modelled
97 using interphase exchange terms. In ‘homogeneous’ approaches, the slip velocity between the
98 phases is neglected; this can be justified by the fact that even in the most extreme cases, the relative
99 velocity between the two phases does not exceed 10% of the local velocity magnitude and only in
100 very localised areas. The most widely utilised mixture approaches employ a transport equation for
101 the mass/volume fraction of the secondary phase. In this type of models, the phase-change rate is
102 controlled using a source term which is typically derived from the Rayleigh-Plesset (R-P) equation,
103 as shown in³⁸⁻⁴¹. A detailed review of such models can be found in⁴² and⁴³. The single-fluid
104 approach for modelling cavitation uses an equation of state (EoS), which relates density and speed
105 of sound with pressure and temperature.

106

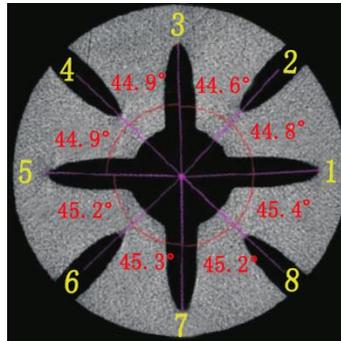
107 Although the internal flow and spray characteristics of diesel injectors has been investigated to some
108 extent (from literature), the complex relationship that governs the transition between the internal
109 flow and spray development is yet to be fully understood. Furthermore, to the best of the author’s
110 knowledge, computational analysis that studies the injection and spray characteristics from
111 asymmetric nozzle holes (taking the effect of each nozzle hole concurrently) hasn’t been researched
112 extensively. Therefore, in this study, orifice-to-orifice variations in injection rate and spray
113 development from a double-layer mini-SAC nozzle are investigated. A customized spray
114 momentum flux experimental test rig was used to obtain the injection rates from each nozzle hole
115 simultaneously, whereas for the spray, a customized test bench was used. For the simulations,
116 independent computational analysis was conducted for the internal nozzle flow utilizing an
117 homogenous mixture cavitation model while the spatial and temporal evolution of the flow from
118 those simulations has been used an initial condition to an Eulerian-Lagrangian spray model
119 resolving the subsequent spray development; the latter has been validated against the obtained
120 experiments and has been further used to elucidate on the effect of nozzle flow on hole-to-hole spray
121 variations.

122 **2. Modelling**

123 **2.1 Geometry model**

124

125 An eight-hole double layer Diesel injector used with heavy-duty vehicles has been utilised.
 126 Information for the actual nozzle geometry was obtained through X-ray Synchrotron radiation
 127 tomography technique. A sample of the obtained data for the geometry of the nozzle is shown in
 128 Fig. 1.

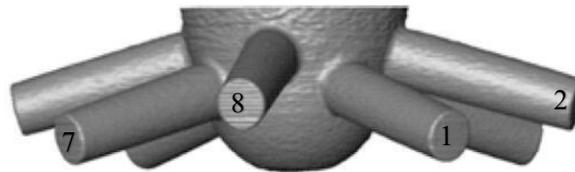


129
 130 Fig.1 Cross-section image of the injection nozzle

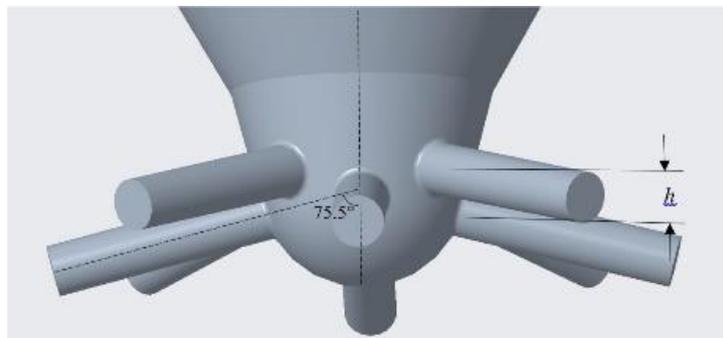
131 The detailed geometric parameters of the nozzle including the hole lengths, inlet rounding corners,
 132 inlet and outlet diameters are shown in Table 1. All the nozzle holes are inclined with an angle of
 133 75.5° as seen in Fig.2 (a) and (b); the inner shape of the needle tip is presented in Fig.2 (c). The
 134 lower layer holes are indicated as 1, 3, 5 and 7 and the upper layered hole as 2, 4, 6 and 8. The mean
 135 mass flow rates of the injector is 38.6 g/s at the working condition of 140MPa. The gap h between
 136 the upper layer and the lower layer nozzle holes is 0.12 mm. To ensure reliability of the results, the
 137 exact replica of the nozzle was numerically reconstructed from the X-ray images, taking into
 138 account all the disparities between holes.

139 Table 1 Geometric parameters

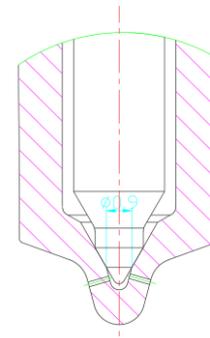
Nozzle holes	$D_{in}/\mu m$	$D_{out}/\mu m$	$r/\mu m$	L/mm
Lower 1,3,5,7	180.2	180.2	31	0.65
Upper 2,4,6,8	180.1	180.2	32	0.65



140
 141 (a) schematic diagram obtained from X-ray technique



142
 143 (b) the whole geometric model



144
 145 (c) 2D diagram showing the shape of the needle and the needle body

Fig.2 3D model of the injection nozzle

146

147 2.2 Coupled two-stage simulation approach

148

149 The coupled method is divided into two parts: the multiphase flow simulation within the injector
150 and the spray jet simulation from the nozzle exit domain. Reynolds-Averaged Navier-Stokes (RANS)
151 equations with the k-zeta-f turbulence model, where adopted in simulating spray development. The
152 spray plume disintegration and development were computed using the models described recently by
153 ⁴⁴. As an interface between the two simulations, the internal flow characteristics were used as the
154 inlet boundary conditions for the subsequent spray simulation.

155

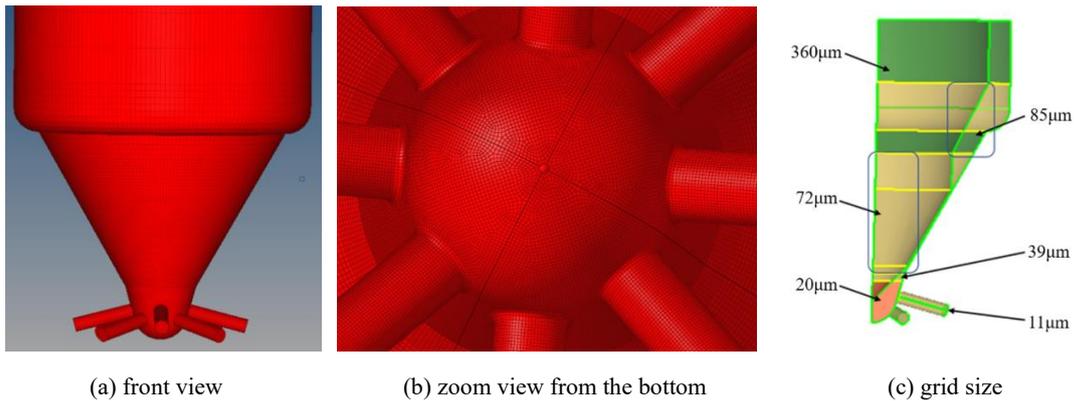
156 In order to achieve this, the calculated flow parameters representing the internal flow characteristics
157 at the nozzle outlet (pressure, velocity field, vapour volume fraction and turbulent kinetic energy
158 and its dissipation rate), were mapped on the grid cells and used as boundary conditions for the
159 subsequent spray simulations.

160

161 2.3 Mathematical model

162 2.3.1 Nozzle modelling

163



164

165 (a) front view

166

(b) zoom view from the bottom

(c) grid size

Fig.3 Diagram of computational grid

167 The whole nozzle model was discretized into ~400,000 hexahedral cells; numerical tests indicated
168 that the mass flow rate was grid independent. In order to capture the transient cavitating flow within
169 the nozzle, the mesh resolution was increased in critical areas, such as nozzle hole inlet and the
170 needle seat, as it can be seen in Fig.3 (c). With the mean fuel flow velocity being around 550 m/s,
171 an estimation of the Taylor length scale yielded around 2.1µm. In other words, the total mesh size
172 that would be required for LES is ~10 million cells, while much smaller time steps will be needed;
173 the difference in CPU time between LES and URANS adopted in this study is approximately 3
174 orders of magnitude, justifying the use of URANS. The inlet pressure, outlet pressure and
175 temperature were set as 140 MPa, 2 MPa and 293.15 K respectively. The thermodynamic properties,
176 of commercial B0 diesel fuel, assumed to be fixed, are listed in Table 2.

177

178

Table 2 Thermodynamic properties

Fuel Properties	Value
Density [kg/m ³]	830

Viscosity [mPa·s]	2.36
Saturation vapor pressure [Pa]	5540
Diesel vapor density [kg/m ³]	0.029
Diesel vapor viscosity [mPa·s]	1.8x10 ⁻³

179

180 Fig.4 shows the assumed transient needle moving applied in the simulation. The needle movement
 181 was considered only in the vertical direction and it was represented by a cell-based mesh
 182 deformation method to ensure mass conservation; its possible eccentric movement was neglected as
 183 this is not known. The number of layer cells in the gap between the needle and the needle seat was
 184 7, while the initial needle lift was set as 0.01 mm; the lift at full needle valve opening was 0.35 mm.

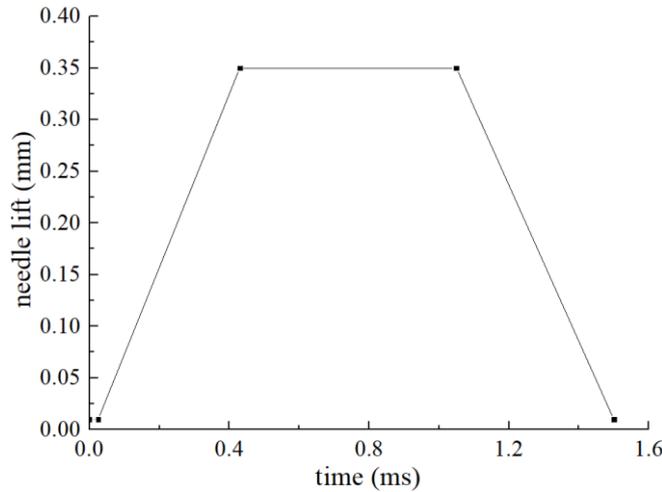


Fig.4 Needle lift curve

185

186

187

188 For the modelling of the internal fuel flow the Navier Stokes equations have been numerically
 189 solved, utilizing the commercial code AVL Fire. The pressure-based SIMPLE algorithm was used
 190 to couple the velocity and pressure fields. The in-nozzle flow simulation were governed by the mass
 191 (1), momentum (2) and energy conservation equations ^{45,46}.

192

$$\frac{\partial \alpha_k \rho_k}{\partial t} + \nabla \cdot \alpha_k \rho_k v_k = \sum_{l=1, l \neq k}^2 \Gamma_{kl} \quad (1)$$

$$\begin{aligned} \frac{\partial \alpha_k \rho_k v_k}{\partial t} + \nabla \cdot \alpha_k \rho_k v_k v_k \\ = -\alpha_k \nabla p + \nabla \cdot \alpha_k (\tau_k + T_k^t) + \alpha_k \rho_k F + \sum_{l=1, l \neq k}^2 M_{kl} + V_k \sum_{l=1, l \neq k}^2 \Gamma_{kl} \end{aligned} \quad (2)$$

$$\begin{aligned} \frac{\partial \alpha_k \rho_k v_k h_k}{\partial t} + \nabla \cdot \alpha_k \rho_k v_k h_k \\ = \nabla \cdot \alpha_k (q_k + q_k^t) + \alpha_k \rho_k q_k''' + \alpha_k \rho_k \cdot v_k + \nabla \cdot \alpha_k (\tau_k + \tau_k^t) \cdot v_k \\ + \alpha_k \frac{\partial p}{\partial t} + \sum_{l=1, l \neq k}^2 H_{kl} + h_k \sum_{l=1, l \neq k}^2 \Gamma_{kl} \end{aligned} \quad (3)$$

193

194 However, the volume fraction expression in Eq. 4 has to be satisfied

195

$$\sum_{k=1}^2 \alpha_k = 1 \quad (4)$$

196

197 k represents the phase k, i.e. 1 for gas phase while 2 for liquid phase. α_k is the volume fraction of
198 phase k, ρ_k is phase k density, v_k is phase k velocity, Γ_{kl} is the interfacial mass exchange between
199 phases k and l, T_k^t is phase k Reynolds stress, and M_{kl} is the momentum interfacial interaction
200 between phases k and l.⁴⁷

201

202 The boundary conditions of the nozzle inlet and all orifices outlets were set as pressure boundary
203 conditions, in order to capture the dynamic effects of cavitation phenomenon within nozzles. The
204 interfacial exchanges in terms of momentum and mass within the fluids were computed with a drag
205 model⁴⁴ and a linear mass exchange model considering cavitation⁴⁷ respectively. More specifically,
206 the mass interfacial exchange was achieved through the linearized Rayleigh's cavitation model,
207 derived from linearizing Equation 5 and 6 below:

208

$$\Gamma_{21} = \rho_2 N''' 4\pi R^2 \dot{R} = -\Gamma_{12} \quad (5)$$

$$\Gamma_{21} = \frac{1}{c_{CR}} \text{sign}(\Delta p) 3.95 \frac{\rho_1}{\sqrt{\rho_2}} N'''^{\frac{1}{3}} \alpha_1^{\frac{2}{3}} |\Delta p|^{\frac{1}{2}} = -\Gamma_{12} \quad (6)$$

211

212 where Δp , R and N''' and are the effective pressure differences, bubble radius and the bubble number
213 density. The bubble radius time derivative from the Rayleigh's equation was performed with the
214 expression

215

$$R\ddot{R} + \frac{3}{2}\dot{R}^2 = \frac{\Delta p}{\rho_2} \quad (7)$$

217

218 The momentum interfacial exchange (M_{kl}) was modeled with the equation

$$M_{kl} = C_D \frac{1}{8} \rho_k A_i''' |v_r| v_r + C_{TD} \rho_k k_k \nabla \alpha_l = -M_{lk} \quad (8)$$

220

221 Where C_D is the drag coefficient of the liquid droplets, A_i''' is the interfacial area density, v_r is
222 the relative velocity and C_{TD} is the turbulence dispersion coefficient.⁴⁸

223

224 The 4-equations k-zeta-f turbulent model, developed from the k- ϵ two-equation model, was adopted
225 for capturing the turbulence phenomenon within the two-phase flow. The k-zeta-f model replicates
226 turbulence and its interactions more accurately and with much more stability than the popular k- ϵ
227 model⁴⁹; however, it requires longer computation time. The basic expressions of the model are the
228 turbulent kinetic energy (Eq. 9), its dissipation rate (Eq.10), the velocity scale (Eq. 11) and the
229 elliptical function (Eq. 12).

230

$$\begin{aligned} & \frac{\partial \alpha_k \rho_k k_k}{\partial t} + \nabla \cdot \alpha_k \rho_k v_k k_k \\ & = \nabla \cdot \alpha_k \left(\mu_k + \frac{\mu_k^t}{\sigma_k} \right) \nabla k_k + \alpha_k P_k - \alpha_k \rho_k \varepsilon_k + \sum_{l=1, l \neq k}^2 K_{kl} + k_k \sum_{l=1, l \neq k}^2 \Gamma_{kl} \end{aligned} \quad (9)$$

$$\begin{aligned} & \frac{\partial \alpha_k \rho_k \varepsilon_k}{\partial t} + \nabla \cdot \alpha_k \rho_k v_k \varepsilon_k \\ & = \nabla \cdot \alpha_k \left(\mu_k + \frac{\mu_k^t}{\sigma_k} \right) \nabla \varepsilon_k + \sum_{l=1, l \neq k}^2 D_{kl} + \varepsilon_k \sum_{l=1, l \neq k}^2 \Gamma_{kl} + \alpha_k C_1 P_k \frac{\varepsilon_k}{k_k} \\ & \quad - \alpha_k C_2 \rho_k \frac{\varepsilon_k^2}{k_k} + \alpha_k C_4 \rho_k \varepsilon_k \nabla \cdot v_k \end{aligned} \quad (10)$$

$$\frac{\partial \alpha_k \rho_k \zeta_k}{\partial t} + \nabla \cdot \alpha_k \rho_k v_k \zeta_k = \nabla \cdot \alpha_k \left(\mu_k + \frac{\mu_k^t}{\sigma_k} \right) \nabla \zeta_k + \zeta_k \sum_{l=1, l \neq k}^2 \Gamma_{kl} - \alpha_k P_k \frac{\zeta_k}{k_k} + \alpha_k f_k \quad (11)$$

$$f_k = L_k^2 \nabla^2 f_k - \frac{1}{T_k} \left(C_1 - 1 + C_2 \frac{P_k}{\varepsilon_k} \right) \left(\zeta_k - \frac{2}{3} \right) \quad (12)$$

231

232 where, k_k is the turbulence kinetic energy at phase k, ε_k is the diffusivity of the turbulence kinetic
 233 energy at phase k, ζ_k is the velocity scales ratio at phase k, f_k is the elliptic function at phase k,
 234 P_k is the production term of the turbulence kinetic energy due to shear and $P_{B,k}$ is the generation
 235 component of the turbulence kinetic energy caused by buoyancy. The Prandtl number for the
 236 turbulence kinetic energy is σ_k , K_{k1} is the component of transmission between phases k and l, σ_ε
 237 is the Prandtl number for the ε equation and C_1, C_2, C_3, C_4 are constants. D_{k1} is the interfacial e
 238 ded with hybrid wall equations. The turbulence for the near-wall regions were modeled
 239 notwithstanding wall equations. Further analyzing of the equations are presented in ⁴⁷.

240

241 The spatial discretization of the momentum equation was performed by a bounded central
 242 differential second-order scheme, while the discretization of the continuity equation was achieved
 243 with the MINMOD scheme. The blending factor was set as 0.5 for momentum discretization in
 244 order to achieve a compromise between computational accuracy and convergence. Time
 245 advancement was performed with a second-order backward differencing scheme, in order to capture
 246 the complex turbulence structures within the nozzle.^{44,50,51} Because of the different resolution in
 247 time and space, the time step interval was set as 1×10^{-6} s, which took the whole simulation 360
 248 CPU·h using 24 processors.

249

250 2.3.2 Spray modelling

251

252 For the spray simulation, the injector was located in the top middle of the spray chamber model,
 253 which was built as a cylinder with a length of 0.04 m and a diameter of 0.08 m. The chamber was
 254 discretized into ~2million cells. The simulation was implemented in a non-evaporating condition at
 255 the same temperature and pressure settings as those from experiment.

256

257 At the exit of orifices the local distribution parameters of the flow field variables, such as the

258 turbulent kinetic energy and the vapor distribution, were captured and used as the inlet boundary
 259 conditions for the subsequent spray simulation. The 3D results of the flow field variables at the time
 260 step of 1.25ms are as shown in Fig. 5. In figure, distinct differences can be observed across the
 261 interface. These differences keep changing and therefore influence the breakup behavior and
 262 penetration of the spray jets from the various orifices.
 263

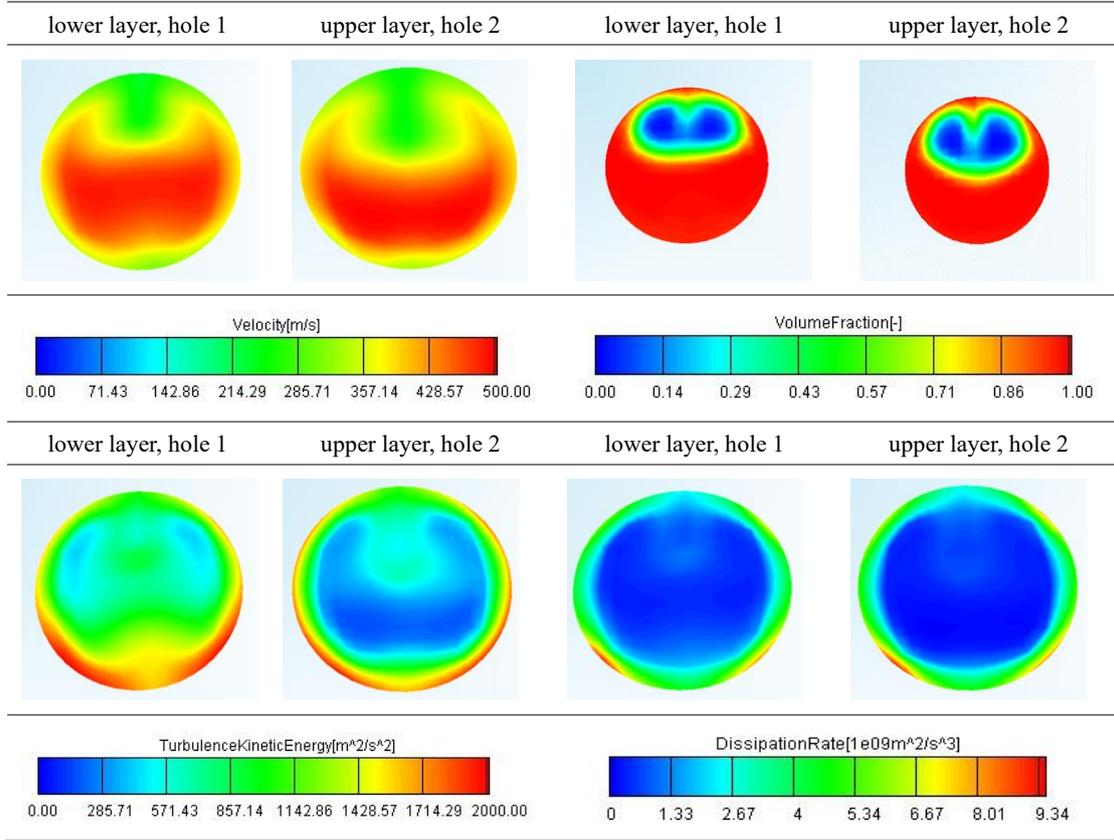


Fig.5 Flow characteristics at the exit of orifices at 1.25ms

264

265

266 *Primary breakup*

267 The blob injection model was selected for the primary breakup model in this study because it could
 268 couple the upstream internal flow characteristics to the downstream spray simulation⁵². The model
 269 considers that the fragment of droplets is dominated by the competitive process between the
 270 turbulence caused by cavitation and the aerodynamic-induced breakup.

271

272 Under the aerodynamic breakup mechanism, the breakup of the liquid core was modeled with:

273

$$\left(\frac{dr}{dt}\right)_a = R_a = -\frac{(r - r_a)}{C_2 \cdot \tau_a} \quad (13)$$

274

275 where r is the actual droplet radius, C_1 and C_2 are constants used for adjusting breakup time and
 276 the characteristic droplet radius, r_a is the characteristic droplet radius, τ_a is the breakup time and
 277 Λ is the dominant aerodynamic wavelength. The subsequent breakup rate of droplets with regards
 278 to turbulent length scale (r_T) was modeled with:

279

$$\frac{dR}{dt} = -\frac{r - C_3 r_T}{C_4 \tilde{\tau}_T} \quad (14)$$

280 where

$$r_T = C_\mu^{0.75} \frac{k^{1.5}}{\varepsilon} \quad (15)$$

$$\tau_T = C_\mu \frac{k}{\varepsilon} \quad (16)$$

281

282 C_μ and C_4 are model constants and $\tilde{\tau}_T = k/\varepsilon$.

283

284 Under the turbulence and cavitation breakup mechanism, the geometric and flow dynamic properties
 285 of the orifices provides the relevant local parameters. This ensures that the transient conditions of
 286 the cavitating flow were captured together with their influence on droplet breakup. By negligible
 287 diffusion effects, the expressions for the induced turbulence in the liquid fuel core are:

288

$$\frac{dk}{dt} = -\varepsilon + S_k \quad (17)$$

$$\frac{d\varepsilon}{dt} = -C \cdot \frac{\varepsilon}{k} \cdot (\varepsilon - S_k) \quad (18)$$

289

290 where S_k is the cavitation source term and C is a constant.

291

292 *Secondary Break up*

293 Secondary breakup of droplets occurs when the aerodynamic breakup mechanism dominates the
 294 turbulent-induced and cavitation breakup mechanism.⁴⁸ For a high pressure diesel engine, the KH-
 295 RT model has been shown to give more accurate results than WAVE and TAB models, hence it was
 296 adopted during this study. The values of the constant for this study has been listed in the Table 3.⁵³

297

298

Table 3 Constant settings of KH-RT model

Model constants	Value
C1	0.61
C2	18
C3	30
C4	2.5
C5	1
C6	0.3
C7	0.03
C8	0.188

299

300 **2.3.3 Mesh sensitive analysis**

301

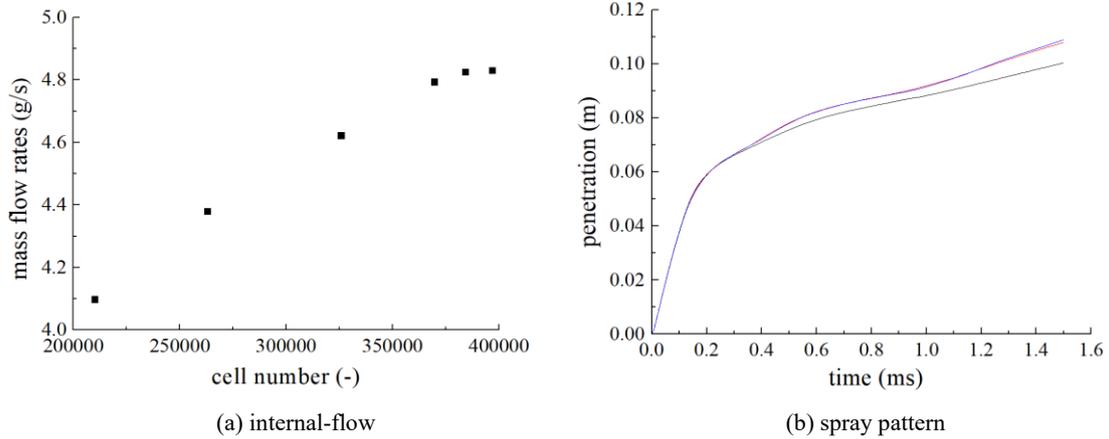


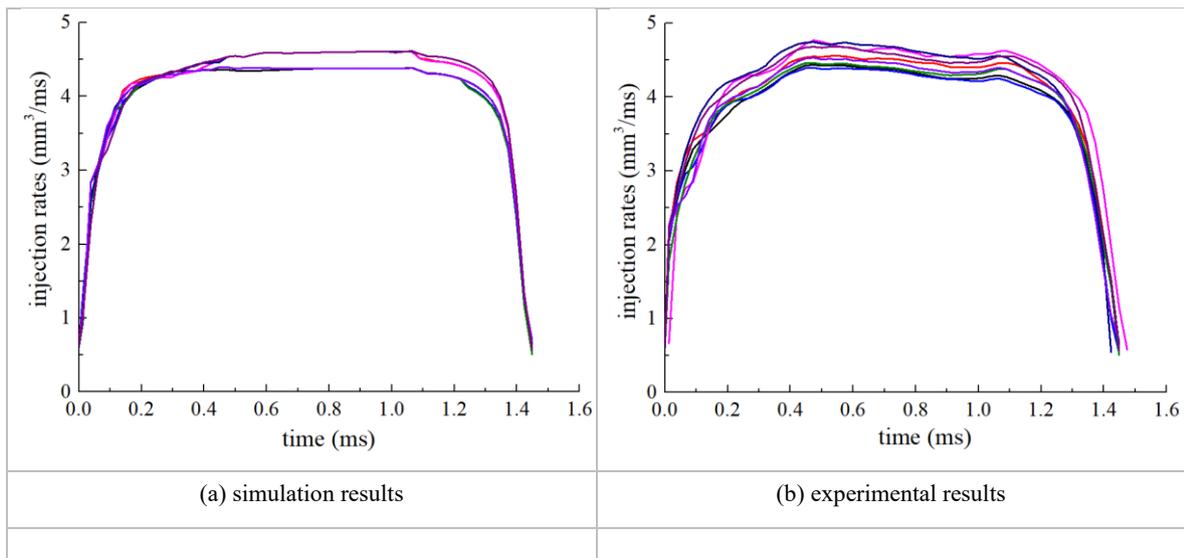
Fig.6 Mesh sensitive analysis for flow and spray domains

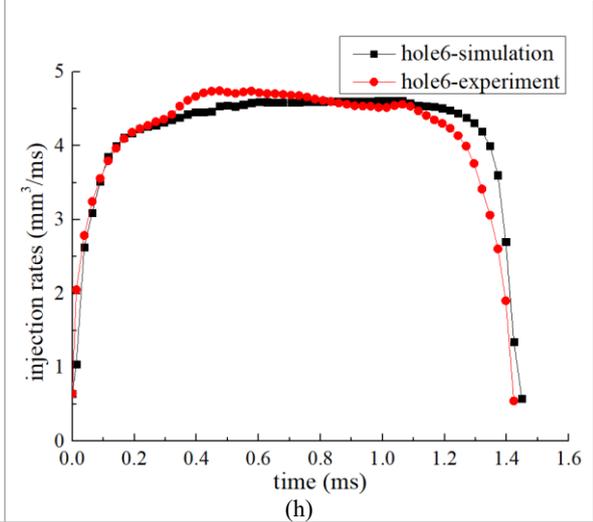
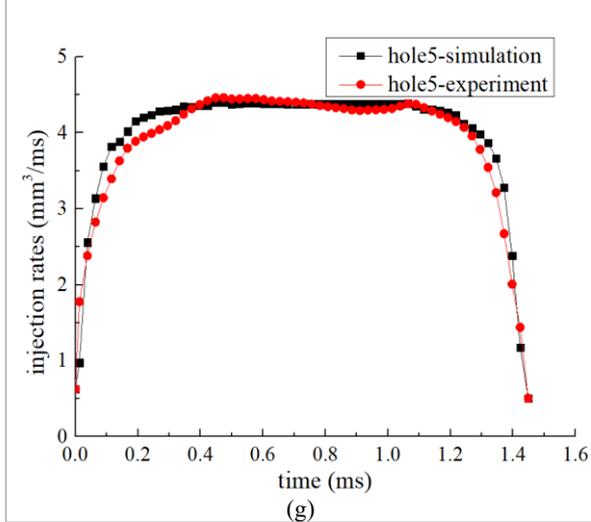
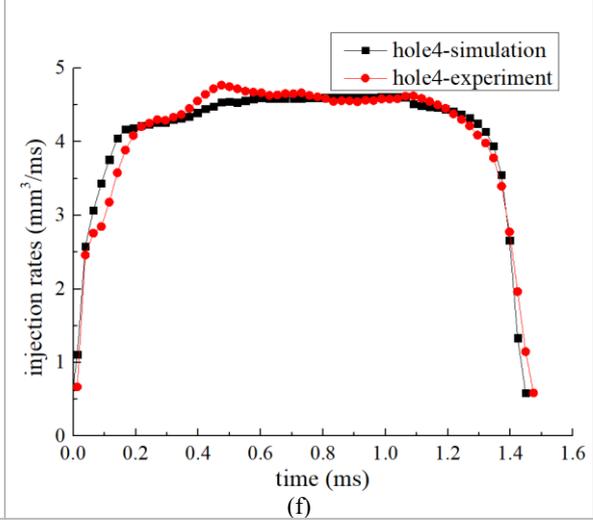
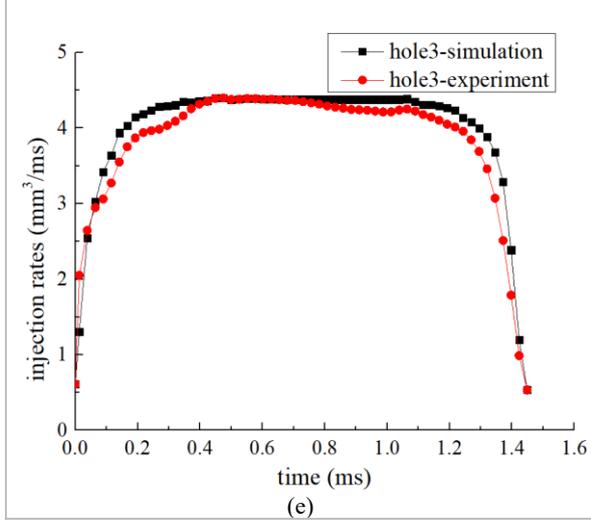
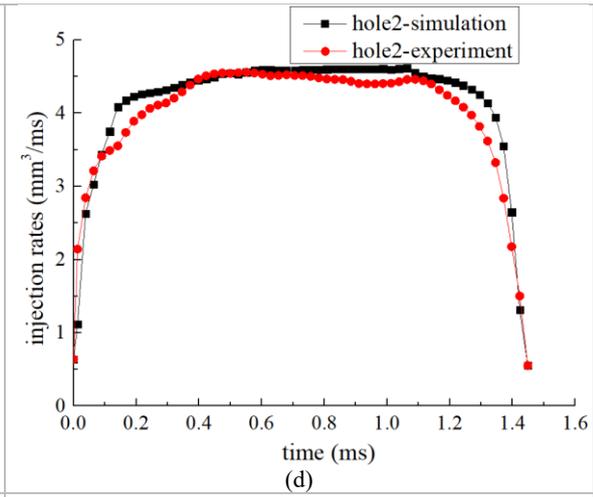
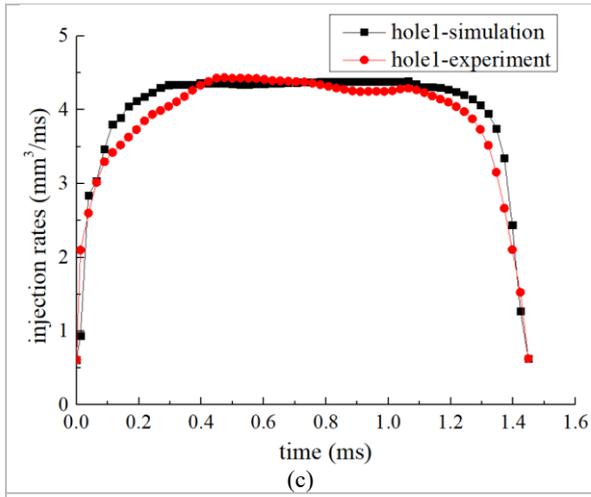
To ensure the simulation results of flow and spray domain are independence of the mesh size, various tests have been performed as shown in Fig. 6 (a) and (b). It can be seen from these figures that grid convergence was attained for the internal flow at around 400,000 cells, while for the spray domain, convergence was attained with about 2 million cells.

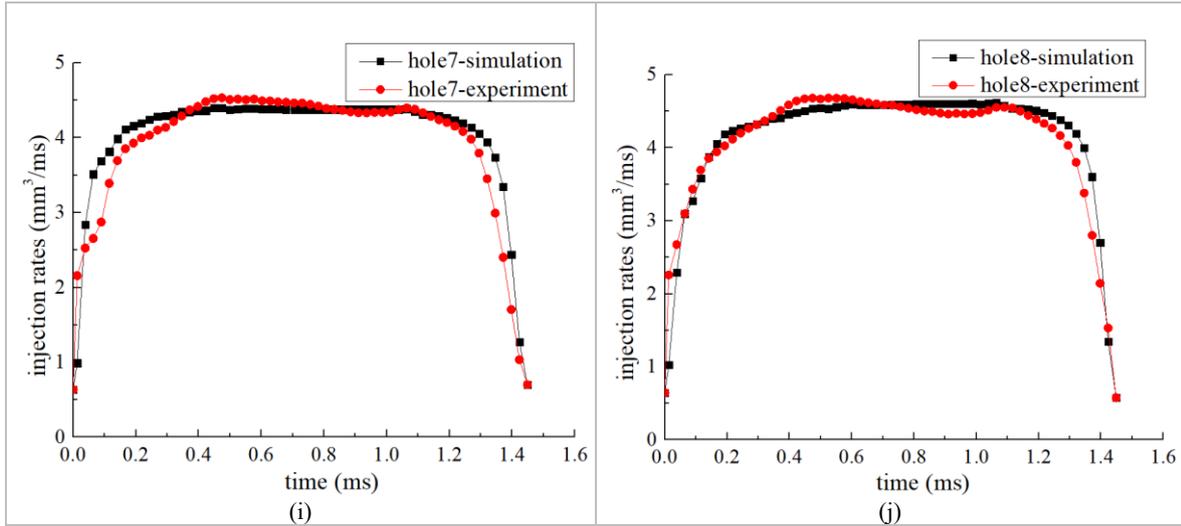
3. Experiment Establishment and validation

3.1 Internal flow characteristics

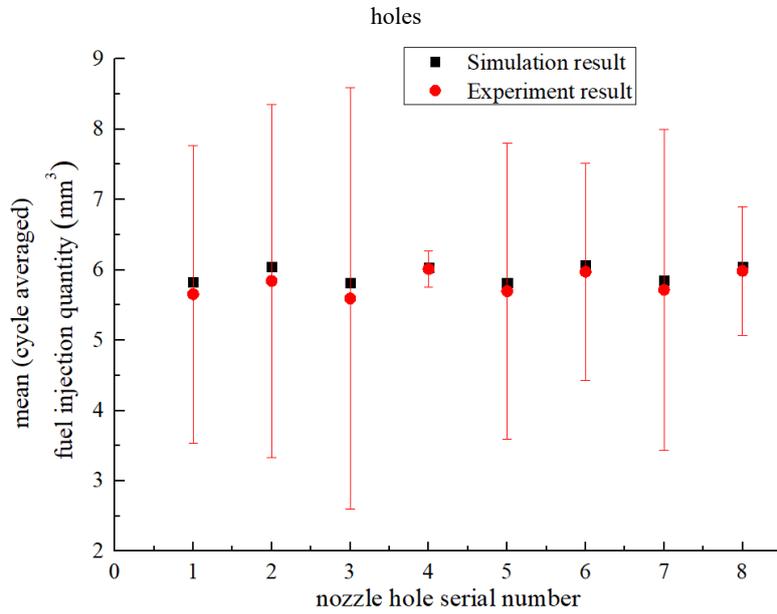
The measurement of injection rates among each nozzle holes were conducted on a customized test rig based on the spray momentum flux. Detailed information about the test method and the test bench are presented in the ⁵⁴⁻⁵⁶. The experiment was conducted with the injection pressure of 140MPa and back pressure of 2MPa. Validation was carried out by comparing experimental and simulation injection rates at the exit of each orifice; the comparisons are presented in Fig. 7.







319 Fig.7 Comparison between simulation and experimental results for the injection rate from individual injection
 320



321
 322 Fig.8 Comparison of hole-to-hole injection quantities and their standard deviation over injection cycles
 323

324 Fig.7 and Fig.8 show the comparison between the simulation and experiment results at the injection
 325 pressure of 140MPa with 2MPa back pressure. From Fig.7, similar trends were present in both
 326 measured and simulated hole-to-hole injection rates. To compute the deviation (error) between the
 327 computational and experimental results, an expression in equation (17) was used:

$$328 \quad \Delta_{hole} = \frac{q_{simulation} - q_{measure}}{q_{measure}} \times 100\% \quad (17)$$

329 Fig.8 shows that, the largest relative error of cycle fuel injection quantities between the simulation
 330 and experiment is less than 3% (at orifice 3). This means that the computational model's accuracy
 331 is within acceptable limits.

332

333 In addition, to quantify the relative discrepancy in cycle fuel injection quantity between the upper
 334 and the lower layer nozzle holes, equation (18) was introduced:

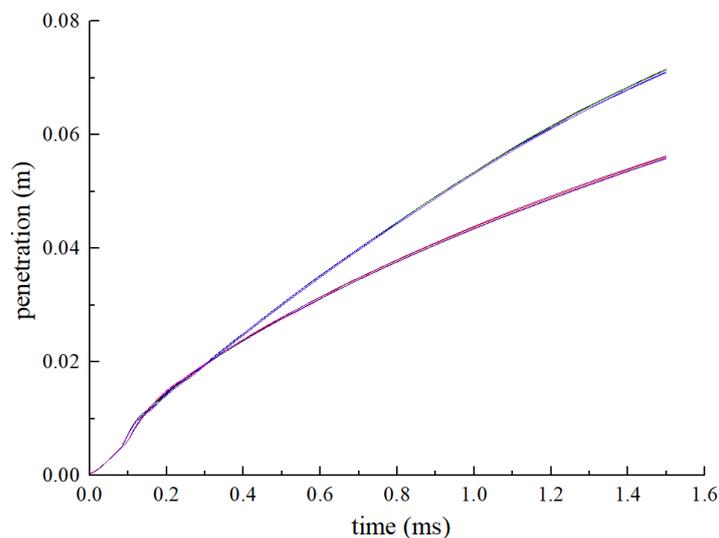
$$335 \quad \Delta = \frac{Q_{lower} - Q_{upper}}{Q_{lower}} \times 100\% \quad (18)$$

336 where Q_{lower} is the total injection quantity of the lower layered holes, Q_{upper} is the total injection
337 quantity of the upper layer holes. The result show that the fuel injection quantities of the lower layer
338 orifices are 6-13% higher than the upper layer orifices. This can be attributed to the smaller flow
339 resistance the fuel experiences as a result of the fuel's gentle entrance into the lower layer orifices.
340

341 3.2 Spray patterns

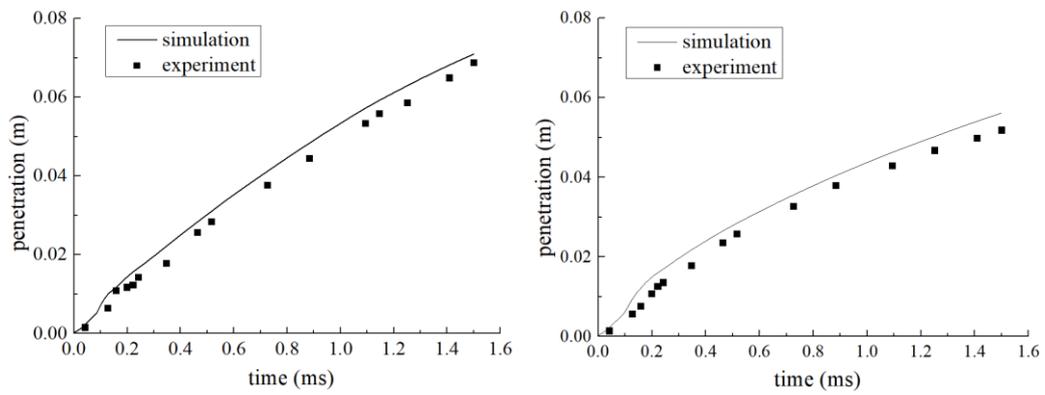
342
343 EFS8400 spray test bench, a constant volume chamber, high speed CCD camera system and the
344 common-rail fuel injection system-BOSCH MOEHWALD-CA4000, were used to acquire spray
345 images and spray jet characteristics through Schlieren method at different injection pressures.⁵⁷
346 Synchronization trigger was adopted to synchronize the working process of the CCD cameras and
347 the fuel injection system. Ambient pressure in the constant volume chamber was provided by stable
348 nitrogen. The maximum pressure of the chamber is 5.2 MPa with a temperature of $293.15 \pm 2K$. Two
349 high speed CCD cameras were installed at the side and the bottom of the constant volume chamber
350 respectively, to photograph the spray shadow through the quartz window. Details on the spray test
351 experimental platform could be found in ⁵⁷, the injection duration was set at 1.5ms.
352 . .

353 Fig.9 shows the simulation results of individual jet penetration for the 8-hole injector at 140 MPa
354 injection pressure and a back pressure of 2MPa (same with internal flow simulation). It is obvious
355 from the figure that, the penetration of the spray jets from the lower layer orifices (1,3,5,7) is much
356 faster than those from the upper layer orifices (2,4,6,8). The difference in spray penetration between
357 the two layers could reach 30% or more. However, the differences in their respective injection rates
358 is between 4% to 8%, as shown in Fig.7.
359



360
361 Fig.9 Simulated penetration results of individual spray plumes
362

363 Although there are some discrepancies in injection rates among the eight nozzle offices, the jet
364 penetration results of the two layered holes (i.e. penetrations of hole 1,3,5,7 and hole 2,4,6,8
365 respectively, as shown in Fig.9) showed acceptable levels of consistency with marginal differences.
366 Therefore, hole 1 from the lower layer and hole 2 from the upper layer were compared with
367 experimental data for validation.



369

370

371

372

(a) hole 1

(b) hole 2

Fig.10 Comparison of hole-to-hole spray penetration

373

374

375

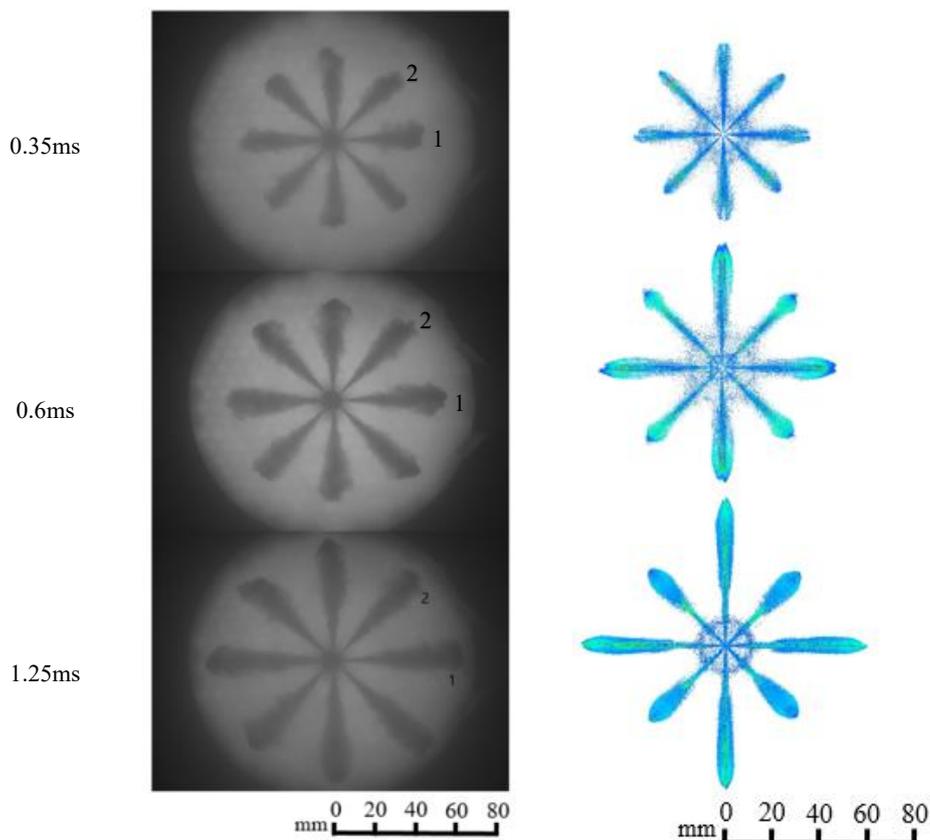
376

377

378

379

Fig. 10 presents the divergence between the experiment and simulated spray jet penetration results. The numerical results showed good consistency against the experimental results, even though they are slightly higher in magnitude. Also, the deviation between the two is larger during the initial stages of injection, and then gradually reduces at longer penetration distances; the deviations are all within acceptable limits (10 %). It should be mentioned that constant inlet boundary condition was set for the internal flow simulation instead of the varying conditions that pertains in reality.



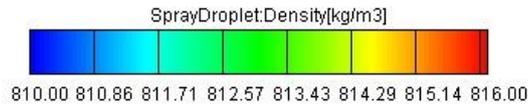


Fig.11 Contrast image of experiment and simulation

380

381

382 The experimental and numerical 3D spray images (at the inlet pressure of 140 MPa with a back
383 pressure of 2 MPa) are shown in Fig. 11. Hole 1 and hole 2 are marked in the experimental images.

384 The others follow the same numbering pattern in the counterclockwise direction. From the images,
385 it is clear that the spray development from simulation and experiment followed similar patterns.

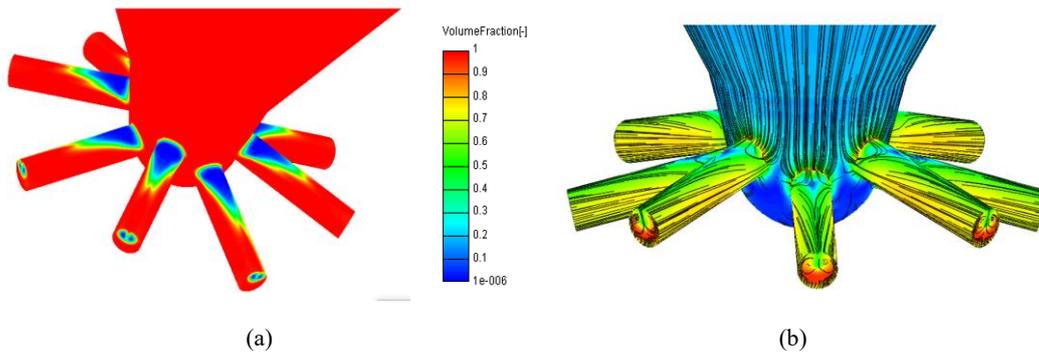
386 Furthermore, the spray penetrations of hole 1,3,5,7 are larger than those of hole 2,4,6,8, while the
387 spray cone angles of hole 1,3,5,7 are also slightly bigger. Fig.9 to Fig.11 show good consistency

388 between computational and experimental results, implying that the coupled model could be used for
389 further analysis in both cavitation flow within the injector and spray jet development.

390 4. Results and discussion

391 4.1 in-nozzle flow characteristics

392



393

394

Fig.12 (a) Cavitation distribution; (b) fuel velocity streamline at full needle lift

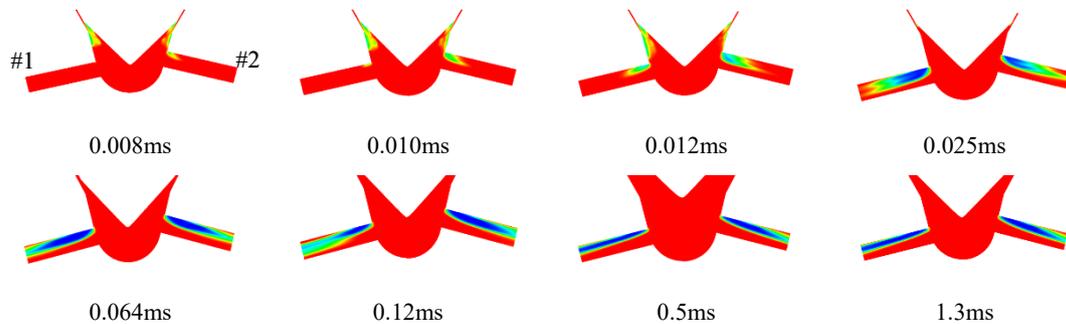
396

397 Cavitation distribution and fuel velocity streamlines in the double-layer nozzle at maximum needle
398 lift are shown in Fig. 12. The cavitation distributions and flow streamlines are both highly symmetric

399 with negligible differences. When the needle valve is fully opened, the cavitation in both the upper
400 and the lower layered holes occurs in the upper part of the orifices. In addition, cavitation occupies

401 a relative larger area in the upper layer nozzle holes than the lower layer ones.

402



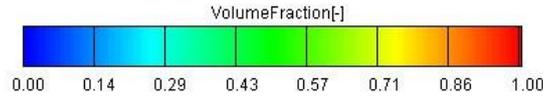


Fig.13 Variation of cavitation development with the time

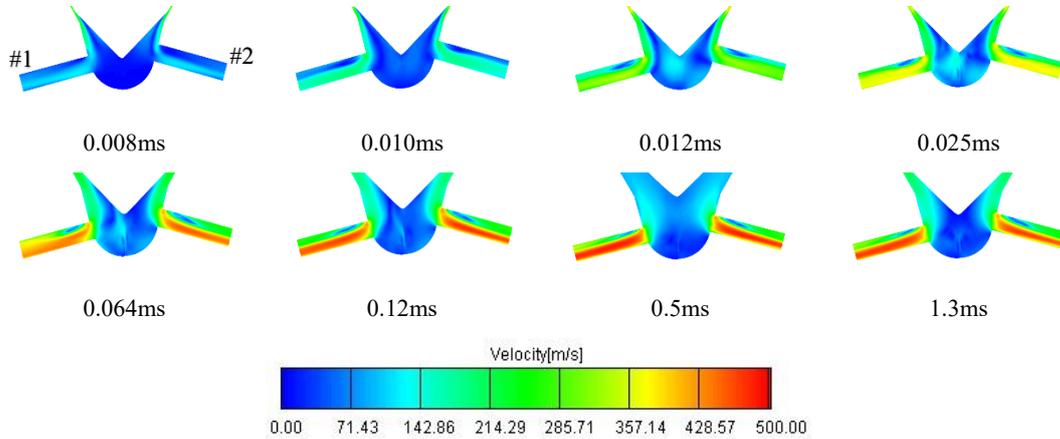


Fig.14 Variation of velocity magnitude with the time

Fig.13 and Fig.14 shows the variation of cavitation and velocity flow field distribution between the upper and lower layer nozzle holes. To better visualize the flow characteristics within the injector, hole 2 is rotated to make the same plane as hole 1. As shown in these figures, cavitation occurs earlier in the upper layered nozzle holes and develops faster as compared to cavitation development in the lower layered nozzle holes. In addition, the fuel velocity in the upper layered nozzle holes is slightly faster. At full needle lift, the fuel flow velocity distributions of the upper layered nozzle holes is also less uniform. The cumulative effect of these discrepancies results in the manifestation of higher degree of cavitation developments in the upper layered nozzle holes than the cavitation developments in the lower layered nozzle holes. Furthermore, the acuteness of the upper layer hole means that cavitation development at their entry sections will be more developed than those at the less acute lower layered holes.

4.2 Spray development

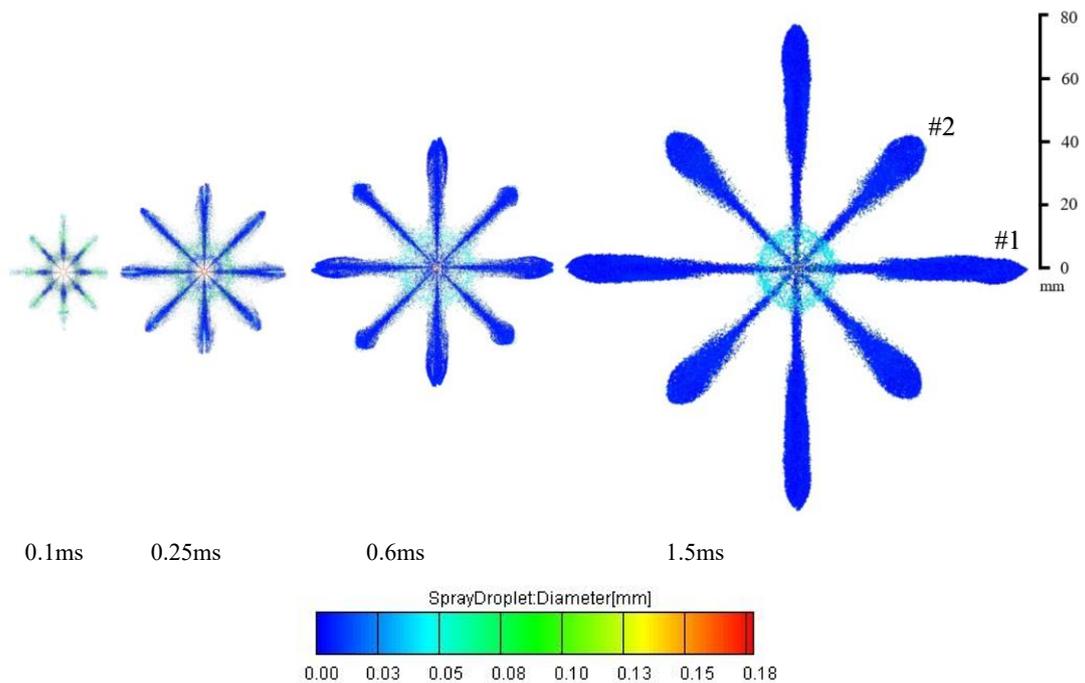


Fig.15 Spray droplet diameter with the time

421
 422
 423
 424
 425
 426
 427
 428
 429
 430
 431
 432
 433

Fig. 15 and 16 represent the spray jet development at different times. In Fig.15, the spray droplet diameter is large at the initial stages of the injection. The spray jet penetration was changed gradually with the evolution of the injection progress. Furthermore, the droplet size continues to decrease due to subsequent droplet breakups. After 0.25ms from the start of injection, the number of the spray droplets increases significantly.

The shape of individual spray jets starts to differentiate after 0.1ms, while the difference become more apparent after 0.25ms. The spray jet penetration from the upper and lower layer holes remain almost the same before 0.25ms. At the end of injection, the droplet distribution from the lower layer holes are more uniform than the distribution from the upper layer nozzle holes.



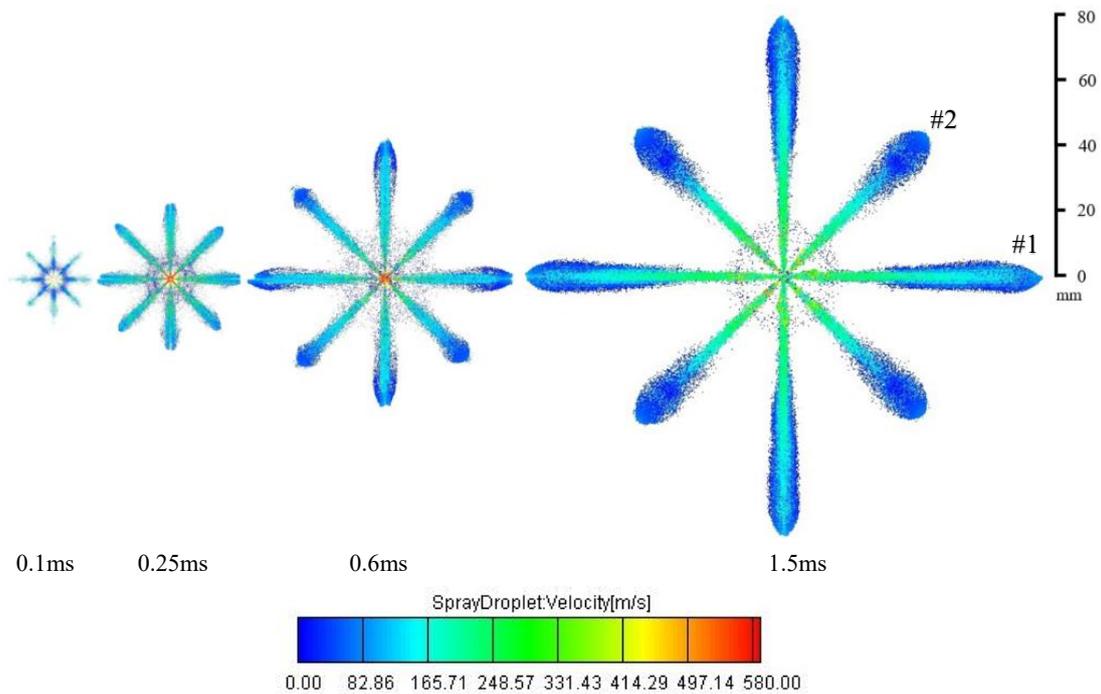


Fig.16 Spray velocity field with the time

434

435

436 The spray velocity field is represented in Fig.16; discrepancies can be seen. In the direction of the
 437 spray jet axis, the droplet velocity decreases outwards along the central axis, because of the larger
 438 aerodynamic influence on the droplets in the far-field of the spray domain. The jet velocity in the
 439 near nozzle domain, increases as the spray progresses, reaching around a maximum speed of 580
 440 m/s at 0.6 m/s after start of injection, and ends at 283.7 m/s. While in the far field of the spray
 441 domain, the velocities of the droplets are much smaller than those around the near nozzle domain
 442 (less than 100 m/s).

443

444 5. Conclusions

445 A double-layer 8-hole heavy-duty diesel engine nozzle geometry derived from X-ray scans and
 446 featuring all geometrical differences between the individual injection holes was used for the
 447 characterization of hole-to-hole variation on spray formation. This was achieved through numerical
 448 simulations. Internal nozzle flow was simulated (using RANS two-phase flow model) and the results,
 449 interfaced as inlet boundary conditions during spray simulation, using the Euler-Lagrangian
 450 approach. The technique was then used to predict spray development after validation. Model
 451 validation was obtained against momentum fluxes from all eight individual holes as well as the
 452 corresponding spray tip penetration rates. The following conclusion were arrived at from further
 453 analysis:

454

455 1) Injection rate as well as spray penetration time histories from both simulation and experiment
 456 follows almost the same trend overall. The accuracy of the established model in predicting flow
 457 characteristics and spray patterns are high and within acceptable limits (less than 5% in flow and
 458 within 10% in spray).

459 2) From both experiments and simulations, the injection rate and the cycle fuel injection quantities

460 of the lower layer nozzle holes were between 4 – 8 % higher than the cycle fuel injection quantities
461 of the upper layer nozzle holes. The differences in spray penetration from the lower layer holes and
462 the upper layer ones reached more than 30%.
463 3) The acuteness of the upper layer nozzle holes contributed to the formation of a higher degree
464 of cavitation development in them and also high spray droplet velocities as compared to the less
465 acute lower layer nozzle holes.

466 **Acknowledgement**

467 This work was supported by the National Natural Science Foundation of China (No. 51476072),
468 and the Chinese Scholarship Council (CSC No. 201808320261).
469

470 **Reference**

- 471 1. Deb M, Sastry GRK, Bose PK, Banerjee R. An experimental study on
472 combustion, performance and emission analysis of a single cylinder, 4-stroke
473 DI-diesel engine using hydrogen in dual fuel mode of operation. *Int J Hydrogen*
474 *Energy*. 2015;40(27):8586-8598.
- 475 2. Reitz RD, Ogawa H, Payri R, et al. IJER editorial: The future of the internal
476 combustion engine. *Int J Engine Res*. 2020;21(1):3-10.
- 477 3. Karathanassis IK, Koukouvinis P, Gavaises M. Comparative evaluation of
478 phase-change mechanisms for the prediction of flashing flows. *Int J Multiph*
479 *Flow*. 2017;95:257-270.
- 480 4. Gavaises M, Papoulias D, Andriotis A, Giannadakis E, Theodorakakos A. Link
481 Between Cavitation Development and Erosion Damage in Diesel Injector
482 Nozzles. *SAE Tech Pap 2007-01-0246*. Vol 2007. ; 2007:776-790.
- 483 5. Payri F, Bermúdez V, Payri R, Salvador FJ. The influence of cavitation on the
484 internal flow and the spray characteristics in diesel injection nozzles. *Fuel*.
485 2004;83(4-5):419-431.
- 486 6. He Z, Zhou H, Duan L, Xu M, Chen Z, Cao T. Effects of nozzle geometries and
487 needle lift on steadier string cavitation and larger spray angle in common rail
488 diesel injector. *Int J Engine Res*. July 2020;1-16.
- 489 7. Prasetya R, Sou A, Oki J, Nakashima A. Three-dimensional flow structure and
490 string cavitation in a fuel injector and their effects on discharged liquid jet. *Int J*
491 *Engine Res*. Mar. 2019;1-14.
- 492 8. Arcoumanis C, Gavaises M, Argueyrolles B, Galzin F. Modeling of Pressure-
493 Swirl Atomizers for GDI Engines. *SAE Trans*. 1999;108:516-532.
- 494 9. Giannadakis E, Papoulias D, Theodorakakos A, Gavaises M. Simulation of
495 cavitation in outward-opening piezo-type pintle injector nozzles. *Proc Inst Mech*
496 *Eng Part D J Automob Eng*. 2008;222(10):1895-1910.
- 497 10. Naseri H, Trickett K, Mitroglou N, et al. Turbulence and Cavitation Suppression
498 by Quaternary Ammonium Salt Additives. *Sci Rep*. 2018;8(1):1-15.
- 499 11. Hayashi T, Suzuki M, Ikemoto M. Visualization of Internal Flow and Spray
500 Formation with Real Size Diesel Nozzle. *12th Triennial International*
501 *Conference on Liquid Atomization and Spray Systems (ICLASS)*, Heidelberg,

- Germany, 2012.
- 503 12. Reid BA, Gavaises M, Mitroglou N, et al. On the formation of string cavitation
504 inside fuel injectors. *Exp Fluids*. 2014;55(1):1-8.
- 505 13. Lockett RD, Bonifacio A. Hydrodynamic luminescence in a model diesel
506 injector return valve. *Int J Engine Res*. Aug. 2019;1-12.
- 507 14. Mitroglou N, McLorn M, Gavaises M, Soteriou C, Winterbourne M.
508 Instantaneous and ensemble average cavitation structures in Diesel micro-
509 channel flow orifices. *Fuel*. 2014;116:736-742.
- 510 15. Fitzgerald RP, Vecchia G Della, Peraza JE, Martin GC, Térmicos M, Politècnica
511 U. Features of Internal Flow and Spray for a Multi-Hole Transparent Diesel Fuel
512 Injector Tip. *ILASS–Europe 2019, 29th Conf Liq At Spray Syst 2-4 Sept 2019,*
513 *Paris, Fr.* 2019;(September):2-4.
- 514 16. Gomez Santos E, Shi J, Gavaises M, Soteriou C, Winterbourn M, Bauer W.
515 Investigation of cavitation and air entrainment during pilot injection in real-size
516 multi-hole diesel nozzles. *Fuel*. 2020;263:116746.
- 517 17. Martin Gold, Richard Pearson, Jack Turner, Dan Sykes, Viacheslav Stetsyuk,
518 Guillaume de Sercey, Cyril Crua, Mithun Girija Murali, Foivos Koukouvinis,
519 Manolis Gavaises. Simulation and Measurement of Transient Fluid Phenomena
520 within Diesel Injection. *SAE Tech Pap 2019-01-0066*, 2019.
- 521 18. Mithun MG, Koukouvinis P, Gavaises M. Numerical simulation of cavitation
522 and atomization using a fully compressible three-phase model. *Phys Rev Fluids*.
523 2018;3(6):0-3.
- 524 19. Ferrari A, Zhang T. Benchmark between Bosch and Zeuch method-based
525 flowmeters for the measurement of the fuel injection rate. *Int J Engine Res*. Mar.
526 2019;1-12.
- 527 20. Karathanassis IK, Trickett K, Koukouvinis P, Wang J, Barbour R, Gavaises M.
528 Illustrating the effect of viscoelastic additives on cavitation and turbulence with
529 X-ray imaging. *Sci Rep*. 2018;8(1):1-15.
- 530 21. Pastor J, Garcia-Oliver JM, Garcia A, Zhong W, Micó C, Xuan T. An
531 Experimental Study on Diesel Spray Injection into a Non-Quiescent Chamber.
532 *SAE Int J Fuels Lubr*. 2017;10(2):394-406.
- 533 22. Matusik KE, Sforzo BA, Seong HJ, Duke D, Kastengren AL, Ilavsky J, Powell
534 CF. X-ray measurements of fuel spray specific surface area and sauter mean
535 diameter for cavitating and non-cavitating diesel sprays. *At Sprays*.
536 2019;29(3):199-216.
- 537 23. Ajrouche H, Nilaphai O, Moreau B, Hespel C, Foucher F, Mounaïm-rousselle C.
538 Engine Combustion Network (ECN): Characterization and comparison of
539 Diesel spray combustion in new high-pressure and high-temperature chamber.
540 *19th Annu Conf Liq At Spray Syst ILASS-Asia*. 2017;(October):1-4.
- 541 24. Koukouvinis P, Gavaises M, Li J, Wang L. Large Eddy Simulation of Diesel
542 injector including cavitation effects and correlation to erosion damage. *Fuel*.
543 2016;175:26-39.
- 544 25. Zhou X, Li T, Wei Y, Wang N. Scaling liquid penetration in evaporating sprays
545 for different size diesel engines. *Int J Engine Res*. Dec. 2019;1-16.

- 546 26. Nsikane DM, Vogiatzaki K, Morgan RE, et al. Novel approach for adaptive
547 coefficient tuning for the simulation of evaporating high-speed sprays injected
548 into a high-temperature and high-pressure environment. *Int J Engine Res.*
549 2020;21(7):1162-1179.
- 550 27. Theodorakakos A, Strotos G, Mitroglou N, Atkin C, Gavaises M. Friction-
551 induced heating in nozzle hole micro-channels under extreme fuel pressurisation.
552 *Fuel.* 2014;123(x):143-150.
- 553 28. Strotos G, Koukouvinis P, Theodorakakos A, Gavaises M, Bergeles G. Transient
554 heating effects in high pressure Diesel injector nozzles. *Int J Heat Fluid Flow.*
555 2015;51:257-267.
- 556 29. Shi J, Guerrassi N, Dober G, Karimi K, Meslem Y. Complex physics modelling
557 of diesel injector nozzle flow and spray supported by new experiments.
558 *THIESEL 2014 Conf. Thermo-Fluid Dynamics. Process Direct Inject Engines.*
559 2014.
- 560 30. Shi J, Aguado PL, Dober G, Guerrassi N, Bauer W, Lai M. Using LES and x-
561 ray imaging to understand the influence of injection hole geometry on Diesel
562 spray formation. *THIESEL 2016 Conf. Thermo-Fluid Dynamics Process Diesel*
563 *Engines Conference Thermo Fluid Dynamics.* 2016.
- 564 31. Shi J, Aguado Lopez P, Gomez Santos E, Guerrassi N, Dober G, Bauer W, Lai
565 M, Wang J. Evidence of vortex driven primary breakup in high pressure fuel
566 injection. *Proc. ILASS–Europe 2017 28th Conf. on Liquid Atomization and Spray*
567 *System (ILASS), Valencia, Spain, 2017.*
- 568 32. Shi J, Lopez PA, Santos EG, et al. High Pressure Diesel Spray development : the
569 effect of nozzle geometry and flow vortex dynamics. *14th Triennial International*
570 *Conf. on Liquid Atomization and Spray Systems (ICLASS), Chicago, IL, USA,*
571 *2018.*
- 572 33. Strotos G, Gavaises M, Theodorakakos A, Bergeles G. Evaporation of a
573 Suspended Multicomponent Droplet Under Convective Conditions. *Proc. of*
574 *CHT-08 International Symposium on Advances in Computational Heat*
575 *Transfer (ICHMT), Marrakech, Morocco, 2008.*
- 576 34. Stefanitsis D, Malgarinos I, Strotos G, Nikolopoulos N, Kakaras E, Gavaises M.
577 Numerical investigation of the aerodynamic breakup of Diesel and heavy fuel
578 oil droplets. *Int J Heat Fluid Flow.* 2017;68:203-215.
- 579 35. Papoutsakis A, Theodorakakos A, Giannadakis E, Papoulias D, Gavaises M.
580 LES Predictions of the Vortical Flow Structures in Diesel Injector Nozzles. *SAE*
581 *Tech Pap 2009-01-0833, 2009.*
- 582 36. Yuan W, Schnerr GH. Numerical Simulation of Two-Phase Flow in Injection
583 Nozzles: Interaction of Cavitation and External Jet Formation. *J Fluids Eng.*
584 2004;125(6):963-969.
- 585 37. Lauer E, Hu XY, Hickel S, Adams NA. Numerical investigation of collapsing
586 cavity arrays. *Phys Fluids.* 2012;24(5):52104.
- 587 38. Kubota A, Kato H, Yamaguchi H. Finite difference analysis of unsteady
588 cavitation on a two-dimensional hydrofoil. *Fifth International Conference on*
589 *Numerical Ship Hydrodynamics (ICNSH), Hiroshima, Japan, 1990.*

- 590 39. Kunz RF, Boger D a, Stinebring DR, et al. A preconditioned Navier - Stokes
591 method for two-phase flows with application to cavitation prediction. *Comput*
592 *Fluids*. 2000;29:849-875.
- 593 40. Schnerr GH, Sauer J. Physical and Numerical Modeling of Unsteady Cavitation
594 Dynamics. *Fourth International Conference on Multiphase Flow (ICMF)*, New
595 Orleans, USA, 2001.
- 596 41. Zwart PJ, Gerber AG, Belamri T. A Two-Phase Flow Model for Predicting
597 Cavitation Dynamics. *Fifth International Conference on Multiphase Flow*
598 *(ICMF)*, Yokohama, Japan, 2004.
- 599 42. Goel T, Thakur S, Haftka R, Shyy W, Zhao J. Surrogate Model-Based Strategy
600 for Cryogenic Cavitation Model Validation and Sensitivity Evaluation. *42nd*
601 *AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*, Sacramento,
602 California, USA, 2006.
- 603 43. Niedzwiedzka A, Schnerr GH, Sobieski W. Review of numerical models of
604 cavitating flows with the use of the homogeneous approach. *Arch Thermodyn*.
605 2016;37(2):71-88.
- 606 44. Cristofaro M, Edelbauer W, Koukouvinis P, Gavaises M. A numerical study on
607 the effect of cavitation erosion in a diesel injector. *Appl Math Model*.
608 2020;78:200-216.
- 609 45. Wang C, Moro A, Xue F, Wu X, Luo F. The influence of eccentric needle
610 movement on internal flow and injection characteristics of a multi-hole diesel
611 nozzle. *Int J Heat Mass Transf*. 2018;117:818-834.
- 612 46. Katz J, Cristofaro M, Edelbauer W, Koukouvinis P, Gavaises M. Large Eddy
613 Simulation of the Internal Injector Flow During Pilot Injection. *Proc. 10th Int*
614 *Symp Cavitation*. 2019:9-12.
- 615 47. Edelbauer W. Numerical simulation of cavitating injector flow and liquid spray
616 break-up by combination of Eulerian–Eulerian and Volume-of-Fluid methods.
617 *Comput Fluids*. 2017;144:19-33.
- 618 48. Moro A, Luo T, Wang C, Luo F. Eccentric needle displacement effect on spray
619 formation from a multi orifice diesel injector. *Heat Mass Transf und*
620 *Stoffuebertragung*. 2019.
- 621 49. Greif D, Edelbauer W, Struel J. Numerical Simulation Study of Cavitating
622 Nozzle Flow and Spray Propagation with Respect to Liquid Compressibility
623 Effects. *SAE Tech Pap 2014-01-1421*, 2014.
- 624 50. Cristofaro M, Edelbauer W, Gavaises M, Koukouvinis P. Numerical simulation
625 of compressible cavitating two-phase flows with a pressure-based solver. *Proc.*
626 *ILASS–Europe 2017 28th Conf. on Liquid Atomization and Spray System*
627 *(ILASS)*, Valencia, Spain, 2017.
- 628 51. Cristofaro M, Edelbauer W, Koukouvinis P, Gavaises M. Influence of Diesel fuel
629 viscosity on cavitating throttle flow simulations at erosive operation conditions.
630 *ACS Omega*. 2020;5(13):7182-7192.
- 631 52. von Berg E, Edelbauer W, Alajbegovic A, et al. Coupled Simulations of Nozzle
632 Flow, Primary Fuel Jet Breakup, and Spray Formation. *J Eng Gas Turbines*
633 *Power*. 2005;127(4):897.

- 634 53. Waidmann W, Boemer A, Braun M. Adjustment and verification of model
635 parameters for Diesel injection CFD simulation. *SAE Tech Pap.* 2006;2006(724).
- 636 54. Luo F, Cui H, Dong S. Transient measuring method for injection rate of each
637 nozzle hole based on spray momentum flux. *Fuel.* 2014;125:20-29.
- 638 55. Luo T, Jiang S, Moro A, Wang C, Zhou L, Luo F. Measurement and validation
639 of hole-to-hole fuel injection rate from a diesel injector. *Flow Meas Instrum.*
640 2018;61:66-78.
- 641 56. Zhou LY, Dong SF, Cui HF, Wu XW, Xue FY, Luo FQ. Measurements and
642 analyses on the transient discharge coefficient of each nozzle hole of multi-hole
643 diesel injector. *Sensors Actuators, A Phys.* 2016;244:198-205.
- 644 57. Yin B, Yu S, Jia H, Yu J. Numerical research of diesel spray and atomization
645 coupled cavitation by Large Eddy Simulation (LES) under high injection
646 pressure. *Int J Heat Fluid Flow.* 2016;59:1-9.
- 647
- 648
- 649
- 650
- 651