



City Research Online

City, University of London Institutional Repository

Citation: Strotos, G., Koukouvinis, P., Theodorakakos, A., Gavaises, M. & Bergeles, G. (2015). Transient heating effects in high pressure Diesel injector nozzles. *International Journal of Heat and Fluid Flow*, 51, pp. 257-267. doi: 10.1016/j.ijheatfluidflow.2014.10.010

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/13564/>

Link to published version: <https://doi.org/10.1016/j.ijheatfluidflow.2014.10.010>

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

1 **Transient heating effects in high pressure Diesel injector nozzles**

2
3 George Strotos^{1,a,*}, Phoevos Koukouvinis^{1,b}, Andreas Theodorakakos^{1,c}, Manolis Gavaises^{1,d} and
4 George Bergeles^{1,e}

5 ¹School of Engineering and Mathematical Sciences, City University London, Northampton
6 Square, EC1V 0HB, London, UK

7 ^a George.Strotos.1@city.ac.uk

8 ^b Foivos.Koukouvinis.1@city.ac.uk

9 ^c Andreas.Theodorakakos.1@city.ac.uk

10 ^d M.Gavaises@city.ac.uk

11 ^e bergeles@fluid.mech.ntua.gr

12 ^{*}Corresponding author

13 14 **Abstract**

15 The tendency of today's fuel injection systems to reach injection pressures up to 3000 bar in
16 order to meet forthcoming emission regulations may significantly increase liquid
17 temperatures due to friction heating; this paper identifies numerically the importance of fuel
18 pressurization, phase-change due to cavitation, wall heat transfer and needle valve motion on
19 the fluid heating induced in high pressure Diesel fuel injectors. These parameters affect the
20 nozzle discharge coefficient (C_d), fuel exit temperature, cavitation volume fraction and
21 temperature distribution within the nozzle. Variable fuel properties, being a function of the
22 local pressure and temperature are found necessary in order to simulate accurately the effects
23 of depressurization and heating induced by friction forces. Comparison of CFD predictions
24 against a 0-D thermodynamic model, indicates that although the mean exit temperature
25 increase relative to the initial fuel temperature is proportional to $(1-C_d^2)$ at fixed needle
26 positions, it can significantly deviate from this value when the motion of the needle valve,
27 controlling the opening and closing of the injection process, is taken into
28 consideration. Increasing the inlet pressure from 2000bar, which is the pressure utilized in
29 today's fuel systems to 3000bar, results to significantly increased fluid temperatures above

30 the boiling point of the Diesel fuel components and therefore regions of potential
31 heterogeneous fuel boiling are identified.

32 **Keywords:** nozzle, cavitation, variable properties, moving needle, fuel heating

33

34 **1 Introduction**

35 The market share for passenger cars is expected to double (ExxonMobil) the coming years, as
36 also the diesel oil consumption. The need for more efficient IC engines which comply with
37 the strict emission legislation to be imposed leads to the development of higher injection
38 pressures, pressures up to 3000bar(Goud M et al., 2012) from 2000 bar, which is the nominal
39 value in today's commercial passenger car fuel injection equipment (FIE). At such elevated
40 pressures high flow velocities develop within the injector which lead to cavitation
41 (Arcoumanis et al., 2000). Cavitation in fuel injectors has been examined both experimentally
42 and numerically as it reduces injection volumetric efficiency and may result to material
43 erosion(Prosperetti and Hao, 1999). On the other hand, it may improve the air-fuel mixing by
44 increasing the spray cone angle(Payri et al., 2004). Flow measurements in cavitating injector
45 nozzles operating under such pressures have not been obtained so far; most of the
46 experimental studies reported are emulating the engine operating conditions as in (Andriotis
47 et al., 2008; Badock et al., 1999; Blessing et al., 2003; Chaves et al., 1995; Payri et al., 2013;
48 Soteriou et al., 2000). Alternatively, computational methodologies seem to be the only way to
49 understand the implications of cavitation under real operating conditions. Several numerical
50 methodologies for simulating cavitation have been proposed. For example, a single-fluid
51 mixture is proposed in(Chen and Heister, 1995) while the two-fluid method is reported in
52 (Alajbegovic et al., 1999; Singhal et al., 2002; Yuan and Schnerr, 2004) where conservation
53 equations are solved for both phases separately and interaction between them is accounted for
54 by using additional source terms. The Eulerian-Lagrangian models of (Brennen, 1995;
55 Giannadakis et al., 2004; Hilgenfeldt et al., 1998; Keller and Miksis, 1980) assume a bubbly
56 phase to be dispersed inside the liquid phase while the Rayleigh-Plesset equation is utilized
57 for predicting the bubble's growth and collapse. The models of(Ando et al., 2011; Fuster and
58 Colonius, 2011; Jamaluddin et al., 2011;Zeravcic et al., 2011) account for compressibility
59 effects. Homogeneous equilibrium models (HEM) assume a perfect mixing between the
60 liquid and the vapor phase while the cavitation bubble's growth is calculated by using a
61 barotropic equation which relates pressure and density (Habchi et al., 2008; Liu et al., 2004;
62 Payri et al., 2012; Salvador et al., 2013).

63 A common feature of cavitation studies in fuel injector nozzles is the assumption of
64 isothermal flow due to the short timescales involved. On the other hand, the flow induced
65 during the discharge of the fuel is characterized by strong velocity gradients which induce
66 wall friction and consequently fuel heating. Studies addressing the complicated effects
67 occurring during the motion of the needle valve that controls the injection process have
68 recently appeared in the literature (Battistoni and Grimaldi, 2012; He et al., 2013; Lee and
69 Reitz, 2010; Margot et al., 2010; Neroorkar et al., 2012; Payri et al., 2009; Zhao et al., 2013).
70 The present study focuses on the thermal effects occurring in high pressure diesel nozzles by
71 solving the energy equation and including the friction induced heating. The CFD model used
72 is an Eulerian-Lagrangian model which has been built upon the in-house CFD cavitation
73 model reported in (Giannadakis et al., 2008); this work is an extension of that presented
74 recently in (Strotos et al., 2014a; Strotos et al., 2014b; Theodorakakos et al., 2014) which
75 additionally examines the effect of needle motion. In the absence of relevant experimental
76 data, the present work aims to quantify the numerical effects of using constant or variable
77 properties, the effect of two-phase flow, the effect of inlet pressure increase and the effect of
78 initial and boundary conditions on temperature distribution within the injector. In the
79 following sections, the mathematical model is presented, followed by the results obtained for
80 high pressure diesel nozzles in steady lift and moving lift cases; the most important
81 conclusions are summarized at the end.

82

83 **2 Numerical model and methodology**

84 **2.1 Equations solved**

85 The flow solver used has been developed by the authors' group and solves the Navier-Stokes
86 equations in an unstructured mesh. Turbulence is modeled with the k-ε model (Lauder and
87 Spalding, 1974); detailed description of the flow equations can be found in (Giannadakis et
88 al., 2008). Here, focus is given into the solution of the energy equation for the liquid phase
89 and the determination of the temperature field. Based on (Städtke, 2007), the conservation
90 equation expressed in terms of the specific total enthalpy is:

$$\begin{aligned}
 & \frac{\partial(a_L \rho h_{tot})}{\partial t} + \nabla \cdot (a_L \rho h_{tot} \mathbf{u}) = \\
 & \nabla \cdot (a_L \kappa_{eff} \nabla T) + \nabla \cdot (a_L \boldsymbol{\tau}_{eff} \cdot \mathbf{u}) + a_L \rho (\mathbf{u} \cdot \mathbf{g}) + \frac{\partial(a_L p)}{\partial t} + S_h
 \end{aligned} \tag{1}$$

91

92 Where the specific total enthalpy is the sum of the specific static enthalpy h , the flow mean
 93 kinetic energy and the turbulent kinetic energy k

$$94 \quad h_{tot} = h + \frac{\mathbf{u} \cdot \mathbf{u}}{2} + k \quad (2)$$

95 The presence of the cavitating phase is taken into account through α_L which represents the
 96 liquid volume fraction in a computational cell, and with the source term S_h (Städtke, 2007)
 97 which accounts for the interaction between the two phases, gas and liquid. This additional
 98 source term for the interaction between the two phases includes the energy exchange due to
 99 mass transfer, the interfacial heat transfer and the work of viscous interfacial forces. Note that
 100 equation (1) reduces to the equation given in (Versteeg and Malalasekera, 2007) for the case
 101 of single phase flow. In (1) the stress tensor $\boldsymbol{\tau}_{eff}$ is given by:

$$102 \quad \boldsymbol{\tau}_{eff} = \mu_{eff} \left(\nabla \mathbf{u} + (\nabla \mathbf{u})^T \right) - \frac{2}{3} \mu_{eff} (\nabla \cdot \mathbf{u}) \mathbf{I} - \frac{2}{3} \rho k \mathbf{I} \quad (3)$$

$$103 \quad \mu_{eff} = \mu_{lam} + \mu_{urb} \quad (4)$$

$$104 \quad \kappa_{eff} = \left(\frac{\mu_{lam}}{Pr_{lam}} + \frac{\mu_{urb}}{Pr_{urb}} \right) c_p \quad (5)$$

105 Where \mathbf{I} is the unit tensor. The turbulent viscosity μ_{urb} is calculated from the k- ϵ turbulence
 106 model and the turbulent Prandtl number Pr_{urb} , is taken equal to 0.85. It has to be noted that
 107 the 2nd RHS term of equation (1) contains both the reversible and the irreversible work of
 108 viscous forces; the latter is commonly known as viscous heating and represents the heating
 109 induced by the friction forces.

110 Following the methodology presented in (Kolev, 2002), the specific enthalpy can be
 111 expressed as

$$112 \quad h = h_0 + c_{pmT} (T - T_0) + h^* \quad (6)$$

$$113 \quad c_{pmT} = \left(\int_{T_0}^T c_p dT \right) / (T - T_0) \quad (7)$$

$$114 \quad h^* = \int_{p_0}^p \left(\frac{\partial h}{\partial p} \right)_{T=T_0} dp \quad (8)$$

115 In these equations h_0 , p_0 , and T_0 , are reference values, h^* is a function of pressure, while c_{pmT}
 116 is the mean heat capacity between the temperature under consideration and a reference
 117 temperature T_0 . For the case of constant properties, c_{pmT} is simply equal to c_p , while $h^*=(p-$
 118 $p_0)/\rho$. The reason for adopting the methodology of (Kolev, 2002) is that the author gives these
 119 thermodynamic properties as a function of pressure and temperature in the range 0-2500bar
 120 and 0-120°C.

121 For reasons of numerical stability, the diffusion term appearing on the RHS of equation (1)
 122 and containing the temperature instead of the total enthalpy is treated in an implicit way.
 123 Solving equation (6) for T and substituting it in (1) after some manipulation the following
 124 transport equation for the specific total enthalpy is derived:

$$125 \quad \frac{\partial(a_L \rho h_{tot})}{\partial t} + \nabla \cdot (a_L \rho h_{tot} \mathbf{u}) = \nabla \cdot (a_L \Gamma \nabla h_{tot}) + \nabla \cdot (a_L \boldsymbol{\tau}_{eff} \cdot \mathbf{u}) + a_L \rho (\mathbf{u} \cdot \mathbf{g}) + \frac{\partial(a_L p)}{\partial t} + S_h$$

$$- \nabla \cdot \left(a_L \Gamma h_{tot} \frac{\nabla c_{pmT}}{c_{pmT}} + a_L \Gamma c_{pmT} \nabla \left(\frac{h_{add}}{c_{pmT}} \right) \right) \quad (9)$$

$$126 \quad \Gamma = \frac{\mu_{lam}}{Pr_{lam}} + \frac{\mu_{turb}}{Pr_{turb}} \quad (10)$$

$$127 \quad h_{add} = h^* + \frac{\mathbf{u} \cdot \mathbf{u}}{2} + k \quad (11)$$

128 In each iteration the total enthalpy equation is solved and the temperature is obtained either
 129 from equation (6) for the case of constant properties, or from an iterative procedure in the
 130 case of variable properties. When the temperature has been determined, the properties are
 131 updated from the known temperature and pressure field. This procedure requires no more than
 132 10 internal iterations to converge while an under-relaxation factor can be also used in
 133 updating the temperature.

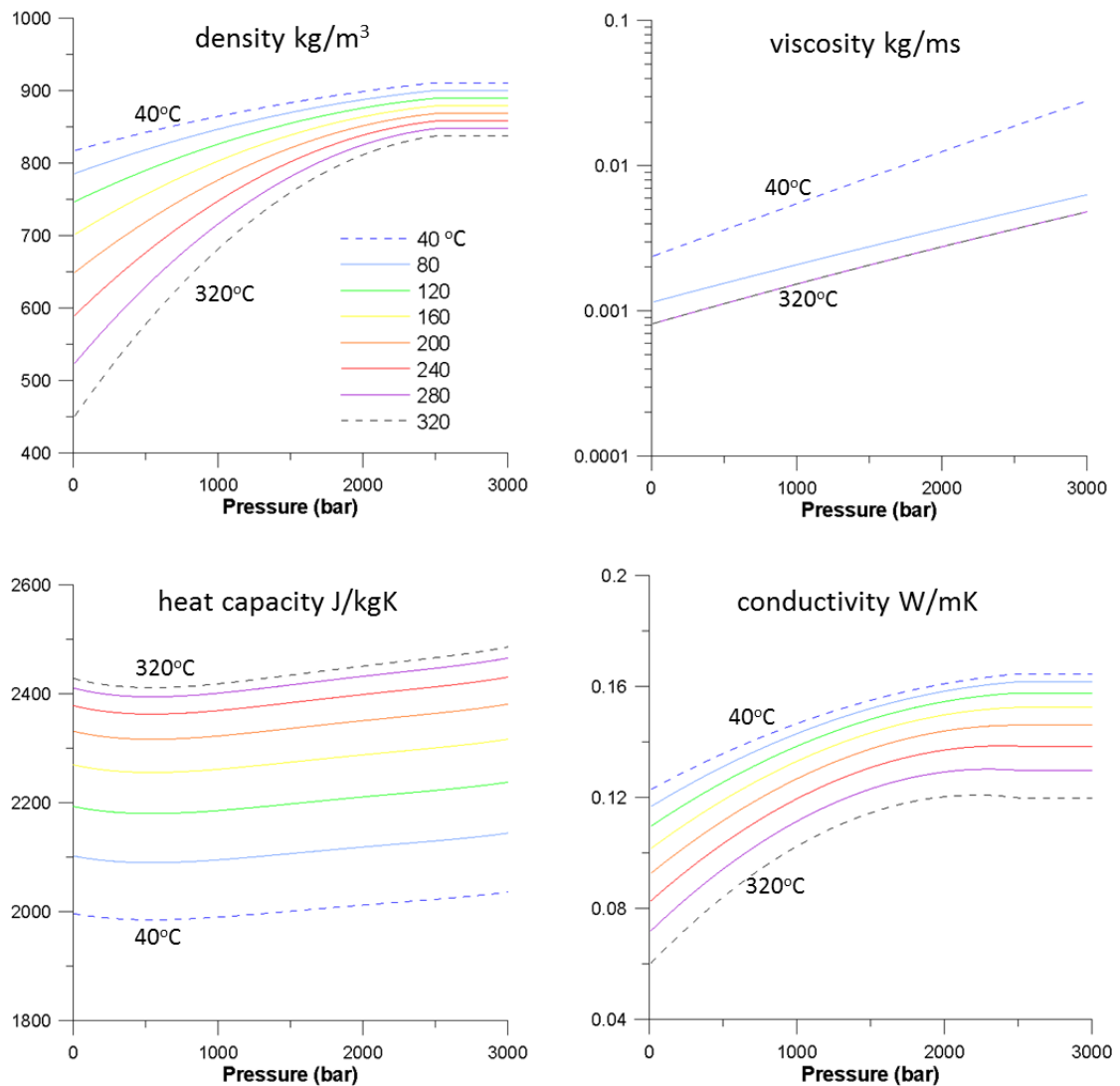
134 Regarding the impact of source term S_h in equation (9), an order of magnitude analysis has
 135 revealed that its impact in fuel heating, could be ignored. The rate of vapor formation is more
 136 than 5 orders of magnitude smaller than the fuel flow rate, while the heat flux due to
 137 vaporization is even smaller compared to the energy of the fuel entering the injector. Thus the
 138 main parameter affecting the fuel heating is the friction forces due to the strong velocity
 139 gradients appearing in the near wall region.

140

141 **2.2 Fuel properties**

142

143 The fuel used is the so-called “summer diesel” and its properties were taken from (Kolev,
144 2002) as function of temperature and pressure. For the purposes of the present work, they
145 have been extrapolated up to 3000bar and 400°C for all cases simulated. The extrapolation
146 method adopted, uses the functions given in (Kolev, 2002) but extends the limits of pressure
147 and temperature up to the point at which the property under consideration reaches a local
148 minimum or maximum; beyond this point, each property is assumed to be equal to the
149 corresponding value of the local minimum or maximum. The fuel properties utilized are
150 shown in Fig.1.



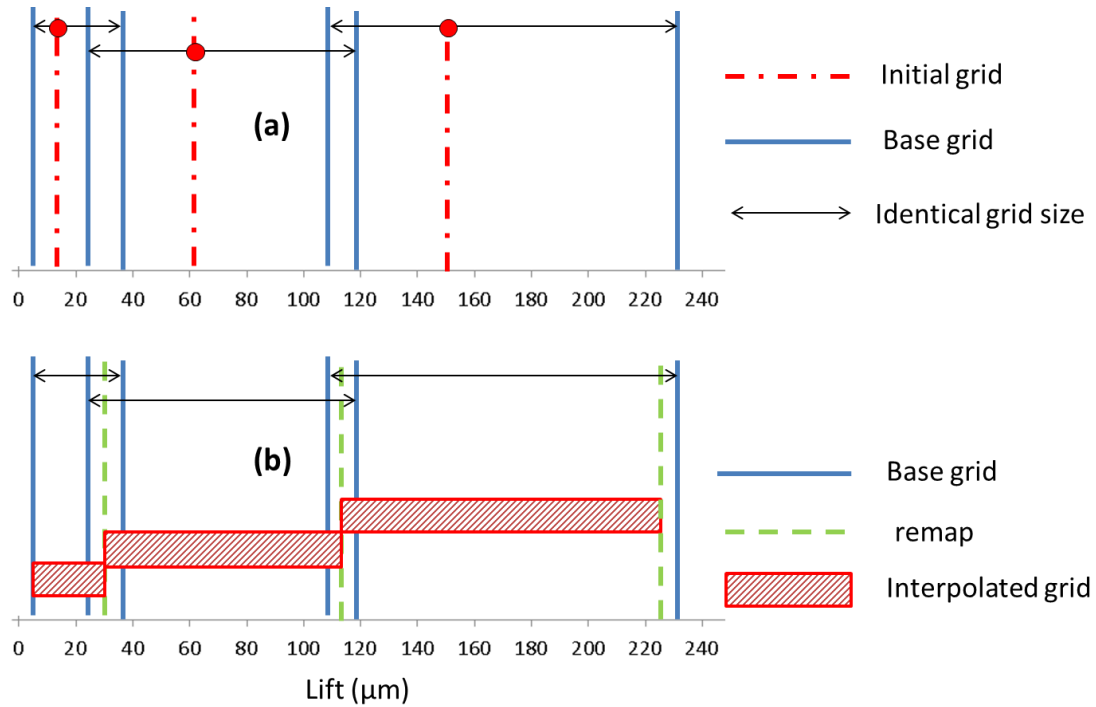
151

152 **Fig.1:** Diesel fuel properties as a function of pressure for selected temperature values
153 (extrapolated from 2400 to 3000bar and 120 to 400°C).

155 **2.3 Implementation of needle motion**

156 The computational technique used to simulate the needle motion is summarized in Fig.2. It
157 can be divided into two stages. In the first stage, three grids (termed as “initial grids”) are
158 constructed at 10, 60 and 150 μm needle lifts. Stretching of the three “initial grids” to both
159 lower and higher needle lifts is performed resulting to three pairs of “base grids”; for the
160 particular nozzle simulated here, the three pairs of ‘base grids’ have been obtained at the
161 following lifts: 5 and 35 μm , 25 and 120 μm , and 110 and 230 μm , respectively. This procedure
162 is graphically represented in Fig.2a. Note that each pair of the “basic grids” have identical
163 number of cells and identical grid topology at the boundary faces of the needle and the nozzle
164 wall. It has also to be noted that overlapping regions exist between 25 and 35 μm and between
165 110 and 120 μm .

166 During the needle motion (Fig.2b), the grid for each needle lift is obtained by linear
167 interpolation between a pair of “base grids”. When the needle lift value falls within an
168 overlapping region, then the obtained solution is remapped to the other pair. Special care has
169 been taken in order to construct grids with similar topology and minimize computational
170 errors when the grid is remapped. With regards to temporal discretization, a fully implicit
171 scheme was used, which is unconditionally stable, while shorter computational time steps
172 have been used in the opening and closing phase of the needle valve in order to ensure that
173 the needle lift does not change more than 1.0 μm /time-step; this limitation was used to avoid
174 abrupt changes in the grid topology. Numerical experiments have indicated that the opening
175 phase is not affected by the chosen time-step. On the other hand, during the closing phase,
176 some differences exist especially at the last stages, in which compressibility effects may
177 however become important but the code used does not account for such phenomena.



178

179 **Fig.2:** Computational technique for the grid adaption in moving lift.

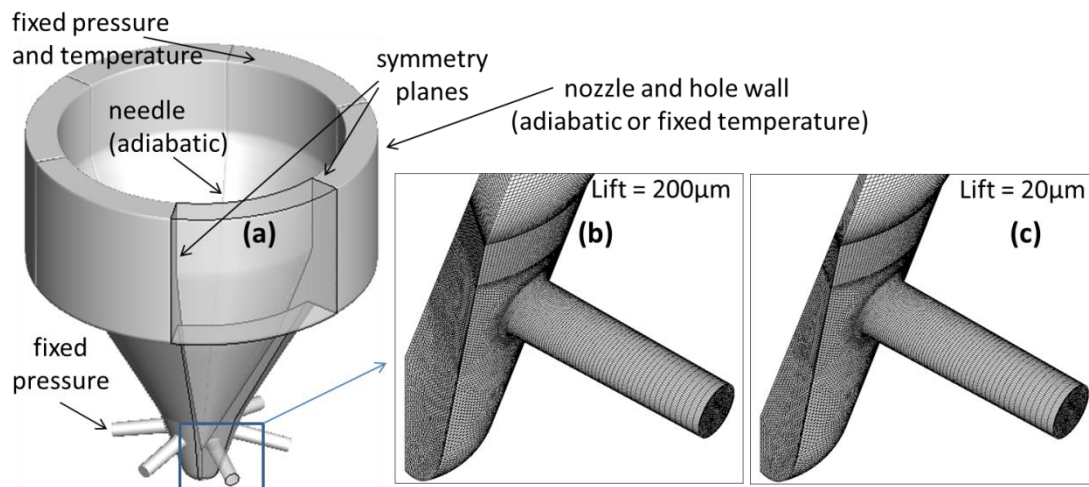
180

181 3 Results and discussion

182 3.1 Cases examined

183 A 6-hole tapered nozzle with 0.175mm hole diameter has been used in the present
 184 investigation; a similar geometry was used in the past for numerically validating an iso-
 185 thermal cavitation model (Giannadakis et al., 2007). The tapered hole has 20 μm rounding at
 186 the inlet of the hole and the k-factor is 1.77 (defined as $(D_{in} - D_{out})/10\mu\text{m}$). For the purposes of
 187 the present simulation, the 1/6th sector of the nozzle was modeled by applying symmetry
 188 boundary conditions at the cross sections; numerical experiments using grid sizes from 0.38M
 189 cells up to 3.4M cells prove that a grid of approximately one million cells was adequate for
 190 grid independent results to be achieved. The maximum variation of discharge coefficient was
 191 approximately 0.02 (i.e. 4%), while the maximum variation for the mean temperature increase
 192 was 1.6 $^{\circ}\text{C}$ for the various lifts examined. The maximum temperature variations between
 193 different grids observed locally at the exit of the hole may reach up to 5 $^{\circ}\text{C}$ which are
 194 considered to be small compared to the overall heating of the fuel. Furthermore, 10-25 cell
 195 layers were used inside the gap between the needle seat and the body of the nozzle, which

196 ensures the capturing of the velocity and thermal boundary layer development. Details of the
197 nozzle geometry and grid details are presented in Fig.3.



198

199 **Fig.3:** Nozzle geometry and grid details. (a)Computational domain and boundary conditions
200 utilized, (b, c) Detail of the computational mesh at 200µm and 20µm needle lift respectively.

201

202 The test cases simulated and the boundary conditions used are listed in Table 1 and Table 2,
203 respectively, covering a wide range of fixed needle lift positions (varying from 5 up to
204 200µm) and transient simulations with a moving lift, while in both cases (fixed or moving lift)
205 the differences between single and two-phase flow are examined. Additionally, the effect of
206 using constant or variable fuel properties is quantified. Two inlet pressures are examined
207 (2000 and 3000bar) with fixed inlet temperature at 80°C; at the nozzle hole exit a fixed
208 pressure equal to 60bar has been utilized. The needle's wall was assumed to be adiabatic,
209 while for the nozzle's wall either adiabatic or constant temperature at 80°C and 300°C
210 boundary conditions were applied, since its temperature is not generally known. The flow
211 field in the near wall region was modelled by using wall functions along with the enhanced
212 wall treatment proposed by (Wolfshtein, 1969); the y^+ values in the wall region were varied
213 between 1 and 30. Regarding the initial conditions for the transient cases, simulations start
214 from a converged velocity field at 5µm lift, while the initial temperature field for the fuel was
215 assumed to be uniform and equal to the inlet temperature for most of the cases examined. The
216 effect of the initial liquid temperature distribution is further examined by considering the last
217 case of Table 1 in which the liquid at the upper part of the injector has 80°C, the liquid at the
218 lower part (including the region of the hole) has a temperature of 120°C and between them
219 there is a region in which the liquid temperature varies linearly between 80°C and 120°C.

220

221

222

223 **Table 1:**Simulation cases

Lift [μm]	Inlet pressure (bar)	phase s	propertie s	Nozzle wall	Initial condition for fuel temperature
5, 20, 40, 80, 200	2000, 3000	single	Constant	Adiabatic	Uniform (80°C)
5, 20, 40, 80, 200	2000, 3000	single	Variable	Adiabatic	Uniform (80°C)
20, 40, 80, 200	2000, 3000	two	Variable	Adiabatic	Uniform (80°C)
Moving lift	2000, 3000	single	Variable	Adiabatic	Uniform (80°C)
Moving lift	2000, 3000	two	Variable	Adiabatic	Uniform (80°C)
Moving lift	2000	single	Variable	Fixed temperature (80/300°C)	Uniform (80°C)
Moving lift	2000	single	Variable	adiabatic	Linear (80-120°C)

224

225 **Table 2:** Summary of boundary conditions.

magnitude	Inlet	exit	Needle wall	Nozzle wall
Static pressure	Fixed (2000, 3000bar)	Fixed (60bar)		
Velocity vector	Zero 1 st gradient	Zero 1 st gradient	No slip	No slip

temperature	Fixed (80°C)	Zero 1 st gradient	adiabatic	Adiabatic or fixed (80/300°C)
-------------	--------------	-------------------------------	-----------	----------------------------------

226

227 **3.2 Steady lift simulations**

228 Due to lack of experimental data for the fuel heating in such high pressure Diesel injectors, a
 229 0-D thermodynamic model is used to estimate the mean fuel heating and validate the present
 230 methodology. **The model combines the continuity equation, the Bernoulli equation and the 1st**
 231 **law of thermodynamics.** Assuming adiabatic nozzle walls and no work exchange in steady lift
 232 conditions, the pressure difference between inlet and exit (Δp) is converted to liquid kinetic
 233 energy and liquid heating for a given nozzle discharge coefficient. It has to be noted that the
 234 0-D model ignores the contribution of turbulence (which is expected to have a minor effect),
 235 as also it is valid only for the case of single phase flow. For the case of constant properties
 236 fluid, it is easy to prove that the fuel increased temperature due to friction heating equals to:

$$237 \quad \Delta T = (1 - C_d^2) \Delta T_{ref} \quad (12)$$

238 where

$$239 \quad \Delta T_{ref} = \frac{\Delta p}{\rho_{in} c_{p,in}} \quad (13)$$

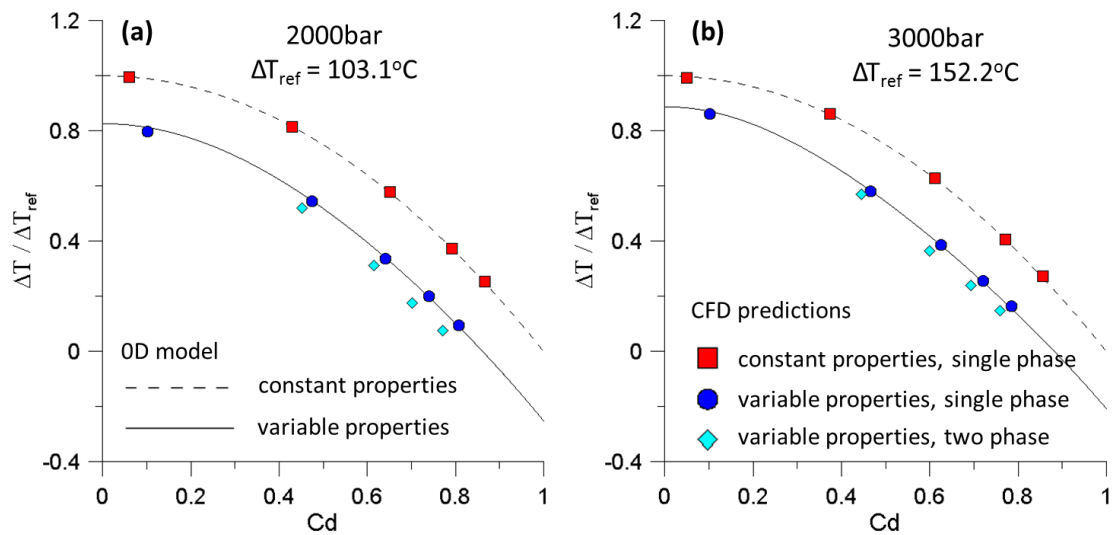
$$240 \quad U_{ref} = \sqrt{\frac{2\Delta p}{\rho_{in} (1 - (A_{out}/A_{in})^2)}} \quad (14)$$

$$241 \quad C_d = \frac{\dot{m}}{\rho_{in} A_{out} U_{ref}} \quad (15)$$

242 The reference temperature difference ΔT_{ref} can serve as a non-dimensional parameter to
 243 compare different cases, thus enabling a direct comparison between cases involving constant
 244 or variable thermodynamic properties. The same comments apply also to the definition of
 245 reference velocity U_{ref} and discharge coefficient C_d which are calculated based on the inlet
 246 properties which are fixed.

247 In Fig.4 the fuel heating for the cases of 2000bar (a) and 3000bar (b) inlet pressure is
 248 presented. The dashed and the solid lines correspond to the 0-D model for constant and
 249 variable properties, respectively. As seen, the assumption of constant properties leads to over-

250 prediction of the fuel heating, while it is important to notice that the variable properties case
 251 leads to fuel sub-cooling for high C_d values. In this case the friction is low and the sub-
 252 cooling due to fuel depressurisation dominates the phenomenon. The difference between the
 253 two curves seems to be rather significant, which implies that variable properties are important
 254 for accurate estimation of the fuel heating. Comparing Fig.4a to Fig.4b it is concluded that the
 255 dimensionless fuel heating is quite similar for different inlet pressures; in dimensional
 256 quantities, the reference temperature difference ΔT_{ref} increases with inlet pressure which
 257 means that more fuel heating is expected for high inlet pressures. For all cases, CFD
 258 predictions are in good agreement with the 0-D model. These predictions were obtained by
 259 changing the needle lift (in the range 5-200 μ m) which in turn results in different values of the
 260 discharge coefficient. It is also important to notice that at the same valve lift, different
 261 discharge coefficient is predicted for constant and variable properties. At low lifts, the
 262 constant properties assumption leads to under-estimation of the discharge coefficient, while at
 263 high lift the constant properties assumption leads to discharge coefficient overestimation. The
 264 effect of two-phase flow is also presented in Fig.4 for the variable properties simulations. As
 265 seen, the discharge coefficient reduces relative to the single phase case due to the partial
 266 blockage of the flow from the bubbles, and also the fuel heating is slightly lower, since the
 267 friction forces are multiplied by the liquid volume fraction. The reduction of friction in
 268 cavitating flows has been also reported in (Payri et al., 2012) and (Javier López et al., 2012).



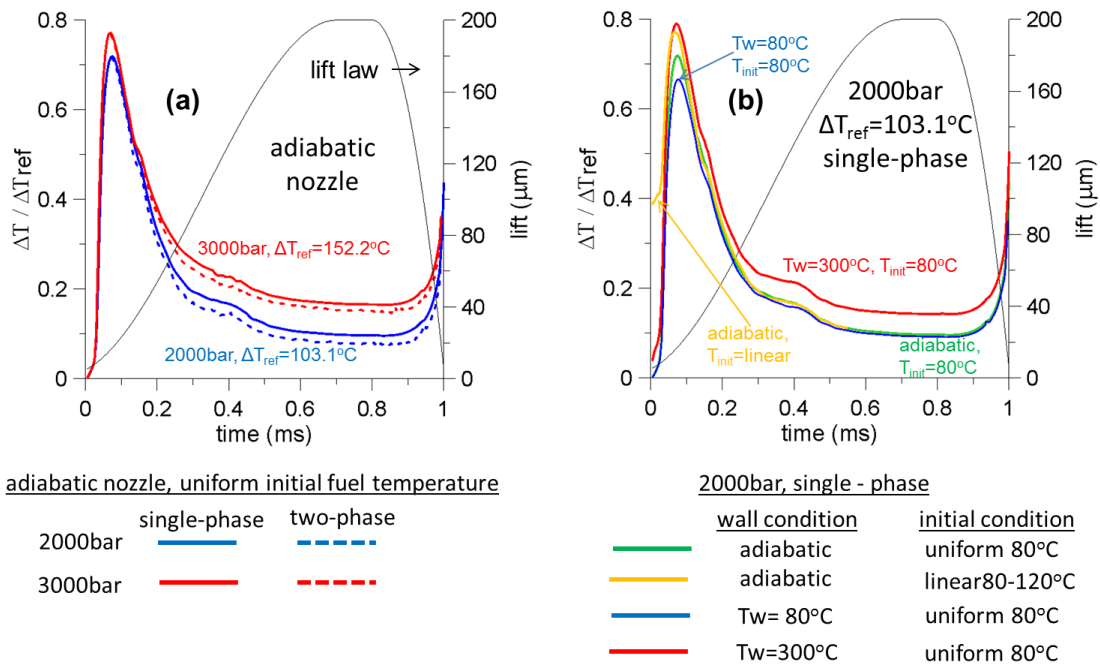
269

270 **Fig.4:** Dimensionless fuel heating for 2000bar (a) and 3000bar (b) inlet pressures. The effect
 271 of constant or variable properties assumption, as also the effect of two-phase flow is
 272 presented.

273

274 **3.3 Moving lift simulations**

275 In this section focus is given to the moving needle simulations which resemble a realistic fuel
276 injection event. In these cases the needle lift law plays an important role, since an injection
277 event has short duration and furthermore the closing phase is usually much shorter than the
278 opening phase. This results in different fuel heating levels during the opening and closing
279 phases of needle motion. In Fig.5a the needle lift law versus time is presented along with the
280 fuel heating for the case of adiabatic nozzle wall with uniform initial fuel temperature
281 distribution; two inlet pressures (2000 and 3000bar) are investigated for single and two phase
282 flows. A strong fuel heating is observed at the initial opening of the needle, which is almost
283 70% higher for the higher inlet pressure of 3000bars (but approximately the same in
284 dimensionless quantities in the order of $0.70-0.75\Delta T_{ref}$); the presence of the vapor phase does
285 not seem to have any noticeable effect on the degree of fuel heating. The mean fuel exit
286 heating reaches its maximum value at approximately $20\mu\text{m}$ needle lift. For the same needle
287 lift, different fuel mass flow quantities are injected from the nozzle in the opening and the
288 closing phases. This is due to the needle motion and the presence of the sac volume in which
289 fuel mass is accumulated in the opening phase; in the closing phase the downward motion of
290 the needle pushes the accumulated fuel mass from the sac volume to the nozzle's exit and
291 thus higher discharge coefficient is calculated. Integrating the instantaneous flow rate reveals
292 that the total mass of the fuel injected during the injection event examined, is 8.33 and
293 10.05mg/hole for 2000bar and 3000bar inlet pressure respectively, while a 2% reduction in
294 the overall mass injected was observed for the case of two-phase flow at 2000bar inlet
295 pressure.



296

297 **Fig.5:**(a) Lift law and fuel heating versus time; solid lines refer to single-phase flow and
 298 dashed lines refer to two-phase flow. (b) Effect of initial and boundary conditions for fuel
 299 heating.

300

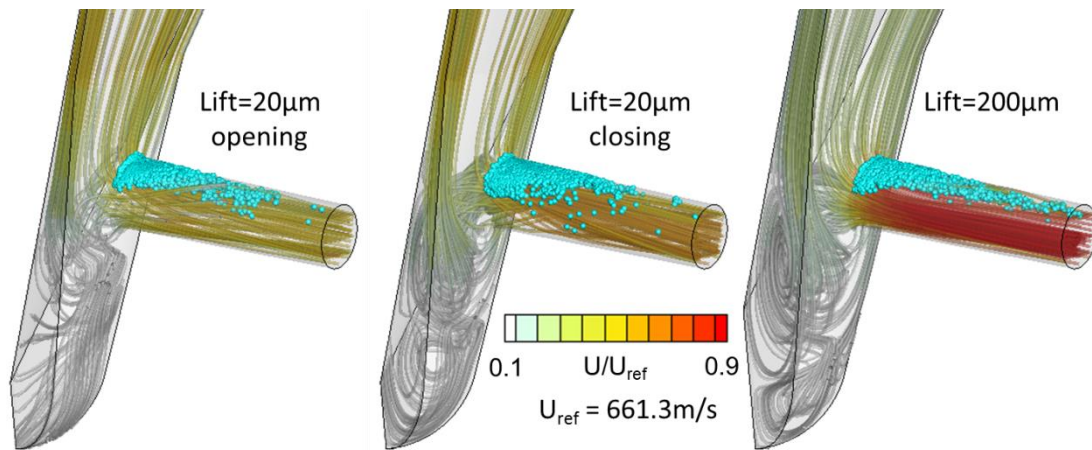
301 The effect of initial and boundary conditions is presented in Fig.5b for the case of 2000bar
 302 inlet pressure and single-phase flow. The case with uniform initial fuel temperature
 303 distribution and constant wall temperature equal to 300°C seems to enhance the fuel heating,
 304 but on the other hand this results to 0.25% overall mass flow reduction, which can be
 305 considered negligible. The case with constant wall temperature equal to the incoming fuel
 306 temperature (80°C) has a minor effect to the fuel heating compared to the adiabatic wall case;
 307 differences in maximum heating of $0.07\Delta T_{ref}$ compared to the adiabatic wall case are observed
 308 only at the initial stages of the opening phase. The case of adiabatic nozzle wall with a more
 309 realistic initial temperature distribution for the fuel is also presented in Fig.5b. The lower part
 310 of the fuel in the injector has initially a temperature of 120°C (equal to the fuel temperature at
 311 the end of the injection presented in Fig.5a), while the upper part of the fuel has the inlet
 312 temperature; between these two regions, a linear temperature distribution for the fuel was
 313 assumed. The curve corresponds to this case starts from a non-zero value, exhibits slightly
 314 higher maximum temperature, but the effect of initial condition is completely eliminated
 315 when the lift exceeds the $80\mu\text{m}$. So, the effect of initial temperature distribution is of minor
 316 importance under the assumptions made. On the other hand, a more realistic approach would
 317 apply a conjugate heat transfer solution between the flow and the injector solid material with

318 increased inlet fuel temperature; such an approach would require excessive computer
319 resources and it was beyond the scope of the paper.

320 **3.4 Flow field regimes**

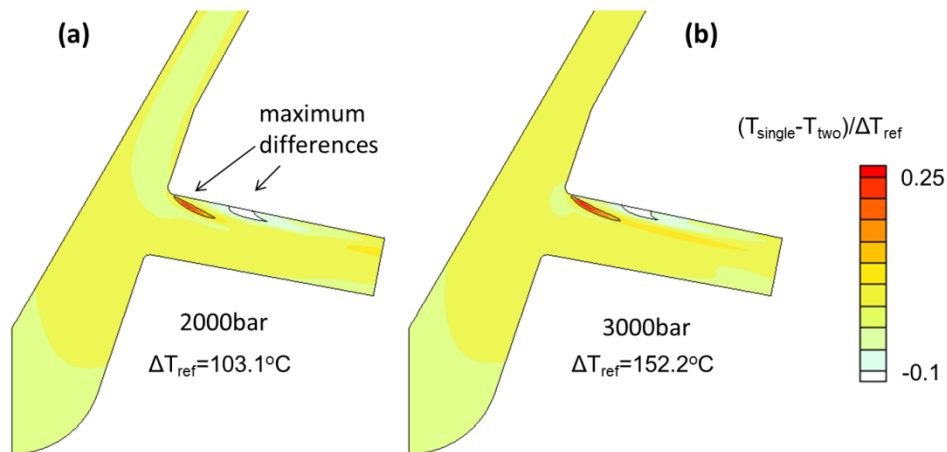
321 In this section the 3-D flow details are presented. In Fig.6 velocity streamlines colored with
322 the velocity magnitude for the case of two-phase flow with 2000bar inlet pressure are
323 presented along with sample cavitation bubbles; the velocity magnitude has been made non-
324 dimensional with the reference velocity (661.3m/s for the case of 2000bar inlet pressure) and
325 similar patterns are observed for the cases with 3000bar inlet pressure. Large vortical
326 structures are observed in the sac volume with a low velocity magnitude. Inside the injector
327 hole, the flow accelerates substantially reaching velocities of the order of 600m/s when the
328 full lift is considered; the present model accounts for compressibility effects (in subsonic
329 flows) as described in (Theodorakakos et al., 2014). At the inlet of the hole the flow turns
330 direction and aligns with the axis of the hole. As a result, the pressure drops below the
331 saturation pressure and bubbles are formed; under the influence of the velocity field, these are
332 carried towards the nozzle exit, while bubble collapse and coalescence also take place. For
333 this particular nozzle design examined, the needle motion does not seem to induce the Coanda
334 effect (Trancossi, 2011) and the flow is directed from the passage to the inlet of the hole. This
335 is attributed to the smooth needle profile, to the prolonged shape of the sac, as also to the
336 elevated position of the hole relative to the sac volume.

337 Turning now our interest into the temperature field, the case of the full needle opening of
338 200 μ m is initially examined since during an injection event the needle lift remains most of the
339 time at the full lift; furthermore no differences were identified between the steady state
340 simulations and the transient simulations for this needle lift case. The effect of two-phase
341 flow in the temperature predictions is presented in Fig.7a,b. In this figure the dimensionless
342 temperature difference between the case of single-phase flow and two-phase flow is
343 presented. As seen, the difference between these two fields is small (but locally may reach
344 values in the order of $0.25\Delta T_{ref}$) and does not alter the overall heating of the fuel as it was
345 shown in Fig.5a.



346

347 **Fig.6:** Dimensionless velocity field along with indicative cavitation bubbles for the case of
 348 two phase flow with 2000bar inlet pressure at 20µm lift (opening and closing phase) and
 349 200µm.

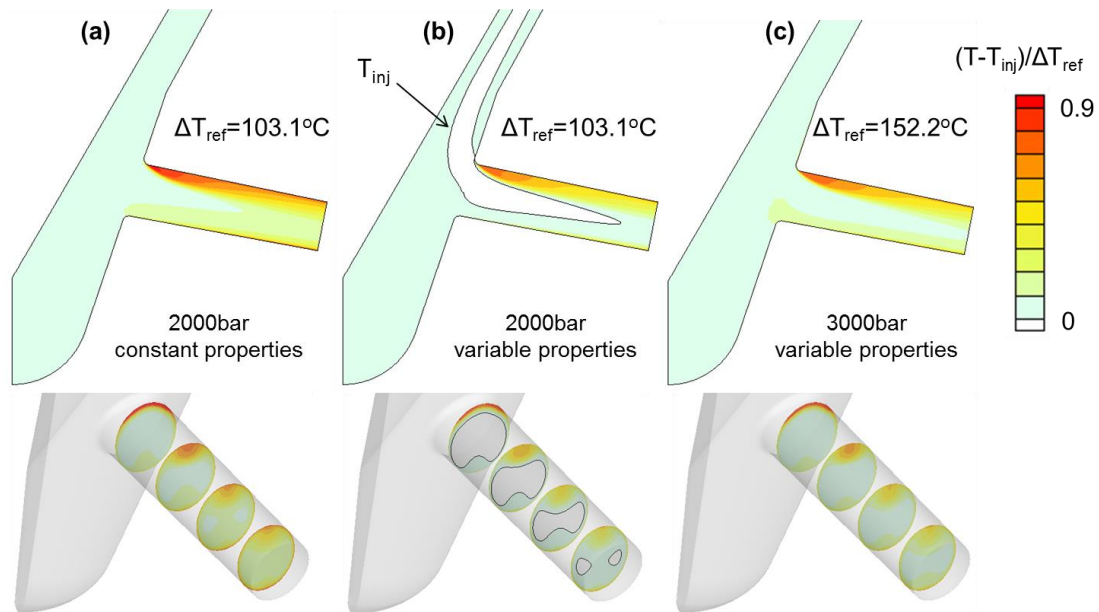


350

351 **Fig.7:** Dimensionless temperature difference between the single-phase and two-phase fields
 352 for the case of 200µm lift. (a) 2000bar inlet pressure, (b) 3000bar inlet pressure.

353 The effect of assuming constant or variable thermodynamic properties is presented in Fig.8
 354 for the cases of single-phase flow with either constant or variable properties with 2000bar
 355 inlet pressure as also the case of single-phase flow with variable properties at 3000bar at
 356 200µm needle lift. The maximum temperature is observed in the upper part of the inlet of the
 357 hole. In this region there are strong velocity gradients which induce friction and thus kinetic
 358 heating; for the case of variable properties the maximum dimensionless local temperature is
 359 approximately equal to ΔT_{ref} irrespective of the inlet pressure, while for the constant properties
 360 case the maximum dimensionless temperature is $1.26\Delta T_{ref}$. The variable properties case
 361 generally exhibits lower heating up values and additionally a sub-cooled liquid core is

362 observed even inside the injector nozzle which vanishes near the nozzle exit; this behavior is
 363 not observed in the case with 3000bar inlet pressure.

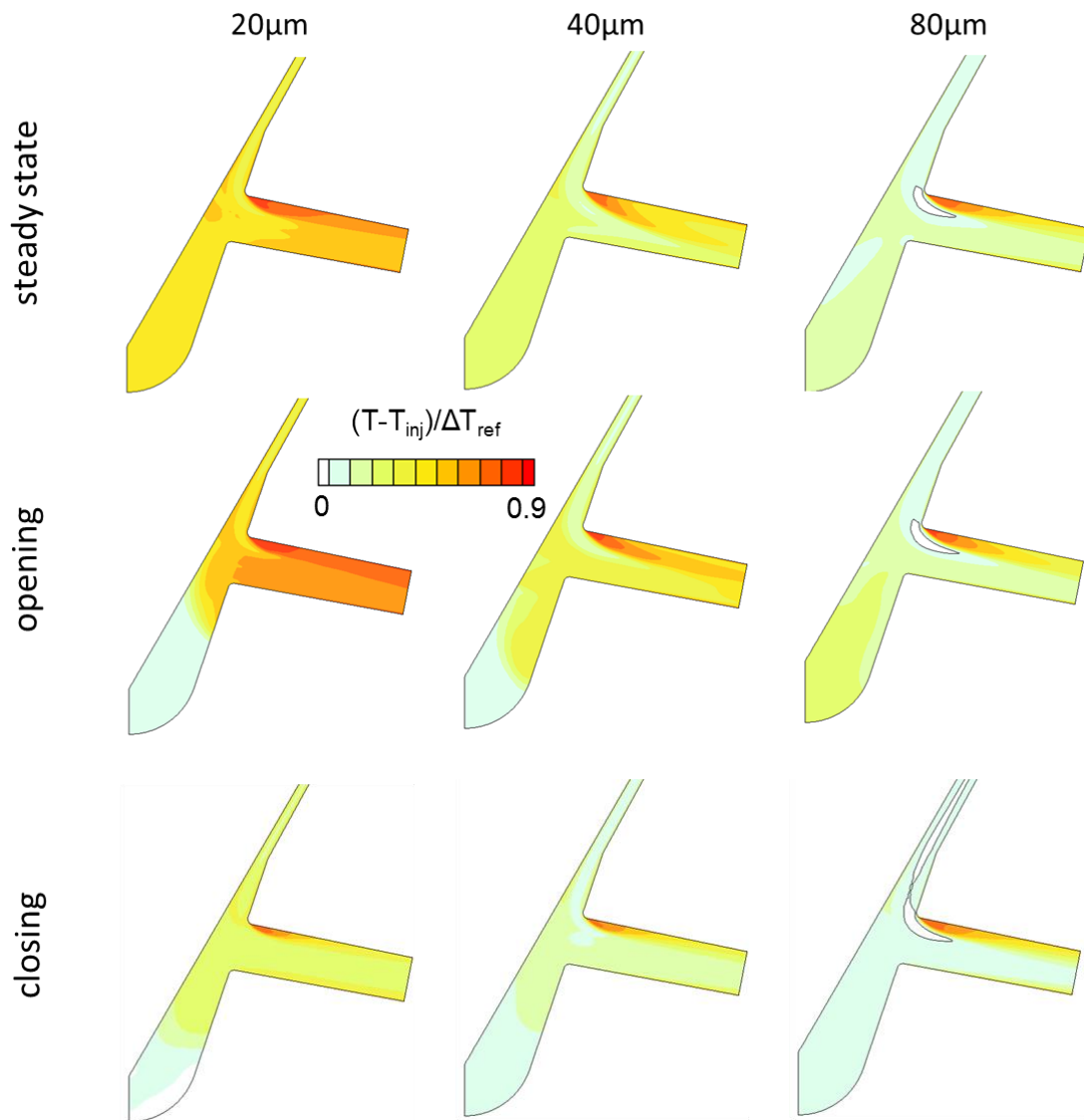


364

365 **Fig.8:** Dimensionless temperature field for the case of single phase flow at 200µm lift. (a)
 366 constant properties, 2000bar, (b) variable properties, 2000bar, (c) variable properties,
 367 3000bar.

368 The transient effects in fuel heating for the case of single phase with 2000bar inlet pressure
 369 are presented in Fig.9 in which a comparison between the predictions for the opening and
 370 closing phase is performed; steady state predictions are also presented. Contrary to the case of
 371 full lift, in all other needle lift positions the temperature field exhibits a different behavior in
 372 opening and closing phase, since the needle motion affects the velocity field due to fuel
 373 incompressibility which in turn affects the temperature field, while the temperature “history”
 374 plays also an important role. The differences between the opening and the closing phase
 375 become more intense when the lift is low; these differences tend to vanish near the full lift.
 376 Initially the fuel in the whole computational domain has a uniform temperature (equal to zero
 377 in non-dimensional units). As the needle opens, the fuel is heated as it flows in the passage
 378 (not shown in Fig.9) and tends to fill the sac volume. Progressively as the lift increases the
 379 fuel is heated to a lower degree and the sac volume is filled with a cooler liquid having the
 380 temperature of the inlet. In the closing phase the downward needle motion pushes the “cold”
 381 fuel from the sac volume towards the hole. This transfer of mass from the sac to the hole,
 382 along with the higher velocities observed due to the needle motion lead to the development of
 383 a thinner thermal boundary layer in the closing phase. Near the inlet of the hole, a sub-cooled
 384 region exists at 80µm needle lift due to fuel depressurization, while this region is more

385 evident in the closing phase. In the absence of Coanda effect, the maximum fuel temperature
 386 is always observed at the upper part of the inlet of hole, reaching values in the order of ΔT_{ref} .



387

388

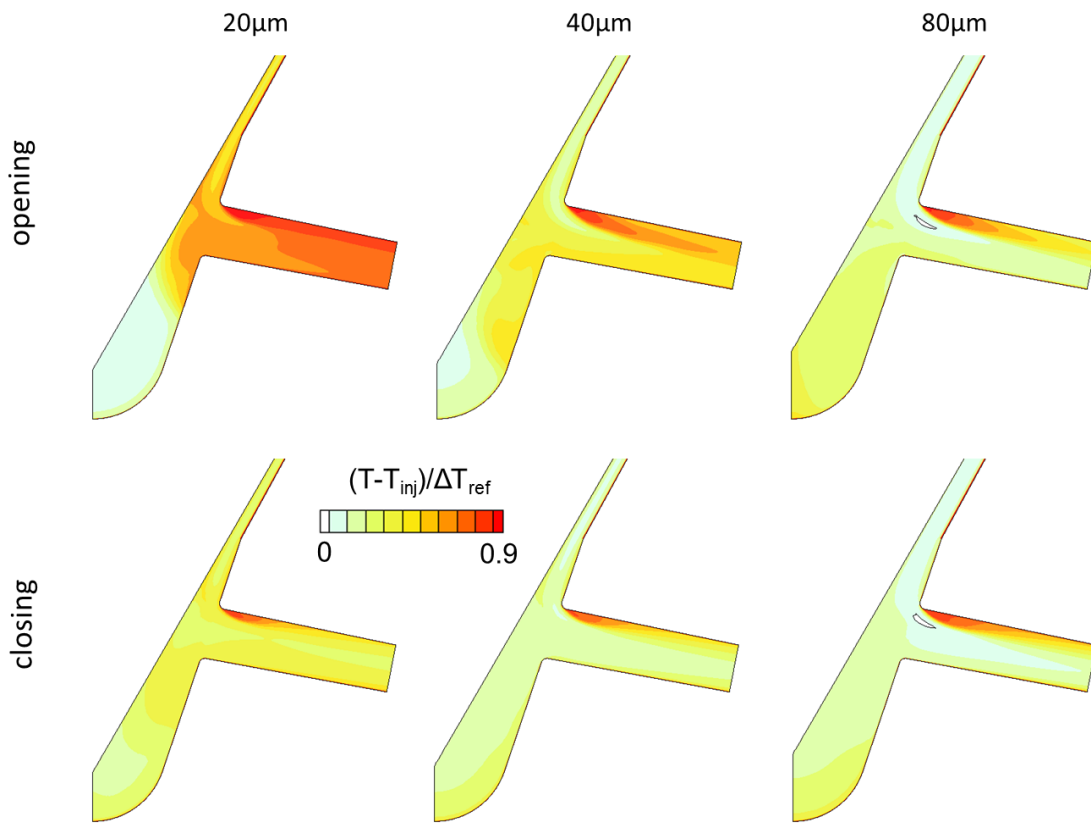
389 **Fig.9:** Temporal evolution of the dimensionless temperature field for the adiabatic nozzle.
 390 Comparison between the opening and the closing phases. Corresponding predictions from
 391 steady state simulations are also presented. 2000bar, single-phase, variable properties.

392

393 It is also of interest to examine the temperature field for the case of constant surface
 394 temperature equal to 300°C. This is presented in Fig.10 for the case of 2000bar inlet pressure
 395 with variable properties. As seen there is a thermal boundary layer developing near the wall
 396 which affects the temperature distribution, mainly in the sac area, when compared with the
 397 corresponding temperature field presented in Fig.9 for the case of adiabatic wall.

398 Furthermore, the sub-cooled region is suppressed for the constant temperature case, while it is
 399 evident that the needle motion affects the temperature field especially at low lifts. Regarding
 400 the case with constant surface temperature equal to 80°C, the temporal evolution of
 401 dimensionless temperature in the opening phase is presented in Fig.11. This case evolves with
 402 the same fashion as the case with adiabatic nozzle wall and differences are observed only at
 403 the initial stages of the opening phase.

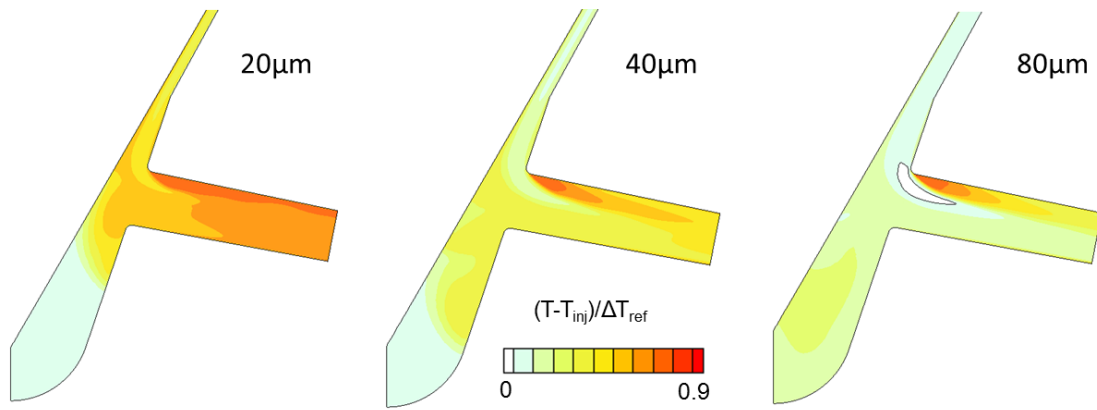
404



405

406 **Fig.10:** Temporal evolution of the dimensionless temperature field for the nozzle with
 407 constant surface temperature (300°C). Comparison between the opening and the closing
 408 phases for the case of 2000bar inlet pressure.

409

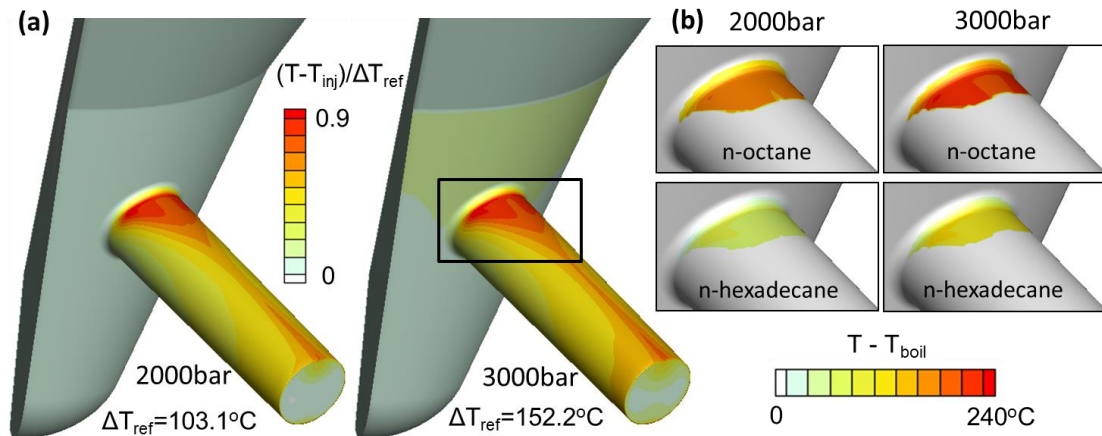


410

411 **Fig.11:** Temporal evolution of the dimensionless temperature field for the nozzle with
 412 constant surface temperature (80°C, opening phase).

413

414 Under the assumption of adiabatic nozzle wall adopted here, the surface of the nozzle may
 415 reach high enough temperatures to induce the onset of heterogeneous boiling. Despite the fact
 416 that the present methodology does not account for such phenomena, an estimation of the
 417 boiling region can be performed by calculating the fuel boiling point according to the local
 418 pressure field and subtract it from the local temperature field. Since Diesel fuel consists of
 419 several components, a light (n-octane C_8H_{18}) and a heavy (n-hexadecane $C_{16}H_{34}$) have been
 420 chosen to estimate the boiling regions of these two components. In Fig.12a the nozzle surface
 421 temperature field for the case of two-phase flow with 2000bar and 3000bar inlet pressure at
 422 200μm is presented. In Fig.12b the enlarged images represent the heterogeneous boiling
 423 regions for two Diesel species at the two corresponding inlet pressures. As seen, there is a
 424 region with a superheat degree ranging from 80 to 240K depending on the component type
 425 and inlet pressure. It is worth mentioning that this region on the nozzle's wall surface is quite
 426 close to the region inside the fluid volume in which cavitation occurs and bubbles are created
 427 but it is not identical.



428

429 **Fig.12:**(a) Dimensionless surface temperature, (b) boiling overheat for a light and a heavy
 430 Diesel component. Two phase flow at $200\mu\text{m}$ lift for the cases of 2000 and 3000bar inlet
 431 pressure.

432

433 4 Conclusions

434 A CFD model accounting for cavitation and thermal effects has been employed for
 435 investigating the flow and temperature field in high pressure Diesel injector nozzles.
 436 Cavitation is considered through a coupled Eulerian-Lagrangian formulation in which the
 437 temperature field is obtained via the solution of the total enthalpy equation accounting for the
 438 viscous heating effects. The thermal model has been initially validated against a 0-D
 439 thermodynamic model for an adiabatic nozzle showing a good performance for a wide range
 440 of steady lift positions. The effect of using constant or variable properties has been quantified
 441 revealing that the constant properties assumption may lead to large deviations in discharge
 442 coefficient and fuel heating predictions, especially in high pressure conditions in which fuel
 443 depressurization may lead to fuel sub-cooling. Transient simulations for moving lift cases
 444 have shown that the needle motion and the temperature history have a serious impact in
 445 predictions and steady lift simulations cannot represent the actual phenomenon, especially at
 446 low lifts. Temperature field exhibits differences in opening and closing phase which
 447 progressively diminish as the lift increases. The effect of cavitation is to reduce the flow rate
 448 due to blockage of the flow by the bubbles and reduce the fuel heating due to friction
 449 reduction. Finally, possible heterogeneous boiling regions have been identified for typical
 450 Diesel components, showing that the boiling region is very close to the cavitation region.

451

452 **5 Acknowledgements**

453 1. The research leading to these results has received funding from the People
454 Programme (Marie Curie Actions) of the European Union's Seventh Framework
455 Programme FP7/2007-2013/ under REA grant agreement n. 324313.

456 2. The scientific developments on the numerical solution of the RANS equations and the
457 modeling of turbulence, scientific advances to which the honored scientists in the
458 present volume, Professors B. E. Launder and K. Hanjalic have substantially
459 contributed, made possible the reliable engineering solution of complex flows and the
460 CFD as an industrial design tool.

461

462 **6 Nomenclature**

463

464 **Romansymbols**

Symbol	Description	Units
A	area	m^2
C_d	Discharge coefficient	-
c_p	Heat capacity	J/kgK
c_{pmT}	Mean heat capacity	J/kgK
D	diameter	μm
g	Gravity acceleration	m/s^2
h	enthalpy	J/kg
\mathbf{I}	Unit tensor	-
k	Turbulent kinetic energy	m^2/s^2
p	pressure	Pa
Pr	Prandtl number	-
S_h	Source term	W/m^3
T	temperature	K
t	time	s
U, \mathbf{u}	velocity	m/s

465

466

467 **Greeksymbols**

Symbol	Description	Units
a	Volume fraction	-
κ	Thermal conductivity	W/mK
μ	viscosity	kg/ms
ρ	density	kg/m^3
τ	Stress tensor	N/m^2

468

Subscripts

Symbol	Description
0	At reference point
add	additional

eff	effective
in	inlet
init	initial
inj	injection
L	liquid
lam	Laminar
m	mean
out	outlet
single	Single phase
tot	total
turb	turbulent
two	Two phase
w	wall

469

470 **References**

- 471 Alajbegovic, A., Grogger, H.A., Philipp, H., 1999. Calculation of Transient Cavitation in
472 Nozzle Using the Two-Fluid Model, Proc. 12th ILASS, Indianapolis, Indiana, USA.
- 473 Ando, K., Colonius, T., Brennen, C., 2011. Numerical simulation of shock propagation in a
474 polydisperse bubbly liquid. *International Journal of Multiphase Flow* 37, 596–608.
- 475 Andriotis, A., Gavaises, M., C., A., 2008. Vortex flow and cavitation in diesel injector
476 nozzles. *Journal of Fluid Mechanics* 610, 195–215.
- 477 Arcoumanis, C., Badami, M., Flora, H., Gavaises, M., 2000. Cavitation in Real-Size Multi-
478 Hole Diesel Injector Nozzles. SAE Paper 2000-01-1249.
- 479 Badock, C., Wirth, R., Fath, A., Leipertz, A., 1999. Investigation of cavitation in real size
480 diesel injection nozzles. *International Journal of Heat and Fluid Flow* 20, 538-544.
- 481 Battistoni, M., Grimaldi, C.N., 2012. Numerical analysis of injector flow and spray
482 characteristics from diesel injectors using fossil and biodiesel fuels. *Applied Energy* 97, 656-
483 666.
- 484 Blessing, M., König, G., Krüger, C., Michels, U., Schwarz, V., 2003. Analysis of Flow and
485 Cavitation Phenomena in Diesel Injection Nozzles and its Effect on Spray and Mixture
486 Formation. SAE Paper 2003-01-1358.
- 487 Brennen, C.E., 1995. *Cavitation and Bubble Dynamics*. Oxford University Press, New York;
488 Oxford.
- 489 Chaves, H., Knapp, M., Kubitzek, A., Obermeier, F., 1995. Experimental Study of Cavitation
490 in the Nozzle Hole of Diesel Injectors Using Transparent Nozzles. SAE Paper 950290.

491 Chen, Y.L., Heister, S.D., 1995. Two-Phase Modeling of Cavitated Flows. *Computers &*
492 *Fluids* 24, 799-809.

493 ExxonMobil, The outlook for energy: A view to 2040.

494 Fuster, D., Colonius, T., 2011. Modelling bubble clusters in compressible liquids. *Journal of*
495 *Fluid Mechanics* 1, 1–38.

496 Giannadakis, E., Gavaises, M., Arcoumanis, C., 2008. Modelling of cavitation in diesel
497 injector nozzles. *Journal of Fluid Mechanics* 616, 153–193.

498 Giannadakis, E., Gavaises, M., Roth, H., Arcoumanis, C., 2004. Cavitation Modelling in
499 Single-Hole Diesel Injector Based on Eulerian-Lagrangian Approach, Proc. THIESEL
500 International Conference on Thermo- and Fluid Dynamic Processes in Diesel Engines,
501 Valencia, Spain.

502 Giannadakis, E., Papoulias, D., Gavaises, M., Arcoumanis, C., Soteriou, C., Tang, W., 2007.
503 Evaluation of the predictive capability of diesel nozzle cavitation models. SAE Technical
504 Paper 2007-01-0245.

505 Goud M, Greif D, Suffa M, Winklhofer E, Gill D, 2012. Virtual erosion prediction, design
506 optimisation and combustion system integration of high pressure fuel injector systems.
507 IMechE Conf Fuel Injection Systems for IC Engines, London.

508 Habchi, C., Dumont, N., Simonin, O., 2008. Multidimensional simulation of cavitating flows
509 in diesel injectors by a homogeneous mixture modeling approach. *Atom. Sprays* 18, 129-162.

510 He, Z., Zhong, W., Wang, Q., Jiang, Z., Fu, Y., 2013. An investigation of transient nature of
511 the cavitating flow in injector nozzles. *Applied Thermal Engineering* 54, 56-64.

512 Hilgenfeldt, S., Brenner, M.P., Grossmann, S., Lohse, D., 1998. Analysis of rayleigh-plesset
513 dynamics for sonoluminescing bubbles. *Journal of Fluid Mechanics* 365, 171–204.

514 Jamaluddin, A., Ball, G., Turangan, C., Leighton, T., 2011. The collapse of single bubbles
515 and approximation of the far-field acoustic emissions for cavitation induced by shock wave
516 lithotripsy. *Journal of Fluid Mechanics* 677, 305–341.

517 Javier López, J., Salvador, F.J., de la Garza, O.A., Arrègle, J., 2012. A comprehensive study
518 on the effect of cavitation on injection velocity in diesel nozzles. *Energy Conversion and*
519 *Management* 64, 415-423.

520 Keller, J., Miksis, M., 1980. Bubble oscillations of large amplitude. *Journal of the Acoustical*
521 *Society of America* 68, 628–633.

522 Kolev, N., 2002. *Multiphase Flow Dynamics 3: Turbulence, Gas Absorption and Release,*
523 *Diesel Fuel Properties.* Springer.

524 Launder, B.E., Spalding, D.B., 1974. The numerical computation of turbulent flows.
525 *Computer Methods in Applied Mechanics and Engineering* 3, 269-289.

526 Lee, W.G., Reitz, R.D., 2010. A Numerical Investigation of Transient Flow and Cavitation
527 Within Minisac and Valve-Covered Orifice Diesel Injector Nozzles. *J. Eng. Gas Turbines*
528 *Power* 132.

529 Liu, T.G., Khoo, B.C., Xie, W.F., 2004. Isentropic one-fluid modelling of unsteady cavitating
530 flow. *Journal of Computational Physics* 201, 80-108.

531 Margot, X., Hoyas, S., Fajardo, P., Patouna, S., 2010. A moving mesh generation strategy for
532 solving an injector internal flow problem. *Mathematical and Computer Modelling* 52, 1143-
533 1150.

534 Neroorkar, K.D., Mitcham, C.E., Plasas, A.H., Grover, R.O., Schmidt, D.P., 2012.
535 *Simulations and Analysis of Fuel Flow in an Injector Including Transient Needle Effects,*
536 *ILASS-Americas, San Antonio.*

537 Payri, F., Bermúdez, V., Payri, R., Salvador, F.J., 2004. The influence of cavitation on the
538 internal flow and the spray characteristics in diesel injection nozzles. *Fuel* 83, 419-431.

539 Payri, F., Margot, X., Patouna, S., Ravet, F., Funk, M., 2009. A CFD Study of the Effect of
540 the Needle Movement on the Cavitation Pattern of Diesel Injectors. *SAE Technical Paper*
541 2009-24-0025.

542 Payri, F., Payri, R., Salvador, F.J., Martínez-López, J., 2012. A contribution to the
543 understanding of cavitation effects in Diesel injector nozzles through a combined
544 experimental and computational investigation. *Computers & Fluids* 58, 88-101.

545 Payri, R., Salvador, F.J., Gimeno, J., Venegas, O., 2013. Study of cavitation phenomenon
546 using different fuels in a transparent nozzle by hydraulic characterization and visualization.
547 *Experimental Thermal and Fluid Science* 44, 235-244.

548 Prosperetti, A., Hao, Y., 1999. Modelling of spherical gas bubble oscillations and
549 sonoluminescence. *Philosophical Transactions of the Royal Society of London Series A-*
550 *Mathematical Physical and Engineering Sciences* 357, 203-223.

551 Salvador, F.J., Martínez-López, J., Romero, J.V., Roselló, M.D., 2013. Computational study
552 of the cavitation phenomenon and its interaction with the turbulence developed in diesel
553 injector nozzles by Large Eddy Simulation (LES). *Mathematical and Computer Modelling* 57,
554 1656-1662.

555 Singhal, A.K., Athavale, M.M., Li, H.Y., Jiang, Y., 2002. Mathematical basis and validation
556 of the full cavitation model. *Journal of Fluids Engineering-Transactions of the ASME* 124,
557 617-624.

558 Soteriou, C., Andrews, R., Smith, M., Torres, N., Sankhalpara, S., 2000. The flow patterns
559 and sprays of variable orifice nozzle geometries for diesel injection. SAE Paper 2000-01-
560 0943.

561 Städtke, 2007. *Gasdynamic Aspects of Two-Phase Flow*. Wiley-VCH Verlag GmbH and Co.
562 KGaA.

563 Strotos, G., Koukouvinis, P., Theodorakakos, A., Gavaises, M., 2014a. Quantification of
564 Friction-induced Heating in tapered Diesel orifices, SIA POWERTRAIN, Rouen, France.

565 Strotos, G., Koukouvinis, P., Theodorakakos, A., Gavaises, M., Wang, L., Li, J., McDavid,
566 R.M., 2014b. Fuel heating in high pressure diesel nozzles, THIESEL, Valencia, Spain.

567 Theodorakakos, A., Mitroglou, N., Strotos, G., Atkin, C., Gavaises, M., 2014. Friction-
568 induced heating in nozzle hole micro-channels under extreme fuel pressurisation. *FUEL* 123,
569 143-150.

570 Trancossi, M., 2011. An Overview of Scientific and Technical Literature on Coanda Effect
571 Applied to Nozzles. SAE Technical Paper 2011-01-2591.

572 Versteeg, H.K., Malalasekera, W., 2007. *An Introduction to Computational Fluid Dynamics:*
573 *The finite volume method (2nd Edition)*.

574 Wolfshtein, M., 1969. The velocity and temperature distribution in one-dimensional flow with
575 turbulence augmentation and pressure gradient. *International Journal of Heat and Mass*
576 *Transfer* 12, 301-318.

577 Yuan, W., Schnerr, G.n.H., 2004. Numerical Simulation of Two-Phase Flow in Injection
578 Nozzles: Interaction of Cavitation and External Jet Formation. *Journal of Fluids Engineering*
579 125, 963-969.

580 Zeravic, Z., Lohse, D., Saarloos, W., 2011. Collective oscillations in bubble clouds. *Journal*
581 *of Fluid Mechanics* 680, 114–149.

582 Zhao, H., Quan, S., Dai, M., Pomraning, E., Senecal, P.K., Xue, Q., Battistoni, M., Som, S.,
583 2013. Validation of a Three-Dimensional Internal Nozzle Flow Model Including Automatic
584 Mesh Generation and Cavitation Effects, ASME 2013 Internal Combustion Engine Division
585 Fall Technical Conference.

586

587