Potential heterogeneous and homogeneous flow boiling conditions in a high-pressure diesel fuel injector

F. Villa[1,2], A. Georgoulas[2,3], R. Salemi[2,3], R. M. McDavid[4], M. Gavaises[5], P. Koukouvinis[5], M. Marengo[2,6]

[1] Laboratoire de Physique des Surfaces et Interfaces, Université de Mons
Avenue Victor Maistriau 19, 7000 Mons, Belgium
[2] Department of Engineering and Applied Sciences, University of Bergamo
viale Marconi 5, I-24044 Dalmine, Italy
[3] Large Power Systems Division Caterpillar UK Engines Co. Ltd,
Peterborough, PE1 5FQ UK
[4] Energy and Transportation Research, Product Development and Global Technology,
Caterpillar Inc., Mossville, IL 61552, USA
[5] School of Engineering and Mathematical Sciences,
City University, London EC1V 0HB, UK
[6] School of Computing, Engineering and Mathematics, Cockcroft Building, C531, Lewes Road
University of Brighton, Brighton BN2 4GJ, UK

The development of direct injection (DI) Diesel engines over the last 20 years has been quite remarkable. The trend in technology is towards diesel fuel pressurisation in the excess of 250 to 300 MPa thus leading to simultaneous reduction of soot and NOx emissions. At high diesel fuel pressures, the excessively high velocity values occurring during the discharge of the liquid through the fuel injector nozzles (up to 660 m/s with a maximum Reynolds number of $1.3 \bullet 10^5$ observed in this study) induce high wall friction, which in turn leads to significant fuel heating with Brinkman number values ranging from 20 to 60. The combination of significant pressure reduction within the sac volume (during low needle lifts) and the nozzles, together with high fuel injector wall temperatures occurring by both heat conduction from the cylinder head and heat convection from the hot gases within the engine cylinder, may potentially lead to heterogeneous flow boiling of the diesel liquid close to the fuel injector walls. In this paper a non-isothermal study of diesel flow within a fuel injector is carried out. The extensive engine cylinder data available from both test results and combustion simulations are characterized to set up the appropriate thermal boundary conditions for the conjugate heat transfer (CHT) simulations carried out in this study. The thermal boundary conditions for the fuel injector simulations are affected by both the thermal state within the engine cylinder (which in turn is primarily affected by combustion) and hence the temperature within the cylinder head. Due to the transient nature of the engine cylinder temperature and heat transfer variations, a sensitivity analysis is performed in order to better understand the effects of such variations on the nozzle wall temperature. The steady state CHT simulations of the fuel injector are carried out, using ANSYS Fluent®, based on the assumption of single phase flow and constant physical properties of the diesel liquid. From the results of CHT simulations, a comparison between the wall nozzle temperature and the saturation temperature of the diesel liquid inside the nozzle, are presented. The second set of simulations presented here use a typical constant wall temperature of 180C for the fuel injector as the boundary conditions for the non-isothermal two phase cavitating flow simulations based on the assumption of variable properties of diesel liquid with respect to both temperature and pressure. These simulations were carried out using City University's CFD code (GFS). Overall, the simulations reveal that at certain locations within the fuel injector geometry (mainly close to the fuel injector walls), the diesel liquid temperature reaches values well above the saturation temperature of the flowing diesel liquid, thus implying the occurrence of heterogeneous flow boiling regions within the fuel injector used for this study.

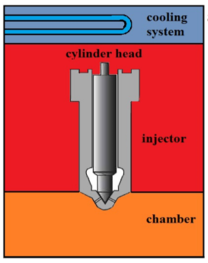
Keywords: Thermal simulation, Heterogeneous Boiling, Diesel injector, Heat transfer, Flow Boiling.

Introduction

The development of direct injection (DI) Diesel engines over the last 20 years has been quite remarkable. The trend in technology is towards fuel pressurisation in the excess of 250 to 300 MPa, thus leading to simultaneous reduction of soot and NOx emissions and hence less demand on the performance and the cost of the after-treatment systems. Typical CFD simulations of both single phase and two-phase cavitating flow usually assume isothermal flow conditions, justified on the basis that the residence time of the diesel liquid within the injection holes is so short that heat transfer to or from the surrounding could be neglected. However, at high inlet fuel pressures, the excessively high velocity values occurring during the discharge of the diesel liquid through the fuel injector nozzles can induce wall friction. This in turn leads to such significant viscous heating that could not possibly be neglected. This together with significant depressurisation occurring within the fuel injector nozzles (or across the injector needle gap at very low lifts), may potentially lead to heterogeneous boiling of the diesel liquid near the wall surface. Additionally the temperature rise of the bulk of the liquid above the saturation temperature may also lead to conditions for homogenous boiling within the flow field. In the literature, Caira et al present the results of their pool boiling experiments at pressure values of up to 70 bar [1]. A higher pressure was achieved by Sakashita and Ono [2] who proposed a correlation for the heat transfer coefficient for boiling at pressures up to 90 bar. However, no data are currently available for higher pressure values, such as those occurring in fuel injection systems, and therefore there is currently no appropriate flow boiling model available for simulating diesel flow under such high pressure values commonly seen in fuel injection systems. In this paper a non-isothermal study of the early development phase of a fuel injector design is carried out. The geometry was provided by the industrial partner for this project [3]. The present paper starts from a description of the thermal conditions within the engine cylinder, using the data available from previous experiments and/or combustion simulations. The boundary conditions for the CHT simulations of the fuel injector are defined mainly by the temperature and the heat transfer coefficient inside the engine cylinder and hence the temperature within the cylinder head. Due to the transient nature of the engine cylinder temperature and heat transfer variations, a sensitivity analysis is performed in order to better understand the effects of such variations on the nozzle wall temperature. The steady state conjugate heat transfer (CHT) simulation of the fuel injector was carried out, using ANSYS Fluent®.

Applied Boundary and Initial Conditions

The diesel fuel injector is in contact with various parts of a diesel engine. All these parts, influence the temperature distribution of the diesel injector wall surfaces and hence the diesel liquid. As it is depicted in Figure 1a, the fuel injector lies within the cylinder head, with the injector tip (the end of the injector) being in direct contact with the combustion chamber (engine cylinder). Additionally, the cylinder head incorporates a cooling system, which is designed to cool it down and reduce its temperature. The heat transfer mechanism that is encountered between the above components is illustrated schematically in Figure 1b. Heat is transferred from the hot gasses inside the combustion chamber, onto the cylinder head and the injector tip by convection (T_{CH}, HTC_{CH}). After the initial warm up period for the engine, the cylinder head reaches a steady-state temperature condition, due to the balance between the heat transfer from the combustion chamber (T_{CH}, HTC_{CH}) and the cooling provided by the cooling system ($T_{CoolSys}$, $HTC_{CoolSys}$). Therefore, in the steady state CHT simulations presented here, the cylinder head is treated as a constant temperature solid body ($T_{cylHead}$). The cylinder head temperature $T_{CylHead}$ and the convective heat transfer from inside the combustion chamber (T_{CH}, HTC_{CH}) provide the main necessary boundary conditions for the conjugate heat transfer simulations (CHT) presented here.



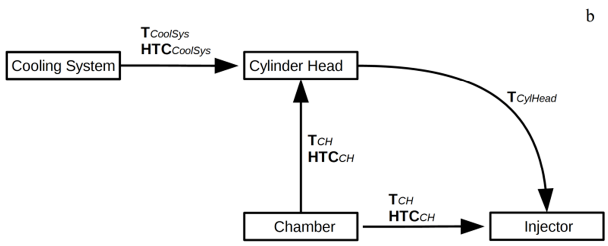


Figure 1. a) Schematic of the system b) Heat transfer between the various parts of the engine.

The thermophysical properties of two solid components of the fuel injector (i.e. the injector tip and the needle) that are involved in the **CHT** simulations are assumed to be constant as summarised in **Errore. L'origine riferimento non è stata trovata.**.

Table 1. Thermophysical properties for the injector tip and needle.

Part	Material	k [W/mK]	Cp [J/kgC]	ρ [kg/m^3]
Needle	AISI 52100	46.6	475	7810
Injector tip	AISI tool steel H10	42.7	477	7810

However, the thermo physical properties of the diesel liquid vary considerably as functions of both pressure and temperature **[4]**. As previously mentioned, before conducting more elaborate CFD simulations with variable fluid properties, the preliminary steady state CHT simulations are performed with the properties of the diesel liquid kept constant with respect to both the temperature and pressure. For these simulations, the fuel properties were evaluated at fuel injector inlet pressure of P_{inj}=240 MPa and T=400 K. The influence of variable fuel properties on the flow field results is explained in the next section of this paper.

Table 2. Thermophysical properties of the diesel liquid at constant temperature (T=400 K)

Fluid	k [W/mK]	Cp [J/KgC]	ρ [Kg/m3]	μ [Kg/m·s]
Diesel	0.137	2426	820.3	1.55E-3

Conjugate heat transfer simulation set-up

The computational domain that is utilized for the steady state CHT simulation is extracted from a three-dimensional geometry of the early development phase of a diesel fuel injector. In order to minimise the computational time while preserving the accuracy of simulations, a 72° section of the actual multi-hole diesel injector encompassing only one of the five injector nozzles, is used for the numerical simulations carried out in this research study. As it can be observed from Figure 2a, the computational domain consists of two distinct parts; a solid part representing the injector tip, and a fluid part that represents the diesel liquid between the injector tip and the needle (the needle lift is fixed at 100 μm).

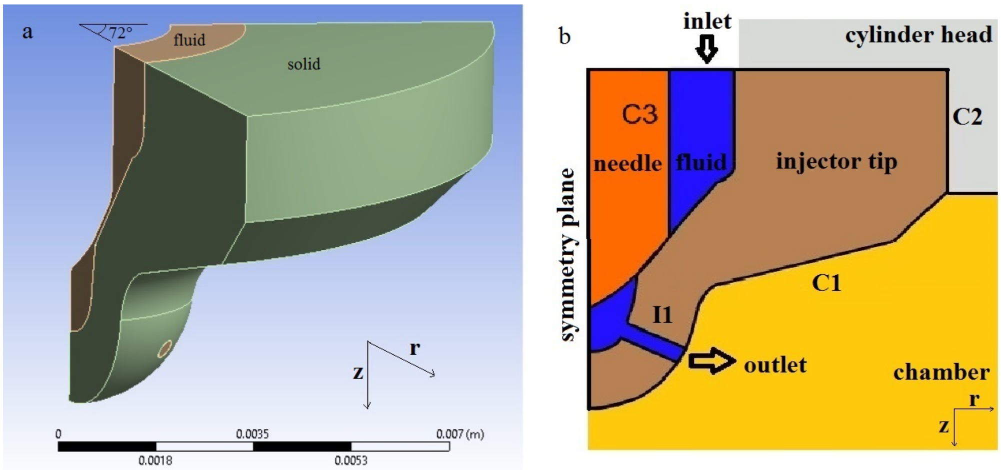


Figure 2. a) The computational domain b) The locations of the boundary conditions

The boundary conditions applied to various locations depicted in Figure 2b are summarised below:

- **C1:** The heat transfer between the combustion chamber and injector tip is due to the convection and radiation from the hot gases inside the engine cylinder. For this purpose, a convective/radiative heat transfer boundary condition is applied on the proposed surface, by using a typical mean hot gas temperature of T_{CH}=700K, a typical heat transfer coefficient of HTC_{CH}=1000 W/m^{2}K, and the C1 surface emissivity of ε=0.8[3].
- **C2:** For this surface a fixed temperature value of $T_{CylHead}$=400K is prescribed, representing the temperature of the cylinder head [3].
- **C3:** The temperature of the needle wall is fixed at T_{needle}=353K. The needle, surrounded by the flowing diesel fluid, has been assumed to readjust to the temperature of the fluid T_{inlet}.
- **Inlet:** The pressure and temperature of the diesel liquid at the inlet are assumed to have constant values of P_{inj}=240 MPa and T_{inlet}=353K, respectively [3].
- **Outlet:** The pressure and temperature of the diesel liquid at the outlet are assumed to have constant values of P_{out}=12 MPa and T_{outlet}=700K [3].

The constructed computational mesh consists of about two million tetrahedral cells within both the fluid and solid parts of the model. In the fluid part the minimum computational cell size is 60μm while in the solid part the minimum cell size is 100μm. An inflation region consisting of parallel layers of prismatic cells is also inserted on the surface of the fluid part that is in contact with the solid part.

Effect of boundary conditions variation on the injector nozzle wall temperature

Before starting the steady state conjugate heat transfer CFD simulation, a parametric analysis is performed, in order to evaluate the sensitivity of the assumed boundary conditions on the simulation results. This study was carried out by conducting a Finite Element Method heat transfer simulation on the solid part only (injector tip).

The parametric analysis examines the parameters shown in **Table 3**. Each of the examined parameters is varied within the prescribed range shown below.

Table 3. The range of variations of the parameters used in the sensitivity analysis

Parameter	Name	Minimum	Base case	Maximum
T_{CH}	Chamber (engine cylinder) temperature [K]	490	700	910
HTC_{CH}	Heat transfer coefficient of the chamber [W/m^{2}K]	700	1000	1300
$T_{CylHead}$	Temperature of the cylinder head [K]	370	400	430
ε	Emissivity of the TIP surface	0.56	0.8	1
T_{inlet}	Temperature of the fuel at the injector inlet [K]	323	353	383

The output of the parametric analysis is the value of θ, defined as:

$$\theta = \frac{T^W_{base} - T^W_{test}}{T^W_{base}} \qquad (1)$$

$T^W_{base} - T^W_{test}$ is the wall temperature difference between the base case and the test case (shown in **Errore. L'origine riferimento non è stata trovata.**). To normalize the results the above difference is divided by T^W_{base}. T^W_{base} and T^W_{test} are the average temperatures of the solid part along line 1 shown in **Errore. L'origine riferimento non è stata trovata.**. Figure 3 shows the impacts of the variations of each boundary condition parameter (within the range specified in Table 3) on the value of θ.

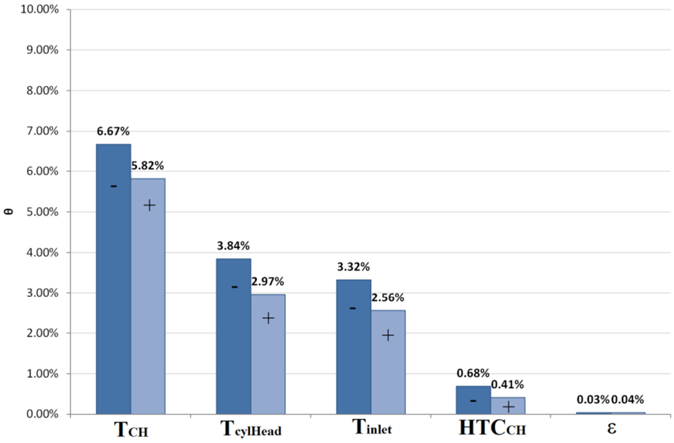


Figure 3. Results of the sensitivity analysis. The dark blue bars refer to the minimum values of the parameter ranges, while the light blue bars refer to the maximum values of the parameter ranges.

The results of the parametric analysis reveal that the T_{CH}, $T_{CylHead}$ and T_{inlet} are the most influential boundary condition parameters on the value of the nozzle wall temperature T^W.

Results of conjugate heat transfer simulation

This section of the paper is devoted to the results of the steady state CHT simulations assuming constant diesel liquid properties with respect to both the pressure and temperature variations. The proposed simulations aim at predicting the temperature variation along the fuel injector nozzle wall (T^W). T^W is evaluated again along line 1 (solid) as illustrated in **Errore. L'origine riferimento non è stata trovata.**a. These wall temperatures are compared with the corresponding saturation temperatures that are evaluated along line 2 (liquid), as shown in **Errore. L'origine riferimento non è stata trovata.**b, in order to highlight the locations in the vicinity of the nozzle wall that could be subjected to heterogeneous flow boiling. Heterogeneous flow boiling occurs if the temperature of the wall (T^W) is greater than or at least equal to the saturation temperature (T_{sat}) of the liquid near the wall, plus a so-called superheat (ΔT_{sh}):

$$T^w \geq T_{sat} + \Delta T_{sh} \qquad (2)$$

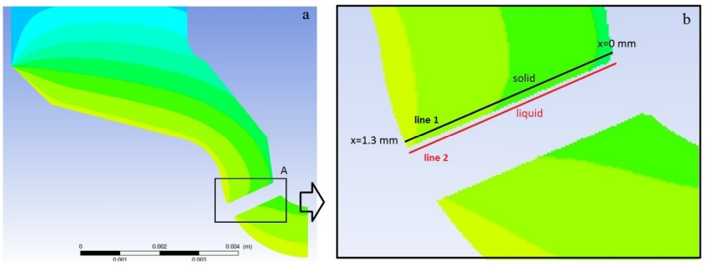


Figure 4. a) A vertical cross-section across the computational domain illustrating the temperature contours through the solid part of the injector tip. b) A zoomed in view of the fuel injector nozzle wall temperatures; The solid wall temperature T^W is evaluated along line 1, representing the temperature values in the first layer of solid cells, while the corresponding values of the diesel liquid saturation temperatures are evaluated along line 2.

The value of the superheat ΔT_{sh} (defined in equation 2) needed for the flow boiling to actually occur, could vary significantly, from only a few degrees to tens of degrees, depending on the roughness and the surface finish of the internal walls [6], [7].
The saturation temperature values along line 2 (**Errore. L'origine riferimento non è stata trovata.**b) are evaluated as function of the locally predicted liquid pressure values [4]. The experimental variations of the saturation temperature of the diesel liquid are available for diesel pressure values within the range of 9 to 3000kPa as shown in **Errore. L'origine riferimento non è stata trovata.**. Equation (3) below, provides the mathematical correlation between the saturation temperature and the diesel liquid pressure within the applicable range shown in Figure 5.

$$T_{sat} = 156.9[\mathrm{K}] + 17.57[K/Pa] \cdot (P_{fluid}(x, y, z))^{0.2246} \qquad (3)$$

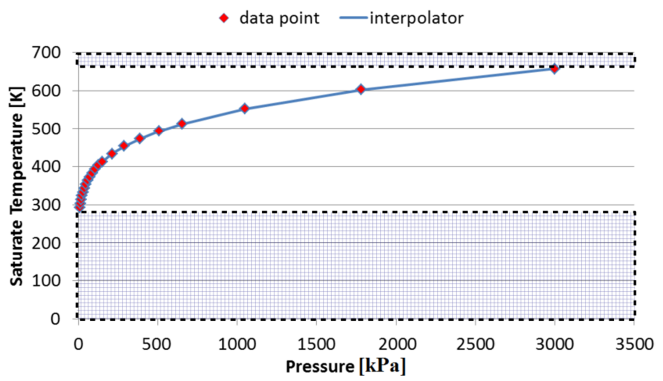


Figure 5. Variations of the saturation temperature of diesel liquid with respect to pressure. The red points are the experimental data [4]. The blue line represents the fitted correlation (equation 3) to the experimental data.

The range of the experimental saturation temperature values for diesel liquid is from 293 to 657K. The corresponding range of the saturation pressure values is from 9 to 3000kPa. According to Kolev[4] the critical temperature and pressure of the diesel fuel (based on the corresponding known values for n-heptan and n-octane) are 658K and 3000kPa respectively [4]. That means although the fuel injector inlet pressure is two orders of magnitude higher than the diesel critical pressure of 3000kPa, within this device, any change of phase from liquid to vapour form would only occur at pressures less than 3000kPa. Furthermore at the other end of the above experimental range, the diesel liquid temperature inside a typical fuel injector is likely to be well above 293K under normal engine operating conditions. Therefore if the diesel liquid pressure falls below 9kPa, by extrapolating equation (3) to pressure values close to absolute zero, the value of ΔT_{sh} (superheat) would be quite high and therefore the occurrence of flow boiling is thought to be quite likely. The grey dashed areas shown in Figure 5 highlight the regions outside the experimental range of the saturation temperature and pressure of diesel liquid.

The simulation results for two different fuel injector inlet pressures (240 and 300 MPa) are shown in **Errore. L'origine riferimento non è stata trovata.**. The dotted lines represent the temperature along line 1 on the solid part of the nozzle (**Errore. L'origine riferimento non è stata trovata.**b). The continuous lines represent the saturation temperature along line 2 on the liquid part of the nozzle (**Errore. L'origine riferimento non è stata**

trovata.b). The comparison between the two lines (for each case) could indicate if there are zones where the temperature of the solid wall (dotted line) is greater than the saturation temperature of the diesel liquid adjacent to the solid wall (continuous line). In such situations equation (2) could become true and these zones could therefore be subjected to potential heterogeneous flow boiling. From the results obtained in this research study, it is evident that at least two zones could potentially be at risk of heterogeneous flow boiling:

- Zone a: close to the entry into the nozzle.
- Zone b: close to the exit from the nozzle.

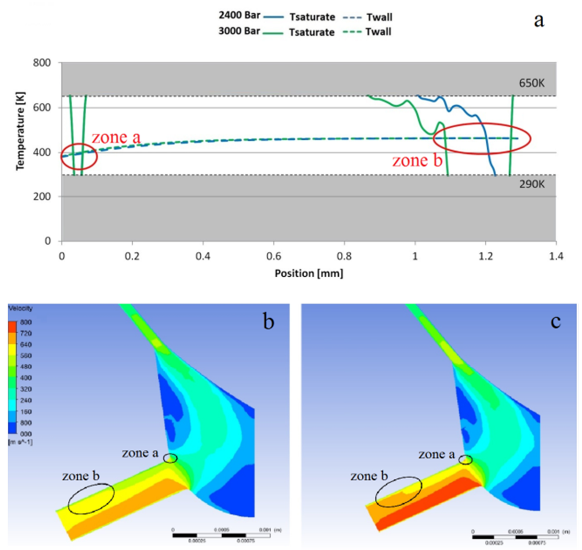


Figure 6 a) Comparison of the nozzle wall temperature T^W along line 1 (dotted line) and the saturation temperature of the diesel liquid along line 2 (continuous line) for two inlet fuel pressures of b) P_{inj} =240 MPa and c) P_{inj} = 300 MPa.

Errore. L'origine riferimento non è stata trovata.b and **Errore. L'origine riferimento non è stata trovata.**c show the diesel liquid velocity contours for the two scenarios considered (P_{inj} = 240 MPa and P_{inj} = 300 MPa). The lower saturation temperatures expected in zones a and b (**Errore. L'origine riferimento non è stata trovata.**) are due to very low pressure values in those two zones. The simulation results in fact highlight the following:

- A significant increase in velocity values together with a significant pressure drop occurring at the sharp entry into the nozzle (zone a).
- A flow recirculation being present close to nozzle exit. This flow recirculation also leads to a pressure reduction in zone b.

Errore. L'origine riferimento non è stata trovata.a and 7b show the effects of the variations of the engine cylinder temperature T_{ch} and the fuel inlet temperature T_{inlet} on the nozzle wall temperature. As it can be observed the variations of these boundary conditions don't have any significant influence on the identified positions of the potential flow boiling regions.

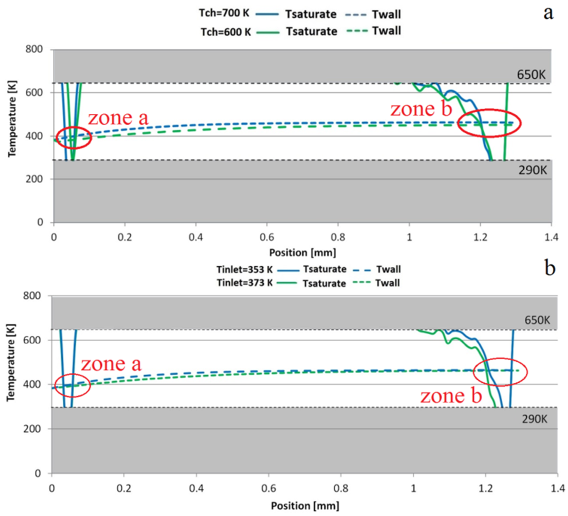


Figure 7 Comparison of the nozzle wall temperature T^W along line 1 (dotted line) and the saturation temperature of the diesel liquid along line 2 (continuous line) based on a) two different engine cylinder temperatures of T_{CH}=700K and T_{CH}=600K and b) two different fuel inlet temperatures of T_{inlet}=353K and T_{inlet}=373K

Transient non-isothermal flow simulation

Using City Univercity's CFD code (GFS), three sets of transient non isothermal cavitating flow simulations are performed, within the fluid region, in order to investigate the possibility of the occurrence of the flow boiling within the flow field under three different sets of boundary conditions. Three sets of boundary conditions are considered:

- r1: $P_{inj}(t)$ varies as shown in Figure 8 and P_{out} = 10 MPa (i.e. a constant value). All walls are assumed to be adiabatic.
- r2: $P_{inj}(t)$ and $P_{out}(t)$ are varied as shown in Figure 8. A constant temperature of 453K is applied to the external walls of the injector including the nozzle hole, but the needle surface is still assumed to be adiabatic.
- r3: P_{inj}=300MPa (i.e. a constant value) and $P_{out}(t)$ varies as shown in Figure 8. A constant temperature of 453K is applied to the external walls of the injector including the nozzle hole, but the needle surface is once again assumed to be adiabatic.

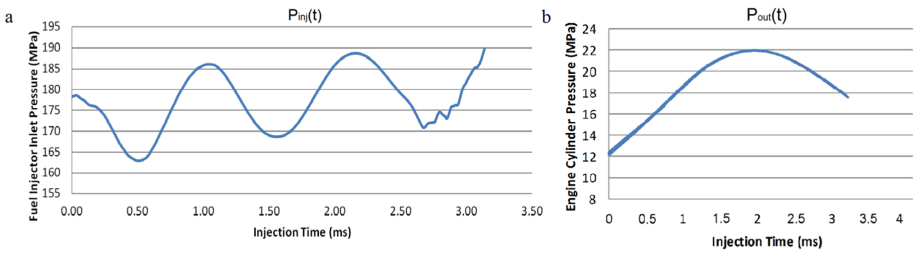


Figure 8 The fuel injector inlet pressure ($P_{inj}(t)$) and the engine cylinder pressure ($P_{out}(t)$) as functions of the injection time [3].

Figure 9 Results of the velocity field and potential flow boiling regions obtained from non-isothermal cavitating two phase flow simulations using City University's CFD code (GFS), under three sets of boundary conditions (r1, r2 and r3) at both the maximum and the minimum axial needle lift positions of 311 and 5µm.
shows the velocity field for the three scenarios (r1,r2 and r3) at two different axial needle lift positions (i.e. at the maximum and minimum needle lifts of 311µm and 5µm). From these results, it is quite evident that the needle axial lift position considerably influences the velocity field. Figure 9 Results of the velocity field and potential flow boiling regions obtained from non-isothermal cavitating two phase flow simulations using City University's CFD code (GFS), under three sets of boundary conditions (r1, r2 and r3) at both the maximum and the minimum axial needle lift positions of 311 and 5µm.

also shows the potential flow boiling regions, based on the positive values of $\Delta T_{boiling}$ defined as the difference between the local temperature of the diesel liquid and its saturation temperature (with the latter calculated as a function of the local fluid pressure as defined by equation 3):

$$\Delta T_{boiling} = T_{fluid}(x, y, z) - T_{sat}(P_{fluid}(x, y, z)) \qquad (4)$$

In regions where $\Delta T_{boiling} > 0$ there is a possibility of flow boiling to occur. The probability of such phenomena obviously increases with increasing values of $\Delta T_{boiling}$. Figure 9 Results of the velocity field and potential flow boiling regions obtained from non-isothermal cavitating two phase flow simulations using City University's CFD code (GFS), under three sets of boundary conditions (r1, r2 and r3) at both the maximum and the minimum axial needle lift positions of 311 and 5μm.

shows strong possibility of the occurrence of the flow boiling within the 5 μm minimum gap between the needle and it seat and some further possibility of the same phenomena occurring within the fuel injector nozzle when the needle is both at its maximum and minimum lift positions. Furthermore with the needle at its minimum lift position, there are potential flow boiling regions within the sac volume.

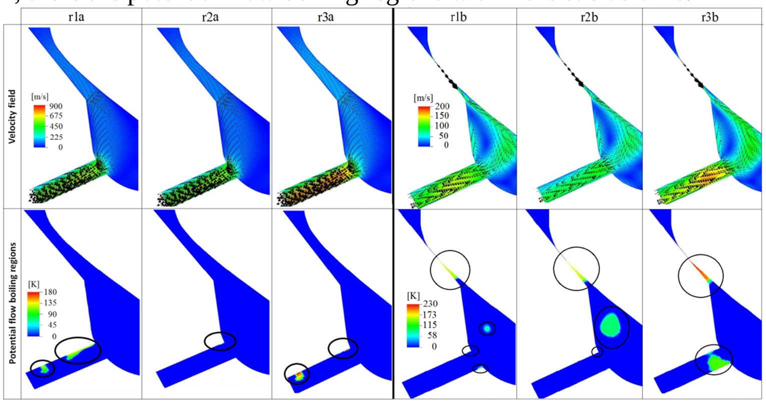


Figure 9 Results of the velocity field and potential flow boiling regions obtained from non-isothermal cavitating two phase flow simulations using City University's CFD code (GFS), under three sets of boundary conditions (r1, r2 and r3) at both the maximum and the minimum axial needle lift positions of 311 and 5μm.

Conclusions

This paper presents the results of a steady state conjugate heat transfer thermal simulation for a diesel fuel injector. After a brief introduction, the boundary conditions used for the CHT simulations are described. In order to estimate the relative influence of each boundary condition on the nozzle wall temperature, a parametric study was performed. This analysis shows that T_{CH} (cylinder gas temperature), T_{inlet} (temperature of the diesel liquid at the inlet) and $T_{CylHead}$ (temperature of the cylinder head) are the most influential parameters on the predicted values of the nozzle wall temperature T^W as shown in Figure 3. Finally the results of the conjugate heat transfer simulation (with constant thermophysical properties) are analysed and discussed in detail. It is shown two zones within the fuel injector nozzle could potentially be subjected to heterogeneous flow boiling; both at the top surface of the nozzle with one zone close to the nozzle inlet (within the first 4% of the nozzle length from the entry) and one close to the nozzle outlet (within the last 10% of the nozzle length from the entry). Based on the boundary condition values used in this study, the average temperature of the nozzle wall T^W is estimated to be about 460 K. This value could be used as thermal boundary condition in future multiphase CFD simulations of the fuel injector. Additionally the results of transient non-isothermal cavitating flow simulations that identify the potential flow boiling regions within the fuel injector are presented. All simulations show potential flow boiling regions at or near the entry to the fuel injector nozzle while at the same time highlight the strong influence of the flow boundary conditions and the axial needle lift position on the size and location of these regions.

Nomenclature

CHT	Conjugate heat transfer
Cp	Thermal capacity
$\mathbf{\Delta T_{boling}}$	Potential flow boiling condition
K	Thermal conductivity
$\mathbf{HTC}_{CH}$	Heat transfer coefficient between chamber and injector tip
$\mathbf{HTC}_{CoolSys}$	Heat transfer coefficient between cylinder head and cooling system

$\boldsymbol{\rho}$	Density
$\mathbf{P}_{inj}$	Injection pressure
$\mathbf{T}_{CH}$	Average temperature of the combustion chamber (engine cylinder)
$\mathbf{T}_{CoolSys}$	Average temperature of the cooling system
$\mathbf{T}_{CylHead}$	Average temperature of the cylinder head
$\mathbf{T}_{needle}$	Average temperature of the needle
$\mathbf{T}_{sat}$	Saturate temperature of the fluid
$\mathbf{T}_{base}^{W}$	Wall temperature on the base case
$\mathbf{T}_{test}^{W}$	Wall temperature on the test case

Acknowledgements

The authors would like to fully acknowledge the funding from the People Programme (Marie Curie Actions) of the European Union's Seventh Framework Programme FP7/People-2012-IAPP/ under REA grant agreement no. 324313.

References

[1] Caira, M., Cipollone, E., Cumo, M., and Naviglio, A., 1987, "Forced convective boiling in high pressure parallel flow in tube bundles," Nuclear Engineering and Design, 99(0), pp. 25-29.
[2] Sakashita, H., and Ono, A., 2009, "Boiling behaviors and critical heat flux on a horizontal plate in saturated pool boiling of water at high pressures," International Journal of Heat and Mass Transfer, 52(3–4), pp. 744-750.
[3] 2014, "Internal communication by Large Power Systems Division Caterpillar UK Engines Co. Ltd."
[4] Kolev, N. I., 2012, Multiphase Flow Dynamics 3.