Fuel 123 (2014) 143–150

Contents lists available at ScienceDirect

Fuel

journal homepage: www.elsevier.com/locate/fuel

Friction-induced heating in nozzle hole micro-channels under extreme fuel pressurisation

Andreas Theodorakakos, George Strotos, Nicholas Mitroglou, Chris Atkin, Manolis Gavaises *

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

HIGHLIGHTS

- CFD simulation of cavitation in Diesel fuel injectors up to 3000 bar.
- The energy conservation equation is solved for temperature variations.
- Simulations include variable fuel properties and induced heating due to wall friction.
- Discharge through a nozzle causes internal cooling and strong temp. gradients.
- Flow found dependent on ρ, μ, thermal conductivity and heat capacity gradients.

ARTICLE INFO

Article history:
Received 2 April 2013
Received in revised form 9 January 2014
Accepted 13 January 2014
Available online 1 February 2014

Keywords:
High fuel pressurisation
Variable fuel properties
Flow-induced boiling
Cavitation
Fuel injection

ABSTRACT

Fuel pressurisation up to 3000 bar, as required by modern Diesel engines, can result in significant variation of the fuel physical properties relative to those at atmospheric pressure and room temperature conditions. The huge acceleration of the fuel as it is pushed through the nozzle hole orifices is known to induce cavitation, which is typically considered as an iso-thermal process. However, discharge of this pressurised liquid fuel through the micro-channel holes can result in severe wall velocity gradients which induce friction and thus heating of the liquid. Simulations assuming variable properties reveal two opposing processes strongly affecting the fuel injection quantity and its temperature. The first one is related to the de-pressurisation of the fuel; the strong pressure and density gradients at the central part of the injection hole induce fuel temperatures even lower than that of the inlet fuel temperature. On the other hand, the strong heating produced by wall friction increases significantly the fuel temperature; local values can exceed the liquid's boiling point and even induce reverse heat transfer from the liquid to the nozzle's metal body. Local values of the thermal conductivity and heat capacity affect the transfer of heat produced at the nozzle surface to the flowing liquid. That creates strong temperature gradients within the flowing liquid which cannot be ignored for accurate predictions of the flow through such nozzles.

1. Introduction

The development of direct injection (DI) Diesel engines over the last 20 years has been remarkable. The market share of Diesels has been increasing for passenger cars while it dominates in the medium and heavy duty vehicles. According to an Energy Outlook Review for 2040 [1], the number of such vehicles is expected to significantly increase (more than double) over the next decades; this increase is linked with the expected growth of the construction sector in the developing countries, the continued development of highly-populated urban areas that require increased transportation of goods by road/rail and the increase in the transportation of goods by sea. It is also expected that the consumption of Diesel fuel from this particular sector will double. As a result, there is great concern about the durability of the fuel injection equipment for medium and heavy duty applications [2], which can be impaired by the appearance of aggressive cavitation within the Diesel injector nozzles. At the same time, increasingly stringent emission legislations such as Euro VI, EPA10, J-PNLT and Stage IV/Tier 4 and the forthcoming regulations on CO_2 are contributing to the development of more efficient IC engines. Advancements towards 2500/3000 bar injection pressure have become a reality [3,4] as this results to simultaneous reduction of soot and NO_x emissions and thus, put less demand on the efficiency and cost of after-treatment systems.

* Corresponding author. Tel.: +44 2070408115.
E-mail address: m.gavaises@city.ac.uk (M. Gavaises).

http://dx.doi.org/10.1016/j.fuel.2014.01.050
0016-2361/

Nomenclature

Symbols

α_L	ratio of liquid volume to cell volume
U	internal energy (J/kg)
c_d	nozzle discharge coefficient
c_p	heat capacity under constant pressure (J/(kg K))
c_{pmT}	mean heat capacity under constant pressure between temperature T and reference temperature T_0 (J/(kg K))
Q_{vis}	viscous work (W/m^3)
h	enthalpy (J/kg)
k	thermal conductivity (W/(mK))
κ	turbulent kinetic energy (m^2/s^2)
p	pressure (Pa)
$\vec{q}$	thermal diffusion vector (W/m^2)
T	temperature (K)
t	time (s)
$\vec{u}$	velocity vector (m/s)
μ	molecular viscosity (kg/(m s))
ρ	density (kg/m^3)
σ	Prandtl number
$\bar{\bar{\tau}}_{eff}$	stress tensor (Pa)
$\mathcal{I}$	unit tensor

Subscripts

0	property at reference pressure p_0 and temperature T_0
eff	effective property
t	turbulent property

Superscripts

$'$	fluctuation

The occurrence of cavitation in Diesel fuel injectors has been documented in the open literature since the late 1990s, for example [5]. Cavitation is known to affect nozzle efficiency, fuel atomisation and spray development. Numerous past studies have examined both experimentally and computationally this phenomenon; however, only limited information is available in the open literature about cavitation effects on system durability and erosion at elevated injection pressures. The cavitating flow in fuel injection systems is typically characterised by a large number (of the order of 10^5–10^6 during a typical injection event) of bubbles exposed to pressure differences reaching 3000 bar. As a result of the violent change in the cavitation bubble size during their collapse, pressures and temperatures may even exceed 1 GPa and 10^4–10^5 K, respectively [6]. Cavitation bubble collapse often produces shock waves strong enough to cause surface erosion. As a result, today's injectors incorporate tapered holes that converge towards the hole exit and which are known to suppress cavitation. Due to the difficulty in obtaining real-time measurements during the injection process, most of the experimental studies reported (selectively [7–11]) refer to experimental devices emulating operating conditions similar to those of Diesel engines. Limited information exists for production injectors during engine operation although recent advances in testing equipment (for example, high speed cameras with 10^6 fps) and use of X-rays [12] are expected to improve our understanding in the near future. Therefore, development and use of computational fluid dynamics models predicting the flow in such systems seem to be the only route for obtaining information about the details of the nozzle flow under realistic operating conditions. Modelling efforts aiming to tackle cavitation under such hostile environments have been a challenge for many years. Within the approach of [13] the cavitating fluid was treated as mixture, assuming the existence of small bubble clouds on a subgrid scale. In the model of [14], instead of treating cavitating fluid as a single mixture, the two-fluid method is employed; two sets of conservation equations are solved, one for the liquid and one for the vapour phase. With this approach the two phases can have different velocities. In [15] another bubble-based cavitation model has been proposed and implemented in commercial CFD codes. Another variant of the bubble model is the approach of [16,17]. In this case the authors modified the classical interface-capturing Volume of Fluid (VOF) method by considering the transported scalar volume fraction to be the local vapour fraction of a bubble cloud. More recent important advances (selectively [18–21]) have proposed models that account for collective compressibility and shock wave interaction effects in poly-dispersed cavitating flows. Recently, more fundamental models have been developed to simulate the shock waves produced during the non-spherical bubble collapse processes [22] but they cannot be directly linked to flows comprising millions of cavitation bubbles. Eulerian–Lagrangian cavitation models are also available. They are based on the implementation of various versions of the Rayleigh–Plesset bubble dynamics equation (selectively [23–25]) for predicting the bubble growth and collapse which is important if heating effects and the implications on erosion are to be considered.

All previous cavitation models assumed iso-thermal flow conditions, which was justified on the basis that cavitation takes place over very short time scales and the fact that the residence time of the liquid within the injection holes is so short that heat transfer with the surrounding could be neglected. However, as the trend in technology is towards pressures in the excess of 2500 bar and possibly reaching 3000 bar, the common assumption of isothermal flow conditions within the context of phase-change through cavitation may no longer be valid. It is now believed that due to this high fuel pressure, the extreme velocities occurring during the discharge of the liquid through the nozzle's micro-channel holes can induce wall friction which leads in turn to significant fuel heating. In some cases this may lead to boiling of the flowing fluid. Boiling heat transfer has been an active research topic in the last decade. However, only limited number of papers deal with pool boiling at high pressures, such as [26] involving experimental campaigns at pressure up to 70 bar. The highest pressure in flow boiling was achieved by [27]; this work proposed a correlation for the evaluation of the heat transfer coefficient for boiling at pressures up to 90 bar. It seems that no data are available for higher pressure systems, such as those used in fuel injection systems, and therefore there is currently no relevant model available for simulating boiling under such pressure conditions.

The present study challenges these assumptions and shows that under such extreme fuel pressurisation and subsequent discharge though the nozzle's micro-channel holes, production of heat caused by wall friction can induce temperatures above the boiling point of the flowing fuel, and thus, significantly alter the flow regimes believed to prevail in such systems. Thus, the main focus of this work is to assess such effects. The group at City University London has developed previously its own CFD/cavitation model based on a coupled Eulerian–Lagrangian stochastic approach [28]; this model provides the platform for the present work. It is also linked with the early computational advances reported in [29] predicting the influence of fuel properties on cavitation when extreme fuel pressurisation up to 2500 bar was utilised. The next section gives the description of the computational model developed to account for variable fuel properties and heating effects;