International Journal of Heat and Fluid Flow 51 (2015) 257–267

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

Transient heating effects in high pressure Diesel injector nozzles

George Strotos *, Phoevos Koukouvinis, Andreas Theodorakakos, Manolis Gavaises, George Bergeles

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

ARTICLE INFO

Article history:
Available online 1 November 2014

Keywords:
Nozzle
Cavitation
Variable properties
Moving needle
Fuel heating

ABSTRACT

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

1. Introduction

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

A common feature of cavitation studies in fuel injector nozzles is the assumption of isothermal flow due to the short timescales

* Corresponding author.
E-mail addresses: George.Strotos.1@city.ac.uk (G. Strotos), Foivos.Koukouvinis.1@city.ac.uk (P. Koukouvinis), Andreas.Theodorakakos.1@city.ac.uk (A. Theodorakakos), M.Gavaises@city.ac.uk (M. Gavaises), bergeles@fluid.mech.ntua.gr (G. Bergeles).

http://dx.doi.org/10.1016/j.ijheatfluidflow.2014.10.010
0142-727X/

Nomenclature

A	area, m^2
C_d	discharge coefficient, –
c_p	heat capacity, J/kg K
c_{pmT}	mean heat capacity, J/kg K
D	diameter, μm
g	gravity acceleration, m/s^2
h	enthalpy, J/kg
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
Greek symbols	
a	volume fraction, –
κ	thermal conductivity, W/mK
μ	viscosity, kg/ms
ρ	density, kg/m^3
τ	stress tensor, N/m^2
Subscripts	
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

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

2. Numerical model and methodology

2.1. Equations solved

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

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

where the specific total enthalpy is the sum of the specific static enthalpy h, the flow mean kinetic energy and the turbulent kinetic energy k.

$$h_{tot}=h+\frac{\mathbf{u}\cdot\mathbf{u}}{2}+k \tag{2}$$

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

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

$$\mu_{eff}=\mu_{lam}+\mu_{turb} \tag{4}$$

$$\kappa_{eff}=\left(\frac{\mu_{lam}}{\mathrm{Pr}_{lam}}+\frac{\mu_{turb}}{\mathrm{Pr}_{tutb}}\right)c_p \tag{5}$$

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

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

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