Fuel 121 (2014) 284–297

Contents lists available at ScienceDirect

Fuel

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

Spray-induced air motion in single and twin ultra-high injection diesel sprays

A. Ghasemi, R.M. Barron *, R. Balachandar

Department of Mechanical, Automotive and Materials Engineering, University of Windsor, Windsor, ON N9B3P4, Canada

HIGHLIGHTS

- Air motion caused by ultra-high injection pressure single and twin diesel sprays is investigated.
- CFD/empirical analysis shows more entrainment at higher injection pressure and ambient density.
- Gas aspiration, recirculation and pushed out zones for single sprays are identified.
- Effects of incidence angle and separation distance of twin sprays on spray parameters are studied.
- Vortical structures are used to characterize the spray-induced air motion in twin sprays.

ARTICLE INFO

Article history:
Received 15 September 2013
Received in revised form 11 December 2013
Accepted 14 December 2013
Available online 1 January 2014

Keywords:
Diesel fuel
Merging spray
Entrainment
Equivalence ratio
Swirling strength

ABSTRACT

The air motion generated by the dispersion of single and twin diesel sprays at ultra-high injection pressures is analyzed using computational fluid dynamics (CFD) modeling. Injection pressures up to 300 MPa are used to generate the sprays in air at ambient densities of 15 and 30 kg/m^3 at 298 K. To validate the models, a single spray injected into an initially quiescent constant volume chamber is simulated using the Eulerian–Lagrangian approach. Reynolds–Averaged Navier–Stokes equations, with the k–ε turbulence model, are solved using an Eulerian formulation for the continuous phase. The discrete droplet phase is treated using a Lagrangian formulation together with spray sub-models. Results are validated with published experimental data. Macroscopic and microscopic characteristics of the single sprays are studied. The CFD results are combined with empirical formulations to evaluate entrainment into a single spray under different injection and ambient conditions. Gas flow field vortex structures are identified based on the swirling strength parameter. In addition, the effects of incidence angle and separation distance of dual interacting sprays on parameters such as tip penetration and Sauter mean diameter (SMD) are investigated. In the case of twin sprays, to evaluate the expansion of the merged spray, a cone angle is defined and compared for different injection point separation distances and incidence angles. Finally, the spray-induced air motion characteristics of the twin sprays are discussed in terms of the vortical structures identified in the gas field.

1. Introduction

The extensive range of applications and the inherent multiphase phenomena that occur in sprays have made them an important class of flows for both industrial applications and academic studies. In direct injection diesel engines, optimization of the spray contributes significantly to higher efficiency and lower emission combustion. Mixture formation is strongly influenced by the atomization process of the fuel spray in a direct injection IC engine. The subsequent processes of ignition, combustion and pollutant formation are also affected by the atomization.

For single sprays, researchers have investigated the use of various injection and ambient conditions in order to achieve higher levels of droplet atomization and mixture formation. Ambient flow conditions like pressure [1], temperature [2], and combustion chamber flow field structure [3] influence spray formation and development, and consequently mixture formation in the chamber. One of the techniques used to provide higher levels of atomization and enhanced mixing in single diesel sprays is to apply ultra-high injection pressures. This is of particular interest when dealing with different biodiesel fuels where break-up is harder to achieve than in conventional diesel fuels due to the larger surface tension and viscosity [4]. Ultra-high injection pressure single sprays interacting with the gas flow field inside the combustion chamber have also been studied [5]. In a single spray, the governing break-up mechanism is a result of the instabilities generated at the gas/liquid

* Corresponding author. Tel.: +1 (519) 253 3000x3019.
E-mail address: AZ3@uwindsor.ca (R.M. Barron).

0016-2361/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.fuel.2013.12.041

Nomenclature

$(A/F)_{st}$	stoichiometric air/fuel ratio
C_a	orifice area contraction
d	distance between nozzles
d_0	initial droplet diameter
d_2	smaller droplet diameter
d_p	particle diameter
D	nozzle diameter
P_{inj}	injection pressure
Re_p	particle Reynolds number
t	time
U_r	slip velocity between phases
U_{rel}	relative velocity between droplets
U_X	spanwise velocity
U_Z	streamwise velocity
We_{coll}	collision Weber number
Z^+	characteristic length scale
X,Y,Z	cartesian coordinates
α_l	fuel volume fraction
Δ	droplet size reduction
ΔP	pressure difference
β	spray incidence angle
θ	spray cone angle
θ_{XZ}	merged spray cone angle (XZ plane)
θ_{YZ}	merged spray cone angle (YZ plane)
λ	swirling strength
μ_a	air viscosity
ν	kinematic viscosity
ρ_a	air density
ρ_l	fuel density
σ	surface tension
τ_p	particle relaxation time
$\bar{\phi}(Z)$	average equivalence ratio

interface due to the aerodynamic forces. Therefore, the choice of an appropriate break-up model is an important task in the simulation of a single spray. For instance, in a classical break-up model such as WAVE, Kelvin–Helmholtz instability determines the rate of break-up. But in a hybrid break-up model like KH–RT (Kelvin–Helmholtz/Rayleigh–Taylor), the competing effect between Kelvin–Helmholtz and Rayleigh–Taylor instabilities governs the break-up rate. Influence of various break-up models on diesel spray formation and break-up has been investigated by Djavareshkian and Ghasemi [6]. There have also been some studies on entrainment and air motion in a single spray. Using particle image velocimetry (PIV), Sepret et al. [7] studied ambient density and nozzle diameter effects on the entrainment into diesel fuel sprays in the quasi-steady and non-stationary regions. They identified three different zones in the spray in which the ambient gas is either entrained, recirculated or pushed out. By evaluating the axial variation of the normal velocity, they suggested that the spray expands with increasing ambient density and more ambient air is engulfed.

In addition to the break-up process discussed above for single sprays, there are other mechanisms that can contribute to the atomization of a liquid jet. The limited literature on collision-induced break-up focuses mainly on the atomization of low Reynolds number liquid jets. For instance, Chen et al. [8] numerically studied the collision-induced break-up resulting from the impingement of a pair of liquid jets. They reported the formation of rim and liquid sheets downstream of the impact point. The role of the instability waves formed in the liquid sheet was discussed in connection with the disintegration of the sheet into ligaments and droplets. Li and Ashgriz [9] also reported on the characteristics of liquid film formation. They discussed two main categories and five sub-regimes that govern the break-up mechanism of the liquid sheet. Ashgriz et al. [10] studied the mixing of two impinging jets. They reported the effect of incidence angle, jet velocity and turbulent dispersion on the mixing. They suggested that the impact momentum of the two jets governs the mixing process prior to atomization. On the other hand, mixing is also influenced by the turbulent dispersion of the droplets after the atomization. In the case of two interacting sprays, collision becomes a very significant break-up mechanism [11]. Due to the major role that collision plays in spray-to-spray interactions, attempts have been made to modify the conventional O'Rourke collision model. The O'Rourke collision model only considers bounce, permanent coalescence and separation of the same kind of droplets (water–water, fuel–fuel) [12] and neglects the formation of satellite droplets [13]. Although it has been shown that collisions in interacting sprays can contribute to atomization and consequently reduce the Sauter mean diameter (SMD) compared to single sprays [14], the O'Rourke collision model shows larger SMD values due to the over-prediction of coalescence. Another observation in interacting sprays is the significant increase in spray volume. Increasing the spray angle leads to a reduction in droplet velocities. On the other hand, impingement distance does not have a major influence on the droplet velocities since it is related to the travelling distance of the droplets [15].

One can observe from the literature that there has been significant attention paid to single sprays in terms of spray formation, break-up and entrainment. However, there has been very little emphasis on the vortex structures formed in the gas field due to the spray dispersion. Identification of vortical structures in the gas field allows for the investigation of the quality of fuel/air mixing which occurs in the vicinity of the vortices. In addition, the study of the interaction of two merging sprays has been rather limited. The objective of this work is to implement an Eulerian–Lagrangian approach to study the characteristics of single and twin sprays. The single spray characteristics such as tip penetration, Sauter mean diameter and spray cone angle are first studied and validated against experimental data [16]. The effect of different injection and ambient pressures on the entrainment, air motion and mixture formation of single sprays are studied. Air motion induced by the single spray dispersion in the initially quiescent gas is investigated by vortex core identification. Characteristics of merged sprays in terms of penetration, expansion and droplet SMD are compared with a single spray. Finally, air motion and mixture formation inside interacting sprays are investigated in terms of vortical structures and liquid volume fraction.

2. Numerical methodology

2.1. Computational procedure

The Eulerian–Lagrangian multiphase approach is adopted to calculate the interaction of the discrete (fuel) and continuous (air) phases. The Eulerian formulation yields a set of partial differential equations for the related fluid flow parameters such as velocity components, pressure, density and temperature as functions of position and time (X, Y, Z, t). The unsteady Reynolds–Averaged Navier–Stokes equations (see Appendix A) are solved for the continuous phase using the standard k–ε turbulence model. The equations are spatially discretized by the finite volume method using the QUICK algorithm for the convective terms in momentum