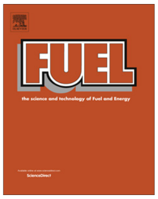
Fuel 188 (2017) 352–366

Contents lists available at ScienceDirect

Fuel

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



Full Length Article

Analysis of diesel spray dynamics using a compressible Eulerian/VOF/LES model and microscopic shadowgraphy

M. Ghiji [a],*, L. Goldsworthy [a], P.A. Brandner [a], V. Garaniya [a], P. Hield [b]

[a] Australian Maritime College, University of Tasmania, Tasmania, Australia
[b] Defence Science and Technology Group, Victoria, Australia

HIGHLIGHTS

- In-and near-nozzle diesel spray dynamics are investigated.
- Structure of the jet at early and quasi-steady stages of the injection is analyzed.
- Starting vortex at early stage of the injection is captured.
- An LES approach is used for numerical analysis of subsonic and supersonic states.
- Onset and development of shock waves for high pressure diesel jet are discussed.

ARTICLE INFO

Article history:
Received 2 March 2016
Received in revised form 4 October 2016
Accepted 6 October 2016
Available online 14 October 2016

Keywords:
Primary atomization
Diesel spray
Large eddy simulation
Cavitation
Shock wave
Compressible flow

ABSTRACT

This paper presents numerical and experimental analysis of diesel engine spray dynamics in the region very close to the nozzle exit. Diesel fuel is injected through a single solid cone injector with sharp-edged nozzle inlet. Numerical investigations are conducted in an Eulerian framework by applying a Volume of Fluid interface capturing technique integrated with Large-Eddy Simulation turbulence modelling. Cavitation is modelled, by allowing liquid fuel to flash to gas at the fuel vapor pressure. The computational domain and settings mimic the experimental injector internal geometry and experimental operating conditions. In-nozzle disturbances are qualitatively well modelled by implementing the no-slip condition at the injector walls as well as cavitation and compressibility effects for each phase. A mesh dependency study is conducted with four different grid resolutions. Data are presented around the start of penetration (SOP) and up to the time when shock waves at the gas-liquid interface are well developed, the quasi-steady stage of injection. At SOP, an umbrella-shaped leading edge is captured in both the numerical and experimental studies however only the experimental images demonstrated a semi-transparent cloud of air-fuel mixture at the leading edge. A previously undescribed toroidal starting vortex near the nozzle exit is captured experimentally and numerically. Development of cavitation, down to the end of nozzle hole leads to the detachment of liquid from the nozzle hole walls and subsequently the diminution of boundary layer effects and thus reduced in-nozzle turbulence, and increased liquid jet velocity.

1. Introduction

Engine emissions are produced during the combustion process which is fundamentally controlled by the dynamics of the fuel injection [1–6]. There is a wide range of fuel injectors based on their shapes and flow characteristics but the purpose of most injectors is still the same, to induce atomization, penetration, turbulence generation and gas-fuel mixing. Undoubtedly, a clear understanding of these processes would assist engineers to design an injector which not only meets strict pollution requirements but also improve engine performance in one of the most extreme environments for multiphase flow. In this harsh environment, shock waves [7] and turbulent eddies [8] are expected, which makes understanding of the spray dynamics a challenge for designers and scientists.

The atomization process which initiates very close to the nozzle hole exit, is called primary atomization and controls the extension of the liquid core and subsequently the secondary atomization in the disperse flow region [9,10]. To date, many theories have been proposed to describe the primary atomization mechanism, including: Aerodynamic shear forces which act through stripping and

---

* Corresponding author.
E-mail address: mghiji@utas.edu.au (M. Ghiji).

http://dx.doi.org/10.1016/j.fuel.2016.10.041
0016-2361/

Kelvin-Helmholtz (K-H) instabilities [11–13]; turbulence-induced disintegration which has a significant effect on jet breakup in higher Reynolds number $Re_l = \rho_l\, V\, D/\mu_l$, where ρ_l is the liquid density, V is the liquid velocity, D is the orifice diameter, and μ_l is the liquid dynamic viscosity [14–17]; relaxation of the velocity profile, creating a "bursting" effect especially in non-cavitating jets and large velocity differentials [18]; cavitation-induced disintegration of the jet due to the reduction of cross-section area at nozzle inlet [19–22]; and liquid bulk oscillation provoking the toroidal surface perturbation [12,23].

For nozzles with small length-to-diameter ratios super-cavitation and hydraulic flip can occur [24]. In these cases, the liquid fuel which has detached at the nozzle inlet remains detached from the walls throughout the entire nozzle passage, and the liquid core is contracted at the nozzle exit compared to the nozzle size, so the mass flow rate is reduced. If the length of the nozzle passage is long enough, or if the injection pressure is not high, the liquid flow can re-attach to the walls downstream of the nozzle hole inlet [25,26]. In this case, the discharge coefficient is higher compared to that of the super-cavitation case.

Based on the Reynolds and Ohnesorge numbers of the flow, the breakup of liquid jets is categorized into four regimes; Rayleigh breakup, first wind-induced breakup, second wind-induced breakup, and atomization [27]. These parameters also change with different fuels. Detailed studies comparing different fuels and the influence on spray structure and formation have been made by Payri et al. [28,29], Desantes et al. [30], Battistoni et al. [31], and Goldsworthy et al. [32]. For diesel propulsion systems, the liquid propellants fall well within the atomization regime. In such regime, average drop diameters are much less than the jet diameter, thus indicating that the scale in which flow instabilities arise is much smaller than the jet diameter. Furthermore, liquid jets within this regime experience stronger axial velocity gradients in the near exit region than the jets in other regimes due to faster relaxation of the liquid surface as it transitions from a no-slip boundary (except in the case of "super-cavitation") to a free surface boundary condition as it exits the injector nozzle.

The existence of shock waves in high pressure diesel spray was first reported by Nakahira et al. [33] and most recently by Huang et al. [7] using the schlieren image technique. Hillamo et al. [34] demonstrated the imaging of shock waves from a diesel spray using the backlit imaging technique. An increase of 15% in the gaseous phase density near the shock front was quantitatively demonstrated by MacPhee et al. [35] using the X-ray radiograph image technique.

In experimentations, isolating and quantifying the various interactive mechanisms involved in primary atomization of a high-pressure liquid jet are very difficult [13,36–40]. Hence, numerical analysis can be employed to get a clearer insight into the effect of each parameter at different stages of the injection process [4,41].

Generated turbulent flows can be represented by eddies with a range of length and time scales. Large eddy simulation (LES) directly resolves large scale eddies and models small eddies, allowing the use of much coarser meshes and longer time steps in LES compared to Direct Numerical Simulation (DNS). LES needs principally finer meshes compared to the ones used for Reynolds Averaged Navier-Stokes (RANS) computations. Since RANS models cannot capture features of the transient spray structure [9,12,42,43] such as droplet clustering and shot to shot variability, LES is applied to overcome these limitations. In addition, the conventional atomization models with Lagrangian Particle Tracking (LPT) limit the grid fineness near the nozzle and do not allow LES to capture the features of the spray and background fluid flow near the nozzle. Refining the grid with the blob atomization method can result in problems with a high liquid fraction in the LPT approach (too much liquid in each cell) [9,42–44]. These limitations motivate the use of the Eulerian approach to model the primary atomization, instead of using LPT atomization models. With ever increasing computational power there is an incentive to use more complex models for primary atomization.

The accuracy of different numerical techniques for modelling the primary atomization of a liquid diesel jet was investigated in detail for low Re ($Re < 5000$) by Herrmann [14] and Desjardins and Pitsch [45]. Herrman [14] demonstrated the importance of the grid resolution on capturing the accurate phase interface geometry of diesel liquid with an injection velocity of 100 m/s and $Re = 5000$. Turbulence was reported as the dominant driving mechanism of atomization within the first 20 nozzle diameters downstream.

The present study focuses on experimental and numerical investigation of the primary atomization in the early stages of injection with increasing injection pressure up to 1200 bar, background pressure of 30 bar, liquid Re of $7\times10^3 \leqslant Re_l \leqslant 46\times10^3$, and liquid Weber number of $4\times10^4 \leqslant We_l \leqslant 2\times10^6$. The liquid Weber number (We_l) is defined as $\rho_l\, V\, D/\sigma$, where σ is the surface tension at the liquid-gas interface. Recent work using X-ray imaging [46–48], especially from the Argonne Laboratory has greatly enhanced our understanding of diesel spray dynamics. The experimental techniques presented here, while less sophisticated are more accessible and give useful data on the spray morphology for comparison with numerical analysis.

A key aim of the present work is to achieve a valid (high-fidelity) Computational Fluid Dynamics (CFD) modelling of diesel spray primary atomization which can be applied by engine developers for improved design of diesel engines. A further aim is to apply the numerical and experimental analysis to enhance understanding of in- and near-nozzle processes.

2. Methodology

Experimental measurements are used to validate the numerical results at various stages of the injection event. The experiments employed a microscopic laser-based backlight imaging (shadowgraphy) technique using a constant volume spray chamber.

Numerical investigations are conducted by applying the VOF phase-fraction interface capturing technique in an Eulerian LES framework where cavitation of the fuel is allowed at a predefined vapor pressure. Enhanced cavitation inception due to nuclei is not modelled. The effects of compressibility of each phase have been included in the numerical model, enabling the investigation of more complex physics associated with a diesel spray process such as viscosity and temperature changes, generation and development of cavitation and gaseous shock waves.

2.1. Experimental set-up

The experimental apparatus consists of a constant volume High-Pressure Spray Chamber (HPSC). The HPSC operating volume is a square-section prism with rounded corners, with the chamber and spray axes vertically oriented. Optical access to the chamber is via three windows of UV quality, optically polished quartz, with viewing area of 200 × 70 mm. The chamber pressure can be varied to emulate the air density occurring in a diesel engine at the start of injection. Diesel fuel is injected axially through a single solid cone fuel spray with an adjustable injection pressure up to 1200 bar from the top of HPSC as shown in Fig. 1. A continuous flow of air through the chamber removes droplets from previous shots. Tests were made to ensure that any turbulence induced by the flushing air did not impact on the spray dynamics, by closing