Fuel 234 (2018) 286–295

Contents lists available at ScienceDirect

Fuel

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

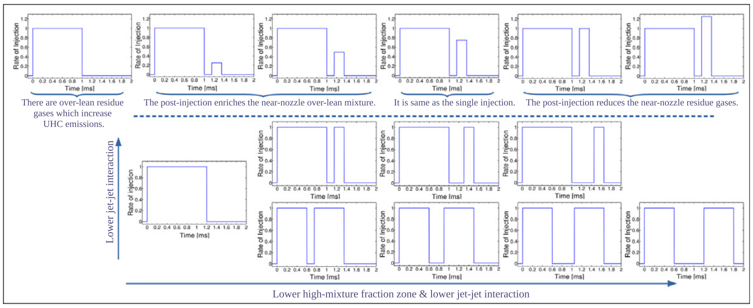
Full Length Article

Jet-jet interaction in multiple injections: A large-eddy simulation study

Ahmad Hadadpour[a], Mehdi Jangi[b,a,*], Xue Song Bai[a]

[a] Division of Fluid Mechanics, Department of Energy Science, Lund University, P.O. Box 118, S 221 00 Lund, Sweden
[b] Department of Mechanical Engineering, School of Engineering, University of Birmingham, Edgbaston, Birmingham B15 2TT, United Kingdom

GRAPHICAL ABSTRACT



ARTICLE INFO

Keywords:
Multiple-injection
Post-injection
Diesel engine
Mixing
LES
Near-nozzle residue

ABSTRACT

This paper reports on studies of multiple-injection strategies of gaseous fuel in a model combustion chamber and the role of jet-jet interactions on the mixing processes in the chamber using large-eddy simulation (LES). A high-pressure non-reacting gas flow injected through a jet with a nozzle diameter of 1.35 mm into a quiescent inert air environment is considered. First, we validate the method and our computational setup by comparing the simulation results of a single injection case with available experimental data. It is shown that the transient ensemble-averaged LES results agree well with the experimental measurements. Second, we simulate and compare fourteen injection strategies in order to understand the effect of the main and the post-injections duration, the dwell time and the mass flow rate of post-injection on the mixing, jet penetration, and near-nozzle mixture. The contribution of each injection in the local mixture composition is quantified by solving transport equations for the mixture fraction of each injection.

The results show that the turbulence generated in the main injection is enhanced when the post-injection flow into the main injection flow. The increase of the local turbulence intensity is in favor of increasing the scalar dissipation rate and enhancing the mixing rate. However, the penetration of the post-injection flow into the main injection flow and the level of the gas flow from the interaction of two injections depend on the dwell time and the momentum of the post-injection.

The results also show that the post-injection modifies the near-nozzle mixture. The comparison of cases with different mass flow rates in the post-injection indicates that the momentum of the post-injection can be optimized either to push away the near-nozzle remaining gas from the main injection and reduce the near-nozzle residue by more than 25% or enrich this fuel-lean region and increase the near-nozzle gasses by more than 43%. These results are very interesting for optimization of the post-injection to reduce engine-out emissions.

---

* Corresponding author at: Department of Mechanical Engineering, School of Engineering, University of Birmingham, Edgbaston, B15 2TT, United Kingdom.
E-mail address: m.jangi@bham.ac.uk (M. Jangi).

https://doi.org/10.1016/j.fuel.2018.07.010
Received 7 February 2018; Received in revised form 27 June 2018; Accepted 3 July 2018
0016-2361/

1. Introduction

Emissions in diesel engines can be reduced by either in-cylinder treatments or after-treatments. One of the common in-cylinder treatments is multiple-injection, which refers to adding a small post-injection of fuel after the end of the main injection [1] or to splitting the main injection into multiple smaller injections [2]. Previous studies showed various effects of the multiple-injection on the engine performance and emissions behavior [3,4]. For instance, Moiz et al. [5] studied the double injection of n-dodecane in a constant volume vessel and reported that the multiple-injection method can be beneficial for combustion efficiency. As another example, O'Connor et al. [6] investigated the effect of post-injection on unburned hydrocarbons (UHC) emissions and reported a 34% reduction of UHC in comparison to a single-injection case at the same load. Moreover, Hessel et al. [7] studied soot formation/oxidation for an engine-like condition and concluded that a short post-injection leads to a lower engine-out soot.

Despite many reports on the effects of the multiple-injection strategies, the underlying physics involved in such flow are still not fully understood. In an extensive review, O'Conor et al. [3] pointed out a number of remaining research questions, and summarised three main possible effects of multiple-injection: enhanced mixing [8–11]; increased combustion temperature [12–15], and the injection duration effects [1,16–18]. Despite many reports on the effects of the multiple-injection strategies, the underlying physics involved in such flow are still not fully understood.

According to detailed soot formation and oxidation modeling [19,20], soot formation is most seen within a range of temperature and equivalent ratio (ϕ). For example for n–heptane combustion at a pressure of 60 bar, the equivalence ratio should be above 2. To gain a deeper understanding of the underlying physics of multiple-injection, we performed a systematic LES study of a non-reacting multiple-injection gas jet. The case is chosen based on a study by Hu et al. [21], in which a transient single-pulse gas injection into ambient air at atmospheric pressure was studied in order to investigate the gas-phase mixing in decelerating jets for practical engine application. We first simulated the same transient single injection case to compare our results with their work and validated our simulation. Then, we used this validated setup to study different multiple-injection strategies.

In this work, we model only the gas phase in order to avoid the uncertainties in the modeling of primary and secondary atomization processes of the spray [22]. Instead, we made a great effort to properly model the mixing processes in the gas-phase by performing detailed LES calculations. The same approach has been used in literature, for instance, see Refs [21,23]. While the size of nozzle diameter, hence the size of the potential core of the injected liquid fuel, in modern engines is of the order of 0.1 mm, in this approach the chosen size of the nozzle is of the order of 1 mm. For instance, in Ref [23] the chosen diameter for the measurement is 1 mm and in Ref [21] the chosen diameter for LES is 1.35 mm. The rationale behind the selection of the parameters for such studies is to choose the right injector diameter and the momentum of the jet that mimic the conditions of spray flows in engines [24,25]. Our previous work on diesel spray [26] also confirmed that the diameter of the gas-phase spray is approximately 10 times larger than the diameter of the liquid injector.

In this study, with the goal of providing insight into gas-phase mixing and interaction of two injections, we strive to answer the following questions: (1) To what degree does the post-injection penetrate into and interact with the main injection? (2) By which mechanism can the post-injection enhance the mixing of injected gas with ambient air? (3) Does the penetration of the main injection change by having post-injection? (4) Is the turbulence in the main injection enhanced by the post-injection? (5) How is the mixture in the near-nozzle region affected by the post-injection strategy? (6) What is the effect of different parameters, such as dwell time, post-injection duration and mass flow rate of the post-injection on the answer to these questions?.

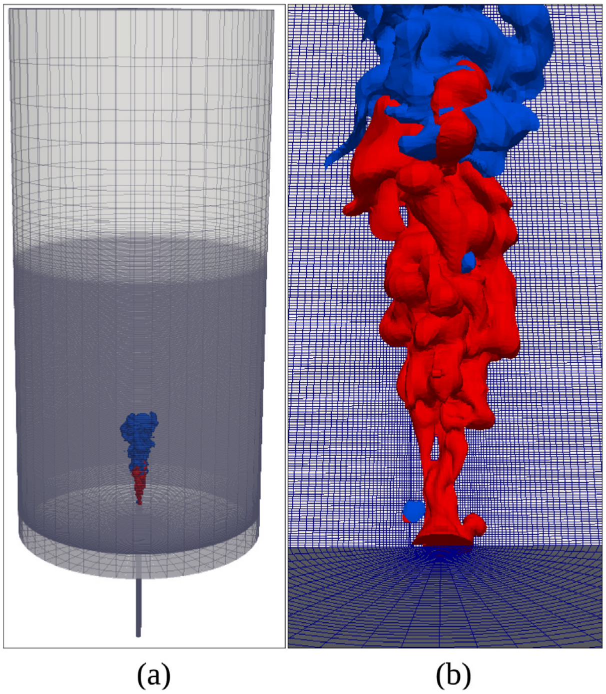


Fig. 1. (a) Computational domain. Domain diameter = 50D and length = 100D, where D = 1.35 mm is the nozzle diameter. The length of auxiliary inlet pipe is 30D; (b) The grid near the nozzle. The red and blue surfaces are iso-countours of mixture fraction in the first injection and second injections, respectively. (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)

2. Case specification and numerical method

As a baseline case for the validation of the numerical method and the solver, we have chosen the study of Hu et al. [21]. They simulated a high-pressure single injection gas jet through a 1.35 mm diameter nozzle into the ambient air at atmospheric pressure and compared the results with an available experiment [27,28]. In these experiments, the gas was injected for 4 ms and the mean axial velocity was measured along the centerline during and after the injection. The experiments reported an ensemble-average of velocity that was calculated from an ensemble of 100–500 measurements at each condition. The nearest data point that was measured in the experiments was at 2.9 mm from the nozzle. In the simulations, the inflowing mass flow rate was selected to match up the mean velocity at 2.9 mm. This gives a velocity of 90 m/s at the nozzle exit and the mass flow rate of 125 mg/s, approximately.

The computational domain is a cylinder with a diameter of 50D and a length of 100D, where D = 1.35 mm is the nozzle diameter (Fig. 1a). The nozzle is placed in the center of the domain base. A cylindrical O-grid mesh is used, which consists of 2.3 million cells (Fig. 1b). The grids are refined toward the nozzle exit and across the jet axis such that 80% of computational cells are located within the refinement cylinder $10D \times 25D$.

It is known that the modeling of turbulent inlet velocity boundary conditions in LES requires special care. For example, imposing a turbulent inlet velocity boundary condition with only white noise fluctuations is inappropriate as such noises will quickly dissipate within a few nozzle diameters and the flow will not develop into large-scale fluctuations. In this work, to achieve an appropriate inlet boundary condition for the injector, in addition to the main domain, a long auxilliary inflow pipe is simulated. A radial velocity component is added to the inlet boundary condition of the pipe to boost the instabilities. This radial velocity creates a large vortex at the early stage of the auxiliary pipe. This vortex brakes down to eddies and boosts the