Applied Thermal Engineering 137 (2018) 721–728

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research Paper

Experimental and analytical study on vapor phase and liquid penetration for a high pressure diesel injector

Raul Payri[a,*], Jhoan S. Giraldo[a], S. Ayyapureddi[b], Z. Versey[b]

[a] CMT – Motores Térmicos, Universitat Politècnica de València, Edificio 6D, 46022 València, Spain
[b] Thermofluids & Integration, Powetrain, Jaguar Land Rover, Abbey Road, Whitley, Conventry CV34LF, UK

HIGHLIGHTS

- A novel automatic software (HEEDS) was used to achieve robust spray model settings with short lead time.
- Lagrangian CFD model was successfully applied without additional model constants for a wide range of operating conditions.
- CFD model reproduces the sensitivities of injection pressure and temperature influence on penetration well, where marginally good for density change.
- Comparison of experimental data with CFD results show better agreement on vapor penetration than on liquid length.

ARTICLE INFO

Keywords:
Macroscopic characterization
Diesel spray
Lagrangian two-phase flow
Robust spray model
RANS

ABSTRACT

In this study, a macroscopic characterization has been performed on a solenoid diesel injector (2200 bar-8 hole nozzle) under various non-reacting but evaporative conditions. For vapor penetration a two pass Schlieren visualization set up was selected. A high speed camera was used to record high speed images of the injection event to analyze the transient evolution of the vapor phase of the spray. The transient liquid penetration of the spray has been measured via MIE-Scattering imaging technique using a high speed camera as well. Unsteady RANS based CFD Simulations have been performed to simulate the experimental conditions and correlation results are presented. Built-in models from commercial code StarCD have been used to model spray formation which includes submodels for turbulence, nozzle flow, break-up and fuel properties. A novel CAE process using an automation and optimization tool has been used to achieve robust model settings, and the final model prediction are compared with the experimental observation for the injector characterization with respect to the non-reacting spray penetration with change in ambient and injection conditions. The model correlates well with the sensitivities for temperature and injection pressures qualitatively however improvements required to capture the density effects mainly related to the mesh orientation, fixed time step size where further analysis required.

1. Introduction

Nowadays, internal combustion engines continue to be an important alternative for energy transformation. The ever more demanding fuel consumption standards and the concerns about the environmental impacts of these engines have pushed the industry into the search of new strategies and technologies. This encourages new studies for improving engine performance and its emissions.

The injection process has been mentioned as an important player in order to improve emissions and engine performance [1–6]. The spray formation includes complex and heterogeneous processes, majorly high-velocity jet flow, liquid droplet break-up, atomization, and evaporation of a dense liquid spray in a turbulent flow environment. The small temporal and spatial scales resulted from this process makes the diesel spray evolution a complicated problem.

To ensure a good mixture between the air and the fuel, the spray must penetrate into the combustion chamber and atomize. There are several parameters that help to characterize the diesel spray from a macroscopic point of view. The liquid length is an indicator of the evaporation capacity of the fuel and it is defined as the distance from the nozzle to the point where are found the ambient conditions necessary for evaporation. Mie scattering imaging technique is widely used by the engine community for the visualization of the fuel spray liquid phase. This technique consists in illuminating the fuel droplets with a light source and collecting the light scattered with a camera [7–9]. The vapor penetration largely determines both the mixing

* Corresponding author.
E-mail addresses: rpayri@mot.upv.es (R. Payri), jghi@mot.upv.es (J.S. Giraldo), sayyapu1@jaguarlandrover.com (S. Ayyapureddi), zversey@jaguarlandrover.com (Z. Versey).

https://doi.org/10.1016/j.applthermaleng.2018.03.097
Received 4 July 2017; Received in revised form 30 January 2018; Accepted 29 March 2018
Available online 06 April 2018

Nomenclature

RANS	Reynolds Average Navier Stokes
MPI2	Modified Max-Planck-Institute
RNG	Renormalization Group
MDO	Multi-disciplinary Optimization
SHERPA	Simultaneous Hybrid Exploration that is Robust, Progressive and Adaptive
CFD	computational fluid dynamics
RMSE	root mean square error
SOI	start of injection
L/D	Nozzle Length-to-Diameter Ratio
Mie	Mie-Scattering optical technique

process and the probability of collision against the chamber walls. It depends essentially on the instantaneous momentum of the spray in the nozzle. The Schlieren technique is able to distinguish gradients in the reflective index of a transparent medium [10,11], which allows clear identification of the vapor phase of the spray in evaporative conditions.

Since diesel combustion is predominantly a mixing-controlled reaction process, modeling the diesel spray formation process accurately is an essential prerequisite for modeling combustion events. The processes involved in the injection event are nonlinear and controlled by multiphase, diffusion phenomena. Modeling the interaction between those complex phenomena poses a huge challenge, and obtaining a unique model which can be robust for a wide range of in-cylinder conditions during fuel injection event is important. For industrial application, it is important that the model is viable with computational time and cost. A wide range of numerical models and sub-models exist in the literature [12–19] by various research groups which are inherently different in many aspects. The database is huge and detailed however still limited to the single-hole injector with moderate injection pressures.

In this work, a diesel multi-hole common rail injector (2200 bar) has been modeled using Lagrangian two-phase flow spray model. Relevant turbulence, nozzle flow, break-up models have been selected. Since the properties of diesel used in tests are unavailable various surrogate fuel properties have been applied. The sensitivities of model settings are included in the study, observations are discussed in Section 4. A novel CAE process using an automation and optimization tool has been presented which was used to investigate a range of model setting combinations to achieve robust spray model settings. The results from simulations obtained with the optimized parameters, are compared with results from visualization and characterization experiments carried out in this study.

2. Model development

2.1. CFD methodology

The numerical simulations are performed using the commercial CFD tool Star-CD. The turbulent flow field is resolved using the k-ϵ equation based Renormalization Group (RNG) turbulence model as this is in-cylinder combustion best practice [20]. The Lagrangian based two-phase flow model has been used to resolve spray formation. The nozzle inflow models are used to capture the nozzle hole exit velocities, the two models considered here are the Effective and MPI2 (modified Max-Planck-Institute) [21,20] models from Star-CD. The advantage of the MPI2 model is that it automatically determines whether cavitation occurs inside the nozzle and distinguishes whether it reaches the nozzle exit or ends inside the nozzle. For all simulations in this paper, the properties of n-dodecane ($C_{12}H_{26}$) were used as a surrogate for diesel fuel, these were taken from the internal Star-CD fuels library [22]. The properties of the surrogate can be seen in Table 1.

The injected liquid with high velocity starts to break-up into smaller droplets, the process comprises of primary breakup (i.e. atomization) and secondary breakup. A range of built-in sub-models is available with-in Star-CD to model this phenomenon. Atomization models differ in the way droplet size distribution and initial velocities are calculated. The difference between the droplet break-up models is the correlations that are used to estimate the time scale of the break-up process and the stable droplet diameter. The Huh atomization model and Reitz -Diwakar droplet break-up models have been used [20]. The Huh model calculates the spray cone angle during simulation so this is not required as an input.

The heat and mass transfer process is modeled using Ranz-Marshall correlation [23] to capture the evaporation process. The drag process and turbulence dispersion are modeled using stand correlation [20]. The inter-droplet collisions are not modeled, as the RNG turbulence model does not take this into account [20]. The droplet-wall interaction is not significant in this bomb case setup, however, the Bai model has been selected to consider any such process [22].

2.2. Computation grid and boundary conditions

The 3D computational domain used to represent the spray chamber fluid volume has been created as shown in Fig.1. The boundary with the injector is defined an adiabatic wall whereas the other boundaries are defined as pressure-outlets. The dimensions of the cuboid are maintained to enclose the non-reacting spray from the multiple injector holes whilst minimizing the influence of the boundary wall on the spray. The location of injector hole is defined at the center of the domain, at a certain depth below the wall surface for the same reason. All the dimensions and characteristics of the computational domain can be found in [22].

The coordinate system seen in Fig. 1 represents each of the 8 holes of the injectors, and was used to set the injection locations within the domain based on the geometrical specification of the injector. A uniform grid with cell size 0.8 mm has been selected as these are the settings used within full combustion models within the JLR 3D-thermofluids diesel combustion group, for which the tuned spray model parameters are required [22].

2.3. Automation and optimization: HEEDS

In this section, HEEDS is briefly explained. For this study the HEEDS MDO (Multi-Disciplinary Optimization) software was utilized in two manners, firstly to carry out a DoE (design of experiment study) into a set of tuning factors to investigate the effects of each on the penetration, and secondly to provide an automated workflow and optimization methodology to target a CFD solution which matches the experimental observations for a specific operating condition. SHERPA is the main algorithm used for optimization; the basic mechanism is that algorithm uses the results from numerical simulations to adapt to a new search

Table 1
n-Dodecane properties @ 298.15 K & 101325 Pa.

Properties	Value	Units
Molecular weight	170	kg/mol
Critical temperature	658.65	K
Critical pressure	1.835×10^6	Pa
Boiling temperature	489.48	K
Density	745.76	kg/m^3
Molecular viscosity	0.00137563	kg/ms
Surface tension coefficient	0.0248679	N/m