International Journal of Multiphase Flow 98 (2018) 110–119

Contents lists available at ScienceDirect

International Journal of Multiphase Flow

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

Modeling and numerical study of primary breakup under diesel conditions

A. Movaghar[a,*], M. Linne[b], M. Herrmann[c], A.R. Kerstein[d], M. Oevermann[a]

[a] Department of Mechanics, Chalmers University of Technology, Gothenburg, Sweden
[b] School of Engineering, University of Edinburgh, Edinburgh, UK
[c] Department of Mechanical and Aerospace Engineering, Arizona State University, Tempe, AZ, USA
[d] Consultant, 72 Lomitas Road, Danville, CA 94526, USA

ARTICLE INFO

Article history:
Received 15 February 2017
Revised 5 July 2017
Accepted 7 September 2017
Available online 14 September 2017

Keywords:
Spray
Primary breakup
Turbulence
One-dimensional turbulence
Direct numerical simulation (DNS)

ABSTRACT

A recently introduced stochastic model for reduced numerical simulation of primary jet breakup is evaluated by comparing model predictions to DNS results for primary jet breakup under diesel conditions. The model uses one-dimensional turbulence (ODT) to simulate liquid and gas time advancement along a lateral line of sight. This one-dimensional domain is interpreted as a Lagrangian object that is advected downstream at the jet bulk velocity, thus producing a flow state expressed as a function of streamwise and lateral location. Multiple realizations are run to gather ensemble statistics that are compared to DNS results. The model incorporates several empirical extensions of the original ODT model that represent the phenomenology governing the Weber number dependence of global jet structure. The model as previously formulated, including the assigned values of tunable parameters, is used here without modification in order to test its capability to predict various statistics of droplets generated by primary breakup. This test is enabled by the availability of DNS results that are suitable for model validation. Properties that are examined are the rate of bulk liquid mass conversion into droplets, the droplet size distribution, and the dependence of droplet velocities on droplet diameter. Quantities of greatest importance for engine modeling are found to be predicted with useful accuracy, thereby demonstrating a more detailed predictive capability by a highly reduced numerical model of primary jet breakup than has previously been achieved.

1. Introduction

Most concepts for current and future high efficiency, low emission internal combustion engines use direct injection of fuel via sprays. Understanding the breakup of the fuel spray is of high interest to further improve engine combustion. When fuel is injected into the engine, the relatively low density ratio between liquid fuel and gas creates strong aerodynamic interactions. The liquid surface becomes unstable and droplets are formed. This process is called primary breakup. Droplets formed from primary breakup break into smaller and smaller droplets in a process called atomization. Fuel droplets evaporate and the fuel vapor mixes with the ambient air to form a fuel–air mixture which ignites either via self-ignition (diesel engine) or spark ignition (gasoline engine). Complete control of fuel–air mixing from primary breakup to turbulent mixing of the fuel vapor with the air in the cylinder is of utmost importance to achieve clean and efficient combustion. As the highly consequential first step in this process, primary breakup plays a special role but is the least well understood.

Due to its technical importance, the breakup of turbulent jets has been investigated experimentally in great detail and many models have been proposed to simulate the breakup process. Eulerian–Lagrangian models are the current workhorses for practical engineering simulations of spray processes including fuel injection in engines. In the majority of these simulations primary breakup is not actually simulated. Instead, simple liquid blobs of the size of the injector diameter are introduced into the simulation. Further breakup of these blobs via secondary breakup is simulated with phenomenological models such as the Taylor analogy breakup (TAB) (O'Rourke and Amsden, 1987; Tanner, 1997) or wave models (Reitz, 1987). Nevertheless, due to the limited understanding of primary breakup, current numerical spray models for Reynolds-averaged Navier–Stokes simulations (RANS) or large-eddy simulations (LES) involve significant simplifications, and tuning is usually necessary every time the flow conditions are changed to achieve satisfactory results.

* Corresponding author.
E-mail address: movaghar@chalmers.se (A. Movaghar).

http://dx.doi.org/10.1016/j.ijmultiphaseflow.2017.09.002
0301-9322/

The limited understanding of primary breakup is due to the fact that experimental observation of the high-density region close to jet inlet is extremely difficult. As a result, much of the underlying physics leading to primary breakup is still unclear. Recently, sophisticated imaging techniques such as ballistic imaging and high speed shadow imaging have been able to provide more details of the primary breakup region (Linne, 2013; Rahm et al., 2015). Those new imaging techniques support the development of more predictive primary breakup models.

Direct numerical simulations (DNS) or high resolution large-eddy simulations (LES) offer an alternative way to study liquid–gas interface dynamics during primary breakup. Spatial and temporal resolution is limited only by the available computational resources, which have improved significantly over the past decades. Ménard et al. (2007) and Lebas et al. (2009) performed detailed simulations of jet breakup using a coupled level set/volume-of-fluid method with a ghost fluid approach. However they did not provide quantitative comparisons such as droplet size distributions with experimental data. Desjardins et al. (2010) and Desjardins et al. (2013) simulated the primary breakup using a conservative level set/ghost fluid approach. They used realistic turbulent boundary conditions at the injector inflow but no droplet size distributions were reported. Herrmann (2011) studied primary breakup of turbulent liquid jet under diesel conditions using the refined level set grid approach. He reported droplet size distributions and results of a grid refinement study providing detailed physical insight into primary breakup for moderate Weber and Reynolds numbers, which is difficult to acquire with experimental studies. However, routine use of DNS for industrial ranges of Weber and Reynolds numbers is still beyond the capacity of today's computers (Herrmann, 2010).

For practical simulations of engineering interest as well as to investigate the physics and scalings of primary breakup beyond the parameter range of DNS studies, a predictive and computationally affordable low-order model for simulating primary breakup is highly desirable. For this purpose, the one-dimensional turbulence (ODT) model has been proposed recently by Movaghar et al. (2017) as a primary breakup model. This stochastic modeling approach provides high lateral resolution by affordably resolving all relevant scales in that direction. The low computational cost of ODT compared to fully resolved three-dimensional DNS overcomes the limitation of DNS to moderate Reynolds and Weber numbers. As Movaghar et al. (2017) showed, after parameter tuning ODT has the capability to reproduce the results of experiments by Wu and Faeth (1995) and Sallam et al. (2002) for cases with high liquid/gas density ratio ($\rho_l/\rho_g > 500$). However, under real engine conditions liquid/gas density ratios are relatively low and aerodynamic effects have a significant effect on primary breakup.

In this work we apply the ODT approach presented in Movaghar et al. (2017) to the simulation of primary breakup of a round turbulent liquid jet injected into stagnant high pressure air under diesel-engine-like conditions. The main results, presented in the form of droplet size and velocity distributions as well as an axial profile of the mass rate of conversion from bulk liquid to droplets, are compared to the DNS study of Herrmann (2011).

The flows investigated in this study are governed by the incompressible Navier-Stokes equations for immiscible two-phase flow. The momentum equation is given by

$$\frac{\partial u}{\partial t} + u \cdot \nabla u = \frac{1}{\rho}\nabla p + \frac{1}{\rho}\nabla \cdot [\mu(\nabla u + \nabla^T u)] + \frac{1}{\rho} T_\sigma, \tag{1}$$

where u is the velocity, ρ the density, p the pressure, μ is the dynamic viscosity and T_σ the surface tension force which is nonzero only at the phase interface. All fluid properties are considered to be constant in each phase, allowing the viscous term to be simplified as shown below.

The one-dimensional turbulence model is outlined briefly in Section 2. A complete description of the ODT formulation used here is provided in Movaghar et al. (2017). The DNS formulation is discussed in detail in Herrmann (2008).

2. One-dimensional turbulence model

2.1. Time advancement processes

ODT is a stochastic model simulating the evolution of turbulent flow along a notional line of sight through a three-dimensional flow. Here it is used to simulate a nominally planar jet. The round-jet interpretation of this planar configuration is explained in Movaghar et al. (2017) and additional details of the execution of the simulations are discussed in Section 3.2. Denoting the jet streamwise direction as x, the ODT line of sight is oriented in the lateral (y) direction. This setup provides high lateral resolution of the relevant physics near the interface.

A Lagrangian picture is adopted, such that time advancement of ODT processes is interpreted as streamwise advancement based on assumed streamwise displacement of the ODT domain at the jet bulk velocity, denoted u_{bulk}. Taking the jet inlet to be the time origin in the ODT simulation, the ODT state at any later time t is interpreted as the state of the jet at streamwise location $x = u_{bulk}t$. Since the ODT state at given t represents the profile in y of all properties that are time advanced during the simulation, a single ODT realization can be interpreted as a representation of the instantaneous state of the jet in the $x - y$ plane. (In Movaghar et al., 2017, an array of y profiles of streamwise velocity, plotted at various x locations, illustrates this interpretation.) Each simulated ODT realization is initialized at the jet inlet with a size-D interval of liquid, where D is the inlet diameter of the round jet represented by the ODT simulation, and gas on both sides of the liquid.

Viscous transport on the ODT line is time advanced by solving

$$\partial u_i(y,t)/\partial t = \nu \partial^2 u_i(y,t)/\partial y^2, \tag{2}$$

where u_i with $i \in 1, 2, 3$ are the three velocity components and ν is the kinematic viscosity. The right-hand side of Eq. (2) is a specialization of the viscous transport in Eq. (1) based on the stated assumption of fixed fluid properties in each phase. A different ν value is needed in each of the phases and the liquid–gas density ratio is involved in the interfacial momentum–flux matching condition. Consistent with the idealized nature of the flow modeling, the gas-phase flow is parameterized rather than time advancing it using Eq. (2), as explained in Movaghar et al. (2017).

In ODT, turbulent advection is modeled by a stochastic sequence of events. These events represent the impact of turbulent eddies on property fields (velocity and any scalars that might be included) along the one-dimensional domain. During each eddy event, an instantaneous map termed the 'triplet map,' representing the effect of a turbulent eddy on the flow, is applied to all property fields. It occurs within the spatial interval $[y_0, y_0 + l]$, where y_0 represents the eddy location on the ODT line and l is the eddy size. A triplet map shrinks each property profile within $[y_0, y_0 + l]$, to one-third of its original length, pastes three identical compressed copies into the eddy range, and reverses the middle copy to ensure the continuity of each profile. The map mimics the eddy-induced folding effect and increase of property gradients. Formally, the new velocity profiles after a map are given by

$$\hat{u}_i(y,t) = u_i(f(y),t), \tag{3}$$