Eulerian Modeling of Internal and External Diesel Injector Flow

M.H. Moulai*, D.P. Schmidt
Department of Mechanical and Industrial Engineering
University of Massachusetts Amherst
Amherst, MA 01002 USA

Abstract
Understanding the relationship between a fuel injector nozzle and the ensuing spray is key to the continued effort to design cleaner diesel engines. Experimental investigation is essential but difficult due to the combination of high injection pressures and tiny length and time scales. Computational models extend our investigative ability. Here, we present results from a homogeneous relaxation model. The in-house CFD code used, implemented in the OpenFOAM framework, is based on the tendency of fuel vapor quality to relax towards thermodynamic equilibrium. Previous work has shown that this model is able to predict flash boiling and cavitation inside the nozzle. Now, as the fuel exits the nozzle, it is used to predict the behavior of a spray under the Engine Combustion Network’s “Spray A” condition. Our results are compared with Kastengren et al.’s x-ray radiography measurements of spray density distribution.

*Corresponding Author: maryammoulai@gmail.com
Introduction

The study of fuel sprays has been an area of interest for some time. Increasingly stringent emissions standards are a compelling reason to understand what influences spray behavior. Investigation has shown that flow through the nozzle largely determines the characteristics of the ensuing spray[1][2][3], and as a result researchers have turned their attention to the flow inside the injector.

There are a number of possible approaches to modeling the flow inside the injector nozzle, which is often undergoing phase change. Lagrangian models are common[4], as well as Eulerian models based on the Rayleigh-Plesset equation[5]. Another family of Eulerian models, homogeneous equilibrium models (HEM), treat the various phases as a homogeneous fluid with one set of governing equations, with scalar fluid properties used to account for the distribution of each phase[6]. The present work is based on the homogeneous relaxation model (HRM), introduced by Bilicki and Kestin[7].

HRMFoam, an in-house CFD code implemented in OpenFOAM[8], uses the homogeneous relaxation model to simulate the thermodynamic nonequilibrium that may occur inside the injector nozzle. Originally a flash boiling model[9][10][11], more recent work has shown that HRMFoam also agrees well with experiential measurements of cavitation[12][13][14].

Ultimately, the purpose of modeling the flow through the nozzle is to gain insight into the resulting spray. Lagrangian particle-tracking methods are commonly used for spray modeling[15]. Ning and Reitz used a homogeneous equilibrium model (HEM) coupled with an Eulerian-Lagrangian spray model to investigate the relationship between nozzle flow and spray characteristics[16]. Demoulin et al. used an Eulerian-Lagrangian approach with both DNS and LES modeling[17]. Purely Eulerian models have also been used, which treat the fuel and gas phases as a single fluid[18][19]. Along a similar tack, the present work attempts to model the entire flow from injection to breakup with HRMFoam. Simulation results are compared to Kastengren et al.’s X-ray radiography measurements of spray density[20].

Method

The nozzle flow is governed by conservation of mass (Eqn.1) and momentum (Eqn. 2):

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot \phi = 0
\]  

(1)

\[
\frac{\partial \rho \vec{U}}{\partial t} + \nabla \cdot (\rho \vec{U}) = -\nabla p + \nabla \tau
\]  

(2)

Schmidt et. al. derived a compressible pressure equation (Eqn. 3) from the continuity equation and a discretized momentum equation[21]:

\[
\frac{1}{\rho} \frac{\partial \rho}{\partial t} + \nabla \cdot \left( \rho \vec{U} \right) = -\nabla p + \nabla \cdot \left( \rho \vec{U} \right)
\]  

(3)

where \(a_p\) is the coefficient of the momentum contribution from the cell in question and \(H(U)\) is the sum of contributions from neighboring cells and source terms[10]. In previous work modeling internal flow with HRMFoam, the compressible terms have been neglected[13][14]. Work by Trask et al.[18] has shown that including compressible terms improves the accuracy of spray modeling[18].

Although the phases are assumed to be perfectly mixed, they are not in thermodynamic equilibrium and cannot be adequately described by an equation of state. Without a state equation to provide closure, the homogeneous relaxation model relies on the assumption, proposed by Bilicki and Kestin, that the instantaneous vapor fraction by mass \(x\) tends towards the equilibrium vapor fraction \(\bar{x}\) over an empirical timescale \(\Theta\)[7]:

\[
\frac{Dx}{Dt} = \frac{\bar{x} - x}{\Theta}
\]  

(4)

The equilibrium vapor fraction is a function of local enthalpy and pressure, and is obtained from a lookup table. The volumetric vapor fraction is then a function of the local and saturation densities:

\[
\alpha = \frac{\rho_l - \rho}{\rho_l - \rho_v}
\]  

(5)

What remains is to define \(\Theta\). Based on Racourceux’s ”Moby Dick” experiments[22], Downar-Zapolski proposed the following correlation for pressures exceeding 10 bar[23]

\[
\Theta = \Theta_0 \alpha^{-0.54} \psi^{-1.76}
\]  

(6)

where

\[
\psi = \frac{p_{sat} - p}{p_{crit} - p_{sat}}
\]  

(7)

and

\[
\Theta = 3.84 \times 10^{-7}
\]  

(8)
In order to model spray breakup, HRM-Foam also includes a transport equation for non-condensible gas, which is discussed in detail in [14]. The quantity \( y \) is introduced, to account for the distribution of noncondensible gas in the flow:

\[
y \equiv \frac{M_g}{M} \tag{9}
\]

where \( M_g \) is the mass of noncondensible gas per mass \( M \) of the overall fluid. This is somewhat analogous to the treatment of fuel vapor. From the volumetric composition of the fluid, where again the subscripts refer to the volume of each phase per volume \( V \) of the overall fluid,

\[
M = V^v + V^l + V^g
\]

\[
\frac{1}{\rho} = \left( \frac{M_v}{\rho_v M_f} + \frac{M_l}{\rho_l M_f} \right) \frac{M_f}{M} + \frac{M_g}{\rho_g M}
\]

\[
\frac{1}{\rho} = \left( \frac{x}{\rho_v} + \frac{1+x}{\rho_l} \right) (1-y) + \frac{y}{\rho_g} \tag{10}
\]

the following expression is derived [14],

\[
\frac{\partial p}{\partial x} = \left( \frac{1}{\rho_v} - \frac{1}{\rho_l} \right) (y-1)\rho^2 \tag{11}
\]

and substituted into 3.

Kastengren et al.’s experiment followed the Engine Combustion Network’s Spray A target condition as closely as possible, using Spray A injector 210677. Their work focused on the ”quasi steady-state,” [20] and so correspondingly the simulations in the present work are static and pressure-driven.

Using snappyHexMesh, a 870k cell hex mesh was generated from stereolithography images of injector 210677 (Fig. 1) made available by the Engine Combustion Network. The nozzle is 83.7 \( \mu \)m in diameter, and approximately 1 mm in length. The nozzle inlet is rounded with a radius of approximately 90 \( \mu \)m, so cavitation is not expected. Mesh cells were concentrated at the nozzle walls and outlet for better resolution of flow structures. The injector nozzle is asymmetrical, as noted by Kastengren et al. [24], and so it is expected that the spray will have asymmetric behavior as well.

A back pressure of 6MPa and an injection pressure of 150MPa were used. In the interest of computational stability, the injection pressure was increased gradually from 6MPa to its final value, and first order upwinding was used for divergence schemes. Slip conditions were applied at the needle and injector walls, to avoid the computational cost of fully resolving the boundary layer.

The diesel fuel was modeled using a surrogate property table, as discussed by Neroorkar et al. [12].

A k-\( \varepsilon \) model was used for turbulent closure. It has been observed [25] that the k-\( \varepsilon \) model may be too diffusive for nozzle applications, and could cause the model to over predict spray angle.

**Results**

The distribution of liquid throughout the domain is shown in Fig. 5. That the fluid in the sac and nozzle is all liquid indicates that the flow is noncavitating, as expected for a tapered nozzle with a rounded inlet. In the spray, a high density core extends from the nozzle outlet, surrounded by a lower density sheath that spreads as the plume develops. A self-similar region, where the jet spreads linearly, may be observed between 1mm and 6mm from the nozzle outlet.

For comparison with Kastengren et al.’s data, the liquid mass of the fluid was projected across the spray at \( y_1=0.1\text{mm} \) and \( y_2=0.6\text{mm} \) from the outlet. These results are shown alongside experimental data in Fig. 2. The mass projection at \( y_1 \) shows a mostly liquid jet exiting the nozzle, with very little diffusion, as observed in the experiments. At \( y_2 \), considerable spreading is evident, with the high density core occupying a small portion of the jet volume. Simulation results at \( y_1 \) are close to the experimental findings. The peak projected mass was under predicted by about 15\%, which may be a result of the overly diffusive k – \( \varepsilon \) RANS modeling.

Simulation results at predicted a spray angle of 2.4\(^{\circ}\), as opposed to 1.7\(^{\circ}\) in the experiments, which is likely also a result of an overly diffusive turbulence model. The simulation captured the asymmetry that Kastengren et al. observed in the down-
stream spray, which is shown in Fig. 3, and is a result of the asymmetry of the nozzle and outlet (see Fig. 4). There is some uncertainty\[26\] regarding the geometry of the Spray A nozzles, which may make it difficult to predict this asymmetry exactly.

Although there is good agreement with respect to spray angle, the peak projected mass at $y_2$ was over predicted by about 30%. This is likely a consequence of the relatively narrow exit plenum (shown in Fig. 1). The boundaries of the domain are close to the area of interest, and may be keeping the liquid core of the jet intact artificially. Further work will explore this issue.

![Figure 3. Axial slice of liquid density 6mm from the nozzle outlet.](image)

**Figure 2.** Projected density plots with experimental results at a) 0.1mm from the outlet and b) 6mm from the outlet.

**Conclusions**

HRMFoam, based on a flash-boiling model and originally developed to simulate internal nozzle flow, was used in this work for a completely coupled nozzle and spray simulation. Spray A nozzle 201677 was modeled under the Engine Combustion Network’s Spray A operating conditions, at steady state. Computational results were compared with X-ray radiography measurements performed by Kastengren et al. HRMFoam captured the eccentricity of the spray and plume development. Mass density projection predictions were close in the near exit region, under predicting by 15% at the peak, which is likely a result of the RANS model. Further downstream, HRMFoam showed the liquid core of the spray remaining intact for longer than in experiments, over predicting the projected mass density at the peak by about 30%. This is likely a result of the relatively narrow computational domain. Future work spray modeling work with HRMFoam will investigate these effects.

**Acknowledgments**

The authors would like to acknowledge the computing resources provided on the Fusion cluster at Argonne National Laboratory, and the financial support of General Motors Research and Development.
Figure 4. Closeup of asymmetrical nozzle outlet.

Nomenclature

- $h$: enthalpy
- $p$: pressure
- $\vec{U}$: velocity vector
- $\mathcal{V}$: volume
- $x$: instantaneous vapor quality
- $\bar{x}$: equilibrium vapor quality
- $\alpha$: vapor fraction of fuel by volume
- $\Theta$: vaporization time scale
- $\rho$: density
- $\tau$: stress tensor
- $\varphi$: mass flux
- $\psi$: difference between local and vapor pressures, dimensionless

Subscripts

- $f$: fuel
- $g$: gas
- $l$: liquid
- $v$: vapor

References


Figure 5. Volume plot of liquid mass fraction of total fluid. Opacity is linearly proportional to liquid concentration (regions with no liquid are transparent).