Assessment of an Eulerian atomization model on diesel spray CFD simulations

J.M. García-Oliver, J.M. Pastor*, A. Pandal
CMT-Motores Térmicos, Universitat Politècnica de València, Spain
jgarciao@mot.upv.es, jopasen@mot.upv.es, adrianpandal26@gmail.com

N. Trask, D. P. Schmidt
Departament of Mechanical and Industrial Enginnering, University of Massachusetts, Armhest, USA
nat.trask@gmail.com, schmidt@acad.umass.edu

Abstract

Diesel spray numerical modeling is still a challenging task due to the complex phenomena taking place and the wide range of spatial and time scales required to capture the physics involved. The aim of this work is to evaluate the so-called *Σ-Y* Eulerian atomization model, originally proposed by Vallet & Borghi, for CFD simulations of diesel engine sprays. This model has emerged as an alternative to the traditional discrete particle methods, widely employed on practical design applications for more than twenty years. Those Lagrangian liquid spray models are not well suited for the description of the primary atomization and the dense two-phase flow occurring at the near field of diesel sprays, where basic model hypothesis cannot be fully accomplished.

The *Σ-Y* model was developed on the basis of an Eulerian representation of the liquid/gas mixture by means of a single-fluid variable density turbulent flow. It is assumed that under high Reynolds and Weber numbers, large scale flow features are independent of viscosity and surface tension; but they affect the smallest scale flow, i.e., the size of the liquid fragments. From this assumption, spray liquid dispersion into the gas phase could be evaluated as the turbulent mixing of a variable density flow**.** The extent of the atomization process is computed from an interphase surface density equation (*Σ*), and then it is not required to presume any particular shape for liquid fragments. Liquid dispersion is calculated by means of liquid mass fraction (*Y*) transport equation using a traditional turbulent diffusion flux closure.

In this work a finite-volume solver for *Σ-Y* model equations has been written using the open-source CFD C++ library OpenFOAM. Model predictions have been compared to experimental data from free diesel sprays injected into a non-vaporizing quiescent vessel with ambient density conditions typical of current automotive DI diesel engines. Spray macroscopic characteristics were obtained by means of high-speed imaging. In order to evaluate droplet velocity field and size, Phase Doppler particle analyser (PDPA) technique measurements were also performed at different spray axial and radial locations. Single-hole conical nozzles were used in those experiments. Nozzle flow characterization has been obtained from mass flow rate and momentum flux measurements.

Different numerical and physical sub-model parameters effects on spray modeling results have been evaluated on a reference test case. The model set-up study indicates that accurate predictions of spray penetration as well on axial and radial velocity profiles can be simultaneously achieved. This result is obtained when κ-ε turbulence model constant $C_{1\varepsilon}$ for dissipation equation is set to typical values employed for round jet simulations. Further parametrical studies indicate that the proposed *Σ-Y* model approach and computational set-up remains valid for a broad range of injection pressure (30 to 130 MPa) and ambient density conditions (10 to 40 kg/m^3). Model accuracy is worse for low ambient density and injection pressure conditions. It is proposed that under these particular conditions, the slip between phases becomes more significant and the single velocity field assumption is less appropriate.

* Corresponding author: jopasen@mot.upv.es