ICLASS 2012, 12th Triennial International Conference on Liquid Atomization and Spray Systems, Heidelberg, Germany, September 2-6, 2012

CFD simulations of multi-hole Diesel injector nozzle flow and sprays for various biodiesel blends

C. A. Chasos*, C. N. Christodoulou and G. N. Karagiorgis
Mechanical Engineering Department, Frederick University, Cyprus
eng.cca@fit.ac.cy, eng.cc@fit.ac.cy and eng.kg@fit.ac.cy

Abstract

There is recent interest for the utilisation of renewable and alternative fuels and this is regulated by the European Union, which for the case of biodiesel currently imposes a lower limit of 7% by volume of biodiesel blending in diesel fuel. The biodiesel physical characteristics, as well as the biodiesel blending percentage in diesel fuel affect the injector and nozzle flow, the spray characteristics and the resulting air/fuel mixture and subsequently the combustion quality and the overall engine performance. In the present study, the computational fluid dynamics (CFD) methodology is applied, where the effects of biodiesel blending percentage in diesel fuel on the Diesel injector internal flow and spray characteristics are investigated and compared in order to identify the connections with the engine performance reported in literature. Four test fuels were examined, namely, pure diesel fuel, 10 % by volume of biodiesel blending in diesel fuel, 50 % by volume of biodiesel blending in diesel fuel and pure biodiesel, denoted B0, B10, B50 and B100, respectively. The fuels physical properties were determined from literature and the European standard for biodiesel fuels. For the investigations, a multi-hole Diesel injector geometry was modelled, and for the test fuels three-dimensional simulations of the injector internal flow were carried out. The commercial CFD code STAR-CD was employed for the investigations, where the Eulerian single-phase modelling methodology was used for the injector flow simulations and the Eulerian/Lagrangian two-phase flow modelling methodology was used for the spray simulations. From the injector internal flow simulations of the test fuels, the nozzle exit velocity was determined. The nozzle exit velocity and the physical properties of test fuels were used in an empirical expression to calculate the injected spray angles. The mass flow rate of the fuel, the nozzle geometry data and the spray angle were used as input in the Reitz-Diwakar spray atomisation model. The transient spray simulations were carried out using the Reitz-Diwakar spray atomisation model, along with the Reitz-Diwakar droplet break-up model and O'Rourke inter-droplet collision model. From the simulations, it was found that cavitation takes place at the nozzle inlet for all test fuels and with increasing biodiesel blending percentage the turbulence level in the injector nozzle is reduced. The resulting spray angle decreases with increasing biodiesel blending percentage by approximately 10 to 20 % from pure diesel B0 to pure biodiesel B100 fuel. Comparisons of the simulated sprays of the test fuels were performed, and B0 and B100 sprays at high pressure and temperature chamber conditions were compared against published spray photographs and penetrations. It was found that the resulting spray penetration increases when the biodiesel blending percentage is increasing. From the validation against the experiments, the shape of the body of the predicted sprays was wider than the experimental sprays, and the spray penetration was slightly overpredicted for both pure diesel and pure biodiesel sprays.

---

*Dr. Charalambos Chasos: eng.cca@fit.ac.cy