SPRAY IN AUTOMOTIVE APPLICATIONS: PART I

Guest Editors

Qingluan Xue

PREFACE

Fuel spray and mixture formation provide the initial conditions for combustion and impact the formation of emissions (soot, NO_x) from internal combustion engines (ICEs). For engine researchers and engineers, spray is a key component for creating a robust engine with high fuel efficiency and low emissions due to its complex stochastic nature. Within the limitation of experimental techniques, computational fluid dynamics (CFD) modeling has assisted in engine research and design by providing detailed transient 3D flow fields, parametric studies, and optimization. Supplied with initial and boundary conditions, CFD solves the Navier-Stokes and Williams spray equations for two-phase reacting flows in ICEs. The Lagrangian-Eulerian (LE) method and KIVA code describe the success of CFD modeling in engine research and design performed by academia and automotive industries worldwide. To further improve the efficiency of the modern engine for sustainability and to reduce design turnaround time, it is critical to further the predictability of CFD for understanding the fuel injection process and spray development. Researchers have continued to improve the LE model, exploring new CFD approaches (such as the Eulerian-Eulerian model), and researching large-eddy simulations (LES).

The goal of this special issue is to report the most recent advances in CFD for addressing the cavitating internal flow in high-pressure injectors, near-field dense sprays, two-phase coupling, and LES analysis. Using today's computational ability and power, it is still not possible to perform direct numerical simulations (DNS) of in-cylinder turbulent flows.

Issue 3 and 4 of *Atomization and Sprays* is an overview of current state-of-the-art CFD approaches and developments in simulating fuel spray processes in automotive research. Researchers from leading groups in mathematical and numerical methods for two-phase flows (cavitation, spray, turbulence, combustion) contributed to this issue, which represents the current state of each topic, and further provides outstanding

1044–5110/15/$35.00

challenges and future work directions. Issue 3 and 4 has five contributions and represent part I. Part II continues in Issue 5 and 6 and focuses more on experiments for model validation.

The paper by Emre et al. reviews, in detail, the advanced Eulerian-Eulerian approach based on a high-order moment method for polydisperse of two-phase flows. It starts from describing the derivations of the moment equations from the fundamental spray equation (Williams-Boltzmann kinetic equation). Their approach to closing the moment equations and their high-order method for polydisperse sprays are thoroughly discussed. Moreover, the implementation and validation of this Eulerian method in ALE formalism in IFP-C3D are provided to demonstrate its promising abilities and computational efficiency for engine sprays. Required future work and directions for Eulerian-Eulerian spray modeling are also described.

The paper by Habchi presents a Gibbs free energy relaxation model (GERM) for cavitation simulations of internal nozzle flows in a two-fluid multi-species approach. The two-fluid model and GERM cavitation model are comprehensively described with an emphasis on the relaxation methods at liquid-gas interfaces. The model and numerical approaches were implemented in IFP-C3D code. A qualitative validation was performed using experimental shadow-images on a single-hole nozzle. Interesting gaseous cavitation and effect of liquid compressibility findings are also presented in this work.

The paper by Torelli et al. describes the implementation of a spherical volume interface approach to couple the dispersed spray in the Lagrangian approach with carrier gas in the Eulerian approach, in order to reduce the grid dependency of traditional LE fuel spray simulations. Using this approach, the interaction between fuel droplet and gas is not limited to the host computational cell as in the traditional LE approach. Instead, an independent spherical volume around the droplet is created for mass, momentum, and energy transfers. The capability of this model is compared to a traditional LE approach. By experimental validation, good accuracy is demonstrated in terms of liquid and vapor penetration, as well as axial and radial mixture fraction profiles.

The paper by Banerjee and Rutland performed large-eddy simulations (LES) of spray-induced turbulent flow with engineering-level mesh resolutions. The work focuses on coherent structure (CS) construction and analysis of the turbulent flow around the liquid spray jet. A λ_2 definition of coherent structure is evaluated in 2D axisymmetry and used to visualize CS. This analysis helps explain the fuel-air mixing mechanism from large eddies breaking down into smaller eddies. The non-viscosity dynamic structure subgrid scale (SGS) and viscosity based Smagorinsky SGS models are compared using CS analysis. The study also shows the advantages of LES over RANS on flow structure prediction through CS visualizations.

The paper by Zhang et al. presents 3D CFD simulations of fuel sprays in stratified charge compression ignition (SCCI) engine conditions. The study focuses on examining the CFD predictability for assisting the investigation of SCCI/HCCI combustion. The predicted fuel distributions quantitatively agree well with experimental data at various injection timings. The numerical study shows that spray-to-spray interaction and spray-wall interaction play important roles in mixture formation. Some parametric studies were performed as well.