W2P136: SPRAY COMBUSTION SIMULATIONS USING FLAMELET-GENERATED MANIFOLD AND LARGE-EDDY SIMULATION

Armin Wehrfritz[1], Ossi Kaario[1], Ville Vuorinen[1], Martti Larmi[1], Cemil Bekdemir[2], Bart Somers[2]
[1]Aalto University, Finland [2]Eindhoven University of Technology, The Netherlands

The present research deals with Large-Eddy Simulations (LES) of diesel fuel sprays in internal combustion engine like conditions. The open-source CFD-toolbox OpenFOAM is used to carry out high-resolution simulations for the 'Spray A' reference case, applying the Flamelet-Generated Manifold (FGM) and implicit LES methods. The liquid fuel phase is modeled using Lagrangian Particle Tracking (LPT). The FGM method has been successfully applied in several fuel spay studies. In a recent study by Bekdemir et al. it has been shown that the FGM method is able to capture the main characteristics of diesel spray combustion very well. The aim of the recent research effort is to investigate the performance of the FGM method in conjunction with the implicit LES approach. Hence, simulations for the non-reacting, as for the reacting case are carried out and compared to experimental data. New inside concerning mixture formation, ignition behavior and combustion characteristics is expected due to the high degree of resolved turbulence and the accurate description of combustion processes.

Armin Wehrfritz: armin.wehrfritz@aalto.fi

W2P137: ASYMPTOTIC ANALYSIS OF FERROFLUID DROPLET COMBUSTION UNDER VERY LARGE MAGNETIC POWER

Cesar Cristaldo, Fernando Fachini, Instituto Nacional de Pesquisas Espaciais, Brazil

In this work, the influence of an external alternating magnetic field on the droplet combustion with dispersed magnetic nanoparticles (ferrofluid) is investigated. The response of the magnetic nanoparticles to the magnetic field produces a heat source. Under some limitation on the magnetic field frequency, the magnetic dipole is fixed on the particle, thus it rotates to align with the external alternating magnetic field. The rotation generates heat by friction with the liquid surrouding the nanoparticle. During the times in the period of oscillation in which the magnetic field has a negligible intensity, the Brownian motion of the fluid molecules is responsible for randomly misalignment the dipole. Under the influence of an external alternating magnetic field, the process of aligment and misalignment is repeated twice during each period. The heating process of the surrounding fluid is known as magnetic hyperthermia. Therefore, the hyperthermia can shorten the ferrofluid droplet heating and can augment the vaporization rate. Consequently, all properties depending on the droplet life time can be improved with the inclusion on the hyperthermia, e.g. design of shorter combustion chambers and the reduction of soot emissions. The current analysis considers the situation of very large magnetic heat power compared with that imposed by the heat transfer from the ambient atmosphere. Under such condition, therefore, a thermal boundary layer is established close to the droplet surface in the liquid side. This boundary layer matches the uniform temperature profile in the droplet core with that in the gas phase. Since in the thermal boundary layer the magnetic as well as the thermal heat power is of the same order, the temperature profile presents a maximum which value is higher than the temperature of the droplet surface. The model is valid up to the maximum reaches the boiling temperature. The numerical simulations show a significant reduction in the heating time and increase of the vaporizarion rate with the magnetic field frequency.

Cesar Cristaldo: profcristaldo@yahoo.com.br

W2P138: EXPERIMENTS AND COMPUTATION OF TRANSIENT IMPINGING TURBULENT JETS – TOWARDS A REFERENCE EXPERIMENT FOR SIMULATION DEVELOPMENT

Sebastian Kaiser[1], Rob Bastiaans[2], Cemil Bekdemir[2], Martin Goschuetz[1], Andreas Kempf[1], Nguyen Thuong[1], Jeroen van Oijen[2], Christof Schultz[1], Irenaeus Wlokas[1]
[1]University of Duisburg-Essen, Germany [2]Eindhoven Univ. of Technology, The Netherlands

This work-in-progress poster presents the initial results of a detailed investigation of flow, turbulence, and mixing in transient impinging jets. While these types of jets are common in many technical applications, in particular internal combustion engines, in terms of detailed, fundamental studies they have received much less attention than free or steady jets.

The flows investigated here are configured to capture the essential features of a Diesel-injection jet impinging on the piston-bowl wall, but are generic enough to be of interest in many other applications. A round jet is started, forms a fully turbulent far-field in a near-quiescent environment, and impinges on an orthogonal wall while inflow is stopped at the nozzle. Four different methods are used in this study, two experimental and two numerical: (1) Unsteady RANS simulation of an incompressible gaseous jet at atmospheric pressure. (2) Large-eddy simulation of the identical jet. (3) Laser-based imaging in a similar, but not identical, gaseous jet. (4) Laser-based imaging of an evaporating Diesel-like jet in an inert high-pressure environment.

For the initial work presented here, cases 1/2, 3, and 4 are not as similar as would be desirable, but the long-term goal is to establish a closely inter-related family of flows with detailed measurements suitable for validation of advanced simulations and submodels. In particular the high-pressure experiment is in direct extension of a target case of the Engine Combustion Network (ECN,http://www.sandia.gov/ecn/, editor: Lyle Pickett) and may be suitable for future incorporation in that international research network.

Presented on the poster are simulation results in terms of scalar and velocity, scalar snap-shots from the atmospheric-pressure experiment, and high-speed Schlieren visualization of the high-pressure jet. While not quantitatively comparable to each other yet, the results give an impression of the complexity of phenomena that can be addressed with these simple configurations. In the future, the flow configurations may be augmented to include features like cross-flow and heat transfer to the wall, which are very important in technical applications.

Sebastian Kaiser: sebastian.kaiser@uni-due.de

W2P139: Maximum Entropy Principle Predictions of Droplet Size Distribution Including Turbulence Effects

Ehsan Movahednejad[1], C.P. Chen[2] [1]Islamic Azad University, Iran [2]University of Alabama, Huntsville, USA

This paper describes the development of a new model to be incorporated into maximum entropy principle (MEP) formalism for prediction of droplet size distribution in primary breakup region. This model encompasses both the deterministic and stochastic aspect of spray droplet formation processes. The deterministic sub-models consider turbulence generation inside the nozzle and the unstable wave growth on jet surface before the liquid bulk breakup using the linear instability analysis. The stochastic stage of droplet formation after the liquid bulk breakup is modeled by statistical means based on the maximum entropy principle (MEP). Finally, these two sub-models are coupled together by source terms of momentum and energy. The main contribution of this research effort is to incorporate the turbulence effect in MEP modeling of droplet size distribution. Comparison between the model prediction and available experimental data for an annular and a gas turbine swirling nozzle indicates good agreement between the two.

Ehsan Movahednejad: emovahed@yahoo.com