High-Fidelity Modeling and Simulation of the NexGen Burner

Manu Kamin & Prashant Khare
University of Cincinnati

This presentation describes a numerical investigation conducted to comprehensively identify the various non-reacting multiphase flow processes through the NexGen burner. The burner geometry consists of a draft tube, stator, turbulator and a burner cone. The stator provides a swirl to the incoming air and the turbulator, which is placed immediately downstream of the stator, increases the unsteadiness of the swirling air. This turbulent swirling air then enters the burner cone where it mixes with the fuel, which subsequently burns. In this research talk, we will discuss the dynamics of airflow through the burner, including its interaction with the liquid fuel. The computations are conducted using an Eulerian–Lagrangian framework, where the gas phase is modeled by the compressible form of the Navier–Stokes equations and the liquid phase is treated in the Lagrangian frame. A modified version of two-way coupling, which takes into account the finite size of the dispersed phase, is used to account for the exchange of mass, momentum, energy and species between the two phases. Turbulence closure is achieved using the large eddy simulation technique (LES). As a first step, to validate the computational framework, the experiments of Ochs (2013) are simulated, which consists of the duct, stator, and turbulator (without the burner cone and fuel spray). The velocity magnitude measured at several locations downstream of the turbulator agrees very well with the present calculations. Next, two cases are conducted with the burner cone added to the rest of the geometry – one with and the other without fuel spray to identify the effects of fuel spray on the gaseous fluid dynamics. Detailed analyses are carried out for both cases to identify the flow structures that are formed inside the burner geometry, swirl, and production of turbulent fluctuations with and without spray injection.