ILASS Americas, 15th Annual Conference on Liquid Atomization and Spray Systems, Dearborn, MI, May 2002

Modeling Liquid Film Formation and Breakup in an Industrial Spray Nozzle

M.P. Fard[1], N. Ashgriz[2], J. Mostaghimi[3]
L.A. Prociw[4] and T.C.J. Hu[5]

[1]Simulent Inc., Toronto, Canada (mpasand@simulent.com)
[2,3]Professors, Mechanical Engineering, University of Toronto
[4,5]Pratt & Whitney Canada Inc.

Abstract

The formation and breakup of a liquid film in a fuel injector spray nozzle is studied using a 3D computational model for the simulation of complex free-surface flows. The model combines the solution of continuity and momentum equations with an algorithm for free surface tracking in presence of an arbitrary nozzle shape. The model simulates the swirling flow inside the injector chamber and predicts the formation of a conical liquid film at the nozzle exit. The disintegration of the liquid film into rings and ligaments is also shown using the numerical model.

Introduction

Industrial spray nozzles are designed based on experimental measurements. Because of the complexity of the flows existing in these systems, there is no accurate technique that can relate the nozzle design to the spray drop size and velocity distribution. Designers and researchers working in any of the above areas, first identify the drop size and velocity distribution for their particular application. Then, they choose a spray nozzle, which provides such spray characteristics. Most spray nozzles are characterized in ambient conditions and may not provide the same results under different conditions. There is no specific tool that one can relate the nozzle design to the spray drop size and velocity distribution. If a computer code makes it possible to see the resulting spray of a specific nozzle design it will be of great interest to any one working in this field. Such a code can be used to: design new nozzles; improve current nozzle designs; obtain the spraying characteristics of a nozzle such as mean droplet size, droplet size distribution, spray angle, spray pattern, mixing/swirling within the spray, droplets velocity in a certain distance from the nozzle exit; and investigate the spraying characteristics of a nozzle under different operating conditions.

The ultimate goal of this study is to develop a computer code for the simulation of the atomization process in industrial spray nozzles. In this paper we present simulation results for liquid film formation and breakup following the inflow of three liquid jets into a swirling spray nozzle. We use a 3D numerical model that combines the solution of Navier-Stokes equations with an algorithm for tracking the liquid free surface in presence of an arbitrary obstacle shape in the computational domain.

Numerical Method

Fluid Flow. Fluid flow in the injector nozzle is modeled using a finite volume solution of the Navier-Stokes equations in a 3D Cartesian coordinate system assuming laminar, incompressible flow. In this paper airflow in the nozzle is not considered. The surface profile of the deforming liquid is defined using the ''fractional volume of fluid'' scheme. In this method, a scalar function f is defined as the fraction of a cell volume occupied by fluid. f is assumed to be unity when a cell is fully occupied by the fluid and zero for an empty cell. Cells with values of $0<f<1$ contain a free surface. Surface tension is modeled as a volume force acting on fluid near the free surface. Details of the fluid flow model are given by Bussmann et al. [1].

Nozzle body. The body of the injector nozzle in the computational domain is a complex internal obstacle that affects the fluid flow. We treat internal obstacles by defining a liquid/obstacle volume fraction Θ, a scalar field whose value is equal to one in the fluid and zero in the obstacle. Cells having a value of Θ satisfying $0<\Theta<1$ are termed "partial flow cells" because a portion Θ of their volume is open to flow and the remaining portion $(1-\Theta)$ is occupied by an obstacle closed to flow. The obstacle is characterized as a fluid with infinite density and zero velocity. In the presence of internal obstacles, the finite volume approximations of the fluid flow equations are modified by defining a volume fraction Θ at the cell center, and area fractions Θ_x, Θ_y and Θ_z at the cell faces in the x, y and z directions, respectively. Boundary conditions that must be imposed on the liquid/obstacle interface are velocity boundary conditions. No-slip conditions on this interface are applied by defining "fictitious" velocities within obstacle cells adjacent to fluid cells. Velocities at the