Abstract Submitted
for the DFD13 Meeting of
The American Physical Society

Numerical Simulation of Liquid Sheet Instability in a Multiphase Flow Domain CHATTERJEE SOUVICK, Virginia Tech, SOUMIK MAHAPATRA, Jadavpur University, ACHINTYA MUKHOPADHYAY, IIT Madras, SWARNENDU SEN, Jadavpur University — Instability of a liquid sheet leading to the formation of droplets is a classical problem finding a wide range of multi-scale applications like gas turbine engines and inkjet printers. Numerical simulation of such a phenomenon is crucial because of its cost and time effective nature. In this work, the hydrodynamics in a custom designed nozzle is analyzed using Volume of Fluid method in Ansys Fluent. This innovative nozzle design includes an annular liquid sheet sandwiched between two air streams such that the inner air channel is recessed to a certain length. Such a recession leads to interaction between the two multiphase streams inside the atomizer resulting to an increased shear layer instability which augments the disintegration process. The numerical technique employed in this work couples Navier Stokes equation with VoF surface tracking technique. A parametric study with the hydrodynamic parameters involved in the problem, as well as the recession length, is performed while monitoring the axial and tangential exit velocities along with the spray cone angle. Comparison between the full 3D model and two different equivalent 2D axisymmetric models have been shown. The two axisymmetric models vary based on conserving different physical parameters between the 2D and 3D cases.

Chatterjee Souvick
Virginia Tech

Date submitted: 01 Aug 2013 Electronic form version 1.4