1093-76-39 **Wanjiao Liu**, Department of Mechanical Engineering, 111 Church Street SE, Minneapolis, MN 55455, and **Sean C Garrick*** (sgarrick@me.umn.edu), Department of Mechanical Engineering, 111 Church Street SE, Minneapolis, MN 55455. *A Lagrangian Volume-of-fluid approach for the simulation of turbulent multiphase flows.*

A new approach is proposed for modeling and simulation of turbulent multiphase flows. These flows are of great interest due to their broad applications. In the past several years, several researchers have successfully carried out direct numerical simulations (DNS) to investigate primary breakup in such flows. DNS is accurate, but requires extensive computational resources. In comparison, large eddy simulation (LES) is more practical, resolving only large-scale flow structures and modeling the small-scale effects. The major difficulty with LES of multiphase turbulent flows is the need to model the interfacial subgrid-scale terms. The subgrid-scale (SGS) surface tension force, for example, plays an important role in small droplet formation process. Our methodology combines the filtered density function (FDF) methodology with a Lagrangian volume of fluid (VOF) method. The FDF is advantageous in that the non-linear surface tension force appears (and other SGS terms) in a closed form and thus needs no modeling. The Lagrangian VOF is advantageous as it is highly accurate with no dispersive or dissipative errors (as opposed to eulerian approaches). We present the methodology as well as results from simulations of turbulent multiphase flows. (Received July 02, 2013)