Abstract Submitted
for the DFD15 Meeting of
The American Physical Society

Numerical simulations of bubbly Taylor-Couette turbulence in co- and counter rotating regime VAMSI SPANDAN, ROBERTO VERZICCO, DETLEF LOHSE, Physics of Fluids, University of Twente — Two-phase Taylor-Couette (flow between two co-axial independently rotating cylinders) is simulated using a two-way coupled Euler-Lagrange approach in which the bubbles are treated as point particles with effective forces such as drag, lift, added mass and buoyancy acting on them. The momentum equations for the fluid and the bubbles are solved in the frame of reference of the outer cylinder. While it is already known that when the outer cylinder is stationary, within a certain Taylor number range ($Ta \sim 10^6 - 10^8$) the bubbles disrupt the plume ejection regions and the coherent vortical structures leading to drag reduction, their effect and arrangement in the gap-width when both cylinders are rotating is still unknown. In this study we focus on studying the effect of bubbles on the angular velocity transport for various rotation rates of the cylinders. We find that the net percentage drag reduction persists even with a rotating outer cylinder, but is there a optimum for various rotation rates ? How does the spatial distribution of bubbles vary with in the co- and counter rotating regime ? These are some questions we attempt to answer in this work.

Vamsi Spandan
Physics of Fluids, University of Twente

Date submitted: 30 Jul 2015 Electronic form version 1.4