DIRECT NUMERICAL SIMULATIONS OF TWO-PHASE TAYLOR-COUETTE TURBULENCE

Vamsi Spandan[1], Rodolfo Ostilla-Monico [1], Roberto Verzicco [1,2], & Detlef Lohse [1]
[1] *Physics of Fluids, University of Twente, Netherlands*
[2] *Dipartimento di Ingegneria Meccanica, University of Rome "Tor-Vergata", Roma, Italy*

Abstract Two-phase Taylor Couette flow is simulated using the Euler-Lagrange approach, where the dispersed phase is treated as point particles with effective forces such as drag, lift, added mass and buoyancy acting on them. Two-way coupling is implemented between the carrier and the dispersed phase allowing us to study the interaction between the point like particles and the large scale flow structure in the carrier phase. Light buoyant particles are observed to be very effective in disrupting the coherent Taylor rolls, thus reducing the overall dissipation in the system and the overall driving torque.

INTRODUCTION

Taylor-Couette (TC) system, which consists of two co-axial independently rotating cylinders with a viscous fluid confined in between is one of the most studied problems in fluid mechanics for various reasons. The system is mathematically well defined through the Navier-Stokes equations and the respective boundary conditions, namely the angular velocities of the inner and the outer cylinder. Additionally it is a closed flow geometry with exact global balances between the driving and dissipation and an ideal system to study the interaction between boundary layers and bulk. Also it is accessible both experimentally and numerically via Direct Numerical Simulations(DNS) with high precision and accuracy due to its simple geometry and high symmetry. A series of studies in the past [4, 1, 2, 3, 5, 6, 7] showed that a low concentration (<5 %) of dispersed phase (bubbles or droplets) in the annular gap can considerably reduce the torque requrired to drive the cylinders. Various theories have been suggested to explain this effect, among them are theories based on effective compressibility introduced through (micro)bubbles, disruption of coherent vortical structures present in the single phase flow and also effects of bubble deformability. However the exact mechanism is still unknown and it is believed that different mechanisms are dominant in different flow regimes.

METHOD

We perform DNS of a two-phase TC (t-TC) system where particles are dispersed into the annular gap of a TC system and study its effect on the dynamics of the carrier phase. The equations governing the motion of the carrier phase are solved on a Eulerian grid using a second order accurate finite difference scheme.
The dispersed phase is advected by a point particle approach (under the condition that the particle size is smaller than the smallest relevant length scale of the flow). Effective forces such as drag, lift, added mass and buoyancy are calculated for each particle and the net total force is equated to the time derivative of the particle velocity.

$$\rho_p V_p \frac{dv}{dt} = (\rho_p - \rho_f)V_p g - C_D \frac{\pi r^2}{2}\rho_f |v-u|(v-u) + \rho_f V_p C_M (\frac{Du}{Dt} - \frac{dv}{dt}) \\ + \rho_f V_p \frac{Du}{Dt} - C_L \rho_f V_p (v-u) \times \omega \quad (1)$$

In the above equations u is the velocity of the carrier phase at the particle position, v is the velocity of individual particle while C_D, C_M, C_L are the drag, lift and added mass coefficients respectively.
Two-way coupling is implemented between the dispersed phase and the carrier phase i.e while the particle motion is governed by the velocity and velocity gradients in the carrier flow at the particle position, the particle also influences the local flow field in the carrier phase. This feedback or back-reaction force from the particle onto the carrier phase is distributed over a finite computational volume (instead of a finite number of computational nodes) to ensure that our solutions are grid independent. Two-way coupling simulations allow us to study the effect of these particles on the torque driving the inner cylinder.

RESULTS

In Figure 1(a) we plot the percentage of the drag reduction (DR) versus the inner cylinder Reynolds number (Re_i) for light buoyant particles injected into the annular gap of a TC system, where we can observe a consistent and statistically steady DR in the low Re_i regime. With increase in Re_i these particles lose their effectiveness in reducing the torque required to drive the inner cylinder. In Figure 1(b) we plot the averaged azimuthal velocity against the normalized radial position. There is a clear difference in the velocity profiles of the two-phase and single phase case. A smaller gradient of the azimuthal velocity near the wall results in a lower shear stress on the driving wall, and thus a lower driving torque.

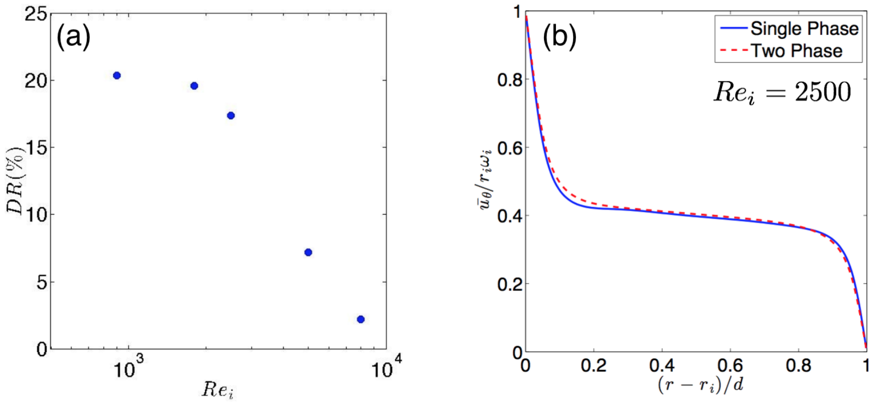

Figure 1. (a) Drag reduction (DR) versus the inner cylinder Reynolds nummber(Re_i) (b) Normalized average azimuthal velocity versus the radial position at $Re_i = 2500$ for both single phase and two-phase cases

In the low Re_i regime the particles are very effective in disrupting the coherent vortical structures in the system which are highly dissipative. This is shown in Figure 2 where we plot contours of the averaged azimuthal and axial velocity fields; the Taylor rolls present in the single phase case are disrupted by buoyant particles in the two-phase case. In another set of simulations we systematically increase the Froude number (in other words reduce the influence of buoyancy of the particles) and study its effect on the total dissipation in the system which is directly related to the driving torque.

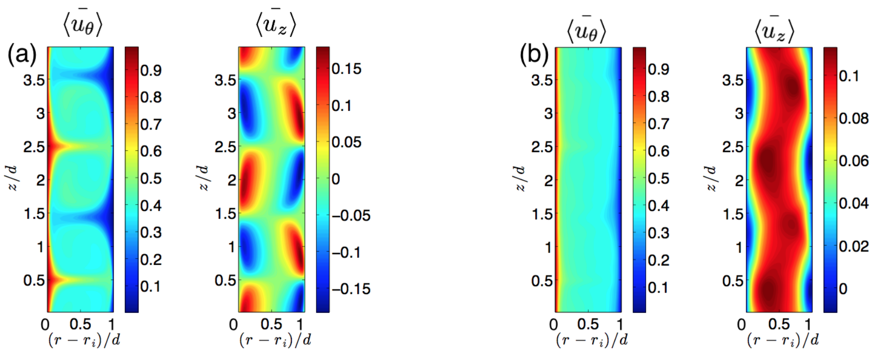

Figure 2. Contour fields of averaged azimuthal and axial velocity fields in the r-z plane at $Re_i = 2500$ (a) Single phase (b) Two phase.

References

[1] Steven L Ceccio. Friction drag reduction of external flows with bubble and gas injection. *Annual Review of Fluid Mechanics*, **42**:183–203, 2010.

[2] Antonino Ferrante and Said Elghobashi. On the physical mechanisms of drag reduction in a spatially developing turbulent boundary layer laden with microbubbles. *Journal of Fluid Mechanics*, **503**:345–355, 2004.

[3] Jiacai Lu, Arturo Fernández, and Gretar Tryggvason. The effect of bubbles on the wall drag in a turbulent channel flow. *Physics of Fluids (1994-present)*, **17**(9):095102, 2005.

[4] NK Madavan, S Deutsch, and CL Merkle. Measurements of local skin friction in a microbubble-modified turbulent boundary layer. *Journal of Fluid Mechanics*, **156**:237–256, 1985.

[5] Thomas H van den Berg, Stefan Luther, Daniel P Lathrop, and Detlef Lohse. Drag reduction in bubbly Taylor-Couette turbulence. *Physical review letters*, **94**(4):044501, 2005.

[6] Thomas H van den Berg, Dennis PM van Gils, Daniel P Lathrop, and Detlef Lohse. Bubbly turbulent drag reduction is a boundary layer effect. *Physical review letters*, **98**(8):084501, 2007.

[7] Dennis PM van Gils, Daniela Narezo Guzman, Chao Sun, and Detlef Lohse. The importance of bubble deformability for strong drag reduction in bubbly turbulent Taylor-Couette flow. *Journal of Fluid Mechanics*, **722**:317–347, 2013.