DLES 13

13th ERCOFTAC Workshop on Direct & Large Eddy Simulation

26-28 October 2022

Udine, Italy

UNIVERSITÀ DEGLI STUDI DI UDINE

ERCOFTAC

European Research Community On Flow, Turbulence And Combustion

WORKSHOP
Direct and Large-Eddy Simulation 13
October 26th-29th 2022, Udine, Italy

DNS OF DISPERSED BUBBLY TAYLOR-COUETTE TURBULENCE

A.D. Franken[1], S. R. Ephrati[1], P. Cifani[1,2], B.J. Geurts[1,3]
[1] Mathematics of Multiscale Modeling and Simulation University of Twente, The Netherlands
[2] Gran Sasso Science Institute, Viale F. Crispi 7, 67100 L'Aquila, Italy
[3] Multiscale Physics, Center for Computational Energy Research, Department of Applied Physics, Eindhoven University of Technology, PO Box 513, 5600 MB Eindhoven, The Netherlands
a.d.franken@utwente.nl

INTRODUCTION

Direct numerical simulation (DNS) of the Navier-Stokes equations describing a gas-liquid flow of dispersed deformable bubbles in water enables quantitative characterization of the modulation of turbulence arising from an immersed phase. This paper adopts the volume-of-fluid method as implemented in the TBFSolver (https://github.com/cifanip/TBFsolver) to simulate dispersed bubbly turbulence, at considerably higher global gas volume fractions compared to literature, using high-performance computing. Attention is given to (i) the fundamental resolution of bubble-bubble and bubble-wall interactions and (ii) to the clustering of bubbles in Taylor-Couette turbulence.

The numerical technique used in the TBFsolver is based on the volume-of-fluid (VOF) method in which the one-fluid formulation is adopted, i.e., a single set of equations is solved on the entire domain. To describe the dispersed embedded flow discontinuous material properties and interfacial terms associated with the bubbles are accounted for using a marker function f_i for each bubble i. Each bubble is given a marker which equals 1 in cells where the bubble fully occupies the cell, 0 where the fluid occupies the cell, and a value between 0 and 1 indicates that the cell contains a bubble interface. The value of the marker function is also referred to as the volume fraction. Given N bubbles, the marker functions are advected via

$$\frac{\partial f_i}{\partial t} + \mathbf{u} \cdot \nabla f_i = 0 \qquad \text{for } i = 1, \ldots, N. \tag{1}$$

The nondimensionalized incompressible Navier-Stokes equations and continuity equation are used to describe the flow:

$$\rho \frac{D\mathbf{u}}{Dt} = -\nabla p + \frac{1}{\mathrm{Fr}^2}\rho\hat{\mathbf{g}} + \frac{1}{\mathrm{Re}}\nabla \cdot (2\mu\mathbf{S}) + \frac{1}{\mathrm{We}}k\mathbf{n}\delta(n) \tag{2}$$

$$\nabla \cdot \mathbf{u} = 0 \tag{3}$$

Here $D\mathbf{u}/Dt$ is the material derivative of the velocity $\mathbf{u}$ with t the time. Moreover, p is the pressure, ρ the mass density, μ the viscosity, k is the curvature, $\hat{\mathbf{g}}$ is the normalized gravity vector, $\mathbf{S}$ the deformation tensor, and $\mathbf{n}$ is the normal vector to the interface. The dimensionless numbers are the Froude number $\mathrm{Fr} = U/\sqrt{gL}$, the Reynolds number $\mathrm{Re} = UL\rho_1/\mu_1$, and the Weber number $\mathrm{We} = LU^2\rho_1/\sigma$. Here L and U denote a characteristic length and a characteristic velocity, respectively. The subscript 1 denotes the continuous phase mass density.

The mass density and viscosity at a certain point follow from the marker functions and the properties of the continuous and dispersed phases. For instance, a cell with volume fraction c would have a density and a viscosity of

$$\rho = \rho_1\ (1 - c) + \rho_2\ c, \tag{4}$$

$$\mu = \mu_1\ (1 - c) + \mu_2\ c. \tag{5}$$

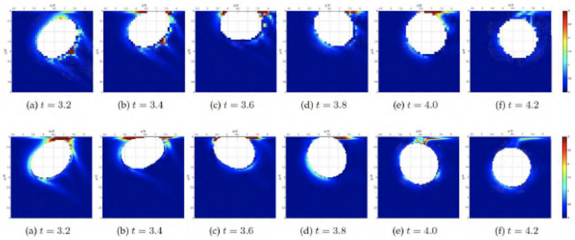
were $c = \max_i f_i$. The continuous surface force (CSF) method is used to model the surface tension term. This method replaces the delta function $\delta(n)\mathbf{n}$ by a smooth term, which is computationally easier but may induce spurious currents. Reducing such spurious currents can effectively be done by accurately computing the curvature of the interface for which a height function method is adopted.

A three-dimensional uniform Cartesian grid is used to discretize the domain and a staggered arrangement of the variables was selected. The spatial discretization of the convective term is based on the finite volume approach. Additionally, the QUICK interpolation scheme is implemented to avoid unphysical oscillations that may occur as the spatial resolution is too low. A second-order finite difference scheme is employed for the diffusive term. Finally, a third-order Runge-Kutta scheme is used to discretize the convection and diffusion terms of the Navier-Stokes equations, while a Crank-Nicolson scheme is employed for the surface tension term.

RESOLVED BUBBLE COLLISIONS

The motion of a bubble impacting a solid wall was simulated in detail. In Figure 1 the energy dissipation in the vicinity of the bubble is plotted at various instances as the bubble rises under an angle of 45 degrees toward the wall and subsequently bounces back. Although the main features are well represented on a grid of 128^3 the findings at 256^3 convey a more precise capturing of the dynamics. This is also expressed by the centre of mass trajectory of the bubbly in Figure 2(a) and the evolution of the y-component of the centre of mass in Figure 2(b). The vertical line in the latter figure corresponds to the moment at which the distance between the bubble centre and the wall is reduced to four grid cells of the finest grid. Despite the modest spatial resolutions of these brief encounters, the overall capturing of the motion appears highly accurate.

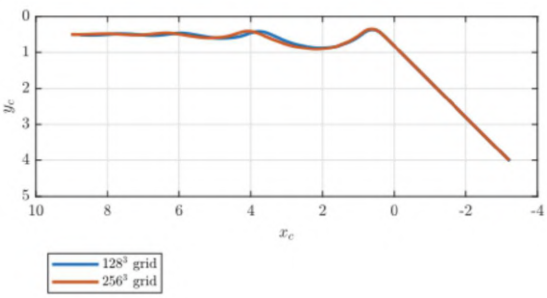
BUBBLY TAYLOR-COUETTE TURBULENCE



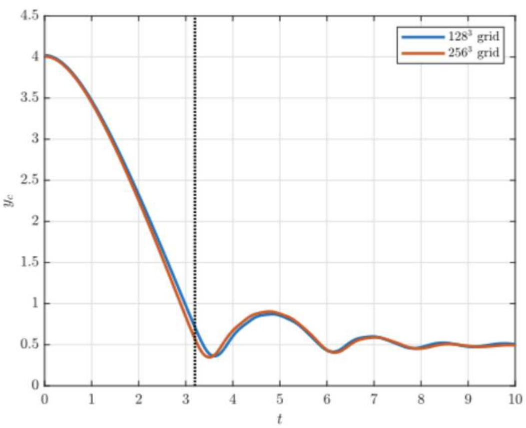
(a)

(b)

Figure 1: Energy dissipation in the vicinity of the bubble at different times using a grid with 128^3 (a) and 256^3 (b) grid cells.



(a)



(b)

Figure 2: Bubble trajectory (a) and y-component of the center of mass motion (b) at 128^3 and 256^3.

The numerical method in the TBFSolver can also be used to simulate a large number of bubbles. We concentrate on turbulence in Taylor-Couette flow as this presents a finite domain in which all conditions can be controlled with high fidelity. To simulate the turbulent flow in this configuration the TBFSolver was extended to cylindrical coordinates. In Figure 3 the domain is shown and a snapshot of the flow with 120 bubbles simulated on a grid with $[N_\theta, N_r, N_z] = [768, 192, 384]$ cells in the circumferential, radial and vertical directions respectively is presented in Figure 4. We observe some degree of clustering in the bubble concentration, which is further quantified in Figure 5. The simulations allow us to study the effects of buoyancy and deformability of large bubbles in turbulent TC flow, which appear to be crucial ingredients for drag reduction.

In the final presentation at the workshop, we expect to have completed further simulations of the Taylor-Couette flow and analysed the drag reduction phenomenon in more detail.

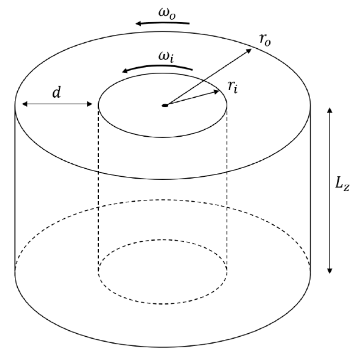


Figure 3: Domain for the Taylor-Couette flow.

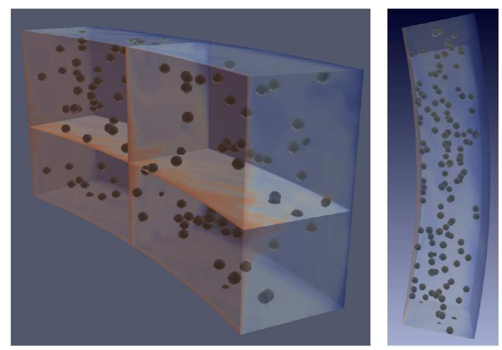


Figure 4: Snapshot of the velocity magnitude in two-phase TC flow - red/blue indicates high/low velocity. Bubbles are seen to cluster near the inner wall as seen clearly in the top-view (right figure).

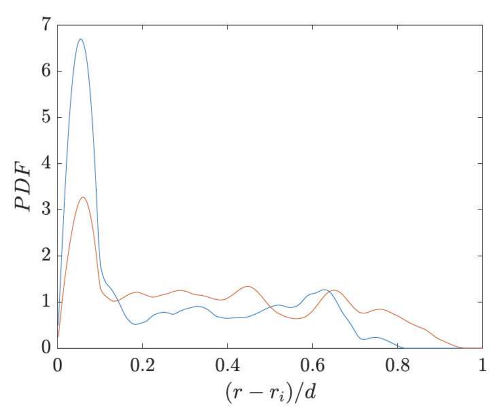


Figure 5: Probability distribution function of the radial coordinates of the gas phase. PDFs are based on averages over 2 flow-through times, taken after 18 (blue) and 48 (red) flow-through times.