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

VOLUME CONSERVATION METHODS FOR VOF-BASED LONG-TERM SIMULATIONS OF TURBULENT BUBBLE-LADEN FLOWS ON COARSE GRIDS

E. Trautner[1,*], J. Hasslberger[1], P. Cifani[2,3] and M. Klein[1]
[1] Department of Aerospace Engineering, University of the Bundeswehr Munich, Germany
[2] Faculty EEMCS, University of Twente, The Netherlands [3] Gran Sasso Science Institute, L'Aquila, Italy
*elias.trautner@unibw.de

INTRODUCTION

Turbulent bubble-laden flows are an integral part of numerous technical applications. The recent increase in computational power has allowed first large-scale Direct Numerical Simulations (DNS) of reduced-complexity bubbly flows at limited Reynolds numbers. However, bubbly flows in technical applications are usually characterized by high Reynolds numbers. Therefore, Large Eddy Simulation (LES) comes into focus for the design of technical devices. However, multiphase flow LES is still in an early development stage.

One of the most severe challenges for this field is the significant under-resolution of the interface dynamics, which leads to a variety of issues with numerical algorithms designed for two-phase flows, e.g. imprecise curvature computation or violation of the volume conservation. This work discusses the latter in the context of Volume-of-Fluid (VOF)-based simulations of turbulent bubble-laden channel flows. In general, the VOF method is known to exhibit significantly better volume conservation properties than, e.g., the Level-set or Front-tracking methods. Still, as will be demonstrated, minimal volume errors on a time-step level can add up to severe errors when bubbly flows are simulated in periodic domains and on coarse grids. The problem is reinforced for long simulation intervals, which are indispensable to compute converged statistics, in particular for quantities involving the gas volume fraction.

This work firstly demonstrates the problem by investigating the volume errors for a number of coarse-grid test cases. Subsequently, two volume conservation strategies for eliminating the problem are presented. Finally, the resulting flow and bubble statistics for the methods are compared.

NUMERICAL METHOD AND FLOW CONFIGURATION

The study is performed using the state-of-the-art code "TBFsolver" [1], which is based on the one-fluid formulation of the incompressible Navier-Stokes equations and the Continuous-Surface-Force approach for computing surface tension. The variables are arranged on a staggered grid and a height function method is used to compute the interface curvature. To avoid numerical coalescence of the bubbles, a multiple-marker formulation [2] is used, i.e., an individual VOF marker function f_b is advected for each bubble b:

$$\frac{\partial f_b}{\partial t} + u_i \frac{\partial f_b}{\partial x_i} = 0. \tag{1}$$

Equation 1 is solved using a geometrical reconstruction and advection algorithm retaining a sharp interface [3]. A split-direction advection is used.

The investigated configuration (see Fig. 1a) is a vertical downflow channel of size $L_x = 8H$, $L_y = 2H$ and $L_z = 4H$, where H is the half width and x, y and z denote the stream-wise, the wall-normal and the span-wise direction. The x and z directions are periodic, whereas no-slip walls are prescribed in y-direction. The flow is controlled by a constant pressure gradient corresponding to a friction Reynolds number of $Re_\tau = 590$. A total of 780 freely deformable bubbles with an initial diameter of $d_b = 0.25H$ are considered, leading to a gas volume fraction of 10%. The domain is discretized using an equidistant cubic mesh. The grid resolution is intentionally coarse (LES-like), such that it corresponds to either $d_b/\Delta = 10$ or $d_b/\Delta = 12.5$. Both the density and dynamic viscosity ratio are set to 20, and the Eötvös number is varied between $Eo = 0.67$ and $Eo = 3.75$. For this setup, the former leads to nearly spherical bubbles, while the bubbles are wobbling and deformed to an ellipsoidal shape for the latter.

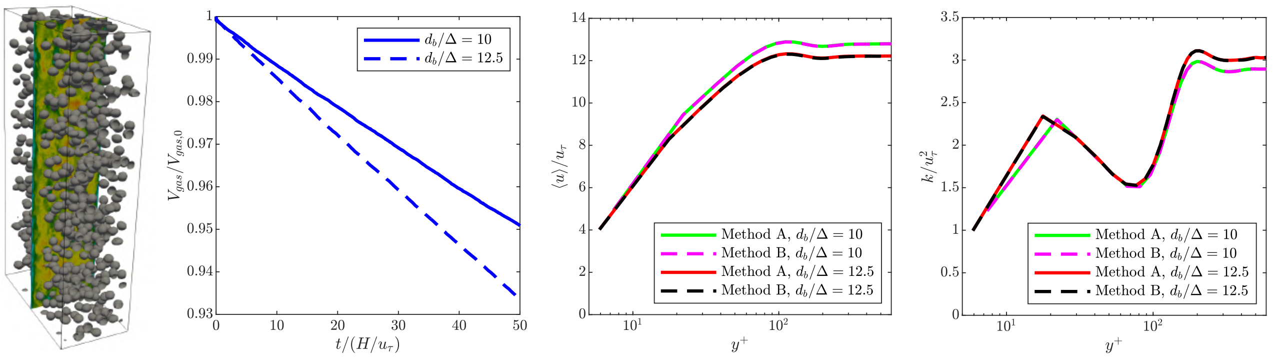
VOLUME CONSERVATION METHODS

The introduced method combines a multiple-marker formulation with a split advection algorithm. In this context, there are three root causes for the violation of volume conservation for the tracked gas phase:

1. Over- ($f_b > 1$) and undershoots ($f_b < 0$) of VOF values in single cells that are clipped to one or zero, respectively.
2. Resetting of cell VOF values to zero in cells with $0 < f_b < 1$, when no cell with $f_b = 1$ is detected within a region of $5 \times 5 \times 5$ cells around the respective cell.
3. Removal of small gas structures that can separate from the respective main bubble when high shear forces overcome the restoring surface tension force.

The first scenario can lead to both an increase and a decrease of gas volume, while the latter two can only lead to volume losses. The procedure explained in the second point is also used in comparable two-phase flow solvers. Under less challenging conditions (DNS-like resolution, lower Re_τ), the observed volume errors are negligible [1].

Figure 1b illustrates the global amount of gas volume loss for $Eo = 0.67$ and the two investigated resolutions. As the figure shows, several percent of the gas volume is lost during the



(a) (b) (c) (d)

Figure 1: Investigated configuration (a); Relative error of the global gas volume over the normalized time (b); Average normalized stream-wise velocity profiles $\langle u \rangle / u_\tau$ (c); Average normalized turbulent kinetic energy profiles k/u_τ^2 (d). u_τ is the friction velocity.

investigated time interval. The global volume error depends on the error per cell, the number of cells and the number of time steps ($CFL = 0.2 = \text{const.}$), which interact in a non-linear manner. For the investigated setup, this gives rise to a slightly larger error for $d_b/\Delta = 12.5$.

On an individual basis, VOF over- and undershoots (1) could be treated by locally redistributing volume to neighbor cells instead of clipping the values to one or zero. However, this does not resolve the issues related to points 2 and 3. Since it is undesirable to perform two different volume correction steps, the methods introduced in the following compensate the total volume error originating from all three issues in each time step. The multiple marker formulation forms the basis for the methods, i.e., the correction is performed for each bubble individually. Both methods rely on the given circumstance that the volume conservation error is minimal for a single time step and only has an impact over a large number of time steps.

Method A uses a straightforward approach to correct the volume error ΔV_b^t for bubble b in time step t. All N cells with $\text{tol} < f_b < (1 - \text{tol})$ are identified. Then, the volume error is corrected by adding or subtracting $\Delta V_b^t/(N \cdot V_{cell})$ to the VOF values of these cells. For the investigated setup, $\text{tol} = 1 \times 10^{-4}$ has been used. This avoids additional over- and undershoots in the correction step.

Method B applies a more sophisticated, parameter-free technique. The evolution of a single bubble's volume can be expressed as

$$\frac{\mathrm{d}V_b}{\mathrm{d}t} = \int_{V_b} (\nabla \cdot \boldsymbol{u}) \, \mathrm{d}V. \tag{2}$$

Consequently, imposing a fictitious velocity field with $\nabla \cdot \boldsymbol{u} \neq 0$ allows to correct the volume error for the bubble. Assuming $\alpha = \nabla \cdot \boldsymbol{u}$, where α is a constant valid for one time step and a single bubble, Eq. 2 can be approximated as

$$\frac{V_b^{\text{corr.}} - V_b^t}{\Delta t} = \alpha V_b^t, \tag{3}$$

where $V_b^{\text{corr.}}$ is the correct bubble volume to be conserved. Since the volume error in Eq. 3 can be measured, α can be computed. Now, a velocity field with $\nabla \cdot \boldsymbol{u} = \alpha$ can be determined as

$$\boldsymbol{u} = \frac{\alpha}{3} \left(x - x_0, \quad y - y_0, \quad z - z_0 \right)^T, \tag{4}$$

where $(x_0,\, y_0,\, z_0)$ denotes the center of the bubble. This velocity field is now used to perform an additional reconstruction and advection step for the bubble's VOF field. This can be interpreted as a slight dilatation (contraction) of the bubble for $V_b^t < V_b^{\text{corr.}}$ ($V_b^t > V_b^{\text{corr.}}$). It can be shown that this procedure preserves the bubble volumes to machine precision.

RESULTS

It is desirable that the volume conservation methods do not exhibit a notable influence on the flow and bubble behavior. One way to evaluate this is to compare the flow statistics resulting for the two different methods. Figure 1c shows the average stream-wise velocity profiles for the setups / methods at $Eo = 0.67$, while the respective average turbulent kinetic energy profiles are given in Fig. 1d. Unlike the grid resolution, which affects both these statistics, the two different volume conservation methods yield quasi-identical profiles at the same resolution. This indicates that, while both techniques successfully conserve the gas volume, their influence on the flow behavior is negligible.

Performing the simulations without applying one of the volume conservation strategies will negatively affect the statistics for long simulation intervals. Due to the continuous loss of gas volume (see Fig. 1b), the flow statistics will slowly deviate from the ones shown in Figs. 1c, 1d.

OUTLOOK

In the current work, four different setups will be investigated. In addition to varying the bubble resolution d_b/Δ, a variation between $Eo = 0.67$ and $Eo = 3.75$ will be performed to additionally evaluate the two proposed methods for deformed, wobbling bubbles. The methods will be assessed based on the flow and bubble statistics. Besides the global flow statistics, the oscillations of individual bubbles will be evaluated on the basis of their surface area fluctuations.

REFERENCES

[1] Cifani, P., Kuerten, J.G.M. and Geurts, B.J.: Highly scalable DNS solver for turbulent bubble-laden channel flow, *Comput. Fluids*, **172**, 67–83 (2018).

[2] Coyajee, E. and Boersma, B.J.: Numerical simulation of drop impact on a liquid-liquid interface with a multiple marker front-capturing method, *J. Comput. Phys.*, **228**, 4444–4467 (2009).

[3] Cifani, P., Michalek, W.R., Priems, G., Kuerten, J.G.M., van der Geld, C.W.M. and Geurts, B.J.: A comparison between the surface compression method and an interface reconstruction method for the VOF approach, *Comput. Fluids*, **136**, 421–435 (2016).