# DIRECT NUMERICAL SIMULATION OF TWO-PHASE TURBULENT PIPE FLOW

Néstor Balcázar[1], Lluís Jofre-Cruanyes[1], Oriol Lehmkuhl [2], Jesús Castro [1] & Joaquim Rigola [1]

[1] *Centre Tecnològic de Transferència de Calor, Universitat Politècnica de Catalunya, ETSEIAT, Colom 11, 08222, Terrassa (Barcelona), Spain, e-mail: cttc@upc.edu*

[2] *Termo Fluids S.L., Av. Jacquard, 97 1-E, 08222, Terrassa (Barcelona), Spain, e-mail: termofluids@termofluids.com*

Abstract DNS of two-phase turbulent flow at $Re_\tau \approx 360$ based on the friction velocity and pipe diameter is performed using the finite volume method and a conservative level set approach to track the interface between two fluids. Fully developed turbulent single-phase solution is used as the initial flow field, and a level set function is introduced to represent a second phase immersed in a liquid environment. The simulation domain $\Omega = [0, \delta] \times [0, 2\pi] \times [0, 2\pi\delta]$ is a circular pipe of diameter $2\delta$ with a periodic boundary condition in the streamwise direction. The simulation results are averaged to obtain the liquid and gas mean velocity distribution and the local void fractions. The liquid phase parameters are compared with the corresponding single-phase turbulent pipe flow.

## INTRODUCTION

The numerical simulation of interfacial and free surface flows is a vast and interesting topic in the areas of engineering and fundamental physics, such as the study of liquid-gas interfaces, formation of droplets and bubbles, sprays, wave motion and others [6]. The continuous improvement of computational power continuously extends the range of affordable problems. More over, the phenomena we consider often happens on scales of space and time where experimental visualization is difficult or impossible. In such cases, numerical simulation may be a useful prod to the intuition of the physicist, the engineer, or the mathematician.

On the other hand, the numerical simulation of turbulent flows is a powerful tool for investigating and understanding the turbulence phenomena and shed light on the physics of turbulent flows such as high Reynolds bubbly flows. The advances in computational fluid dynamics (CFD) together with the increasing capacity of parallel computers have made possible to tackle such complex problems by using high performance numerical techniques such as direct numerical simulation (DNS) [5, 1]. Thus, considering the actual state-of-the-art in interface tracking methods [2] and DNS of complex flows, the present work propose an step forward into this line by aiming at the resolution of turbulent multiphase flows such as "Turbulent two-phase flow".

## MATHEMATICAL FORMULATION AND NUMERICAL METHOD

The time dependent Navier Stokes equations in a finite time interval $[0, T]$ and in a three dimensional domain $\Omega$ are written in conservative form as

$$\frac{\partial}{\partial t}(\rho \mathbf{v}) + \nabla \cdot (\rho \mathbf{v}\mathbf{v}) = -\nabla p + \nabla \cdot \mu \nabla \mathbf{v} + \nabla \cdot \mu (\nabla \mathbf{v})^T + \sigma \kappa \nabla \phi + \rho \mathbf{g} \qquad \text{in } \Omega \times [0, T] \tag{1}$$

$$\nabla \cdot \mathbf{v} = 0 \qquad \text{in } \Omega \times [0, T] \tag{2}$$

$$\beta = \beta_1 \phi + \beta_2 (1 - \phi) \qquad \text{with } \beta \in \{\rho, \mu\} \text{ in } \Omega \times [0, T] \tag{3}$$

where $\mathbf{v}$ is the velocity field, $\rho$ is the density, $\mu$ is the dynamic viscosity, $\mathbf{g}$ is the gravity acceleration, $p$ is the pressure, $\delta_\Gamma = \delta(\mathbf{x} - \mathbf{x}_\Gamma)$ is the Dirac delta function, $\mathbf{x} \in \Omega$, $\mathbf{x}_\Gamma \in \Gamma$, $\Gamma = \{\mathbf{x} \mid \phi(\mathbf{x}, t) = 0.5\}$ is the interface between two fluids, $\phi(\mathbf{x}, 0) = 1/\left(1 + e^{d(\mathbf{x},0)/\varepsilon}\right)$ is the conservative level set (CLS) function [2], $d(\mathbf{x}, t) = min\{\|\mathbf{x} - \mathbf{x}_\Gamma\|\}$ is a signed distance function introduced by [3, 4], $\sigma$ is the surface tension coefficient and $\kappa$ is the interface curvature given by

$$\kappa = -\nabla \cdot \left( \frac{\nabla \phi}{\|\nabla \phi\|} \right) \tag{4}$$

The evolution of $\phi$ in a free divergence velocity field is given by the advection equation

$$\frac{\partial \phi}{\partial t} + \nabla \cdot \phi \mathbf{v} = 0 \tag{5}$$

A reinitialization equation is added in order to mantain the profile and thickness of the interface constant,

$$\frac{\partial \phi}{\partial \tau} + \nabla \cdot \phi (1 - \phi) \mathbf{n} = \nabla \cdot \varepsilon \nabla \phi \tag{6}$$

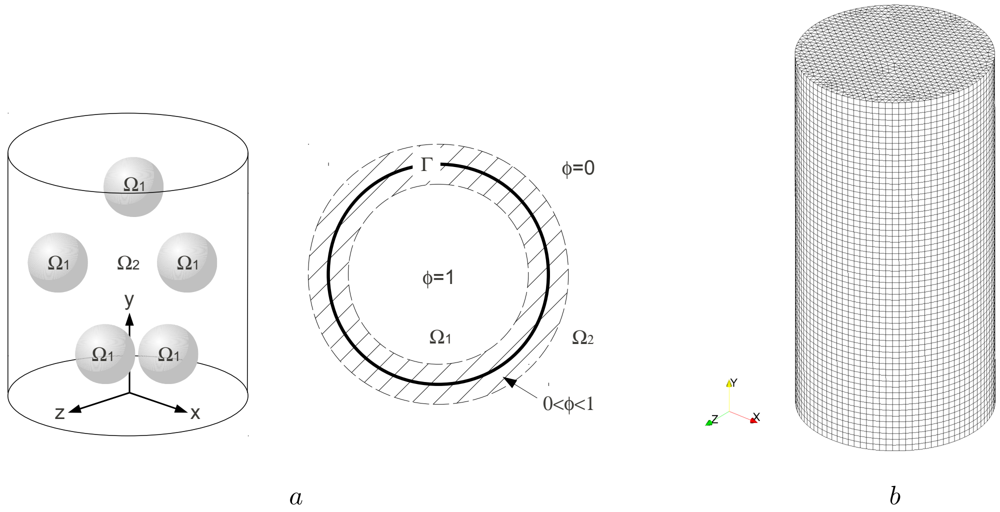


**Figure 1.** (**a**.) Simulation domain $\Omega$. Subdomain $\Omega_1$ is ocupied by the light fluid immersed in the heavier fluid wich fill the subdomain $\Omega_2$. The duct is periodic in the streamwise direction. (**b**.) Unstructured mesh.

where $\tau$ is a pseudo-time, $\varepsilon$ is a smooth parameter proportional to mesh cell size and $\mathbf{n} = (\nabla\phi/\|\nabla\phi\|)_{\tau=0}$ is a vector field of interface normals.

The DNS simulations of gas-liquid two phase flow corresponds to Reynolds number $Re = 5300$ based on the bulk velocity of liquid and tube diameter, including the effect of bubble deformation and coalescence in the flow regime. The two phase pipe flow simulation is performed using the finite volume method based approach and a Level Set Method for interface capturing. The Navier Stokes equations and interface propagation equation will be solved on sufficient fine grids so that all details, including the shape of the bubbles and the flow aroud them, are fully solved. To make the simulation as simple as possible, the density of the second phase is taken to be one-tenth of the liquid density. The flow is assumed to be periodic in the streamwise direction, driven by an imposed constant pressure gradient and gravity buoyancy force acting in the vertical direction.

The complex interaction between gas phase and turbulent liquid flow is numerically resolved and compared with time-averaged flow characteristics of single phase flow.

## References

[1] Igor A. Bolotnov, Kenneth E. Jansen, Donald A. Drew, Assad A. Oberai, and Richard T. Lahey Jr. Detached direct numerical simulations of turbulent two-phase bubbly channel flow. *International Journal of Multiphase Flow* **37**: 647–659, 2011.

[2] Olsson E., Kreiss G., A conservative level set method for two phase flow, *J. Comput. Phys.* **210**: 225-246, 2005.

[3] Osher S., Sethian J.A., Fronts propagating with curvature-dependent speed: Algorithms based on Hamilton-Jacobi formulations, *J. Comput. Phys.* **79** 175–210, 1988.

[4] Sussman M., Smereka P., Osher S., A Level Set Approach for Computing Solutions to Incompressible Two-Phase Flow, *J. Comput. Phys.* **144** 46-159, 1994.

[5] Ivette Rodríguez, Ricard Borell, Oriol Lehmkuhl, Carlos D. Pérez Segarra, Assensi Oliva, Direct numerical simulation of the flow over a sphere at $Re = 3700$, *J. Fluid Mech.*, **679** 263-287, 2011.

[6] Néstor Balcázar, Lluis Jofre, Oriol Lehmkuhl, Joaquim Rigola, Jesús Castro, Numerical Simulation of Incompressible Two Phase Flow by Conservative Level Set Method, In Conference on Modelling Fluid Flow (CMFF'12), 1-8, 2012.