11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20–25, 2014, Barcelona, Spain

DIRECT NUMERICAL SIMULATION OF THE FLOW AROUND A SPHERICAL BUBBLE IN A TURBULENT PIPE FLOW

L. Jofre[1,*], N. Balcázar[1,*], O. Lehmkuhl[1,2], R. Borrell[2] and J. Castro[1]

[1]Heat and Mass Transfer Technological Center, Technical University of Catalonia,
C/ Colom 11, 08222 Terrassa, Spain. E-mail: cttc@cttc.upc.edu
[2]Termo Fluids S.L., Av. Jacquard 97 E, 08222 Terrassa, Spain.
E-mail: termofluids@termofluids.com

Key words: *Bubble, DNS, Multiphase, Turbulence, Pipe Flow.*

This work aims at investigating the flow over a high-Reynolds-number clean spherical bubble fixed on the axis of a turbulent pipe flow. In particular, the first- and second-order statistics of the flow over the bubble and the forces acting on it will be fully analyzed. In fact, Merle et al. [1] state that the flow over the bubble is influenced by all the length and time scales down to the Kolmogorov microscales. Hence, the use of kinetic-energy-conserving schemes [2] is a requirement that must be considered in order to obtain trustable results.

The problem under consideration falls into the classification of bubbly flow, which differs in three important aspects from bluff body flows. First, when the liquid is pure enough has the possibility to slip along the surface of the bubbles, in contrast to the flow over rigid bodies where the no-slip condition prevails. Second, due to the very small relative density of bubbles compared to that of the liquid, almost all the inertia is contained in the liquid, making inertia induced hydrodynamic forces particularly important in the prediction of bubble motion. Third, the shape of the bubbles may change with the local forces, adding new degrees of freedom to an already complex problem.

In detail, the problem consists in a spherical bubble, with diameter d and density ρ_{bl}, placed fixed at the center of a circular pipe, having diameter D and length L, that contains a fluid of density $\rho_{fl} = 10\rho_{bl}$. The physics of the problem depends on the bulk and bubble Reynolds numbers. The bulk Reynolds number, is defined as $Re = \rho_{fl} u_b D/\mu_{fl}$, where u_b refers to the bulk velocity. Similarly, the bubble Reynolds number is expressed as $Re_{bl} = \rho_{bl} u_c d/\mu_{bl}$, being u_c the time-averaged centerline velocity of the flow. In particular, this test chooses ρ_{bl}, Re, Re_{bl} and u_b as 1, 6000, 500 and 1, so that the size of the bubble is comparable to the Taylor microscale of the flow and is about ten times the Kolmogorov microscale. The relation between pipe length and diameter is $L = 5D$ and the bubble's diameter is chosen as $L = 78d$. In this way, the pipe is long enough to include even the largest-scale structures, and the velocity defect in the bubble's wake is significantly decreased before re-entering through the inlet boundary due to the periodic condition.

The Navier-Stokes equations in the variable-density incompressibility limit have been discretized by means of second-order finite-volume conservative mesh schemes. Such schemes preserve not only mass and momentum, but also the kinetic energy. In addition, the grid spacing has been selected in order to satisfy requirements for the correct resolution of both the pipe and bubble's boundary layer and bubble's wake. Hence, the mesh is made up of 5.4M cells, resulting from rotating 128 times a 2-D grid, which is discretized by means of 128 points (concentrated near the pipe wall and bubble) in the radial direction, and 330 points (accumulated at the bubble) in the y direction. In detail, the 2-D mesh contains the first radial point near the pipe wall at $r^+ = 0.94$ — similarly to the grid spacing used for the direct numerical simulation of the turbulent pipe flow at $Re = 5300$ by Eggels et al. [3]. Furthermore, the mesh is generated such that at least three cells lie within the bubble's boundary layer, which, according to Legendre and Magnaudet [4], is a necessary condition in order to properly solve all the scales in the vicinity of the bubble.

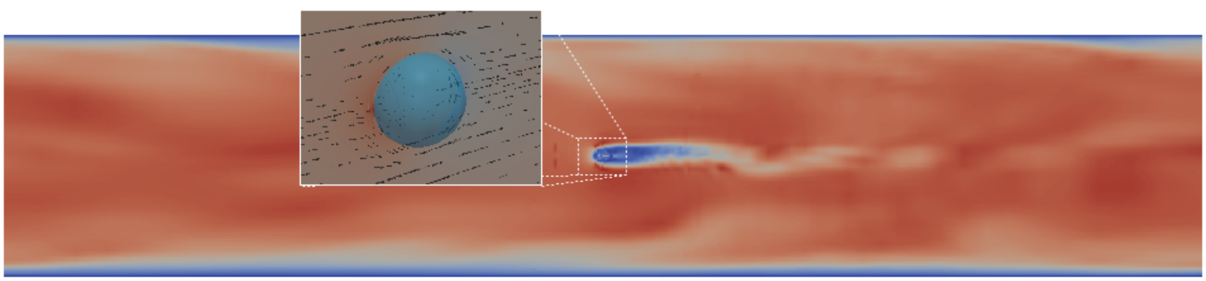

Figure 1: Flow around a spherical bubble in a turbulent pipe flow.

REFERENCES

[1] A. Merle, D. Legendre and J. Magnaudet. Forces on a High-Reynolds-Number Spherical Bubble in a Turbulent Flow. *J. Fluid Mech.*, Vol. **532**, 53–62, 2005.

[2] L. Jofre, O. Lehmkuhl, J. Ventosa, F. X. Trias and A. Oliva. Conservation Properties of Unstructured Finite-Volume Mesh Schemes for the Navier-Stokes Equations. *Numer. Heat Tr. B-Fund.*, Vol. **65**, 53–79, 2014.

[3] J. G. M. Eggels, F. Unger, M. H. Weiss, J. Westerweel, R. J. Adrian, R. Friedrich and F. T. M. Nieuwstadt. Fully Developed Turbulent Pipe Flow: a Comparison between Direct Numerical Simulation and Experiment. *J. Fluid Mech.*, Vol. **268**, 175–209, 1994.

[4] D. Legendre and J. Magnaudet. The Lift Force on a Spherical Bubble in a Viscous Linear Shear Flow. *J. Fluid Mech.*, Vol. **368**, 81–126, 1998.