

# The 3D Hysing benchmark

Lionel GAMET

January 16, 2019

## 1 Introduction

This case is a reference test case for VoF simulations. It is an elementary quantitative benchmark configuration that was originally established in 2D by Hysing et al. [1]. Adelsberger et al. [2] have published a 3D equivalent of the same benchmark. The case consists in a single rising bubble in a quiescent liquid. The test case number 2 as described by Hysing et al. [1] is modeled here.

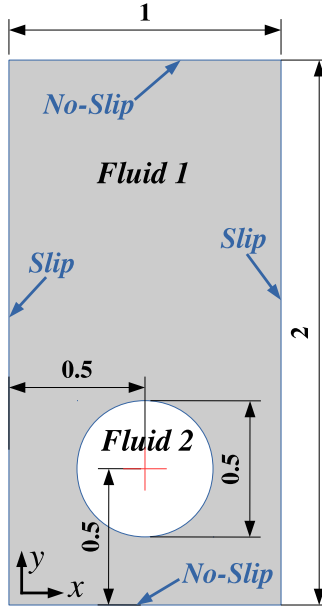


Figure 1: Configuration and boundary conditions for 3D bubble benchmark.

The 3D case setup is schematized in Figure 1. First phase (liquid) properties are  $\rho_1 = 1000 \text{ kg/m}^3$ ,  $\mu_1 = 10 \text{ kg/(ms)}$ , while second phase (gas) takes  $\rho_2 = 1 \text{ kg/m}^3$ ,  $\mu_2 = 0.1 \text{ kg/(ms)}$  as physical parameters. The surface tension is  $\sigma = 1.96 \text{ kg/s}^2$ . Gravity is taken as  $g = 0.98 \text{ m/s}^2$ . The chosen values are not representative of existing fluids. This test case has originally been designed for

comparison purposes between CFD solvers. The domain is of size  $1 \times 2 \times 1$  in the  $x$ ,  $y$  and  $z$  directions.

## 2 Setting up the test case in OpenFOAM

This test case uses incompressible VoF solvers in OpenFOAM, namely `interFoam` or `interIsoFoam` solvers.

A uniform Cartesian grid built with `blockMesh` is used for 3D simulations. The default grid size is  $20 \times 40 \times 20$ . When running the grid sensitivity script (see section 3), finer grid levels will be used. The bubble is initialized as a sphere in 3D using the `setAlphaField` utility.

**NB: We use the `invertAlpha` option in `setAlphaFieldDict` to reverse the initial field from a drop to a bubble. This option is only available starting from OpenFOAM+ v1812.**

In the `fvSchemes` file, a Crank-Nicolson second order time scheme with blending coefficient 0.9 is chosen. `Gauss limitedLinearV 1` is used to treat the convective term, and `Gauss vanLeer` is used for the  $\alpha$  convective term for MULES simulations. The `Gauss linear` scheme is used by default for all gradient terms.

In the `fvSolution` file, the GAMG implicit solver is used for pressure terms, while the smooth solver is used for the velocity. The PIMPLE algorithm uses 3 `nOuterCorrectors` and 3 PISO correctors (`nCorrectors=3`). It was found that `momentumPredictor` needed to be set to true to get a correct solution in terms of rising velocity and bubble sphericity, in particular with `isoAdvector`. Both `isoAdvector` and MULES numerical parameters are present and appear separately in `fvSolution`, so that both `interFoam` and `interIsoFoam` can be run from the same input files.

A constant time step is used, starting at  $\Delta t = 0.002$  s for the  $20 \times 40$  coarsest level and reduced for finer grids to keep the maximum CFL number below 0.05. Computations are run up to time  $t = 3.5$  s in 3D.

Post-processing quantities of interest are described in details in [1, 2]. These are the vertical position of the bubble centroid, the bubble rise velocity, the bubble sphericity, bubble volume and area. All these quantities are computed through a coded `functionObject` inlined in the `controlDict` file. The results appear in the log file of the solver. We use a `sampledIsoSurfaceCell` object, defined as the isosurface  $\alpha = 0.5$ , to compute the bubble area. We also use volume integrals of the gas fraction overall the computational domain to compute the bubble volume, centroid and velocity. A bubble equivalent diameter named  $D_A$  is computed from the bubble volume. A bubble equivalent diameter named  $D_B$  is computed from the bubble area. The sphericity is defined in 3D as the ratio  $D_A^2/D_B^2$ . This number takes the value 1 at  $t = 0$  as the bubble is initialized as a perfect sphere, and then decreases with time as the bubble rises and deforms.

Finally, the bubble shape is output at `writeTime` frequency through a `surfaces` sampling `functionObject`, based upon an isosurface  $\alpha = 0.5$ . Both interpolated

and constant iso bubble shapes are output.

### 3 Running the case

The `Allrun` script will run the default grid size at  $20 \times 40 \times 20$ . The grid is first constructed by running `blockMesh`. Then the `setAlphaField` utility is run to initialize the bubble as a sphere. Finally, the application (default to `interFoam`) is run. Post-processing quantities of interest (bubble volume, area, centroid, velocity and sphericity) are extracted from the solver log file through a `grep` command.

The `Allrun_sensitivity` is a script at a level above `Allrun`. It is used to run a grid sensitivity for both solvers `interFoam` and `interIsoFoam` with a number of cells 20, 40, 80, 160 and 320. As the test case generates large grids, the automatic running has been commented in the sensitivity script. The sensitivity script will thus only generate running directories that you can execute manually later, or on another computer.

For comparisons, reference 3D data can be found at the bibliography link.

### References

- [1] S. Hysing, S. Turek, D. Kuzmin, N. Parolini, E. Burman, S. Ganesan, and L. Tobiska, “Quantitative benchmark computations of two-dimensional bubble dynamics,” *I.J.N.M.F.*, vol. 60, no. 11, pp. 1259–1288, 2009. [Online]. Available: <http://www.featflow.de/en/benchmarks/cfdbenchmarking/bubble.html>
- [2] J. Adelsberger, P. Esser, M. Griebel, S. Groß, M. Klitz, and A. Rüttgers, “3D incompressible two-phase flow benchmark computations for rising droplets,” 2014, proceedings of the 11th World Congress on Computational Mechanics (WCCM XI), Barcelona, Spain, also available as INS Preprint No. 1401 and as IGPM Preprint No. 393. [Online]. Available: <http://wissrech.ins.uni-bonn.de/research/projects/risingbubblebenchmark/>