Hands-on session 13 – Bubble Column Polydispersed

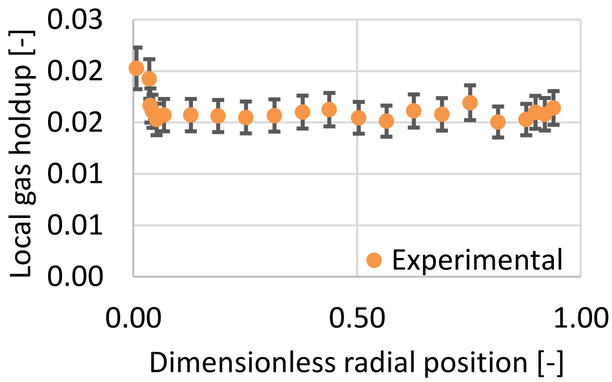
Goal

The purpose of this tutorial is to provide guidelines for the simulation of polydispersed bubbly flows. This tutorial demonstrates how to do the following:

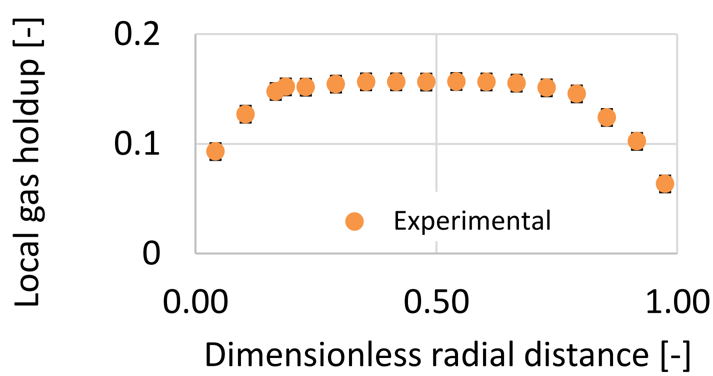
1. set up a new phase representing the large bubbles
2. use the Eulerian multiphase model
3. solve the problem using appropriate solver settings
4. postprocess the resulting data

1 Introduction

We know that at low flow rates the bubble adjust themselves homogeneously in the space. Indeed, the typical volume fraction distribution far away from the injector is basically flat as in the plot below.



Above a certain velocity we will have bubbles with multiple size, from small to relatively large. Large bubbles get deformed and experience a lift force towards the center of the domain, while small bubbles experience a lift force towards the wall. Hence, the typical volume fraction distribution at large velocity will be like the one below.

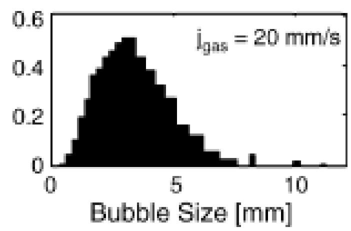


If the velocities are not too high, and the domain is not too small there will be no phenomena of coalescence and break-up and we could treat the bubbles as two separated groups. The differentiation between the two groups point would be the diameter size where the lift force changes sign which depends on the model employed, e.g. Tomiyama, Ziegenhein, Hessenkemper.

1.1 Problem description

In this tutorial you will simulate two cases:

- Case 1: increase the surface velocity to 20 mm/s. Evaluate the mass flow rate of air to be introduced as large bubbles assuming that the large bubbles can be represented by a constant bubble diameter equal to 5.5mm diameter while the small group is still represented by a constant value equal to 3.0 mm. To evaluate the flow rate for each group, use the bubble distribution provided by Keppler et al. 2007, in the picture below:



- Case 2: increase the surface velocity to 30 mm/s. As there is no experimental value, please assume that the large bubble diameter is constant and equal to 7.0 mm and that the small bubble group represents 60% of the total mass flow rate, and the large bubble group represents 40% of the total mass flow rate.

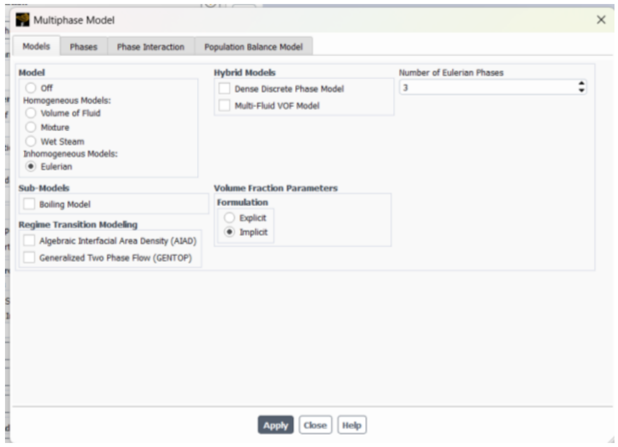
Please:

1. evaluate the gas holdup;
2. evaluate the interfacial area;
3. observe and analyze the disperse phase dynamics;

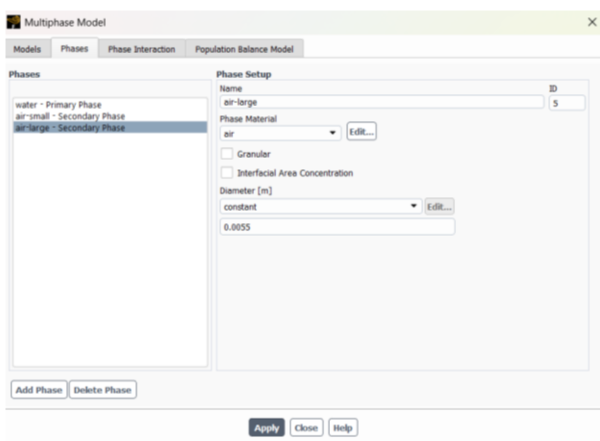
2 Simulation of the bi-dispersed

2.1 Definition of the large bubble phase

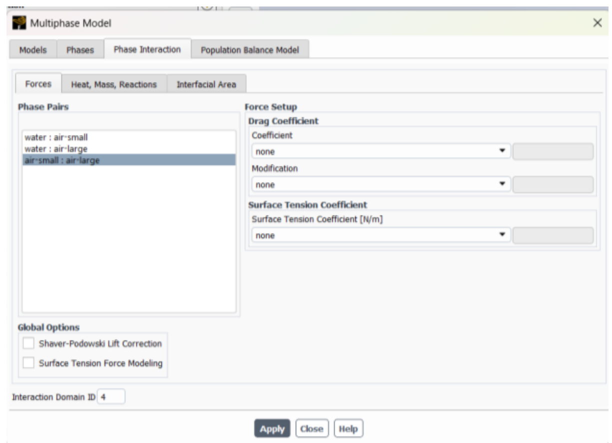
Open the previous case and rename it as "caseName_polydispersed". Open the multi-phase model and, in the Models tab, increase the Number of Eulerian phases to 3.



Click on the Phase tab and rename the existing air phase to air-small and the new phase as air-large. This approach is basically a three-phase approach without interaction between the air phases. Set the diameter of the air-large phase to 0.0055 m.

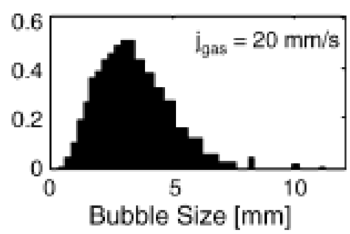


In phase interaction make sure that in air-small - air-large phase pairs there is no interaction. The degassing boundary will be changed automatically to outlet as the software does not allow degassing for two different phases.



2.2 Inlet mass flow rate definition

In the experimental work of Krepper, which this tutorial refers to, the bubble distribution at surface velocity 2 cm/s is given as below. Evaluate the mass flow rate of air to be introduced as large bubbles assuming that the large bubbles can be represented by a constant bubble diameter equal to 5.5mm diameter while the small group is still represented by a constant value equal to 3.0 mm. To evaluate the flow rate for each group, use the bubble distribution provided by Keppler et al. 2007, in the picture below:



2.3 Other settings

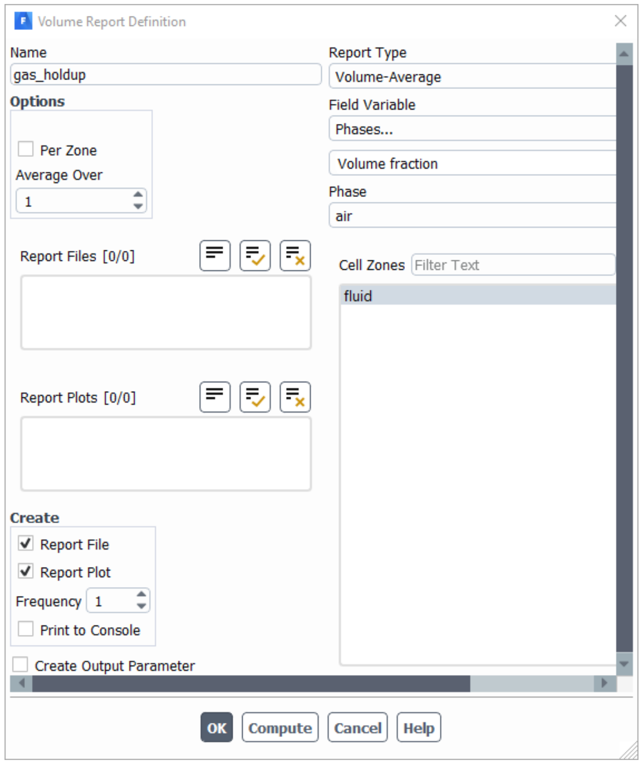
Leave all other settings as in the previous tutorial.

2.4 Report settings

Reports and graphics are intended to provide insights in the global and local flow phenomena.

Report tab → New > Volume Report of type Volume- Average

Enter gas holdup for name. Enable report file and report plot. Select phases and volume fraction from the variable drop-down list and search for air. In the list of cell zones select fluid.



Surface tab → Create → Surfaces → Iso-Surface

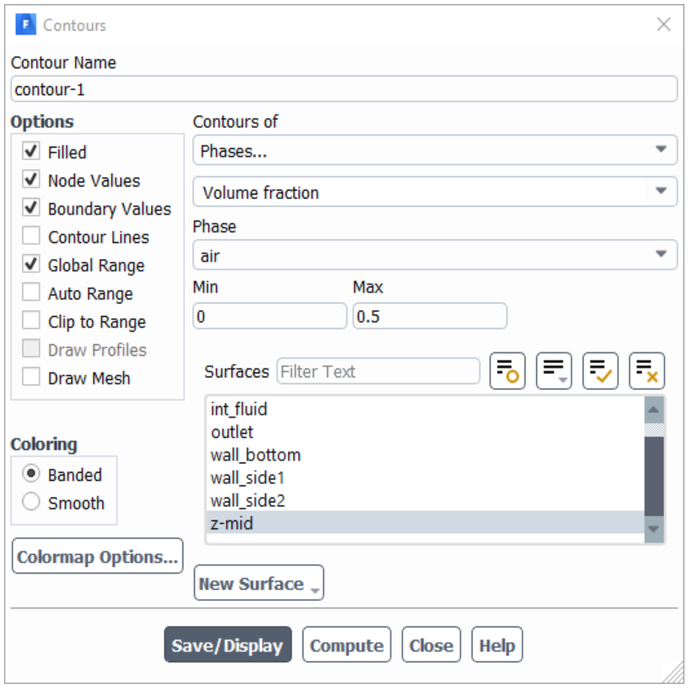
In the Iso-Surface dialog box, select mesh and Z-Coordinate from the surface of constant dropdown lists. Enter 0.01 for Iso-Values. Enter z-mid for new surface name. To activate this tab you need to initialize the solution. In the Solution Initialization task page, select inlet on the Compute from drop- down list. Click Initialize.

Graphics tab → Mesh →New

In the mesh display dialog box, select only faces in the options group box. From the list under surfaces select only wall_side1 and wall_side_2. Click Save/Display and close the Mesh/Display dialog box. Similarly create another mesh object, selecting the zones inlet and wall_bottom, while keeping both faces and edges enabled

Graphics tab → Contours →New

Enable filled under options. Disable auto range and clip to range. Select Phases and Volume fraction from the. Contours of drop-down lists.Select air from the Phase drop-down list. Enter 0 for Min and 0.5 for Max. From the list under Surfaces select z-mid. Click Save/Display and close the Contours dialog box.



Results → Scene → New

In the Scene dialog box that opens, under Graphics select the check boxes for contour-1, mesh-1 and mesh-2.

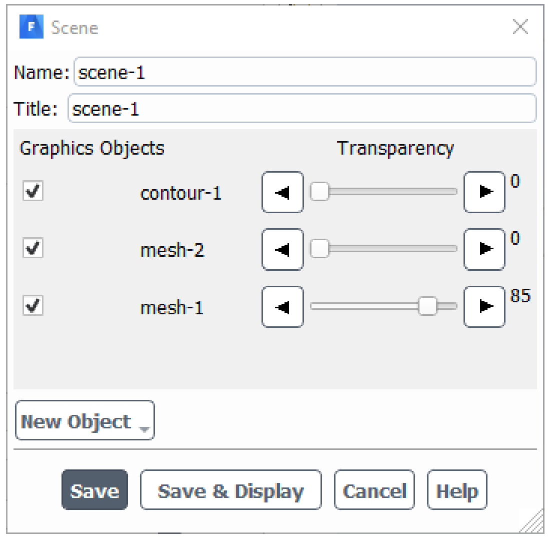
Note: Mesh-1 is the mesh object with wall_side1 and wall_side2. The transparency does not have to be the exact value, just something close. You can experiment with other values to find what you think has the best appearance. Click Save & Display and then you can close the Scene dialog box.

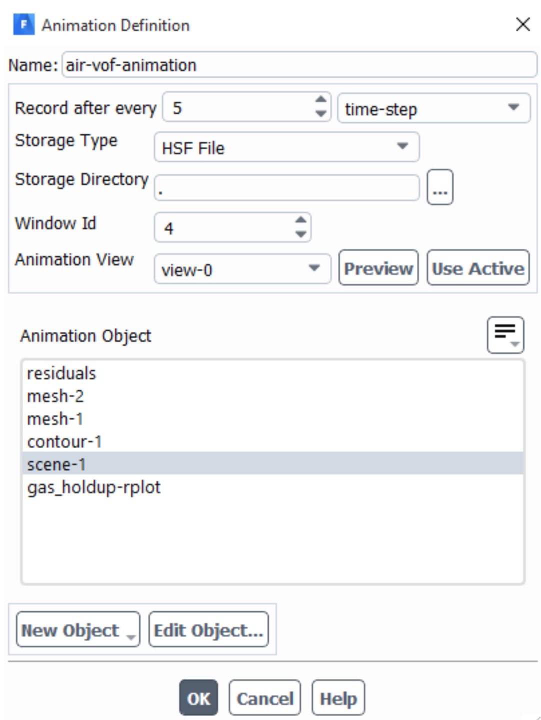
Solution tab → Activities group → Autosave .

In the Autosave dialog box, enter 250 for Save Data File Every (Time Steps).

Solution tab → Activities group → Create → Solution Animations

Rename the animation object to air-volume-fraction animation; Enter 5 for record after every; Select scene-1 from the list under Animation Object; Re-orient the view as desired and click on Use Active


Scene
Name: scene-1
Title: scene-1
Graphics Objects
Transparency
contour-1
0
mesh-2
0
mesh-1
85
New Object
Save
Save & Display
Cancel
Help


Animation Definition
Name: air-vof-animation
Record after every
5
time-step
Storage Type
HSF File
Storage Directory
.
Window Id
4
Animation View
view-0
Preview
Use Active
Animation Object
residuals
mesh-2
mesh-1
contour-1
scene-1
gas_holdup-rplot
New Object
Edit Object...
OK
Cancel
Help

2.5 Initialization and calculation settings

Solution Initialization task page

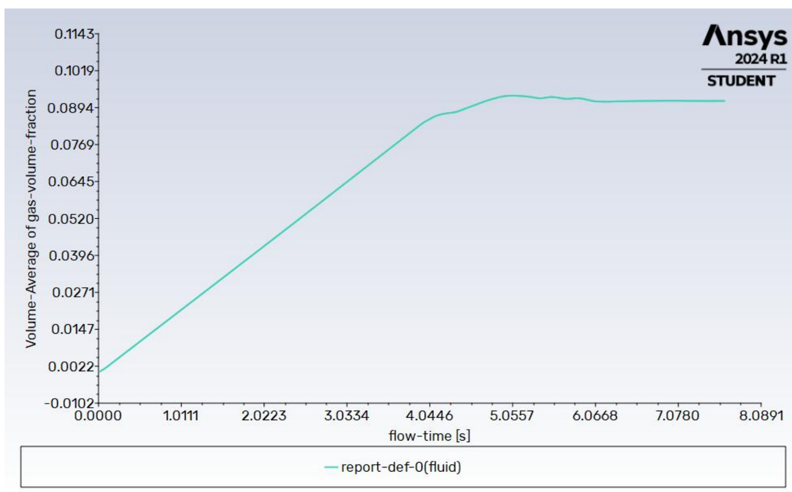
The column is initially filled with water, as in experiments, and null velocities are set. Click Initialize. Save the case and data files as

Run task page

In the run Calculation task page, enter 0.02 s for time step size (20 iterations for time step). Also, enter 500 for number of time steps.

2.6 Post-processing

Gas holdup. Gas holdup is the volume fraction of gas in the total volume of gas–liquid phase in the bubble column, which is one of the most important parameters to characterize the hydrodynamic characteristics of the bubble column. The plot from shows the initial transient startup behavior of the flow has begun to stabilize (approximately 4.7 %).

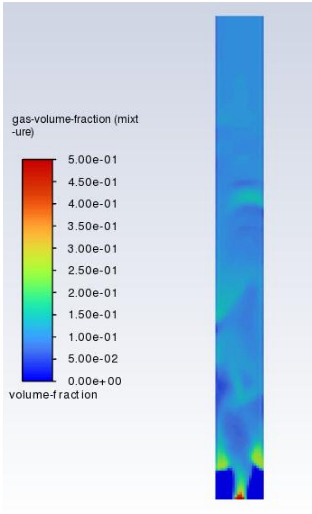


Interfacial area. It can be computed based on $a_i = 6\frac{\varepsilon_G}{d_{23}}\left[\frac{1}{mm}\right] \rightarrow \frac{6}{3mm}\varepsilon_G = 2\varepsilon_G\left[\frac{1}{mm}\right] = 0.094\left[\frac{1}{mm}\right]$. In practical applications, a large contact area between the phases ensures high interfacial heat and mass transfer and, eventually, high reaction rate.

Flow phenoemna. You can check the contours in the window of contours of gas holdup (volume fraction of air) as well as the gas holdup evolution over time.

Results tab → Animation group → Solution Playback

In the playback dialog box, select animation-1 from the list under animation sequences.



You can check the contours in the window of contours of gas holdup (volume fraction of air) as well as the gas holdup evolution over time. After a certain period, the oscillatory motion of the bubble plume is restricted to the lower part, while in the upper part the gas volume fraction distribution is more uniformly distributed. This has to be attributed to the developing flow field and acting bubble lift and turbulent dispersion forces. The transient nature of this flow with a lower oscillating region and a more uniform region in the upper part was nicely captured by the model.

Volume Fraction distribution

Create few lines in the domain and show the volume fraction. Do you obtain the typical volume fraction of polydispersed regimes with larger volume fraction in the center of the domain?

2.7 Case 2: superficial velocity 30 mm/s

Next increase the velocity to 30 mm/s and change the diameter of the large bubbles to 7 mm/s. As no experimental data are available assign 60% of the total flow rate to the small bubbles and 40% to the large bubbles.