Tutorial: Hydrodynamics of Bubble Column Reactors

Introduction

The purpose of this tutorial is to provide guidelines and recommendations for solving a gas-liquid bubble column problem using the multiphase mixture model, including advice on solver settings.

This tutorial demonstrates how to do the following:

- Set up a transient bubble column.
- Use the mixture multiphase model.
- Solve the problem using appropriate solver settings.
- Postprocess the resulting data.

Prerequisites

This tutorial is written with the assumption that you have completed Tutorial 1 from the ANSYS FLUENT 12.0 Tutorial Guide, and that you are familiar with the ANSYS FLUENT navigation pane and menu structure. Some steps in the setup and solution procedure will not be shown explicitly.

In this tutorial, you will use mixture multiphase model. This tutorial will not cover the mechanics of using this model. Instead, it will focus on the multiphase modeling of a gas liquid bubble column. For more information refer to Section 24.4 Setting Up the Mixture Model in the ANSYS FLUENT 12.0 User's Guide.

Problem Description

The representation of the problem is shown in Figure 1. The gas-liquid bubble flow is in a flat bubble column with rectangular cross-section and essentially two-dimensional flow pattern.

The main dimensions of the experimental apparatus are a length of 0.5 m and height of 1.5 m, which give a height-to-length ratio of 3. The gas is injected into the water column through a small inlet located 15 cm away from the left side wall. The inlet is 4 cm wide, and the air velocity at the inlet is very small, 6.6e-4 m/s. The mean diameter for the air bubbles is 3 mm.

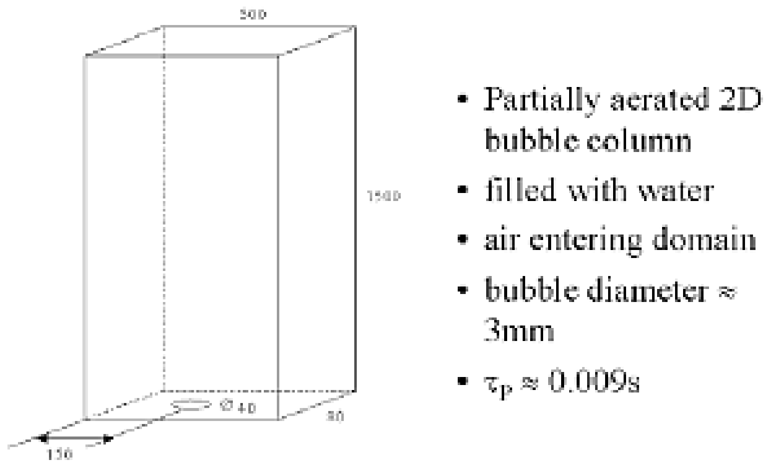


Figure 1: Schematic of the Gas-Liquid Bubble Column

Preparation

1. Copy the mesh file, `becker.msh` to your working folder.

2. Use FLUENT Launcher to start the 2D version of ANSYS FLUENT.

   For more information about FLUENT Launcher *see Section* 1.1.2 Starting ANSYS FLUENT Using FLUENT Launcher *in the* ANSYS FLUENT *12.0 User's Guide.*

   Note: *The* Display Options *are enabled by default. Therefore, after you read in the mesh, it will be displayed in the embedded graphics window.*

Setup and Solution

Step 1: Mesh

1. Read the mesh file (`becker.msh`).

   File ⟶ Read ⟶ Mesh...

   As the mesh is read, ANSYS FLUENT *will report the progress in the console. The mesh size will be reported as 1443 cells.*

Step 2: General

1. Check the mesh.

   General ⟶ Check

   ANSYS FLUENT *performs various checks on the mesh and reports the progress in the console. Make sure that the minimum volume reported is a positive number.*

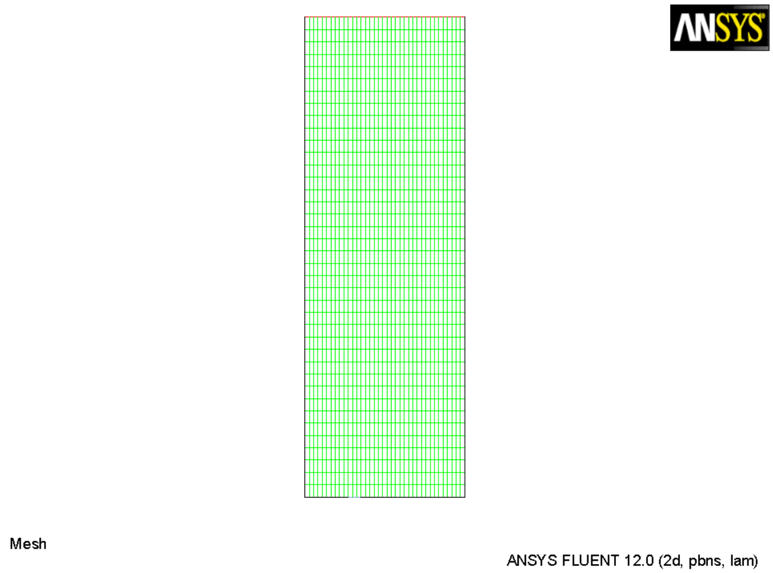


Figure 2: Mesh Display

2. Enable the transient solver by selecting Transient from the Time list.

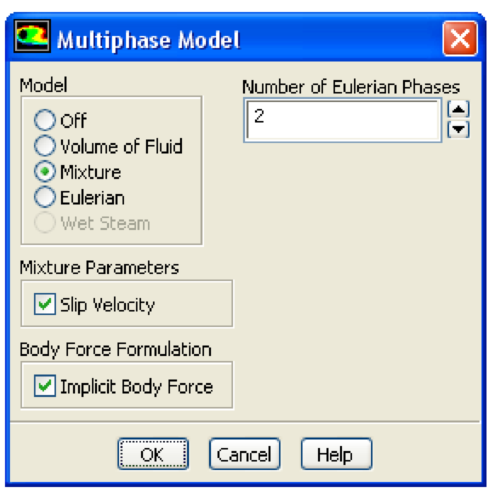
   General ⟶ Transient

Step 3: Models

The multiphase model is available only with the pressure based solver.

1. Select the mixture multiphase model.

   Models ⟶ Multiphase ⟶ Edit...



   (a) Select Mixture from the Model list.

(b) Enable Implicit Body Force.

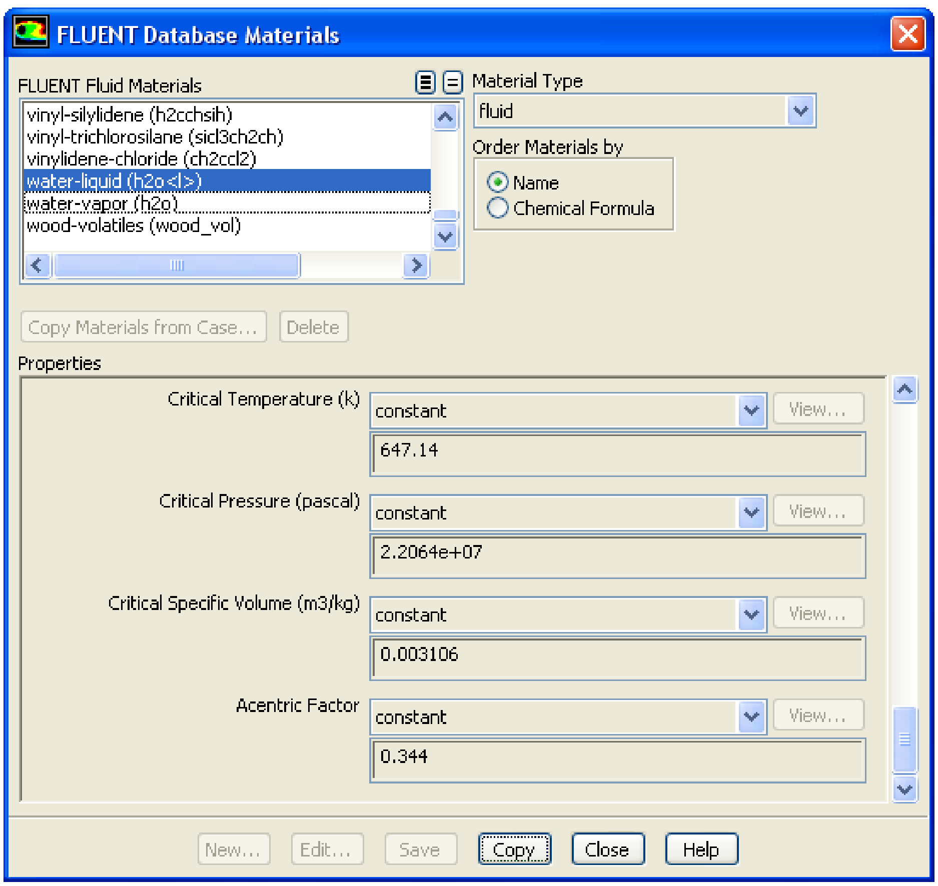
(c) Click OK to close the Multiphase Model dialog box.

Step 4: Materials

You will need to add the fluid water to the list of fluid materials by copying it from the ANSYS FLUENT *materials database.*

Materials ⟶ Create/Edit...

1. Click the FLUENT Database... button to open the FLUENT Database Materials dialog box.



(a) Ensure that fluid is selected from the Material Type drop-down list.

(b) Select water-liquid (h2o<l>) from the FLUENT Fluid Materials selection list.

(c) Click Copy and close the FLUENT Database Materials dialog box.

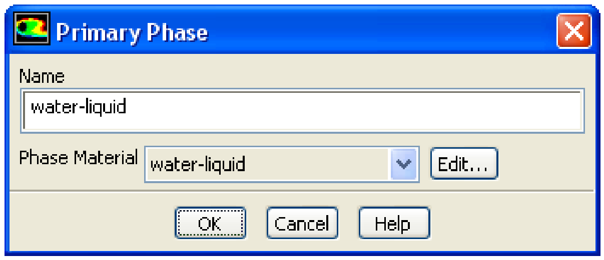
The properties will be downloaded from the database into your local list, and a copy of the properties will now be displayed in the Create/ Edit Materials *dialog box.*

2. Click Change/Create and close the Create/Edit Materials dialog box.

Step 5: Phases

1. Define the primary phases.

   Phases ⟶ phase-1 ⟶ Edit...



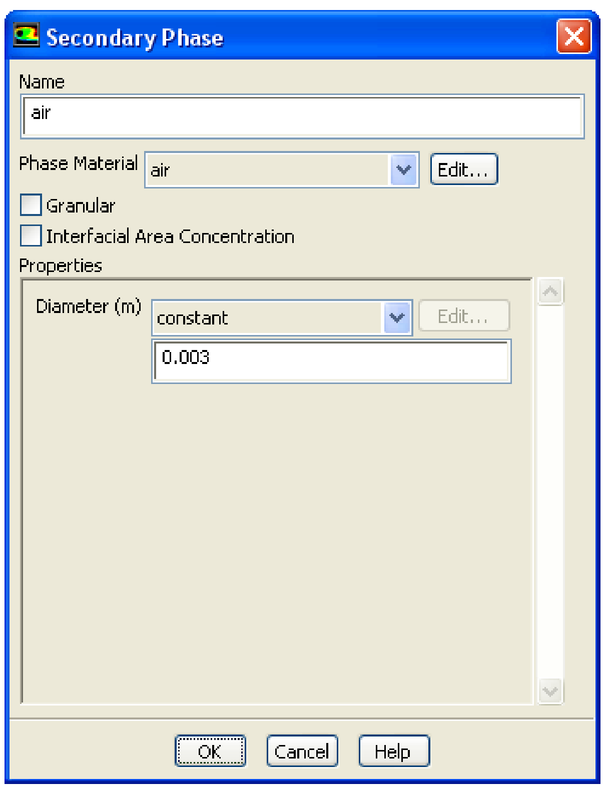
   (a) Enter `water-liquid` for Name.

   (b) Select water-liquid from the Phase Material drop-down list.

   (c) Click OK to close the Primary Phase dialog box.

2. Define the secondary phases.

   Phases ⟶ phase-2 ⟶ Edit...



   (a) Enter `air` for the Name.

   (b) Retain selection of air from the Phase Material drop-down list.

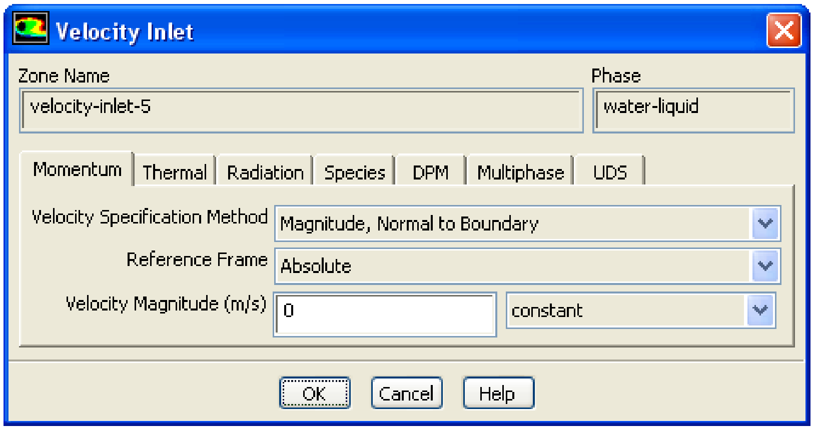
(c) Enter 0.003 for Diameter.

(d) Click OK to close the Secondary Phase dialog box.

Step 6: Boundary Conditions

1. Set the boundary conditions for velocity-inlet-5.

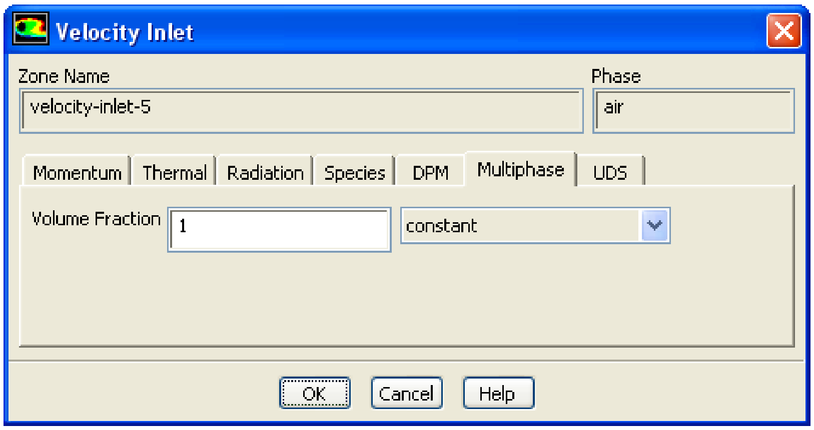
Boundary Conditions ⟶ velocity-inlet-5



(a) Select water-liquid from the Phase drop-down list and click Edit... to open the Velocity Inlet dialog box.

i. Retain the default settings and click OK to close the Velocity Inlet dialog box.

(b) Select air from the Phase drop-down list and click Edit... to open the Velocity Inlet dialog box.



i. Enter 0.66e-3 m/s for Velocity Magnitude.

ii. Click the Multiphase tab and enter 1 for Volume Fraction.

iii. Click OK to close the Velocity Inlet dialog box.

2. Set the boundary conditions for pressure-outlet-4.

   Boundary Conditions ⟶ pressure-outlet-4

   (a) Select mixture from the Phase drop-down list and click Edit... to open the Pressure Outlet dialog box.

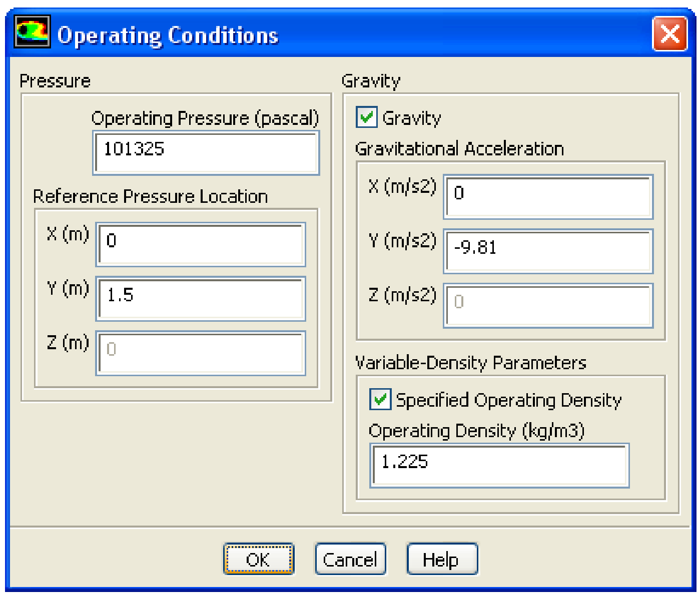
      i. Retain the default settings and click OK to close the Pressure Outlet dialog box.

   (b) Select air from the Phase drop-down list and click Edit... to open the Pressure Outlet dialog box.

      i. Retain the default settings for air and click OK to close the Pressure Outlet dialog box.

3. Set the operating conditions

   Boundary Conditions ⟶ Operating Conditions...



   (a) Set the Reference Pressure Location in the Y direction to `1.5 m`.

   (b) Enable Gravity.

   (c) Enter `-9.81 m/s`^{2} for Gravitational Acceleration in the Y direction.

   (d) Set the operating density.

      i. Enable Specified Operating Density.

      ii. Retain 1.225 kg/m^3 for Operating Density.

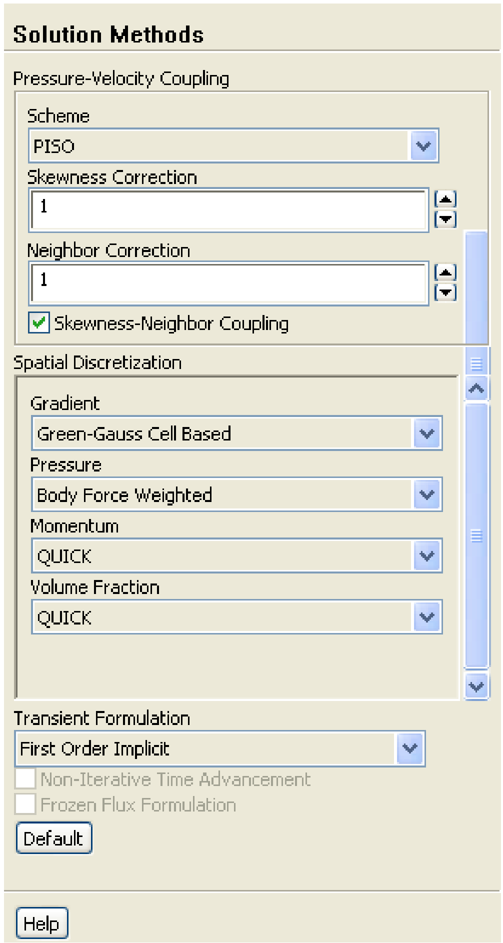
      The operating density is set to the density of the lighter phase. This excludes the buildup of hydrostatic pressure within the lighter phase, improving the round-off accuracy for the momentum balance.

   (e) Click OK to close the Operating Conditions dialog box.

Step 7: Solution

1. Set the solution control parameters.

   Solution Methods



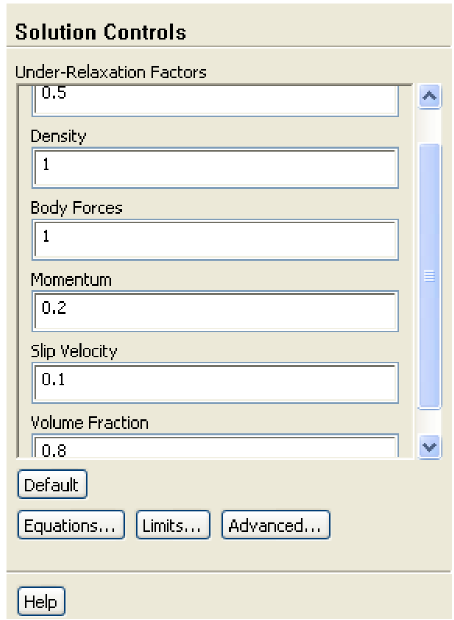
   (a) Select PISO from the Scheme drop-down list in the Pressure-Velocity Coupling group box.

   (b) Select Body Force Weighted from the Pressure drop-down list.

   (c) Select QUICK from the Momentum and Volume Fraction drop-down lists.

2. Set the Under-Relaxation Factors.

   Solution Controls



   (a) Enter 0.5 for Pressure.

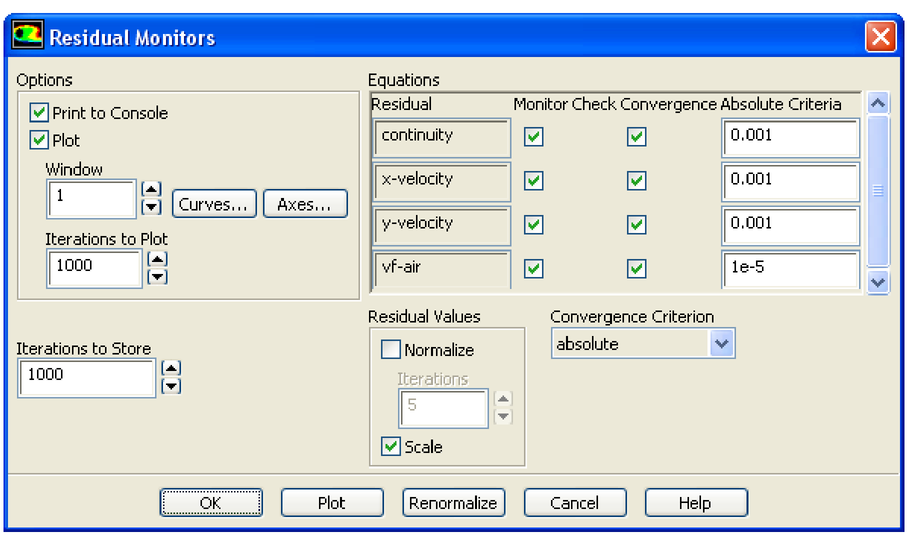
   (b) Enter 0.2 for Momentum.

   (c) Enter 0.8 for Volume Fraction.

   Note: *Since this is a laminar flow, a turbulence model has not been activated. If the flow is turbulent, select the appropriate k-epsilon turbulence model and enter* 0.8 *for under-relaxation parameters for* Turbulent Kinetic Energy *and* Turbulent Dissipation Rate.

3. Enable the plotting of residuals during the calculation.

   Monitors ⟶ Residuals ⟶ Edit...



(a) Ensure that Plot is enabled from the Options list.

(b) Set the Convergence Criterion for vf-air to `1e-5`.

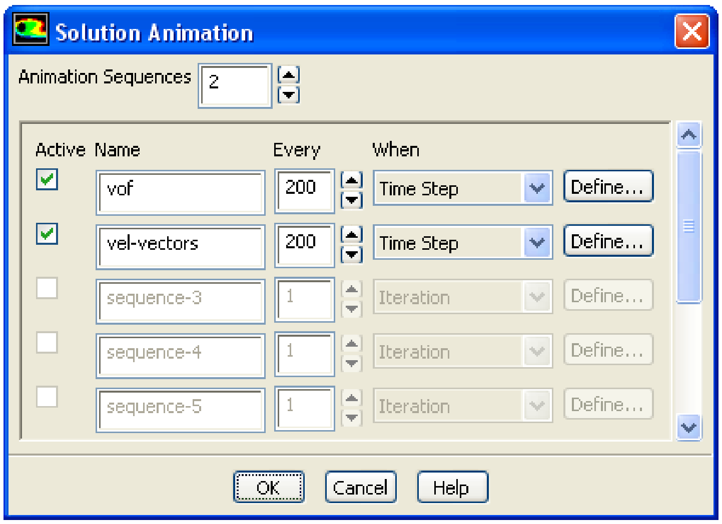
(c) Click OK to close the Residual Monitors dialog box.

4. Initialize the flow field.

   Solution Initialization ⟶ Initialize

5. Create an animation sequence for the velocity vector and volume fraction display.

   Calculation Activities (Solution Animations) ⟶ Create/Edit



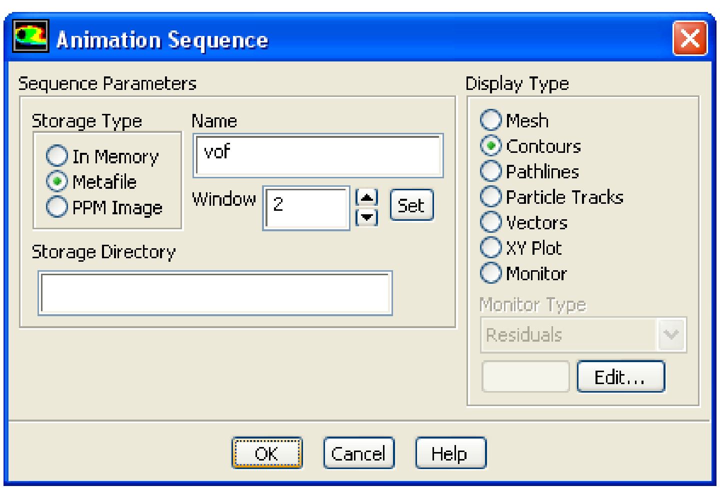
(a) Set the number of Animation Sequences to 2.

(b) Enter `vof` for the first animation and `vel-vectors` for the second animation Name.

(c) Enter 200 for Every for `vof` and `vel-vectors`.

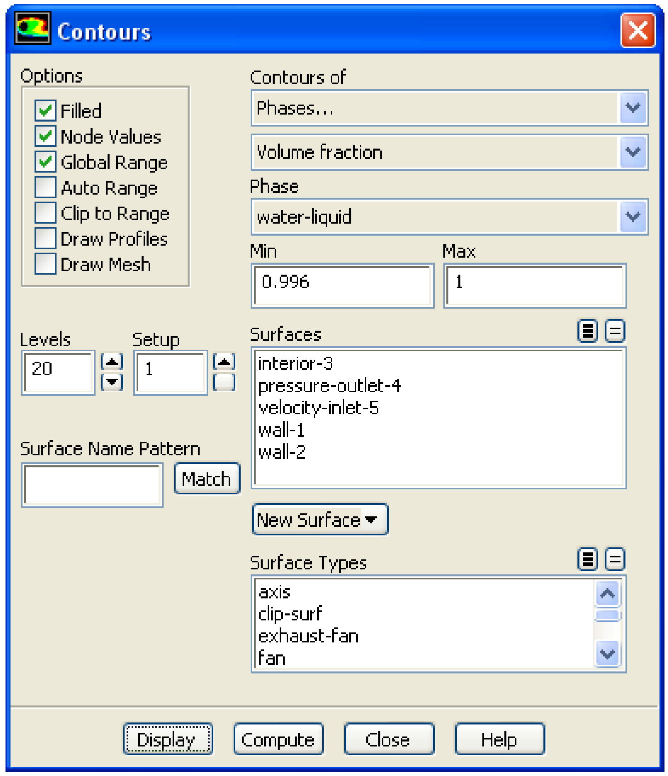
(d) Select Time Step from the When drop-down list for `vof` and `vel-vectors`.

(e) Click the Define... button for `vof` to open the Animation Sequence dialog box.



i. Set Window number to 2 and click Set.

ii. Select Contours from the Display Type list to open the Contours dialog box.



A. Select Phases... and Volume fraction from the Contours of drop-down lists.

B. Enable Filled, disable Auto Range and Clip to Range from the Options list.

C. Ensure that water-liquid is selected from the Phase drop-down list.

D. Set the range from `0.996` for Min to `1` for Max.

E. Click Display and close the Contours dialog box.

iii. Click OK to close the Animation Sequence dialog box.

(f) Click the Define... button for `vel-vectors` to open the Animation Sequence dialog box.

i. Set Window number to 3 and click the Set button.

ii. Select Vectors from the Display Type list to open the Vectors dialog box.

A. Ensure that Auto Range and Clip to Range are disabled from the Options list.

B. Ensure that Velocity from the Vectors of drop-down list and mixture from the Phase drop-down list are selected.

C. Ensure that Velocity... and Velocity Magnitude are selected from the Color by drop-down list.

D. Set the range from `2.26e-4` to `1.32e-1`.

E. Click Display and close the Vectors dialog box.

iii. Click OK to close the Animation Sequence dialog box.

The active checkbox for `vof` *and* `vel-vectors` *will be enabled in the* Solution Animation *dialog box.*

iv. Click OK to close the Solution Animation dialog box.

6. Autosave the data file.

   Calculation Activities

   (a) Enter 200 for Autosave Every (Time Steps).

7. Save the case file (`becker.cas.gz`).

   File ⟶ Write ⟶ Case...

8. Set the time stepping parameters.

   Run Calculation

   (a) Set Time Step Size to `0.01`.

   (b) Set the Number of Time Steps to 5000.

   (c) Click Calculate.

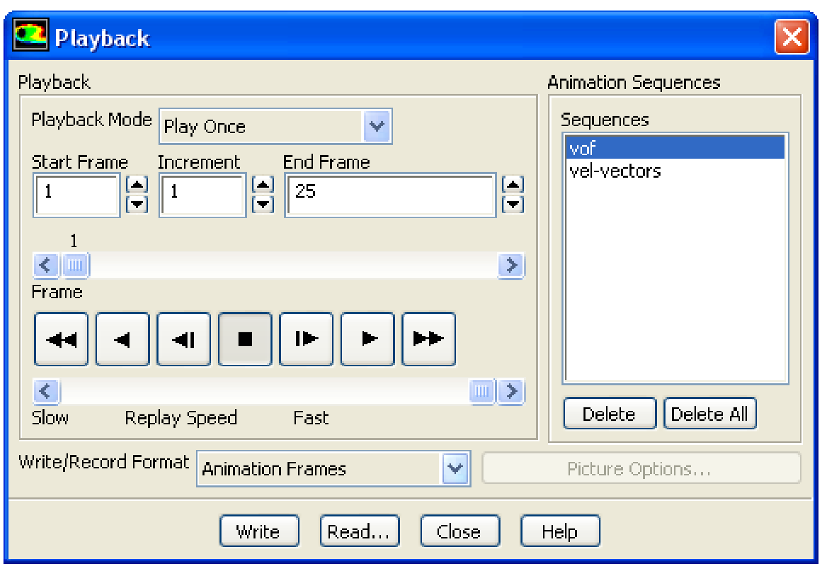
9. Save the data file (`becker.dat.gz`).

   File ⟶ Write ⟶ Data...

Step 8: Postprocessing

1. Play back the animations of the volume fraction and velocity vectors.

   Graphics and Animations ⟶ Solution Animation Playback ⟶ Set Up...



(a) Select vof from the Sequences selection list and click the ▶ button.

(b) Select vel-vectors from the Sequences selection list and click the ▶ button.

(c) Close the Playaback dialog box.

2. Display filled contours of volume fraction of water (Figure 3).

   Graphics and Animations ⟶ Contours ⟶ Set Up...

   (a) Ensure Filled is enabled.

   (b) Ensure that Auto Range and Clip to Range are disabled from the Options list.

   (c) Retain selection of Phases... and Volume fraction from the Contours of drop-down list.

   (d) Ensure that water-liquid is selected from the Phase drop down list.

   (e) Retain `0.996` for Min.

   (f) Retain `1` for Max.

   (g) Click Display.

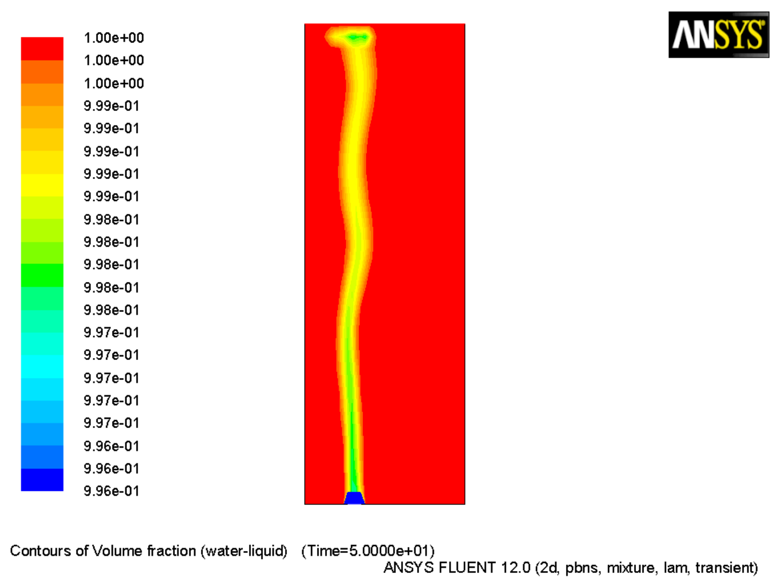


Figure 3: Contours of Volume Fraction of Water

   (h) Close the Contours dialog box.

3. Display velocity vectors (Figure 4).

   Graphics and Animations ⟶ Vectors ⟶ Set Up...

   (a) Ensure that Auto Range and Clip to Range are disabled from the Options list.

   (b) Retain selection of Velocity from the Vectors of drop-down list and mixture from the Phases drop-down list.

(c) Retain selection of Velocity... and Velocity Magnitude from the Color by drop-down list.

(d) Set the range from `2.26e-4` for Min to `1.32e-1` for Max respectively.

(e) Click Display.

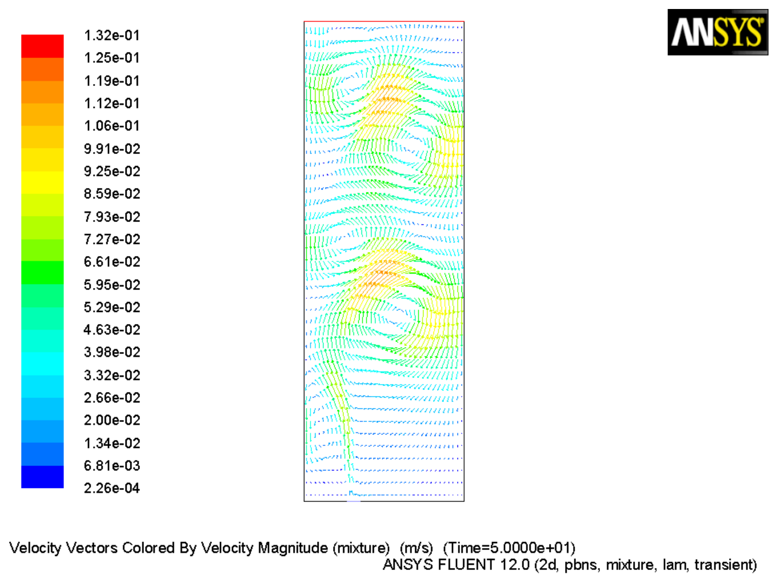


Figure 4: Velocity Vectors for the Bubble Column

(f) Close the Vectors dialog box.

Summary

In this tutorial, a transient bubble column was set up and solved using the Multiphase-Mixture model in ANSYS FLUENT.

For more information about the applicability of the different multiphase models, refer to the ANSYS FLUENT 12.0 User's Guide.