Tutorial 19. Using the Mixture and Eulerian Multiphase Models

Introduction

This tutorial examines the flow of water and air in a tee junction. Initially you will solve the problem using the less computationally intensive mixture model, and then turn to the more accurate Eulerian model. The results of these two approaches can then be compared.

This tutorial will demonstrate how to do the following:

- Use the mixture model with slip velocities.
- Set boundary conditions for internal flow.
- Calculate a solution using the pressure-based solver.
- Use the Eulerian model.
- Display the results obtained using the two approaches for comparison.

Prerequisites

This tutorial assumes that you are familiar with the menu structure in FLUENT and that you have completed Tutorial 1. Some steps in the setup and solution procedure will not be shown explicitly.

Problem Description

This problem considers an air-water mixture flowing upwards in a duct and then splitting in a tee-junction. The ducts are 25 mm in width, the inlet section of the duct is 125 mm long, and the top and the side ducts are 250 mm long. The schematic of the problem is shown in Figure 19.1.

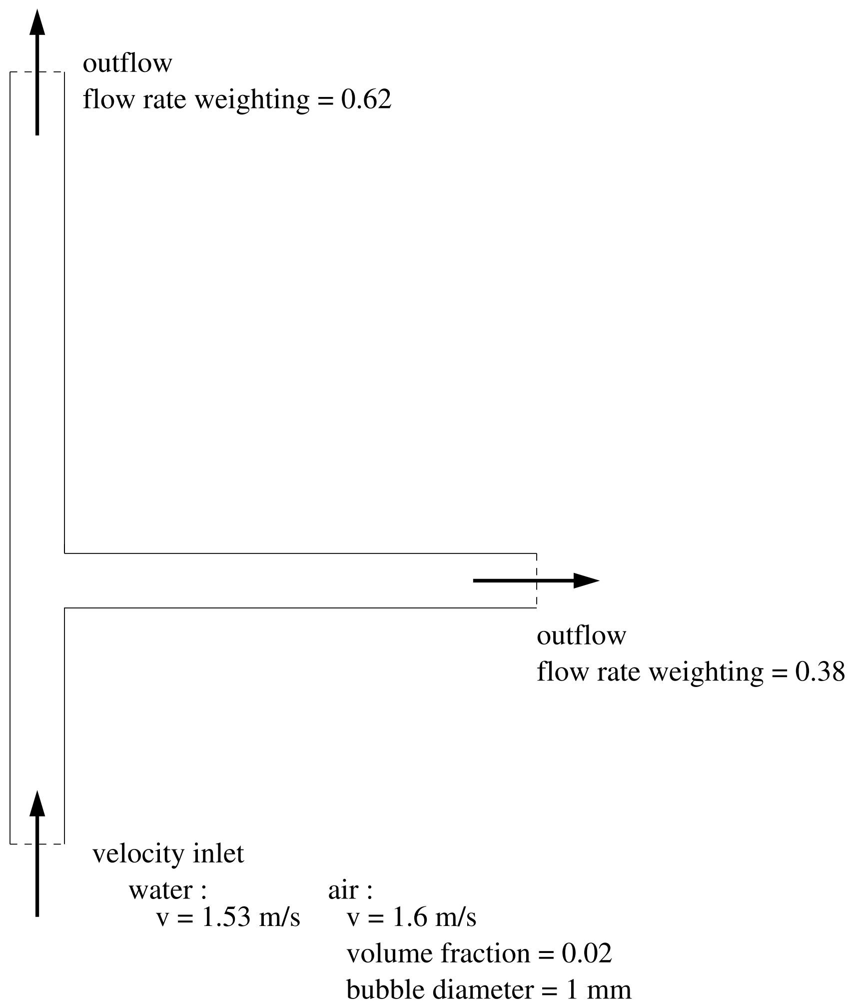


Figure 19.1: Problem Specification

Setup and Solution

Preparation

1. Download `mix_eulerian_multiphase.zip` from the Fluent Inc. User Services Center or copy it from the FLUENT documentation CD to your working folder (as described in Tutorial 1).

2. Unzip `mix_eulerian_multiphase.zip`.

   The file `tee.msh` *can be found in the* `mix_eulerian_multiphase` *folder created after unzipping the file.*

3. Start the 2D (2d) version of FLUENT.

© Fluent Inc. September 21, 2006

Step 1: Grid

1. Read the grid file `tee.msh`.

   File ⟶ Read ⟶ Case...

   As FLUENT *reads the grid file, it will report its progress in the console.*

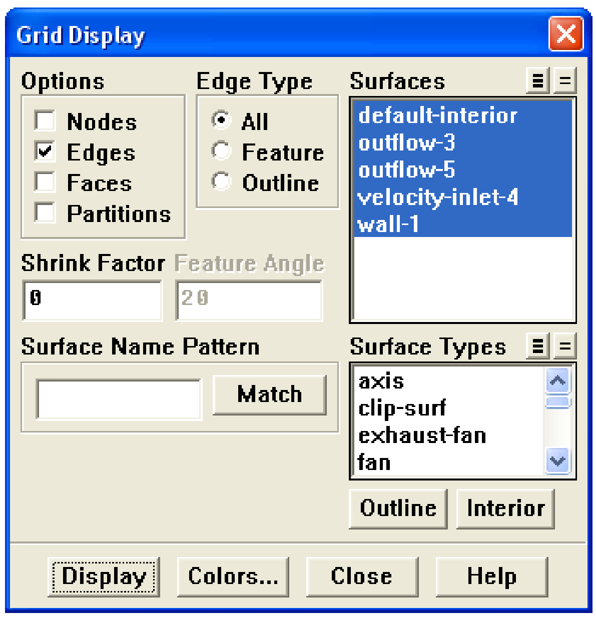
2. Check the grid.

   Grid ⟶ Check

   FLUENT *will perform various checks on the mesh and will report the progress in the console. Pay particular attention to the reported minimum volume and ensure it is a positive number.*

3. Display the grid (Figure 19.2).

   Display ⟶ Grid...



   (a) Retain the default settings.

   (b) Click Display and close the Grid Display panel.

   Extra: *You can use the right mouse button to probe for grid information in the graphics window. If you click the right mouse button on any node in the grid, information will be displayed in the* FLUENT *console about the associated zone, including the name of the zone. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.*

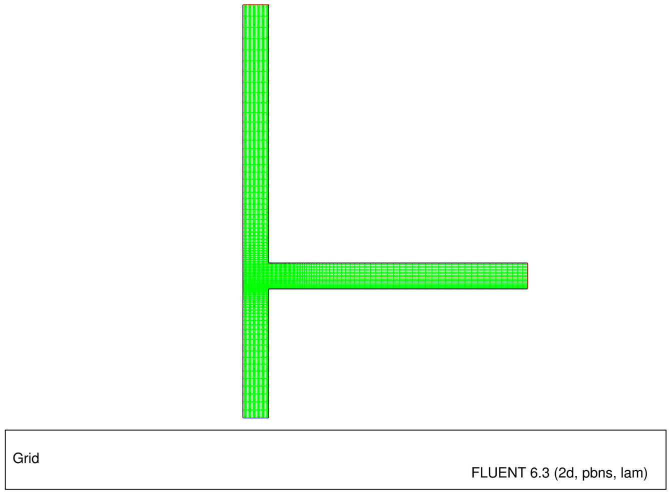


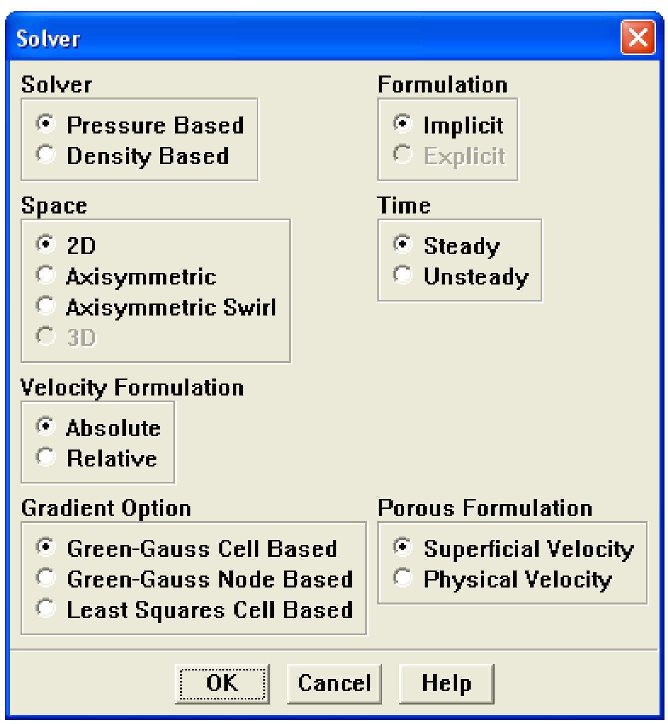
Figure 19.2: Grid Display

Step 2: Models

1. Retain the default settings for the pressure-based solver.

   The pressure-based solver must be used for multiphase calculations.

   Define ⟶ Models ⟶Solver...



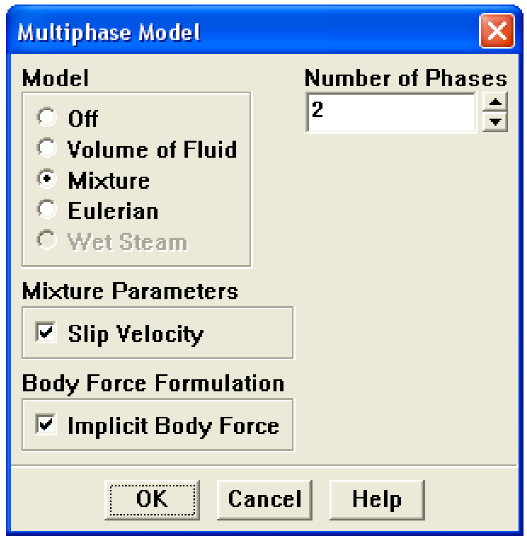
© Fluent Inc. September 21, 2006

2. Select the mixture multiphase model with slip velocities.

   Define ⟶ Models ⟶ Multiphase...

   (a) Select Mixture from the Model list.

   The Multiphase Model *panel will expand to show the inputs for the mixture model.*



   (b) Make sure that the Slip Velocity option is enabled in the Mixture Parameters group box.

   You need to solve the slip velocity equation since there will be significant difference in velocities for the different phases.

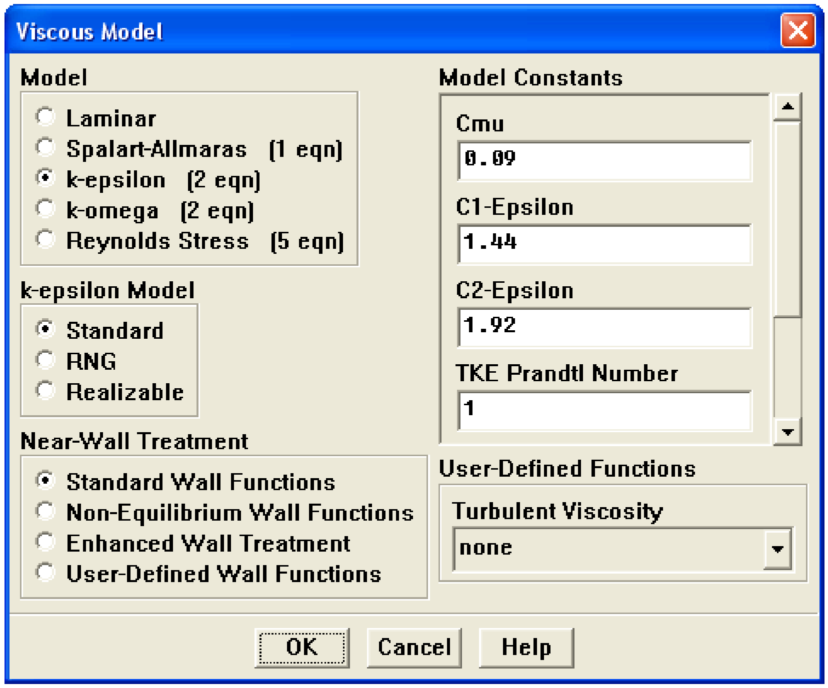
   (c) Enable the Implicit Body Force option in the Body Force Formulation group box.

   This treatment improves solution convergence by accounting for the partial equilibrium of the pressure gradient and body forces in the momentum equations. It is used in VOF and mixture problems, where body forces are large in comparison to viscous and connective forces.

   (d) Click OK to close the Multiphase Model panel.

3. Select the standard k-ϵ turbulence model with standard wall functions.

   Define ⟶ Models ⟶ Viscous...



   (a) Select k-epsilon from the Model list.

   (b) Retain the default selection of Standard from the k-epsilon Model list.

   The standard k-ϵ model is quite effective in accurately resolving mixture problems when standard wall functions are used.

   (c) Retain the default selection of Standard Wall Functions from the Near-Wall Treatment list.

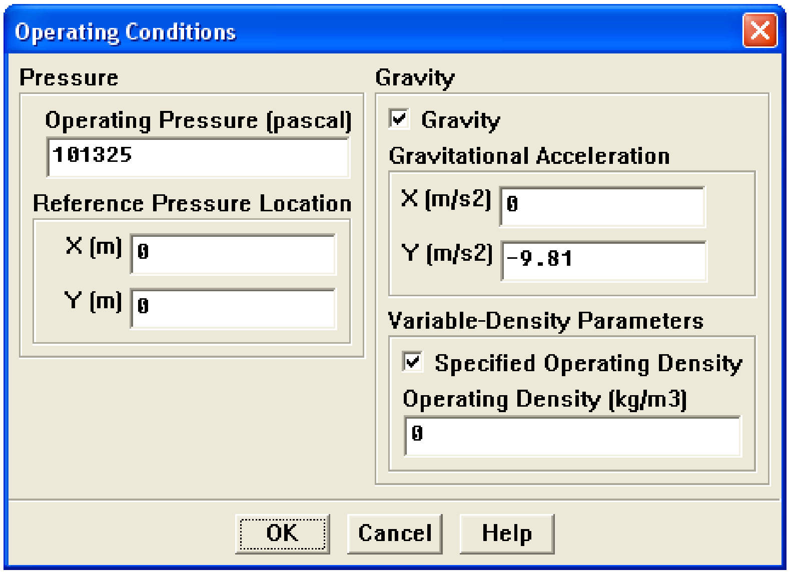
   This problem does not require a particularly fine grid, and standard wall functions will be used.

   (d) Click OK to close the Viscous Model panel.

© Fluent Inc. September 21, 2006

4. Set the gravitational acceleration.

   Define ⟶ Operating Conditions...



   (a) Enable the Gravity option.

   The Operating Conditions *panel will expand to show additional inputs.*

   (b) Enter `-9.81` m/s^2 for Y in the Gravitational Acceleration group box.

   (c) Enable the Specified Operating Density option.

   (d) Enter `0` kg/m^3 for Operating Density.

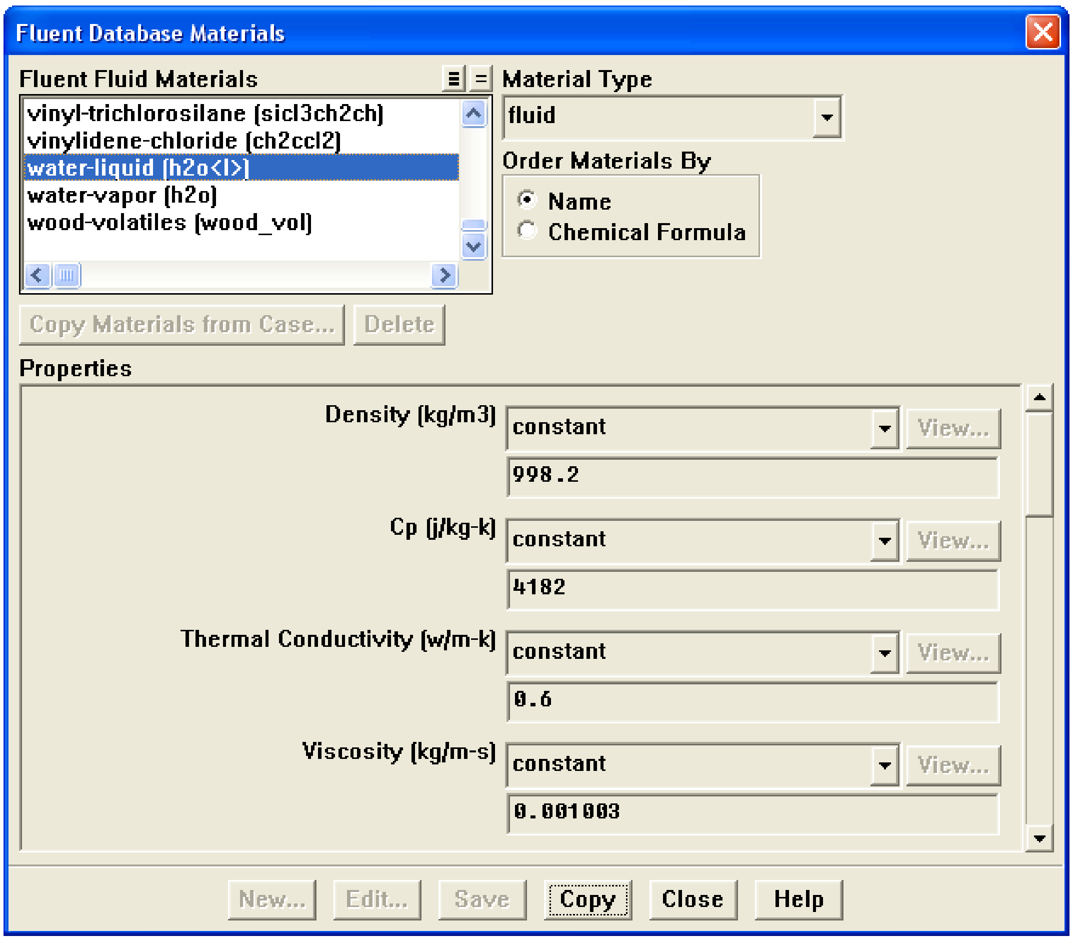
   (e) Click OK to close the Operating Conditions panel.

Step 3: Materials

1. Copy the properties for liquid water from the materials database so that it can be used for the primary phase.

   Define ⟶Materials...

   (a) Click the Fluent Database... button to open the Fluent Database Materials panel.



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

      Scroll down the list to find water-liquid (h2o<l>).

   ii. Click Copy to copy the properties for liquid water to your model.

   iii. Close the Fluent Database Materials panel.

   (b) Close the Materials panel.

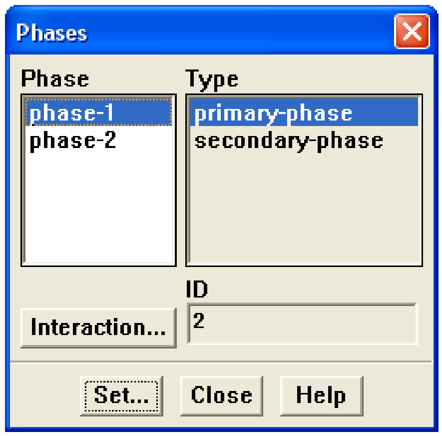
© Fluent Inc. September 21, 2006

Step 4: Phases

In the following steps you will define the liquid water and air phases that flow in the tee junction.

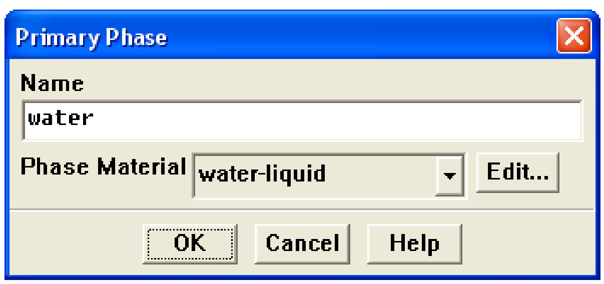
1. Specify liquid water as the primary phase.

   Define ⟶Phases...

   (a) Select phase-1 from the Phase selection list.

   (b) Click Set... to open the Primary Phase panel.

   i. Enter `water` for Name.

   ii. Select water-liquid from the Phase Material drop-down list.

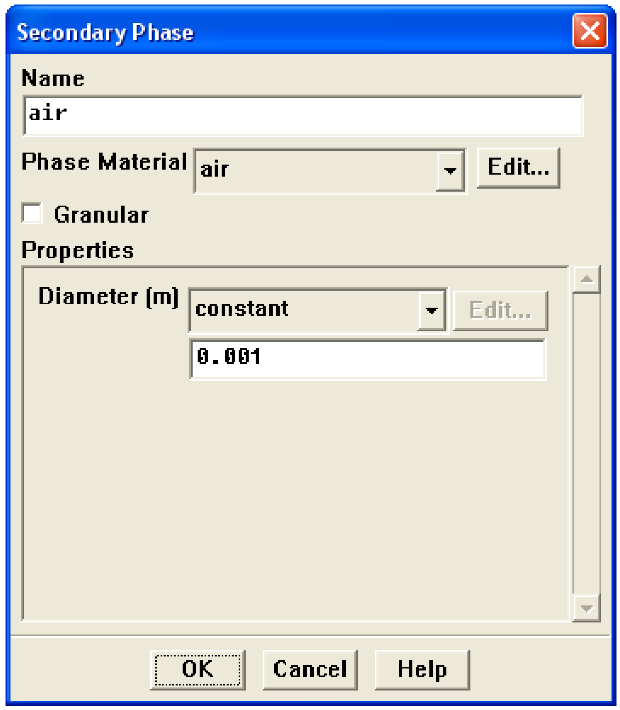
   iii. Click OK to close the Primary Phase panel.

2. Specify air as the secondary phase.

   Define ⟶Phases...

   (a) Select phase-2 from the Phases selection list.

   (b) Click Set... to open the Secondary Phase panel.



   i. Enter `air` for Name.

   ii. Retain the default selection of air from the Phase Material drop-down list.

   iii. Enter `0.001` m for Diameter.

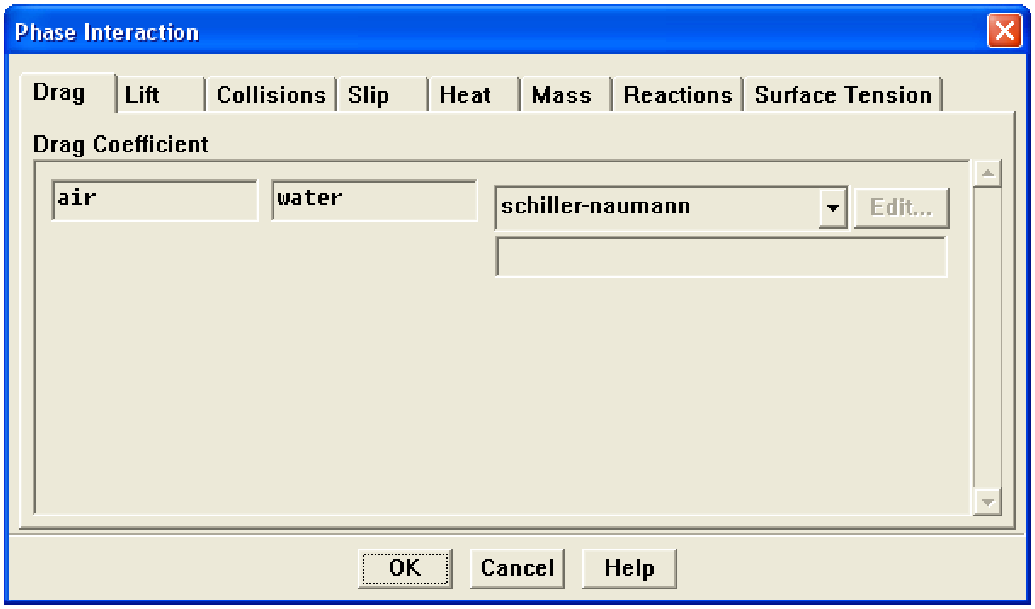
   iv. Click OK to close the Secondary Phase panel.

© Fluent Inc. September 21, 2006

3. Check that the drag coefficient is set to be calculated using the Schiller-Naumann drag law.

   Define ⟶Phases...

   (a) Click the Interaction... button to open the Phase Interaction panel.



   i. Retain the default selection of schiller-naumann from the Drag Coefficient drop-down list.

      The Schiller-Naumann drag law describes the drag between the spherical particle and the surrounding liquid for a wide range of conditions. In this case, the bubbles have an approximately spherical shape with a diameter of 1 mm.

   ii. Click OK to close the Phase Interaction panel.

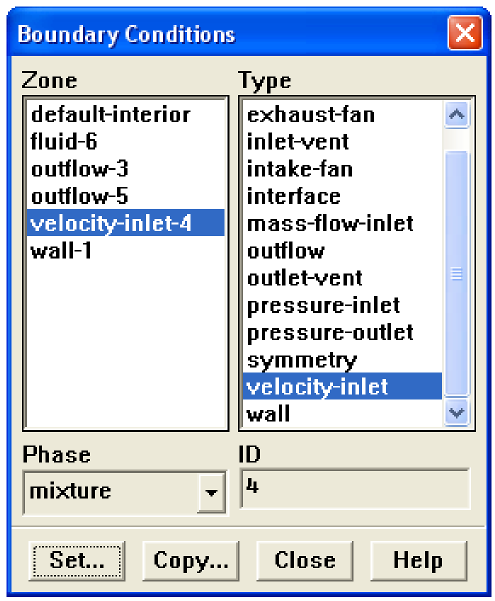
   (b) Close the Phases panel.

Step 5: Boundary Conditions

For this problem, you need to set the boundary conditions for three boundaries: the velocity inlet and the two outflows. Since this is a mixture multiphase model, you will set the conditions at the velocity inlet that are specific for the mixture (i.e., conditions that apply to all phases) and also conditions that are specific to the primary and secondary phases.

1. Set the boundary conditions at the velocity inlet (velocity-inlet-4) for the mixture.

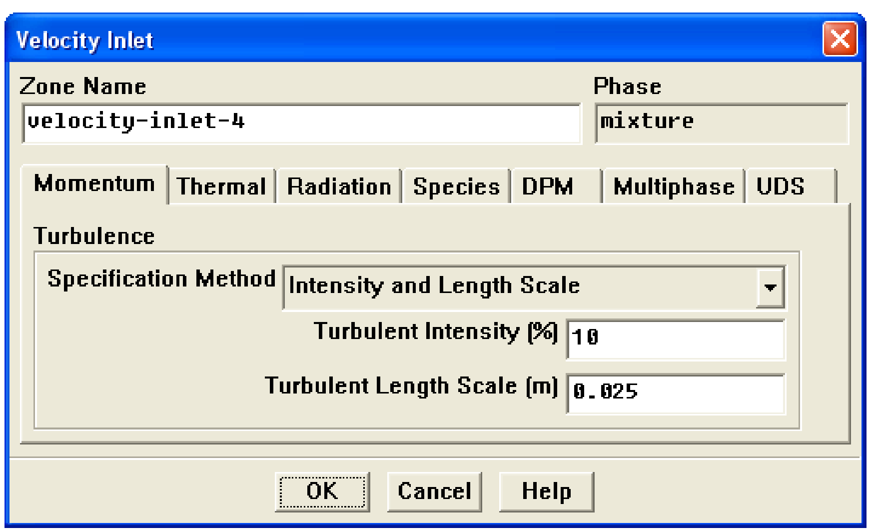
   Define →Boundary Conditions...



   (a) Select velocity-inlet-4 from the Zone selection list.

   (b) Retain the default selection of mixture in the Phase drop-down list.

   (c) Click Set... to open the Velocity Inlet panel.



 © Fluent Inc. September 21, 2006

i. Select Intensity and Length Scale from the Specification Method drop-down list.

ii. Retain the default value of 10% for Turbulent Intensity.

iii. Enter 0.025 m for Turbulent Length Scale.

iv. Click OK to close the Velocity Inlet panel.

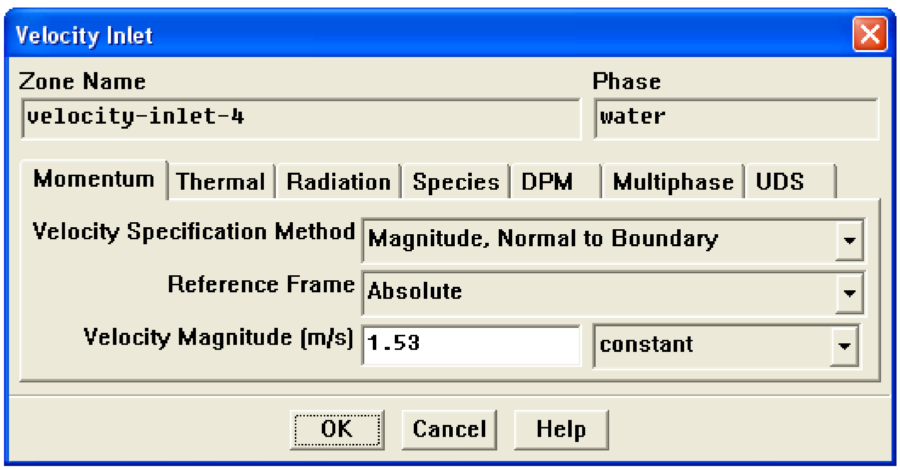
2. Set the boundary conditions at the velocity inlet (velocity-inlet-4) for the primary phase (water).

   Define ⟶Boundary Conditions...

   (a) Make sure that velocity-inlet-4 is selected from the Zone selection list.

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

   (c) Click Set... to open the Velocity Inlet panel.



i. Retain the default selection of Magnitude, Normal to Boundary from the Velocity Specification Method drop-down list.

ii. Retain the default selection of Absolute from the Reference Frame drop-down list.

iii. Enter 1.53 m/s for Velocity Magnitude.

iv. Click OK to close the Velocity Inlet panel.

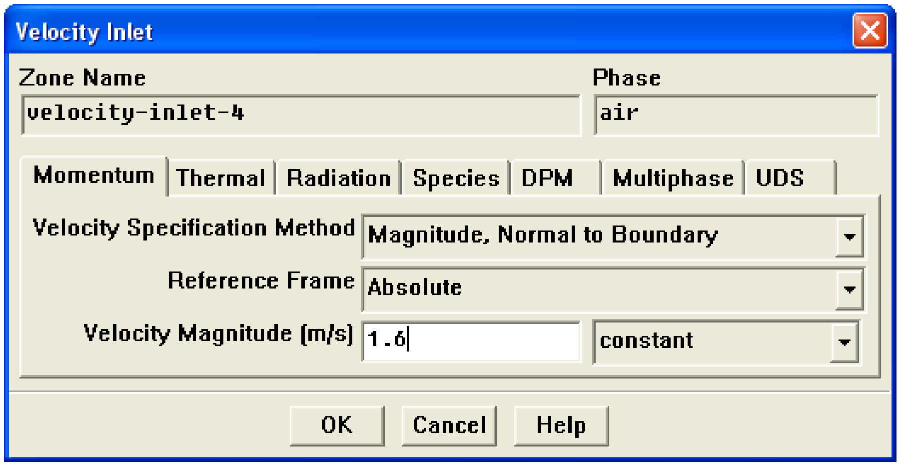
3. Set the boundary conditions at the velocity inlet (velocity-inlet-4) for the secondary phase (air).

   Define ⟶Boundary Conditions...

   (a) Make sure that velocity-inlet-4 is selected from the Zone selection list.

   (b) Select air from the Phase drop-down list.

(c) Click Set... to open the Velocity Inlet panel.



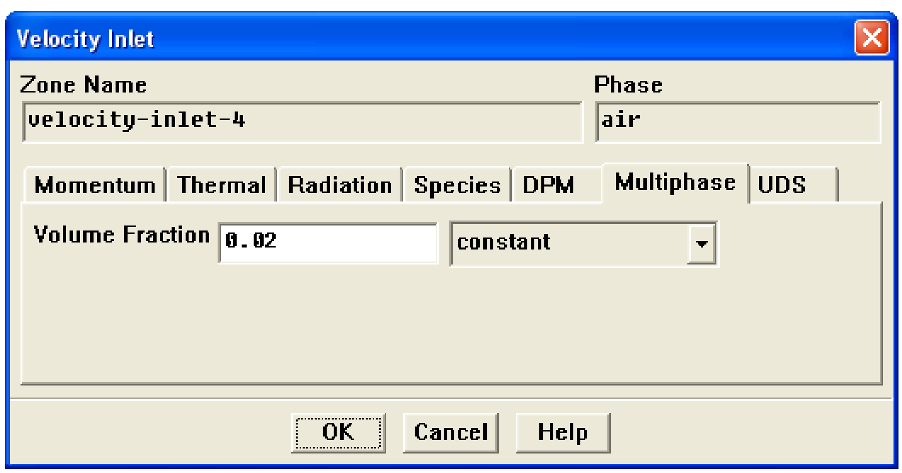
i. Retain the default selection of Magnitude, Normal to Boundary from the Velocity Specification Method drop-down list.

ii. Retain the default selection of Absolute from the Reference Frame drop-down list.

iii. Enter `1.6` m/s for Velocity Magnitude.

In multiphase flows, the volume rate of each phase is usually known. Volume rate divided by the inlet area gives the superficial velocity, which is the product of the inlet physical velocity and the volume fraction. When you have two phases, you must enter two physical velocities and the volume fraction of the secondary phase. Here it is assumed that bubbles at the inlet are moving with faster physical speed and their relative velocity with respect to water is $1.6 - 1.53 = 0.07$ m/s.

iv. Click the Multiphase tab and enter `0.02` for Volume Fraction.



v. Click OK to close the Velocity Inlet panel.

© Fluent Inc. September 21, 2006

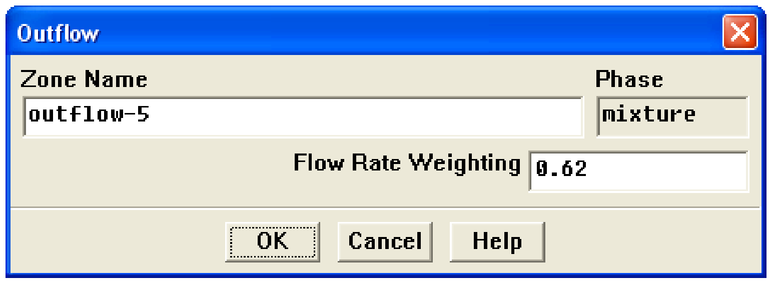
4. Set the boundary conditions at outflow-5 for the mixture.

   Define ⟶Boundary Conditions...

   (a) Select outflow-5 from the Zone selection list.

   (b) Select mixture from the Phase drop-down list.

   (c) Click Set... to open the Outflow panel.



      i. Enter 0.62 for Flow Rate Weighting.

      ii. Click OK to close the Outflow panel.

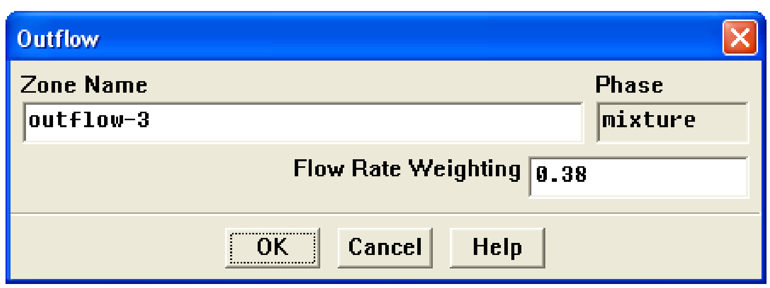
5. Set the boundary conditions at outflow-3 for the mixture.

   Define ⟶Boundary Conditions...

   (a) Select outflow-3 in the Zone selection list.

   (b) Make sure that mixture is selected in the Phase drop-down list.

   (c) Click Set... to open the Outflow panel.



      i. Enter 0.38 for Flow Rate Weighting.

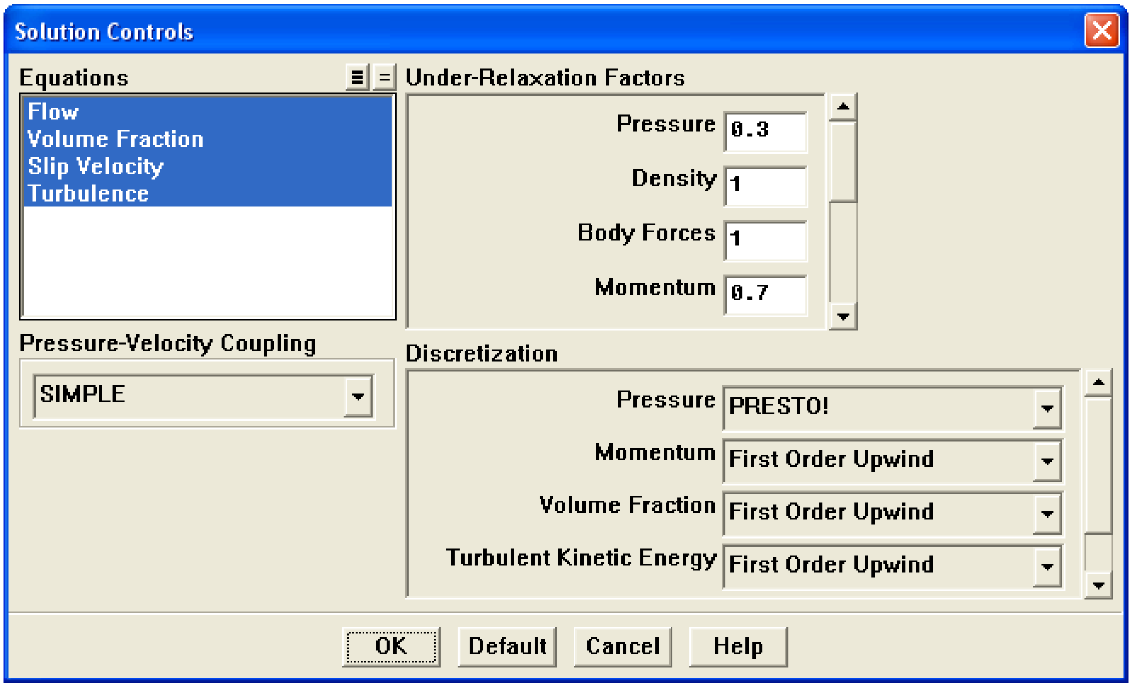
      ii. Click OK to close the Outflow panel.

   (d) Close the Boundary Conditions panel.

Step 6: Solution Using the Mixture Model

1. Set the solution parameters.

   Solve ⟶ Controls ⟶ Solution...



   (a) Retain the default values in the Under-Relaxation Factors group box.

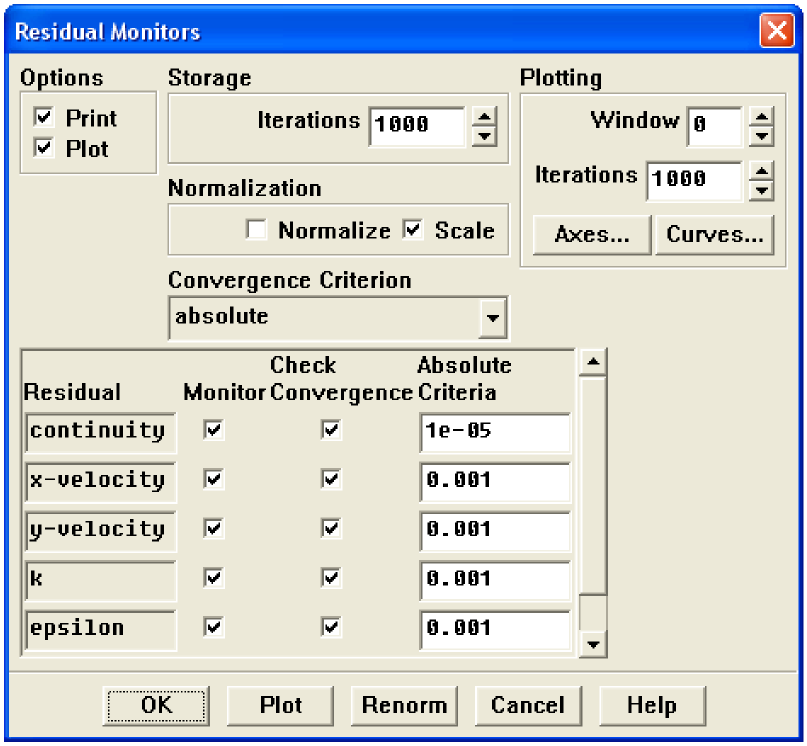
   (b) Select PRESTO! from the Pressure drop-down list in the Discretization group box.

   (c) Click OK to close the Solution Controls panel.

© Fluent Inc. September 21, 2006

2. Enable the plotting of residuals during the calculation.

   Solve ⟶ Monitors ⟶ Residual...



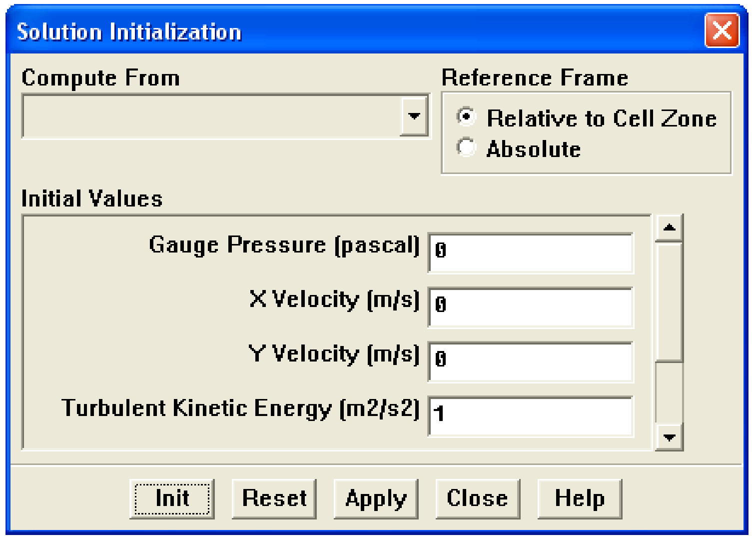
   (a) Enable Plot in the Options group box.

   (b) Enter `1e-05` for the Absolute Criteria of continuity, as shown in the previous panel.

   (c) Click OK to close the Residual Monitors panel.

3. Initialize the solution.

   Solve ⟶ Initialize ⟶ Initialize...



   (a) Retain the default settings.

   (b) Click Init and close the Solution Initialization panel.

4. Save the case file (`tee.cas.gz`).

   File ⟶ Write ⟶ Case...

5. Start the calculation by requesting 1200 iterations.

   Solve ⟶ Iterate...

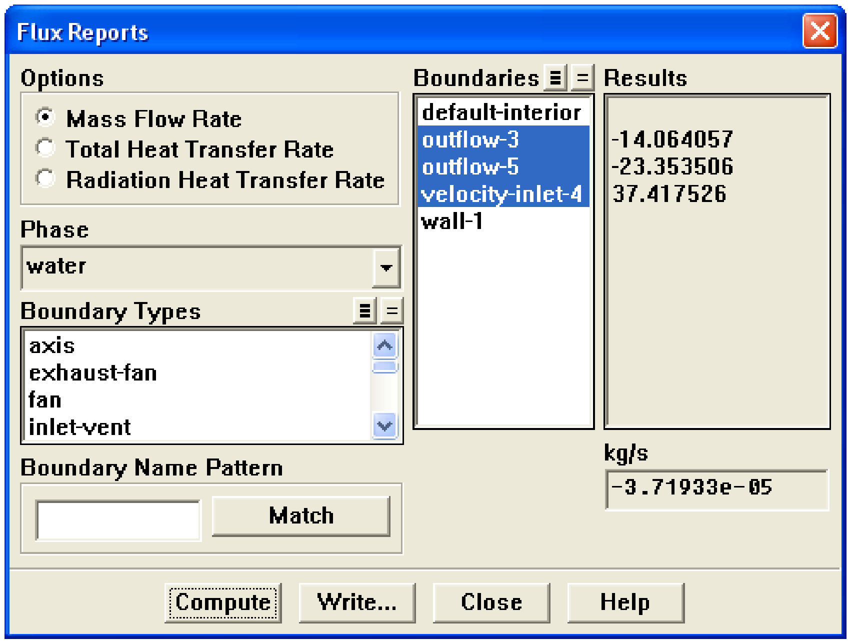
6. Save the case and data files (`tee.cas.gz` and `tee.dat.gz`).

   File ⟶ Write ⟶ Case & Data...

© Fluent Inc. September 21, 2006

7. Check the total mass flow rate for each phase.

   Report ⟶Fluxes...



   (a) Retain the default selection of Mass Flow Rate from the Options list.

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

   (c) Select outflow-3, outflow-5 and velocity-inlet-4 from the Boundaries selection list.

   (d) Click Compute.

   Note that the net mass flow rate is almost zero, indicating that total mass is conserved.

   (e) Select air from the Phase drop-down list and click Compute again.

   Note that the net mass flow rate is almost zero, indicating that total mass is conserved.

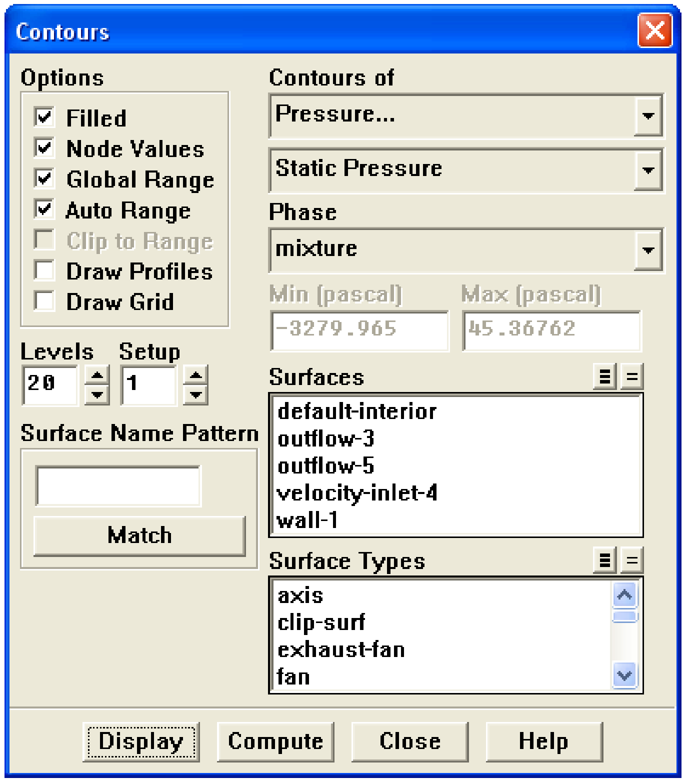
   (f) Close the Flux Reports panel.

© Fluent Inc. September 21, 2006

Step 7: Postprocessing for the Mixture Solution

1. Display the static pressure field in the tee (Figure 19.3).

   Display ⟶Contours...



   (a) Enable Filled in the Options group box.

   (b) Retain the default selection of Pressure... and Static Pressure from the Contours of drop-down lists.

   (c) Click Display.

2. Display contours of velocity magnitude (Figure 19.4).

   Display ⟶Contours...

   (a) Select Velocity... and Velocity Magnitude from the Contours of drop-down lists.

   (b) Click Display.

© Fluent Inc. September 21, 2006

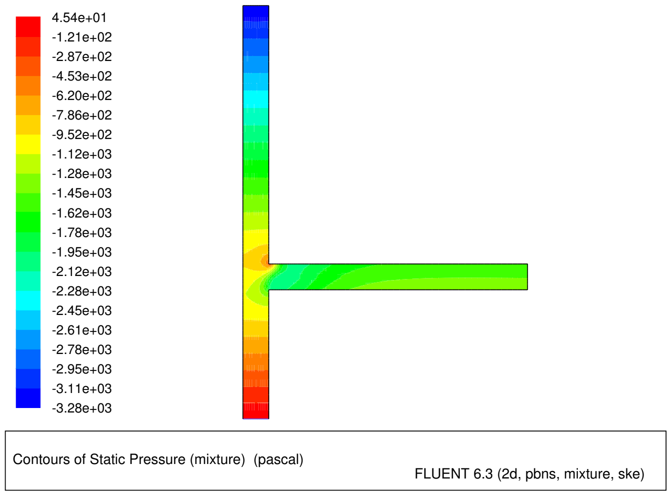


Figure 19.3: Contours of Static Pressure

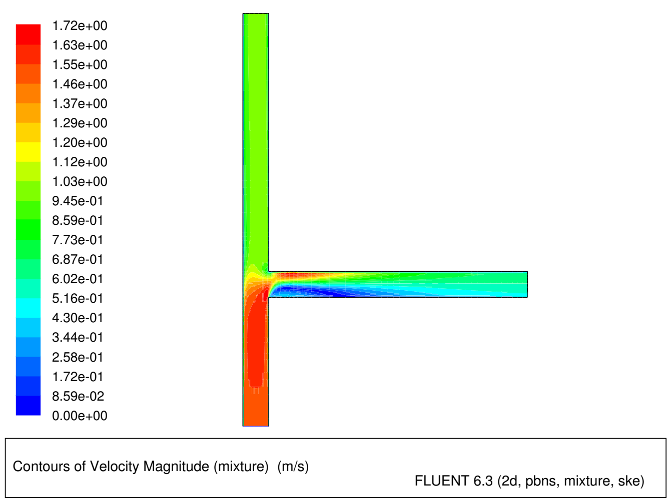


Figure 19.4: Contours of Velocity Magnitude

3. Display the volume fraction of air (Figure 19.5).

   Display ⟶Contours...

   (a) Select Phases... and Volume fraction from the Contours of drop-down lists.

   (b) Select air from the Phase drop-down list.

   (c) Click Display and close the Contours panel.

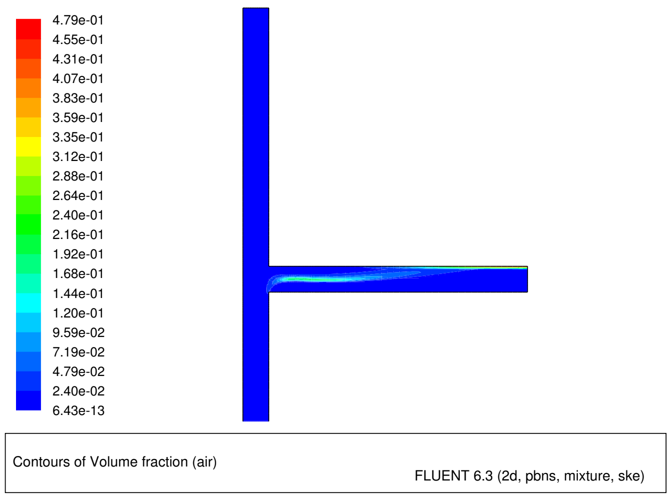


Figure 19.5: Contours of Air Volume Fraction

When gravity acts downwards, it induces stratification in the side arm of the tee junction. In Figure 19.5, you can see that the gas (air) tends to concentrate on the upper part of the side arm. In this case, gravity acts against inertia that tends to concentrate gas on the low pressure side, thereby creating gas pockets. In the vertical arm, the gas travels upward faster than the water due to the effect of gravity, and therefore there is less separation. The outflow split modifies the relation between inertia forces and gravity to a large extent, and has an important role in flow distribution and on the gas concentration.

© Fluent Inc. September 21, 2006

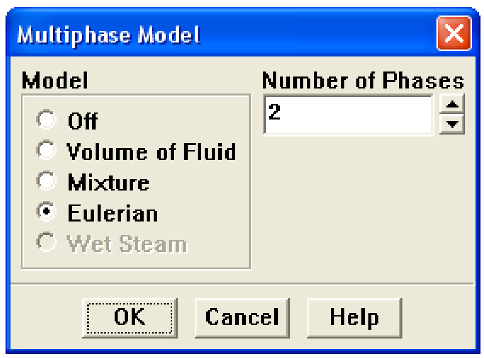
Step 8: Setup and Solution for the Eulerian Model

The mixture model is a simplification of the Eulerian model and is valid only when bubble velocity is in the same direction as water velocity. This assumption can be violated in the recirculation pattern. The Eulerian model is expected to make a more realistic prediction in this case.

You will use the solution obtained using the mixture model as an initial condition for the calculation using the Eulerian model.

1. Select the Eulerian model.

   Define ⟶ Models ⟶ Multiphase...

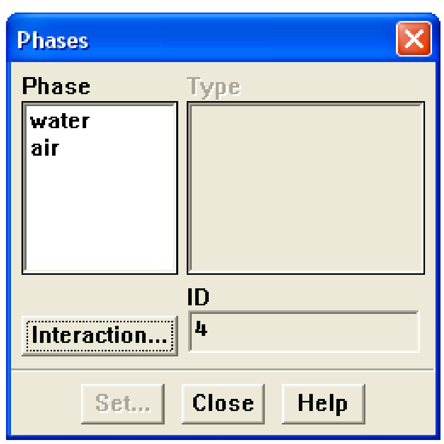


   (a) Select Eulerian from the Model list.

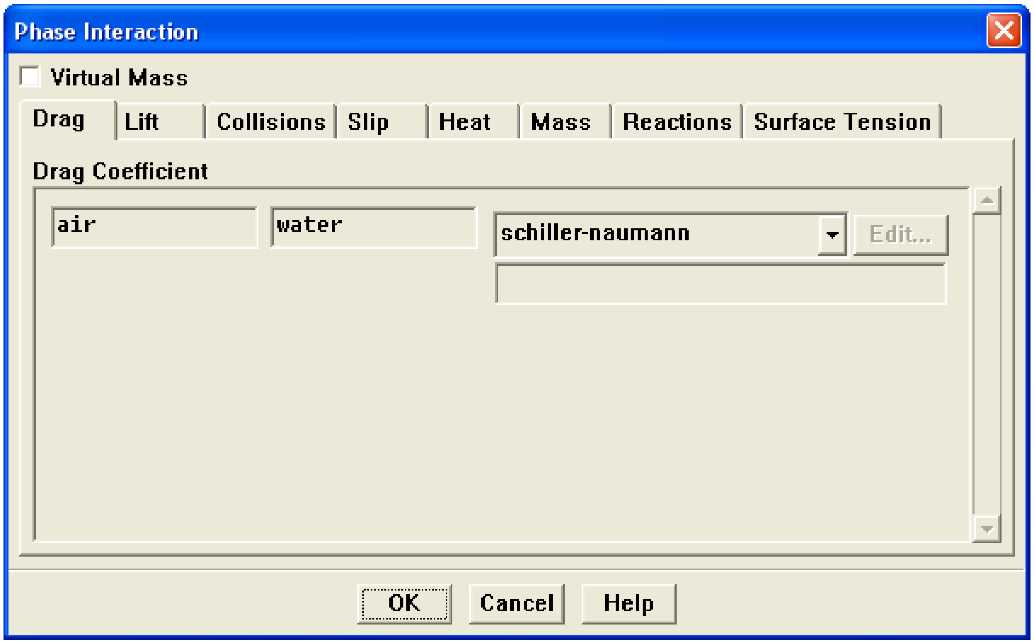
   (b) Click OK to close the Multiphase Model panel.

2. Specify the drag law to be used for computing the interphase momentum transfer.

   Define ⟶ Phases...



(a) Click the Interaction... button to open the Phase Interaction panel.



i. Retain the default selection of schiller-naumann from the Drag Coefficient drop-down list.

ii. Click OK to close the Phase Interaction panel.

Note: *For this problem, there are no parameters to be set for the individual phases other than those that you specified when you set up the phases for the mixture model calculation. If you use the Eulerian model for a flow involving a granular secondary phase, you will need to set additional parameters. There are also other options in the* Phase Interaction *panel that may be relevant for other applications.*

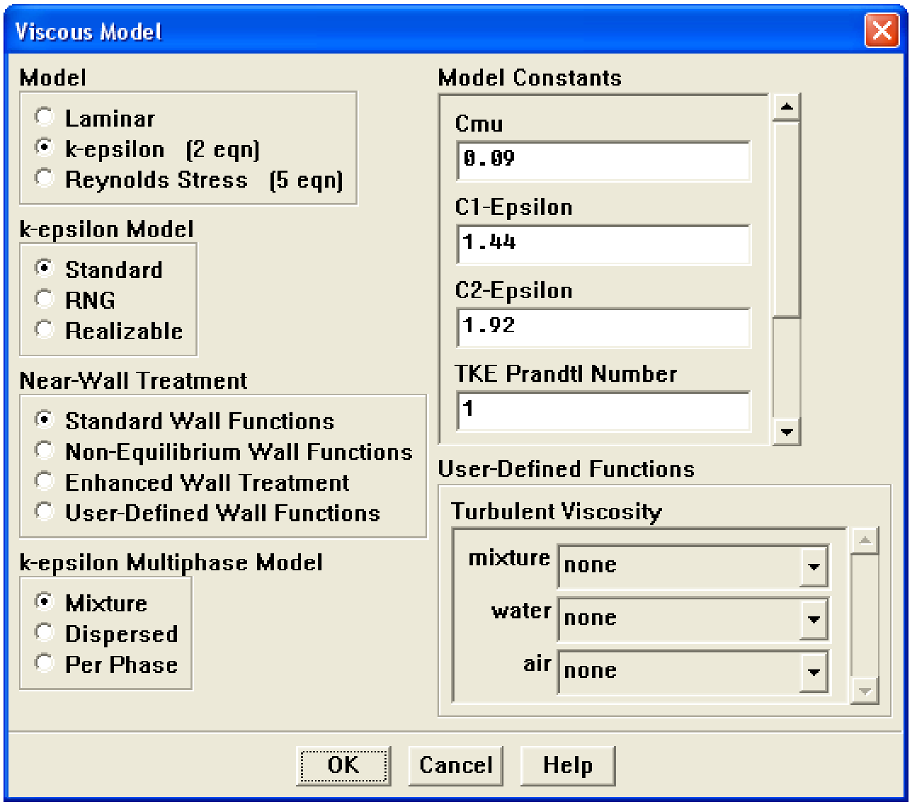
(b) Close the Phases panel.

See Section 23.9 of the User's Guide for complete details on setting up an Eulerian multiphase calculation.

© Fluent Inc. September 21, 2006

3. Select the multiphase turbulence model.

   Define $\longrightarrow$ Models $\longrightarrow$ Viscous...



   (a) Retain the default selection of Mixture from the k-epsilon Multiphase Model list.

   (b) Click OK to close the Viscous Model panel.

   The mixture turbulence model is applicable when phases separate, for stratified (or nearly stratified) multiphase flows, and when the density ratio between phases is close to 1. In these cases, using mixture properties and mixture velocities is sufficient to capture important features of the turbulent flow.

   See Chapter 23 of the User's Guide for more information on turbulence models for the Eulerian multiphase model.

4. Continue the solution by requesting 1000 additional iterations.

   Solve $\longrightarrow$ Iterate...

   The solution will converge after approximately 55 iterations.

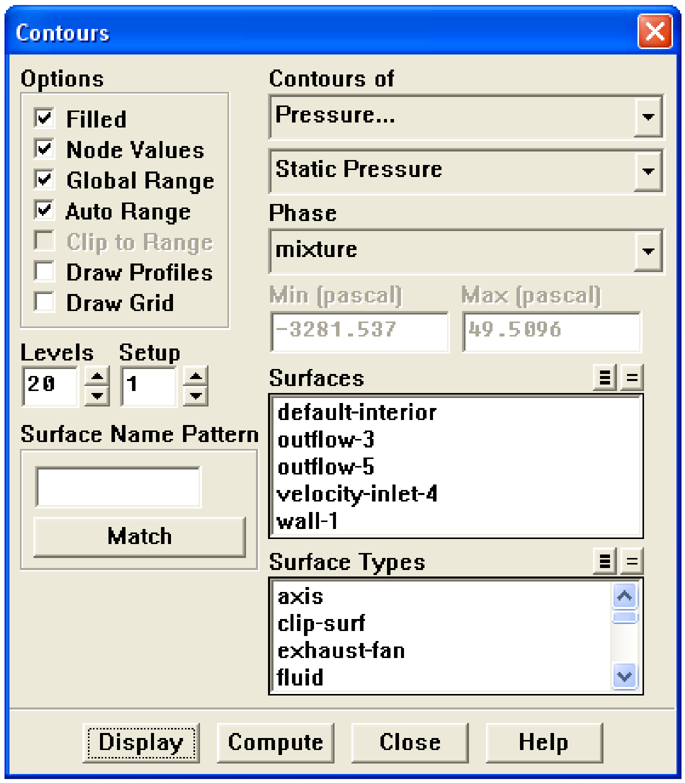
5. Save the case and data files (`tee2.cas.gz` and `tee2.dat.gz`).

   File $\longrightarrow$ Write $\longrightarrow$ Case & Data...

Step 9: Postprocessing for the Eulerian Model

1. Display the static pressure field in the tee for the mixture (Figure 19.6).

   Display ⟶Contours...



   (a) Select Pressure... from the Contours of drop-down list.

   By default, Dynamic Pressure *will be displayed in the lower* Contours of *drop-down list. This will change automatically to* Static Pressure *after you select the appropriate phase in the next step.*

   (b) Select mixture from the Phase drop-down list.

   The lower Contours of *drop-down list will now display* Static Pressure.

   (c) Click Display.

2. Display contours of velocity magnitude for water (Figure 19.7).

   Display ⟶Contours...

   (a) Select Velocity... and Velocity Magnitude from the Contours of drop-down lists.

   (b) Retain the default selection of water from the Phase drop-down list.

   Since the Eulerian model solves individual momentum equations for each phase, you can choose the phase for which solution data is plotted.

© Fluent Inc. September 21, 2006

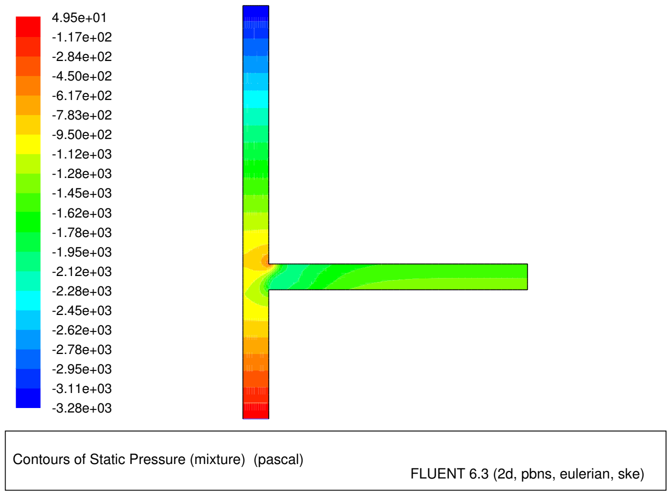


Figure 19.6: Contours of Static Pressure

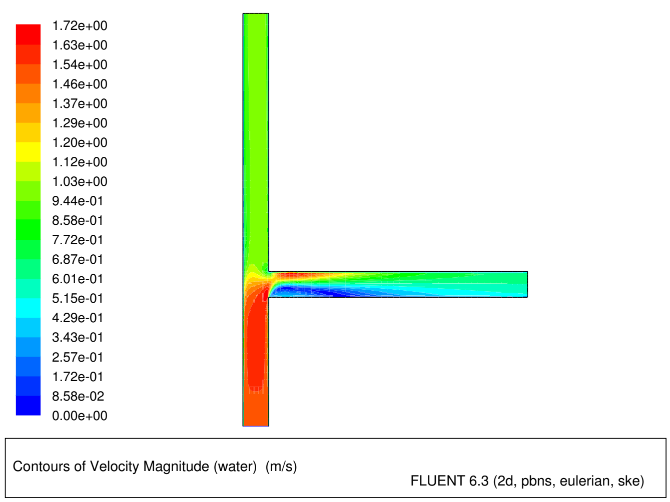


Figure 19.7: Contours of Water Velocity Magnitude

(c) Click Display.

3. Display the volume fraction of air (Figure 19.8).

   Display ⟶Contours...

   (a) Select Phases... and Volume fraction from the Contours of drop-down lists.

   (b) Select air from the Phases drop-down list.

   (c) Click Display and close the Contours panel.

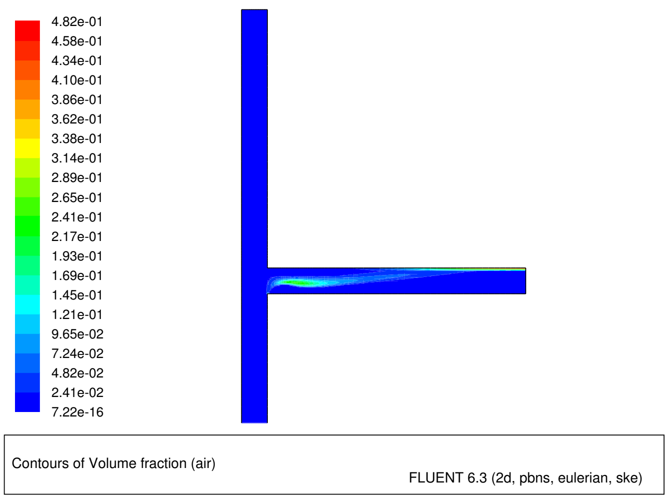


Figure 19.8: Contours of Air Volume Fraction

Summary

This tutorial demonstrated how to set up and solve a multiphase problem using the mixture model and the Eulerian model. You learned how to set boundary conditions for the mixture and both phases. The solution obtained with the mixture model was used as a starting point for the calculation with the Eulerian model. After completing calculations for each model, you displayed the results to allow for a comparison of the two approaches. See Chapter 23 of the User's Guide for more information about the mixture and Eulerian models.

Further Improvements

This tutorial guides you through the steps to reach an initial set of solutions. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the grid. Grid adaption can also ensure that the solution is independent of the grid. These steps are demonstrated in Tutorial 1.

 © Fluent Inc. September 21, 2006