Tutorial 17. Using the Mixture and Eulerian Multiphase Models

Introduction: This tutorial examines the flow of water and air in a tee junction. First you will solve the problem using the less computationally-intensive mixture model, and then you will turn to the more accurate Eulerian model. Finally, you will compare the results obtained with the two approaches.

In this tutorial you will learn how to:

- Use the mixture model with slip velocities
- Set boundary conditions for internal flow
- Calculate a solution using the segregated solver
- Use the Eulerian model
- Compare the results obtained with the two approaches

Prerequisites: This tutorial assumes that you are familiar with the menu structure in FLUENT and that you have solved or read 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 geometry and data for the problem are shown in Figure 17.1.

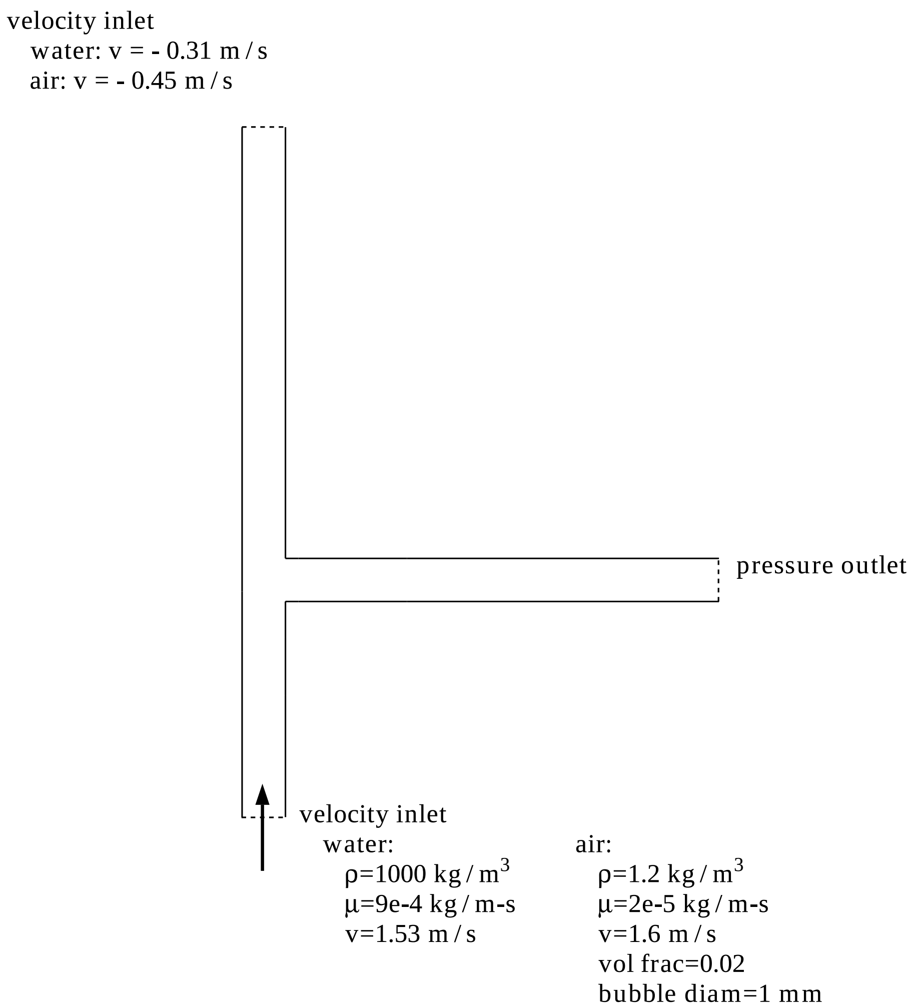


Figure 17.1: Problem Specification

Preparation

1. Copy the file `tee/tee.msh` from the FLUENT documentation CD to your working directory (as described in Tutorial 1).
2. Start the 2D version of FLUENT.

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 window.*

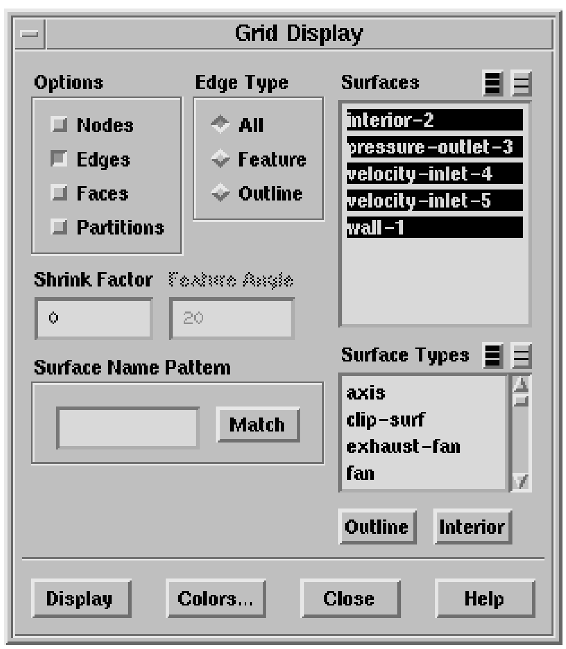
2. Check the grid.

   Grid ⟶ Check

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

3. Display the grid.

   Display ⟶ Grid...



   (a) Display the grid using the default settings (Figure 17.2).

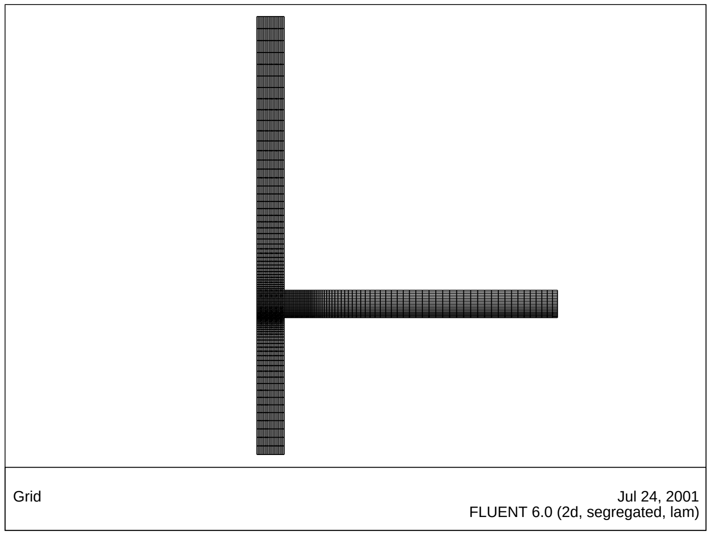


Figure 17.2: The Grid in the Tee Junction

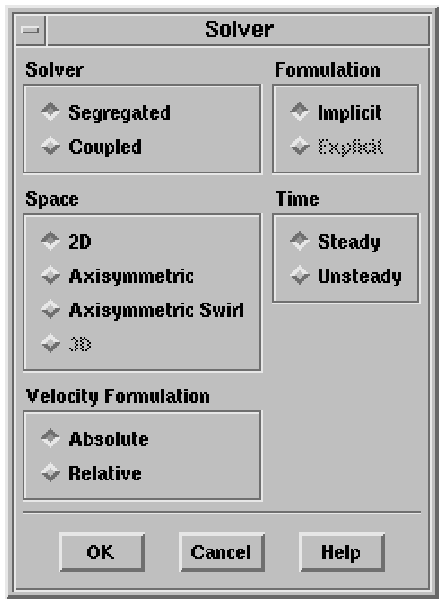
Extra: *You can use the right mouse button to check which zone number corresponds to each boundary. If you click the right mouse button on one of the boundaries in the graphics window, its zone number, name, and type will be printed in the* FLUENT *console window. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.*

Step 2: Models

1. Keep the default settings for the 2D segregated steady-state solver.

   Define → Models → Solver...

   The segregated solver must be used for multiphase calculations.



2. Enable the multiphase mixture model with slip velocities.

   Define → Models → Multiphase...

   (a) Select Mixture as the Model.

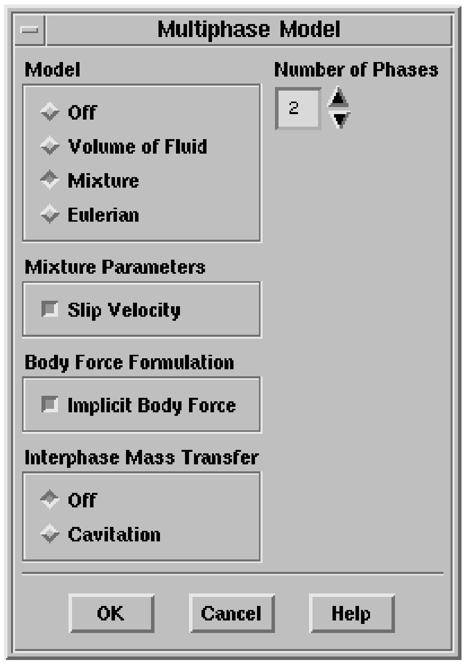
   The panel will expand to show the inputs for the mixture model.

   (b) Under Mixture Parameters, keep the Slip Velocity turned on.

   Since there will be significant difference in velocities for the different phases, you need to solve the slip velocity equation.

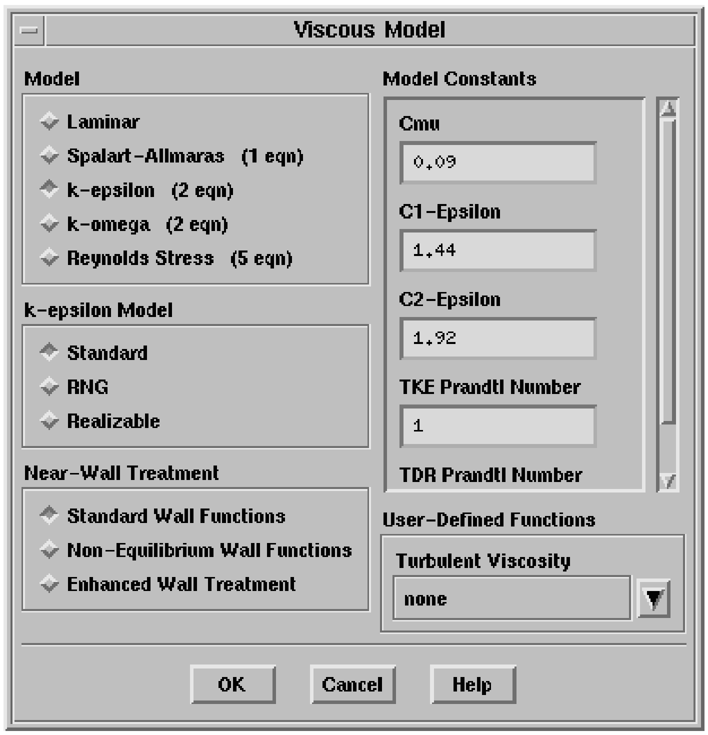
(c) Under Body Force Formulation, select Implicit Body Force.

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 when body forces are large in comparison to viscous and convective forces, namely in VOF and mixture problems.



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

   Define ⟶ Models ⟶ Viscous...



   (a) Select k-epsilon as the Model.

   (b) Under k-epsilon Model, keep the default selection of Standard.

   The standard k-ϵ model has been found to be quite effective in accurately resolving mixture problems when standard wall functions are used.

   (c) Keep the default selection of Standard Wall Functions under Near-Wall Treatment.

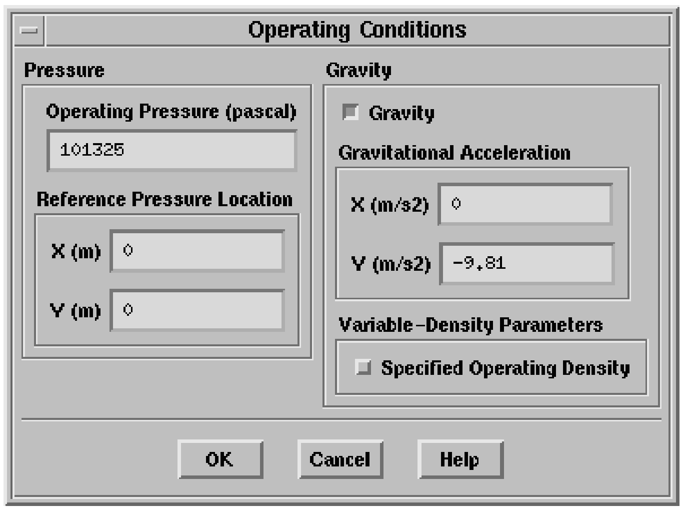
   This problem does not require a particularly fine grid, and standard wall functions will be used.

4. Set the gravitational acceleration.

   $\boxed{\text{Define}}$ ⟶Operating Conditions...

   (a) Turn on Gravity.

   The panel will expand to show additional inputs.



   (b) Set the Gravitational Acceleration in the Y direction to -9.81 m/s^2.

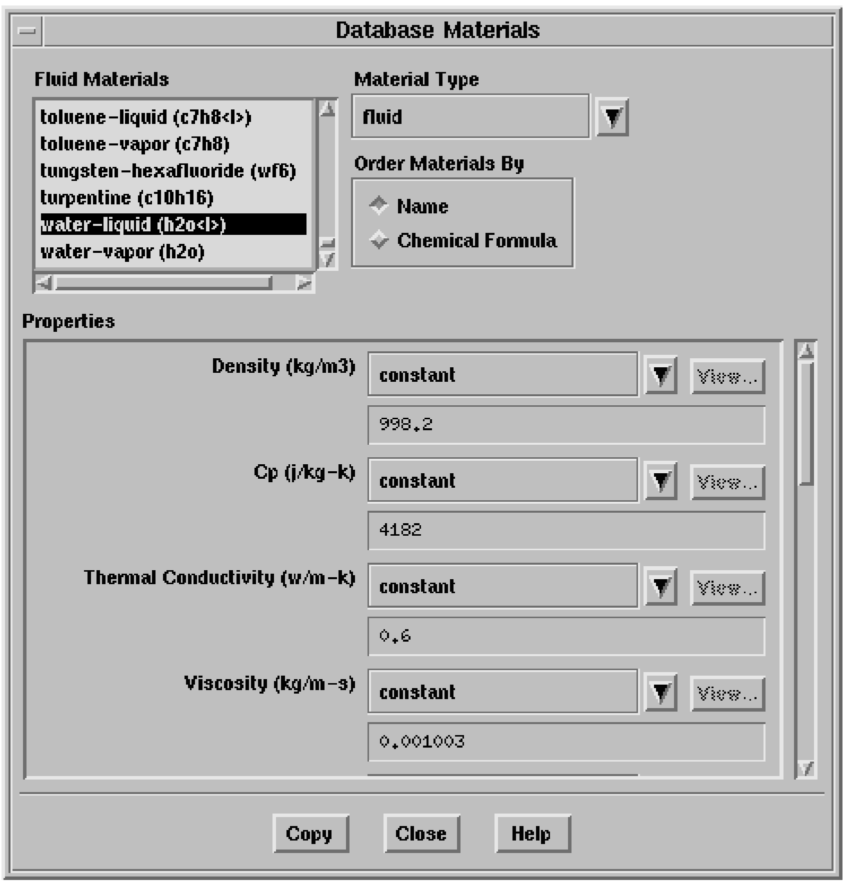
Step 3: Materials

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

   $\boxed{\text{Define}}$ ⟶Materials...

   (a) Click the Database... button in the Materials panel.

   The Database Materials *panel will open.*



(b) In the list of Fluid Materials, select water-liquid (h2o<l>).

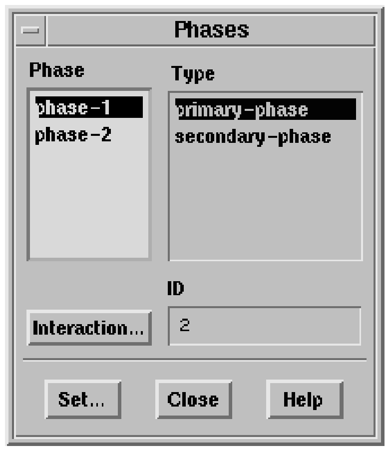
(c) Click Copy to copy the information for liquid water to your model.

(d) Close the Database Materials panel and the Materials panel.

Step 4: Phases

1. Define the liquid water and air phases that flow in the tee junction.

   Define →Phases...



   (a) Specify liquid water as the primary phase.

      i. Select phase-1 and click the Set... button.

      ii. In the Primary Phase panel, enter `water` for the Name.

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

(b) Specify air as the secondary phase.

i. Select phase-2 and click the Set... button.

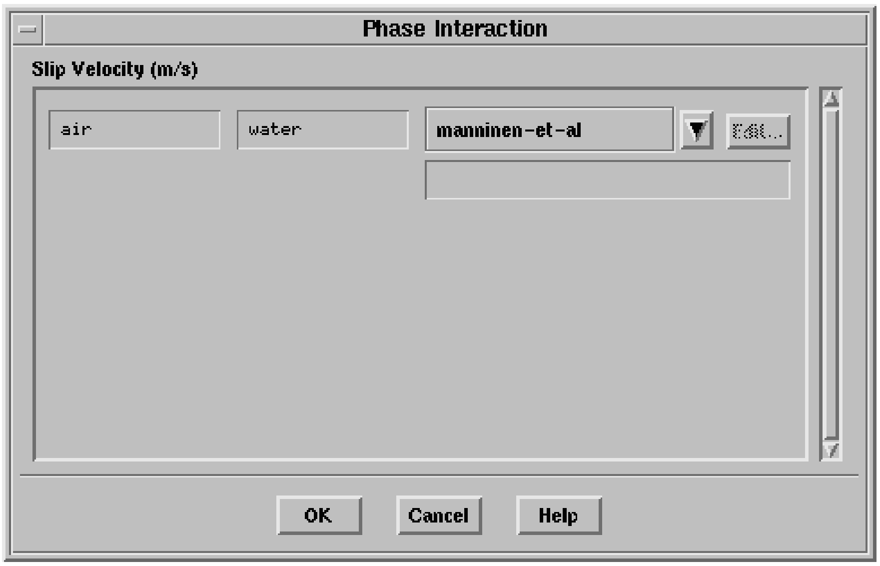
ii. In the Secondary Phase panel, enter `air` for the Name.

iii. Select air from the Phase Material drop-down list.

iv. Set the Diameter to `0.001` m.

2. Check the slip velocity formulation to be used.

   (a) Click the Interaction... button in the Phases panel.



   (b) In the Phase Interaction panel, keep the default selection of manninen-et-al in the Slip Velocity drop-down list.

Step 5: Boundary Conditions

For this problem, you need to set the boundary conditions for three boundaries: the upper and lower velocity inlets and the pressure outlet.

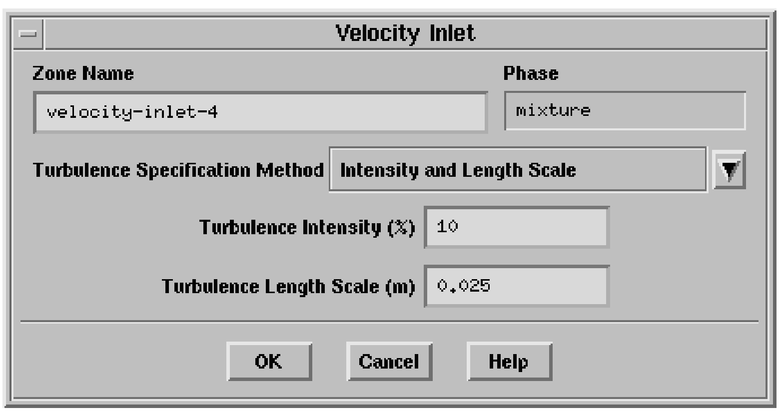
Define ⟶Boundary Conditions...

1. Set the conditions for the lower velocity inlet (velocity-inlet-4).

   For the multiphase mixture model, you will specify conditions at a velocity inlet for the mixture (i.e., conditions that apply to all phases) and also conditions that are specific to the primary and secondary phases.

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

      i. In the Boundary Conditions panel, keep the default selection of mixture in the Phase drop-down list and click Set....

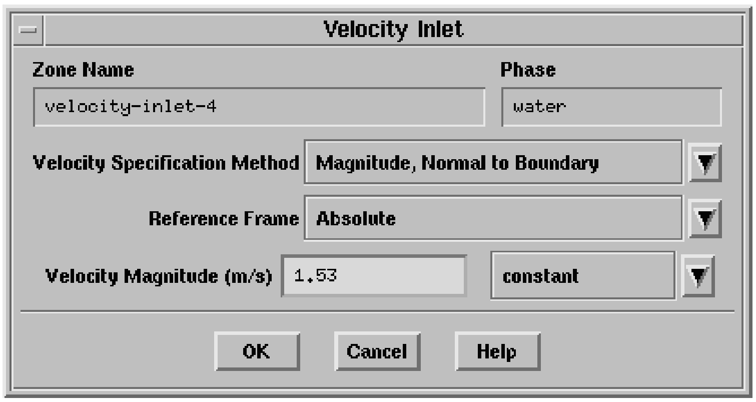


      ii. In the Turbulence Specification Method drop-down list, select Intensity and Length Scale.

      iii. Set Turbulence Intensity to 10% and Turbulence Length Scale to `0.025` m.

(b) Set the conditions for the primary phase.

i. In the Boundary Conditions panel, select water from the Phase drop-down list and click Set....

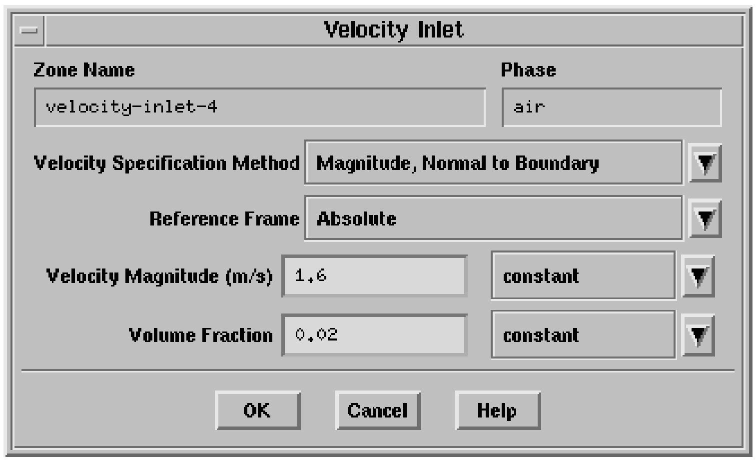


ii. Keep the default Velocity Specification Method and Reference Frame.

iii. Set the Velocity Magnitude to `1.53`.

(c) Set the conditions for the secondary phase.

i. In the Boundary Conditions panel, select air from the Phase drop-down list and click Set....



ii. Keep the default Velocity Specification Method and Reference Frame.

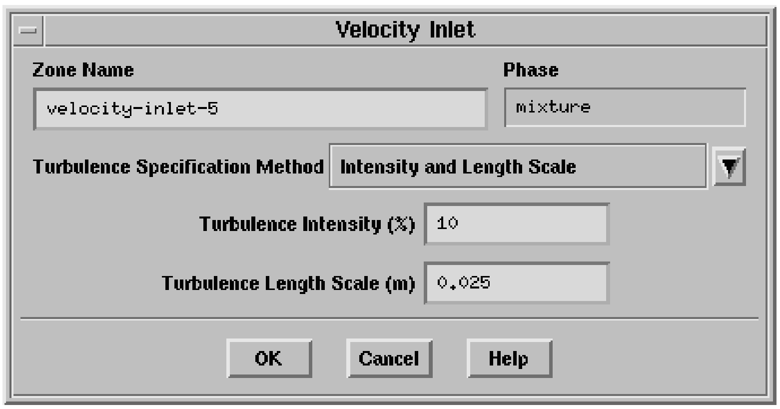
iii. Set the Velocity Magnitude to `1.6`.

iv. Set the Volume Fraction to `0.02`.

2. Set the conditions for the upper velocity inlet (velocity-inlet-5).

   (a) Set the conditions at velocity-inlet-5 for the mixture.

      i. In the Boundary Conditions panel, select mixture in the Phase drop-down list and click Set....

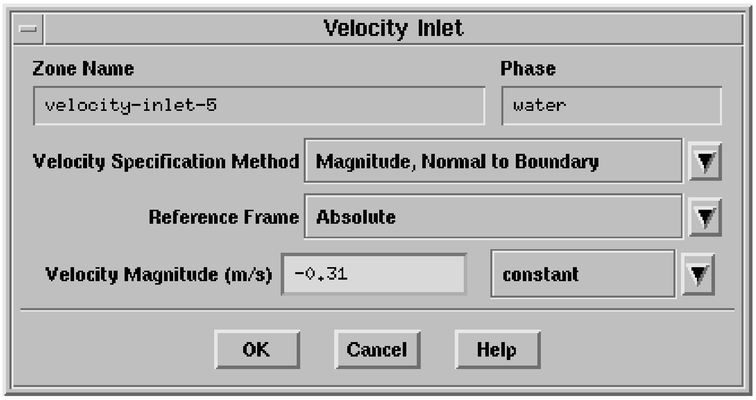


      ii. In the Turbulence Specification Method drop-down list, select Intensity and Length Scale.

      iii. Set Turbulence Intensity to 10% and Turbulence Length Scale to `0.025` m.

(b) Set the conditions for the primary phase.

i. In the Boundary Conditions panel, select water from the Phase drop-down list and click Set....



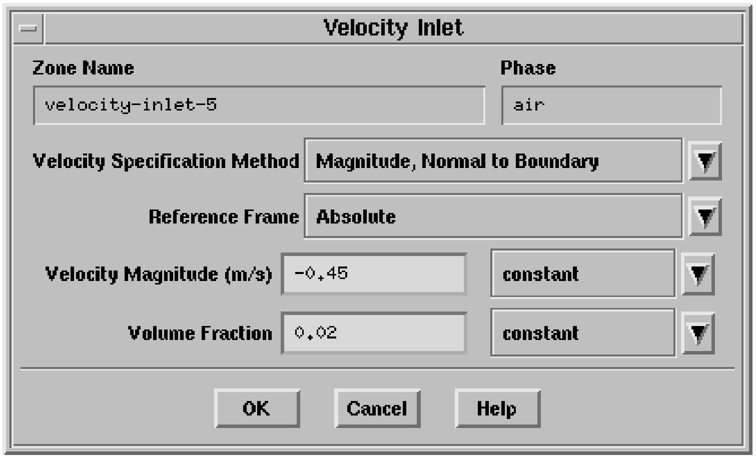
ii. Keep the default Velocity Specification Method and Reference Frame.

iii. Set the Velocity Magnitude to `-0.31`.

In this problem, outflow characteristics at the upper velocity inlet are assumed to be known, and therefore imposed as a boundary condition.

(c) Set the conditions for the secondary phase.

i. In the Boundary Conditions panel, select air from the Phase drop-down list and click Set....



ii. Keep the default Velocity Specification Method and Reference Frame.

iii. Set the Velocity Magnitude to -0.45.

iv. Set the Volume Fraction to 0.02.

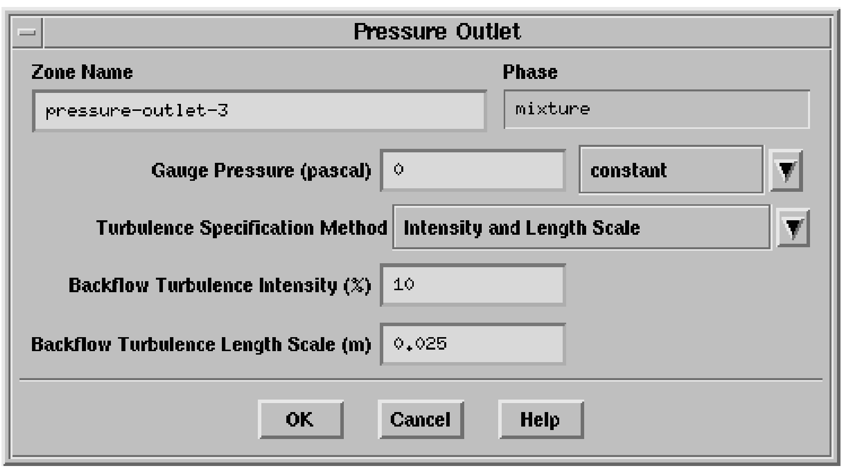
3. Set the boundary conditions for the pressure outlet (pressure-outlet-3).

   For the multiphase mixture model, you will specify conditions at a pressure outlet for the mixture and for the secondary phase. There are no conditions to be set for the primary phase.

   The turbulence conditions you input at the pressure outlet will be used only if flow enters the domain through this boundary. You can set them equal to the inlet values, as no flow reversal is expected at the pressure outlet. In general, however, it is important to set reasonable values for these downstream scalar values, in case flow reversal occurs at some point during the calculation.

   (a) Set the conditions at pressure-outlet-3 for the mixture.

      i. In the Boundary Conditions panel, select mixture in the Phase drop-down list and click Set....



      ii. In the Turbulence Specification Method drop-down list, select Intensity and Length Scale.

      iii. Set the Backflow Turbulence Intensity to 10%.

      iv. Set the Backflow Turbulence Length Scale to 0.025.

(b) Set the conditions for the secondary phase.

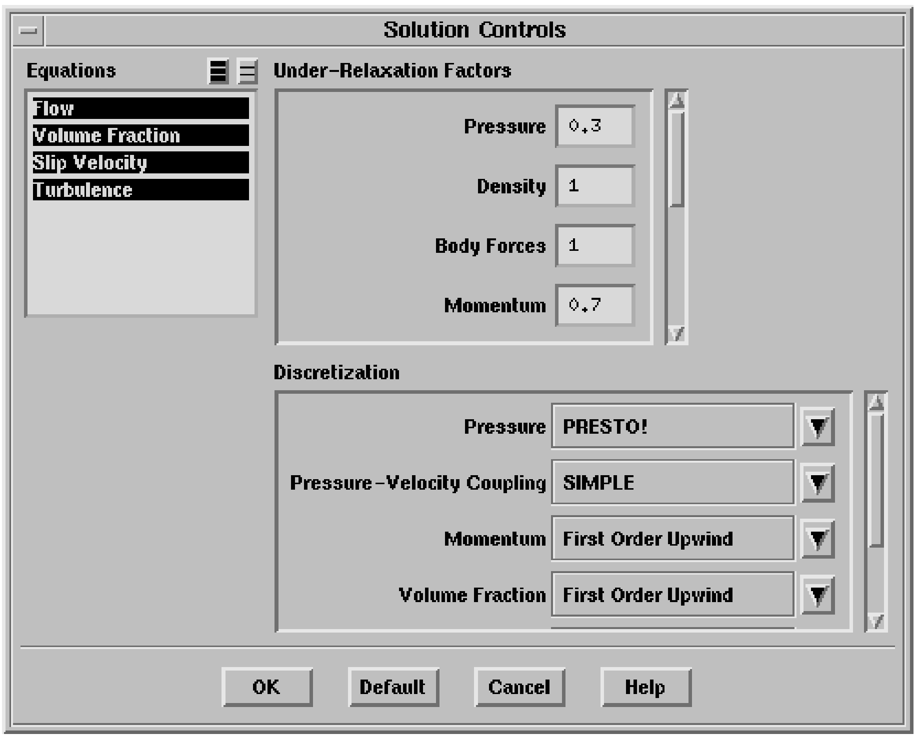
i. In the Boundary Conditions panel, select air from the Phase drop-down list and click Set....

ii. Set the Backflow Volume Fraction to `0.02`.

Step 6: Solution Using the Mixture Model

1. Set the solution parameters.

   Solve ⟶ Controls ⟶ Solution...



   (a) Keep all default Under-Relaxation Factors.

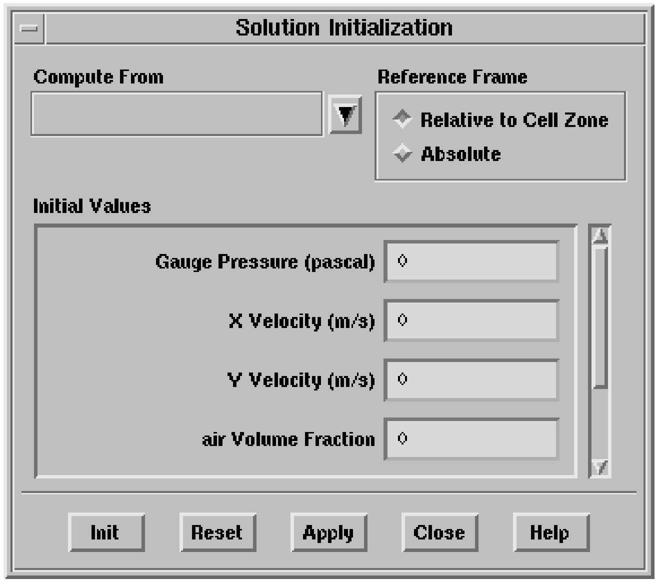
   (b) Under Discretization, select PRESTO! in the Pressure drop-down list.

2. Enable the plotting of residuals during the calculation.

   Solve ⟶ Monitors ⟶ Residual...

3. Initialize the solution.

   Solve ⟶ Initialize ⟶ Initialize...



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

   File ⟶ Write ⟶ Case...

5. Start the calculation by requesting 1000 iterations.

   Solve ⟶ Iterate...

   The solution will converge in approximately 600 iterations.

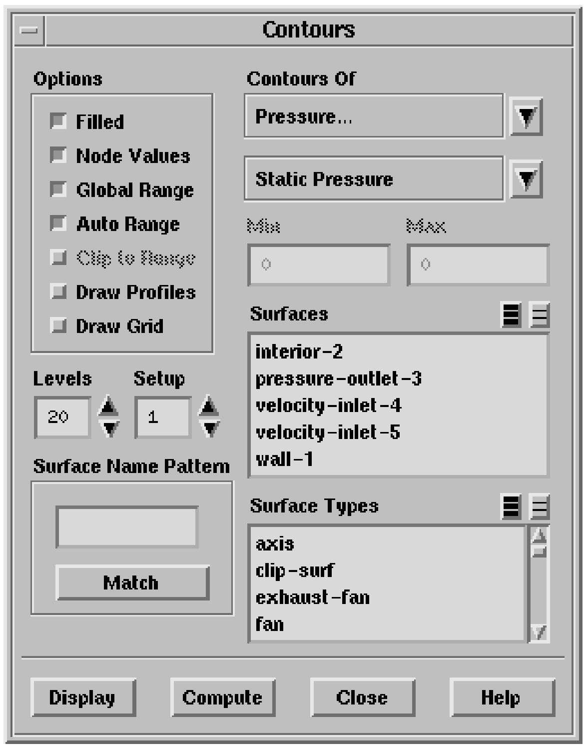
6. Save the case and data files (`tee.cas` and `tee.dat`).

   File ⟶ Write ⟶ Case & Data...

Step 7: Postprocessing for the Mixture Solution

1. Display the pressure field in the tee.

   Display ⟶Contours...



   (a) Select Pressure... and Static Pressure in the Contours Of drop-down lists.

   (b) Select Filled under Options.

   (c) Click Display.

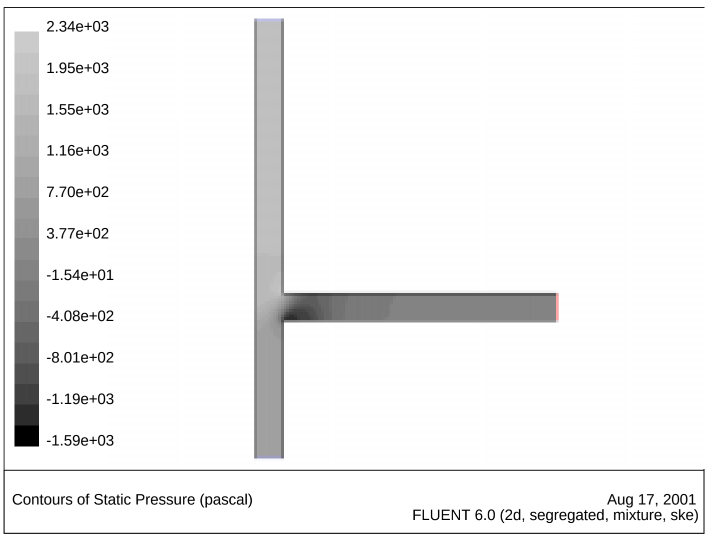


Figure 17.3: Contours of Static Pressure

2. Display contours of velocity magnitude (Figure 17.4).

   Display ⟶Contours...

   (a) Select Velocity... and Velocity Magnitude in the Contours Of drop-down lists.

   (b) Click Display.

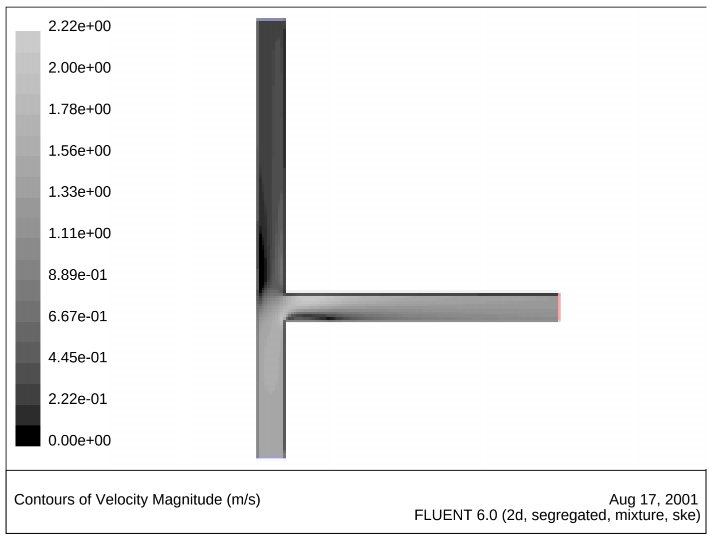


Figure 17.4: Contours of Velocity Magnitude

3. Display the volume fraction of air.

   Display ⟶Contours...

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

   (b) Click Display.

   In Figure 17.5, note the small bubble of air that separates at the sharp edge of the horizontal arm of the tee junction, and the small layer of air that floats in the same area above the water, marching towards the pressure outlet.

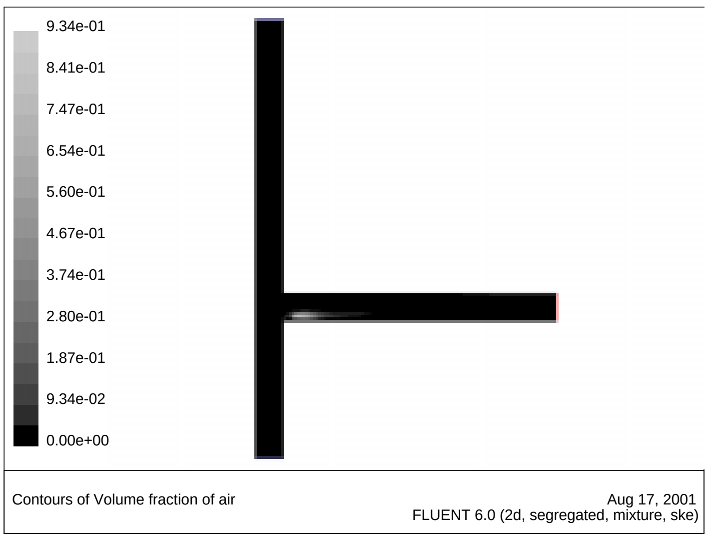


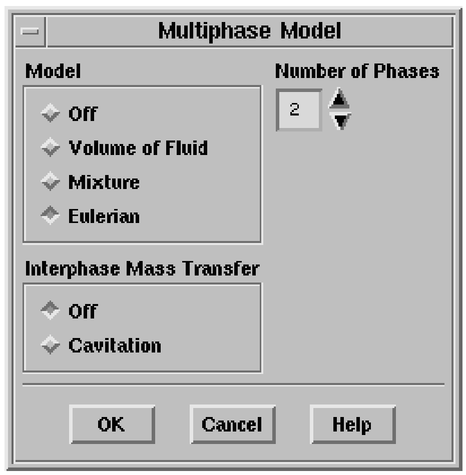
Figure 17.5: Contours of Air Volume Fraction

Step 8: Setup and Solution for the Eulerian Model

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

1. Turn on the Eulerian model.

   Define ⟶ Models ⟶ Multiphase...

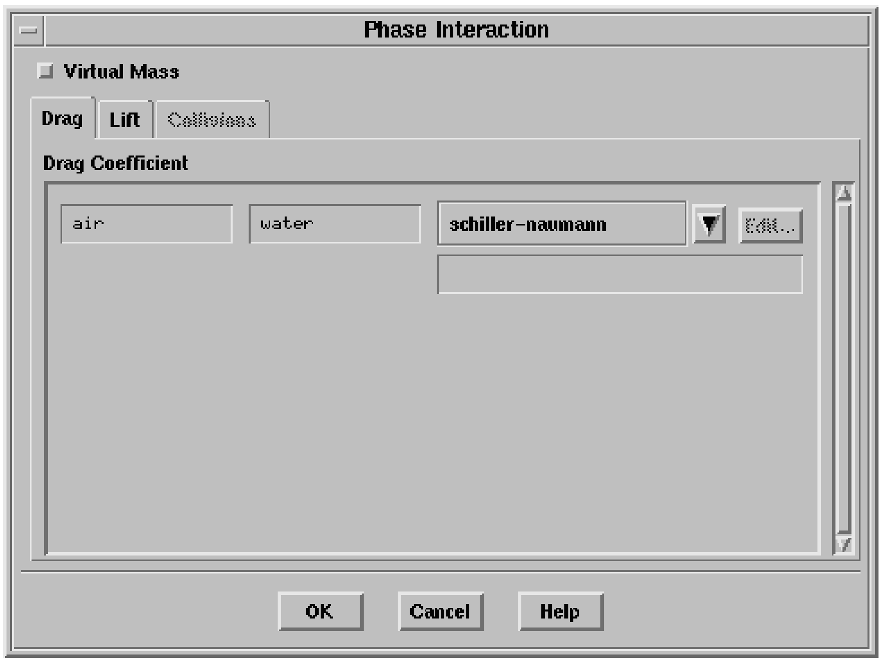


   (a) Under Models, select Eulerian.

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

   Define ⟶Phases...

   (a) Click the Interaction... button in the Phases panel.

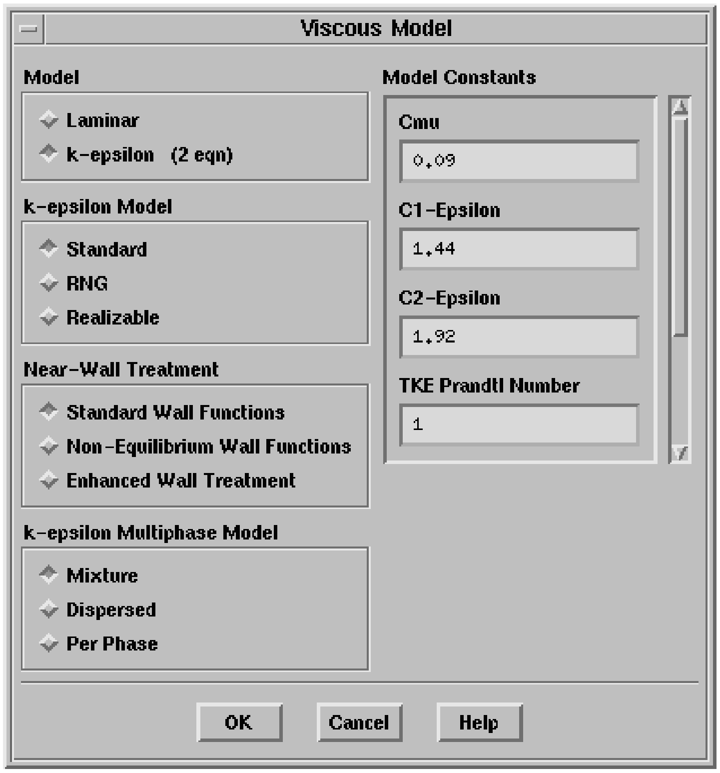


   (b) In the Phase Interaction panel, keep the default selection of schiller-naumann in the Drag Coefficient drop-down list.

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, there are additional parameters that you need to set. There are also other options in the* Phase Interaction *panel that may be relevant for other applications. See the User's Guide for complete details on setting up an Eulerian multiphase calculation.*

3. Select the multiphase turbulence model.

   Define ⟶ Models ⟶ Viscous...



   (a) Under k-epsilon Multiphase Model, keep the default selection of Mixture.

   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 section 20.4.7 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 about 300 additional iterations.

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

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

Step 9: Postprocessing for the Eulerian Model

1. Display the pressure field in the tee.

   Display ⟶Contours...

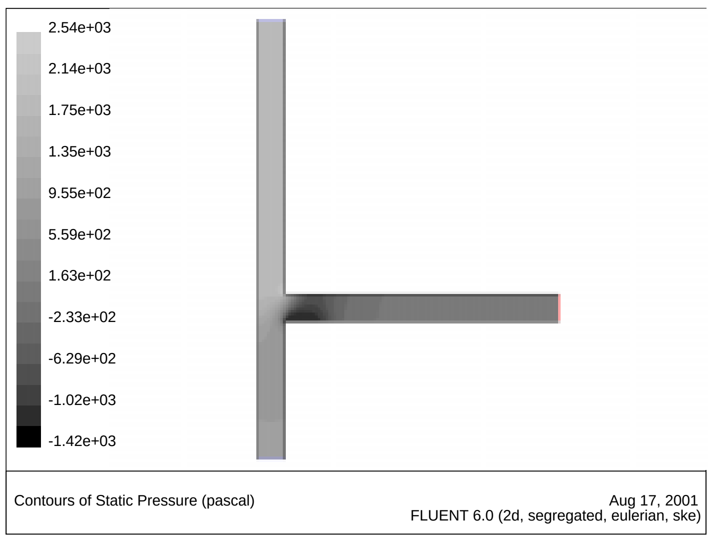


Figure 17.6: Contours of Static Pressure

2. Display contours of velocity magnitude for the water (Figure 17.7).

   Display ⟶Contours...

   (a) In the Contours Of drop-down lists, select Velocity... and water Velocity Magnitude.

   Because the Eulerian model solves individual momentum equations for each phase, you have the choice of which phase to plot solution data for.

   (b) Click Display.

3. Display the volume fraction of air.

   Display ⟶Contours...

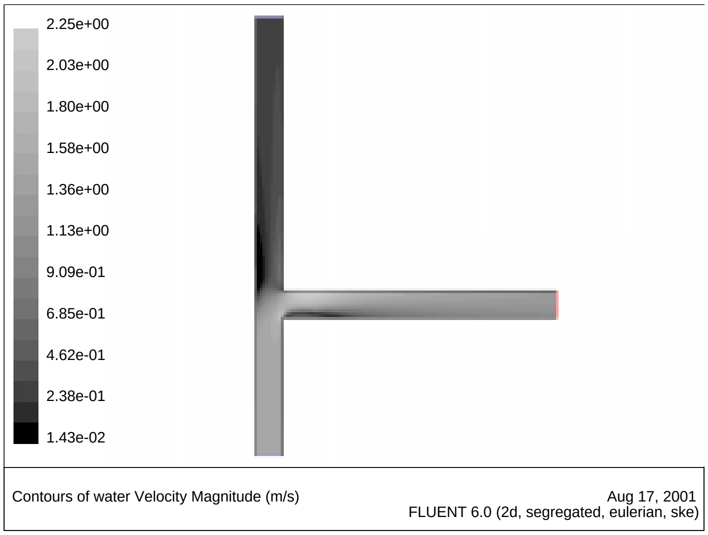


Figure 17.7: Contours of Water Velocity Magnitude

Figure 17.8: Contours of Air Volume Fraction

Note that the air bubble at the tee junction in Figure 17.8 is slightly different from the one that you observed in the solution obtained with the mixture model (Figure 17.5). The Eulerian model generally offers better accuracy than the mixture model, as it solves separate sets of equations for each individual phase, rather than modeling slip velocity between phases. See Sections 20.3 and 20.4 of the User's Guide for more information about the mixture and Eulerian models.

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 with both models, you compared the results obtained with the two approaches.