
Tutorial 20. 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 demonstrates 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 coupled solver with the mixture model.
- Use the Eulerian model.
- Calculate a solution using the multiphase coupled solver with the Eulerian model.
- Display the results obtained using the two approaches for comparison.

Prerequisites

This tutorial is written with the assumption that you have completed [Tutorial 1](#), 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.

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 20.1](#).

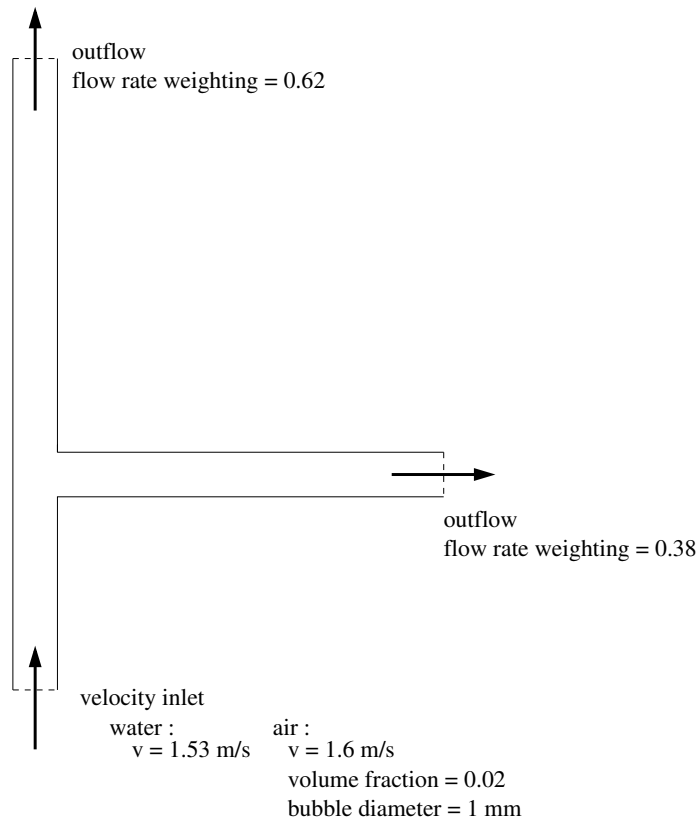


Figure 20.1: Problem Specification

Setup and Solution

Preparation

1. Download `mix_eulerian_multiphase.zip` from the [User Services Center](#) 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. Use FLUENT Launcher to start the 2D version of ANSYS FLUENT with Double Precision enabled.

For more information about FLUENT Launcher, see Section 1.1.2 in the separate [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.*

Step 1: Mesh

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

File → Read → Mesh...

As ANSYS FLUENT reads the mesh file, it will report the progress in the console.

Step 2: General Settings

◆ General

1. Check the mesh.

◆ General → Check

ANSYS FLUENT will perform various checks on the mesh and will report the progress in the console. Ensure that the reported minimum volume is a positive number.

2. Examine the mesh (Figure 20.2).

Extra: *You can use the right mouse button to probe for mesh information in the graphics window. If you click the right mouse button on any node in the mesh, information will be displayed in the ANSYS 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.*

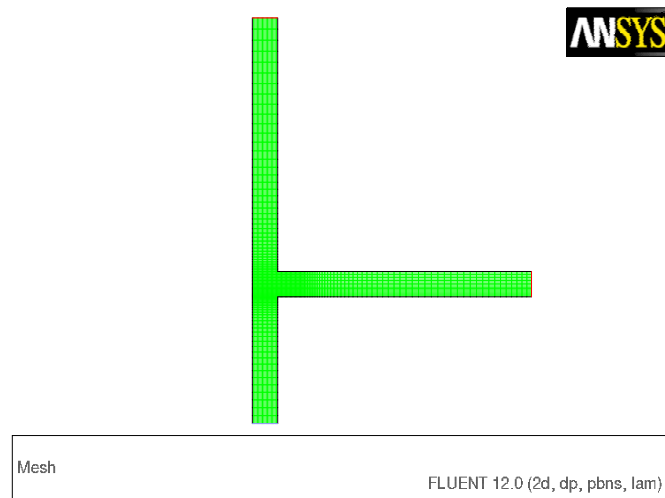
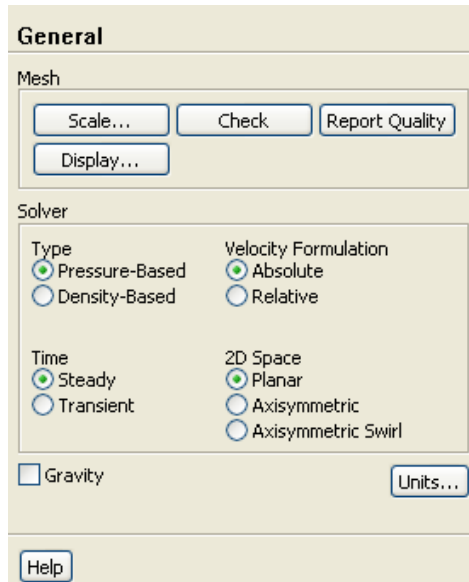


Figure 20.2: Mesh Display

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

◆ General



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

Step 3: Models

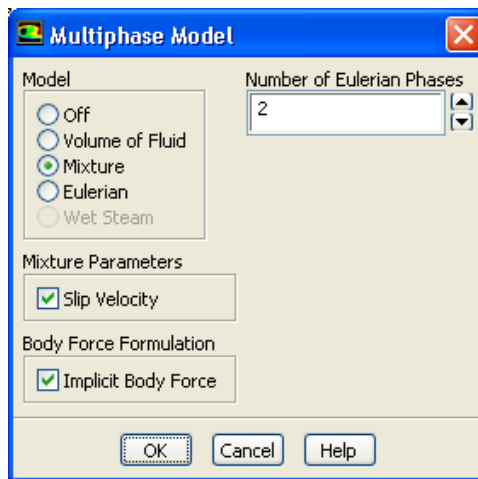
◆ Models

1. Select the mixture multiphase model with slip velocities.

◆ Models → Multiphase → Edit...

- (a) Select Mixture in the Model list.

The Multiphase Model dialog box will expand to show the inputs for the mixture model.



- (b) Ensure that Slip Velocity is enabled in the Mixture Parameters group box.

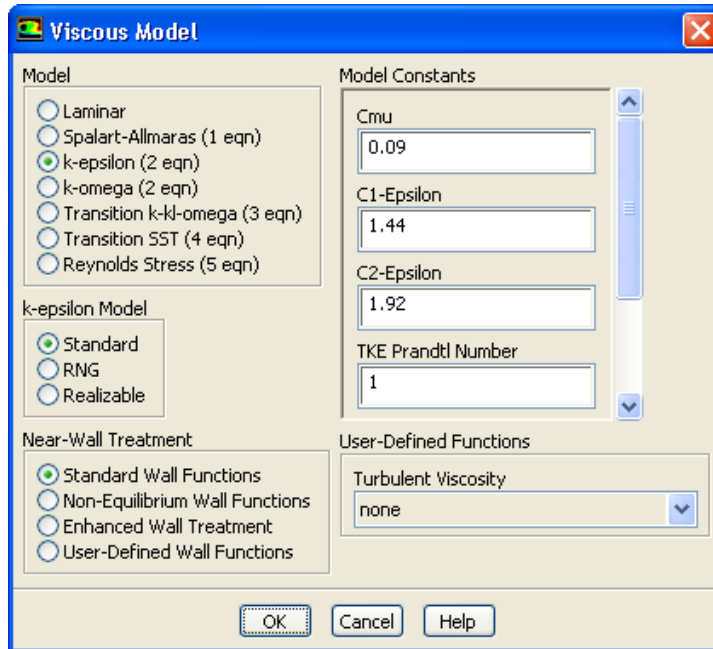
You need to solve the slip velocity equation since there will be significant difference in velocities for the different phases.

- (c) Enable Implicit Body Force 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 dialog box.

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



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

- (b) Retain the default selection of Standard in 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 in the Near-Wall Treatment list.

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

- (d) Click OK to close the Viscous Model dialog box.

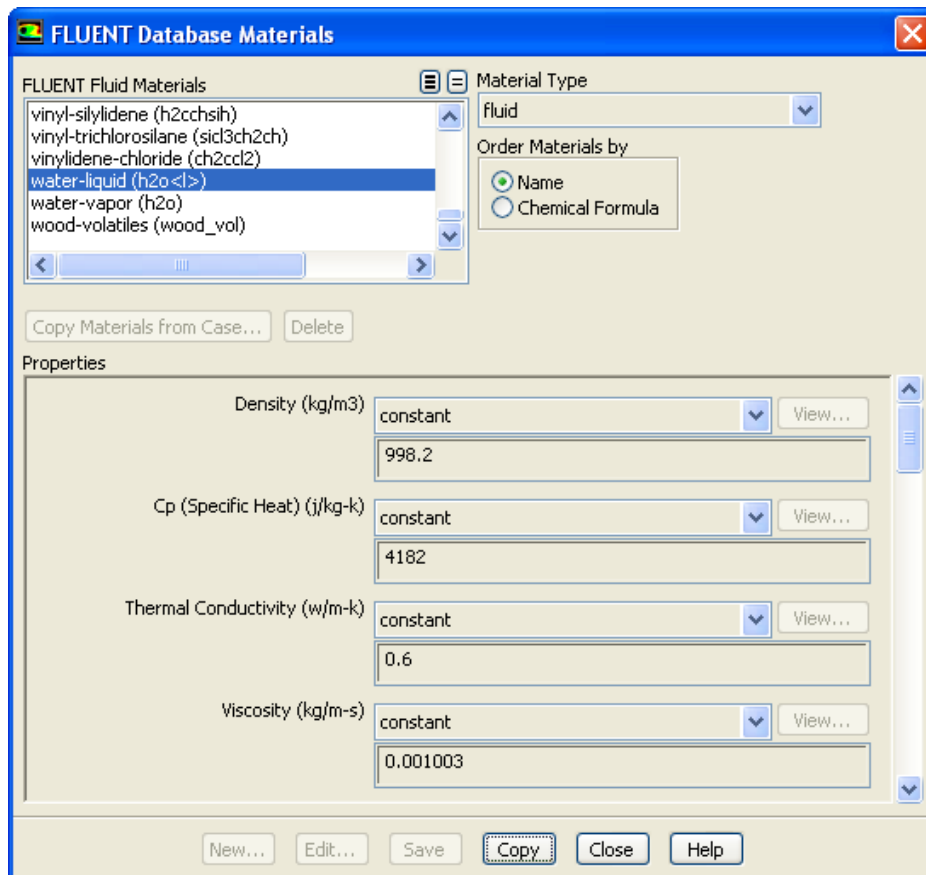
Step 4: Materials

Materials

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

Materials → Fluid → Create/Edit...

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



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

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

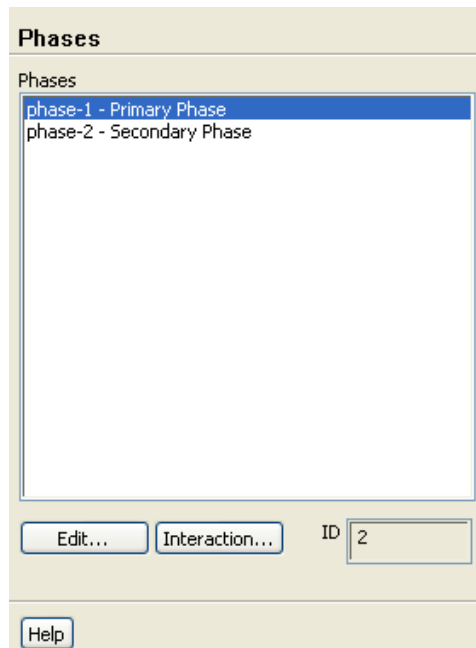
- Click Copy to copy the properties for liquid water to your model.
- Close the FLUENT Database Materials dialog box.

- Close the Create/Edit Materials dialog box.

Step 5: Phases

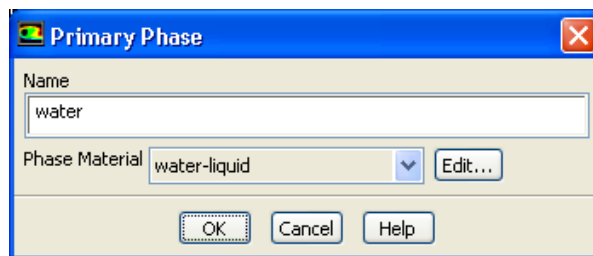
◆ Phases

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



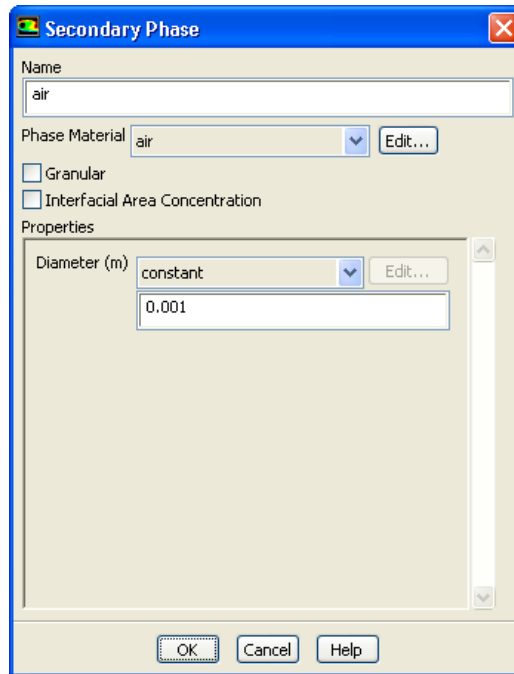
1. Specify liquid water as the primary phase.

◆ Phases → ☰ phase-1 → Edit...

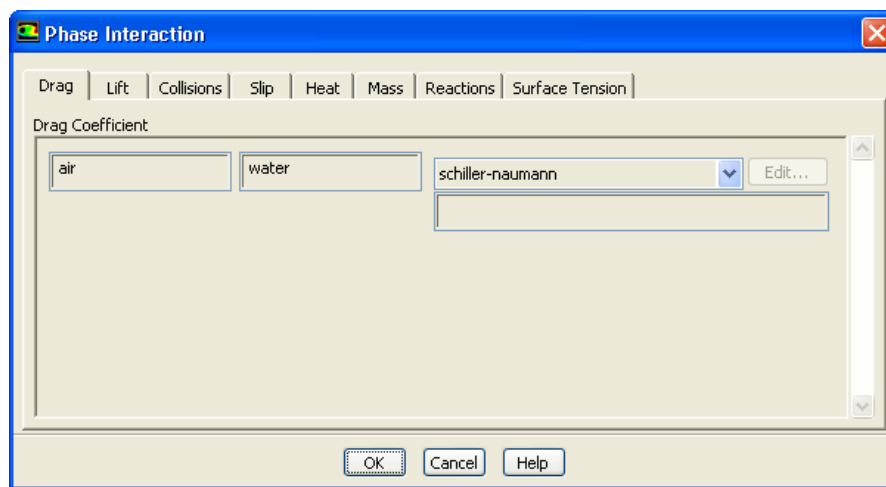


- (a) Enter **water** for Name.
- (b) Select **water-liquid** from the Phase Material drop-down list.
- (c) Click **OK** to close the Primary Phase dialog box.

- Specify air as the secondary phase.



- Enter `air` for Name.
 - Retain the default selection of `air` from the Phase Material drop-down list.
 - Enter `0.001` m for Diameter.
 - Click OK to close the Secondary Phase dialog box.
- Check that the drag coefficient is set to be calculated using the Schiller-Naumann drag law.



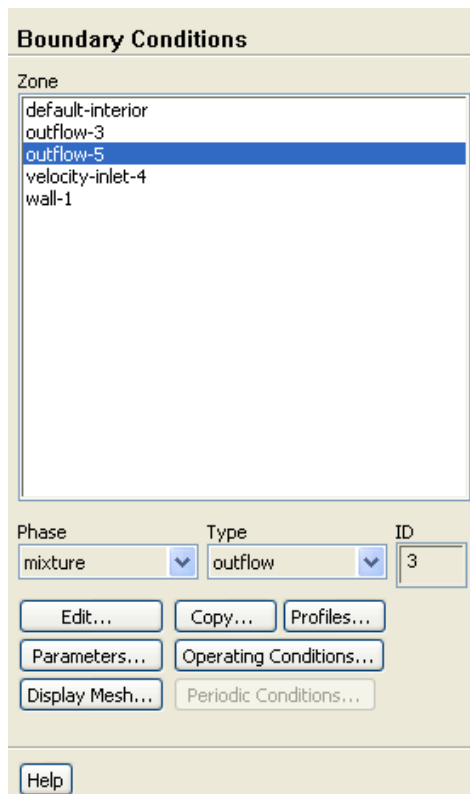
- (a) Retain the default selection of schiller-naumann from the Drag Coefficient drop-down list.

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.

- (b) Click OK to close the Phase Interaction dialog box.

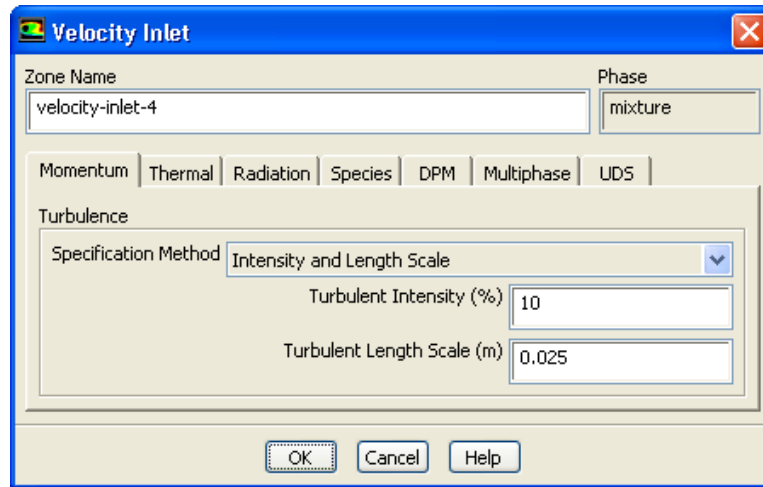
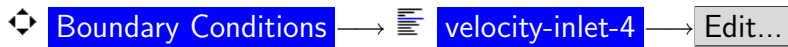
Step 6: Boundary Conditions

◆ Boundary Conditions



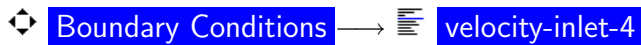
For this problem, you need to set the boundary conditions for three boundaries: the velocity inlet and the two outflows. Since this is a mixture multiphase model, you will set the conditions at the velocity inlet that are specific for the mixture (i.e., conditions that apply to all phases) and also conditions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

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

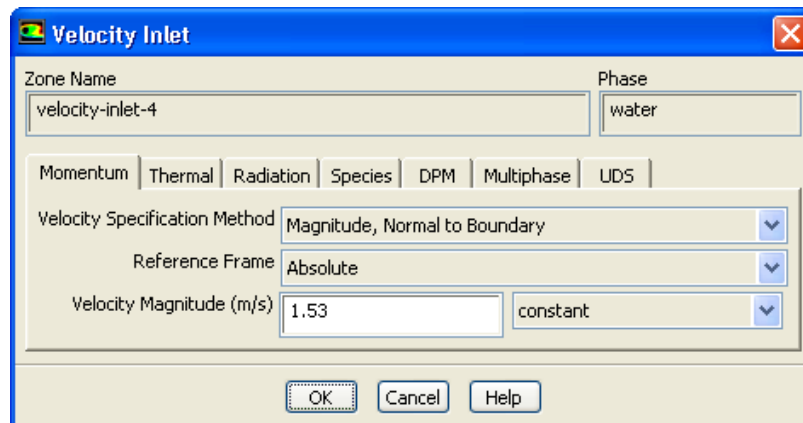


- (a) Select Intensity and Length Scale from the Specification Method drop-down list.
- (b) Retain the default value of 10% for Turbulent Intensity.
- (c) Enter 0.025 m for Turbulent Length Scale.
- (d) Click OK to close the Velocity Inlet dialog box.

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



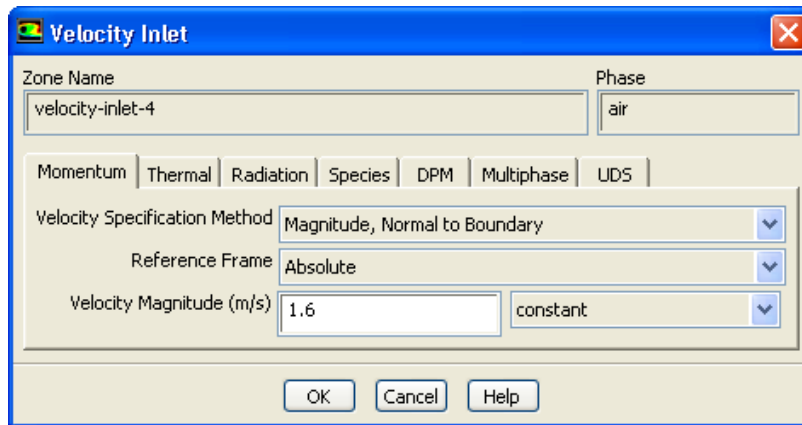
- (a) Select water from the Phase drop-down list.
- (b) Click Edit... to open the Velocity Inlet dialog box.



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

◆ **Boundary Conditions** → **velocity-inlet-4**

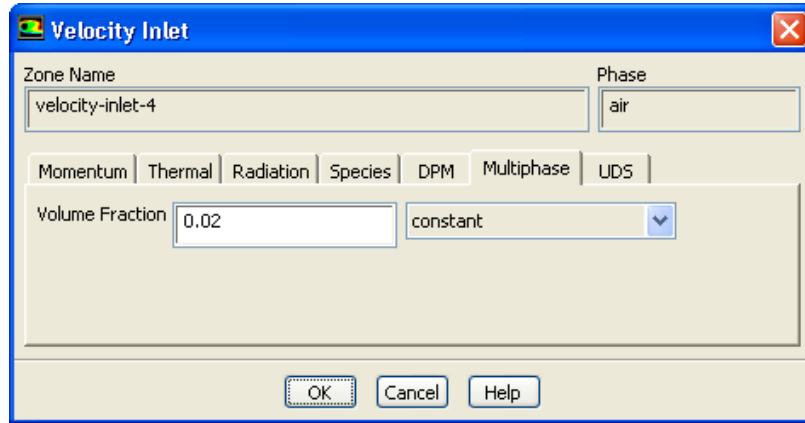
- (a) Select air from the Phase drop-down list.
- (b) Click Edit... to open the Velocity Inlet dialog box.



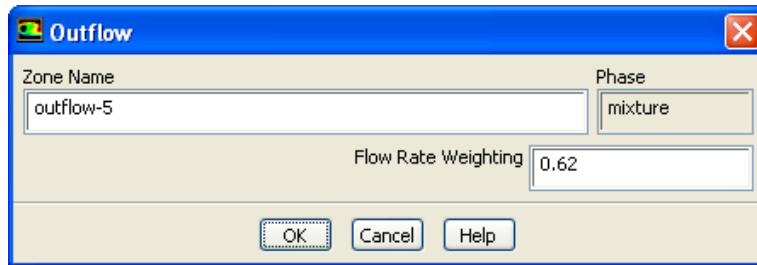
- i. Retain the default selection of Magnitude, Normal to Boundary from the Velocity Specification Method drop-down list.
- ii. Retain the default selection of Absolute from the Reference Frame drop-down list.
- iii. Enter 1.6 m/s for Velocity Magnitude.

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.

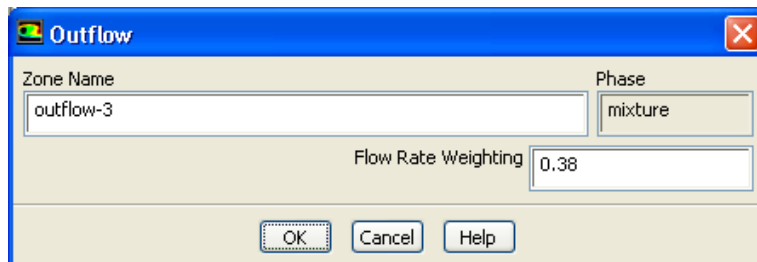
- iv. Click the Multiphase tab and enter 0.02 for Volume Fraction.



- v. Click OK to close the Velocity Inlet dialog box.
4. Set the boundary conditions at outflow-5 for the mixture.
- ◆ **Boundary Conditions** → **outflow-5**
 - (a) Select mixture from the Phase drop-down list.
 - (b) Click Edit... to open the Outflow dialog box.



- i. Enter 0.62 for Flow Rate Weighting.
 - ii. Click OK to close the Outflow dialog box.
5. Set the boundary conditions at outflow-3 for the mixture.
- ◆ **Boundary Conditions** → **outflow-3** → **Edit...**



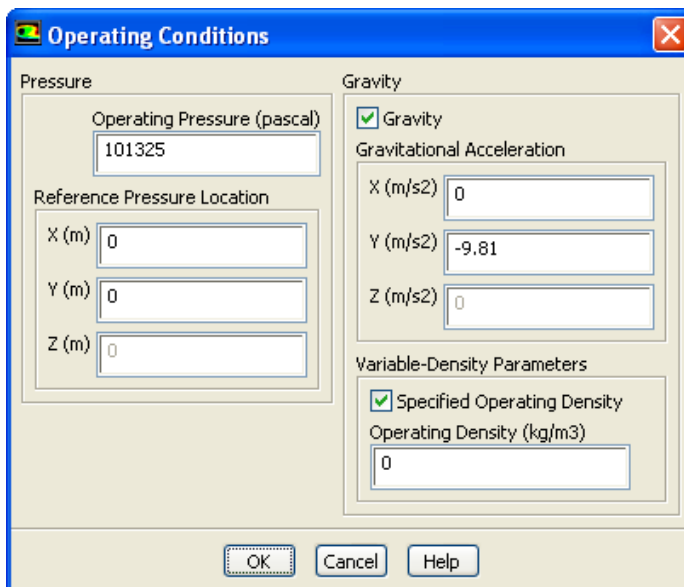
- (a) Enter 0.38 for Flow Rate Weighting.
- (b) Click OK to close the Outflow dialog box.

Step 7: Operating Conditions

◆ Boundary Conditions

1. Set the gravitational acceleration.

◆ Boundary Conditions → Operating Conditions...



- (a) Enable Gravity.

The Operating Conditions dialog box will expand to show additional inputs.

- (b) Enter -9.81 m/s^2 for Y in the Gravitational Acceleration group box.
- (c) Enable Specified Operating Density.
- (d) Enter 0 kg/m^3 for Operating Density.
- (e) Click OK to close the Operating Conditions dialog box.

Step 8: Solution Using the Mixture Model

1. Set the solution parameters.

◊ **Solution Methods**

The image shows a software dialog box titled "Solution Methods". It is divided into several sections:

- Pressure-Velocity Coupling:** Contains a "Scheme" drop-down menu with "Coupled" selected.
- Spatial Discretization:** Contains five drop-down menus:
 - Gradient: "Least Squares Cell Based"
 - Pressure: "PRESTO!"
 - Momentum: "First Order Upwind"
 - Volume Fraction: "First Order Upwind"
 - Turbulent Kinetic Energy: "First Order Upwind"
- Transient Formulation:** Contains a drop-down menu (currently blank) and two unchecked checkboxes: "Non-Iterative Time Advancement" and "Frozen Flux Formulation".

At the bottom of the dialog, there are two buttons: "Default" and "Help".

- (a) Select Coupled from the Scheme drop-down list.
- (b) Select PRESTO! from the Pressure drop-down list.

2. Set the solution controls.

◆ **Solution Controls**

- (a) Enter 40 for Courant Number.
- (b) Enter 0.5 for both Momentum and Pressure in the Explicit Relaxation Factors group box.

Solution Controls

Courant Number
40

Explicit Relaxation Factors

Momentum 0.5

Pressure 0.5

Under-Relaxation Factors

Slip Velocity
0.4

Volume Fraction
0.4

Turbulent Kinetic Energy
0.8

Turbulent Dissipation Rate
0.8

Turbulent Viscosity
1

Default

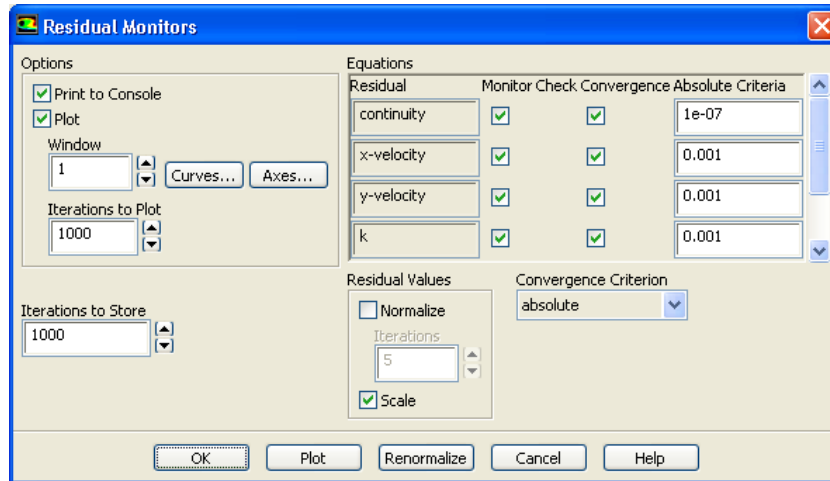
Equations... Limits... Advanced...

Help

- (c) Enter 0.4 for both Slip Velocity and Volume Fraction in the Under-Relaxation Factors group box.

3. Enable the plotting of residuals during the calculation.

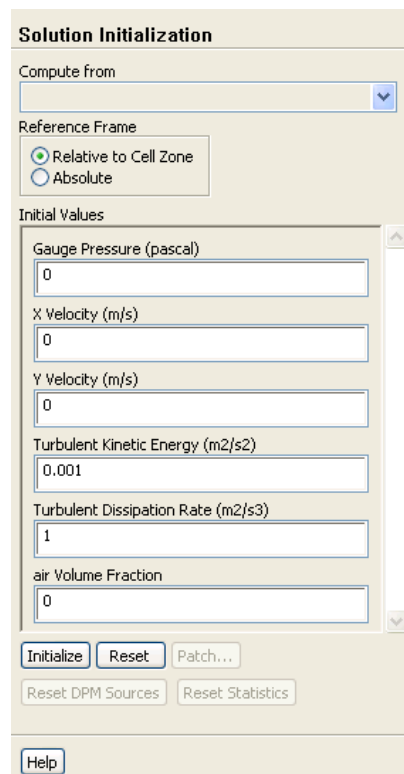
◆ **Monitors** → **Residuals** → **Edit...**



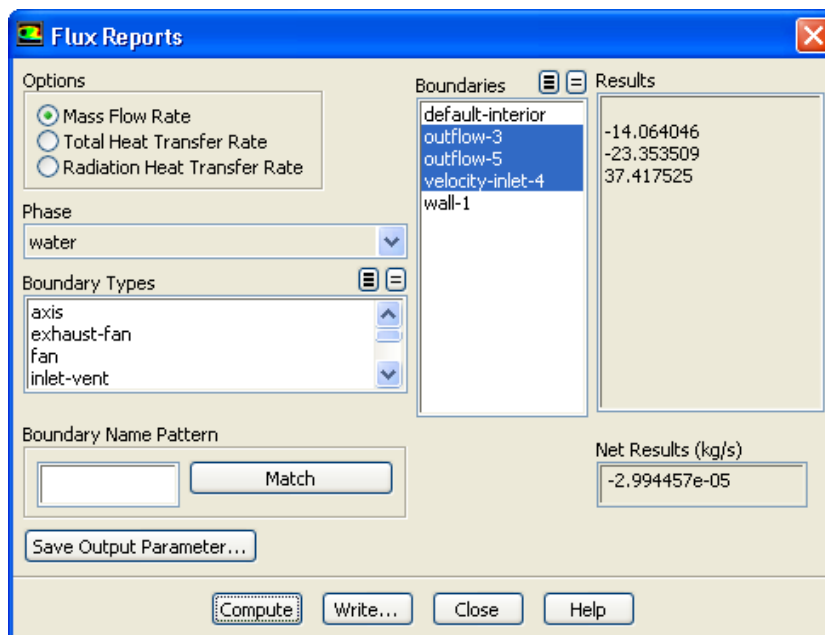
- (a) Ensure that Plot is enabled in the Options group box.
- (b) Enter 1e-07 for Absolute Criteria for continuity.
- (c) Click OK to close the Residual Monitors dialog box.

4. Initialize the solution.

◆ **Solution Initialization**



- (a) Enter $0.001 \text{ m}^2/\text{s}^2$ for Turbulent Kinetic Energy.
 - (b) Click Initialize.
5. Save the case file (tee.cas.gz).
- File → Write → Case...
6. Start the calculation by requesting 400 iterations.
- ◆ Run Calculation
7. Save the case and data files (tee.cas.gz and tee.dat.gz).
- File → Write → Case & Data...
8. Check the total mass flow rate for each phase.
- ◆ Reports → Fluxes → Set Up...



- (a) Retain the default selection of Mass Flow Rate in 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.

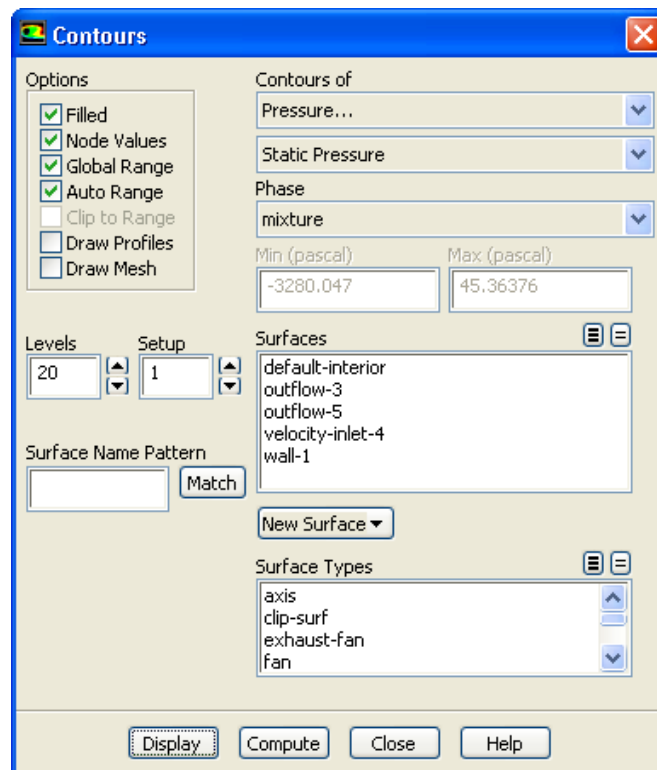
Note that the net mass flow rate is almost zero, indicating that total mass is conserved.

- (f) Close the Flux Reports dialog box.

Step 9: Postprocessing for the Mixture Solution

◆ Graphics and Animations

1. Display the static pressure field in the tee (Figure 20.3).



- (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.

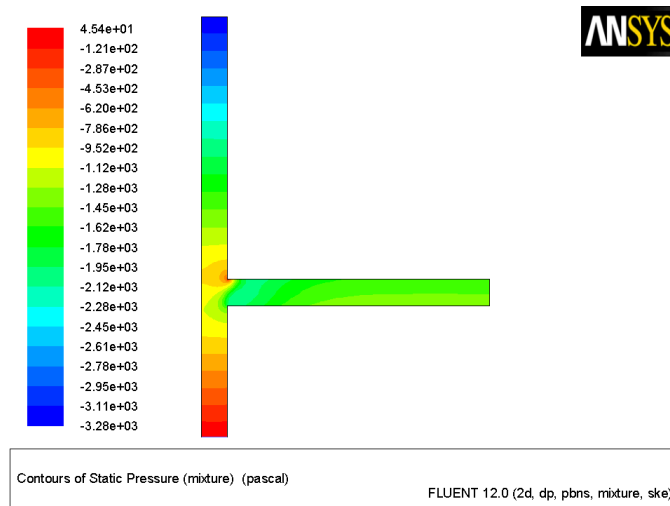


Figure 20.3: Contours of Static Pressure

2. Display contours of velocity magnitude (Figure 20.4).



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

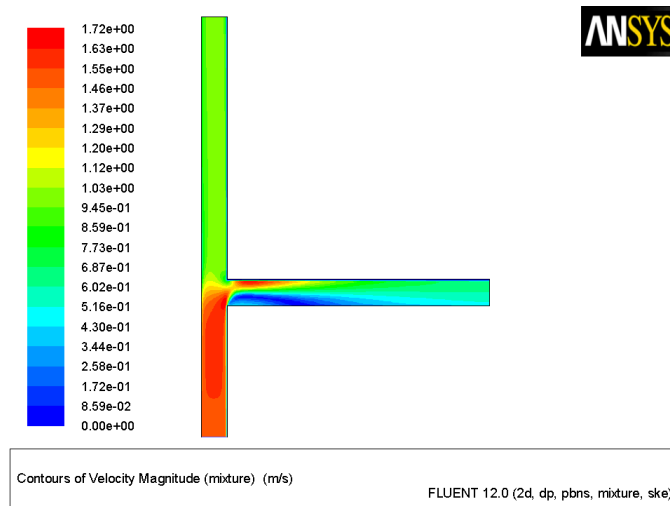


Figure 20.4: Contours of Velocity Magnitude

3. Display the volume fraction of air (Figure 20.5).

◆ **Graphics and Animations** → **Contours** → **Set Up...**

- Select Phases... and Volume fraction from the Contours of drop-down lists.
- Select air from the Phase drop-down list.
- Click Display and close the Contours dialog box.

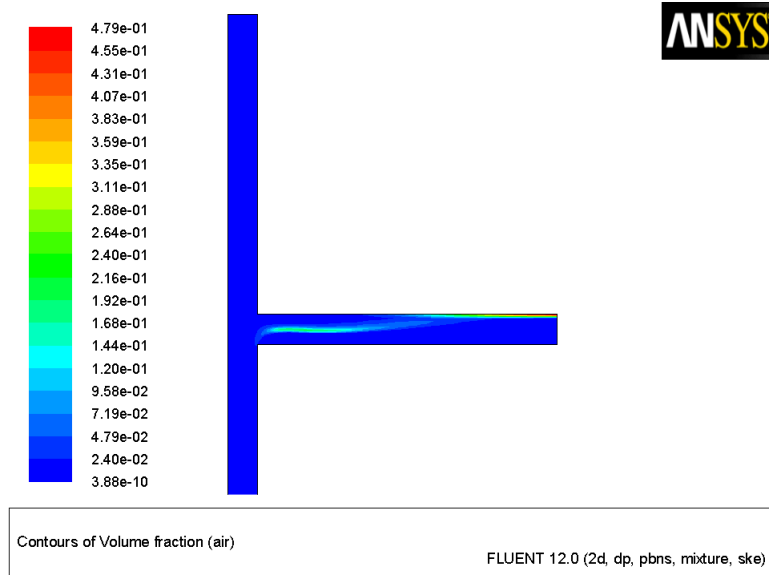


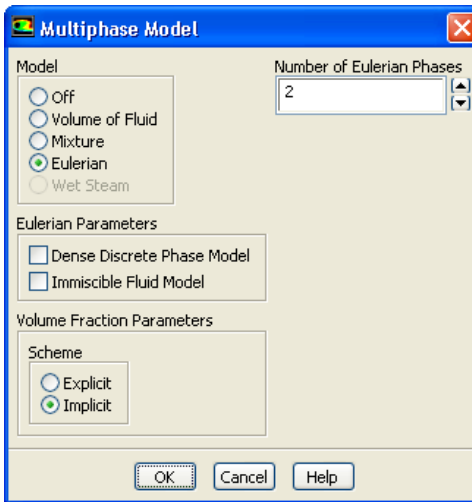
Figure 20.5: Contours of Air Volume Fraction

When gravity acts downwards, it induces stratification in the side arm of the tee junction. In Figure 20.5, 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.

Step 10: Setup and Solution for the Eulerian Model

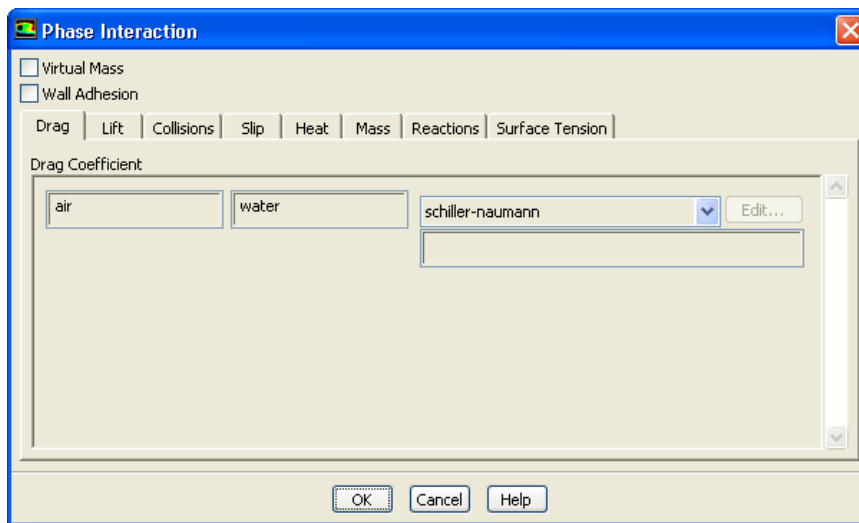
The mixture model is a simplification of the Eulerian model and is valid only when bubble velocity is in the same direction as water velocity. This assumption can be violated in the recirculation pattern. The Eulerian model is expected to make a more realistic prediction in this case. You will use the solution obtained using the mixture model as an initial condition for the calculation using the Eulerian model.

1. Select the Eulerian multiphase model.



- (a) Select Eulerian in the Model list.
- (b) Click OK to close the Multiphase Model dialog box.

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



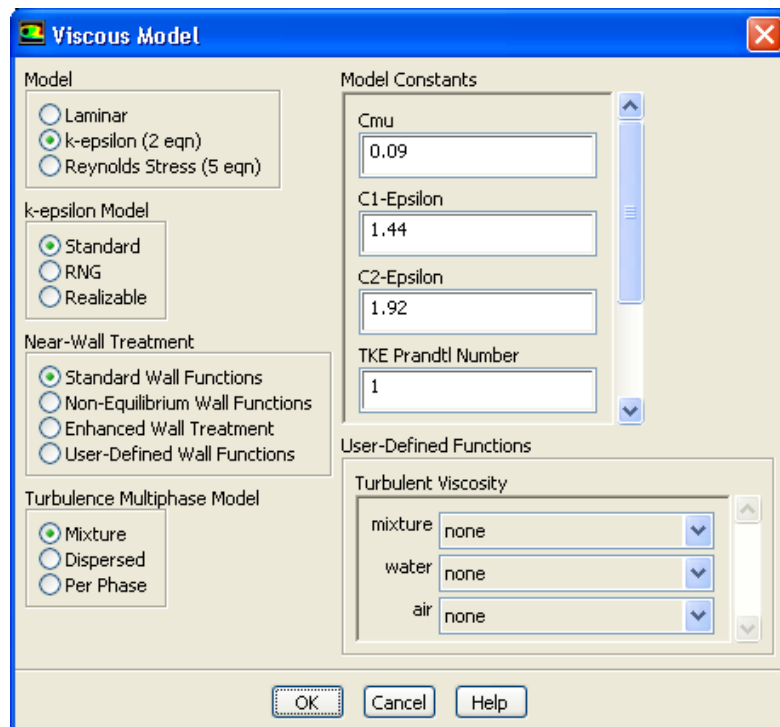
- (a) Retain the default selection of schiller-naumann from the Drag Coefficient drop-down list.
- (b) Click OK to close the Phase Interaction dialog box.

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 dialog box that may be relevant for other applications.

For details on setting up an Eulerian multiphase calculation, see Section 24.2 in the separate User's Guide.

3. Select the multiphase turbulence model.

◆ **Models** → **Viscous** → **Edit...**



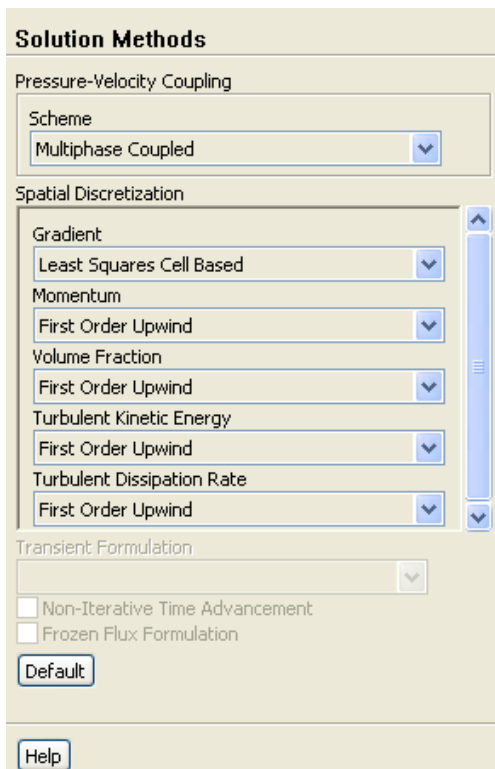
- (a) Retain the default selection of Mixture in the Turbulence Multiphase Model list.
- (b) Click OK to close the Viscous Model dialog box.

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.

For more information on turbulence models for the Eulerian multiphase model, see Chapter 24 in the separate [User's Guide](#).

4. Change the solution parameters.

◆ **Solution Methods**



- (a) Select **Multiphase Coupled** from the Scheme drop-down list.

5. Change the solution controls.

◆ **Solution Controls**

Solution Controls

Courant Number
40

Explicit Relaxation Factors

Momentum 0.5
Pressure 0.5

Under-Relaxation Factors

Body Forces
1

Volume Fraction
0.4

Turbulent Kinetic Energy
0.8

Turbulent Dissipation Rate
0.8

Turbulent Viscosity
1

Default
Equations... Limits... Advanced...
Help

- (a) Enter 40 for Courant Number.
 - (b) Enter 0.5 for both Momentum and Pressure in the Explicit Relaxation Factors group box.
 - (c) Retain the value of 0.4 for Volume Fraction in the Under-Relaxation Factors group box.
6. Continue the solution by requesting 1200 additional iterations.

◆ **Run Calculation**

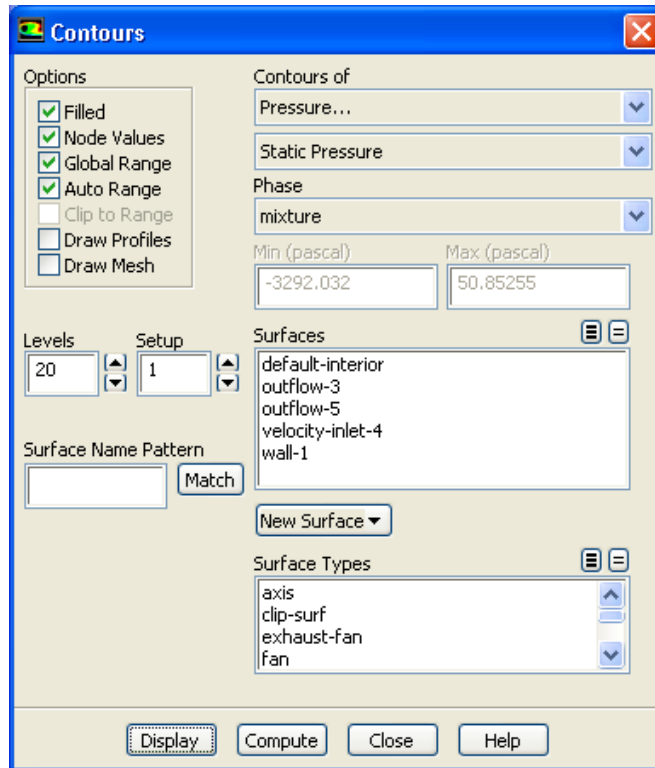
7. Save the case and data files (tee2.cas.gz and tee2.dat.gz).

File → Write → Case & Data...

Step 11: Postprocessing for the Eulerian Model

◆ Graphics and Animations

1. Display the static pressure field in the tee for the mixture (Figure 20.6).



- (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 automatically change 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.

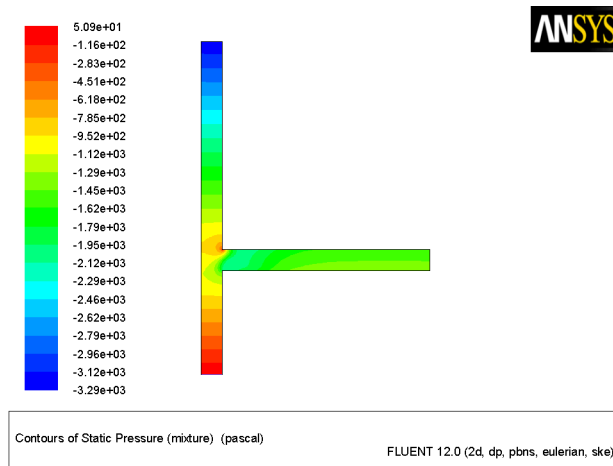


Figure 20.6: Contours of Static Pressure

2. Display contours of velocity magnitude for water (Figure 20.7).



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

- (c) Click Display.

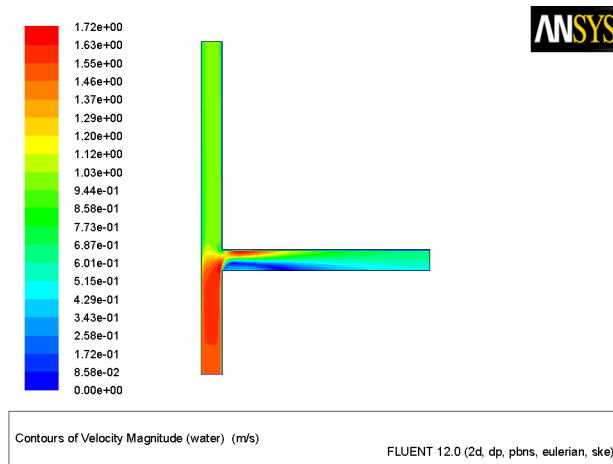


Figure 20.7: Contours of Water Velocity Magnitude

3. Display the volume fraction of air (Figure 20.8).

◆ **Graphics and Animations** → **Contours** → **Set Up...**

- Select Phases... and Volume fraction from the Contours of drop-down lists.
- Select air from the Phase drop-down list.
- Click Display and close the Contours dialog box.

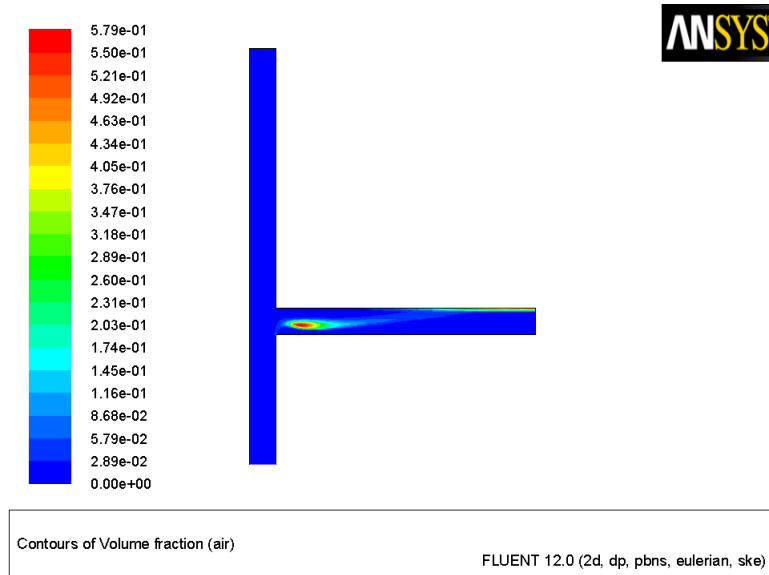


Figure 20.8: Contours of Air Volume Fraction

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. For more information about the mixture and Eulerian models, see Chapter 24 in the separate [User's Guide](#).

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 mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in [Tutorial 1](#).

