

---

## Chapter 22: Using the Mixture and Eulerian Multiphase Models

---

This tutorial is divided into the following sections:

- 22.1. Introduction
- 22.2. Prerequisites
- 22.3. Problem Description
- 22.4. Setup and Solution
- 22.5. Summary
- 22.6. Further Improvements

### 22.1. 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.

### 22.2. Prerequisites

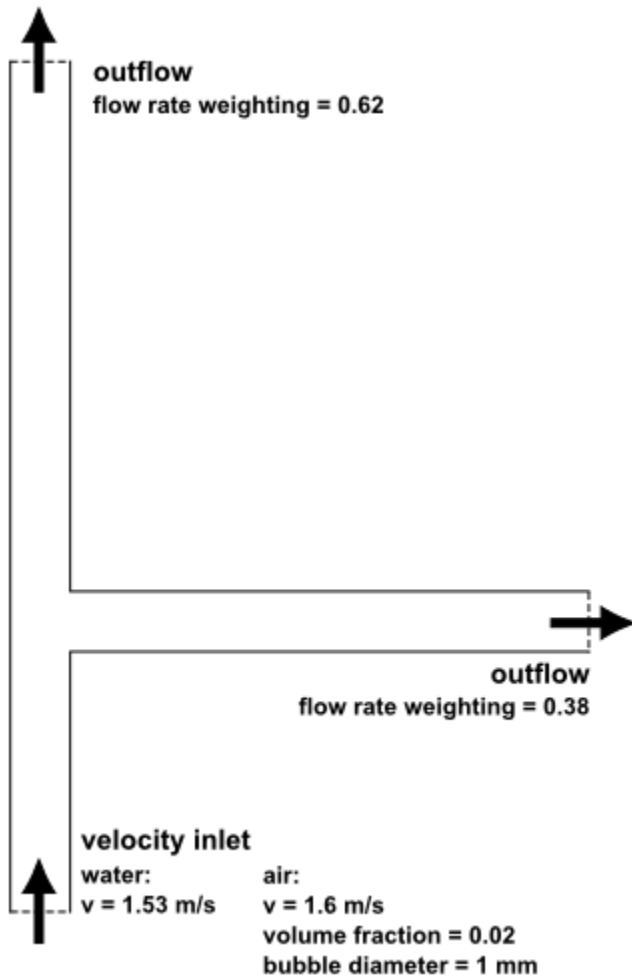
This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- *Introduction to Using ANSYS FLUENT in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow* (p. 1)
- *Parametric Analysis in ANSYS Workbench Using ANSYS FLUENT* (p. 77)
- *Introduction to Using ANSYS FLUENT: Fluid Flow and Heat Transfer in a Mixing Elbow* (p. 131)

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.

### 22.3. 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 22.1](#) (p. 906).

**Figure 22.1 Problem Specification**

## 22.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

- 22.4.1. Preparation
- 22.4.2. Step 1: Mesh
- 22.4.3. Step 2: General Settings
- 22.4.4. Step 3: Models
- 22.4.5. Step 4: Materials
- 22.4.6. Step 5: Phases
- 22.4.7. Step 6: Boundary Conditions
- 22.4.8. Step 7: Operating Conditions
- 22.4.9. Step 8: Solution Using the Mixture Model
- 22.4.10. Step 9: Postprocessing for the Mixture Solution
- 22.4.11. Step 10: Setup and Solution for the Eulerian Model
- 22.4.12. Step 11: Postprocessing for the Eulerian Model

### 22.4.1. Preparation

1. Extract the file `mix_eulerian_multiphase.zip` from the `ANSYS_Fluid_Dynamics_Tutorial_Inputs.zip` archive which is available from the [Customer Portal](#).

**Note**

For detailed instructions on how to obtain the `ANSYS_Fluid_Dynamics_Tutorial_Inputs.zip` file, please refer to [Preparation \(p. 3\)](#) in *Introduction to Using ANSYS FLUENT in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow* (p. 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.

For more information about FLUENT Launcher, see [Starting ANSYS FLUENT Using FLUENT Launcher](#) in the User's Guide.

4. Enable **Double Precision**.

**Note**

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

**Note**

The double precision solver is recommended for modeling multiphase flow simulations.

**22.4.2. 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.*

**22.4.3. Step 2: General Settings** **General**

1. Check the mesh.

 **General** → **Check**

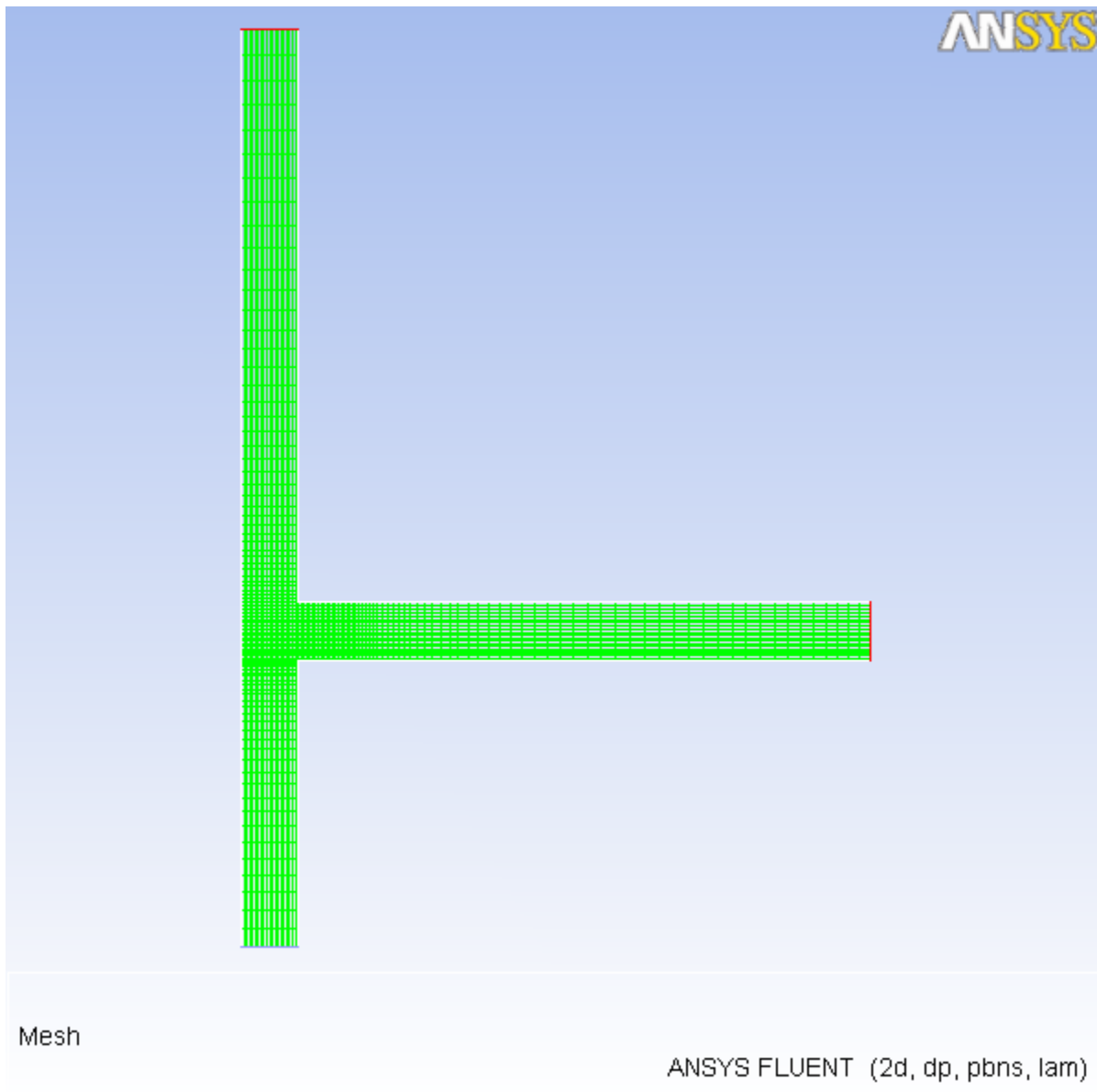
*ANSYS FLUENT will perform various checks on the mesh and will report the progress in the console. Ensure that the reported minimum volume is a positive number.*

2. Examine the mesh ([Figure 22.2 \(p. 908\)](#)).

### Extra

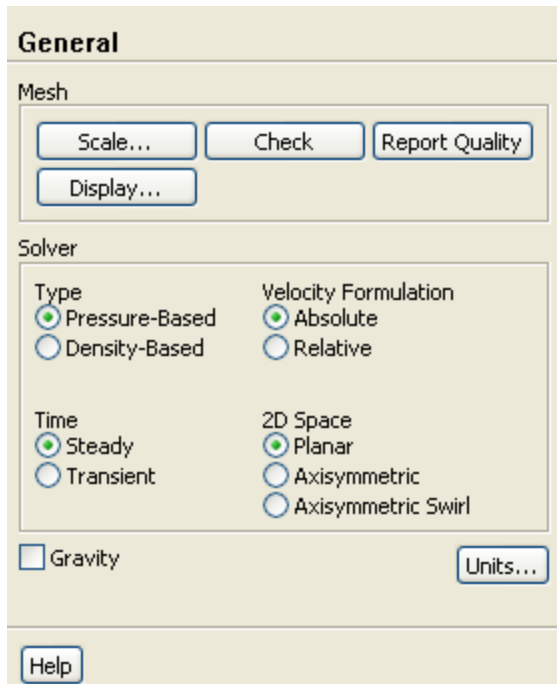
You can use the right mouse button to probe for mesh information in the graphics window. If you click the right mouse button on any node in the mesh, information will be displayed in the ANSYS FLUENT console about the associated zone, including the name of the zone. This feature is especially useful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

**Figure 22.2 Mesh Display**



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

### ◆ General



## 22.4.4. Step 3: Models

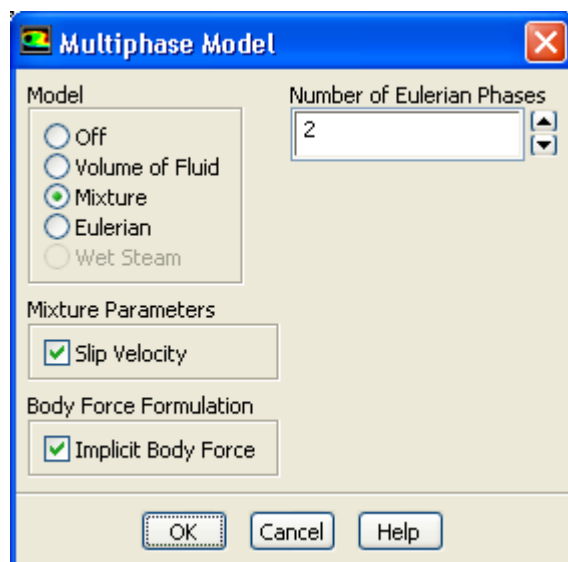
### Models

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.

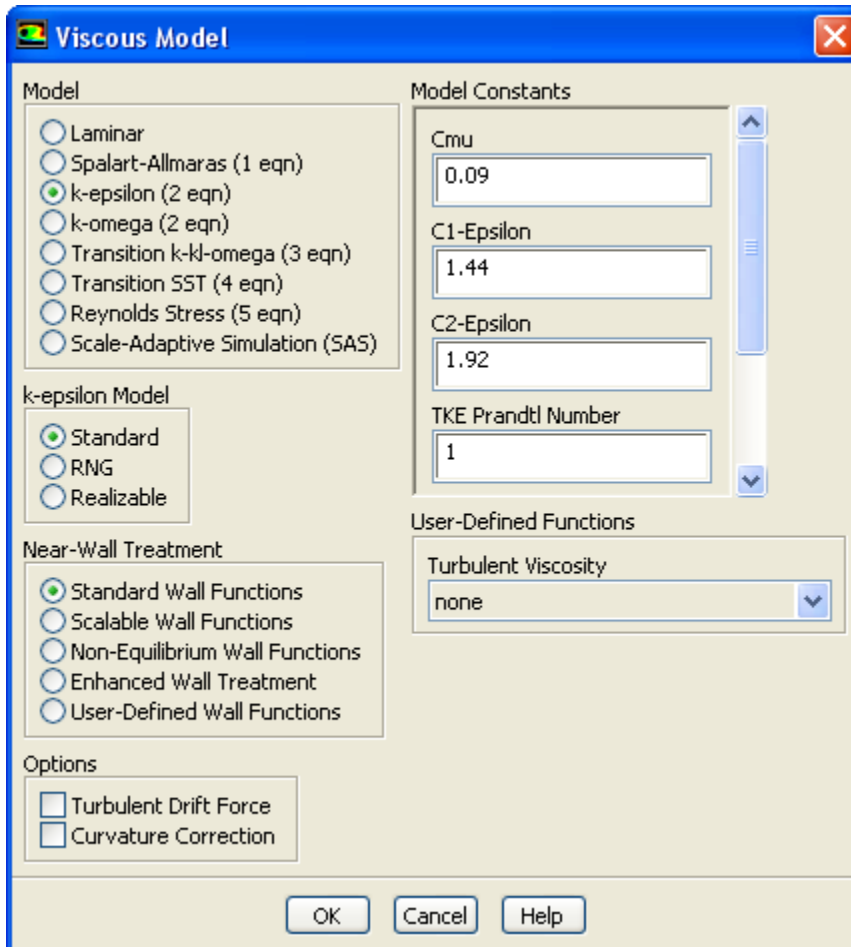
- c. Enable **Implicit Body Force** in the **Body Force Formulation** group box.

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  $k$ - $\epsilon$  turbulence model with standard wall functions.

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



- a. Select **k-epsilon** in the **Model** list.  
 b. Retain the default selection of **Standard** in the **k-epsilon Model** list.

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

- c. Select **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.

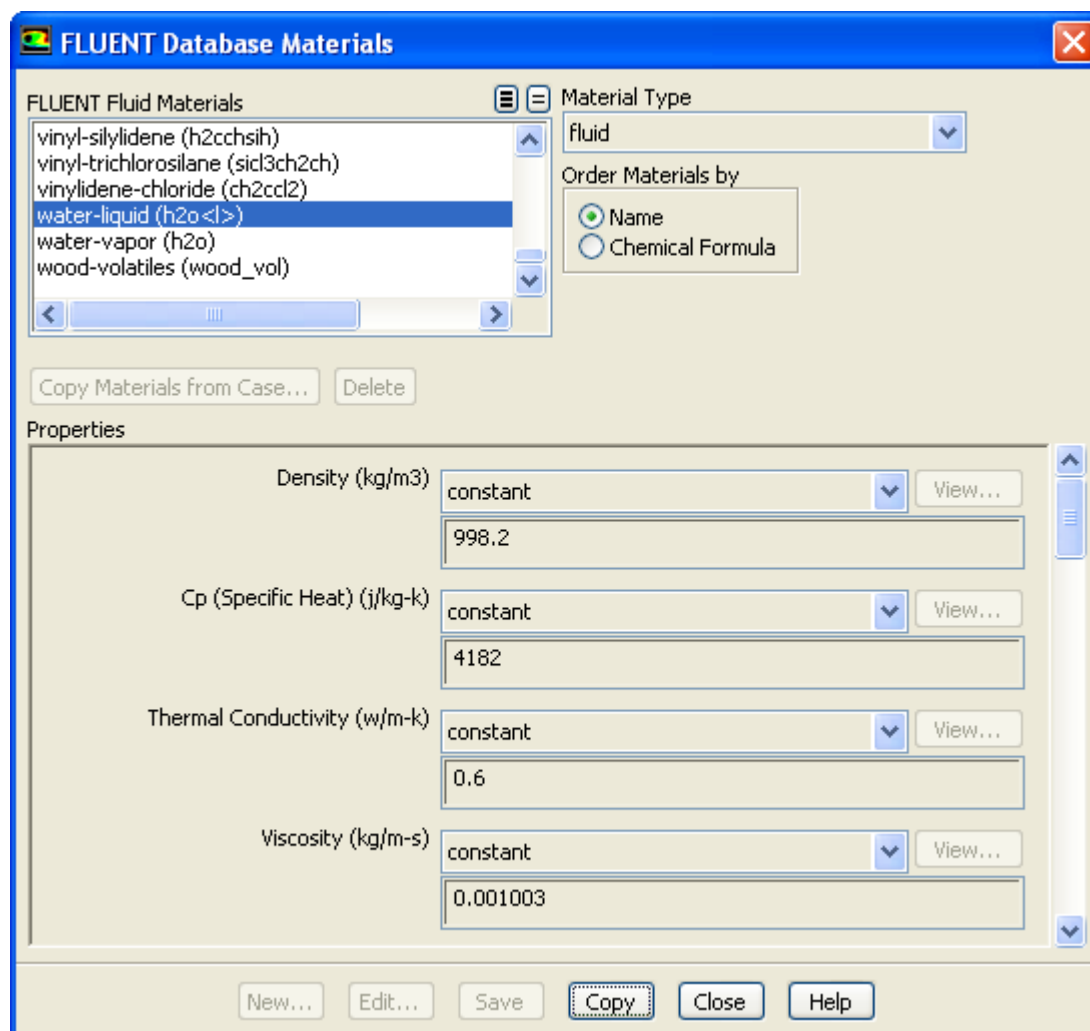
## 22.4.5. Step 4: Materials

### 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.

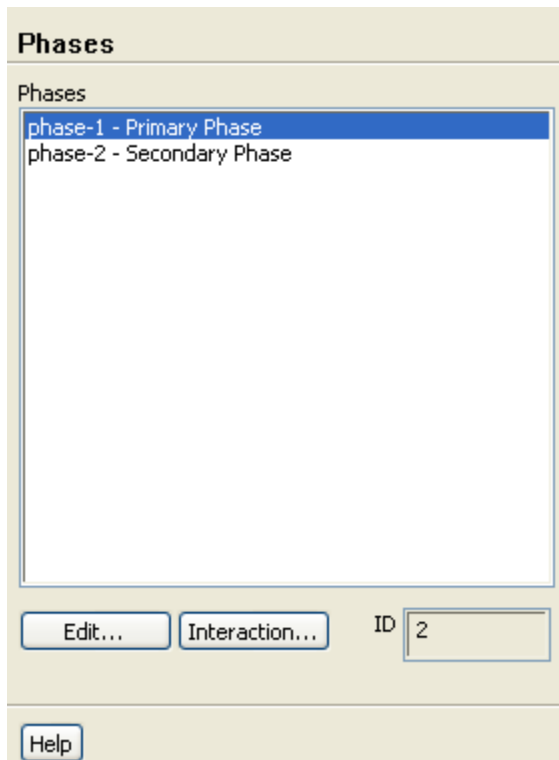


- Select **water-liquid (h2o < l >)** from the **FLUENT Fluid Materials** selection list.  
*Scroll down the list to find **water-liquid (h2o < l >)**.*
  - Click **Copy** to copy the properties for liquid water to your model.
  - Close the **FLUENT Database Materials** dialog box.
- Close the **Create/Edit Materials** dialog box.

## 22.4.6. Step 5: Phases

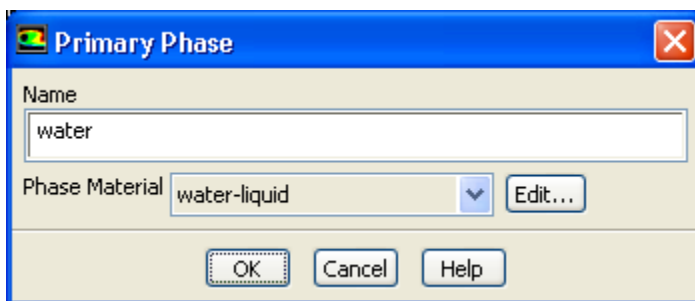
### Phases

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.

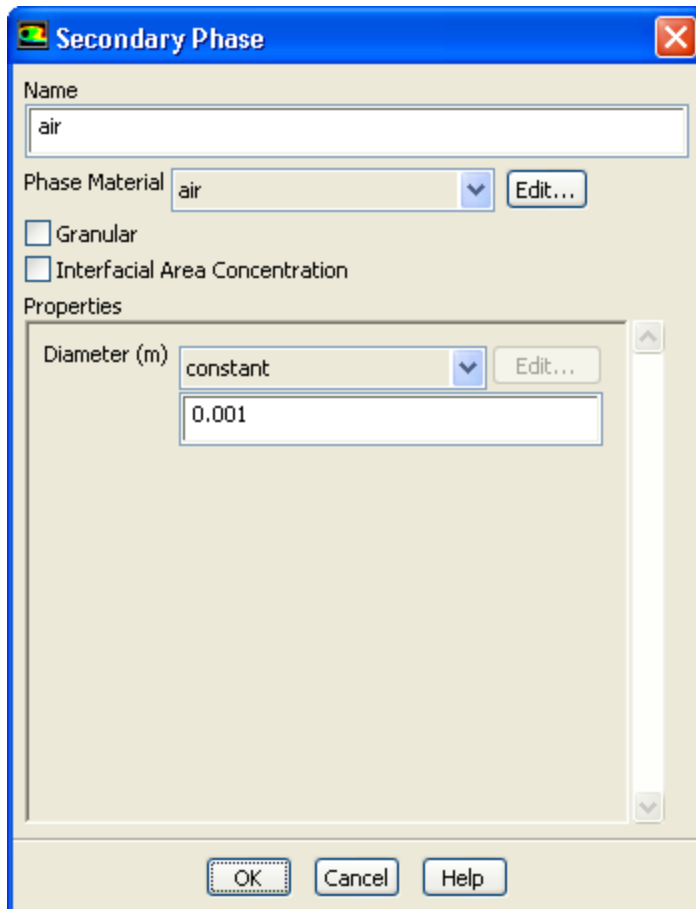
Phases → phase-1 → Edit...



- 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.
2. Specify air as the secondary phase.

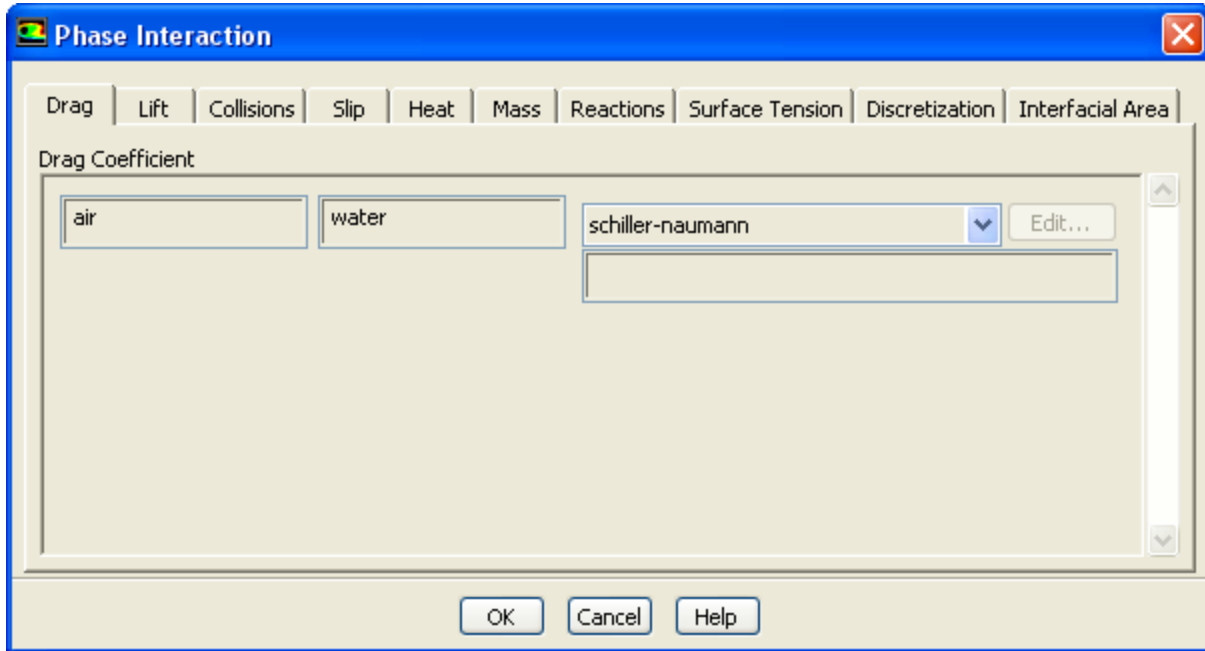
Phases → phase-2 → Edit...





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

◀ **Phases** → **Interaction...**



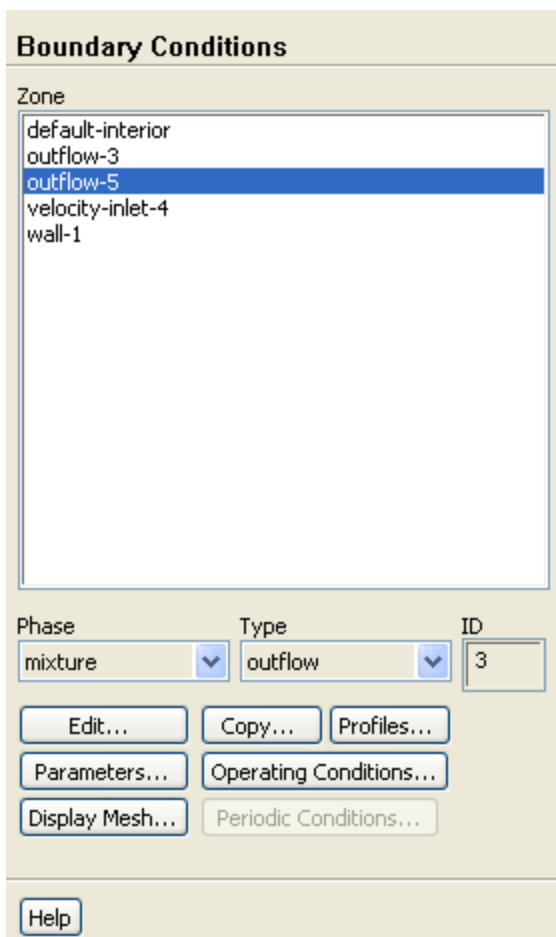
- a. Retain the default selection of **schiller-naumann** from the **Drag Coefficient** drop-down list.

*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.

### 22.4.7. Step 6: Boundary Conditions

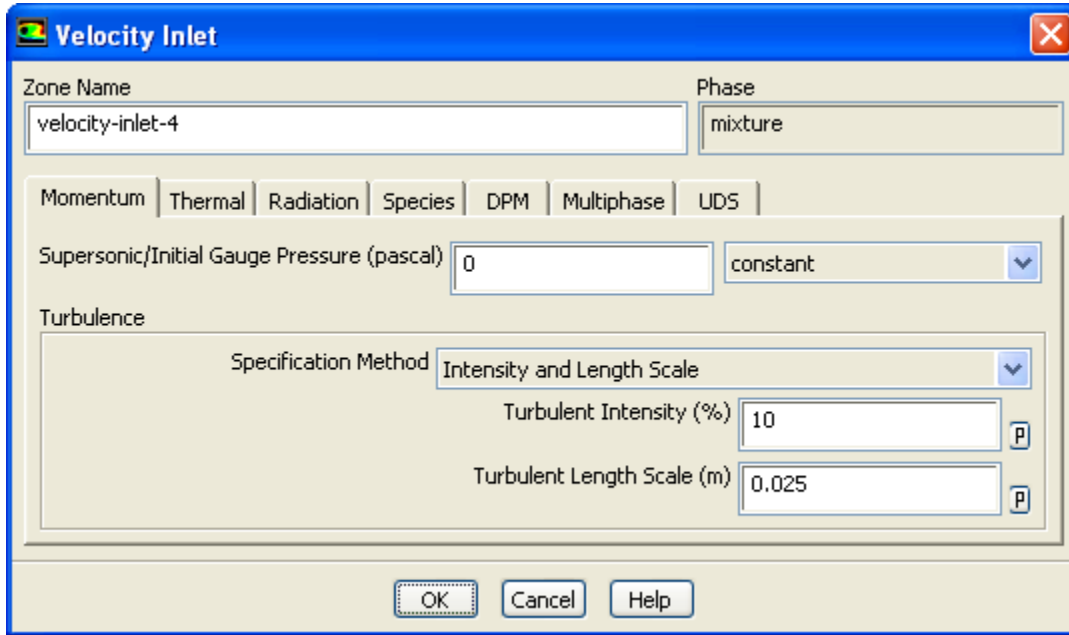
#### Boundary Conditions



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.

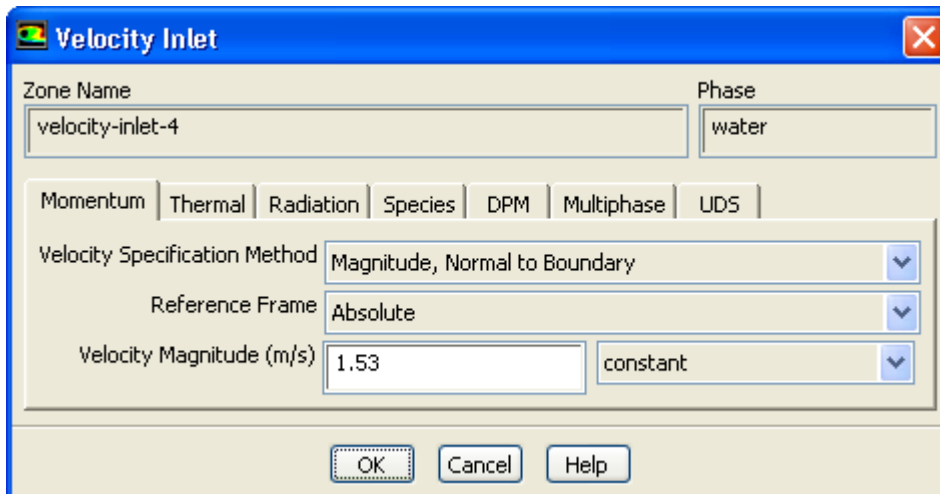
 **Boundary Conditions** →  **velocity-inlet-4** → **Edit...**



- 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).

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

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

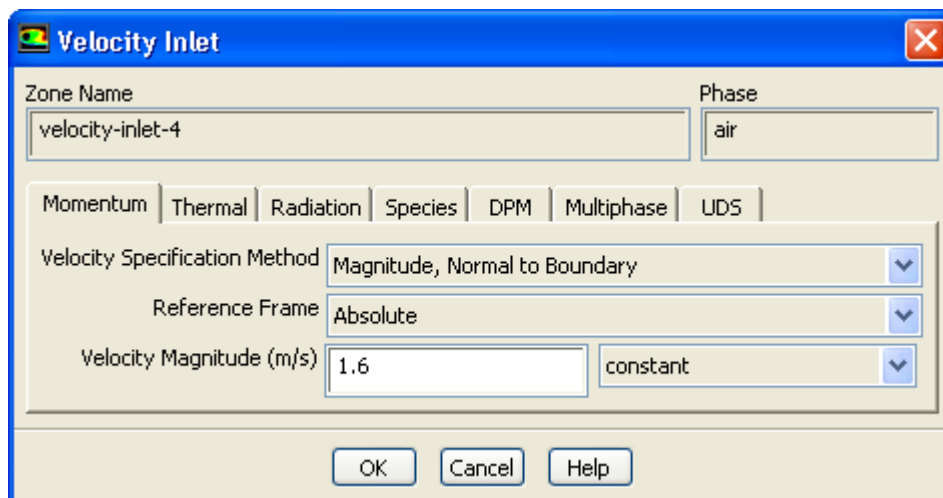


- i. Retain the default selection of **Magnitude, Normal to Boundary** from the **Velocity Specification Method** drop-down list.
- ii. Retain the default selection of **Absolute** from the **Reference Frame** drop-down list.
- iii. Enter 1.53 m/s for **Velocity Magnitude**.

- 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**

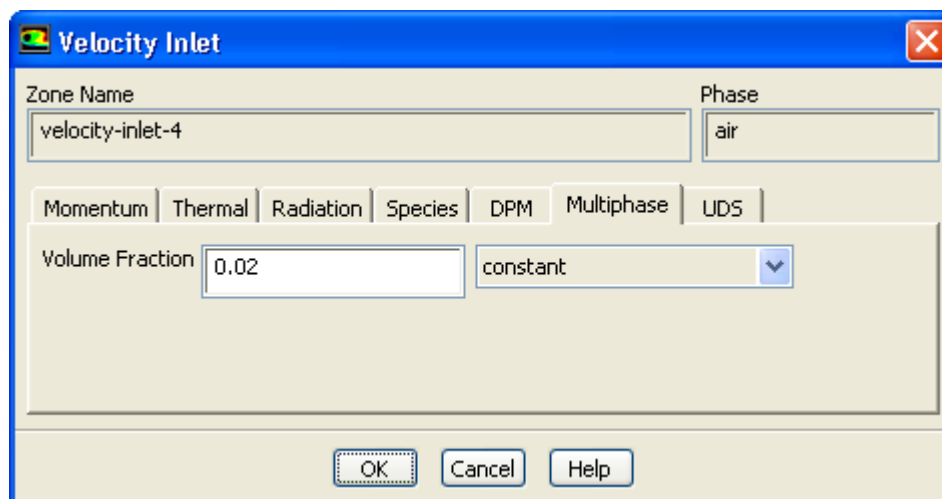
- a. Select **air** from the **Phase** drop-down list.
- b. Click **Edit...** to open the **Velocity Inlet** dialog box.



- 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  $1.6 \text{ m/s}$  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  $1.6 - 1.53 = 0.07 \text{ m/s}$ .*

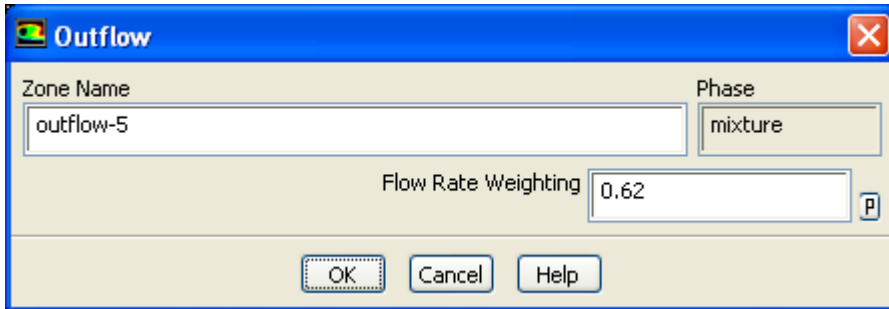
- iv. Click the **Multiphase** tab and enter  $0.02$  for **Volume Fraction**.




- v. Click **OK** to close the **Velocity Inlet** dialog box.
4. Set the boundary conditions at **outflow-5** for the mixture.

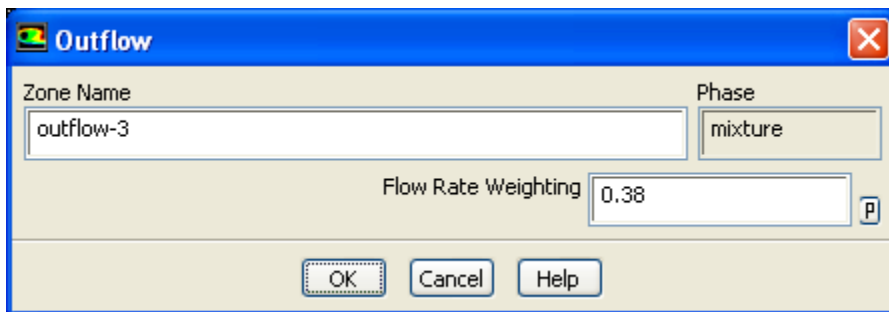
◆ **Boundary Conditions** →  **outflow-5**

- 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...**



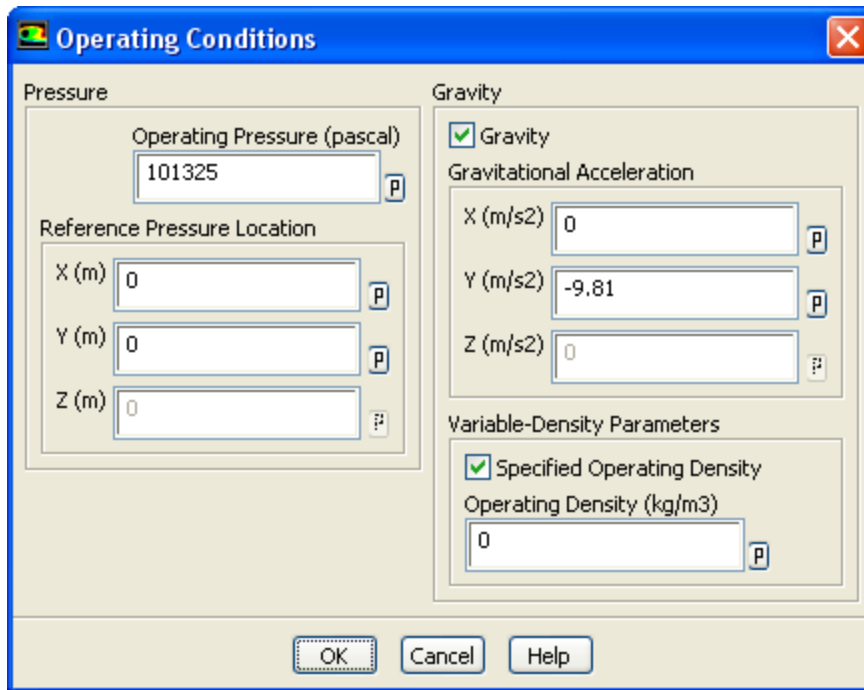
- a. Enter 0.38 for **Flow Rate Weighting**.
- b. Click **OK** to close the **Outflow** dialog box.

## 22.4.8. Step 7: Operating Conditions

◆ **Boundary Conditions**

- 1. Set the gravitational acceleration.

◆ **Boundary Conditions** → **Operating Conditions...**



- a. Enable **Gravity**.

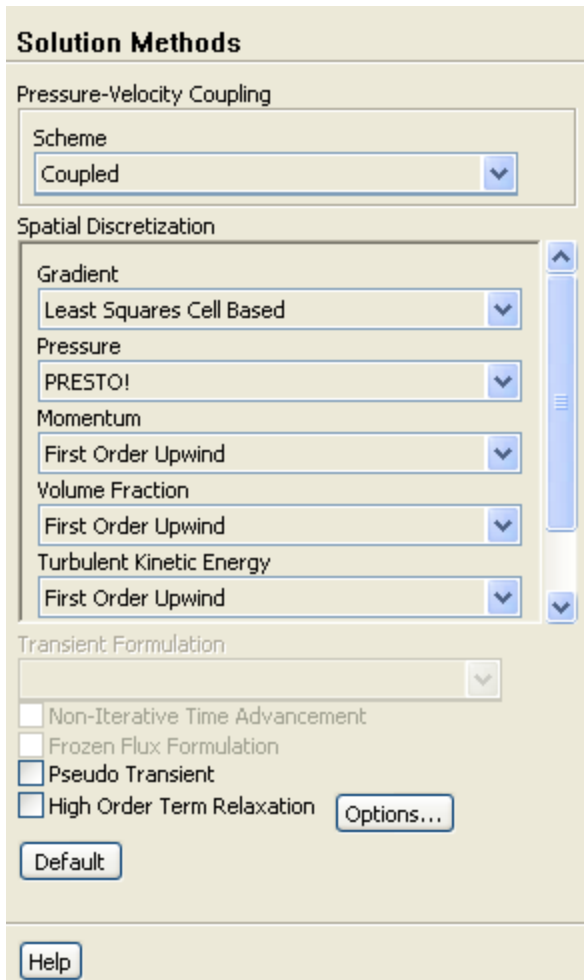
*The **Operating Conditions** dialog box will expand to show additional inputs.*

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

## 22.4.9. Step 8: Solution Using the Mixture Model

1. Set the solution parameters.

### Solution Methods



- 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 **Flow Courant Number**.
- b. Enter 0.5 for both **Momentum** and **Pressure** in the **Explicit Relaxation Factors** group box.



**Solution Controls**

Flow 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

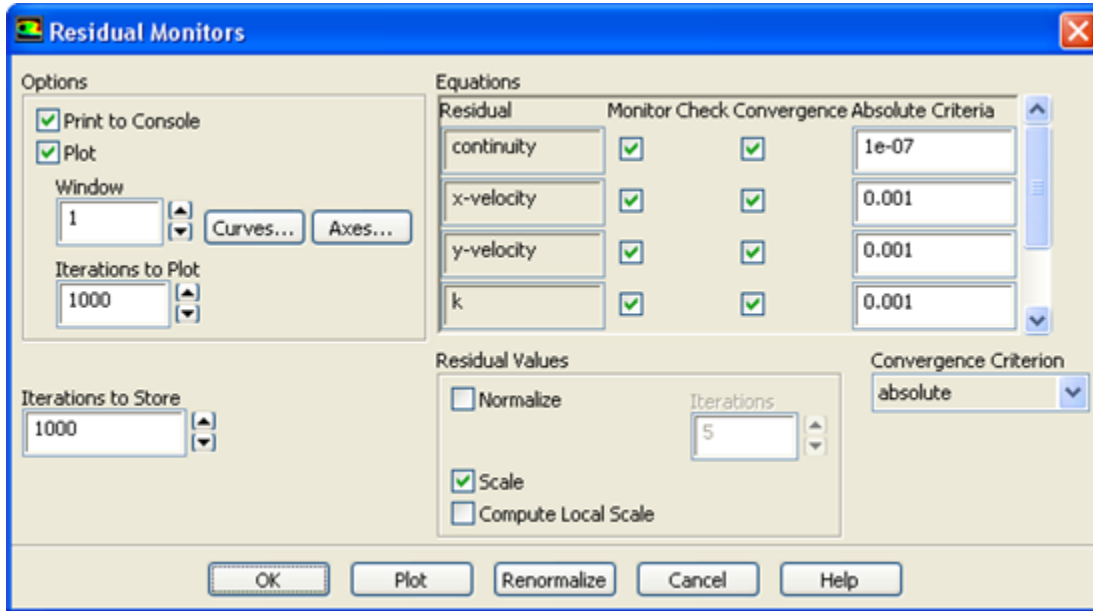
Default

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

Help

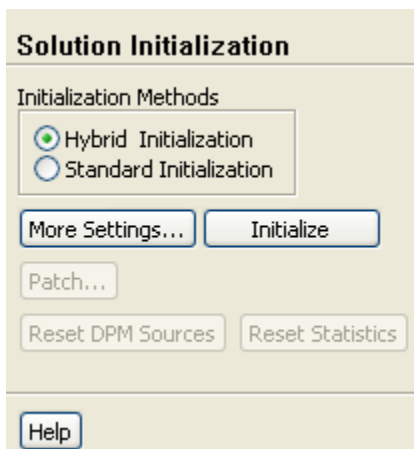
- c. Enter 0.4 for both **Slip Velocity** and **Volume Fraction** in the **Under-Relaxation Factors** group box.
3. Enable the plotting of residuals during the calculation.

 **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. Select **Hybrid Initialization** from the **Initialization Methods** group box.
- b. Click **Initialize**.

### Note

For flows in complex topologies, hybrid initialization will provide better initial velocity and pressure fields than standard initialization. This in general will help in improving the convergence behavior of the solver.

5. Save the case file (tee.cas.gz).

**File** → **Write** → **Case...**

- Start the calculation by requesting 400 iterations.

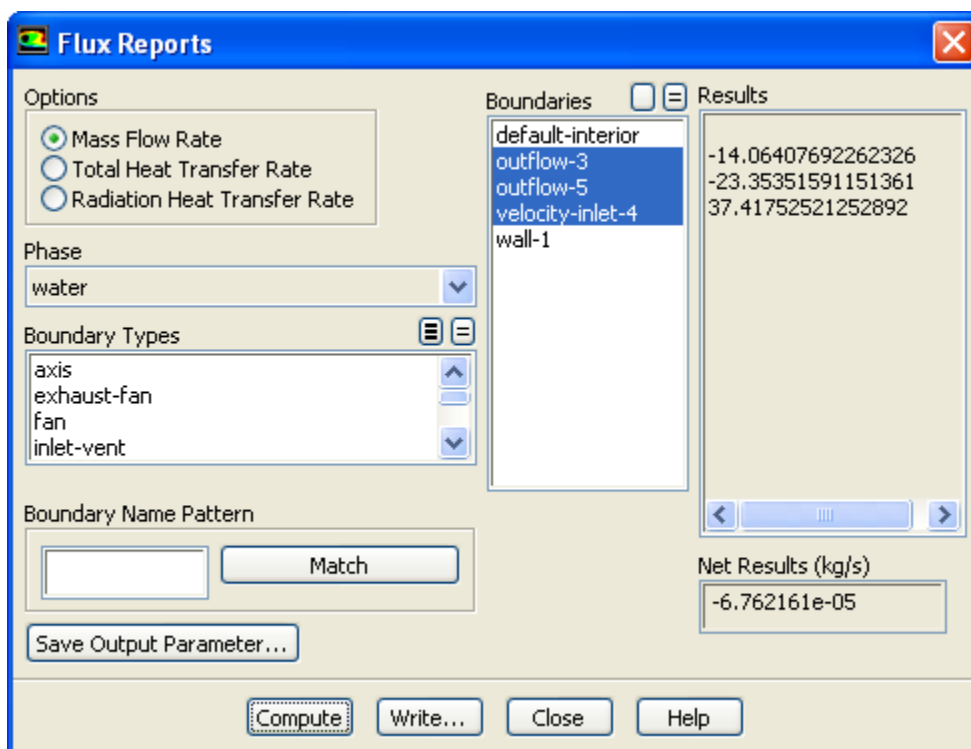
### Run Calculation

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

### File → Write → Case & Data...

- Check the total mass flow rate for each phase.

### Reports → Fluxes → Set Up...



- Retain the default selection of **Mass Flow Rate** in the **Options** list.
- Select **water** from the **Phase** drop-down list.
- Select **outflow-3**, **outflow-5**, and **velocity-inlet-4** from the **Boundaries** selection list.
- Click **Compute**.

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

- 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.*

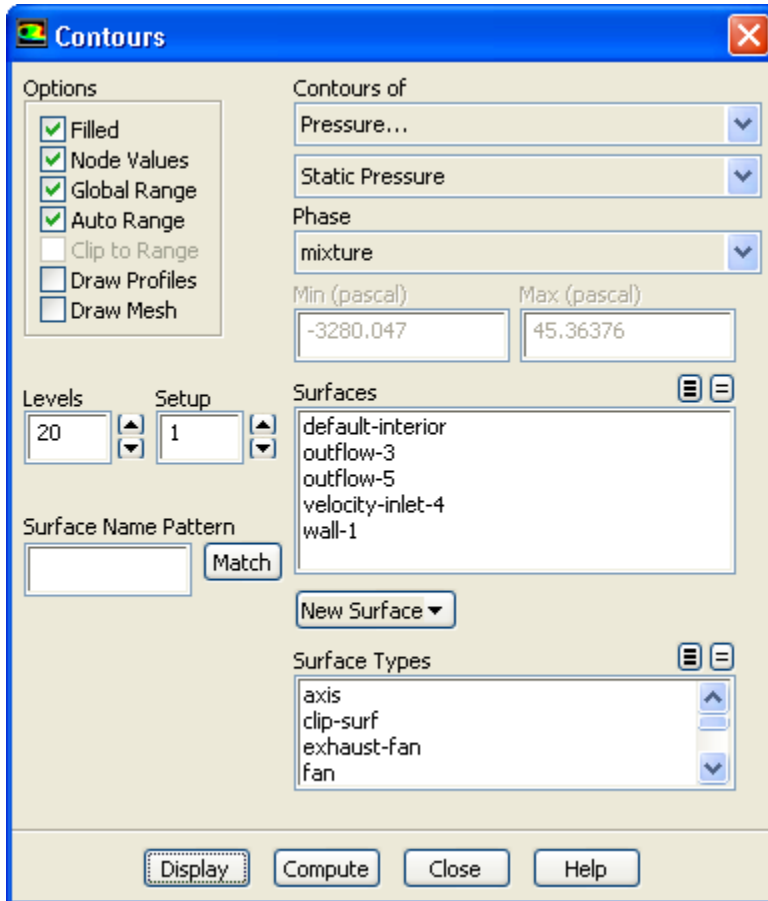
- Close the **Flux Reports** dialog box.

## 22.4.10. Step 9: Postprocessing for the Mixture Solution

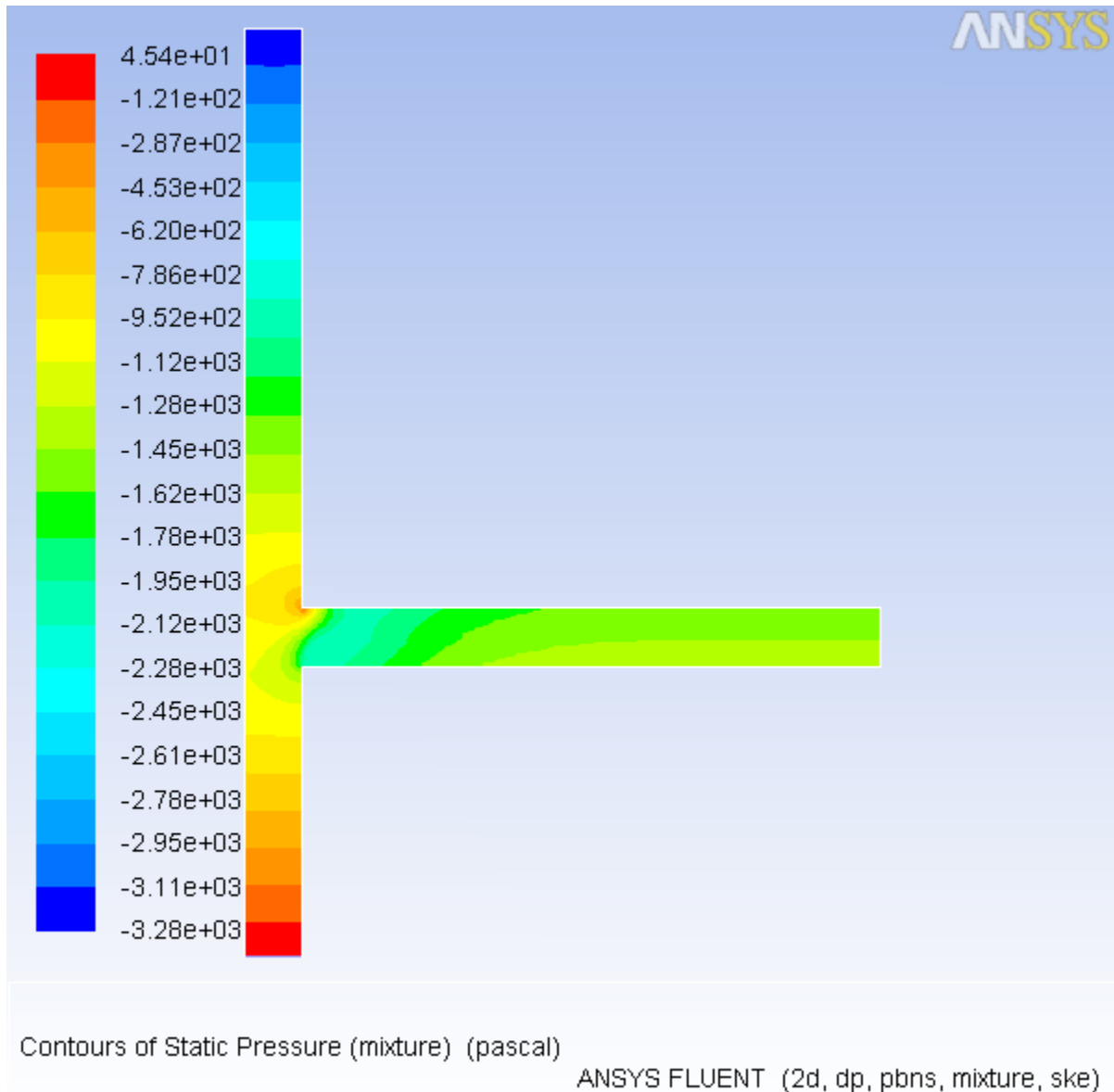
### Graphics and Animations

- Display the static pressure field in the tee (*Figure 22.3* (p. 925)).


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



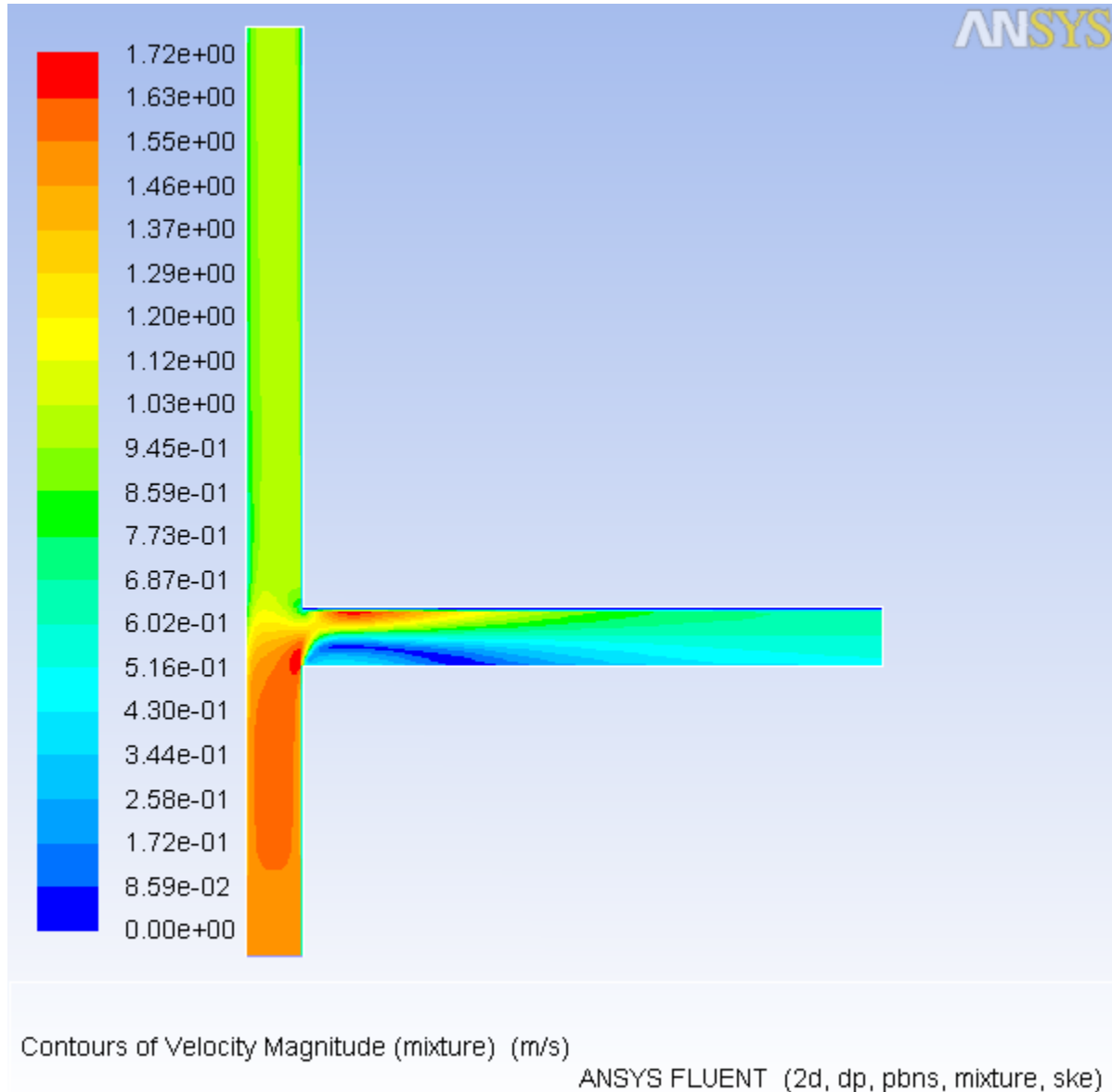
- Enable **Filled** in the **Options** group box.
- Retain the default selection of **Pressure...** and **Static Pressure** from the **Contours of** drop-down lists.
- Click **Display**.

**Figure 22.3 Contours of Static Pressure**

2. Display contours of velocity magnitude (*Figure 22.4* (p. 926)).

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

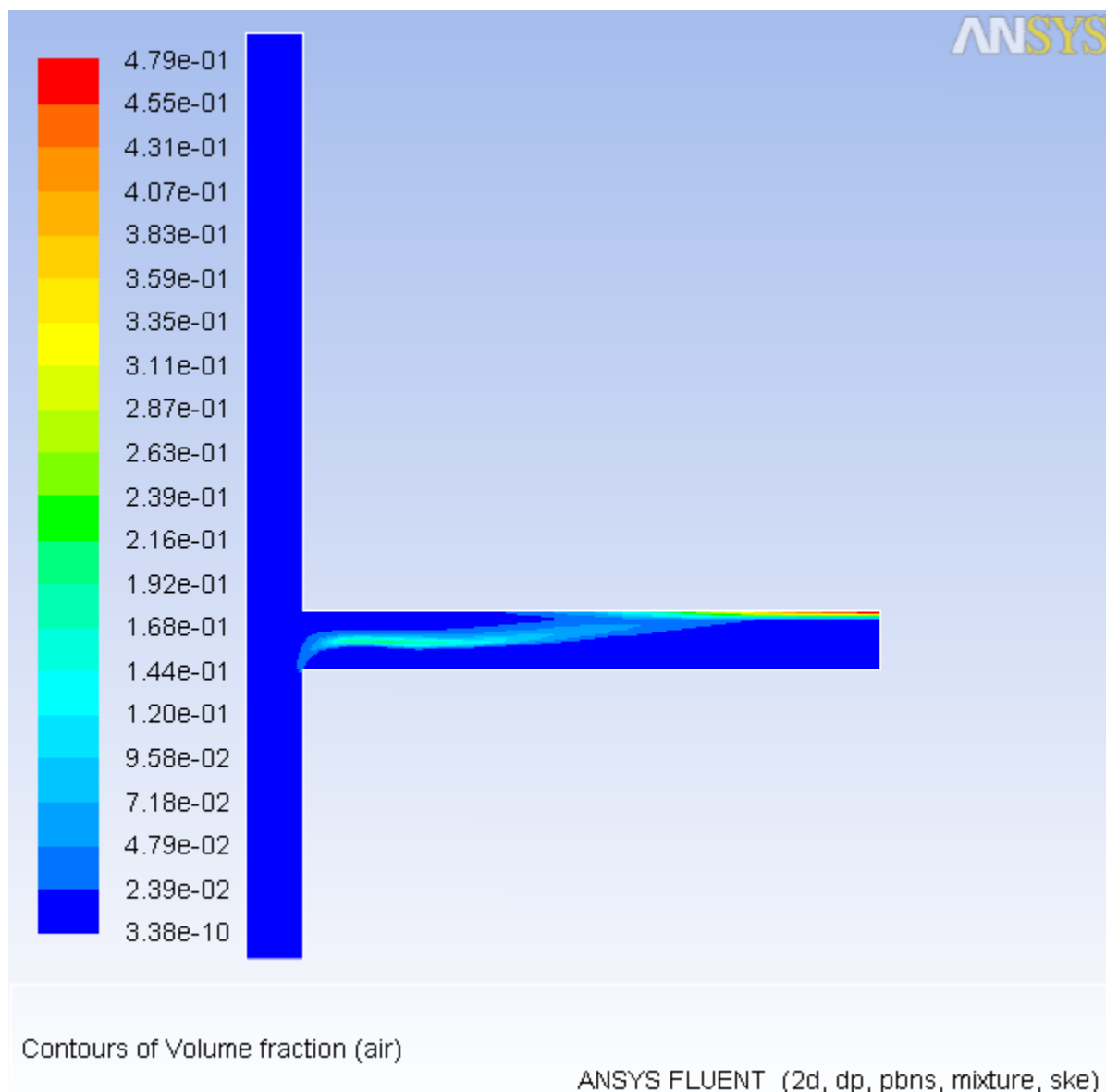
- a. Select **Velocity...** and **Velocity Magnitude** from the **Contours of** drop-down lists.
- b. Click **Display**.

**Figure 22.4 Contours of Velocity Magnitude**

3. Display the volume fraction of air (*Figure 22.5* (p. 927)).

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

- 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** dialog box.

**Figure 22.5 Contours of Air Volume Fraction**

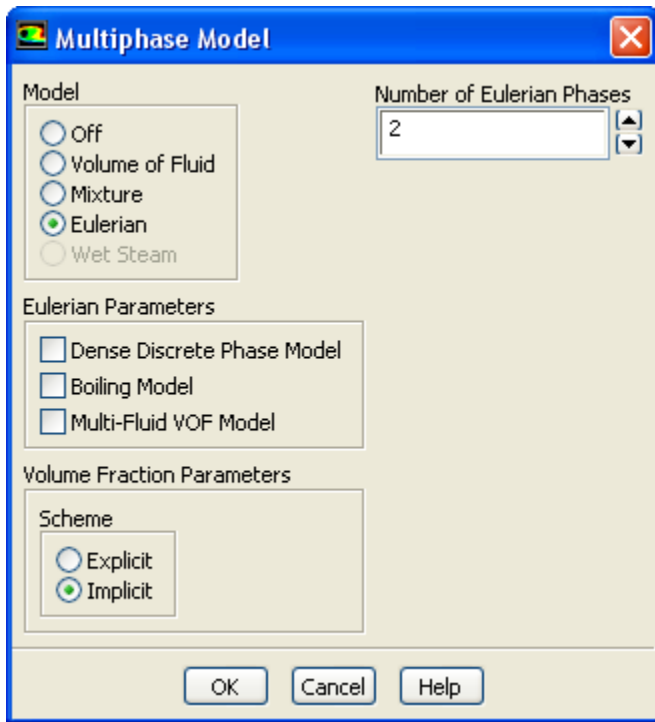
When gravity acts downwards, it induces stratification in the side arm of the tee junction. In [Figure 22.5 \(p. 927\)](#), you can see that the gas (air) tends to concentrate on the upper part of the side arm. In this case, gravity acts against inertia that tends to concentrate gas on the low pressure side, thereby creating gas pockets. In the vertical arm, both the gas and the water have velocities in the same direction, and therefore there is no 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.

### 22.4.11. 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 inertia can be neglected. 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.

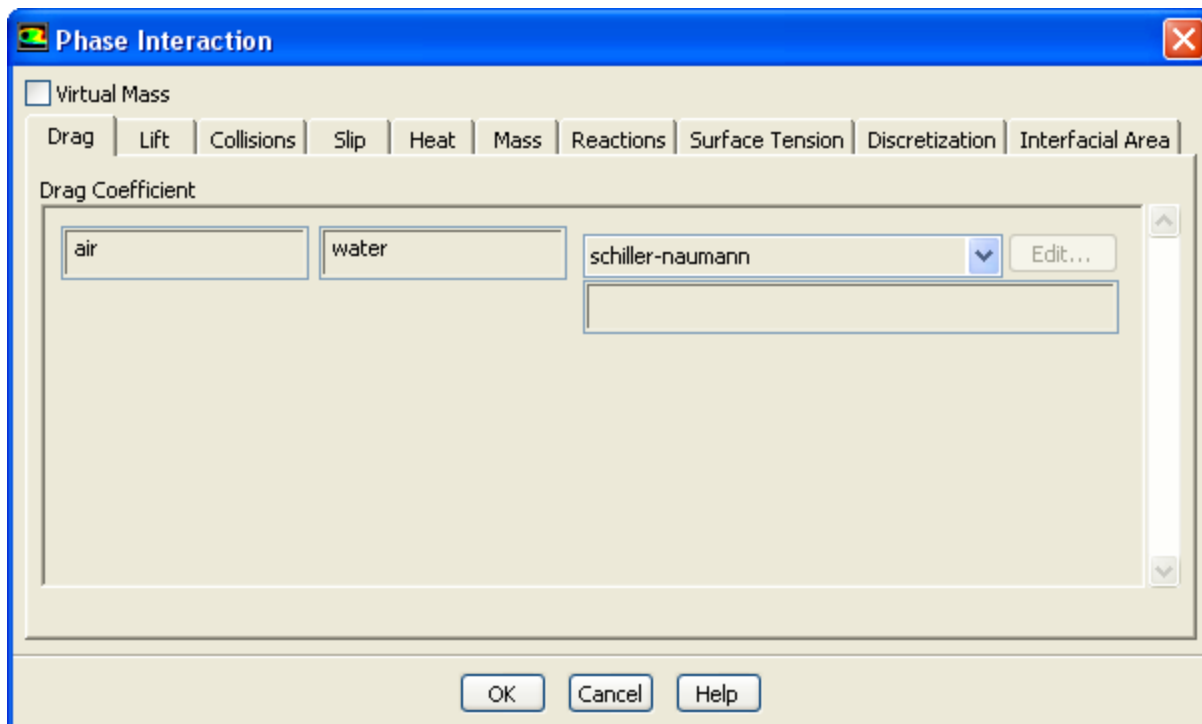
 **Models** →  **Multiphase** → **Edit...**



- 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.

 **Phases** → **Interaction...**





- 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.

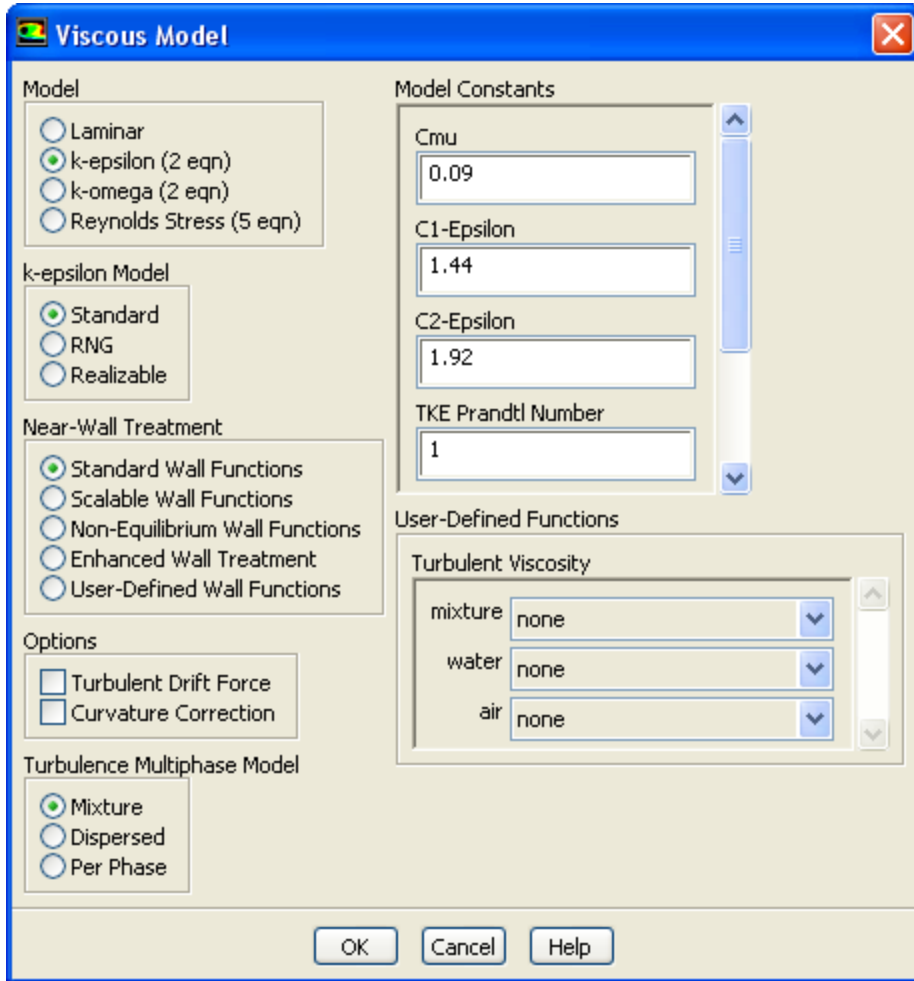
### Note

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 [Steps for Using a Multiphase Model](#) in the 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 "[Modeling Multiphase Flows](#)" in the User's Guide.

4. Change the solution parameters.

## Solution Methods

**Solution Methods**

Pressure-Velocity Coupling

Scheme  
Coupled   
 Coupled with Volume Fractions

Spatial Discretization

Gradient  
Least Squares Cell Based

Momentum  
First Order Upwind

Volume Fraction  
First Order Upwind

Turbulent Kinetic Energy  
First Order Upwind

Turbulent Dissipation Rate  
First Order Upwind

Transient Formulation

Non-Iterative Time Advancement  
 Frozen Flux Formulation  
 Pseudo Transient  
 High Order Term Relaxation

- a. Select **Coupled** from the **Scheme** drop-down list.
5. Change the solution controls.

### Solution Controls

**Solution Controls**

Flow 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 **Flow 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).

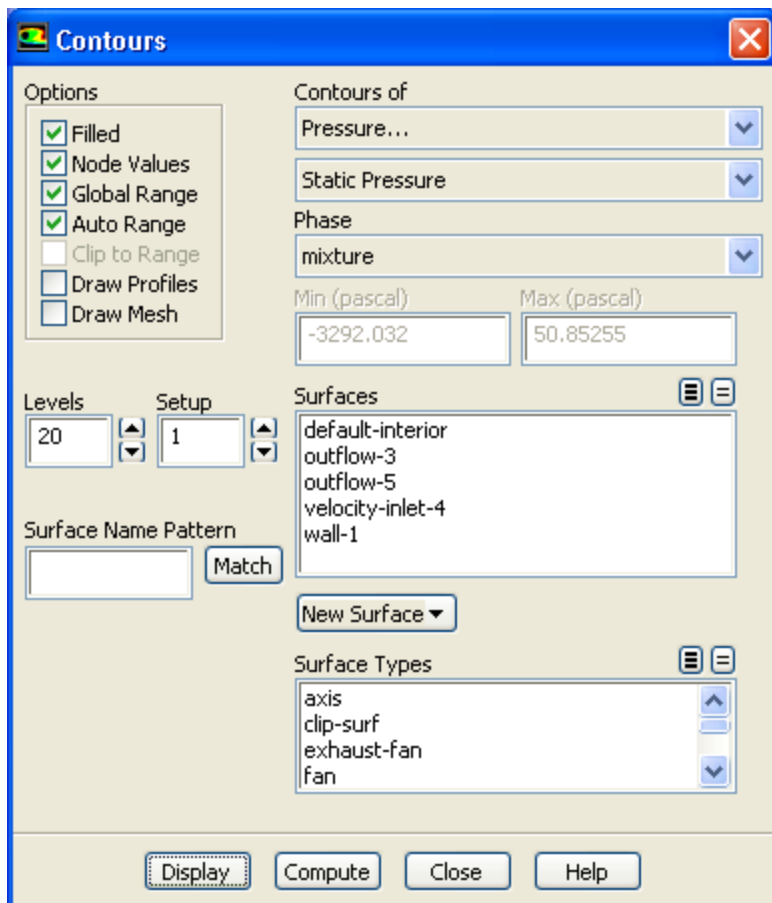
**File** → **Write** → **Case & Data...**

## 22.4.12. Step 11: Postprocessing for the Eulerian Model

### ◆ Graphics and Animations

1. Display the static pressure field in the tee for the mixture (*Figure 22.6* (p. 934)).

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



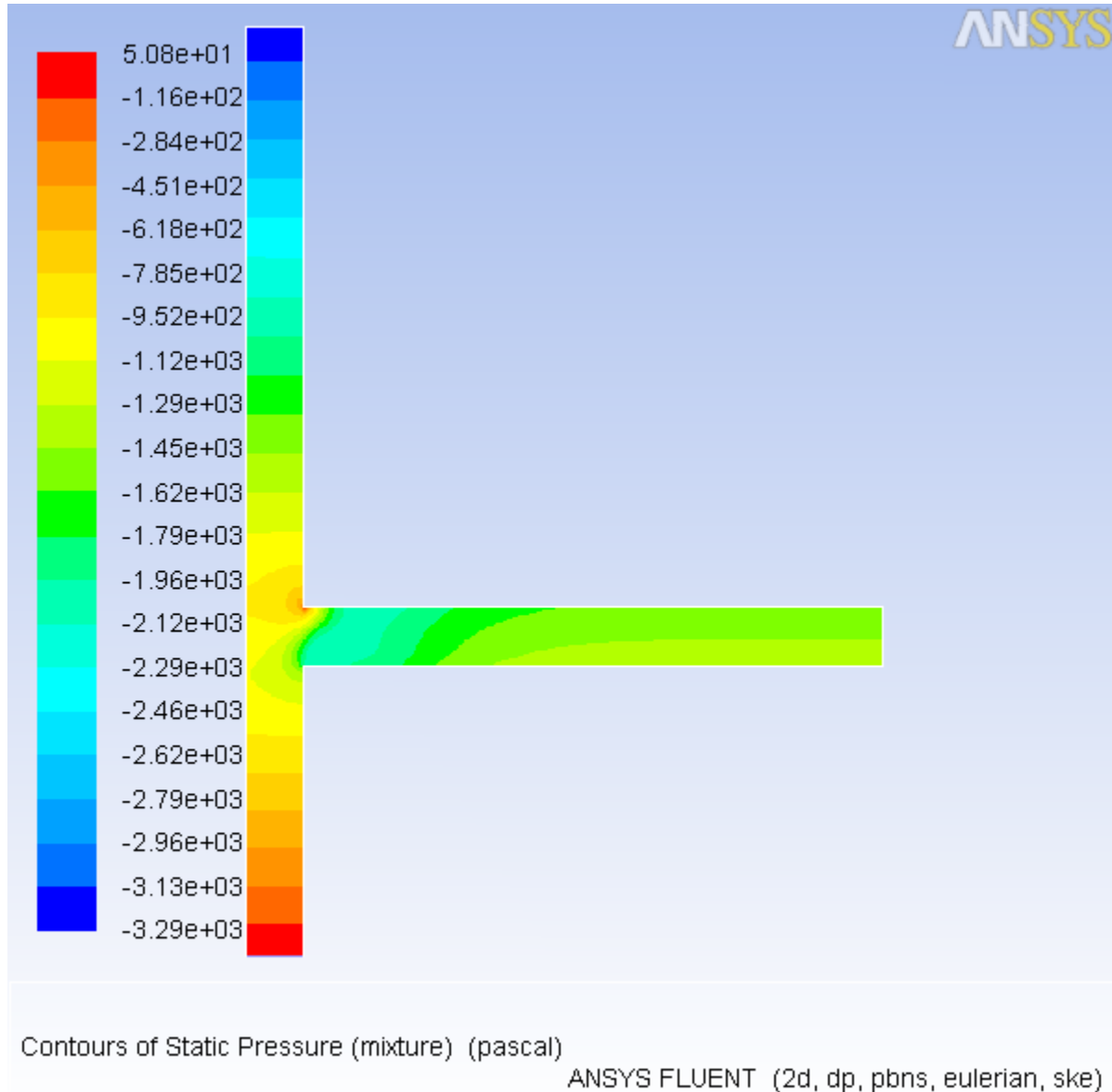
- a. Select **Pressure...** from the **Contours of** drop-down list.

*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 22.6 Contours of Static Pressure**

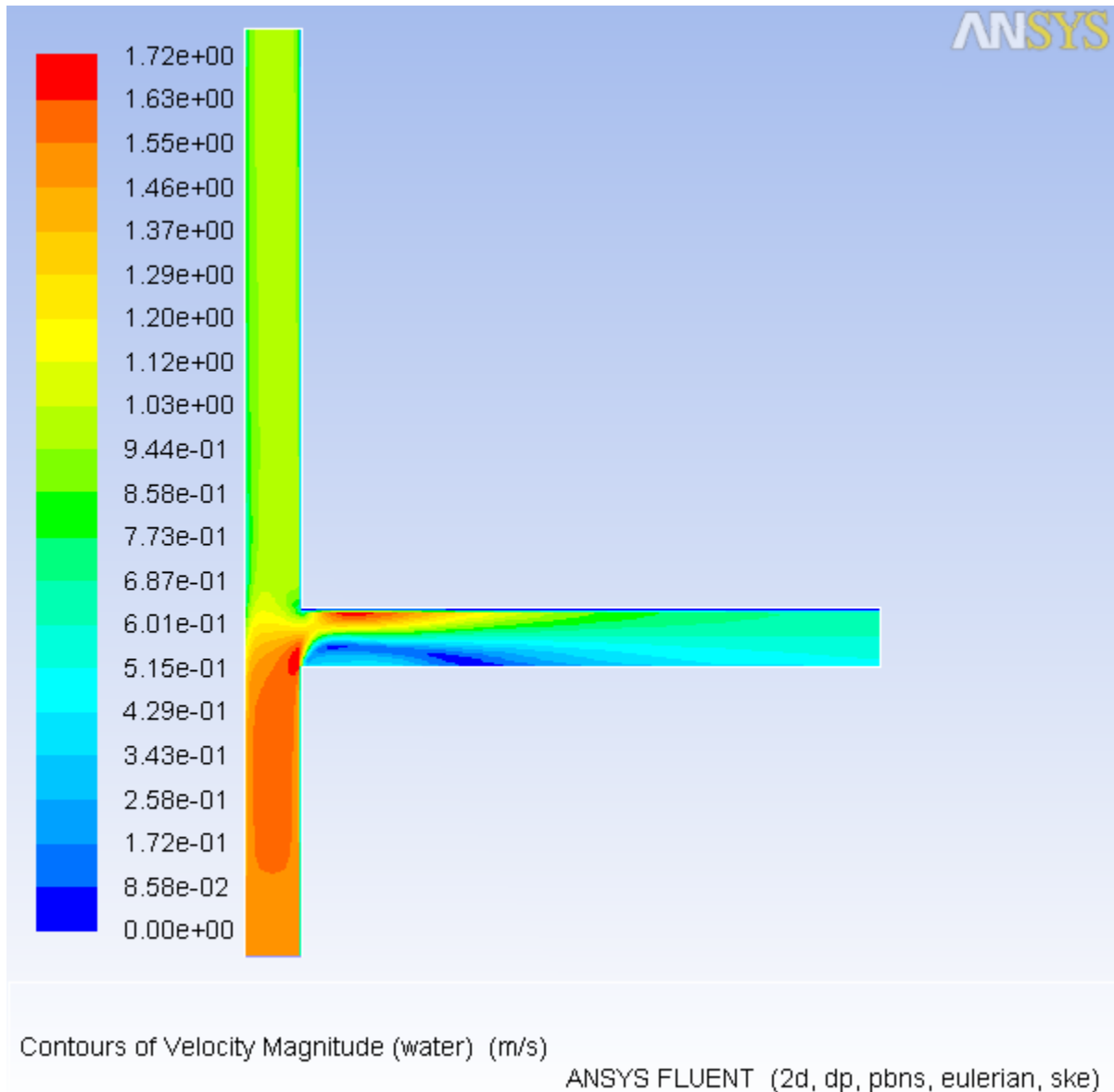
2. Display contours of velocity magnitude for water (*Figure 22.7* (p. 935)).

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

- 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.*

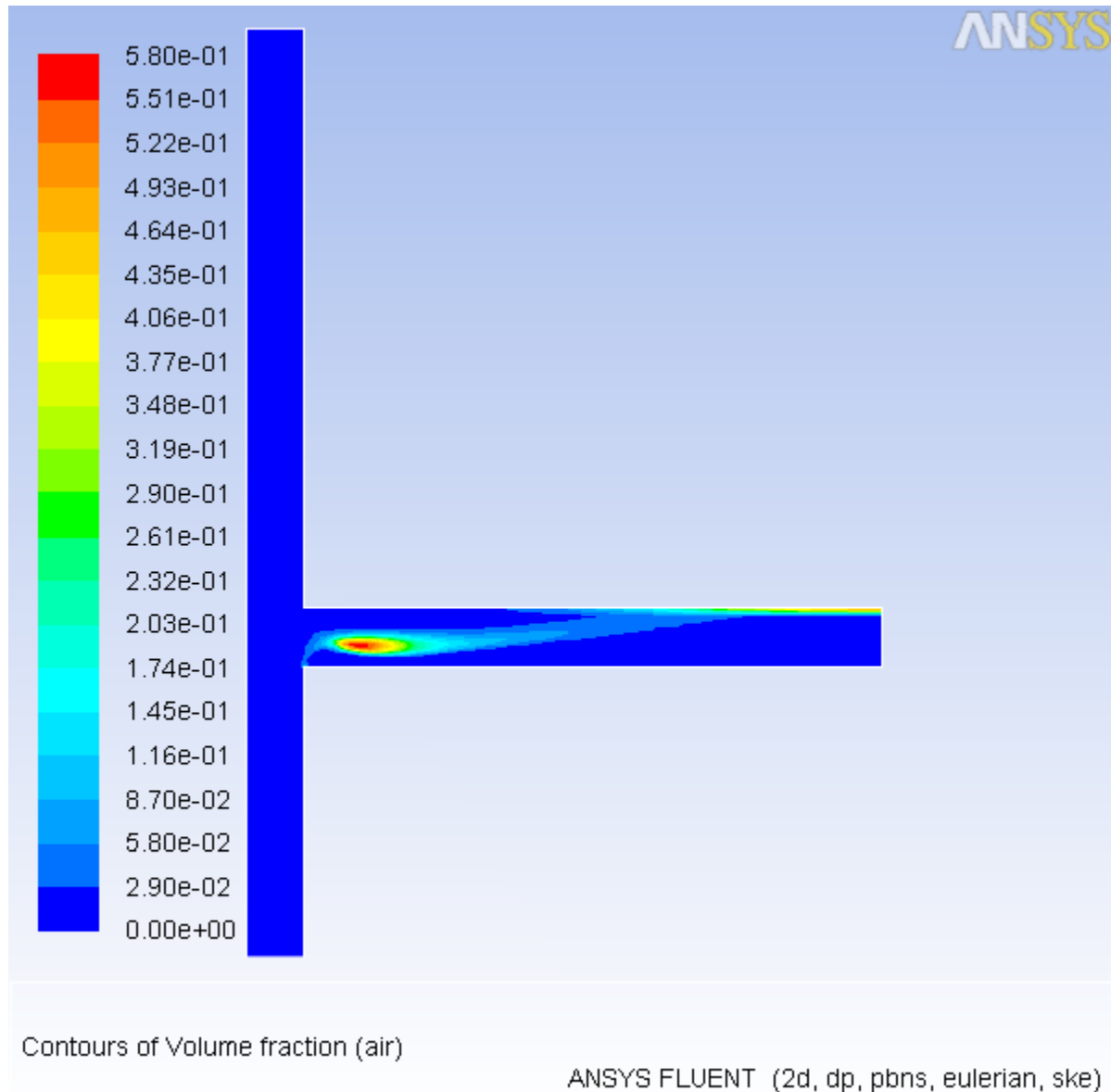
- c. Click **Display**.

**Figure 22.7 Contours of Water Velocity Magnitude**

3. Display the volume fraction of air (*Figure 22.8* (p. 936)).

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

- 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** dialog box.

**Figure 22.8 Contours of Air Volume Fraction**

## 22.5. 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 "Modeling Multiphase Flows" in the User's Guide.

## 22.6. 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 *Introduction to Using ANSYS FLUENT: Fluid Flow and Heat Transfer in a Mixing El-bow* (p. 131).

