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.

Step 1: Grid

1. Read the grid file tee.msh.

 $\mathsf{File} \longrightarrow \mathsf{Read} \longrightarrow \mathsf{Case...}$

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

2. Check the grid.

 $Grid \longrightarrow Check$

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).

 $\mathsf{Display} \longrightarrow \mathsf{Grid}...$

Grid Display		X	
Options ☐ Nodes ☑ Edges ☐ Faces ☐ Partitions	Edge Type	Surfaces = = default-interior outflow-3 outflow-5 velocity-inlet-4 wall-1	
Shrink Factor F	eature Angle 20		
Surface Name Pattern		Surface Types 🔳 🖃	
Match		axis clip-surf exhaust-fan fan	
		Outline Interior	
Display Colors Close Help			

- (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. <u>The pressure-based solver must be used for multiphase calculations.</u>

```
\boxed{\mathsf{Define}} \longrightarrow \boxed{\mathsf{Models}} \longrightarrow \\ \mathsf{Solver}...
```

Solver		
Solver © Pressure Based © Density Based	Formulation Implicit C Explicit	
Space • 2D • Axisymmetric • Axisymmetric Swirl • 3D Velocity Formulation • Absolute • Relative	Time Steady Unsteady	
Gradient Option © Green-Gauss Cell Ba © Green-Gauss Node E © Least Squares Cell E	Porous Formulation ased Based Based	
OK Cancel Help		

2. Select the mixture multiphase model with slip velocities.

Define \longrightarrow Models \longrightarrow Multiphase...

(a) Select Mixture from the Model list.

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

Multiphase Model	
Model C Off C Volume of Fluid G Mixture C Eulerian C Wet Steam Mixture Parameters Mixture Parameters Slip Velocity Body Force Formulation Miniplicit Body Force	Number of Phases
OK Canc	el Help

(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.

 $\boxed{\mathsf{Define}} \longrightarrow \mathsf{Models} \longrightarrow \mathsf{Viscous}...$

Viscous Model		
Model	Model Constants	
⊂ Laminar ⊂ Spalart-Allmaras (1 eqn) ∙ k-epsilon (2 eqn)	Cmu 0.09	
⊙ k-omega (2 eqn)	C1-Epsilon	
C Reynolds Stress (5 eqn)	1.44	
k-epsilon Model • Standard	C2-Epsilon	
C RNG C Realizable	TKE Prandtl Number	
Near-Wall Treatment	· ·	
Standard Wall Functions	User-Defined Functions	
© Non-Equilibrium Wall Functions Turbulent Viscosity		
C Enhanced Wall Treatment	none 👻	
© User-Defined Wall Functions		
OK Cancel Help		

- (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.

4. Set the gravitational acceleration.

Define → Operating Conditions...

Operating Conditions		
Pressure Operating Pressure (pascal) 101325 Reference Pressure Location X (m) 0 Y (m) 0	Gravity ✓ Gravity Gravitational Acceleration × (m/s2) g Y (m/s2) -9.81 Variable-Density Parameters ✓ Specified Operating Density Operating Density (kg/m3) g	
OK Cancel Help		

(a) Enable the **Gravity** option.

The Operating Conditions panel will expand to show additional inputs.

- (b) Enter -9.81 $\rm m/s^2$ 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.

 $Define \longrightarrow Materials...$

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

Fluent Database Materials		
Fluent Fluid Materials = vinyl-trichlorosilane (sicl3ch2ch) vinylidene-chloride (ch2ccl2) water-liquid (h2o<1>) water-vapor (h2o) wood-volatiles (wood_vol) v Copy Materials from Case Delete	Material Type fluid Order Materials By Image: Constraint of the second seco	
Density (kg/m3) Cp (j/kg-k)	constant ▼ View 998.2 constant ▼ View 4182	
Thermal Conductivity (w/m-k) Viscosity (kg/m-s)	constant View Ø.6 View Constant View Ø.001003 View	
New Edit Save Copy Close Help		

- 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.

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.

 $Define \longrightarrow Phases...$

Phases	×
Phase	Туре
phase-1	primary-phase
phase-2	secondary-phase
	ID
Interaction	2
Set	Close Help

- (a) Select phase-1 from the Phase selection list.
- (b) Click Set... to open the Primary Phase panel.

Primary Phase 🔀		
Name		
water		
Phase Material water-liquid 🗾 Edit		
OK Cancel Help		

- i. Enter water for Name.
- ii. Select water-liquid from the Phase Material drop-down list.
- iii. Click OK to close the $\mathsf{Primary}\ \mathsf{Phase}\ \mathrm{panel}.$

2. Specify air as the secondary phase.

 $Define \longrightarrow Phases...$

- (a) Select phase-2 from the Phases selection list.
- (b) Click Set... to open the Secondary Phase panel.

Secondary Phase	e	×
Name		_
air		
Phase Material	air 🝷 Edit	
🗆 Granular		
Properties		
Diameter (m)	constant 💌 Edit	
	0.001	
]		-
	OK Cancel Help	

- i. Enter air for Name.
- ii. Retain the default selection of air from the Phase Material drop-down list.
- iii. Enter $0.001 \ \mathrm{m}$ for Diameter.
- iv. Click OK to close the Secondary Phase panel.

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

Define \longrightarrow Phases...

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

Phase Interaction		
Drag Lift Collisions Slip Heat Mass Reactions Surface Tension Drag Coefficient air water schiller-naumann Edit		
OK Cancel Help		

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.
 - $Define \longrightarrow Boundary \ Conditions...$



- (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.

Velocity Inlet		
Zone Name	Phase	
velocity-inlet-4	mixture	
Momentum Thermal Radiation Species DPM	Multiphase UDS	
Specification Method Intensity and Length Scale		
Turbulent Intensity (%) 10		
Turbulent Length Scale (m) 0.025		
OK Cancel Help		

- 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 $0.025~\mathrm{m}$ 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 \longrightarrow 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.

Velocity Inlet		
Zone Name	Phase	
verocity-iniet-4	water	
Momentum Thermal Radiation Species DPM	Multiphase UDS	
Velocity Specification Method Magnitude, Normal to Boundary		
Reference Frame Absolute		
Velocity Magnitude (m/s) 1.53	constant 🗸	
OK Cancel Help		

- 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 dropdown 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.

Velocity Inlet	
Zone Name velocity-inlet-4	Phase air
Momentum Thermal Radiation Species DPM	Multiphase UDS
Velocity Specification Method Magnitude, Normal	to Boundary 🗾
Reference Frame Absolute	•
Velocity Magnitude (m/s) 1.6	constant 🔹
OK Cancel Help	

- 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 dropdown 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.

Velocity Inlet	
Zone Name velocity-inlet-4	Phase air
Momentum Thermal Radiation Species DPM	Multiphase UDS
Volume Fraction 0.02 constant	•
OK Cancel Help	 J

v. Click OK to close the Velocity Inlet panel.

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.

Outflow	×
Zone Name outflow-5	Phase Mixture
,	Flow Rate Weighting 0.62
	OK Cancel Help

- i. Enter 0.62 for Flow Rate Weighting.
- ii. Click OK to close the $\mathsf{Outflow}$ 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.

Outflow		X
Zone Name		Phase
outflow-3		mixture
	Flow Rate Weighting 0.38	}
	OK Cancel Help	

- i. Enter $0.38~{\rm for}$ Flow Rate Weighting.
- ii. Click OK to close the $\mathsf{Outflow}$ panel.
- (d) Close the Boundary Conditions panel.

Step 6: Solution Using the Mixture Model

1. Set the solution parameters.

```
Solve \longrightarrow Controls \longrightarrow Solution...
```

Solution Controls	
Equations 📃 📃	Under-Relaxation Factors
Flow Volume Fraction	Pressure 0.3
Slip Velocity Turbulence	Density 1
	Body Forces 1
	Momentum 0.7
Pressure-Velocity Coupling	Discretization
SIMPLE	Pressure PRESTO!
	Momentum First Order Upwind 👻
	Volume Fraction First Order Upwind 👻
	Turbulent Kinetic Energy First Order Upwind
0	Default Cancel Help

- (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.

2. Enable the plotting of residuals during the calculation.

 $\fbox{Solve} \longrightarrow \fbox{Monitors} \longrightarrow \r{Residual} \dots$

Residual Monit	ors				
Options	Storage			Plotting	
✓ Print✓ Plot	Iterat	tions 100	0 1	Wind	ow 0
	Normalizatio	n		Iterations	1000 붗
	🗆 No	rmalize 🛛	Scale	Axes	Curves
	Convergence	Criterion			
	absolute		•		
Residual	Che Monitor Con	ck vergence	Absolute Criteria	-	
continuity			1e-05	_	
x-velocity			0.001		
y-velocity			0.001	_	
k		\mathbf{V}	0.001		
epsilon	~	$ \overline{\bullet} $	0.001	•	
0	Plot	Reno	rm Ca	ncel H	elp

- (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…
---------	--------------	---------------

Solution Initialization	×
Compute From Reference Frame Reference Frame Relative to Ce Absolute	ll Zone
Initial Values Gauge Pressure (pascal) 0 X Velocity (m/s) 0 Y Velocity (m/s) 0 Turbulent Kinetic Energy (m2/s2) 1	
Init Reset Apply Close Help	

- (a) Retain the default settings.
- (b) Click ${\sf Init} \mbox{ and } {\sf close} \mbox{ the Solution Initialization panel.}$
- 4. Save the case file (tee.cas.gz).

 $\mathsf{File} \longrightarrow \mathsf{Write} \longrightarrow \mathsf{Case...}$

- Start the calculation by requesting 1200 iterations.
 Solve → Iterate...
- 6. Save the case and data files (tee.cas.gz and tee.dat.gz).

$File \longrightarrow Write \longrightarrow Case \And D$	ata
--	-----

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

 $\mathsf{Report} \longrightarrow \mathsf{Fluxes}...$

Flux Reports		
Options Mass Flow Rate Total Heat Transfer Rate Radiation Heat Transfer Rate	Boundaries = = default-interior outflow-3 outflow-5	Results -14.064057 -23.353506
Phase water	velocity-inlet-4 wall-1	37.417526
axis exhaust-fan fan inlet-vent		
Boundary Name Pattern Match		kg/s -3.71933e-05
Compute Write	Close	Help

- (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.

Step 7: Postprocessing for the Mixture Solution

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

Display \longrightarrow Contours...

Contours	
Options	Contours of
🗹 Filled	Pressure
✓ Node Values ✓ Global Range	Static Pressure 🔹
🗹 Auto Range	Phase
Clip to Range	mixture -
Draw Profiles	Min (pascal) Max (pascal) -3279.965 45.36762
Levels Setup	Surfaces I
Surface Name Pattern	default-interior outflow-3 outflow-5 velocity-inlet-4 wall-1
	Surface Types 📃 =
	axis clip-surf exhaust-fan fan
Display	mpute Close Help

- (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 \longrightarrow Contours...

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



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 \longrightarrow 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.

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.

$\boxed{\text{Define}} \longrightarrow \boxed{\text{Models}} \longrightarrow \boxed{\text{Multiphase}} \dots$

Multiphase Model	\mathbf{X}
Model C Off C Volume of Fluid C Mixture C Eulerian C Wet Steam	Number of Phases
OK Can	cel Help

- (a) Select Eulerian from the Model list.
- (b) Click OK to close the Multiphase Model panel.
- Specify the drag law to be used for computing the interphase momentum transfer.
 Define → Phases...

9	hases		
	Phase	Туре	
	water air		
	Interaction	ID 4	
	Set	Close	Help

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

Phase Interaction			K	
🗆 Virtual Mass				
Drag Lift Col	lisions Slip Hea	t Mass Reactions Surface Tension		
Drag Coefficient				
air	water	schiller-naumann 🔻 Edit		
, ·	,			
OK Cancel Help				

- 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.

3. Select the multiphase turbulence model.

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

Viscous Model		
Viscous Model Model C Laminar C k-epsilon (2 eqn) C Reynolds Stress (5 eqn) k-epsilon Model Standard RNG C RNG C Realizable Near-Wall Treatment Standard Wall Functions C Non-Equilibrium Wall Functions C Enhanced Wall Treatment C User-Defined Wall Functions	Model Constants Cmu	
k-epsilon Multiphase Model Mixture Dispersed Per Phase	mixture none water none air none	
OK Cancel Help		

- (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).

 $\mathsf{File} \longrightarrow \mathsf{Write} \longrightarrow \mathsf{Case} \And \mathsf{Data...}$

Step 9: Postprocessing for the Eulerian Model

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

Display \longrightarrow Contours...

Contours	X	
Options	Contours of	
🗹 Filled	Pressure	
✓ Node Values ✓ Global Range	Static Pressure 🔹	
🗹 Auto Range	Phase	
Clip to Range	mixture 🗸	
Draw Grid	Min (pascal) Max (pascal) -3281.537 49.5096	
Levels Setup	Surfaces <u>=</u>	
Surface Name Pattern	default-interior outflow-3 outflow-5 velocity-inlet-4 wall-1	
Match	Surface Types	
	axis clip-surf exhaust-fan fluid	
Display Compute Close Help		

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

By default, Dynamic Pressure will be displayed in the lower Contours of dropdown 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 \longrightarrow 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.



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).

 $\mathsf{Display} \longrightarrow \mathsf{Contours...}$

- (a) Select Phases... and Volume fraction from the Contours of drop-down lists.
- (b) Select air from the Phases drop-down list.
- (c) Click $\mathsf{Display}$ and close the $\mathsf{Contours}$ 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.