# Tutorial: Heat and Mass Transfer with the Mixture Model and Evaporation-Condensation Model

# Introduction

The purpose of this tutorial is to demonstrate the solution of a multiphase problem involving heat and mass transfer using the mixture multiphase model along with Evaporation-Condensation model.

This tutorial demonstrates how to do the following:

- Use the mixture model in ANSYS FLUENT to solve a mixture multiphase problem.
- Use Evaporation-Condensation model
- Solve the case using appropriate solver settings.
- Postprocess the resulting data.

# Prerequisites

This tutorial is written with the assumption that you have completed Tutorial 1 from the ANSYS FLUENT 13.0 Tutorial Guide, 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.

In this tutorial you will use the mixture multiphase model. This tutorial will not cover the mechanics of using this model. Instead, it will focus on the application of the model to solve a mixture multiphase problem involving heat and mass transfer. For more information refer to Section 26.4 Setting Up the Mixture Model in the ANSYS FLUENT 13.0 User's Guide.

# **Problem Description**

The problem to be solved in this tutorial is shown in Figure 1. Initially, the container contains water (the primary phase) at a temperature near the boiling point (372 K). The center portion of the bottom wall of the container is at a temperature of 573 K, which is higher than the boiling temperature. Because of conduction, the temperature of the fluid near this wall will increase beyond the saturation temperature (373 K). Vapor bubbles will form and rise due to buoyancy, establishing a pattern similar to a bubble column with vapor escaping at the top and water recirculating in the container.

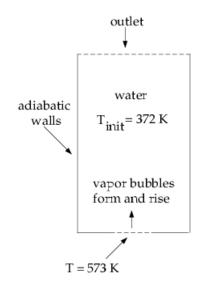


Figure 1: Problem Specification

# Preparation

- 1. Copy the mesh file (boil.msh.gz) to the working folder.
- Use FLUENT Launcher to start the 2D version of ANSYS FLUENT. For more information about FLUENT Launcher see Section 1.1.2 Starting ANSYS FLU-ENT Using FLUENT Launcher in the ANSYS FLUENT 13.0 User's Guide.
- 3. Enable Double-Precision in the Options list.

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

### Setup and Solution

#### Step 1: Mesh

1. Read the mesh file boil.msh.gz.

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

#### Step 2: General

1. Check the mesh.  $\bigcirc$  General  $\longrightarrow$  Check

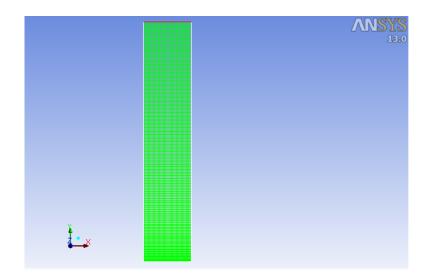


Figure 2: Mesh Display

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

2. Enable the transient solver by selecting Transient from the Time list.

#### Step 3: Models

1. Define the multiphase model.

4 Models  $\rightarrow \equiv$  Multiphase

💶 Multiphase Mod	el 🛛 🔀
Model Off Volume of Fluid Mixture Eulerian Wet Steam Mixture Parameters Slip Velocity Body Force Formulation Implicit Body Force	
ОК	Cancel Help

Edit...

- (a) Select Mixture from the Model list.
- $(b)\ {\rm Enable}\ {\rm Implicit}\ {\rm Body}\ {\rm Force}\ {\rm from}\ {\rm the}\ {\rm Body}\ {\rm Force}\ {\rm Formulation}\ {\rm list}.$
- (c) Click OK to close the Multiphase Model dialog box.
- 2. Enable energy equation.

♦ Models → Energy –	$\rightarrow$ Edit
	🖻 Energy 🛛 🔀
	Energy Energy Equation
	OK Cancel Help

After you enable the Energy Equation an information dialog box will open reminding you to confirm the property values that have changed.

#### Step 4: Materials

- 1. Copy water-liquid (h2o < l > from the database.
  - (a) Click the FLUENT Database... to open the FLUENT Database Materials dialog box.
    - i. Select water-liquid (h2o<l>) from the FLUENT Fluid Materials selection list.
    - ii. Click Copy and close the FLUENT Database Materials dialog box.

💶 Creat	e/Edit Materials						
Name water-liq	uid		Material Type fluid			~	Order Materials by
Chemical F	Formula		FLUENT Fluid Mater	ials			Chemical Formula
h2o <l></l>			water-liquid (h2o<			~	FLUENT Database
			Mixture				User-Defined Database
			none			~	
Properties	i						
	Viscosity (kg/m-s)	constant		▼ E	Edit	^	
		0.0009					
Mo	olecular Weight (kg/kgmo)	constant		<b>•</b> E	Edit		
		18.0152					
Standar	d State Enthalpy (j/kgmo)	constant		<b>~</b> E	Edit		
		0					
R	eference Temperature (k)	constant		<b>v</b> E	Edit		
		298.15					
1						*	
	Cha	nge/Create	Delete	Close	Hel	p	

- (b) Enter 1000  $\text{kg/m}^3$  for Density.
- (c) Enter 0.0009 kg/m-s for Viscosity.
- (d) Enter 18.0152 for Molecular Weight.
- (e) Enter 0 for Standard State Enthalpy.
- (f) Enter 298.15 K for Reference Temperature.
- (g) Click Change/Create and and close Create/Edit Materials dialog box.
- 2. Copy water-vapor (h2o from the database.
  - (a) Click the FLUENT Database... to open the FLUENT Database Materials dialog box.
    - i. Select water-vapor (h2o) from the FLUENT Fluid Materials selection list.
    - ii. Click Copy and close the FLUENT Database Materials dialog box.

ame	Materi	al Type			Order Materials by
water-vapor	fluid			•	Name
nemical Formula	ELLEN	T Fluid Materials			Chemical Formula
120		-vapor (h2o)			FLUENT Database
	Mixtur				User-Defined Database.
	none				×
operties					
Viscosity (kg/m-s)	constant	*	Edit	<u>^</u>	
	1.34e-05				
Molecular Weight (kg/kgmo)	constant	¥	Edit		
	18.0152				
Standard State Enthalpy (j/kgmo)	constant	*	Edit		
	2.992325e+07				
Deference Temperature (1)					
Reference Temperature (k)	constant	*	Edit		
	298.15				
				~	

- (b) Enter 0.5542 kg/m<sup>3</sup> for Density.
- (c) Enter 2014 j/kg-K for Cp (Specific Heat).
- (d) Enter 0.0261 w/m-K for Thermal Conductivity.
- (e) Enter 1.34e-05 kg/m-s for Viscosity.
- (f) Enter 18.0152 for Molecular Weight.
- (g) Enter 2.992325e+07 for Standard State Enthalpy.
- (h) Enter 298.15 K for Reference Temperature.
- (i) Click Change/Create and close the Create/Edit Materials dialog box.

Note that the standard state enthalpies of vapor and liquid phase are set such that their difference equals to latent heat of vaporization. The unit of standard state enthalpy is kj/kmol and usually the latent heat value is available in kj/kg. while specifying it here, multiply that value by molecular wight.

For correct specification of latent heat it is important to set the reference temperature to 298.15~K

#### Step 5: Phases

1. Define the primary phase.  $\clubsuit$  Phases  $\longrightarrow$  phase-1  $\longrightarrow$  Edit...

💶 Primary Phase	×
Name liquid	
Phase Material water-liquid 💽 Edit	
OK Cancel Help	

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

Phases

	y Phase	
Name		
vapor		
Phase Material	water-vapor 🗸 Edit	
🔄 Granular		
	Area Concentration	
Properties		-
Diameter (m)	constant 🗸 Edit	-
	0.0002	

- (a) Enter vapor for Name.
- (b) Ensure that air is selected from the Phase Material drop-down list.

- (c) Enter 0.0002 m for Diameter.
- (d) Click OK to close the Secondary Phase dialog box.
- 3. Select the Evaporation-Condensation Model

 $\clubsuit$  Phases  $\longrightarrow$  Interaction...

Phase Interaction			×
Drag Lift Collisions Sli	ip Heat Mass	Reactions Surface Tension Discretization	
Number of Mass Transfer Mecha	nisms 1		
Mass Transfer			
From	То		~
Phase	Phase	Mechanism	
1 liquid	✓ vapor	evaporation-condensation V Edit	
			~
OK Cancel Help			

- (a) Click the Mass tab.
- (b) Select liquid from the From Phase drop-down list.
- (c) Select vapor from the To Phase drop-down list.
- (d) Select evaporation-condensation from the Mechanism drop-down list and Click the Edit... button for Evaporation-Condensation to open the Evaporation-Condensation dialog box.

💶 Evaporation-Condensation 🔀
Model Constants
Evaporation Frequency
0.1
Condensation Frequency
0.1
Evaporation-Condensation Properties
Saturation Temperature (k)
constant 🗸 Edit
373.15
OK Cancel Help

- (e) Click OK to close the Evaporation-Condensation dialog box.
- (f) Click OK to close the Phase Interaction dialog box.

#### Step 6: Cell Zone Conditions



1. Select mixture from the Phase drop-down list and click the Edit... button to open the Fluid dialog box.

🖴 Fluid	
Zone Name	Phase
fluid	mixture
Frame Motion Source Terms Key Motion Fixed Values	
Porous Zone	
Reference Frame   Mesh Motion   Porous Zone   Embedded LES   F	Reaction   Source Terms   Fixed Values   Multiphase
Rotation-Axis Origin       X (m)       0       r (m)       0       constant	
OK Cancel (	Help

2. Keep the default settings and Click OK to close the fluid panel.

#### Step 7: Boundary Conditions

1. Set the boundary conditions for the pressure outlet.

 $\clubsuit$  Boundary Conditions...  $\longrightarrow$  poutlet

(a) Select mixture from the Phase drop-down list and click the Edit... button to open the Pressure Outlet dialog box.

Pressure Outlet	
Zone Name poutlet	Phase mixture
Momentum Thermal Radiation Species DPM Backflow Total Temperature (k)	Multiphase UDS
OK Canc	el Help

- i. Retain the default value of 0 for the  $\mathsf{Gauge}\ \mathsf{Pressure}.$
- ii. Click on Thermal tab and enter 372 K as the Backflow Total Temperature.
- iii. Click OK to close the Pressure Outlet dialog box.
- (b) Select vapor from the Phase drop-down list and click the Edit... button to open the Pressure Outlet dialog box.
  - i. Click on the Multiphase tab.
  - ii. Retain the default value of 0 for the Backflow Volume Fraction.
  - iii. Click OK to close the Pressure Outlet dialog box.
- 2. Set the boundary conditions for the wall zone, (wall-hot).

```
➡ Boundary Conditions... → ■ wall-hot
```

(a) Select mixture from the drop-down list for Phase and click the Edit... button to open the Wall dialog box.

💶 Wall			×
Zone Name		Phase	
wall-hot		mixture	
Adjacent Cell Zone			
fluid			
Momentum Therma	Radiation Species DPM Multiphase	UDS	
Thermal Conditions			
O Heat Flux	Temperature (k	573	constant 💌
<ul> <li>Temperature</li> <li>Convection</li> </ul>		Wall Thickne	ess (m)
Radiation	Hash Consultion Robe (when	A []	
Mixed	Heat Generation Rate (w/m3	0	constant 💙
Material Name aluminum	Edit		
	OK	el Help	

- i. Click the Thermal tab and select Temperature from the Thermal Conditions list.
- ii. Enter 573 K for Temperature.
- iii. Click OK to close the Wall dialog box.
- 3. Set the boundary conditions for the adiabatic wall, wall-1.

 $\textcircled{} Boundary Conditions...} \longrightarrow \fbox{} wall-1 \longrightarrow \fbox{} Edit$ 

- (a) Click on Thermal tab.
- (b) Retain the default value of 0 for Heat Flux to specify an adiabatic wall.
- (c) Click OK to close the Wall dialog box.

4. Similarly check for the other adiabatic wall, wall-2.

-					-	
Ψ	Boundary	Conditions	wall-2	$\longrightarrow$	Edit	

#### **Step 8: Operating Conditions**

Boundary Conditions — Operating Conditions...

Operating Conditions		
Operating Conditions  Pressure  Operating Pressure (pascal)  101325 P  Reference Pressure Location  X (m)  V (m)  P  Y (m)  P  P	Gravity Gravity Gravitational Acceleration X (m/s2) Y (m/s2) -9.81 Z (m/s2) T T T T T T T T T T T T T	
Z (m)	Boussinesq Parameters Operating Temperature (k) 288.16 P Variable-Density Parameters Specified Operating Density Operating Density (kg/m3) 0.5542 P	
OK Cancel Help		

- 1. Enable Gravity.
- 2. Enter -9.81  $m/s^2$  for Gravitational Acceleration in the Y direction.
- 3. Enable Specified Operating Density.
- 4. Enter 0.5542  $\text{kg/m}^3$  for Operating Density.
- 5. Click OK to close the Operating Conditions dialog box.

#### Step 9: Solution

- 1. Set the solution control parameters.
  - Solution Methods

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
SIMPLE	
Spatial Discretization	
Gradient	^
Green-Gauss Cell Based 🗸	
Pressure	
Body Force Weighted	
Momentum	
QUICK 🖌	
Volume Fraction	
QUICK 🖌	
Energy	
QUICK 💙	v
Transient Formulation	
First Order Implicit	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
Default	

- (a) Select Green Gauss Cell Based from the Gradient drop-down list.
- $(b) \ \mbox{Select Body Force Weighted} \ \mbox{from the Pressure drop-down list.}$
- (c) Select  $\mathsf{QUICK}$  from the Momentum, Volume Fraction, and Energy drop-down lists.

 $2. \ {\rm Set} \ {\rm the} \ {\rm Under-Relaxation} \ {\rm Factors}.$ 

Solution Controls	
Under-Relaxation Factors	
1	^
Body Forces	
1	
Momentum	
0.2	
Vaporization Mass	
1	
Slip Velocity	
0.1	
Volume Fraction	
0.4	~
Default	
Equations Limits Advanced	

- (a) Enter 0.5 for Pressure.
- (b) Enter 0.2 for Momentum.
- (c) Enter 0.4 for Volume Fraction.
- 3. Initialize the flow field using a  $\mathsf{Temperature}$  of 372 k.

 $\blacklozenge \mathsf{Solution\ Initialization\ } \longrightarrow \mathsf{Initializat}$ 

4. Mark the boundary region next to wall-hot.

 $\mathsf{Adapt} \longrightarrow \mathsf{Boundary}...$ 

💶 Boundary Adap	tion	$\mathbf{X}$
Options Cell Distance Normal Distance Volume Distance Contours Manage Controls	Number of Cells          1         Distance Threshold (m)         0         Boundary Volume (m3)         1         Growth Factor         1	Boundary Zones E E poutlet wall-1 wall-2 wall-hot
Adapt	Mark Apply Close	Help

- (a) Retain the Number of Cells as 1.
- (b) Deselect all zones from the Boundary Zones selection list, and then select wall-hot.
- (c) Click Mark to mark the cells for refinement.
- $(\mathrm{d})~\mathrm{Close}$  the Boundary Adaption dialog box.
- 5. Patch a temperature slightly higher than the saturation temperature of 373 K.
  - ◆ Solution Initialization → Patch...

💶 Patch		
Reference Frame  Relative to Cell Zone Absolute  Phase  Variable  Pressure X Velocity Y Velocity Temperature	Value (k) 373.15 Use Field Function Field Function	Zones to Patch E
Patch	n Close Help	

- (a) Select Temperature from the Variable selection list.
- (b) Enter  $373.15~\mathrm{K}$  for Value.
- (c) Select boundary-r0 from the Registers To Patch selection list.

- (d) Click Patch and close the Patch dialog box.
- 6. Save data files every 100 time steps.
  - Calculation Activities
  - (a) Enter 100 for Autosave Every (Time Steps).
- 7. Define commands for setting up animations.

Calculation Activities (Execute Commands)—

Create/Edit
Create/Luit

💶 Exe	cute Commands	;			×
Defined	Commands 5				
	Name	Every	When	Command	
	command-1	10	Time Step 🔽	dis cont vap vof 0 0.1	
	command-2	10	Time Step 😽 👻	dis view r-v front	
	command-3	10	Time Step 🛛 👻	dis s-p vof-%06t.jpg	
	command-4	10	Time Step 🔽	dis vec li vel li vm 0 0.4 , ,	
	command-5	10	Time Step 🛛 👻	dis s-p vel-%06t.jpg	~
OK Define Macro Cancel Help					

- (a) Set Defined Commands to 5.
- (b) Enable Active for all 5 commands.
- (c) Set Every for all commands to 10.
- (d) Select Time Step from the When drop-down lists for all commands.
- (e) Enter the commands as shown in the table.

Name	Command	
command-1	dis con vap vof 0 0.1	
command-2	dis view r-v front"	
command-3	dis s-p vof%06t.jpg	
command-4 dis vec li vel li vm 0 0.4 , ,		
command-5	nd-5 dis s-p vel%06t.jpg	

- (f) Click OK to close the Execute Commands dialog box.
- 8. Start the calculation.

# Run Calculation

- (a) Enter 0.01 as the Time Step Size.
- (b) Enter 1000 for Number of Time Steps.

- (c) Save the initial case and data files (boil-init.cas.gz and boil-init.dat.gz). File  $\longrightarrow$  Write  $\longrightarrow$  Case & Data...
- $(d)\ {\rm Click}\ {\sf Calculate}.$
- 9. Save the case and data files (boil.cas.gz and boil.dat.gz).

#### Step 10: Postprocessing

1. Read the data file for the 300th time step (boil-1-00300.dat).

2. Display filled contours of liquid velocity magnitude (Figure 3).

$$\textcircled{\ } \textbf{Graphics and Animations} \longrightarrow \textbf{E} \textbf{Contours} \longrightarrow \textbf{Set Up...}$$

- (a) Select Filled from the  $\mathsf{Options}\ \mathrm{list}.$
- (b) Select Velocity... and Velocity Magnitude from the Contours of drop-down lists.
- (c) Select liquid from the  $\mathsf{Phase}$  drop-down list and click  $\mathsf{Display}.$

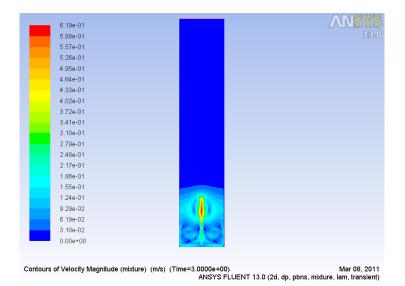


Figure 3: Contours of liquid Velocity Magnitude

- 3. Display filled contours of vapor volume fraction (Figure 4).
  - (a) Select Phases... and Volume fraction from the Contours of drop-down lists.
  - (b) Select vapor from the Phase drop-down list and click Display.

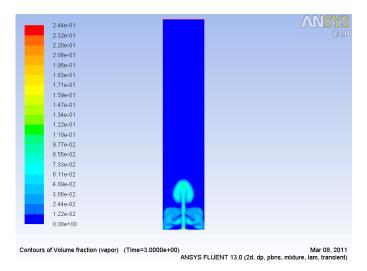
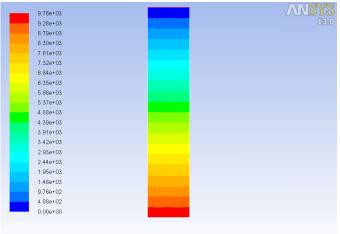


Figure 4: Contours of vapor Volume Fraction

- 4. Display filled contours of static pressure (Figure 5).
  - (a) Select Pressure... and Static Pressure from the Contours Of drop-down lists and click Display.



Contours of Static Pressure (mixture) (pascal) (Time=3.0000e+00) Mar 08, 2011 ANSYS FLUENT 13.0 (2d, dp, pbns, mixture, lam, transient)

Figure 5: Contours of Static Pressure

- 5. Display filled contours of static temperature (Figure 6).
  - (a) Select Temperature... and Static Temperature from the Contours of drop-down lists.
  - (b) Disable Auto Range and enable Clip to Range from the Options list.
  - (c) Enter  $372~\mathrm{K}$  for Min and  $375~\mathrm{K}$  for Max.
  - (d) Click Display.

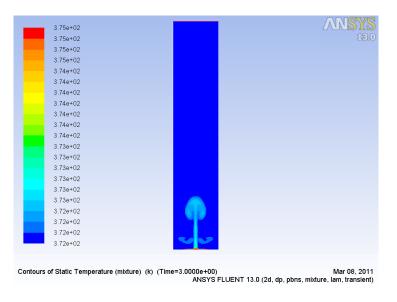


Figure 6: Contours of Static Temperature

### Summary

Application of the mixture multiphase model along with Evaporation-Condensation model has been demonstrated in this tutorial.