Tutorial: Modeling Nucleate Boiling Using ANSYS FLUENT

Introduction

The purpose of this tutorial is to demonstrate the modeling of forced convection subcooled nucleate boiling using the in-built boiling model available under Eulerian multiphase model.

This tutorial demonstrates how to do the following:

- Generate a single phase flow solution and use the fully-developed outlet as an inlet for the multiphase calculation.
- Define solution-dependent material properties as piecewise-linear functions of temperature.
- Use outlet profiles from one simulation as inlet conditions for another simulation.
- Set up the Eulerian multiphase model to predict boiling.
- Run the calculation to obtain a steady-state solution.
- Postprocess the resulting data.

Prerequisites

This tutorial is written with the assumption that you have completed Tutorial 1 from 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 Eulerian multiphase model for boiling flow. For details about this model, see Section 26.5, Setting Up the Eulerian Model in ANSYS FLUENT 13.0 User's Guide.

For more details about boiling model, see 26.5.8, Including the boiling Model in ANSYS FLUENT 13.0 User's Guide.

© ANSYS, Inc. February 17, 2011

Problem Description

In this tutorial, you will consider upward, vertical flow in a pipe with a heated wall. The flow domain is shown schematically in Figure 1. The pipe is 15.4 mm in diameter and 2 m in length. The wall provides heat to the fluid at the rate of 345.6 kW/m^2. As the wall temperature rises above the fluid saturation temperature, steam bubbles are formed and they migrate away from the wall. Since the bulk flow is subcooled, the bubbles condense near the center of the pipe.

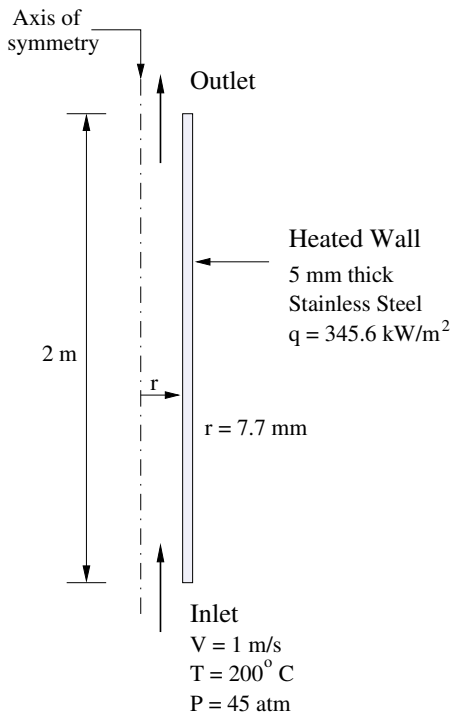


Figure 1: Schematic of the Problem

This tutorial will guide you through the complete problem setup in ANSYS FLUENT. Outlet profiles of velocity magnitude and turbulence quantities generated for a simulated flow field without boiling will be used as inlet information to the boiling simulation. This is done to ensure a fully-developed profile of these quantities at the inlet.

© ANSYS, Inc. February 17, 2011

Setup and Solution—Single Phase Flow

Preparation

1. Copy the file `boiling-conjugate.msh` to your working folder.
2. Use FLUENT Launcher to start the 2D version of ANSYS FLUENT.

   For more information about FLUENT Launcher *see Section 1.1.2, Starting* ANSYS FLUENT *Using FLUENT Launcher in* ANSYS FLUENT *13.0 User's Guide.*
3. Enable Double-Precision in the Options list.

Step 1: Mesh

1. Read the mesh file (`boiling-conjugate.msh`).

   File ⟶ Read ⟶ Mesh...

   As the mesh file is read, ANSYS FLUENT *will report the progress in the console.*

Step 2: General Settings

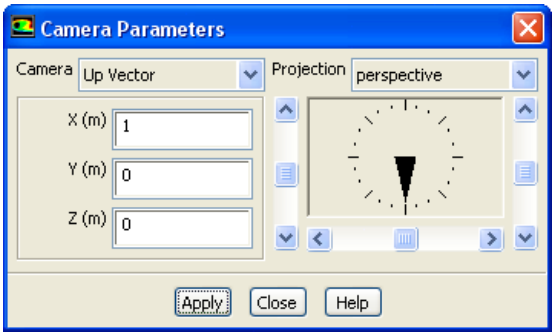
1. Define the solver settings.

   General ⟶ Axisymmetric
2. Check the mesh.

   General ⟶ Mesh ⟶ Check
3. Customize the display.

   Display ⟶ Views...

   (a) Click the Camera... button to open the Camera Parameters dialog box.



(b) Select Up Vector from the Camera drop-down list.

(c) Set the vector (X, Y, Z) to (1, 0, 0).

(d) Click Apply and close the Camera Parameters dialog box.

The graphics window now displays the pipe in the vertical position. The camera setting only controls the display of the mesh and/or solution. This is just a display feature and does not affect the calculations. In this case, the flow is still in the x-direction.

(e) Select axis from the Mirror Planes list.

(f) Click Apply.

(g) Click Save to save the view.

The saved view, view-0 *is now available in the* Views *list.*

(h) Close the Views dialog box.

(i) Zoom and pan in the graphics display window (see Figure 2).

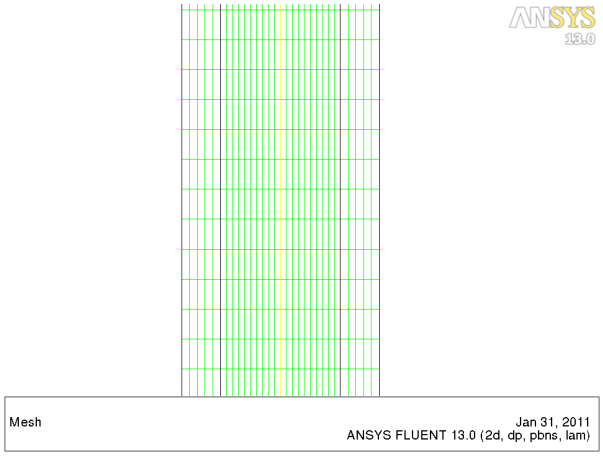


Figure 2: Mesh Display

Step 3: Models

1. Enable the Energy Equation.

   Models → Energy → Edit...

2. Select the renormalization group k-epsilon turbulence model.

   Models → Viscous → Edit...

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

   The Viscous Model *dialog box expands to show additional options.*

   (b) Select RNG from the k-epsilon Model group box.

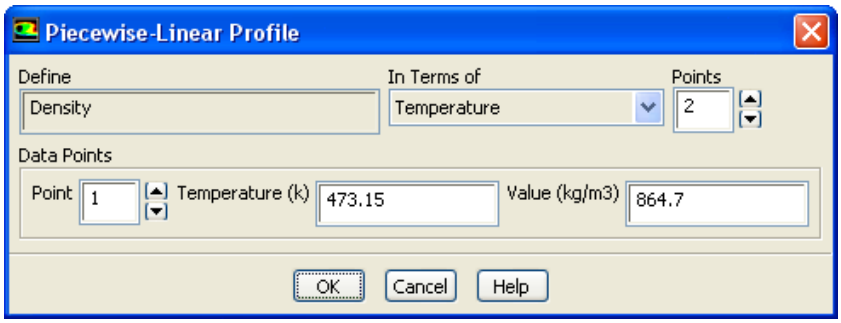
   (c) Click OK to close the Viscous Model dialog box.

© ANSYS, Inc. February 17, 2011

Step 4: Materials

Materials → Fluid → Create/Edit...

1. Copy water-liquid (h2o<l>) from the database.
2. Click on FLUENT Database... to open FLUENT Database Materials dialog box.
   (a) Select water-liquid (h2o<l>) from the FLUENT Fluid Materials list.
   (b) Click Copy and close FLUENT Database Materials dialog box.
3. Define density as a piecewise-linear profile of temperature.
   (a) Select piecewise-linear from the Density drop-down list to open the Piecewise-Linear Profile dialog box.



   (b) Set Points to 2.

   This tells ANSYS FLUENT *that you will be defining the linear profile using two data points.*

   (c) Set Point to 1 in the Data Points group box.
   (d) Enter 473.15 K and 864.7 kg/m^3 for Temperature and Value, respectively.
   (e) Set Point to 2 in the Data Points group box.
   (f) Enter 543.15 K and 770.6 kg/m^3 for Temperature and Value, respectively.
   (g) Click OK to close the Piecewise-Linear Profile dialog box.
4. Similarly, define piecewise-linear profiles for Cp (Specific Heat), Thermal Conductivity, and Viscosity as shown in Table 1.

	Points	Point 1		Point 2	
		Temperature (k)	Value	Temperature (k)	Value
Cp (j/kg-k)	2	473.15	4494	543.15	5067
Thermal conductivity (w/m-k)	2	473.15	0.664	543.15	0.5928
Viscosity (kg/m-s)	2	473.15	1.339e-04	543.15	9.995e-05

Table 1: Physical Properties of water-liquid

© ANSYS, Inc. February 17, 2011

5. Click Change/Create and close the Create/Edit Materials dialog box.

Step 5: Cell Zone Conditions

Cell Zone Conditions → fluid → Edit...

1. Select water-liquid from the Material Name drop-down list

Step 6: Boundary Conditions

1. Verify that the Type for the axis zone is set to axis.

   Cell Zone Conditions → axis

2. Define the boundary conditions at the inlet.

   Boundary Conditions → inlet → Edit...

   (a) Retain the default selection of Magnitude, Normal to Boundary as the Velocity Specification Method.

   (b) Enter 1 m/s for Velocity Magnitude.

   (c) Select Intensity and Hydraulic Diameter from the Specification Method drop-down list.

   (d) Enter 4% and 0.0154 m for Turbulent Intensity and Hydraulic Diameter, respectively.

   (e) Click the Thermal tab and enter 473.15 K for Temperature.

   (f) Click OK to close the Velocity Inlet dialog box.

3. Define the boundary conditions at the outlet.

   (a) Retain the default value of 0 atm for Gauge Pressure.

   (b) Retain Normal to Boundary from the Backflow Direction Specification Method drop-down list.

   (c) Select Intensity and Hydraulic Diameter from the Specification Method drop-down list.

   (d) Enter 4% and 0.0154 m for Backflow Turbulent Intensity and Backflow Hydraulic Diameter, respectively.

   The backflow conditions should be set to a reasonable value so that ANSYS FLUENT *will introduce the fluid through an outlet if necessary.*

   (e) Click the Thermal tab and enter 530.55 K for Backflow Total Temperature.

   (f) Click OK to close the Pressure Outlet dialog box.

© ANSYS, Inc. February 17, 2011

Step 7: Operating Conditions

Boundary Conditions → Operating Conditions...

1. Enter `4.5e6` Pa for Operating Pressure.
2. Enable Gravity.
   (a) Enter `-9.81` m/s^2 for Gravitational Acceleration in the X direction.
   This indicates that the gravity vector is opposed to the direction of flow.
3. Click OK to close the Operating Conditions dialog box.

Step 8: Single Phase Solution and Outlet Profile

1. Retain the default solution control parameters.
   Solution Controls
2. Specify residual monitor settings.
   Monitors → Residuals → Edit...
   (a) Reduce the Absolute Criteria for continuity to `1e-08`.
   (b) Click OK to close the Residual Monitors dialog box.
3. Initialize the solution using the values at the inlet.
   Solution Initialization
4. Run the calculation for 500 iterations.
   Run Calculation
   The solution converges in 320 iterations approximately (see Figure 3).

© ANSYS, Inc. February 17, 2011

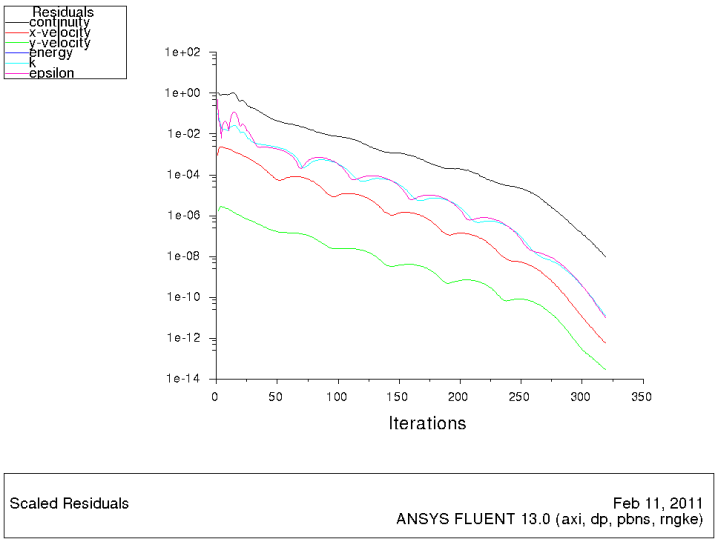


Figure 3: Residual Plot of the Single-Phase Flow Solution

5. Plot the fully developed outlet profile.

   Plots → XY Plot → Set Up...

   (a) Set the Plot Direction (X, Y) to (0, 1).

   (b) Select Velocity... and Velocity Magnitude in the Y Axis Function drop-down lists.

   (c) Select outlet from the Surfaces list and click Plot (see Figure 4).

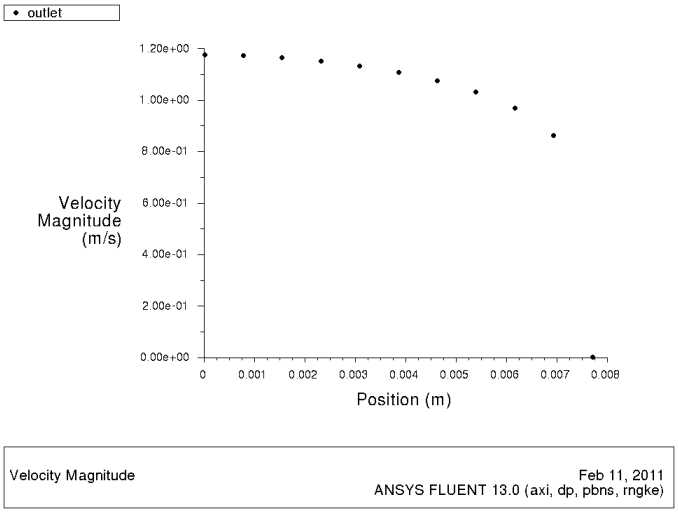


Figure 4: Fully-Developed Outlet Velocity Profile

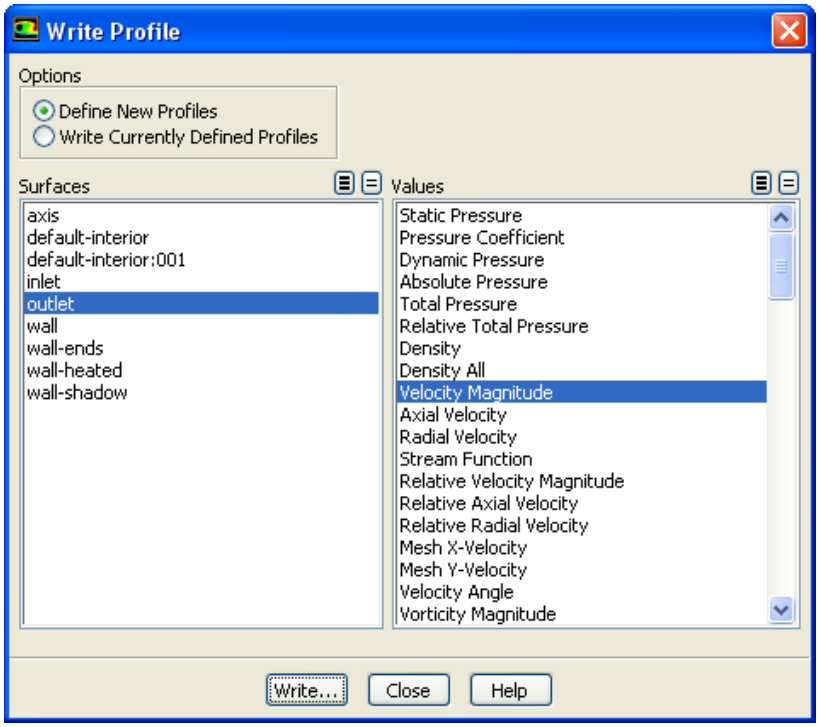
© ANSYS, Inc. February 17, 2011

(d) Close the Solution XY Plot dialog box.

6. Write a data profile at the outlet.

   File ⟶ Write ⟶ Profile...

   In this step, you will write a file containing selected flow solution information on specified surfaces. You will use this information as inlet boundary conditions in the boiling model.



   (a) Retain Define New Profiles in the Options group box.

   (b) Select outlet from the Surfaces list.

   (c) Select Velocity Magnitude, Turbulent Kinetic Energy (k), and Turbulent Dissipation Rate (Epsilon) from the Values list.

   Scroll down the Values *list to select* Turbulent Kinetic Energy (k)*, and* Turbulent Dissipation Rate (Epsilon)*.*

   (d) Click Write... to open the Select File dialog box.

   (e) Enter `liquid-outlet.prof` for File Name and click OK.

   This writes a profile file to the working folder. This file is used to supply inlet information for the boiling calculation.

   (f) Close the Write Profile dialog box.

7. Read the newly created profile into memory.

   File ⟶ Read ⟶ Profile...

   After creating the profile, you need to read it into memory to make the profile variables available as boundary conditions. ANSYS FLUENT *displays the progress of reading the profile in the console.*

8. Save the case and data files (`boiling-single-phase.cas/dat.gz`).

Setup and Solution—Boiling Flow

Step 1: Models

1. Enable the Eulerian multiphase model. In the same panel under Eulerian parameters, select Boiling Model. This will pop up a message that boiling model should be used with turbulent flow. Click ok. Note that under Boiling Model Options RPI Boiling model is selected by default.

   Models → Multiphase → Edit...

2. Ensure that Mixture is selected from the Turbulence Multiphase Model group box.

   Models → Viscous → Edit...

Step 2: Materials

Materials → water-liquid → Create/Edit...

1. Enter additional properties for water-liquid.
   (a) Enter `18.0152` for Molecular Weight.
   (b) Enter `0` for Standard State Enthalpy.
   (c) Enter `298.15` K for Reference Temperature.
   (d) Click Change/Create

2. Copy the fluid material (water-vapor) from the database.
   (a) Click on FLUENT Database... to open FLUENT Database Materials dialog box.
      i. Select fluid in the Material Type drop-down list.
      ii. Select water-vapor (h2o) from the FLUENT Fluid Materials list.
      iii. Click Copy to add the selected material to the current case setup.
      iv. Close the FLUENT Database Materials dialog box.

3. Modify the properties of water-vapor.
   (a) Specify the properties as shown in Table 2.

   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 avaiable 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*

© ANSYS, Inc. February 17, 2011

Property	Value
Density (kg/m^3)	23.75
Cp (Specific Heat) (j/kg-k)	4221
Thermal Conductivity (w/m-k)	0.0528
Viscosity (kg/m-s)	1.79e-05
Molecular Weight (kg/kgmol)	18.0152
Standard State Enthalpy (j/kgmol)	2.992325e7
Reference Temperature (K)	298.15

Table 2: Physical Properties of water-vapor

4. Create the wall material (stainless-steel) by modifying the properties of aluminum.

   (a) Select Solid from Material Type drop-down list.

   (b) Rename aluminum to `stainless-steel` and delete the entry for Chemical Formula.

   (c) Enter the properties of stainless-steel as shown in Table 3.

Property	Value
Density (kg/m^3)	8030
Cp (j/kg-k)	502.48
Thermal Conductivity (w/m-k)	16.27

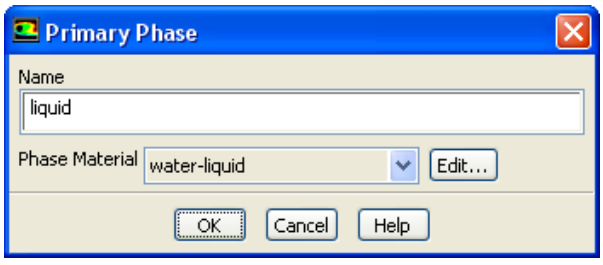
Table 3: Physical Properties of stainless-steel

   (d) Click Change/Create and close Create/Edit Materials dialog box.

   A Question *dialog box appears asking change/create* stainless-steel *and overwrite* aluminum. *Click* Yes.

Step 3: Phases

1. Define the primary (liquid) phase.



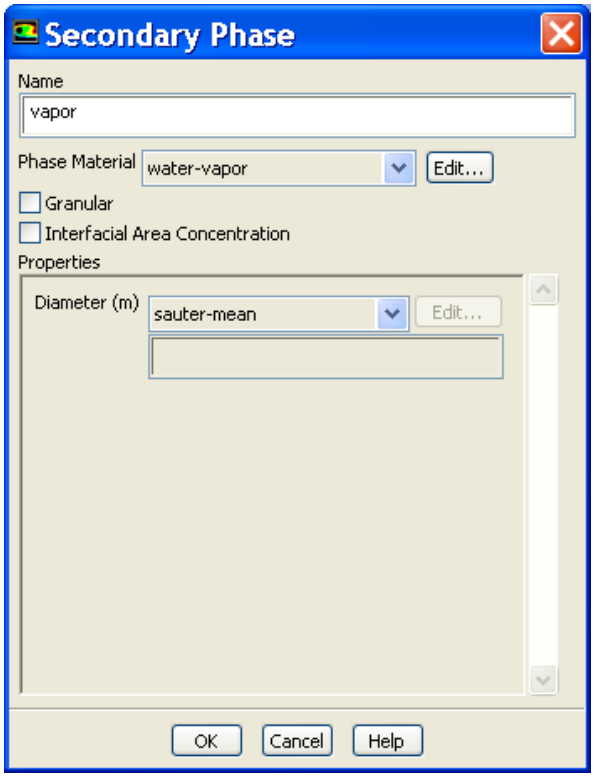
   (a) Enter `liquid` for Name.

   (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 (vapor) phase.

   Phases → phase-2-Secondary Phase → Edit...



   (a) Enter vapor for Name and select water-vapor from the Phase Material drop-down list.

   (b) Select sauter-mean from the Diameter drop-down list.

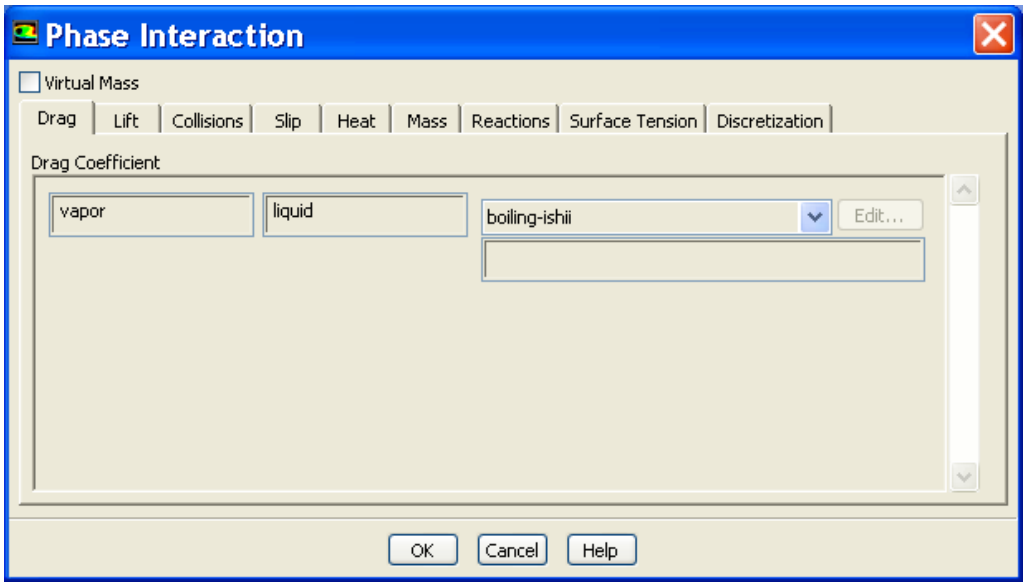
   (c) Click OK to close the Secondary Phase dialog box.

3. Define the interaction between the phases.

   Phases → Interaction...

   (a) Click the Drag tab and select boiling-ishii from the Drag Coefficient drop-down list.

© ANSYS, Inc. February 17, 2011

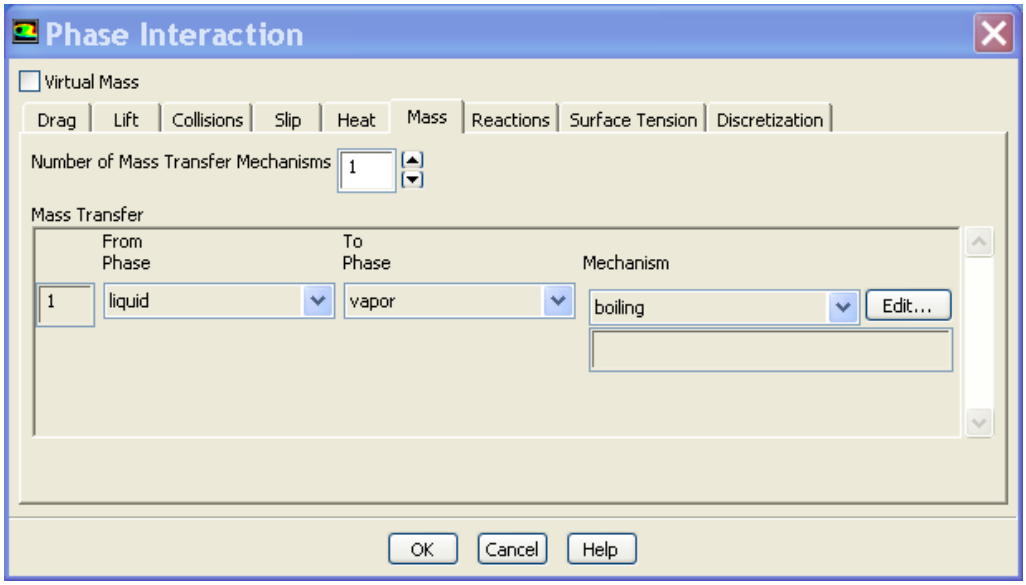


(b) Similarly, click the Lift tab and select boiling-moraga from the Lift Coefficient drop-down list.

(c) Click the Heat tab and select ranz-marshall from the Heat Transfer Coefficient drop-down list.

(d) Click the Mass tab and set the Number of Mass Transfer Mechanisms to 1.

The dialog box expands to show additional inputs.



i. Select liquid and vapor from the From Phase and To Phase drop-down lists, respectively.

ii. Select boiling from the Mechanism drop-down list.

iii. Click Edit button next to the Mechanism drop-down list. This will open a panel for boiling parameters. Set 530.55K for saturation temperature and

click OK to close this panel.

iv. Click the Surface Tenstion tab and set a constant value of `0.02382` n/m.

v. Click OK to close the Phase Interaction dialog box.

4. Enable turbulence dispersion force

   (a) Execute following TUI command to enable turbulence dispersion force:

   /define/models/viscous/multiphase-turbulence/multiphase-options yes

Step 4: Operating Conditions

Boundary Conditions → Operating Conditions...

1. Enable Specified Operating Density in the Variable-Density Parameters group box.
2. Enter `23.75` kg/m^3 for Operating Density.
3. Click OK to close the Operating Conditions dialog box.

Step 5: Solution Initialization

Before setting the boundary conditions, you need to initialize the flow field because currently memory has single-phase flow solution later on you will be setting two-phase boundary conditions on the walls. Else, it will give an error.

Initialize the flow field.

Solution Initialization

In order to maintain flexibility in multiphase problems, ANSYS FLUENT *allows the specification of separate boundary conditions for each phase. Boundary conditions for the mixture phase are required as well. This allows each phase to be treated differently at various surfaces.*

Step 6: Cell Zone Conditions

1. Set the cell zone conditions for fluid.

   Cell Zone Conditions → fluid

   (a) Select vapor from the Phase drop-down list and click Edit....

      i. Enable Fixed Values.

      ii. Click the Fixed Values tab.

         A. Select constant from the Temperature drop-down list.

            By applying the constant value for the temperature in the vapor phase, it is assumed that the vapor is always in the saturated condition. This approach is recommended for subcooled nucleate boiling. The saturation temperature is 530.55 K approximately.

© ANSYS, Inc. February 17, 2011

B. Enter 530.55 K for Temperature.

iii. Click OK to close the Fluid dialog box.

2. Set the cell zone conditions for solid.

Cell Zone Conditions → solid

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

i. Select stainless-steel from the Material Name drop-down list.

ii. Click OK to close the Solid dialog box.

Step 7: Boundary Conditions

1. Define the boundary conditions at the outlet.

Boundary Conditions → outlet

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

i. Ensure that the Gauge Pressure to 0.

ii. Select Intensity and Hydraulic Diameter from the Specification Method drop-down list.

iii. Enter 1% and 0.0154 m for Backflow Turbulent Intensity and Backflow Hydraulic Diameter respectively.

iv. Click OK to close the Pressure Outlet dialog box.

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

i. Click the Thermal tab and enter 530.55 K for Backflow Total Temperature.

The backflow conditions should be set to a reasonable value so that ANSYS FLUENT *will introduce the fluid through an outlet if necessary.*

ii. Click OK to close the Pressure Outlet dialog box.

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

i. Click the Thermal tab and enter 530.55 K for Backflow Total Temperature.

ii. Click the Multiphase tab and enter 0 for Backflow Volume Fraction.

iii. Click OK to close the Pressure Outlet dialog box.

2. Define wall boundary conditions.

(a) Select wall-heated from the Zone list and mixture in the Phase drop-down list, and click Edit....

i. Click the Thermal tab and select stainless-steel from the Material Name drop-down list.

ii. Enter 345600 w/m^2 for Heat Flux.

iii. Click OK to close the Wall dialog box.

3. Define the boundary conditions at the inlet.

   Boundary Conditions → inlet

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

      i. Select K and Epsilon from the Specification Method drop-down list.

      ii. Select outlet turb-kinetic-energy from the Turbulent Kinetic Energy drop-down list.

      iii. Select outlet turb-diss-rate from the Turbulent Dissipation Rate drop-down list.

      iv. Click OK to close the Velocity Inlet dialog box.

   (b) Select liquid from the Phase drop-down list and 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. Select the outlet velocity-magnitude from the Velocity Magnitude drop-down list.

      iii. Click the Thermal tab and ensure that Temperature is `473.15` K.

      The inlet temperature (T = 473.15 K) is below the saturation temperature (T_{sat} = 530.55 K), indicating a subcooled liquid entering the flow domain. The difference in the two values ($T - T_{sat}$) is defined as the degree of subcooling. In this case, the liquid at the inlet is subcooled by 57.4 degrees.

      iv. Click OK to close the Velocity Inlet dialog box.

   (c) Select vapor from the Phase drop-down list and 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. Enter `0` for Velocity Magnitude.

      iii. Click the Thermal tab and enter `530.55` K for Temperature.

      iv. Click the Multiphase tab and enter `0` for Volume Fraction.

      The inlet vapor velocity and volume fraction are set to zero because there is no vapor entering the flow domain. Instead, vapor is evolved at the walls as a result of nucleate boiling. The vapor temperature is set to the saturation temperature (530.55 K) as the vapor is assumed to be saturated throughout the flow domain.

      v. Click OK to close the Velocity Inlet dialog box.

Step 8: Solution

1. Define Solution Method

© ANSYS, Inc. February 17, 2011

(a) For pressure-Velocity Coupling, select Full Multiphase Coupled scheme from the drop dowm menu.

(b) Leave Spatial discretization schemes to default

2. Define the solution control parameters.

   Solution Controls

   (a) Set Courant Number to 10

   (b) Set Explicit Relaxation Factors for Pressure and Momentum to 1.0 and 1.0 respectively.

   (c) Specify the Under-Relaxation Factors as shown in Table 4.

Flow Variable	**Under-Relaxation Factor**
Density	1.0
Body Forces	0.5
Vaporization Mass	0.5
Volume Fraction	0.3
Turbulent Kinetic Energy	0.3
Turbulent Dissipation Rate	0.3
Turbulent Viscosity	0.5
Energy	0.6

Table 4: Under-Relaxation Factors

3. Set Residual Monitors

   Monitors (Residuals)⟶ Edit...

   (a) Switch on Compute Local Scle and from the drop down menu of Reporting option, choose local scale.

   (b) Click Ok to close the residuals panel

4. Set additional solution monitors.

   For multiphase flow modeling, the default residual plot can sometimes be misleading as to whether or not the solution is converged. Hence, additional monitors should always be used to evaluate other quantities such as overall mass balance, heat balance, etc. For this simulation, define monitors for global mass balance and average liquid outlet temperature.

© ANSYS, Inc. February 17, 2011

(a) Set a monitor for global mass balance (`surf-mon-1`).

Monitors (Surface Monitors)⟶Create...

i. Enable Plot and Write.

ii. Select Flow Time from the X Axis drop-down list.

iii. Select Time Step from Every drop-down list.

iv. Select Mass Flow Rate in the Report Type drop-down list.

v. Select inlet and outlet from Surfaces list.

(b) Set a monitor for average outlet vapor fraction (surf-mon-2).

Monitors (Surface Monitors)⟶Create...

i. Enable Plot and Write.

ii. Select Flow Time from the X Axis drop-down list.

iii. Select Time Step from Every drop-down list.

iv. Select Area-Weighted Average in the Report Type drop-down list.

v. Select Phases... and Volume fraction from the Field Variable drop-down lists.

vi. Select vapor in the Phase drop-down list.

vii. Select outlet from Surfaces list.

5. Initialize the solution from the inlet zone.

Solution Initialization

6. Save the case and data files (`boiling-init.cas/dat.gz`).

7. Start the calculation.

Run Calculation

(a) Set the Number of Iterations to 2000.

8. Click Calculate.

Figure 5 shows the residual plot for the boiling simulation. Figures 6 and Figure 7 show the surface monitor plots of mass flow balance and average outlet vapor fraction, respectively.

© ANSYS, Inc. February 17, 2011

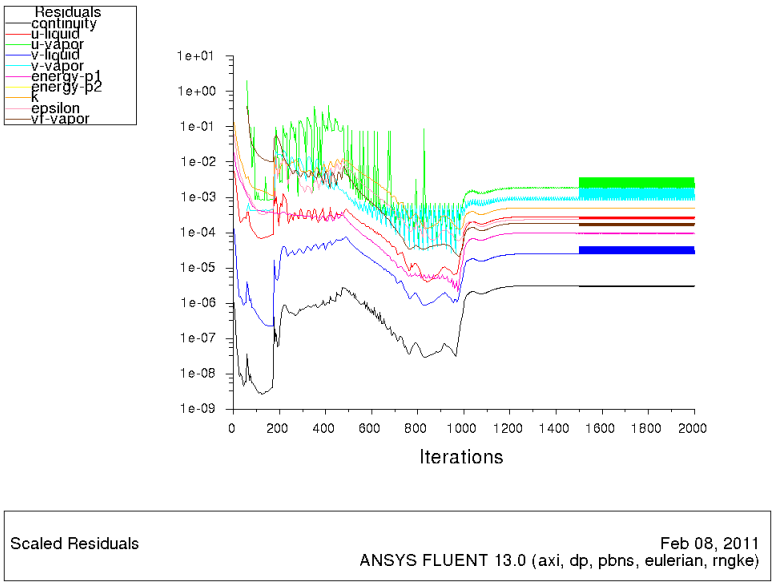


Figure 5: Residual Plot for Boiling Simulation

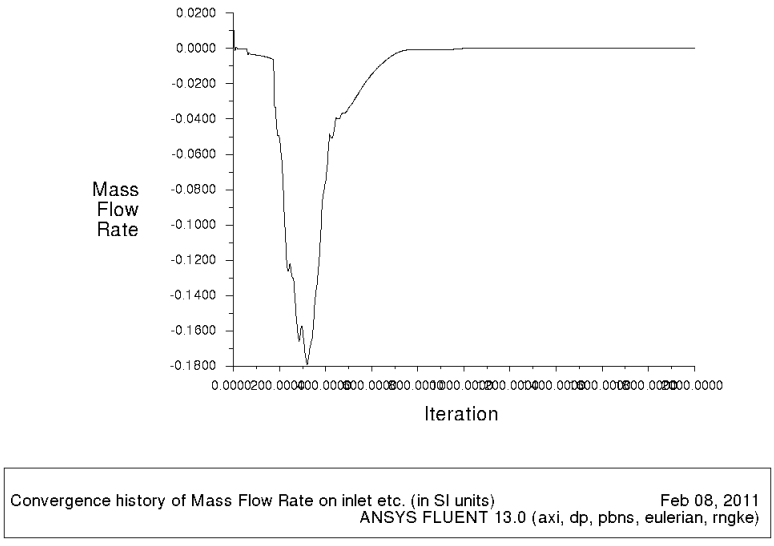


Figure 6: Convergence History of Mass Balance

© ANSYS, Inc. February 17, 2011

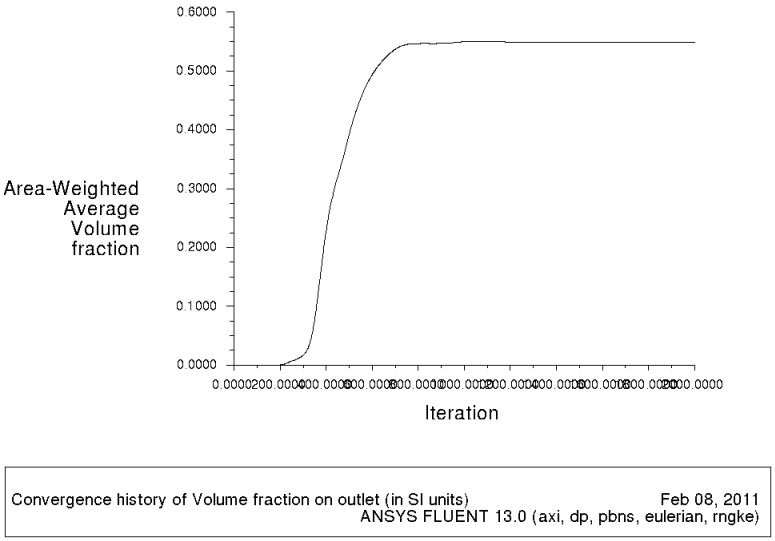


Figure 7: Convergence History of Average Outlet Vapor Fraction

Step 9: Postprocessing

1. Create a zone surface for the fluid zone.

   Surface ⟶ Zone...

   (a) Select fluid from the Zone list.

   (b) Click Create and close the Zone Surface dialog box.

2. Plot the vapor volume fraction along the wall of the pipe.

   Plots ⟶ XY Plot ⟶ Set Up...

   (a) Ensure that Node Values in the Options group box is enabled.

   (b) Set the Plot Direction (X, Y) to (1, 0).

   (c) Select Phases... and Volume fraction from the Y Axis Function drop-down lists.

   (d) Select vapor from the Phase drop-down list.

   (e) Select wall from the Surfaces list.

   The wall *zone is selected since it is adjacent to the fluid zone where boiling occurs.*

   (f) Click Plot (see Figure 8).

   Figure 8 shows that the vapor is evolved along the pipe wall as the fluid is heated and boiling occurs. The location of boiling initialization is observed to be around $x = 0.5$ *m.*

© ANSYS, Inc. February 17, 2011

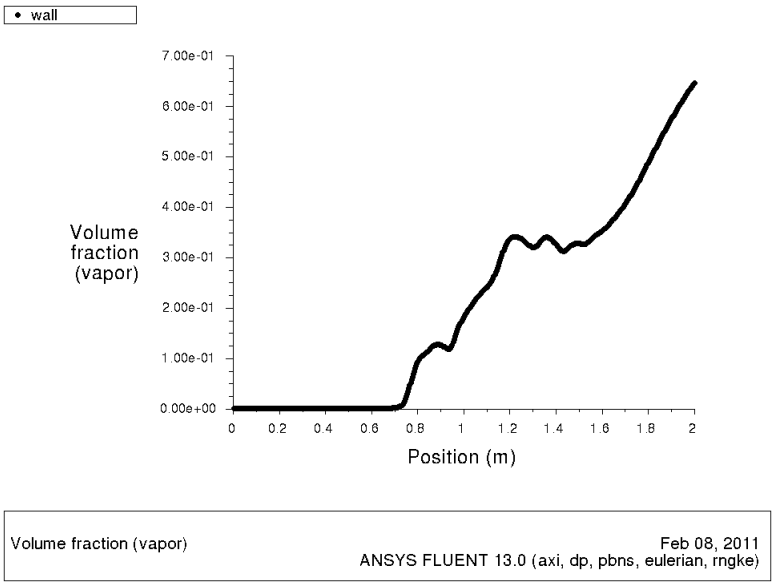


Figure 8: Profile of Vapor Volume Fraction Along the Wall

3. Create custom surfaces.

   (a) Create a line surface at $x = 0.2$ m.

   Surface ⟶ Iso-Surface...

      i. Select Mesh... and X-Coordinate from the Surface of Constant drop-down lists.

      ii. Click Compute.

      The minimum and maximum permissible values are displayed in the Min *and* Max *fields.*

      iii. Enter `0.2` for Iso-Values.

      iv. Enter `x=0.2m` for New Surface Name.

      v. Click Create.

   (b) Similarly, create surfaces at $x = 0.4$ m, $x = 0.6$ m, $x = 0.8$ m, $x = 1.0$ m, $x = 1.2$ m, $x = 1.4$ m, $x = 1.6$ m, $x = 1.8$ m, and $x = 2.0$ m.

4. Plot the profiles of vapor volume fraction at each location along the pipe.

   Plots ⟶ XY Plot ⟶ Set Up...

   (a) Enable Node Values in the Options group box.

   (b) Set the Plot Direction (X, Y) to (`0`, `1`).

   (c) Select Phases... and Volume fraction from the Y Axis Function drop-down lists.

   (d) Select vapor from the Phase drop-down list.

   (e) Select the surfaces created in the earlier step from the Surfaces list.

   (f) Click Plot (see Figure 9).

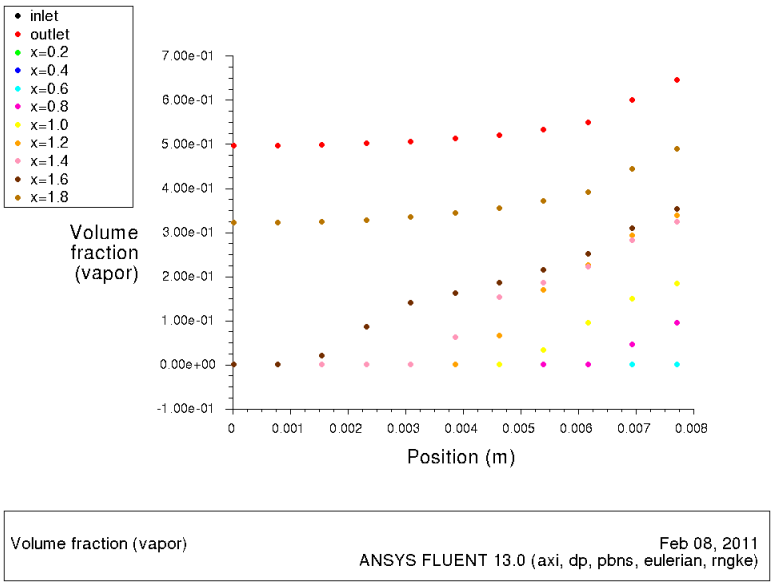


Figure 9: Profiles of Vapor Volume Fraction at Various Locations Along the Pipe

Figure 9 shows radial profiles of vapor volume fraction at various locations along the pipe.

Summary

This tutorial demonstrated that the subcooled nucleate boiling can be simulated using in built RPI Wall Boiling Model in ANSYS FLUENT.

© ANSYS, Inc. February 17, 2011