
Tutorial: Horizontal Film Boiling

Introduction

The purpose of this tutorial is to provide guidelines and recommendations for setting up and solving film boiling problems and advice on mesh resolution and solver settings.

In such problems, the wall temperature is much higher than the saturation temperature of the liquid in contact with the wall, and the entire wall surface is immersed in vapor. Due to the boiling mass exchange occurring at the vapor-liquid interface, bubbles of gas are periodically produced and emitted upward. Such a regime is known as film boiling.

In this tutorial, you will need to compile the user-defined function (UDF), as it cannot be run through the FLUENT interpreter for this version of FLUENT.

Prerequisites

This tutorial assumes that you are familiar with the FLUENT interface, and have a good understanding of the basic setup and solution procedures. Some steps will not be shown explicitly.

In this tutorial, you will use the volume of fluid (VOF) multiphase model. If you have not used this model before, refer to [Section 24.2: Volume of Fluid \(VOF\) Model](#) in the FLUENT 6.2 User's Guide and [Tutorial 16: Using the VOF Model](#) in the FLUENT 6.2 Tutorial Guide. If you have not used UDFs, refer to the [FLUENT 6.2 UDF Manual](#).

This tutorial does not cover the mechanics of using this model. Instead, it focuses on the application of this model to a film boiling regime.

Problem Description

The problem to be solved in this tutorial is shown in Figure 1.

The wall has a temperature 10K above saturation. Initially, the linear temperature profile (from T_{wall} to T_{sat} in the positive Y direction) is patched on the liquid domain.

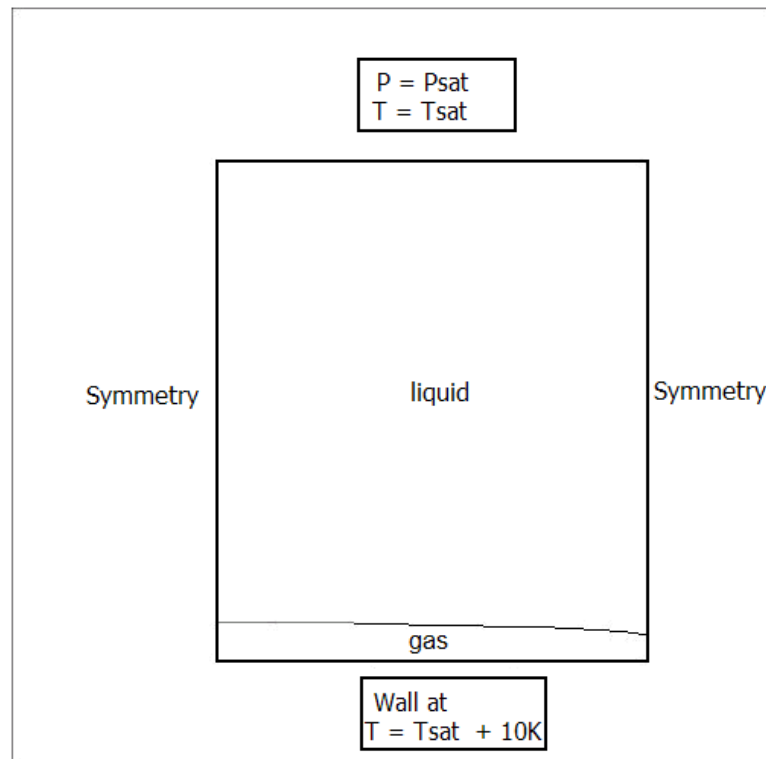


Figure 1: Problem Schematic

Preparation

1. Copy the files, `test-2d.msh` and `boiling.c` to your working directory.
2. Start the 2D double precision (2ddp) version of FLUENT.

Setup and Solution

Step 1: Grid

1. Read the mesh file, `test-2d.msh`.
2. Check and display the grid (Figure 2).

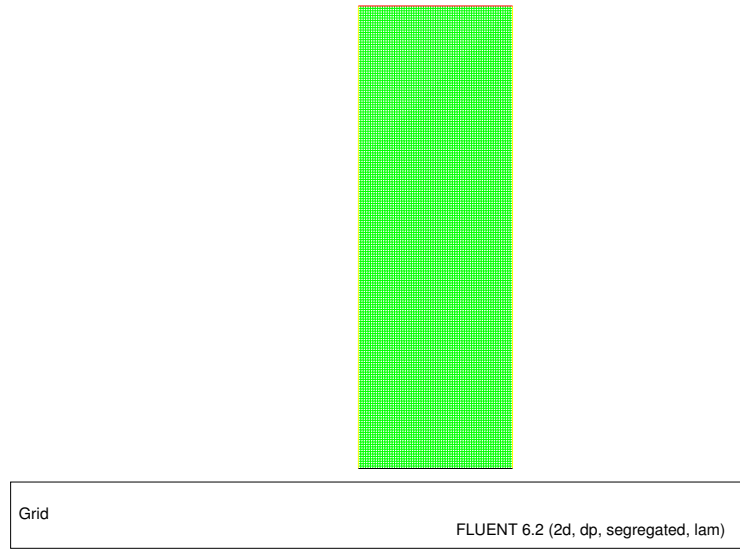


Figure 2: Grid Display

Step 2: Models

1. Define the solver settings.
 - (a) Under **Time**, select **Unsteady**.
 - (b) Retain the default settings for the other parameters.
2. Define the multiphase model.
 - (a) Select the **Volume of Fluid** multiphase model.
 - (b) Enable the **Implicit Body Force** formulation.
3. Enable the **Energy Equation**.

Step 3: Materials

1. Create two new materials, **liquid** and **vapor**, and specify the properties (in SI units) as given in the following table:

Parameters	liquid	vapor
Density (kg/m^3)	200	5
Cp (J/kg-K)	400	200
Thermal Conductivity (W/m-K)	40	1
Viscosity (kg/m-s)	0.1	0.005

Step 4: Phases

1. Define the primary (vapor) phase.
 - (a) Under Phase, select phase-1, and click Set....
 - (b) Select vapor in the Phase Material drop-down list.
 - (c) Under Name, enter gas.
2. Similarly, define the secondary (liquid) phase, specifying liquid as the Phase Material and fluid for Name.
3. Specify the interphase interaction.
 - (a) Click Interaction....
 - (b) In the Phase Interaction panel, click the Surface Tension tab.
 - (c) Select constant in the drop-down list for Surface Tension Coefficients and enter a value of 0.1 n/m.

Step 5: Operating Conditions

1. Enable Gravity.
2. Set the Gravitational Acceleration in the Y direction to -9.81 m/s^2 .
3. Enable Specified Operating Density and set the Operating Density to 5 kg/m^3 .

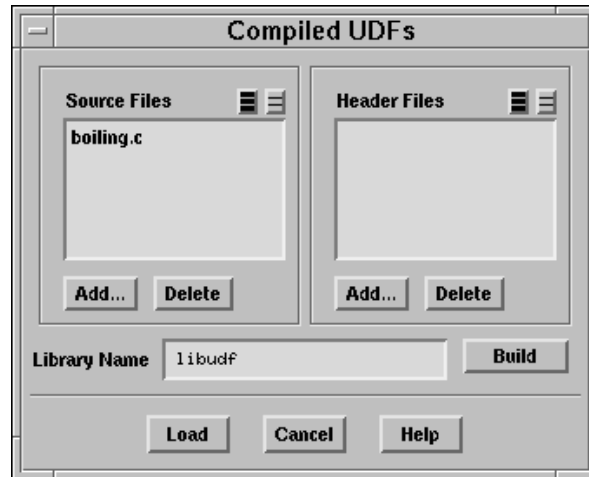
Step 6: User-Defined Function

In this problem, the UDF is used to specify the mass transfer between the phases. If you want to use a mass transfer model that is not available in FLUENT you can define it using a UDF. The mass transfer terms for each phase (vapor and liquid), are equal and have opposite signs. They are specified as source terms in volume fraction equations and have units of kg-sec/m^3 . In addition, there is an energy source temperature to take into account the latent heat absorbed/released, which has units of Watt/m^3 and is prescribed for mixture energy equation.

For more information on compiled UDFs, refer to [Section 8: Compiling UDF Source Files](#) in the FLUENT 6.2 UDF Manual.

1. Compile the UDF, boiling.c, using the Compiled UDFs panel.

Define → User-Defined → Functions → Compiled...



Make sure that the UDF source file, `boiling.c`, is in the same directory that contains your case and data files.

- (a) Under Source Files, click Add....
This will open the Select File panel.
- (b) In the Select File panel, select `boiling.c`.
- (c) In the Compiled UDFs panel, enter the name of the library directory (e.g., `libudf`) and click Build to build a shared library for the source file.
- (d) Click Load to link your shared library to the FLUENT executable.

2. Define Function Hooks.

Define → User-Defined → Function Hooks...

- (a) Select `my_init_function::libudf` in the Initialization drop-down list to hook this function to the initialization function.
- (b) Select `area_density::libudf` in the Adjust drop-down list.

3. In the User-Defined Memory panel, set the Number of User-Defined Memory Locations (UDMLs) to 3.

Define → User-Defined → Memory...

This step is necessary because the UDF uses three UDMLs.

Step 7: Boundary Conditions

1. Set the boundary conditions for fluid.

- (a) In the Boundary Conditions panel, retain mixture for Phase and click Set....
 - i. Enable Source Terms and click the Source Terms tab.
 - ii. Select `udf energy::libudf` in the Energy drop-down list.
- (b) In the Boundary Conditions panel, change the Phase to fluid and click Set....

- i. Enable Source Terms.
 - ii. Select `udf liquid::libudf` in the Mass drop-down list.
 - (c) In the Boundary Conditions panel, change the Phase to gas and click Set....
 - i. Enable Source Terms.
 - ii. Select `udf gas::libudf` in the Mass drop-down list.
2. Set the boundary conditions for heat.
 - (a) In the Boundary Conditions panel, change the Phase to mixture and click Set....
 - (b) Under Thermal Conditions, select Temperature and specify a value of 510 k.

This indicates 10 K superheat with respect to the saturation temperature 500 K.
3. Set the boundary conditions for the outlet.
 - (a) In the Boundary Conditions panel, retain mixture for Phase and click Set....
 - (b) Enter 500 k for Backflow Total Temperature.

This prevents gas from entering the outlet when reversed flow occurs. It also sets the saturation temperature for the liquid in case reversed flow occurs.
 - (c) In the Boundary Conditions panel, change the Phase to fluid and click Set....
 - (d) Enter 1 for Backflow Volume Fraction.

Step 8: Solution

1. Set the surface tension calculation options using the following TUI commands:

You may need to press the <Enter> key to get the > prompt.

```
> solve
/solve> set
/solve/set> st
Use node based smoothing
[no]

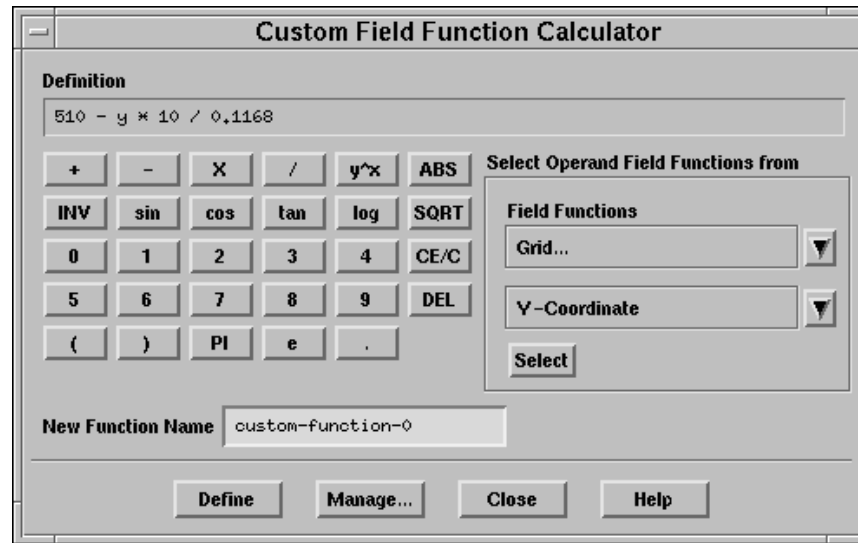
Number of smoothings
[1]

Smoothing relaxation Factor
[1]

use vof gradients at the nodes for curvature calcualtion? [yes] no
/solve/set>
```

2. Set the solution parameters.
 - (a) Set the Under-Relaxation Factors for Pressure and Momentum to 0.5.

- (b) Under Discretization, select PRESTO! for Pressure, and QUICK for Momentum and Energy.
 - (c) Under Pressure-Velocity Coupling, select PISO.
3. Initialize the solution.
 - (a) Initialize with pressure and velocity components at zero.
 - (b) Set the fluid Volume Fraction to 1.
 - (c) Set the value for Temperature to 500 k.
 - (d) Click Init and close the panel.
4. Define a custom field function for initializing the temperature profile.
 - (a) Define the custom field function, $510 - y * 10 / 0.1168$.



Select y by selecting Grid... and Y-Coordinate in the Field Functions drop-down lists.

This function is needed to initialize the temperature profile, ranging from 510 K at $y=0$ (for heat wall) to 500 K at $y=0.1168$ m (for outlet).

5. Patch a temperature using custom-function-0 for the fluid zone.
 - (a) Under Variable, select Temperature.
 - (b) Enable Use Field Function, and under Field Function, select custom-function-0.
 - (c) Under Zones to Patch, select fluid, and click Patch.
6. Monitor the solution during iteration.
 - (a) Enable the plotting of residuals.

- (b) Define a surface monitor.
 - i. Increase the number of Surface Monitors to 1.
 - ii. Enable Plot, Print, and Write.
 - iii. Under Every, select Time Step and click Define... to open the Define Surface Monitor panel.
 - A. Select Wall Fluxes... and Surface Nusselt Number in the Report of drop-down lists.
 - B. Under Surfaces, select heat.
 - C. Select Area-Weighted Average in the Report Type drop-down list.
 - D. Select Flow Time in the X Axis drop-down list.
 - E. Under File Name, enter `nusselt-1.out`.
- (c) Define a volume monitor.
 - i. Increase the number of Volume Monitors to 1.
 - ii. Enable Plot, Print, and Write.
 - iii. Under Every, select Time Step and click Define... to open the Define Volume Monitor panel.
 - A. Select Phases... and Volume fraction in the Field Variable drop-down lists.
 - B. Select gas in the Phase drop-down list.
 - C. Under Cell Zones, select fluid.
 - D. Select Volume-Average in the Report Type drop-down list.
 - E. Select Flow Time in the X Axis drop-down list.
 - F. Under File Name, enter `vol-mon-1.out`.
- 7. Set Reference Values for Length to 0.0778 m and Temperature to 500 k.
Report → Reference Values...
- 8. Enable autosaving of the data file after every 100 time steps.
File → Write → Autosave...
 - (a) Set the Autosave Data File Frequency to 100.
- 9. Start the calculation.
 - (a) Set Time Step Size to 0.001.
 - (b) Set the Number of Time Steps to 4000.
 - (c) Click Iterate.

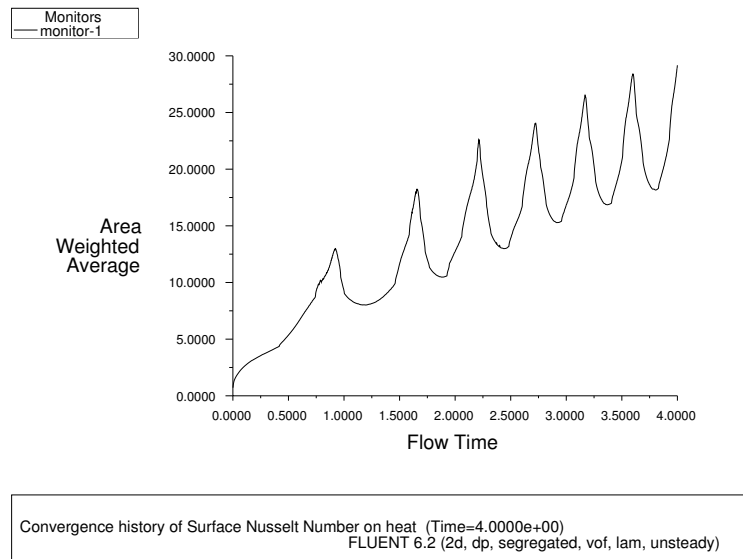


Figure 3: Convergence History of Surface Nusselt Number

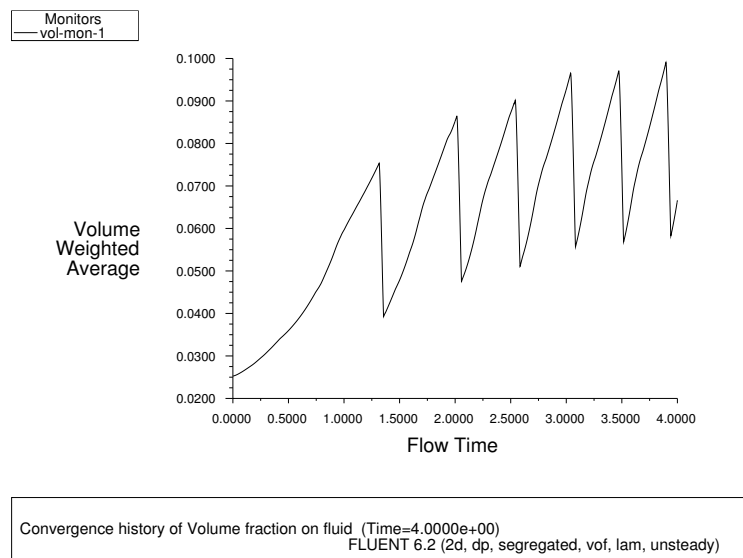


Figure 4: Convergence History of Volume Fraction

Step 9: Postprocessing

1. Display filled contours of volume fraction of gas after 900 time steps ($t = 0.9s$). See Figure 5.

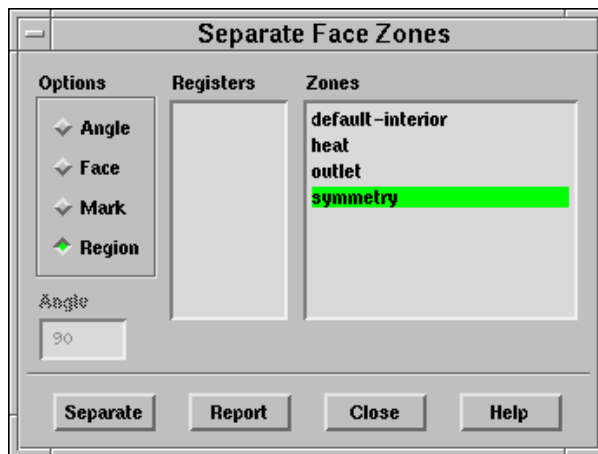
To display the contours, you should read the data file corresponding to 900 time steps ($t = 0.9s$).

- (a) Under Options, select Filled.
- (b) Decrease Levels to 2.
- (c) Select Phases... and Volume fraction in the Contours of drop-down lists.
- (d) Select gas in the Phase drop-down list and click Display.

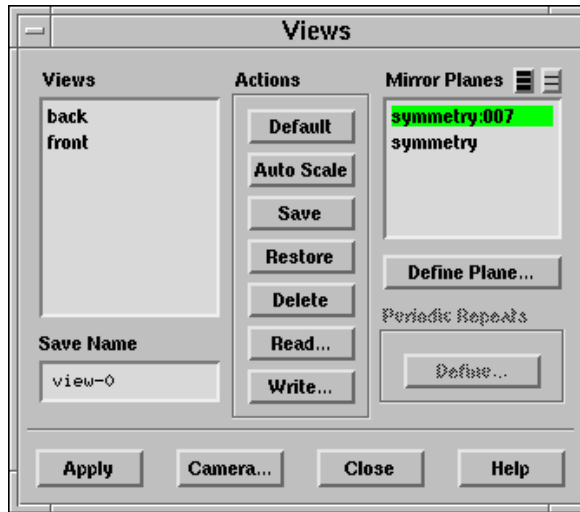
The contour display in Figure 5 shows a mirrored view across the symmetry boundary. To obtain such a view, separate the single symmetry zone into two symmetry zones and use one of these zones to mirror the display.

- (e) Separate the symmetry zone into two symmetry zones.

Grid → Separate → Faces...



- i. Under Zones, select symmetry and under Options, select Region.
 - ii. Click Separate.
- (f) In the Views panel, under Mirror Planes, select symmetry:007 and click Apply to mirror the view.



2. Display filled contours of volume fraction of gas after 1100 time steps ($t = 1.1s$). See Figure 6.
3. Display filled contours of user-defined memory after 2000 time steps ($t = 2.0s$). See Figure 7.
 - (a) Increase Levels to 9.
 - (b) Select User Defined Memory... and User Memory 1 in the Contours of drop-down lists and click Display.
4. Display contours of static temperature after 2000 time steps ($t = 2.0s$). See Figure 8.
 - (a) Under Options, deselect Filled.
 - (b) Select Temperature... and Static Temperature in the Contours of drop-down lists and click Display.
5. Display contours of volume fraction of fluid after 2000 time steps ($t = 2.0s$). See Figure 9.

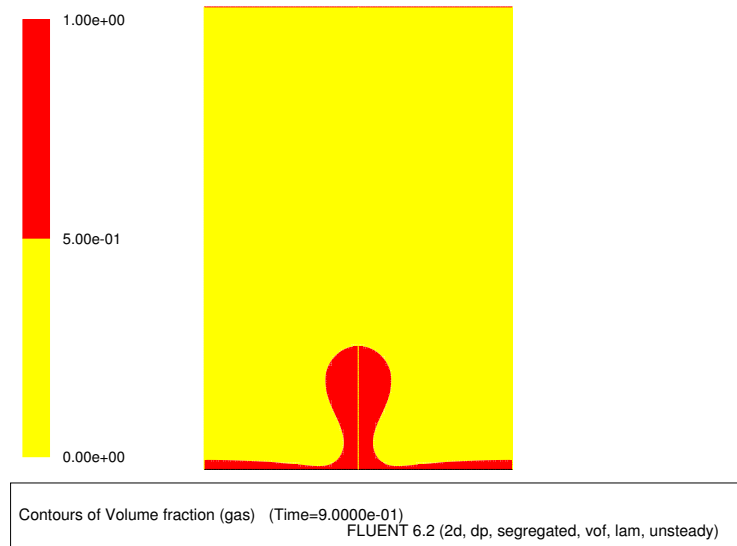


Figure 5: Contours of Gas Volume Fraction at $t = 0.9\text{s}$

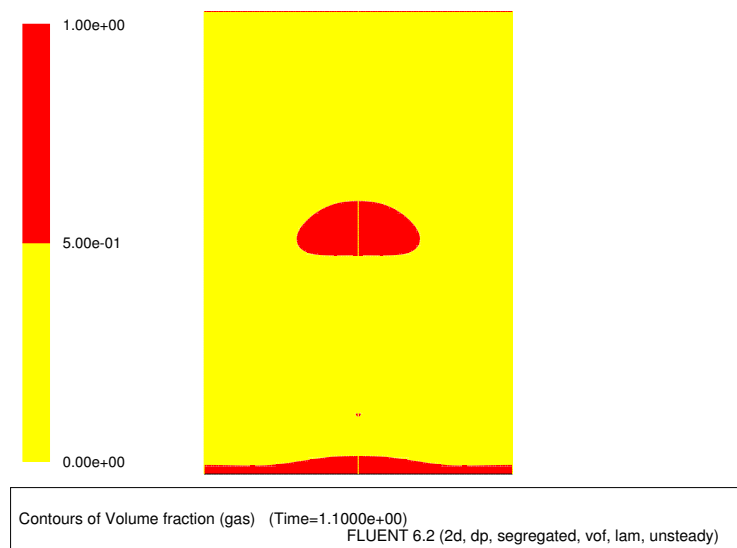


Figure 6: Contours of Gas Volume Fraction at $t = 1.1\text{s}$

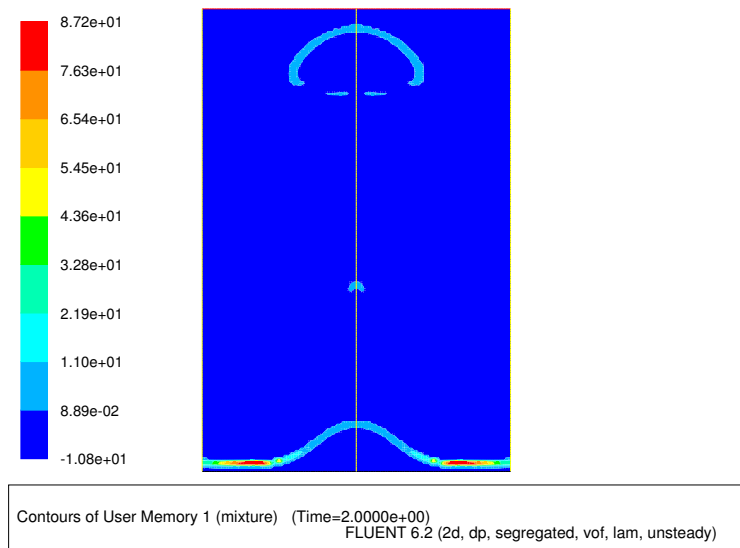


Figure 7: Contours of User-Defined Memory at $t = 2.0s$

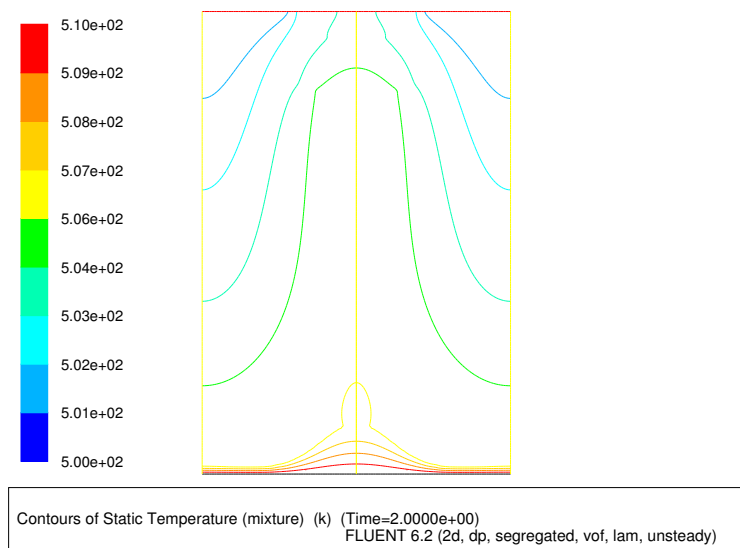


Figure 8: Contours of Static Temperature at $t = 2.0s$

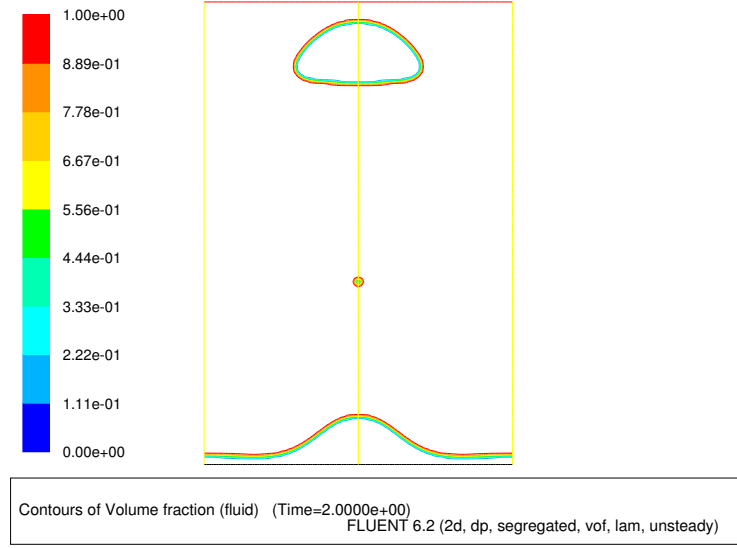


Figure 9: Contours of Fluid Volume Fraction at $t = 2.0s$

Results

This problem requires a UDF for implementing mass exchange source terms in mass equations for each phase (liquid and vapor). It also requires a source term in the energy equation, to account for latent heat transfer.

The general form of the mass source in the vapor phase is:

$$S_{\alpha_s} = -\frac{(q_g'' - q_l'') \cdot \nabla \alpha_1}{L} \quad (1)$$

Here, q'' is the heat flux across the interface per unit area of the interface, the subscripts g and l refer to vapor and liquid, respectively, and L is the latent heat.

As a first order approximation, the heat flux difference is represented as:

$$(q_g'' - q_l'') \approx -(\alpha_l k_l + \alpha_g k_g) \nabla T \quad (2)$$

In this approximation, Equation 1 becomes,

$$S_{\alpha_g} = \frac{(k_l \alpha_l + k_g \alpha_g)(\nabla T \cdot \nabla \alpha_l)}{L} \quad (3)$$

Since there is no internal mass source, mass source for liquid phase becomes:

$$S_{\alpha_l} = S_{\alpha_g} \quad (4)$$

Latent heat source for the energy equation becomes:

$$S_E = -S_{\alpha_g} \cdot L \quad (5)$$

Interfacial properties include surface tension $0.1N/m$, latent heat $1e^5 J/kg$, saturation temperature $T_{sat} = 500K$. The length scale of the problem is the most dangerous wavelength of Taylor-Raleigh instability:

$$\lambda_0 = 2\pi \left(\frac{3\sigma}{(\rho_l - \rho_g)g_y} \right)^{1/2} = 0.0787m \quad (6)$$

The velocity scale of the problem is:

$$\sqrt{g_y \lambda_0} = 0.878m/s \quad (7)$$

Hence the time scale will be:

$$\sqrt{\lambda_0/g_y} = 0.09s \quad (8)$$

The domain horizontal width is $\lambda_0/2$ and the vertical height is $3\lambda_0/2$. The mesh resolution is $64(hor) \times 192(ver)$. The initial shape of the vapor-liquid interface must be perturbed to initiate bubble growth. Therefore, there is another initialization UDF which fills with gas all the cells satisfying the following condition:

$$y \leq 0.00292 + 0.0006 \cos(2\pi x/\lambda_0) \quad (9)$$

Here $x(y)$ is horizontal (vertical) coordinate in meters.

Nusselt number is an important dimensionless group characterizing boiling heat transfer and is defined as:

$$Nu = \frac{|q''| \lambda_0}{k_l(T_{wall} - T_{sat})} \quad (10)$$

Since the time scale of this problem is 0.1s, the time step is 0.001, i.e., 100 time steps resolution. In all, the problem should run for about 1200 time steps to capture the first bubble emission.

Postprocessing the results will show void fraction and temperature profiles at 900th and 1100th time steps, corresponding to 0.9 and 1.1 seconds. These times correspond to the minimum and maximum thickness of the boiling film at the heated wall. They correspond to minimum and maximum values of Nusselt number, respectively.

Summary

Application of the VOF model in a film boiling regime has been demonstrated in this tutorial. Also, UDFs have been used to enhance the standard features of FLUENT.

- `DEFINE_ADJUST` (area_density) is a general purpose macro that can be used to adjust or modify FLUENT variables. In this tutorial, it calculates the dot product and stores this value in Equation 3.

For information, see [Section 4.2.1: DEFINE_ADJUST](#) in the FLUENT 6.2 UDF Manual.

- `DEFINE_SOURCE` (gas) uses `UDMI(0)` and the user input for `L` to compute the mass source term (Equation 3) for the gas phase and stores it in `UDMI(1)`. This macro also computes the energy source (Equation 5) and stores it in `UDMI(2)`.

For information, see [Section 4.3.17: DEFINE_SOURCE](#) in the FLUENT 6.2 UDF Manual.

- `DEFINE_SOURCE` (liquid) assigns `UDMI(1)` with a negative sign as a liquid mass source in accordance with Equation 4.
- `DEFINE_SOURCE` (energy) assigns `UDMI(2)` as a latent heat source.
- `DEFINE_INIT` (my_init_function) initializes the gas void fraction in accordance with Equation 7.

For information, see [Section 4.2.5: DEFINE_INIT](#) in the FLUENT 6.2 UDF Manual.