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 to advise 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 23.3: Volume of Fluid (VOF) Model in the FLUENT 6.3 User's Guide and Tutorial 17: Using the VOF Model in the FLUENT 6.3 Turorial Guide. If you have not used UDFs, refer to the FLUENT 6.3 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 10 K above saturation. Initially, the linear temperature profile (from T_{wall} to T_{sat} in the positive Y direction) is patched on the liquid domain.

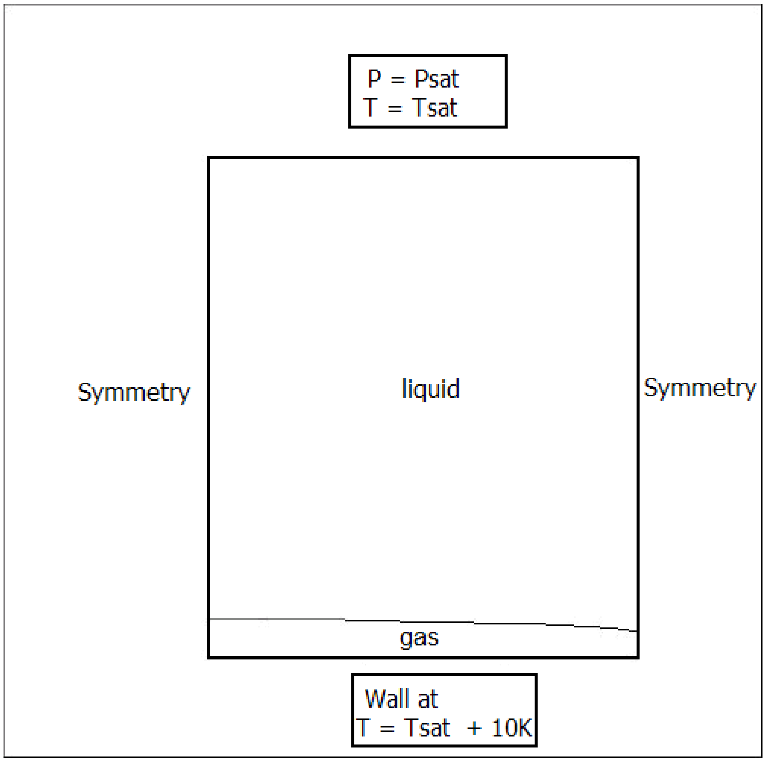


Figure 1: Problem Schematic

Preparation

1. Copy the files, test-2d.msh and boiling.c to your working folder.
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).

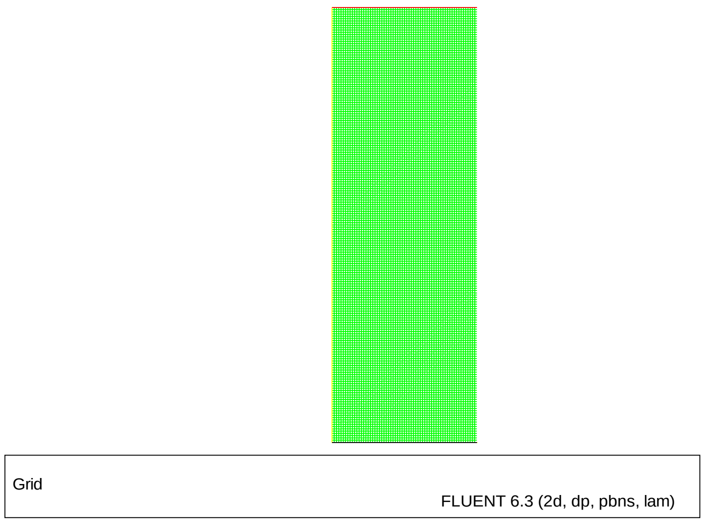


Figure 2: Grid Display

Step 2: Models

1. Define the solver settings.
   (a) Select Unsteady from the Time drop down list.
   (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

Define −→Phases...

1. Define the primary (vapor) phase.
   (a) Select phase-1 from Phase drop-down list and click the Set... button to open the Primary Phase panel.
      i. Select vapor from the Phase Material drop-down list.
      ii. Enter gas for Name.
      iii. Click OK to close the Primary Phase panel.
2. Similarly, define the secondary (liquid) phase, specifying liquid as the Phase Material and fluid for Name.
3. Specify the interphase interaction.
   (a) Click the Interaction... button to open the Phase Interaction panel.
      i. Click the Surface Tension tab.
      ii. Select constant from the drop-down list for Surface Tension Coefficients and enter a value of 0.1 N/m.
      iii. Click OK to close the Phase Interaction panel.

Step 5: Operating Conditions

Define −→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.

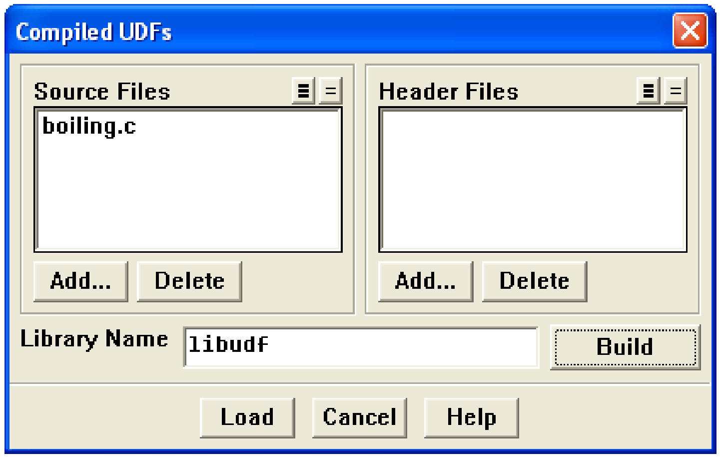
Step 6: User-Defined Function

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 W/m3 and is prescribed for mixture energy equation.*

For more information on compiled UDFs, refer to Section 8: Compiling UDF Source Files in the FLUENT 6.3 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 folder that contains your case and data files.

   (a) Click the Add... button in the Source Files Group Box to open the Select File Dialog Box.

   (b) Select (boiling.c) and click OK to close the Select File Dialog Box.

   (c) 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 the shared library to the FLUENT executable.

2. Define function hooks.

   Define −→ User-Defined −→Function Hooks...