Created in COMSOL Multiphysics 6.0

Heat Transfer by Free Convection

Introduction

This example describes a fluid flow problem with heat transfer in the fluid. An array of heating tubes is submerged in a vessel with fluid flow entering at the bottom. Figure 1 shows the setup.

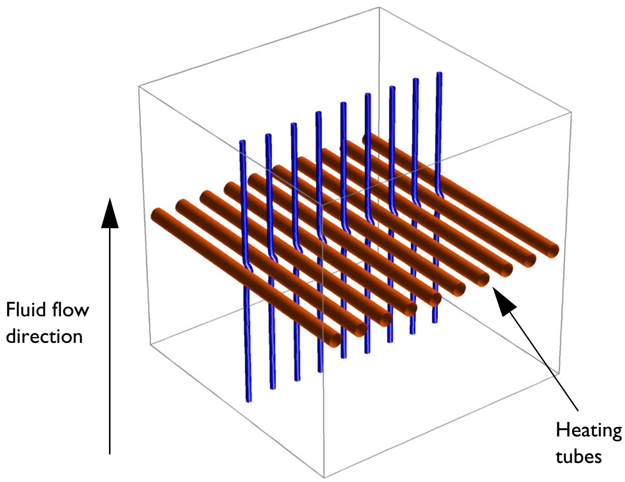

Figure 1: Heating tubes and direction of the fluid flow

Model Definition

The first consideration when modeling should always be the true dimension of the problem. Sometimes there are no variations in the third dimensions, and it can be extrapolated from the solution of a related 2D case. By neglecting any end effects from the walls of the vessel, the solution is constant in the direction of the heating tubes. You can therefore reduce the model to a 2D domain.

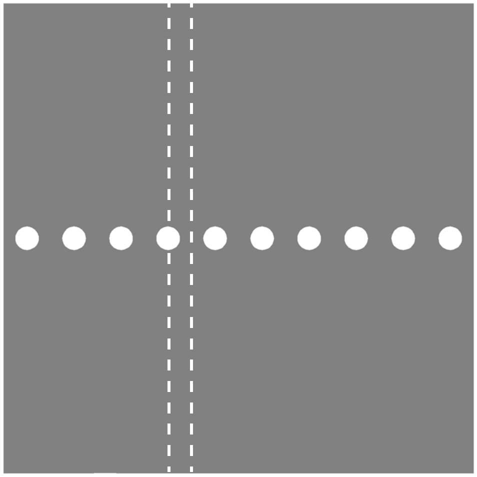

Figure 2: Using symmetry to reduce computation time and complexity. The model describes one section of the array of heating tubes (indicated by the dashed lines).

The next step is finding symmetries. In this case, using symmetry planes, it suffices to model the thin domain indicated in Figure 2.

GOVERNING EQUATIONS

This is a multiphysics model because it involves fluid dynamics coupled with heat transfer. The pressure p and the velocity fields u and v are the solution of the Navier-Stokes equations, while the temperature T is solved through the heat equation. These variables are all related through bidirectional multiphysics couplings.

Results

The analysis of the coupled thermal-fluid model provides the velocity field, pressure distribution, and temperature distribution in the fluid. Figure 3 shows a plot of the velocity field and the temperature.

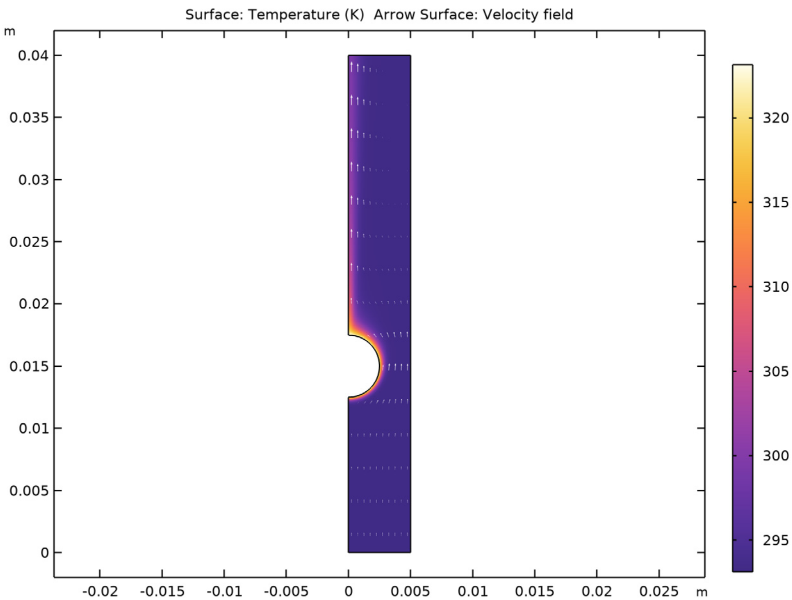

Figure 3: The velocity field and temperature distribution in the fluid.

Using integration to find the mean temperature at the outlet shows that the temperature increases roughly by 1 K from the inlet to the outlet.

Notes About the COMSOL Implementation

To build a model in COMSOL Multiphysics using the above equations, use two physics interfaces: the Laminar Flow interface for laminar single-phase fluid flow and the Heat Transfer in Fluids interface for heat transfer.

In this model, the equations are coupled in both directions. The buoyancy force lifting the fluid is entered in the compressible Navier-Stokes equations via a force term **F** depending on temperature through the density. At the same time, the heat equation accounts for convective heat transfer.

Use the Nonisothermal Flow multiphysics node to automatically use the temperature field computed in Heat Transfer in Fluids in Laminar Flow and the velocity and pressure from Laminar Flow into Heat Transfer in Fluids.

Application Library path: `COMSOL_Multiphysics/Multiphysics/free_convection`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Fluids (ht)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies>Stationary**.

8 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
v_in	5[mm/s]	0.005 m/s	Inlet velocity
T_in	20[degC]	293.15 K	Inlet temperature
T_heat	50[degC]	323.15 K	Heater temperature

GEOMETRY 1

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.005`.

4 In the **Height** text field, type `0.04`.

5 Click **Build All Objects**.

Circle 1 (c1)

1 In the **Geometry** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `0.0025`.

4 Locate the **Position** section. In the **y** text field, type `0.015`.

5 Click **Build All Objects**.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **r1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.

5 Select the object **c1** only.

6 Click **Build All Objects**.

DEFINITIONS

Define a nonlocal coupling for computing average values over the outlet.

Average 1 (aveop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 4 only.

5 In the **Operator name** text field, type `avgout`.

Using this coupling, define a variable, `DeltaT`, for the temperature rise from inlet to outlet.

Variables 1

1 In the **Definitions** toolbar, click **Local Variables**.
2 In the **Settings** window for **Variables**, locate the **Variables** section.
3 In the table, enter the following settings:

Name	Expression	Unit	Description
DeltaT	avgout(T)-T_in	K	Temperature rise

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2 Go to the **Add Material** window.
3 In the tree, select **Built-in>Water, liquid**.
4 Click **Add to Component** in the window toolbar.
5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

Add a **Nonisothermal flow** multiphysics node to set up the velocity in heat transfer and to account for the multiphysics stabilization.

MULTIPHYSICS

Nonisothermal Flow 1 (nitf1)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain>Nonisothermal Flow**.

LAMINAR FLOW (SPF)

Since the water density is temperature dependent, the flow is set to weakly compressible.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.
2 In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.
3 From the **Compressibility** list, choose **Weakly compressible flow**.

Symmetry 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
2 Select Boundaries 1, 3, and 5 only.

Inlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2 Select Boundary 2 only.
3 In the **Settings** window for **Inlet**, locate the **Velocity** section.

4 In the U_0 text field, type v_in.

Outlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 Select Boundary 4 only.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the p text field, type spf.rhoref*g_const*(0.04[m]-y).

Volume Force 1

1 In the **Physics** toolbar, click **Domains** and choose **Volume Force**.

2 Select Domain 1 only.

3 In the **Settings** window for **Volume Force**, locate the **Volume Force** section.

4 Specify the **F** vector as

0	x
-g_const*spf.rho	y

Show advanced physics options as follows, and enable pseudo time stepping for the laminar flow equation for better convergence.

5 Click the **Show More Options** button in the **Model Builder** toolbar.

6 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

7 Click **OK**.

8 In the **Model Builder** window, click **Laminar Flow (spf)**.

9 In the **Settings** window for **Laminar Flow**, click to expand the **Advanced Settings** section.

10 Find the **Pseudo time stepping** subsection. From the **Use pseudo time stepping for stationary equation form** list, choose **On**.

HEAT TRANSFER IN FLUIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Fluids (ht)**.

Temperature 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Temperature**, locate the **Temperature** section.

4 In the T_0 text field, type `T_in`.

Temperature 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundaries 6 and 7 only.

3 In the **Settings** window for **Temperature**, locate the **Temperature** section.

4 In the T_0 text field, type `T_heat`.

Outflow 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outflow**.

2 Select Boundary 4 only.

Symmetry 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 Select Boundaries 1, 3, and 5 only.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the T text field, type `T_in`.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.

STUDY 1

In the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht)

The default plots visualize the velocity, pressure, and temperature fields. To reproduce the plots in the Figure 3, modify the temperature plot.

Arrow Surface 1

1 In the **Model Builder** window, right-click **Temperature (ht)** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, locate the **Arrow Positioning** section.

3 Find the **x grid points** subsection. In the **Points** text field, type `10`.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.

5 In the **Temperature (ht)** toolbar, click **Plot**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

Finally, evaluate the temperature rise.

Global Evaluation 1

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>DeltaT - Temperature rise - K**.

3 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The value should be close to 1 K.