Model created in COMSOL Multiphysics 6.4

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.

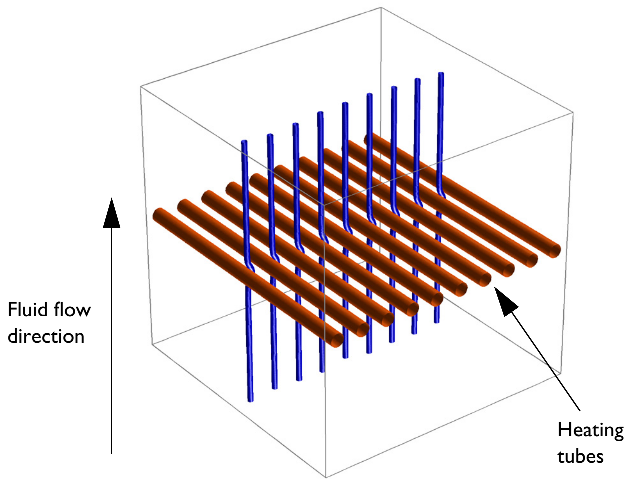

Figure 1: Heating tubes and direction of the fluid flow.

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.

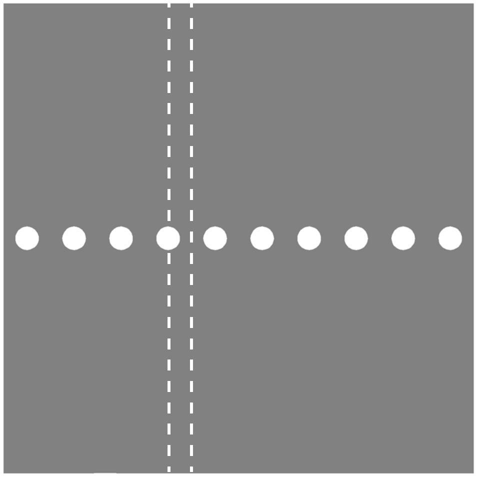

Figure 2: Using symmetry to reduce computation time and complexity. The model describes one section of the array of heating tubes (indicated by the dashed lines).

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.

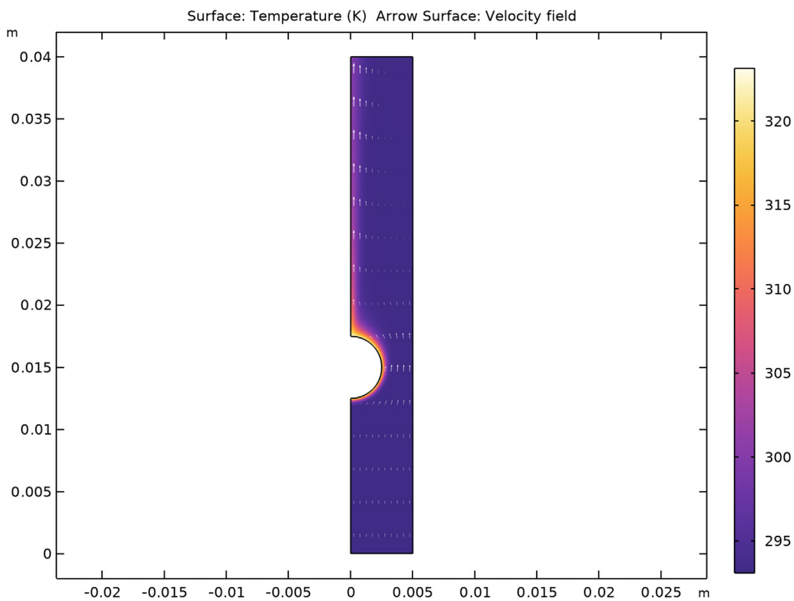

Figure 3: The velocity field and temperature distribution in the fluid.

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, the Nonisothermal Laminar Flow interface is used to simulate the coupling between heat transfer and fluid flow.

It combines the Heat Transfer in Fluids and Laminar Flow interfaces. The Nonisothermal Flow multiphysics coupling is automatically added. It couples the heat transfer and flow interfaces and provides options to include flow heating in the model.

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** > **Nonisothermal Flow** > **Laminar Flow**.

3 Click **Add**.

4 Click **Study**.

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

6 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 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

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 **Materials** 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 the **Add to Component** button in the window toolbar.

5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

LAMINAR FLOW (SPF)

1 In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.

2 Select the **Include gravity** checkbox.

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

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)`.

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

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

5 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Advanced Physics Options**.

6 Click **OK**.

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

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

9 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 **Study** 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

1 Go to the **Table 1** window.

The value should be close to 1 K.