

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.

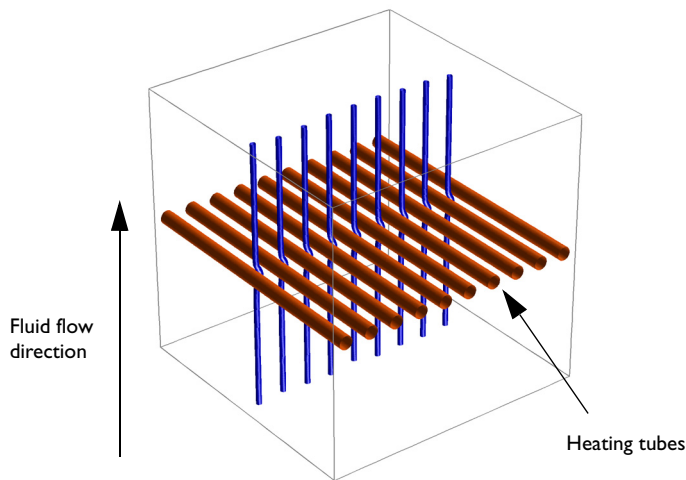


Figure 1: Heating tubes and direction of the fluid flow

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. Neglecting any end effects from the walls of the vessel, the solution is constant in the direction of the heating tubes, therefore you can reduce the model to a 2D domain.

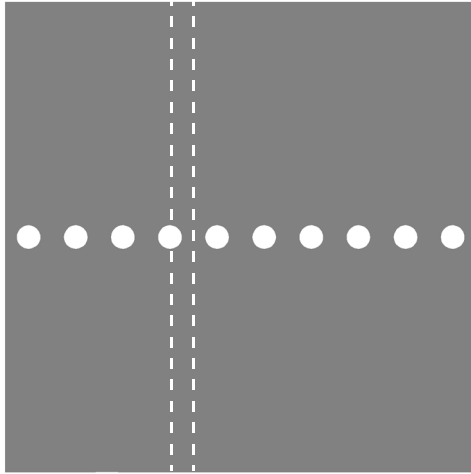


Figure 2: Using symmetry to reduce computation time and complexity. The model describes one section of the array of heating tubes (indicated by the dashed lines).

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 more than one kind of physics. The incompressible Navier-Stokes equations from fluid dynamics work together with a heat transfer equation. There are four unknown field variables (dependent variables):

- The velocity field components, u and v
- The pressure, p
- The temperature, T

They are all related through bidirectional multiphysics couplings.

The incompressible Navier-Stokes equations consist of a momentum balance (a vector equation) and a mass conservation and incompressibility condition:

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \eta \nabla^2 \mathbf{u} + \mathbf{F}$$

$$\nabla \cdot \mathbf{u} = 0$$

Here

- \mathbf{u} is the velocity field.
- p is the pressure.
- \mathbf{F} is a volume force.
- ρ is the fluid density.
- η is the dynamic viscosity.
- ∇ is the vector differential operator.

The heat equation is an energy conservation equation that says that the change in energy is equal to the heat source minus the divergence of the diffusive heat flux:

$$\rho C_p \left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T \right) + \nabla \cdot (-k \nabla T) = Q$$

where C_p is the heat capacity of the fluid and ρ is fluid density. Q represents a source term. The velocity field comes from the incompressible Navier-Stokes equation.

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. Without heating, you would expect an exit y -velocity that is slightly lower toward the left side, behind the heating tube (wake

effect). In this case, however, you see that the y -velocity is higher on the left side. This is because of the buoyancy effect of the free convection.

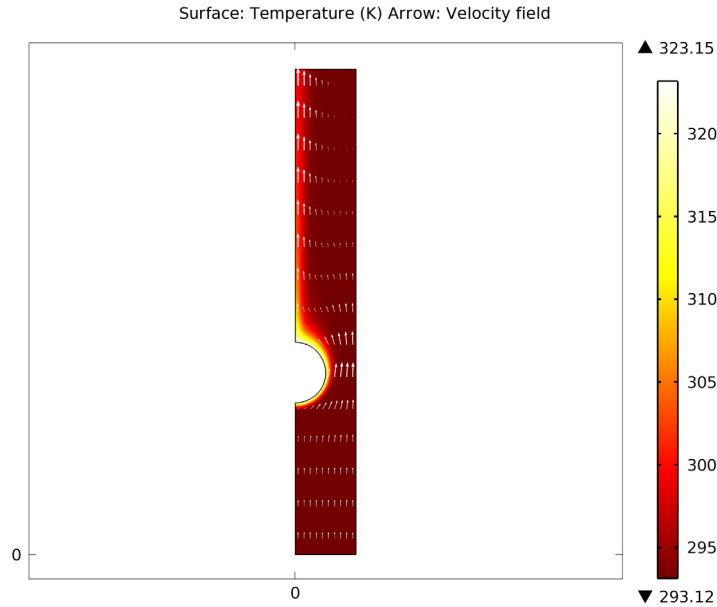


Figure 3: The velocity field and temperature distribution in the fluid.

Using integration to find the mean temperature at the outlet shows that the temperature increases roughly 1.1 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 interlace: the Laminar Flow interface for laminar single-phase fluid flow and the Heat Transfer interface for heat transfer.

In this model, the equations are coupled in both directions. First you add free convection to the fluid flow with the *Boussinesq approximation*. This approximation ignores variations in density with temperature, except that the variations give rise to a buoyancy force lifting the fluid. This force enters the \mathbf{F} term in the incompressible Navier-Stokes equations.

At the same time, the heat equation must account for the velocity field. The velocity field from the Laminar Flow appears automatically as a predefined option in the model input for the velocity field that determines the convective heat transfer.

Model Library path: COMSOL_Multiphysics/Multiphysics/free_convection

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click the **2D** button.
- 3 Click **Next**.
- 4 In the **Add physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.
- 5 Click **Add Selected**.
- 6 In the **Add physics** tree, select **Heat Transfer>Heat Transfer in Fluids (ht)**.
- 7 Click **Add Selected**.
- 8 Click **Next**.
- 9 In the **Studies** tree, select **Preset Studies for Selected Physics>Stationary**.
- 10 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2 Go to the **Settings** window for Parameters.
- 3 Locate the **Parameters** section. In the **Parameters** table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
v_in	5[mm/s]	Inlet velocity
T_in	20[degC]	Inlet temperature
T_heat	50[degC]	Heater temperature
alpha0	0.18e-3[1/K]	Thermal expansion coefficient

GEOMETRY I

Rectangle 1

- 1 In the **Model Builder** window, right-click **Model I>Geometry I** and choose **Rectangle**.
- 2 Go to the **Settings** window for Rectangle.

- 3 Locate the **Size** section. In the **Width** edit field, type 0.005.
- 4 In the **Height** edit field, type 0.04.
- 5 Click the **Build All** button.

Circle 1

- 1 In the **Model Builder** window, right-click **Geometry 1** and choose **Circle**.
- 2 Go to the **Settings** window for Circle.
- 3 Locate the **Size and Shape** section. In the **Radius** edit field, type 0.0025.
- 4 Locate the **Position** section. In the **y** edit field, type 0.015.
- 5 Click the **Build All** button.

Difference 1

- 1 In the **Model Builder** window, right-click **Geometry 1** and choose **Boolean Operations>Difference**.
- 2 Go to the **Settings** window for Difference.
- 3 Locate the **Difference** section. Under **Objects_to_add**, click **Activate Selection**.
- 4 Select the object **r1** only.
- 5 Under **Objects_to_subtract**, click **Activate Selection**.
- 6 Select the object **c1** only.
- 7 Click the **Build All** button.

DEFINITIONS

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

Average 1

- 1 In the **Model Builder** window, right-click **Model 1>Definitions** and choose **Model Couplings>Average**.
- 2 Go to the **Settings** window for Average.
- 3 Locate the **Source Selection** section. From the **Geometric entity level** list, select **Boundary**.
- 4 Select Boundary 4 only.
- 5 Locate the **Operator Name** section. In the **Operator name** edit field, type avgout.
Using this operator, define a variable, ΔT , for the temperature rise from inlet to outlet.

Variables 1

- 1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.
- 2 Go to the **Settings** window for Variables.
- 3 Locate the **Variables** section. In the **Variables** table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
DeltaT	avgout(T) - T_in	Temperature rise

MATERIALS

- 1 In the **Model Builder** window, right-click **Model 1 > Materials** and choose **Open Material Browser**.
- 2 Go to the **Material Browser** window.
- 3 Locate the **Materials** section. In the **Materials** tree, select **Built-In > Water, liquid**.
- 4 Right-click and choose **Add Material to Model** from the menu.

LAMINAR FLOW*Symmetry 1*

- 1 In the **Model Builder** window, right-click **Model 1 > Laminar Flow** and choose **Symmetry**.
- 2 Select Boundaries 1, 3, and 5 only.

Inlet 1

- 1 In the **Model Builder** window, right-click **Laminar Flow** and choose **Inlet**.
- 2 Select Boundary 2 only.
- 3 Go to the **Settings** window for Inlet.
- 4 Locate the **Velocity** section. In the U_0 edit field, type v_{in} .

Outlet 1

- 1 In the **Model Builder** window, right-click **Laminar Flow** and choose **Outlet**.
- 2 Select Boundary 4 only.

Initial Values 1

- 1 In the **Model Builder** window, click **Initial Values 1**.
- 2 Go to the **Settings** window for Initial Values.

3 Locate the **Initial Values** section. Specify the \mathbf{u} vector as

0	x
v_in	y

Volume Force 1

1 In the **Model Builder** window, right-click **Laminar Flow** and choose **Volume Force**.

2 Select Domain 1 only.

3 Go to the **Settings** window for Volume Force.

4 Locate the **Volume Force** section. Specify the \mathbf{F} vector as

0	x
g_const*spf.rho*alpha0*(T-T_in)	y

Note that the constant temperature expansion coefficient α_0 is valid only in a region near $T = T_0$. To simulate large temperature differences, you would need to use a temperature-dependent expression, $\alpha(T)$.

HEAT TRANSFER

Heat Transfer in Fluids 1

1 In the **Model Builder** window, expand the **Model 1 > Heat Transfer** node, then click **Heat Transfer in Fluids 1**.

2 Go to the **Settings** window for Heat Transfer in Fluids.

3 Locate the **Model Inputs** section. From the p list, select **Pressure (spf/fp1)**.

4 From the \mathbf{u} list, select **Velocity field (spf/fp1)**.

Temperature 1

1 In the **Model Builder** window, right-click **Heat Transfer** and choose **Temperature**.

2 Select Boundary 2 only.

3 Go to the **Settings** window for Temperature.

4 Locate the **Temperature** section. In the T_0 edit field, type T_{in} .

Temperature 2

1 In the **Model Builder** window, right-click **Heat Transfer** and choose **Temperature**.

2 Select Boundaries 6 and 7 only.

3 Go to the **Settings** window for Temperature.

4 Locate the **Temperature** section. In the T_0 edit field, type T_{heat} .

Outflow 1

- 1 In the **Model Builder** window, right-click **Heat Transfer** and choose **Outflow**.
- 2 Select Boundary 4 only.

Initial Values 1

- 1 In the **Model Builder** window, click **Initial Values 1**.
- 2 Go to the **Settings** window for Initial Values.
- 3 Locate the **Initial Values** section. In the **T** edit field, type T_{in} .

STUDY 1

- 1 In the **Model Builder** window, right-click **Model 1>Mesh 1** and choose **Build All**.
- 2 Right-click **Study 1** and choose **Show Default Solver**.
- 3 Expand the **Study 1>Solver Configurations** node.

Solver 1

- 1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solver 1** node.
- 2 In the **Model Builder** window, expand the **Stationary Solver 1** node, then click **Fully Coupled 1**.
- 3 Go to the **Settings** window for Fully Coupled.
- 4 Click to expand the **Damping and Termination** section.
- 5 From the **Damping method** list, select **Automatic highly nonlinear**.
- 6 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS*Temperature (ht)*

The default plots show the velocity, the pressure, the temperature and temperature gradient on the surface. To reproduce the plots in the [Figure 3](#), modify 2D Plot Group 3.

- 1 In the **Model Builder** window, right-click **Results>Temperature (ht)** and choose **Arrow Surface**.
- 2 Go to the **Settings** window for Arrow Surface.
- 3 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** edit field, type 10.
- 4 In the upper-right corner of the **Expression** section, click **Replace Expression**.

- 5 From the menu, choose **Laminar Flow>Velocity field (u,v)**.
- 6 Locate the **Coloring and Style** section. From the **Color** list, select **White**.
- 7 Click the **Plot** button.
- 8 Click the **Zoom Extents** button on the Graphics toolbar.

Derived Values

Finally, evaluate the temperature rise.

- 1 In the **Model Builder** window, right-click **Results>Derived Values** and choose **Global Evaluation**.
- 2 Go to the **Settings** window for Global Evaluation.
- 3 In the upper-right corner of the **Expression** section, click **Replace Expression**.
- 4 From the menu, choose **Definitions>Temperature rise (DeltaT)**.
- 5 Click the **Evaluate** button.

The value of the temperature rise be close to 1.1 K.