Modeling Exercise

Define the physics for a model of heat transfer by free convection using the manual approach with predefined couplings

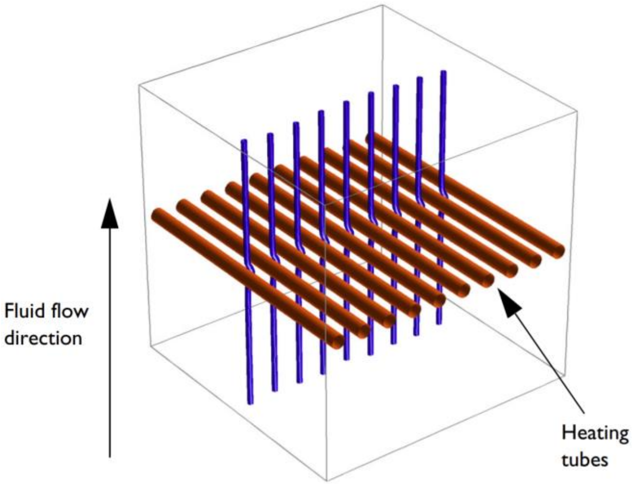
Introduction

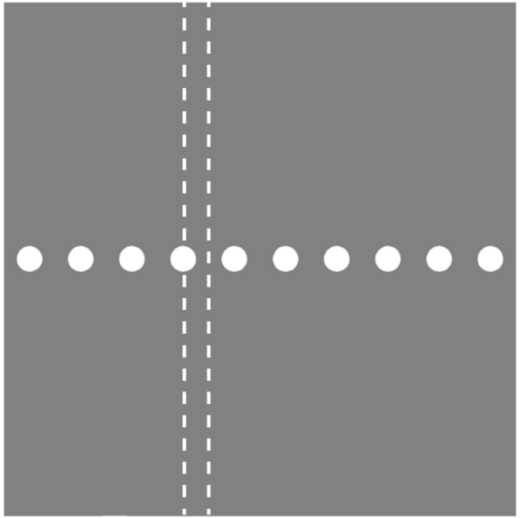
- This model exercise demonstrates the concept of multiphysics modeling in COMSOL Multiphysics®
- Define the physics for the model using the manual approach with predefined couplings
  - Run a single-physics simulation for the *Laminar Flow (spf)* interface, followed by a multiphysics simulation including the *Heat Transfer in Fluids* interface and *Nonisothermal Flow* multiphysics coupling for the nonisothermal flow
    - Enables more quickly and easily locating and resolving any errors that may have been made in the definition of the physics phenomena involved before computing the full multiphysics model
- Important information for setting up the model can be found in the model specifications
  - Refer to this when building the model

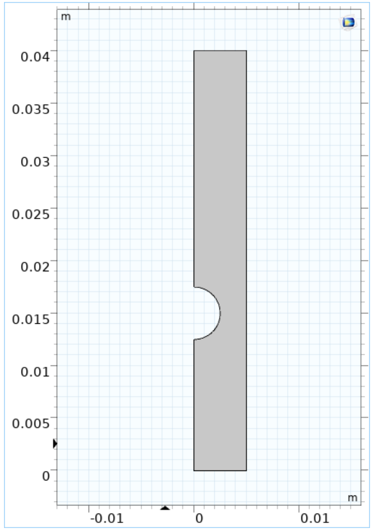
Model Overview

- An array of heating tubes are submerged in a vessel of water with the fluid entering from the bottom
  - The model is reduced from 3D to 2D and further simplified by exploiting symmetry due to the array
- As fluid enters the vessel and travels past the heating element, heat is transferred through convection
  - An instance of nonisothermal flow
- The buoyancy force lifting the fluid is incorporated through a force term that depends on the temperature through the density
  - Modeled through a *Volume Force* domain feature
- Results include the velocity field, pressure distribution, and temperature distribution

Model Overview



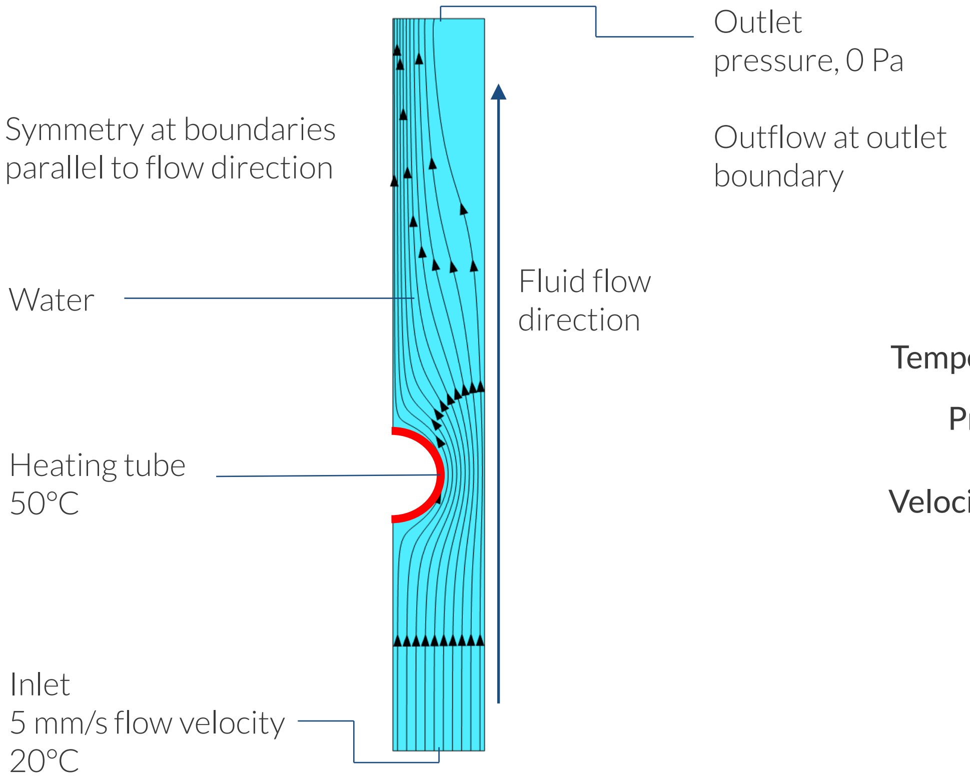




A cross section (center) of the 3D model geometry (left) is taken, and symmetry of the array is exploited to result in the model geometry (right)

COMSOL

Model Specifications



Volume force lifting the fluid:

x	0
y	-g_const*spf.rho

Initial conditions:

Temperature	T	20[degC]
Pressure	p	spf.rhoref*g_const*(0.04[m]-y)
Velocity field	u	0 m/s
	v	0 m/s

Details and specifications for the free convection model setup

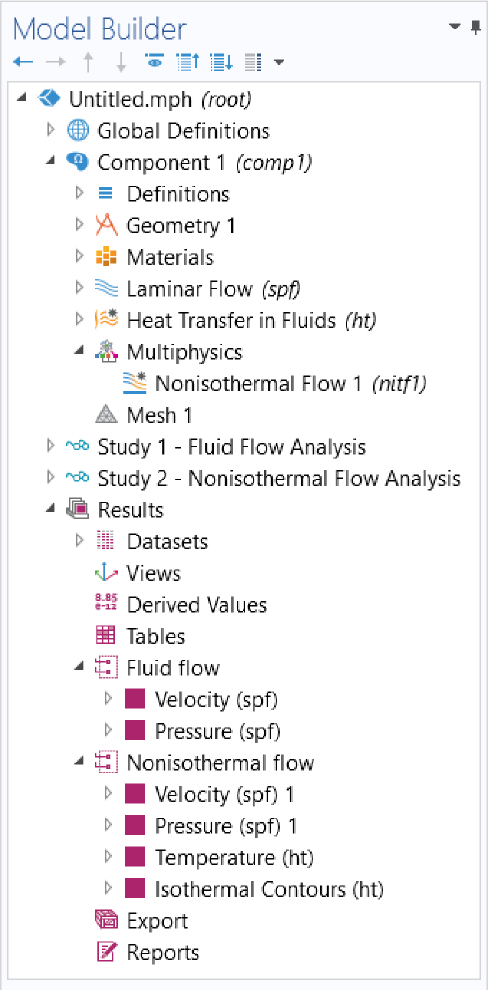
Manual with Predefined Couplings Approach

Define the physics for the model using predefined multiphysics couplings

Procedure:

1. Add the physics interface
2. Define the physics settings
3. Add the multiphysics couplings
   - Only applicable when multiple physics interfaces have been added
4. Compute the study
5. Check the results
6. Repeat steps 1–4 for each subsequent combination of physics

The model tree for the free convection tutorial model when the manual approach with predefined couplings has been used



COMSOL

Modeling Workflow

A general outline of the steps that can be used to set up, build, and compute this model to complete this modeling exercise is provided here:

Fluid Flow Analysis

1. Set up the model
   - Add 2D model component
2. Import geometry
3. Assign materials
4. Define the physics
   - Add *Laminar Flow (spf)* interface
5. Build the mesh
6. Run the study
   - Add *Stationary* study
7. Check the results

Nonisothermal Flow Analysis

1. Define the physics
   - Add *Heat Transfer in Fluids* interface
   - Add *Nonisothermal Flow* multiphysics coupling
2. Run the study
   - Add *Stationary* study
3. Check the results

* = Build the model using the procedure here and refer to the model specifications for values