SOLUTION

Modeling Exercise

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

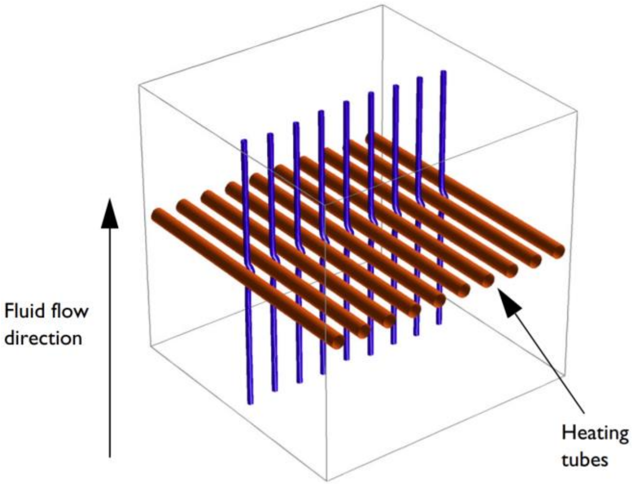
Introduction

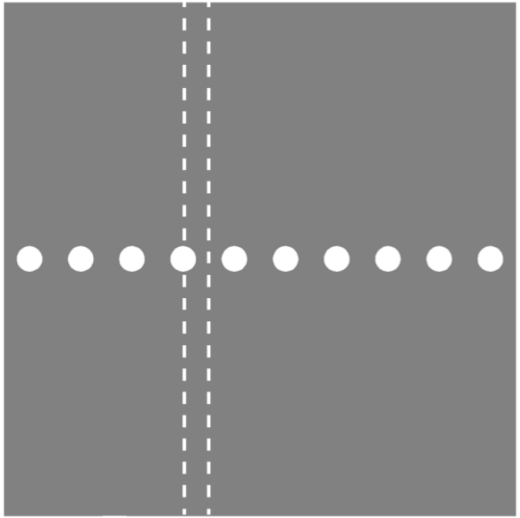
- This model exercise demonstrates the concept of multiphysics modeling in COMSOL Multiphysics®
- Define the physics for the model using the manual approach with predefined couplings
  - Run a single-physics simulation for the *Laminar Flow (spf)* interface, followed by a multiphysics simulation including 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

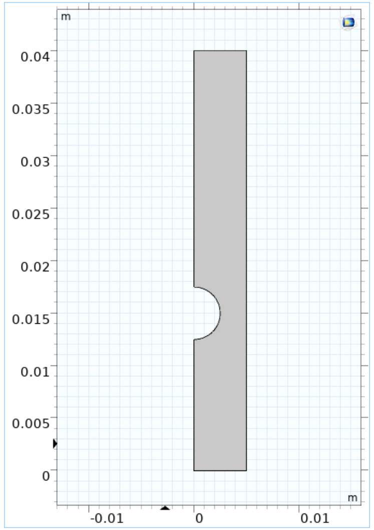
Model Overview

- 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

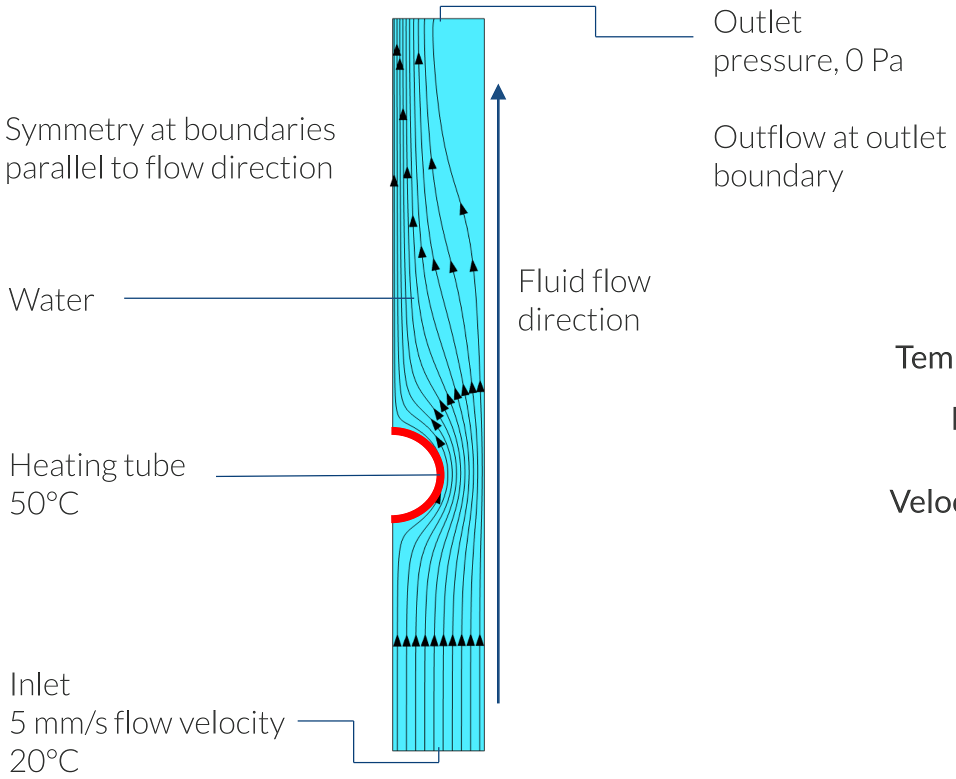






A cross section (center) of the 3D model geometry (left) is taken, and symmetry of the array is exploited to result in the model geometry (right)

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

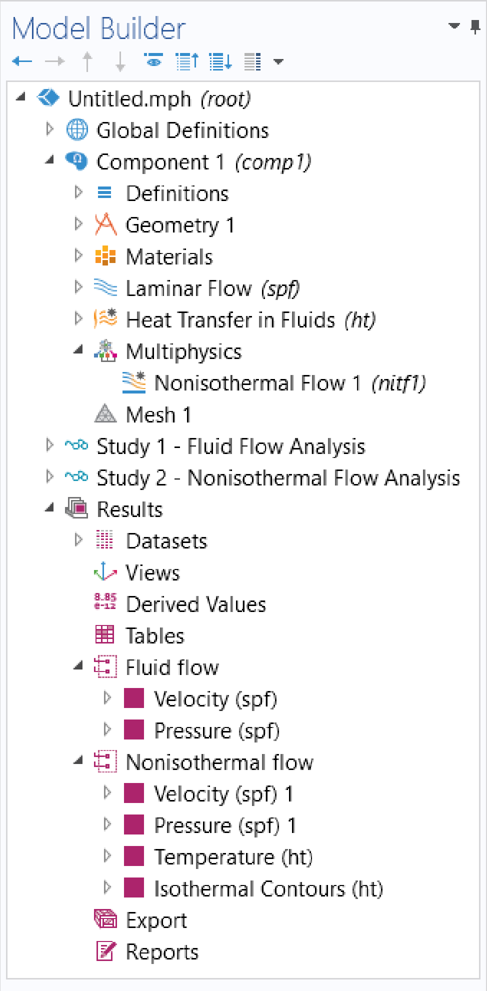
Details and specifications for the free convection model setup

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 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

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

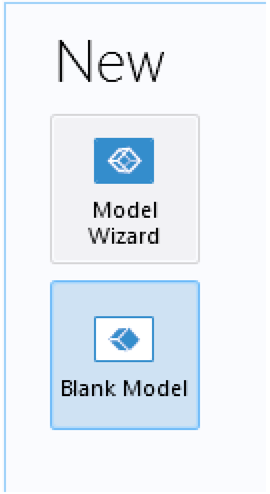
Nonisothermal Flow Analysis

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

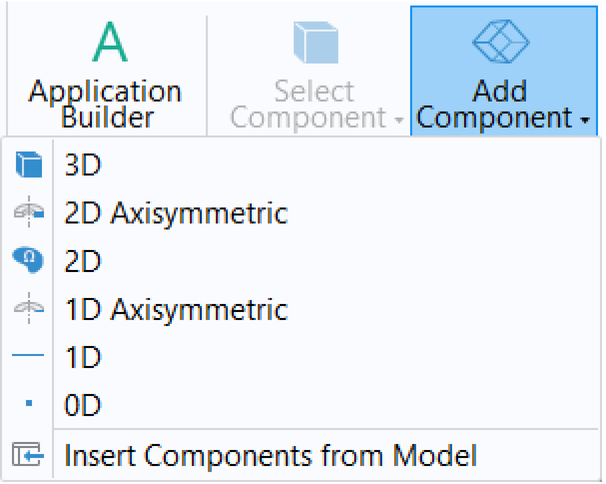
* = Build the model using the procedure here and refer to the model specifications for values

Model Setup

- Open the software
- Choose a *Blank Model*
- Add a 2D model component

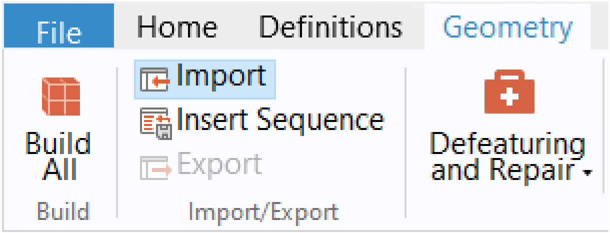


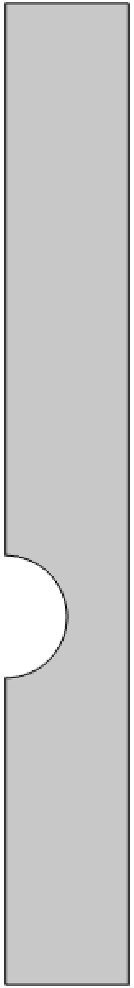
Screenshots of the steps performed to set up the model



Import Geometry

- Download the geometry file *free_convection.mphbin*
- Import the geometry
- Build *Form Union* operation to finalize the geometry

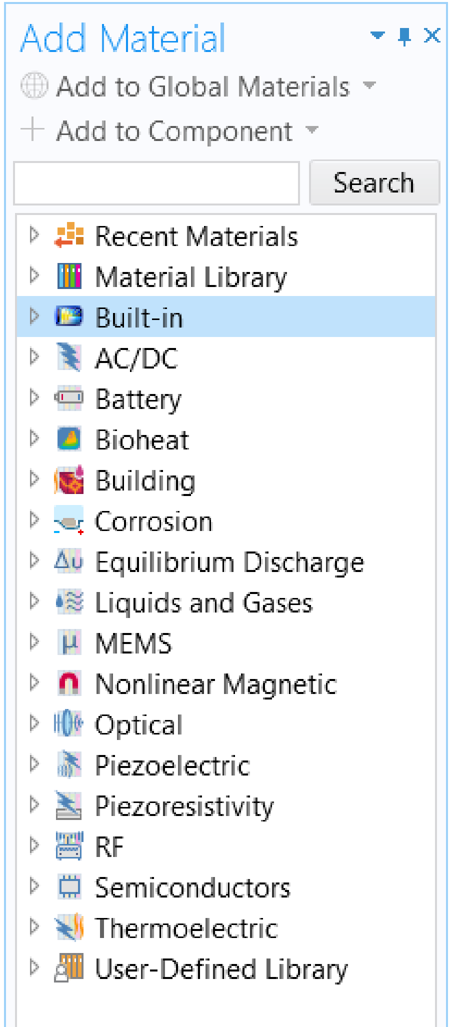


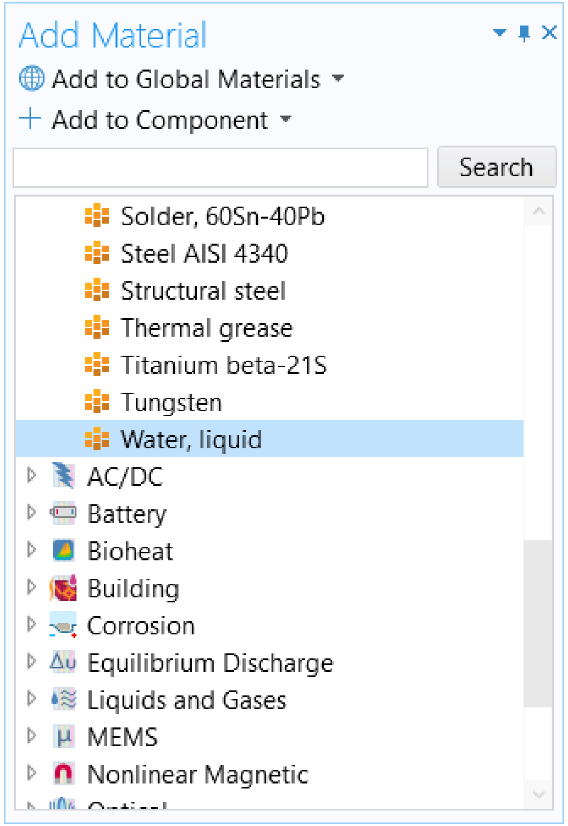


The Import button used and the free convection model geometry

Assign Materials

- Fluid domain
  - Apply *Water, liquid*





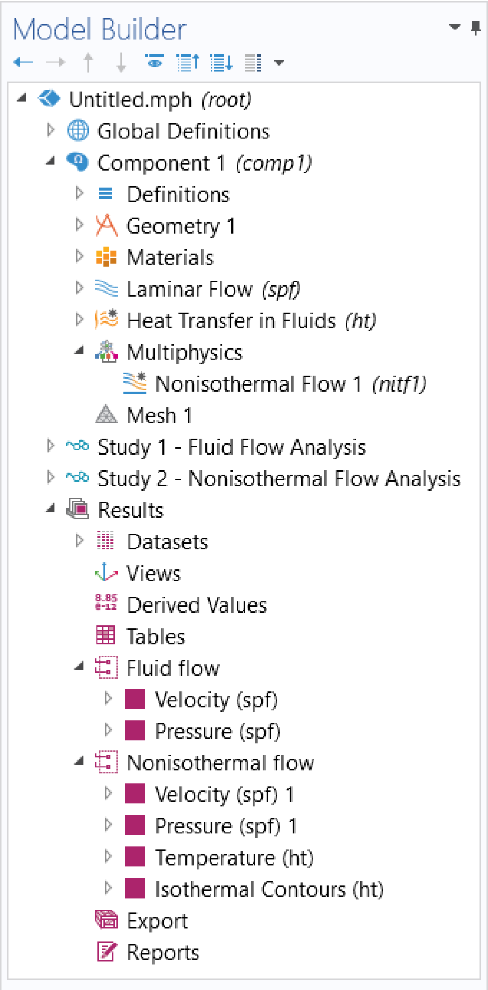
*The **Add Material** window, under which we add the **Water, liquid** material to our model*

Manual with Predefined Couplings Approach

Perform two stationary studies, the first for the fluid flow and the second including heat transfer with fluid flow.

Procedure:

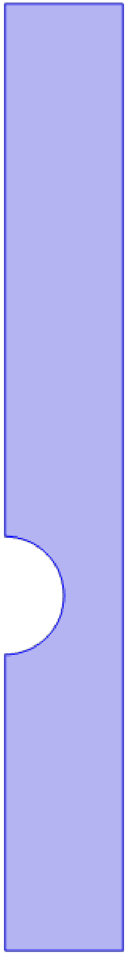
1. Fluid flow analysis
   - Add and define settings for the *Laminar Flow (spf)* interface
2. Nonisothermal flow analysis
   - Add and define settings for the *Heat Transfer in Fluids* interface
   - Add the *Nonisothermal Flow* multiphysics coupling

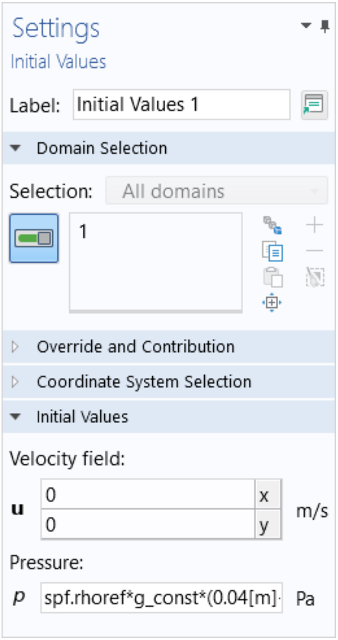


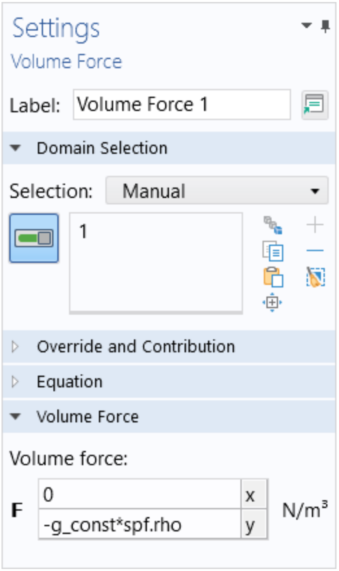
The model tree for the free convection tutorial model when the manual approach with predefined couplings has been used

Laminar Flow

- Active in all domains
- Update *Initial Values* node*
  - Defines initial conditions
- Add *Symmetry* boundary condition
  - Defines symmetry boundaries
- Add *Inlet* boundary condition*
  - Defines where fluid flows into domain
- Add *Outlet* boundary condition
  - Defines where fluid flows out of domain
- Add *Volume Force* node*
  - Defines buoyancy force lifting the fluid





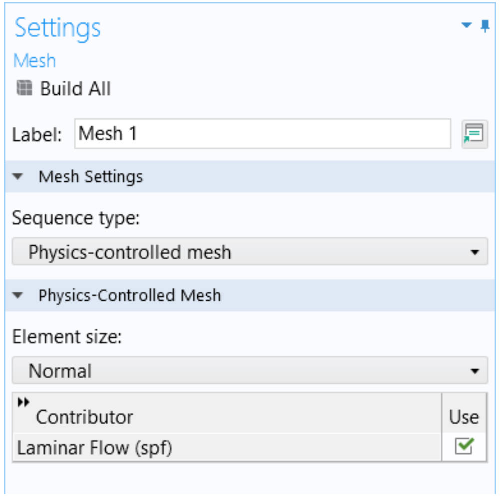


Settings for the Initial Values *and* Volume Force *nodes and their geometry selection*

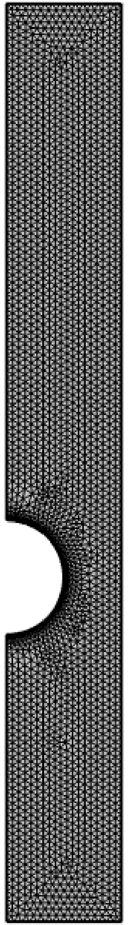
* = Refer to model specifications for values

Build the Mesh

Build the mesh using
the default settings

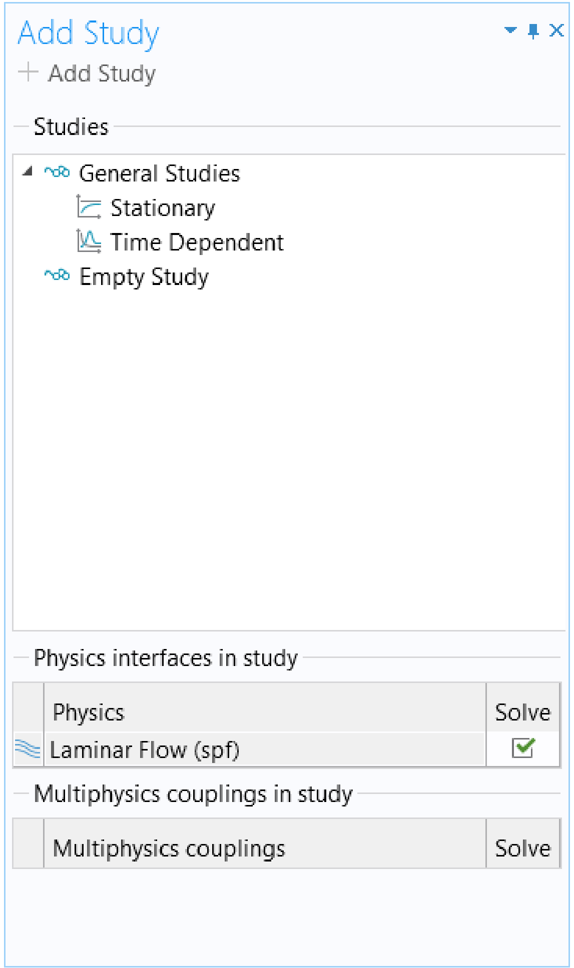


The setting used to generate the mesh for the free convection model, also pictured



Run the Study: Fluid flow

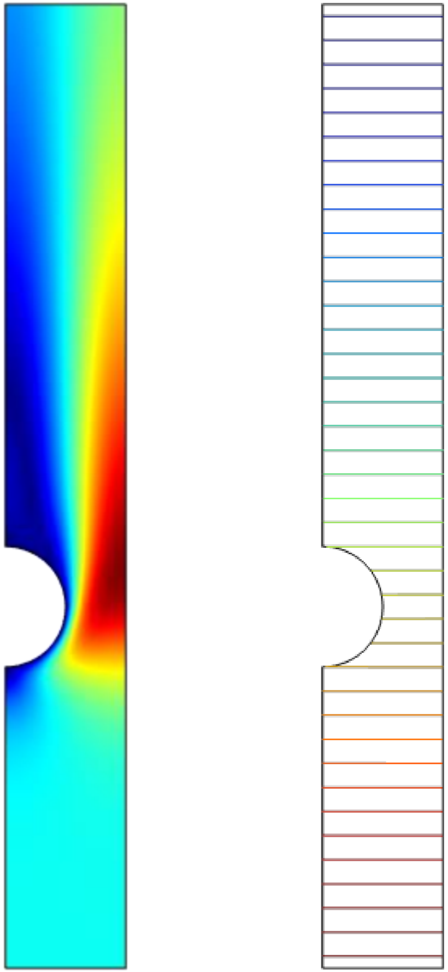
- Add a *Stationary* study
  - Study 1
- Change label for *Study 1* node to *Fluid Flow Analysis*
- Compute the model



Settings for the first Stationary study added to the model

Postprocess Results: Fluid Flow

- Default plots generated by the software
  - Velocity
  - Pressure

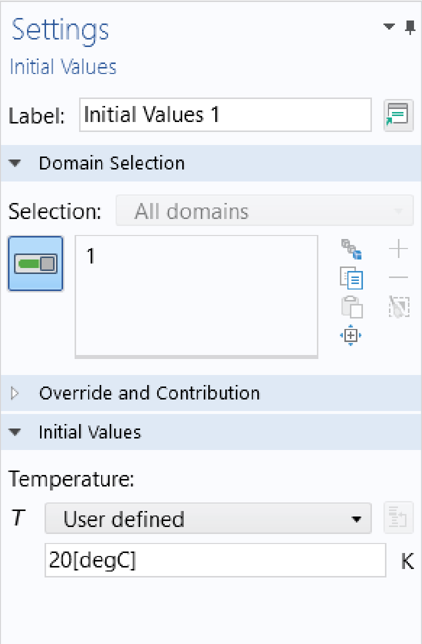


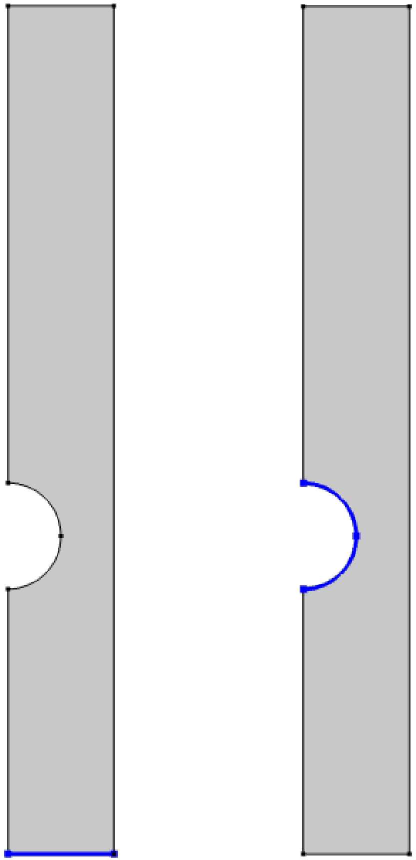
Results plots for velocity magnitude (left) and pressure (right)

PHYSICS SETTINGS

Heat Transfer in Fluids

- Active in all domains
- Update *Initial Values* node*
  - Defines initial conditions
- Add *Temperature* boundary condition*
  - Defines temperature at inlet
- Add *Temperature* boundary condition*
  - Defines temperature of heater
- Add *Outflow* boundary condition
  - Defines outlet boundary
- Add *Symmetry* boundary condition
  - Defines symmetry boundaries





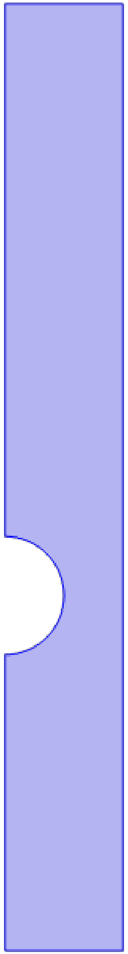
Settings for the initial values (left) and the geometry selections for the inlet (center) and heater (right)

* = Refer to model specifications for values

MULTIPHYSICS SETTINGS

Nonisothermal Flow

- Active in all domains
- Couples the *Laminar Flow (spf)* and *Heat Transfer in Fluids* interfaces
  - *Laminar Flow (spf)*
    - Incorporates the temperature field computed in the heat transfer interface
  - *Heat Transfer in Fluids*
    - Incorporates the pressure and velocity fields computed in the fluid flow interface

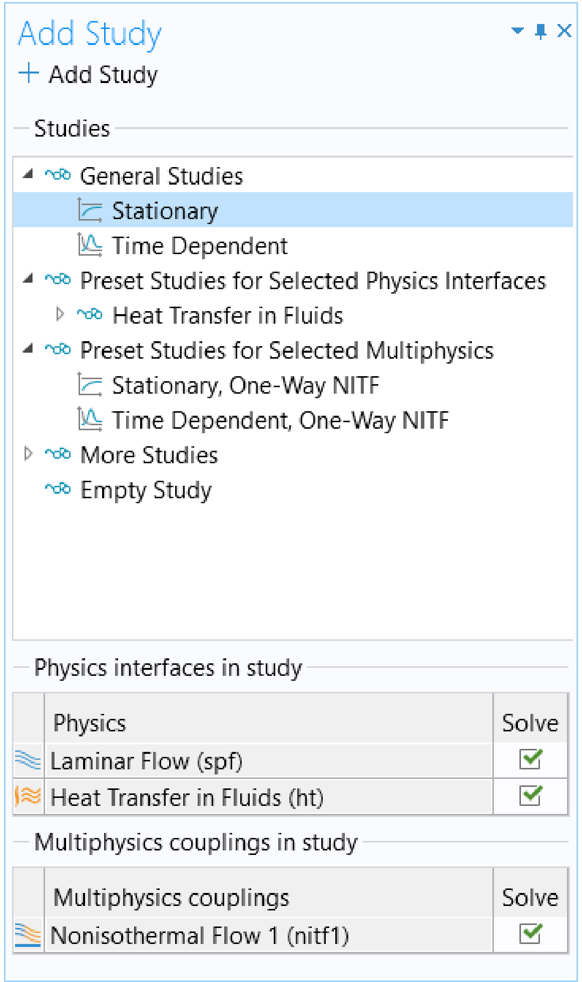


Geometry selection for the Nonisothermal Flow *multiphysics coupling node*

Run the Study: Nonisothermal Flow

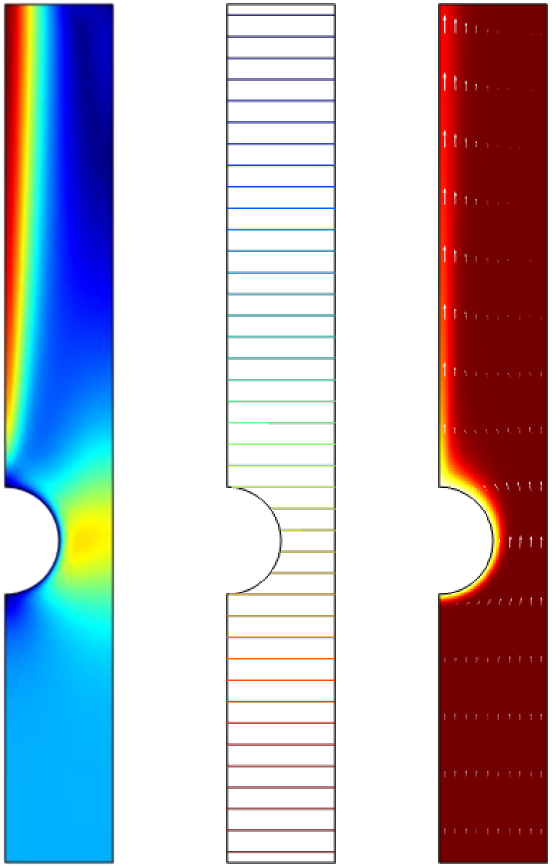
- Add a *Stationary* study
  - Study 2
- Change label for *Study 2* to *Nonisothermal Flow Analysis*
- Compute the model

The settings for the second Stationary *study being added to the model*



Postprocess Results: Nonisothermal Flow

- Default plots generated by the software
  - Velocity
  - Pressure
  - Temperature
- Add arrows to *Temperature* plot to visualize the velocity field
  - Add an *Arrow Surface* plot
  - Use expression that represents the velocity field
    - *x* component: *u*
    - *y* component: *v*
  - Change the arrow color to *White*
  - Change number of *x grid points* to *10*



Results plots for velocity magnitude (left), pressure (center), and temperature (right)