Modeling Exercise

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

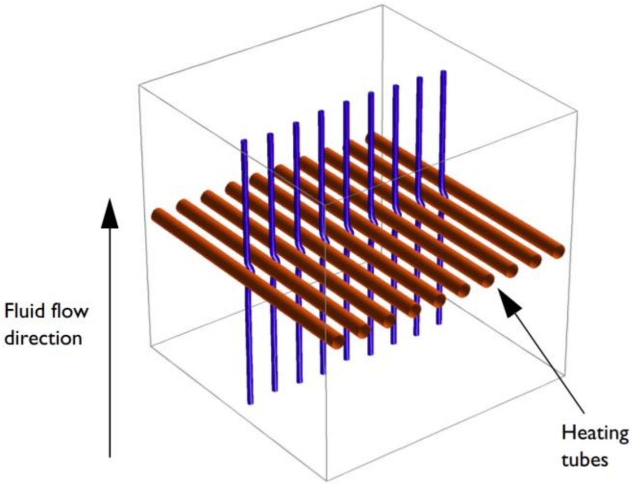
Introduction

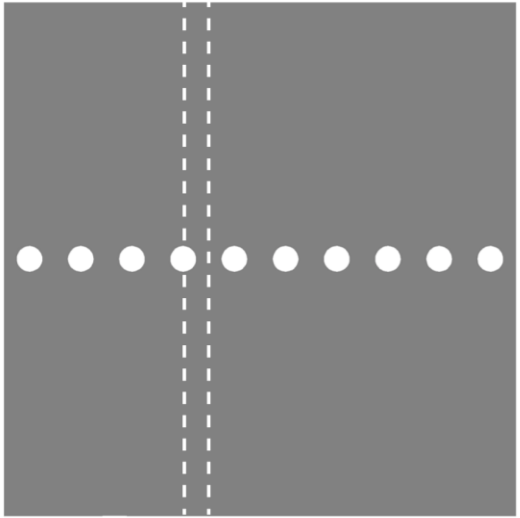
- This model exercise demonstrates the concept of multiphysics modeling in COMSOL Multiphysics®
- Define the physics for the model using the manual approach with user-defined couplings
  - Add and define the physics settings for the *Laminar Flow (spf)* interface, followed by the *Heat Transfer in Fluids* interface, and then manually couple the physics to simulate nonisothermal flow by including dependent variables of each physics interface as input to the other
    - Enables you to manually implement couplings between physics interfaces for which no coupling features are available
- Important information for setting up the model can be found in the model specifications
  - Refer to this when building the model

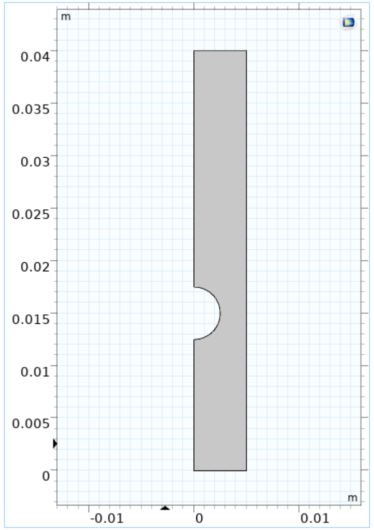
Model Overview

- 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

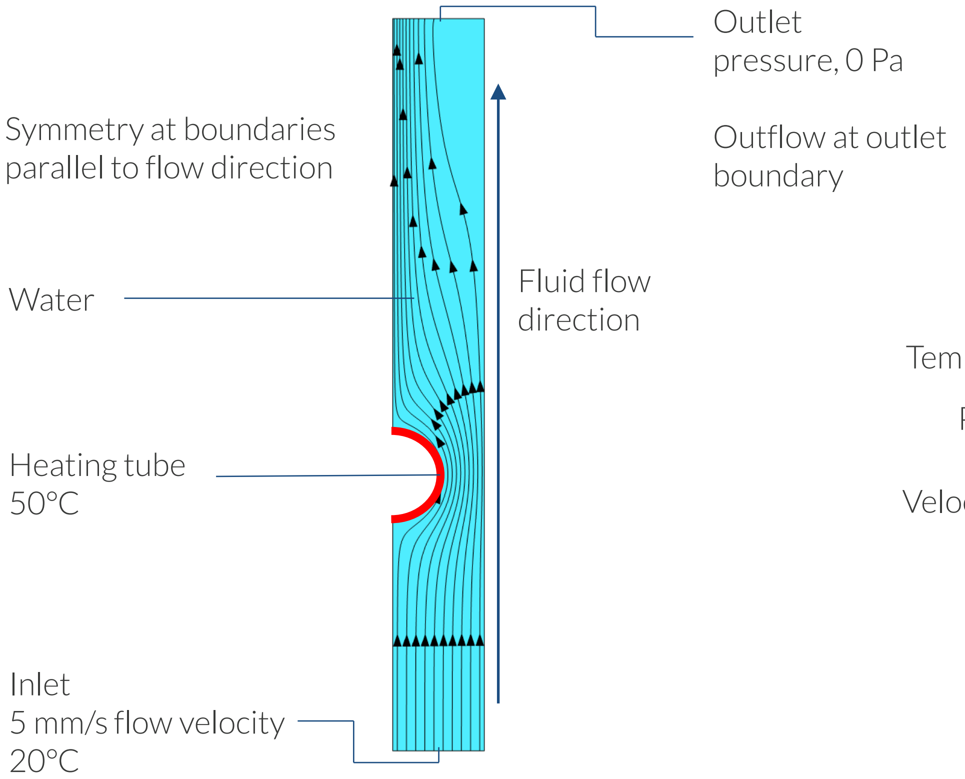






A cross section (center) of the 3D model geometry (left) is taken, and symmetry of the array is exploited to result in the model geometry (right)

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]

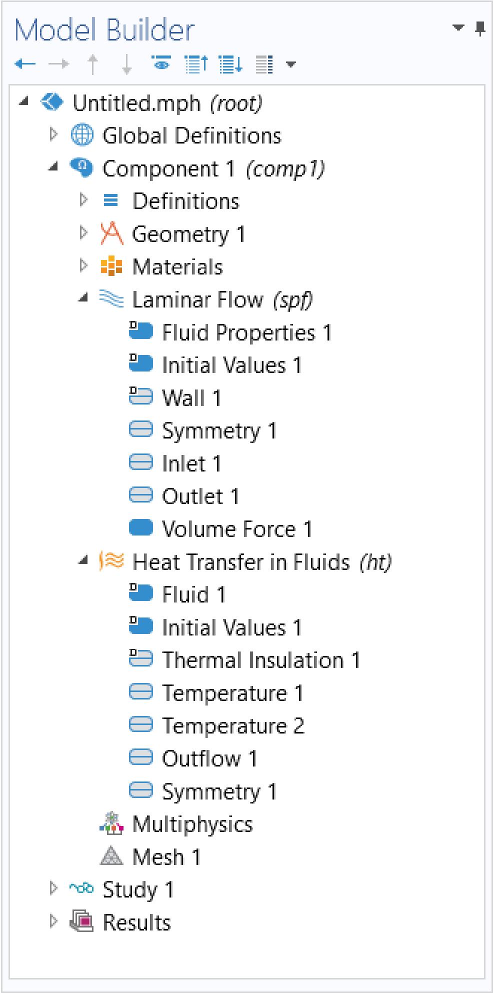
Details and specifications for the free convection model setup

Manual Approach with User-Defined Couplings

Define the physics for the model using user-defined multiphysics couplings

Procedure:

1. Add the physics interface
2. Define the physics settings
3. Repeat steps 1 and 2 for each subsequent physics interface
4. Define the multiphysics couplings



The model tree after the physics interfaces and the features to enable coupling the physis have been implemented

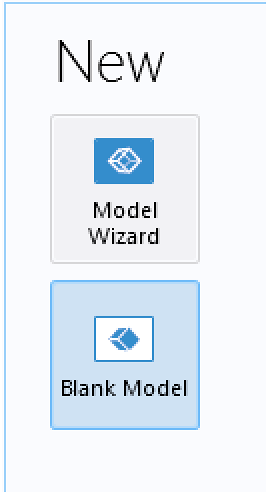
Modeling Workflow

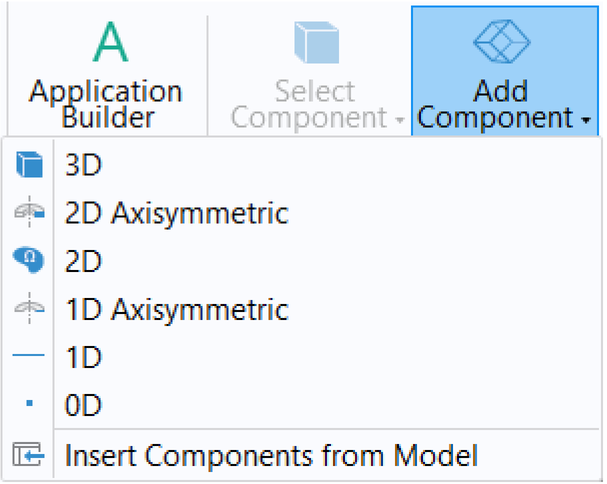
Outline of the steps used to set up, build, and compute this model to complete this modeling exercise are provided here.

1. Set up the model
2. Import geometry
3. Assign materials
4. Define the physics
   - Add *Laminar Flow (spf)* interface
   - Add *Heat Transfer in Fluids* interface
   - Implement user-defined multiphysics coupling
5. Build the mesh
6. Run the study
7. Postprocess results

Model Setup

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

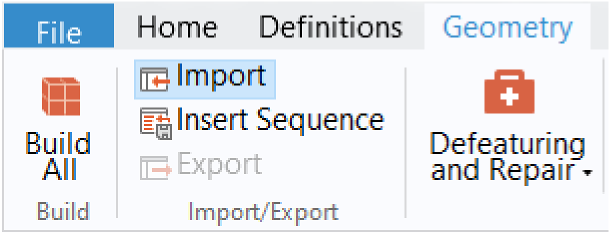


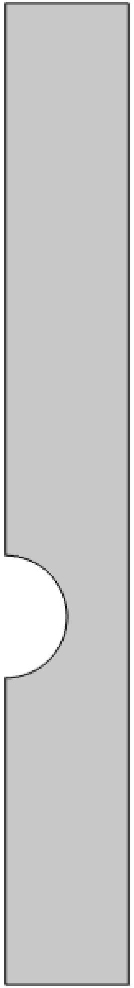


Screenshots of the steps performed to set up the model

Import Geometry

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

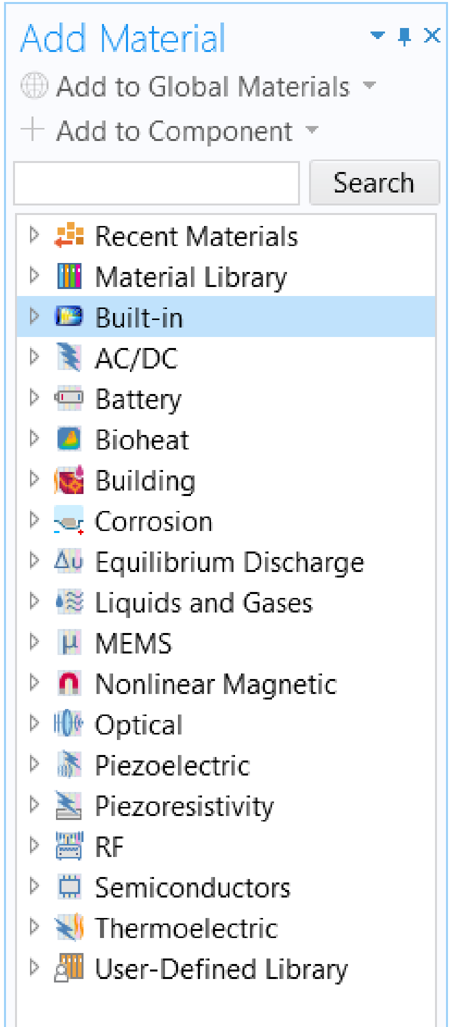


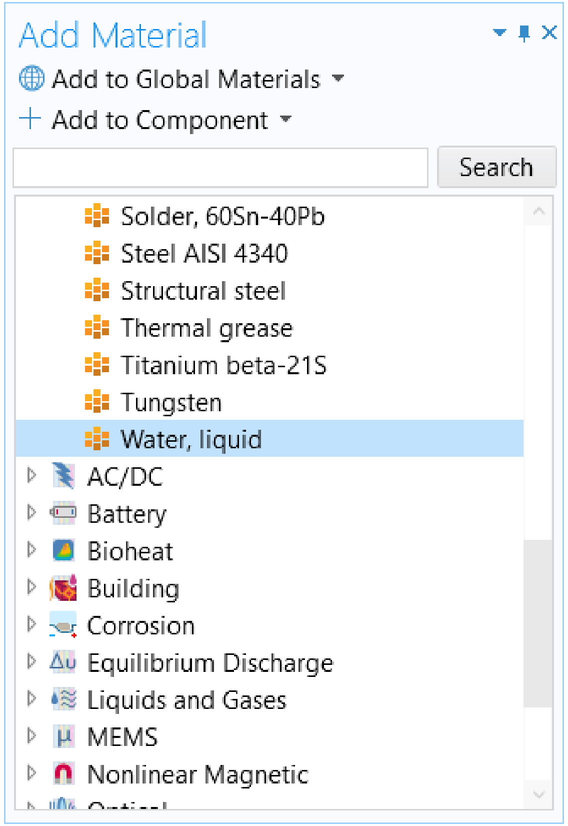


The Import button used and the free convection model geometry

Assign Materials

- Fluid domain
  - Apply *Water, liquid*





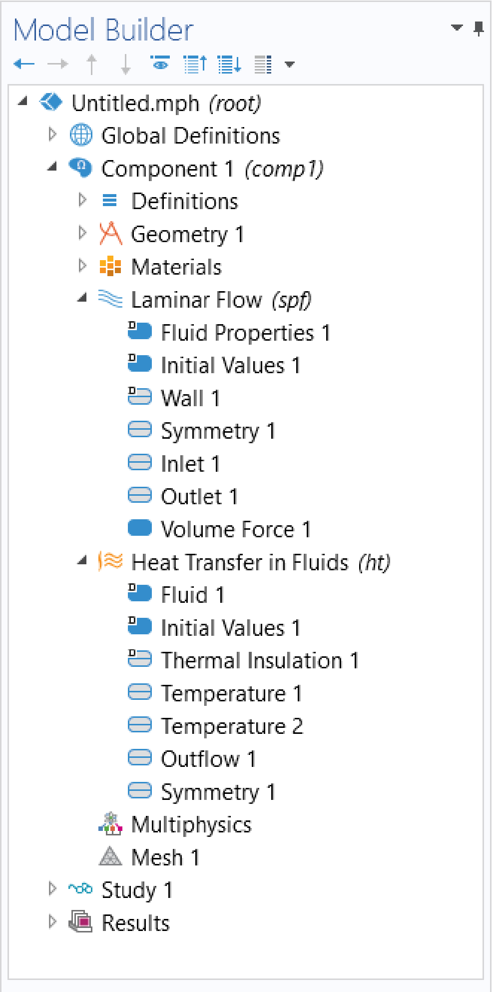
The **Add Material** *window, under which we add the* **Water, liquid** *material to our model*

Manual Approach with User-Defined Couplings

Define the physics for the model using user-defined multiphysics couplings

Procedure:

1. Add the physics interfaces
   - *Laminar Flow (spf)*
   - *Heat Transfer in Fluids*
2. Define the physics settings
3. Repeat steps 1 and 2 for each subsequent physics interface
4. Define the multiphysics couplings
   - *Laminar Flow (spf)* > *Fluid Properties* node
   - *Heat Transfer in Fluids* > *Fluid* node

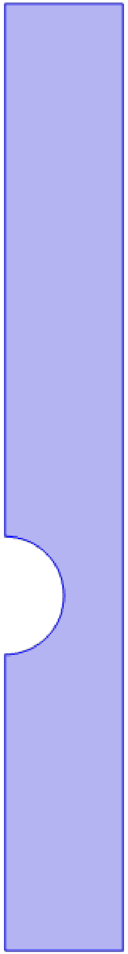


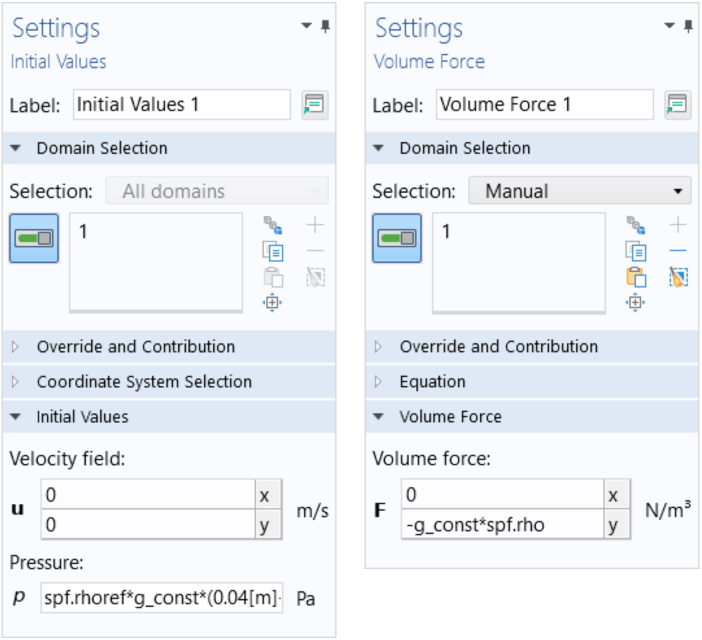
The model tree after the physics interfaces and the features to enable coupling the physics have been implemented

PHYSICS SETTINGS

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





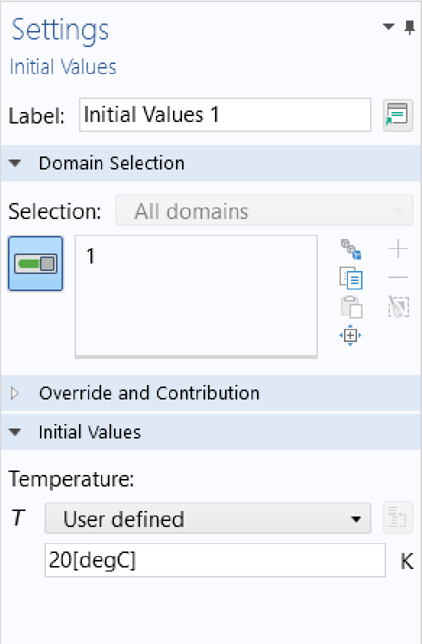
Settings for the Initial Values *and* Volume Force *nodes and their geometry selection*

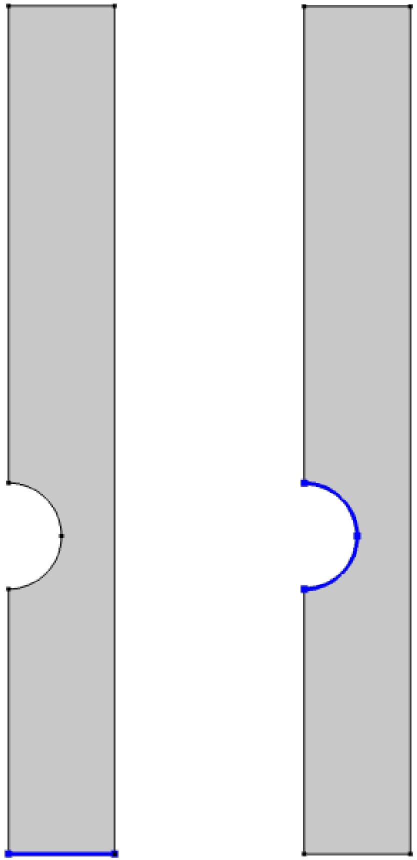
* = Refer to model specifications for values

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





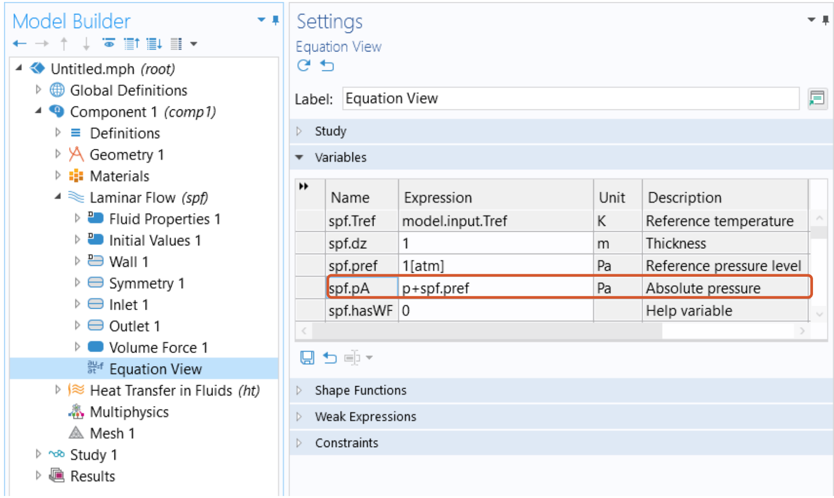
Settings for the initial values (left) and the geometry selections for the inlet (center) and heater (right)

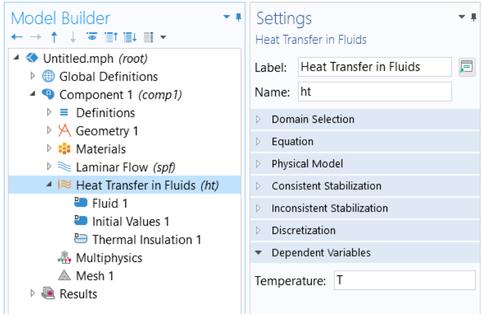
* = Refer to model specifications for values

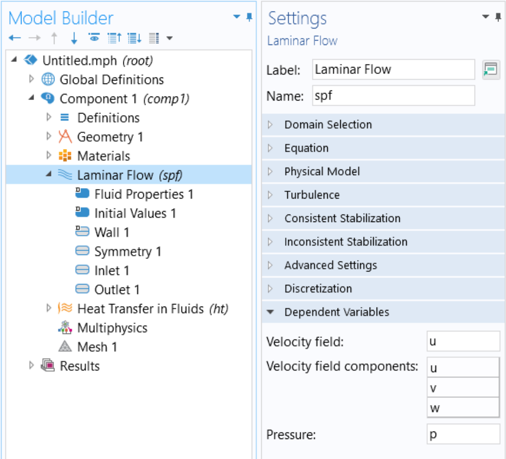
MULTIPHYSICS SETTINGS

Nonisothermal Flow

- *Laminar Flow (spf)* interface
  - Include temperature from heat transfer interface as input
    - Fluid properties depend on temperature
- *Heat Transfer in Fluids* interface
  - Include absolute pressure from fluid flow interface as input
  - Include velocity field from fluid flow interface as velocity field for convective heat transfer







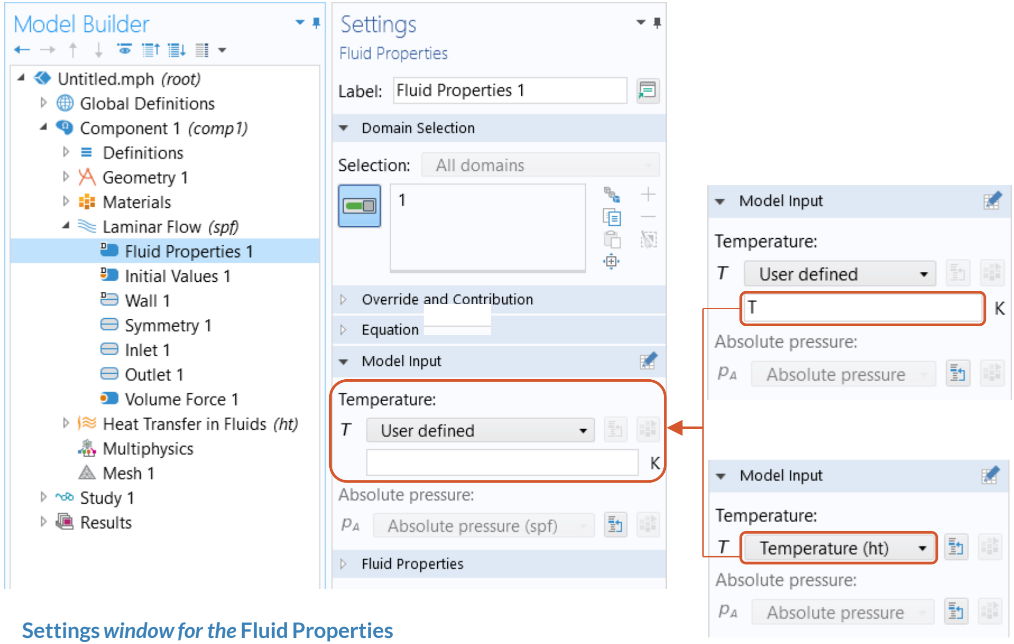
Equation View *node for the* Laminar Flow (spf) *interface (top) and the dependent variables for the* Laminar Flow (spf) *interface (left) and* Heat Transfer in Fluids *interface (right)*

MULTIPHYSICS SETTINGS

Nonisothermal Flow

Laminar Flow (spf) interface

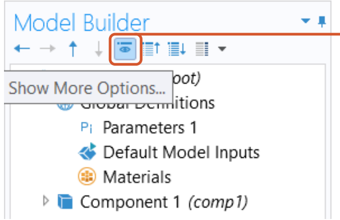
1. Select *Fluid Properties* node
2. Change the model input for temperature to *User defined*
3. For the expression enter the temperature field from the *Heat Transfer in Fluids* interface
   - Alternatively select *Temperature (ht)* from the *Temperature* model input drop-down menu



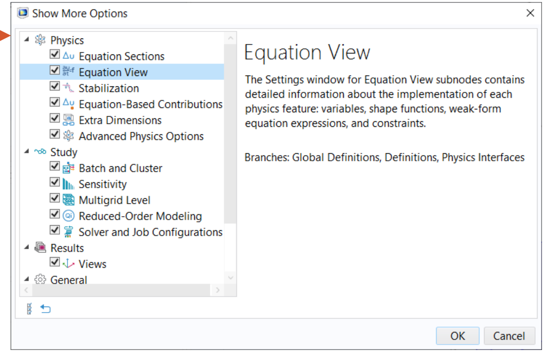
Settings window for the **Fluid Properties** *node and the user-defined expressions and options that can couple the physics*

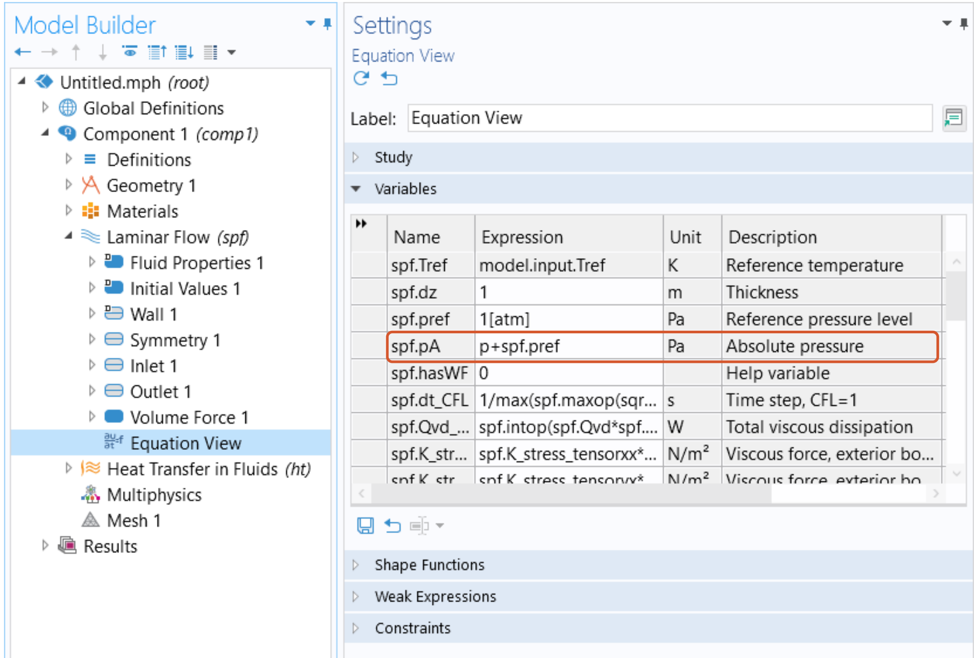
Nonisothermal Flow

- Access predefined physics variables to use as input to other physics interfaces
  - Enable displaying *Equation View* nodes through the *Model Builder* toolbar
  - Select *Equation View* node for the *Laminar Flow (spf)* interface node
  - Include absolute pressure from fluid flow physics as input to heat transfer physics



The Equation View node for the Laminar Flow (spf) *interface node*



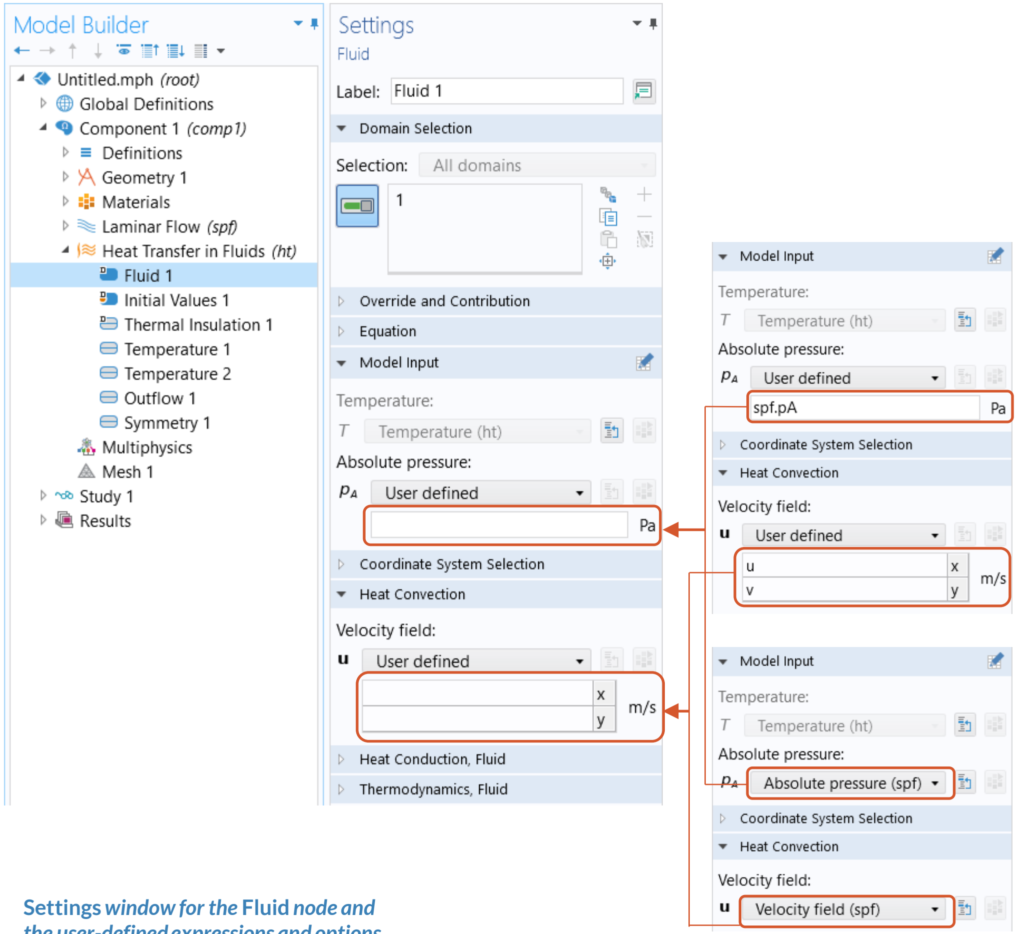


MULTIPHYSICS SETTINGS

Nonisothermal Flow

Heat Transfer in Fluids interface

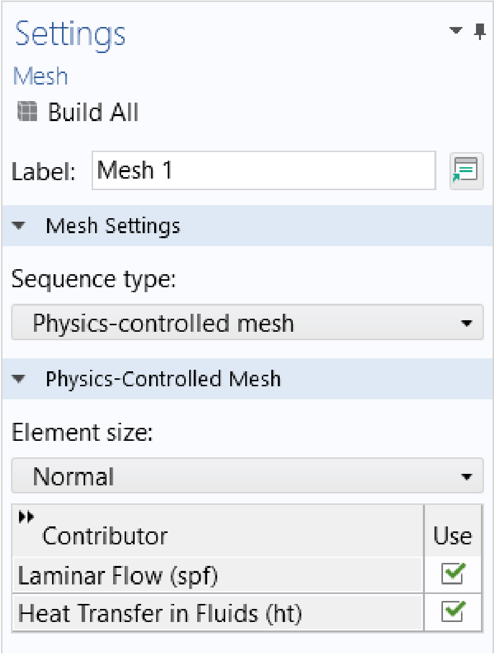
1. Select *Fluid* node
2. Change the model inputs for the absolute pressure and velocity field to *User defined*
3. For the expression enter the absolute pressure and velocity field components from the *Laminar Flow (spf)* interface
   - Alternatively select *Absolute pressure (spf)* and *Velocity field (spf)* from the respective drop-down menus

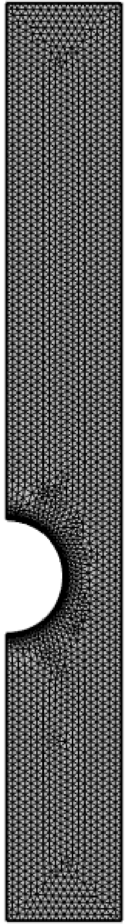


Settings window for the Fluid node and the user-defined expressions and options that can couple the physics

Build the Mesh

Build the mesh using
the default settings

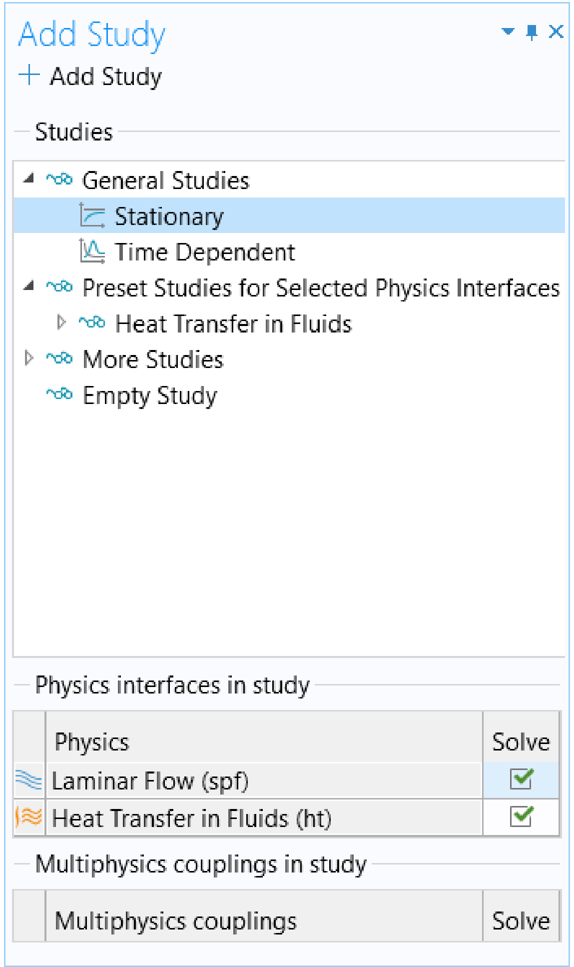




The setting used to generate the mesh for the free convection model, also pictured

Run the Study

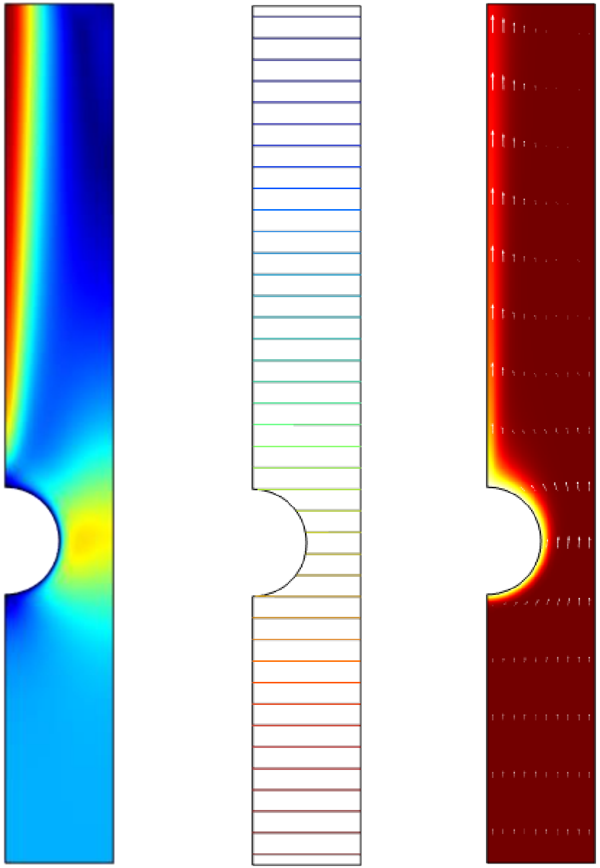
- Add a *Stationary* study
- Compute the model



Settings for the Stationary study being added to the model

Postprocess Results

- Default plots generated by the software
  - Velocity
  - Pressure
  - Temperature
- Add arrows to *Temperature* plot to show the velocity field
  - Add an *Arrow Surface* plot
  - Use an expression that represents the velocity field
  - 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)