

Modeling Exercise

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

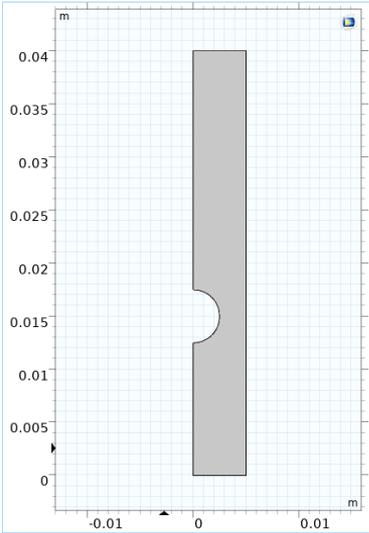
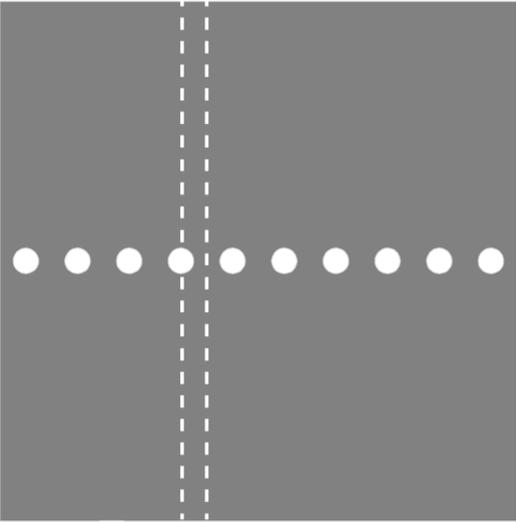
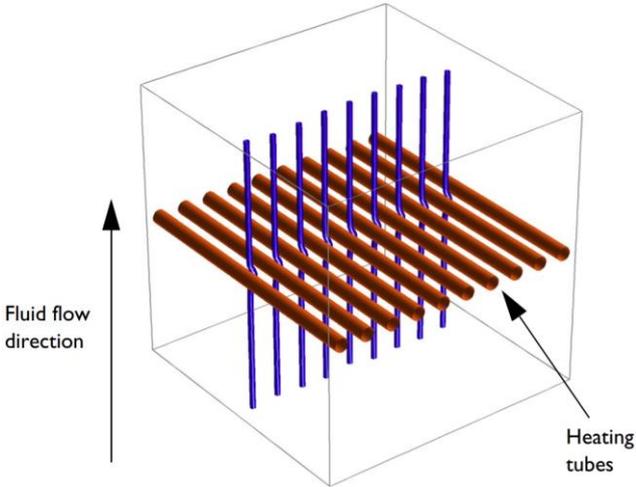
Introduction

- 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

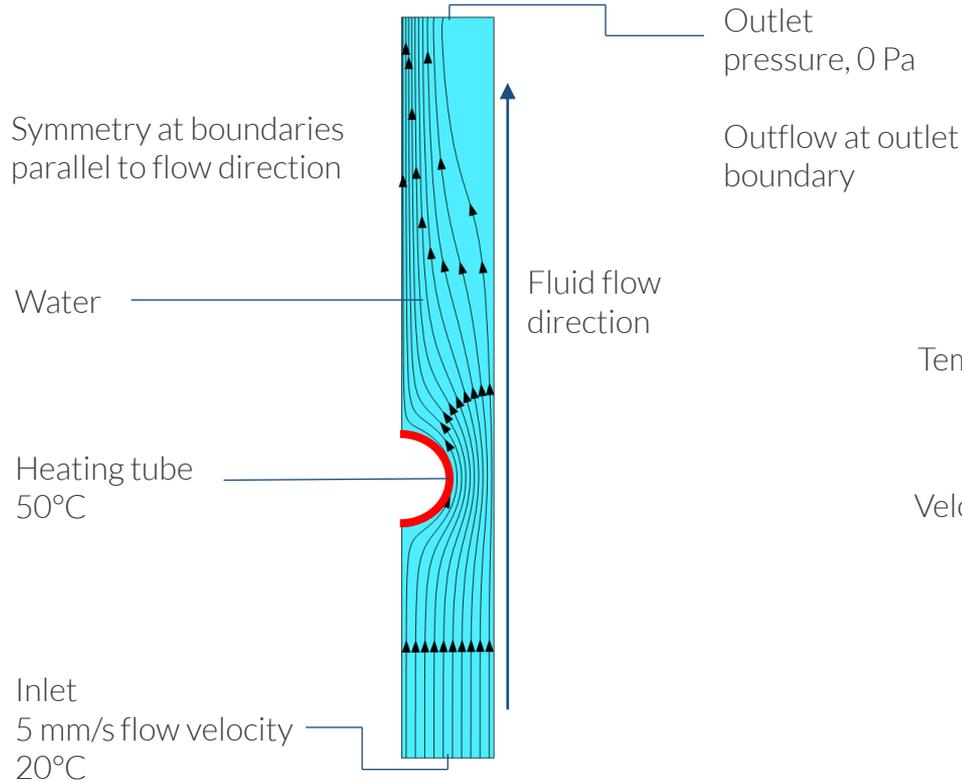
- An array of heating tubes are submerged in a vessel of water with the fluid entering from the bottom
 - The model is reduced from 3D to 2D and further simplified by exploiting symmetry due to the array
- As fluid enters the vessel and travels past the heating element, heat is transferred through convection
 - 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

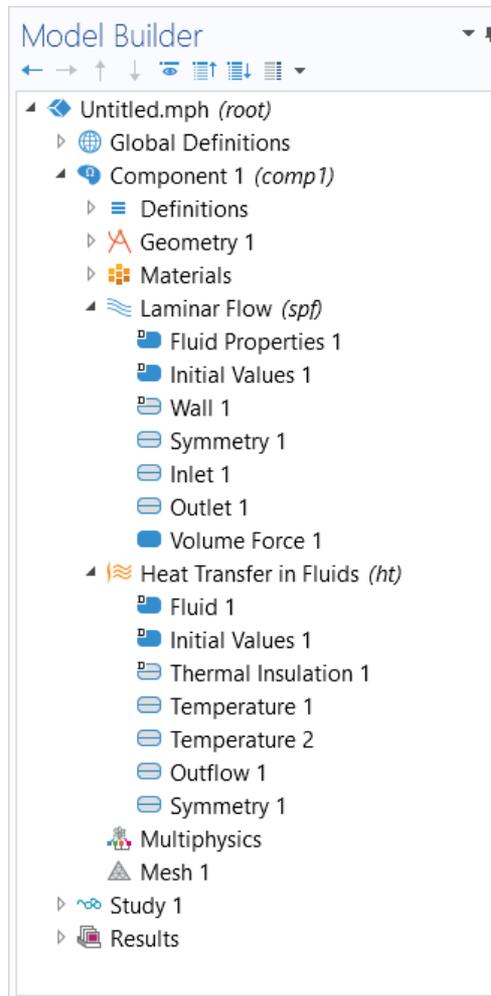
Manual Approach with User-Defined Couplings

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 physics have been implemented



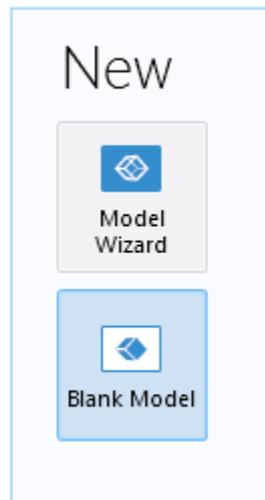
Modeling Workflow

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

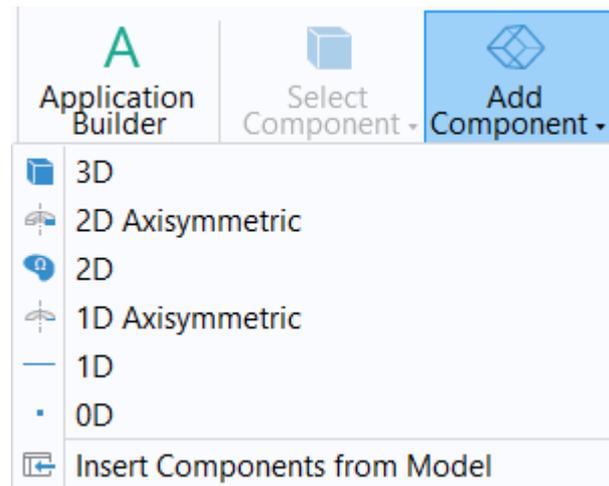
1. Set up the model
2. Import geometry
3. Assign materials
4. Define the physics
 - Add *Laminar Flow (spf)* interface
 - Add *Heat Transfer in Fluids* interface
 - Implement user-defined multiphysics coupling
5. Build the mesh
6. Run the study
7. Postprocess results

Model Setup

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

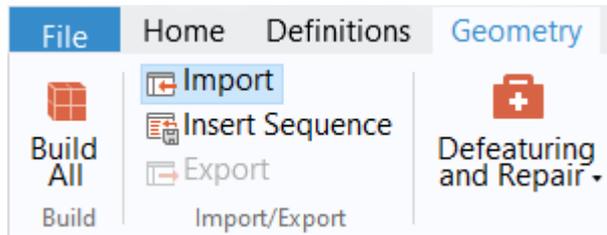


Screenshots of the steps performed to set up the model



Import Geometry

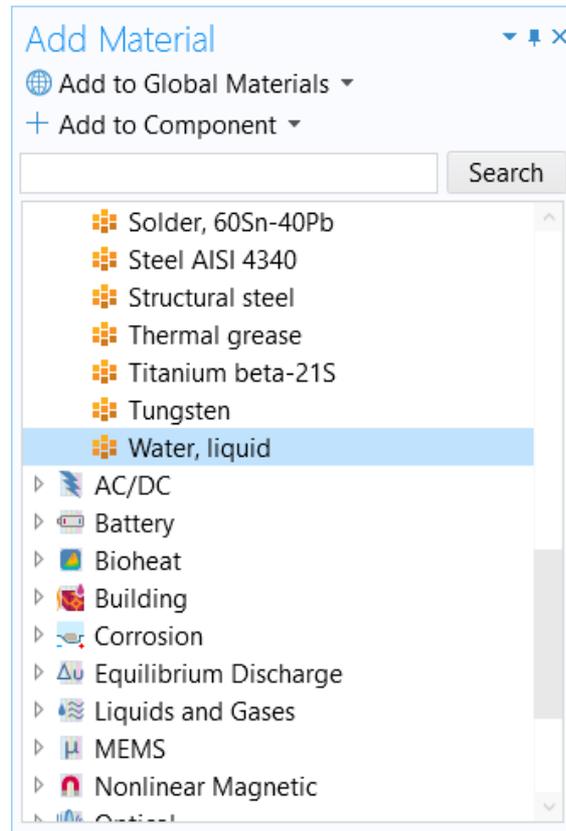
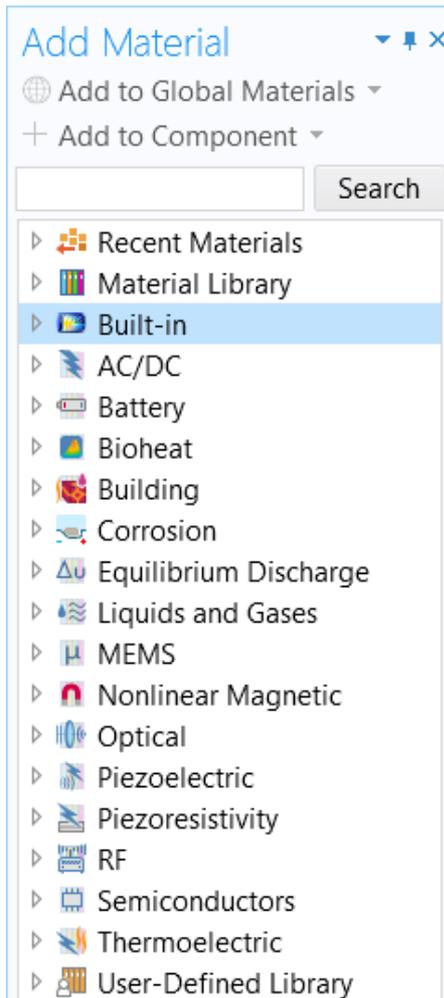
- Download the geometry file *free_convection.mphbin*
- Import the geometry
- Build *Form Union* operation to finalize the geometry



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

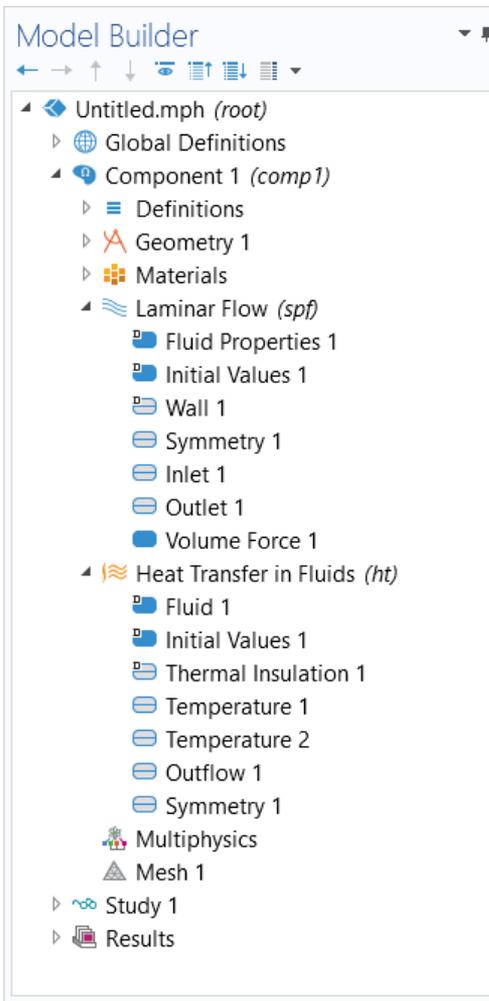
Manual Approach with User-Defined Couplings

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
Initial Values

Label: Initial Values 1

Domain Selection

Selection: All domains

1

Override and Contribution

Coordinate System Selection

Initial Values

Velocity field:

u	0	x	m/s
	0	y	

Pressure:

p spf.rhoref*g_const*(0.04[m]) Pa

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

Settings
Volume Force

Label: Volume Force 1

Domain Selection

Selection: Manual

1

Override and Contribution

Equation

Volume Force

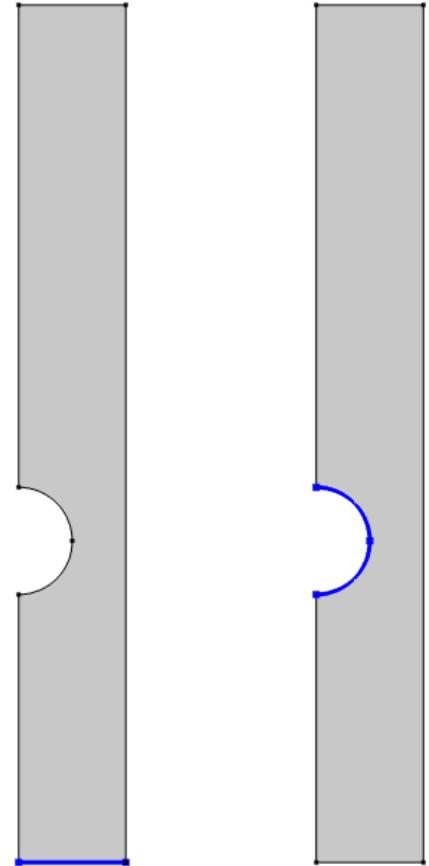
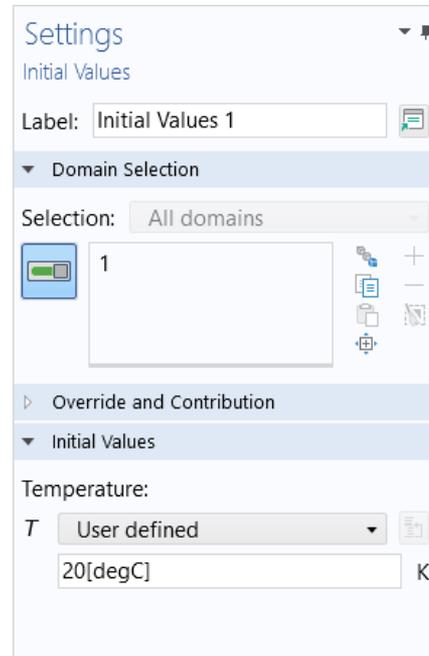
Volume force:

F	0	x	N/m ³
	-g_const*spf.rho	y	
		z	

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)

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

The screenshot shows the Model Builder on the left and the Settings window on the right. The Model Builder tree shows the hierarchy: Untitled.mph (root) > Global Definitions > Component 1 (comp1) > Definitions > Materials > Laminar Flow (spf) > Equation View. The Settings window is in Equation View, showing a table of variables:

Name	Expression	Unit	Description
spf.Tref	model.input.Tref	K	Reference temperature
spf.dz	1	m	Thickness
spf.pref	1[atm]	Pa	Reference pressure level
spf.pA	p+spf.pref	Pa	Absolute pressure
spf.hasWF	0		Help variable

The left screenshot shows the Model Builder with the Heat Transfer in Fluids (ht) interface selected. The Settings window shows the Name: ht and the Temperature: T dependent variable.

The right screenshot shows the Model Builder with the Laminar Flow (spf) interface selected. The Settings window shows the Name: spf and the Velocity field and Pressure dependent variables:

Velocity field:	u
Velocity field components:	u
	v
	w
Pressure:	p

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

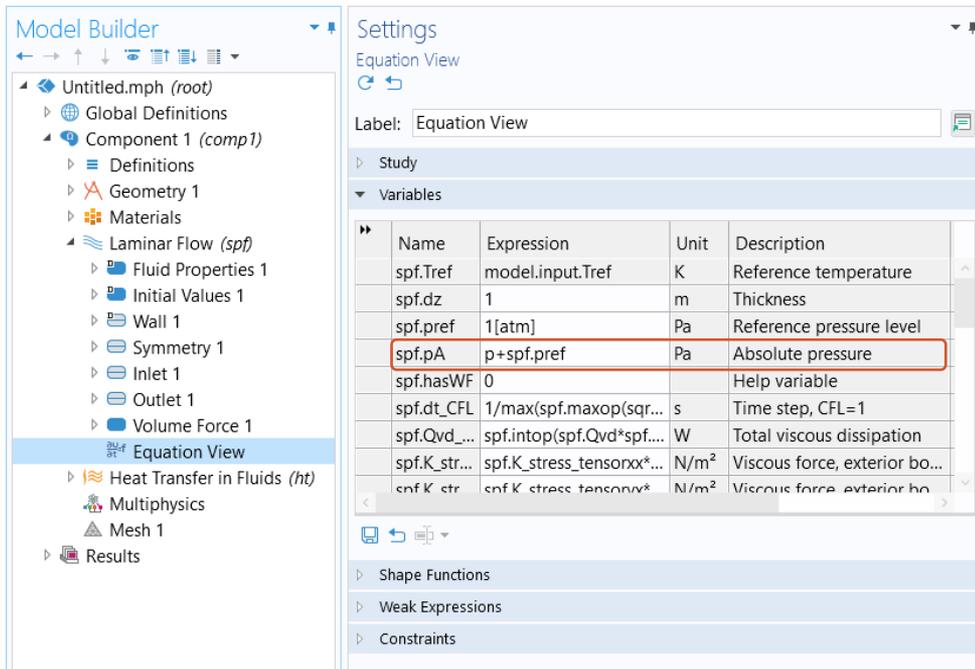
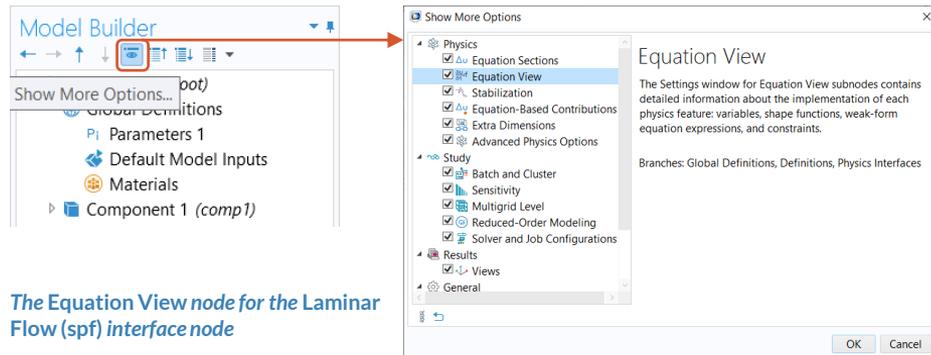
The screenshot displays the COMSOL Model Builder interface. On the left, the 'Model Builder' tree shows the 'Fluid Properties 1' node selected under the 'Laminar Flow (spf)' interface. The 'Settings' window for 'Fluid Properties' is open, showing the 'Model Input' section. The 'Temperature' dropdown is set to 'User defined', and the 'Absolute pressure' dropdown is set to 'Absolute pressure'. A red box highlights the 'Temperature' dropdown and the 'User defined' text. A red arrow points from this box to a second 'Model Input' window on the right, which shows the 'Temperature' dropdown set to 'Temperature (ht)'. This second window also has a red box around the 'Temperature (ht)' dropdown.

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

MULTIPHYSICS SETTINGS

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



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

The screenshot displays the COMSOL Model Builder interface. On the left, the tree view shows the project structure: 'Untitled.mph (root)' > 'Global Definitions' > 'Component 1 (comp1)' > 'Definitions' > 'Geometry 1' > 'Materials' > 'Laminar Flow (spf)' > 'Heat Transfer in Fluids (ht)' > 'Fluid 1'. The 'Fluid 1' node is selected. The main window shows the 'Settings' for 'Fluid 1'. The 'Domain Selection' is set to 'All domains'. The 'Model Input' section is expanded, showing 'Temperature' set to 'Temperature (ht)', 'Absolute pressure' set to 'User defined', and 'Velocity field' set to 'User defined'. Red boxes highlight the 'User defined' dropdowns and the input fields for the velocity field components (u, v) and absolute pressure. A second 'Model Input' section is also visible on the right, showing 'Absolute pressure' set to 'Absolute pressure (spf)' and 'Velocity field' set to 'Velocity field (spf)'.

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

Settings

Mesh

Build All

Label: Mesh 1

Mesh Settings

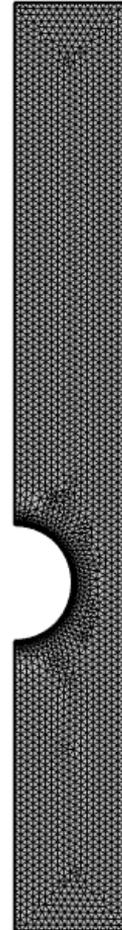
Sequence type:
Physics-controlled mesh

Physics-Controlled Mesh

Element size:
Normal

Contributor	Use
Laminar Flow (spf)	<input checked="" type="checkbox"/>
Heat Transfer in Fluids (ht)	<input checked="" type="checkbox"/>

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*

Add Study

+ Add Study

– Studies

- General Studies
 - Stationary
 - Time Dependent
- Preset Studies for Selected Physics Interfaces
 - Heat Transfer in Fluids
- More Studies
 - Empty Study

– Physics interfaces in study

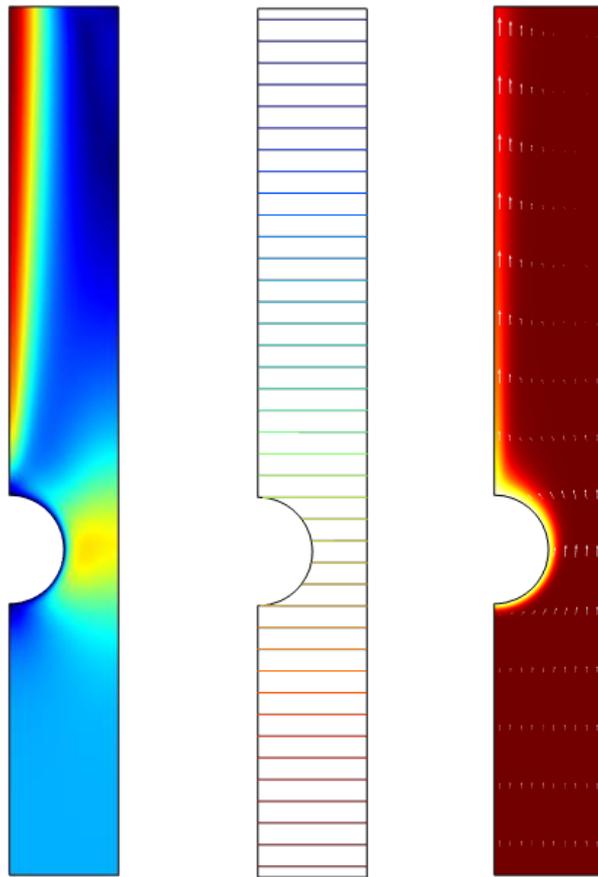
Physics	Solve
Physics	
Laminar Flow (spf)	<input checked="" type="checkbox"/>
Heat Transfer in Fluids (ht)	<input checked="" type="checkbox"/>

– Multiphysics couplings in study

Multiphysics couplings	Solve
Multiphysics couplings	<input type="checkbox"/>

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)