Modeling Exercise

Define the physics for a model of heat transfer by free convection using the fully automatic approach

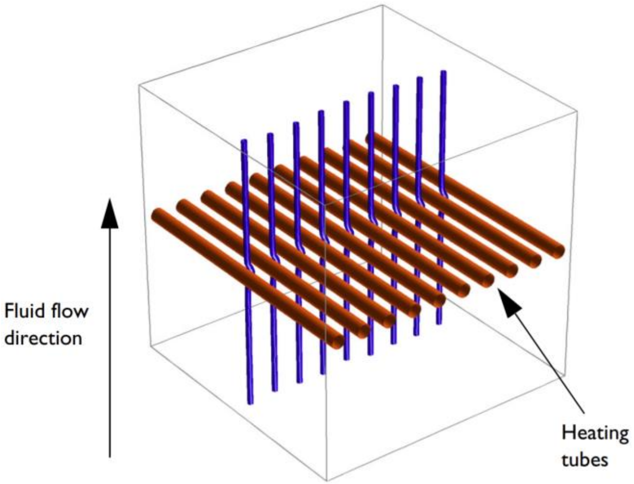
Introduction

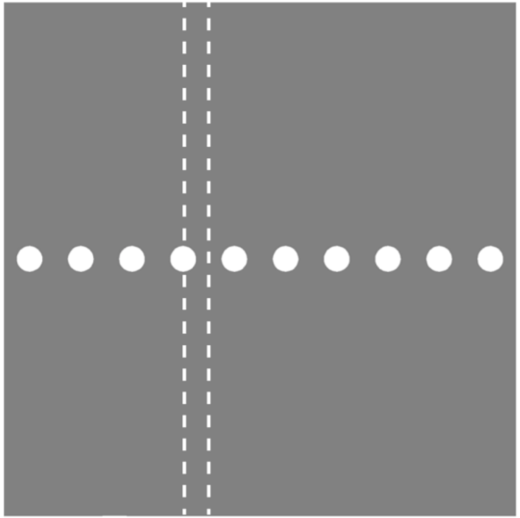
- This model exercise demonstrates the concept of multiphysics modeling in COMSOL Multiphysics®
- Define the physics for the model using the fully automatic approach
  - Add the *Nonisothermal Laminar Flow* multiphysics interface
    - Completely streamlines defining the physics by automatically including the physics phenomena involved and the appropriate settings for the combination of physics phenomena involved
- Important information for setting up the model can be found in the model specifications
  - Refer to this when building the model

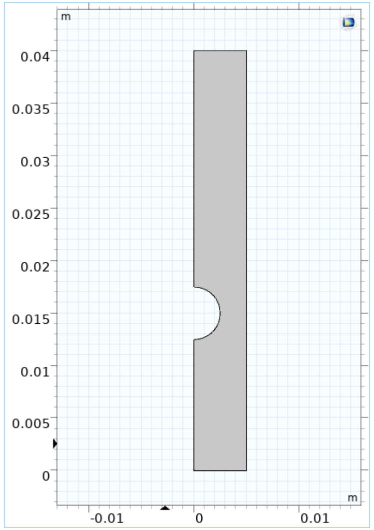
Model Overview

- 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

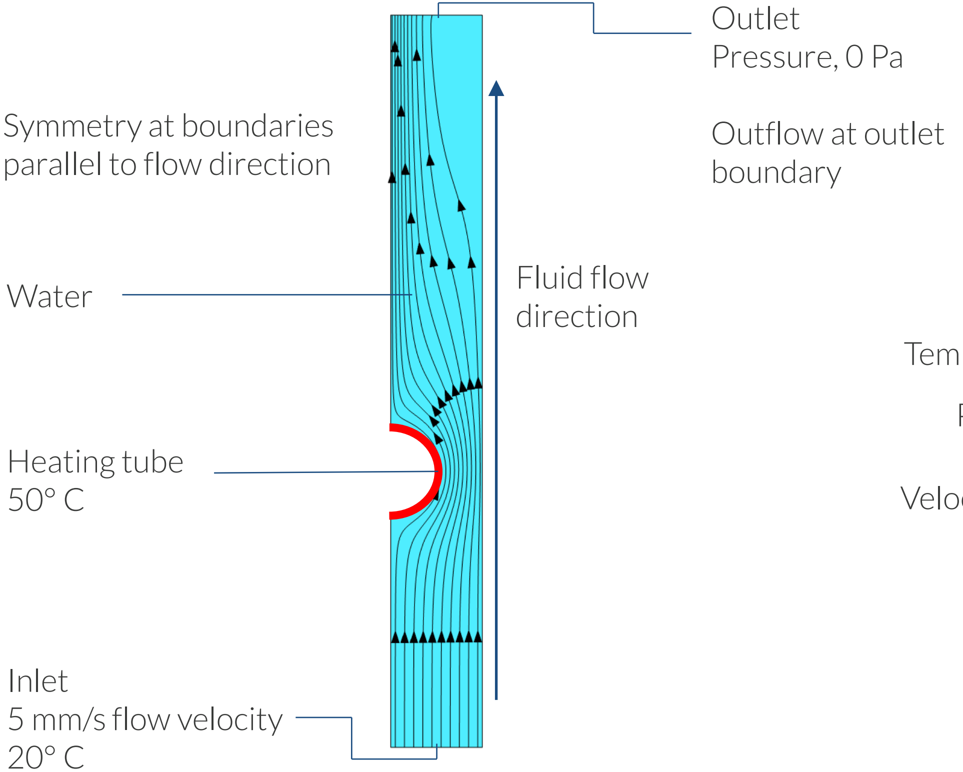






A cross section (center) of the 3D model geometry (left) is taken, and symmetry of the array is exploited to result in the model geometry (right).

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]

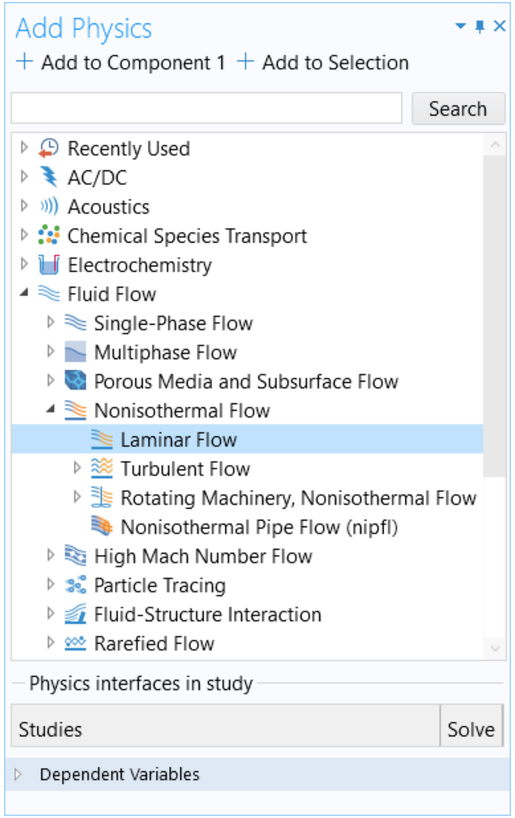
Details and specifications for the free convection model setup.

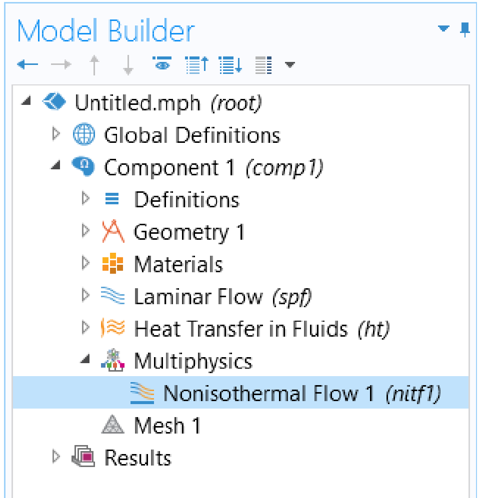
Fully Automatic Approach

Define the physics for the model using a predefined multiphysics interface

Procedure:

1. Add the physics interface
2. Define the physics settings





The Add Physics window and the model tree after the multiphysics interface has been added.

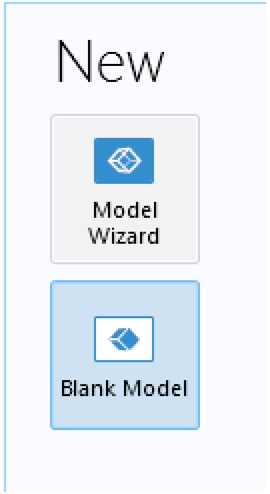
Modeling Workflow

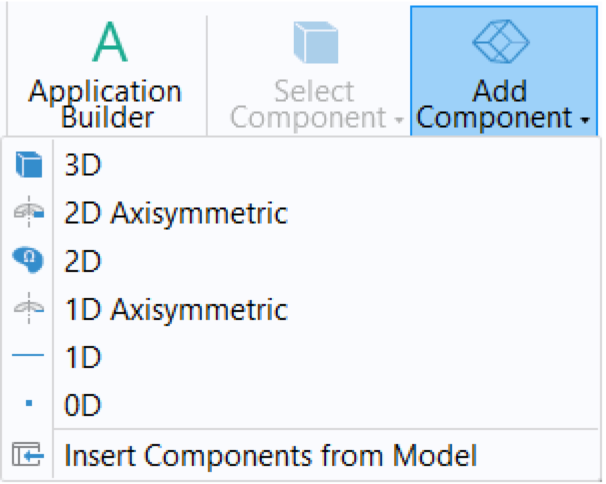
An outline of the steps used to set up, build, and compute this model to complete this modeling exercise is provided here.

1. Set up the model
2. Import geometry
3. Assign materials
4. Define the physics
   - Add *Nonisothermal Laminar Flow* multiphysics interface
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

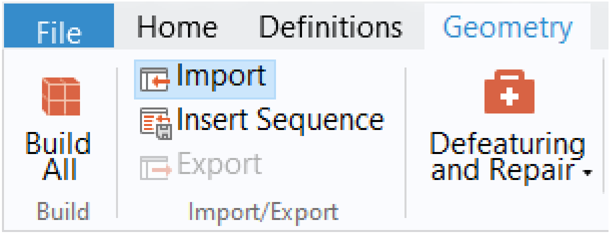


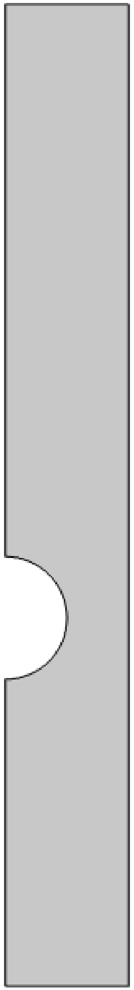


Screenshots of the steps performed to set up the model.

Import Geometry

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

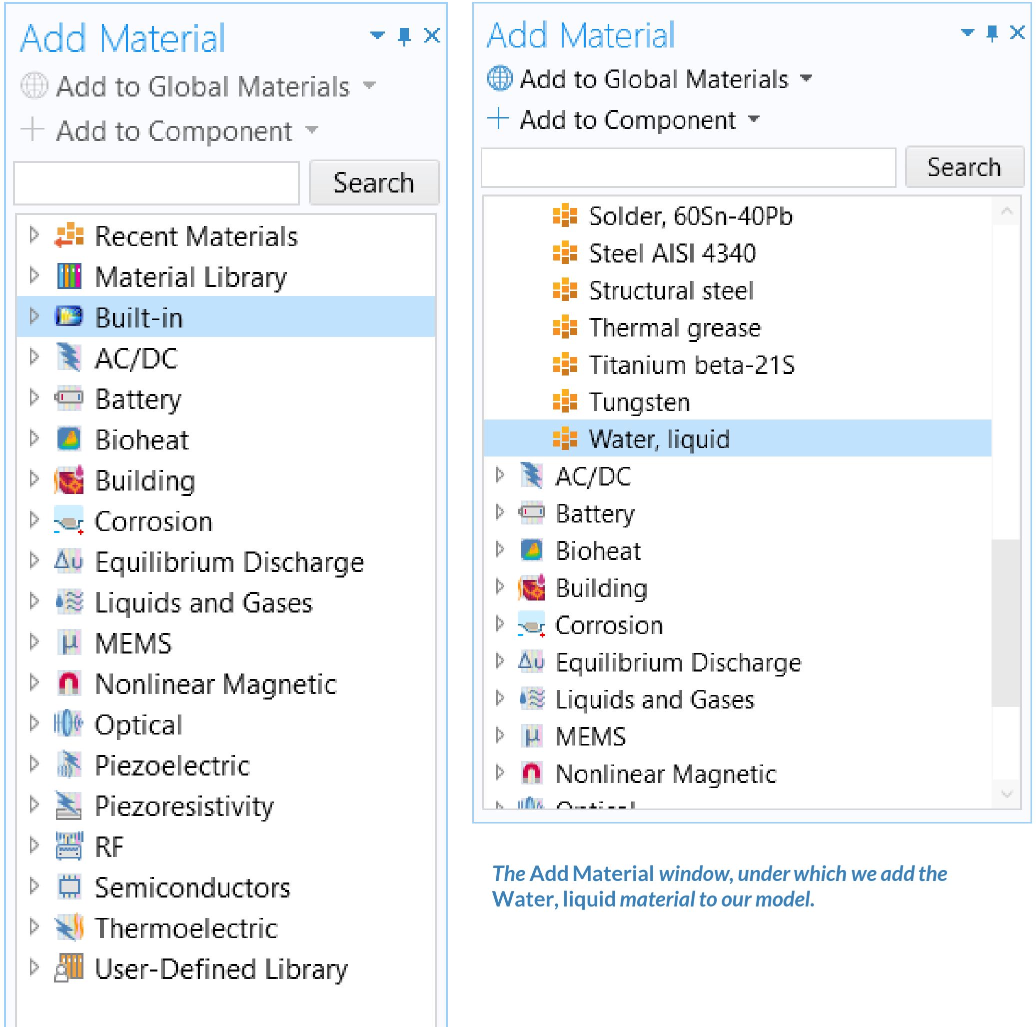




The Import button used and the free convection model geometry.

Assign Materials

- Fluid domain
  - Apply *Water, liquid*



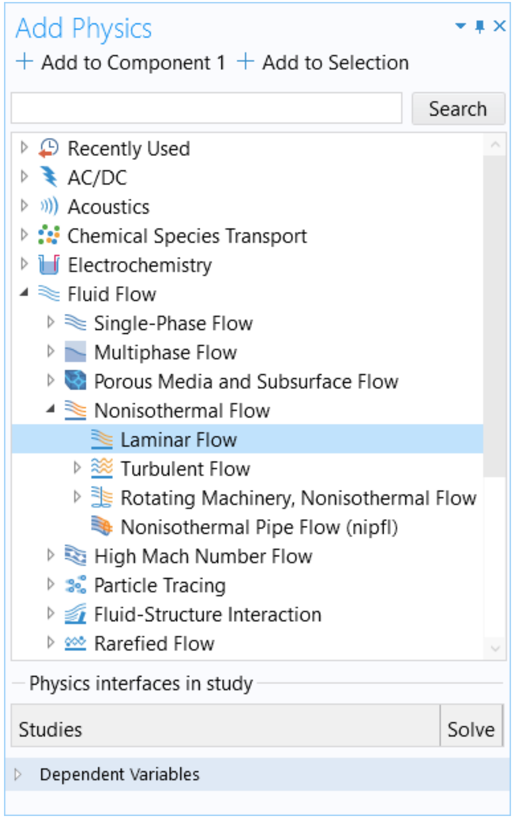
The **Add Material** *window, under which we add the* **Water, liquid** *material to our model.*

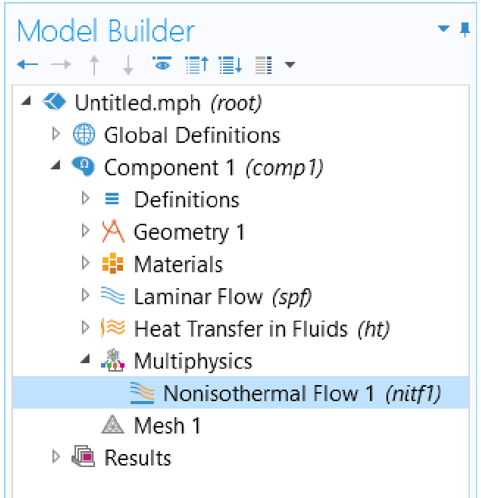
Fully Automatic Approach

Define the physics for the model using a predefined multiphysics interface

Procedure:

1. Add the physics interface
   - *Nonisothermal Laminar Flow* multiphysics interface
2. Define the physics settings
   - *Laminar Flow (spf)* interface
   - *Heat Transfer in Fluids* interface
   - *Nonisothermal Flow* multiphysics coupling





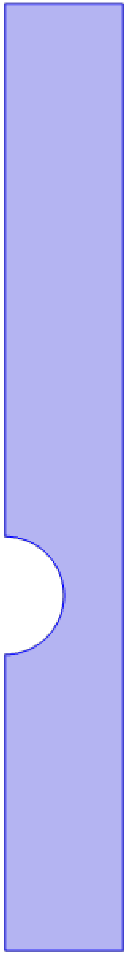
The Add Physics window and the model tree after the multiphysics interface has been added.

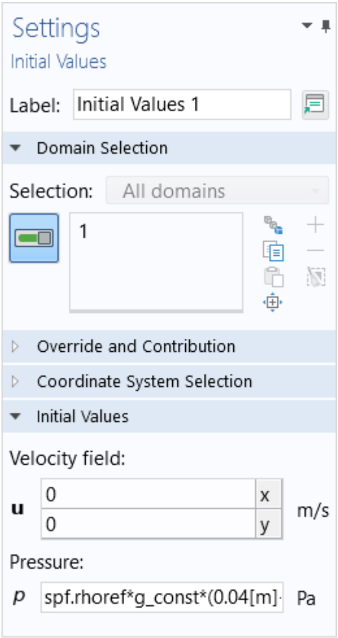
PHYSICS SETTINGS

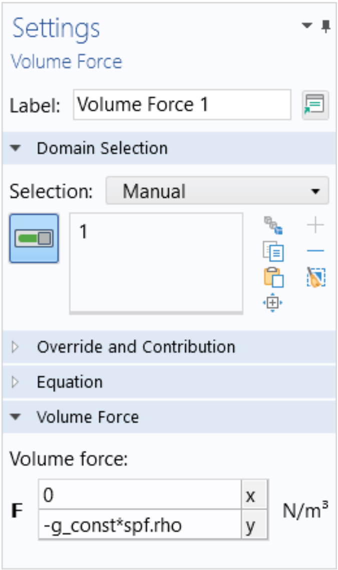
Laminar Flow

- 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

* = Refer to model specifications for values







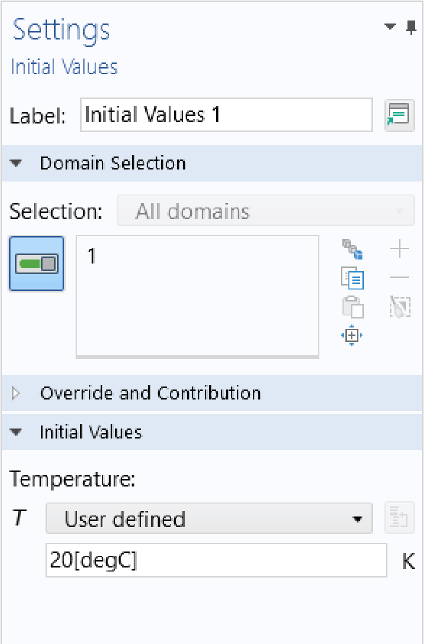
Settings for the Initial Values *and* Volume Force *nodes and their geometry selection.*

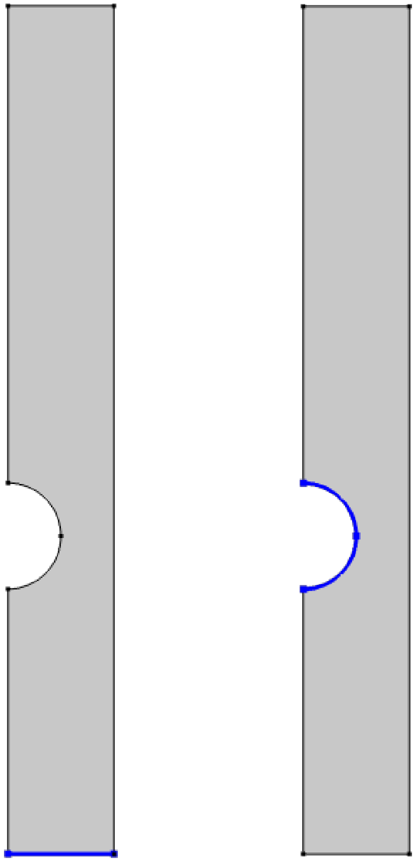
PHYSICS SETTINGS

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

* = Refer to model specifications for values



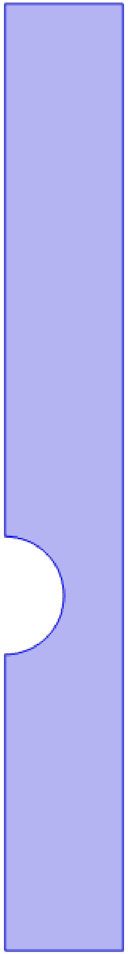


Settings for the Initial Values node (left) and the geometry selections for the inlet (center) and heater (right).

MULTIPHYSICS SETTINGS

Nonisothermal Flow

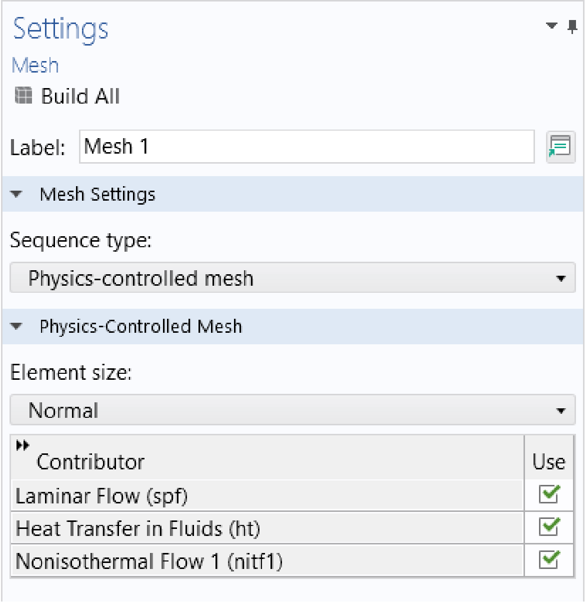
- Active in all domains
- Couples the *Laminar Flow (spf)* and *Heat Transfer in Fluids* interfaces
  - *Laminar Flow (spf)*
    - Incorporates the temperature field computed in the heat transfer interface
  - *Heat Transfer in Fluids*
    - Incorporates the pressure and velocity fields computed in the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

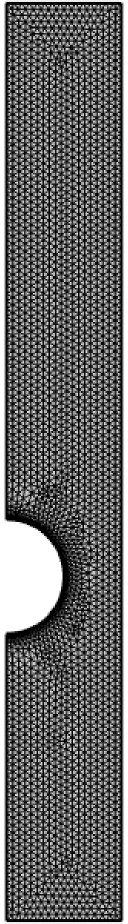


Geometry selection for the Nonisothermal Flow *multiphysics coupling node.*

Build the Mesh

Build the mesh using the default settings



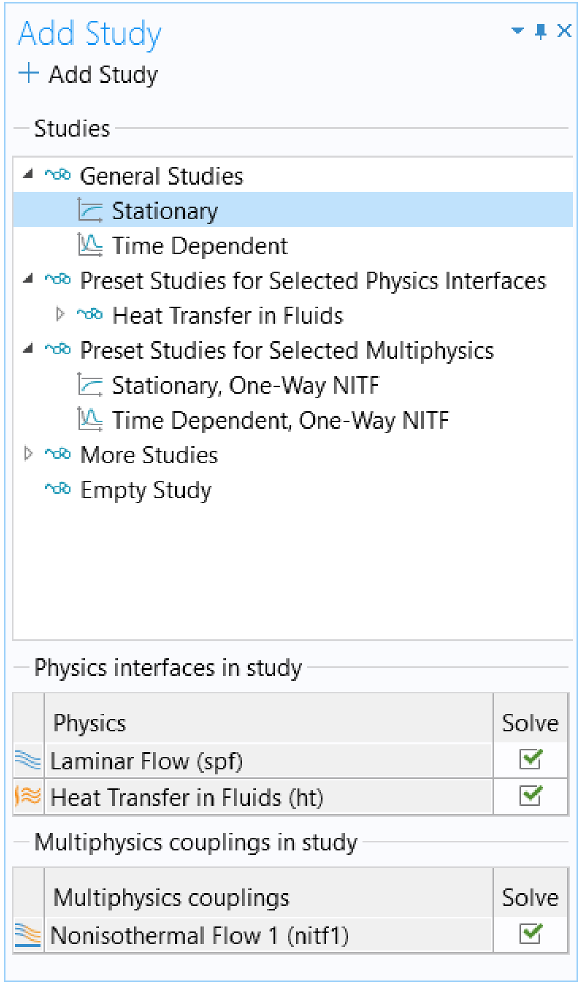


The settings used to generate the mesh for the model and the resulting mesh.

Run the Study

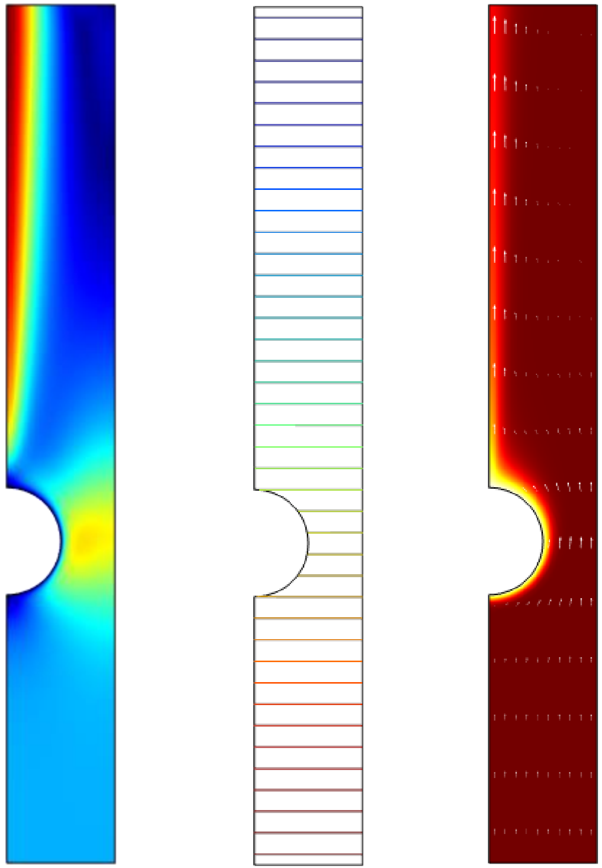
- Add a *Stationary* study
- Compute the model

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).