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

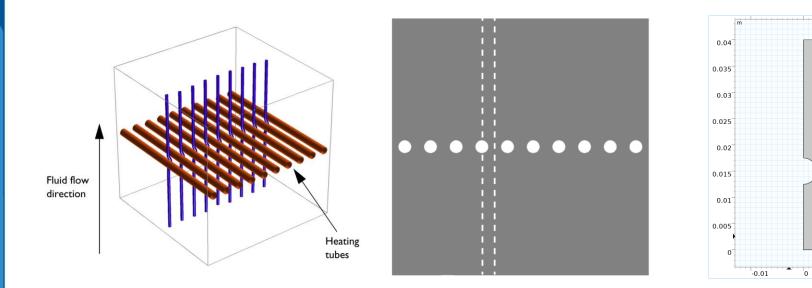


Model Overview

- 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

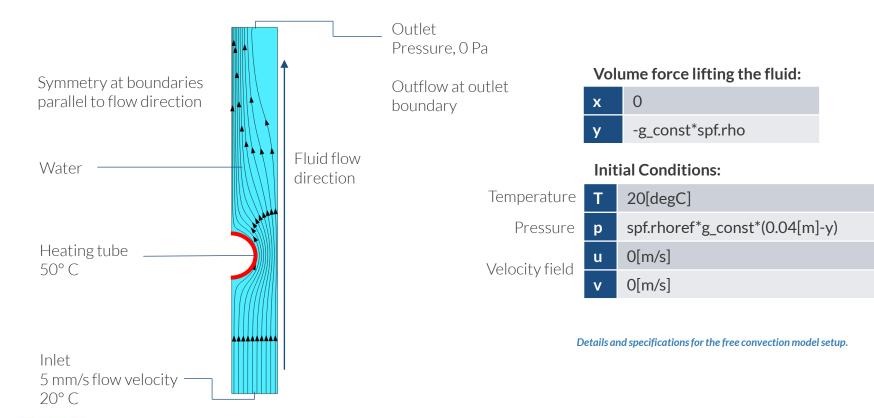


0.01

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).

ICOMSOL

Model Specifications



COMSOL

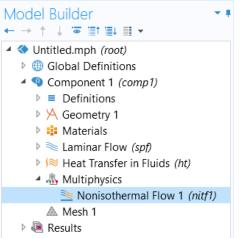
Fully Automatic Approach

Define the physics for the model using a predefined multiphysics interface

Procedure:

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

Add Physics **- +** × + Add to Component 1 + Add to Selection Search Recently Used AC/DC Acoustics Chemical Species Transport Electrochemistry 🔺 📚 Fluid Flow Single-Phase Flow Multiphase Flow Porous Media and Subsurface Flow A Nonisothermal Flow Naminar Flow Turbulent Flow Rotating Machinery, Nonisothermal Flow Nonisothermal Pipe Flow (nipfl) Model Builder High Mach Number Flow Particle Tracing Fluid-Structure Interaction ⊳ Rarefied Flow Physics interfaces in study Þ Solve Studies Materials Dependent Variables The Add Physics window and the model tree after the multiphysics interface has been added.





Modeling Workflow

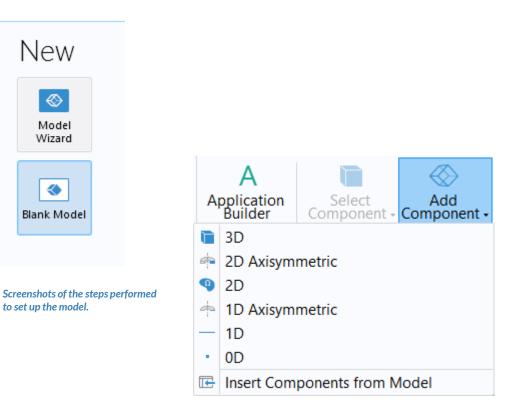
An outline of the steps used to set up, build, and compute this model to complete this modeling exercise is provided here.

- 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

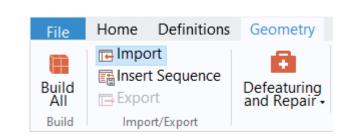
- Open the software
- Choose a Blank Model
- Add a 2D model component





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

	Material • • • •
+ Ac	Id to Component 👻
	Search
▷ 🚚	Recent Materials
Þ 🚻	Material Library
Þ 📴	Built-in
▷ 🖹	AC/DC
🗅	Battery
Þ 🚺	Bioheat
⊳ 📢	Building
Þ 🤜	Corrosion
⊳ Δυ	Equilibrium Discharge
⊳ ∙≋	Liquids and Gases
Þμ	MEMS
Þ 👖	Nonlinear Magnetic
⊳ ⊪0€	Optical
۸	Piezoelectric
۵ 🛓	Piezoresistivity
▷ 🚟	RF
▷ 🛱	Semiconductors
Þ 剩	Thermoelectric
⊳ 🎢	User-Defined Library

Add to Component =	
+ Add to Component 👻	
	Search
🚦 Solder, 60Sn-40Pb	
📫 Steel AISI 4340	
Structural steel	
Thermal grease	
🚦 Titanium beta-21S	
🚦 Tungsten	
🚦 Water, liquid	
AC/DC	
🖻 📟 Battery	
🕨 🧧 Bioheat	
🖻 📑 Building	
Corrosion	
Equilibrium Discharge	
Liquids and Gases	
MEMS	
🕨 🙍 Nonlinear Magnetic	

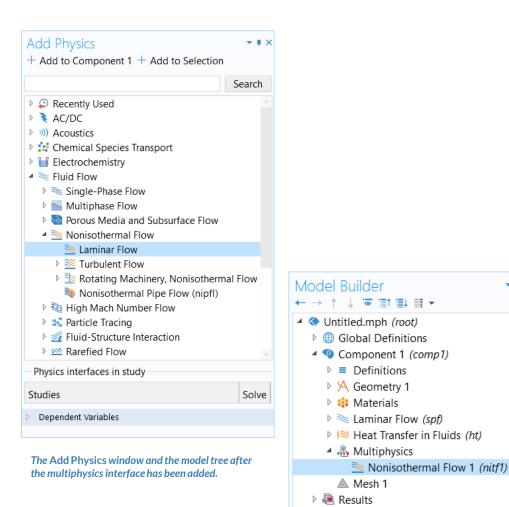
The Add Material window, under which we add the Water, liquid material to our model.

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



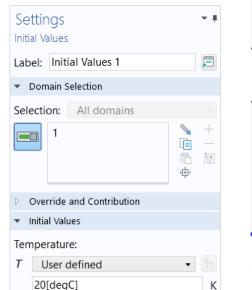
Laminar Flow

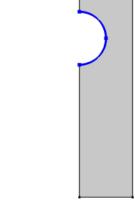
- Active in all domains
- Update Initial Values node*
 - Defines initial conditions
- Add Symmetry boundary condition
 - Defines symmetry boundaries
- Add Inlet boundary condition*
 - Defines where fluid flows into domain
- Add Outlet boundary condition
 - Defines where fluid flows out of domain
- Add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

	ettings		* #		ettir	-				
Init	ial Values			Vol	ume	For	ce			
Lat	oel: Initial Values 1		,=	Lab	oel:	Vol	ume Force	1		
•	Domain Selection			•	Dom	nain S	election			
Sel	ection: All domains			Sel	ecti	on:	Manual			
		r t t	+ 			1				
⊳	Override and Contribution			⊳	Over	rride	and Contrib	ution		
⊳	Coordinate System Selection			⊳	Equa	ation				
•	Initial Values			•	Volu	me F	orce			
Vel	ocity field:			Vol	ume	e for	ce:			
u	0	х	m/s	F	0				х	
	0	у	111/3		-g_	cons	st*spf.rho		у	
Pre	ssure:									
р	spf.rhoref*g_const*(0.04[r	n]·	Pa							

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

Settings Mesh Build All	~ #
Label: Mesh 1	Ę
 Mesh Settings 	
Sequence type:	
Physics-controlled mesh	•
 Physics-Controlled Mesh 	
Element size:	
Normal	•
* Contributor	Use
Laminar Flow (spf)	
Heat Transfer in Fluids (ht)	
Nonisothermal Flow 1 (nitf1)	

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





Run the Study

- Add a *Stationary* study
- Compute the model

+ Add Study	
Studies	
▲ 👒 General Studies	
🖂 Stationary	
🖄 Time Dependent	
Preset Studies for Selected Physic	cs Interfaces
 Meat Transfer in Fluids Preset Studies for Selected Multiple 	physics
∠ Stationary, One-Way NITF	physics
La Time Dependent, One-Way N	ITF
More Studies	
· ·· wore studies	
∞ Empty Study	
∞ Empty Study	Solve
 Empty Study Physics interfaces in study 	Solve
 Empty Study Physics interfaces in study Physics 	
 Empty Study Physics interfaces in study Physics Laminar Flow (spf) 	
 Physics interfaces in study Physics Laminar Flow (spf) Heat Transfer in Fluids (ht) 	



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

