COMSOL

Introduction to the Porous Media Flow Module

Version: COMSOL 5.5

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries, contact Technical Support, or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located at the COMSOL Access page at www.comsol.com/support/case. Other useful links include:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Contents

Introduction

Porous media are encountered in many natural and man-made systems. The need for advanced porous media modeling spans many industries and application areas such as processes in fuel cells, drying of pulp and paper, food production, filtration processes, and so on.

The Porous Media Flow Module extends the COMSOL Multiphysics modeling environment to the quantitative investigation of mass, momentum and energy transport in porous media. It is designed for researchers, engineers, teachers, and students, and it suits both single-physics and multiphysics modeling.

The contents of the Porous Media Flow Module are a set of fundamental building blocks which cover a wide array of physics questions. The physics interfaces it offers work on their own or linked to each other. They can also be coupled to physics interfaces already built into COMSOL Multiphysics, or to new equations you create.

The physics interfaces, options, and functionalities in this module are tailored to account for processes in porous media. The Heat Transfer interfaces, for example, include options to automate the calculation of effective thermal properties for multicomponent systems. The fluid flow equations represent a wide range of possibilities. Included are Richards' equation, which describes nonlinear flow in variably saturated porous media. The options for saturated porous media include Darcy's law for slow flow and the Brinkman equations where shear is non negligible. The Laminar Flow and Creeping Flow interfaces cover free flows at different Reynolds numbers. The module also treats the transport of chemical species. The Transport of Diluted Species interface account for the transport of species in free flow, saturated, and partially saturated porous media. A number of examples link these physics interfaces together.

You can use the Porous Media Flow Module in combination with essentially any of the other products in the COMSOL Multiphysics product suite. The module is a general interface for computing optimal solutions to engineering problems to, for example, improve a design so that it minimizes energy consumption or maximizes the output.

Porous Media Flow Module Interfaces

The Porous Media Flow Module contains a number of physics interfaces that predefine equations or sets of equations adapted to mass, momentum and energy transport in porous media. You can take the equations in these physics interfaces and their variables and then modify them, link them together, and couple them to physics interfaces represented elsewhere in COMSOL Multiphysics.

The physics interfaces are based on the laws for conservation of mass, momentum, and energy in porous media. The flow models contain different combinations and formulations of the conservation laws that apply to the physics of the fields. These laws of physics are translated into partial differential equations and are solved together with the specified initial and boundary conditions.

A physics interface defines a number of features. These features are used to specify the fluid properties, initial conditions, boundary conditions, and possible constraints. Each feature represents an operation describing a term or condition in the conservation equations. Such a term or condition can be defined on a geometric entity of the component, such as a domain, boundary, edge (for 3D components), or point.

Another useful tool in these physics interfaces is the ability to describe material properties such as density and viscosity by entering expressions that describe them as a function of other parameters, such as species concentration, pressure, or temperature. Many materials in the material libraries use temperature- and pressure-dependent property values.

Figure 1 shows the group of physics interfaces available with this module in addition to the COMSOL Multiphysics basic license. Use these interfaces to model chemical species transport, fluid flow, heat transfer and solid mechanics, to

make modeling easier, something that is briefly discussed next. See also Physics Interface Guide by Space Dimension and Study Type.
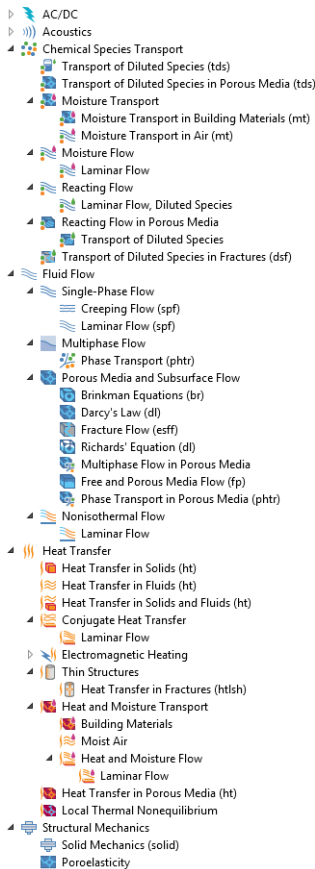

Figure 1: The physics interfaces for the Porous Media Flow Module as shown in the Model Wizard for a 3D application.

The module handles time-dependent and stationary problems for 1D, 2D, and 3D geometries with axisymmetry for 1D and 2D. The predefined physics interfaces cover four main categories: Chemical Species Transport (🔵), Fluid Flow (〰️), Heat Transfer (〰️), and Structural Mechanics (🔩), as discussed in the next pages.

Chemical Reaction and Mass Transport

The Transport of Diluted Species interface ( ) simulates chemical species transport through convection (when coupled to fluid flow), diffusion, and reactions, for mixtures where one component, a solvent, is present in excess.

The Transport of Diluted Species in Porous Media interface ( ) is tailored to model solute transport in saturated and partially saturated porous media. This physics interface characterizes the rate and transport of individual or multiple and interacting chemical species for systems containing fluids, solids, and gases. The equations supply predefined options to describe mass transfer by convection, adsorption, dispersion, diffusion, volatilization and reactions. You define the convective velocity from either of the included physics interfaces, or you set it to a predefined velocity profile.

The Laminar Flow, Diluted Species interface ( ) under the Reacting Flow branch combines the functionality of the Single-Phase Flow and Transport of Diluted Species interfaces. This multiphysics interface is primarily applied to model flow at low to intermediate Reynolds numbers in situations where the mass transport and flow fields are coupled.

The Transport of Diluted Species interface ( ) under the Reacting Flow in Porous Media branch combines the Brinkman Equations and the Transport of Diluted Species in Porous Media interfaces. This multiphysics interface is primarily applied to model the transport of diluted reacting mixtures in porous media.

The Transport of Diluted Species in Fractures interface ( ) is used to model the transport of a solutes along thin porous fractures, taking into account diffusion, dispersion, convection, and chemical reactions. The fractures are defined by boundaries in 2D and 3D, and the solutes are diluted in a solvent. The mass transport equation solved along the fractures is the tangential differential form of the convection-diffusion-reaction equation. Different effective diffusivity models are available.

The Moisture Transport in Building Materials interface ( ) is used to compute the relative humidity field in building materials. It simulates moisture transport by taking in account moisture storage, liquid transport by capillary suction forces.

The Moisture Transport in Air interface ( ) is used to compute the relative humidity distribution in air. It simulates moisture transport by vapor convection and diffusion in moist air and the evaporation or condensation on walls.

The Laminar Flow interface under the Moisture Flow folder ( ) combines all features from the Moisture Transport in Air and Single-Phase Flow interfaces. The Moisture Flow multiphysics coupling is automatically added. The fluid properties may depend on vapor concentration. The physics interface supports low Mach numbers (typically less than 0.3). Similarly to Conjugate Heat Transfer coupling,

the moisture transport interface can be coupled with additional turbulence models for the free flow with the use of the CFD Module.

Fluid Flow

Flow in porous media normally occur at low Reynolds numbers. The Reynolds number (Re) is a measure of the ratio of the fluid viscous to the inertial forces acting on the fluid and is given by: Re=$\rho\,UL/\mu$, where ρ is the fluid density, U is a characteristic velocity, L is a characteristic length scale, and μ is the dynamic viscosity.

The Creeping Flow interface ( ) approximates the Navier-Stokes equations for the case when the Reynolds number is significantly less than 1. This is often referred to as Stokes flow and it is appropriate for use when viscous flow is dominant.

The Laminar Flow interface ( ) describes fluid motion when turbulences are not present, and the Reynolds number is less than approximately 1000. The physics interface solves the Navier-Stokes equations for incompressible, weakly compressible, or compressible flows where the Mach number (Ma) is less than 0.3.

The Phase Transport interface ( )under the Multiphase Flow branch is used to simulate the transport of multiple immiscible phases in free flow. This interface solves for the averaged volume fractions of the phases.

Porous Media Flow

The Darcy's Law interface ( ) describes fluid movement through interstices in a porous medium. This physics interface can be used to model low velocity flows, for which the pressure gradient is the major driving force, and the flow is mostly influenced by the frictional resistance within the pores. Its use is within very low flows, or media where the permeability and porosity are very small. You can also set up multiple Darcy's Law interfaces to model multiphase flows involving more than one mobile phase. The Darcy's Law interface can also add non-Darcian effects such as Ergun or Forchheimer drag terms, which are important for Reynolds numbers bigger than 100.

The Fracture Flow interface ( ) is a variant of Darcy's law that defines the flow along interior boundaries representing fractures within a porous (or solid) medium.

The Richards' Equation interface ( ) analyzes flow in variably saturated porous media. With variably saturated flow, the permeability changes as fluids move through the medium, filling some pores and draining others. Richards' equation appears similar to the saturated flow equation set out in Darcy's Law, but it is notoriously nonlinear due to changes in material properties from unsaturated to

saturated conditions. The analytic formulas of van Genuchten and Brooks and Corey are frequently employed with variably saturated flow modeling.

The Brinkman Equations interface ( ) is used to model incompressible, weakly compressible, or compressible flows where the Mach number (Ma) is less than 0.3. You can select the Stokes-Brinkman flow feature to reduce the equations' dependence on inertial effects, when the Reynolds number is significantly less than one. The Brinkman Equations interface extends Darcy's law to describe the dissipation of the kinetic energy by viscous shear, similar to the Navier-Stokes equation. The Brinkman Equations interface can also add a Ergun or Forchheimer drag term, which are viscous drags proportional to the square of the velocity. These non-Darcian effects are important for Reynolds numbers bigger than 100.

The Free and Porous Media Flow interface ( ) is useful for modeling problems where free flow is connected to porous media, such as in fixed-bed reactors and catalytic converters. The Free and Porous Media Flow interface is used over at least two different domains, a free channel and a porous medium. The physics interface adds functionality that allows the equations to be optimized according to the flow properties of the relevant domain. For example, you can select the Stokes-Brinkman flow feature to reduce the equations' dependence on inertial effects in the porous domain, or just the Stokes' flow feature to reduce the equations' dependence on inertial effects in the free channel.

The Phase Transport in Porous Media interface ( ) is used to simulate the transport of multiple immiscible phases through a porous medium. The interface solves for the averaged volume fractions (saturations) of the phases, and it does not track the interface between the different phases, although microscopic interfacial effects are taken into account in the macroscopic equations through capillary pressure functions.

The Multiphase Flow in Porous Media interface ( ) combines the functionality of the Darcy's Law and Phase Transport in Porous Media interfaces. This multiphysics interface is intended to model flow and transport of multiple immiscible phases in a porous medium.

As always, the physics interfaces give you direct access to defining, with either constants or expressions, the material properties that describe the porous media flow. This includes the density, dynamic viscosity, permeability, and porosity.

Heat Transfer

The Heat Transfer interfaces apply to systems consisting of solids, fluids, and fluid-solid mixtures, and are able to calculate effective properties for porous media consisting of several fluids, gases and solid components, such as a rock formation with different mineral proportions.

The Heat Transfer in Solids interface (![icon]) describes, by default, heat transfer by conduction. The physics interface is also able to account for the heat flux due to translation in solids as well as for solid deformation, including volume or surface changes. In the case of irreversible thermally induced transformation, it accounts for the enthalpy and material properties changes.

The Heat Transfer in Fluids interface (![icon]) accounts for conduction and convection in gases and liquids as the default heat transfer mechanisms. The coupling to the flow field in the convection term may be entered manually, or it may be selected from a list that couples the heat transfer to an existing Fluid Flow interface. The Heat Transfer in Fluids interface can be solved simultaneously with the Laminar Flow interface, or when the flow field has already been calculated and the heat transfer problem is solved afterward, typically for simulations of forced convection.

The Heat Transfer in Solids and Fluids interface (![icon]) contains solids and fluids domains by default. It is aimed to simplify the setup of models where capabilities of Heat Transfer in Solids interface (![icon]) and Heat Transfer in Fluids interface (![icon]) are used, in particular in conjugate heat transfer applications.

The Conjugate Heat Transfer branch (![icon]) combine features from the Heat Transfer and Single-Phase Flow interfaces to describe heat transfer in solids and fluids and non isothermal flow in fluids. The heat transfer process is tightly coupled with the fluid flow problem via a predefined multiphysics coupling. Additional turbulence models for the free flow are available with the use of the CFD Module.

The Heat Transfer in Fractures interface (![icon]) under the Thin Structures branch (![icon]) describes heat transfer in fractures and thin porous media. It provides efficient models defined at the boundaries level representing thin three-dimensional domains. The simplest model assumes that the temperature changes through the fracture thickness can be neglected, while the general model computes the temperature variation across the fracture.

The Building Materials interface (![icon]) combines the Heat Transfer in Building interface (![icon]) with the Moisture Transport in Building Materials interface (![icon]). It can be used to model different moisture variations phenomena in building components such as drying of initial construction moisture, condensation due to migration of moisture from outside to inside, or moisture accumulation by interstitial condensation due to diffusion.

The Moist Air interface (![icon]) combines the Heat Transfer in Moist Air interface (![icon]) with the Moisture Transport in Air interface (![icon]). It is used to simulate the coupling between heat transfer and vapor transport in air and the evaporation and condensation on walls.

The Heat Transfer in Porous Media interface (![icon]) combines the heat conduction and convection in a solid-fluid system. This physics interface provides mixing rules for calculating the effective heat transfer properties, expressions for heat dispersion in porous media. Dispersion is caused by the tortuous path of the liquid in the porous medium, which would not be described if only the mean convective term was taken into account. This physics interface may be used for a wide range of porous materials, from porous structures to the simulation of heat transfer in soils and rocks, and also to model heat transfer in fractures.

The Local Thermal Nonequilibrium (LTNE) multiphysics interface (![icon]) is intended to simulate heat transfer in porous media where the temperatures in the porous matrix and the fluid are not in equilibrium. The absence of thermal equilibrium can result from fast transient changes, but it can also be observed in stationary scenarios. Typical applications are rapid heating (or cooling) of porous media using a hot (or cold) fluid, or internal heat generation in one of the phases. This effect can be observed in nuclear devices, electronics systems, and fuel cells.

These features interact seamlessly and can be used in combination in a single application. Surface-to-surface radiation can also be included in the energy equation, although this requires a license for the Heat Transfer Module.

Structural Mechanics

The Poroelasticity multiphysics interface (![icon]) combines a transient formulation of Darcy's law with a linear elastic material included in the Solid Mechanics interface. The poroelasticity coupling means that the pore fluid affects the compressibility of the porous medium, as well as changes in volumetric strains affect the porosity and the fluid flow.

Physics Interface Guide by Space Dimension and Study Type

The table lists the physics interfaces available with this module in addition to those included with the COMSOL basic license.

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
![icon] **Chemical Species Transport**				
Transport of Diluted Species	![icon]	tds	all dimensions	stationary; time dependent
Transport of Diluted Species in Porous Media		tds	all dimensions	stationary; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Transport of Diluted Species in Fractures		dsf	3D, 2D, 2D axisymmetric	stationary; time dependent
Moisture Transport				
Moisture Transport in Building Materials		mt	all dimensions	stationary; time dependent
Moisture Transport in Air		mt	all dimensions	stationary; time dependent
Moisture Flow				
Laminar Flow		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Reacting Flow				
Laminar Flow, Diluted Species		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Reacting Flow in Porous Media				
Transport of Diluted Species		rfds	3D, 2D, 2D axisymmetric	stationary; time dependent
Fluid Flow				
Single-Phase Flow				
Creeping Flow		spf	3D, 2D, 2D axisymmetric	stationary; time dependent
Laminar Flow		spf	3D, 2D, 2D axisymmetric	stationary; time dependent
Phase Transport				
Phase Transport		phtr	3D, 2D, 2D axisymmetric	stationary; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
🟦 **Porous Media and Subsurface Flow**				
Brinkman Equations	🟦	br	3D, 2D, 2D axisymmetric	stationary; time dependent
Darcy's Law	🟦	dl	all dimensions	stationary; time dependent
Fracture Flow	🟦	esff	3D, 2D, 2D axisymmetric	stationary; time dependent
Richards' Equation	🟦	dl	all dimensions	stationary; time dependent
Multiphase Flow in Porous Media	🟦	—	3D, 2D, 2D axisymmetric	stationary; time dependent
Free and Porous Media Flow	🟦	fp	3D, 2D, 2D axisymmetric	stationary; time dependent
Phase Transport in Porous Media	🟦	phtr	3D, 2D, 2D axisymmetric	stationary; time dependent
〰 **Heat Transfer**				
Heat Transfer in Solids	🟧	ht	all dimensions	stationary; time dependent
Heat Transfer in Fluids	🟧	ht	all dimensions	stationary; time dependent
Heat Transfer in Solids and Fluids	🟧	ht	all dimensions	stationary; time dependent
🟧 **Thin Structures**				
Heat Transfer in Fractures	🟧	htlsh	3D, 2D, 2D axisymmetric	stationary; time dependent; thermal perturbation, frequency domain
🟥 **Heat and Moisture Transport**				
Building Materials	🟥	—	all dimensions	stationary; time dependent; thermal perturbation, frequency domain

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Moist Air		—	all dimensions	stationary; time dependent; thermal perturbation, frequency domain
Heat and Moisture Flow				
Laminar Flow		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Heat Transfer in Porous Media		ht	all dimensions	stationary; time dependent
Local Thermal Nonequilibrium		—	all dimensions	stationary; time dependent; thermal perturbation, frequency domain
Structural Mechanics				
Poroelasticity		poro	3D, 2D, 2D axisymmetric	stationary; time dependent
[1] This physics interface is included with the core COMSOL package but has added functionality for this module.				
[1] Requires the addition of the Heat Transfer Module or CFD Module. [2] Requires the addition of the CFD Module.				

Tutorial Example — Packed Bed Latent Heat Storage

Thermal energy storage (TES) units are used to accumulate thermal energy from solar, geothermal, or waste heat sources. The simplest TES units are built from water tanks, where the solar energy is stored as sensible heat. These systems are called sensible heat storage (SHS) units. The thermal capacity of these tanks can be further increased by including latent heat, which gives rise to latent heat storage (LHS) units. Typically, LHS tanks contain spherical capsules filled with paraffin wax as phase change material. Paraffin is a suitable phase change material, as it is relatively inexpensive, reliable and nontoxic, and it is commercially available for a wide rage melting temperatures.

This example is inspired by the experimental investigation found in Ref. 1. It models the flow through a packed-bed storage tank, and it includes the effects of heat transfer with phase change and local thermal nonequilibrium (LTNE) while charging the LHS unit.
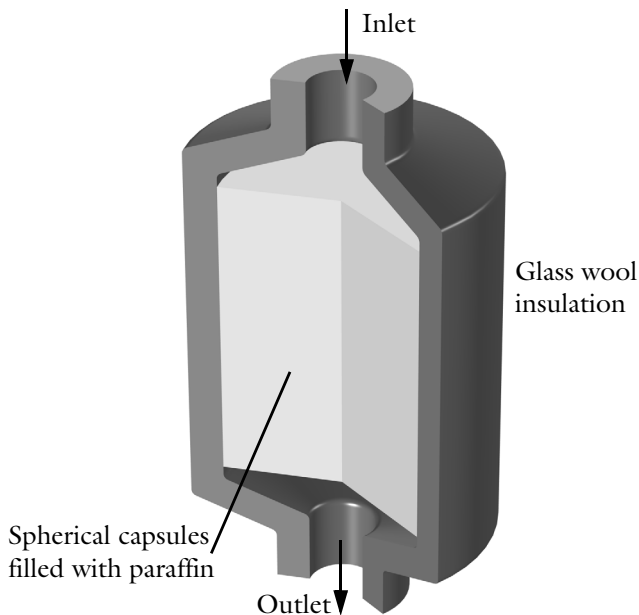
Figure 2: Latent heat storage unit.

MODEL DEFINITION

Paraffin-filled spherical capsules with a diameter of d_p = 55 mm are stored in a tank of 36 cm in diameter and 47 cm in height. The porosity of this bed is ε_p = 0.49. The model geometry is shown in Figure 2.

The material properties of paraffin wax are listed in the following table.

MATERIAL PROPERTY	PARAFFIN, SOLID	PARAFFIN, LIQUID
Density ρ (kg/m^3)	861	778
Heat Capacity, C_p (J/(kg·K))	1850	2384
Thermal conductivity, k (W/(m·K))	0.4	0.15
Melting temperature, T_m (°C)	60	
Latent heat of fusion, L (J/kg)	213	

Geometry, material properties, and operating conditions are taken from Ref. 1.

The initial temperature in the tank is set to 32 °C. Warm water flows through the tank with a flow rate of m_{in} = 2 l/min. During thermal charging the water is continuously heated up by a solar collector that delivers a power of Q_u = 375 W. The temperature difference at the tank's inlet and outlet is given by the relation

$$\frac{Q_u}{m_{in}} = \rho C_p (T_{in} - T_{out}) \tag{1}$$

here, T_{in} and T_{out} are the inlet and outlet temperatures, and ρ and C_p are the density and heat capacity of water.

Ergun equation describes non-Darcian flow through the packed bed, which estimates the pressure drop as a function of the velocity field $\mathbf{u}$

$$\nabla p = -\frac{\mu}{\kappa}\mathbf{u} - \frac{1{,}75(1-\varepsilon_p)}{d_p\varepsilon_p^3}\rho|\mathbf{u}|\mathbf{u} \tag{2}$$

here, μ (Pa·s) and ρ (kg/m^3) are the viscosity and density of water, d_p (m) is the spheres' diameter, and ε_p the bed porosity. The permeability κ (m^2) of the packed bed is given by

$$\kappa = \frac{d_p^2\varepsilon_p^3}{150(1-\varepsilon_p)^2} \tag{3}$$

The Reynolds number in the packed bed can be estimated as

$$Re = \frac{d_p v\rho}{(1-\varepsilon_p)\mu} \tag{4}$$

The maximum velocity in the tank, v, is about 6 mm/s, which implies a Reynolds number of about 600. For this Reynolds number the flow field is assumed to be independent of the temperature distribution, such that a stationary field can be computed before running the thermal simulation. This is a reasonable simplification that reduces the computational effort.

The relative large diameter of the capsules as compared to the tank dimensions suggests a significant temperature difference between the encapsulated paraffin and the surrounding water flow, thus a local thermal nonequilibrium (LTNE) approach is considered in the example.

The Local Thermal Nonequilibrium multiphysics interface is used to couple the Heat Transfer in Solids with the Heat Transfer in Fluids interfaces. The heat transferred from the paraffin-filled capsules to the water is modeled with a heat source

$$Q_f = \frac{q_{sf}}{\varepsilon_p}(T_s - T_f) \tag{5}$$

here, T_s and T_f are the paraffin and water temperatures, and q_{sf} (W/(m3·K)) is the interstitial convective heat transfer coefficient, which for spherical capsules reads

$$q_{sf} = \frac{6(1 - \varepsilon_p)}{d_p} h_{sf} \tag{6}$$

The interstitial heat transfer coefficient h_{sf} follows a Nusselt number correlation see the Theory for the Local Thermal Nonequilibrium Interface section in the documentation for more information. Convection inside the capsules is neglected, thus the paraffin wax is treated as a solid or immobile liquid.

RESULTS

The tank reaches a temperature of 70 °C after approximately 13 hours. The resulting velocity and temperature distribution is shown in Figure 3
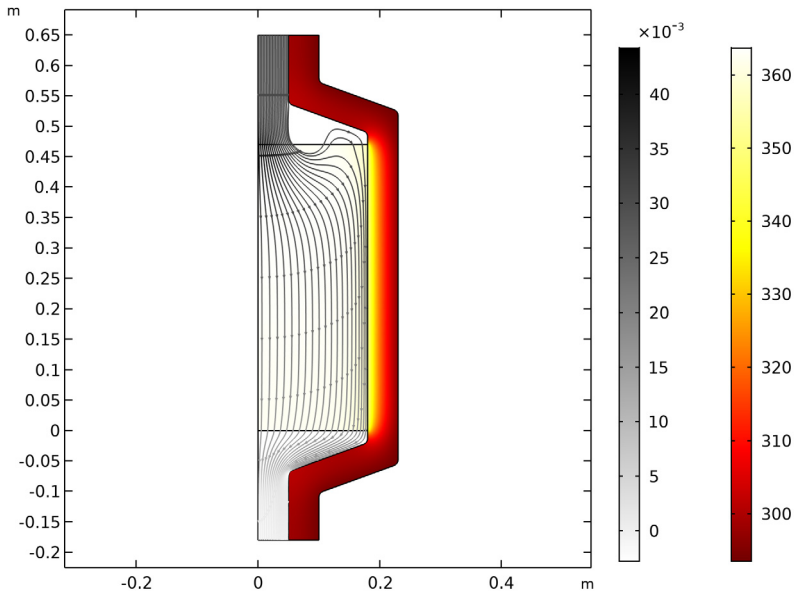
Figure 3: Velocity field (streamlines) with the gray color indicating the pressure and temperature field (color) after 12 hours of thermal charging.

Figure 4 shows the evolution of the paraffin temperature, water temperature and the weighted average (porous medium) temperature at three different points located in the central axis of the tank. During the phase change, the encapsulated paraffin is not in thermal equilibrium with the surrounding water. Measuring the water temperature alone at the inlet or outlet does not give accurate information about neither the temperature inside the capsules nor the phase in which the paraffin wax is.
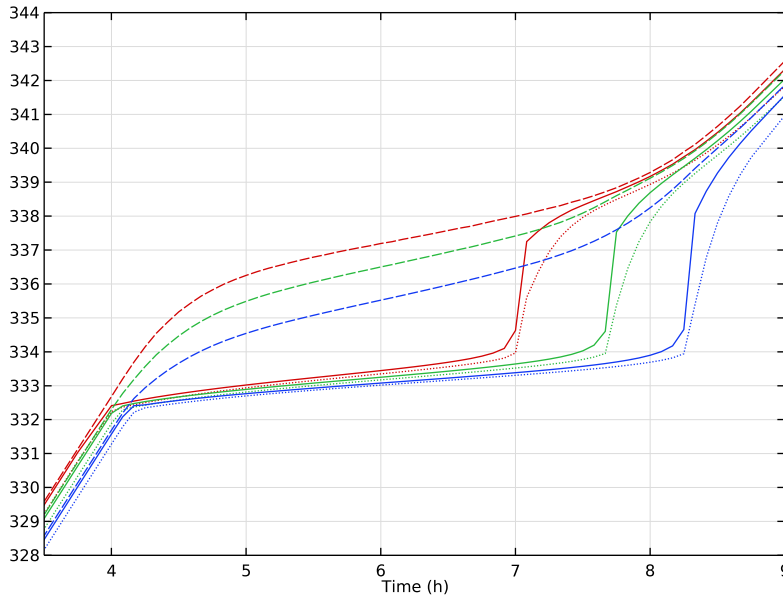
Figure 4: Evolution of water (dashed), paraffin (dotted) and average porous medium temperature (solid) during phase change for top (red), center (green) and low (blue) position.

The evolution of the phase distribution changes as the paraffin wax warms up. Liquid paraffin appears after approximately 4 hours, when water heats up the paraffin wax to its melting temperature of 60°C. The wax is completely molten after about 10 hours. The latent heat storage tank is considered fully charged as soon as a temperature of 70°C is reached everywhere in the tank, which happens after approximately 13 hours.
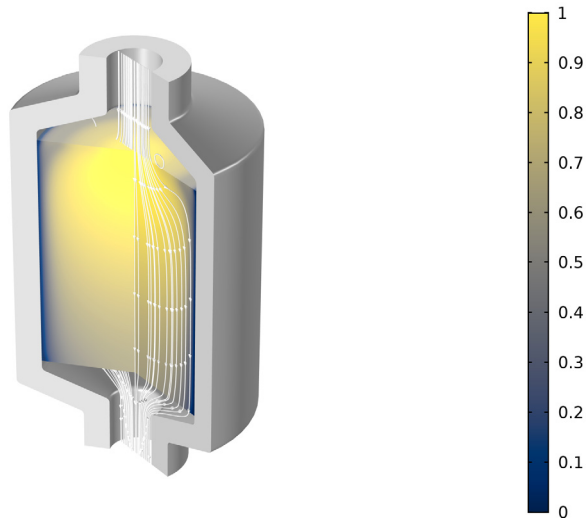
Figure 5: Paraffin solid phase (blue) and liquid phase (yellow) after 7 hours of thermal charging the LHS unit.

Figure 5 shows the paraffin phase distribution after 7 hours. Near the walls, where the flow velocity is negligible, the phase transition has not yet begun and the paraffin wax is still solid, while it is melted in the center of the tank.

REFERENCES

1. N. Nallusamy et al, "Study on performance of a packed bed latent heat thermal energy storage unit integrated with solar water heating system," J*ournal of Zhejiang University-SCIENCE A*, vol. 7, pp. 1422–1430, 2006.

Model Wizard

Note: These instructions are for the user interface on Windows but apply, with minor differences, also to Linux and Mac.

1 To start the software, double-click the COMSOL icon on the desktop. When the software opens, you can choose to use the Model Wizard to create a new

COMSOL model or Blank Model 🔷. to create one manually. For this tutorial, click the Model Wizard 🔷 button.

If COMSOL is already open, you can start the Model Wizard by selecting New 📄 from the File menu and then click Model Wizard 🔷.

The Model Wizard guides you through the first steps of setting up a model. The next window lets you select the dimension of the modeling space.

2 In the Select Space Dimension window click 2D Axisymmetric 🔄.

3 In the Select Physics tree, expand the Fluid Flow folder, and under the Porous Media and Subsurface Flow folder double-click Free and Porous Media Flow 🟦 to add it to the Added physics interfaces list. You can also click Add or right-click the icon and choose Add Physics ➕.

4 In the Select Physics tree under the Heat Transfer folder, double-click Local Thermal Nonequilibrium 🔴 to add it to the Added physics interfaces list. You can also click Add or right-click and choose Add Physics ➕.

5 Click Done ☑.

Geometry

You can build the geometry from geometric primitives. Here, use instead a file containing the sequence of geometry features that has been provided for convenience.

The location of the file used in this exercise varies based on your installation. For example, if the installation is on your hard drive, the file path might be similar to `C:\Program Files\COMSOL\COMSOL55\Multiphysics\applications\`.

1 On the Geometry toolbar choose Import 🔳.

2 In the Settings window for Import, locate the Import section, and browse to the model's Application Libraries folder. Double-click the file `packed_bed_latent_heat_storage.mphbin`.

3 In the Settings window for Import, click Import.

4 Click Build All Objects ⊞ to display the geometry.
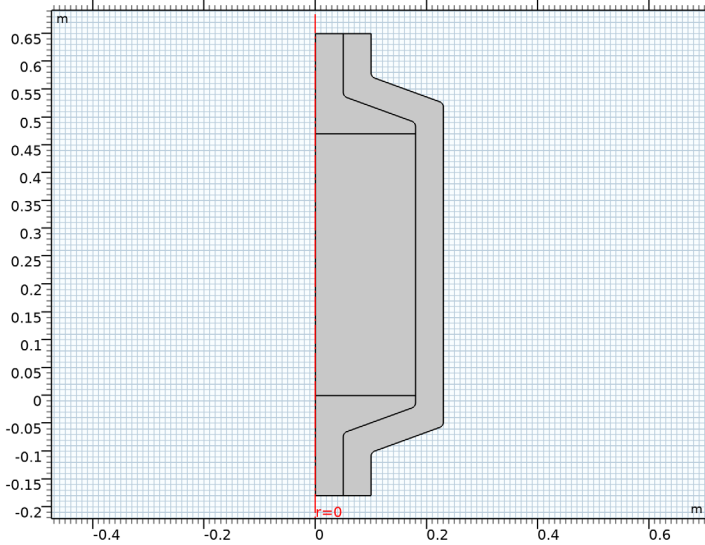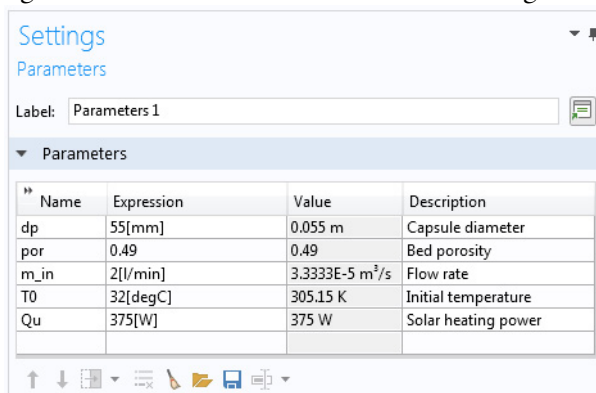
Global Definitions — Parameters

Add the parameters needed in this example.

Parameters

1 In the Model Builder window under Global Definitions ⊕ click Parameters P_i .

2 In the Settings window for Parameters enter the following data

Name	Expression	Value	Description
dp	55[mm]	0.055 m	Capsule diameter
por	0.49	0.49	Bed porosity
m_in	2[l/min]	3.3333E-5 m³/s	Flow rate
T0	32[degC]	305.15 K	Initial temperature
Qu	375[W]	375 W	Solar heating power

Materials

Define the material nodes for water, paraffin, and the insulating glass.

Water, liquid

1 In the Home toolbar, click
Add Material 🔩 .

2 Go to the Add Material window,
expand the Built-in folder and select
Water, liquid. Click
Add to Component ✚ in the window
toolbar to add it o the Materials list.
You can also double-click on it or
right-click and choose
Add to Component 1 (comp 1) 🖘 .

3 On the Home toolbar click
Add Material 🔩 again to close the
window.
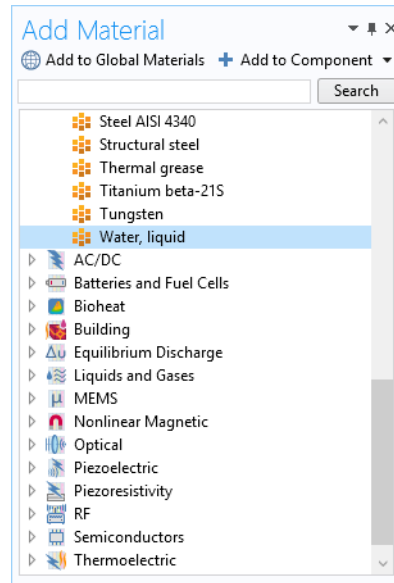
Paraffin, solid

4 In the Model Builder window, under
Component 1 (comp 1) 🖘,
right-click Materials 🔩 and choose
Blank Material 🔩 .

5 In the Label text field under the Settings window for Material, type
`Paraffin, solid`.

Repeat the same procedure to create two more material nodes, call them
`Paraffin, liquid` and `Glass_wool` respectively. After setting up the physics
interfaces you can fill in the required material properties in these nodes.

Definitions

Average 1 (aveop1)

Create an average operator to evaluate the inlet temperature.

1 In the Definitions toolbar click Nonlocal Couplings 🔗 and choose
Average ᴬᵛ .

2 In the Settings window for Average locate the Source Selection section, and from the Geometric entity level list choose Boundary.

3 Select Boundary **2** only.

Minimum 1 (minop1)

Create a minimum operator to evaluate the minimum temperature in the packed bed.

4 In the Definitions toolbar click Nonlocal Couplings 🔗 and choose Minimum MIN .

5 Select Domain **2** only.

Variables 1

As water is pumped through a closed loop, its temperature increases over time during the charging process. Define the variables `deltaT` and `T_in` as functions of the outlet temperature using Equation 1. Also, define the minimum temperature in the packed bed with the help of the minimum operator `minop1`, which is used later as stop condition in the solver settings.

6 In the Definitions toolbar click Local Variables a= .

7 In the Settings window for Variables, locate the Variables section and enter the following settings.

Settings

Variables

Label: Variables 1

Geometric Entity Selection

Geometric entity level: Entire model

ON

▼ Variables

Name	Expression	Unit	Description
deltaT	Qu/m_in/aveop1(ht2.Cp)/aveop1(ht2.rho)	K	Temperature increase
T_in	aveop1(T2)+deltaT	K	Inlet temperature
T_min	minop1(T)	K	Minimum temperature
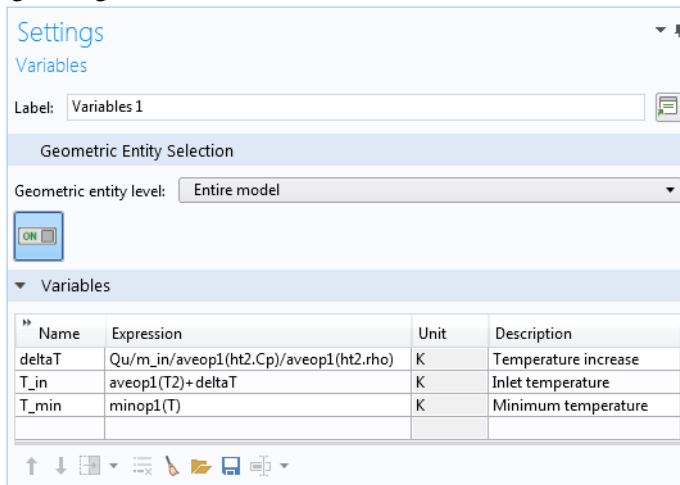
The variables `ht2.Cp` and `ht2.rho` are the heat capacity and density of water as defined by the Heat Transfer in Fluids interface.

Explicit 1

Create a selection for the tank outer boundary to apply a heat flux condition.

8 In the Definitions toolbar click Explicit ⬝⬝ .

9 In the Settings window for Explicit locate the Input Entities section, and from the Geometric entity level list choose Boundary.

10 Select the Group by continuous tangent check box.

11 Select Boundaries 14–17, 21–23, 26, 27, 30, and 31. In the Label text field type Heat flux boundary.

Free and Porous Media Flow

Fluid and Matrix Properties 1

1 In the Model Builder window, under Component 1 (comp 1) ⬥, click Free and Porous Media Flow (fp) 🔲 and select Domains 1–3. These are the free flow and packed bed domains.

2 In the Physics toolbar click Domains ⬤ and choose Fluid and Matrix Properties ⬤. Select Domain 2 only.

3 In the Settings window for Fluid and Matrix Properties locate the Fluid Properties section. From the Fluid material list choose Water, liquid (mat1).

4 Locate the Porous Matrix Properties section. From the ε_p list choose User defined and type por.

5 From the Permeability model list choose Non-Darcian, and from the Non-Darcian flow model list choose Ergun. In the Particle diameter d_p field type dp.

Next, apply the boundary conditions for the flow.

Inlet 1

1 In the Physics toolbar click Boundaries ⬤ and choose Inlet ⬤. Select Boundary 7 only.

2 In the Settings window for Inlet locate the Boundary Condition section, from the list choose Fully developed flow. Click the Flow rate button, and in the Flow rate V_0 field type m_in.

Outlet 1

1 In the Physics toolbar click Boundaries ⬤ and choose Outlet ⬤. Select Boundary 2 only. The default outlet condition specifies a zero relative pressure, and suppresses back flow effects.

Solid 1

Continue with setting up the heat transfer properties for the insulating glass wool.

1 In the Model Builder window, under Component 1 (comp1) ⬡ click Heat Transfer in Solids (ht) 🔶 and select Domains 2 and 4 only.

2 In the Model Builder window, under Component 1 (comp1)>Heat Transfer in Solids (ht) 🔶 click Solid 1 ⬤.

3 In the Settings window for Solid, locate the Solid Material section and from the list, choose Glass wool (mat4).

Solid 2

Continue with setting up the paraffin properties.

4 In the Physics toolbar click Domains ⬤ and choose Solid ⬤. Select Domain 2 only.

5 In the Physics toolbar click Attributes ⬡ and choose Phase Change Material ⬤.

6 In the Settings window for Phase Change Material locate the Density section.

7 From the ρ list, choose User defined. In the associated text field type 820[kg/m^3], which is the average density of liquid and solid paraffin. The paraffin density is assumed to remain constant, which is a reasonable assumption.

8 Locate the Phase Change section. In the $T_{\text{pc}, 1 \to 2}$ field type 60[degC]. In the in the $\Delta T_{1 \to 2}$ field type 2[K], and in the $L_{1 \to 2}$ field type 213[kJ/kg].

9 Locate the Phase 1 section. From the Material, phase 1 list choose Paraffin, solid (mat2).

10 Locate the Phase 2 section. From the Material, phase 2 list choose Paraffin, liquid (mat3).

Initial Values 1

11 In the Model Builder window, click Initial Values 1. In the Settings window locate the Initial Values section and type T0 in the T text field.

Heat Flux 1

Apply the boundary conditions for the heat transfer in the packed bed.

12 In the Physics toolbar click Boundaries ⬢ and choose Heat Flux ⬢.

13 In the Settings window for Heat Flux locate the Boundary Selection section, from the Selection list choose Heat flux boundary.

14 Locate the Heat Flux section and click the Convective heat flux button. In the Heat transfer coefficient h field type `5[W/m^2/K]`.

Heat Transfer in Fluids 2 (ht2)

Continue with setting up the heat transfer properties for the water flow.

1 In the Model Builder window under Component 1 (comp1) ⛵ click Heat Transfer in Fluids 2 (ht2) 〰 and select Domains 1–3 only.

Fluid 1

2 In the Model Builder window, under Component 1 (comp1)>Heat Transfer in Fluids 2 (ht2) 〰 click Fluid 1 ●.

3 In the Settings window for Fluid, locate the Fluid Material section and from the list choose Water, liquid (mat1).

Initial Values 1

4 In the Model Builder window, under Component 1 (comp1)>Heat Transfer in Fluids 2 (ht2) 〰 click Initial Values 1 ●.

5 In the Settings window for Initial Values, locate the Initial Values section.

6 In the T2 text field type `T0`.

Next, apply the boundary conditions for heat transfer in water.

Outflow 1

7 In the Physics toolbar click Boundaries ⊖ and choose Outflow ⊖. Select Boundary 2 only.

Temperature 1

8 In the Physics toolbar click Boundaries ⊖ and choose Temperature ⊖. Select Boundary 7 only, which corresponds to the water inlet.

9 In the Settings window for Temperature locate the Temperature section, in the T_0 field type `T_in`.

Now, fill out the remaining material properties. Since you set up the physics, the software automatically detects which properties are required for the simulation.

Paraffin, solid (mat2)

10 In the Model Builder window, under Component 1 (comp1)>Materials 🔡 click Paraffin, solid (mat2).

11 In the Settings window for Material locate the Material Contents section and enter the following settings:

|	Property	Variable	Value	Unit	Property group
☑	Thermal conductivity	k_iso ;...	0.4	W/(m·K)	Basic
☑	Heat capacity at constant pressure	Cp	1850	J/(kg·K)	Basic

Paraffin, liquid (mat3)

12 In the Model Builder window, under Component 1 (comp1)>Materials click Paraffin, liquid (mat3).

13 In the Settings window for Material locate the Material Contents section and enter the following settings:

	Property	Variable	Value	Unit	Property group
☑	Thermal conductivity	k_iso ;...	0.15	W/(m·K)	Basic
☑	Heat capacity at constant pressure	Cp	2384	J/(kg·K)	Basic

Glass wool (mat4)

14 In the Model Builder window, under Component 1 (comp1)>Materials click Glass wool (mat4) and select Domain 4 only.

15 In the Settings window for Material locate the Material Contents section and enter the following settings:

	Property	Variable	Value	Unit	Property group
☑	Thermal conductivity	k_iso ;...	0.025	W/(m·K)	Basic
☑	Density	rho	1250	kg/m³	Basic
☑	Heat capacity at constant pressure	Cp	850	J/(kg·K)	Basic

Add Multiphysics

1 In the Physics toolbar, click Add Multiphysics to couple the Free and Porous Media Flow with the Heat Transfer in Fluids interface.

2 Go to the Add Multiphysics window and find the
Select the physics interfaces you want to couple subsection. In the table, enter
the following settings:
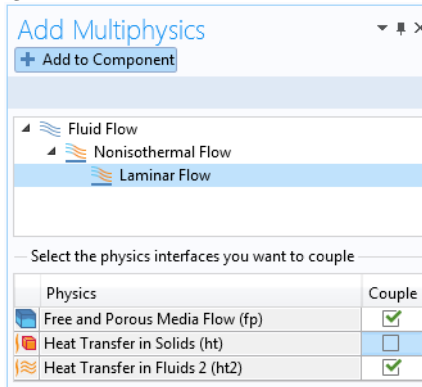
3 In the tree, select Fluid Flow>Nonisothermal Flow>Laminar Flow ≋.

4 Click ✚ Add to Component. In the Physics toolbar, click again
Add Multiphysics 🔬 to close the window.

Local Thermal Nonequilibrium 1 (ltne1)

5 In the Model Builder window under Component 1 (comp1)>Multiphysics 🔬
click Local Thermal Nonequilibrium 1 (ltne1) 🔶.

6 In the Settings window for Local Thermal Nonequilibrium locate the
Local Thermal Nonequilibrium Settings section, and in the θ_p text field type
`1-por`.

7 From the Interstitial convective heat transfer coefficient list choose Spherical pellet bed, and in the r_p text field type dp/2.

▼ Coupled Interfaces

Heat transfer in solids:

| Heat Transfer in Solids (ht) | ▼ |

Heat transfer in fluids:

| Heat Transfer in Fluids 2 (ht2) | ▼ |

▼ Local Thermal Nonequilibrium Settings

Solid volume fraction:

θ_p | 1-por | 1

Interstitial convective heat transfer coefficient:

| Spherical pellet bed | ▼ |

Average pellet radius:

r_p | dp/2 | m

Dynamic viscosity:

μ | From material | ▼ |

Mesh 1

1 In the Model Builder window under Component 1 (comp1) click Mesh 1.

2 In the Settings window for Mesh locate the Physics-Controlled Mesh section, and from the Element size list choose Fine.

3 Click Build All, to see the automatically generated mesh.

The figure below shows the boundary layer mesh at the walls. Zoom in to the mesh using the zoom function on the Graphics toolbar ⊕ to confirm that it matches the figure.
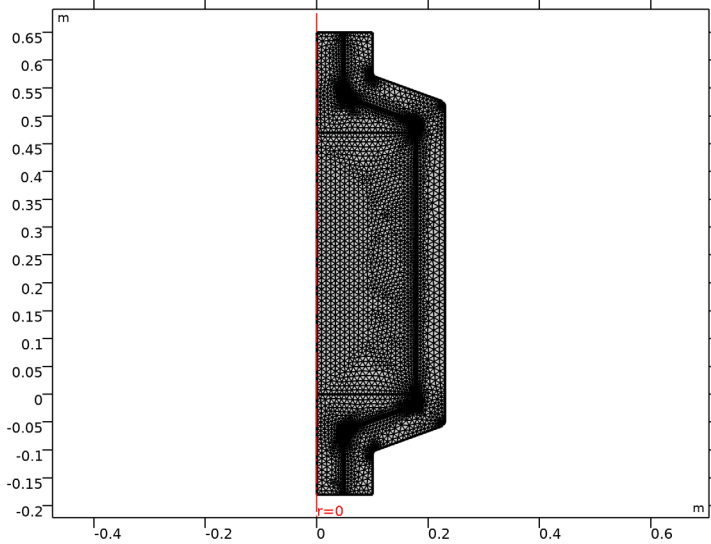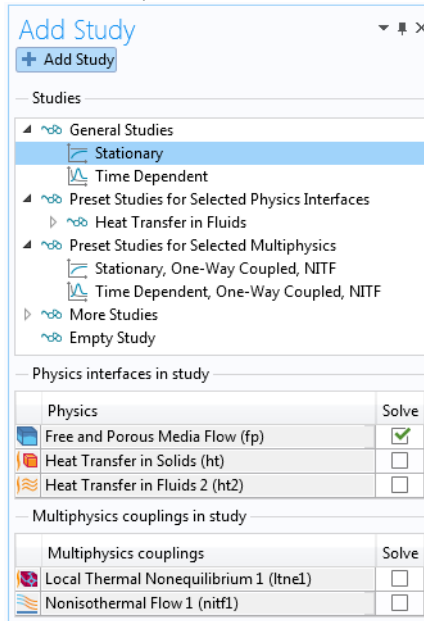

Add Study

Stationary

Add a stationary study step to solve for the velocity profile.

1 In the Home toolbar click Add Study ⁸. In the Add Study window find the Studies subsection, expand the General Studies ⁸ folder, and select Stationary ⟋ .

2 Find the Physics interfaces in study subsection, and the
Multiphysics couplings in study subsection. Enter the following settings in the
tables to solve fist the stationary flow field:

3 Click ✚ Add Study in the window toolbar.

4 To clear the default plots generated by the solver, in the Model Builder window
click Study 1 ⌒⌒ .

5 In the Settings window for Study locate the Study Settings section, and clear the
Generate default plots check box.
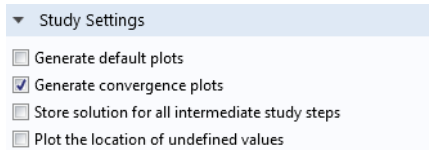
Next, add a Time Dependent step to solve the coupled heat transfer problem.

Time Dependent

6 In the Study toolbar click Study Steps ⌒ and choose Time Dependent ⌒ .

7 In the Settings window for Time Dependent locate the Study Settings section,
and from the Time unit list choose h.

8 In the Times text field type
`range(0,0.25,3.75) range(4,5[min],8.75) range(9,0.25,24)`. This

will carry on the simulation time for 24 hours. The time steps are chosen such that the phase change is properly resolved.

9 From the Tolerance list choose User controlled, and in the Relative tolerance text field type `1e-4`.

10 Locate the Physics and Variables Selection section and enter the following settings in the tables.

▾ Physics and Variables Selection		
☐ Modify model configuration for study step		

» Physics interface	Solve for	Discretization
Free and Porous Media Flow (fp)	☐	Physics settings ▾
Heat Transfer in Solids (ht)	☑	Physics settings ▾
Heat Transfer in Fluids 2 (ht2)	☑	Physics settings ▾

» Multiphysics couplings	Solve for
Local Thermal Nonequilibrium 1 (ltne1)	☑
Nonisothermal Flow 1 (nitf1)	☑

11 In the Study toolbar, click Show Default Solver ▸ .

12 In the Solver Configurations ▸ node, expand the Solution 1 (sol1) ▸ node and click the Time-Dependent Solver 1 ⚙ node.

13 In the Settings window for Time-Dependent Solver, click to expand the Time Stepping section, and from the Steps taken by solver list choose Strict. This option forces the time-dependent solver to strictly use the time steps specified before.

The time it takes to charge the tank is not known a priori. In order to reduce the computational time, a stop condition for the time dependent solver is used to stop the simulation after a temperature of 70°C is reached in every point inside the packed bed.

14 Right-click the Time-Dependent Solver 1 ⚙ node and choose Stop Condition 🛑 .

15 In the Settings window for Stop Condition, locate the Stop Expressions section and click ➕ Add.

16 In the table enter the following settings.
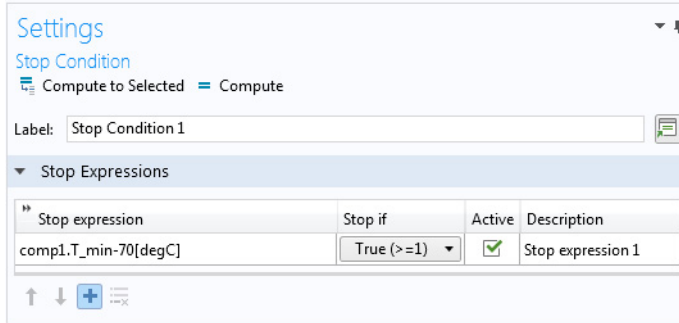
17 In the Home toolbar, click Compute $=$. A warning message appears and states that the stop condition was fulfilled after about 47000 seconds (about 13 hours).

Results

As we removed the default plot generation from the solver settings, follow the steps to generate Figure 3 and Figure 4.

2D PLOT GROUP 1

1 In the Home toolbar, click Add Plot Group ⊞ and choose 2D Plot Group ■ . In the Settings window for 2D Plot Group, type Temperature and velocity field in the Label text field.

**2** Click to expand the Title section, and from the Title type list choose None.

Surface 1

**3** Right-click Temperature and velocity field ■ and choose Surface ■ .

4 In the Settings window for Surface locate the Expression section, and in the Expression text field type T.

5 Locate the Coloring and Style section. From the Color table list choose ThermalLight.

6 Right-click Surface 1 ■ and choose Selection ⚙ . Select Domain 4 only. This plots the temperature field in the container.

Surface 2

7 In the Model Builder window, right-click the Temperature and velocity field ■ node and choose Surface ■ . You can also click Surface ■ in the Temperature and velocity field toolbar.

8 In the Settings window for Surface, locate the Expression section, and in the Expression text field type `ltne1.T`. This plots for the average temperature in the packed bed.

9 Click to expand the Inherit Style section, and from the Plot list choose Surface 1.

Streamline 1

10 Right-click the Temperature and velocity field ■ node and choose Streamline ≋ . You can also click Streamline ≋ in the Temperature and velocity field toolbar.

11 In the Settings window for Streamline, locate the Selection section and select Boundary **7** only. This is the inlet on the upper boundary.

12 Locate the Coloring and Style section. Find the Point style subsection, and from the Type list choose Arrow.

13 Right-click the Streamline 1 ≋ node and choose Color Expression 🎨 .

14 In the Settings window for Color Expression locate the Expression section, and in the Expression text field type `p`.

15 Locate the Coloring and Style section, and from the Color table list choose GrayScale. Select the Reverse color table check box.

1D PLOT GROUP 2

Add a new data set to create Figure 4. Then evaluate and plot the different temperatures over time with the help of table graphs.

Cut Point 2D 1

1 In the Results toolbar, click Cut Point 2D ▣ .

2 In the Settings window for Cut Point 2D, locate the Point Data section. In the r text field type `0`, and in the z text field type `0.05 0.235 0.42`. These points will evaluate temperatures in the packed bed at three different positions along the symmetry axis.

3 In the Results toolbar, click Point Evaluation ⸬ .

4 In the Settings window for Point Evaluation locate the Data section, and from the Dataset list choose Cut Point 2D 1.

5 Locate the Expressions section. In the table enter the following settings:
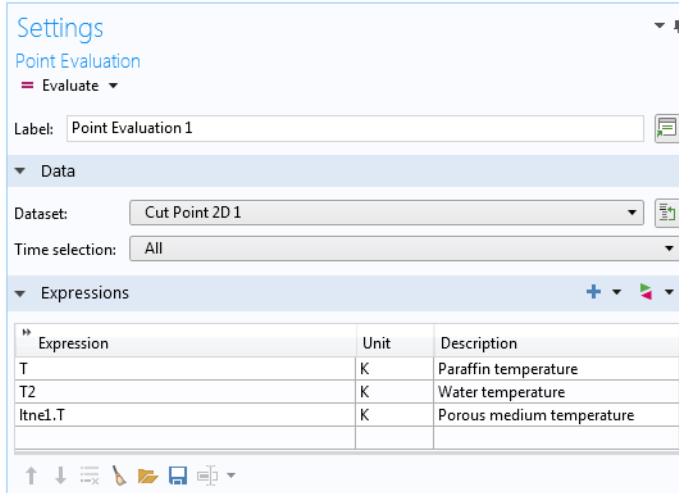
6 Click Evaluate $=$. A table is created in the window toolbar with the data for the temperature variables evaluated at the cut-points for all the time steps.

Table Graph 1

7 Go to the Table window and click Table Graph ⊞ in the window toolbar. This generates a 1D Plot Group 2 node with a Table Graph 1 ⊞ node.

8 In the Settings window for Table Graph locate the Data section, and from the Plot columns list choose Manual.

9 In the Columns list choose
`Paraffin temperature (K), Point: (0, 0.05)`,
`Paraffin temperature (K), Point: (0, 0.235)`, and
`Paraffin temperature (K), Point: (0, 0.42)`.

10 Locate the Coloring and Style section. Find the Line style subsection, and from the Line list choose Dotted.

Table Graph 2

11 Right-click the Table Graph 1 ⊞ node and choose Duplicate 🔁 .

12 In the Settings window for Table Graph locate the Data section, and in the Columns list choose `Water temperature (K), Point: (0, 0.05)`,
`Water temperature (K), Point: (0, 0.235)`, and
`Water temperature (K), Point: (0, 0.42)`.

13 Locate the Coloring and Style section. Find the Line style subsection, and from the Line list choose Dashed. From the Color list choose Cycle (reset).

37

Table Graph 3

14 Right-click Table Graph 2 ⊞ node and choose Duplicate ⤴ .

15 In the Settings window for Table Graph locate the Data section, and in the Columns list choose
```
Porous medium temperature (K), Point: (0, 0.05),
Porous medium temperature (K), Point: (0, 0.235), and
Porous medium temperature (K), Point: (0, 0.42).
```

16 Locate the Coloring and Style section. Find the Line style subsection and from the Line list choose Solid.

1D Plot Group 2

17 In the Model Builder under Results click 1D Plot Group 2 ∿ .

18 In the Settings window for 1D Plot Group, type `Temperature evolution` in the Label text field to rename the plot.

19 Locate the Axis section and select the Manual axis limits check box. In the x minimum text field, type `3.5`, and in the x maximum text field type `9`. This zooms into the time range between 3.5 and 9 hours. In the y minimum text field type `328`, and in the y maximum text field type `344`. to adjust the temperature range.

20 Click Plot ⊡ in the Temperature evolution toolbar to visualize the data.

You clearly see that paraffin and water are not in thermal equilibrium, especially during phase change of paraffin.