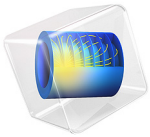


Packed Bed Latent Heat Storage

This model is licensed under the COMSOL Software License Agreement 6.2.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

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, often found in households, 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 as phase change material. Paraffin is a suitable phase change material to include the effect of latent heat, as it is relatively inexpensive, reliable, and nontoxic, and it is commercially available for a wide range of 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 while charging the LHS unit.

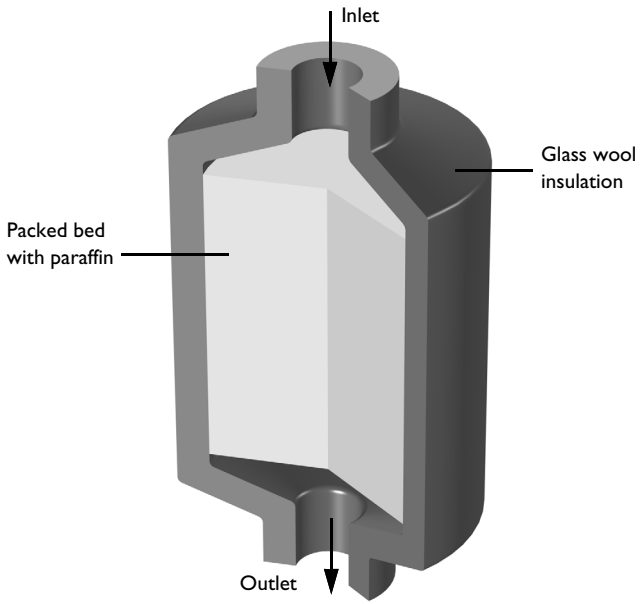


Figure 1: Model setup.

Model Definition

The model geometry is shown in Figure 1. Geometry, material properties, and operating conditions are taken from Ref. 1. The thermo-physical properties of paraffin are listed in Table 1.

TABLE 1: THERMO-PHYSICAL PROPERTIES OF PARAFFIN.

MATERIAL PROPERTY	PARAFFIN, SOLID	PARAFFIN, LIQUID
Melting temperature, T_m (°C)	60	
Latent heat of fusion, L (J/kg)	213	
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

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 $\varepsilon_p = 0.49$. The temperature is initially set to 32°C. Warm water flows through the tank with a flow rate of $V_{in} = 2$ l/min, and during thermal charging it 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

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

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 the flow through the packed bed, which estimates the pressure drop as a function of the velocity field $\mathbf{u}$

$$\nabla p = -\frac{\mu}{\kappa}\mathbf{u} - \frac{1.75(1-\varepsilon_p)}{d_p \varepsilon_p^3}\rho|\mathbf{u}|\mathbf{u}$$

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 porosity. The permeability κ (m^2) of the packed bed is given by

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

The Reynolds number can be estimated as

$$\mathrm{Re} = \frac{d_\mathrm{p} v \rho}{(1-\varepsilon_\mathrm{p})\mu} \tag{2}$$

The maximum velocity in the bed, 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. 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 this example.

The heat transferred from the paraffin-filled capsules to the water is modeled with a heat source

$$Q_\mathrm{f} = \frac{q_\mathrm{sf}}{\varepsilon_\mathrm{p}}(T_s - T_\mathrm{f})$$

here, T_s and T_f are the paraffin and water temperatures, and q_sf (W/(m^3·K)) is the interstitial convective heat transfer coefficient, which for spherical capsules reads

$$q_\mathrm{sf} = \frac{6(1-\varepsilon_\mathrm{p})}{d_\mathrm{p}} h_\mathrm{sf}$$

The interstitial heat transfer coefficient h_sf follows a Nusselt number correlation (see the section *Local Thermal Nonequilibrium* under *Theory for Heat Transfer in Porous Media* in the *Heat Transfer Module User's Guide* for more information). Convection inside the capsules is neglected, thus paraffin is treated as a solid or immobile liquid.

The tank reaches a temperature of 70°C after approximately 11 hours. The resulting velocity and temperature distribution is shown in Figure 2.

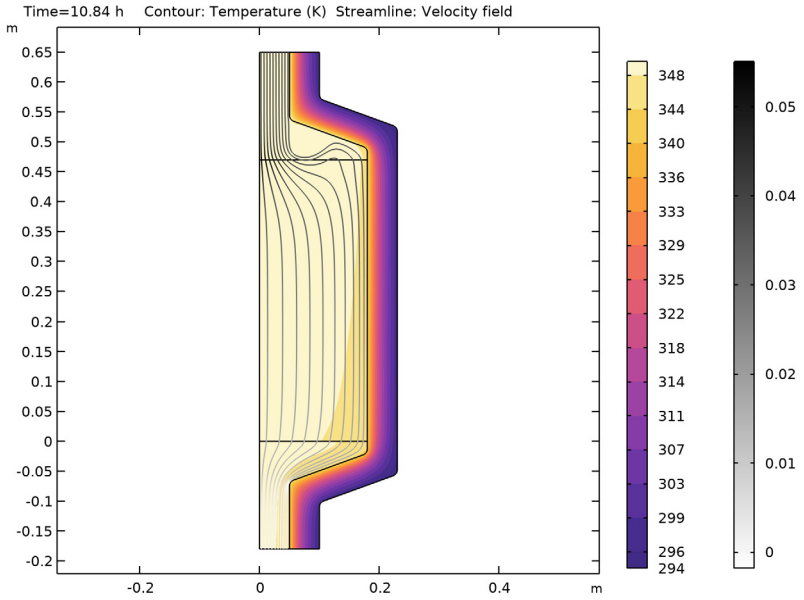


Figure 2: Velocity field (streamlines) with the gray color indicating the pressure and temperature field (color contours) after 13 hours.

Figure 3 shows the evolution of the paraffin temperature, the water temperature, and the weighted average (porous-medium) temperature. During the phase change, the encapsulated paraffin is not in thermal equilibrium with the surrounding water. Measuring the water temperature at the inlet or the outlet does not give accurate information about neither the temperature inside the capsules nor the phase in which the paraffin wax is.

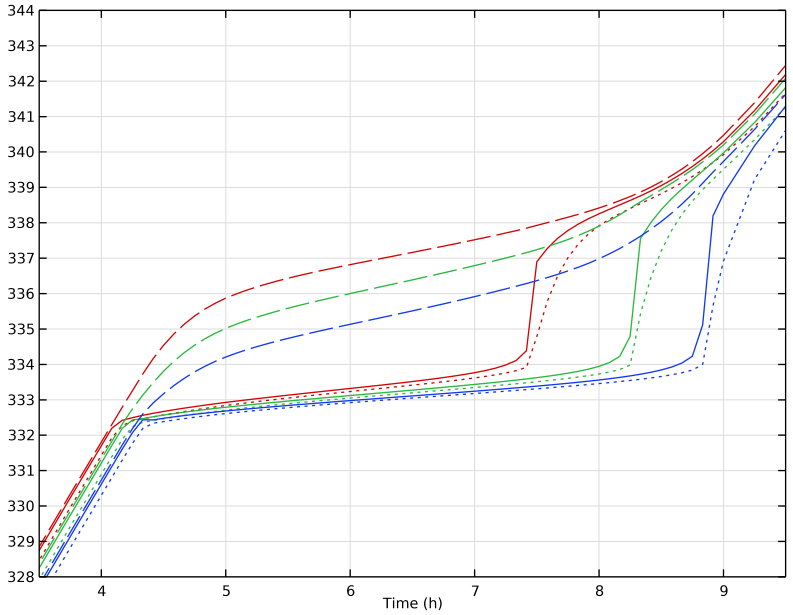


Figure 3: Evolution of water (dashed), paraffin (dotted) and average porous medium temperature (solid) during phase change for top (red), center (green) and low (blue) position.

Figure 4 shows the phase distribution after 7 hours. Near the walls, where the flow velocity is negligible, the phase transition has not yet begun while it is already completed in the center of the tank.

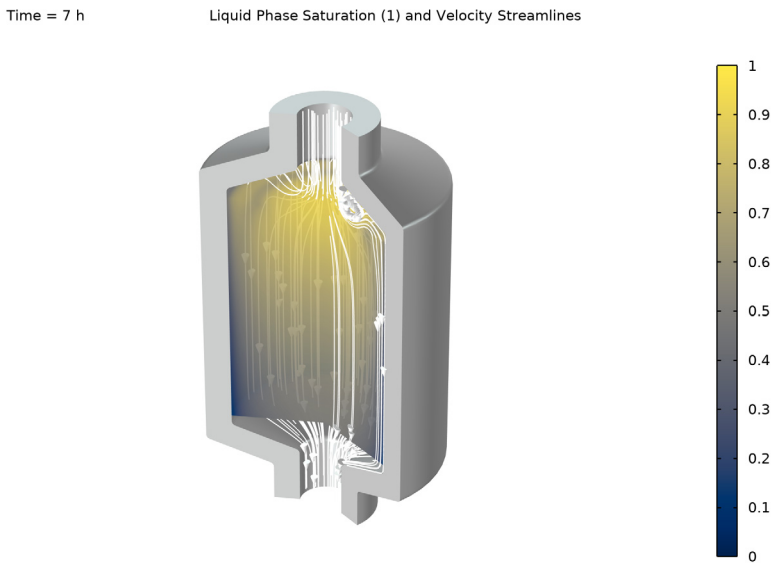


Figure 4: Distribution of solid phase (blue) and liquid phase (yellow) after 7 hours.

The evolution of the paraffin phase distribution is visualized in Figure 5. It starts at about 4 hours when water is heated up to the melting temperature of 60°C. Paraffin 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, which happens after approximately 11 hours.

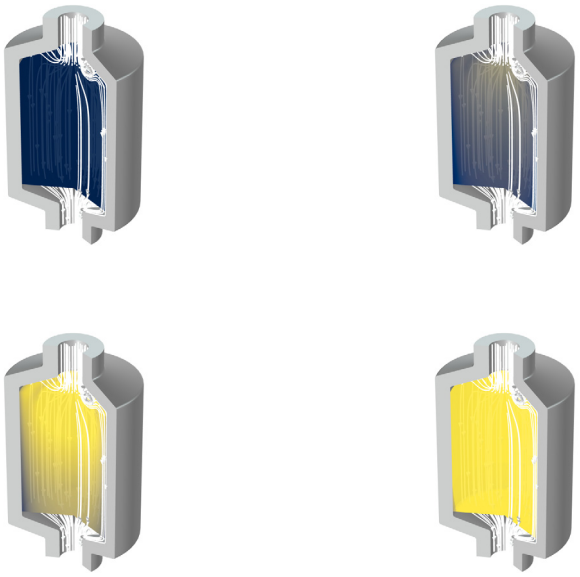


Figure 5: Phase distribution after 4, 6, 8, and 10 hours.

Notes About the COMSOL Implementation

The time it takes to thermally charge the tank is not known a priori. To avoid calculating too many time steps, a stop condition is used which stops the simulation after a temperature of 70°C is reached everywhere inside the tank.

Reference

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

Application Library path: `Porous_Media_Flow_Module/Heat_Transfer/packed_bed_latent_heat_storage`

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1. In the **Model Wizard** window, click **2D Axisymmetric**.
2. In the **Select Physics** tree, select **Fluid Flow>Porous Media and Subsurface Flow> Free and Porous Media Flow, Brinkman (fp)**.
3. Click **Add**.
4. In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Solids and Fluids (ht)**.
5. Click **Add**.
6. Click **Study**.
7. In the **Select Study** tree, select **General Studies>Stationary**.
8. Click **Done**.

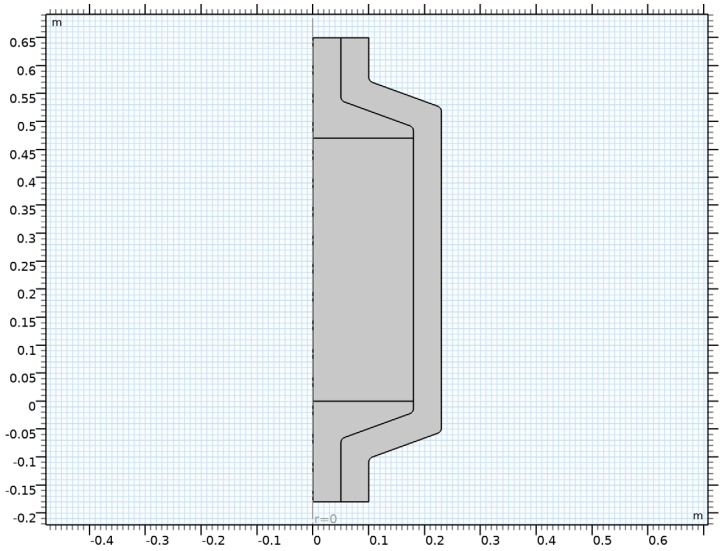
GEOMETRY 1

Import the geometry from a file.

Import 1 (imp1)

1. In the **Home** toolbar, click **Import**.
2. In the **Settings** window for **Import**, locate the **Import** section.
3. Click **Browse**.
4. Browse to the model's Application Libraries folder and double-click the file `packed_bed_latent_heat_storage.mphbin`.
5. Click **Import**.

6 Click **Build All Objects**.



GLOBAL DEFINITIONS

Add parameters that will be used to set up the model.

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
dp	55[mm]	0.055 m	Diameter of encapsulated PCM
por	0.49	0.49	Bed porosity
V_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
rho_av	(861[kg/m^3]+ 778[kg/m^3])/2	819.5 kg/m³	Average density of paraffin

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>Water, liquid**.

4 Click **Add to Component** in the window toolbar.

5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Paraffin, solid

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Paraffin, solid` in the **Label** text field.

Paraffin, liquid

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Paraffin, liquid` in the **Label** text field.

Glass Wool

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Glass Wool` in the **Label** text field.

3 Select Domain 4 only.

Porous Material 1 (pmat1)

1 Right-click **Materials** and choose **More Materials>Porous Material**.

2 Select Domain 2 only.

Continue with setting up the physics interfaces. After that you can fill the required material properties.

FREE AND POROUS MEDIA FLOW, BRINKMAN (FP)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Free and Porous Media Flow, Brinkman (fp)**.

2 Select Domains 1–3 only.

Porous Medium 1

1 In the **Physics** toolbar, click **Domains** and choose **Porous Medium**.

2 Select Domain 2 only.

According to Equation 2, Ergun's equation gives the best description of the flow behavior in the bed.

3 In the **Settings** window for **Porous Medium**, locate the **Porous Medium** section.

4 From the **Flow model** list, choose **Non-Darcian flow**.

Porous Matrix 1

1 In the **Model Builder** window, click **Porous Matrix 1**.

2 In the **Settings** window for **Porous Matrix**, locate the **Matrix Properties** section.

3 From the **Permeability model** list, choose **Ergun**.

4 In the d_p text field, type `dp`.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

Continue with setting up the heat transfer interface.

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>** **Heat Transfer in Solids and Fluids (ht)** click **Fluid 1**.

2 Select Domains 1 and 3 only.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the T text field, type `T0`.

Porous Medium 1

1 In the **Physics** toolbar, click **Domains** and choose **Porous Medium**.

2 Select Domain 2 only.

3 In the **Settings** window for **Porous Medium**, locate the **Porous Medium** section.

4 From the **Porous medium type** list, choose **Local thermal nonequilibrium**.

5 From the **Interstitial convective heat transfer coefficient** list, choose **Spherical pellets**.

6 In the d_{pe} text field, type `dp`.

For a porous medium not in thermal equilibrium the initial temperatures for each component of the porous medium need to be specified.

Initial Values 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>** **Heat Transfer in Solids and Fluids (ht)>Porous Medium 1>Fluid 1** node, then click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the T text field, type `T0`.

Continuity 1

The fluid phase temperature field is considered by default to be continuous with the surrounding domains.

Porous Matrix 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids and Fluids (ht)>Porous Medium 1** click **Porous Matrix 1**.

2 In the **Settings** window for **Porous Matrix**, locate the **Matrix Properties** section.

3 From the **Define** list, choose **Solid phase properties**.

Initial Values 1

1 In the **Model Builder** window, expand the **Porous Matrix 1** node, then click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the T text field, type `T0`.

The solid phase temperature field is considered by default to be insulated from the surrounding domains.

Porous Matrix 1

In the **Model Builder** window, click **Porous Matrix 1**.

Phase Change Material 1

1 In the **Physics** toolbar, click **Attributes** and choose **Phase Change Material**.

In this solid-solid phase change process, the density is assumed to remain constant which is a reasonable simplification. The mean density value of liquid and solid paraffin is used, which is calculated in the parameter list.

2 In the **Settings** window for **Phase Change Material**, locate the **Phase Change** section.

3 In the $T_{1\to 2}$ text field, type `60[degC]`.

4 In the $\Delta T_{1\to 2}$ text field, type `2[K]`.

5 In the $L_{1\to 2}$ text field, type `213[kJ/kg]`.

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

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

MULTIPHYSICS

Nonisothermal Flow 1 (nitf1)

In the **Physics** toolbar, click  **Multiphysics Couplings** and choose **Domain> Nonisothermal Flow**, to couple the **Free and Porous Media Flow** with the **Heat Transfer in Solids and Fluids** interface.

MATERIALS

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

Paraffin, solid (mat2)

1. In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Paraffin, solid (mat2)**.
2. In the **Settings** window for **Material**, locate the **Material Contents** section.
3. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	0.4	W/(m·K)	Basic
Heat capacity at constant pressure	Cp	1850	J/(kg·K)	Basic

Paraffin, liquid (mat3)

1. In the **Model Builder** window, click **Paraffin, liquid (mat3)**.
2. In the **Settings** window for **Material**, locate the **Material Contents** section.
3. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	0.15	W/(m·K)	Basic
Heat capacity at constant pressure	Cp	2384	J/(kg·K)	Basic

Glass Wool (mat4)

1. In the **Model Builder** window, click **Glass Wool (mat4)**.
2. In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	0.025	W/(m·K)	Basic
Density	rho	850	kg/m³	Basic
Heat capacity at constant pressure	Cp	1250	J/(kg·K)	Basic

Porous Material 1 (pmat1)

1 In the **Model Builder** window, click **Porous Material 1 (pmat1)**.

2 In the **Settings** window for **Porous Material**, locate the **Phase-Specific Properties** section.

3 Click **Add Required Phase Nodes**.

Fluid 1 (pmat1.fluid1)

1 In the **Model Builder** window, click **Fluid 1 (pmat1.fluid1)**.

2 In the **Settings** window for **Fluid**, locate the **Fluid Properties** section.

3 From the **Material** list, choose **Water, liquid (mat1)**.

Solid 1 (pmat1.solid1)

1 In the **Model Builder** window, click **Solid 1 (pmat1.solid1)**.

2 In the **Settings** window for **Solid**, locate the **Solid Properties** section.

3 In the θ_s text field, type `1-por`.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	rho_av	kg/m³	Basic
Porosity	epsilon	0.49	1	Porous model

FREE AND POROUS MEDIA FLOW, BRINKMAN (FP)

Continue by applying the boundary conditions.

In the **Model Builder** window, under **Component 1 (comp1)** click **Free and Porous Media Flow, Brinkman (fp)**.

Inlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

4 From the list, choose **Fully developed flow**.

5 Locate the **Fully Developed Flow** section. Click the **Flow rate** button.

6 In the V_0 text field, type `V_in`.

7 Locate the **Boundary Selection** section. Click **Create Selection**.

8 In the **Create Selection** dialog box, type `Inlet` in the **Selection name** text field.

9 Click **OK**. This creates a selection for the inlet boundary and will be used again during the model setup.

Outlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Outlet**, locate the **Boundary Selection** section.

4 Click **Create Selection**.

5 In the **Create Selection** dialog box, type `Outlet` in the **Selection name** text field.

6 Click **OK**. This creates a selection for the outlet boundary and will be used again during the model setup.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids and Fluids (ht)**.

Inflow 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inflow**.

2 In the **Settings** window for **Inflow**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Inlet**.

4 Locate the **Upstream Properties** section. In the T_{ustr} text field, type `T_in`.

The water temperature increases over time during the charging process. While water is pumped through a closed loop, it is heated by a solar system. Therefore you later define the variable `T_in` as a function of the outlet temperature and the solar heating power using Equation 1.

Outflow 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outflow**.

2 In the **Settings** window for **Outflow**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Outlet**.

DEFINITIONS (COMPI)

The tank is cooled by the surroundings. Create a selection for the outer boundary to apply the heat flux condition.

Heat Flux Boundary

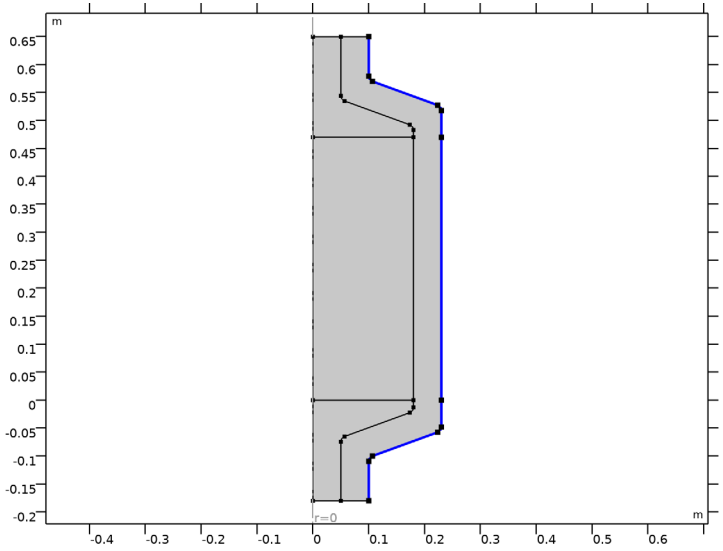
1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Heat Flux Boundary` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select the **Group by continuous tangent** check box.

Select one of the outer boundaries.



HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

Heat Flux 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 In the **Settings** window for **Heat Flux**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Heat Flux Boundary**.

4 Locate the **Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.

5 In the h text field, type `5`.

This value is a good approximation to account for cooling by ambient air.

DEFINITIONS (COMP1)

Now, define the variable `T_in`.

Average 1 (aveop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Outlet**.

This operator is used to compute the average outlet temperature.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`deltaT`	`Qu/V_in/aveop1(ht.Cp)/aveop1(ht.rho)`	K	`Temperature increase`
`T_in`	`aveop1(T)+deltaT`	K	`Inlet temperature`

The expressions `ht.Cp` and `ht.rho` refer to the heat capacity and density of water as defined by the **Heat Transfer in Fluids** interface.

The default physics-controlled mesh automatically takes into account that there are steep gradients for the velocity close to the walls. Use a fine mesh size to also resolve the thermal effects well.

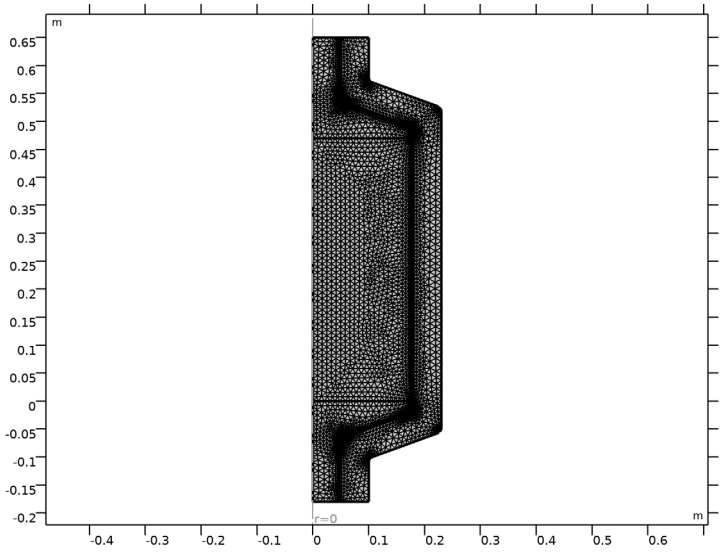
MESH 1

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.

3 From the **Element size** list, choose **Fine**.

4 Click **Build All**.



STUDY 1

Step 1: Stationary

Since the flow field can be assumed to be independent of time, it is calculated in a first stationary step and then used as input for the heat transport in the subsequent time-dependent step.

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Heat Transfer in Solids and Fluids (ht)**.

4 In the table, clear the **Solve for** check box for **Nonisothermal Flow 1 (nitf1)**.

Step 2: Time Dependent

1 In the **Study** toolbar, click **Study Steps** and choose **Time Dependent> Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Free and Porous Media Flow, Brinkman (fp)**.

4 Locate the **Study Settings** section. From the **Time unit** list, choose **h**.

5 In the **Output times** text field, type `range(0,0.25,3.75) range(4,5[min],9) range(9.25,0.25,24)`.

The time stepping is chosen such that the phase change is resolved properly.

6 From the **Tolerance** list, choose **User controlled**.

7 In the **Relative tolerance** text field, type `1e-4`.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Steps taken by solver** list, choose **Strict**.

This forces the solver to use at least the time steps specified above.

DEFINITIONS (COMP1)

Use a stop condition for the time-dependent solver to force the charging process to stop when the minimum temperature in the tank reaches 70°C. This requires another coupling operator for the minimum temperature.

Minimum 1 (minop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Minimum**.

2 Select Domain 2 only.

Variables 1

1 In the **Model Builder** window, click **Variables 1**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`T_min`	`minop1(ht.porous.pm.T)`	K	`Minimum temperature`

STUDY 1

Solution 1 (sol1)

1 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** right-click **Time-Dependent Solver 1** and choose **Stop Condition**.

2 In the **Settings** window for **Stop Condition**, locate the **Stop Expressions** section.

3 Click + **Add**.

4 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
`comp1.T_min > 70[degC]`	True (>=1)	√	`Stop expression 1`

5 Locate the **Output at Stop** section. From the **Add solution** list, choose **Step after stop**.

6 In the **Home** toolbar, click = **Compute**.

The solver automatically stops when the stop condition is fulfilled. A warning message appears and states that the stop condition is fulfilled after about 39,000 s (about 10.8 hours).

RESULTS

Automatically 5 default plots are created. A velocity, pressure and temperature plot in 2D, a velocity plot in 3D on the revolved geometry, and a 2D plot of temperature and velocity field. To create a plot matching Figure 2 use one of the predefined plots.

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Windows** and choose **Add Predefined Plot**.

2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Study 1/Solution 1 (sol1)>Heat Transfer in Solids and Fluids> Isothermal Contours (ht)**.

4 Click **Add Plot** in the window toolbar.

RESULTS

Isothermal Contours & Velocity Streamlines

In the **Settings** window for **2D Plot Group**, type `Isothermal Contours & Velocity Streamlines` in the **Label** text field.

Contour 1

1 In the **Model Builder** window, expand the **Isothermal Contours & Velocity Streamlines** node, then click **Contour 1**.

2 In the **Settings** window for **Contour**, locate the **Coloring and Style** section.

3 From the **Contour type** list, choose **Filled**.

4 Locate the **Levels** section. In the **Total levels** text field, type 15.

Isothermal Contours & Velocity Streamlines

In the **Isothermal Contours & Velocity Streamlines** toolbar, click **Streamline**.

Streamline 1

1. In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.
2. In the **Number** text field, type `10`.
3. Locate the **Selection** section. From the **Selection** list, choose **Inlet**.

Color Expression 1

1. In the **Isothermal Contours & Velocity Streamlines** toolbar, click **Color Expression**.
2. In the **Settings** window for **Color Expression**, locate the **Expression** section.
3. In the **Expression** text field, type `p`.
4. Locate the **Coloring and Style** section. Click **Change Color Table**.
5. In the **Color Table** dialog box, select **Linear>GrayScale** in the tree.
6. Click **OK**.
7. In the **Settings** window for **Color Expression**, locate the **Coloring and Style** section.
8. From the **Color table transformation** list, choose **Reverse**.

Cut Point 2D 1

To create Figure 3, begin by creating a new dataset, then use a **Point Evaluation** node to evaluate the different temperatures before plotting them.

1. In the **Results** toolbar, click **Cut Point 2D**.
2. In the **Settings** window for **Cut Point 2D**, locate the **Point Data** section.
3. In the **r** text field, type `0`.
4. In the **z** text field, type `0.05 0.47/2 0.42`.

Point Evaluation 1

1. In the **Results** toolbar, click **Point Evaluation**.
2. In the **Settings** window for **Point Evaluation**, locate the **Data** section.
3. From the **Dataset** list, choose **Cut Point 2D 1**.
4. Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`ht.porous.pm.T`	`K`	`Paraffin temperature`
`ht.porous.fluid.T`	`K`	`Water temperature`
`T`	`K`	`Porous Medium temperature`

5 Click = **Evaluate**.

TABLE 1

1 Go to the **Table 1** window.

2 Click **Table Graph** in the window toolbar.

RESULTS

Table Graph 1

1 In the **Model Builder** window, under **Results>1D Plot Group 7** click **Table Graph 1**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Plot columns** list, choose **Manual**.

4 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)**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

6 Right-click **Results>1D Plot Group 7>Table Graph 1** and choose **Duplicate**.

Table Graph 2

1 In the **Model Builder** window, click **Table Graph 2**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 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)**.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

5 From the **Color** list, choose **Cycle (reset)**.

6 Right-click **Table Graph 2** and choose **Duplicate**.

7 In the **Model Builder** window, click **Table Graph 2**.

8 Locate the **Data** section. 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)**.

9 Locate the **Coloring and Style** section. From the **Line** list, choose **Solid**.

Temperature Evolution

1 In the **Model Builder** window, under **Results** click **1D Plot Group 7**.

2 In the **Settings** window for **1D Plot Group**, type `Temperature Evolution` in the **Label** text field.

3 Locate the **Axis** section. Select the **Manual axis limits** check box.

4 In the **x minimum** text field, type `3.5`.

5 In the **x maximum** text field, type `9.5`.

6 In the **y minimum** text field, type `328`.

7 In the **y maximum** text field, type `344`.

8 In the **Temperature Evolution** toolbar, click **Plot**.

Compare with Figure 3. You can clearly see that paraffin and water are not in thermal equilibrium, especially during phase change of paraffin.

Now follow the steps below to visualize the phase distribution and velocity field in a 3D plot as in Figure 4 and Figure 5.

3D Plot Group 8

In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

Volume 1

1 Right-click **3D Plot Group 8** and choose **Volume**.

2 In the **Settings** window for **Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Heat Transfer in Solids and Fluids>Phase change>ht.theta2 - Phase indicator, phase arg.n - 1**.

3 Locate the **Coloring and Style** section. Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Linear>Cividis** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Volume**, click to expand the **Range** section.

7 Select the **Manual color range** check box.

8 In the **Maximum** text field, type `1`.

Transparency 1

1 Right-click **Volume 1** and choose **Transparency**.

2 In the **Settings** window for **Transparency**, locate the **Transparency** section.

3 In the **Transparency** text field, type `0.1`.

Surface 1

1 In the **Model Builder** window, right-click **3D Plot Group 8** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Gray**.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

2 Select Domain 4 only.

Material Appearance 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Material Appearance**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Aluminum**.

3D Plot Group 8

In the **3D Plot Group 8** toolbar, click **Streamline**.

Streamline 1

1 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

2 From the **Positioning** list, choose **Uniform density**.

3 In the **Separating distance** text field, type `0.06`.

4 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

5 Select the **Number of arrows** check box. In the associated text field, type `120`.

6 From the **Color** list, choose **White**.

Liquid Phase

1 In the **Model Builder** window, under **Results** click **3D Plot Group 8**.

2 In the **Settings** window for **3D Plot Group**, type `Liquid Phase` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Liquid Phase Saturation (1) and Velocity Streamlines`.

5 In the **Parameter indicator** text field, type `Time = eval(t,h) h`.

6 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

7 Locate the **Data** section. From the **Time (h)** list, choose **7**.

8 In the **Liquid Phase** toolbar, click **Plot**.

9 Click the **Show Grid** button in the **Graphics** toolbar.

10 Click the **Show Axis Orientation** button in the **Graphics** toolbar.

11 Click the **Zoom Extents** button in the **Graphics** toolbar.

To reproduce the sequence shown in Figure 5, just select, in turn, from the **Time (h)** list the values **4**, **6**, **8**, and **10** and plot.