Model created in COMSOL Multiphysics 6.4

Phase Change in a Semi-Infinite Soil Column

Introduction

The freezing of subsurface water has a high impact on groundwater flow and subsurface heat transfer. This example models the freezing of a soil column over time. This is a benchmark model with an existing analytical solution, derived by Lunardini in 1985.

Lunardini developed an exact analytical solution for the propagation of subfreezing temperatures in a semi-infinite, initially unfrozen porous medium with time. He therefore divided the porous medium in three zones: A totally frozen zone (for temperatures $T < T_{\mathrm{m}}$), a so-called mushy or partially frozen zone ($T_{\mathrm{m}} < T < T_{\mathrm{f}}$), and a totally unfrozen or liquid water zone ($T > T_{\mathrm{f}}$).

This example uses the **Phase Change Material** subfeature from the **Heat Transfer in Porous Media** interface.

This example demonstrates how to model a phase change between water and ice using a user-defined phase transition function. The solution should be equal to that of Lunardini.

Model Definition

In this example, the soil column is approximated by a line interval of 10 m length. The initial temperature is 4°C and the temperature at one end is set to –6°C while the other end is thermally isolated.

The following equation is solved:

$$(\rho C_{\mathrm{p}})_{\mathrm{eff}}\frac{\partial T}{\partial t} + \nabla\cdot(-k_{\mathrm{eff}}\nabla T) = Q \tag{1}$$

Here, T is the temperature (K) and Q is a heat source (W/m^3). The effective values are defined as

$$(\rho C)_{\mathrm{eff}} = \varepsilon_{\mathrm{p}}\rho_{\mathrm{f}}C_{\mathrm{p,f}} + \theta_{\mathrm{s}}\rho_{\mathrm{s}}C_{\mathrm{p,s}} + \theta_{\mathrm{imf}}\rho_{\mathrm{imf}}C_{\mathrm{p,\,imf}} \tag{2}$$

$$k_{\mathrm{eff}} = \varepsilon_{\mathrm{p}}k_{\mathrm{f}} + \theta_{\mathrm{s}}k_{\mathrm{s}} + \theta_{\mathrm{imf}}k_{\mathrm{imf}} + k_{\mathrm{disp}} \tag{3}$$

where ρ (kg/m^3) is the density (of the fluid, solid, and immobile fluid), C_{p} (J/(kg·K)) the heat capacity at constant pressure, and k the thermal conductivity (W/(m·K)).

Results and Discussion

Figure 1 shows the temperature profile after 24, 48, and 72 h and compares the computed results (solid line) with the analytical solution (Ref. 1).

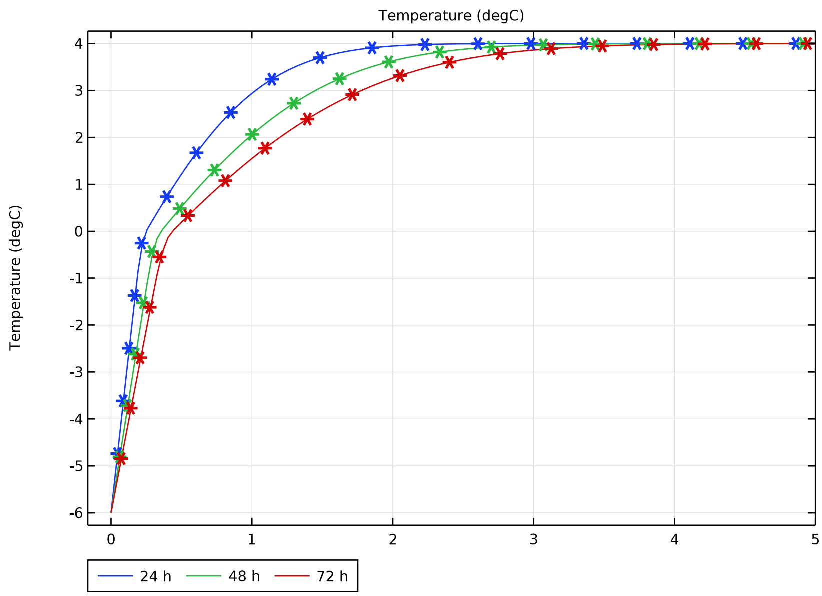

Figure 1: Computed (solid line) compared to analytical solution (asterisks) after 24, 48, and 72 h.

The results match very well.

Reference

1. N. Tubini, S. Gruber, and R. Rigon, "A method for solving heat transfer with phase change in ice or soil that allows for large time steps while guaranteeing energy conservation," *The Cryosphere,* vol. 15, pp. 2541–2568, 2021.

Application Library path: `Porous_Media_Flow_Module/Verification_Examples/phase_change_lunardini`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **1D**.

2 In the **Select Physics** tree, select **Heat Transfer** > **Porous Media** > **Heat Transfer in Porous Media (ht)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Time Dependent**.

6 Click **Done**.

GEOMETRY 1

Interval 1 (i1)

The model domain is approximated by a 1D line segment of 10 m length.

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Geometry 1** and choose **Interval**.

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

3 In the table, enter the following settings:

Coordinates (m)
0
10

4 Click **Build All Objects**.

GLOBAL DEFINITIONS

Now, enter the parameters used in the model. You can import them from an external file.

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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `phase_change_lunardini_parameters.txt`.

HEAT TRANSFER IN POROUS MEDIA (HT)

Follow the steps below to set up the physics.

Fluid 1

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

Phase Change Material 1

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

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

3 From the **Phase transition function** list, choose **User defined**. In the $L_{1 \to 2}$ text field, type `L`.

4 In the $\alpha_{1 \to 2}$ text field, type `f_phtr(T)`, which is yet to be defined.

5 Locate the **Phase 1** section. From the k_1 list, choose **User defined**. In the associated text field, type `k_ice`.

6 From the ρ_1 list, choose **User defined**. In the associated text field, type `rho_ice`.

7 From the $C_{p,1}$ list, choose **User defined**. In the associated text field, type `Cv/rho_ice`.

8 Locate the **Phase 2** section. From the k_2 list, choose **User defined**. In the associated text field, type `k_water`.

9 From the ρ_2 list, choose **User defined**. In the associated text field, type `rho_water`.

10 From the $C_{p,2}$ list, choose **User defined**. In the associated text field, type `Cv/rho_water`.

DEFINITIONS

Now, define the phase transition function as follows.

Interpolation 1 (int1)

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

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type `f_phtr`.

4 In the table, enter the following settings:

t	f(t)
-4	Sw_res
-1	Sw_res

t	f(t)
0	1
6	1

5 Locate the **Units** section. In the **Function** table, enter the following settings:

Function	Unit
f_phtr	1

6 In the **Argument** table, enter the following settings:

Argument	Unit
t	degC

7 Click **Plot**.

HEAT TRANSFER IN POROUS MEDIA (HT)

Add the soil properties next.

Porous Matrix 1

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

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

3 From the ε_p list, choose **User defined**. In the associated text field, type `por`.

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

5 Locate the **Heat Conduction, Porous Matrix** section. From the k_s list, choose **User defined**. In the associated text field, type `k_solid`.

6 Locate the **Thermodynamics, Porous Matrix** section. From the ρ_s list, choose **User defined**. In the associated text field, type `rho_solid`.

7 From the $C_{p,s}$ list, choose **User defined**. In the associated text field, type `Cv/rho_solid`.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Heat Transfer in Porous Media (ht)** click **Initial Values 1**.

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

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

Temperature 1

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

2 Select Boundary 1 only.

3 In the **Settings** window for **Temperature**, locate the **Temperature** section.

4 In the T_0 text field, type `T_in`.

MESH 1

As the model is cooled from one end of the domain, the mesh is created to resolve the area with the highest temperature gradient best.

Edge 1

In the **Mesh** toolbar, click **Edge**.

Distribution 1

1 Right-click **Edge 1** and choose **Distribution**.

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

3 From the **Distribution type** list, choose **Predefined**.

4 In the **Number of elements** text field, type `100`.

5 In the **Element ratio** text field, type `10`.

Edge 1

In the **Model Builder** window, right-click **Edge 1** and choose **Build All**.

STUDY 1

Now set up the study with output times after 1, 2, and 3 days.

Step 1: Time Dependent

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

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 From the **Time unit** list, choose **h**.

4 In the **Output times** text field, type `0 24 48 72`.

The time step has to be small enough to catch the temperature decrease and the phase change correctly. Therefore, restrict the maximum time step to 2 minutes.

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 **Maximum step constraint** list, choose **Constant**.

5 In the **Maximum step** text field, type `2[min]`.

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

RESULTS

Temperature (ht)

Per default, the temperature is plotted. With the next steps you can change the temperature unit to degC and add a legend to the plot.

1 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

2 From the **Time selection** list, choose **From list**.

3 In the **Times (h)** list, choose **24**, **48**, and **72**.

4 Locate the **Legend** section. From the **Layout** list, choose **Outside graph axis area**.

5 From the **Position** list, choose **Bottom**.

Line Graph 1

1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Unit** field, type `degC`.

4 Click to expand the **Legends** section. Select the **Show legends** checkbox.

Table 1: Analytical Solution

The analytical solution provided by Ref. 1 is available in a text file. Load it into the model and plot it to compare.

1 In the **Results** toolbar, click **Table**.

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

3 Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `phase_change_lunardini_analytical_solution.txt`.

5 In the **Label** text field, type `Table 1: Analytical Solution`.

Table Graph 1

1 In the **Model Builder** window, right-click **Temperature (ht)** and choose **Table Graph**.

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

3 Find the **Line style** subsection. From the **Line** list, choose **None**.

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

5 Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.

6 From the **Positioning** list, choose **Interpolated**.

7 In the **Number** text field, type 25.

Temperature (ht)

1 In the **Model Builder** window, click **Temperature (ht)**.

2 In the **Settings** window for **1D Plot Group**, locate the **Axis** section.

3 Select the **Manual axis limits** checkbox.

4 In the **x maximum** text field, type 5.

5 In the **Temperature (ht)** toolbar, click **Plot**. Compare with Figure 1