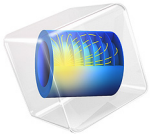


Frozen Inclusion

Introduction

This benchmark example simulates phase change in porous media. The model studies the melting process of an ice inclusion within a porous medium, and it demonstrates how to couple Darcy's Law and the Heat Transfer in Porous Media interfaces including phase change. It is one of the test cases from the INTERFROST project — a comparison project for coupled thermo-hydraulic systems with respect to climate change in permafrost regions (see Ref. 2).

Model Definition

The model setup is shown in Figure 1. A flow channel 3 meters in length and 1 meter in width contains a squared ice inclusion of 33.3 cm in length, centered at the coordinates x = 1 m and y = 0.5 m.

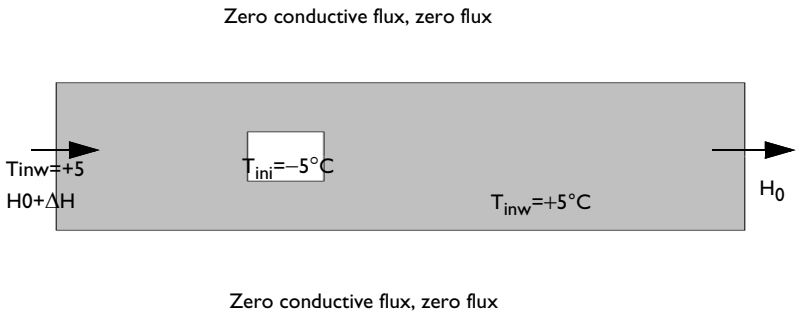


Figure 1: Model geometry, initial temperature distribution, and boundary conditions.

The initial temperature of the ice inclusion is T_{ini} = −5°C. The temperature in the porous channel is T_{inw} = +5°C. The fluid flow is driven by a hydraulic head gradient of ΔH = 3% over the length of the channel.

The initial pressure and velocity field are computed from a steady state flow simulation for a uniformly distributed temperature and permeability (as if the ice inclusion was not present). Considering the symmetry of the benchmark test model, only the lower half of the domain is modeled.

The following simplifications have been made in the INTERFROST benchmark and therefore are also assumed in this example: No thermal dispersion is included in the heat

transfer equation, and water density and dynamic viscosity are considered constant with respect to the temperature (see Table 1).

MODEL EQUATIONS

The Porous Medium feature in Darcy's Law interface includes an option to define a user-defined storage S_p (SI unit: 1/Pa) which in this case is defined as

$$S_p = S_w \beta \varepsilon_p \tag{1}$$

where S_w is the water saturation, ε_p is the porosity, and β the effective compressibility; which is a combined value of water, ice, and solid matrix compressibility. For simplicity, gravity is neglected. The variable Q_m is a mass source which represents the additional liquid water due to the melting of the ice inclusion:

$$Q_m = S_w \varepsilon_p (\rho_i - \rho_w) \frac{\partial S_w}{\partial t}. \tag{2}$$

here ρ_i and ρ_w are the ice and liquid water densities. The liquid water saturation S_w depends on the phase change as:

$$S_w = S_{w,\,res} + (1 - S_{w,\,res}) f_{phtr} \tag{3}$$

Here, $S_{w,\,res}$ it the residual liquid water saturation and f_{phtr} is the phase transition function which is defined as follows:

$$f_{phtr} = \begin{cases} e^{-\left(\frac{(T - 273.15)}{W}\right)^2} & (T < 273.15) \\ 1 & (T \geq 273.15) \end{cases} \tag{4}$$

In the INTERFROST benchmark it is assumed that the mushy ice zone extends from 0°C to –1°C. Therefore, W, which corresponds to the slope of the phase transition function, is set to 0.5 K.

The heat transfer in the porous medium is described by the following equation:

$$(\rho C)_{eq} \frac{\partial T}{\partial t} + \rho C_w \mathbf{u} \cdot \nabla T + \nabla \cdot (-k_{eq} \nabla T) = Q \tag{5}$$

here $(\rho C)_{eq}$ is the equivalent value of density ρ (kg/m^3) and heat capacity C at constant pressure (J/kg·K), k_{eq} is the effective thermal conductivity (W/m·K), T is the temperature (K), and Q is a heat source (W/m^3). The variable C_w is the effective fluid's heat capacity at constant pressure.

MODEL DATA

The benchmark example describes the ice-to-water phase change of an ice inclusion within a porous channel. The values for thermal properties of water, ice, and solid matrix are presented in Table 1.

TABLE 1: MATERIAL PROPERTIES OF ICE, WATER, AND SOLID MATRIX.

MATERIAL PROPERTY	ICE	WATER	SOLID MATRIX
Density	920 kg/m^3	1000 kg/m^3	2650 kg/m^3
Heat capacity at constant pressure	2060 J/(kg·K)	4182 J(kg·K)	835 J(kg·K)
Thermal conductivity	2.14 W/(m·K)	0.6 W/(m·K)	9 W/(m·K)

A latent heat of melting, L= 334 kJ/kg is used. Additional physical parameters used in the example are summarized in Table 2.

TABLE 2: ADDITIONAL PHYSICAL PARAMETERS.

Parameter	Description	Value
μ_w	Dynamic viscosity of water	$1.793 \cdot 10^{-3}$ Pa·s
ε	Porosity	0.37
β	Effective compressibility	10^{-8} Pa^{-1}
S_{wres}	Residual water saturation	0.05
Ω	Impedance factor	50
k_{int}	Intrinsic permeability	$1.3 \cdot 10^{-10}$ m^2

Results and Discussion

The simulation is run for 56 hours which is the time until the frozen inclusion is completely melted and the cooler temperatures have been convected out of the channel. However, it is also interesting to display the effect of the molten inclusion on the pressure and temperature distribution; Figure 2 shows the pressure distribution after 9 hours of

simulated time when there is still some ice present. As gravity has been neglected, only the pressure effects of the ice inclusion are shown.]

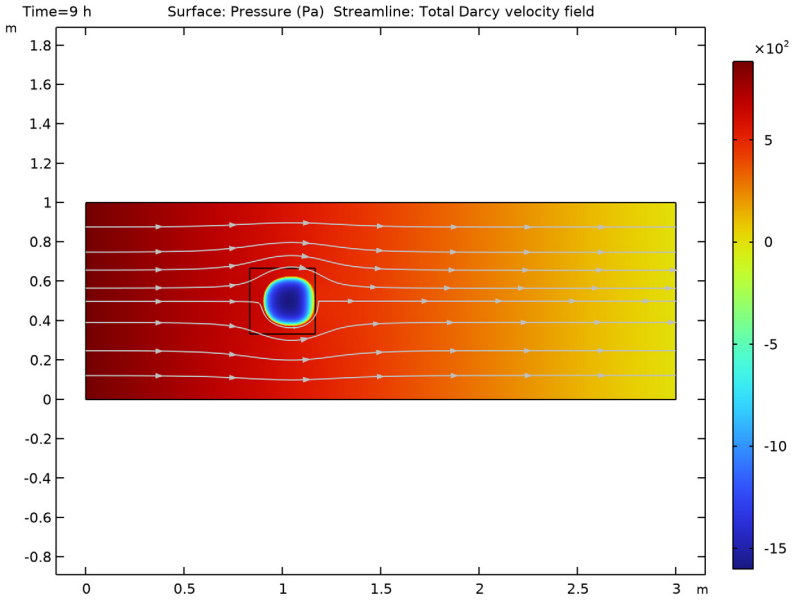


Figure 2: Pressure distribution after 9 hours when there is still some ice present in the model domain.

The temperature field is shown at the same time (after 9 hours) in Figure 3 and the liquid water saturation in Figure 4. Both figures indicate the ice inclusion and its influence on the temperature field. The low temperatures are convected in the downstream part of the domain.

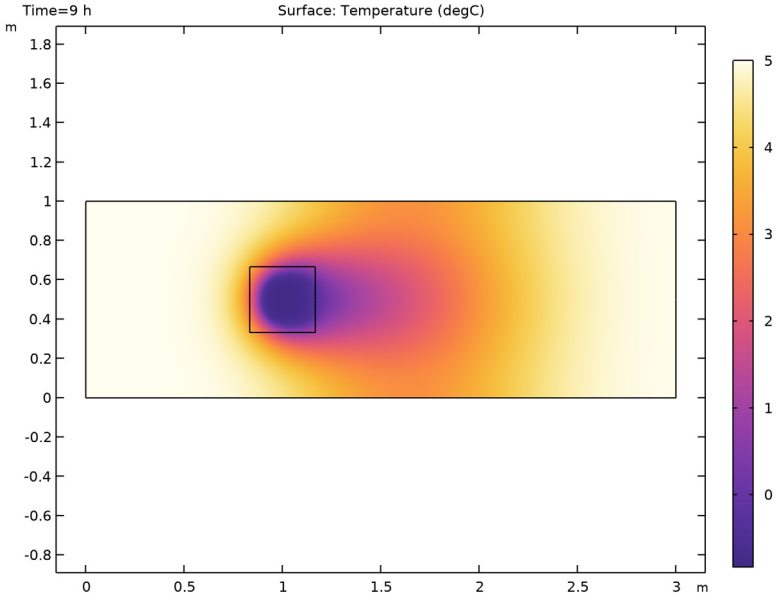


Figure 3: Surface plot showing the temperature distribution after 9 hours.

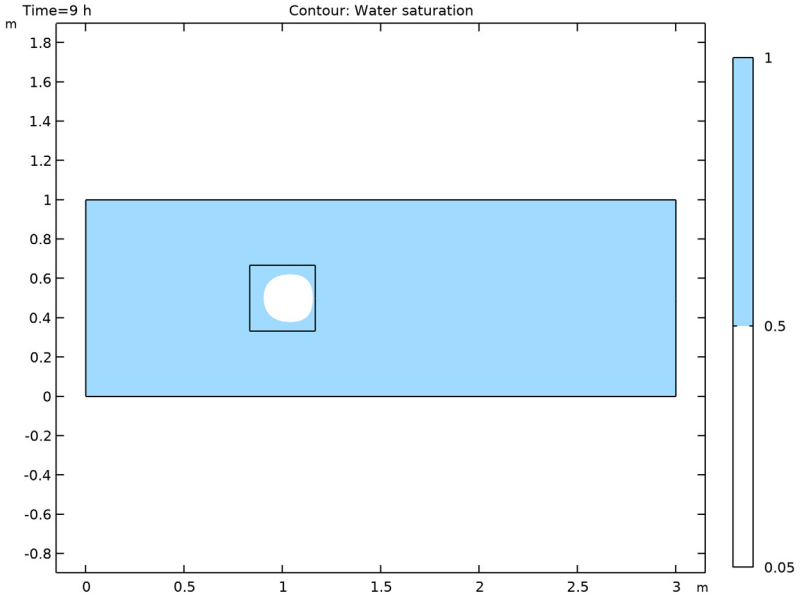


Figure 4: Surface plot showing the liquid water saturation after 9 hours. Effectively the areas with liquid water appear in blue whereas the areas of ice appear in white.

The plots below have been used as performance measures in the INTERFROST benchmark, and therefore are also included here:

The evolution of the minimum of the simulated temperature field as a function of time is shown in Figure 5.

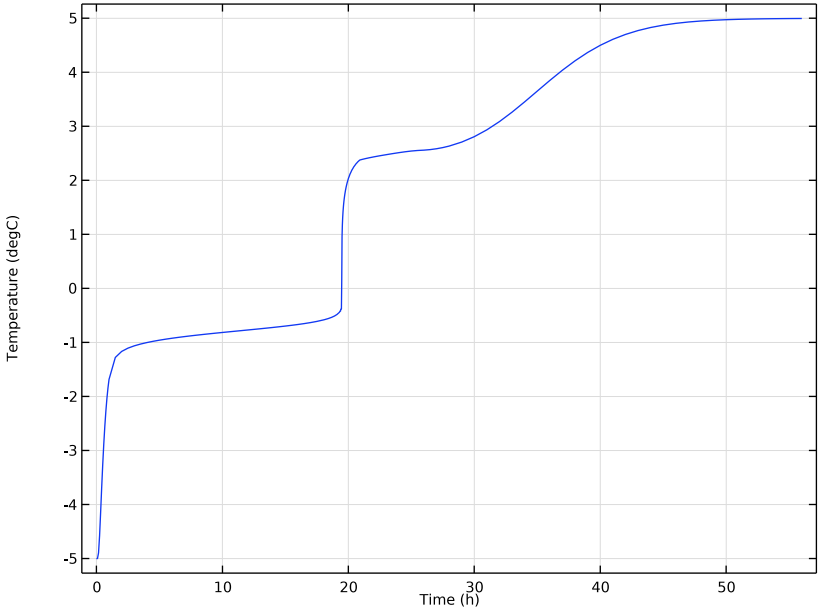


Figure 5: The minimum temperature in the model domain as a function of time.

Figure 6 shows the time evolution of the total heat flux exiting the system at the downstream boundary.

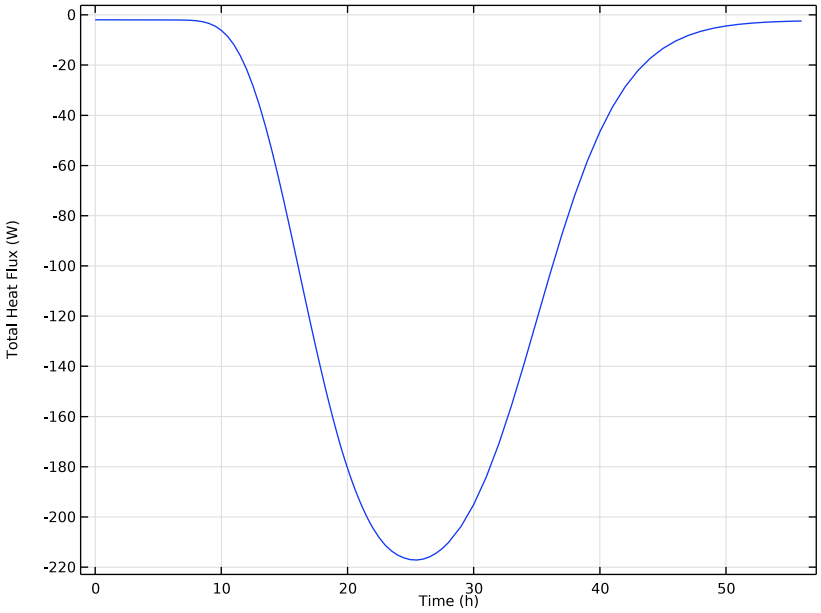


Figure 6: Total heat flux exiting the system via the outflow boundary as a function of time.

Figure 7 displays the evolution of the total liquid water volume in the domain as a function of time.

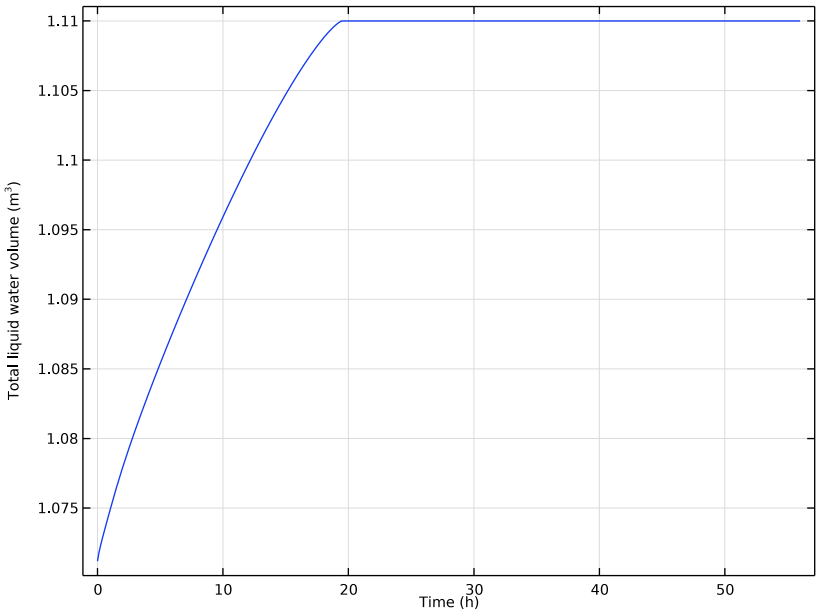


Figure 7: Total liquid water volume in the domain as a function of time.

Notes About the COMSOL Implementation

A residual liquid phase is present in soils even for temperatures significantly below 0°C. This residual water is not subject to phase change or convection. The **Porous Material** definition as well as the **Porous Medium** node within the **Heat Transfer in Porous Media** interface allow to define such **Immobile Fluids**. According to Ref. 1, the residual saturation $S_{w,\,res}$ is set to 0.05.

References

1. C. Grenier and others, "Groundwater flow and heat transport for systems undergoing freeze-thaw: Intercomparison of numerical simulators for 2D test cases," *Advances in Water Resources*, vol. 114, pp. 196–218, 2018.

2. https://wiki.lsce.ipsl.fr/interfrost/doku.php?id=test_cases:five

Application Library path: `Porous_Media_Flow_Module/Heat_Transfer/frozen_inclusion`

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 **2D**.
2. In the **Select Physics** tree, select **Fluid Flow>Porous Media and Subsurface Flow> Darcy's Law (dl)**.
3. Right-click and choose **Add Physics**.
4. In the **Select Physics** tree, select **Heat Transfer>Porous Media> Heat Transfer in Porous Media (ht)**.
5. Right-click and choose **Add Physics**.
6. Click **Study**.
7. In the **Select Study** tree, select **General Studies>Stationary**.
8. Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

   The parameters defining the material properties of the solid matrix, water, and ice have to be declared. They are available in a file and you can import them as follows:
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 `frozen_inclusion_parameters.txt`.

GEOMETRY 1

Now, the model domain is drawn as two rectangles, one representing the porous domain, the other the frozen inclusion within the porous domain. Create the geometry as follows and note that due to symmetry only half of the model domain is drawn.

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `3`.

4 In the **Height** text field, type `0.5`.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.333`.

4 In the **Height** text field, type `0.333/2`.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **x** text field, type `1`.

7 In the **y** text field, type `0.5-0.333/4`.

8 In the **Geometry** toolbar, click **Build All**.

MATERIALS

Define the materials in the next step. Introduce them as empty material nodes, first. As soon as the physics has been defined, the material node menu will show you which properties are needed for the simulation. You can then just fill in the values defined in the **Parameters** list.

Solid Matrix

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 `Solid Matrix` in the **Label** text field.

Water

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

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

Ice

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

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

Porous Material 1 (pmat1)

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

2 In the **Settings** window for **Porous Material**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **All domains**.

4 Locate the **Phase-Specific Properties** section. 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 (mat2)**.

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 From the **Material** list, choose **Solid Matrix (mat1)**.

4 In the θ_s text field, type `1-epsilon_p`.

With the following steps you define the residual liquid water as immobile fluid and its volume as a product of porosity and predefined residual water saturation.

Immobile Fluid 1 (pmat1.imfluid1)

1 In the **Model Builder** window, right-click **Porous Material 1 (pmat1)** and choose **Immobile Fluid**.

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

3 From the **Material** list, choose **Water (mat2)**.

4 In the θ_{imf} text field, type `epsilon_p*Sw_res`.

HEAT TRANSFER IN POROUS MEDIA (HT)

Continue with setting up the physics. Since some heat transfer variables are needed to define the saturation variables as explained in the Model Definition section, start with Heat Transfer in Porous Media (ht).

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

2 In the **Settings** window for **Heat Transfer in Porous Media**, locate the **Physical Model** section.

3 In the T_{ref} text field, type `T_initial_w`.

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 **Define** list, choose **Solid phase properties**, because the properties of a pure solid material are given.

Now add the immobile fluid to the **Porous Medium** node. It will automatically refer to the **Porous Material** settings you have made earlier.

Porous Medium 1

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

Immobile Fluids 1

In the **Physics** toolbar, click **Attributes** and choose **Immobile Fluids**.

Fluid 1

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

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

3 From the p_A list, choose **Absolute pressure (dl)**.

4 Locate the **Heat Convection** section. From the $\mathbf{u}$ list, choose **Total Darcy velocity field (dl/porous1)**.

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**, click to expand the **Sketch** section.

Inflate the sketch to visualize which material you have to choose for which temperatures. In this case, material 1 should be ice, material 2 water. Enter a phase transition function that has yet to be defined.

3 Locate the **Phase Change** section. From the **Phase transition function** list, choose **User defined**. In the $\alpha_{1\to 2}$ text field, type `f_phtr(T)`.

4 In the $L_{1\to 2}$ text field, type `L`.

5 Locate the **Phase 1** section. From the **Material, phase 1** list, choose **Ice (mat3)**.

6 Locate the **Phase 2** section. From the **Material, phase 2** list, choose **Water (mat2)**.

Now define the phase transition function.

GLOBAL DEFINITIONS

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
W	0.5[K]	0.5 K	Parameter for phase transition function

DEFINITIONS (COMP1)

Analytic 1 (an1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Component 1 (comp1)>Definitions** and choose **Functions>Analytic**.

3 In the **Settings** window for **Analytic**, type `f_phtr` in the **Function name** text field.

4 Locate the **Definition** section. In the **Expression** text field, type `exp(-((T-273.15)/W)^2)*(T<273.15)+1*(T>=273.15)`.

5 In the **Arguments** text field, type `T`.

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

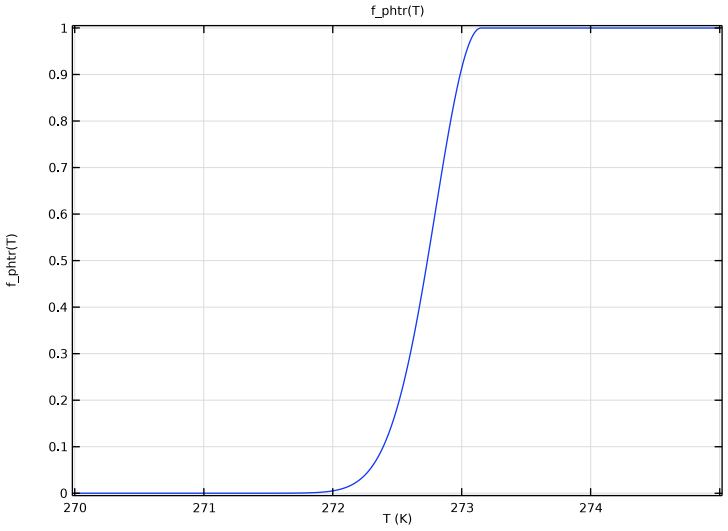
Argument	Unit
T	K

To ensure that you have entered everything correctly, plot the function as follows.

7 Locate the **Plot Parameters** section. In the table, enter the following settings:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	T	270	275	0	K

8 Click **Plot**.



Now continue setting up the physics for the **Heat Transfer in Porous Media** interface.

HEAT TRANSFER IN POROUS MEDIA (HT)

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_initial_w`.

Initial Values 2

1 In the **Physics** toolbar, click **Domains** and choose **Initial Values**.

2 Select Domain 2 only.

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

4 In the T text field, type `T_initial_i`.

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_initial_w`.

Outflow 1

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

2 Select Boundary 9 only.

Symmetry 1

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

2 Select Boundaries 3, 6, and 8 only.

DEFINITIONS (COMP1)

For Darcy's Law the water saturation has to be defined as temperature-dependent variable. More precisely it depends on the phase transition function which determines the fraction of water as a function of temperature during phase change. The permeability is then defined as a function of water saturation.

Variables 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Component 1 (comp1)>Definitions** and choose **Variables**.

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

4 In the table, enter the following settings:

Name	Expression	Unit	Description
Sw	`Sw_res+(1-Sw_res)*f_phtr(T)`		Water saturation
kr	`max(10^(-Omega*epsilon_p*(1-Sw)),1e-6)`		Relative permeability
kappa_w	`k_int*kr`	m^2	Permeability

DARCY'S LAW (DL)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Darcy's Law (dl)**.

2 In the **Settings** window for **Darcy's Law**, click to expand the **Discretization** section.

3 From the **Pressure** list, choose **Linear**.

Change the shape function to linear to reduce the computational time and increase numerical stability.

Porous Medium 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Darcy's Law (dl)** click **Porous Medium 1**.

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

3 From the **Storage model** list, choose **User defined**. In the S_p text field, type `Sw* epsilon_p*beta`.

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 ε_p list, choose **User defined**. In the associated text field, type `epsilon_p*Sw`.

Mass Source 1

1 In the **Physics** toolbar, click **Domains** and choose **Mass Source**.

The mass source accounts for the addition of water during melting as described in the Model Definition section.

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

3 From the **Selection** list, choose **All domains**.

4 Locate the **Mass Source** section. In the Q_m text field, type `dl.epsilon*(rho_i-rho_w)*d(Sw,t)`. Remember that `dl.epsilon` was defined as `epsilon_p*Sw` to represent only the pore space that is filled with water.

In the INTERFROST intercomparison experiment, the flow is driven by an imposed gradient of the hydraulic head. In this case, a pressure gradient is imposed which is calculated from the hydraulic head.

Inlet 1

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

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

3 From the **Boundary condition** list, choose **Pressure**.

4 Select Boundary 1 only.

5 Locate the **Pressure** section. In the p_0 text field, type `g_const*delta_H_dLx*3[m]*rho_w`.

Outlet 1

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

2 Select Boundary 9 only.

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

4 From the **Boundary condition** list, choose **Pressure**.

Symmetry 1

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

2 Select Boundaries 3, 6, and 8 only.

MATERIALS

Having defined the physics, now fill in the empty expressions in the **Materials** node.

Porous Material 1 (pmat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Porous Material 1 (pmat1)**.

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Permeability	kappa_iso ; kappaii = kappa_iso, kappaij = 0	`kappa_w`	m²	Basic

Solid Matrix (mat1)

1 In the **Model Builder** window, click **Solid Matrix (mat1)**.

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
Density	rho	`rho_s`	kg/m³	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`lambda_s`	W/(m·K)	Basic
Heat capacity at constant pressure	Cp	`Cs`	J/(kg·K)	Basic

Water (mat2)

1 In the **Model Builder** window, click **Water (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
Density	rho	rho_w	kg/m³	Basic
Dynamic viscosity	mu	mu_w	Pa·s	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	lambda_w	W/(m·K)	Basic
Heat capacity at constant pressure	Cp	Cw	J/(kg·K)	Basic

Ice (mat3)

1 In the **Model Builder** window, click **Ice (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	lambda_i	W/(m·K)	Basic
Density	rho	rho_i	kg/m³	Basic
Heat capacity at constant pressure	Cp	Ci	J/(kg·K)	Basic

MESH 1

When generating the mesh, note that it should be fine enough to capture strong gradients within the phase change region. In this case, a maximum element size of 5 mm is enforced in the domain of the frozen inclusion.

Free Triangular 1

1 In the **Mesh** toolbar, click  **Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.

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

4 Select Domain 2 only.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section.

5 Select the **Maximum element size** check box. In the associated text field, type `0.005`.

6 Click **Build Selected**.

Free Triangular 2

In the **Mesh** toolbar, click **Free Triangular**.

Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extra fine**.

Free Triangular 2

In the **Model Builder** window, right-click **Free Triangular 2** and choose **Build All**.

STUDY 1

Step 1: Stationary

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

The initial pressure and velocity field results from a steady state flow simulation which is performed in this first step.

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 Porous Media (ht)**.

4 In the **Home** toolbar, click **Compute**.

Step 2: Time Dependent

Now add the time dependent study step. To ensure that the phase change is represented properly, the time steps have to be chosen small enough to catch the increase of temperature as soon as the ice has totally melted. This is done by defining the output times accordingly and by forcing the time step to be smaller than the output intervals.

1 In the **Study** toolbar, click **Study Steps** and choose **Time Dependent> 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 `range(0,5,60)[min] range(1.5,0.5,18) range(18.1,0.025,22) range(22.5,0.5,28) range(29,1,56)`.

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

6 In the **Relative tolerance** text field, type `0.005`.

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 ensures that the time step cannot be bigger than the output intervals. However, it can still be smaller if necessary.

5 Select the **Initial step** check box. In the associated text field, type `1[s]`.

Step 2: Time Dependent

In the **Model Builder** window, under **Study 1** right-click **Step 2: Time Dependent** and choose **Compute Selected Step**.

RESULTS

The default plot shows the pressure distribution and velocity field at the final time. To see the pressure and velocity field for the full model domain at an earlier time, when there is still some ice present (see Figure 2), follow the steps below. First, create the full dataset from the simplified symmetric model domain.

Mirror 2D 1

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets** and choose **More 2D Datasets>Mirror 2D**.

3 In the **Settings** window for **Mirror 2D**, locate the **Axis Data** section.

4 In row **Point 1**, set **Y** to `0.5`.

5 In row **Point 2**, set **X** to `3`.

6 In row **Point 2**, set **Y** to `0.5`.

7 Click to expand the **Advanced** section. Select the **Remove elements on the symmetry axis** check box.

Pressure (dl)

1 In the **Model Builder** window, under **Results** click **Pressure (dl)**.

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

3 From the **Dataset** list, choose **Mirror 2D 1**.

4 From the **Time (h)** list, choose **9**.

Streamline 1

1 In the **Model Builder** window, expand the **Pressure (dl)** node, then click **Streamline 1**.

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

3 From the **Positioning** list, choose **Magnitude controlled**.

4 In the **Minimum distance** text field, type `0.02`.

5 In the **Maximum distance** text field, type `0.1`.

6 In the **Pressure (dl)** toolbar, click **Plot**.

ADD PREDEFINED PLOT

Add the surface plot of the temperature field as shown in Figure 3. To do this, choose the **Temperature** plot from the **Add Predefined Plot** window as follows.

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 Porous Media> Temperature (ht)**.

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

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

RESULTS

Temperature (ht)

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

2 From the **Dataset** list, choose **Mirror 2D 1**.

3 From the **Time (h)** list, choose **9**.

Surface 1

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

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

3 From the **Unit** list, choose **degC**.

4 In the **Temperature (ht)** toolbar, click **Plot**.

Liquid Water Saturation

To reproduce the plot of the liquid water saturation (Figure 4), follow the instructions below.

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 2D 1**.

4 From the **Time (h)** list, choose **9**.

Contour 1

1 Right-click **Liquid Water Saturation** and choose **Contour**.

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

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

4 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

5 In the **Levels** text field, type `0.05 0.5 1`.

6 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Filled**.

7 Click **Change Color Table**.

8 In the **Color Table** dialog box, select **Aurora>JupiterAuroraBorealis** in the tree.

9 Click **OK**.

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

11 From the **Color table transformation** list, choose **Reverse**.

12 In the **Liquid Water Saturation** toolbar, click **Plot**.

Surface Minimum 1

To reproduce the next plots the resulting data has to be further evaluated. Follow the instructions below to derive the minimum temperature as a function of time.

1 In the **Results** toolbar, click **More Derived Values** and choose **Minimum> Surface Minimum**.

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

3 From the **Selection** list, choose **All domains**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Heat Transfer in Porous Media>Temperature>T - Temperature - K**.

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

Expression	Unit	Description
`T`	`degC`	`Temperature`

6 Click = **Evaluate**.

TABLE 1

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

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

RESULTS

Minimum Temperature

1 In the **Model Builder** window, under **Results** click **1D Plot Group 4**.

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

Line Integration 1

Following the next steps you will derive and plot the total heat flux exiting the system via the downstream boundary as a function of time.

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Integration> Line Integration**.

2 Select Boundary 9 only.

3 In the **Settings** window for **Line Integration**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)> Heat Transfer in Porous Media>Boundary fluxes>ht.ntflux - Normal total heat flux - W/m²**.

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

Expression	Unit	Description
`ht.ntflux*dl.d*2`	`W`	`Total Heat Flux`

As the model is a 2D approximation of the thawing process and due to symmetry only half the domain was modeled, the heat flux value has to be multiplied by the thickness of the model, `dl.d`, and by a factor of 2 to get the correct value.

5 Click ▼ next to = **Evaluate**, then choose **New Table**.

TABLE 2

1 Go to the **Table 2** window.

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

RESULTS

Total Heat Flux

1 In the **Model Builder** window, under **Results** click **1D Plot Group 5**.

2 In the **Settings** window for **1D Plot Group**, type `Total Heat Flux` in the **Label** text field.

Surface Integration 2

The final plot, the evolution of total liquid water volume in the domain as shown in Figure 7 can be produced as follows.

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Integration> Surface Integration**.

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

3 From the **Selection** list, choose **All domains**.

4 Locate the **Expressions** section. Click **Clear Table**.

5 In the table, enter the following settings:

Expression	Unit	Description
`Sw*epsilon_p*dl.d*2`	`m^3`	`Total liquid water volume`

`Sw*epsilon_p` gives the pore space filled with water, `dl.d` is the thickness of the model and the factor of 2 is due to symmetry.

6 Click next to **Evaluate**, then choose **New Table**.

TABLE 3

1 Go to the **Table 3** window.

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

RESULTS

Total Liquid Water Volume

1 In the **Model Builder** window, under **Results** click **1D Plot Group 6**.

2 In the **Settings** window for **1D Plot Group**, type `Total Liquid Water Volume` in the **Label** text field.

Compare your results with those presented in Figure 7.