Ground Heat Recovery for Radiant Floor Heating

Introduction

Geothermal heating is an environmentally friendly and energy efficient method to supply modern and well insulated houses with heat. The investment costs are higher than for gas or oil heating so there is a need to investigate the possibilities of arranging heat collectors in the subsurface.

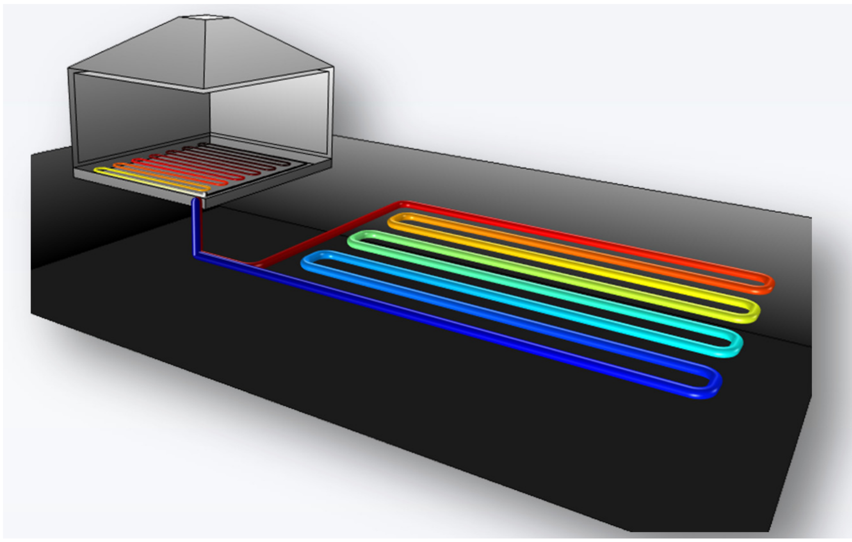


Figure 1: Example of heat recovery coils in a garden, connected to a dwelling via a heat pump.

This example compares three different patterns embedded in the subsurface. Typical thermal properties of an uppermost soil layer in a garden are used for the calculations.

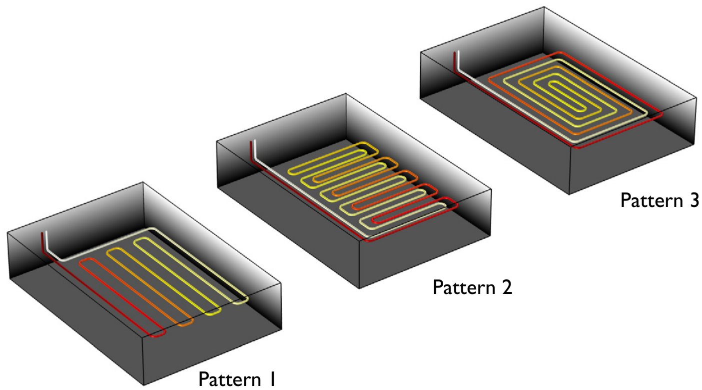


Figure 2: Patterns for the heat collectors.

Model Definition

The three patterns for the pipe arrangement are shown in Figure 2. The geometry subsequence functionality in COMSOL Multiphysics offers the possibility to perform the analysis over different pipe arrangements within the same model using a parametric sweep.

This model uses functions and events to describe the real operating conditions. For the subsurface, a temperature gradient with depth is prescribed. At the surface, a time dependent temperature is applied which corresponds to typical temperature variations in central Europe.

Assuming that the fluid properties are temperature-independent, the inlet temperature of the heat collector required to reach a certain heat extraction rate, $P(t)$ is given by

$$T_{\mathrm{in}} = T_{\mathrm{out}} - \frac{P}{\rho C_p \dot{V}} \tag{1}$$

where ρ and C_p the density and specific heat capacity for the fluid inside the pipes and $\dot{V}$ the volumetric flow rate, here equal to 1 l/s.

The dynamic heat extraction is triggered according to a typical daily heat demand of a single-family house. The heat extraction process is active each day until the demanded heat is extracted. When the demand is reached, the heat extraction is stopped but the fluid flow in the pipes still continue at a lower rate. In this application, this flow rate during inaction of the pump is set to 1/10th of the operating flow rate, that is, 0.1 l/s.

Results and Discussion

The pipe temperature for the third pattern after two days is shown below.

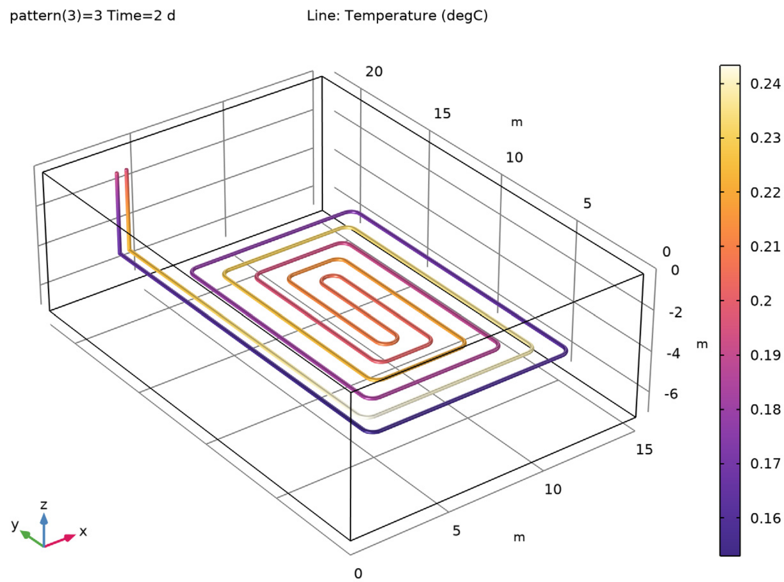


Figure 3 shows the outlet temperature over time for the third pattern. It is important to make sure that the fluid inside the pipes stays above a certain value. In this example, water is used as working fluid. However, during winter, the ambient temperature above the surface may reach values below 0 °C. In practical situations, antifreezes — such as glycerol

— are added to water and this application assumes that such additives do not modify its thermal properties.

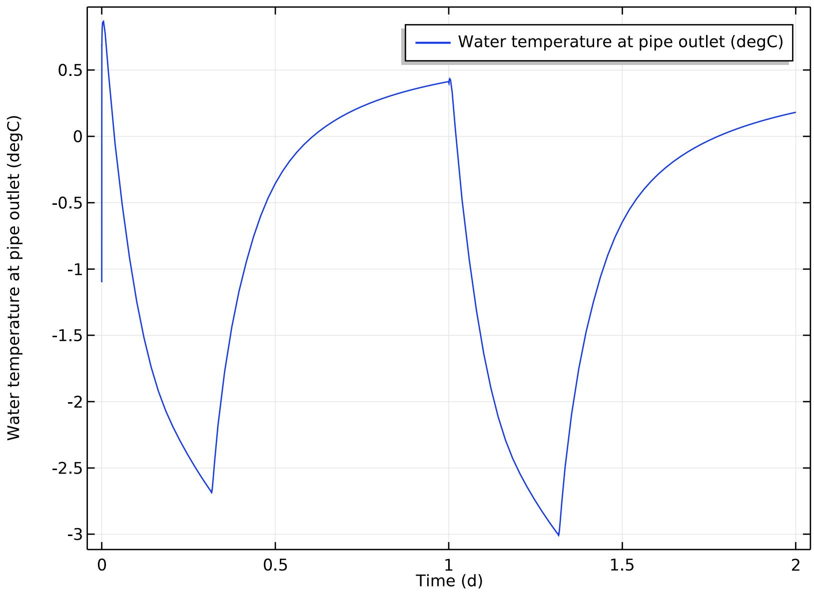


Figure 3: Outlet temperatures for the third pattern.

The temperature variations in Figure 3 correspond to the moments when the heater is turned on or off. The graph of the heater state in time is shown in Figure 4.

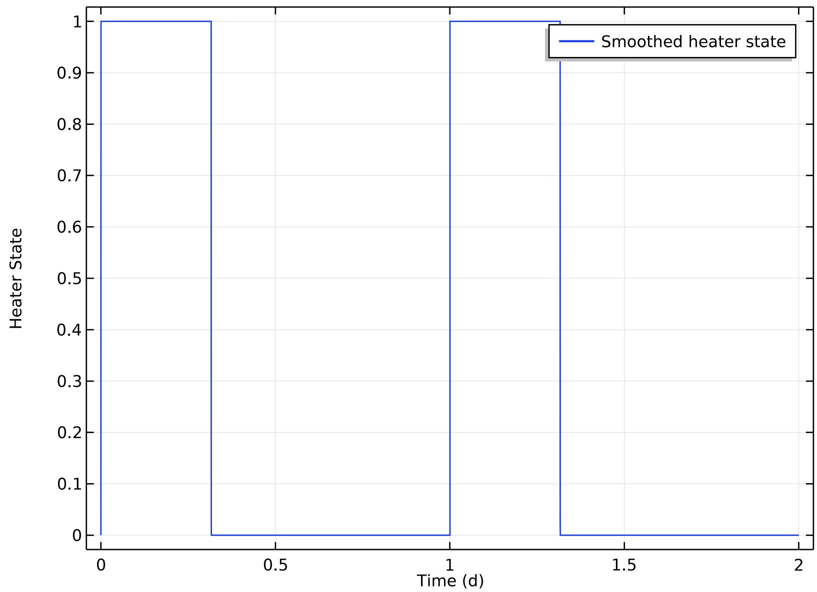


Figure 4: Heater state in time for the third pattern. A value of 0 indicates that the heater is turned off while 1 indicates that the heater is turned on.

The heater state itself depends on the daily heat production. As soon as the heat production reaches the daily demand, the heater state is turned off until the next day.

Figure 5 shows the heat production the two days of simulation time. Just after the daily requirement of 30 kWh, the production is stopped until the next day.

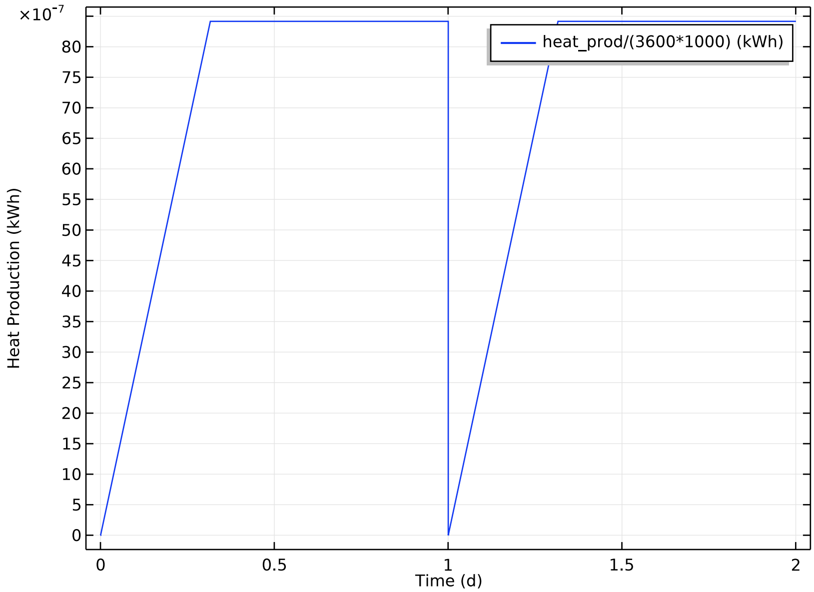


Figure 5: Heat production for the third pattern.

Notes About the COMSOL Implementation

This model uses several interpolation functions, which are based on rough estimations and local temperature variations, which may differ for your case. The functions needs to be replaced by your own measured data. Then, you need to make sure that the time stepping is still fine enough to resolve them.

Application Library path: `Pipe_Flow_Module/Heat_Transfer/ground_heat_recovery`

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 **3D**.
2 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Pipes (htp)**.
5 Click **Add**.
6 In the **Select Physics** tree, select **Mathematics>ODE and DAE Interfaces>Events (ev)**.
7 Click **Add**.
8 In the **Select Physics** tree, select **Mathematics>ODE and DAE Interfaces>Global ODEs and DAEs (ge)**.
9 Click **Add**.
10 Click **Study**.
11 In the **Select Study** tree, select **General Studies>Time Dependent**.
12 Click **Done**.

GEOMETRY 1

This example demonstrates how to use part import and programming to perform the simulation for different geometries. Start by importing the parts from a file, as shown below.

1 In the **Geometry** toolbar, click **Parts** and choose **Load Part**.
2 Browse to the model's Application Libraries folder and double-click the file `ground_heat_recovery_geom_sequence.mph`.
3 In the **Load Part** dialog box, in the **Select parts** list, choose **Pattern 1**, **Pattern 2**, and **Pattern 3**.
4 Click **OK**.

GLOBAL DEFINITIONS

Parameters 1

Define the parameter that will be used to call the different patterns.

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
pattern	3	3	Parameter for selecting the pattern

This parameter is now used to import the different patterns into the work plane based on a logical expression.

Add the remaining parameters.

4 In the table, enter the following settings:

Name	Expression	Value	Description
d_pipe	36[mm]	0.036 m	Pipe diameter in soil
flowrate_pipe	1[l/s]	0.001 m^3/s	Flow rate inside pipes
heat_demand	30[kW*h]	1.08E8 J	Daily heat demand
power	4[kW]	4000 W	Heat pump power
dt	30[s]	30 s	Smoothed heater state transition zone
depth	4[m]	4 m	Depth of heat exchanger
Tz_depth	0.5[K/m]	0.5 K/m	Temperature gradient
month	1	1	Month index
humidity	1	1	Soil humidity
k_soil	0.18[W/(m*K)]+(1.5-0.18)*humidity	1.5 W/(m·K)	Soil thermal conductivity

GEOMETRY 1

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

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

3 In the **z-coordinate** text field, type `-depth`.

4 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

5 In the **New Cumulative Selection** dialog box, type `Pipes` in the **Name** text field.

6 Click **OK**.

The above step ensures that, independently of the pattern you choose, the pipes are selected correctly.

7 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Work Plane 1 (wp1)>If 1 (if1)

1 In the **Work Plane** toolbar, click **Programming** and choose **If + End If**.

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

3 In the **Condition** text field, type `pattern==1`.

Work Plane 1 (wp1)>Import 1 (imp1)

1 In the **Work Plane** toolbar, click **Import**.

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

3 From the **Source** list, choose **Geometry sequence**.

4 From the **Geometry** list, choose **Pattern 1**.

Work Plane 1 (wp1)>Else If 1 (elseif1)

1 In the **Model Builder** window, right-click **Plane Geometry** and choose **Programming>Else If**.

2 In the **Settings** window for **Else If**, locate the **Else If** section.

3 In the **Condition** text field, type `pattern==2`.

Work Plane 1 (wp1)>Import 2 (imp2)

1 In the **Work Plane** toolbar, click **Import**.

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

3 From the **Source** list, choose **Geometry sequence**.

4 From the **Geometry** list, choose **Pattern 2**.

Work Plane 1 (wp1)>Else 1 (else1)

Right-click **Plane Geometry** and choose **Programming>Else**.

Work Plane 1 (wp1)>Import 3 (imp3)

1 In the **Work Plane** toolbar, click **Import**.

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

3 From the **Source** list, choose **Geometry sequence**.

4 From the **Geometry** list, choose **Pattern 3**.

Polygon 1 (pol1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **More Primitives> Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

x (m)	**y (m)**	**z (m)**
`2.5`	`20`	`0`
`2.5`	`20`	`-depth`

4 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Pipes**.

Copy 1 (copy1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **pol1** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

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

5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Pipes**.

Block 1 (blk1)

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

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

3 In the **Width** text field, type `15`.

4 In the **Depth** text field, type `22`.

5 In the **Height** text field, type `depth+3[m]`.

6 Locate the **Position** section. In the **z** text field, type `-(depth+3[m])`.

7 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

8 In the **New Cumulative Selection** dialog box, type `Ground` in the **Name** text field.

9 Click **OK**.

10 In the **Settings** window for **Block**, click **Build All Objects**.

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

DEFINITIONS

Hide for Physics 1

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

2 Right-click **View 1** and choose **Hide for Physics**.

3 In the **Settings** window for **Hide for Physics**, locate the **Geometric Entity Selection** section.

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

5 Select Boundaries 1, 2, and 4 only.

GLOBAL DEFINITIONS

Define the functions for the surface temperature and the initial temperature.

Yearly Temperature Profile

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, type `Yearly Temperature Profile` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `T_z0`.

4 Click **Load from File**.

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

6 Locate the **Interpolation and Extrapolation** section. From the **Interpolation** list, choose **Piecewise cubic**.

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

Argument	Unit
t	a

8 In the **Function** table, enter the following settings:

Function	Unit
T_z0	degC

9 Click **Create Plot**.

RESULTS

Temperature Profile

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

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

3 Locate the **Plot Settings** section.

4 Select the **x-axis label** check box. In the associated text field, type `Month`.

Function 1

1 In the **Model Builder** window, expand the **Temperature Profile** node, then click **Function 1**.

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

3 In the **Expression** text field, type `T_z0(root.t[a])[1/degC]`.

4 Locate the **x-Axis Data** section. In the **Expression** text field, type `root.t*12`.

5 In the **Lower bound** text field, type `1`.

6 In the **Upper bound** text field, type `12`.

7 In the **Description** text field, type `Month`.

8 Click to expand the **Coloring and Style** section. From the **Width** list, choose **3**.

Line Graph 1

1 In the **Model Builder** window, right-click **Temperature Profile** and choose **Line Graph**.

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

3 From the **Dataset** list, choose **Grid 1D 1**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `T_z0(root.t[a])[1/degC]`.

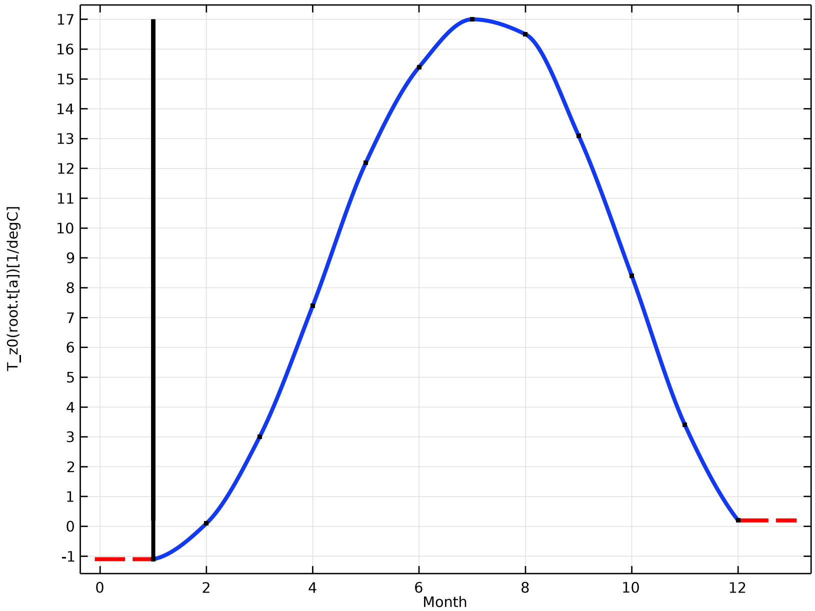
5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `month`.

7 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Black**.

8 From the **Width** list, choose **3**.

9 In the **Temperature Profile** toolbar, click **Plot**.



GLOBAL DEFINITIONS

Depth Temperature Gradient

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Analytic**.

2 In the **Settings** window for **Analytic**, type `Depth Temperature Gradient` in the **Label** text field.

3 In the **Function name** text field, type `T0`.

4 Locate the **Definition** section. In the **Expression** text field, type `T_z0(month[a]/12)[1/degC]+Tz_depth[m/K]*(-z)`.

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

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

Argument	Unit
z	m

7 In the **Function** text field, type `degC`.

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

Argument	Lower limit	Upper limit	Unit
z	-10	0	m

9 Click **Create Plot**.

RESULTS

Temperature Gradient

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

2 Locate the **Title** section. From the **Title type** list, choose **None**.

3 Locate the **Plot Settings** section.

4 Select the **x-axis label** check box. In the associated text field, type `Depth (m)`.

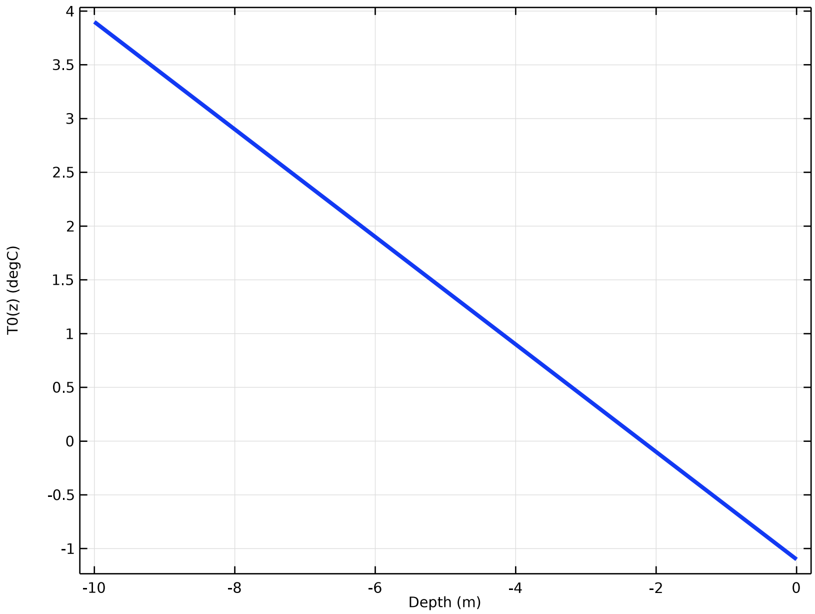
Function 1

1 In the **Model Builder** window, expand the **Temperature Gradient** node, then click **Function 1**.

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

3 From the **Width** list, choose **3**.

4 In the **Temperature Gradient** toolbar, click **Plot**.



GLOBAL DEFINITIONS

Smoothed Heaviside Function

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Step**.

2 In the **Settings** window for **Step**, type `Smoothed Heaviside Function` in the **Label** text field.

3 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `1`.

4 Click **Plot**.

MATERIALS

Next, add the material properties for the subsurface and the fluid inside the pipes. The thermal and hydrodynamic properties for common cooling fluids are similar to the water properties. Use water from the built-in material library as the fluid inside the pipes.

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

Water, liquid (mat1)

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

2 From the **Geometric entity level** list, choose **Edge**.

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

Soil

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

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

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Ground**.

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

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`k_soil`	W/(m·K)	Basic
Density	rho	1742	kg/m³	Basic
Heat capacity at constant pressure	Cp	1175	J/(kg·K)	Basic

Set up the Heat Transfer in Solids interface. Set the initial conditions to the depth dependent temperature `T_0` which you defined as a function before.

HEAT TRANSFER IN SOLIDS (HT)

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (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 `T0(z)`.

The surface temperature, also defined as a function `T_z0`, varies in time.

Temperature 1

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

2 Select Boundary 4 only.

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

4 In the T_0 text field, type `T_z0(t+month[a]/12)`.

Heat Flux 1

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

2 Select Boundary 3 only.

3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

4 In the q_0 text field, type `-ht.k_iso*Tz_depth`.

HEAT TRANSFER IN PIPES (HTP)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Pipes (htp)**.

2 In the **Settings** window for **Heat Transfer in Pipes**, locate the **Edge Selection** section.

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

Heat Transfer 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Pipes (htp)** click **Heat Transfer 1**.

2 In the **Settings** window for **Heat Transfer**, locate the **Heat Convection and Conduction** section.

3 In the u text field, type `flowrate_pipe*(1/10+heater_state_smoothed*9/10)/htp.A`.

Pipe Properties 1

1 In the **Model Builder** window, click **Pipe Properties 1**.

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

3 From the list, choose **Circular**.

4 In the d_i text field, type `d_pipe`.

DEFINITIONS

Use a nonlocal integration coupling for evaluating the outlet temperature. Then the inlet temperature for the pipes can be evaluated according to Equation 1.

Integration 1 (intop1)

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

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

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

4 Select Point 11 only.

Integration 2 (intop2)

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

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

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

4 Select Point 7 only.

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	Description
heater_state_smoothed	if(heater_state,1,step1((mod(t+dt/2,1[d])-dt)/dt))*if(t_stop>0,1-step1((t-(t_stop+dt/2))/dt),1)	Smoothed heater state
T_out	intop1(T2)	Water temperature at pipe outlet
T_in	nojac(T_out-power*heater_state_smoothed/(intop1(htp.rho)*intop1(htp.Cp)*flowrate_pipe))	Inlet temperature for pipes

HEAT TRANSFER IN PIPES (HTP)

Temperature 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Pipes (htp)** click **Temperature 1**.

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

3 In the T_{in} text field, type `T_in`.

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 $T2$ text field, type `T0(z)`.

Heat Outflow 1

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

2 Select Point 11 only.

Wall Heat Transfer 1

1 In the **Physics** toolbar, click **Edges** and choose **Wall Heat Transfer**.

2 In the **Settings** window for **Wall Heat Transfer**, locate the **Edge Selection** section.

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

Internal Film Resistance 1

In the **Physics** toolbar, click **Attributes** and choose **Internal Film Resistance**.

Wall Heat Transfer 1

In the **Model Builder** window, click **Wall Heat Transfer 1**.

Wall Layer 1

1 In the **Physics** toolbar, click **Attributes** and choose **Wall Layer**.

2 In the **Settings** window for **Wall Layer**, locate the **Specification** section.

3 From the k list, choose **User defined**.

4 In the text field, type `400`.

5 From the Δw list, choose **User defined**.

6 In the text field, type `3.25[mm]`.

MULTIPHYSICS

Pipe Wall Heat Transfer 1 (pwhtc1)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Edge>Pipe Wall Heat Transfer**.

EVENTS (EV)

In the **Model Builder** window, under **Component 1 (comp1)** click **Events (ev)**.

Discrete States 1

1 In the **Physics** toolbar, click **Global** and choose **Discrete States**.

2 In the **Settings** window for **Discrete States**, locate the **Discrete States** section.

3 In the table, enter the following settings:

Name	Initial value (u0)
heater_state	0

Indicator States 1

1 In the **Physics** toolbar, click **Global** and choose **Indicator States**.

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

3 In the table, enter the following settings:

Name	g(v,vt,vtt,t)	Initial value (u0)
heat_diff	heat_demand-heat_prod	heat_demand

Explicit Event 1

1 In the **Physics** toolbar, click **Global** and choose **Explicit Event**.

2 In the **Settings** window for **Explicit Event**, locate the **Event Timings** section.

3 In the t_i text field, type `dt`.

4 In the T text field, type `24[h]`.

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

Variable	Expression
heater_state	1
t_stop	t
heat_prod	0

Implicit Event 1

1 In the **Physics** toolbar, click **Global** and choose **Implicit Event**.

2 In the **Settings** window for **Implicit Event**, locate the **Event Conditions** section.

3 In the **Condition** text field, type `heat_diff<0`.

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

Variable	Expression
heater_state	0

GLOBAL ODES AND DAES (GE)

Global Equations 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Global ODEs and DAEs (ge)** click **Global Equations 1**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)
heat_prod	power*heater_state-heat_prodt	0	0

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `energy` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Energy (J)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, type `power` in the text field.

12 Click **Filter**.

13 In the tree, select **General>Power (W)**.

14 Click **OK**.

Global Equations 2

1 In the **Global ODEs and DAEs** toolbar, click **Global Equations**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)
`t_stop`	`heater_state-t_stopt`	0	0

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `time` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Time (s)**.

8 Click **OK**.

DEFINITIONS

Define the probes to monitor the outlet temperature, heat production, and heater state while solving.

Outlet Temperature

1 In the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, type `Outlet Temperature` in the **Label** text field.

3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>T_out - Water temperature at pipe outlet - K**.

4 Locate the **Expression** section. From the **Table and plot unit** list, choose **degC**.

5 Click to expand the **Table and Window Settings** section. From the **Output table** list, choose **New table**.

6 From the **Plot window** list, choose **New window**.

Heat Production

1 In the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, type `Heat Production` in the **Label** text field.

3 Locate the **Expression** section. From the **Table and plot unit** list, choose **kWh**.

4 Locate the **Table and Window Settings** section. From the **Output table** list, choose **New table**.

5 From the **Plot window** list, choose **New window**.

Heater State

1 In the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, type `Heater State` in the **Label** text field.

3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>heater_state_smoothed - Smoothed heater state**.

4 Locate the **Table and Window Settings** section. From the **Output table** list, choose **New table**.

5 From the **Plot window** list, choose **New window**.

MESH 1

Edge 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Edge**.

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

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

Size 1

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

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

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

4 Select Points 7 and 11 only.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.

6 Click the **Custom** button.

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

8 Select the **Maximum element size** check box. In the associated text field, type `0.25`.

Size 2

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

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

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

4 Click the **Custom** button.

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

6 Select the **Maximum element size** check box. In the associated text field, type `0.5`.

7 Select the **Minimum element size** check box. In the associated text field, type `0.02`.

Free Tetrahedral 1

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

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 **Fine**.

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

STUDY 1

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
pattern (Parameter for selecting the pattern)	1 2 3	

Step 1: Time Dependent

1 In the **Model Builder** window, 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 **d**.

4 Click **Range**.

5 In the **Range** dialog box, type `3[h]` in the **Step** text field.

6 In the **Stop** text field, type `2`.

7 Click **Replace**.

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 Select the **Initial step** check box. In the associated text field, type `0.1[s]`.

For better accuracy, force the time-dependent solver to use time steps shorter than 30 min.

5 From the **Maximum step constraint** list, choose **Constant**.

6 In the **Maximum step** text field, type `30[min]`.

To solve the problem efficiently, relax the nonlinear setting.

7 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** node, then click **Fully Coupled 1**.

8 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

9 In the **Damping factor** text field, type `1`.

10 From the **Jacobian update** list, choose **Minimal**.

11 In the **Study** toolbar, click **Compute**.

RESULTS

Line 1

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

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

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

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

Surface 1

In the **Model Builder** window, right-click **Temperature (pwhtc1)** and choose **Surface**.

Slice 1

In the **Model Builder** window, right-click **Slice 1** and choose **Disable**.

Surface 1

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

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

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

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Line 1**.

Line 1

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

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

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

4 Locate the **Coloring and Style** section. In the **Radius scale factor** text field, type `5`.

5 In the **Temperature (pwhtc1)** toolbar, click **Plot**.

Outlet Temperature

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

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

This plot group corresponds to that of Figure 3.

3 In the **Outlet Temperature** toolbar, click **Plot**.

Heat Production

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 9**.

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

3 Locate the **Plot Settings** section.

4 Select the **y-axis label** check box. In the associated text field, type `Heat Production (kWh)`.

5 In the **Heat Production** toolbar, click **Plot**.

Heater State

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 10**.

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

3 Locate the **Plot Settings** section.

4 Select the **y-axis label** check box. In the associated text field, type `Heater State`.

5 In the **Heater State** toolbar, click **Plot**.