



Model created in COMSOL Multiphysics 6.4

Artificial Ground Freezing for Tunnel Excavation

Introduction

Artificial ground freezing is used in tunnel excavation when the soil is water-bearing and conventional stabilization methods are insufficient. Because it is an expensive technique, it is typically applied only to short, critical sections where it provides reliable ground support and ensures watertight conditions.

Numerical modeling helps in predicting the freezing process, optimizing the design, and reducing both risks and costs. For the most accurate predictions, a full Thermo-Hydraulic-Mechanical (THM) model should be used. This approach captures the coupled thermal, hydraulic, and mechanical processes — such as heat transfer, phase change, water migration, and changes in soil properties.

Model Definition

THM modeling of the freezing process captures the following processes:

- Heat conduction in the frozen soil
- Heat convection due to groundwater flow
- Phase change during the freezing process
- Water migration (cryosuction)
- Poroelasticity and thermal expansion

HEAT CONDUCTION AND CONVECTION

The time-dependent heat transfer equation for porous media is

$$(\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} + \rho_f C_{p,f} \mathbf{u} \cdot \nabla T + \nabla \cdot (-k_{\text{eff}} \nabla T) = Q$$

The effective volumetric heat capacity $(\rho C_p)_{\text{eff}}$ and the effective thermal conductivity k_{eff} of the porous medium is a combination of the fluid (f) and solid-matrix (s) properties:

$$(\rho C_p)_{\text{eff}} = \varepsilon_p \rho_f C_{p,f} + (1 - \varepsilon_p) \rho_s C_{p,s}$$

$$k_{\text{eff}} = \varepsilon_p k_f + (1 - \varepsilon_p) k_s$$

where ε_p represents the porosity.

The fluid is transported by the Darcy velocity \mathbf{u} and undergoes phase change between water and ice, which alters its properties. This effect is accounted for by using the apparent heat capacity method, described below.

PHASE CHANGE

The phase change is a key effect that leads to a strongly coupled THM system. The apparent heat capacity method describes the phase transition within a temperature interval, incorporating the latent heat as an additional term in the heat capacity.

The volume fractions of ice, θ_1 , and water, θ_2 , are computed using a phase transition function $\alpha_{12} = \theta_2$. This phase transition function corresponds to the soil freezing characteristic curve (SFC), which describes the relationship between temperature and the unfrozen water content in the soil.

A common relation is the van Genuchten-type relationship (Ref. 1):

$$\theta_2 = \begin{cases} [1 + |\alpha\psi(T)|^n]^{\frac{1}{m}} & T < T_f \\ 1 & T \geq T_f \end{cases} \quad (1)$$

The van Genuchten parameters α , n , and $m = 1 - 1/n$ are soil dependent and need to be determined by curve fitting to measurement data. The cryogenic suction head $\psi(T)$ describes the relation between ice pressure, water pressure, and temperature

WATER MIGRATION

The cryogenic suction head can be derived from the generalized Clapeyron equation

$$\psi(T) = \frac{\lambda}{g} \ln \frac{T}{T_f} \quad (2)$$

with the latent heat λ and the freezing temperature T_f . This describes the strong suction of water toward the freezing front.

POROELASTICITY AND THERMAL EXPANSION

The flow velocity is determined by Darcy's law:

$$\mathbf{u} = -\frac{\kappa}{\mu}(\nabla p - \rho \mathbf{g})$$

In this example, the permeability κ depends on the water saturation as described by the van Genuchten relationship:

$$\kappa = \kappa_0 \theta_2 [1 - (1 - \theta_2^{1/n})^n]^2$$

where κ_0 is the intrinsic permeability, and n is the same van Genuchten parameter used for the phase change relationship.

The fluid pressure in the pores acts as a load on the soil. Biot's poroelasticity model reads

$$\nabla \cdot (\mathbf{S} - \alpha_B(p_A - p_{\text{ref}})) + \mathbf{f}_V = 0$$

where \mathbf{S} is the stress tensor, α_B the Biot–Willis coefficient, p_A and p_{ref} are the absolute and reference pressure, and \mathbf{f}_V stands for volumetric loads like gravity. Deformation also leads to changes in porosity, which are taken into account by the poroelasticity coupling.

In addition to the pore pressure, ice formation leads to thermal strains due to the volumetric expansion of ice which is about 9%:

$$\varepsilon_{\text{th}} = 0.09 \cdot \theta_2 \varepsilon_p \cdot$$

MODEL SETUP

The construction site spans an area measuring 20 m in width, 40 m in length, and approximately 20 m in depth, with a variable surface topography. Eight freezing pipes, each with a diameter of 230 mm, are arranged in a semicircular pattern with a slight incline following the ground profile. The tunnel has a diameter of 5.6 m, and the freezing pipes are positioned 3.8 m from the tunnel center. This configuration allows for the formation of a sufficiently thick ice wall to support the excavation. The underlying material consists of an impermeable rock, where artificial ground freezing is not required. All other model parameters are listed in [Table 1](#).

TABLE 1: MODEL PARAMETER

NAME	EXPRESSION	DESCRIPTION
por	0.4	Porosity
kappa0	1e-10[m ²]	Permeability
lambda	334[kj/kg]	Latent heat
T0	8[degC]	Ground temperature
Tc	-30[degC]	Cooling temperature
eta	0.05	Suction potential
n	0.46	Van-Genuchten parameter

The material properties for ice, water and the solid matrix are listed in [Table 2](#).

TABLE 2: MATERIAL PROPERTIES

MATERIAL PROPERTY	WATER	ICE	POROUS MATRIX
Density (kg/m ³)	1000	920	2800
Dynamic Viscosity (Pa·s)	1e-3	-	-
Heat capacity at constant pressure (J/(kg·K))	4182	2060	800
Thermal conductivity (W/(m·K))	0.6	2.14	2.4
Young's Modulus (MPa)	-	-	50
Poisson's ratio (l)	-	-	0.3
Biot-Willis coefficient (l)	-	-	0.95

The Heat Transfer in Porous Media interface is used to describe the heat transport by conduction and convection, as well as accounting for phase change using [Equation 1](#). The surface temperature and the groundwater temperature are set to T_0 . Heat flux boundary conditions are applied at the top and pipes surfaces, where the inner temperature is set to T_c .

For Darcy's law, constant head boundary conditions are applied to generate a slow groundwater flow. To account for the amount of frozen water, a mass source is added according to

$$Q_m = \epsilon_p(\rho_{ice} - \rho_{water}) \frac{\partial \theta_2}{\partial t}$$

The Solid Mechanics interface assumes a linear elastic material under gravity loading. The bottom surface is fixed, while the vertical surfaces use roller conditions. The pipes are defined using a Rigid Connector boundary condition, which allows the pipes to move while keeping their shape.

First, a stationary study is computed without cooling to reach the initial state under gravity, an initial Darcy velocity contribution, and an initial (constant) temperature distribution. A subsequent time dependent study is used to model the freezing process over 30 days.

Results and Discussion

Figure 1 shows the deformation after 30 days, together with the flow field and the isosurface for ice with $\theta_1 = 0.5$ around the pipes.

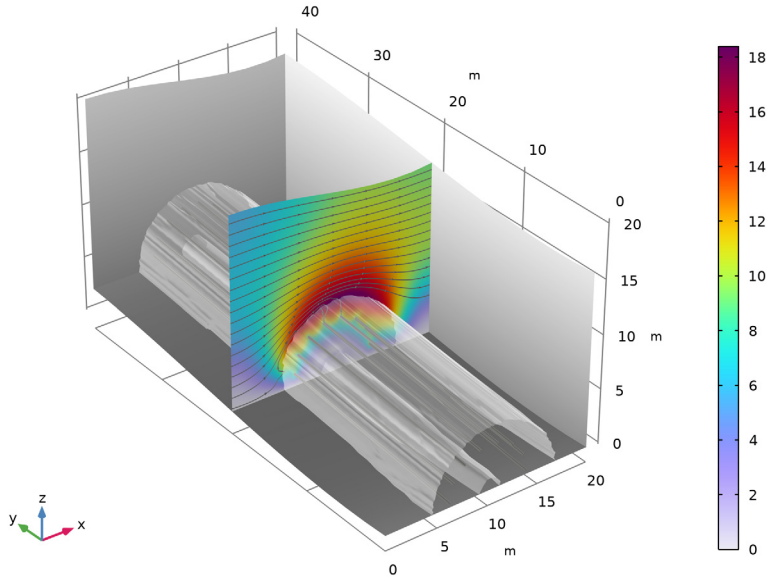


Figure 1: Cut plane showing the deformation in cm and the flow field using streamlines. The isosurface indicates the ice front.

The plot for the ice volume fraction at different times is pictured in Figure 2.

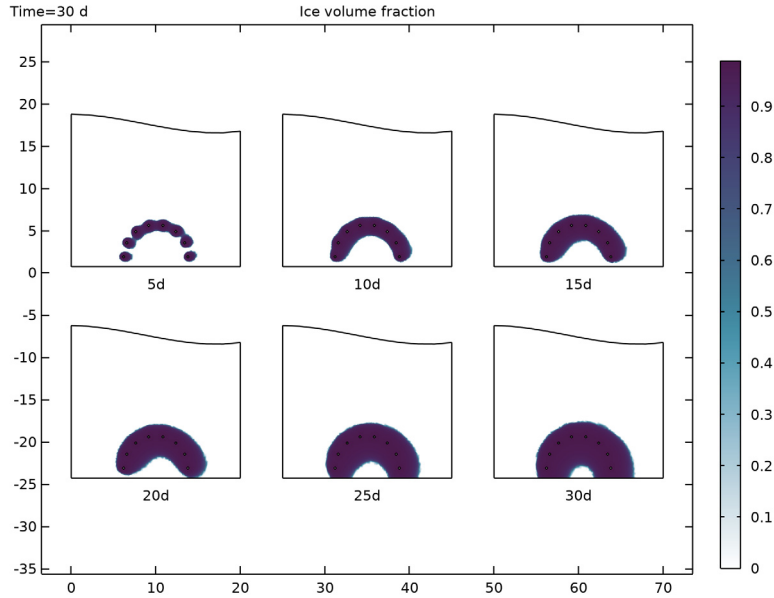


Figure 2: Ice volume fraction at different times.

The ice thickness in the upstream direction and near the bottom surface is the most critical for stability. Therefore, the ice thickness is computed at different locations based on different criteria.

One approach is to use ice saturation as the criterion. In this case, a solid ice wall is assumed to form when the ice saturation exceeds a certain threshold (here, $S_i > 0.9$).

Alternatively, a temperature-based criterion can be used, where the frozen zone is defined as the region where the temperature drops below a certain value (here, $T < -5$ °C).

The choice of the criterion and threshold values depends on the specific soil properties. The results are shown in Figure 3.

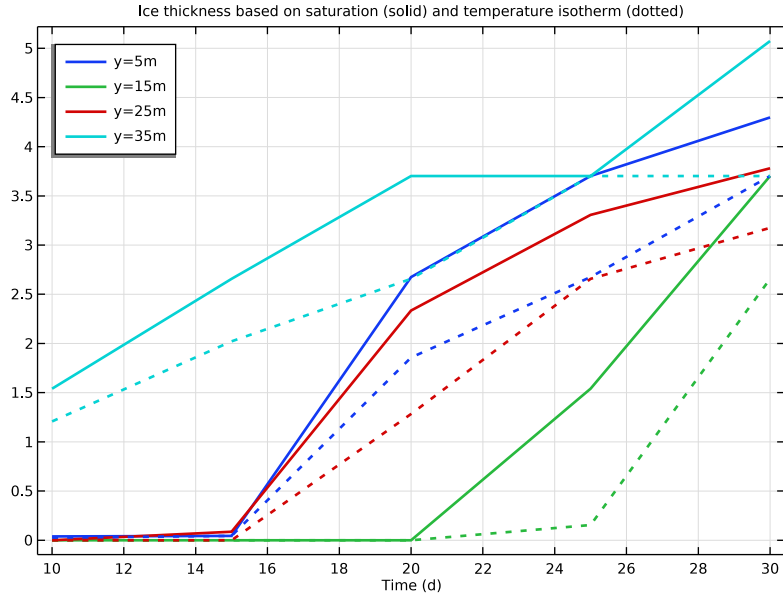


Figure 3: Ice thickness at different locations based on the saturation (solid) or temperature isotherm (dotted).

Reference


I. L.Sun, X. Tang, K. Ramzy Aboyanah, Q. Zhao, Q. Liu, and G. Grasselli “A coupled cryogenic thermo-hydro-mechanical model for frozen medium: Theory and implementation in FDEM,” *J. Rock Mech. Geotech. Eng.*, vol. 16, no. 11, pp. 4335–4353, 2024; doi.org/10.1016/j.jrmge.2023.09.007.

Application Library path: Subsurface_Flow_Module/Poromechanics/artificial_ground_freezing


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**.

Focus on heat transfer first, as it is the key component in THM modeling and has impact on all other coupled processes. Once the Heat Transfer in Porous Media interface is properly set up, add the Poroelasticity multiphysics interface.

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 > Stationary**.

6 Click  **Done**.

GEOMETRY I

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `artificial_ground_freezing_geom_sequence.mph`.

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

4 Click the  **Wireframe Rendering** button in the **Graphics** toolbar.

GLOBAL DEFINITIONS

Parameters I

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

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 `artificial_ground_freezing_parameters.txt`.

Next, define the functions for the cryogenic suction head (Equation 2) and the soil freezing characteristic curve (Equation 1). These are essential for capturing the phase change.

DEFINITIONS

Cryogenic Suction Head

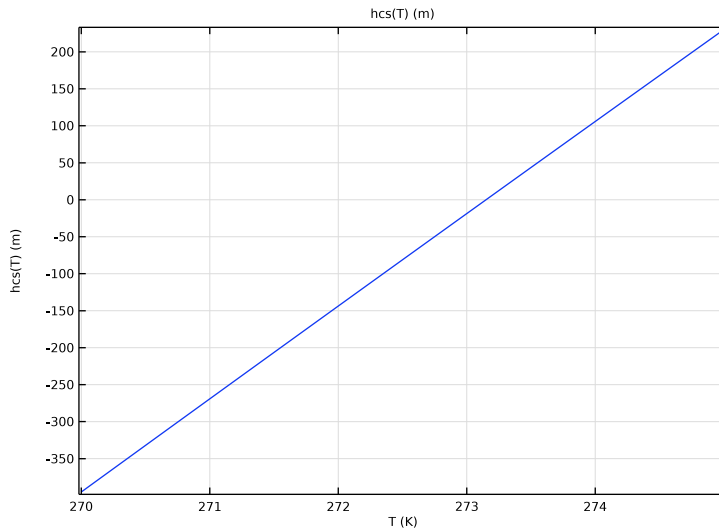
- 1 In the **Definitions** toolbar, click  **Analytic**.
- 2 In the **Settings** window for **Analytic**, type Cryogenic Suction Head in the **Label** text field.
- 3 In the **Function name** text field, type `hcs`.
- 4 Locate the **Definition** section. In the **Expression** text field, type $\lambda \cdot \log(T / 273.15[K]) / g_const$.
- 5 In the **Arguments** text field, type `T`.
- 6 Locate the **Units** section. In the **Function** text field, type `m`.
- 7 In the table, enter the following settings:

Argument	Unit
T	K


- 8 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

9 Click  **Plot**.



Soil Freezing Characteristic Curve

- 1 In the **Definitions** toolbar, click  **Analytic**.
- 2 In the **Settings** window for **Analytic**, type Soil Freezing Characteristic Curve in the **Label** text field.
- 3 In the **Function name** text field, type sfc.
- 4 Locate the **Definition** section. In the **Expression** text field, type $\text{if}(T < 273.15[\text{K}], (1 + \text{abs}(\text{eta} * \text{hcs}(T))^{1/(1-n)})^{-n}, 1)$.
- 5 In the **Arguments** text field, type T.
- 6 Locate the **Units** section. In the table, enter the following settings:


Argument	Unit
T	K

- 7 Click to expand the **Local Parameters** section. In the table, enter the following settings:

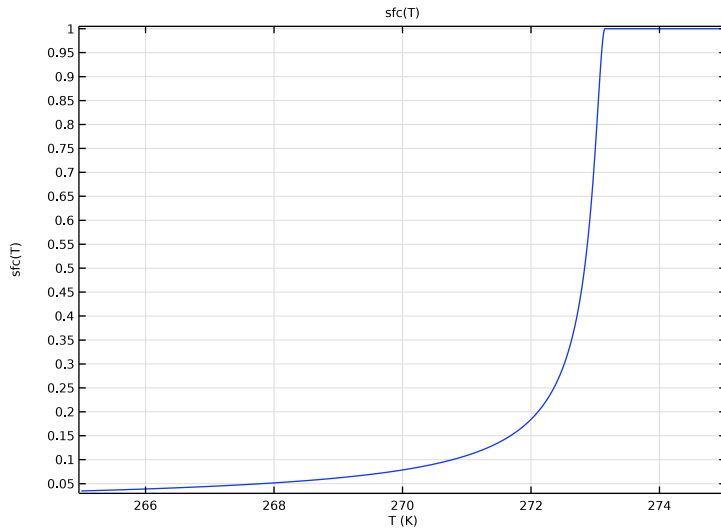
Name	Expression
eta	0.05
n	0.46

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

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	T	265	275	0	K

9 Click  **Plot** and compare with the plot below.

Note the steep gradient near the freezing point. This can be numerically challenging as similar steep gradients will appear in multiple parts of the model.



Define variables for the water and ice saturation.

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
Sw	nojac(sfc(T))		Water volume fraction

The `nojac` operator helps improve convergence by excluding certain terms from the Jacobian matrix. When a function has very steep gradients, it can make the solver unstable. By using `nojac`, the steep part of the function is temporarily ignored in this prediction, making it easier for the solver to handle.

4 In the table, enter the following settings:

Name	Expression	Unit	Description
Si	1-Sw		Ice volume fraction
Qm	por*(mat2.def.rho-mat1.def.rho)*d(sfc(T), TIME)		Mass source for freezing

To account for the mass transfer between liquid water and ice during phase change, the time derivative of the freezing curve must be explicitly evaluated.

MATERIALS

Continue with adding materials for water, ice, and the porous material. The material properties are populated later, after the physics is set up. Then COMSOL automatically detects which material properties are required.

Water

- 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 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 Select Domain 1 only.

HEAT TRANSFER IN POROUS MEDIA (HT)

Fluid 1

In the **Model Builder** window, under **Component 1 (comp1)** > **Heat Transfer in Porous Media (ht)** > **Porous Medium 1** 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 $\alpha_1 \rightarrow_2$ text field, type Sw.

- 4 In the $L_{1 \rightarrow 2}$ text field, type lambda.
- 5 Locate the **Phase 1** section. From the **Material, phase 1** list, choose **Ice (mat2)**.
- 6 Locate the **Phase 2** section. From the **Material, phase 2** list, choose **Water (mat1)**.

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**.

The initial temperature is 8°C. This is also the ambient temperature and groundwater temperature.


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_0 .

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_0 .


Heat Flux: Surface

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Heat Flux**.
- 2 In the **Settings** window for **Heat Flux**, type Heat Flux: Surface in the **Label** text field.
- 3 Locate the **Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.
- 4 In the h text field, type 5.
- 5 In the T_{ext} text field, type T_0 .
- 6 Select Boundary 4 only.

For the cooling pipes, apply a heat flux boundary condition with a high heat transfer coefficient and a cooling temperature of -30°C. To ensure a smooth transition from the initial temperature to the cooling temperature, which improves solver stability, use a step function to decrease the pipe temperature from T_0 to T_c .


DEFINITIONS

Step 1 (step1)

- 1 In the **Definitions** toolbar, click  **More Functions** and choose **Step**.
- 2 In the **Settings** window for **Step**, locate the **Parameters** section.
- 3 In the **Location** text field, type 4.
- 4 In the **From** text field, type T0.
- 5 In the **To** text field, type Tc.
- 6 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type 6.



HEAT TRANSFER IN POROUS MEDIA (HT)

Heat Flux: Pipes

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Heat Flux**.
- 2 In the **Settings** window for **Heat Flux**, type Heat Flux: Pipes in the **Label** text field.
- 3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Pipes**.
- 4 Locate the **Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.
- 5 In the h text field, type 180.
- 6 In the T_{ext} text field, type step1 ($t[1/h]$).

The heat transfer setup has been completed. Proceed with the poroelasticity multiphysics coupling.

ADD PHYSICS

- 1 In the **Physics** toolbar, click  **Add Physics** to open the **Add Physics** window.
- 2 Go to the **Add Physics** window.
- 3 In the tree, select **Structural Mechanics** > **Poroelasticity** > **Poroelasticity, Solid**.
- 4 Click the **Add to Component 1** button in the window toolbar.
- 5 In the **Physics** toolbar, click  **Add Physics** to close the **Add Physics** window.

SOLID MECHANICS (SOLID)


Gravity 1

In the **Physics** toolbar, click  **Global** and choose **Gravity**.


Fixed Constraint 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Fixed Constraint**.
- 2 Select Boundary 3 only.

Roller 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Roller**.
- 2 Select Boundaries 1, 2, 5, and 38 only.

Rigid Connector 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Rigid Connector**.
- 2 In the **Settings** window for **Rigid Connector**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Pipes**.

Rigid means the pipes can move as a whole, but its shape stays fixed.

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**, locate the **Gravity Effects** section.
- 3 Select the **Include gravity** checkbox.
- 4 Click to expand the **Discretization** section. From the **Pressure** list, choose **Linear**.

Darcy's law only applies to the liquid water in the pores. If a pore is filled with ice, no flow can occur there. This is ensured with the following step.

Porous Matrix 1


- 1 In the **Model Builder** window, under **Component 1 (comp1)** > **Darcy's Law (dl)** > **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 $\text{por} * S_w$.

Initial Values 1


- 1 In the **Model Builder** window, under **Component 1 (comp1)** > **Darcy's Law (dl)** click **Initial Values 1**.
- 2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
- 3 Click the **Hydraulic head** button.

Add the mass source term to account for liquid water turning into ice.

Mass Source 1

- 1 In the **Physics** toolbar, click  **Domains** and choose **Mass Source**.
- 2 Select Domain 1 only.
- 3 In the **Settings** window for **Mass Source**, locate the **Mass Source** section.
- 4 In the Q_m text field, type Q_m .

Hydraulic Head 1



- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Hydraulic Head**.
- 2 Select Boundary 1 only.
- 3 In the **Settings** window for **Hydraulic Head**, locate the **Hydraulic Head** section.
- 4 In the H_0 text field, type 10[cm].

Hydraulic Head 2

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Hydraulic Head**.
- 2 Select Boundary 38 only.

ADD MULTIPHYSICS

Now set up the remaining couplings by linking the Darcy velocity to the heat transfer interface to capture convection, and by adding the thermal expansion of ice.

- 1 In the **Physics** toolbar, click  **Add Multiphysics** to open the **Add Multiphysics** window.
- 2 Go to the **Add Multiphysics** window.
- 3 In the tree, select **No Predefined Multiphysics Available for the Selected Physics Interfaces**.
- 4 Find the **Select the physics interfaces you want to couple** subsection. In the table, clear the **Couple** checkbox for **Darcy's Law (dl)**.
- 5 In the tree, select **Structural Mechanics > Thermal-Structure Interaction > Thermal Stress, Solid**.
- 6 Click the **Add to Component** button in the window toolbar.
- 7 In the **Physics** toolbar, click  **Add Multiphysics** to close the **Add Multiphysics** window.

MULTIPHYSICS

Thermal Expansion 1 (te1)

- 1 In the **Settings** window for **Thermal Expansion**, locate the **Thermal Expansion Properties** section.
- 2 From the **Input type** list, choose **Thermal strain**.
- 3 From the dL list, choose **User defined**. In the associated text field, type 0.09*Si*por.

HEAT TRANSFER IN POROUS MEDIA (HT)

Fluid 1


- 1 In the **Model Builder** window, under **Component 1 (comp1) > Heat Transfer in Porous Media (ht) > Porous Medium 1** click **Fluid 1**.
- 2 In the **Settings** window for **Fluid**, locate the **Heat Convection** section.

3 From the **u** list, choose **Total Darcy velocity field (dl/porous1)**.

Define the permeability using a van Genuchten like relationship for the dependence of the permeability on the water saturation.

DEFINITIONS

Permeability

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

2 In the **Settings** window for **Analytic**, type Permeability in the **Label** text field.

3 In the **Function name** text field, type kappa.

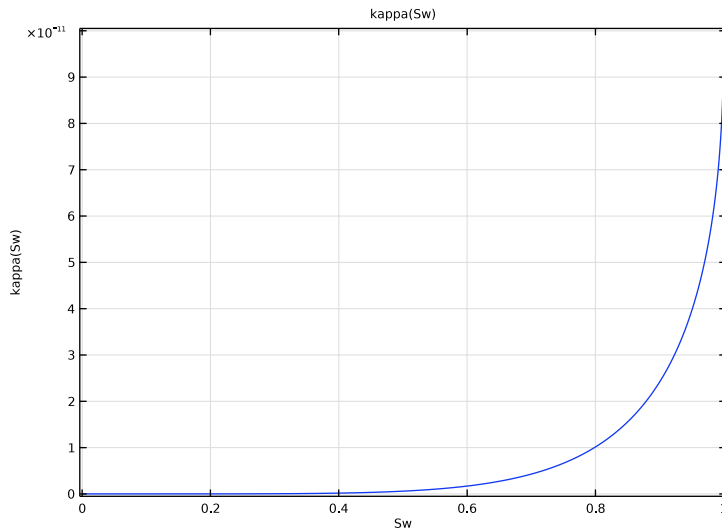
4 Locate the **Definition** section. In the **Expression** text field, type $\text{kappa}0 * \text{Sw} * (1 - (1 - \text{Sw}^{(1/n)})^n)^2$.

5 In the **Arguments** text field, type Sw.

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

Name	Expression
n	0.46


7 Click  **Plot**.



The physics setup is complete, and the material properties can now be specified.

MATERIALS

Porous Material 1 (pmat1)

- 1 In the **Model Builder** window, expand the **Component 1 (comp1) > Materials > Porous Material 1 (pmat1)** node, then 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 (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	2800	kg/m ³	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	2.4	W/(m·K)	Basic
Heat capacity at constant pressure	Cp	800	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 **Homogenized Properties** section.
- 3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	2800	kg/m ³	Basic
Young's modulus	E	50 [MPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3		Young's modulus and Poisson's ratio

Property	Variable	Value	Unit	Property group
Biot-Willis coefficient	alphaB	0.95	l	Poroelastic material
Permeability	kappa_iso ; kappaii = kappa_iso, kappaij = 0	kappa (Sw)	m ²	Basic

Water (mat1)

- 1 In the **Model Builder** window, click **Water (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
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	0.6	W/(m·K)	Basic
Density	rho	1000	kg/m ³	Basic
Heat capacity at constant pressure	Cp	4182	J/(kg·K)	Basic
Dynamic viscosity	mu	1e-3	Pa·s	Basic

Ice (mat2)


- 1 In the **Model Builder** window, click **Ice (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	2.14	W/(m·K)	Basic
Density	rho	920	kg/m ³	Basic
Heat capacity at constant pressure	Cp	2060	J/(kg·K)	Basic

MESH 1

Free Triangular 1

- 1 In the **Mesh** toolbar, click  **More Generators** and choose **Free Triangular**.
- 2 Select Boundary 2 only.


- 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** checkbox. In the associated text field, type 2.5.
- 6 Click  **Build All**.

STUDY I



Use a stationary study step to compute the system with the pipe cooling turned off, so that gravity loads, groundwater flow, and heat balance are established. The results from this stationary study then serve as the initial conditions for the subsequent time-dependent simulation of the freezing process.

- 1 In the **Model Builder** window, click **Study I**.
- 2 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 3 Clear the **Generate default plots** checkbox.

Step 2: Time Dependent

- 1 In the **Study** toolbar, click  **Time Dependent**.
- 2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 3 From the **Time unit** list, choose **d**.
- 4 In the **Output times** text field, type range (0, 1, 30).

Step 1: Stationary

- 1 In the **Model Builder** window, click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
- 3 Select the **Modify model configuration for study step** checkbox.
- 4 In the tree, select **Component 1 (comp1) > Heat Transfer in Porous Media (ht) > Heat Flux: Pipes**.
- 5 Click  **Disable**.
- 6 In the **Study** toolbar, click  **Get Initial Value**.

Before computing the study, which takes about 1.5 hours to solve, prepare the solver and create a plot to monitor the freezing process during computation.


RESULTS

In the **Model Builder** window, expand the **Results** node.

Cut Plane 1

- 1 In the **Model Builder** window, expand the **Results > Datasets** node.
- 2 Right-click **Results > Datasets** and choose **Cut Plane**.
- 3 In the **Settings** window for **Cut Plane**, locate the **Plane Data** section.
- 4 From the **Plane** list, choose **XZ-planes**.
- 5 In the **Y-coordinate** text field, type 20.

Ice Saturation and Stress

- 1 In the **Results** toolbar, click  **2D Plot Group**.
- 2 In the **Settings** window for **2D Plot Group**, type Ice Saturation and Stress in the **Label** text field.
- 3 Click to expand the **Plot Array** section. From the **Array type** list, choose **Square**.

Surface 1

- 1 Right-click **Ice Saturation and Stress** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type Si.
- 4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Pelagic**.

Streamline 1


- 1 In the **Model Builder** window, right-click **Ice Saturation and Stress** and choose **Streamline**.
- 2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1) > Darcy's Law > Velocity and pressure > dl.u,...,dl.w - Total Darcy velocity field (spatial frame)**.
- 3 Locate the **Streamline Positioning** section. From the **Entry method** list, choose **Coordinates**.
- 4 In the **x** text field, type 0.
- 5 In the **y** text field, type range (0, 2, 20).
- 6 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Type** list, choose **Arrow**.
- 7 Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

Surface 2

- 1 Right-click **Ice Saturation and Stress** and choose **Surface**.

- 2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component I (comp1) > Solid Mechanics > Stress > solid.misesGp - von Mises stress - N/m²**.
- 3 Locate the **Coloring and Style** section. From the **Color table** list, choose **Prism**.

Ice Saturation and Stress

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

STUDY I

Step 2: Time Dependent


- 1 In the **Model Builder** window, under **Study I** click **Step 2: Time Dependent**.
- 2 In the **Settings** window for **Time Dependent**, click to expand the **Results While Solving** section.
- 3 Select the **Plot** checkbox.
- 4 From the **Update at** list, choose **Time steps taken by solver**.
- 5 In the table, enter the following settings:

Plot group	Plot window
Ice Saturation and Stress	Graphics

Solver Configurations


In the **Model Builder** window, expand the **Study I > Solver Configurations** node.

Solution 1 (sol1)

- 1 In the **Model Builder** window, expand the **Study I > Solver Configurations > Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.
- 2 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
- 3 From the **Steps taken by solver** list, choose **Intermediate**.
- 4 Select the **Initial step** checkbox. In the associated text field, type 1 [h] to capture the initial temperature drop in and around the pipes.
- 5 In the **Study** toolbar, click  **Compute**.

RESULTS

Freezing Front and Deformation

- 1 In the **Results** toolbar, click  **3D Plot Group**.

- 2 In the **Settings** window for **3D Plot Group**, type Freezing Front and Deformation in the **Label** text field.

Isosurface 1

- 1 Right-click **Freezing Front and Deformation** and choose **Isosurface**.
- 2 In the **Settings** window for **Isosurface**, locate the **Expression** section.
- 3 In the **Expression** text field, type Si.
- 4 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.
- 5 In the **Levels** text field, type 0.5.
- 6 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
- 7 From the **Color** list, choose **White**.
- 8 Clear the **Color legend** checkbox.

Transparency 1

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


Surface 1

- 1 In the **Model Builder** window, right-click **Freezing Front and Deformation** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Cut Plane 1**.
- 4 From the **Solution parameters** list, choose **From parent**.
- 5 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1) > Solid Mechanics > Displacement > solid.disp - Displacement magnitude - m**.
- 6 Locate the **Expression** section. From the **Unit** list, choose **cm**.
- 7 Locate the **Coloring and Style** section. From the **Color table** list, choose **Prism**.

Freezing Front and Deformation

In the **Model Builder** window, click **Freezing Front and Deformation**.

Streamline Surface 1

- 1 In the **Freezing Front and Deformation** toolbar, click  **More Plots** and choose **Streamline Surface**.
- 2 In the **Settings** window for **Streamline Surface**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Cut Plane 1**.
- 4 From the **Solution parameters** list, choose **From parent**.

- 5 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1) > Darcy's Law > Velocity and pressure > dl.u,....,dl.w - Total Darcy velocity field (spatial frame)**.
- 6 Locate the **Streamline Positioning** section. From the **Entry method** list, choose **Coordinates**.
- 7 In the **x** text field, type 0.
- 8 In the **y** text field, type 20.
- 9 In the **z** text field, type range (0, 1, 18).
- 10 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Type** list, choose **Arrow**.
- 11 From the **Color** list, choose **Custom**.
- 12 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button and select a dark gray tone.

Surface 2

- 1 Right-click **Freezing Front and Deformation** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type z.
- 4 Locate the **Coloring and Style** section. Clear the **Color legend** checkbox.
- 5 From the **Coloring** list, choose **Gradient**.
- 6 From the **Bottom color** list, choose **Custom**.
- 7 Click **Define custom colors**.
- 8 Set the RGB values to 128, 128, and 128, respectively.
- 9 Click **Add to custom colors**.
- 10 Click **Show color palette only** or **OK** on the cross-platform desktop.

Selection 1

- 1 Right-click **Surface 2** and choose **Selection**.
- 2 Select Boundaries 3, 5, and 38 only.

Surface 3

In the **Model Builder** window, right-click **Freezing Front and Deformation** and choose **Surface**.

Selection 1



- 1 In the **Model Builder** window, right-click **Surface 3** and choose **Selection**.

- 2 In the **Settings** window for **Selection**, locate the **Selection** section.
- 3 From the **Selection** list, choose **Pipes**.

Material Appearance 1


- 1 In the **Model Builder** window, right-click **Surface 3** 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 **Steel**.

Freezing Front and Deformation

- 1 In the **Model Builder** window, under **Results** click **Freezing Front and Deformation**.
- 2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.
- 3 Clear the **Plot dataset edges** checkbox.
- 4 In the **Freezing Front and Deformation** toolbar, click  **Plot**.
- 5 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 6 Click to expand the **Quality** section. From the **Recover** list, choose **Within domains**.
- 7 In the **Freezing Front and Deformation** toolbar, click  **Plot**.

Plot the ice thickness at different times.

Ice Saturation at Different Times

- 1 In the **Results** toolbar, click  **2D Plot Group**.
- 2 In the **Settings** window for **2D Plot Group**, type Ice Saturation at Different Times in the **Label** text field.
- 3 Locate the **Plot Array** section. From the **Array type** list, choose **Square**.
- 4 From the **Displacement** list, choose **Absolute**.
- 5 In the **Row displacement** text field, type -25.
- 6 In the **Column displacement** text field, type 25.

Surface 1

- 1 Right-click **Ice Saturation at Different Times** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type Si.
- 4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Pelagic**.

Solution Array 1

- 1 Right-click **Surface 1** and choose **Solution Array**.


- 2 In the **Settings** window for **Solution Array**, locate the **Data** section.
- 3 From the **Time selection** list, choose **Interpolated**.
- 4 In the **Times (d)** text field, type range (5, 5, 30).
- 5 Locate the **Plot Array** section. From the **Array shape** list, choose **Square**.

Add the time stamps in the next step.



Ice Saturation at Different Times

In the **Model Builder** window, under **Results** click **Ice Saturation at Different Times**.

Table Annotation 1

- 1 In the **Ice Saturation at Different Times** toolbar, click  **More Plots** and choose **Table Annotation**.
- 2 In the **Settings** window for **Table Annotation**, locate the **Data** section.
- 3 From the **Source** list, choose **Local table**.
- 4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
10	0	5d
35	0	10d
60	0	15d
10	-25	20d
35	-25	25d
60	-25	30d



- 5 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.
- 6 From the **Anchor point** list, choose **Upper middle**.
- 7 In the **Ice Saturation at Different Times** toolbar, click  **Plot**.
- 8 Click the  **Zoom Extents** button in the **Graphics** toolbar.

Compare with [Figure 2](#). Note that the thinnest part of the ice wall is in upstream direction close to the surface. To compute the ice thickness in this area at different times, follow the steps below.



Surface 1

- 1 In the **Results** toolbar, click  **More Datasets** and choose **Surface**.
- 2 Select Boundary 3 only.

Cut Line 2D I

- 1 In the **Results** toolbar, click  **Cut Line 2D**.
- 2 In the **Settings** window for **Cut Line 2D**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Surface I**.
- 4 Locate the **Line Data** section. In row **Point 1**, set **x** to -20.
- 5 In row **Point 2**, set **x** to -10.
- 6 In row **Point 1**, set **y** to 5.
- 7 In row **Point 2**, set **y** to 5.
- 8 Select the **Additional parallel lines** checkbox.
- 9 In the **Distances** text field, type 10 20 30.
- 10 Click  **Plot**. The lines are at the bottom surface in upstream direction.
- 11 Click to expand the **Advanced** section.

Line Integration I

- 1 In the **Results** toolbar, click  **More Derived Values** and choose **Integration > Line Integration**.
- 2 In the **Settings** window for **Line Integration**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Cut Line 2D I**.
- 4 From the **Time selection** list, choose **Interpolated**.
- 5 In the **Times (d)** text field, type range (10, 5, 30).
- 6 Locate the **Multiple Curves** section. Select the **Evaluate each line separately** checkbox.
- 7 Locate the **Expressions** section. Click  **Clear Table**.

Because of the soil freezing characteristic curve, a completely frozen state is never reached, and a small fraction of liquid water always remains. To evaluate when the frozen wall can be considered stable, different criteria can be used depending on soil type, flow and requirements. For example, an ice saturation of about 0.9 or, alternatively, the -5°C isotherm can be applied as indicators of a stable ice wall.

- 8 In the table, enter the following settings:

Expression	Unit	Description
$1*(S_i > 0.9)$	m	Ice thickness based on ice saturation
$1*(T < -5[\text{degC}])$	m	Ice thickness based on temperature isotherm

- 9 Click  **Evaluate**.

TABLE 1

- 1 Go to the **Table 1** window.
- 2 Click the **Table Graph** button in the window toolbar.

RESULTS

Table Graph 1


- 1 In the **Settings** window for **Table Graph**, locate the **Data** section.
- 2 From the **Plot columns** list, choose **Manual**.
- 3 In the **Columns** list, choose **Ice thickness based on ice saturation (m)**,
Ice thickness based on ice saturation (m), Distance=10,
Ice thickness based on ice saturation (m), Distance=20, and
Ice thickness based on ice saturation (m), Distance=30.
- 4 Locate the **Coloring and Style** section. From the **Width** list, choose **2**.
- 5 Click to expand the **Legends** section. Select the **Show legends** checkbox.
- 6 From the **Legends** list, choose **Manual**.
- 7 In the table, enter the following settings:

Legends
y=5m
y=15m
y=25m
y=35m

Table Graph 2

- 1 Right-click **Results > ID Plot Group 4 > Table Graph 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Table Graph**, locate the **Data** section.
- 3 In the **Columns** list, choose **Ice thickness based on temperature isotherm (m)**,
Ice thickness based on temperature isotherm (m), Distance=10,
Ice thickness based on temperature isotherm (m), Distance=20, and
Ice thickness based on temperature isotherm (m), Distance=30.
- 4 Locate the **Legends** section. Clear the **Show legends** checkbox.
- 5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.
- 6 From the **Color** list, choose **Cycle (reset)**.

Ice Thickness

- 1** In the **Model Builder** window, under **Results** click **ID Plot Group 4**.
- 2** In the **Settings** window for **ID Plot Group**, type Ice Thickness 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 Ice thickness based on saturation (solid) and temperature isotherm (dotted).
- 5** Locate the **Legend** section. From the **Position** list, choose **Upper left**.
- 6** In the **Ice Thickness** toolbar, click  **Plot**.