18
Freeze pipes in tunnel construction [ULT]

This tutorial illustrates change in coupling of groundwater flow and thermal flow as a result of ground freezing. A tunnel is constructed with the use of freeze pipes. By first installing freeze pipes in the soil, the soil freezes and becomes watertight so that tunnel construction can take place. This method of construction requires a lot of energy for the cooling of the soil, so by being able to model the cooling behaviour while groundwater flow is present an optimal freezing system can be designed.

Objectives

- Modelling soil freezing, coupling between thermal flow and groundwater flow
- Modelling unfrozen water content.
- Using the command line for structure definition.

Geometry

In this tutorial a tunnel with a radius of 3.0 m will be constructed in a 30 m deep soil layer. A groundwater flow from left to right is present, influencing the thermal behaviour of the soil. First the soil will be subjected to the low temperatures of the freeze pipes, and once the soil has frozen sufficiently, tunnel construction can take place. The latter is not included in this tutorial.

Because groundwater flow causes an asymmetric temperature distribution, the whole geometry needs to be modelled, where in previous examples only half of the geometry was sufficient. The geometry of the project is shown in
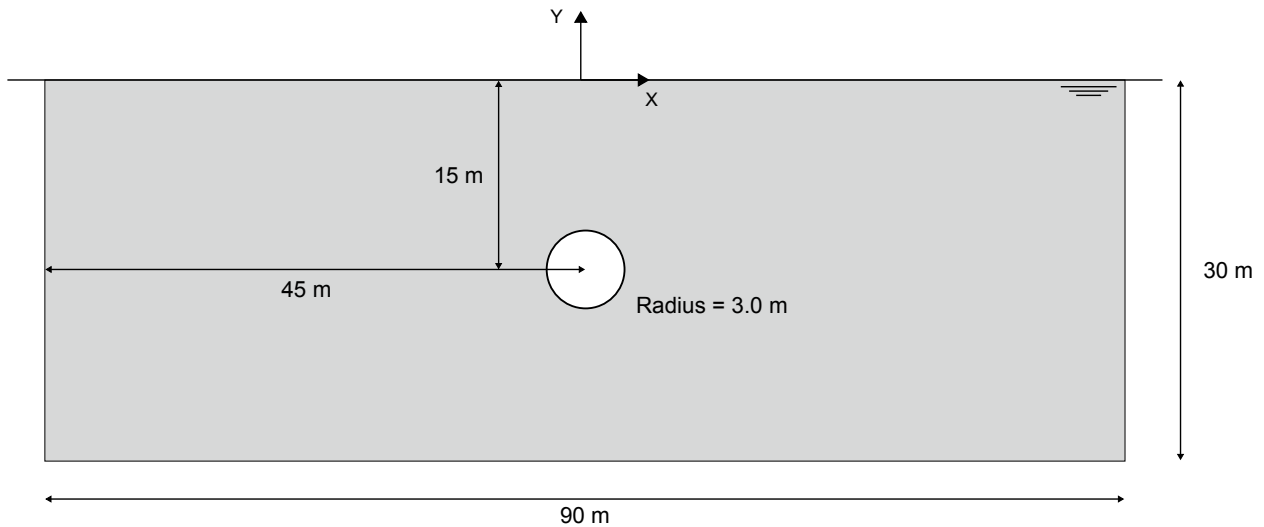

Figure 243: Geometry of the project

18.1 Create new project

1. Start the Input program and select **Start a new project** from the **Quick start** dialog box.
2. In the **Project** tabsheet of the **Project properties** window, enter an appropriate title.
3. In the **Model** tabsheet, the default options for **Model** and **Elements** are used for this project. Also the default options for the units are used in this tutorial. Note that the unit of **Mass** is set automatically to **tonnes**.
4. Set the model **Contour** to x_{min} = -45 m, x_{max} = 45 m, y_{min} = -30 m and y_{max} = 0 m.
5. In the **Constants** tabsheet, set T_{water} and T_{ref} to 283 K, other constants keep their default values. A description of constants can be found in the Reference Manual.
6. Click **OK** to close the **Project properties** window.

18.2 Define the soil stratigraphy

To define the soil stratigraphy:

1. Click the **Create borehole** button and create a borehole at x = 0.
   The **Modify soil layers** window pops up.
2. Create a single soil layer with top level at 0 m and bottom level at -30 m. Set the head at ground level (0 m).

18.3 Create and assign material data sets

1. Click the **Materials** button in the **Modify soil layers** window.
2. Define a data set for soil with the parameters given in Table 39 (on page 273), for the **General Parameters** and **Groundwater** tabsheets.

Freeze pipes in tunnel construction [ULT]

Create and assign material data sets

Table 39: Material properties

Parameter	Name	Sand	Unit
General			
Soil model	-	Mohr-Coulomb	-
Drainage type	-	Drained	-
Unsaturated unit weight	γ_{unsat}	18	kN/m^3
Saturated unit weight	γ_{sat}	18	kN/m^3
Initial void ratio	e_{init}	0.5	-
Mechanical			
Young's modulus	E'_{ref}	$100 \cdot 10^3$	kN/m^2
Poisson's ratio	ν	0.3	-
Cohesion	c'_{ref}	0	kN/m^2
Friction angle	φ'	37	°
Dilatancy angle	ψ	0	°
Groundwater			
Classification type	-	Standard	-
Soil - class	-	Medium	-
Flow parameters - Use defaults	-	None	-
Horizontal permeability	k_x	1	m/day
Vertical permeability	k_y	1	m/day
Thermal			
Specific heat capacity	c_s	860	kJ/t/K
Thermal conductivity	λ_s	$4 \cdot 10^{-3}$	kW/m/K
Soil density	ρ_s	2.6	t/m^3
Thermal expansion type	-	Isotropic	-
Thermal expansion	α_{sv}	$0.015 \cdot 10^{-3}$	1/K

Thermal			
Unfrozen water saturation method	-	User defined (see table below)	-

Interfaces			
Strength determination	-	Rigid	-
Thermal resistance	$R_{thermal}$	0	m^2K/kW

Initial			
K_0 determination	-	Automatic	-

To model the amount of (fluid) water available to flow through the soil at certain temperatures, a curve for unfrozen water content needs to be determined by defining a table with values for unfrozen water content at certain temperatures. The same curve can be applied in other projects, hence the table can be saved and loaded into the soil properties of other projects. For more information, refer to the Reference Manual.

3. Click the **Thermal** tab. Enter the values as given in .

Table 40: Input for unfrozen water content curve for sand

#	Temperature [K]	Unfrozen water content [-]
1	273.0	1.00
2	272.0	0.99
3	271.6	0.96
4	271.4	0.90
5	271.3	0.81
6	271.0	0.38
7	270.8	0.15
8	270.6	0.06
9	270.2	0.02
10	269.5	0.00

4. Select the option **User defined** from the drop down menu for **Unfrozen water content** at the bottom of the tabsheet.

5. Add rows to the table by clicking the **Add row** button . Complete the data using the values given in .

6. Enter the values for **Interfaces** and **Initial** tabsheets as given in .

7. Click **OK** to close the dataset.

8. Assign the material dataset to the soil layer.

> **Note:**
>
> The table can be saved by clicking the **Save** button in the table. The file must be given an appropriate name. For convenience, save the file in the same folder as the project is saved.

18.4 Define the structural elements

The freeze pipes are modelled by defining lines with a length similar to the freeze pipe diameter (10 cm), containing a convective boundary condition. For simplicity, in this tutorial only 12 cooling elements are defined, while in reality more elements may be implemented in order to achieve a sufficient share of frozen soil.

1. Proceed to **Structures mode** .
2. Click the **Create line** button �’ in the side toolbar.
3. Click the command line and type

    ```
    line 0 -12.05 0 -11.95
    ```

    Press **Enter** to create the first freezing pipe. For more information regarding command line, see Reference Manual .
4. The remaining freeze pipes will be created by copying the first freeze pipe 12 times. To do so select the line representing the freeze pipe that was just created.
5. From the side toolbar select the **Create array** option ⣿. Set the **Array pattern** to **Polar** as we want to create a circle of freeze pipes, the **Center point** is (x y) = (0 -15), the **Total number of items** is 12 (the original plus 11 copies) and finally the **Angle to fill** must be set to 360 degrees as we want to create a full circle. Press OK to create the 11 additional freeze pipes.
6. Multi select the 12 lines representing the freeze pipes using the **Select lines** option ▨ from the side toolbar.
7. Right click the selected lines and select **Thermal flow BC** ⟋ to create the thermal flow boundary conditions for the freeze pipes.
8. For the selected freeze pipes, in the **Selection explorer** expand the subtree for the **ThermalFlowBC**.
9. The **Behaviour** is set to **Convection**, the T_{fluid} to 250 K and the **Transfer coefficient** to 1.0 kW/m^2/K.
10. Click the **Create line** button �’ in the side toolbar.
11. Select the **Create thermal flow BC** option ⟋ in the expanded menu. In the drawing area create a thermal boundary condition along the perimeter of the model, hence from (x y) = (-45 0) to (45 0), (45 -30), (-45 -30) and back to (-45 0).
12. Select the four boundaries just have been created and right-click on them. From the menu that pops up select **Create** and **Create groundwater flow BC** to add groundwater flow boundary conditions to the thermal flow boundary conditions.

PLAXIS 2D allows different types of **Thermal** boundary conditions to be applied. In this tutorial the freeze pipes will be modelled as convective boundary conditions.

1. Multi select the created boundaries.
2. For the **ThermalFlowBC**, set the **Behaviour** to **Temperature** and T_{ref} to 283 K.

To assign the groundwater boundary conditions, the following steps are followed:

1. Multi select the top and bottom boundary.
2. For the **GWFlowBC**, set the **Behaviour** to **Closed**.
3. Select the left boundary, set the **Behaviour** to **Inflow** with a q_{ref} value of 0.1 m/day.
4. The right boundary has the default behaviour of **Seepage**.

The tunnel is created with the help of the **Tunnel designer**. Because deformations are not considered in this calculation, there is no need to assign a plate material to the tunnel. The generated tunnel will only be used for generating a more dense and homogeneous mesh around the freezing pipes. The tunnel will not be activated during any calculation phase, but PLAXIS 2D will detect the line elements and will generate the mesh according to these elements. Changing the coarseness factor of the pipe elements will cause a denser, but not a more homogeneous mesh.

1. Click the **Create tunnel** button in the side toolbar and click on (0 -18) in the drawing area.
2. The option **Circular** is selected for **Shape type**. Note that the default option is **Free**.
3. The default option of **Define whole tunnel** is used in this example.
4. Proceed to the **Segments** tab and set **Radius** to 3 m to the two multi selected segments.
5. Click on **Generate** to generate the defined tunnel in the model. Close the **Tunnel designer** window.

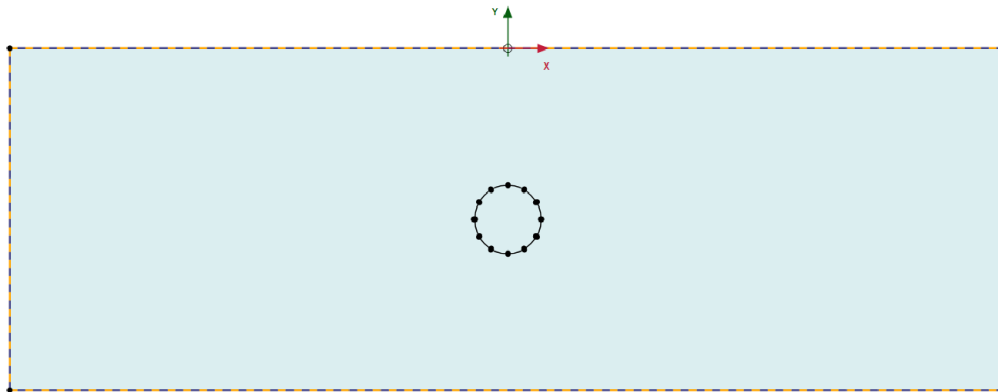The geometry of the model is shown in Figure 244 (on page 276) .

Figure 244: Geometry of the model

18.5 Generate the mesh

1. Proceed to the **Mesh mode**.
2. Click the **Generate mesh** button to generate the mesh. The default element distribution of **Medium** is used for this example.
3. Click the **View mesh** button to view the mesh.

   The resulting mesh is shown in Figure 245 (on page 277):
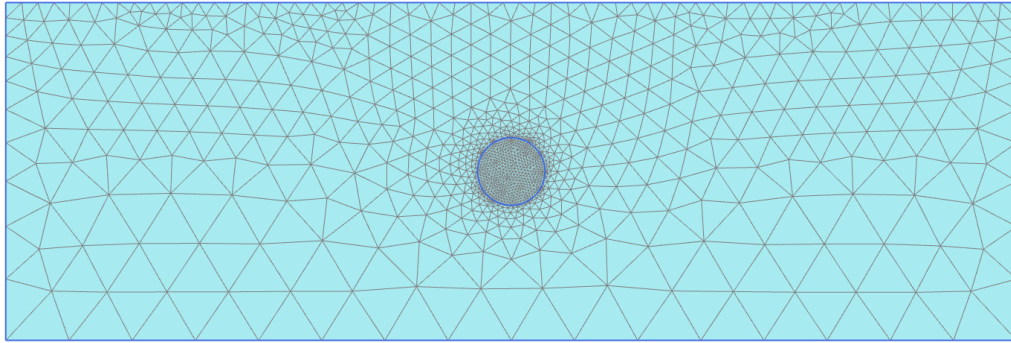
Figure 245: The generated mesh

4. Click the **Close** tab to close the Output program.

18.6 Define and perform the calculation

The calculations for this tutorial are carried out in the **Flow only mode**.

18.6.1 Initial phase

1. Click on the **Staged construction** tab to proceed with the definition of the calculation phases.
2. Double click on **Initial phase** in the **Phases explorer**.
3. In the **Phases** window select the **Flow only** option from the **Calculation type** drop-down menu.
4. Choose the **Earth gradient** option for the **Thermal calculation type**.
5. In the **Staged construction** activate the **ThermalFlow** under the **Model conditions** subtree and set the value for T_{ref} to 283 K, h_{ref} to 0 m and 0 K/m for the **Earth gradient**.
6. The model for initial phase is shown in .
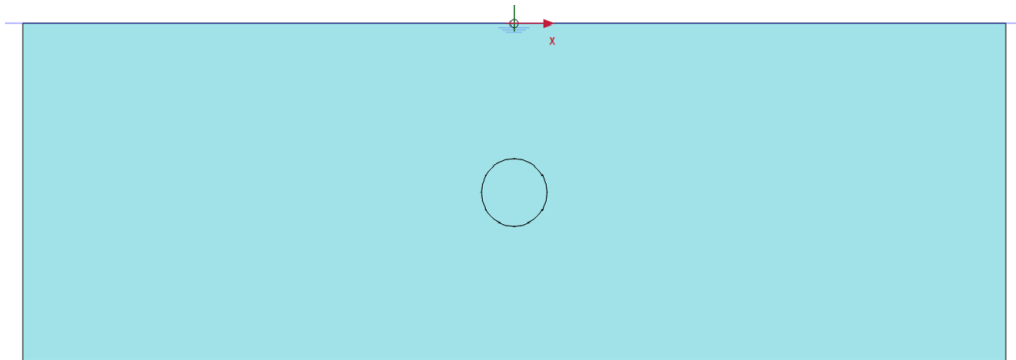
Figure 246: Initial phase

18.6.2 Phase 1: Transient calculation

1. Click the **Add phase** button ⊞ to create a new phase.
2. Double click the new phase in the **Phases explorer**.
3. In the **Phases** window, enter an appropriate name for the phase ID (e.g. `Transient calculation`).
4. Set **Transient groundwater flow** as the option for the **Pore pressure calculation type**.
5. Set **Transient thermal flow** as the option for the **Thermal calculation type**.
6. Set **Time interval** to 180 days and the **Max number of steps stored** to 100. This is to be able to view intermediate time steps after the calculation.
7. In **Staged construction** mode, activate all the thermal boundary conditions by clicking the check box for the **Thermal flow BCs** in the **Model explorer**.
8. In the **Model explorer**, activate the four groundwater flow boundary conditions corresponding to the left, top, right and bottom boundary conditions in the **Groundwater flow BCs** subtree.

18.6.3 Execute the calculation

Before starting the calculation it is suggested that you select nodes or stress points for a later generation of curves.

1. Click the **Select points for curves** button in the side toolbar. Select at least the node in the middle of the model and some characteristic points for curves (for example between two freezing pipes).
2. Click the **Calculate** button to calculate the project.
3. After the calculation has finished, save the project by clicking the **Save** button .

18.7 Results

Interesting results from this calculation can be the point in time when there is no groundwater flow in between two freezing pipes, groundwater flow over the whole model and temperature distribution for both steady state and transient calculations.

To view the results in the Output program:

1. Click the **View calculation results** button on the toolbar.
2. Select the menu **Stresses** > **Heat flow** > **Temperature**.
3. Figure 247 (on page 279) shows the spatial distribution of the temperature for transient calculation in the final step. Note that the element contours have been switched off to better show that the temperature of the soil inside the tunnel is below freezing.
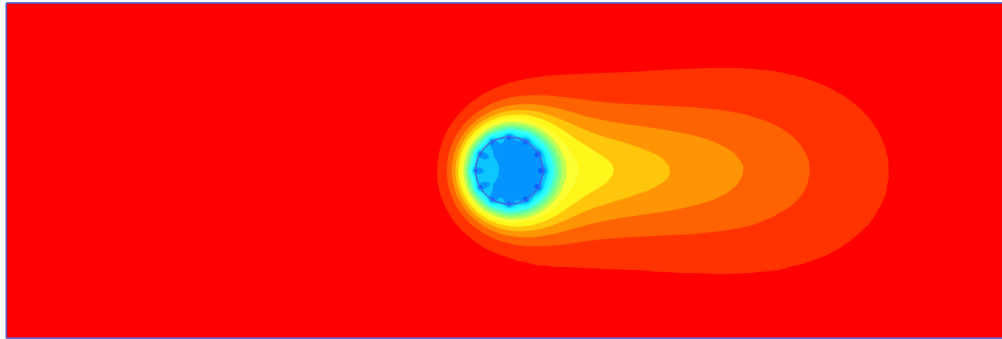
Figure 247: Temperature distribution for transient phase

1. Select the menu **Stresses** > **Groundwater flow** > **|q|**.
2. Select the menu **View** > **Arrows** or click the corresponding button ➔ in the toolbar to display the results arrows.

In the Output program, it is possible to view the results for the intermediate saved steps. More information is available in the Reference Manual . It is possible to view the progression of the freezing of the tunnel.

Figure 248 (on page 279) shows the distribution of the of groundwater flow field for an intermediate step for the transient calculation (around 38 days).
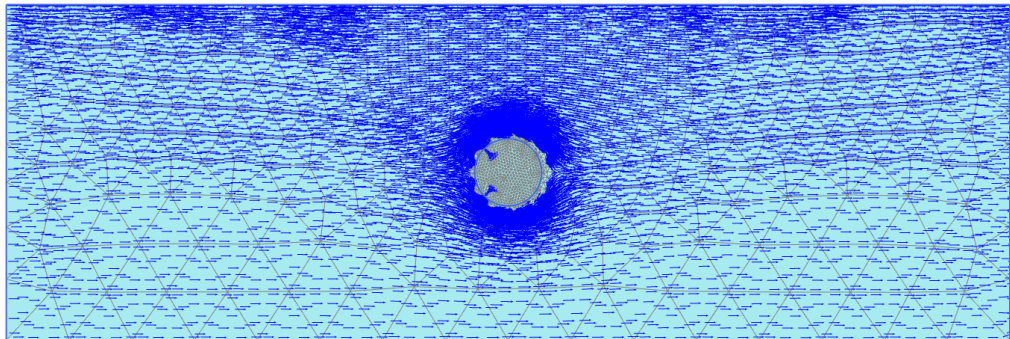

Figure 248: Groundwater flow field for transient phase for an intermediate step (t_{approx} 38 days)

Figure 249 (on page 280) shows the groundwater flow field for the last time step for the transient flow calculation. Here it is clearly noticeable that the entire tunnel area is frozen and no flow occurs.
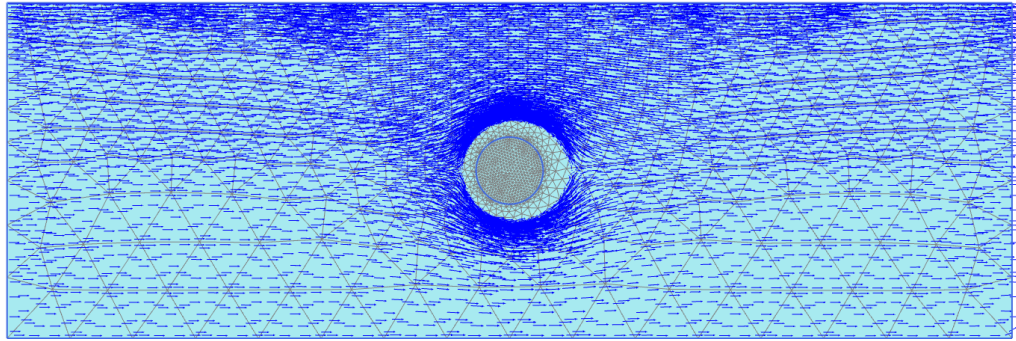
Figure 249: Groundwater flow field after 180 days

Figure 250 (on page 280) shows the change of temperature in time for the point in the middle of the tunnel. It can be seen that the temperature drops quite fast until about 273 K when the pore water starts to change from water to ice. During this process the temperature remains almost constant and only until after all pore water has turned to ice (at t = 122 s) the ice temperature drops further.
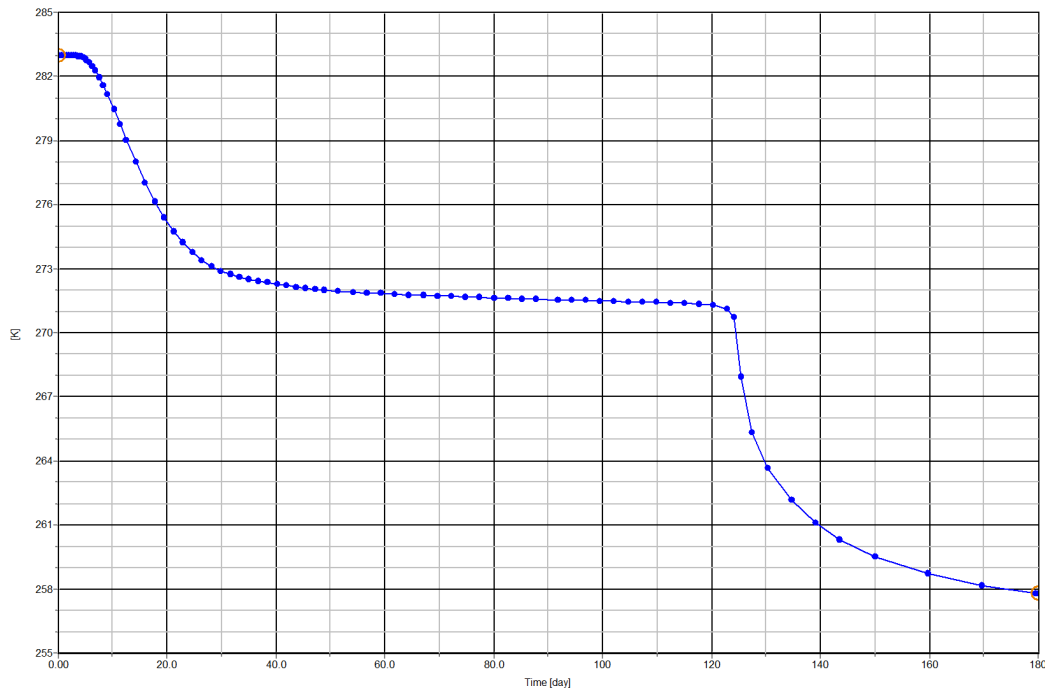

Figure 250: Temperature drop in time at the center of the tunnel