16 FREEZE PIPES IN TUNNEL CONSTRUCTION

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.

In this tutorial a tunnel with a radius of 3.0 m will be constructed in a 30 m deep soil layer (Figure 16.1). 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.

Objectives:

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

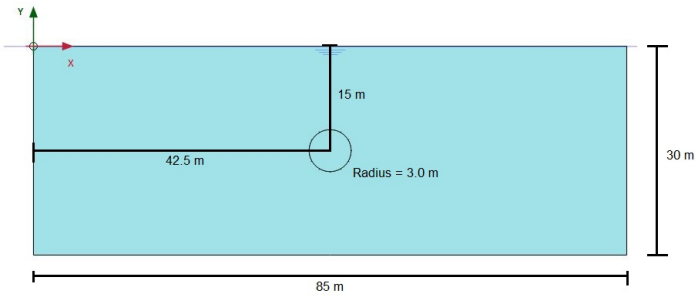

Figure 16.1 Input geometry

16.1 INPUT

General settings

- Start the input program and select *Start a new project* from the *Quick select* dialog box.
- In the *Project* tabsheet of the *Project properties* window, enter an appropriate title.
- 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*.
- Set the model dimensions to x_{min} = 0.0 m, x_{max} = 85.0 m, y_{min} = -30.0 m and y_{max} =

0.0 m.

- 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. Click *Ok* to exit the *Project properties* window.

Definition of soil stratigraphy

Create a borehole at x = 0. The *Modify soil layers* window pops up.

- Create a single soil layer with top level at 0.0 m and bottom level at -30.0 m. Set the head at ground level (0.0 m).

Click the *Materials* button in the *Modify soil layers* window.

- Define a data set for soil with the parameters given in Table 16.1, for the *General*, *Parameters* and *Groundwater* tabsheets.

Table 16.1 Soil properties of the sand

Parameter	Name	Sand	Unit
General			
Material model	*Model*	Mohr-Coulomb	-
Type of material behaviour	*Type*	Drained	-
Soil unit weight above phreatic level	γ_{unsat}	18.0	kN/m^3
Soil unit weight below phreatic level	γ_{sat}	18.0	kN/m^3
Initial void ratio	e_{init}	0.5	-
Parameters			
Young's modulus	E'	$1 \cdot 10^5$	kN/m^2
Poisson's ratio	ν	0.3	-
Cohesion	c_{ref}'	0.0	kN/m^2
Angle of internal friction	ϕ'	37.0	°
Dilatancy angle	ψ	0.0	°
Groundwater			
Data set	-	Standard	-
Type	-	Medium	-
Horizontal permeability	k_x	1.00	m/day
Vertical permeability	k_y	1.00	m/day
Change of permeability	c_k	$1.0 \cdot 10^{15}$	-
Thermal			
Specific heat capacity	c_s	860	kJ/t/K
Thermal conductivity	λ_s	$4.0 \cdot 10^{-3}$	kW/m/K
Soil density	ρ_s	2.6	t/m^3
X-component of thermal expansion	α_x	$5.0 \cdot 10^{-6}$	1/K
Y-component of thermal expansion	α_y	$5.0 \cdot 10^{-6}$	1/K
Z-component of thermal expansion	α_z	$5.0 \cdot 10^{-6}$	1/K
Unfrozen water content	-	User defined (Table 16.2)	-
Interfaces			
Interface strength	-	Rigid	-
Thermal resistance	R	0	m^{2}K/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 Section 6.1.4 of the Reference Manual.

- Click the *Thermal* tab. Enter the values as given in the Table 16.1.
- Select the option *User defined* from the drop down menu for *Unfrozen water content* at the bottom of the tabsheet.
- Add rows to the table by clicking the *Add row* button to introduce a new row in the table. Complete the data using the values given in the Table 16.2.
- Enter the values for *Interfaces* and *Initial* tabsheets as given in Table 16.1.
- Click *OK* to close the dataset.
- Assign the material dataset to the soil layer.

Hint: 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.

Table 16.2 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

Definition of 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.

- Proceed to *Structures* mode.
- Click the *Create line* button in the side toolbar.
- Click the command line and type "line 45.141 -13.475 45.228 -13.425". Press <Enter> to create the first freezing pipe. For more information regarding command line, see Section 3.6 of the Reference Manual.
- Similarly create the remaining freeze pipes according to Table 16.3.
- Multi select the created lines using the *Select lines* from the side toolbar.
- Right click the selected lines and select *Thermal flow BC* to create the thermal flow boundary conditions for the freeze pipes.

Hint: A file containing the commands for the definition of the lines, is available in the PLAXIS knowledge base(http://kb.plaxis.nl/search/site/LineCoordinatesCommands.txt). This can be downloaded and copied in the *Commands runner*, to get the pipes.

- For the selected freeze pipes, in the *Selection explorer* expand the subtree for the *ThermalFlowBC*.
- The *Behaviour* is set to *Convection*, the T_{fluid} to 250 K and the *Transfer coefficient* to 1.0 kW/m^2/K.

Table 16.3 Coordinates of the end points of the freezing pipes, modelled as lines

Line number	X_{point1}	Y_{point1}	X_{point2}	Y_{point2}
1	45.141	-13.475	45.228	-13.425
2	44.025	-12.359	44.075	-12.272
3	42.500	-11.950	42.500	-11.850
4	40.975	-12.359	40.925	-12.272
5	39.859	-13.475	39.772	-13.425
6	39.450	-15.000	39.350	-15.000
7	39.859	-16.525	39.772	-16.575
8	40.975	-17.641	40.925	-17.728
9	42.500	-18.050	42.500	-18.150
10	44.025	-17.641	44.075	-17.728
11	45.141	-16.525	45.228	-16.575
12	45.550	-15.000	45.650	-15.000

Click the *Create line* button in the side toolbar.

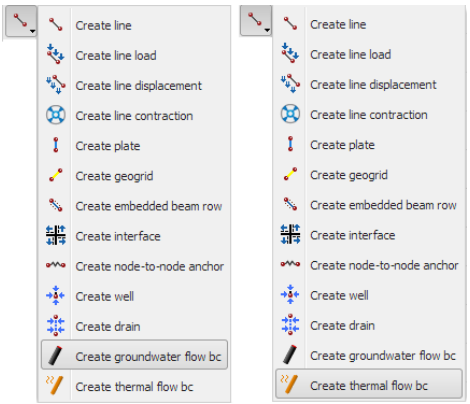

Figure 16.2 The *Create thermal and groundwater flow bc* option in the *Create line* menu

Select the *Create thermal flow BC* option in the expanded menu. In the draw area create a thermal boundary condition from (0.0 0.0) to (85.0 0.0) (Figure 16.2).

Click the *Create line* button in the side toolbar, select the *Create groundwater flow BC* option in the expanded menu. In the draw area create a groundwater flow boundary condition from (0.0 0.0) to (85.0 0.0) (Figure 16.2).

- Similarly follow the above steps to create thermal and groundwater flow boundary for the following lines (85.0 0.0) to (85.0 -30.0); (85.0 -30.0) to (0.0 -30.0) and finally (0.0 -30.0) to (0.0 0.0).

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

- Multi select the created boundaries.
- 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:

- Multi select the top and bottom boundary.
- For the *GWFlowBC*, set the *Behaviour* to *Closed*.
- Select the left boundary, set the *Behaviour* to *Inflow* with a q_{ref} value of 0.1 m/day.
- 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 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.

- Click the *Create tunnel* button in the side toolbar and click on (42.5 -18) in the draw area.
- The option *Circular* is selected for *Shape type*. Note that the default option is *Free*.
- The default option of *Define whole tunnel* is used in this example.
- Proceed to the *Segments* tab and set *Radius* to 3.0 m to the two multi selected segments.
- 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 16.3.

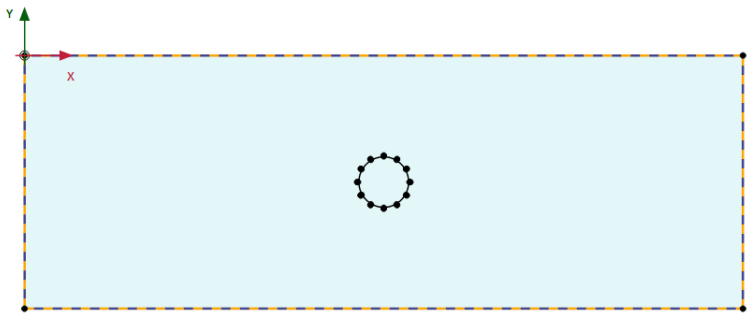

Figure 16.3 Geometry of the model

16.2 MESH GENERATION

- Proceed to the *Mesh* mode.
- Click the *Generate mesh* button. The default element distribution of *Medium* is used for this example.
- View the generated mesh. The resulting mesh is shown in Figure 16.4.
- Click on the *Close* tab to close the Output program.

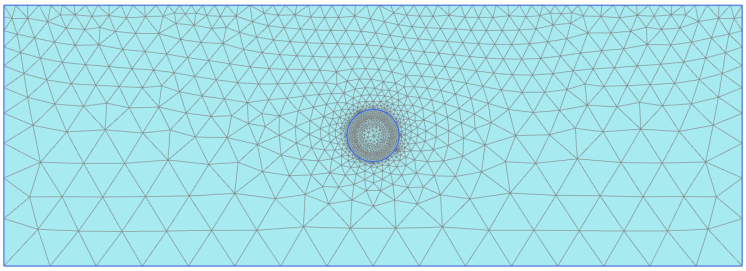

Figure 16.4 The generated mesh

16.3 CALCULATIONS

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

Initial phase

- Proceed to *Staged construction* mode.
- Double click on *Initial phase* in the *Phases explorer*.
- In the *Phases* window select the *Flow only* option from the *Calculation type* drop-down menu.
- Choose the *Earth gradient* option for the *Thermal calculation type*.
- 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*.

Phase 1

- Add a new phase.
- Double click the new phase in the *Phases explorer*.
- In the *Phases* window, enter an appropriate name for the phase ID (For eg: "Transient calculation").
- Set *Transient groundwater flow* as the option for the *Pore pressure calculation type*.
- Set *Transient thermal flow* as the option for the *Thermal calculation type*.
- 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.

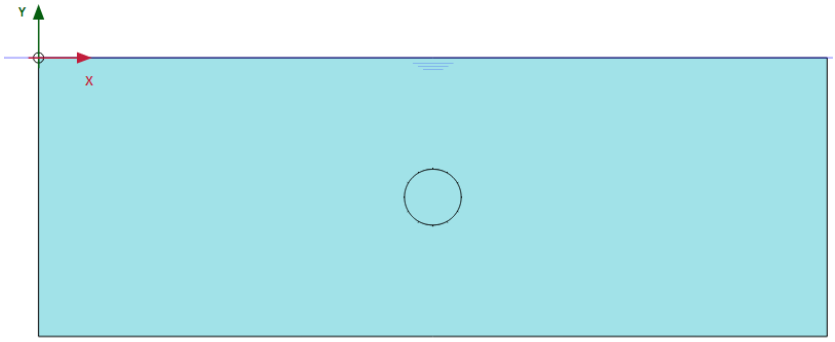

Figure 16.5 Initial phase

- In *Staged construction* mode, activate all the thermal boundary conditions by clicking the check box for the *Thermal flow BCs* in the *Model explorer*.
- 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.
- In the *Model explorer*, deactivate the *ThermalFlow* condition under the *Model conditions* subtree.

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

Click the *Select points for curves* button in the side toolbar. Select some characteristic points for curves (for example between two freezing pipes).

Calculate the project by clicking the *Calculate* button in the *Staged construction* mode.

Save the project after the calculation has finished.

16.4 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:

Click the *View calculation results* button on the toolbar.

- From the *Stresses* menu, select *Heat flow* → *Temperature*.
- Figure 16.6 shows the spatial distribution of the temperature for transient calculation in the final step.
- From the *Stresses* menu, select *Groundwater flow* → $|q|$.

Select the *Arrows* option in the *View* menu 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.

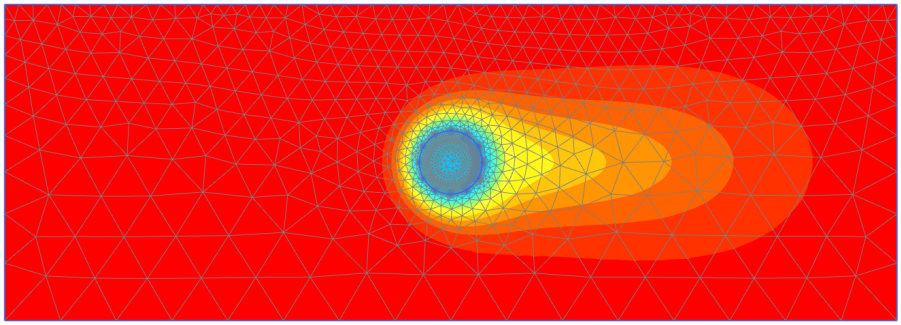

Figure 16.6 Temperature distribution for transient phase

More information is available in Section 8.4.9 of the Reference Manual. It is possible to view the progression of the freezing of the tunnel.

- Figure 16.7 shows the distribution of the of groundwater flow field for an intermediate step for the transient calculation (around 80 days).
- Figure 16.8 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.

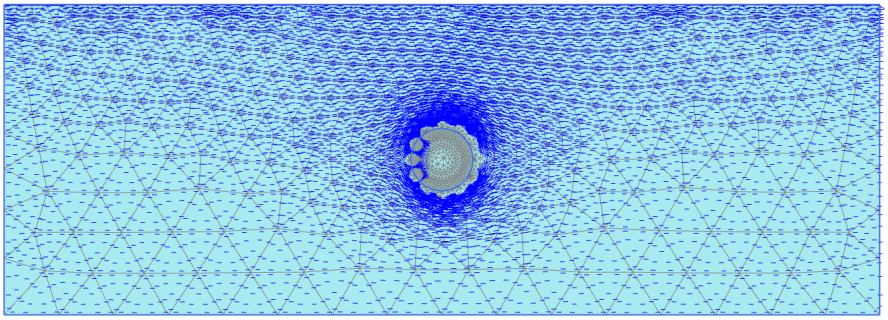

Figure 16.7 Groundwater flow field for transient phase for an intermediate step (t $\approx$ 38 days)

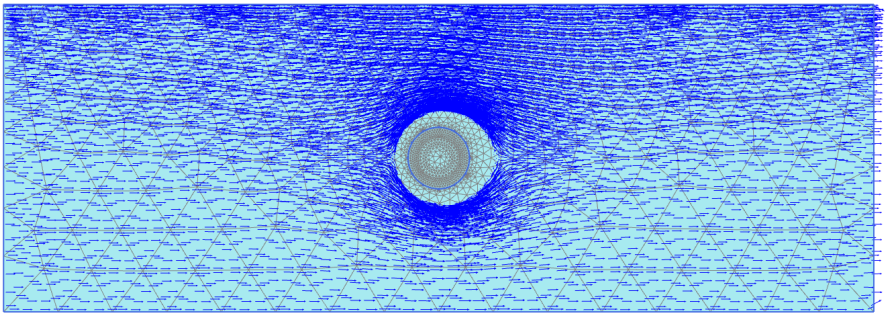

Figure 16.8 Groundwater flow field after 180 days