PLAXIS

CONNECT Edition V21.01

PLAXIS 2D - Tutorial Manual - Thermal expansion of a navigable lock [ULT]

Last Updated: March 16, 2021

1

Thermal expansion of a navigable lock [ULT]

A navigable lock is temporarily 'empty' due to maintenance. After some time there is significant increase of the air temperature, which causes thermal expansion of the inner side of the lock, while the soil-side of the concrete block remains relatively cold. This leads to backward bending of the wall and, consequently, to increased lateral stress in the soil behind the wall and increased bending moments in the wall itself.

Objectives

This example demonstrates the use of the **Thermal** module to analyse this kind of situations.

- Defining a thermal temperature function
- Use of thermal expansion
- Performing a fully coupled analysis for THM calculation

Geometry

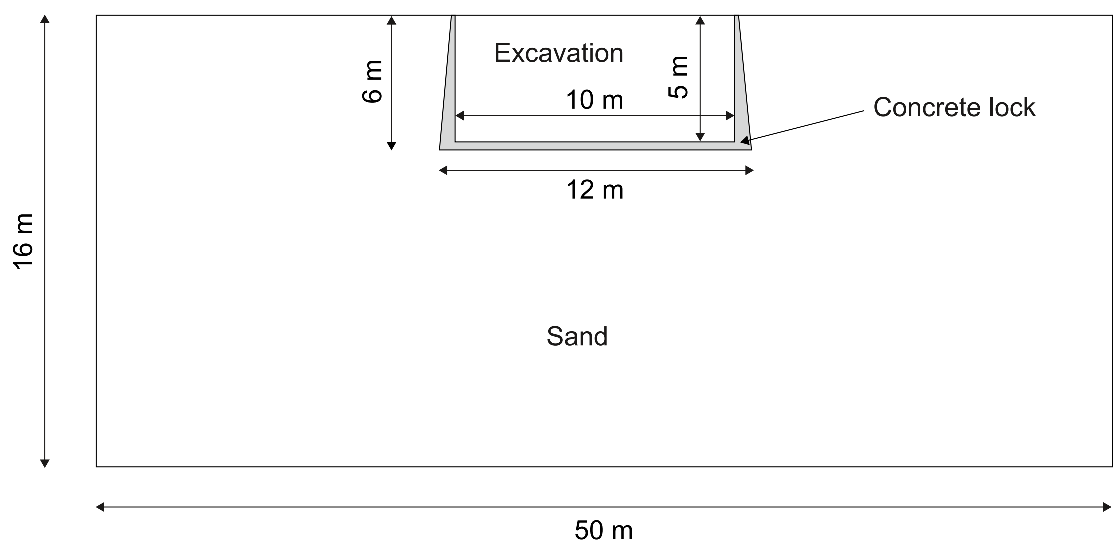

Figure 1: Geometry of the project

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.
4. Set the model **Contour** to x_{min} = 0 m, x_{max} = 25 m, y_{min} = -16 m and y_{max} = 0 m.
5. Click **OK** to close the **Project properties** window.

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 0.0 m and bottom level at -16 m. Set the head at -4 m.

Create and assign material data sets

Two data sets need to be created; one for the sand layer and one for the concrete block.

1. Open the **Material sets** window.
2. Define a data set for the **Sand** layer with the parameters given in Table 39 (on page 3), for the **General**, **Parameters**, **Groundwater**, **Thermal** and **Initial** tabsheets.
3. Create another dataset for **Concrete** according to the Table 39 (on page 3).
4. Assign the material dataset **Sand** to the borehole soil layer.

Table 1: Material properties

Parameter	Name	Sand	Concrete	Unit
General				
Material model	-	HS small	Linear elastic	-
Drainage type	-	Drained	Non-porous	-
Soil unit weight above phreatic level	γ_{unsat}	20	24	kN/m^3
Soil unit weight below phreatic level	γ_{sat}	20	-	kN/m^3
Initial void ratio	e_{init}	0.5	0.5	-
Parameters				
Young's modulus	E'	-	$25\cdot 10^6$	kN/m^2
Poisson's ratio	ν	-	0.15	-

Parameter	Name	Sand	Concrete	Unit
Secant stiffness in standard drained triaxial test	E_{50}^{ref}	$40 \cdot 10^3$	-	kN/m^2
Tangent stiffness for primary oedometer loading	E_{oed}^{ref}	$40 \cdot 10^3$	-	kN/m^2
Unloading / reloading stiffness	E_{ur}^{ref}	$1.2 \cdot 10^5$	-	kN/m^2
Power for stress-level dependency of stiffness	m	0.5	-	-
Cohesion	c_{ref}'	2	-	kN/m^2
Friction angle	φ'	32	-	°
Dilatancy angle	ψ	2	-	°
Shear strain at which $G_s = 0.722\ G_0$	$\gamma_{0.7}$	$0.1 \cdot 10^{-3}$	-	-
Shear modulus at very small strains	G_0^{ref}	$80 \cdot 10^3$	-	kN/m^2
Groundwater				
Data set	-	USDA	-	-
Model	-	Van Genuchten	-	-
Soil - Type	-	Sand	-	-
Flow parameters - Use defaults	-	From data set	-	-
Thermal				
Specific heat capacity	c_s	860	900	kJ/t/K
Thermal conductivity	λ_s	$4 \cdot 10^{-3}$	$1 \cdot 10^{-3}$	kW/m/K
Solid thermal expansion	-	Linear	Linear	-
Soil density	ρ_s	2.6	2.5	t/m^3
X-component of thermal expansion	α_x	$0.5 \cdot 10^{-6}$	$0.01 \cdot 10^{-3}$	1/K
Y-component of thermal expansion	α_y	$0.5 \cdot 10^{-6}$	$0.01 \cdot 10^{-3}$	1/K
Z-component of thermal expansion	α_z	$0.5 \cdot 10^{-6}$	$0.01 \cdot 10^{-3}$	1/K
Interfaces				
Interface strength	-	Rigid	Manual	-

Parameter	Name	Sand	Concrete	Unit
Strength reduction factor	R_{inter}	1.0	0.67	-
Initial				
K_0determination	-	Automatic	Automatic	-

Define the structural elements

The lock will be modelled as a concrete block during the staged construction.

1. Proceed to **Structures** mode.
2. Click the **Create soil polygon** button in the side toolbar and select the **Create soil polygon** option.
3. Define the lock in the drawing area by clicking on (0 -5), (5 -5), (5 0), (5.5 0), (6 -6), (0 -6) and (0 -5).

Note:

The **Snapping options** can be selected, and the **Spacing** can be set to 0.5 to easily create the polygon.

The **Concrete** material will be assigned later in the **Staged construction**.

4. Click the **Create line** button in the side toolbar.
5. Select the **Create thermal flow bc** option in the expanded menu.
6. Create thermal boundaries at vertical boundaries and the bottom boundary (X_{min}, X_{max} and Y_{min}).
7. The vertical boundaries have the default option of **Closed** for the **Behaviour**.
8. Select the bottom boundary, in the **Selection explorer** set the **Behaviour** to **Temperature**.
9. Set the reference temperature, T_{ref} to 283.4 K.

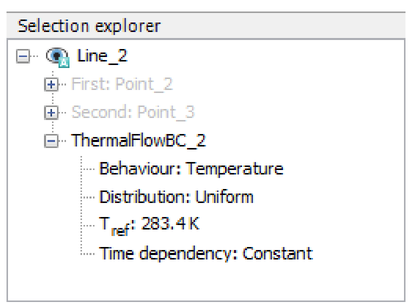

*Figure 2: Thermal boundary condition in the **Selection explorer***

The geometry of the model is now complete.

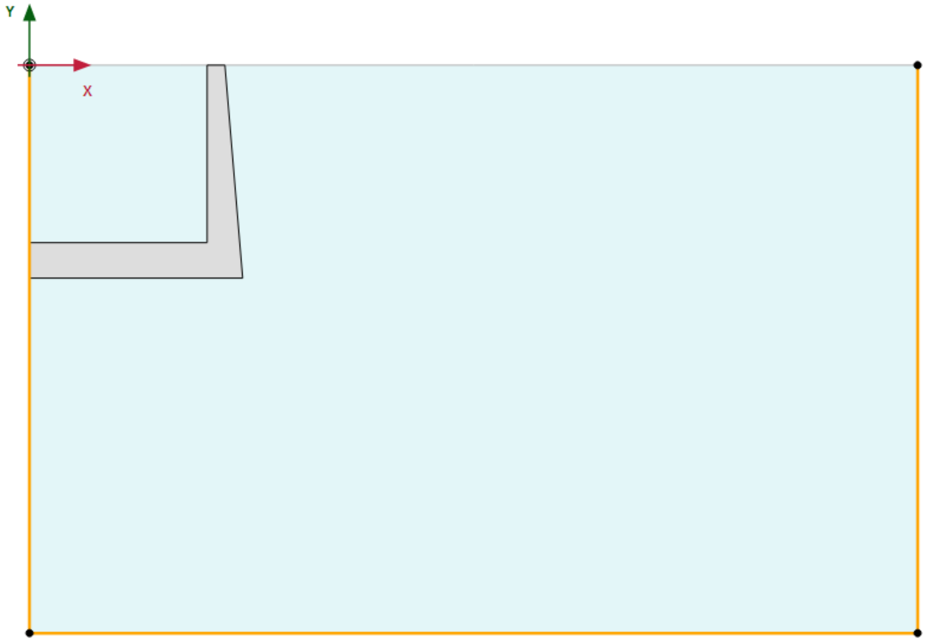

Figure 3: Geometry of the model

Generate the mesh

1. Proceed to the **Mesh** mode.
2. Select the polygon representing the concrete block, and in the **Selection explorer** set the **Coarseness factor** to 0.25.
3. Click the **Generate mesh** button to generate the mesh. The default element distribution of **Medium** is used for this example.
4. Click the **View mesh** button to view the mesh.
   The resulting mesh is shown:

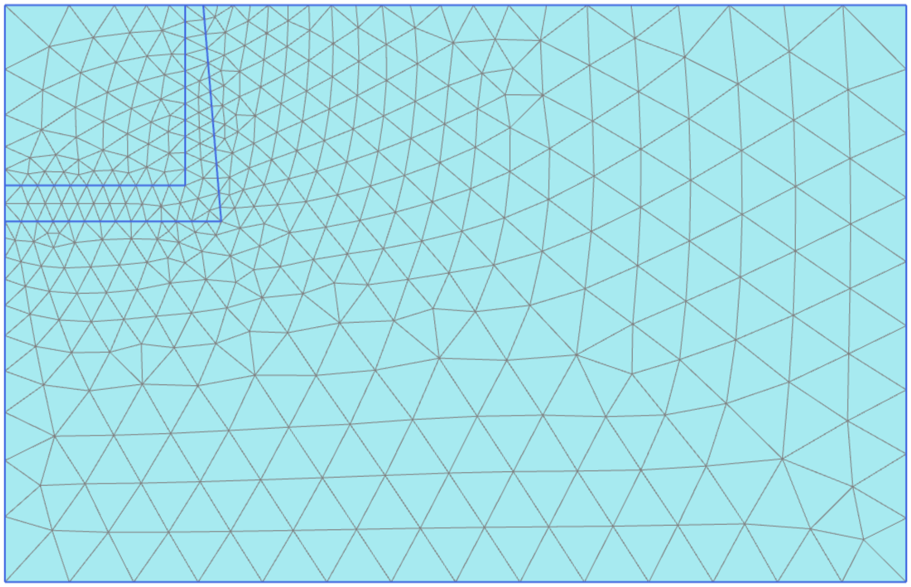

Figure 4: The generated mesh

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

Define and perform the calculation

The calculations for this tutorial is carried out in three phases. The concrete lock is activated in a plastic calculation, after which the temperature increase is defined as a fully coupled flow deformation analysis.

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. The default options for **Calculation type** and **Pore pressure calculation type** are used in this example.
4. Select **Earth gradient** for the **Thermal calculation type** option and close the **Phases** window.
5. In the **Staged construction** activate the **ThermalFlow** under the **Model conditions** subtree and set the value for T_{ref} to 283 K. The default values for h_{ref} and **Earth gradient** are valid.

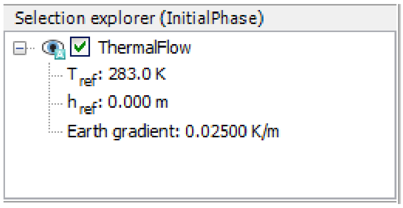

Figure 5: Thermal flow in the Selection explorer

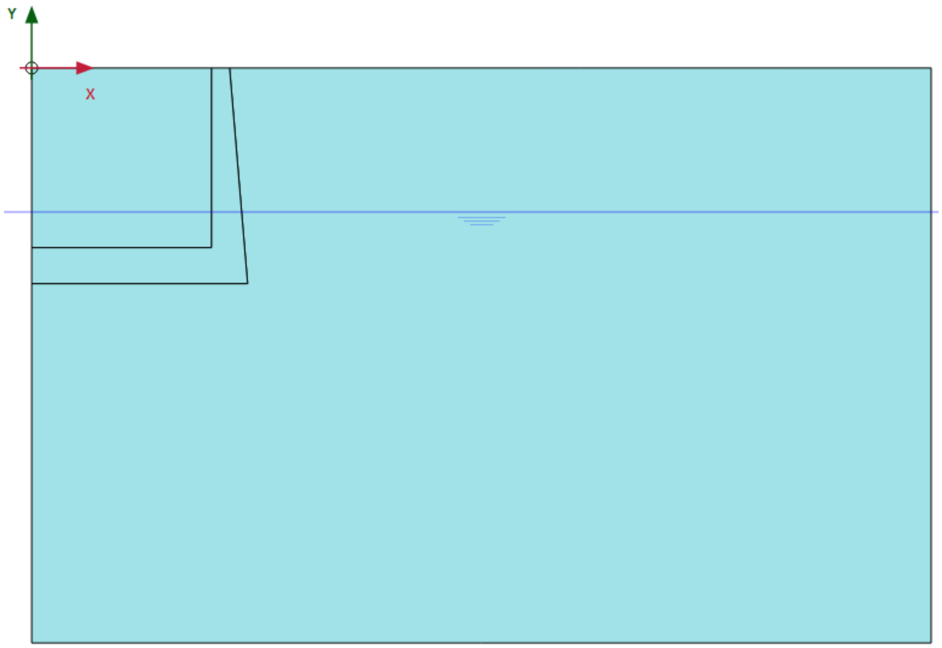

Figure 6: Initial phase

Phase 1: Construction

1. Click the **Add phase** button to create a new phase (Phase_1).
2. Double click on Phase_1 in the **Phases explorer**.
3. In the **Phases** window, enter an appropriate name for the phase ID and select **Steady state groundwater flow** as **Pore pressure calculation type**.
4. Set the **Steady state thermal flow** for the **Thermal calculation type**.
5. Make sure that the **Reset displacements to zero** and **Ignore suction** options are selected.
6. In the **Staged construction** mode, assign the **Concrete** dataset to the created polygon which represents the navigable lock.

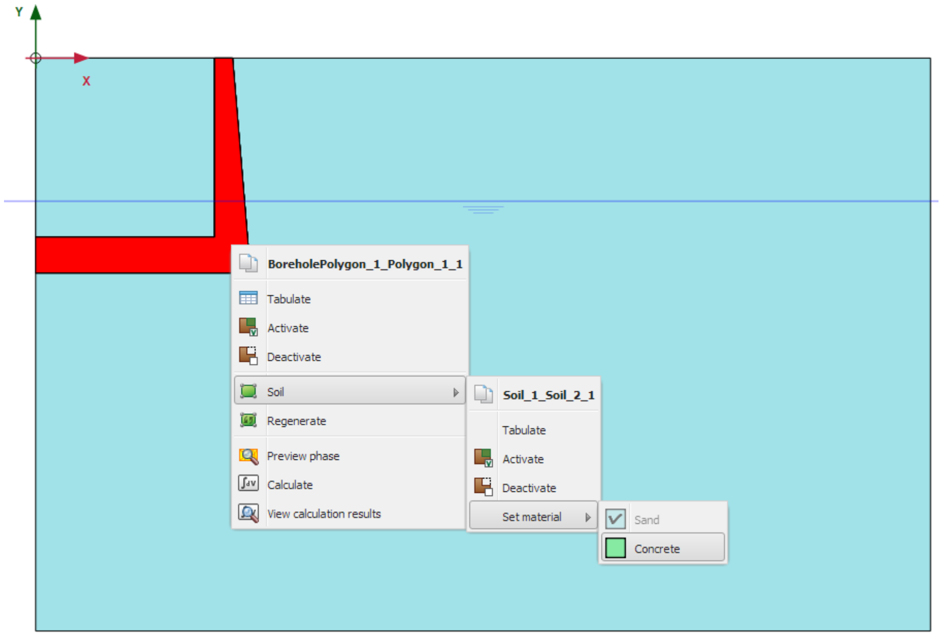

*Figure 7: Assigning **Concrete** soil data set to the navigable lock*

7. Right click the soil cluster which is cut-off by the polygon and select the option **Deactivate** from the appearing menu.
8. In the **Selection explorer**, set the **WaterConditions** of this cluster to **Dry**.
9. In the **Model explorer**, activate all the **Thermal flow boundary conditions**.
10. In the **Model explorer**, activate the **Model conditions** > **Climate** condition.
11. Set the **Air temperature** to 283 K and the **Surface transfer** to 1 kW/m^2/K.
    This will define the thermal conditions at the ground surface and the inside of the lock.

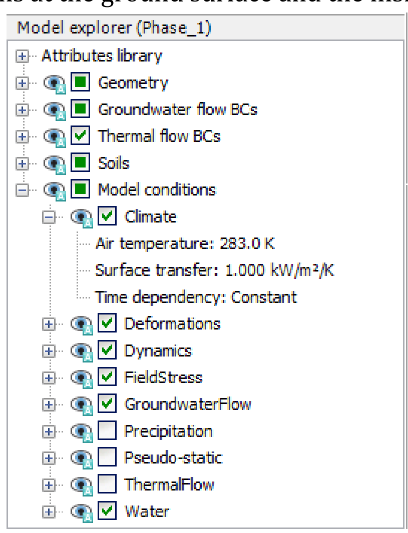

Figure 8: Model conditions for Phase_1

12. Deactivate the **ThermalFlow** option. This is because the thermal flow boundary conditions, including climate condition, are used in a steady state thermal flow calculation, instead of the earth gradient option.

The following figure shows the model at the end of **Phase_1**

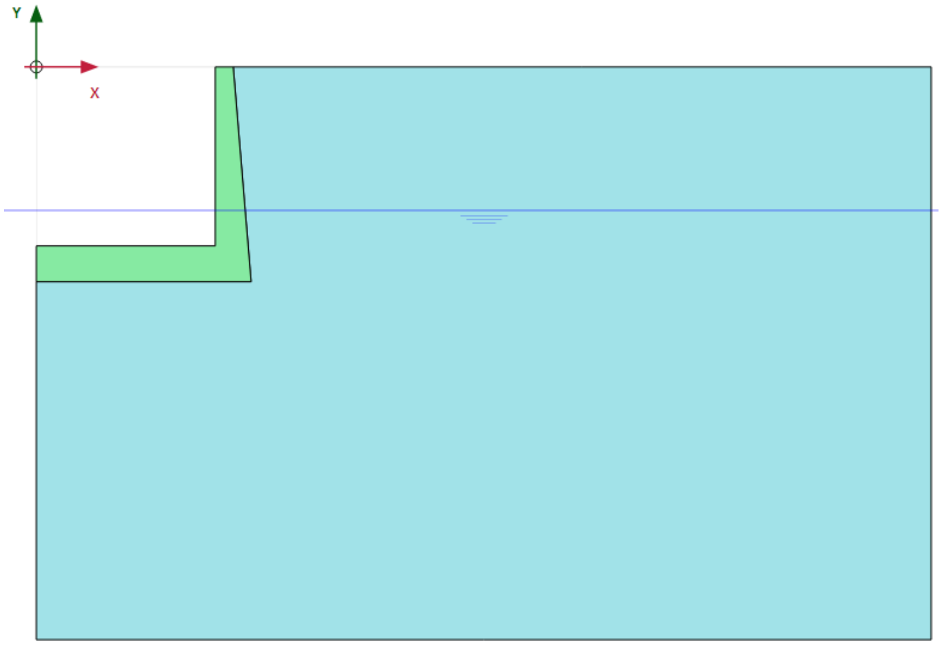

Figure 9: The model at the end of Phase_1

Phase 2: Heating

1. Click the **Add phase** button to create a new phase (Phase_2).
2. Double click on Phase_2 in the **Phases explorer**.
3. Set the **Calculation type** to **Fully coupled flow deformation**.
4. The **Thermal calculation type** is set to **Use temperatures from previous phase**. This is to indicate that temperature needs to be considered and that the initial temperature is taken from the previous phase.
5. The **Time interval** is set to 10 days.
6. Make sure that the **Reset displacements to zero** and **Reset small strain** options are selected in the **Deformation control parameters** subtree. The **Ignore suction** option is unchecked by default.
7. A temperature function is defined for the **Time dependency** in **Climate** which is used for this phase. Follow these steps to create a temperature function.
   a. Right-click the **Thermal functions** option in the **Attributes library** in the **Model explorer** and select **Edit** option in the appearing menu.
   The **Thermal functions** window is displayed.
   b. In the **Temperature functions** tabsheet add a new function by clicking on the corresponding button.
   The new function is highlighted in the list and options to define the function are displayed.
   c. The default option of **Harmonic** is used for this signal.
   d. Assign a value of 15 for the **Amplitude** and 40 days for the **Period.**
   A graph is displayed showing the defined function. Since the time interval of the phase is 10 days, only a quarter of a temperature cycle is considered in this phase, which means that after 10 days the temperature has increased by 15 K.

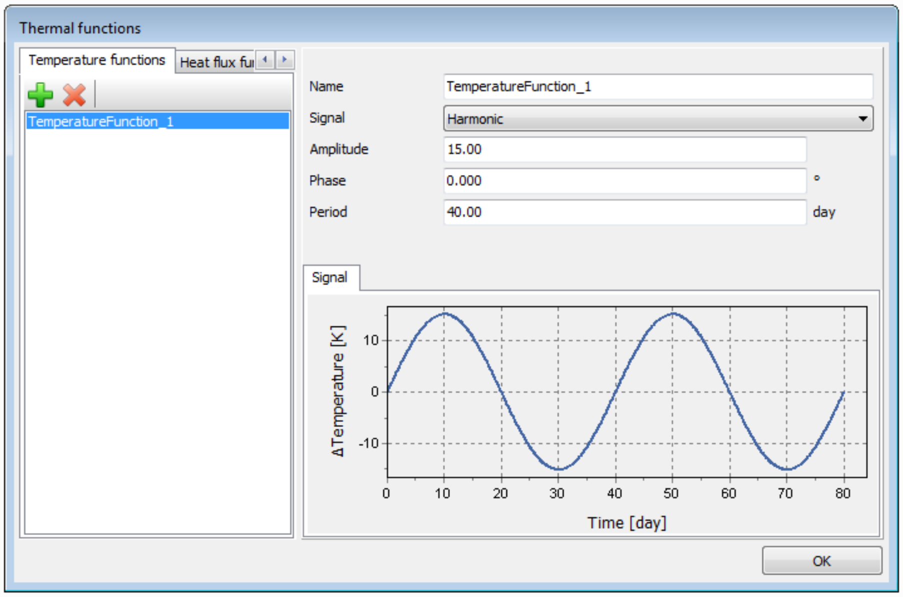

Figure 10: The temperature function

e. Click **OK** to close the **Thermal functions** window.

8. Expand the subtree **Model conditions** in the **Model explorer**.
9. In the **Climate** option, set the **Time dependency** to **Time dependent** and assign the temperature function which was created.

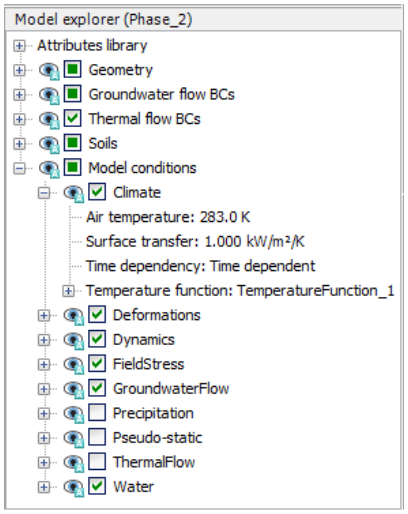

Figure 11: Model conditions for Phase_2

The calculation definition is now complete.

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 and select some characteristic points for curves (for example at the top of the excavation, (5.0, 0.0)).
2. Click the **Calculate** button to calculate the project, a warning regarding different stress type used in the **Fully coupled flow deformation** analysis will appear. This warning appears because the **Fully coupled flow deformation** analysis always calculates with suction while the other calculation types by default do not calculate suction, and mixing phases with and without suction may lead to unexpected results. However, since in this tutorial we are dealing with sand the influence of suction will be very small and thus the warning can be ignored.
3. After the calculation has finished, save the project by clicking the **Save** button.

Results

In the **Phases explorer**, select the **Initial phase** and click the **View calculation results** button on the toolbar. In the Output program, select the menu **Stresses** > **Heat flow** > **Temperature**.

The figure below shows the initial temperature distribution, which is obtained from the reference temperature at the ground surface and the earth gradient. This gives a temperature of 283.0 K at the ground surface and 283.4 at the bottom of the model.

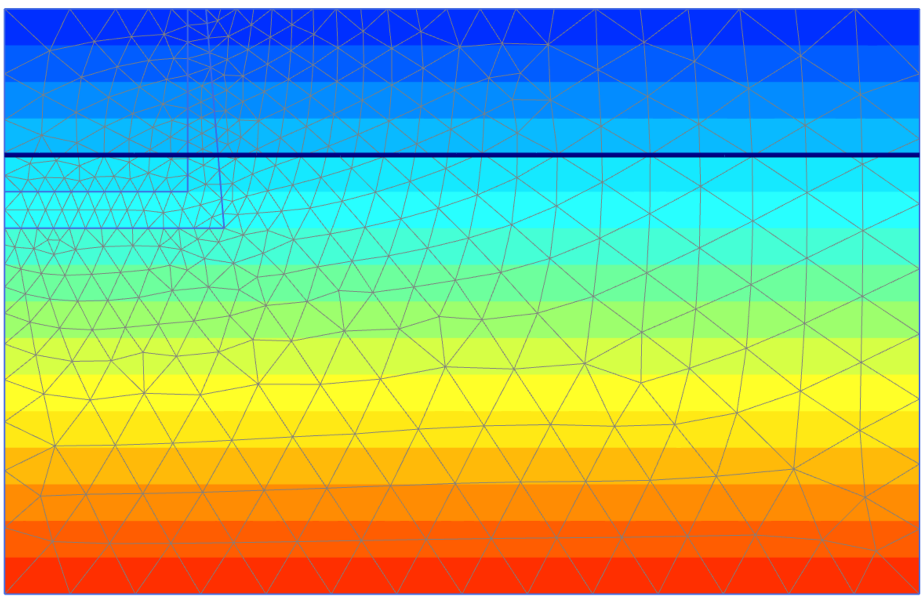

Figure 12: Initial temperature distribution

The figure below shows the temperature distribution obtained from Phase_1 using a steady-state thermal flow calculation. In fact, the temperatures at the top and bottom are equal to the temperatures as defined in the **Initial phase**; however, since the temperature at the ground surface is now defined in terms of **Climate** conditions (air temperature), this temperature is also applied at the inner side of the lock and affects the temperature distribution in the ground.

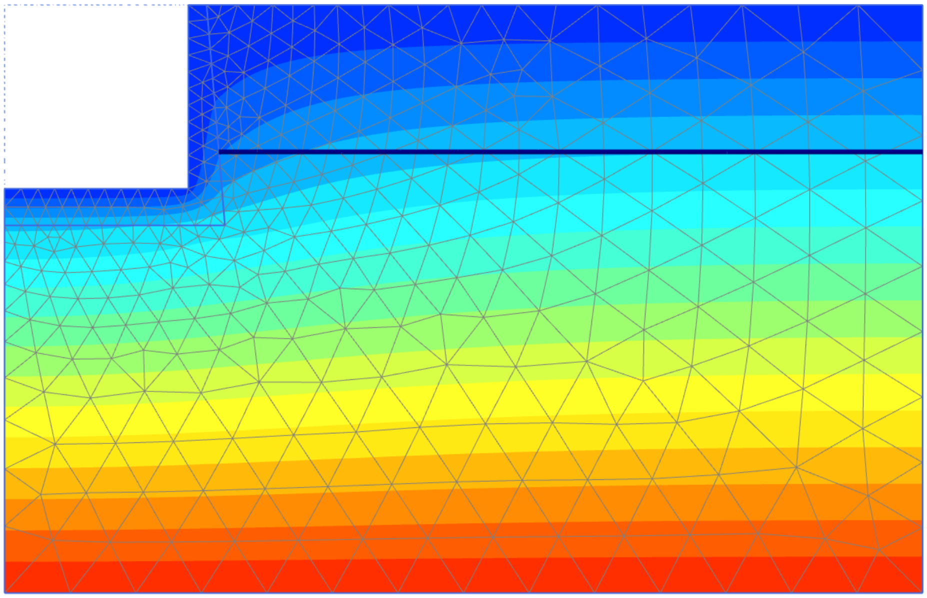

Figure 13: Steady-state temperature distribution in Phase_1

The most interesting results are obtained in Phase_2 in which the air temperature in the **Climate** condition increases gradually from 283 K to 298 K (defined by a quarter of a harmonic cycle with an amplitude of 15K). The figure below shows the temperature at the ground surface as a function of time.

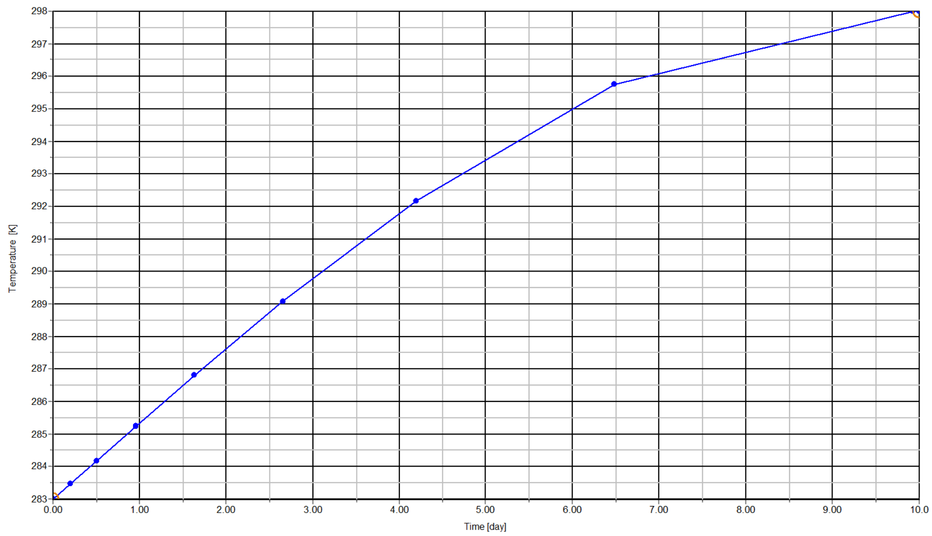

Figure 14: Temperature distribution in Point A as a function of time

As a result of the short increase in temperature at the inside of the concrete block, while the outer side (soil side) remains 'cold', the wall will bend towards the soil. The figure below shows the deformed mesh at the end of Phase_2.

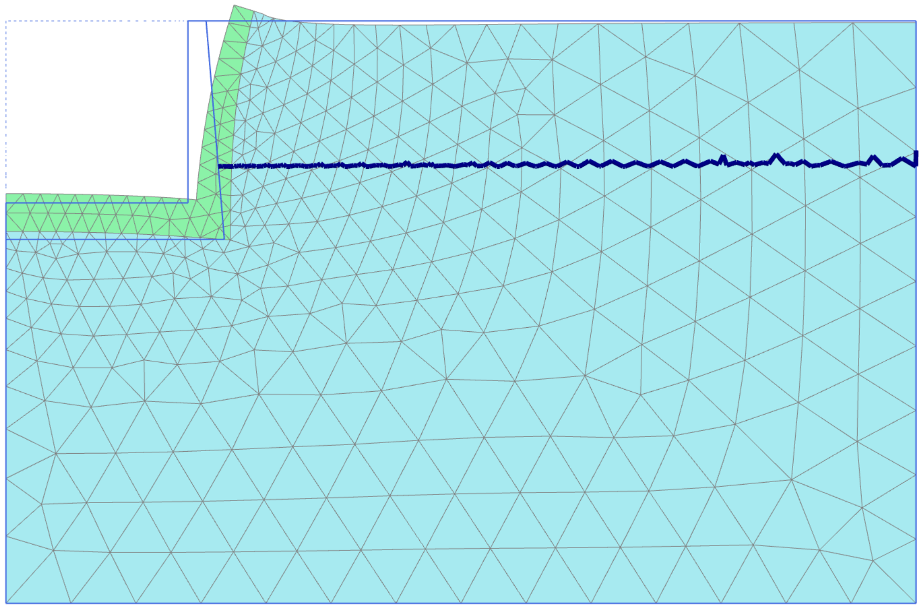

Figure 15: Deformed mesh at the end of Phase_2

As a result of this backward bending, the lateral stresses in the soil right behind the concrete block will increase, tending towards a passive stress state.

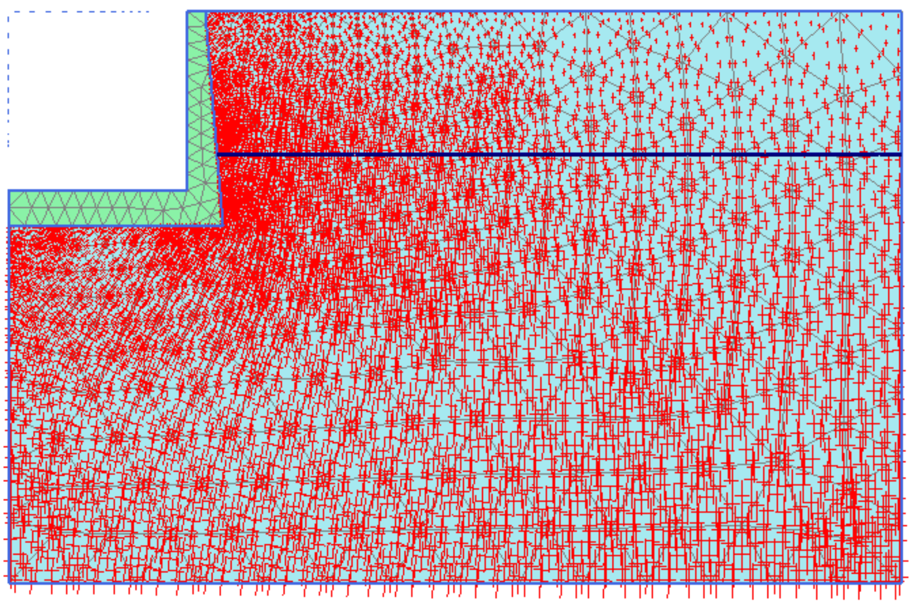

Figure 16: Effective principal stresses at the end of Phase_2 in the Principal directions

Note: Note that the visualisation is different for Figure 220 (on page 14), because it displays the stresses in the porous materials. This can be changed in **View** > **Settings** on the tab **Results** (see the Reference Manual for more information).

Note: Note that Figure 220 (on page 14) shows the principal stresses for all stress points whereas by default the principal stresses are only shown for the 3 center stress points. This can be changed using the  and  buttons on the navigation bar.