15 THERMAL EXPANSION OF A NAVIGABLE LOCK

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.

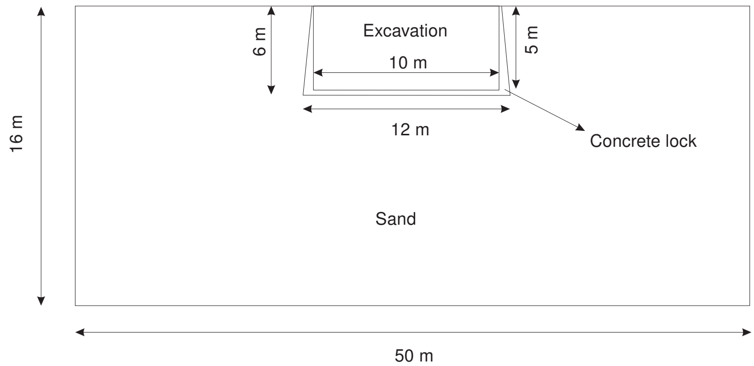

Figure 15.1 Concrete lock

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

Objectives:

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

15.1 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.
- Set the model dimensions to x_{min} = 0.0 m, x_{max} = 25.0 m, y_{min} = -16.0 m and y_{max} = 0.0 m.
- 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 -16.0 m. Set the head at -4.0 m.

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

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

- Define a data set for the *Sand* layer with the parameters given in Table 15.1, for the *General*, *Parameters*, *Groundwater*, *Thermal* and *Initial* tabsheets.
- Create another dataset for *Concrete* according to the Table 15.1.
- Assign the material dataset *Sand* to the borehole soil layer.

Table 15.1 Soil properties of the sand

Parameter	Name	Sand	Concrete	Unit
General				
Material model	*Model*	HS small	Linear elastic	-
Type of material behaviour	*Type*	Drained	Non-porous	-
Soil unit weight above phreatic level	γ_{unsat}	20.0	24.0	kN/m^3
Soil unit weight below phreatic level	γ_{sat}	20.0	-	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	-
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}'	2.0	-	kN/m^2
Angle of internal friction	ϕ'	32.0	-	°
Dilatancy angle	ψ	2.0	-	°
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}	$8 \cdot 10^4$	-	kN/m^2
Groundwater				
Data set	-	USDA	-	-
Model	-	Van Genuchten	-	-
Soil type	-	Sandy clay	-	-
Use defaults	-	From data set	-	-
Thermal				
Specific heat capacity	c_s	860	900	kJ/t/K
Thermal conductivity	λ_s	$4.0 \cdot 10^{-3}$	$1.0 \cdot 10^{-3}$	kW/m/K
Soil density	ρ_s	2.6	2.5	t/m^3
X-component of thermal expansion	α_x	$0.5 \cdot 10^{-6}$	$0.1 \cdot 10^{-4}$	1/K
Y-component of thermal expansion	α_y	$0.5 \cdot 10^{-6}$	$0.1 \cdot 10^{-4}$	1/K
Z-component of thermal expansion	α_z	$0.5 \cdot 10^{-6}$	$0.1 \cdot 10^{-4}$	1/K
Interfaces				
Interface strength	-	Rigid	Manual	-
Strength reduction factor inter.	R_{inter}	1.0	0.67	-
Initial				
K_0 determination	-	Automatic	Automatic	

Definition of structural elements

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

- Proceed to *Structures* mode.

- Click the *Create soil polygon* button in the side toolbar and select the *Create soil polygon* option in the appearing menu.
- Define the lock in the drawing area by clicking on (0.0 -5.0), (5.0 -5.0), (5.0 0.0), (5.5 0.0), (6.0 -6.0), (0.0 -6.0) and (0.0 -5.0).

> **Hint:** 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*.

- Click the *Create line* button in the side toolbar.
- Select the *Create thermal flow bc* option in the expanded menu (Figure 15.2).
- Create thermal boundaries at vertical boundaries and the bottom boundary (X_{min}, X_{max} and Y_{min}).

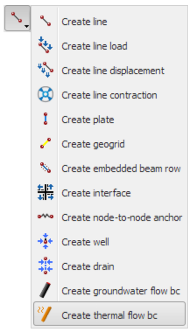

Figure 15.2 The *Create thermal bc* option in the *Create line* menu

- The vertical boundaries have the default option of *Closed* for the *Behaviour*.
- Select the bottom boundary, in the *Selection explorer* set the *Behaviour* to *Temperature*.
- Set the reference temperature, T_{ref} to 283.4 K (Figure 15.3).

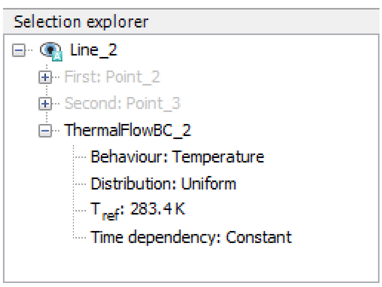

Figure 15.3 Thermal boundary condition in the *Selection explorer*

The geometry of the model is now complete (Figure 15.4).

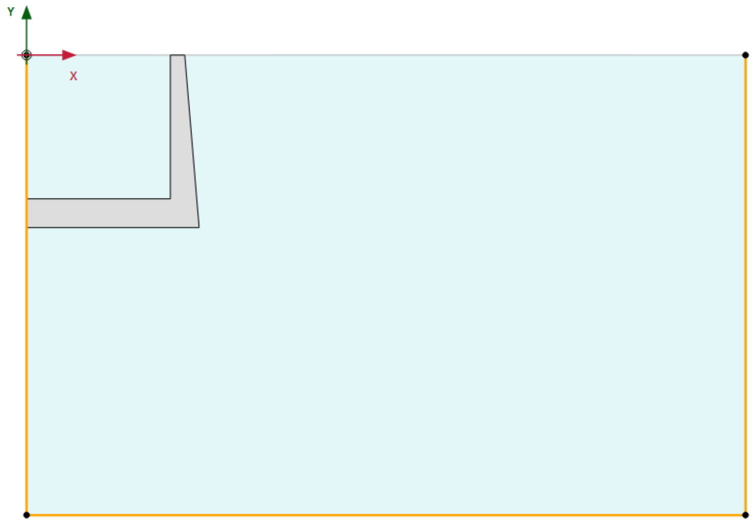

Figure 15.4 Geometry of the model

15.2 MESH GENERATION

- Proceed to the *Mesh* mode.
- Select the polygon representing the concrete block, and in the *Selection explorer* set the *Coarseness factor* to 0.25.
- 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 15.5.
- Click on the *Close* tab to close the Output program.

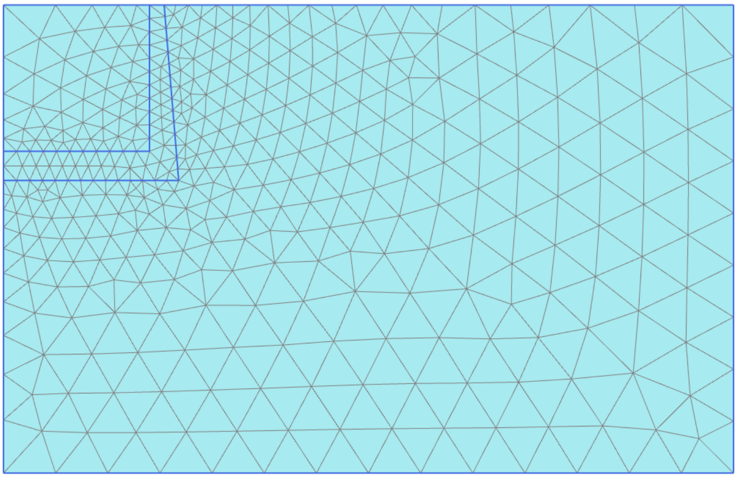

Figure 15.5 The generated mesh

15.3 CALCULATIONS

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

- Proceed to *Staged construction* mode.
- Double click on *Initial phase* in the *Phases explorer*.
- The default options for *Calculation type* and *Pore pressure calculation type* are used in this example.
- Select *Earth gradient* for the *Thermal calculation type* option and close the *Phases* window.
- 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 15.6).

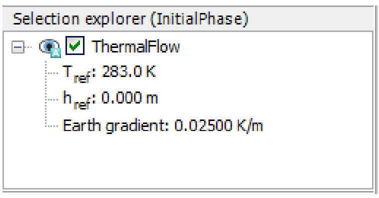

Figure 15.6 Thermal flow in the *Selection explorer*

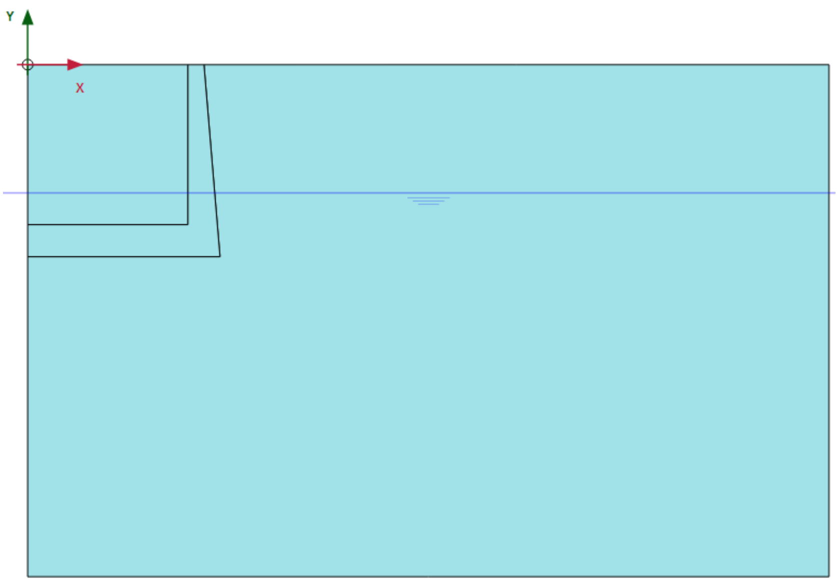

Figure 15.7 Initial phase

Phase 1: Construction

- Add a new phase (Phase_1).
- Double click on *Phase_1* in the *Phases explorer*.
- In the *Phases* window, enter an appropriate name for the phase ID and select *Steady state groundwater flow* as *Pore pressure calculation type*.
- Set the *Steady state thermal flow* for the *Thermal calculation type*.
- Make sure that the *Reset displacements to zero* and *Ignore suction* options are selected.
- In the *Staged construction* mode, assign the *Concrete* dataset to the created polygon which represents the navigable lock (Figure 15.8).

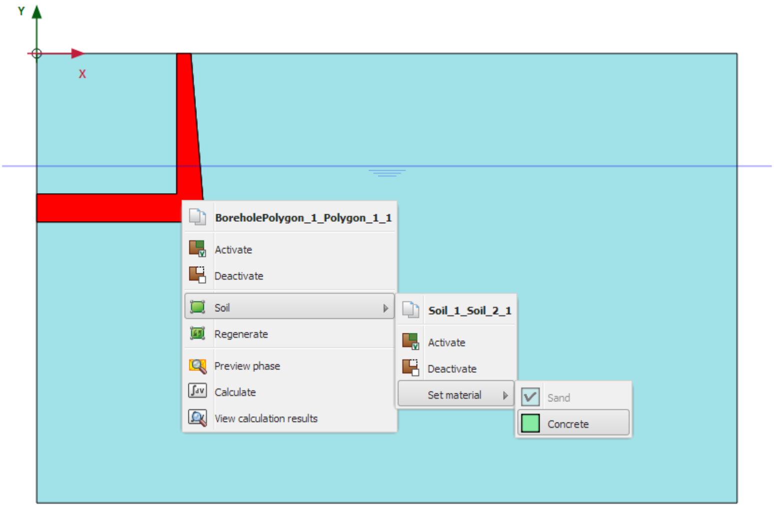

Figure 15.8 Assigning *Concrete* soil data set to the navigable lock

- Right click the soil cluster which is cut-off by the polygon and select the option *Deactivate* from the appearing menu.
- In the *Selection explorer*, set the *WaterConditions* of this cluster to *Dry*.
- Multi-select the vertical and bottom horizontal wall of the excavation.
- In the *Selection explorer*, activate the *Groundwater flow boundary condition*.
- Set the *Behaviour* to *Head* and the h_{ref} to -5.0 m (Figure 15.9). This will simulate an 'empty' lock.
- In the *Model explorer*, activate all the *Thermal flow boundary conditions*.
- In the *Model explorer*, activate the *Climate* condition under the subtree *Model conditions*.
- Set the *Air temperature* to 283.0 K and the *Surface transfer* to 1.0 $kW/m^2/K$ (Figure 15.10). This will define the thermal conditions at the ground surface and the inside of the lock.

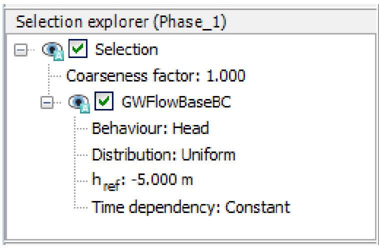

Figure 15.9 Groundwater flow boundary condition in the *Selection explorer*

- 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.
- Figure 15.11 shows the model at the end of *Phase_1*.

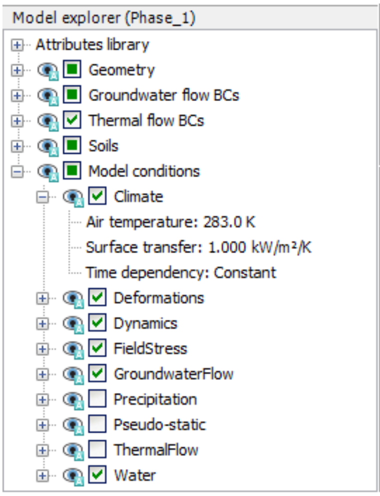

Figure 15.10 Model conditions for Phase_1

Phase 2: Heating

Add a new phase (Phase_2).

- Double click on *Phase_2* in the *Phases explorer*.

Set the *Calculation type* to *Fully coupled flow deformation*.

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.

- The *Time interval* is set to 10 days.
- 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.

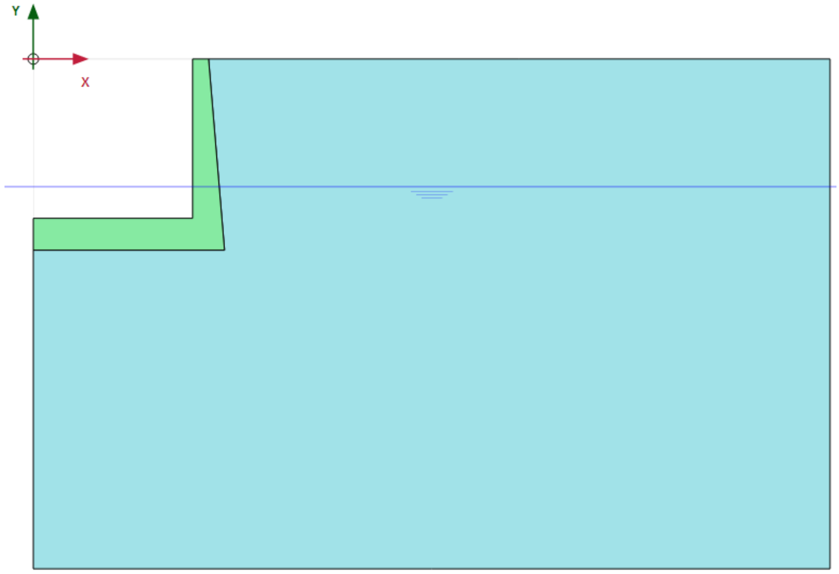

Figure 15.11 The model at the end of *Phase_1*

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.

- 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.

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.

- The default option of *Harmonic* is used for this signal.
- Assign a value of 15.0 for the *Amplitude* and 40 days for the *Period*. A graph is displayed showing the defined function (Figure 15.12). 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.
- Click *OK* to close the *Thermal functions* window.
- Expand the subtree *Model conditions* in the *Model explorer*.
- In the *Climate* option, set the *Time dependency* to *Time dependent* and assign the temperature function which was created (Figure 15.13).

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 at the top of the excavation, (5.0, 0.0)).

Calculate the project by clicking the *Calculate* button and ignore the warnings regarding different stress type used in the *Fully coupled flow deformation* analysis.

Save the project after the calculation has finished.

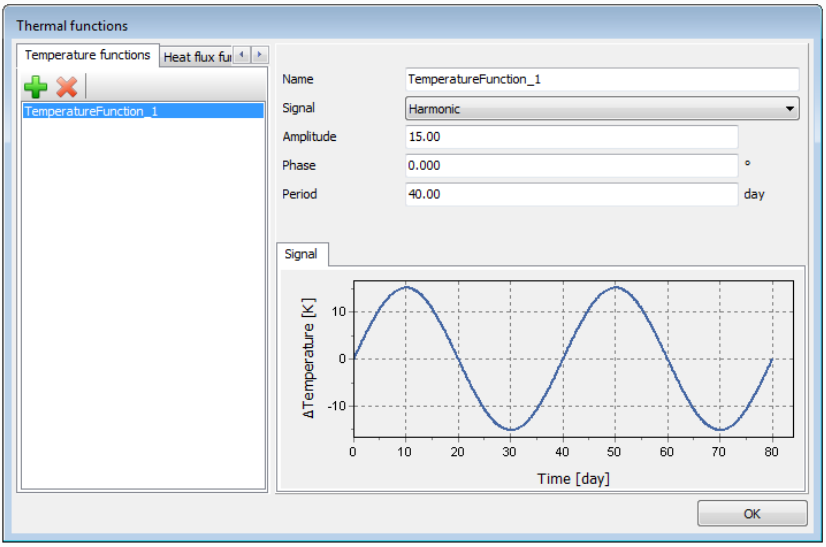

Figure 15.12 The temperature function

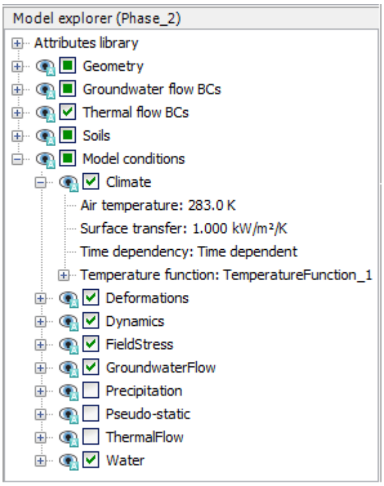

Figure 15.13 Model conditions for Phase_2

15.4 RESULTS

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

Figure 15.14 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 15.15 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

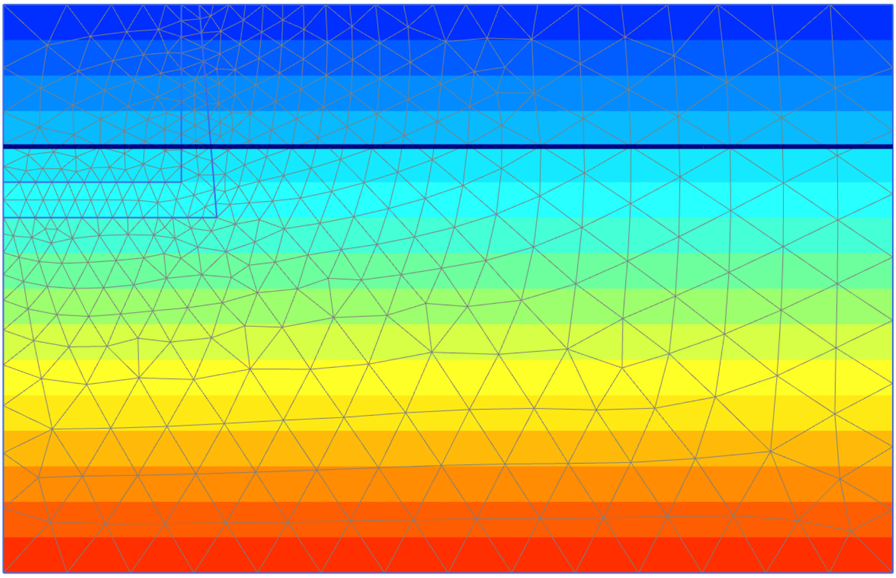

Figure 15.14 Initial temperature distribution

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.

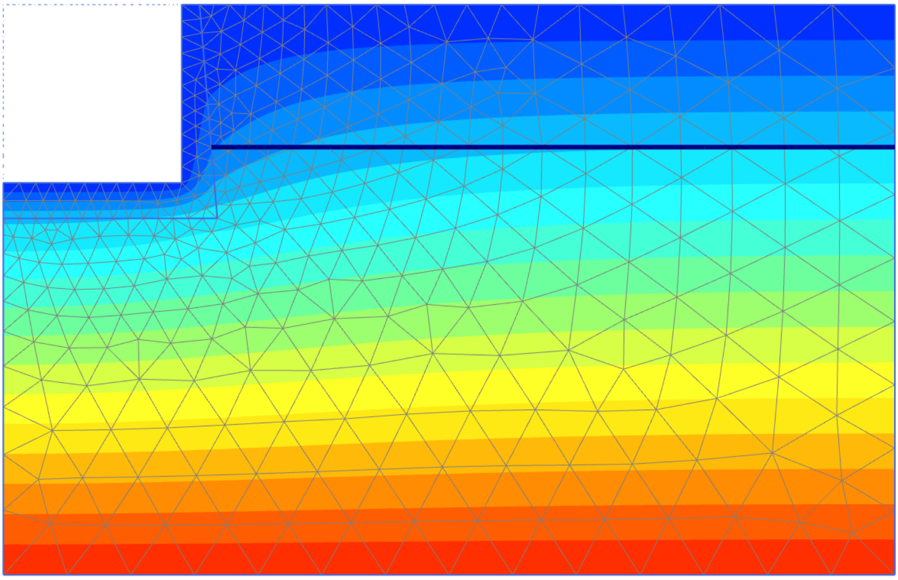

Figure 15.15 Steady-state temperature distribution in *Phase_1*

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). Figure 15.16 shows the temperature at the ground surface as a function of time.

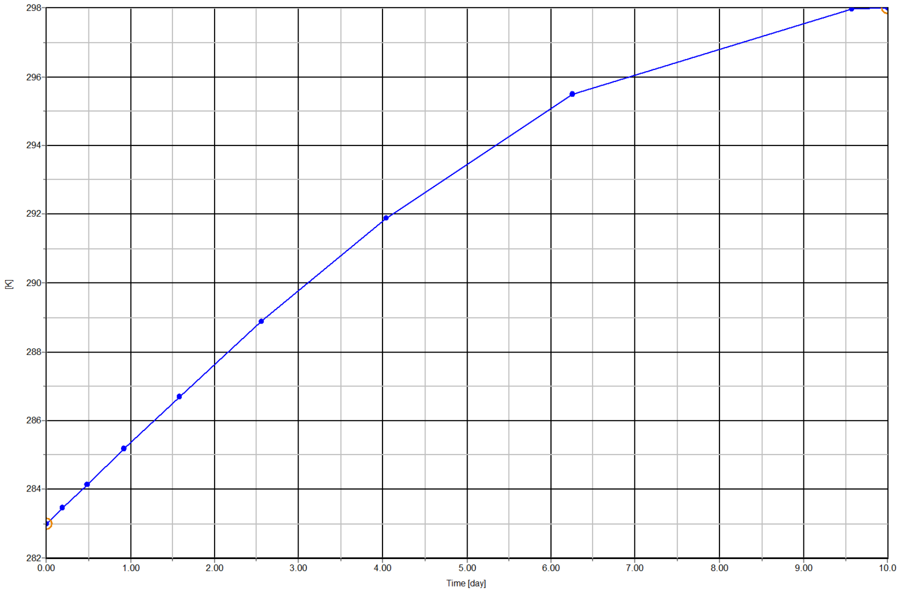

Figure 15.16 Temperature distribution in Point A as a function of time

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. Figure 15.17 shows the 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 15.18). Note that the visualisation is different for Figure 15.18, because it displays the stresses in the porous materials. This can be changed in the *Settings* window under the tab *Results* (Section 8.5.2 of the Reference Manual).

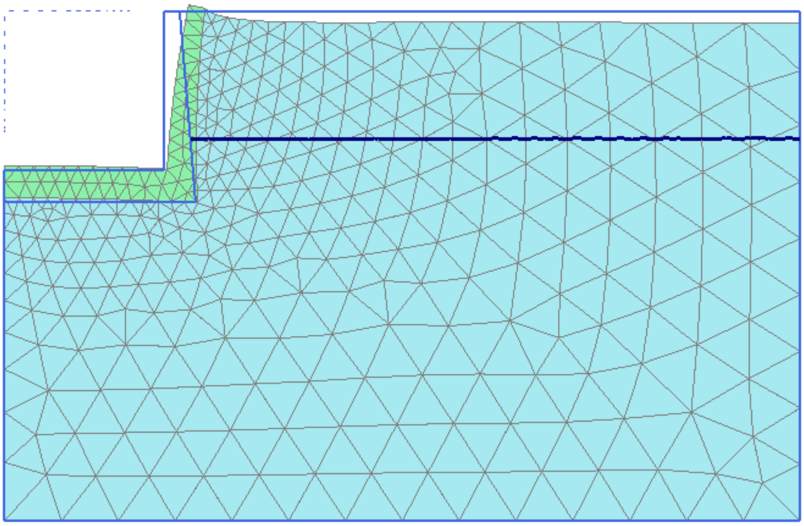

Figure 15.17 Deformed mesh at the end of *Phase_2*

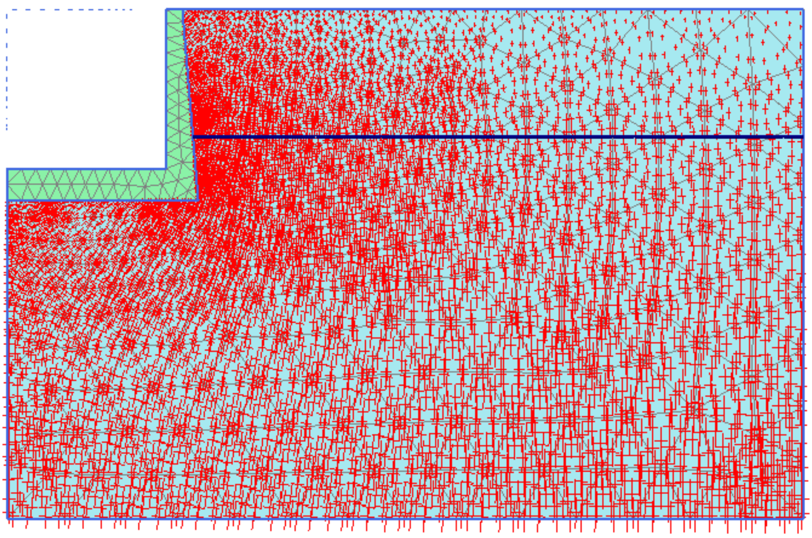

Figure 15.18 Effective principal stresses at the end of *Phase_2* in the *Principal directions*