

18

Thermal expansion of a navigable lock [ULT]

18.1 | Introduction

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

18.2 | Geometry

The geometry of the project is shown in [Figure 18-1 \(p. 272\)](#).

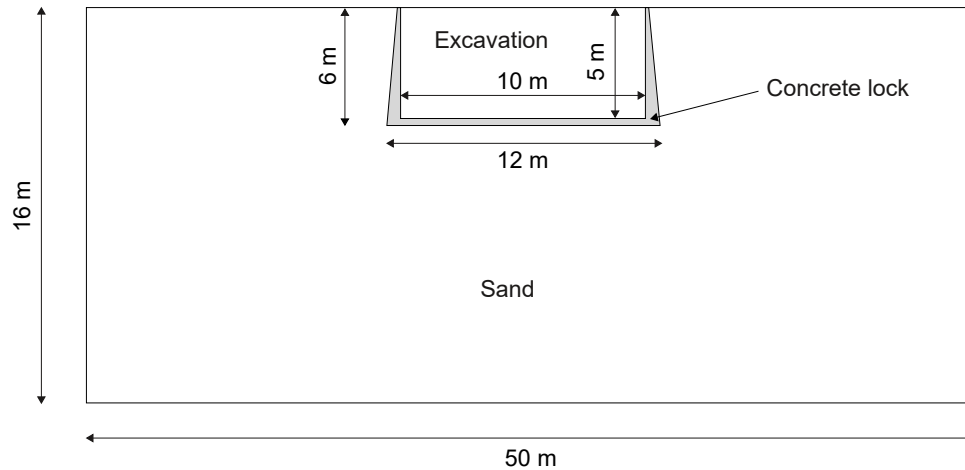



Figure 18-1: Geometry of the project

18.3 | 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.

18.4 | 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.

18.5 | 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 18-1 \(p. 273\)](#), for the **General, Mechanical, Groundwater, Thermal, Interfaces** and **Initial** tabsheets.
3. Create another dataset for **Concrete** according to [Table 18-1 \(p. 273\)](#).
4. Assign the material dataset **Sand** to the borehole soil layer.


Table 18-1: Material properties

Parameter	Name	Sand	Concrete	Unit
General				
Soil model	-	HS small	Linear elastic	-
Drainage type	-	Drained	Non-porous	-
Unsaturated unit weight	γ_{unsat}	20	24	kN/m ³
Saturated unit weight	γ_{sat}	20	-	kN/m ³
Mechanical				
Young's modulus	E_{ref}	-	$25 \cdot 10^6$	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	-	-
Shear modulus at very small strains	G_0^{ref}	$80 \cdot 10^3$	-	kN/m ²
Shear strain at which $G_s = 0.722 G_0$	$\gamma_{0.7}$	$0.1 \cdot 10^{-3}$	-	-
Cohesion	c'_{ref}	2	-	kN/m ²
Friction angle	φ'	32	-	°
Dilatancy angle	ψ	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				
Thermal conductivity	λ_s	$4 \cdot 10^{-3}$	$1 \cdot 10^{-3}$	kW/m/K
Soil density	ρ_s	2.6	2.5	t/m ³
Thermal expansion type	-	Isotropic	Isotropic	-
Volumetric Thermal expansion	α_{sv}	$1.5 \cdot 10^{-6}$	$0.03 \cdot 10^{-3}$	1/K
Interfaces				
Strength determination	-	Rigid	Manual	-
Interface reduction factor	R_{inter}	1.0	0.67	-
Initial				
K_0 determination	-	Automatic	Automatic	-

18.6 | 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).

✓ **Tip:**

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 which is shown in [Figure 18-2 \(p. 275\)](#).

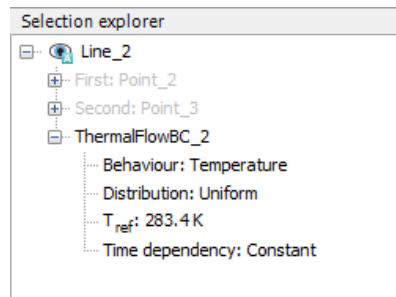


Figure 18–2: Thermal boundary condition in the Selection explorer

The geometry of the model is now complete as shown in [Figure 18–3 \(p. 275\)](#) .

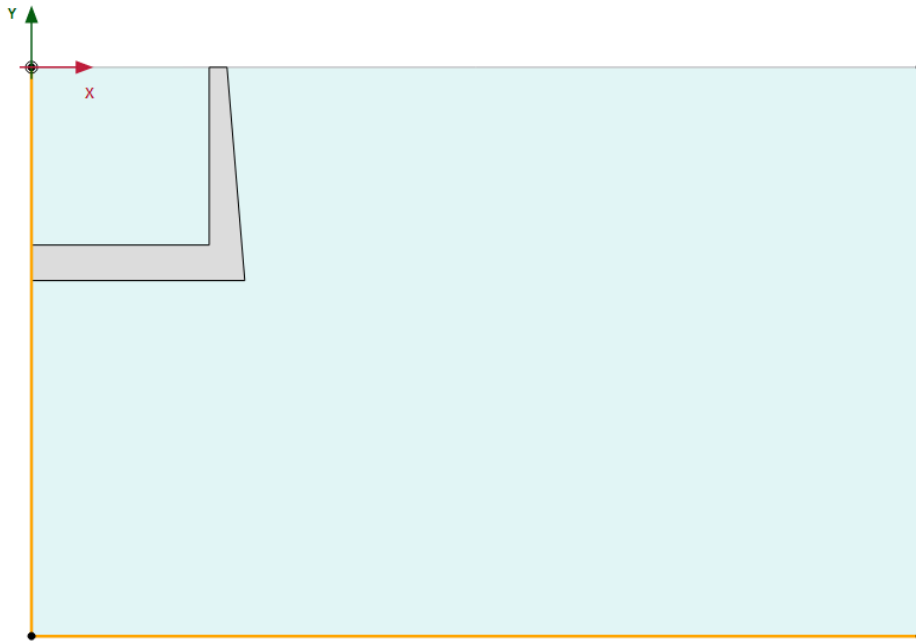




Figure 18–3: Geometry of the model

18.7 | 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 is shown in [Figure 18–4 \(p. 276\)](#):

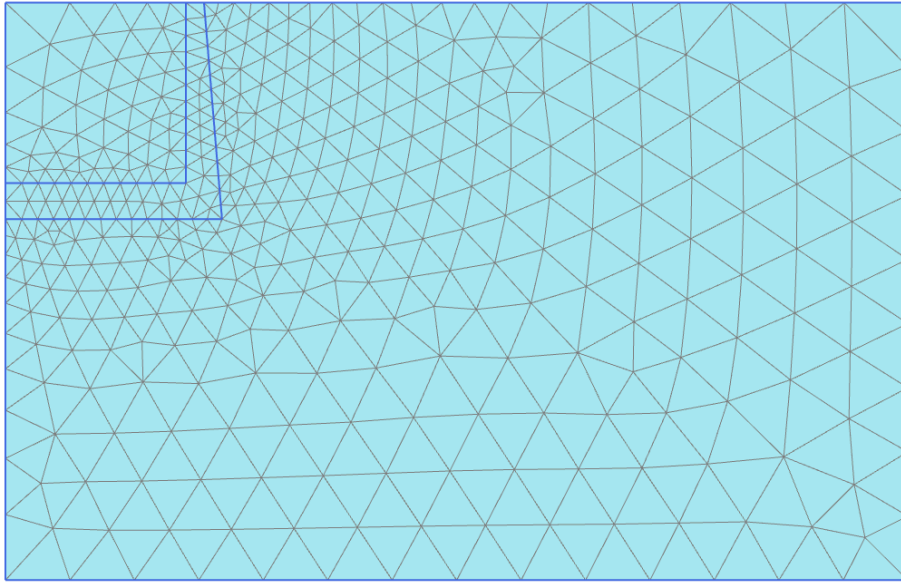


Figure 18–4: The generated mesh

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

18.8 | 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.

18.8.1 | 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 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. The Thermal flow parameters are shown in [Figure 18–5 \(p. 277\)](#) and the model for initial phase is shown in [Figure 18–6 \(p. 277\)](#).

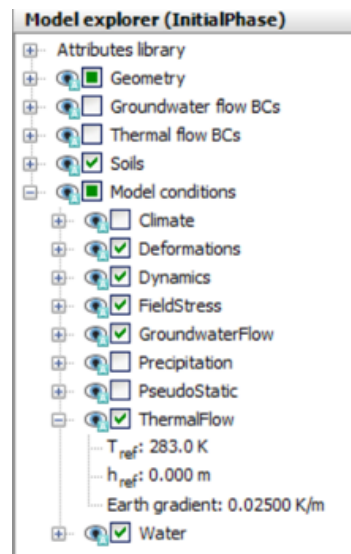


Figure 18–5: Thermal flow in the Model explorer

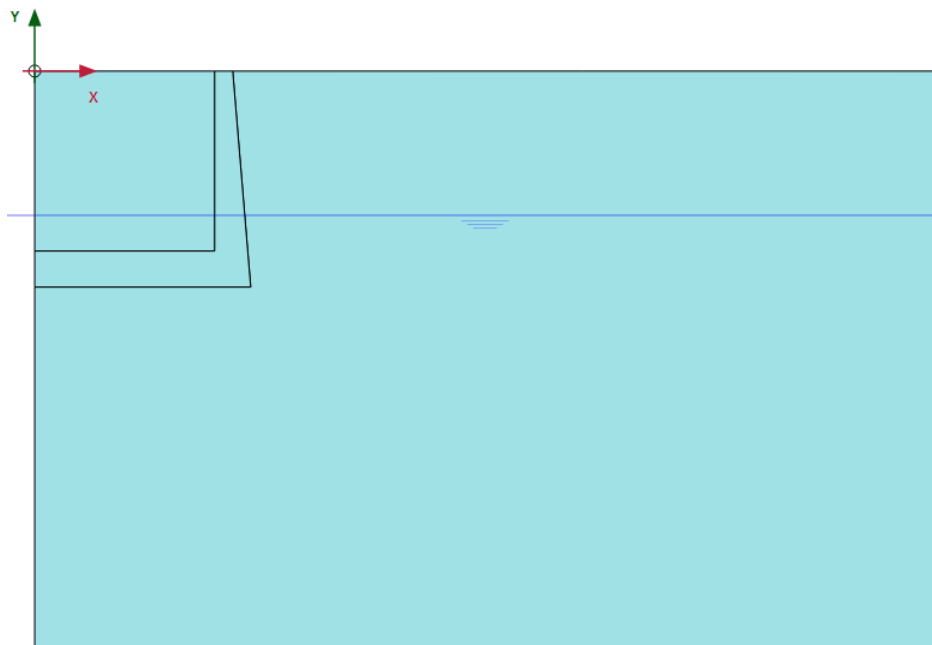





Figure 18–6: Initial phase

18.8.2 | 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 as shown in [Figure 18–7 \(p. 278\)](#).

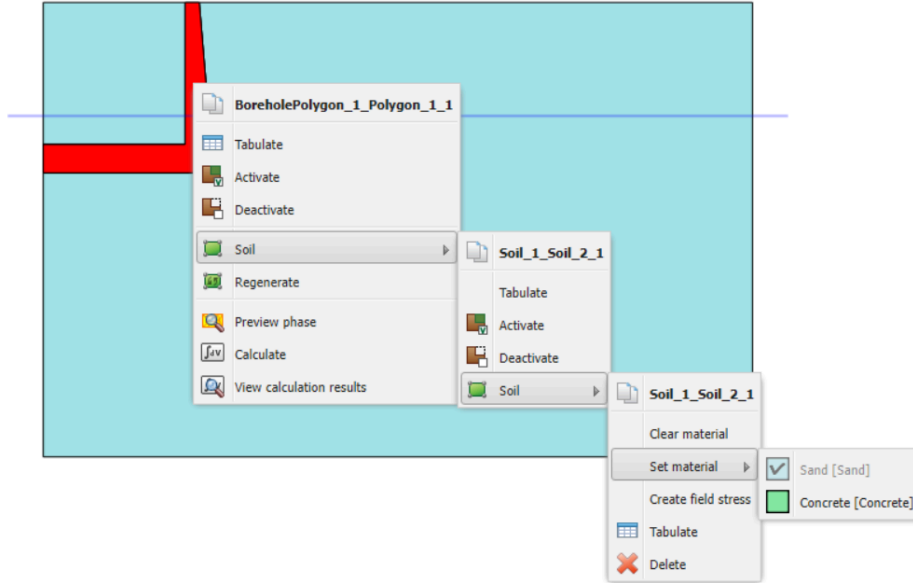


Figure 18–7: Concrete set as material for polygon

- 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/m²/K as shown in [Figure 18–8 \(p. 279\)](#).

This will define the thermal conditions at the ground surface and the inside of the lock.

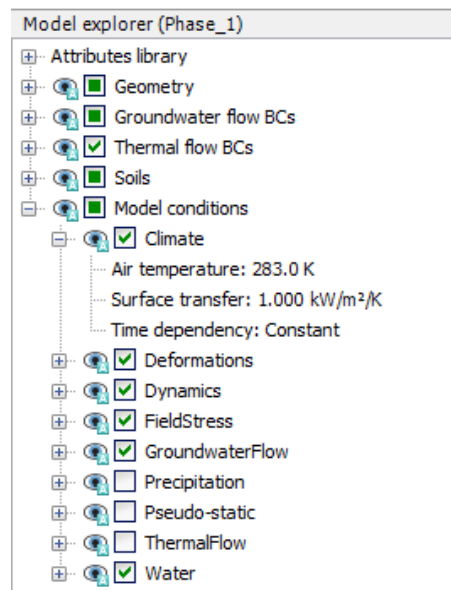


Figure 18–8: Model conditions for Phase_1

- 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.

[Figure 18–9 \(p. 279\)](#) shows the model at the end of **Phase_1**.

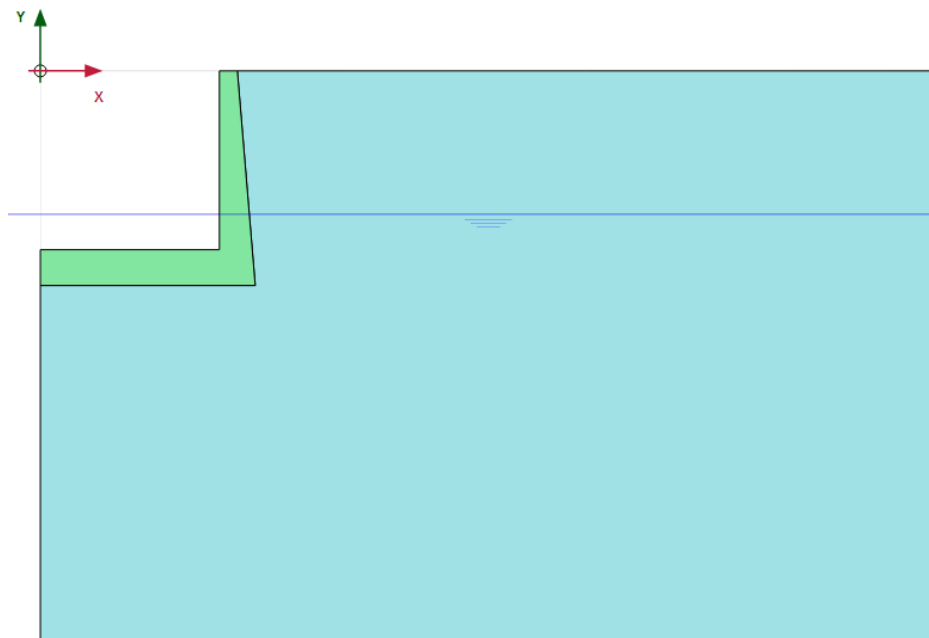






Figure 18–9: The model at the end of Phase_1

18.8.3 | 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 in [Figure 18–10 \(p. 280\)](#) 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.

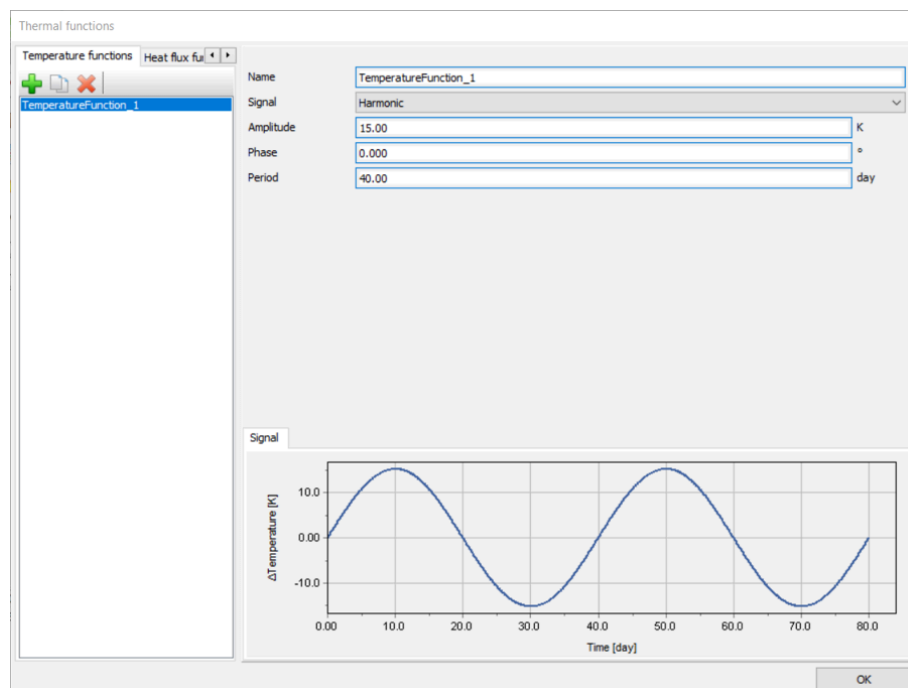


Figure 18–10: The temperature function

- e. Click **OK** to close the **Thermal functions** window.
- 8 Expand the subtree **Model conditions** in the **Model explorer** shown in [Figure 18–11 \(p. 281\)](#).

- 9 In the **Climate** option, set the **Time dependency** to **Time dependent** and assign the temperature function which was created.

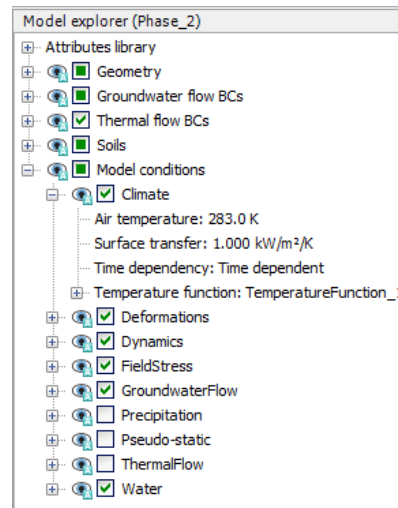





Figure 18–11: Model conditions for Phase_2

The calculation definition is now complete.

18.8.4 | 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 .

18.9 | 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**.

[Figure 18–12 \(p. 282\)](#) 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.

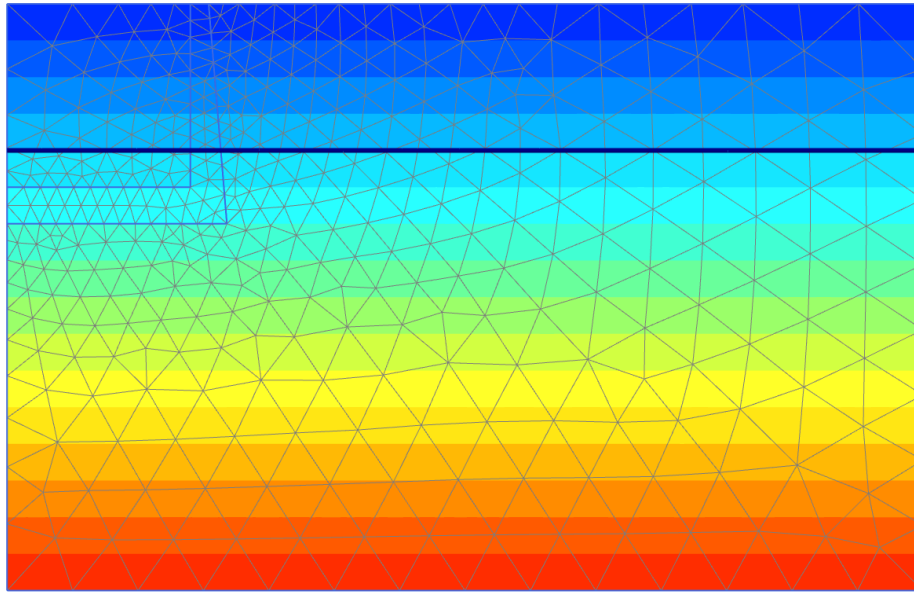


Figure 18-12: Initial temperature distribution

[Figure 18-13 \(p. 282\)](#) 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.

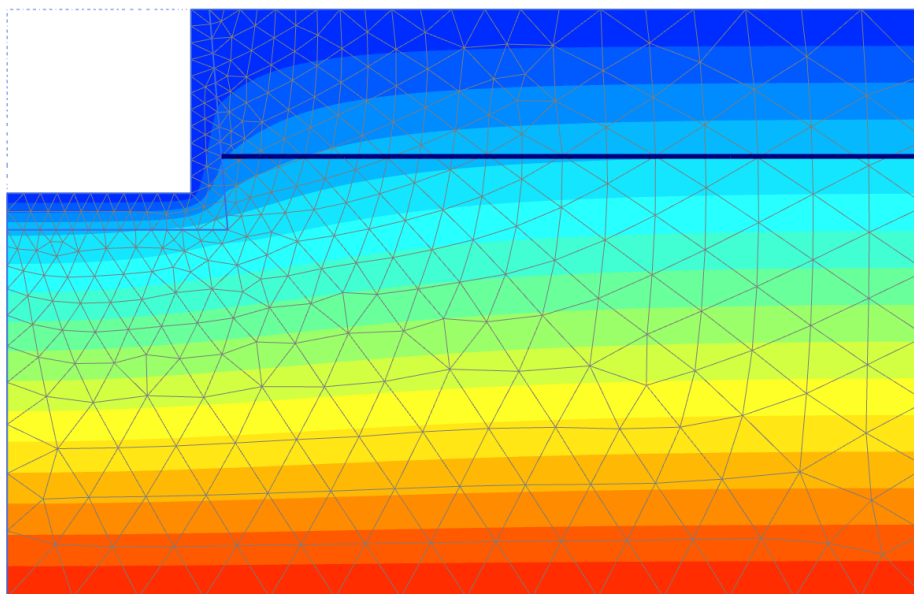


Figure 18-13: Steady-state temperature distribution in Phase_1

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). [Figure 18-14 \(p. 283\)](#) shows the temperature at the ground surface as a function of time.

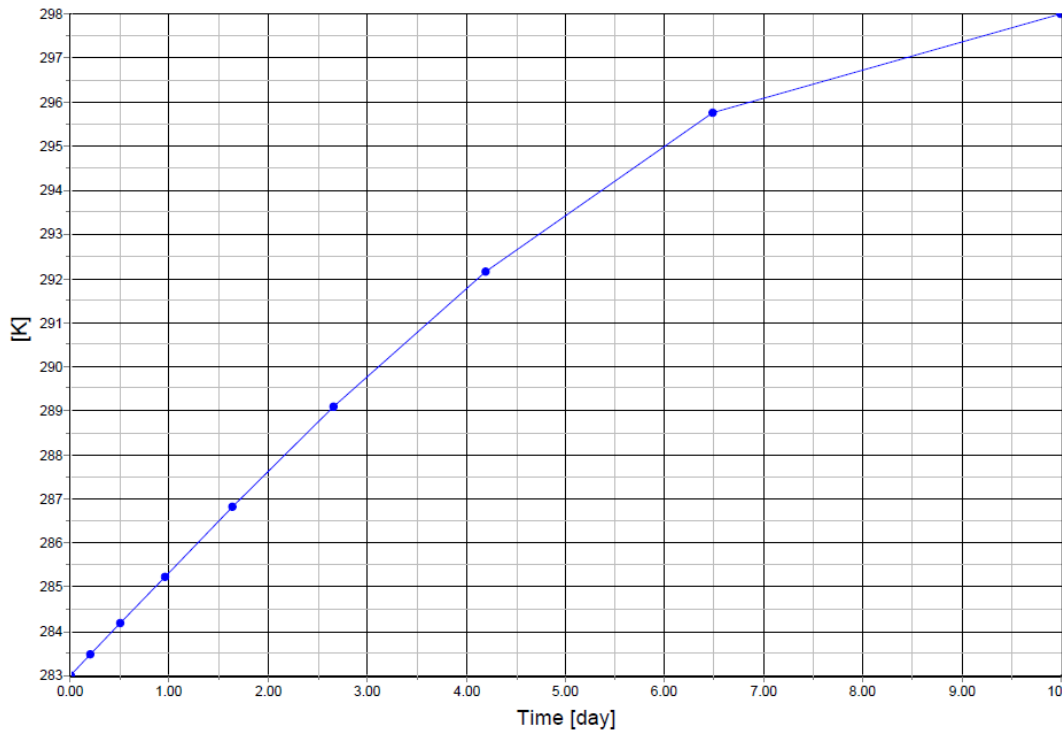


Figure 18-14: Temperature distribution in Point A as a function of time

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. [Figure 18-15 \(p. 283\)](#) shows the deformed mesh at the end of Phase_2.

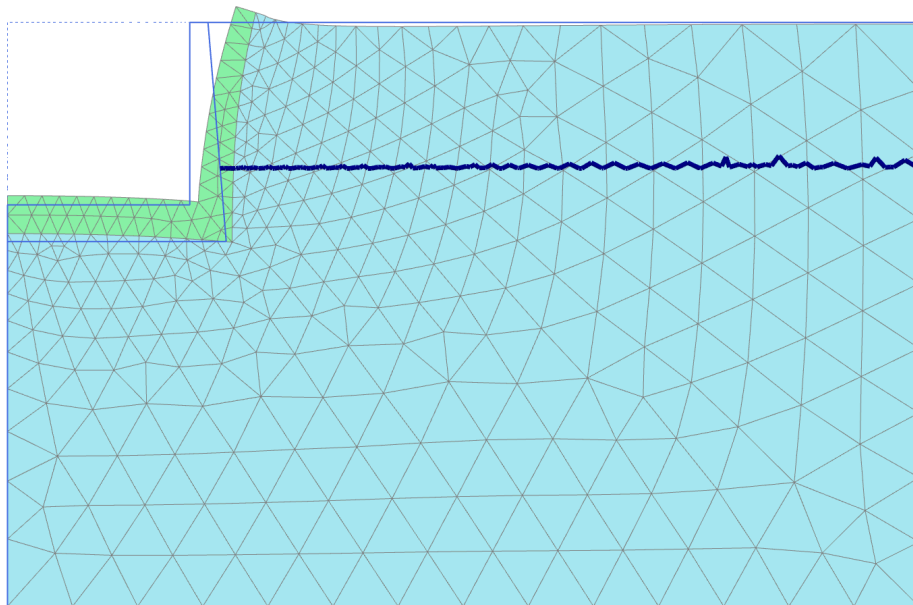


Figure 18-15: Deformed mesh at the end of Phase_2

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.

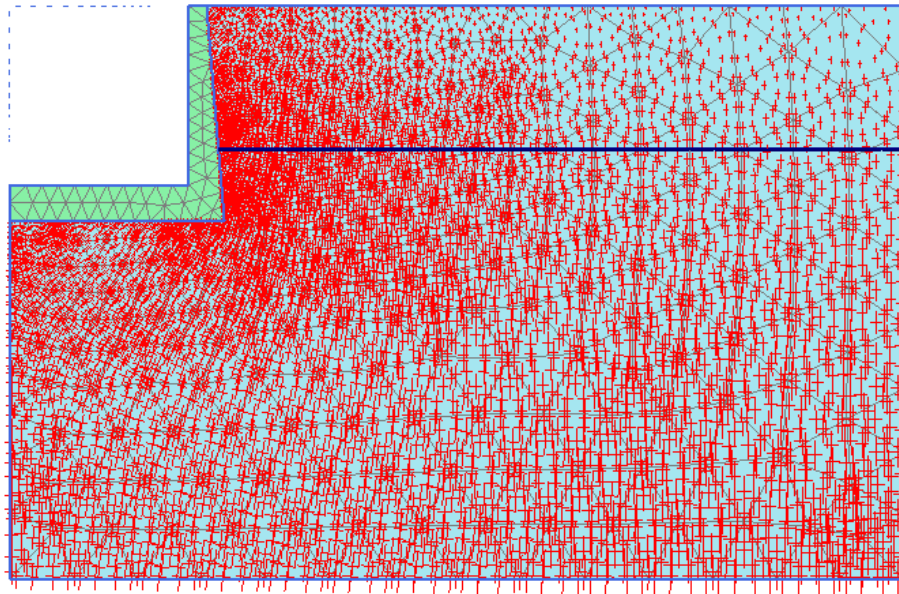




Figure 18–16: Effective principal stresses at the end of Phase_2 in the Principal directions

Note:

- Note that the visualisation is different for [Figure 18–16 \(p. 284\)](#), 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 that [Figure 18–16 \(p. 284\)](#) 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.