3 DRY EXCAVATION USING A TIE BACK WALL

This example involves the dry construction of an excavation. The excavation is supported by concrete diaphragm walls. The walls are tied back by prestressed ground anchors.

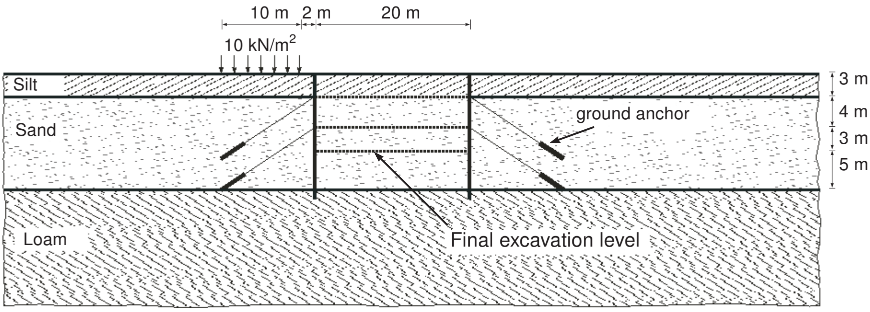

Figure 3.1 Excavation supported by tie back walls

PLAXIS allows for a detailed modelling of this type of problem. It is demonstrated in this example how ground anchors are modelled and how prestressing is applied to the anchors. Moreover, the dry excavation involves a groundwater flow calculation to generate the new water pressure distribution. This aspect of the analysis is explained in detail.

Objectives:

- Modelling ground anchors.
- Generating pore pressures by groundwater flow.
- Displaying the contact stresses and resulting forces in the model.
- Scaling the displayed results.

3.1 INPUT

The excavation is 20 m wide and 10 m deep. 16 m long concrete diaphragm walls of 0.35 m thickness are used to retain the surrounding soil. Two rows of ground anchors are used at each wall to support the walls. The anchors have a total length of 14.5 m and an inclination of 33.7° (2:3). On the left side of the excavation a surface load of 10 kN/m^2 is taken into account.

The relevant part of the soil consists of three distinct layers. From the ground surface to a depth of 23 m there is a fill of relatively loose fine sandy soil. Underneath the fill, down to a minimum depth of 15 m, there is a more or less homogeneous layer consisting of dense well-graded sand. This layer is particular suitable for the installation of the ground anchors. The underlying layer consists of loam and lies to a large depth. 15 m of this layer is considered in the model. In the initial situation there is a horizontal phreatic level at 3 m below the ground surface (i.e. at the base of the fill layer).

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 keep the default options for *Model* (*Plane strain*), and *Elements* (*15-Node*).
- Set the model dimensions to x_{min} = 0.0 m, x_{max} = 100.0 m, y_{min} = 0.0 m, y_{max} = 30.0 m.
- Keep the default values for units and the constants and press *OK* to close the *Project properties* window.

Definition of soil stratigraphy

To define the soil stratigraphy:

- Create a borehole at x = 0. The *Modify soil layers* window pops up.
- Add three soil layers to the borehole. Locate the ground level at y = 30 m by assigning 30 to the *Top* level of the uppermost layer. The bottom levels of the layers are located at 27, 15 and 0 m, respectively.
- Set the *Head* to 23 m. The layer stratigraphy is shown in Figure 3.2.

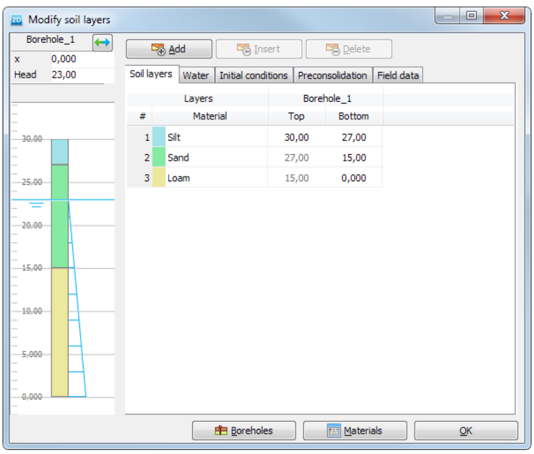

Figure 3.2 The *Modify soil layers* window

- Define three data sets for soil and interfaces with the parameters given in Table 3.1.
- Assign the material data sets to the corresponding soil layers (Figure 3.2).

Table 3.1 Soil and interface properties

Parameter	Name	Silt	Sand	Loam	Unit
General					
Material model	*Model*	Hardening soil	Hardening soil	Hardening soil	-
Type of material behaviour	*Type*	Drained	Drained	Drained	-
Soil unit weight above phreatic level	γ_{unsat}	16	17	17	kN/m^3
Soil unit weight below phreatic level	γ_{sat}	20	20	19	kN/m^3
Parameters					
Secant stiffness in standard drained triaxial test	E_{50}^{ref}	$2.0 \cdot 10^4$	$3.0 \cdot 10^4$	$1.2 \cdot 10^4$	kN/m^2
Tangent stiffness for primary oedometer loading	E_{oed}^{ref}	$2.0 \cdot 10^4$	$3.0 \cdot 10^4$	$8.0 \cdot 10^3$	kN/m^2
Unloading / reloading stiffness	E_{ur}^{ref}	$6.0 \cdot 10^4$	$9.0 \cdot 10^4$	$3.6 \cdot 10^4$	kN/m^2
Power for stress-level dependency of stiffness	m	0.5	0.5	0.8	-
Cohesion	c_{ref}'	1.0	0.0	5.0	kN/m^2
Friction angle	φ'	30	34	29	°
Dilatancy angle	ψ	0.0	4.0	0.0	°
Poisson's ratio	ν_{ur}'	0.2	0.2	0.2	-
K_0-value for normal consolidation	K_0^{nc}	0.5	0.4408	0.5152	-
Groundwater					
Data set	-	USDA	USDA	USDA	-
Model	-	Van Genuchten	Van Genuchten	Van Genuchten	-
Soil type	-	Silt	Sand	Loam	-
$< 2\mu m$	-	6.0	4.0	20.0	%
$2\mu m - 50\mu m$	-	87.0	4.0	40.0	%
$50\mu m - 2mm$	-	7.0	92.0	40.0	%
Use defaults	-	From data set	From data set	From data set	-
Permeability in horizontal direction	k_x	0.5996	7.128	0.2497	m/day
Permeability in vertical direction	k_y	0.5996	7.128	0.2497	m/day
Interfaces					
Interface strength	—	Manual	Manual	Rigid	-
Strength reduction factor inter.	R_{inter}	0.65	0.70	1.0	-
Consider gap closure	—	Yes	Yes	Yes	-
Initial					
K_0 determination	—	Automatic	Automatic	Automatic	-
Over-consolidation ratio	*OCR*	1.0	1.0	1.0	-
Pre-overburden pressure	*POP*	0.0	0.0	25.0	kN/m^2

3.1.1 DEFINITION OF STRUCTURAL ELEMENTS

- In the *Structures* mode, model the diaphragm walls as plates passing through (40.0 30.0) - (40.0 14.0) and (60.0 30.0) - (60.0 14.0).
- Multi-select the plates in the model.
- In the *Selection explorer* click on *Material*. The view will change displaying a drop-down menu and a plus button next to it (Figure 3.3).
- Click the plus button. A new empty material set is created for plates.
- Define the material data set for the diaphragm walls according to the properties are listed in Table 3.2. The concrete has a Young's modulus of 35 GN/m^2 and the wall is 0.35 m thick.

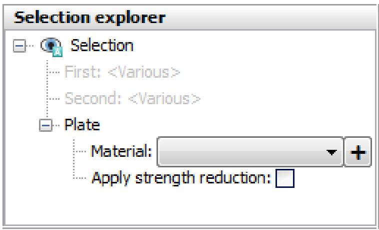

Figure 3.3 Material assignment in the *Selection explorer*

Table 3.2 Properties of the diaphragm wall (plate)

Parameter	Name	Value	Unit
Material type	*Type*	Elastic; Isotropic	-
Prevent punching	—	Yes	-
Normal stiffness	*EA*	$1.2 \cdot 10^7$	kN/m
Flexural rigidity	*EI*	$1.2 \cdot 10^5$	kNm^2/m
Weight	*w*	8.3	kN/m/m
Poisson's ratio	ν	0.15	-

- Assign positive and negative interfaces to the geometry lines created to represent the diaphragm walls.

The soil is excavated in three stages. The first excavation layer corresponds to the bottom of the silt layer and it is automatically created. To define the remaining excavation stages:

- Define the second excavation phase by drawing a line through (40.0 23.0) and (60.0 23.0).
- Define the third excavation phase by drawing a line through (40.0 20.0) and (60.0 20.0).

A ground anchor can be modelled by a combination of a node-to-node anchor and an embedded beam. The embedded pile simulates the grouted part of the anchor whereas the node-to-node anchor simulates the free length. In reality there is a complex three-dimensional state of stress around the grout body which cannot be simulated in a 2D model.

- Define the node-to-node anchors according to Table 3.3.

Table 3.3 Node to node anchor coordinates

Anchor location		First point	Second point
Top	Left	(40.0 27.0)	(31.0 21.0)
	Right	(60.0 27.0)	(69.0 21.0)
Bottom	Left	(40.0 23.0)	(31.0 17.0)
	Right	(60.0 23.0)	(69.0 17.0)

- Create an *Anchor* material data set according to the parameters specified in Table 3.4.
- Multi-select the anchors in the drawing area. Assign the material data set by selecting the corresponding option in the *Material* drop-down menu in the *Selection explorer*.
- Define the grout body using the *Embedded beam row* button according to Table 3.5.
- Create the *Grout* material data set according to the parameters specified in Table 3.6 and assign it to the grout body.

Table 3.4 Properties of the anchor rod (node-to-node anchor)

Parameter	Name	Value	Unit
Material type	Type	Elastic	-
Normal stiffness	*EA*	$5.0 \cdot 10^5$	kN
Spacing out of plane	L_s	2.5	m

Table 3.5 Grout coordinates

Grout location		First point	Second point
Top	Left	(31.0 21.0)	(28.0 19.0)
	Right	(69.0 21.0)	(72.0 19.0)
Bottom	Left	(31.0 17.0)	(28.0 15.0)
	Right	(69.0 17.0)	(72.0 15.0)

Table 3.6 Properties of the grout body (embedded beam rows)

Parameter	Name	Value	Unit
Material type	Type	Elastic	- Stiffness
E	$7.07 \cdot 10^6$	kN/m^2	
Unit weight	γ	0	kN/m^3
Beam type	Type	Predefined	-
Predefined beam type	Type	Massive circular beam	-
Diameter	D	0.3	m
Pile spacing	$L_{spacing}$	2.5	m
Skin resistance	$T_{skin,start,max}$	400	kN/m
	$T_{skin,end,max}$	400	kN/m
Base resistance	F_{max}	0	kN
Interface stiffness factor	-	Default values	-

- Set the *Behaviour* of the embedded beam rows to *Grout body* (Figure 3.4). The connection with the anchor will be automatically established.

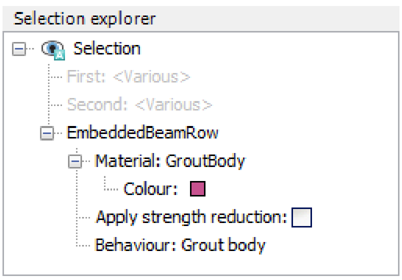

Figure 3.4 Embedded beam rows in the *Selection explorer*

- Multi-select (keep the <Ctrl> key pressed while selecting) the top node-to-node anchors and embedded beams. Right-click and select the *Group* option in the appearing menu.
- In the *Model explorer* expand the *Groups subtree*. Note that a group is created composed of the elements of the top ground anchors.
- Click on *Group_1* in the *Model explorer* and type a new name (e.g 'GroundAnchor_Top').
- Follow the same steps to create a group and to rename the bottom ground anchors.

Although the precise stress state and interaction with the soil cannot be modelled with this 2D model, it is possible in this way to estimate the stress distribution, the deformations and the stability of the structure on a global level, assuming that the grout body does not slip relative to the soil. With this model it is certainly not possible to evaluate the pullout force of the ground anchor.

Create a line load between (28.0 30.0) and (38.0 30.0).

3.2 MESH GENERATION

- Proceed to the *Mesh* mode.

Create the mesh. Use the default option for the *Element distribution* parameter (*Medium*).

View the mesh. The resulting mesh is displayed in Figure 3.5.

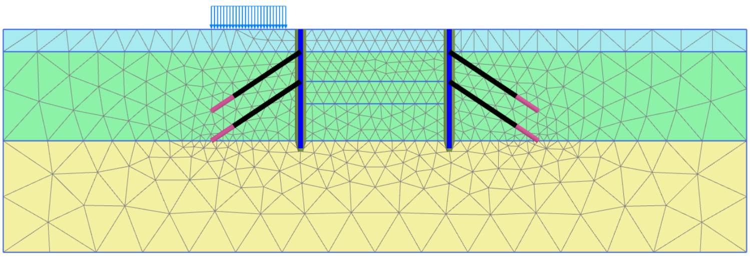

Figure 3.5 The generated mesh

3.3 CALCULATIONS

The calculation of this project consists of six phases. In the initial phase (Phase 0), the initial stresses are generated. In Phase 1, the walls are constructed and the surface loads are activated. In Phase 2, the first 3 m of the pit is excavated without connection of anchors to the wall. At this depth the excavation remains dry. In Phase 3, the first anchor is installed and pre-stressed. Phase 4 involves further excavation to a depth of 7 m. At this depth the excavation still remains dry. In Phase 5, the second anchor is installed and pre-stressed. Phase 6 is a further excavation to the final depth of 10 m including the dewatering of the excavation.

Before defining the calculation phases, the water levels to be considered in the calculation can be defined in the *Flow conditions* mode. The water level is lowered in the final excavation phase. At the side boundaries, the groundwater head remains at a level of 23.0 m. The bottom boundary of the problem should be closed. The flow of groundwater is triggered by the fact that the pit is pumped dry. At the bottom of the excavation the water pressure is zero, which means that the groundwater head is equal to the vertical level (head = 20.0 m). This condition can be met by drawing a new general phreatic level and performing a groundwater flow calculation. Activating the interfaces during the groundwater flow calculation prevents flow through the wall.

Initial phase:

The initial stress field is generated by means of the *K0 procedure* using the default K_0-values in all clusters defined automatically by the program.

- Proceed to the *Staged construction* mode.
- Initially, all structural components and loads are inactive. Hence, make sure that the plates, the node-to-node anchors, the embedded beam rows and the surface loads are deactivated.
- In the *Phases explorer* double-click the initial phase. The default parameters for the initial phase will be used. The *Phreatic* option is selected as *Pore pressure calculation type*. Note that when the pore pressures are generated by phreatic level, the full geometry of the defined phreatic level is used to generate the pore pressures.
- Click *OK* to close the *Phases* window.
- In the *Model explorer* expand the *Model conditions* subtree.
- Expand the *Water* subtree. The water level created according to the head value specified in the borehole, (*BoreholeWaterLevel_1*), is automatically assigned to *GlobalWaterLevel* (Figure 3.6).

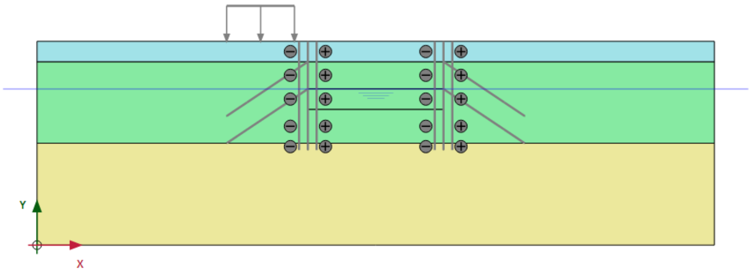

Figure 3.6 Configuration of the initial phase

Phase 1: Activation of wall and load

Add a new phase.

- In the *Staged constructions* mode activate all walls and interfaces by clicking on the checkbox in front of them in the *Model explorer*. The active elements in the project are indicated by a green check mark.
- Activate the distributed load.
- After selecting the line load assign a value of -10 to $q_{y,start,ref}$ in the *Selection explorer* (Figure 3.7).

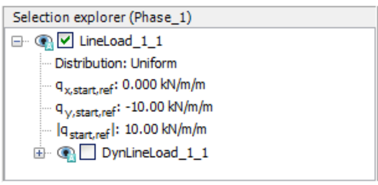

Figure 3.7 Line load in the *Selection explorer*

- The model for the phase 1 in the *Staged construction* mode is displayed in Figure 3.8.

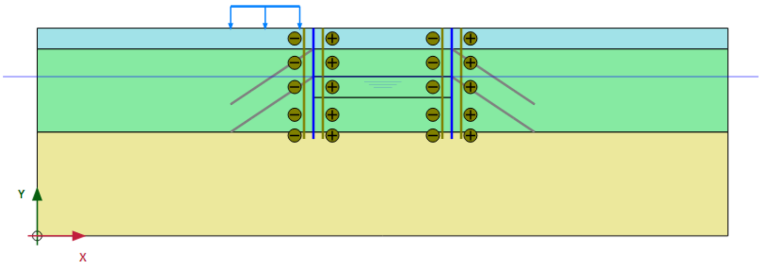

Figure 3.8 Configuration of Phase 1 in the *Staged construction* mode

Phase 2: First excavation

Add a new phase.

- In the *Staged construction* mode de-activate the upper cluster of the excavation (Figure 3.9).

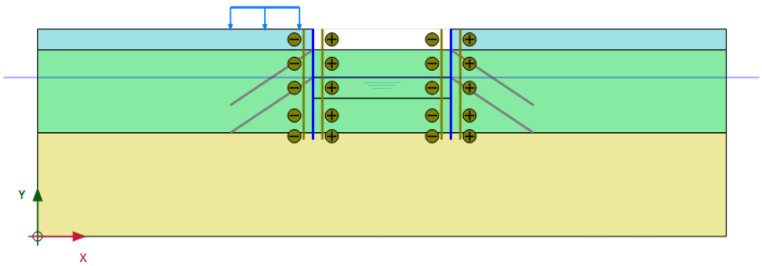

Figure 3.9 Configuration of Phase 2 in the *Staged construction* mode

Phase 3: First anchor row

Add a new phase.

- Activate the upper ground anchors by clicking on the checkbox in front of *GroundAnchors_Top* under the *Groups* subtree in the *Model explorer*.

Multi-select the top node-to-node anchors.

- In the *Selection explorer* set the *Adjust prestress* parameter to *True* and assign a pre-stress force of 500 kN.

> **Hint:** A pre-stress force is exactly matched at the end of a finished staged construction calculation and turned into an anchor force. In successive calculation phases the force is considered to be just an anchor force and can therefore further increase or decrease, depending on the development of the surrounding stresses and forces.

- The model for the phase 3 in the *Staged construction* mode is displayed in Figure

3.10.

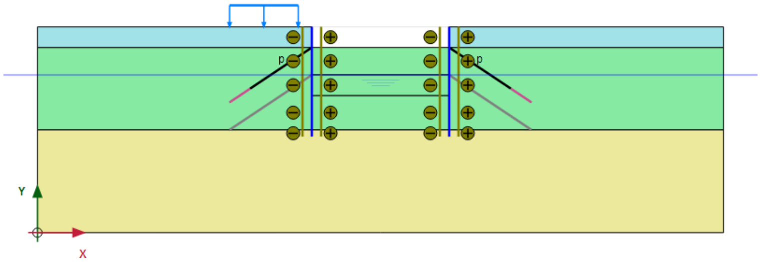

Figure 3.10 Configuration of Phase 3 in the *Staged construction* mode

Phase 4: Second excavation

Add a new phase.

- Deactivate the second cluster of the excavation. The model for the phase 4 in the *Staged construction* mode is displayed in Figure 3.11. Note that the anchors are not pre-stressed anymore.

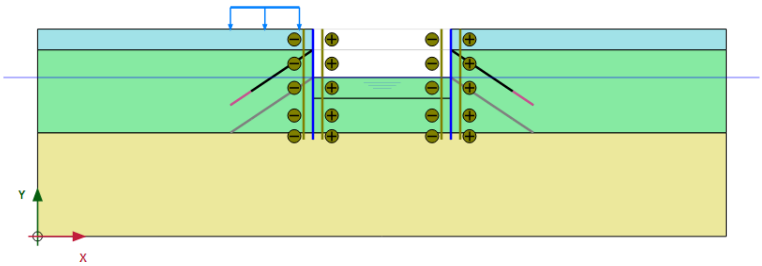

Figure 3.11 Configuration of Phase 4 in the *Staged construction* mode

Phase 5: Second anchor row

Add a new phase.

- Activate the lower ground anchors.

Select the bottom node-to-node anchors.

- In the *Selection explorer* set the *Adjust prestress* parameter to *True* and assign a pre-stress force of 1000 kN.
- The model for the phase 5 in the *Staged construction* mode is displayed in Figure 3.12.

Phase 6: Final excavation

Add a new phase.

In the *General* subtree of the *Phases* window select the *Steady state groundwater flow* option as *Pore pressure calculation type*. The default values of the remaining parameters is valid.

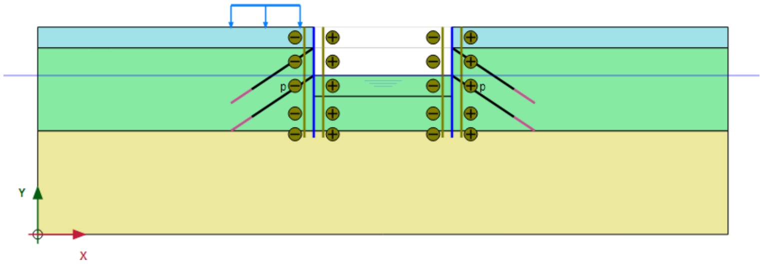

Figure 3.12 Configuration of Phase 5 in the *Staged construction* mode

- Deactivate the third cluster of the excavation.
- Click the *Flow conditions* tab to display the corresponding mode.
- In the *Model explorer* expand the Attributes library.
- Expand the *Water levels* subtree.
- Click the *Create water level* button in the side toolbar and draw a new phreatic level. Start at (0.0 23.0) and draw the phreatic level through (40.0 20.0), (60.0 20.0) and end in (100.0 23.0).
- In the *Model explorer* expand the *User water levels* subtree. Click on *UserWaterLevel_1* and type 'LoweredWaterLevel' to rename the water level created in the *Flow conditions* mode (Figure 3.13).

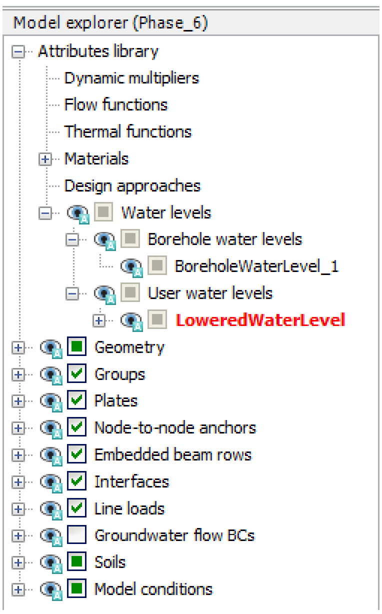

Figure 3.13 Water levels in the *Model explorer*

- Expand the *GroundwaterFlow* subtree under the *Model conditions* in the *Model explorer*. The default boundary conditions (Figure 3.14) are valid.
- In the *Water* subtree assign the *LoweredWaterLevel* to *GlobalWaterLevel*. The model and the defined water levels are displayed in Figure 3.15.

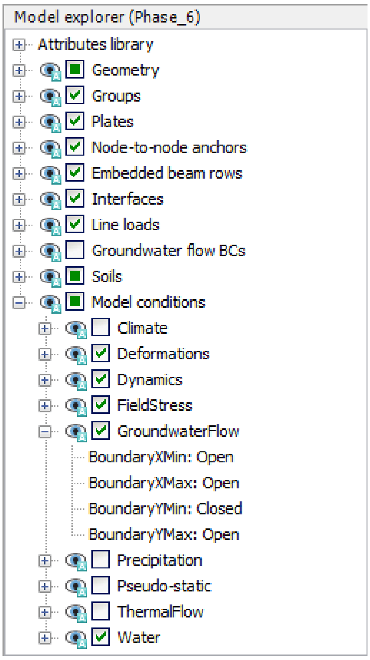

Figure 3.14 The *GroundwaterFlow* subtree under the *Model conditions* in the *Model explorer*

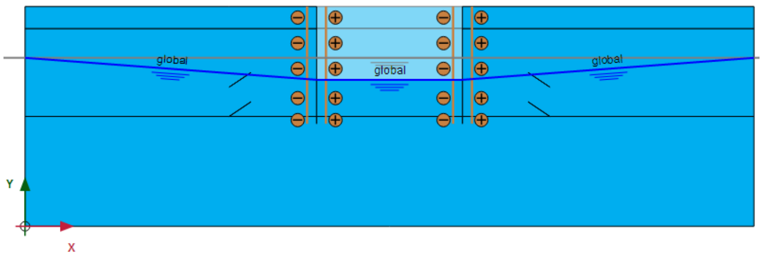

Figure 3.15 Configuration of Phase 6 in the *Flow conditions* mode

> **Hint:** Note that for *Groundwater flow* (steady or transient) the intersection points of the water level with the active model boundaries are important. The program calculates flow boundary conditions in terms of a groundwater head corresponding to the water level. The 'internal' part of the water level is not used and will be replaced by the phreatic level resulting from the groundwater flow calculation. Hence, the water level tool is just a convenient tool to create boundary conditions for a flow calculation.

- Select some characteristic points for curves (for example the connection points of the ground anchors on the diaphragm wall, such as (40.0 27.0) and (40.0 23.0)).
- Calculate the project by clicking the *Calculate* button in the *Staged construction* mode.
- Save the project after the calculation has finished.

3.4 RESULTS

Figures 3.16 to 3.20 show the deformed meshes at the end of calculation phases 2 to 6.

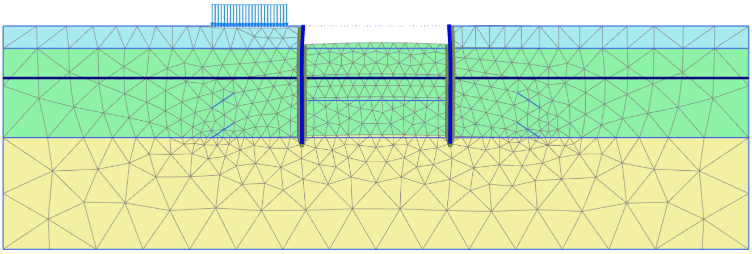

Figure 3.16 Deformed mesh (scaled up 50.0 times) - Phase 2

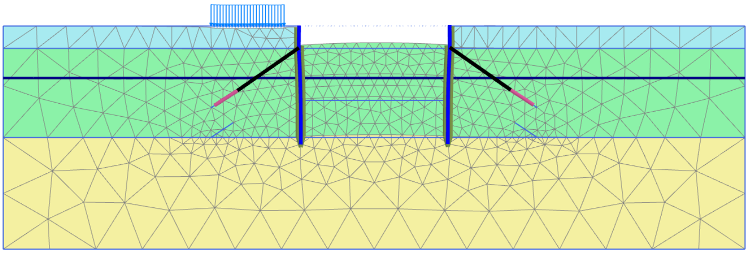

Figure 3.17 Deformed mesh (scaled up 50.0 times) - Phase 3

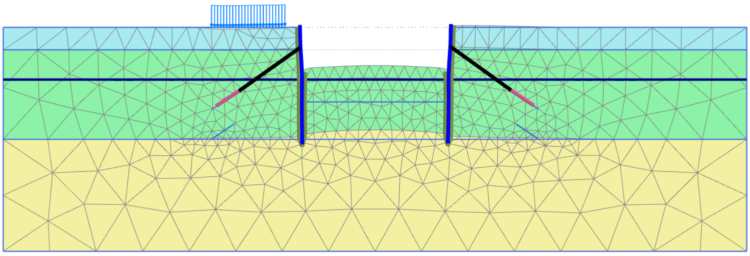

Figure 3.18 Deformed mesh (scaled up 50.0 times) - Phase 4

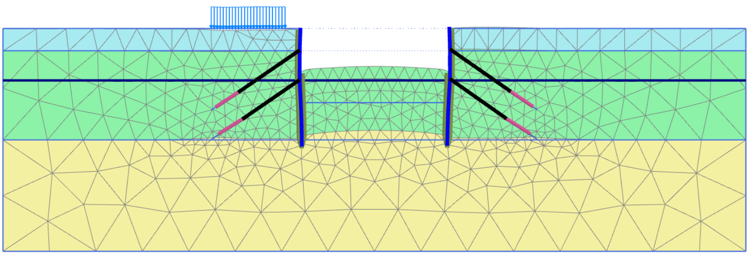

Figure 3.19 Deformed mesh (scaled up 50.0 times) - Phase 5

Figure 3.21 shows the effective principal stresses in the final situation. The passive stress state beneath the bottom of the excavation is clearly visible. It can also be seen

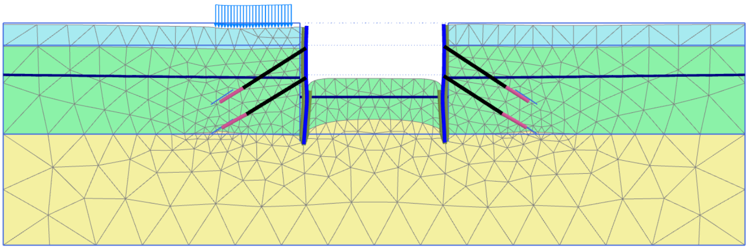

Figure 3.20 Deformed mesh (scaled up 50.0 times) - Final phase

that there are stress concentrations around the grout anchors.

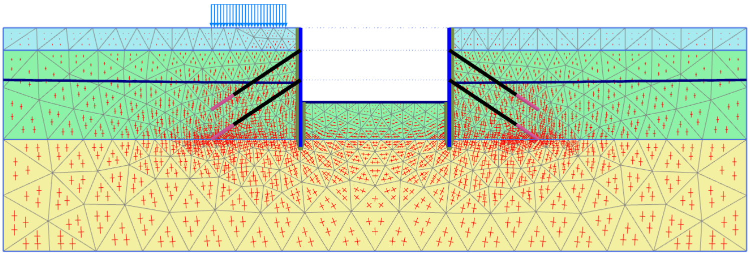

Figure 3.21 Principal effective stresses (final stage)

Figure 3.22 shows the bending moments in the diaphragm walls in the final state. The two dips in the line of moments are caused by the anchor forces.

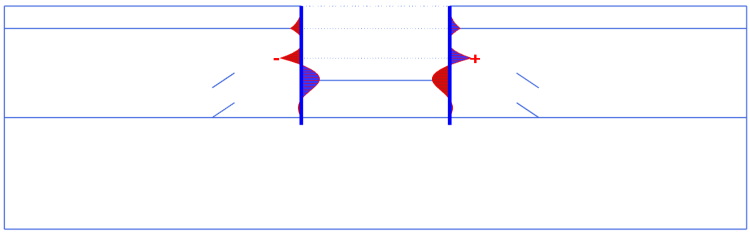

Figure 3.22 Bending moments in the diaphragm walls in the final stage

The anchor force can be viewed by double clicking the anchor. When doing this for the results of the third and the fifth calculation phase, it can be checked that the anchor force is indeed equal to the specified pre-stress force in the calculation phase they are activated. In the following phases this value might change due to the changes in the model.