Dry excavation using a tie back wall [ADV]

7.1 Introduction

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.

PLAXIS 2D allows for 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 with a groundwater flow calculation.
- Displaying the contact stresses and resulting forces in the model.
- Scaling the displayed results.

7.2 Geometry

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/m² is taken into account.

The relevant part of the soil consists of three distinct layers. From the ground surface to a depth of 3 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 particularly 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 as shown in Figure 7-1 (p. 122).

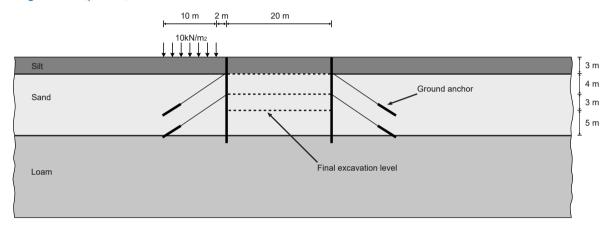


Figure 7-1: Excavation supported by tie back walls

7.3 Create new project

- Start the Input program and select Start a new project from the Quick start 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 **Contour** to $x_{min} = 0$ m, $x_{max} = 100$ m, $y_{min} = 0$ m, $y_{max} = 30$ m.
- Keep the default values for units and the constants and press OK to close the Project properties window.

7.4 Define the soil stratigraphy

To define the soil stratigraphy:

Click the **Create borehole** button \blacksquare and 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 7–2 (p. 123):

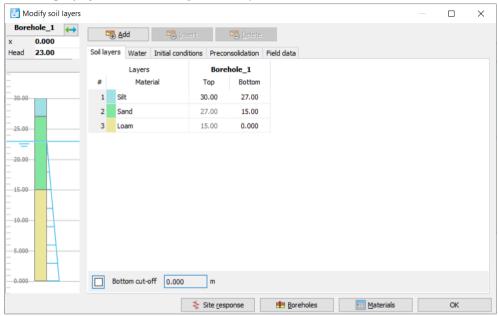


Figure 7-2: The Modify soil layers window

7.5 Create and assign material data sets

Three data sets need to be created. The materials have the following properties as shown in Table 7-1 (p. 123):

Table 7-1: Soil and interface properties

oedometer loading

Tangent stiffness for primary

| Parameter | Name | Silt | Sand | Loam | Unit |
|--|--------------------------------|----------------------|----------------------|----------------------|-------------------|
| General | | <u> </u> | | | |
| Soil model | Model | Hardening soil | Hardening soil | Hardening soil | - |
| Drainage type | Туре | Drained | Drained | Drained | - |
| Unsaturated unit weight | Yunsat | 16 | 17 | 17 | kN/m ³ |
| Saturated unit weight | Y _{sat} | 20 | 20 | 19 | kN/m ³ |
| Mechanical | | | | | |
| Secant stiffness in standard drained triaxial test | E ₅₀ ^{ref} | 20 · 10 ³ | 30 · 10 ³ | 12 · 10 ³ | kN/m ² |

 $E_{oed}^{\ ref}$

 $20 \cdot 10^{3}$

 $8 \cdot 10^3$

 $30 \cdot 10^{3}$

kN/m²

| Mechanical | | | | | |
|--|------------------------------|----------------------|----------------------|----------------------|-------------------|
| Unloading / reloading stiffness | E _{ur} ref | 60 · 10 ³ | 90 · 10 ³ | 36 · 10 ³ | kN/m ² |
| Poisson's ratio | V _{ur} | 0.2 | 0.2 | 0.2 | - |
| Power for stress-level dependency of stiffness | m | 0.5 | 0.5 | 0.8 | - |
| Cohesion (constant) | c' _{ref} | 1 | 0 | 5 | kN/m ² |
| Friction angle | φ' | 30 | 34 | 29 | 0 |
| Dilatancy angle | Ψ | 0 | 4 | 0 | 0 |
| K ₀ -value for normal consolidation | K ₀ ^{nc} | 0.5 | 0.4408 | 0.5152 | 1 |
| Groundwater | | | | | |
| Classification type | - | USDA | USDA | USDA | - |
| SWCC fitting method | - | Van Genuchten | Van Genuchten | Van Genuchten | - |
| Soil class | - | Silt | Sand | Loam | - |
| < 2μm | - | 6.0 | 4.0 | 20.0 | % |
| 2μm - 50μm | - | 87.0 | 4.0 | 40.0 | % |
| 50μm - 2mm | - | 7.0 | 92.0 | 40.0 | % |
| Flow parameters - 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 | | | | | |
| Strength determination | - | Manual | Manual | Rigid | - |
| Interface reduction factor | R _{inter} | 0.65 | 0.70 | 1.0 | - |
| Consider gap closure | - | Yes | yes | yes | |
| Initial | | | | | |

| Interfaces | | | | | |
|----------------------------|--------------------|--------|--------|-------|---|
| Strength determination | - | Manual | Manual | Rigid | - |
| Interface reduction factor | R _{inter} | 0.65 | 0.70 | 1.0 | - |
| Consider gap closure | - | Yes | yes | yes | |

| Initial | | | | | |
|------------------------------|-----|-----------|-----------|-----------|-------------------|
| K ₀ determination | - | Automatic | Automatic | Automatic | - |
| Pre-overburden pressure | POP | 0 | 0 | 25 | kN/m ² |
| Over-consolidation ratio | OCR | 1.0 | 1.0 | 1.0 | - |

¹⁾ Define three data sets for soil and interfaces with the parameters given in Table 7–1 (p. <u>123)</u>.

Assign the material data sets to the corresponding soil layers (Figure 7–2 (p. 123)).

7.6 Define the structural elements

The creation of diaphragm walls, excavation levels, ground anchor and surface load is described below.

Click the **Structures** tab to proceed with the input of structural elements in the **Structures mode**

7.6.1 To define the diaphragm wall and interfaces:

A diaphragm wall with the following material properties as shown in Table 7-2 (p. 125) has to be defined:

Table 7-2: Material properties of the diaphragm wall (plate)

| Parameter | Name | Value | Unit |
|------------------|------|---------|--------|
| General | | | |
| Material type | - | Elastic | - |
| Weight | W | 8.3 | kN/m/m |
| Prevent punching | - | True | - |

| Mechanical | | | |
|-------------------|--------|-----------------------|--------|
| Isotropic | - | True | - |
| Axial stiffness | EA₁ | 12 · 10 ⁶ | kN/m |
| Bending stiffness | EI | 120 · 10 ³ | kNm²/m |
| Poisson's ratio | v (nu) | 0.15 | - |

- In the **Structures mode**, model the diaphragm walls as plates passing through (40 30) (40 14) and (60 30) - (60 14).
- 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 as shown in Figure 7-3 (p. 125):



Figure 7-3: Material assignment in the Selection explorer

Click the Add 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 7-2 (p. 125). The concrete has a Young's modulus of 35 GN/m² and the wall is 0.35 m thick.
- Assign positive and negative interfaces to the geometry lines created to represent the diaphragm walls.

7.6.2 Definition of excavation levels

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 \$\sqrt{\text{through (40 23)}} and (60 23).
- Define the third excavation phase by drawing a line \$\square\$ through (40 20) and (60 20).

7.6.3 Defining the ground anchor

A ground anchor can be modelled by a combination of a node-to-node anchor and an embedded beam. The embedded beam 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.

The coordinates of the anchor and grout body are listed in Table 7–3 (p. 126), Table 7–4 (p. 126) and Table 7-5 (p. 126) and material properties of grout body are shown in Table 7-6 (p. 127).

Table 7-3: Node to node anchor coordinates

| Anchor location | Name | First point | Second point |
|-----------------|-------|-------------|--------------|
| Тор | Left | (40 27) | (31 21) |
| | Right | (60 27) | (69 21) |
| Bottom | Left | (40 23) | (31 17) |
| | Right | (60 23) | (69 17) |

Table 7-4: Properties of the anchor rod (node-to-node anchor)

| Parameter | Name | Value | Unit |
|----------------------|----------------------|-----------------------|------|
| Material type | - | Elastic | - |
| Axial stiffness | EA | 500 · 10 ³ | kN |
| Out-of-plane spacing | L _{spacing} | 2.5 | m |

Table 7-5: Grout coordinates

| Anchor location | Name | First point | Second point |
|-----------------|-------|-------------|--------------|
| Тор | Left | (31 21) | (28 19) |
| ТОР | Right | (69 21) | (72 19) |

| Anchor location | Name | First point | Second point |
|-----------------|-------|-------------|--------------|
| Bottom | Left | (31 17) | (28 15) |
| Bottom | Right | (69 17) | (72 15) |

Table 7-6: Properties of the grout body (embedded beam)

| Parameter | Name | Value | Unit |
|---------------|------|---------|-------------------|
| General | | | |
| Material type | - | Elastic | - |
| Unit weight | γ | 0 | kN/m ³ |

| Mechanical | | | |
|----------------------------|-------------------------------|------------------------|-------------------|
| Pile spacing | $L_{spacing}$ | 2.5 | m |
| Beam type | - | Predefined | - |
| Predefined beam type | - | Solid circular beam | - |
| Diameter | D | 0.3 | m |
| Stiffness | Е | 7.07 · 10 ⁶ | kN/m ² |
| | Distribution | Linear | - |
| Axial Skin resistance | T _{skin, start, max} | 400 | kN/m |
| | T _{skin, end, max} | 400 | kN/m |
| Lateral resistance | Lateral resistance | Unlimited | - |
| Base resistance | F _{max} | 0 | kN |
| Interface stiffness factor | Default values | Yes | - |

- Define the node-to-node anchors according to Table 7-3 (p. 126).
- Create an **Anchor** material data set according to the parameters specified in <u>Table 7–4 (p.</u> 126).
- 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** button \(^{\strace{1}{2}}\) according to \(\frac{Table 7-5}{2}\) (p. 126).
- 5 Create the **Grout** material data set according to the parameters specified in Table 7-6 (p. 127) and assign it to the grout body.
- Set the **Behaviour** of the embedded beam to **Grout body** as shown in Figure 7-4 (p. 128).

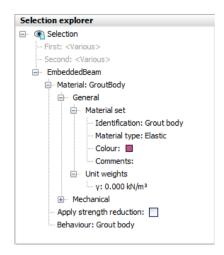


Figure 7-4: Embedded beam in the Selection explorer

The connection with the anchor will be automatically established.

- Multi-select (keep Ctrl pressed while selecting) the top node-to-node anchors and embedded beams. Right-click on the selected region and select the Group option in the context 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).
- 10 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.

7.6.4 Definition of distributed load

Create a line load * between (28 30) and (38 30).

7.7 Generate the mesh

In order to generate the mesh, follow these steps:

- Proceed to the Mesh mode.
- Click the **Generate mesh** button on the side toolbar. Use the default option for the Element distribution parameter (Medium).
- Click the **View mesh** button <a> to view the mesh as shown in Figure 7–5 (p. 129).

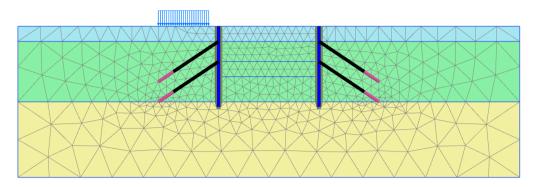


Figure 7-5: The generated mesh

7.8 Define and perform the calculation

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 prestressed. 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.

7.8.1 Initial phase

The initial stress field is generated in all the clusters by means of the KO procedure using the default K_0 -values which is 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 .

The model for initial phase is shown in Figure 7–6 (p. 130).

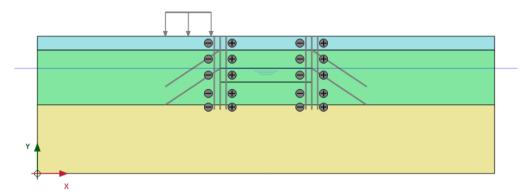


Figure 7-6: Configuration of the initial phase

7.8.2 Phase 1: Activation of wall and load

- Click the **Add phase** button **to create 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 $q_{v, start, ref}$ in the **Selection explorer** shown in Figure 7-7 (p. 130):

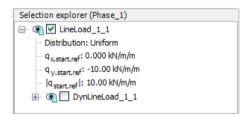


Figure 7-7: Line load in the Selection explorer

The model for the phase 1 in the **Staged construction mode** is displayed in Figure 7–8 (p. 130):

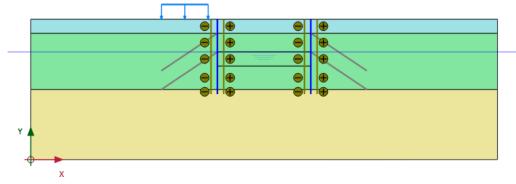


Figure 7-8: Configuration of Phase 1 in the Staged construction mode

7.8.3 Phase 2: First excavation

- Click the **Add phase** button in the **Phases explorer** to add a new phase.
- In the Staged construction mode deactivate the upper cluster of the excavation

The model for the first excavation phase is shown in Figure 7–9 (p. 131):

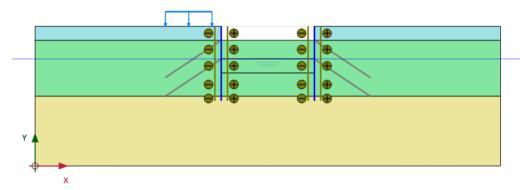


Figure 7-9: Configuration of Phase 2 in the Staged construction mode

7.8.4 Phase 3: First anchor row

- Click the **Add phase** button **in the Phases explorer**.
- 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 prestress force of 500 kN.
 - Note: 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 shown in Figure 7-10 (p. 132):

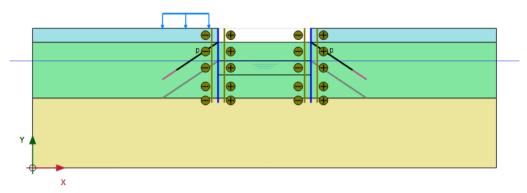


Figure 7-10: Configuration of Phase 3 in the Staged construction mode

7.8.5 Phase 4: Second excavation

- Click the **Add phase** button to 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 7–11 (p. 132):

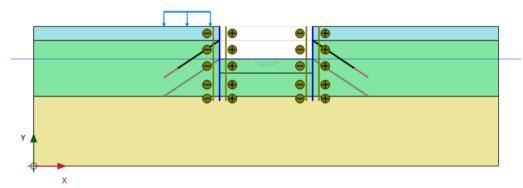


Figure 7-11: Configuration of Phase 4 in the Staged construction mode

Note that the anchors are not pre-stressed anymore.

7.8.6 Phase 5: Second anchor row

- Click the **Add phase** button to 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 prestress force of 1000 kN.

The model for the phase 5 in the **Staged construction** mode is displayed in Figure 7–12 (p. 133):

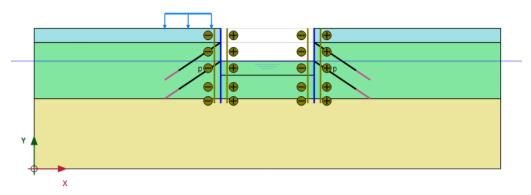


Figure 7-12: Configuration of Phase 5 in the Staged construction mode

7.8.7 Phase 6: Final excavation

- Click the **Add phase** button **to add a new phase**.
- In the Phases window, within General > Pore pressure calculation type select the Steady state groundwater flow option 🖹. The default values of the remaining parameters are valid.
- 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 23) and draw the phreatic level through (40 20), (60 20) and end in (100 23).
- 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 as shown in Figure 7–13 (p. 133).

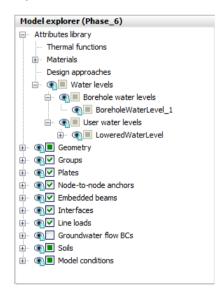


Figure 7–13: Water levels in the Model explorer

In the Model explorer expand Model conditions > GroundwaterFlow. The default boundary conditions are valid which is shown in Figure 7-14 (p. 134).

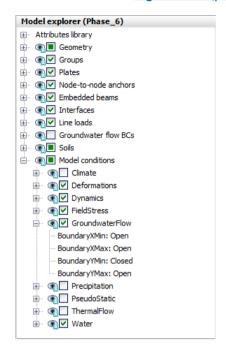


Figure 7–14: The GroundwaterFlow subtree under the Model conditions in the Model explorer

In the Water subtree assign the LoweredWaterLevel to GlobalWaterLevel.

The model and the defined water levels are displayed in Figure 7–15 (p. 134):

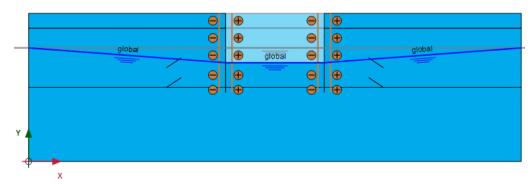


Figure 7-15: Configuration of Phase 6 in the Flow conditions mode

1 Note: 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.

7.8.8 Execute the calculation

- Click the **Select points for curves** button \checkmark in the side toolbar.
- Select some characteristic points for curves (for example the connection points of the ground anchors on the diaphragm wall, such as (40 27) and (40 23).
- Click the **Calculate** button **I** to calculate the project.
- After the calculation has finished, save the project by clicking the Save button ...

7.9 Results

The deformed meshes at the end of calculation phase 2 to phase 6 are shown in Figure 7–16 (p. 135), Figure 7–17 (p. 135), Figure 7–18 (p. 136), Figure 7–19 (p. 136) and Figure 7–20 (p. 136):

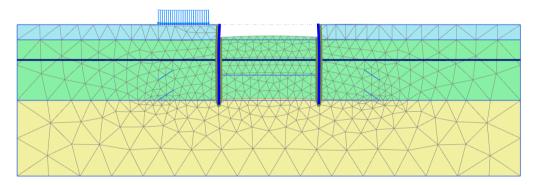


Figure 7-16: Deformed mesh (scaled up 50.0 times) - Phase 2

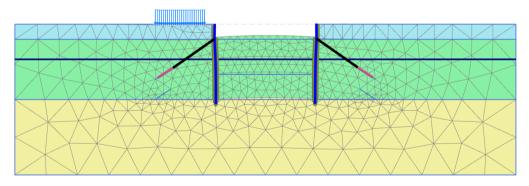


Figure 7-17: Deformed mesh (scaled up 50.0 times) - Phase 3

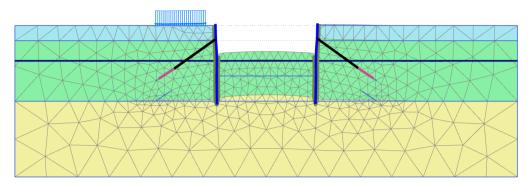


Figure 7-18: Deformed mesh (scaled up 50.0 times) - Phase 4

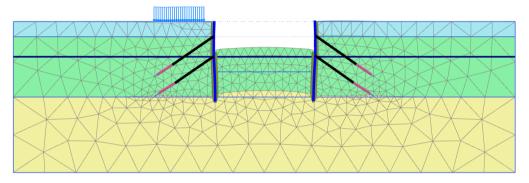


Figure 7-19: Deformed mesh (scaled up 50.0 times) - Phase 5

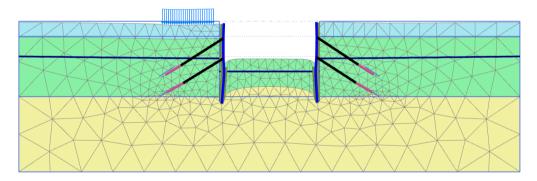


Figure 7-20: Deformed mesh (scaled up 50.0 times) - Final phase

Figure 7–21 (p. 136) 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 that there are stress concentrations around the grout anchors.

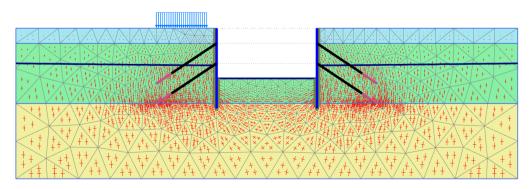


Figure 7-21: Principal effective stresses (final stage)

Figure 7–22 (p. 137) 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.

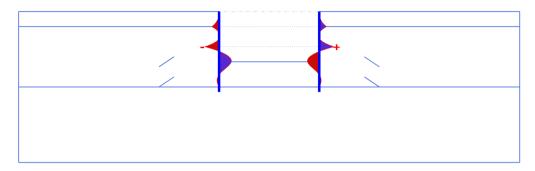


Figure 7–22: Bending moments in the diaphragm walls in the final stage

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.