TIED-BACK EXCAVATION

Using the HSsmall model

INTRODUCTION

A building pit was constructed in the south of the Netherlands. The pit is 15 m deep and 30 m wide. A diaphragm wall is constructed using 100 cm diameter bored piles; the wall is anchored by two rows of pre-stressed ground anchors. In this exercise the construction of this building pit is simulated and the deformation and bending moments of the wall are evaluated.

The upper 40 m of the subsoil consists of a more or less homogeneous layer of medium dense fine sand with a unit weight of 18 kN/m^3. Triaxial test data of a representative soil sample is given in figure 2. Underneath this layer there is very stiff layer of gravel, which is not to be included in the model. The groundwater table is very deep and does not play a role in this analysis.

AIMS

- Using interface elements
- Using ground anchors
- Pre-stressing of anchors
- Combination of structural elements

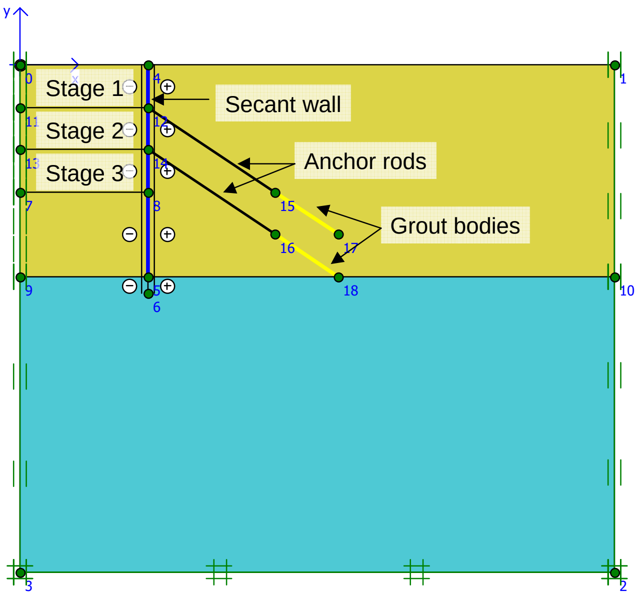


Figure 1: Geometry for tied-back excavation

MATERIAL PARAMETERS

Determination of stiffness & strength properties (sand)

In this exercise the HSsmall model is used and the model parameters for the sand layer have been extracted from the triaxial test data (see figure 2). The HSsmall model takes into account the stress-dependency of soil stiffness, elasto-plastic behaviour under both compression loading and shear loading and increased stiffness in areas with very low strain levels. The soil parameters can be found in table 1, while the determination of the soil parameters can be found in appendix A.

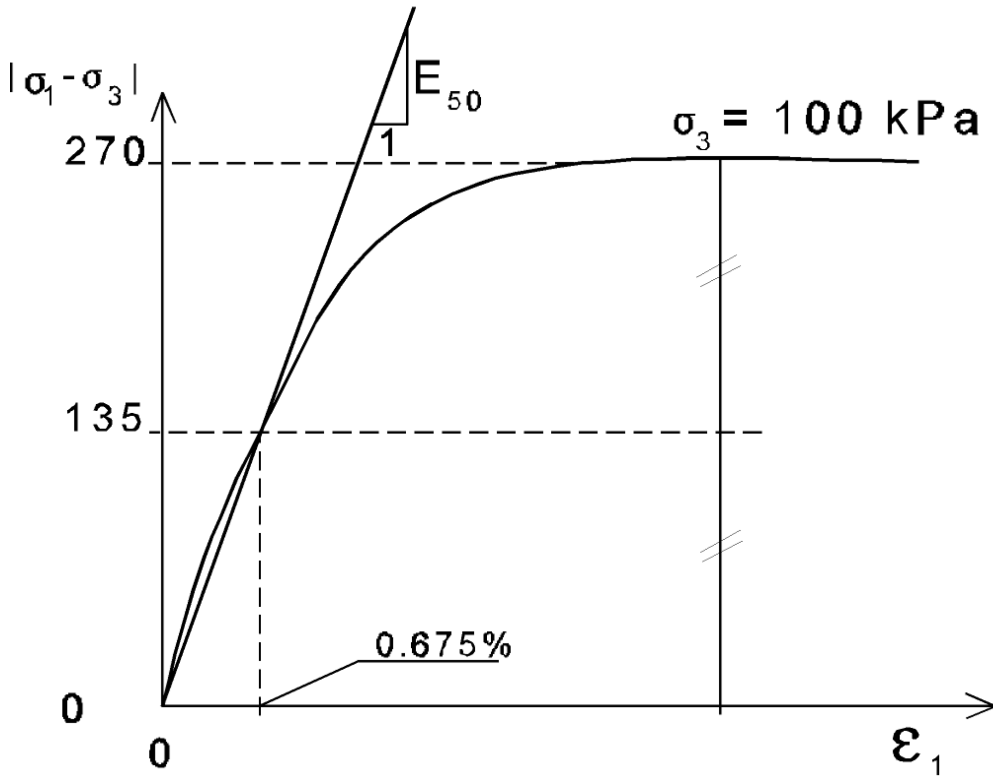


Figure 2: Triaxial test data for the sand layer

Secant wall

The secant wall consists of 100cm diameter bored piles with an intermediate distance of 80cm, hence there is a 20cm overlap of the piles. This configuration is taken this into account for the determination of the cross sectional area (A) and moment of inertia (I) per meter out-of-plane (see Appendix B). The concrete stiffness is E_c=2.7•107 kN/m2 with a specific weight γ=16 kN/m3, which leads to the material parameters as given in Table 2. The determination of the stiffness parameters can be found in Appendix A.

Table 1: Soil parameters for the HSsmall model

Parameter	Symbol	Sand(R_{inter}=0.6)	Sand(R_{inter}=1.0)	Unit
Material model	Model	HSsmall	HSsmall	–
Type of behaviour	Type	Drained	Drained	–
Unsaturated weight	γ_{unsat}	18.0	18.0	kN/m^3
Saturated weight	γ_{sat}	18.0	18.0	kN/m^3
Drained triaxial test stiffness	E_{50}^{ref}	2.0·10^4	2.0·10^4	kN/m^2
Drained primary oedometer stiffness	E_{oed}^{ref}	2.0·10^4	2.0·10^4	kN/m^2
Unloading/reloading stiffness	E_{ur}^{ref}	8.0·10^4	8.0·10^4	kN/m^2
Power for stress-dependent stiffness	m	0.5	0.5	–
Cohesion	c'	1.0	1.0	kN/m^2
Friction angle	φ'	35	35	º
Dilatancy angle	ψ	5	5	º
Small-strain shear modulus	G_0^{ref}	10.0·10^4	10.0·10^4	kN/m^2
Threshold shear strain	$\gamma_{0.7}$	1.5·10^{-4}	1.5·10^{-4}	–
Unloading/reloading Poisson's ratio	ν'_{ur}	default	default	–
Reference stress	p_{ref}	default	default	kN/m^2
Coefficient for lateral stress under primary loading	K_0^{NC}	default	default	–
Interface strength reduction	R_{inter}	0.6	rigid	–
Coefficient for lateral initial stress	K_0	automatic	automatic	–

Ground anchors

The anchors are made of 32mm diameter steel bars at an intermediate distance of 1m. The steel bars have a stiffness of E_s=2.1*10^8 kN/m^2. The anchors have an ultimate strength of 605 kN per anchor. In combination with a secant wall the anchors may be prestressed to a maximum level of 60% of the ultimate strength, hence up to 363 kN per anchor. The maximum compression force of the anchor is not important as the anchors will not be loaded under compression. The grout body that forms the bonded length of the anchor behaves relatively weak under tension compared to the steel bar inside. Therefore it is assumed that both stiffness and strength of the bonded part of the anchor are fully determined by the steel bar. This leads to the material properties for both the anchor rod (free length) and grout body (bonded length) as given in tables 3 and 4.

Table 2: Properties of the secant wall (plate)

Parameter	Symbol	Secant wall	Unit
Material behaviour	Material type	Elastic	–
Axial stiffness	EA	$2*10^7$	kN/m
Flexural stiffness	EI	$1.67*10^6$	kN/m^2/m
Weight	w	15.0	kN/m/m
Poisson's ratio	ν	0.15	–

Table 3: Properties of the anchor rods (node-to-node anchors)

Parameter	Symbol	Anchor rod	Unit
Material behaviour	Material type	Elastoplastic	–
Axial stiffness	EA	$1.7*10^5$	kN
Spacing	$L_{spacing}$	1.0	m
Max. tension force	$\|F_{max,tens}\|$	605	kN
Max. compression force	$\|F_{max,comp}\|$	605	kN

Table 4: Properties of the grout bodies (geotextiles)

Parameter	Symbol	Grout body	Unit
Material behaviour	Material type	Elastoplastic	–
Axial stiffness	EA	$1.7*10^5$	kN/m
Max. tension force	N_p	605	kN/m

GEOMETRY INPUT

- Start a new project

Project properties

- Accept the default values in the *Project* tab sheet of the *Project properties* (15-node elements). For the dimensions see figure 3.

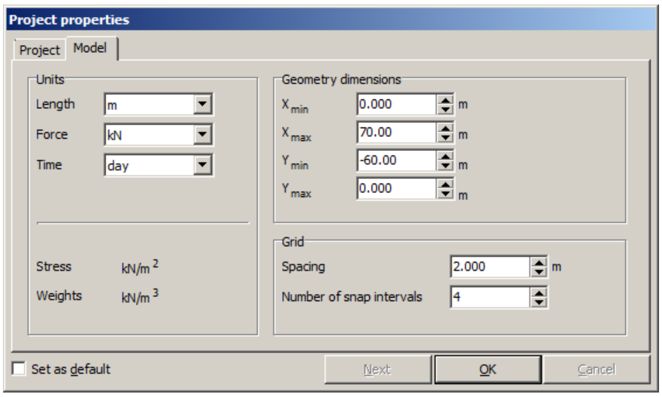


Figure 3: Project propeties, tabsheet Model

Geometry

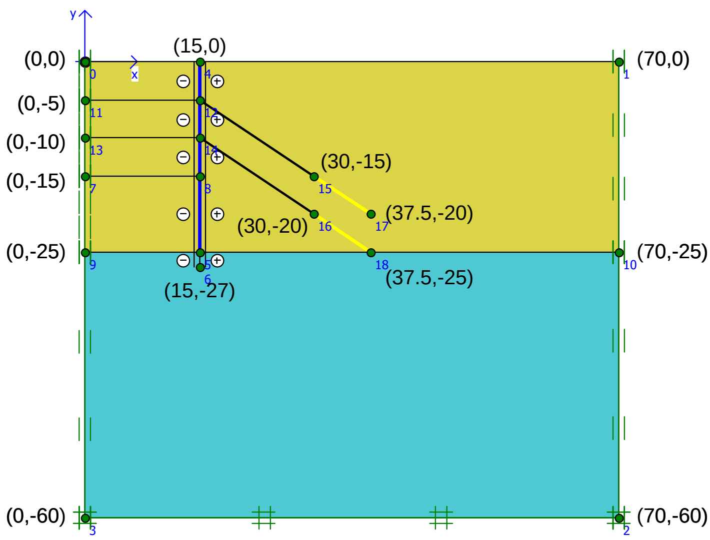


Figure 4: Geometry of the model

- Click the *Geometry line* button and draw the geometry contour and soil layers as specified in figure 4.

- Click the *Plate* button and draw the secant wall from (15, 0) to (15, -25).
- Click the *Interface* button and draw the interface from (15,0) to (15, -27) and back to (15,0). This creates an interface on both sides of the secant wall.
- Click the *Geotextile* button and insert both grout bodies.
- Click the *Node-to-node* anchor button and insert both anchor rods. These anchors connect the beginning of the grout bodies to the wall.
- Finally, click the *Geometry line* button again to introduce the two levels of excavation.

Hints:	As interfaces can be introduced on both sides of a geometry line, one should pay attention to the arrows on the cursor. These arrows indicate where the program will locate the interfaces.
>	Please note that the interface is extended for a short distance underneath the beam. This is done to overcome a singular point at the bottom of the wall.

Hint:	It is not necessary to create a geometry line before creating plates, geogrids or anchors. When drawing a plate or geogrid, a geometry line is automatically added. Anchors do not create corresponding geometry lines. This is not necessary since anchors do not interact with the underlying soil.

Fixities

- Click the *Standard fixities* button to apply standard boundary conditions.

Material properties

- Enter the material properties for the four soil data sets, as determined in table 1of this exercise.
- After entering all properties for the three soil types, drag and drop the properties to the appropriate clusters.
- Enter material properties for the plates, anchors and 'geogrids' as indicated in tables 2, 3and 4.

Mesh generation

- From the *Mesh* menu, set the *Global coarseness* to *Medium* and press the *Generate* button. This will result in a mesh as shown in figure 5.

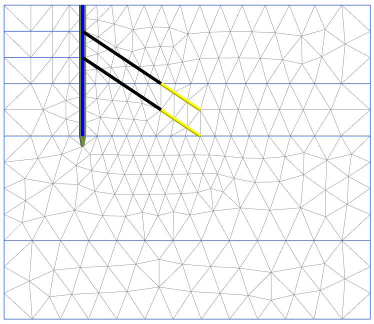


Figure 5: Medium finite element mesh

- Select the geogrid and plate elements and press *Refine line* from the *Mesh* menu. This will result in a refinement around the selected lines as shown in Figure 6.

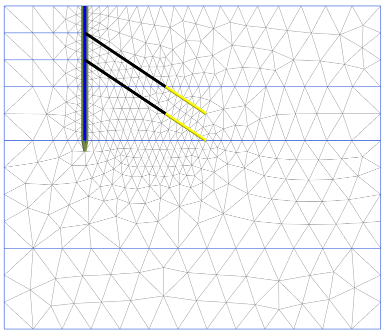


Figure 6: Refined finite element mesh

CALCULATION

- When starting the calculation program choose the *Classical mode*.

The entire construction process consists of five phases. Define the phases, as shown graphically below. For each phase, use the *Plastic calculation*, *Staged construction*.

Initial phase

For the initial phase choose the K_0procedure for calculating the initial stresses. As the phreatic line is located below the geometry the generation of initial pore pressures can be skipped and since it's not necessary to switch off any soil for the initial situation it is not needed to define the initial phase.

Phase 1

- In the first phase, the diaphragm wall is activated and the first excavation takes place.
- Note that though the the interfaces along the wall are activated automatically with the activation of the wall, the extensions below the diaphragm wall have to be activated manually.

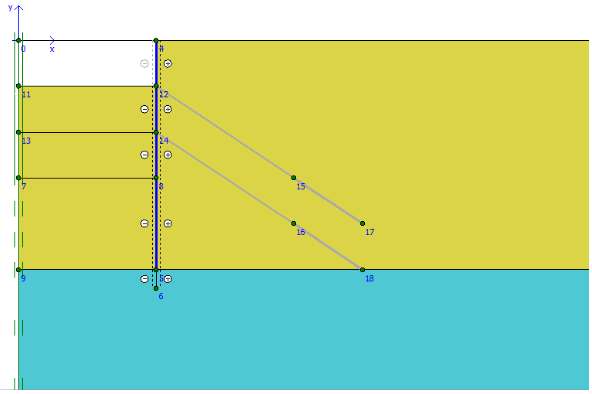


Figure 7: Phase 1: activation of the wall and 1st excavation

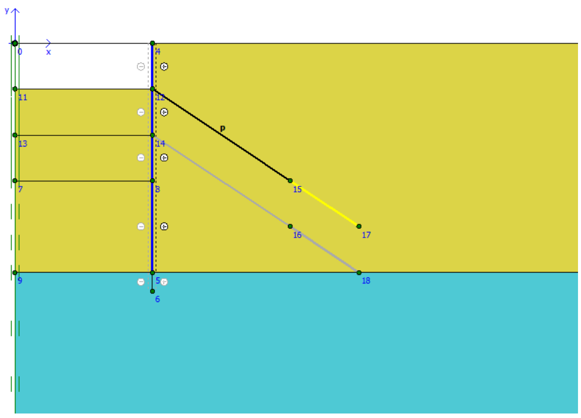


Figure 8: Phase 2: activation and presstressing of the 1st anchor

Phase 2

In the second phase, a new option is used, namely the prestressing of anchors.

- First the grout-body (the geogrid) is switched on by clicking on the 'geogrid' element. The element will appear in yellow as soon as it is switched on. The light grey colour indicates non-active elements.

- Now that the grout-body is active, the anchor element needs to be prestressed. By double clicking on a node-to-node anchor a window will appear as shown in figure 9.
- Select the option *Adjust prestress*, fill in a prestress force of 300 kN/m (tension) and press *OK*.
- In the geometry a black node-to-node anchor indicates that the anchor is activated. The letter P indicates that a prestress force will be active in the anchor.

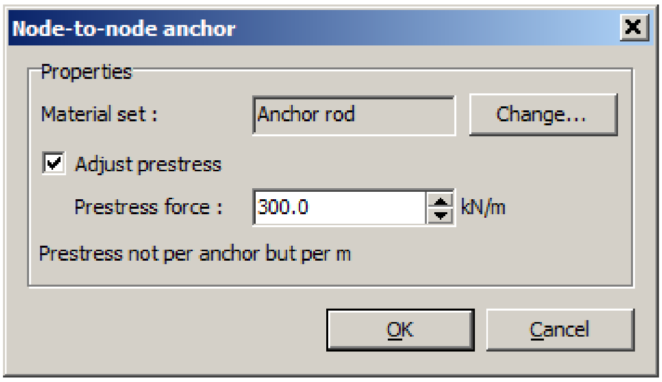


Figure 9: Node-to-node anchor properties

Phase 3, 4 and 5

Now define the remaining phases according to figures 10, 11 and 12.

- In phase 3 excavate the second part of the excavation
- In phase 4 activate the lower anchor and prestress it to 300 kN/m
- In phase 5 excavate the remaining 3rd part.

Hint: When processing an anchor in a certain calculation phase the anchor force will exactly match the prestress force at the end of that phase. In following calculation phases without prestressing, the anchor force will be influenced by the excavation process

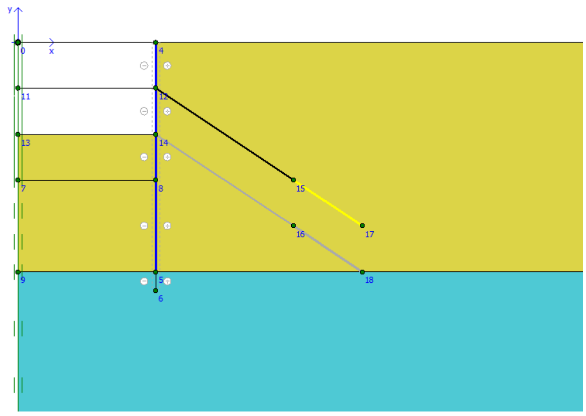


Figure 10: Phase 3: Second excavation

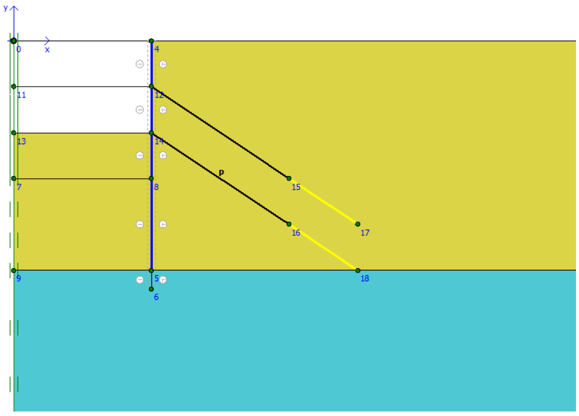


Figure 11: Phase 4: Activation and prestressing of 2nd anchor

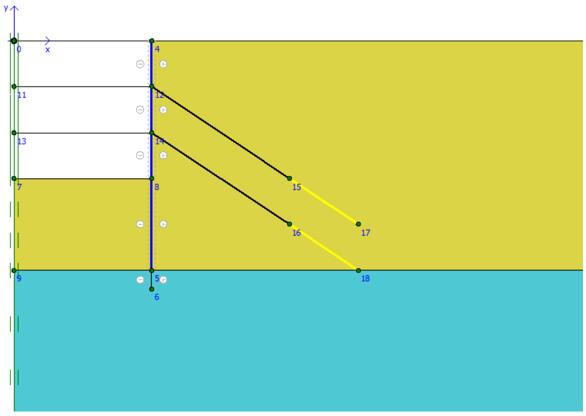


Figure 12: Phase 5: Final excavation

INSPECT OUTPUT

The results of fase 5 is presented in Figure 13. After this final stage the excavation bottom heave calculated is about 5 cm.

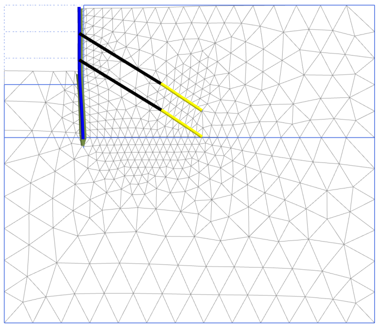


Figure 13: Deformed mesh (phase 5)

- By double clicking on the node-to-node anchors, Plaxis will present a table, in which the stress in all anchors may be inspected. Anchor forces are approximately 340 kN where the lower anchor has a slightly higher anchor force than the upper anchor.

When double-clicking on one of the geogrids the change of axial forces within the grout body can be investigated. What is immediately noticeable is that the axial force at the connection with the anchor rod is significantly lower than the force in the anchor rod itself. This is due the fact that the end of the anchor rod is not only connected to the grout body, but also to several soil elements surrounding the end of the anchor rod. Therefore part of the anchor force is transferred directly to those soil elements while part of the anchor force is transferred to the geotextile representing the grout body. The amount of force transferred to the soil depends on the stiffness of the soil; in this exercise it is 25-35% of the anchor force. However, this effect has very little influence on other calculation results. That is, it is not so important for other calculation results how the anchor rod transfers its force; directly to the soil or by means of the grout body.

- By double-clicking on the wall the structural forces in the wall can be inspected. The maximum bending moment should be in the order of 350 kNm/m (figure 14)

- When double-clicking on an interface only the results of part of the interface can be seen. In order to see the results for the whole interface chain, keep Ctrl + Shift pressed on the keyboard while double-clicking on the interface. In figure 15the left side are the passive earth pressures and the right side are the active earth pressures. It can be seen that only a small part of the maximum passive earth pressures has been mobilized at this stage.

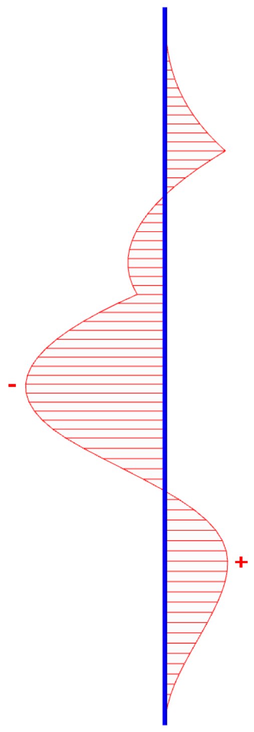


Figure 14: Bending moments in the secant wall

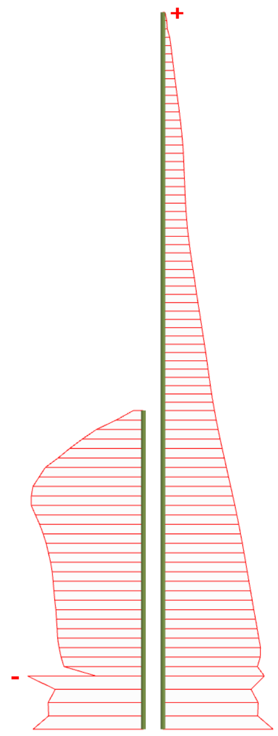


Figure 15: Effective normal stresses in the interface

Geometry size

For any project the geometry has to be made sufficiently large so that the boudary conditions have no influence on the calculation results. This means in practice that close to the boundaries (with exception of a axis of symmetry) displacements should be small and stresses should be undisturbed. When using the HSsmall model there is an interesting plot that can be used to check this.

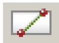
- From the *Stresses* menu choose the option *State parameters* and then *G/G$_{ur}$*.

This plot shows the actual shear stiffness divided by the unloading/reloading shear stiffnes at engineering strain level. For areas with very small deformations the stiffness will be high (small strain stiffness) and so the value of G/G$_{ur}$> 1. Hence, the geometry is sufficiently large if next to the boundaries, with exception of the axis of symmetry, G/G$_{ur}$> 1, which indeed is the case.

Hint:	State parameters are additional quantities that relate to the state of the material in the current calculation step, taking into account the stress history. Examples of state parameters are the isotropic overconsolidation pressure (p$_p$) and the hardening parameter γ_p that specifies the maximum shear strain level reach in the stress history.

Surface settlements

In Plaxis Output it is possible to see calculation results in a user-defined cross section. This feature will be used to check the surface settlements behind the secant wall.

- Click the *Cross section* button . The *Cross section points* window appears, see figure 16.

It is possible to draw a cross section by hand and check in the *Cross section points* window what the coordinates are of the start and end point of the cross section. However, it is also possible to position the cross section at a specific location by defining the coordinates of the start and end point manually.

- Move the mouse to the *Cross section points* window and fill in the coordinates (15, -0.1) for the first point and (70, -0.1) for the second point and press *OK*. This will create a cross section from the secant wall until the right boundary of the model just below the soil surface. The cross section will open in a new window.
- From the *Deformations* menu select *Total displacements* and then u_y to see the vertical displacements of the soil surface. The maximum settlement is 12-13 mm, see figure 17.

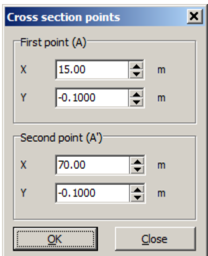


Figure 16: Cross section points window

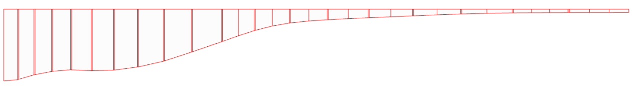


Figure 17: Vertical displacements behind the secant wall

APPENDIX A: DETERMINATION OF SOIL PARAMETERS FROM TRIAXIAL TEST

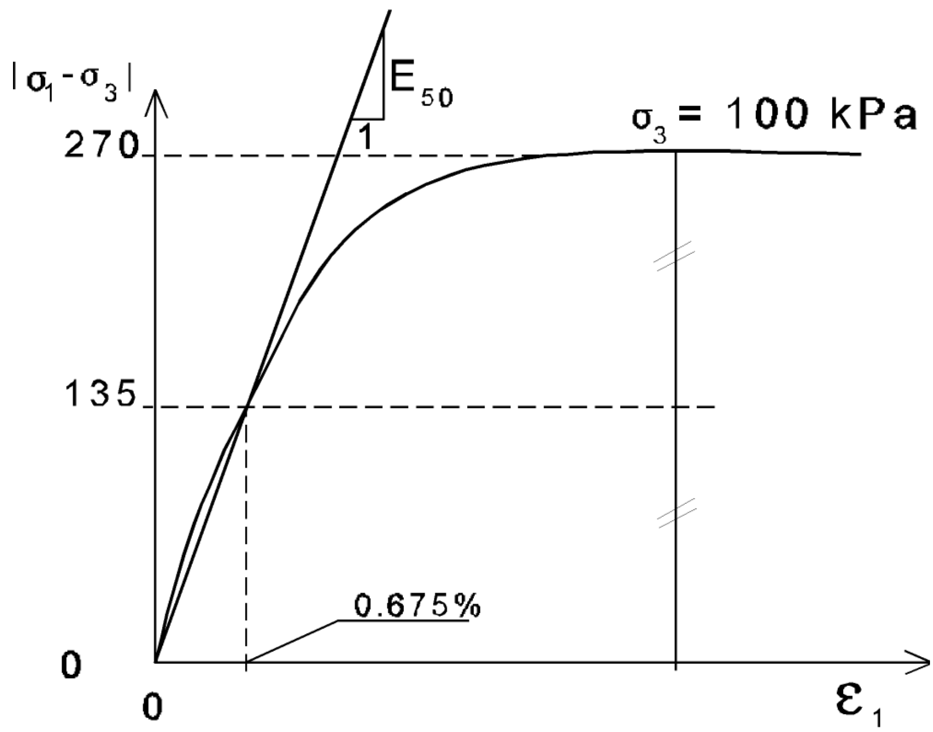


Figure 18: Triaxial test for sand layer

Strength parameters

Fill in $\sigma_1$and $\sigma_3$in the Mohr-Coulomb criteria:

$$\sigma_1 - \sigma_3 = (\sigma_1 + \sigma_3)\sin\varphi + 2c\cos\varphi$$

Since the cohesion will be small, assume c = 0:

$\frac{\sigma_1-\sigma_3}{\sigma_1+\sigma_3} = \sin\varphi$

$\frac{370-100}{370+100} = \sin\varphi$

φ = 35^o

$\psi = \varphi - 30$ = 5^o

For reasons of numerical stability, use c = 1 kPa

Stiffness parameters

Since excavation is considered in this exercise, the input of Young's modulus E should be based on unloading, rather than on primary loading. For the same reason, Poisson's ratio should also be based on unloading, which results in a somewhat lower value.

The triaxial test has a cell pressure σ_3 = 100 kPa. This corresponds with reference pressure, so E_{50}= E_{50}^{ref}.

$$E_{50}^{ref} = \frac{\Delta\sigma_v}{\Delta\epsilon_v} = \frac{135}{0.675\%} = 2.0 \cdot 10^4 kPa$$

For Sand it can be assumed that

$E_{oed}^{ref} = E_{50}^{ref} = 2.0 \cdot 10^4 kPa$

$E_{ur}^{ref} \approx 4 * E_{50}^{ref} = 8.0 \cdot 10^4 kPa$

$m = 0.5$

Additionally it is assumed that:

$G_0^{ref} = 1.25 \cdot E_{ur}^{ref} = 1 \cdot 10^5 kPa$

$\gamma_{0.7} = 1.5 \cdot 10^{-4}$

APPENDIX B: MATERIAL PROPERTIES SECANT WALL

For a plane strain model material properties for the secant wall have to specified per meter length of the wall. In order to do so we first recognize the secant wall as consisting of repetitive parts at a certain intermediate distance, as shown in Figure 19.

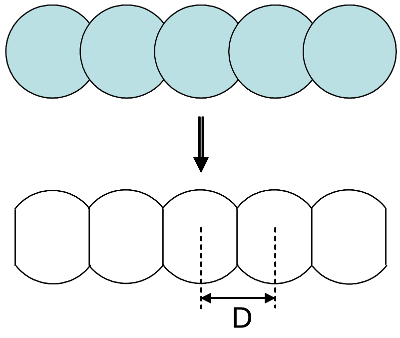


Figure 19: Secant wall as repetitive equal sections

Compared to the original bored piles the repetitive sections have a reduced cross sectional area. Though it can be analytically derived how much the reduction is, the fastest way to determine this is to draw the repetitive section on paper with a fine grid based on the original bored piles with a diameter of 1000mm and an overlap of 200mm and count squares. Using this method the cross sectional area of the repetitive section is determined as A_s = 0.74 m^2. Since the sections are at a distance D apart where D is given as 800mm, the cross sectional area of the wall per meter is given as:

$A_{wall} = \frac{A_s}{D} = \frac{0.74}{0.8} = 0.93$ m^2/m

For the moment of inertia is assumed that the influence of the reduced cross sectional area is negligble as the reduction is close to the axis of bending and symmetric. Therefore the moment of inertia per meter wall is determined as:

$I_{wall} = \frac{I_{pile}}{D} = \frac{\pi r^4}{4D} = \frac{\pi \cdot (0.5)^4}{4 \cdot 0.8} = 61.3 \cdot 10^{-3}$ m^4/m

With $E_{concrete} = 2.7 \cdot 10^7$ kN/m^2 this gives

EA = (2.7$\cdot 10^7$)(0.93) = **2.5**$\cdot 10^7$ **kN/m**
EI = (2.7$\cdot 10^7$)(61.3$\cdot 10^{-3}$) = **1.67**$\cdot 10^6$ **kNm^2/m**

And for the weight:

w = $\gamma \cdot$A = 16 $\cdot$ 0.93 = **15 kN/m/m**