

## TABLE OF CONTENTS

<b>1</b>	<b>Introduction</b> .....	1 - 1
<b>2</b>	<b>Getting started</b> .....	2 - 1
	2.1 Installation.....	2 - 1
	2.2 General modelling aspects.....	2 - 1
	2.3 Input procedures.....	2 - 3
	2.3.1 Input of geometry objects.....	2 - 3
	2.3.2 Input of text and values.....	2 - 3
	2.3.3 Input of selections.....	2 - 4
	2.3.4 Structured input.....	2 - 5
	2.4 Starting the program.....	2 - 6
	2.4.1 General settings.....	2 - 6
	2.4.2 Creating a geometry model.....	2 - 8
<b>3</b>	<b>Settlement of circular footing on sand (Lesson 1)</b> .....	3 - 1
	3.1 Geometry.....	3 - 1
	3.2 Rigid footing.....	3 - 2
	3.2.1 Creating the input.....	3 - 2
	3.2.2 Performing calculations.....	3 - 14
	3.2.3 Viewing output results.....	3 - 18
	3.3 Flexible footing.....	3 - 21
<b>4</b>	<b>Submerged construction of an excavation (Lesson 2)</b> .....	4 - 1
	4.1 Geometry.....	4 - 2
	4.2 Calculations.....	4 - 11
	4.3 Viewing output results.....	4 - 14
<b>5</b>	<b>Undrained river embankment (Lesson 3)</b> .....	5 - 1
	5.1 Geometry model.....	5 - 1
	5.2 Calculations.....	5 - 4
	5.3 Output.....	5 - 9
<b>6</b>	<b>Dry excavation using a tie back wall (Lesson 4)</b> .....	6 - 1
	6.1 Input.....	6 - 1
	6.2 Calculations.....	6 - 5
	6.3 Output.....	6 - 9
<b>7</b>	<b>Construction of a road embankment (Lesson 5)</b> .....	7 - 1
	7.1 Input.....	7 - 1
	7.2 Calculations.....	7 - 4
	7.3 Output.....	7 - 5
	7.4 Safety analysis.....	7 - 7

<b>8</b>	<b>Construction of a shield tunnel (Lesson 4)</b> .....	8 - 1
	8.1 Geometry.....	8 - 2
	8.2 Calculations.....	8 - 6
	8.3 Output .....	8 - 7
<b>A</b>	<b>Appendix A - Menu tree</b> .....	A - 1
	A.1 Input menu .....	A - 1
	A.2 Calculations menu.....	A - 2
	A.3 Output menu.....	A - 3
	A.4 Curves menu .....	A - 4
<b>B</b>	<b>Appendix B - Calculation scheme for initial stresses due to soil weight</b> .....	B - 1

## 6 DRY EXCAVATION USING A TIE BACK WALL (LESSON 4)

This example involves the dry construction of an excavation. The excavation is supported by concrete diaphragm walls. The walls are tied back by pre-stressed ground anchors. PLAXIS allows for a detailed modelling of this type of problem. It is demonstrated in this example how ground anchors are modelled and how pre-stressing 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.

### 6.1 INPUT

The excavation is 20 m wide and 10 m deep. 15 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 upper anchor has a total length of 14.5 m and an inclination of  $33.7^\circ$  (2:3). The lower anchor is 10 m long and is installed at an angle of  $45^\circ$ . The excavation is symmetric so only one half of the problem needs to be modelled.

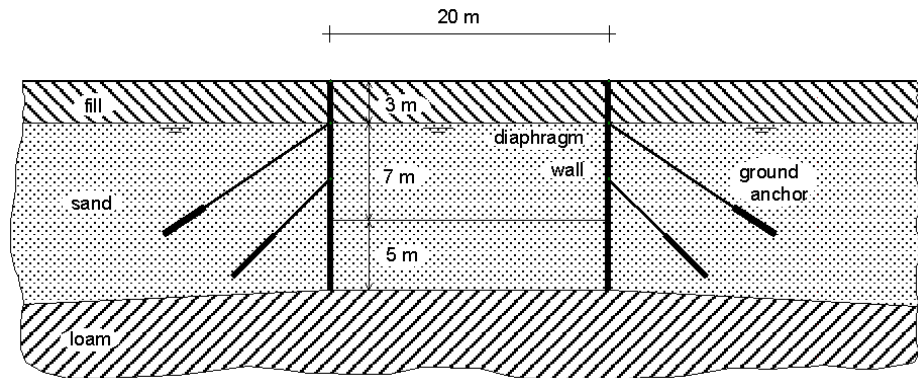


Figure 6.1 Excavation supported by tie back walls

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 more or less homogeneous layer consisting of dense well graded sand. This layer is particularly suitable for the installation of the ground anchors. 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) Below the sand layer there is a loam layer which extends to large depth.

### Geometry model



The symmetric problem can be modelled with a geometry model of 32 m width and 20 m depth. The proposed geometry model is given in Fig. 6.2. A ground anchor can be modelled by a combination of a node-to-node anchor and a geotextile (yellow line). The geotextile simulates the grout body whereas the node-to-node anchor simulates the anchor rod. In reality there is a complex three dimensional state of stress around the grout body.



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 pull-out force of the ground anchor.

The diaphragm wall is modelled as a beam. The interfaces around the beam are used to model soil-structure interaction effects. They are extended under the wall for 1.0 m. Interfaces should not be used around the geotextiles that represent the grout body.

**Hint:**

In general, it is a good habit to extend interfaces around corners of structures in order to allow for sufficient freedom of deformation and to obtain a more accurate stress distribution. When doing so, make sure that the strength of the extended part of the interface is equal to the soil strength and that the interface permeability does not influence the flow field, if applicable (see material properties).

The excavation is constructed in three excavation stages. The separation between the stages is modelled by geometry lines. Create the basic geometry model as presented in Fig. 6.2. The *standard fixities* can be used to generate the proper boundary conditions.

### Material properties

The soil consists of three distinct layers. Enter three data sets for soil & interfaces with the parameters given in table 6.1. Note that the values for the permeability of the interface do not correspond with the default setting. The beam elements used to model the walls are, on their own, fully permeable. Therefore, the interfaces around the wall must be used to block the flow through the wall for groundwater calculations and consolidation analyses. This can be achieved by setting the permeability parameter of the interface to *Impermeable*. In that case a very low (but non zero) value of the interface permeability is used. For the interfaces in the loam layer below the wall (the extended part of the interfaces) the strength reduction factor is set to *Rigid* (no reduction) and the permeability parameter is set to *Neutral*.

The latter means that the flow in these elements is neither blocked (*Impermeable*), nor drained.

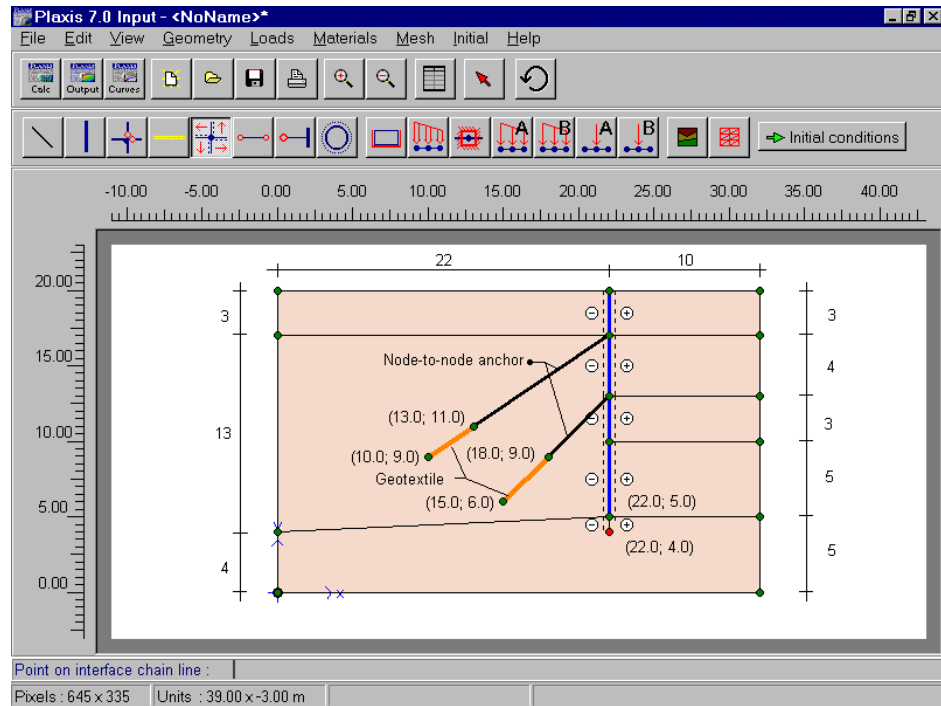


Figure 6.2 Geometry model of building pit

**Hint:** The extended part of an interface is not used for soil-structure interaction and should therefore have the same strength as the surrounding soil. This can be achieved with a strength reduction factor  $R_{inter} = 1.0$ , which is automatically adopted in the *Rigid* selection. In addition, the extended part of an interface should not influence the flow field. This is achieved by setting the interface permeability parameter to *Neutral*. Hence, the extended part of an interface generally has the settings *Rigid* and *Neutral*. If necessary, a separate material data set must be created for the extended part of an interface.

The properties of the concrete diaphragm wall are entered in a material set of the *Beam* type. The concrete has a Young's modulus of 35 GPa and the wall is 0.35 m thick. The properties are listed in table 6.2.

Table 6.1. Soil and interface properties

Parameter	Name	Fill	Sand	Loam	Unit
Material model	<i>Model</i>	MC	MC	MC	-
Type of material behaviour	<i>Type</i>	drained	drained	drained	-
Dry soil weight	$\gamma_{dry}$	16	17	17	kN/m <sup>3</sup>
Wet soil weight	$\gamma_{wet}$	20	20	19	kN/m <sup>3</sup>
Horizontal permeability	$k_x$	1.0	0.5	0.1	m/day
Vertical permeability	$k_y$	1.0	0.5	0.1	m/day
Young's modulus	$E_{ref}$	8000	30000	20000	kN/m <sup>2</sup>
Poisson's ratio	$\nu$	0.30	0.30	0.33	-
Cohesion	$c_{ref}$	1.0	1.0	8.0	kN/m <sup>2</sup>
Friction angle	$\phi$	30	34	29	°
Dilatancy angle	$\psi$	0.0	4.0	0.0	°
Interface reduction factor	$R_{inter}$	0.65	0.70	Rigid	-
Interf. Permeability parameter	<i>Perm.</i>	Imperm	Imperm	Neutral	-

Table 6.2. Properties of the diaphragm wall (beam)

Parameter	Name	Value	Unit
Type of behaviour	<i>Material type</i>	Elastic	-
Normal stiffness	$EA$	12·10 <sup>6</sup>	kN/m
Flexural rigidity	$EI$	0.12·10 <sup>6</sup>	kNm <sup>2</sup> /m
Equivalent thickness	$d$	0.346	m
Weight	$w$	8.3	kN/m/m
Poisson's ratio	$\nu$	0.15	-

For the properties of the ground anchors, two material data sets are needed: One of the *Anchor* type and one of the *Geotextile* type. The *Anchor* data set contains the properties of the anchor rod and the *Geotextile* data set contains the properties of the grout body. The data are listed in tables 6.3 and 6.4.

Table 6.3. Properties of the anchor rod (node-to-node anchors)

Parameter	Name	Value	Unit
Type of behaviour	<i>Material type</i>	Elastic	-
Normal stiffness	$EA$	2·10 <sup>5</sup>	kN
Spacing out of plane	$L_s$	2.5	m
Maximum force	$F_{max}$	1·10 <sup>15</sup>	kN

Table 6.4. Property of the grout body (geotextile)

Parameter	Name	Value	Unit
Normal stiffness	<i>EA</i>	$1 \cdot 10^5$	kN/m

**Mesh generation**

For the generation of the mesh it is advisable to set the *Global coarseness* parameter to *Medium*. In addition, it is expected that stress concentrations will occur around the two grout bodies and in the lower part of the wall, and so a local refinement is proposed here. Select the two geotextiles and the lower part of the beam simultaneously (use the <Shift> key) and select *Refine line* from the *Mesh* menu. This process results in a mesh of approximately 260 elements.

**Initial conditions**

In the initial conditions, a water weight of  $10 \text{ kN/m}^3$  is entered. The initial water pressures are generated on the basis of a horizontal general phreatic line at a level of  $y = 17 \text{ m}$  (through points (0; 17.0) and (32.0; 17.0)).

Initially, all structural components are deactivated. Hence, deselect the beam, the two node-to-node anchors and the two geotextiles. The initial stress field is generated by means of the *K<sub>0</sub>-procedure* using the default *K<sub>0</sub>*-values in all clusters.

**6.2 CALCULATIONS**

The calculation consists of five phases. In the first phase the wall is constructed and the first 3 m of the excavation are constructed without connection of anchors to the wall. At this depth the excavation remains dry. In the second phase the first anchor is installed and pre-stressed. The third phase involves further excavation to a depth of 7 m, including the de-watering of the excavation. This involves a groundwater flow analysis to calculate the new pore water distribution, which is a part of the definition of the third calculation phase. In the fourth phase the second anchor is installed and pre-stressed and the fifth phase is a further excavation (and de-watering) of to the final depth of 10 m.

All calculation phases are defined as *Plastic* calculations of the *Load advancement ultimate level* type using *Staged construction* as *Loading input* and standard settings for all other parameters. The instructions given below are limited to a description of how the phases are defined within the *Staged construction* mode.

**Phase 1:**

- Activate the wall
- De-activate the upper cluster of the excavation.

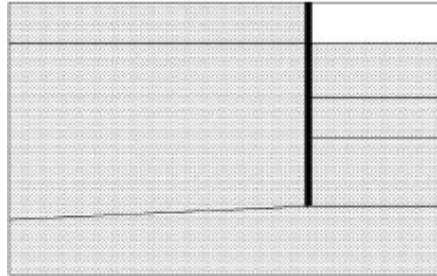


Figure 6.3a Phase 1

**Phase 2:**

- Activate the upper geotextile
- Double click the upper node-to-node anchor. A node-to-node anchor properties window appears with the anchor pre-stress options. Select the *Adjust pre-stress force* box and enter a pre-stress force of 120 kN/m. Press <OK> to close the window.

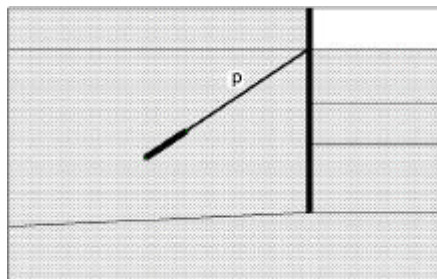


Figure 6.3b Phase 2

**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.

**Phase 3:**

- Deactivate the second cluster of the excavation.

Now the boundary conditions for the groundwater flow calculation have to be entered. At the left boundary, the groundwater head remains at a level of 17.0 m. The bottom boundary of the problem should be closed. The right boundary is a symmetry line, so flow cannot occur through this line and therefore this boundary is also closed. The top is also closed since no outflow is expected at this boundary. 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 = 13.0 m). This condition is automatically satisfied if the clusters of the pit are inactive during the groundwater flow calculation. Flow through the wall is prevented because the permeability parameter of the interfaces was set to *Impermeable* (see *Material properties*).

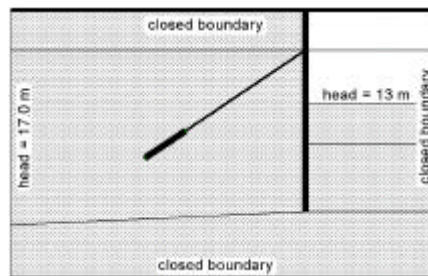




Figure 6.3c Phase 3

In order to prescribe correctly these boundary conditions, follow these steps:

-  Click on the 'switch' to go to the water pressures mode.
-  Click on the *Selection* button.
- Double click on one of the three geometry lines of the left boundary. A window appears showing the groundwater head on the boundary points. Enter a value of 17.0 for both points and press <OK>.
- Do the same for the other two geometry lines of the left boundary.
- Select the *Closed flow boundary* button (black line) from the toolbar. Click on the lower left point of the geometry; proceed to the lower right point and click again; proceed to the upper right point and click again; proceed to the upper left point and click again.
- Click on the *Generate water pressures* button. Select *Groundwater flow* from the *Generate by* box and click <OK> to start the groundwater flow calculation (the *Iterative procedure* can remain at the *Standard setting*).

- After the groundwater calculation has finished, press the <OK> button in the calculation window. The window closes and the flow field is presented in the Output window.

**Hint:** The results of a groundwater calculation can be viewed as *Pore pressures*, *Flow field* and *Groundwater head*. These options are available from the *Stress* menu.

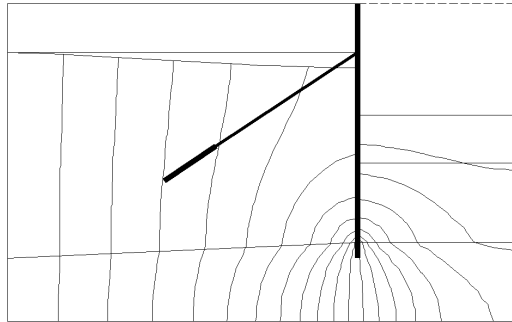


Figure 6.4 Groundwater head contours resulting from groundwater calculation

- Click on the <Update> button to return to the staged construction mode.
- Within the staged construction mode, click on the <Update> button to return to the Calculation program.

**Phase 4:**

- Activate the lower geotextile
- Double click the lower node-to-node anchor. In the Anchor window, select the *Adjust pre-stress force* box and enter a pre-stress force of 200 kN/m. Press <OK> to close the window.

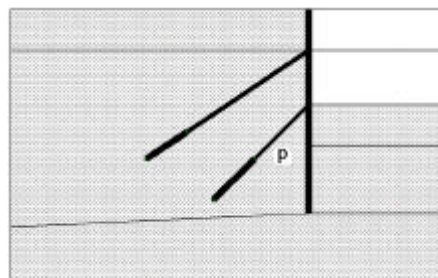


Figure 6.3d Phase 4

**Phase 5:**

- Deactivate the third cluster of the excavation.
- Click on the 'switch' to go to the water pressures mode.
- The boundary conditions were already defined in phase 3. They are still valid for the current groundwater calculation. However it is now necessary to lower the water level within the excavation to the new construction depth. In order to do this, click on the *Generate water pressures* button. Select *Groundwater flow* from the *Generate by* box and click <OK> to start the groundwater flow calculation.
- After the groundwater calculation has finished, press the <OK> button in the calculation window and view the results in the Output window. Click on the <Update> button to return to the staged construction mode.

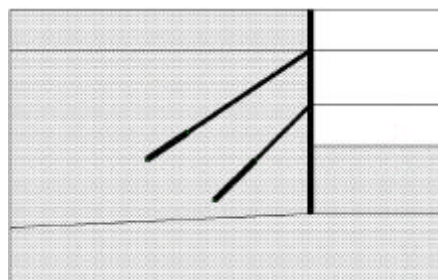


Figure 6.3e Phase 5

After all calculation phases have been defined, some points for load-displacement curves should be selected (for example the connection points of the ground anchors on the diaphragm wall). Start the calculation by clicking on the <Calculate> button.

**6.3 OUTPUT**

Figs. 6.5 a to e show the deformed meshes at the end of the five respective calculation phases. In the final situation, the wall has moved about 7 cm forward. Behind the wall there is a small settlement trough.

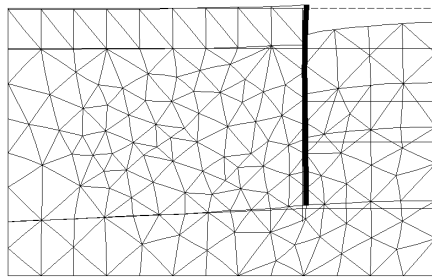


Figure 6.5a Deformed mesh, phase 1

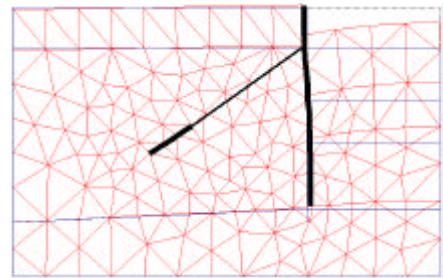


Figure 6.5b Deformed mesh, phase 2

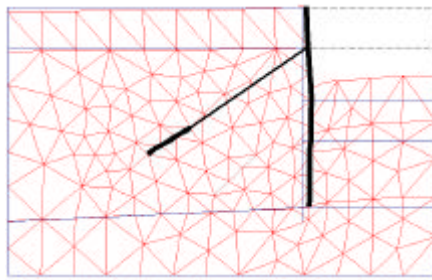


Figure 6.5c Deformed mesh, phase 3

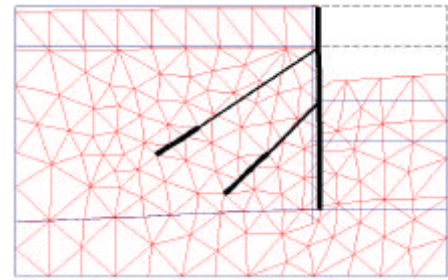


Figure 6.5d Deformed mesh, phase 4

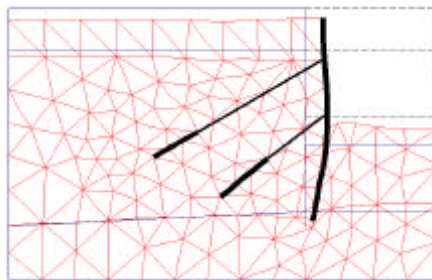


Figure 6.5e Deformed mesh, final stage

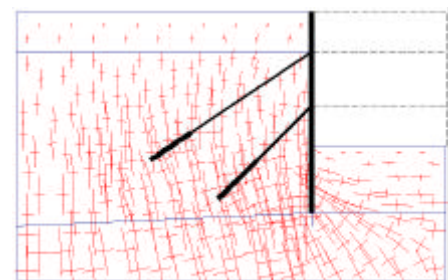


Figure 6.6 Effective stresses, final stage

Fig. 6.6 shows the principal effective 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.

Fig. 6.7 shows the bending moments in the diaphragm wall in the final state. The two dips in the line of moments are caused by the anchor forces.

The anchor force can be viewed by double clicking on the anchor. When doing this for the results of the second and the fourth calculation phase, it can be checked that the anchor force is indeed equal to the specified pre-stress force.

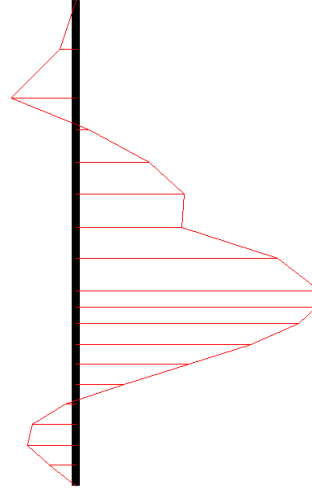


Figure 6.7 Bending moment in the diaphragm wall in the final stage

