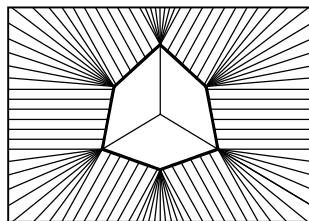


# **PLAXIS Version 8**

## **Tutorial Manual**





**TABLE OF CONTENTS**

<b>1</b>	<b>Introduction.....</b>	<b>1-1</b>
<b>2</b>	<b>Getting started.....</b>	<b>2-1</b>
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>	<b>3-1</b>
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.....	3-20
<b>4</b>	<b>Submerged construction of an excavation (lesson 2) .....</b>	<b>4-1</b>
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>	<b>5-1</b>
5.1	Geometry model.....	5-1
5.2	Calculations .....	5-4
5.3	Output .....	5-8
<b>6</b>	<b>Dry excavation using a tie back wall (lesson 4) .....</b>	<b>6-1</b>
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>	<b>7-1</b>
7.1	Input.....	7-1
7.2	Calculations .....	7-4
7.3	Output .....	7-5

---

7.4	Safety analysis .....	7-7
7.5	Updated mesh analysis.....	7-11
<b>8</b>	<b>Settlements due to tunnel construction (lesson 6) .....</b>	<b>8-1</b>
8.1	Geometry .....	8-2
8.2	Calculations .....	8-6
8.3	Output .....	8-8

**Appendix A - Menu structure**

**Appendix B - Calculation scheme for initial stresses due to soil weight**

## 1 INTRODUCTION

PLAXIS is a finite element package that has been developed specifically for the analysis of deformation and stability in geotechnical engineering projects. The simple graphical input procedures enable a quick generation of complex finite element models, and the enhanced output facilities provide a detailed presentation of computational results. The calculation itself is fully automated and based on robust numerical procedures. This concept enables new users to work with the package after only a few hours of training.

This Tutorial Manual is intended to help new users become familiar with PLAXIS. The various lessons deal with a wide range of interesting practical applications and cover most of the program features. However, the use of soil models is limited to the basic Mohr-Coulomb model. Users are expected to have a basic understanding of soil mechanics and should be able to work in a Windows environment. It is strongly recommended that the lessons are followed in the order that they appear in the manual. The tutorial lessons are also available in the examples folder of the PLAXIS program directory and can be used to check your results.

The Tutorial Manual does not provide theoretical background information on the finite element method, nor does it explain the details of the various soil models available in the program. The latter can be found in the Material Models Manual, as included in the full manual, and theoretical background is given in the Scientific Manual. For detailed information on the available program features, the user is referred to the Reference Manual. In addition to the full set of manuals, short courses are organised on a regular basis at several places in the world to provide hands-on experience and background information on the use of the program.

## TUTORIAL MANUAL

## 2 GETTING STARTED

This chapter describes some of the notation and basic input procedures that are used in PLAXIS. In the manuals, menu items or windows specific items are printed in *Italics*. Whenever keys on the keyboard or text buttons on the screen need to be pressed, this is indicated by the name of the key or button in brackets, (for example the <Enter> key).

### 2.1 INSTALLATION

For the installation procedure the user is referred to the General Information section in this manual.

### 2.2 GENERAL MODELLING ASPECTS

For each new project to be analysed it is important to create a geometry model first. A geometry model is a 2D representation of a real three-dimensional problem and consists of points, lines and clusters. A geometry model should include a representative division of the subsoil into distinct soil layers, structural objects, construction stages and loadings. The model must be sufficiently large so that the boundaries do not influence the results of the problem to be studied. The three types of components in a geometry model are described below in more detail.

#### **Points:**

Points form the start and end of lines. Points can also be used for the positioning of anchors, point forces, point fixities and for local refinements of the finite element mesh.

#### **Lines:**

Lines are used to define the physical boundaries of the geometry, the model boundaries and discontinuities in the geometry such as walls or shells, separations of distinct soil layers or construction stages. A line can have several functions or properties.

#### **Clusters:**

Clusters are areas that are fully enclosed by lines. PLAXIS automatically recognises clusters based on the input of geometry lines. Within a cluster the soil properties are homogeneous. Hence, clusters can be regarded as parts of soil layers. Actions related to clusters apply to all elements in the cluster.

After the creation of a geometry model, a finite element model can automatically be generated, based on the composition of clusters and lines in the geometry model. In a finite element mesh three types of components can be identified, as described below.

---

**Elements:**

During the generation of the mesh, clusters are divided into triangular elements. A choice can be made between 15-node elements and 6-node elements. The powerful 15-node element provides an accurate calculation of stresses and failure loads. In addition, 6-node triangles are available for a quick calculation of serviceability states. Considering the same element distribution (for example a default coarse mesh generation) the user should be aware that meshes composed of 15-node elements are actually much finer and much more flexible than meshes composed of 6-node elements, but calculations are also more time consuming. In addition to the triangular elements, which are generally used to model the soil, compatible plate elements, geogrid elements and interface elements may be generated to model structural behaviour and soil-structure interaction.

**Nodes:**

A 15-node element consists of 15 nodes and a 6-node triangle is defined by 6 nodes. The distribution of nodes over the elements is shown in Figure 2.1. Adjacent elements are connected through their common nodes. During a finite element calculation, displacements ( $u_x$  and  $u_y$ ) are calculated at the nodes. Nodes may be pre-selected for the generation of load-displacement curves.

**Stress points:**

In contrast to displacements, stresses and strains are calculated at individual Gaussian integration points (or stress points) rather than at the nodes. A 15-node triangular element contains 12 stress points as indicated in Figure 2.1a and a 6-node triangular element contains 3 stress points as indicated in Figure 2.1b. Stress points may be pre-selected for the generation of stress paths or stress-strain diagrams.

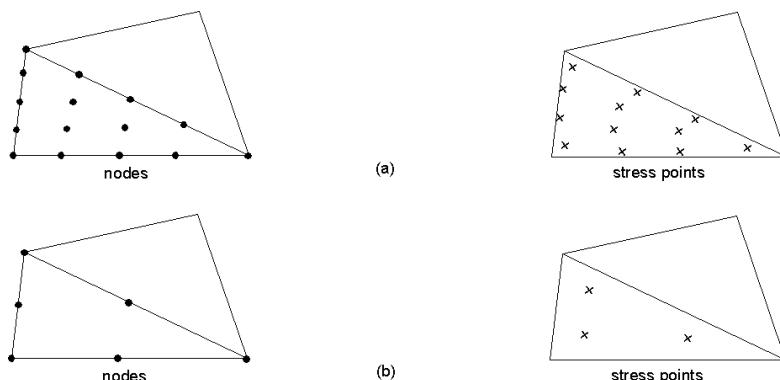


Figure 2.1 Nodes and stress points

## 2.3 INPUT PROCEDURES

In PLAXIS, input is specified by a mixture of mouse clicking and moving, and by keyboard input. In general, distinction can be made between four types of input:

Input of geometry objects	(e.g. drawing a soil layer)
Input of text	(e.g. entering a project name)
Input of values	(e.g. entering the soil unit weight)
Input of selections	(e.g. choosing a soil model)

The mouse is generally used for drawing and selection purposes, whereas the keyboard is used to enter text and values.

### 2.3.1 INPUT OF GEOMETRY OBJECTS

The creation of a geometry model is based on the input of points and lines. This is done by means of a mouse pointer in the draw area. Several geometry objects are available from the menu or from the toolbar. The input of most of the geometry objects is based on a line drawing procedure. In any of the drawing modes, lines are drawn by clicking on the left mouse button in the draw area. As a result, a first point is created. On moving the mouse and left clicking with the mouse again, a new point is created together with a line from the previous point to the new point. The line drawing is finished by clicking the right mouse button, or by pressing the <Esc> key on the keyboard.

### 2.3.2 INPUT OF TEXT AND VALUES

As for any software, some input of values and text is required. The required input is specified in the edit boxes. Multiple edit boxes for a specific subject are grouped in windows. The desired text or value can be typed on the keyboard, followed by the <Enter> key or the <Tab> key. As a result, the value is accepted and the next input field is highlighted. In some countries, like The Netherlands, the decimal dot in floating point values is represented by a comma. The type of representation that occurs in edit boxes and tables depends on the country setting of the operating system. Input of values must be given in accordance with this setting.

Many parameters have default values. These default values may be used by pressing the <Enter> key without other keyboard input. In this manner, all input fields in a window can be entered until the <OK> button is reached. Pressing the <OK> button confirms all values and closes the window. Alternatively, selection of another input field, using the mouse, will result in the new input value being accepted. Input values are confirmed by left clicking the <OK> button with the mouse.

Pressing the <Esc> key or left clicking the <Cancel> button will cancel the input and restore the previous or default values before closing the window.

---

The *spin edit* feature is shown in Figure 2.2. Just like a normal input field a value can be entered by means of the keyboard, but it is also possible to left-click on the ▲ or ▼ arrows at the right side of each spin edit to increase or decrease its value by a predefined amount.

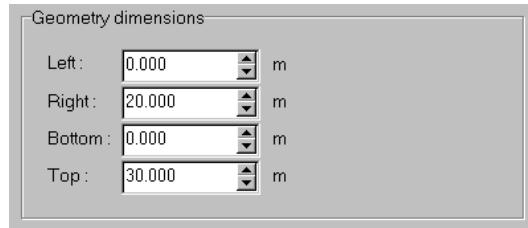


Figure 2.2 Spin edits

### 2.3.3 INPUT OF SELECTIONS

Selections are made by means of radio buttons, check boxes or combo boxes as described below.

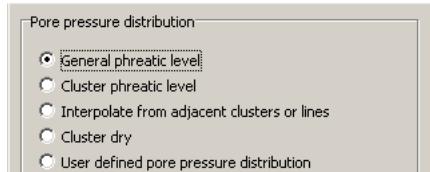


Figure 2.3 Radio buttons

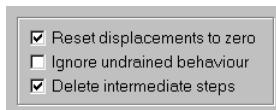


Figure 2.4 Check boxes

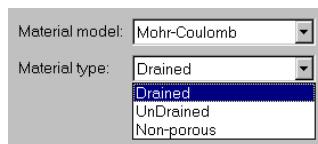


Figure 2.5 Combo boxes

**Radio buttons:**

In a window with radio buttons only one item may be active. The active selection is indicated by a black dot in the white circle in front of the item. Selection is made by clicking the left mouse button in the white circle or by using the up and down arrow keys on the keyboard. When changing the existing selection to one of the other options, the 'old' selection will be deselected. An example of a window with radio buttons is shown in Figure 2.3. According to the selection in Figure 2.3 the *Pore pressure distribution* is set to *General phreatic level*.

**Check boxes:**

In a window with check boxes more than one item may be selected at the same time. The selection is indicated by a black tick mark in a white square. Selection is made by clicking the left mouse button in the white square or by pressing the space bar on the keyboard. Another click on a preselected item will deselect the item. An example of three check boxes is shown in Figure 2.4.

**Combo boxes:**

A combo box is used to choose one item from a predefined list of possible choices. An example of a window with combo boxes is shown in Figure 2.5. As soon as the ▼ arrow at the right hand side of the combo box is left clicked with the mouse, a pull down list occurs that shows the possible choices. A combo box has the same functionality as a group of radio buttons but it is more compact.

### 2.3.4 STRUCTURED INPUT

The required input is organised in a way to make it as logical as possible. The Windows environment provides several ways of visually organising and presenting information on the screen. To make the reference to typical Windows elements in the next chapters easier, some types of structured input are described below.

**Page control and tab sheets:**

An example of a page control with three tab sheets is shown in Figure 2.6. In this figure the second tab sheet for the input of the model parameters of the *Mohr-Coulomb* soil model is active. Tab sheets are used to handle large amounts of different types of data that do not all fit in one window. Tab sheets can be activated by left-clicking on the corresponding tab or using <Ctrl><Tab> on the keyboard.

**Group boxes:**

Group boxes are rectangular boxes with a title. They are used to cluster input items that have common features. In Figure 2.6, the active tab sheet contains three group boxes named *Stiffness*, *Strength* and *Alternatives*.

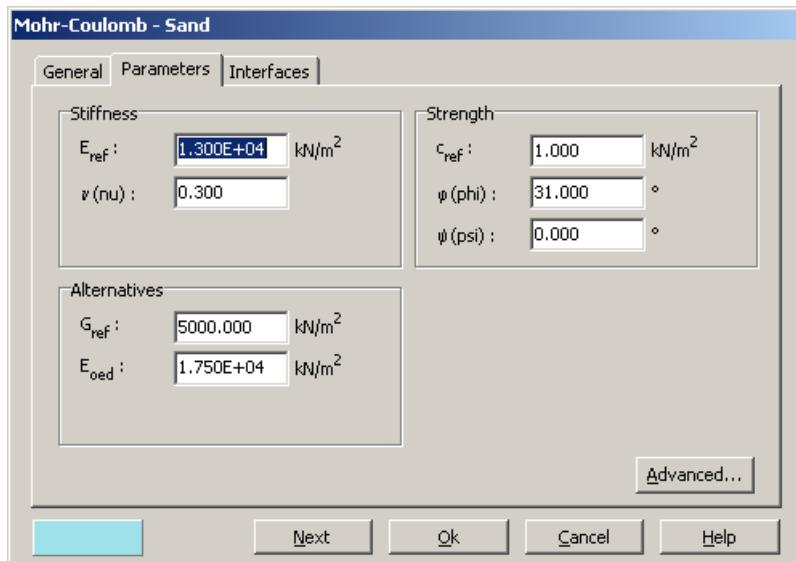


Figure 2.6 Page control and tab sheets

## 2.4 STARTING THE PROGRAM

It is assumed that the program has been installed using the procedures described in the General Information part of the manual. It is advisable to create a separate directory in which data files are stored. PLAXIS can be started by double clicking on the *Plaxis input* icon in the PLAXIS program group. The user is asked whether to define a new problem or to retrieve a previously defined project. If the latter option is chosen, the program lists four of the most recently used projects from which a direct choice can be made. Choosing the item <>more files<> that appears first in this list will give a file requester from which the user can choose any previously defined project for modification.

### 2.4.1 GENERAL SETTINGS

If a new project is to be defined, the *General settings* window as shown in Figure 2.7 appears. This window consists of two tab sheets. In the first tab sheet miscellaneous

settings for the current project have to be given. A filename has not been specified here; this can be done when saving the project.

The user can enter a brief description of the problem as the title of the project as well as a more extended description in the *Comments* box. The title is used as a proposed file name and appears on output plots. The comments box is simply a convenient place to store information about the analysis. In addition, the type of analysis and the type of elements must be specified. Optionally, a separate acceleration, in addition to gravity, can be specified for a pseudo-static simulation of dynamics forces.

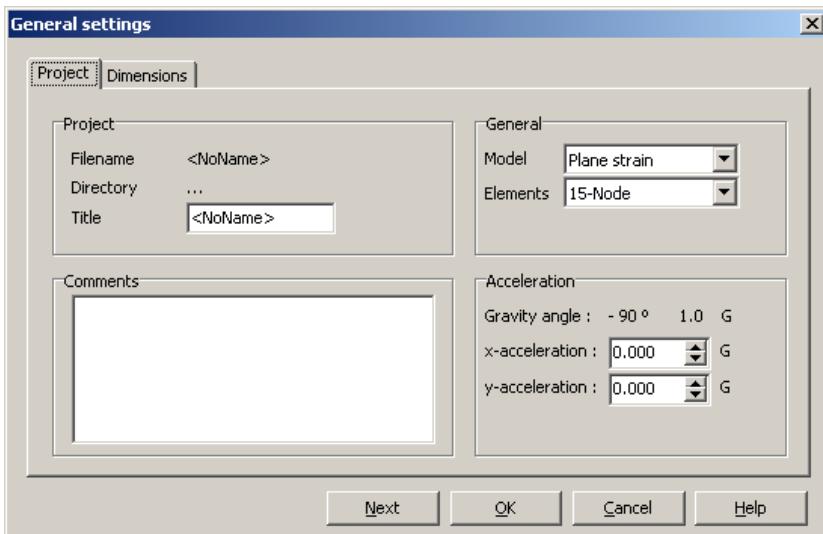


Figure 2.7 General settings - *General* tab sheet

The second tab sheet is shown in Figure 2.8. In addition to the basic units of *Length*, *Force* and *Time*, the minimum dimensions of the draw area must be given here, such that the geometry model will fit the draw area. The general system of axes is such that the *x*-axis points to the right, the *y*-axis points upward and the *z*-axis points towards the user. In PLAXIS a two-dimensional model is created in the (*x,y*)-plane. The *z*-axis is used for the output of stresses only. *Left* is the lowest *x*-coordinate of the model, *Right* the highest *x*-coordinate, *Bottom* the lowest *y*-coordinate and *Top* the highest *y*-coordinate of the model.

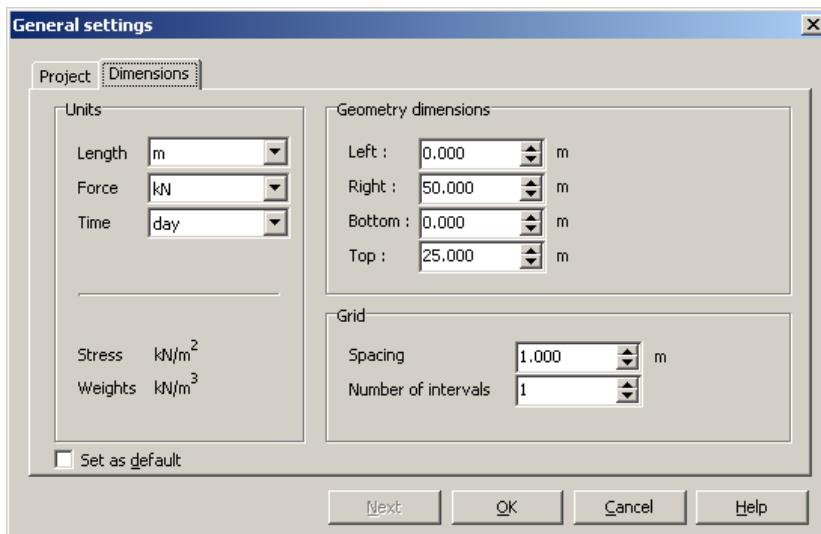


Figure 2.8 General settings - *Dimensions* tab sheet

In practice, the draw area resulting from the given values will be larger than the values given in the four spin edits. This is partly because PLAXIS will automatically add a small margin to the dimensions and partly because of the difference in the width/height ratio between the specified values and the screen.

#### 2.4.2 CREATING A GEOMETRY MODEL

When the general settings are entered and the <OK> button is clicked, the main *Input* window appears. This main window is shown in Figure 2.9. The most important parts of the main window are indicated and briefly discussed below.

##### **Main menu:**

The main menu contains all the options that are available from the toolbars, and some additional options that are not frequently used.

##### **Tool bar (General):**

This tool bar contains buttons for general actions like disk operations, printing, zooming or selecting objects. It also contains buttons to start the other programs of the PLAXIS package (Calculations, Output and Curves).

##### **Tool bar (Geometry):**

This tool bar contains buttons for actions that are related to the creation of a geometry model. The buttons are ordered in such a way that, in general,

following the buttons on the tool bar from the left to the right results in a completed geometry model.

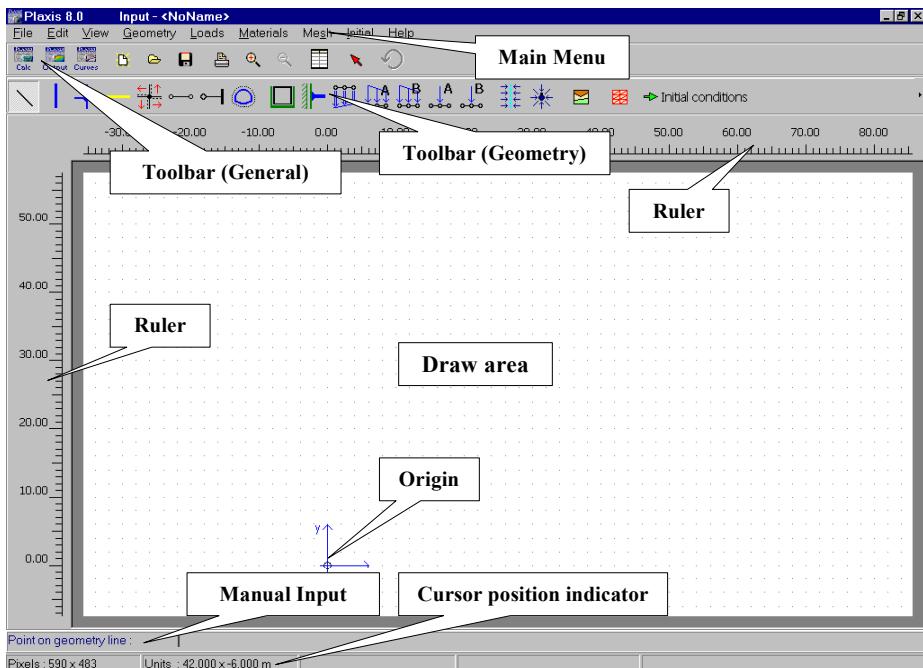


Figure 2.9 Main window of the Input program

### **Rulers:**

At both the left and the top of the draw area, rulers indicate the physical coordinates, which enables a direct view of the geometry dimensions.

### **Draw area:**

The draw area is the drawing sheet on which the geometry model is created. The draw area can be used in the same way as a conventional drawing program. The grid of small dots in the draw area can be used to snap to regular positions.

### **Origin:**

If the physical origin is within the range of given dimensions, it is represented by a small circle, with an indication of the  $x$ - and  $y$ -axes.

### **Manual input:**

If drawing with the mouse does not give the desired accuracy, then the *Manual input* line can be used. Values for *x*- and *y*-coordinates can be entered here by typing the corresponding values separated by a space. The manual input can also be used to assign new coordinates to a selected point or refer to an existing geometry point by entering its point number.

### **Cursor position indicator:**

The cursor position indicator gives the current position of the mouse cursor both in physical units and screen pixels.

Some of the objects mentioned above can be removed by deselecting the corresponding item from the *View* menu.

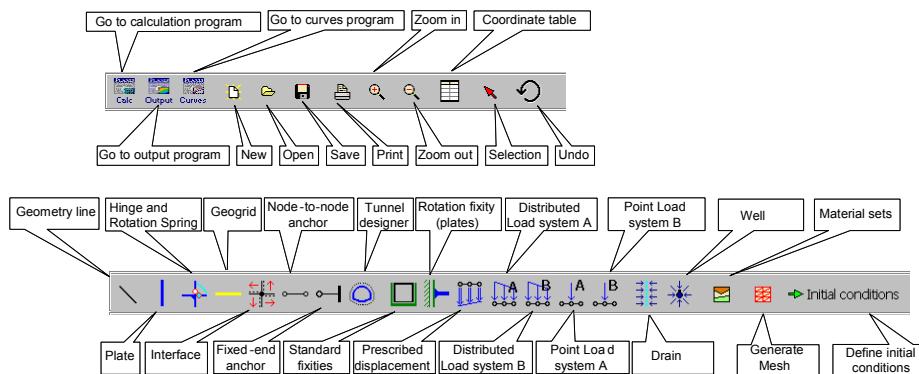


Figure 2.10 Toolbars

For both toolbars, the name and function of the buttons is shown after positioning the mouse cursor on the corresponding button and keeping the mouse cursor still for about a second; a hint will appear in a small yellow box below the button. The available hints for both toolbars are shown in Figure 2.10. In this Tutorial Manual, buttons will be referred to by their corresponding hints.

Help can be obtained from the user interface by pressing <F1> on the keyboard. This will provide background information on the selected part of the program.

For detailed information on the creation of a complete geometry model, the reader is referred to the various lessons that are described in this Tutorial Manual.

### 3 SETTLEMENT OF CIRCULAR FOOTING ON SAND (LESSON 1)

In the previous chapter some general aspects and basic features of the PLAXIS program were presented. In this chapter a first application is considered, namely the settlement of a circular foundation footing on sand. This is the first step in becoming familiar with the practical use of the program. The general procedures for the creation of a geometry model, the generation of a finite element mesh, the execution of a finite element calculation and the evaluation of the output results are described here in detail. The information provided in this chapter will be utilised in the later lessons. Therefore, it is important to complete this first lesson before attempting any further tutorial examples.

#### 3.1 GEOMETRY

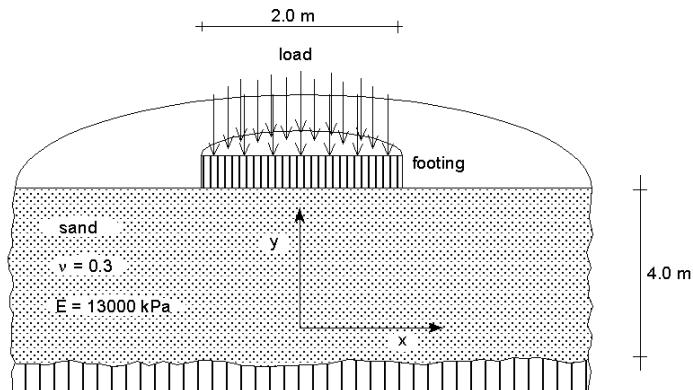


Figure 3.1 Geometry of a circular footing on a sand layer

A circular footing with a radius of 1.0 m is placed on a sand layer of 4.0 m thickness as shown in Figure 3.1. Under the sand layer there is a stiff rock layer that extends to a large depth. The purpose of the exercise is to find the displacements and stresses in the soil caused by the load applied to the footing. Calculations are performed for both rigid and flexible footings. The geometry of the finite element model for these two situations is similar. The rock layer is not included in the model; instead, an appropriate boundary condition is applied at the bottom of the sand layer. To enable any possible mechanism in the sand and to avoid any influence of the outer boundary, the model is extended in horizontal direction to a total radius of 5.0 m.

## 3.2 CASE A: RIGID FOOTING

In the first calculation, the footing is considered to be very stiff and rough. In this calculation the settlement of the footing is simulated by means of a uniform indentation at the top of the sand layer instead of modelling the footing itself. This approach leads to a very simple model and is therefore used as a first exercise, but it also has some disadvantages. For example, it does not give any information about the structural forces in the footing. The second part of this lesson deals with an external load on a flexible footing, which is a more advanced modelling approach.

### 3.2.1 CREATING THE INPUT

Start PLAXIS by double-clicking the icon of the Input program. A *Create/Open project* dialog box will appear in which you can select an existing project or create a new one. Choose a *New project* and click on the <OK> button. Now the *General settings* window appears, consisting of the two tab sheets *Project* and *Dimensions* (see Figure 3.3 and Figure 3.4).

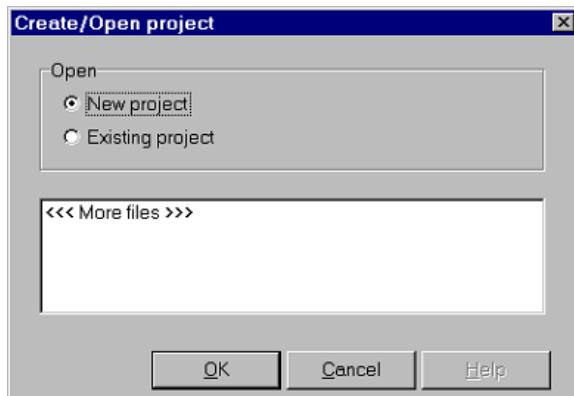


Figure 3.2 *Create/Open project* dialog box

### ***General Settings***

The first step in every analysis is to set the basic parameters of the finite element model. This is done in the *General settings* window. These settings include the description of the problem, the type of analysis, the basic type of elements, the basic units and the size of the draw area. To enter the appropriate settings for the footing calculation follow these steps:

- In the *Project* tab sheet, enter “Lesson 1” in the *Title* box and type “Settlements of a circular footing” in the *Comments* box.

- In the *General* box the type of the analysis (*Model*) and the basic element type (*Elements*) are specified. Since this lesson concerns a circular footing, choose *Axisymmetry* from the *Model* combo box and select *15-node* from the *Elements* combo box.

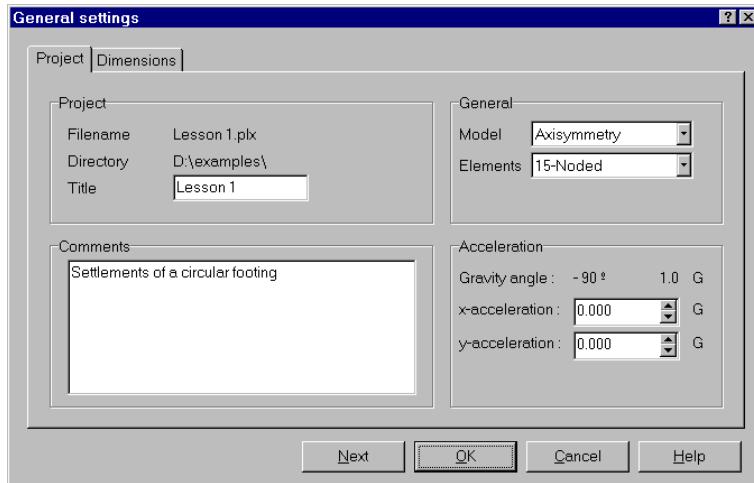


Figure 3.3 Project tab sheet of the *General settings* window

- The *Acceleration* box indicates a fixed gravity angle of  $-90^\circ$ , which is in the vertical direction (downward). In addition to the normal gravity, independent acceleration components may be entered for pseudo-dynamic analyses. These values should be kept zero for this exercise. Click on the <Next> button below the tab sheets or click on the *Dimensions* tab.
- In the *Dimensions* tab sheet, keep the default units in the *Units* box (Unit of *Length* = m; Unit of *Force* = kN; Unit of *Time* = day).
- In the *Geometry dimensions* box the size of the required draw area must be entered. When entering the upper and lower coordinate values of the geometry to be created, PLAXIS will add a small margin so that the geometry will fit well within the draw area. Enter 0.0, 5.0, 0.0 and 4.0 in the *Left*, *Right*, *Bottom* and *Top* edit boxes respectively.
- The *Grid* box contains values to set the grid spacing. The grid provides a matrix of dots on the screen that can be used as reference points. It may also be used for snapping to regular points during the creation of the geometry. The distance between the dots is determined by the *Spacing* value. The spacing of snapping points can be further divided into smaller intervals by the *Number of intervals* value. Enter 1.0 for the spacing and 1 for the intervals.

- Click on the <OK> button to confirm the settings. Now the draw area appears in which the geometry model can be drawn.

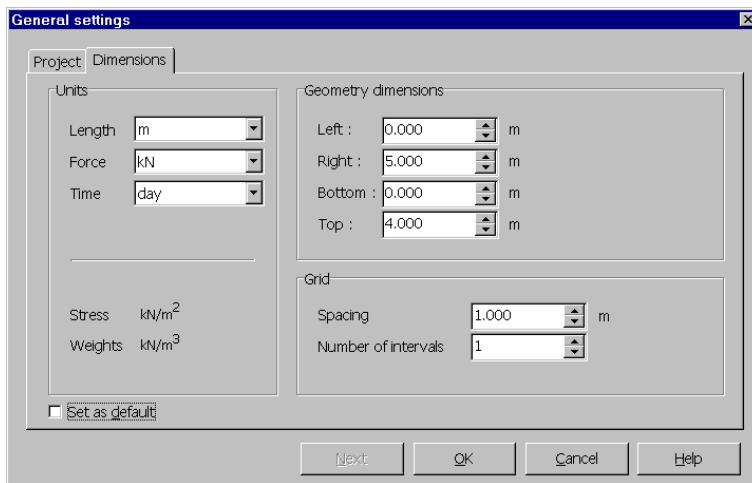


Figure 3.4 Dimensions tab sheet of the *General settings* window

**Hint:** In the case of a mistake or for any other reason that the general settings need to be changed, you can access the *General settings* window by selecting the *General settings* option from the *File* menu.

### Geometry Contour

Once the general settings have been completed, the draw area appears with an indication of the origin and direction of the system of axes. The x-axis is pointing to the right and the y-axis is pointing upward. A geometry can be created anywhere within the draw area. To create objects, you can either use the buttons from the toolbar or the options from the *Geometry* menu. For a new project, the *Geometry line* button is already active. Otherwise this option can be selected from the second toolbar or from the *Geometry* menu. In order to construct the contour of the proposed geometry, follow these steps:

-  Select the Geometry *line* option (already pre-selected).
- Position the cursor (now appearing as a pen) at the origin of the axes. Check that the units in the status bar read 0.0 x 0.0 and click the left mouse button once. The first geometry point (number 0) has now been created.

- Move along the x-axis to position (5.0; 0.0). Click the left mouse button to generate the second point (number 1). At the same time the first geometry line is created from point 0 to point 1.
- Move upward to position (5.0; 4.0) and click again.
- Move to the left to position (0.0; 4.0) and click again.
- Finally, move back to the origin (0.0; 0.0) and click the left mouse button again. Since the latter point already exists, no new point is created, but only an additional geometry line is created from point 3 to point 0. PLAXIS will also detect a cluster (area that is fully enclosed by geometry lines) and will give it a light colour.
- Click the right mouse button to stop drawing.

**Hint:**

Mispositioned points and lines can be modified or deleted by first choosing the *Selection* button from the toolbar. To move a point or line, select the point or the line and drag it to the desired position. To delete a point or a line, select the point or the line and press the <Delete> button on the keyboard.



Unwanted drawing operations can be removed by pressing the *Undo* button from the toolbar or by selecting the *Undo* option from the *Edit* menu or by pressing <Ctrl><Z> on the keyboard.

&gt;

Lines can be drawn perfectly horizontal or vertical by holding down the <Shift> key on the keyboard while moving the cursor.

The proposed geometry does not include plates, hinges, geogrids, interfaces, anchors or tunnels. Hence, you can skip these buttons on the second toolbar.

**Hint:**

The full geometry model has to be completed before a finite element mesh can be generated. This means that boundary conditions and model parameters must be entered and applied to the geometry model first.

### **Boundary Conditions**

Boundary conditions can be found in the centre part of the second toolbar and in the *Loads* menu. For deformation problems two types of boundary conditions exist: Prescribed displacements and prescribed forces (loads).

In principle, all boundaries must have one boundary condition in each direction. That is to say, when no explicit boundary condition is given to a certain boundary (a free boundary), the natural condition applies, which is a prescribed force equal to zero and a free displacement.

To avoid the situation where the displacements of the geometry are undetermined, some points of the geometry must have prescribed displacements. The simplest form of a prescribed displacement is a fixity (zero displacement), but non-zero prescribed displacements may also be given. In this problem the settlement of the rigid footing is simulated by means of non-zero prescribed displacements at the top of the sand layer.

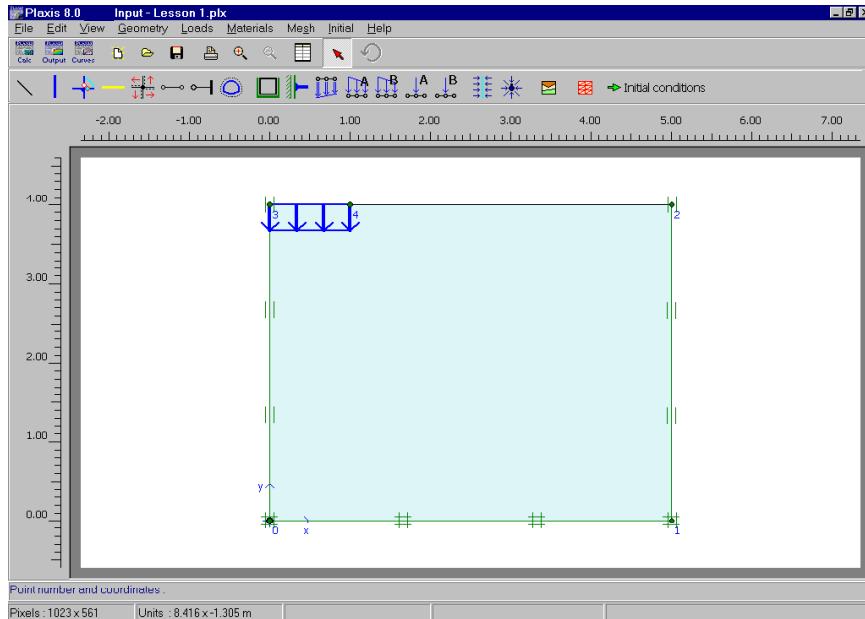


Figure 3.5 Geometry model in the Input window

To create the boundary conditions for this lesson, follow these steps:

- Click on the *Standard fixities* button on the toolbar or choose the *Standard fixities* option from the *Loads* menu to set the standard boundary conditions.
- As a result PLAXIS will generate a full fixity at the base of the geometry and roller conditions at the vertical sides ( $u_x=0$ ;  $u_y=\text{free}$ ). A fixity in a certain direction appears on the screen as two parallel lines perpendicular to the fixed direction. Hence, roller supports appear as two vertical parallel lines and full fixity appears as crosshatched lines.

**Hint:** The *Standard fixities* option is suitable for most geotechnical applications. It is a fast and convenient way to input standard boundary conditions.



Select the *Prescribed displacements* button from the toolbar or select the corresponding option from the *Loads* menu.

- Move the cursor to point (0.0; 4.0) and click the left mouse button.
- Move along the upper geometry line to point (1.0; 4.0) and click the left mouse button again.
- Click the right button to stop drawing.

In addition to the new point (4), a prescribed downward displacement of 1 unit (1.0 m) in a vertical direction and a fixed horizontal displacement are created at the top of the geometry. Prescribed displacements appear as a series of arrows starting from the original position of the geometry and pointing in the direction of movement.

**Hint:** The input value of a prescribed displacement may be changed by first clicking on the *Selection* button and then double-clicking on the line at which a prescribed displacement is applied. On selecting *Prescribed displacements* from the *Select* dialog box, a new window will appear in which the changes can be made.

> The prescribed displacement is actually activated when defining the calculation stages (Section 3.2.2). Initially it is not active.

### Material data sets

In order to simulate the behaviour of the soil, a suitable soil model and appropriate material parameters must be assigned to the geometry. In PLAXIS, soil properties are collected in material data sets and the various data sets are stored in a material database. From the database, a data set can be appointed to one or more clusters. For structures (like walls, plates, anchors, geogrids, etc.) the system is similar, but different types of structures have different parameters and therefore different types of data sets.

PLAXIS distinguishes between material data sets for *Soil & Interfaces*, *Plates*, *Anchors* and *Geogrids*.

The creation of material data sets is generally done after the input of boundary conditions. Before the mesh is generated, all material data sets should have been defined and all clusters and structures must have an appropriate data set assigned to them.

Table 3.1 Material properties of the sand layer

Parameter	Name	Value	Unit
Material model	<i>Model</i>	Mohr-Coulomb	-
Type of material behaviour	<i>Type</i>	Drained	-
Soil unit weight above phreatic level	$\gamma_{unsat}$	17.0	kN/m <sup>3</sup>
Soil unit weight below phreatic level	$\gamma_{sat}$	20.0	kN/m <sup>3</sup>

Parameter	Name	Value	Unit
Permeability in horizontal direction	$k_x$	1.0	m/day
Permeability in vertical direction	$k_y$	1.0	m/day
Young's modulus (constant)	$E_{ref}$	13000	kN/m <sup>2</sup>
Poisson's ratio	$\nu$	0.3	-
Cohesion (constant)	$c_{ref}$	1.0	kN/m <sup>2</sup>
Friction angle	$\phi$	31.0	°
Dilatancy angle	$\psi$	0.0	°

The input of material data sets can be selected by means of the *Material Sets* button on the toolbar or from the options available in the *Materials* menu.

To create a material set for the sand layer, follow these steps:

-  Select the *Material Sets* button on the toolbar.
- Click on the <New> button at the lower side of the *Material Sets* window. A new dialog box will appear with three tab sheets: *General*, *Parameters* and *Interfaces* (see Figure 3.6 and Figure 3.7).

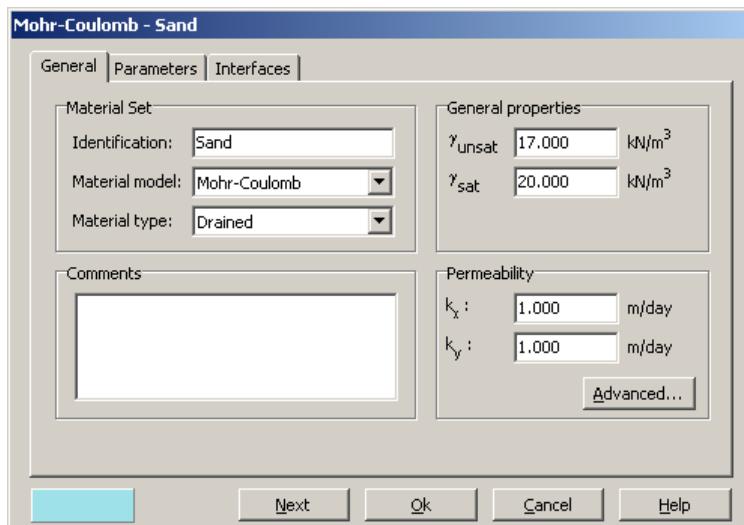


Figure 3.6 *General* tab sheet of the soil and interface data set window

- In the *Material Set* box of the *General* tab sheet, write “Sand” in the *Identification* box.

- Select *Mohr-Coulomb* from the *Material model* combo box and *Drained* from the *Material type* combo box (default parameters).
- Enter the proper values in the *General properties* box and the *Permeability* box according to the material properties listed in Table 3.1.
- Click on the <Next> button or click on the *Parameters* tab to proceed with the input of model parameters. The parameters appearing on the *Parameters* tab sheet depend on the selected material model (in this case the Mohr-Coulomb model).

See the Material Models manual for a detailed description of different soil models and their corresponding parameters.

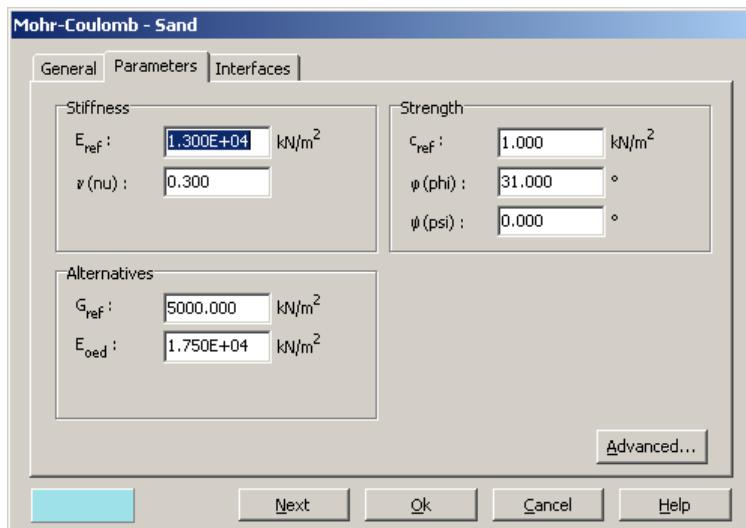


Figure 3.7 *Parameters* tab sheet of the soil and interface data set window

- Enter the model parameters of Table 3.1 in the corresponding edit boxes of the *Parameters* tab sheet.
- Since the geometry model does not include interfaces, the third tab sheet can be skipped. Click on the <OK> button to confirm the input of the current material data set. Now the created data set will appear in the tree view of the *Material Sets* window.
- Drag the data set “Sand” from the *Material Sets* window (select it and keep the left mouse button down while moving) to the soil cluster in the draw area and drop it there (release the left mouse button). Notice that the cursor changes shape to indicate whether or not it is possible to drop the data set. Correct

assignment of a data set to a cluster is indicated by a change in colour of the cluster.

- Click on the <OK> button in the *Material Sets* window to close the database.

**Hint:** PLAXIS distinguishes between a project database and a global database of material sets. Data sets may be exchanged from one project to another using the global database. The data sets of all lessons in this Tutorial Manual are stored in the global database during the installation of the program. To copy an existing data set, click on the <Global> >>> button of the *Material Sets* window. Drag the appropriate data set (in this case “Lesson 1 sand”) from the tree view of the global database to the project database and drop it there. Now the global data set is available for the current project. Similarly, data sets created in the project database may be dragged and dropped in the global database.

**Hint:** Existing data sets may be changed by opening the material sets window, selecting the data set to be changed from the tree view and clicking on the <Edit> button. As an alternative, the material sets window can be opened by double clicking a cluster and clicking on the <Change> button behind the *Material set* box in the properties window. A data set can now be assigned to the corresponding cluster by selecting it from the project database tree view and clicking on the <Apply> button.

> The program performs a consistency check on the material parameters and will give a warning message in the case of a detected inconsistency in the data.

### ***Mesh Generation***

When the geometry model is complete, the finite element model (or mesh) can be generated. PLAXIS allows for a fully automatic mesh generation procedure, in which the geometry is divided into elements of the basic element type and compatible structural elements, if applicable. The mesh generation takes full account of the position of points and lines in the geometry model, so that the exact position of layers, loads and structures is accounted for in the finite element mesh. The generation process is based on a robust triangulation principle that searches for optimised triangles and which results in an unstructured mesh. Unstructured meshes are not formed from regular patterns of elements. The numerical performance of these meshes, however, is usually better than structured meshes with regular arrays of elements. In addition to the mesh generation itself, a transformation of input data (properties, boundary conditions, material sets, etc.) from the geometry model (points, lines and clusters) to the finite element mesh (elements, nodes and stress points) is made.

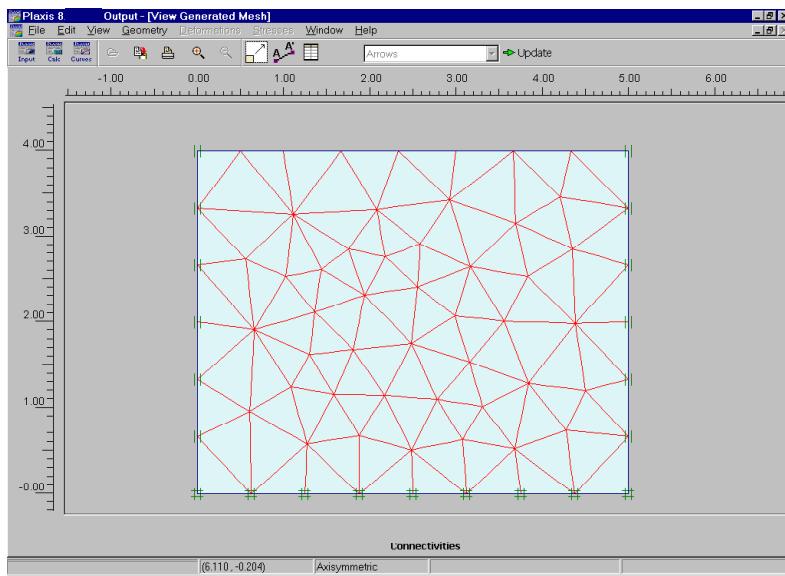


Figure 3.8 Axisymmetric finite element mesh of the geometry around the footing

In order to generate the mesh, follow these steps:



Click on the *Generate mesh* button in the toolbar or select the *Generate* option from the *Mesh* menu.

After the generation of the mesh a new window is opened (Output window) in which the generated mesh is presented (see Figure 3.8).

- Click on the <Update> button to return to the geometry input mode.

**Hint:** The <Update> button must always be used to return to the geometry input, even if the result from the mesh generation is not satisfactory.

**Hint:** By default, the *Global coarseness* of the mesh is set to *Coarse*, which is adequate as a first approach in most cases. The *Global coarseness* setting can be changed in the *Mesh* menu. In additional options are available to refine the mesh globally or locally.  
 > At this stage of input it is still possible to modify parts of the geometry or to add geometry objects. If modifications are made at this stage, then the finite element mesh has to be regenerated.

If necessary, the mesh can be optimised by performing global or local refinements. Mesh refinements are considered in some of the other lessons. Here it is suggested that the current finite element mesh is accepted.

### ***Initial Conditions***

Once the mesh has been generated, the finite element model is complete. Before starting the calculations, however, the initial conditions must be generated. In general, the initial conditions comprise the initial groundwater conditions, the initial geometry configuration and the initial effective stress state. The sand layer in the current footing project is dry, so there is no need to enter groundwater conditions. The analysis does, however, require the generation of initial effective stresses by means of the  $K_0$ -procedure.

The initial conditions are entered in separate modes of the Input program. In order to generate the initial conditions properly, follow these steps:

 Click on the *Initial conditions* button on the toolbar or select the *Initial conditions* option from the *Initial* menu.

- First a small window appears showing the default value of the unit weight of water, which is 10 ( $\text{kN/m}^3$ ). Click <OK> to accept the default value, after which the groundwater conditions mode appears. Note that the toolbar and the background of the geometry have changed compared to the geometry input mode.

The initial conditions option consists of two different modes: The water pressures mode and the geometry configuration mode. Switching between these two modes is done by the 'switch' in the toolbar.

 Since the current project does not involve water pressures, proceed to the geometry configuration mode by clicking on the right hand side of the 'switch' (*Initial stresses and geometry configuration*). A phreatic level is automatically placed at the bottom of the geometry.

 Click on the *Generate initial stresses* button (red crosses) in the toolbar or select the *Initial stresses* option from the *Generate* menu. The  $K_0$ -procedure dialog box appears.

- Keep the total multiplier for soil weight,  $\Sigma Mweight$ , equal to 1.0. This means that the full weight of the soil is applied for the generation of initial stresses. Accept the default values of  $K_0$  as suggested by PLAXIS and click on the <OK> button.

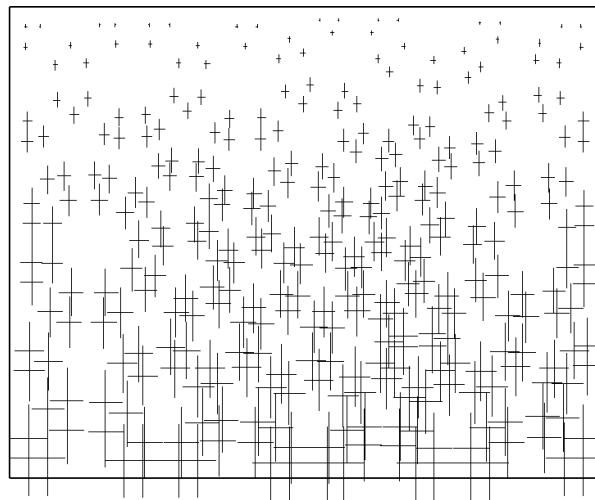


Figure 3.9 Initial stress field in the geometry around the footing

**Hint:** The  $K_0$ -procedure may only be used for horizontally layered geometries with a horizontal ground surface and, if applicable, a horizontal phreatic level. See Appendix A or the Reference Manual for more information on the  $K_0$ -procedure.

> The default value of  $K_0$  is based on Jaky's formula:  $K_0 = 1 - \sin\phi$ . If the value was changed, the default value can be regained by entering a negative value for  $K_0$ .

- After the generation of the initial stresses the Output window is opened in which the effective stresses are presented as principal stresses (see Figure 3.9).

The length of the lines indicates the relative magnitude of the principal stresses and the orientation of the lines indicates the principal directions. Click on the <Update> button to return to the geometry configuration mode of the Input program.



After the generation of the initial stresses, the calculation can be defined. After clicking on the <Calculate> button, the user is asked to save the data on the hard disk. **Click on the <Yes> button.** The file requester now appears. Enter an appropriate file name and click on the <Save> button.

### 3.2.2 PERFORMING CALCULATIONS

After clicking on the <Calculate> button and saving the input data, the Input program is closed and the Calculations program is started. The Calculations program may be used to define and execute calculation phases. It can also be used to select calculated phases for which output results are to be viewed.

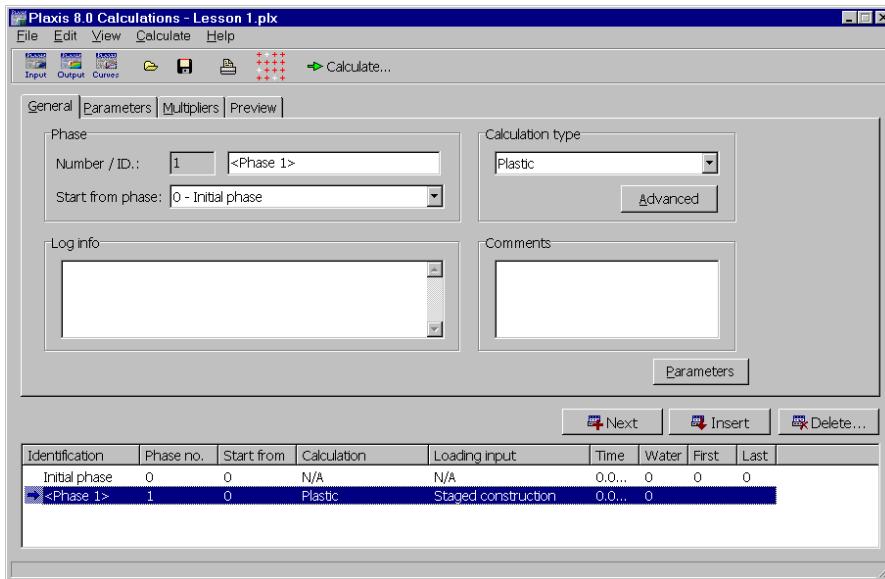


Figure 3.10 The *Calculations* window with the *General* tab sheet

The *Calculations* window consists of a menu, a toolbar, a set of tab sheets and a list of calculation phases, as indicated in Figure 3.10.

The tab sheets (*General*, *Parameters* and *Multipliers*) are used to define a calculation phase. This can be a loading, construction or excavation phase, a consolidation period or a safety analysis. For each project multiple calculation phases can be defined. All defined calculation phases appear in the list at the lower part of the window. The tab sheet *Preview* can be used to show the actual state of the geometry. A preview is only available after calculation of the selected phase.

When the Calculations program is started directly after the input of a new project, a first calculation phase is automatically inserted. In order to simulate the settlement of the footing in this analysis, a plastic calculation is required. PLAXIS has a convenient procedure for automatic load stepping, which is called Load Advancement. This procedure can be used for most practical applications. Within the plastic calculation, the prescribed displacements are activated to simulate the indentation of the footing. In order to define the calculation phase, follow these steps:

- In the *Phase ID* box write (optionally) an appropriate name for the current calculation phase (for example “Indentation”) and select the phase from which the current phase should start (in this case the calculation can only start from phase 0 - *Initial phase*).
- In the *General* tab sheet, select *Plastic* from the combo box of the *Calculation type* box.
- Click on the <Parameters> button or click on the *Parameters* tab.

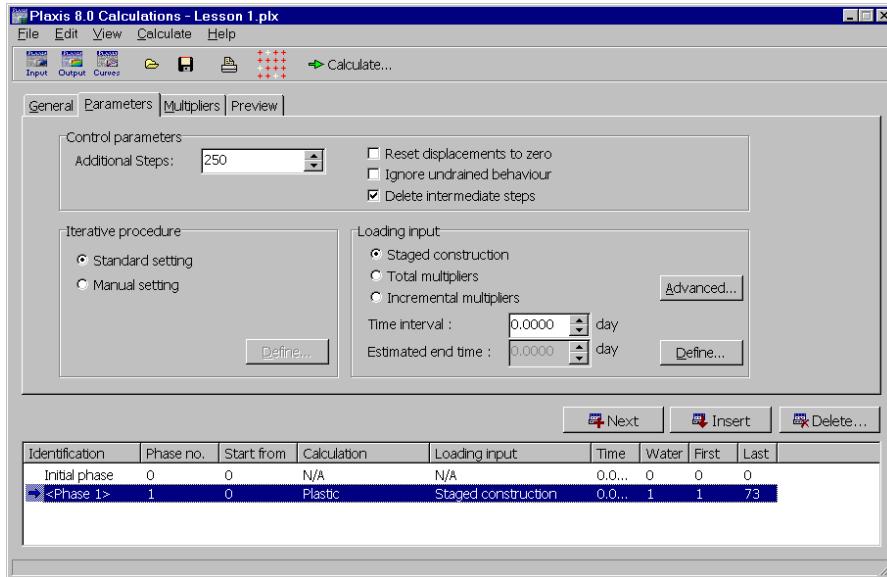


Figure 3.11 The *Calculations* window with the *Parameters* tab sheet

- The *Parameters* tab sheet contains the calculation control parameters, as indicated in Figure 3.11. Keep the default value for the maximum number of *Additional steps* (250) and select the *Standard setting* from the *Iterative procedure* box. See the Reference Manual for more information about the calculation control parameters.
- From the *Loading input* box, select *Staged Construction*.
- Click on the <Define> button.
- The *Staged Construction* window appears, showing the currently active geometry configuration. Select the prescribed displacement by double clicking on the top line. A dialog box will appear.

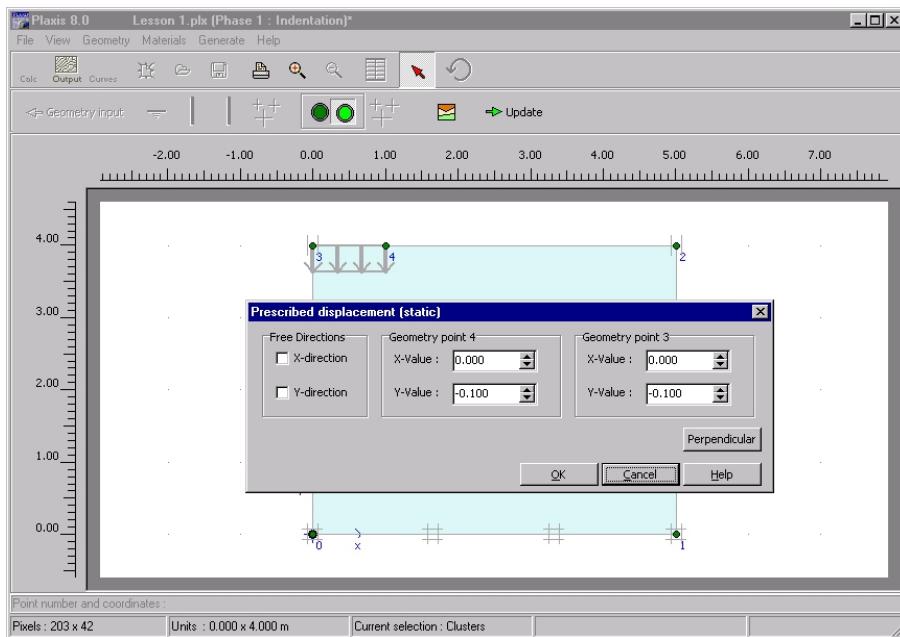


Figure 3.12 The *Prescribed Displacements* dialog box in the *Staged Construction* window

- In the *Prescribed Displacement* dialog box the magnitude and direction of the prescribed displacement can be specified, as indicated in Figure 3.12. In this case enter a *Y-value* of -0.1 in both input fields, signifying a downward displacement of 0.1 m. All *X-values* should remain zero. Click <OK>.
- Now click on the <Update> button to return to the *Parameters* tab sheet of the calculations window.

The calculation definition is now complete. Before starting the first calculation it is advisable to select nodes or stress points for a later generation of load-displacement curves or stress and strain diagrams. To do this, follow these steps:

-  Click on the *Select points for curves* button on the toolbar. As a result, a window is opened, showing all the nodes in the finite element model.
- Select the node at the top left corner. The selected node will be indicated by 'A'. Click on the <Update> button to return to the Calculations window.
  - In the *Calculations* window, click on the <Calculate> button. This will start the calculation process. All calculation phases that are selected for execution, as indicated by the blue arrow (→) (only one phase in this case) will, in principle, be executed in the order controlled by the *Start from phase* parameter.

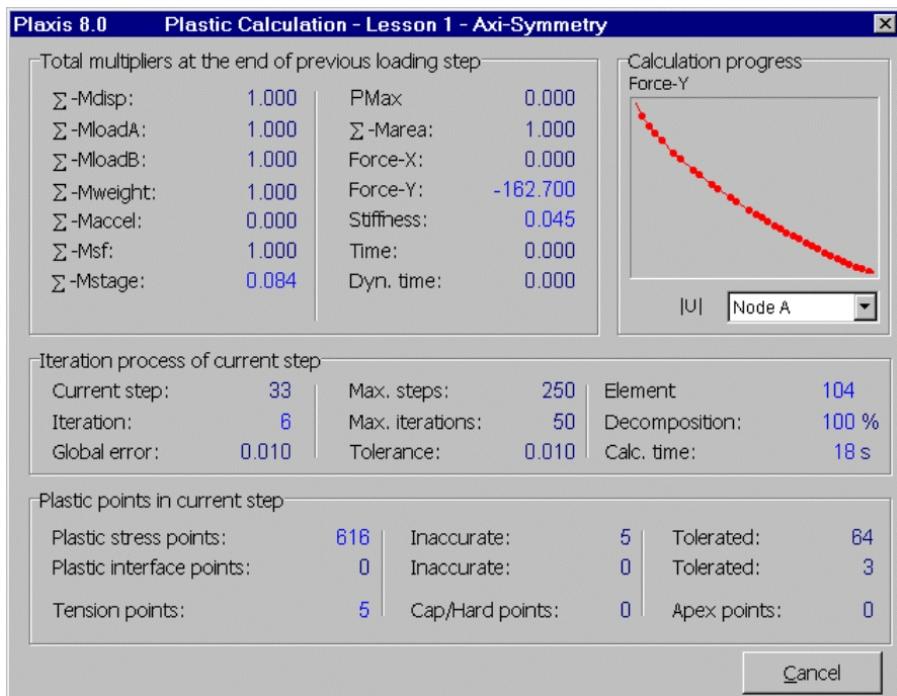


Figure 3.13 The calculations info window

**Hint:** The <Calculate> button is only visible if a calculation phase that is selected for execution is focused in the list.

During the execution of a calculation a window appears which gives information about the progress of the actual calculation phase (see Figure 3.13). The information, which is continuously updated, comprises a load-displacement curve, the level of the load systems (in terms of total multipliers) and the progress of the iteration process (iteration number, global error, plastic points, etc.). See the Reference Manual for more information about the calculations info window.

When a calculation ends, the list of calculation phases is updated and a message appears in the corresponding *Log info* memo box. The *Log info* memo box indicates whether or not the calculation has finished successfully. The current calculation should give the message 'Prescribed ultimate state fully reached'.

To check the applied load that results in the prescribed displacement of 0.1 m, click on the *Multipliers* tab and select the *Reached values* radio button. In addition to the reached values of the multipliers in the two existing columns, additional information is presented

at the left side of the window. For the current application the value of *Force-Y* is important. This value represents the total reaction force corresponding to the applied prescribed vertical displacement, which corresponds to the total force under 1.0 radian of the footing (note that the analysis is axisymmetric). In order to obtain the total footing force, the value of *Force-y* should be multiplied by  $2\pi$  (this gives a value of about 1100 kN).

- Hint:** Calculation phases may be added, inserted or deleted using the <Next>, <Insert> and <Delete> buttons half way the Calculations window.
- > Check the list of calculation phases carefully after each execution of a (series of) calculation(s). A successful calculation is indicated in the list with a green check mark (✓) whereas an unsuccessful calculation is indicated with a red cross (✗). Calculation phases that are selected for execution are indicated by a blue arrow (→).
  - > When a calculation phase is focused that is indicated by a green check mark or a red cross, the toolbar shows the <Output> button, which gives direct access to the Output program. When a calculation phase is focused that is indicated by a blue arrow, the toolbar shows the <Calculate> button.

### 3.2.3 VIEWING OUTPUT RESULTS

Once the calculation has been completed, the results can be evaluated in the Output program. In the *Output* window you can view the displacements and stresses in the full geometry as well as in cross sections and in structural elements, if applicable.

The computational results are also available in tabulated form. To view the results of the footing analysis, follow these steps:

- Click on the last calculation phase in the *Calculations* window. In addition, click on the <Output> button in the toolbar. As a result, the Output program is started, showing the deformed mesh (which is scaled to ensure that the deformations are visible) at the end of the selected calculation phase, with an indication of the maximum displacement (see Figure 3.14).
- Select *Total displacements* from the *Deformations* menu. The plot shows the total displacements of all nodes as arrows, with an indication of their relative magnitude.
- The presentation combo box in the toolbar currently reads *Arrows*. Select *Shadings* from this combo box. The plot shows colour shadings of the total displacements. An index is presented with the displacement values at the colour boundaries.

- Select *Contours* from the presentation combo box in the toolbar. The plot shows contour lines of the total displacements, which are labelled. An index is presented with the displacement values corresponding to the labels.

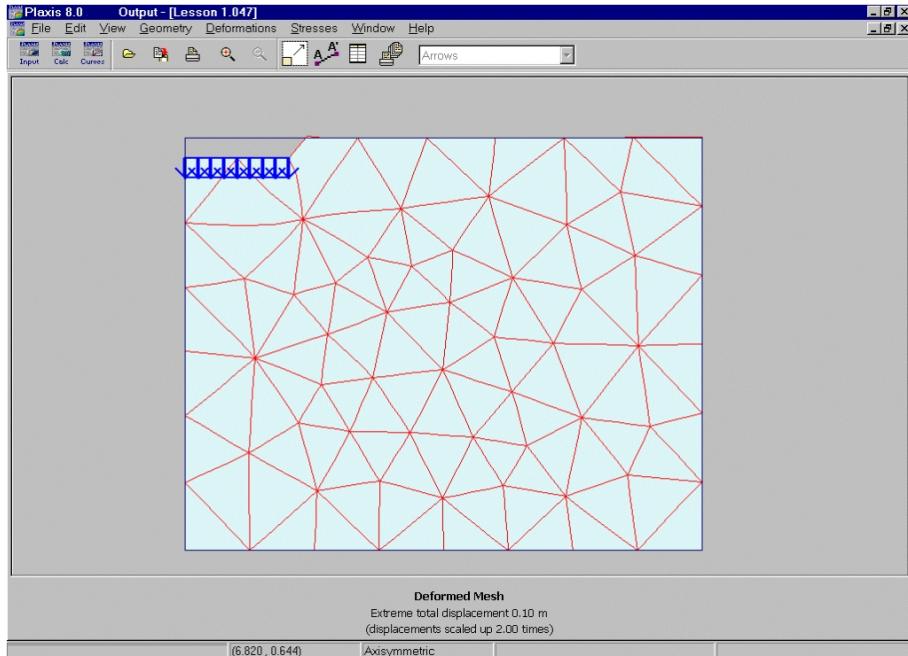


Figure 3.14 Deformed mesh

**Hint:** In addition to the total displacements, the *Deformations* menu allows for the presentation of *Incremental displacements*. The incremental displacements are the displacements that occurred within one calculation step (in this case the final step). Incremental displacements may be helpful in visualising an eventual failure mechanism.

- Select *Effective stresses* from the *Stresses* menu. The plot shows the effective stresses as principal stresses, with an indication of their direction and their relative magnitude (see Figure 3.15).

**Hint:** The plots of stresses and displacements may be combined with geometrical features, as available in the *Geometry* menu.

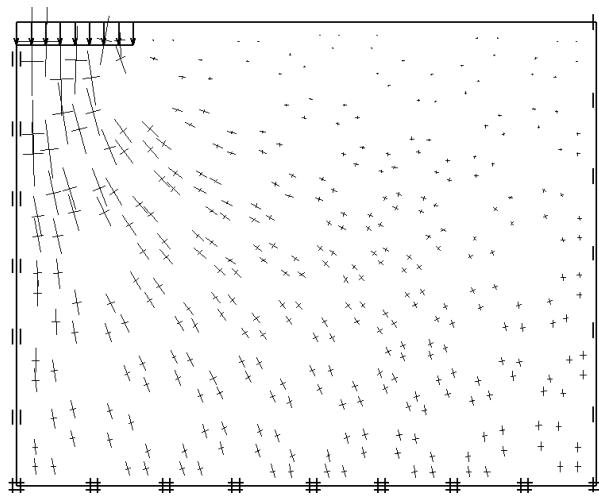


Figure 3.15 Principal stresses



Click on the *Table* button on the toolbar. A new window is opened in which a table is presented, showing the values of the Cartesian stresses in each stress point of all elements.

### 3.3 CASE B: FLEXIBLE FOOTING

The project is now modified so that the footing is modelled as a flexible plate. This enables the calculation of structural forces in the footing. The geometry used in this exercise is the same as the previous one, except that additional elements are used to model the footing. The calculation itself is based on the application of load rather than prescribed displacements. It is not necessary to create a new model; you can start from the previous model, modify it and store it under a different name. To perform this, follow these steps:

#### *Modifying the geometry*



Click on the *Go to Input* button at the left hand side of the toolbar.

- Select the previous file (“lesson1” or whichever name it was given) from the *Create/Open project* window.
- Select the *Save as* option of the *File* menu. Enter a non-existing name for the current project file and click on the <Save> button.

- Select the geometry line on which the prescribed displacement was applied and press the <Del> key on the keyboard. Select *Prescribed displacement* from the *Select items to delete* window and click on the <Delete> button.



Click on the *Plate* button in the toolbar.

- Move to position (0.0; 4.0) and press the left mouse button.
- Move to position (1.0; 4.0) and press the left mouse button, followed by the right mouse button to finish the drawing. A plate from point 3 to point 4 is created which simulates the flexible footing.

### ***Modifying the boundary conditions***



Click on the *Distributed load - load system A* button in the toolbar.

- Click on point (0.0; 4.0) and then on point (1.0; 4.0).
- Press the right mouse button to finish the input of distributed loads. Accept the default input value of the distributed load (1.0 kN/m<sup>2</sup> perpendicular to the boundary). The input value will later be changed to the real value when the load is activated.

### ***Adding material properties for the footing***



Click on the *Material sets* button.

- Select *Plates* from the *Set type* combo box in the *Material Sets* window.
- Click on the <New> button. A new window appears where the properties of the footing can be entered.
- Write “Footing” in the *Identification* box and select the *Elastic* material type.
- Enter the properties as listed in Table 3.2.
- Click on the <OK> button. The new data set now appears in the tree view of the *Material Sets* window.
- Drag the set “Footing” to the draw area and drop it on the footing. Note that the cursor changes shape to indicate that it is valid to drop the material set.
- Close the database by clicking on the <OK> button

Table 3.2. Material properties of the footing

Parameter	Name	Value	Unit
Normal stiffness	$EA$	$5 \cdot 10^6$	kN/m
Flexural rigidity	$EI$	8500	$\text{kNm}^2/\text{m}$
Equivalent thickness	$d$	0.143	m
Weight	$w$	0.0	kN/m/m
Poisson's ratio	$\nu$	0.0	-

**Hint:** If the *Material Sets* window is displayed over the footing and hides it, move the window to another position so that the footing is clearly visible.

**Hint:** The equivalent thickness is automatically calculated by PLAXIS from the values of EA and EI. It cannot be entered by hand.

### ***Generating the mesh***



Click on the *Mesh generation* button to generate the finite element mesh. A warning appears, suggesting that the water pressures and initial stresses should be regenerated after regenerating the mesh. Press the <OK> button.

- After viewing the mesh, click on the <Update> button.

**Hint:** Regeneration of the mesh results in a redistribution of nodes and stress points. In general, existing stresses will not correspond with the new position of the stress points. Therefore it is important to regenerate the initial water pressures and initial stresses after regeneration of the mesh.

### ***Initial conditions***

Back in the *Geometry input* mode, click on the <Initial conditions> button.

Since the current project does not involve pore pressures, proceed to the *Geometry configuration* mode by clicking on the 'switch' in the toolbar.

- Click on the *Generate initial stresses* button, after which the  $K_0$ -procedure dialog box appears.
- Keep  $\Sigma M weight$  equal to 1.0 and accept the default value of  $K_0$  for the single cluster.
  - Click on the <OK> button to generate the initial stresses.

- After viewing the initial stresses, click on the <Update> button.
- Click on the <Calculate> button and confirm the saving of the current project.

### **Calculations**

- In the *General* tab sheet, select for the *Calculation type: Plastic*.
- Enter an appropriate name for the phase identification and accept *0 - initial phase* as the phase to start from.
- In the *Parameters* tab sheet, select the *Staged construction* option and click on the <Define> button.
- The plot of the active geometry will appear. Click on the load to activate it. A *Select items* dialog box will appear. Activate both the plate and the load by checking the check boxes to the left..
- While the load the load is selected, click on the <Change> button at the bottom of the dialog box. The *distributed load – load system A* dialog box will appear to set the loads. Enter a *Y-value* of  $-350 \text{ kN/m}^2$  for both geometry points. Note that this gives a total load that is approximately equal to the footing force that was obtained from the first part of this lesson.
- $(350 \text{ kN/m}^2 \times \pi \times (1.0 \text{ m})^2 \approx 1100 \text{ kN})$ .
- Close the dialog boxes and press <Update>.



- Check the nodes and stress points for load-displacement curves to see if the proper points are still selected (the mesh has been regenerated so the nodes might have changed!). The top left node of the mesh should be selected.
- Check if the calculation phase is marked for calculation by a blue arrow. If this is not the case double click on the calculation phase or right click and select *Mark calculate* from the pop-up menu. Click on the <Calculate> button to start the calculation.

### **Viewing the results**

- After the calculation the results of the final calculation step can be viewed by clicking on the <Output> button. Select the plots that are of interest. The displacements and stresses should be similar to those obtained from the first part of the exercise.
- Double-click on the footing. A new window opens in which either the displacements or the bending moments of the footing may be plotted (depending on the type of plot in the first window).
- Note that the menu has changed. Select the various options from the *Forces* menu to view the forces in the footing.

**Hint:** Multiple (sub-)windows may be opened at the same time in the Output program. All windows appear in the list of the *Window* menu. PLAXIS follows the Windows standard for the presentation of sub-windows (*Cascade*, *Tile*, *Minimize*, *Maximize*, etc). See your Windows manual for a description of these standard possibilities.

### ***Generating a load-displacement curve***

In addition to the results of the final calculation step it is often useful to view a load-displacement curve. Therefore the fourth program in the PLAXIS package is used. In order to generate the load-displacement curve as given in Figure 3.17, follow these steps:



Click on the *Go to curves program* button on the toolbar. This causes the Curves program to start.

- Select *New chart* from the *Create / Open project* dialog box.
- Select the file name of the latest footing project and click on the <Open> button.

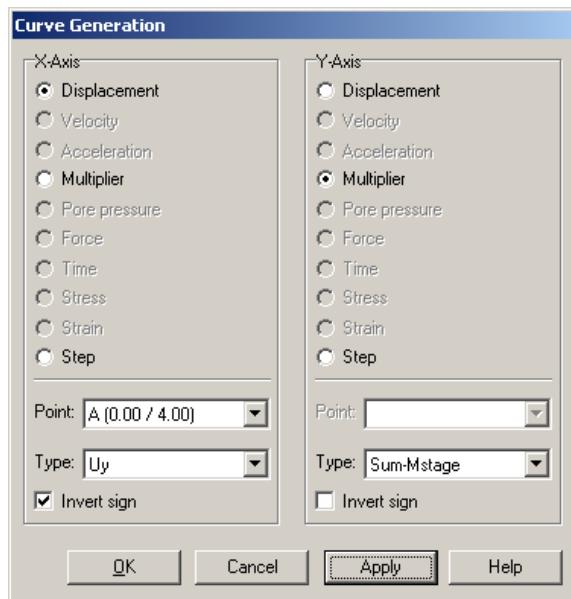


Figure 3.16 Curve generation window

A *Curve generation* window now appears, consisting of two columns (x-axis and y-axis), with multi select radio buttons and two combo boxes for each column. The

combination of selections for each axis determines which quantity is plotted along the axis.

- For the *X-axis* select the *Displacement* radio button, from the *Point* combo box select *A (0.00 / 4.00)* and from the *Type* combo box  $U_y$ . Also select the *Invert sign* check box. Hence, the quantity to be plotted on the x-axis is the vertical displacement of point A (i.e. the centre of the footing).
- For the *Y-axis* select the *Multiplier* radio button and from the *Type* combo box select  $\Sigma M_{stage}$ . Hence, the quantity to be plotted on the y-axis is the amount of the specified changes that has been applied. Hence the value will range from 0 to 1, which means that 100% of the prescribed load ( $350 \text{ kN/m}^2$ ) has been applied and the prescribed ultimate state has been fully reached.
- Click on the <OK> button to accept the input and generate the load-displacement curve. As a result the curve of Figure 3.17 is plotted in the *Curves* window.

**Hint:** The *Curve settings* window may also be used to modify the attributes or presentation of a curve.

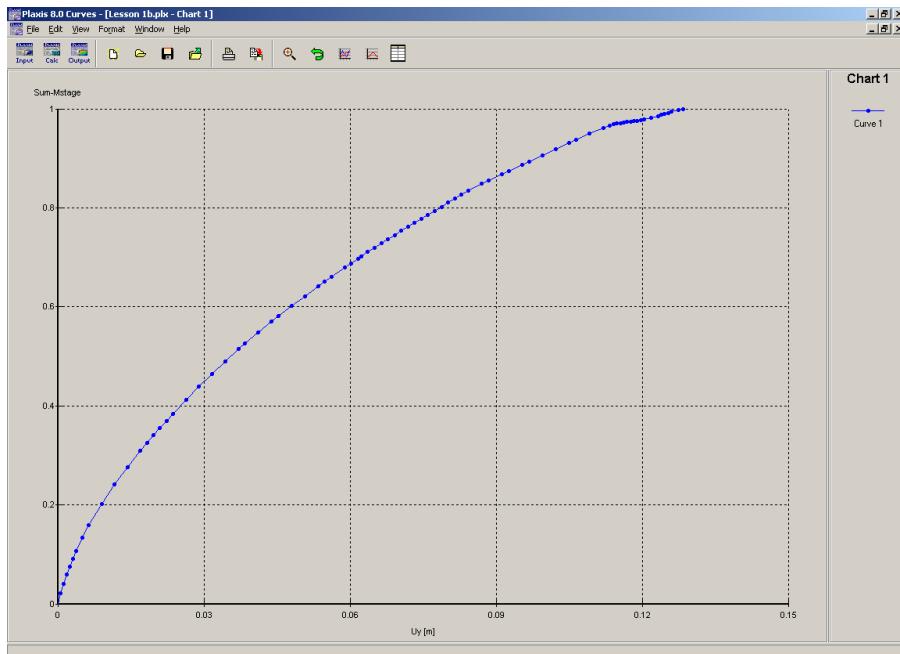


Figure 3.17 Load-displacement curve for the footing

- Hint:** To re-enter the *Curve generation* window (in the case of a mistake, a desired regeneration or modification) you can click on the *Change curve settings* button from the toolbar. As a result the *Curve settings* window appears, on which you should click on the <Regenerate> button. Alternatively, you may open the *Curve settings* window by selecting the *Curve* option from the *Format* menu.
- > The *Frame settings* window may be used to modify the settings of the frame. This window can be opened by clicking on the *Change frame settings* button from the toolbar or selecting the *Frame* option from the *Format* menu.

### ***Comparison between Case A and Case B***

When comparing the calculation results obtained from Case A and Case B, it can be noticed that the footing in Case B, for the same maximum load of 1100 kN, exhibited more deformation than that for Case A. This can be attributed to the fact that in Case B a finer mesh was generated due to the presence of a plate element. (By default, PLAXIS generates smaller soil elements at the contact region with a plate element) In general, geometries with coarse meshes may not exhibit sufficient flexibility, and hence may experience less deformation. The influence of mesh coarseness on the computational results is pronounced more in axisymmetric models. If, however, the same mesh was used, the two results would match quite well.

#### 4 SUBMERGED CONSTRUCTION OF AN EXCAVATION (LESSON 2)

This lesson illustrates the use of PLAXIS for the analysis of submerged construction of an excavation. Most of the program features that were used in Lesson 1 will be utilised here again. In addition, some new features will be used, such as the use of interfaces and anchor elements, the generation of water pressures and the use of multiple calculation phases. The new features will be described in full detail, whereas the features that were treated in Lesson 1 will be described in less detail. Therefore it is suggested that Lesson 1 should be completed before attempting this exercise.

This lesson concerns the construction of an excavation close to a river. The excavation is carried out in order to construct a tunnel by the installation of prefabricated tunnel segments. The excavation is 30 m wide and the final depth is 20 m. It extends in longitudinal direction for a large distance, so that a plane strain model is applicable. The sides of the excavation are supported by 30 m long diaphragm walls, which are braced by horizontal struts at an interval of 5.0 m. Along the excavation a surface load is taken into account. The load is applied from 2 meter from the diaphragm wall up to 7 meters from the wall and has a magnitude of  $5 \text{ kN/m}^2/\text{m}$ .

The upper 20 m of the subsoil consists of soft soil layers, which are modelled as a single homogeneous clay layer. Underneath this clay layer there is a stiffer sand layer, which extends to a large depth.

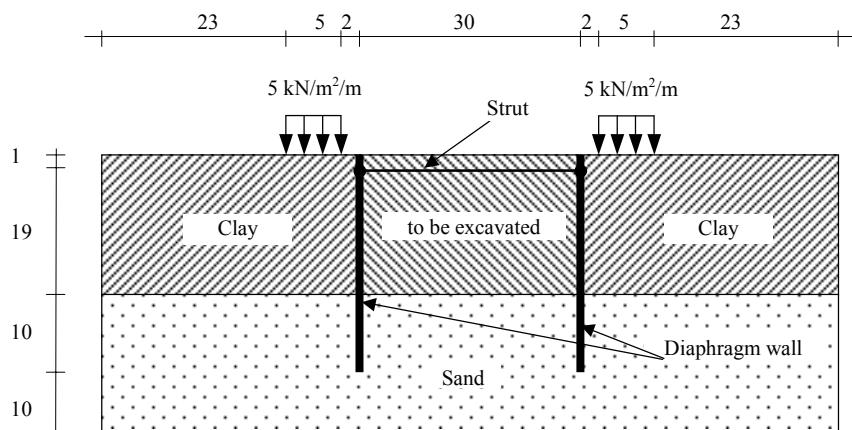


Figure 4.1 Geometry model of the situation of a submerged excavation

The bottom of the problem to be analysed is taken at 40 m below the ground surface. Since the geometry is symmetric, only one half (the left side) is considered in the analysis. The excavation process is simulated in three separate excavation stages. The diaphragm wall is modelled by means of a plate, such as used for the footing in the

previous lesson. The interaction between the wall and the soil is modelled at both sides by means of interfaces. The interfaces allow for the specification of a reduced wall friction compared to the friction in the soil. The strut is modelled as a spring element for which the normal stiffness is a required input parameter.

For background information on these new objects, see the Reference Manual.

## 4.1 GEOMETRY

To create the geometry model, follow these steps:

### **General settings**

- Start the Input program and select *New project* from the *Create / Open project* dialog box.
- In the *Project* tab sheet of the *General settings* window, enter an appropriate title and make sure that *Model* is set to *Plane strain* and that *Elements* is set to *15-node*.
- In the *Dimensions* tab sheet, keep the default units (*Length* = m; *Force* = kN; *Time* = day) and enter for the horizontal dimensions (*Left*, *Right*) 0.0 and 45.0 respectively and for the vertical dimensions (*Bottom*, *Top*) 0.0 and 40.0. Keep the default values for the grid spacing (*Spacing*=1m; *Number of intervals* = 1).
- Click on the <OK> button after which the worksheet appears.

### **Geometry contour, layers and structures**



The geometry contour: Select the *Geometry line* button from the toolbar (this should, in fact, already be selected for a new project). Move the cursor to the origin (0.0; 0.0) and click the left mouse button. Move 45 m to the right (45.0; 0.0) and click again. Move 40 m up (45.0; 40.0) and click again. Move 45 m to the left (0.0; 40.0) and click again. Finally, move back to the origin and click again. A cluster is now detected. Click the right mouse button to stop drawing.

- The separation between the two layers: The *Geometry line* button is still selected. Move the cursor to position (0.0; 20.0). Click on the existing vertical line. A new point (4) is now introduced. Move 45 m to the right (45.0; 20.0) and click on the other existing vertical line. Another point (5) is introduced and now two clusters are detected.



The diaphragm wall: Select the *Plate* button from the toolbar. Move the cursor to position (30.0; 40.0) at the upper horizontal line and click. Move 30 m down (30.0; 10.0) and click. In addition to the point at the toe of the wall, another point is introduced at the intersection with the middle horizontal line (layer separation). Click the right mouse button to finish the drawing.



The separation of excavation stages: Select the *Geometry line* button again. Move the cursor to position (30.0; 38.0) at the wall and click. Move the cursor 15 m to the right (45.0; 38.0) and click again. Click the right mouse button to finish drawing the first excavation stage. Now move the cursor to position (30.0; 30.0) and click. Move to (45.0; 30.0) and click again. Click the right mouse button to finish drawing the second excavation stage.

**Hint:**

Within the geometry input mode it is not strictly necessary to select the buttons in the toolbar in the order that they appear from left to right. In this case, it is more convenient to create the wall first and then enter the separation of the excavation stages by means of a *Geometry line*.

- > When creating a point very close to a line, the point is usually snapped onto the line, because the mesh generator cannot handle non-coincident points and lines at a very small distance. This procedure also simplifies the input of points that are intended to lie exactly on an existing line.
- > If the pointer is substantially mispositioned and instead of snapping onto an existing point or line a new isolated point is created, this point may be dragged (and snapped) onto the existing point or line by using the *Selection* button.
- > In general, only one point can exist at a certain coordinate and only one line can exist between two points. Coinciding points or lines will automatically be reduced to single points or lines. The procedure to drag points onto existing points may be used to eliminate redundant points (and lines).



The interfaces: Click on the *Interface* button on the toolbar or select the *Interface* option from the *Geometry* menu. The shape of the cursor will change into a cross with an arrow in each quadrant. The arrows indicate the side at which the interface will be generated when the cursor is moved in a certain direction.

- Move the cursor (the centre of the cross defines the cursor position) to the top of the wall (30.0; 40.0) and click the left mouse button. Move to the bottom of the wall (30.0; 10.0) and click again. According to the position of the 'down' arrow at the cursor, an interface is generated at the left hand side of the wall. Similarly, the 'up' arrow is positioned at the right side of the cursor, so when moving up to the top of the wall and clicking again, an interface is generated at the right hand side of the wall. Move back to (30.0; 40.0) and click again. Click the right mouse button to finish drawing.

**Hint:**

The selection of an interface is done by selecting the corresponding geometry line and subsequently selecting the corresponding interface (positive or negative) from the *Select* dialog box.

**Hint:** Interfaces are indicated as dotted lines along a geometry line. In order to identify interfaces at either side of a geometry line, a positive sign ( $\oplus$ ) or negative sign ( $\ominus$ ) is added. This sign has no physical relevance or influence on the results.



The strut: Click on the *Fixed-end anchor* button on the toolbar or select the *Fixed-end anchor* option from the *Geometry* menu. Move the cursor to a position 1 metre below point 6 (30.0; 39.0) and click the left mouse button. A properties window appears in which the orientation angle and the equivalent length of the anchor can be entered. Enter an *Equivalent length* of 15 m (half the width of the excavation) and click on the <OK> button (the orientation angle remains  $0^\circ$ ).

**Hint:** A fixed-end anchor is represented by a rotated T with a fixed size. This object is actually a spring of which one end is connected to the mesh and the other end is fixed. The orientation angle and the equivalent length of the anchor must be directly entered in the properties window. The equivalent length is the distance between the connection point and the position in the direction of the anchor rod where the displacement is zero. By default, the equivalent length is 1.0 unit and the angle is zero degrees (i.e. the anchor points in the positive x-direction).  
> Clicking on the 'middle bar' of the corresponding T selects an existing fixed-end anchor.



The surface load: Click on the *Distributed load – load system A*. Move the cursor to (23.0; 40.0) and click. Move the cursor 5 m to the right to (28.0; 40.0) and click again. Right click to finish drawing. Click on the *Selection* tool and double click on the distributed load and select *Distributed Load (System A)* from the list. Enter *Y-values* of  $-5 \text{ kN/m}^2$ .

### Boundary Conditions



To create the boundary conditions, click on the *Standard fixities* button on the toolbar. As a result, the program will generate full fixities at the bottom and vertical rollers at the vertical sides. These boundary conditions are in this case appropriate to model the conditions of symmetry at the right hand boundary (center line of the excavation). The geometry model so far is shown in Figure 4.2.

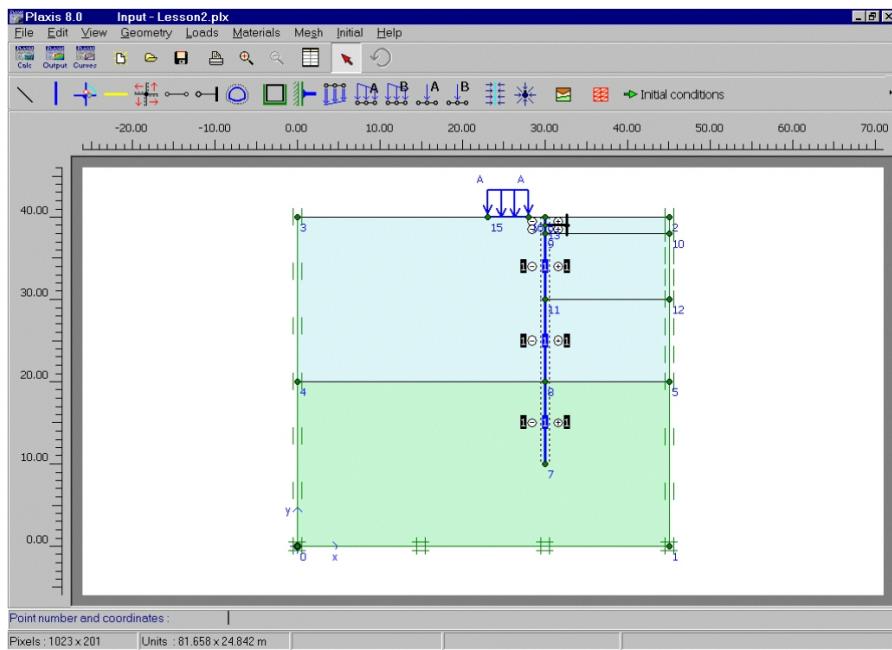


Figure 4.2 Geometry model in the Input window

### **Material properties**

After the input of boundary conditions, the material properties of the soil clusters and other geometry objects are entered in data sets. Interface properties are included in the data sets for soil (Data sets for *Soil & interfaces*). Two data sets need to be created; one for the clay layer and one for the sand layer. In addition, a data set of the *Plate* type is created for the diaphragm wall and a data set of the *Anchor* type is created for the strut. To create the material data sets, follow these steps:



Click on the *Material sets* button on the toolbar. Select *Soil & interfaces* as the *Set type*. Click on the <New> button to create a new data set.

- For the clay layer, enter 'Clay' for the *Identification* and select *Mohr-Coulomb* as the *Material model*. Since only long-term effects of the excavation are considered here, we will not take into account the undrained behaviour. Hence, the material type is set to *Drained*.
- Enter the properties of the clay layer, as listed Table 4.1, in the corresponding edit boxes of the *General* and *Parameters* tab sheet.

Click on the *Interfaces* tab. In the *Strength* box, select the *Manual* radio button. Enter a value of 0.5 for the parameter  $R_{inter}$ . This parameter relates the strength of the soil to the strength in the interfaces, according to the equations:

$$\tan \varphi_{interface} = R_{inter} \tan \varphi_{soil} \quad \text{and} \quad c_{inter} = R_{inter} c_{soil}$$

where:

$$c_{soil} = c_{ref} \text{ (see Table 4.1)}$$

Hence, using the entered  $R_{inter}$ -value gives a reduced interface friction and interface cohesion (adhesion) compared to the friction angle and the cohesion in the adjacent soil.

Table 4.1. Material properties of the sand and clay layer and the interfaces

Parameter	Name	Clay layer	Sand layer	Unit
Material model	<i>Model</i>	Mohr-Coulomb	Mohr-Coulomb	-
Type of material behaviour	<i>Type</i>	Drained	Drained	-
Soil unit weight above phreatic level	$\gamma_{unsat}$	16	17	kN/m <sup>3</sup>
Soil unit weight below phreatic level	$\gamma_{sat}$	18	20	kN/m <sup>3</sup>
Permeability in hor. direction	$k_x$	0.001	1.0	m/day
Permeability in ver. direction	$k_y$	0.001	1.0	m/day
Young's modulus (constant)	$E_{ref}$	10000	40000	kN/m <sup>2</sup>
Poisson's ratio	$\nu$	0.35	0.3	-
Cohesion (constant)	$c_{ref}$	5.0	1.0	kN/m <sup>2</sup>
Friction angle	$\varphi$	25	32	°
Dilatancy angle	$\psi$	0.0	2.0	°
Strength reduction factor inter.	$R_{inter}$	0.5	0.67	-

- For the sand layer, enter 'Sand' for the *Identification* and select *Mohr-Coulomb* as the *Material model*. The material type should be set to *Drained*.
- Enter the properties of the sand layer, as listed Table 4.1, in the corresponding edit boxes of the *General* and *Parameters* tab sheet.
- Click on the *Interfaces* tab. In the *Strength* box, select the *Manual* radio button. Enter a value of 0.67 for the parameter  $R_{inter}$ . Close the data set.
- Drag the 'Sand' data set to the lower cluster of the geometry and drop it there. Assign the 'Clay' data set to the remaining four clusters (in the upper 20 m). By default, interfaces are automatically assigned the data set of the adjacent cluster.

**Hint:** Instead of accepting the default data sets of interfaces, data sets can directly be assigned to interfaces in their properties window. This window appears after double clicking the corresponding geometry line and selecting the appropriate interface from the *Select* dialog box. On clicking the <Change> button behind the *Material set* parameter, the proper data set can be selected from the *Material sets* tree view.

> In addition to the *Material set* parameter in the properties window, the *Virtual thickness factor* can be entered. This is a purely numerical value, which can be used to optimise the numerical performance of the interface. Non-experienced users are advised not to change the default value. For more information about interface properties see the Reference Manual.

- Set the *Set type* parameter in the *Material sets* window to *Plates* and click on the <New> button. Enter “Diaphragm wall” as an *Identification* of the data set and enter the properties as given in Table 4.2. Click on the <OK> button to close the data set.
- Drag the *Diaphragm wall* data set to the wall in the geometry and drop it as soon as the cursor indicates that dropping is possible.

Table 4.2. Material properties of the diaphragm wall (Plate)

Parameter	Name	Value	Unit
Type of behaviour	<i>Material type</i>	Elastic	
Normal stiffness	<i>EA</i>	$7.5 \cdot 10^6$	kN/m
Flexural rigidity	<i>EI</i>	$1.0 \cdot 10^6$	$\text{kNm}^2/\text{m}$
Equivalent thickness	<i>d</i>	1.265	m
Weight	<i>w</i>	10.0	kN/m/m
Poisson's ratio	<i>v</i>	0.0	-

**Hint:** The radio button *Rigid* in the *Strength* box is a direct option for an interface with the same strength properties as the soil ( $R_{inter} = 1.0$ ).

- Set the *Set type* parameter in the *Material sets* window to *Anchors* and click on the <New> button. Enter “Strut” as an *Identification* of the data set and enter the properties as given in Table 4.3. Click on the <OK> button to close the data set.
- Drag the *Strut* data set to the anchor in the geometry and drop it as soon as the cursor indicates that dropping is possible. Close the *Material sets* window.

Table 4.3. Material properties of the strut (anchor)

Parameter	Name	Value	Unit
Type of behaviour	<i>Material type</i>	Elastic	
Normal stiffness	$EA$	$2 \cdot 10^6$	kN
Spacing out of plane	$L_s$	5.0	m
Maximum force	$F_{max,comp}$	$1 \cdot 10^{15}$	kN
	$F_{max,tens}$	$1 \cdot 10^{15}$	kN

### Mesh Generation

In this lesson some simple mesh refinement procedures are used. In addition to a direct global mesh refinement, there are simple possibilities for local refinement within a cluster, on a line or around a point. These options are available from the *Mesh* menu. In order to generate the proposed mesh, follow these steps:



Click on the *Generate mesh* button in the toolbar. A few seconds later, a coarse mesh is presented in the Output window. Click on the <Update> button to return to the geometry input.

- From the *Mesh* menu, select the *Global coarseness* option. The *Element distribution* combo box is set to *Coarse*, which is the default setting. In order to refine the global coarseness, one could select the next item from the combo box (*Medium*) and click on the <Generate> button. Alternatively, the *Refine global* option from the *Mesh* menu could be selected. As a result, a finer mesh is presented in the Output window. Click on the <Update> button to return.
- Corner points of structural elements may cause large displacement gradients. Hence, it is good to make those areas finer than other parts of the geometry. Click in the middle of the lowest part of the wall (single click). The selected geometry line is now indicated in red. From the *Mesh* menu, select the option *Refine line*. As a result, a local refinement of the indicated line is visible in the presented mesh. Click on the <Update> button to return.

**Hint:** The mesh settings are stored together with the rest of the input. On re-entering an existing project and not changing the geometry configuration and mesh settings, the same mesh can be regenerated by just clicking on the *Generate mesh* button on the toolbar. However, any slight change of the geometry will result in a different mesh.  
> The *Reset all* option from the *Mesh* menu may be used to restore the default setting for the mesh generation (*Global coarseness = Coarse* and no local refinement).

### Initial conditions

The initial conditions of the current project require the generation of water pressures, the deactivation of structures and loads and the generation of initial stresses. Water pressures (pore pressures and water pressures on external boundaries) can be generated in two different ways: A direct generation based on the input of phreatic levels and groundwater heads or an indirect generation based on the results of a groundwater flow calculation. The current lesson only deals with the direct generation procedure. Generation based on groundwater flow is presented in the second part of the Lesson 4 (see Section 6.2).

Within the direct generation option there are several ways to prescribe the water conditions. The simplest way is to define a general phreatic level, under which the water pressure distribution is hydrostatic, based on the input of a unit water weight. The general phreatic level is automatically assigned to all clusters for the generation of pore pressures. It is also used to generate external water pressures, if applicable. Instead of the general phreatic level, individual clusters may have a separate phreatic level or an interpolated pore pressure distribution. The latter advanced options will be demonstrated in the first part of Lesson 3 (see Section 5.2). Here only a general phreatic level is defined at 2.0 m below the ground surface.

In order to generate the proper initial pore pressures, follow these steps:

 Click on the *Initial conditions* button on the toolbar.

**Hint:** When a project is newly created, the water weight is presented directly on entering the *Groundwater mode*. On re-entering an existing project the input of the water weight can be accessed by selecting the *Water weight* option from the *Geometry* menu in the *Groundwater mode*.

 Click <OK> to accept the default value of the unit weight of water, which is 10 kN/m<sup>3</sup>. The *Groundwater conditions* mode then becomes active, in which the *Phreatic level* button is already selected. By default, a *General* phreatic level is generated at the bottom of the geometry.

- Move the cursor to position (0.0; 38.0) and click the left mouse button. Move 45 m to the right (45.0; 38.0) and click again. Click the right mouse button to finish drawing. The plot now indicates a new *General* phreatic level 2.0 m below the ground surface.

**Hint:** An existing phreatic level may be modified using the *Selection* button from the toolbar. On deleting the *General* phreatic level (selecting it and pressing the <Del> key on the keyboard), the default general phreatic level will be created again at the bottom of the geometry. The graphical input or modification of phreatic levels does not affect the existing geometry.

- > To create an accurate pore pressure distribution in the geometry, an additional geometry line can be included corresponding with the level of the groundwater head or the position of the phreatic level in a problem.



Click on the *Generate water pressures* button (blue crosses) on the toolbar. Now the *Water pressure generation* window appears.

- From the *Water pressure generation* window, select the *Phreatic level* radio button in the *Generate by* box and click the <OK> button.
- After the generation of water pressures, the result is displayed in the Output window. Click on the <Update> button to return to the *Groundwater conditions* mode.

After the generation of water pressures and before the generation of initial effective stresses, parts of the geometry that are not active in the initial state must be deactivated. This option is used initially to deactivate geometry parts (clusters or structural objects) that are to be constructed at later calculation stages. PLAXIS will automatically deactivate loads and structural elements in the initial geometry configuration.

In the current project, the diaphragm wall and the anchor are initially not present and should be deactivate for the initial geometry. The  $K_0$ -procedure for the generation of initial stresses will not take into account the deactivate geometry clusters.



Proceed to the *Geometry configuration* mode by clicking on the 'switch' in the toolbar.

- Check that the wall and the strut in the geometry are inactive. Inactive elements are coloured grey. Make sure that all soil clusters remain active.

**Hint:**

Inactive clusters are white, just like the background, whereas active clusters have the colour of the corresponding material set. Inactive structural objects are grey, whereas active structures have the basic colour as used during the creation of the geometry model.



Click on the *Generate initial stresses* button in the toolbar. The  $K_0$ -procedure dialog box appears.

- Keep the total multiplier for soil weight equal to 1.0. Accept the default values for  $K_0$  and click on the <OK> button.
- After the generation of the initial effective stresses, the result is displayed in the Output window. Click on the <Update> button to return to the *Initial configuration* mode.
- Click on the <Calculate> button. Select <Yes> in response to the question about saving the data and enter an appropriate file name.

## 4.2 CALCULATIONS

In practice, the construction of an excavation is a process that can consist of several phases. First, the wall is installed to the desired depth. Then some excavation is carried out to create space to install an anchor or a strut. Then the soil is gradually removed to the final depth of the excavation. Special measures are usually taken to keep the water out of the excavation. Props may also be provided to support the retaining wall.

In PLAXIS, these processes can be simulated by means of the *Staged construction* calculation option. *Staged construction* enables the activation or deactivation of weight, stiffness and strength of selected components of the finite element model. The current lesson explains the use of this powerful calculation option for the simulation of excavations.

**Hint:** The *Staged construction* option is not only intended to simulate excavations or constructions, but it can also be used to change the water pressure distribution, to change material properties (to simulate soil improvement, for example) or to improve the accuracy of previous computational results.

The excavation, as considered in this example, is to be carried out in five phases. The separation of the three excavation phases was taken into account during the creation of the geometry model by introducing geometry lines in the appropriate positions. In order to define the five calculation phases, follow these steps:

### Phase 1: External load

- In addition to the initial phase, the first calculation phase has already been automatically created by the program. In the *General* tab sheet, accept all defaults.
- In the *Parameters* tab sheet, keep the default value for the *Control parameters* and the *Iterative procedure*. Select *Staged construction* from the *Loading input* box.
- Click on the <Define> button. The *Staged construction* window now appears, showing the currently active part of the geometry, which is the full geometry except for the wall, strut and load. Click on the wall to activate it (the wall should become blue). In addition, click on the load to activate it. The load has been defined in *Input* as  $-5 \text{ kN/m}^2$ . You can check this by clicking on the <Change> button.

**Hint:** You can also enter or change the values of the load at this time by double clicking on the load and entering a value. If a load is applied on a structural object such as a plate, load values can be changed by clicking on the load or the object. As a result a window appears in which you can select the load. Then click on the <Change> button to modify the load values.

- Click on the <Update> button to finish the definition of the construction phase. As a result, the *Staged construction* window is closed and the *Calculations* window reappears. The first calculation phase has now been defined and saved.

#### **Phase 2: First excavation stage**

- Within the *Calculations* window, click on the <Next> button. A new calculation phase appears in the list.
- Note that the program automatically presumes that the current phase should start from the previous one.
- In the *General* tab sheet, accept all defaults. Enter the *Parameters* tab sheet and click on the <Define> button to define the next *Staged construction* step. The staged construction window now reappears. The load and the wall should be active and marked as blue elements. Click on the top right cluster in order to deactivate it and simulate the first excavation step.
- Click on the <Update> button to finish the definition of the first excavation phase.

#### **Phase 3: Installation of strut**

- To enter the third calculation phase, click on <Next> and proceed as described above. Again click <Define> to enter the *Staged construction* window. Now activate the strut by clicking on the horizontal line. The strut should turn black to indicate it is active.
- Click on <Update> to return to the calculation program and define another calculation phase.

#### **Phase 4: Second (submerged) excavation stage**

- Keep all default settings and enter the *Staged construction* window. This phase will simulate the excavation of the second part of the building pit. Deactivate the second cluster from the top on the right side of the mesh. It should be the topmost active cluster. Go to <Update> and define the final stage.

**Hint:** Note that in PLAXIS the pore pressures are not automatically deactivated when deactivating a soil cluster. Hence, in this case, the water remains in the excavated area and a submerged excavation is simulated.

#### **Phase 5: Third excavation stage**

- The final calculation stage is to simulate the excavation of the last clay layer inside the pit. Deactivate the third cluster from the top on the right hand side of the mesh. Click on <Update> to return to the *Calculations* window.

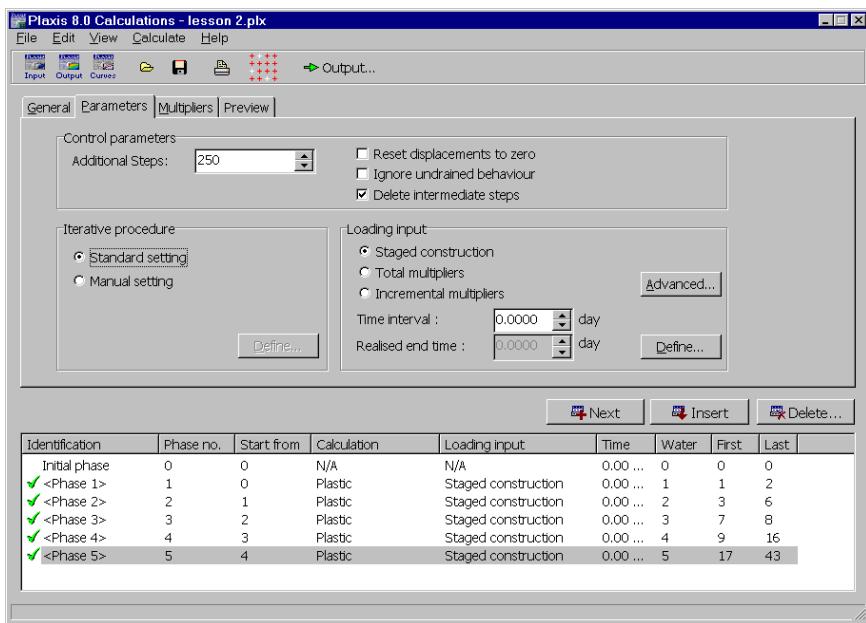


Figure 4.3 The *Calculations* window with the *Parameters* tab sheet

The calculation definition is now complete. Before starting the calculation it is suggested that you select nodes or stress points for a later generation of load-displacement curves or stress and strain diagrams. To do this, follow the steps given below.

- Click on the *Select points for curves* button on the toolbar.
- Select some nodes on the wall at points where large deflections can be expected (e.g. 30.0;30.0) and click on the <Update> button.
- In the *Calculations* window, click on the *Calculate* button.

The calculation process should now start. The program searches for the first calculation phase that is selected for execution, which is <Phase 1>.

**Hint:** To select the desired nodes, it may be convenient to use the *Zoom in* option on the toolbar to zoom into the area of interest.

During a *Staged construction* calculation, a multiplier called  $\Sigma M_{stage}$  is increased from 0.0 to 1.0. This parameter is displayed on the calculation info window. As soon as  $\Sigma M_{stage}$  has reached the value 1.0, the construction stage is completed and the calculation phase is finished. If a *Staged construction* calculation finishes while

$\Sigma M_{stage}$  is smaller than 1.0, the program will give a warning message. The most likely reason for not finishing a construction stage is that a failure mechanism has occurred, but there can be other causes as well. See the Reference Manual for more information about *Staged construction*.

**Hint:** The *Staged construction* window is similar to the *Initial conditions* window of the Input program. The main difference is that *Initial conditions* is used to create an initial situation, whereas *Staged construction* is used as a type of loading.

In this example, all calculation phases should successfully finish, which is indicated by the green check boxes in the list. In order to check the values of the  $\Sigma M_{stage}$  multiplier, click on the *Multipliers* tab and select the *Reached values* radio button. The  $\Sigma M_{stage}$  parameter is displayed at the bottom of the *Other* box that pops up. Verify that this value is equal to 1.0. You also might wish to do the same for the other calculation phase.

### 4.3 VIEWING OUTPUT RESULTS

In addition to the displacements and the stresses in the soil, the Output program can be used to view the forces in structural objects. To examine the results of this project, follow these steps:

- Click on the final calculation phase in the *Calculations* window.
- Click on the <Output> button on the toolbar. As a result, the Output program is started, showing the deformed mesh (scaled up) at the end of the selected calculation phase, with an indication of the maximum displacement (Figure 4.4). The loads shown inside the excavation represent the remaining water pressures.
- Select *Total increments* from the *Deformations* menu. The plot shows the displacement increments of all nodes as arrows. The length of the arrows indicates the relative magnitude.
- The presentation combo box in the toolbar currently reads *Arrows*. Select *Shadings* from this combo box. The plot should now show colour shadings of the displacement increments. From this plot a zone of intense shearing is visible behind the wall.
- Select *Effective stresses* from the *Stresses* menu. The plot shows the magnitude and direction of the principal effective stresses. The orientation of the principal stresses indicates a large passive zone under the bottom of the excavation and a small passive zone behind the strut (see Figure 4.5).

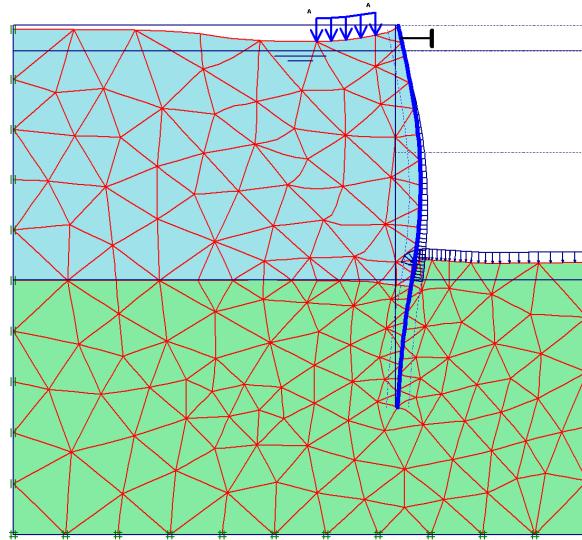


Figure 4.4 Deformed mesh after submerged excavation

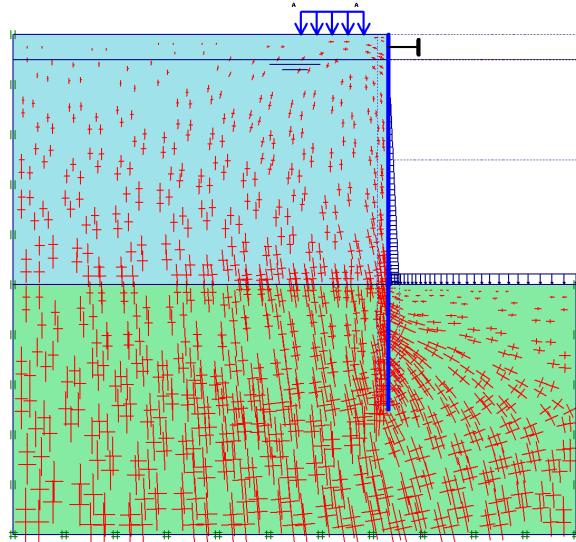


Figure 4.5 Principal stresses after excavation

To plot the shear forces and bending moments in the wall follow the steps given below.

- Double click on the wall. A new window is opened showing the bending moments in the wall, with an indication of the maximum moment (see Figure 4.6). Note that the menu has changed.

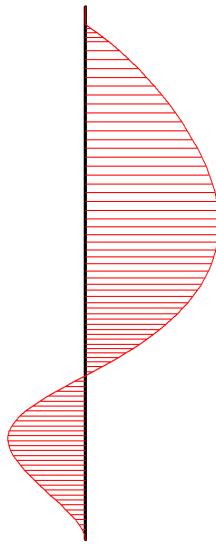


Figure 4.6 Bending moments in the wall

- Select *Shear forces* from the *Forces* menu. The plot now shows the shear forces in the wall.

**Hint:** The *Window* menu may be used to switch between the window with the forces in the wall and the stresses in the full geometry. This menu may also be used to *Tile* or *Cascade* the two windows, which is a common option in a Windows environment.

- Select the first window (showing the effective stresses in the full geometry) from the *Window* menu. Double click on the strut. A new window is now opened showing the strut force in kN/m. This value must be multiplied by the out of plane spacing of the struts to calculate the individual strut forces (in kN).
- Click on the *Go to curves program* button on the toolbar. As a result, the load-displacement curves program is started.

- Select *New chart* from the *Create / Open project* dialog box and select the file name of the excavation project from the file requester.
- In the *Curve generation* window, select for the *X-axis* the *Displacement* radio button and point *A* (e.g. 30.0;30.0) and from the *Type* combo box select the item  $|U|$ . Select for the *Y-axis* the *Multiplier* radio button and from the *Type* combo box  $\Sigma M_{stage}$ .
- Click on the <OK> button to accept the input and generate the load-displacement curve. As a result the curve of Figure 4.7 is plotted.

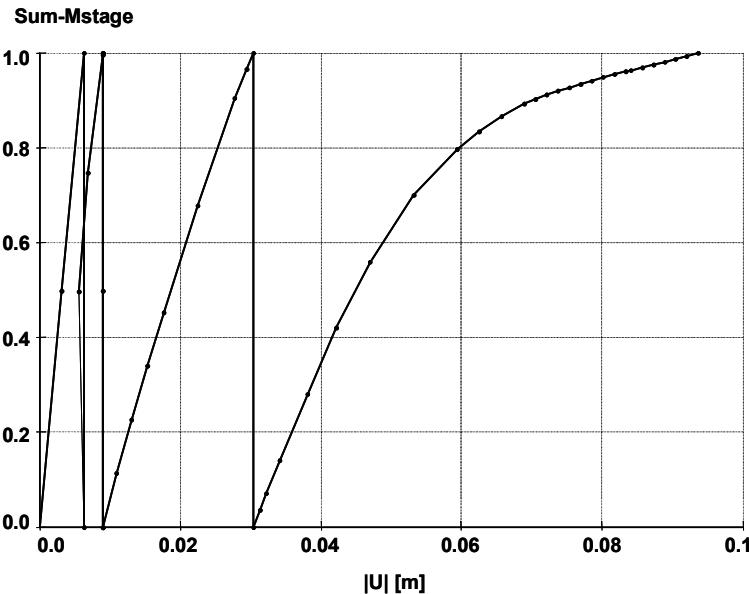


Figure 4.7 Load-displacement curve of deflection of wall

The curve shows the construction stages. For each stage, the parameter  $\Sigma M_{stage}$  changes from 0.0 to 1.0. The decreasing slope of the curve of the last stage indicates that the amount of plastic deformation is increasing. The results of the calculation indicate, however, that the excavation remains stable at the end of construction.



## 5 UNDRAINED RIVER EMBANKMENT (LESSON 3)

River embankments may be subjected to varying water levels. The change in water level and the resulting change in the pore pressure distribution influences the stability of the embankment. PLAXIS may be used to analyse the influence of pore pressure changes on the deformation and stability of geotechnical structures. This feature is used here to study the behaviour of the river embankment during the increase of the water level, as shown in Figure 5.1. A special problem related to such a situation is the possible uplift of the lowlands behind the embankment. This is due to the fact that the light soft soil layers cannot sustain the high pore pressures that arise in the permeable sand layer below. This effect may reduce the stability of the embankment.

The embankment in Figure 5.1 is 5 m high and consists of relative impervious clay. The upper 6 m of the subsoil consists of soft soil layers, of which the top 3 m is modelled as a clay layer and the lower 3 m as a peat layer. The soft soil layers are nearly impermeable, so a short-term variation in the river water level hardly influences the pore pressure distribution in these layers. Below the soft soil layers there is a deep permeable sand layer, of which the upper 4 m are included in the finite element model. It is assumed that the water in the sand layer is in contact with the river, which means that the hydraulic head in the sand layer follows the river water level variation closely.

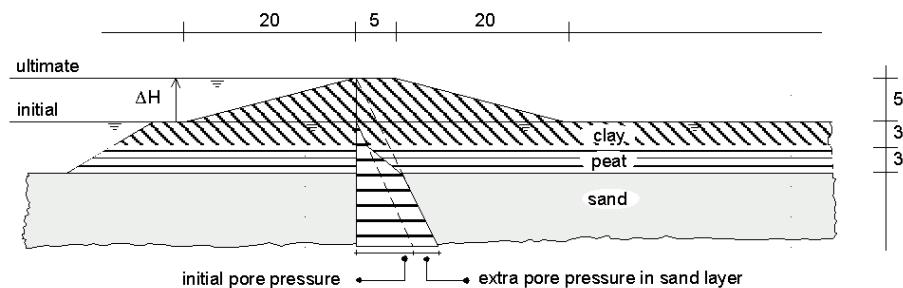


Figure 5.1 Geometry of the river embankment subjected to a changing water level

### 5.1 GEOMETRY MODEL

The geometry of Figure 5.1 is modelled with a plane strain geometry model. The finite element mesh is based on the 15-node elements. The units used in this example are meters for length, kiloNewton for force and day for time. The dimensions of the geometry are 65 m in horizontal direction and 15 m in vertical direction. The full geometry can be created using the *Geometry line* option. The *Standard fixities* option is

used to define the boundary conditions. The suggested geometry model is shown in Figure 5.2.

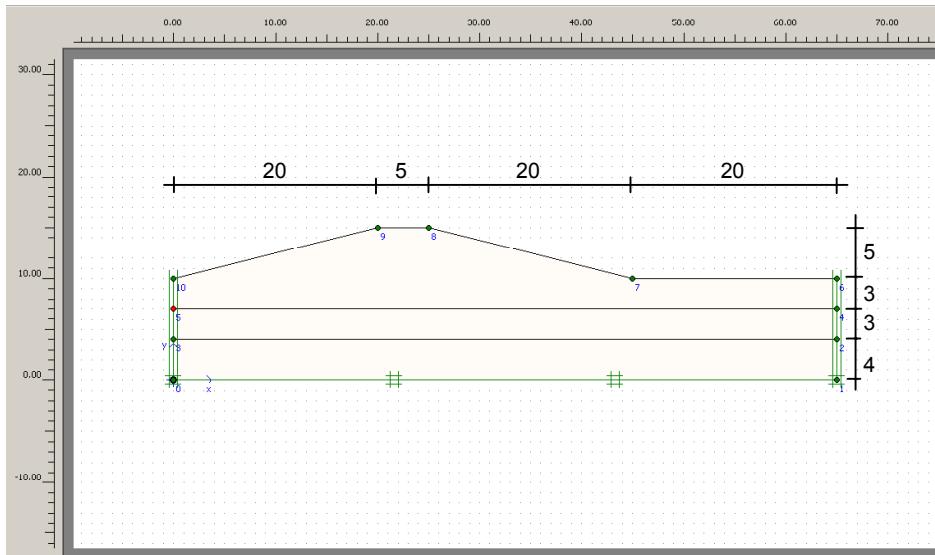


Figure 5.2 Geometry model of the river embankment project

### Material sets

Three material layers are adopted for the soil. The properties are given in Table 5.1.

Table 5.1. Material properties of the river embankment and subsoil

Parameter	Name	Clay	Peat	Sand	Unit
Material model	<i>Model</i>	MC	MC	MC	-
Type of behaviour	<i>Type</i>	Undr.	Undr.	Drained	-
Soil unit weight above p.l.	$\gamma_{unsat}$	16	8	17	kN/m <sup>3</sup>
Soil unit weight below p.l.	$\gamma_{sat}$	18	11.5	20	kN/m <sup>3</sup>
Horizontal permeability	$k_x$	0.001	0.01	1.0	m/day
Vertical permeability	$k_y$	0.001	0.001	1.0	m/day
Young's modulus	$E_{ref}$	2000	500	20000	kN/m <sup>2</sup>
Poisson's ratio	$\nu$	0.35	0.35	0.3	-
Cohesion	$c_{ref}$	2.0	7.0	1.0	kN/m <sup>2</sup>
Friction angle	$\varphi$	24	20	35	°
Dilatancy angle	$\psi$	0.0	0.0	0.0	°

Open the material data base and create three data sets named 'Clay', 'Peat' and 'Sand' with the model parameters as listed above. The interface properties are not relevant in this example. Note that the *Material type* of the clay and peat layers is *Undrained*, whereas the sand layer is *Drained*. Drag the data sets to the respective layers in the geometry model (see Figure 5.1).

### **Mesh generation**

This example, in which an uplift situation is modelled, is sensitive to the degree of refinement of the mesh. Therefore the *Global coarseness* is set to *Medium* in the *Mesh* menu. In addition, larger displacement gradients may be expected at the right hand embankment toe. In order to model that part of the geometry more accurately, select the geometry point of the embankment toe and select *Refine around point* from the *Mesh* menu. As a result, the element size around the embankment toe is modified to half the average element size. The generated mesh is shown in Figure 5.3.

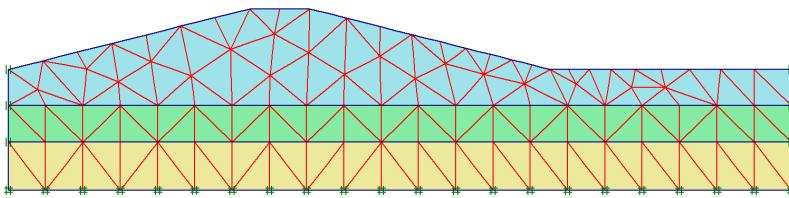


Figure 5.3 Finite element mesh of river embankment project

### **Initial conditions**

The geometry contains a **non-horizontal** soil surface. Therefore the *K<sub>0</sub>-procedure* cannot be used to calculate the initial stress field. Instead the initial stresses must be calculated by means of 'Gravity loading'. This is a calculation option that will be explained in Section 5.2. The activation of water pressures is always done together with the soil weight, but the generation of water pressures may be done in advance. In order to generate the proper initial water pressures, follow these steps:

- Click on the <Initial conditions> button.
- Accept the default value of the water weight (10 kN/m<sup>3</sup>).
- Enter a general phreatic level from point (0.0; 10.0) to point (65.0; 10.0).
- Generate the pore pressures from the phreatic level by clicking on the *Generate water pressures* button and subsequently clicking the <OK> button.
- In the Output window, check the pore pressure distribution and click on the <Update> button.

- Back in the Input window, click directly on the <Calculate> button. **Do not** generate the initial stresses according to the  $K_0$ -procedure.
- Save the input under an appropriate name.

**Hint:** If initial stresses were generated by mistake, they can be reset by entering the  $K_0$ -procedure, entering a value of  $\Sigma M weight = 0$  and pressing the <Generate> button.

## 5.2 CALCULATIONS

The calculation consists of two phases. First the initial stress field has to be calculated since this has not been done during the input of the initial conditions. The calculation of the initial stresses can be done in a plastic calculation where the multiplier for the soil weight is increased from 0.0 to 1.0. A calculation of this sort is called *Gravity loading*. This procedure is recommended when the soil surface, the layering or the phreatic level is non-horizontal. *Gravity loading* always results in an equilibrium stress state, whereas the  $K_0$ -procedure does not in the case of a non-horizontally layered subsoil. During *Gravity loading* both the soil weight and the pore pressures (that were generated previously) are activated.

**Hint:** Since the initial stresses are not subject to undrained behaviour, it is important that undrained behaviour is disabled during gravity loading. This can be done by selecting *Ignore undrained behaviour* in the *Parameters* tab sheet of the *Calculations* window.

> In contrast to the  $K_0$ -procedure, the calculation of initial stresses by means of gravity loading results in displacements. These displacements are not realistic, because the embankment is modelled as it appears in reality and the calculation of the initial stresses should not influence the displacements computed later in the analysis. These unrealistic displacements can be reset to zero at the start of the next calculation phase by selecting *Reset displacements to zero* in the next phase.

The second calculation phase is the increase of the river water level, and the pore pressure, in the sand layer. This is done in the *Staged construction* mode. In order to define the two calculation phases correctly, follow this procedure:

### **Phase 1: Gravity loading**

- For the first calculation phase, accept all default values of the *General* tab sheet and advance to the next tab sheet.

- In the *Parameters* tab sheet, select *Ignore undrained behaviour* in the *Control parameters* box. Select *Total multipliers* in the *Loading input* box and click on the <Define> button.
- In the *Multipliers* tab sheet, enter a value of 1.0 for  $\Sigma Mweight$  (the multiplier for the soil weight).

### **Phase 2: Raising the water level**

- Click on the <Next> button to create the next calculation phase. In this phase the 'Ultimate' situation as indicated in Figure 5.1 will be defined.
- In the *General* tab sheet, accept all default values. The default setting is such that the current phase starts from the results obtained from the previous phase.
- In the *Parameters* tab sheet, select *Reset displacements to zero* in the *Control parameters* box. This will eliminate the non-physical displacements resulting from the first calculation phase. This operation, however, does not affect the stresses.
- Select *Staged construction* in the *Loading input* box and click on the <Define> button.
- In the *Geometry configuration* window, click on the left button of the 'Switch' to arrive in the water pressure mode.
- Enter a general phreatic level through the points (0.0; 15.0), (20.0; 15.0), (45.0; 10.0). This general phreatic level is only meant to generate the external water pressures on the left side of the embankment (see also Figure 5.4). Individual water conditions will now be assigned to the different layers.

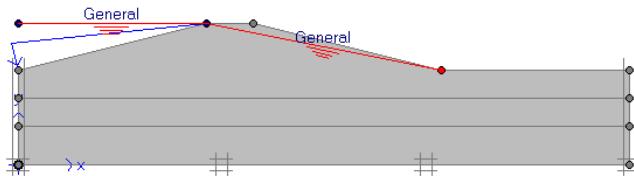


Figure 5.4 General phreatic level for generation of external water pressures

- Click on the *Selection* button and select the cluster of the clay layer (including the embankment).
- While the clay layer cluster is marked, click on the *Phreatic level* button and draw a phreatic level through the points (0.0; 10.0), (65.0; 10.0). This 'Cluster defined' phreatic level only applies to the indicated cluster (see Figure 5.5).

Figure 5.5 has been edited to only shows the phreatic level for the clay layer. The general phreatic level is not indicated.

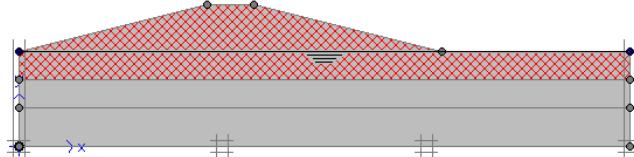


Figure 5.5 Phreatic level for clay layer

- Click on the *Selection* button and select the cluster of the sand layer.
- While the sand layer is marked, click on the *Phreatic level* button and draw a phreatic level through the points (0.0; 15.0), (65.0; **13.0**). This phreatic level only applies to the sand layer cluster. Figure 5.6 only shows the phreatic level for the sand layer; other phreatic levels are not indicated.

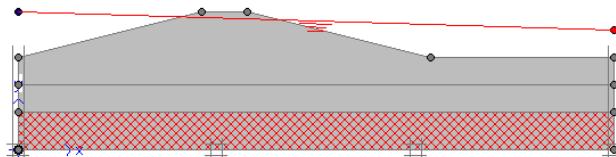


Figure 5.6 Phreatic level for sand layer

- Click on the *Selection* button and double-click, or click using the right mouse button, the intermediate peat layer. As a result, a *Cluster Pore pressures distribution* window appears. In the *Pore pressure distribution* box, there are five radio buttons. By default the *General phreatic level* is selected. The other options are *Cluster phreatic level*, *Interpolate from adjacent clusters or lines*, *Cluster dry* and *User defined pore pressure distribution*. The *Cluster phreatic level* option is automatically selected if a separate phreatic level is entered, as described above. For the current cluster (the peat layer) you should select the option *Interpolate from adjacent clusters or lines* (see also Figure 5.7). This will result in a linear distribution from the pressure at the bottom of the upper clay layer to the pressure at the top of the sand layer. Click the <OK> button to close the window.

**Hint:** The phreatic level corresponding to a particular cluster is indicated in red as soon as the cluster is selected. Clicking outside the geometry results in an indication of the general phreatic level. If a cluster is selected where the *Interpolate...* option applies, no phreatic level is indicated.

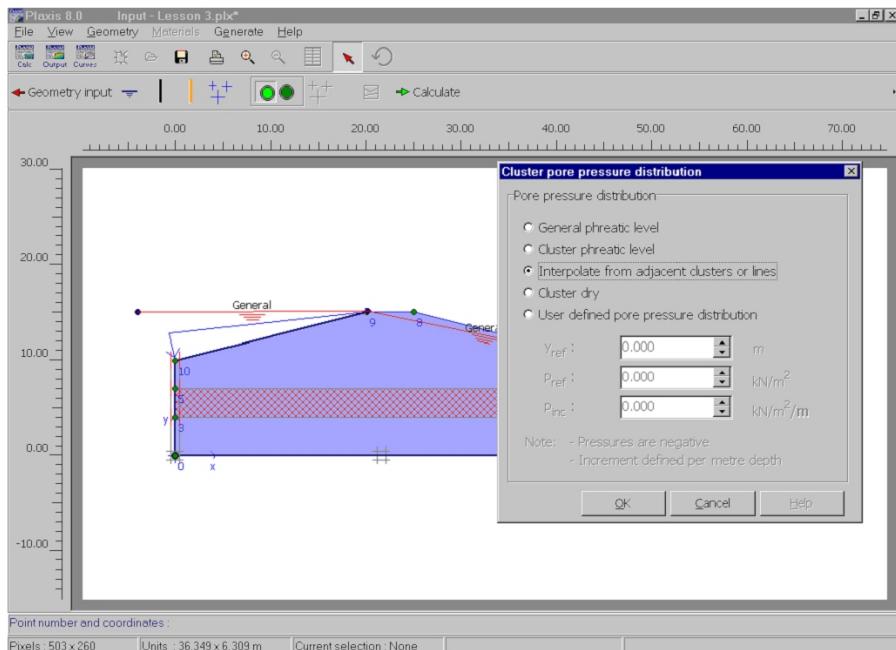


Figure 5.7 Definition of pore pressures for peat layer

- Click on the *Generate water pressures* button to generate the water pressures according to the ultimate situation.
- The pore pressure distribution is presented as principal stresses (by means of crosses) in the Output window. Click on the *Cross section* button and draw a vertical line through the top of the embankment to the bottom of the geometry. As a result, the pore pressure distribution over all three layers is displayed in a separate window. In addition to the hydrostatic part of the pore pressure distribution in the clay and sand layers, the plot shows the linear increase in pore pressure through the peat layer.

**Hint:** A cross section can be drawn perfectly horizontal or vertical by holding down the <Shift> key while drawing the cross section.

- Click on the <Update> button to return to the geometry configuration.
- In the geometry configuration, click on the <Update> window to return to the Calculations window.
- Click on the *Select points for curves* button. In the Output window, select suitable points for load-displacement curves (for example the toe and crest points of the embankment) and click on the <Update> button.
- In the Calculations window, click on the <Calculate> button to start the calculations.

### 5.3 OUTPUT

After the calculation has finished, click on the <Output> button to view the results of the second calculation phase. The Output program will now display the deformations of the embankment due to the change of the water level. The plot clearly shows the uplift of the soft soil layers behind the embankment and the movement of the embankment itself. This becomes even clearer if you select *Total increments* from the *Deformations* menu (see Figure 5.8).

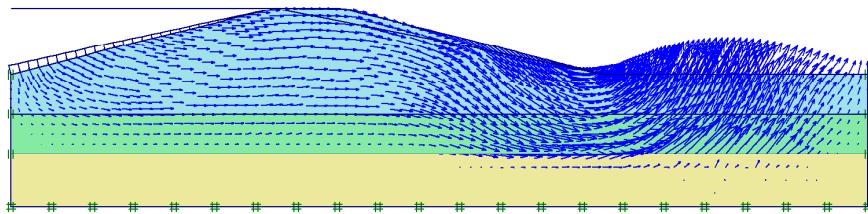


Figure 5.8 Displacement increments due to the change in water level

On selecting *Effective stresses* from the *Stresses* menu, it can be seen that at the top of the sand layer at the right hand side of the model the effective stresses are nearly zero (see Figure 5.9). This is due to the increase in pore pressures in the sand layer. From the stress plot it can also be seen that the movement of the embankment causes a passive stress state in the clay layer behind the embankment.

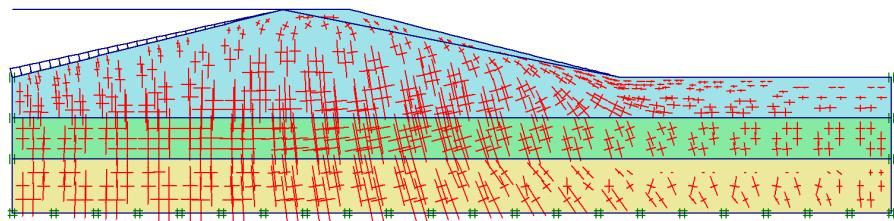


Figure 5.9 Effective stresses in embankment after the increase of the water level

The undrained behaviour in the clay and peat layers causes excess pore pressures to develop. The excess pore pressures can be viewed by selecting *Excess pore pressures* from the *Stresses* menu (see Figure 5.10).

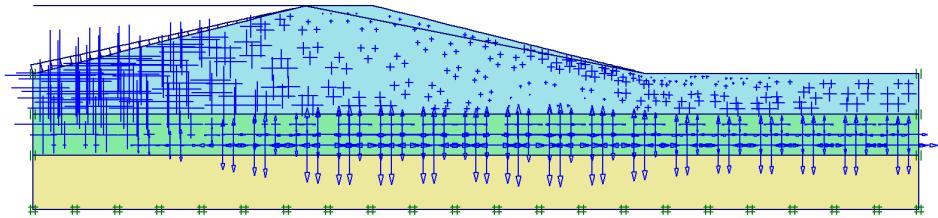


Figure 5.10 Excess pore pressures after the increase of the water level



## 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  $45^\circ$ . On the left side of the excavation a surface load of  $10 \text{ kN/m}^2$  is taken into account and on the right side a surface load of  $5 \text{ kN/m}^2$ .

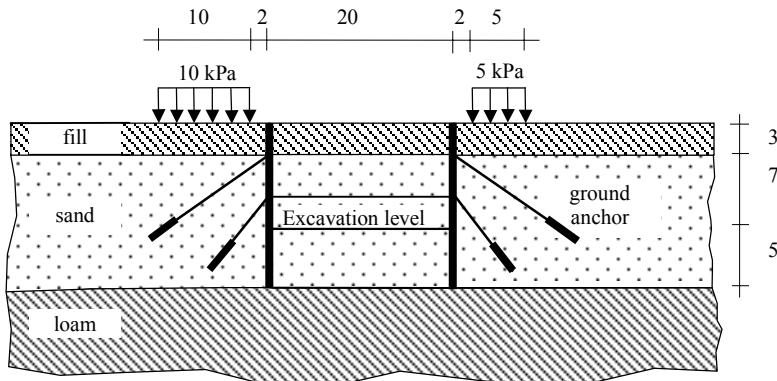


Figure 6.1 Excavation supported by tie back walls

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 a 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 problem can be modelled with a geometry model of 80 m width and 20 m height. The proposed geometry model is given in Figure 6.2. A ground anchor can be modelled by a combination of a node-to-node anchor and a geogrid (yellow line). The geogrid 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 pullout force of the ground anchor.

The diaphragm walls are modelled as plates. The interfaces around the plates 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 geogrid that represent the grout body that.

**Hint:** In general, it is a good habit to extend interfaces around corners of structures 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 does not influence the flow field, if applicable.

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 Figure 6.2. The *standard fixities* can be used to generate the proper boundary conditions.

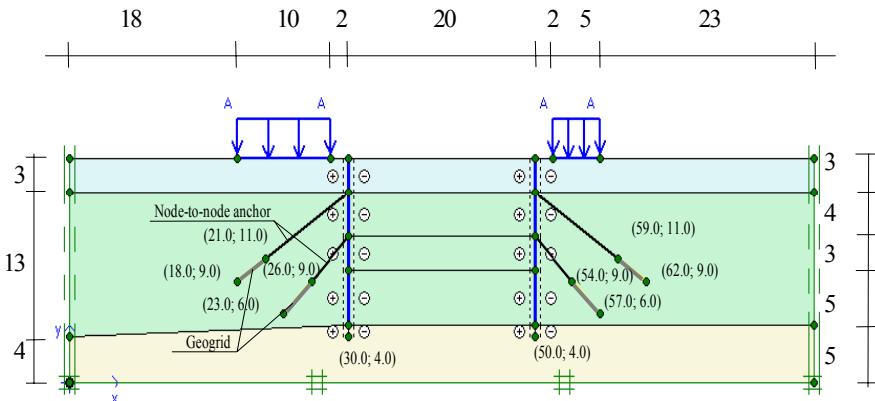


Figure 6.2 Geometry model of building pit

### Material properties

The soil consists of three distinct layers. Enter three data sets for soil & interfaces with the parameters given in Table 6.1.

**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. If necessary, a separate material data set must be created for the extended part of an interface. In addition, the extended part of an interface should not influence the flow field. This is achieved by deactivating the interface when generating the pore pressures.

The properties of the concrete diaphragm wall are entered in a material set of the *Plate* 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	-
Soil unit weight above p.l.	$\gamma_{unsat}$	16	17	17	kN/m <sup>3</sup>
Soil unit weight below p.l.	$\gamma_{sat}$	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	-

Table 6.2. Properties of the diaphragm wall (plate)

Parameter	Name	Value	Unit
Type of behaviour	<i>Material type</i>	Elastic	-
Normal stiffness	$EA$	$12 \cdot 10^6$	kN/m
Flexural rigidity	$EI$	$0.12 \cdot 10^6$	kNm <sup>2</sup> /m
Equivalent thickness	$d$	0.346	m

Parameter	Name	Value	Unit
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 *Geogrid* type. The *Anchor* data set contains the properties of the anchor rod and the *Geogrid* 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 anchor)

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

Table 6.4. Property of the grout body (geogrid)

Parameter	Name	Value	Unit
Normal stiffness	$EA$	$1 \cdot 10^5$	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 so a local refinement is proposed here. Select the four geogrids simultaneously (use the <Shift> key) and select *Refine line* from the *Mesh* menu. This process results in a mesh of approximately 590 elements.

### ***Initial conditions***

In the initial conditions, a water weight of 10 kN/m<sup>3</sup> is entered. The initial water pressures are generated on the basis of a horizontal general phreatic level at a level of y = 17 m (through points (0; 17.0) and (80.0; 17.0)).

Initially, all structural components are inactive. Hence, make sure that the plates, the node-to-node anchors and the geogrids are deactivated. The surface loads are also initially inactive. 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.

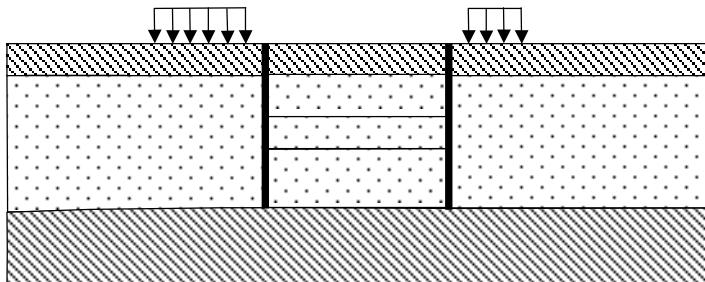
## 6.2 CALCULATIONS

The calculation consists of six phases. In the first phase the walls are constructed and the surface loads are activated. In the second phase the first 3 m of the pit is excavated without connection of anchors to the wall. At this depth the excavation remains dry. In the third phase the first anchor is installed and pre-stressed. The fourth 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 fifth phase the second anchor is installed and pre-stressed and the sixth phase is a further excavation (and de-watering) of to the final depth of 10 m.

All calculation phases are defined as *Plastic* calculations 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:**

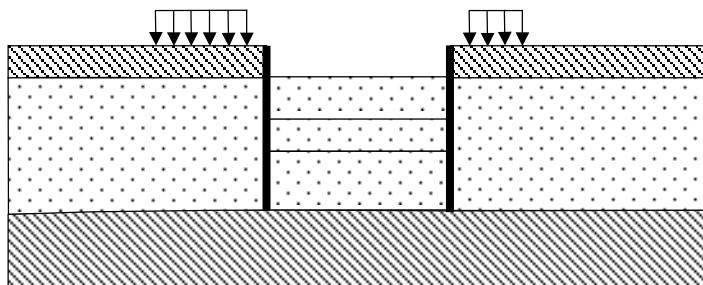
- Activate the walls.
- Activate the surface loads and assign a load value. Enter a *Y-value* = -10 kPa for the load on the left side and -5 kPa for the load on the right side of the excavation.



Phase 1

### **Phase 2:**

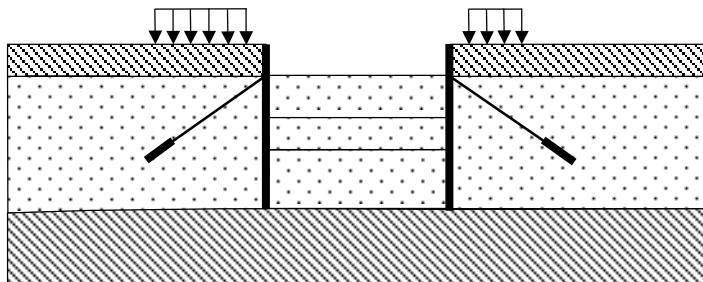
- De-activate the upper cluster of the excavation.



Phase 2

**Phase 3:**

- Activate the upper geogrids
- Double click the upper node-to-node anchors. 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.



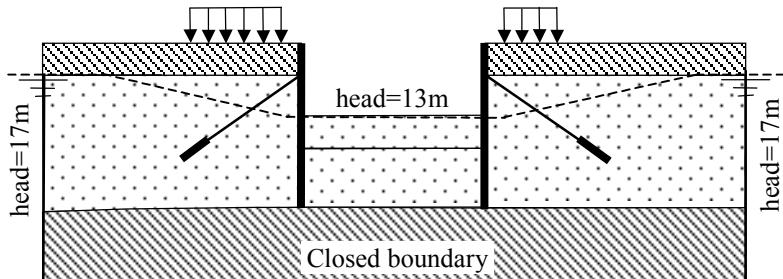
Phase 3

**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 4:**

- Deactivate the second cluster of the excavation.

Now the boundary conditions for the groundwater flow calculation have to be entered. At the side boundaries, the groundwater head remains at a level of 17.0 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 = 13.0 m). This condition can be met 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.



Phase 4

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

- Click on the 'switch' to go to the water pressures mode.
- 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
- Click on the *Selection* button.
- The interfaces on both sides of the wall should be activated by default in the water pressures mode, marking them as *impermeable*. Clicking on an interface in the water pressures mode activates or deactivates the interface during groundwater calculations. An active interface is marked with an orange circle and is considered *impermeable* during groundwater calculations. Do not switch the interfaces below the walls to *impermeable*, those should remain *permeable* (inactive during the groundwater flow calculation).
- Click on the *General phreatic level* button and draw a new phreatic level. Start in (0.0; 17.0) and draw the phreatic level through (30.0; 13.0), (50.0; 13.0) and end in (80.0; 17.0).
- Click on the *Generate water pressures* button. Select *Groundwater calculation* 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.

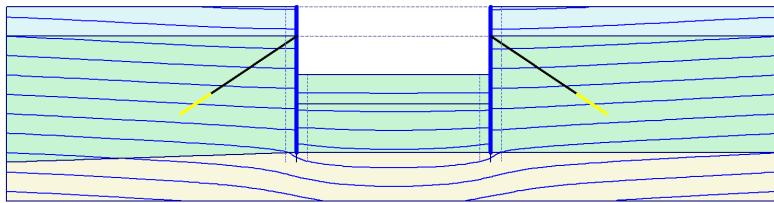
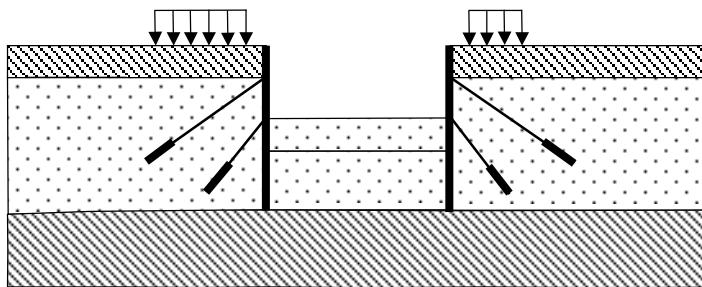


Figure 6.3 Active pore pressure contours resulting from groundwater calculation

- 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 5:

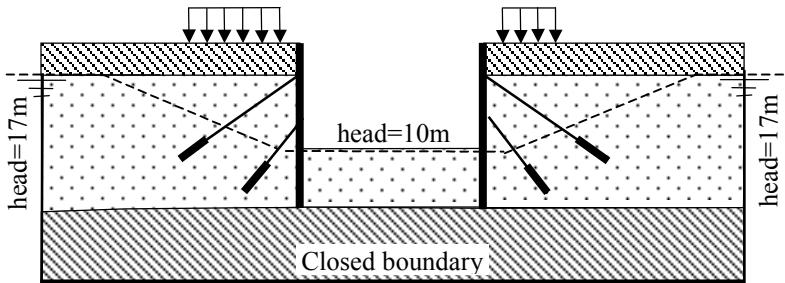
- Activate the lower geogrids
- Double click the lower node-to-node anchors. In the Anchor window, select the *Adjust pre-stress force* box and enter a pre-stress force of 200 kN/m. Press <OK> to close the window.



Phase 5

### **Phase 6:**

- 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, draw a new *General phreatic level* from (0.0; 17.0) through points (30.0; 10.0), (50.0; 10.0) and (80.0; 17.0). Click on the *Generate water pressures* button and 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.

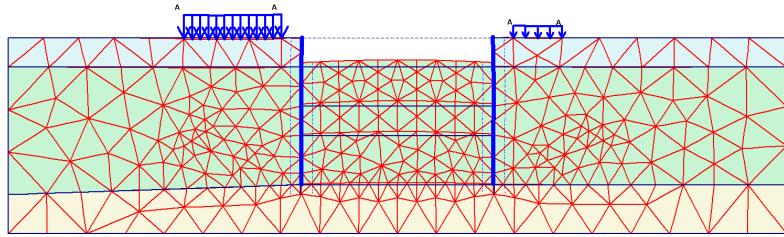


Phase 6

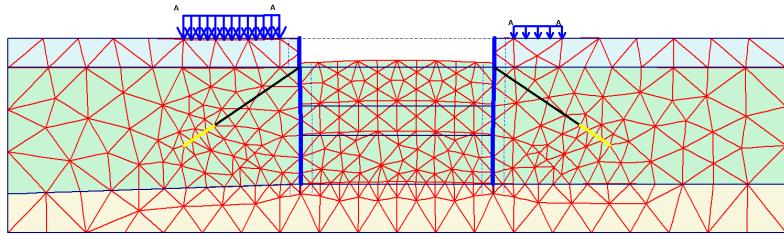
After all calculation phases have been defined, some points for load-displacement curves should be selected (for example the connection points of the ground anchors on the diaphragm wall). Start the calculation by clicking on the <Calculate> button.

### **6.3 OUTPUT**

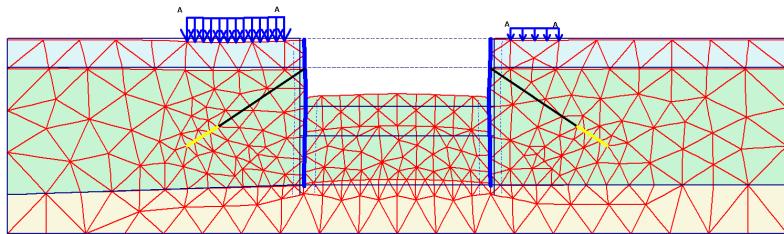
Figure 6.4 (a) to (e) show the deformed meshes at the end of calculation phases 2 to 6. In the final situation, the walls have moved about 8 cm forward. Behind the wall there is a small settlement trough.



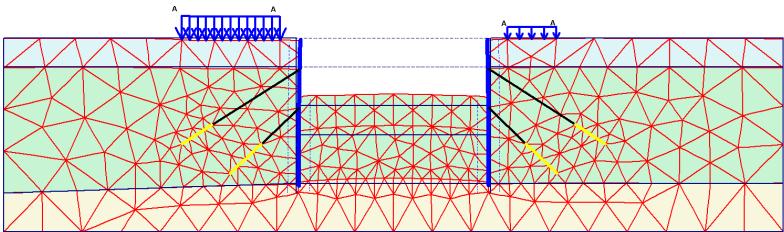
(a) phase 2



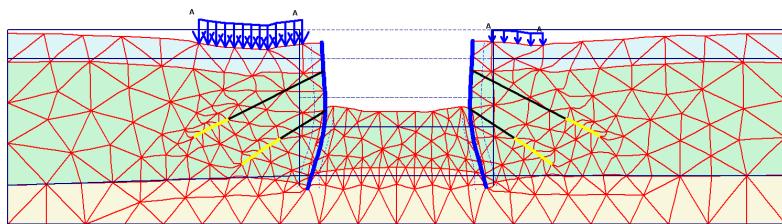
(b) phase 3



(c) phase 4



(d) phase 5



(e) final stage

Figure 6.4 Deformed mesh stages (a) to (e)

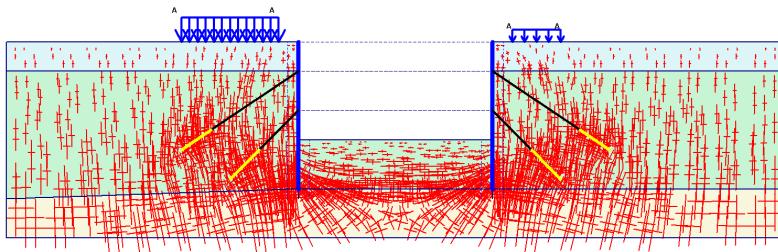


Figure 6.5 Effective stresses, final stage

Figure 6.5 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.

Figure 6.6 shows the bending moments in the left 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 third and the fifth calculation phase, it can be checked that the anchor force is indeed equal to the specified pre-stress force.

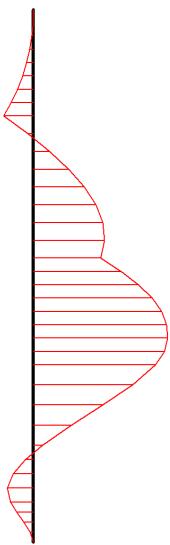


Figure 6.6 Bending moments in the left diaphragm wall in the final stage

## 7 CONSTRUCTION OF A ROAD EMBANKMENT (LESSON 5)

The construction of an embankment on soft soil with a high groundwater level leads to an increase in pore pressure. As a result of this 'undrained behaviour' the effective stress remains low and intermediate consolidation periods have to be adopted in order to construct the embankment safely. During consolidation the excess pore pressures dissipate so that the soil can obtain the necessary shear strength to continue the construction process.

This lesson concerns the construction of a road embankment in which the mechanism described above is analysed in detail. In the analysis three new calculation options are introduced, namely a consolidation analysis, an updated mesh analysis and the calculation of a safety factor by means of phi-c-reduction.

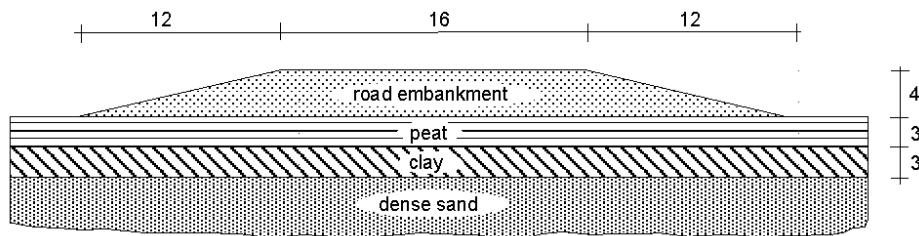


Figure 7.1 Situation of a road embankment on soft soil

### 7.1 INPUT

Figure 7.1 shows a cross section of a road embankment. The embankment is 16.0 m wide and 4.0 m high. The slopes have a slope of 1:3. The problem is symmetric, so only one half is modelled (in this case the right half is chosen). The embankment itself is composed of loose sandy soil. The subsoil consists of 6.0 m of soft soil. The upper 3.0 m of this soft soil layer is modelled as a peat layer and the lower 3.0 m as clay. The phreatic level coincides with the original ground surface. Under the soft soil layers there is a dense sand layer, which is not included in the model.

#### ***Geometry model***

The embankment shown in Figure 7.1 can be analysed with a plane strain model. For this example 15-node elements are utilised. The standard units for *Length*, *Force* and *Time* are used (m, kN and day). A total width of 40 m is considered in the geometry model, starting from the embankment centre. The full geometry can be drawn using the *Geometry line* option. The deformations of the deep sand layer in Figure 7.1 are assumed to be zero. Hence, this layer is not included in the model and a fixed base is used instead.

The *Standard fixities* can be used to define the boundary conditions. The geometry model is shown in Figure 7.2.

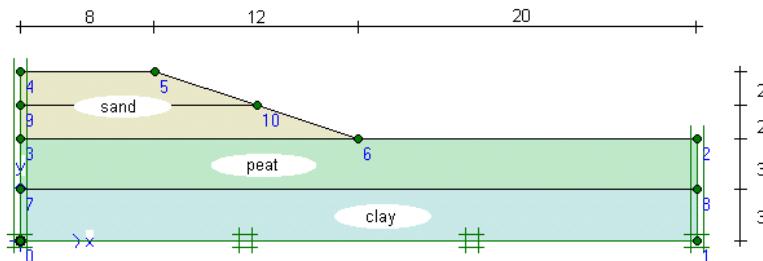


Figure 7.2 Geometry model of road embankment project

Table 7.1. Material properties of the road embankment and subsoil

Parameter	Name	Clay	Peat	Sand	Unit
Material model	<i>Model</i>	MC	MC	MC	-
Type of behaviour	<i>Type</i>	Undrained	Undrained	Drained	-
Soil unit weight above phreatic level	$\gamma_{unsat}$	15	8	16	kN/m <sup>3</sup>
Soil unit weight below phreatic level	$\gamma_{sat}$	18	11	20	kN/m <sup>3</sup>
Horizontal permeability	$k_x$	$1 \cdot 10^{-4}$	$2 \cdot 10^{-3}$	1.0	m/day
Vertical permeability	$k_y$	$1 \cdot 10^{-4}$	$1 \cdot 10^{-3}$	1.0	m/day
Young's modulus	$E_{ref}$	1000	350	3000	kN/m <sup>2</sup>
Poisson's ratio	$\nu$	0.33	0.35	0.3	-
Cohesion	$c_{ref}$	2.0	5.0	1.0	kN/m <sup>2</sup>
Friction angle	$\varphi$	24	20	30	°
Dilatancy angle	$\psi$	0.0	0.0	0.0	°

### Material sets and mesh generation

The properties of the different soil types are given in Table 7.1. Three material sets are to be created, containing the data according to the table. The clay and the peat layer are undrained. This type of behaviour leads to an increase of pore pressures during the construction of the embankment. Assign the data to the corresponding clusters in the geometry model. After the input of material parameters, a simple finite element mesh may be generated using the *Medium coarseness* setting. Generate the mesh by clicking on the *Generate mesh* button.

### **Initial conditions**

In the *Initial conditions* the water weight is set to 10 kN/m<sup>3</sup>. The water pressures are fully hydrostatic and based on a general phreatic level through the points (0.0; 6.0) and (40.0; 6.0).

In addition to the phreatic level, attention must be paid to the boundary conditions for the consolidation analysis that will be performed during the calculation process. Without giving any additional input, all boundaries are draining so that water can freely flow out of all boundaries and excess pore pressures can dissipate in all directions. In the current situation, however, the left vertical boundary must be closed because this is a line of symmetry, so horizontal flow should not occur. The right vertical boundary should also be closed because there is no free outflow at that boundary. The bottom is open because below the soft soil layers the excess pore pressures can freely flow into the deep and permeable sand layer (which is not included in the model). The upper boundary is obviously open as well. In order to create the appropriate consolidation boundary conditions, follow these steps:

-  Click on the *Closed consolidation boundary* button (yellow line) in the toolbar.
- Move to the upper point of the left boundary (0.0; 10.0) and click on this point. Move to the lower point of the left boundary (0.0; 0.0) and click again. Click the right mouse button to finish this closed boundary.
- Move to the upper point of the right boundary (40.0; 6.0) and click. Move to the lower point (40.0; 0.0) and click again. Finish this closed boundary.
- Click on the *Generate water pressures* button to generate the water pressures and the consolidation boundary conditions.

After the generation of the water pressures, click on the 'switch' to modify the initial geometry configuration. In the initial situation the embankment is not present. In order to generate the initial stresses therefore, the embankment must be deactivated first.

<b>Hint:</b>	Closed consolidation boundaries can only be defined by clicking on existing geometry points. The program will automatically find intermediate geometry points.
>	Consolidation boundary conditions must be generated in the boundary nodes of the mesh. This is done together with the generation of water pressures. Hence, after introducing or changing consolidation boundaries, always click on the <i>Generate water pressures</i> button.

Click once in the two clusters that represent the embankment, just like in a staged construction calculation. When the embankment has been deactivated (the corresponding clusters should have the background colour), the remaining active geometry is horizontal with horizontal layers, so the *K<sub>0</sub>-procedure* can be used to calculate the initial stresses. The suggested *K<sub>0</sub>*-values of the clay and peat layer (based

on Jaky's formula:  $K_0 = 1 - \sin\phi$ ) can be accepted. After the generation of the initial stresses the input is complete and the calculations can be defined.

## 7.2 CALCULATIONS

The embankment construction consists of two phases, each taking 5 days. After the first construction phase a consolidation period of 200 days is introduced to allow the excess pore pressures to dissipate. After the second construction phase another consolidation period is introduced from which the final settlements may be determined. Hence, a total of four calculation phases have to be defined.

A consolidation analysis introduces the dimension of time in the calculations. In order to correctly perform a consolidation analysis a proper time step must be selected. The use of time steps that are smaller than a critical minimum value can result in stress oscillations. The consolidation option in PLAXIS allows for a fully automatic time stepping procedure that takes this critical time step into account. Within the automatic time stepping procedure there are three main possibilities: Either consolidate for a predefined period, including the effects of changes to the active geometry (*Staged construction*), consolidate until all excess pore pressures in the geometry have reduced to a predefined minimum value (*Minimum pore pressure*) or consolidate for a given number of steps, using incremental multipliers to globally increase load systems in time or to apply rate loading (*Incremental multiplier*). The first two possibilities will be used in this exercise.

To define the calculation phases, follow these steps:

- The first calculation stage is a *Consolidation* analysis, *Staged construction*. In the *General* tab sheet select *Consolidation* from the *Calculation type* combo box. In the *Parameters* tab sheet, enter a *Time interval* of 5 days. Select *Staged construction* for the *Loading input* and click on the <Define> button. Activate the first part of the embankment in the Geometry configuration window and click on the <Update> button.

Back in the Calculation window, click on the <Next> button to introduce the next calculation phase.

- The second phase is also a *Consolidation* analysis, *Staged construction*. This time no changes to the geometry are made as only a consolidation analysis to ultimate time is required. Enter a time interval of 200 days and click on the <Next> button to introduce the next calculation phase.
- The third phase is once again a *Consolidation* analysis, *Staged construction*. After selecting *Staged construction* in the *Parameters* tab sheet enter a *Time interval* of 5 days. Click on the <Define> button and activate the second part of the embankment. Click <Update> and enter the next phase.

- The fourth phase is a consolidation analysis to a minimum pore pressure. In the *Parameters* tab sheet, select *Minimum pore pressure* from the *Loading input* box and accept the default value of  $1 \text{ kN/m}^2$  for the minimum pressure.

Before starting the calculation, click on the *Select points for curves* button and select the following points: As Point A, select the toe of the embankment. The second point (Point B) will be used to plot the development (and decay) of excess pore pressures. To this end, a point somewhere in the middle of the soft soil layers is needed, close to (but not actually on) the left boundary. After selecting these points, start the calculation.

During a consolidation analysis the development of time can be viewed in the upper part of the calculation info window. In addition to the multipliers, a parameter  $PP_{max}$  occurs, which indicates the current maximum excess pore pressure. This parameter is of interest in the case of a *Minimum pore pressure* consolidation analysis, where all pore pressures are specified to reduce below a predefined value.

### 7.3 OUTPUT

After the calculation has finished, select the third and the fourth phase simultaneously (hold the *<Ctrl>* key on the keyboard while selecting these phases) and click on the *<Output>* button. The Output window now shows the two deformed meshes, one after the undrained construction of the final part of the embankment and one after full consolidation.

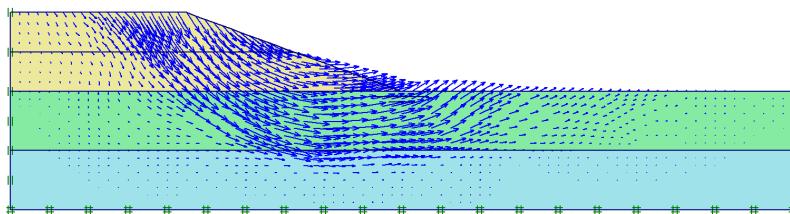


Figure 7.3 Displacement increments after undrained construction of embankment

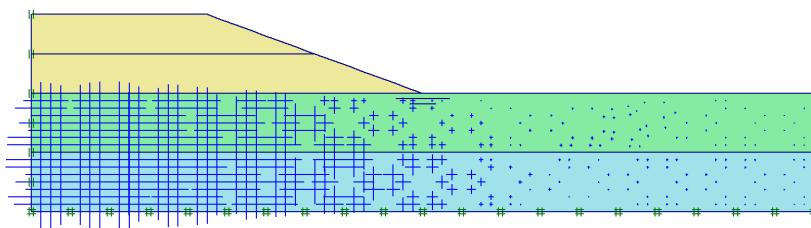


Figure 7.4 Excess pore pressures after undrained construction of embankment

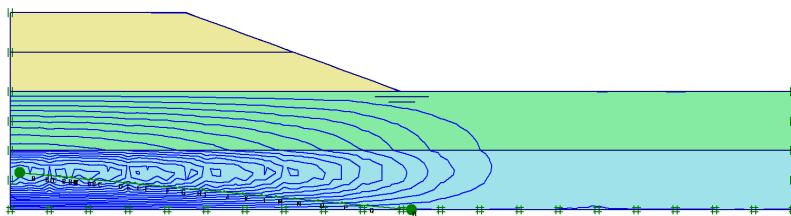


Figure 7.5 Excess pore pressure contours after consolidation to  $P_{excess} < 1.0 \text{ kN/m}^2$

Considering the results of the third phase (undrained construction), the deformed mesh shows the uplift of the embankment toe and hinterland due to the undrained behaviour. On evaluating the total displacement increments, it can be seen that a failure mechanism is developing (see Figure 7.3). In addition, Figure 7.4 shows the excess pore pressures distribution. It is clear that the highest excess pore pressure occurs under the embankment centre.

It can be seen that the settlement of the original soil surface and the embankment increases considerably during the fourth phase. This is due to the dissipation of the excess pore pressures, which causes consolidation of the soil. Figure 7.5 shows the remaining excess pore pressure distribution after consolidation. Check that the maximum value is below  $1.0 \text{ kN/m}^2$ .

The *Curves* program can be used to view the development, with time, of the excess pore pressure under the embankment. In order to create such a curve, follow these steps:

- Click on the *Go to curves program* button in the upper left corner of the Output window.
- Select *New chart* and select the current project from the file requester.
- In the *Curve generation* window, select *Time* for the x-axis. For the y-axis, select *Pore pressure – Excess pore pressure* and select the point in the middle of the soft soil layers (Point B) from the *Point* combo box. After clicking on the <OK> button, a curve similar to Figure 7.6 should appear.

Figure 7.6 clearly shows the four calculation phases. During the undrained construction phases the excess pore pressure increases with a small increase in time while during the consolidation periods the excess pore pressure decreases with time. In fact, consolidation already occurs during construction of the embankment, as this involves a small time interval. From the curve it can be seen that more than 700 days are needed to reach full consolidation.

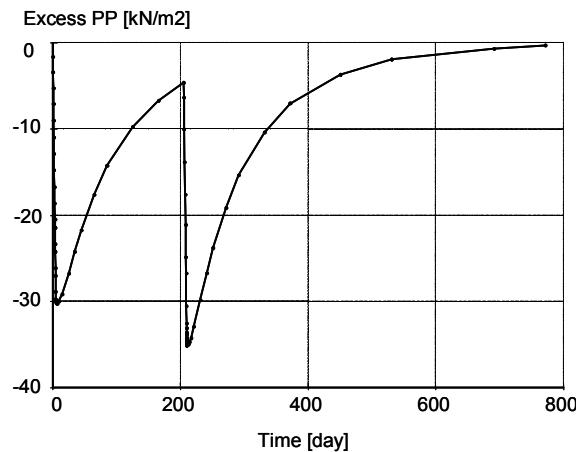


Figure 7.6 Development of excess pore pressure under the embankment

#### 7.4 SAFETY ANALYSIS

In the design of an embankment it is important to consider not only the final stability, but also the stability during the construction. It is clear from the output results that a failure mechanism starts to develop after the second construction phase.

It is interesting to evaluate a global safety factor at this stage of the problem, and also for other stages of construction.

In structural engineering, the safety factor is usually defined as the ratio of the collapse load to the working load. For soil structures, however, this definition is not always useful. For embankments, for example, most of the loading is caused by soil weight and an increase in soil weight would not necessarily lead to collapse. Indeed, a slope of purely frictional soil will not fail in a test in which the self weight of the soil is increased (like in a centrifuge test). A more appropriate definition of the factor of safety is therefore:

$$\text{Safety factor} = \frac{S_{\text{maximum available}}}{S_{\text{needed for equilibrium}}}$$

Where  $S$  represents the shear strength. The ratio of the true strength to the computed minimum strength required for equilibrium is the safety factor that is conventionally used in soil mechanics. By introducing the standard coulomb condition, the safety factor is obtained:

$$\text{Safety factor} = \frac{c - \sigma_n \tan \varphi}{c_r - \sigma_n \tan \varphi_r}$$

Where  $c$  and  $\varphi$  are the input strength parameters and  $\sigma_n$  is the actual normal stress component. The parameters  $c_r$  and  $\varphi_r$  are reduced strength parameters that are just large enough to maintain equilibrium. The principle described above is the basis of the method of *Phi-c-reduction* that can be used in PLAXIS to calculate a global safety factor. In this approach the cohesion and the tangent of the friction angle are reduced in the same proportion:

$$\frac{c}{c_r} = \frac{\tan \varphi}{\tan \varphi_r} = \Sigma M_{sf}$$

The reduction of strength parameters is controlled by the total multiplier  $\Sigma M_{sf}$ . This parameter is increased in a step-by-step procedure until failure occurs. The safety factor is then defined as the value of  $\Sigma M_{sf}$  at failure, provided that at failure a more or less constant value is obtained for a number of successive load steps.

The *Phi-c-reduction* calculation option is available in PLAXIS from the *Calculation type* list box on the *General* tab sheet. If the *Phi-c-reduction* option is selected the *Loading input* on the *Parameters* tab sheet is automatically set to *Incremental multipliers*.

To calculate the global safety factor for the road embankment at different stages of construction, follow these steps:

- Click on the *Go to calculations program* button to focus the Calculations window.
- We first want to calculate the safety factor after the first construction stage. Therefore introduce a new calculation phase and select Phase 1 in the *Start from phase* list box.
- In the *General* tab sheet, select a *Phi-c-reduction* calculation.
- In the *Parameters* tab sheet the number of Additional steps is automatically set to 100 (instead of the default value of 250). In order to exclude existing deformations from the resulting failure mechanism, select the Reset displacements to zero option. The *Incremental multipliers* option is already selected in the *Loading input* box. Click on the <Define> button to enter the *Multipliers* tab sheet.
- In the *Multipliers* window, check that the first increment of the multiplier that controls the strength reduction process,  $M_{sf}$ , is set to 0.1. The first safety calculation has now been defined.

**Hint:** The default value of *Additional steps* in a *Phi-c-reduction* calculation is 100. In contrast to an *Ultimate level* calculation, the number of additional steps is always fully executed. In most *Phi-c-reduction* calculations, 100 steps are sufficient to arrive at a state of failure. If not, the number of additional steps can be increased to a maximum of 1000.

> For most phi-c-reduction calculations  $Msf = 0.1$  is an adequate first step to start up the process. During the calculation process, the development of the total multiplier for the strength reduction,  $\Sigma Msf$ , is automatically controlled by the load advancement procedure.

- We now want to define the calculation of the safety factor after the second construction stage. Therefore introduce a new calculation phase and select Phase 3 as the phase to start from . The can be done in the General tab sheet of the Calculation program by clicking on the combo box *start from phase* and choosing Phase 3.
- In the *General* tab sheet, select *Phi-c-reduction* from the *Loading type* combo box.
- In the *Parameters* tab sheet, select the *Reset displacements to zero* option, select *Incremental multipliers* and click on the <Define> button.
- In the *Multipliers* window, check that  $Msf$  is set to 0.1.
- Finally we want to know the final safety factor of the embankment. Therefore introduce one more calculation stage and let it start from the fourth calculation phase.
- In the *General* tab sheet, select *Phi-c-reduction* as the loading type.
- In the *Parameters* tab sheet, select the *Reset displacements to zero* option. In addition, select the *Ignore undrained behaviour* option, because in this case the long term behaviour is considered. Select *Incremental multipliers* and click on the <Define> button.
- In the *Multipliers* window, check that  $Msf$  is set to 0.1.

Before starting the calculations, make sure that only the new calculation phases are selected for execution ( $\rightarrow$ ); the others should be indicated with the  $\sqrt{}$ -sign.

### ***Evaluation of results***

Additional displacements are generated during a phi-c-reduction calculation. The total displacements do not have a physical meaning, but the incremental displacements in the final step (at failure) give an indication of the likely failure mechanism. In order to view the mechanisms in the three different stages of the embankment construction, select the phases 5, 6 and 7 simultaneously (use the <Ctrl> key) and click on the <Output> button. Select for all windows the *Total increments* from the *Deformations* menu and change

the presentation from *Arrows to Shadings*. The resulting plots give a good impression of the failure mechanisms (see Figure 7.7). The magnitude of the displacement increments is not relevant.

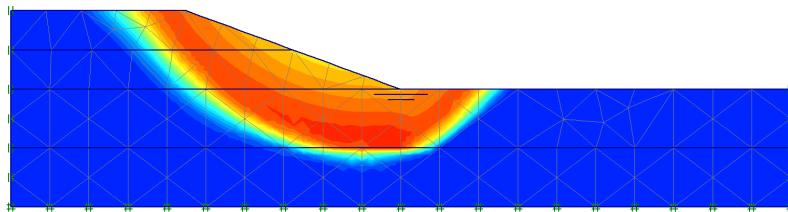


Figure 7.7 Shadings of the total displacement increments indicating the most applicable failure mechanism of the embankment in the final stage

The safety factor can be obtained from the *Calculation info* option of the *View* menu. The *Multipliers* tab sheet of the *Calculation information* window represents the actual values of the load multipliers. The value of  $\Sigma M_{sf}$  represents the safety factor, provided that this value is indeed more or less constant during the previous few steps.

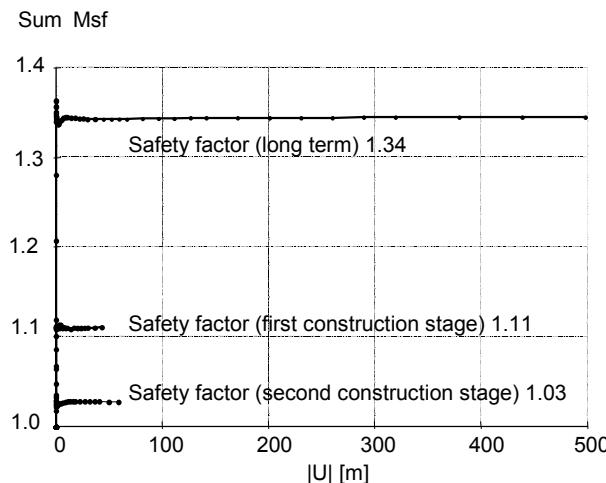


Figure 7.8 Evaluation of safety factor for three stages of the construction process

The best way to evaluate the safety factor, however, is to plot a curve in which the parameter  $\Sigma M_{sf}$  is plotted against the displacements of a certain node. Although the displacements are not relevant, they indicate whether or not a failure mechanism has

developed. In order to evaluate the safety factors for the three situations in this way, follow these steps:

- Click on the *Go to curves program* button to start the *Curves* program.
- Select a *New chart* and select the road embankment file from the file requester.
- In the *Curve generation* window, select the total displacement of the embankment toe (Point A) for the x-axis. For the y-axis, select *Multipliers* and select  $\Sigma M_{sf}$  from the *Type* combo box. As a result, the curve of Figure 7.8 appears.

The maximum displacements plotted are not relevant. It can be seen that for all curves a more or less constant value of  $\Sigma M_{sf}$  is obtained. Hovering the mouse cursor over a point on the curves, a box showing the exact value of  $\Sigma M_{sf}$  can be obtained.

## 7.5 UPDATED MESH ANALYSIS

As can be seen from the output of the *Deformed mesh* at the end of consolidation (stage 4), the embankment settles over half a metre within two years of the start of construction. Part of the sand fill that was originally above the phreatic level will settle below the phreatic level. As a result of buoyancy forces the effective weight of the soil that settles below the water level will change, which leads to a reduction of the effective overburden in time. This effect can be simulated in PLAXIS using the *Updated mesh* and *Updated water pressures* options. For the road embankment the effect of using these options will be investigated.

Open the current project in the *Input* program and select *Save as* from the *File* menu. Save the project under a different name. Now click on the *Go to Calculation program* button and open the new project. All phases will be marked for calculation. To change the calculation to an Updated mesh analysis, do the following:

- Select Phase 1 and click on the *Advanced* button below the *Calculation type* list box. Check the *Updated mesh* and *Updated water pressures* options in the *Advanced general settings* window. Click <OK> to return to the calculation window.
- Repeat this step for all calculation phases 2, 3 and 4.
- Delete phases 5, 6 and 7.
- Start the calculation.

When the calculation has finished, open the *Curves* program to compare the settlements for the two different calculation methods.

- Select a *New chart* and select the road embankment calculation using updated mesh from the file requester.

- In the *Curve generation* window select time for the x-axis and select the total displacement of the embankment toe (Point A) for the y-axis.

To compare those with the displacements from the calculation without the updated mesh option, add a curve from the previous calculation, without the updated mesh option.

- From the *File* menu, choose to *Add curve, from another project*.
- From the file requester, select the road embankment calculation without updated mesh.
- In the *Curve generation* window, select the same point (Point A), for which deformation versus time will be plotted. Click <OK> to add the curve.

The default graph generated also includes the displacements calculated during the *Phi-c-reduction* stages. These displacements are not of interest at the moment and can be removed from the curve.

- Select *Curve* from the *Format* menu and click on the *Phases* button.
- In the *Select phases* window, deselect the *Phi-c-reduction phases*, i.e. phases 5, 6 and 7. Click <OK> to return to the *Curve settings* window and click <OK> again to update the graph.

Now only the displacements during the construction and consolidation phases are plotted. To change the vertical scale of the graph:

- Select *Graph* from the *Format* menu and change the *Scaling* of the *Y-axis* to *Manual*. Enter a maximum value of 0.5. Click on <OK> to update the graph.

In Figure 7.9 it can be seen that the settlements are less when the *Updated mesh* and *Updated water pressures* options are used. This is partly because the *Updated mesh* procedure includes second order deformation effects by which changes of the geometry are taken into account, and partly because the *Updated water pressures* procedure results in smaller effective weights of the embankment. This last effect is caused by the buoyancy of the soil settling below the (constant) phreatic level. The use of these procedures allows for a realistic analysis of settlements, taking into account the positive effects of large deformations.

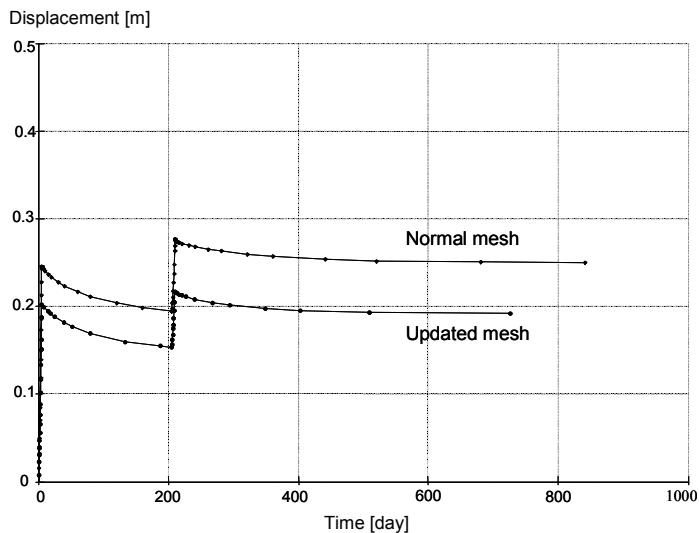


Figure 7.9 Settlements of the toe of the embankment using updated mesh calculation



## 8 SETTLEMENTS DUE TO TUNNEL CONSTRUCTION (LESSON 6)

PLAXIS has special facilities for the generation of circular and non-circular tunnels and the simulation of a tunnel construction process. In this chapter the construction of a shield tunnel in medium soft soil and the influence on a pile foundation is considered. A shield tunnel is constructed by excavating soil at the front of a tunnel boring machine (TBM) and installing a tunnel lining behind it. In this procedure the soil is generally over-excavated, which means that the cross sectional area occupied by the final tunnel lining is always less than the excavated soil area. Although measures are taken to fill up this gap, one cannot avoid stress re-distributions and deformations in the soil as a result of the tunnel construction process. To avoid damage to existing buildings or foundations on the soil above, it is necessary to predict these effects and to take proper measures. Such an analysis can be performed by means of the finite element method. This lesson shows an example of such an analysis.

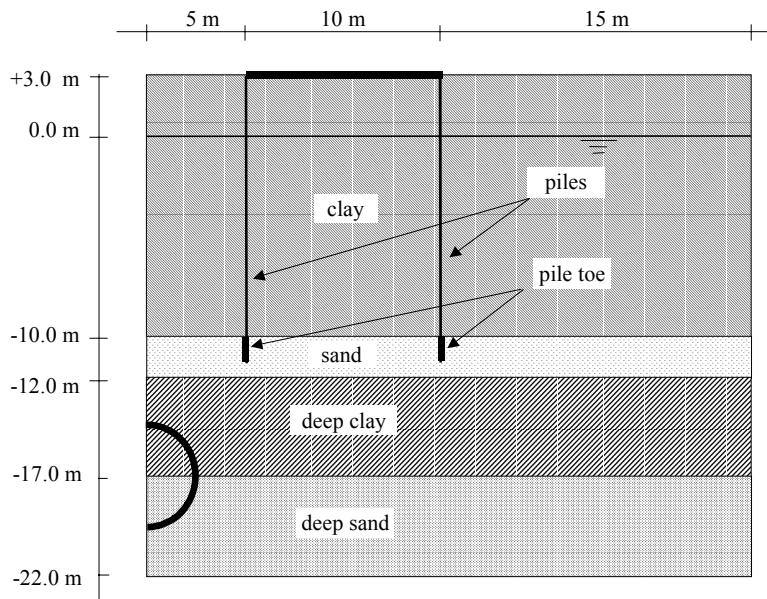


Figure 8.1 Geometry of the tunnel project with an indication of the soil layers

The tunnel considered in this lesson has a diameter of 5.0 m and is located at an average depth of 20 m. The soil profile indicates four distinct layers: The upper 13 m consists of soft clay type soil with stiffness that increases approximately linearly with depth. Under the clay layer there is a 2.0 m thick fine sand layer. This layer is used as a foundation layer for old wooden piles on which traditional brickwork houses were built. The pile foundation of such a building is modelled next to the tunnel. Displacements of these

piles may cause damage to the building, which is highly undesirable. Below the sand layer there is a 5.0 m thick deep loamy clay layer.

This is one of the layers in which the tunnel is constructed. The other part of the tunnel is constructed in the deep sand layer, which consists of dense sand and some gravel. This layer is very stiff. As a result only 5.0 m of this layer is included in the finite element model; the deeper part is considered to be fully rigid and modelled by appropriate boundary conditions. The pore pressure distribution is hydrostatic. The phreatic level is located 3 m below the ground surface (at a level of  $y = 0$  m). Since the situation is more or less symmetric, only one half (the right half) is taken into account in the plane strain model. From the centre of the tunnel the model extends for 30 m in horizontal direction. The 15-node element is adopted for this example.

## 8.1 GEOMETRY

The basic geometry including the four soil layers, as shown in Figure 8.1 (but excluding the tunnel and the foundation elements), can be created using the geometry line option. Since the ground surface is located at 3.0 m above the reference level, the *Top* parameter is taken at +3.0 m in the *General settings* and the *Bottom* at -22.0 m. For the generation of the tunnel we will use the tunnel designer, which is a special tool within PLAXIS that enables the use of circle segments (arcs) and lines to model the geometry of a tunnel. The tunnel considered here is the right half of a circular tunnel and will be composed of four sections. After generating the basic geometry, follow these steps to design the circular tunnel:



Click on the *Tunnel* button in the toolbar. The *Tunnel designer* window appears, with a number of options in its toolbar for creating tunnel shapes. Select *Half tunnel - Right half* from the toolbar.

- The tunnel designer will show a default (half) tunnel shape composed of three sections of which the lower one (Section 1) is selected, as indicated in Figure 8.2. The right side of the window shows some geometrical values.
- Keep the *Type of tunnel* on the default value of a *Bored tunnel*. Make sure that the lower tunnel section is selected (if not, select it by clicking with the mouse in the lower section).
- The values in the table represent the properties of the first tunnel section. For a circular (bored) tunnel the radius can be entered here. Enter a radius of 2.5 m. The result of this action is directly visible in the drawing.
- The value below the radius represents the angle over which the section extends. Enter an angle of 90 degrees (which is the maximum angle of one section).
- The local x- and y-coordinates of the first arc centre point is always located at the local origin ( $x=0$ ;  $y=0$ ) for a bored tunnel.

- Make sure that the options *Shell* and *Interface* are selected for this section.
- Proceed to the next section (2) by pressing the right arrow at the bottom of the window. Alternatively, you may click on the second tunnel section in the designer window.
- Enter an angle of 90 degrees. It is not necessary, nor possible, to enter the radius of the second tunnel segment. This value is automatically adopted from the first tunnel segment in case of a circular tunnel.
- Make sure that the *Shell* and *Interface* options are selected for section 2.

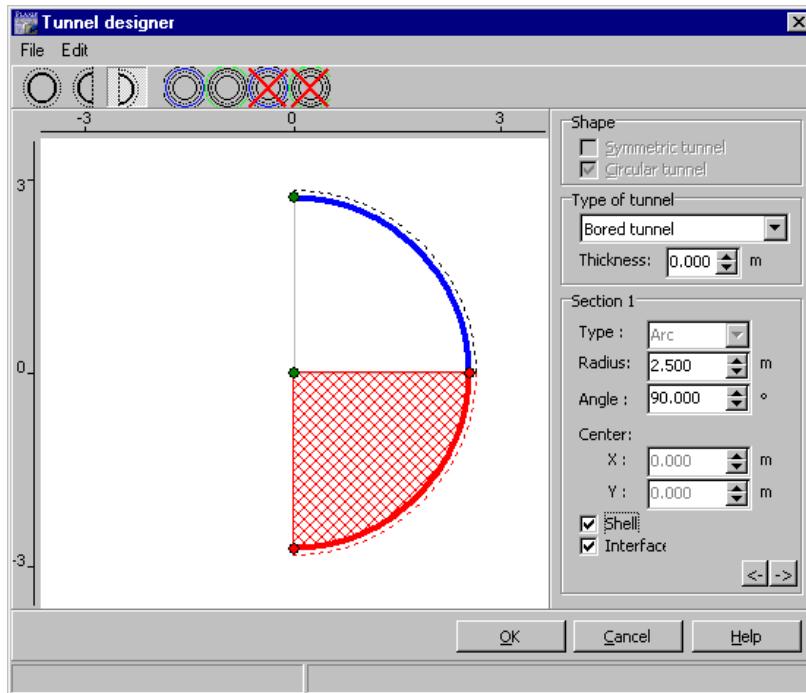


Figure 8.2 Tunnel designer with current tunnel model

**Hint:** A shell and interface can be assigned directly to all tunnel sections by clicking on the corresponding buttons at the top of the tunnel window.  
 > A tunnel lining consists of curved plates. The lining properties can be specified in the material database for plates. Similarly, a tunnel interface is nothing more than a curved interface.

- > In the tunnel as considered here the sections do not have a specific meaning since the tunnel lining is homogeneous and the tunnel will be constructed at once.
- > In general, the meaning of sections becomes significant when:
  - > It is desired to excavate or construct the tunnel (lining) in different stages.
  - > Different tunnel sections have different lining properties.
  - > One would consider hinge connections in the lining (hinges can be added after the design of the tunnel in the general drawing area).
  - > The tunnel shape is composed of arcs with different radii (for example NATM tunnels).

- In the *Shape* group box there are two parameters. The *Symmetric tunnel* parameter can be selected for whole tunnels. The *Circular tunnel* option is automatically selected for bored tunnels.
- The tunnel has now been completely defined. Press the <OK> button to close the tunnel designer.
- Back in the draw area, the tunnel must be included in the geometry model. This is done by entering the global position of the origin of the local tunnel axes. Click on the existing point at position (0.0; -17.0) (5.0 m above the bottom of the geometry model). The tunnel will be drawn with its centre at this location.

The wooden piles below the building are end bearing piles. Only a small part of the total bearing capacity results from skin friction. To correctly model this behaviour, the piles will be modelled using a combination of plates and node-to-node anchors. The building itself will be represented by a stiff plate founded on the node-to-node anchors.

- Draw three separate plates from (5.0;-10.0) to (5.0;-11.0), from (15.0;-10.0) to (15.0;-11.0) and from (5.0;3.0) to (15.0;3.0).
- Connect the top of the pile toes to the foundation plate using node-to-node anchors, as indicated in Figure 8.1.

### ***Boundary conditions***

- Click on the *Standard fixities* button to apply the appropriate boundary conditions. In addition to the standard displacement fixities, fixed rotations are introduced to the upper and lower point of the tunnel lining.

**Hint:** In the *Standard fixities* option, a plate that extends to a geometry boundary that is fixed in at least one direction obtains fixed rotations, whereas a plate that extends to a free boundary obtains a free rotation.

### **Material properties**

The material properties for the four different soil layers are listed in Table 8.1. For all layers the material behaviour is set to drained since we are interested in the long term deformations.

For the upper clay layer we use the advanced option to let the stiffness increase with depth. Therefore a value of  $E_{increment}$  is entered in the *Advanced* parameters window. The value of  $E_{ref}$  becomes a reference value at the reference level  $y_{reference}$ . Below  $y_{reference}$  the actual value of  $E$  increases with depth according to:  $E(y) = E_{ref} + E_{increment} (y_{reference} - y)$ .

The data sets of the two lower soil layers include appropriate parameters for the tunnel interfaces. In the other data sets the interface properties just remain at their default values. Enter four data sets with the properties as listed in Table 8.1 and assign them to the corresponding clusters in the geometry model. To enter the advanced parameters for the Clay data set, click on the <Advanced> button in the *Parameters* tab sheet.

Table 8.1. Material properties of soil in the tunnel project

Parameter	Name	Clay	Sand	Dp. Clay	Dp.sand	Unit
Material model	<i>Model</i>	MC	MC	MC	MC	-
Material beh.	<i>Type</i>	Drained	drained	Drained	drained	-
Soil unit weight above phreatic level	$\gamma_{unsat}$	15	16.5	16	17	kN/m <sup>3</sup>
Soil unit weight below phreatic level	$\gamma_{sat}$	18	20	18.5	21	kN/m <sup>3</sup>
H. permeability	$k_x$	$1 \cdot 10^{-4}$	1.0	$1 \cdot 10^{-2}$	0.5	m/day
V. permeability	$k_y$	$1 \cdot 10^{-4}$	1.0	$1 \cdot 10^{-2}$	0.5	m/day
Young's modulus	$E_{ref}$	1000	80000	10000	120000	kN/m <sup>2</sup>
Increase $E$	$E_{incr}$	650	-	-	-	kN/m <sup>3</sup>
Reference level	$y_{ref}$	0.0	-	-	-	m
Poisson's ratio	$\nu$	0.33	0.3	0.33	0.3	-
Cohesion	$c_{ref}$	5.5	1.0	4.0	1.0	kN/m <sup>2</sup>
Friction angle	$\varphi$	24	31	25	33	°
Dilatancy angle	$\psi$	0.0	1.0	0.0	3.0	°
Interface strength	$R_{inter}$	rigid	rigid	0.7	0.7	-

In addition to the four data sets for the soil and interfaces, three plate and one anchor data set have to be created. The properties for those plates are listed in Table 8.2 and Table 8.3. Assign the Lining data set to the tunnel lining and the Pile toe data set to the two pile toes. The building data set is assigned to the foundation plate representing the building. The weight of this beam also represents the load of the entire building. Assign the Pile data set to the two node-to-node anchors.

Table 8.2. Material properties of the plates

Parameter	Name	Lining	Pile toe	Building	Unit
Type of behaviour	<i>Type</i>	Elastic	Elastic	Elastic	
Normal stiffness	<i>EA</i>	$1 \cdot 10^7$	$2 \cdot 10^6$	$1 \cdot 10^{10}$	kN/m
Flexural rigidity	<i>EI</i>	$1.43 \cdot 10^5$	$8 \cdot 10^3$	$1 \cdot 10^{10}$	$\text{kNm}^2/\text{m}$
Equivalent thickness	<i>d</i>	0.35	0.219	3.464	m
Weight	<i>w</i>	8.4	2.0	25	$\text{kN}/\text{m}/\text{m}$
Poisson's ratio	<i>v</i>	0.15	0.2	0.0	-

Table 8.3. Material properties of the anchors

Parameter	Name	Pile	Unit
Material type	<i>Type</i>	Elastic	
Normal stiffness	<i>EA</i>	$2 \cdot 10^6$	kN
Spacing between anchors	<i>L<sub>spacing</sub></i>	1	m

### ***Mesh generation***

In this example the 15-node element is used as the basic element type. This means that the mesh is more accurate than when using the 6-node element. The global coarseness parameter can remain at its default value (*Coarse*). It can be expected that stress concentrations occur around the tunnel and the pile toes. Therefore the mesh should be refined in these areas. Select the two clusters inside the tunnel and use the *Refine cluster* option in the *Mesh* menu. Select the two plates representing the pile toes and select *Refine line* from the *Mesh* menu.

### ***Initial conditions***

The water weight should be taken 10 kN/m<sup>3</sup>. The water pressures can be generated on the basis of a general phreatic level at a level of  $y = 0.0$  m.

Before generating the initial stresses, make sure that the building, piles, pile toes and tunnel lining are deactivated. The  $K_0$ -procedure can be used to generate the initial effective stresses with the appropriate values of  $K_0$ .

## **8.2 CALCULATIONS**

To simulate the construction of the tunnel it is clear that a staged construction calculation is needed in which the tunnel lining is activated and the soil clusters inside the tunnel are deactivated. Deactivating the soil inside the tunnel only affects the soil stiffness and strength and the effective stresses. Without additional input the water

pressures remain. To remove the water pressure inside the tunnel the two soil clusters in the tunnels must be set to *dry* in the water conditions mode and the water pressures should be regenerated. To create this input, follow these steps.

- The first calculation phase is used to activate the building. Select a *Plastic* calculation using *Staged construction*. Within staged construction mode activate the pile toes, anchors and the foundation plate. Click on <Update> to return to the calculation window.
- The second calculation phase is a plastic calculation, *Staged construction*. On the *Parameters* tab sheet, select the *reset displacements to zero* checkbox. Click on the <Define> button and activate the tunnel lining and deactivate the two soil clusters inside the tunnel.
- Click on the 'switch' to proceed to the water pressures mode. Click on the *Selection* button and select both soil clusters inside the tunnel simultaneously (using the <Shift> key). Double-click on one of the clusters while holding the <Shift> key. This will show the *Cluster pore pressure distribution* window. In this window select *Cluster dry* and click <OK> to return to the water pressure mode.
- Click on the *Generate water pressures* button to generate the water pressures. In the resulting plot it can be seen that there are indeed no water pressures inside the tunnel. Click on the <Update> button to return to the water pressures mode.
- Within the water pressures mode, click on the <Update> button to return to the calculations window.

In addition to the installation of the tunnel lining, the excavation of the soil and the dewatering of the tunnel, the volume loss is simulated by applying a contraction to the tunnel lining. This contraction will be defined in a staged construction calculation phase:

- Click on the <Next> button to introduce a next calculation phase.
- Select a plastic calculation, *Staged construction* and click on the <Define> button.
- Double-click on the centre of the tunnel to open the Tunnel contraction window. Enter a contraction of 2% and click <OK> to return to the geometry mode and <Update> to return to the calculations window.
- Select some characteristic points for load-displacement curves (for example the corner point at the ground surface above the tunnel and the corner points of the building).
- Start the calculations.

**Hint:** The contraction of the tunnel lining by itself does not introduce forces in the tunnel lining. Eventual changes in lining forces as a result of the contraction procedure are due to stress redistributions in the surrounding soil or to changing external forces.

### 8.3 OUTPUT

After the calculation, select the last two calculation phases and click on the <Output> button. The Output program is started, showing the deformed meshes at the end of the calculation phases.

As a result of the second calculation phase (removing soil and water out of the tunnel) there is some settlement of the soil surface and the tunnel lining shows some deformation. In this phase the axial force in the lining is the maximum axial force that will be reached. The lining forces can be viewed by double clicking the lining and selecting force related options from the *Force* menu (see Figure 8.3).

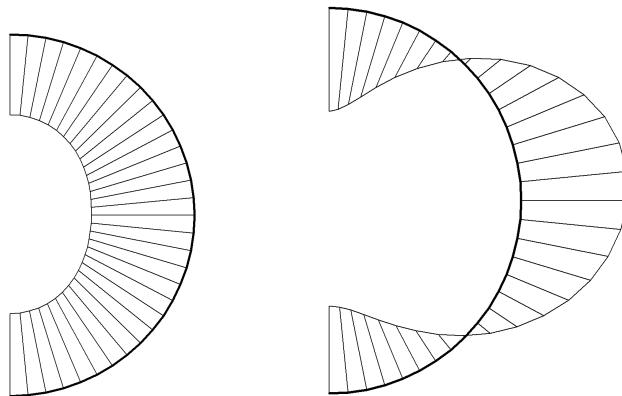


Figure 8.3 Axial forces and Bending moments in the lining after the second phase.

The third calculation phase shows the results due to the simulation of the volume loss. The deformed mesh indicates a settlement trough at the ground surface, which is influenced by the presence of the building. (see Figure 8.4). The plot of effective stresses, Figure 8.5, shows that arching occurs around the tunnel. This arching reduces the stresses acting on the tunnel lining. As a result, the axial force in phase is lower than that after the second calculation phase. The bending moments, however, are larger (see Figure 8.6). The influence of the tunnel contraction on the foundation can be seen in a plot of the relative shear stresses or plots of the displacements of the pile toes.

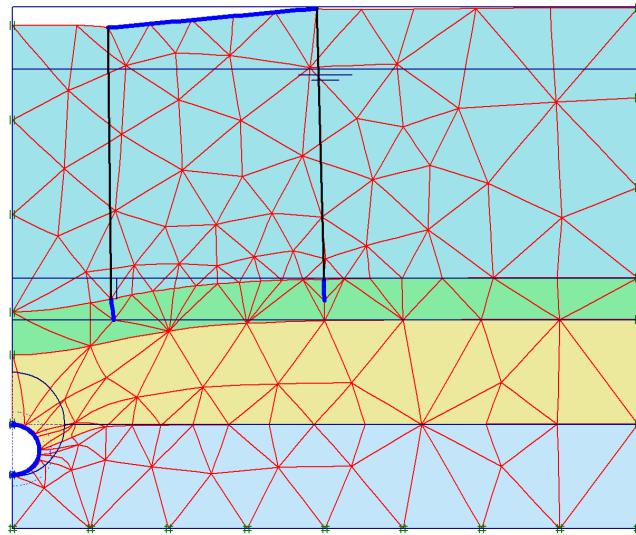


Figure 8.4 Deformed mesh after construction of the tunnel

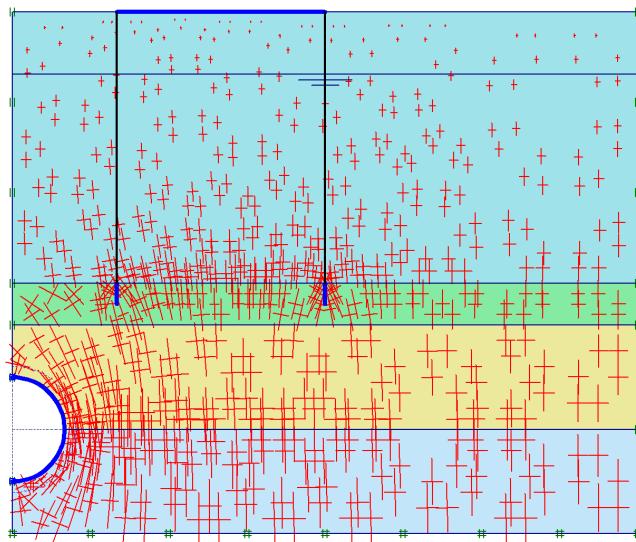


Figure 8.5 Effective stresses after construction of the tunnel

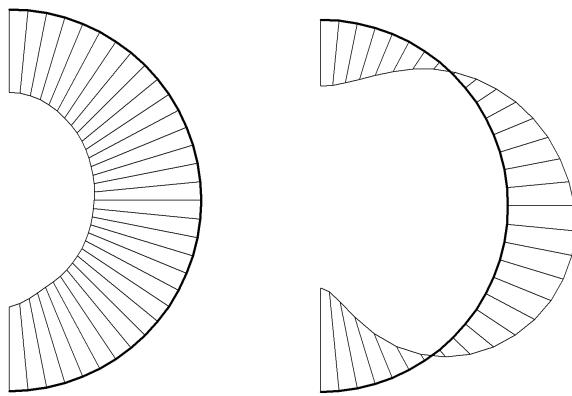


Figure 8.6 Axial forces and bending moments in the lining after the third phase

---

## APPENDIX A - MENU TREE

### A.1 INPUT MENU

INPUT MENU		File	Edit	View	Geometry	Loads	Materials	Mesh
New	Undo	Zoom in		Geometry line	Standard fixities		Soil & interfaces	Basic element type
Open	Copy	Zoom out		Plate	Total fixities	Plates		Global coarseness
Save		Reset view		Geogrid	Vertical fixities	Geogrids		Refine global
Save as		Table		Node-to-node anchor	Horizontal fixities	Anchors		Refine cluster
Print		Rulers		Fixed-end anchor	Rotation fixity (plates)			Refine line
Work directory		Cross hair		Interface	Prescribed displacements			Refine around point
Pack Project				Grid	Tunnel	Distributed load system A		Reset All
Import		Axes			Hinge & rotation spring	Distributed load system B	Generate	
General settings		Snap to grid		Drain	Point load system A			
(Recent projects)				Point numbers	Well	Point load system B		
Exit				Chain numbers				

**INITIAL CONDITIONS MENU**

File	View	Geometry	Generate	Help
Save	Zoom in	Water weight	Water pressures	Help topics
Save as	Zoom out	Phreatic level	Initial stresses	About
Print	Reset view	Closed flow boundary		
General settings	Rulers	Closed consolidation boundary		
Exit	Cross hair			
	Grid			
		Snap to grid		
		Point numbers		
		Chain numbers		

## A.2 CALCULATIONS MENU

### CALCULATION MENU

File	Edit	View	Calculate	Help
Open		Next phase	Current project	Help topics
Save		Insert phase	Multiple projects	About
Pack project		Delete phase(s)		
Print		Copy to clipboard		
Work directory		Select all		
(Recent projects)				
Exit				

**A.3 OUTPUT MENU****OUTPUT MENU (1)**

<b>File</b>	<b>Edit</b>	<b>View</b>	<b>Geometry</b>	<b>Deformations</b>	<b>Stresses</b>	<b>Window</b>	<b>Help</b>
Open	Copy	Zoom in	Structures	Deformed mesh	Effective stresses	Cascade	Help top.
Close	Scale	Zoom out	Materials	Total displacements	Total stresses	Tile	About
Close all	Interval	Reset view	Phreatic level	Horizontal displ. (x)	Cartesian effective stresses		(Active window)
Print	Scan line	Cross section	Loads	Vertical displacements	Cartesian total stresses		
Report generation		Table	Fixities		Overconsolidation ratio		
Work directory		Rulers	Presc. displacements	Hor. increments (x)	Plastic points		
Pack project	(Recent projects)	Title	Connectivity plot	Vertical increments	Active pore pressures		
		Legend	Elements	Total strains	Excess pore pressures		
		Grid	Nodes	Cartesian strains	Groundwater head		
		General info	Stress points	Incremental strains	Flow field		
		Load info	Element numbers	Increm. cartesian strains	Degree of saturation		
		Material info	Node numbers				
			Calculation info	Stress point numb.			
				Material set numb.			
				Cluster numbers			

**OUTPUT MENU (2)***Cross section:*

<b>Deformations</b>	<b>Stresses</b>	<i>Plates:</i>	<b>Deformations</b>	<b>Forces</b>
Total displacements	Effective normal stresses		Total displacements	Axial forces
Horizontal displacem. (x)	Total normal stresses		Horizontal disp.	Shear forces
Vertical displacements	Shear stresses		Vertical disp.	Bending moments
Total increments	Cartesian effective stresses		Total increments	Hoop forces
Horizontal increments (x)	Cartesian total stresses		Horizontal iner. (x)	Force envelopes
Vertical increments	Overconsolidation ratio		Vertical increments	
Normal strain	Effective mean stresses (p)			
Shear strain	Total mean stresses (p)			
Cartesian strains	Deviatoric stresses (q)			
Normal strain increments	Active pore pressure	<i>Interfaces:</i>		
Shear strain increments	Excess pore pressure			
Cartesian strain increments	Groundwater flow			
	Groundwater head			
	Degree of saturation			

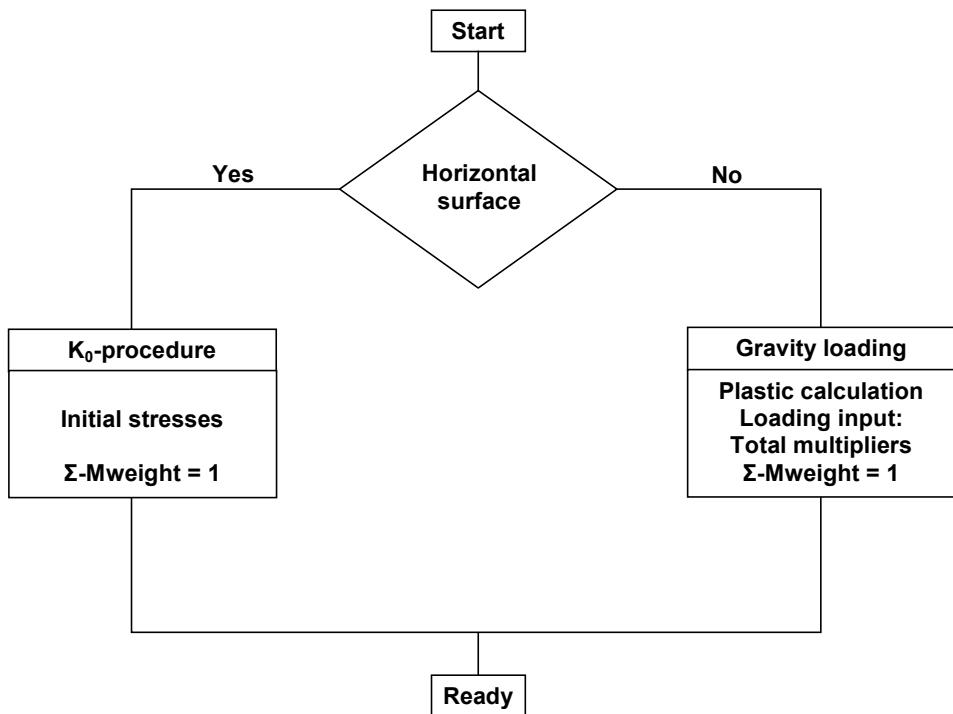
<b>Deformations</b>	<b>Stresses</b>	<i>Plates:</i>	<b>Deformations</b>	<b>Forces</b>
Total displacements	Effective normal stresses		Total displacements	Effective normal stresses
Horiz. displ. (x)	Shear stresses		Horiz. displ. (x)	Shear stresses
Vertical displ.	Shear stresses (z)		Vertical disp.	Shear stresses (z)
Total increments	Relative shear stresses		Total increments	Relative shear stresses
Horizontal increments	Active pore pressures		Horizontal increments	Active pore pressures
Vertical increments	Excess pore pressures		Vertical increments	Excess pore pressures
Relative displ.				
Relative increm.				

#### A.4 CURVES MENU

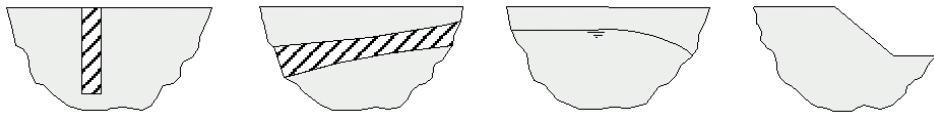
##### CURVES MENU

File	Edit	View	Format	Window	Help
New	Copy	Zoom in	Curves	Cascade	Help topics
Open		Recent view	Chart	Tile horizontally	About
Save		Table		Tile vertically	
Add curve		Legend		(Active windows)	
Delete chart		Value indication			
Close					
Close all					
Work directory					
Print					
(Recent projects)					
Exit					

**APPENDIX B - CALCULATION SCHEME  
FOR INITIAL STRESSES DUE TO SOIL WEIGHT**



Examples of non-horizontal surfaces, and non-horizontal weight stratifications are:



## TUTORIAL MANUAL

---