PLAXIS 3D FOUNDATION
Tutorial Manual
version 1.5

TABLE OF CONTENTS

1 INTRODUCTION

PLAXIS 3D FOUNDATION is a three-dimensional finite element program especially developed for the analysis of foundation structures, including off-shore foundations. It combines simple graphical input procedures, which allow the user to automatically generate complex finite element models, with advanced output facilities and robust calculation procedures. The program is designed such that a user can analyse complex constructions after only a few hours of training.

This Tutorial Manual is designed to help new users get acquainted with the program. The lessons deal with a number of different constructions, which gradually introduce the various features of the program and show some of its possible applications. Users are expected to have a basic understanding of soil mechanics and should be able to work with a Windows environment. It is helpful, but not required, that users are familiar with the standard PLAXIS (2D) deformation analysis program, as many of the interface objects are similar. It is recommended that the lessons are made in the order in which they appear in the manual. The input files and results of the lessons are also available from the examples folder of the PLAXIS 3D FOUNDATION 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, whereas 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.

2 GETTING STARTED

This chapter describes some of the notation and basic input procedures that are used in the PLAXIS 3D FOUNDATION program. 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 3D project to be analysed it is important to create a geometry model first. A geometry model is a representation of a real three-dimensional problem and is defined by work planes and boreholes. 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 two components of a geometry model are described below in more detail.

Boreholes

Boreholes are objects in the geometry model, defining the ground surface, soil layers and the water table at a certain location. Multiple boreholes can be used to define the stratigraphy of the soil for the project. During 3D mesh generation, the position of soil layers is interpolated between bore-holes and the mesh is generated such that the boundaries between soil layers always coincide with the boundaries of elements.

Work planes

Work planes are horizontal planes, with different y-coordinates, presenting a top-view of the geometry model at the specified vertical level. They are used to create the loads and structures in the model. Each work plane holds the same geometry lines, but the distance between work planes may vary, as defined by the input of y-coordinates. Work planes may be used to activate or deactivate point loads, line loads, distributed loads or structural elements.

Within work planes, points, lines and clusters can be used to describe a 2D geometry model. These three components are described below.

Points

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

Lines

Lines are used to define the physical boundaries of the geometry, contours of structures and discontinuities in the geometry such as walls or beams, or excavation areas. 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 properties are homogeneous. Hence, clusters can be regarded as homogeneous parts of a structure or soil layer. Actions related to clusters apply to all elements in the cluster.

After the creation of the 2D geometry model in a work plane, a 2D finite element mesh composed of 6-node triangles can automatically be generated, based on the composition of clusters and lines in the geometry model. If the 2D mesh is satisfactory, an extension into the third dimension can be made. In a 3D finite element mesh three types of components can be identified, as described below.

Elements

During the generation of the mesh, the geometry is divided into 15-node wedge elements. These elements are composed of the 6-node triangular faces in the work planes, as generated by the 2D mesh generation, and 8-node quadrilateral faces in y-direction. In addition to the volume elements, which are generally used to model the soil, compatible 3 node line elements, 6-node plate elements and 16-node interface elements may be generated to model structural behaviour and soil-structure interaction respectively.

Nodes

The wedge elements as used in the 3D Foundation program consist of 15 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, u_y and u_z) 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 wedge element contains 6 stress points as indicated in Figure 2.1. Stress points may be preselected for the generation of stress and strain diagrams.

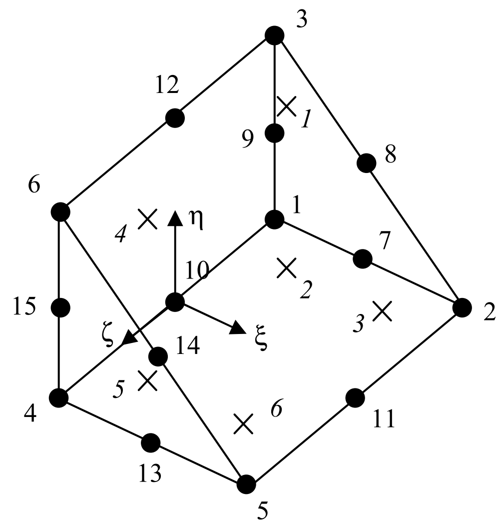


Figure 2.1 Distribution of nodes (●) and stress points (×) in a 15-node wedge element

2.3 INPUT PROCEDURES

In PLAXIS, input is specified by using the mouse and also by keyboard input. In general, four types of input may be identified:

- Input of geometry objects (e.g. drawing a geometry line)
- 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.

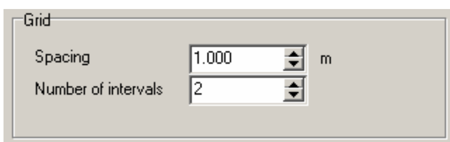


Figure 2.2 Spin edits

2.3.3 INPUT OF SELECTIONS

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

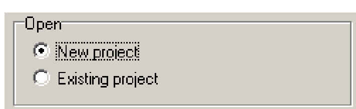


Figure 2.3 Radio buttons

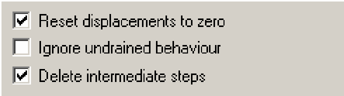


Figure 2.4 Check boxes

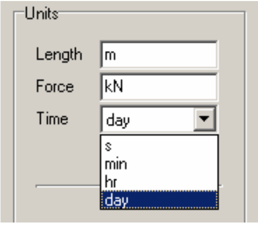


Figure 2.5 Combo boxes

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 project that will be opened is set to a *New project*.

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 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.

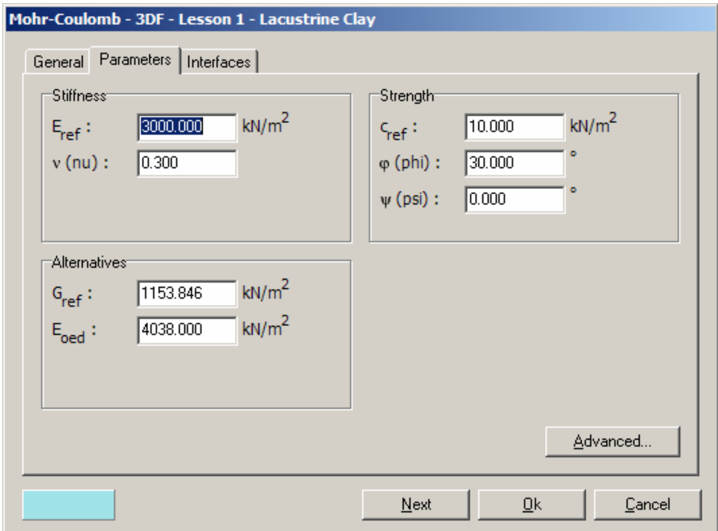


Figure 2.6 Page control and tab sheets

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 which 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*.

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. The PLAXIS 3D FOUNDATION program can be started by selecting the *PLAXIS 3D FOUNDATION input* icon in the PLAXIS 3D FOUNDATION 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.

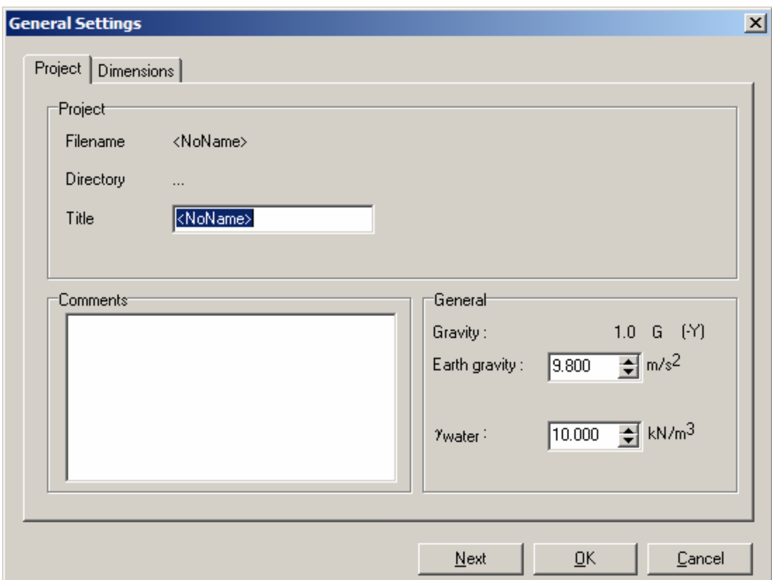


Figure 2.7 General settings - *Project* tab sheet

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 proposed as the default filename and appears on output plots. The comments box is simply a convenient place to store information about the analysis. Also, the magnitude of the standard *earth gravity* and the *unit weight of water* can be specified here.

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 and the *z*-axis points downwards in the draw area. *Xmin* is the lowest *x*-coordinate of model, *Xmax* the highest *x*-coordinate, *Zmin* the lowest *z*-coordinate and *Zmax* the highest *z*-coordinate of the model. Y-coordinates are not entered in the General settings dialog, but during the input of bore holes and work planes.

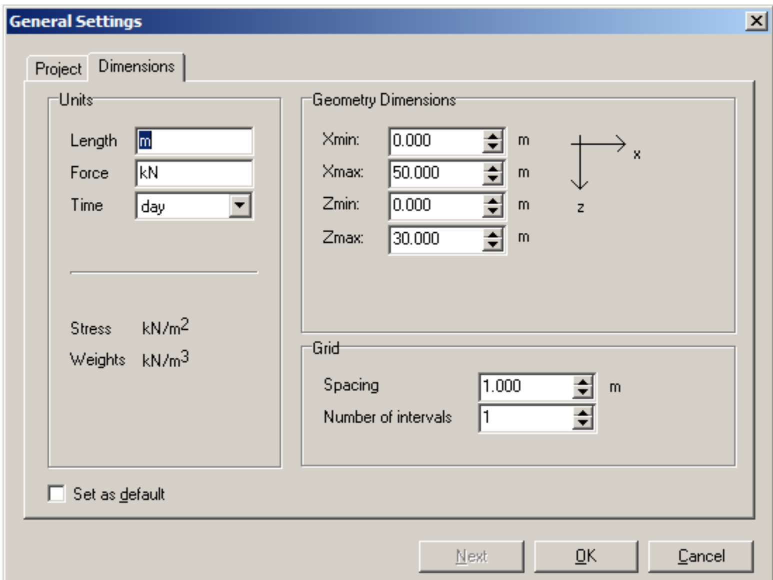


Figure 2.8 General settings - *Dimensions* tab sheet

In practice, the draw area resulting from the given values of *Xmin*, *Xmax*, *Zmin* and *Zmax* 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 MODEL

When the general settings are entered and the <OK> button is clicked, the main 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. For a more extensive description the reader is referred to the Reference Manual.

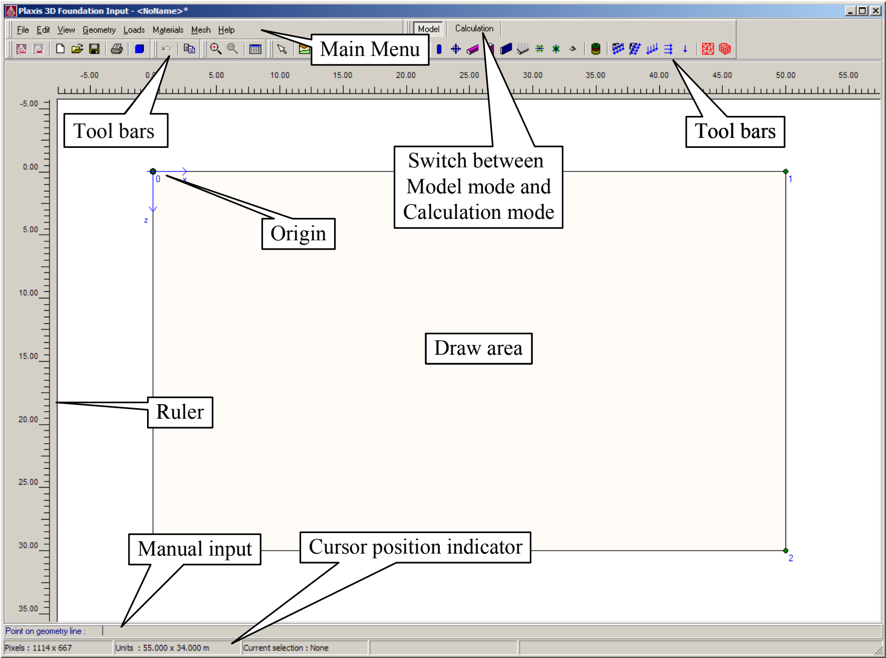


Figure 2.9 Main window of the Input program

Input menu

The Input menu contains all input items and operation facilities of the Input program. Most items are also available as buttons in the tool bar.

Tool bar (File)

This tool bar contains buttons for file operations, corresponding with the options in the File menu. It also contains buttons to start the other sub-programs of the PLAXIS 3D FOUNDATION package (Output, Curves).

Tool bar (Edit)

This tool bar contains buttons for editing operations, corresponding with the options in the Edit menu.

Tool bar (View)

This tool bar contains buttons for viewing operations such as zooming into a particular part of the draw area. The buttons correspond with the options in the View menu.

Tool bar (General)

This tool bar contains buttons for functionalities that apply to the Model mode as well as to the Calculation mode, among which the use of the selection tool and the selection of a work plane.

Tool bar (Model)

This tool bar contains buttons related to the creation of a geometry model, such as Geometry lines, Piles, Beams, Walls, Floors, Line fixities, Springs, Bore holes and loads, as well as options for 2D and 3D mesh generation.

Tool bar (Calculation)

This tool bar contains buttons related to the definition of calculation phases.

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 z-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 z-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.

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.

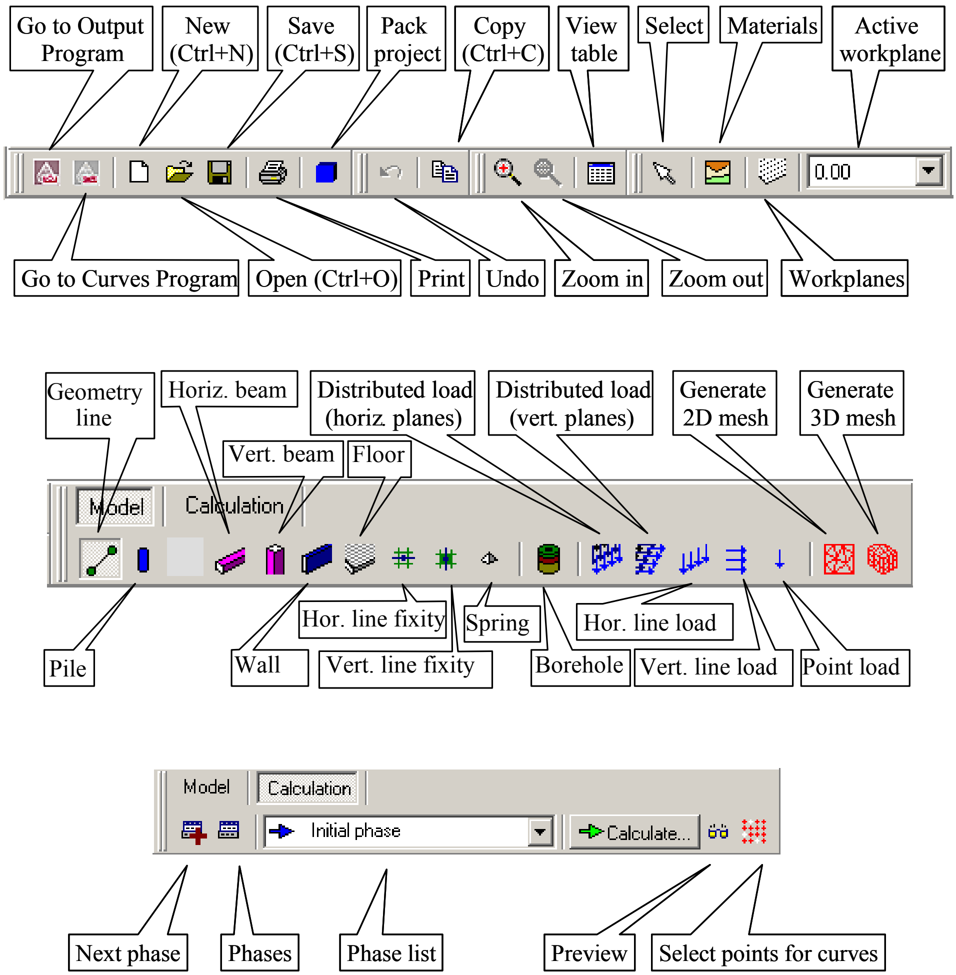


Figure 2.10 Toolbars

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.

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

3 RAFT FOUNDATION ON OVERCONSOLIDATED CLAY (LESSON 1)

In the previous chapter some general aspects and basic features of the PLAXIS 3D FOUNDATION program were presented. In this chapter a first application is considered, namely the settlement of a raft foundation on clay. This is the first step in becoming familiar with the practical use of the program.

The general procedures for the creation of a geometry, 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 later lessons. Therefore, it is important to complete this first lesson before attempting any further tutorial examples.

3.1 GEOMETRY

This exercise deals with the construction and loading of a raft foundation on a lightly overconsolidated Lacustrine Clay. Below the clay layer there is a stiff rock layer that forms a natural boundary for the considered geometry. The rock layer is not included in the geometry; instead an appropriate boundary condition is applied at the bottom of the clay layer. The purpose of the exercise is to find the settlement of the raft foundation and the structural forces in the floor slab.

The building is composed of a basement level and 4 floors above ground level, see Figure 3.1. In this exercise only the basement will be modelled. The loads from the upper floors are transferred to the floor slab by columns. Each column bears a load of 6000 kN, as sketched in Figure 3.2.

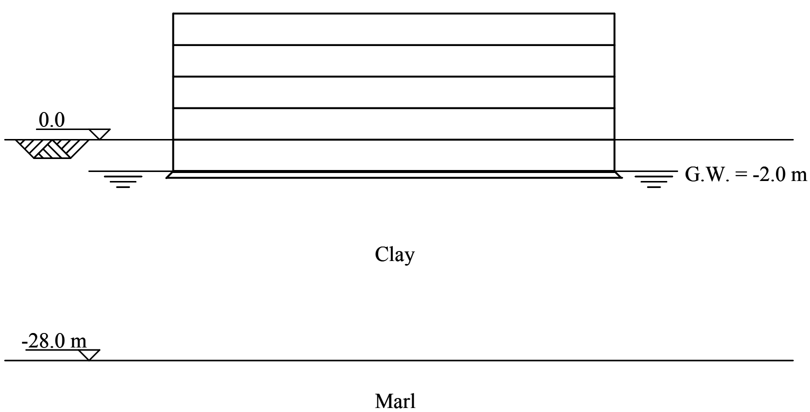


Figure 3.1 Side view of building on raft foundation

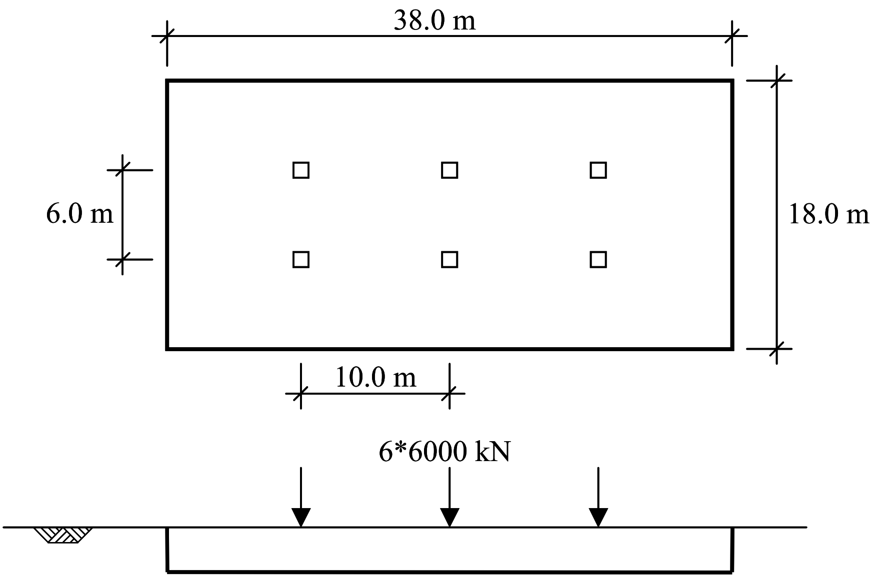


Figure 3.2 Top view and simplified geometry of the building

3.2 CREATING THE INPUT

Start the PLAXIS 3D FOUNDATION input program. A *Create/Open project* dialog box will appear in which you can select an existing project or create a new one. Choose *New Project* and click the <OK> button (see Figure 3.3). Now the *General settings* window appears, consisting of the two tab sheets *Project* and *Dimensions* (see Figure 3.4 and Figure 3.5).

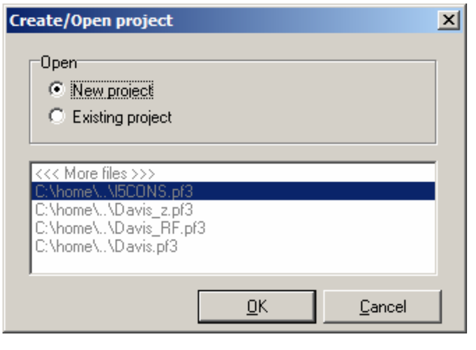


Figure 3.3 Create/Open dialog box

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 basic units and the size of the draw area. To enter the appropriate settings for the raft foundation calculation follow these steps:

- In the *Project* tab sheet, enter 'Lesson 1' in the Title box and type 'Settlements of a raft foundation' in the *Comments* box.
- The *General* box indicates a fixed gravity of 1.0 *G*, in the vertical direction downward (-*Y*). Also, the value of the acceleration of gravity (1.0 *G*) can be entered in the *Earth gravity* box. This should be kept to the default value of 9.8 m/s^2 for this exercise.
- In the γ_{water} box the unit weight of water can be entered. Keep this to the default value of 10 kN/m^3. Click the <Next> button below or click the *Dimensions* tab sheet.

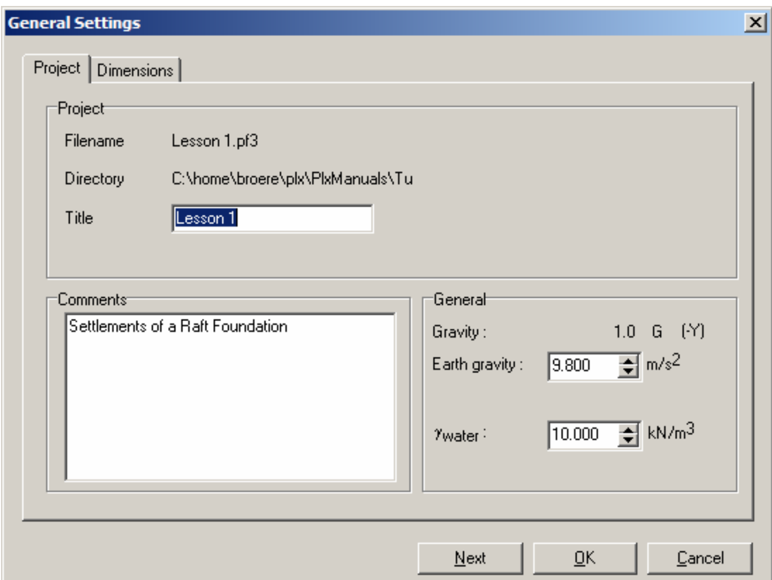


Figure 3.4 Project tab sheet of the General settings window

- 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, a small margin is automatically added so that the geometry will fit well within the draw area. Enter –50.0, 50.0, -30.0, 30.0 for *Xmin*, *Xmax*, *Zmin*, *Zmax* respectively.

- The *Grid* box contains the settings for 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 a 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 the <OK> button to confirm the settings. Now the draw area appears in which the geometry model can be drawn.

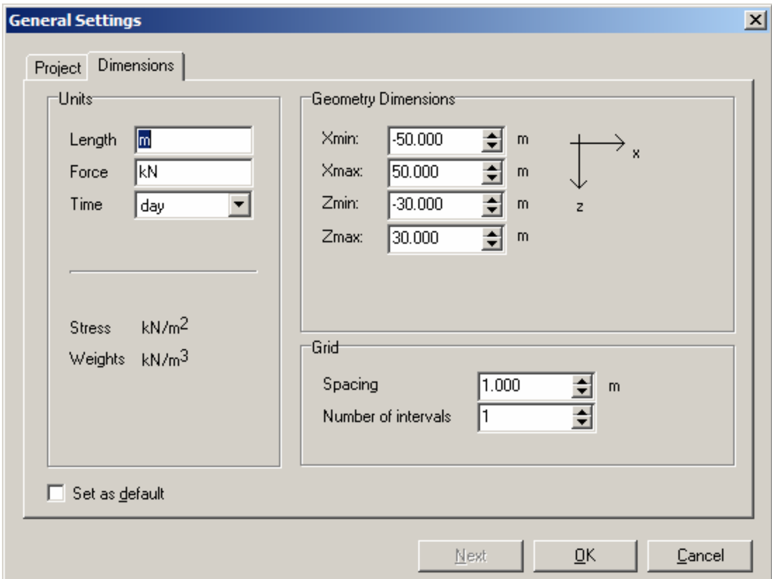


Figure 3.5 Dimensions tab sheet of the General settings window

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.

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 z-axis is pointing downwards on the screen. The y-axis is perpendicular to the draw area, pointing towards the user. A first cluster (area that is fully enclosed by geometry lines) with dimensions equal to the dimensions entered in the *General settings* window is also automatically created. A geometry model can be created in this draw area, which will be considered later on. First, we will consider the extension into the vertical direction (y-direction) by the definition of work planes.

Work planes

Work planes are horizontal layers with different *y*-coordinates, at which the structural objects, loads and construction stages can be defined. Work planes are needed at each level where a discontinuity in the geometry or the loading occurs in the initial situation or in the construction process. They are defined in the *Work planes* window. This window can be opened with the *Work planes* button, found on the toolbar just left of the *Active Work plane* combo-box, or from the *Work planes* option in the *Geometry* menu. One work plane has already been automatically created at $y = 0.0$. For this first project, we need to define two additional work planes. To do this, follow these steps:

Open the *Work planes* window by clicking the *Work planes* button on the toolbar. A new window will appear, in which the automatically generated work plane at $y = 0$ is shown. A vertical cross section is shown, indicating in red the currently selected *work plane*.

- Click the <Add> button. A new work plane is inserted at $y = -3.0$ m, below the current work plane.
- Select the new work plane ($y = -3.0$ m) by clicking on the entry in the table on the left and enter a value $y = -28.0$ m. Press <Enter> to accept the new value.
- Click the <Insert> button, and change the *y*-coordinate of the new work plane to $y = -2.0$ m. After accepting this value, the overview of work planes is updated, as shown in Figure 3.6.

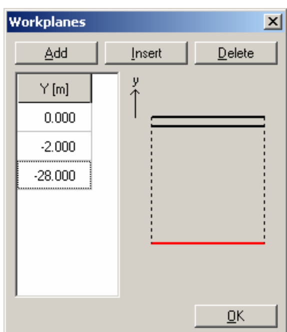


Figure 3.6 Work planes window

- Click the <OK> button to close the *Work planes* window and return to the draw area.

Building Geometry

To create objects, you can either use 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 lower right toolbar or from the *Geometry* menu. For this project, we will start by defining the basement walls and floor. First we need to select the correct work plane.

- From the *Active work planes* combo-box, select the work plane at *y* = 0.0 m. Click on the arrow to open the combo box and click on the entry 0.0.

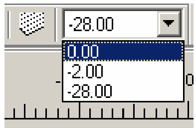


Figure 3.7 Active work plane combo box

In order to construct the outside walls of the building, follow these steps:

Select the *Wall* option (the fifth button on the geometry toolbar).

- Position the cursor at the coordinate (-19.0, 9.0). Check that the units in the status bar read –19.0 x 9.0 and click the left mouse button once. The first geometry point in addition to the default geometry boundary points (number 4) has now been created.
- Move to the right to position (19.0, 9.0). Click the left mouse button to generate the second point (number 5). At the same time the first wall is created from point 4 to point 5, below the current work plane. This is indicated by a thick blue line. Interfaces are automatically added to both sides of the wall. These are indicated by black dashed lines. Interfaces are used to model the soil-structure interaction.
- Move upwards to position (19.0, -9.0) and click again.
- Move to position (-19.0, -9.0) and click again.
- Finally, move back to point 4, position (-19.0, 9.0) 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 7 to point 4.
- Click the right mouse button to stop drawing.

Hint: When adding a geometry line to a work plane, this geometry line will be repeated in all other work planes. When structural objects such as floors and beams are added, these will be added to the currently active work plane only. Walls will be added below the currently active work plane. In addition, a corresponding geometry line will be added to all other work planes.

- After the walls have been defined, we can easily add the basement floor.
- From the *Active work planes* combo box, select the work plane at *y* = -2.0 m.

Select the *Floor* option from the geometry toolbar.

- Click anywhere within the area enclosed by the walls. This area should now be coloured olive (green), indicating that a floor has been added.

Hint:	Incorrectly positioned 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 line and drag it to the desired position. To delete a point or line, select the point or 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> from the keyboard.
>	Lines can be drawn perfectly horizontal or vertical by holding down the <Shift> key on the keyboard while moving the mouse.

Loads

After the basement geometry has been fully defined, the load resulting from the building itself has to be modelled. This load is transferred by columns from the top floors onto the basement floor. For this exercise, there is no need to model the columns themselves. The loads can be modelled using point loads acting directly on the basement floor. As Plaxis 3D Foundation cannot handle single unconnected points during mesh generation, extra geometry lines have to be added before entering the actual loads.

Select the Geometry Line tool from the geometry toolbar.

- Make sure the work plane at y=-2.0 m is selected and move the cursor to (-10.0, -3.0) and click.
- Now move the cursor to (10.0, -3.0) and click again.
- Right-click to end drawing this geometry line.
- Also draw a geometry line from (-10.0, 3.0) to (10.0, 3.0).

The columns are spaced 10 metres apart, as indicated in Figure 3.2. To enter the loads:

Select the *Point load* tool from the geometry toolbar.

- Move the cursor to (-10.0, 3.0) and click to add the point load. The point load has a default value of 1 kN, acting in the downward y-direction, perpendicular to the work plane. Such a load is represented by a blue circle with a letter A next to it.
- Repeat this action at locations (-10.0, -3.0), (0.0, 3.0), (0.0, -3.0), (10.0, 3.0), and (10.0, -3.0).

This way, all loads are defined with a default value of 1 kN acting in the negative y-direction. Later on, in the definition of calculation stages, the value of all loads will be changed to the desired values.

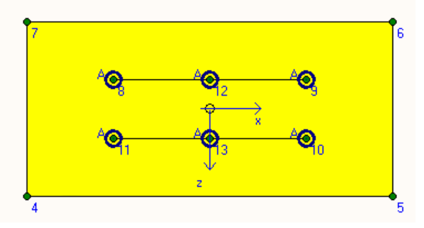


Figure 3.8 Overview of the load input

Borehole

All structural information on the geometry in the vertical direction has been entered using work planes. Information on the soil layers and the water table is entered in a different way, using so called *Boreholes*. Boreholes are locations in the draw area at which the information on the location of soil layers and the water table is given. If multiple boreholes are defined, PLAXIS 3D FOUNDATION will automatically interpolate between the boreholes, and derive the position of the soil layers from the borehole information.

Hint:	PLAXIS 3D FOUNDATION can also deal with layers that are discontinuous, i.e. only locally present in the model area. In such cases, so called degenerated elements are used. See section 3.3.11 of the Reference Manual for more information.

In the current example, only one soil layer is present, and only a single borehole is needed to define the soil layer. In order to define the borehole, follow these steps:

 Select the *Borehole* tool from the geometry toolbar.

- Click at a location in the cluster that represents the soil. It is suggested to click on (-50.0 , -30.0).
- This places a borehole at location (-50.0, -30.0) and opens the *Borehole* window. The top and bottom boundaries of the borehole will be determined automatically. They correspond to y = 0.0 m and to the lowest work plane in the model, in this case y = -28.0 m. For this exercise, these values are correct and no change is needed.
- In the *Water level* box, set the water level to y = -2.0 m. Do not close the window.
- All geometry objects have now been defined. Before the mesh can be generated, material properties have to be assigned to all objects.

Material Data Sets

In order to simulate the behaviour of the soil and the structures, a suitable material 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 assigned to one or more clusters or structural objects. Different objects have different types of data sets, and these types cannot be mixed.

PLAXIS 3D FOUNDATION distinguishes between material data sets for *Soils & Interfaces*, *Beams*, *Walls*, *Floors* and *Springs*. The input of material data sets is generally done after the input of all geometry objects. Before the mesh can be generated material data sets have to be assigned to all clusters and structures.

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

Soils & Interfaces

To create a material set for the clay layer, using the last of the three possible methods, follow these steps:

- The *Borehole* window should still be open. If it is not open, choose the *Select* tool from the toolbar and double-click on the borehole at (-50,-30).

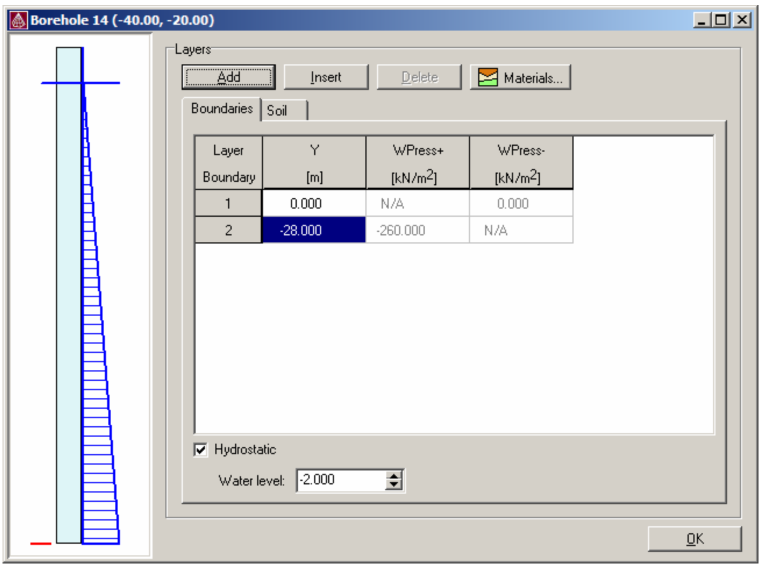


Figure 3.9 The borehole window

Table 1 Material properties of the clay layer

Parameter	Name	Value	Unit
Material model	*Model*	Mohr-Coulomb	-
Type of material behaviour	*Type*	Drained	-
Unit weight of soil above phreatic level	γ_{unsat}	17.0	kN/m^3
Unit weight of soil below phreatic level	γ_{sat}	18.0	kN/m^3
Young's modulus (constant)	E_{ref}	3000	kN/m^2
Poisson's ratio	ν	0.3	-
Cohesion (constant)	c_{ref}	10.0	kN/m^2
Friction angle	φ	30.0	°
Dilatancy angle	ψ	0.0	°

- This opens the *Borehole* window and shows the layers in the selected borehole. In this case only a single layer is present, with a bottom at –28 m and a top at 0 m.

Open the *Material Sets* window by clicking on the *Materials* button.

- Click the <New> button on the lower side of the *Material Sets* window. A new dialog box will appear with three tab sheets: *General*, *Parameters* and *Interfaces* (see Figure 3.10 and Figure 3.11).
- In the *Material Set* box of the *General* tab sheet, write 'Lacustrine Clay' in the *Identification* box.
- Select *Mohr-Coulomb* as the material model from the *Material model* combo box and *Drained* from the *Material type* combo box (default parameters).
- Enter the unit weights in the *General properties* box according to the material data set as listed in Table 1.
- Click the <Next> button or click 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). The Mohr-Coulomb model involves only five basic parameters (E, ν, c, φ, ψ). See the Material Models manual for a detailed description of the different soil models and their corresponding parameters.
- Enter the model parameters of Table 1 in the corresponding edit boxes of the *Parameters* tab sheet.

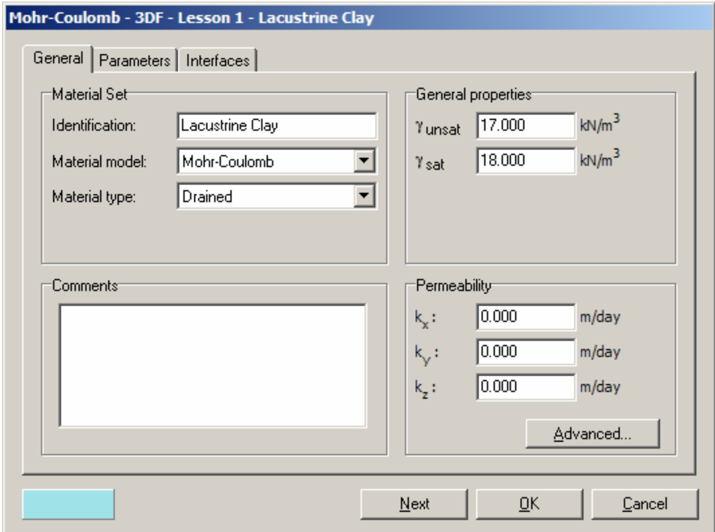


Figure 3.10 *General* tab sheet of the soil and interfaces data set window

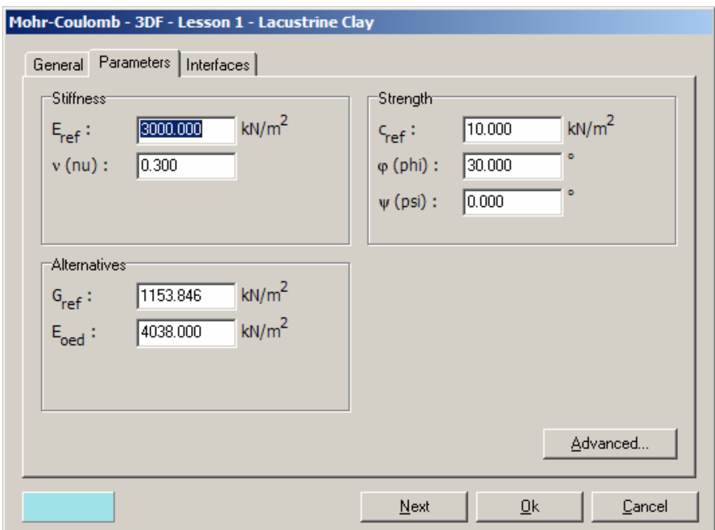


Figure 3.11 *Parameters* tab sheet of the soil and interfaces data set window

- On the third tab sheet the parameters for *Interfaces* can be entered. Click the *Interfaces* tab or the <Next> button to proceed to the *Interfaces* tab. For this exercise the interface *Strength* will be left on the default setting *Rigid*. Click 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 set 'Lacustrine Clay' from the *Material Sets* window to the soil column in the *Borehole* window 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 the data set to the soil layer is indicated by a change in the colour of the layer.
- The *Soil* tab sheet may be skipped in this exercise. Click the <OK> button in the *Material Sets* window to close the database.
- Click the <OK> button in the *Borehole* window to return to the draw area.

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 the <Global> button of the *Material Sets* window. Drag the appropriate data set (in this case '3DF Lesson 1 Lacustrine Clay') 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 from the general tool bar.

\> 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.

Walls

Walls and floors also need a material data set. To create the data set for the wall, follow these steps:

- Set the *Active work plane* to $y = 0.0$ m.

Open the *Material sets* window by clicking on the corresponding button on the toolbar. In the material data set window, set the *Set type* to *Walls* (see Figure 3.12). Click the <New> button. Enter 'Basement Wall' as the *Material set* identification.

- Select *Linear* for the properties and enter the properties given in Table 2. As the material is considered homogeneous, the same value needs to be entered for each triplet of E, G and ν. To this end, first select the *Isotropic* option.
- Now enter the isotropic values in the boxes labelled E_1 and ν_{12}. The value for G_{12} is automatically updated.

- Click the <OK> button to close the data set.
- Drag and drop the Basement Wall data set to the wall in the draw area. If the *Material sets* window partially obscures the walls and floor of the building, the window can be dragged aside. The cursor will change to indicate if the material set can be dropped.
  The walls will briefly flash red and change colour from light blue to dark blue, indicating that a material data set has been assigned successfully.

Table 2 Material properties of the Basement Wall (wall)

Parameter	Name	Basement Wall	Unit
Type of Behaviour	Type	Linear	-
Thickness	d	0.3	m
Weight	γ	24	kN/m^3
Young's modulus	E	$1 \cdot 10^7$	kN/m^2
Shear modulus	G	$4.167 \cdot 10^6$	kN/m^2
Poisson's ratio	ν	0.2	-

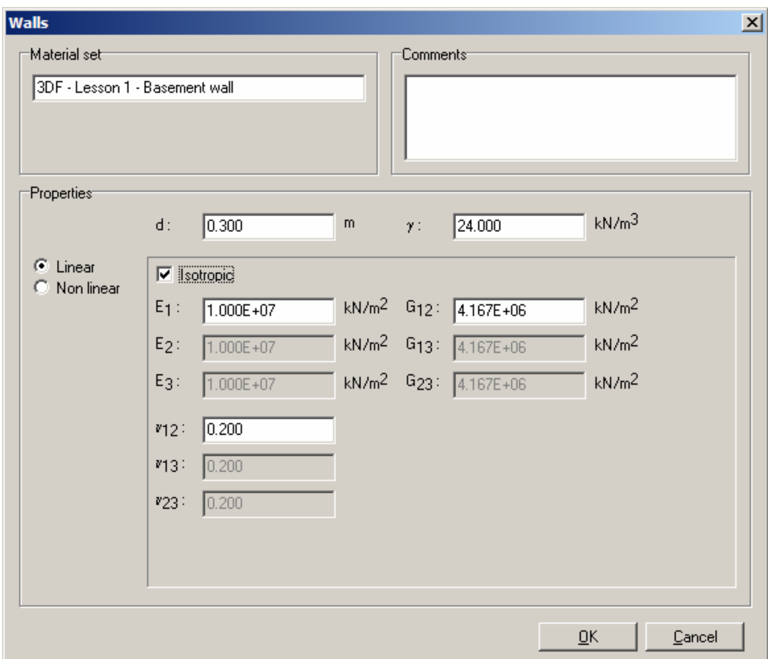


Figure 3.12 Input for the Walls material data set

Hint: To check whether the correct material set has been assigned to a structural element, double click the object, and select it from the select window that appears. A window will appear, see Figure 3.13, indicating the type of object and the material data set assigned to it. The *Change* button can be used to assign a different material data set to the object.

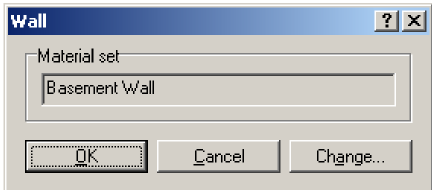


Figure 3.13 Inspecting the material data set for a structural element

Floor

The material properties for the floor are entered in a similar manner. Without first closing the *Material sets* window, do the following:

- Set the Active work plane to y = -2.0 m.
- Set the *Set type* in the *Material sets* window to *Floors*. Click the <New> button and enter 'Basement Floor' as the *Material set* identification.
- Enter the properties given in Table 3 and click <OK> to close the data set.
- Drag and drop the Basement Floor data set to the floor cluster in the draw area. The floor will briefly flash red and change colour from light green to dark green. Close the *Material sets* window by clicking <OK>.

Table 3 Material properties of the Basement Floor (floor)

Parameter	Name	Basement Floor	Unit
Type of behaviour	*Type*	Linear	-
Thickness	d	0.5	m
Weight	γ	24	kN/m^3
Young's modulus	E	$1 \cdot 10^7$	kN/m^2
Shear modulus	G	$4.167 \cdot 10^6$	kN/m^2
Poisson's ratio	ν	0.2	-

2D Mesh Generation

When the geometry model is complete and all borehole information has been entered, it is recommended to generate a 2D finite element mesh before generating a full 3D mesh. PLAXIS 3D FOUNDATION allows for a fully automatic mesh generation procedure, in which the geometry is divided into volume elements and compatible structure elements, if applicable. The mesh generation takes full account of the position of lines and points in the geometry model, so that the exact position of layers, loads and structures is accounted for in the finite element mesh. The 2D mesh generation process is based on a robust triangulation principle that searches for optimised triangles and which results in an unstructured mesh. Although unstructured meshes do not form regular patterns of elements, the numerical performance of these meshes is usually better than structured meshes with regular arrays of elements. In addition to the mesh generation itself, a transformation of the input data (properties, boundary conditions, material sets, etc.) from the geometry model (points, lines and clusters) to the finite element mesh (elements and nodes) is made.

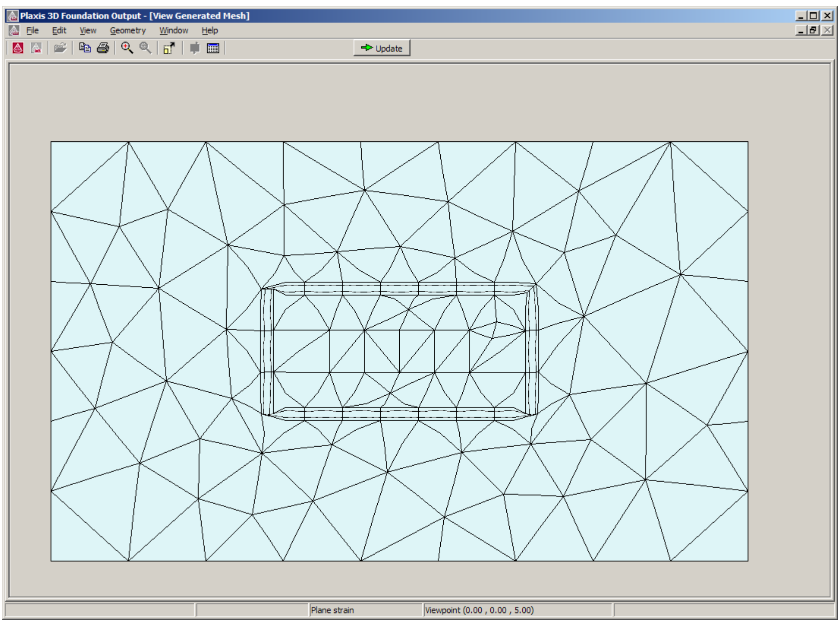


Figure 3.14 2D finite element mesh of the geometry

To generate the mesh, follow these steps:

Click the *Generate 2D mesh* button in the toolbar or select the *Generate 2D mesh* 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 3.14).

This window can be used to inspect the generated mesh and to decide if mesh refinement is required.

- Click the <Update> button to return to the *Input* window.
- Large displacement gradients are expected around and under the basement. Hence, it is appropriate to have a finer mesh around and under the building. Select the cluster that represents the basement floor. The floor is now indicated in red as shown in Figure 3.15. From the *Mesh* menu, select the option *Refine cluster*. As a result, a local refinement of the indicated cluster is visible in the presented mesh. Click the <Update> button to return.

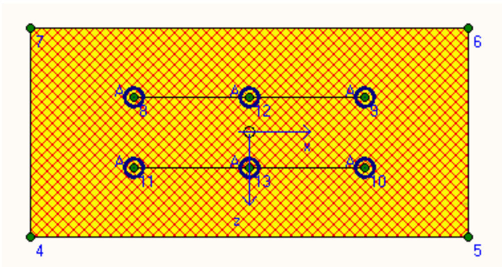


Figure 3.15 The selected Floor cluster

- If necessary, the mesh can be further optimised by performing global or local refinements. These mesh refinements are considered in a later lesson. Here it is suggested that the current mesh is accepted.

Hint: By default, the Global coarseness of the mesh is set to *Coarse*, which is adequate as a first approach in many cases. The *Global coarseness* setting can be changed in the *Mesh* menu. In addition, 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, the finite element mesh has to be regenerated.

> The automatically generated mesh may not be perfectly suitable for the intended calculation. Therefore it is recommended that the user inspect the mesh and makes refinements if necessary.

3D Mesh Generation

After generation of the 2D mesh, the model must be extended to a fully 3D mesh. This can be done by clicking on the 3D mesh generation button or selecting the corresponding option from the *Mesh* menu. The information in vertical direction on layer boundaries, construction levels and changes in the geometry has already been entered, using work planes and boreholes. No extra information is needed in order to generate the 3D mesh. The 3D mesh is created by connecting the corners of the 2D triangular elements to the corresponding points of the corresponding elements in the next work plane. In this way a 3D mesh, composed of 15-noded wedge elements is formed. Where needed extra element levels are automatically introduced, so that the size of the elements in *y*-direction is about equal to the average element size defined for the 2D mesh. If multiple boreholes are used and not all layers are present in all boreholes, so-called degenerated elements will be introduced, to deal with the transition between the various layers. See section 3.3.11 Boreholes of the Reference Manual for more details.

To generate the 3D mesh, follow these steps:

Click the *Generate 3D mesh* button on the toolbar or select *Generate 3D mesh* from the *Mesh* menu. The 3D mesh generation procedure is started and the 3D mesh is displayed in the *Output* window. An additional element level is automatically introduced between the work planes at –2 m and –28 m to reduce the element size in *y*-direction. The 3D mesh and the various structural elements are shown in two different windows. The arrow keys of the keyboard allow the user to rotate the model so that it can be viewed from any direction (see Figure 3.16).

- Click the <Update> button to return to the *Input* program.

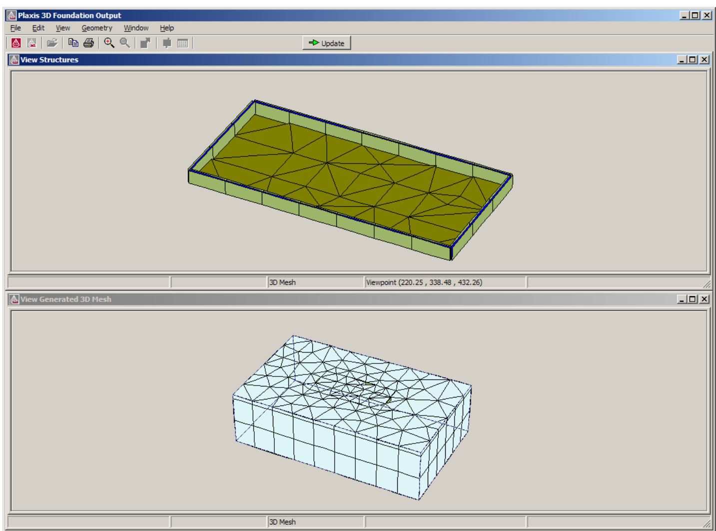
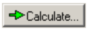


Figure 3.16 Generated 3D mesh in the Output window

Hint:	If the distance between two adjacent work planes is significantly larger than the average (2D) element size, the 3D mesh generation procedure will automatically generate *intermediate element levels* (see Figure 3.16) to avoid badly shaped elements.

3.3 PERFORMING CALCULATIONS

Once the mesh has been generated, the finite element model is complete. Before the actual calculation is started, the calculation stages have to be defined. The first of these will be the definition of initial conditions. Other stages can be the excavation and construction of the basement or the activation of loads.



Click the *Calculation* button to switch to calculation mode.

- The program proposes to save the model. Press <Yes> to do so and save the model under its predefined name (Lesson 1.PF3).

The definition of calculation stages, such as switching on and off parts of the geometry, assigning different material data sets to clusters or structural elements or changing the magnitude of loads, is done in the calculation mode of the Input program. Clicking the <Calculation> button will hide the geometry model toolbar and show the calculation toolbar. This toolbar, see Figure 3.17, contains a calculation phases combo box and a number of buttons to define calculation phases, perform the calculation and activate the *Output* program, in order to view the results. The calculation mode of the Input program is a separate mode where calculation phases are defined and calculations can be started. Clicking the *Phases* button will open the Phases window (see Figure 3.18).

Figure 3.17 The Calculations toolbar

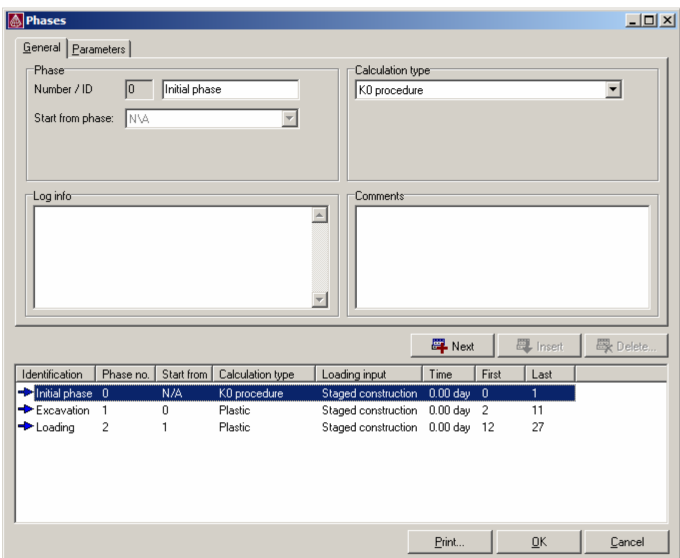


Figure 3.18 The Phases window with the General tab sheet

In the Phases window you can add and delete calculation phases, and set the parameters for the calculation procedure. All defined calculation phases appear in the list at the lower part of the window.

As in all PLAXIS programs the 3D Foundation program has convenient procedures for automatic load stepping and for the activation and deactivation of loads and parts of the geometry (staged construction). These procedures can be used for many practical applications. Staged construction is a very useful type of loading input. In this special PLAXIS feature it is possible to change the geometry and load configuration by deactivating or reactivating loads, volume elements or structural objects as created in the geometry input. Staged construction provides an accurate and realistic simulation of the various loading, construction and excavation processes. This option can also be used to reassign material data sets to clusters or structural elements.

Initial Conditions

Before starting the actual calculations the initial conditions must be generated. In general, the initial conditions comprise the initial geometry configuration and the initial stress state. The initial water conditions for the clay layer have been entered already in the *Borehole* window. These conditions are also taken into account to calculate the initial effective stress state. When a new project has been defined, a first calculation phase, named ‘Initial phase’, is automatically created and selected in the *Phase list* combo-box and the *Phases* window. All structural elements and loads that are present in the geometry are initially automatically switched off, only the soil clusters are initially active.

In PLAXIS 3D FOUNDATION two methods are available to generate the initial stresses, *gravity loading* or the *K0 procedure*. By default *gravity loading* is used, which requires no additional actions except for running the calculation. In this example, however, the *K0 procedure* will be used.

> **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 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-sinφ. If the value was changed, the default value can be regained by entering a negative value for K_0.

In order to generate the initial stresses according to the *K0 procedure*, follow these steps:

Click on the *Phases window* button to open the *Phases window*.

- In the *General* tab sheet, the *Calculation type* is set to the default of *Gravity loading*. In order to generate initial stresses according to the K_0 procedure, select *K0 procedure* from the *Calculation type* combo box.

- Click the <Parameters> button or select the *Parameters* tab.
- If the *K0 procedure* calculation type is selected, the Parameters tab sheet contains the *K0 procedure* input fields. In this tab sheet, accept the default values of K_0 as suggested by PLAXIS and click <OK>.

Defining Construction Stages

After the definition of the initial conditions, the construction of the foundation and the loading can be modelled. This will be done in two separate calculation phases, which need to be added. To do this, follow these steps:

Click the *Next phase* button. This will add a new phase and open the *Phases* window and create a new calculation phase. This new phase will be named <Phase 1> and is automatically selected.

- In the *General* tab sheet, write (optionally) an appropriate name for the new phase in the *ID* box (for example 'Excavation') and select the phase from which the current phase should start (in this case the calculation phase can only start from phase *0 – Initial phase*, which contains the initial stress state).
- Leave the *Calculation type* to *Plastic* and click the *Parameters* tab to open the *Parameters* tab sheet.

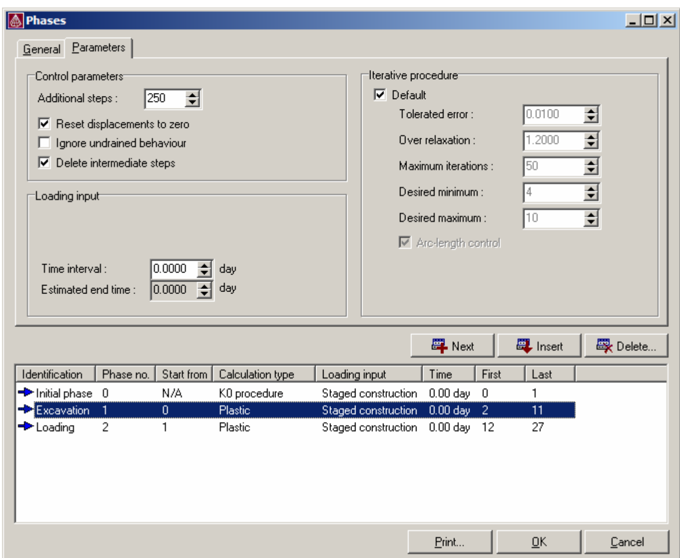


Figure 3.19 The Parameters tab sheet of the Phases window

- The *Parameters* tab sheet (see Figure 3.19) contains the calculation control parameters. Keep the default settings in the *Iterative procedure* box and keep the number of *Additional steps* to 250.

- The calculation parameters for the first phase have now been set. Click <OK> to close the phases window.
- Back in the main window of the Input program, the *Phase list* combo box will show ‘Excavation’. The draw area can now be used to define the construction stages for the current project. To do this, follow these steps:
- Select the work plane at y = 0.0 m from the *Active work plane* combo-box.
- Click on the geometry line corresponding to the basement wall. This should activate the basement wall. An active wall is coloured blue, an inactive wall is coloured grey.
- Also click on the soil cluster enclosed by the walls. A dialog will appear, including a tick box, which allows you to switch off the *Soil below*. Click on the tick box to switch off the cluster. It is not necessary to switch off the water, since the water starts at the level y = –2.0 m, which is just below the excavation. Click <OK> to close the dialog. The cluster will now be coloured white, to indicate it is switched off.
- Now select the work plane at y = -2.0 m.
- Click on the floor element to activate it. A dialog will appear, including a set of check boxes (see Figure 3.20). Click on the check box in front of *Floor* to switch on the floor. Click <OK> to close the dialog box. The floor should be coloured dark green (olive), indicating it is active.

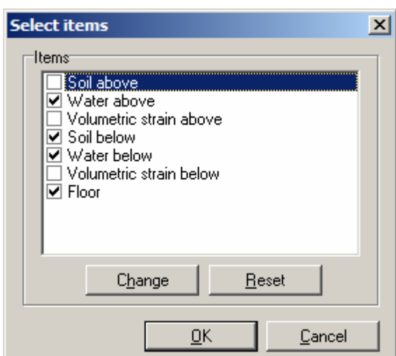


Figure 3.20 The Select items window with the floor element activated

This completes the calculation definition for the first phase. To define the second calculation stage, follow these steps:

- Click the *Next phase* button to enter a second calculation phase. The phases window will open and a new phase is added, named <Phase 2> by default. Name this phase ‘Loading’ and make sure this phase starts from *Phase 1* ‘Excavation’

- In the *Parameters* tab sheet, make sure that the option *Reset displacements to zero* is ***not*** checked.
- Keep the default parameters for the other options and close the Phases window .
- Make sure *Phase 2* ‘Loading’ is selected in the *Phase list*.
- Make sure the work plane at y = -2.0 m is still selected.
- Double click the grey circle indicating the inactive point load at (-10.0, -3.0). This will open a *Point load* window, where the magnitude and direction of the load can be entered. Enter load y = -6000 kN (6000 kN acting in the downward y-direction) and click <OK>. Make sure the load is active, indicated by a blue arrow. If necessary, the load can be activated or deactivated by clicking once on the load.
- Repeat this action for all point loads, so they all have a y-value of -6000 kN.

Click the *Preview* button to check the definition of the calculation stages. The preview option enables a direct visual check of the situation to be calculated before the calculation is started. The preview should show the excavated basement with walls and the activated loads acting on the basement floor. If the loads are not visible, the geometry can be rotated using the arrow keys on the keyboard. Also make sure the *Loads* option is selected from the *Geometry* menu.

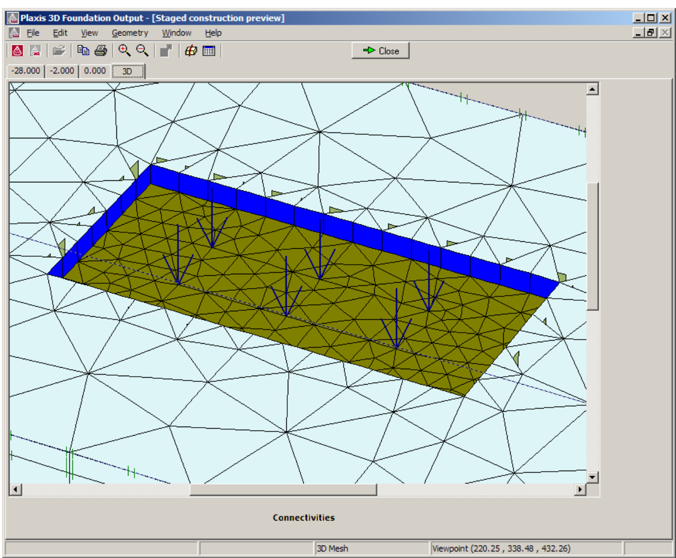
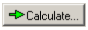


Figure 3.21 Preview of the geometry and the activated loads

- After the preview click the <Close> button to return to the main window. If the situation is unsatisfactory, click the <Close> button and correct the situation in the main window. The calculation definition is now fully complete. The calculation can now be started.



Click the *Calculate* button. This will start the calculation process.

All calculation phase that are marked for calculation, as indicated by a blue arrow (three phases in this case) will, in principle, be executed in the order controlled by the *Start from phase* parameter.

During the execution of a calculation a window appears which gives information about the progress of the actual calculation phase (see Figure 3.22). The information, which is continuously updated, shows amongst others the calculation progress, the current step number, the global error in the current iteration and the number of plastic points in the current calculation step. It will take several minutes to perform the calculation. When a calculation ends, the window is closed and focus is returned to the main window. Also, the *Phase list* is updated, showing a green tick mark to indicate that the calculation was finished successfully.

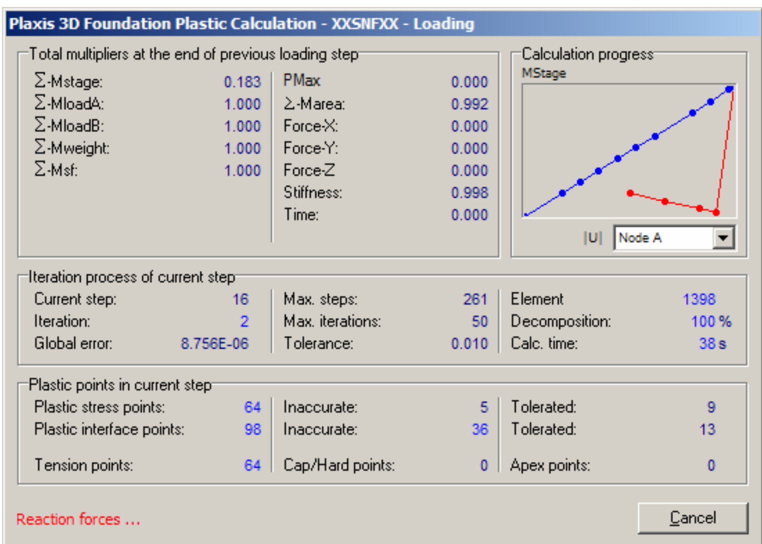


Figure 3.22 The calculations information window

Hint: Calculation phases may be added, inserted or deleted using the <Next>, <Insert> and <Delete> buttons in the Phase list 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 tick mark (√) whereas an unsuccessful calculation is indicated with a red cross (×). Calculation phases that are selected for execution are indicated by a blue arrow (→).

3.4 VIEWING OUTPUT RESULTS

Once the calculation has been completed, the results can be evaluated in the *Output* program. In the *Output* program the displacement and stresses in the full 3D model as well as in the individual work planes or structural elements can be viewed. The computational results are also available in tabular form. To view the results for the current analysis, follow these steps:

Select the last calculation phase (Loading) in the *Phase list* combo box.

Note that the *Calculate* button has changed into an *Output* button. Click the *Output* button to open the Output program. The Output program will by default show the 3D deformed mesh at the end of the selected calculation phase. The deformations are scaled to ensure that they are clearly visible.

- 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 (see Figure 3.23). A legend is presented with the displacement values at the colour boundaries. In case the floor is shown this will partially hide the displacements of the soil. In order to obtain the plot shown below, switch off the floor by selecting the *Structures* option from the *Geometry* menu.

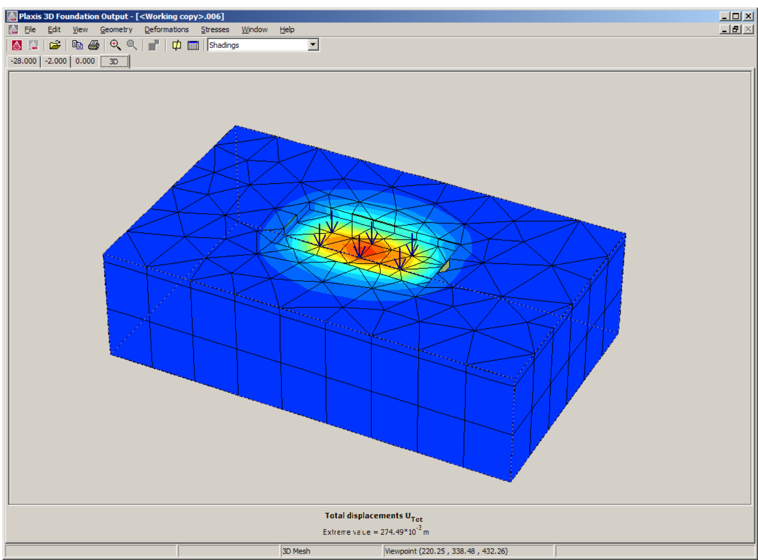


Figure 3.23 Shadings of total displacements

Hint: The arrow keys may be used to change the orientation of a 3D model on the screen. By default, the orientation is such that the positive x-direction is to the right, the positive y-direction is upwards and the positive z-direction is towards the user. The ← and → keys may be used to rotate the model around the y-axis whereas the ↑ and ↓ keys may be used to rotate the model in its current orientation around the horizontal screen axis.

- Select *Contour lines* from the presentation combo box in the toolbar. The plot shows contour lines of the total displacements. A legend is presented with the displacement values corresponding to the labels.

Apart from the entire 3D geometry, output can also be obtained at the work planes. The work planes can be selected using the tabs at the top of the output window. Each tab corresponding to a work plane is labeled by the *y*-coordinate of the work plane. The right-most tab, labeled 3D, will show the 3D geometry view.

- Select the middle work plane (y = -2.0) by clicking the second tab. The plot now shows contours of total displacements in the work plane with labels corresponding to the index.

Hint: In addition to the *Total displacements*, the *Deformations* menu allows for the presentation of *Incremental displacements* and *Phase displacements*. The incremental displacements are the displacements that occurred in one calculation step (in this case the final step). *Incremental displacements* may be helpful in visualising failure mechanisms. *Phase displacements* are the displacements that occurred in one calculation phase (in this case the last phase). *Phase displacements* can be used to inspect the impact of a single construction phase, without the need to reset displacements before starting the phase.

- When the middle plane is active, double click on the floor plate. This will open a new window, showing the deformed plane of the floor. Select *Bending moments M11* from the *Forces* menu to show the bending moments over the longest direction in the floor plate. If the floor is not visible, it cannot be selected. In this case, select the *Structures* option from the *Geometry* menu.

To view the bending moments in tabulated form, click the Table button. A new window is opened in which a table is presented, showing the values of bending moments in each node of the floor.

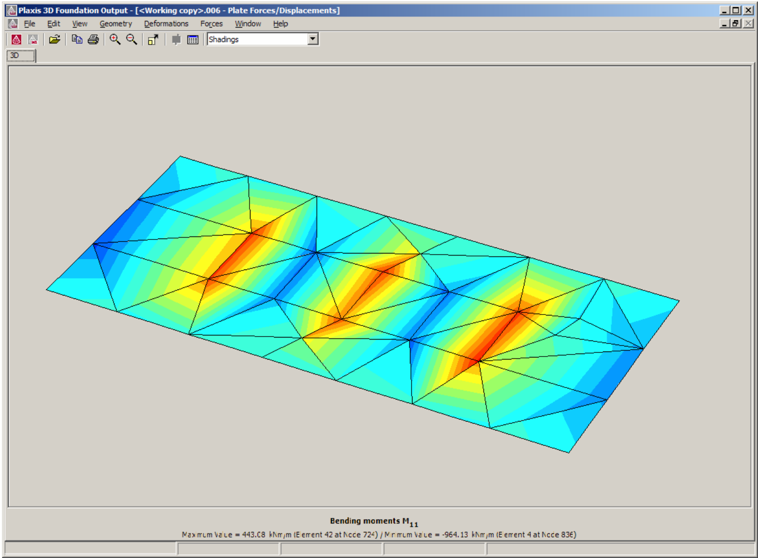


Figure 3.24 Bending moments in the floor plate

4 LOAD CAPACITY OF A BORED PILE (LESSON 2)

In this lesson a load test on a bored pile will be simulated. During a full scale field test in Brazil, a total of 6 piles were tested (das Neves, 2001)†. The diameter of the piles ranged from 35 to 50 cm and they were loaded in compression as well as tension. The piles were monitored using extensometers as well as load cells.

One of the piles was a 40 cm diameter pile with a length of 10 m, which was ultimately loaded to failure in compression. It has been constructed in silty and clayey sands, which can be divided in several layers, as sketched in Figure 4.1. The water table is located just below the foot of the pile.

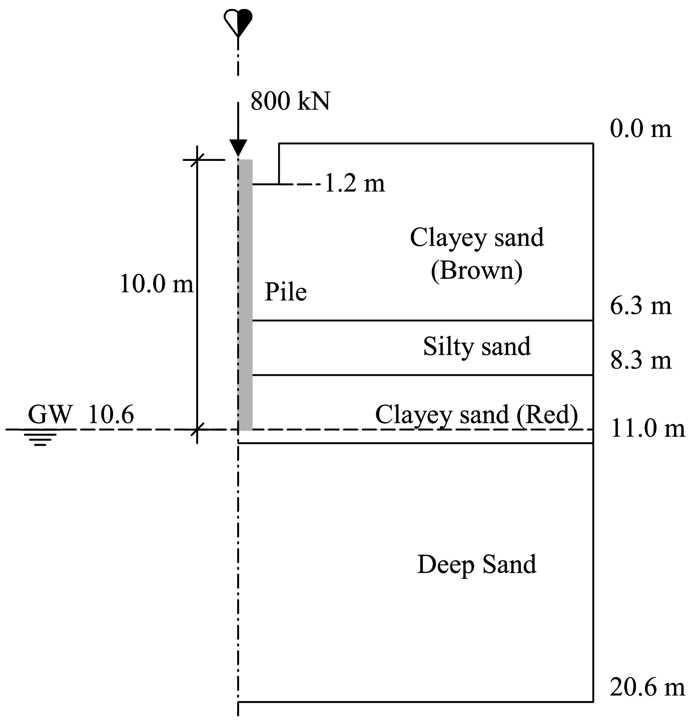


Figure 4.1 Cross section of the test site

The purpose of the analysis is to demonstrate the general set-up of a 3D model including piles modelled as volumetric elements. In order to restrict the calculation time needed, the model is rather coarse, but it will show the load-settlement behaviour of a bored pile. The mesh is too coarse however, to reliably predict the bearing capacity of the pile.

† M. das Neves et al. (2001) Étude du comportement de pieux forés. Bulletin des Laboratoires de Ponts et Chaussées (231), pp. 39-54 & 55-67

4.1 GEOMETRY

The pile is placed at the centre of a small excavation with a depth of 1.20 m. The top of the pile extends 60 cm above the bottom of the excavation. The subsoil can be divided in four layers. The water table coincides with the top of the deep sand layer, which starts slightly below the pile toe. In order to avoid any influence of the boundaries, the model is extended 10 m below the pile toe and 10 m to all sides.

To create the geometry and finite element mesh, follow these steps:

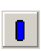
General settings

- 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 for the project and keep the other settings to their default.
- In the *Dimensions* tab sheet, keep the standard units (*Length* = m, *Force* = kN, *Time* = day) and enter for the dimensions *Xmin* = -10, *Xmax* = 10, *Zmin* = -10 and *Zmax* = 10. In the *Grid* box enter *Spacing* = 1 m and *Number of intervals* = 2.
- Click the <OK> button and the draw area appears.

Work planes

- Open the *Work planes* window. Insert 4 new work planes. In addition to the default work plane at y = 0.0 m, assign values of y = 10.0, 19.4, 20.0 and 20.6 m to work planes. To do this, click in the table of work plane levels and type the desired value. You may note that once you press <Enter> or click on another work plane, the list of values is automatically sorted.
- Click <OK> to close the work plane window.

Pile geometry



Make sure the work plane y = 20.6 m is selected. Select the *Pile* tool and the *Pile* designer will appear (see Figure 4.2). Select a *Massive Circular pile* for the *Pile type*.

- From the combo box, select a *Massive circular pile* and set the *Diameter* to 0.4 m.

A pile is composed of different *Sections*. A section can be a circular *Arc* or a straight *Line*. In a *User-defined pile* both Arcs and Lines can be mixed together. Using the left and right arrows at the bottom right of the *Section* box, one can switch between the different sections. Alternatively, clicking on a section in the preview of the pile will select the clicked section.

- Leave the *Angle* for each section of the pile to the default value of 60 º.
- Make sure that the pile has an interface along the entire circumference by selecting *Outside interface.*
- Click <OK> to close the *Pile designer*.
- The cursor is now shaped as a pile to indicate that the pile is about to be placed in the geometry. The position of the pointer corresponds with the axis of the pile. Move the cursor to (0,0) and click once. The pile is now placed in the geometry between the active work plane and the work plane just below it.
- Change the active work plane to y = 20.0 and click on (0,0) once more to add the pile also between this work plane and the work plane below it. Similarly, add the pile to the work plane y = 19.4.
- Right click to end adding piles to the geometry.

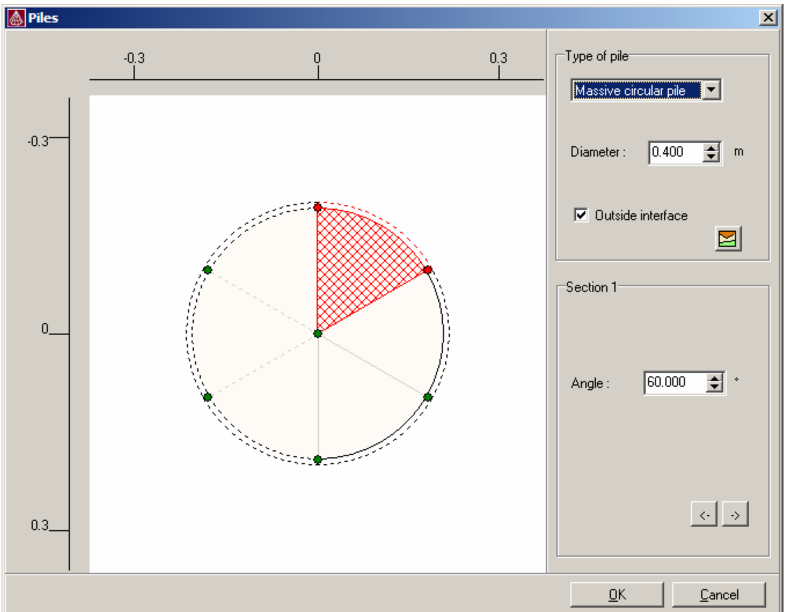


Figure 4.2 Pile designer

Hint:	The point where the pile is inserted in the work plane is called the Pile reference point. An existing pile may be edited by double clicking the pile reference point. As a result, the pile designer appears in which the existing pile is presented.

Excavation around the pile cap

During the pile load test, a small excavation is used to make the pile cap more accessible. To model this excavation, an additional cluster must be defined. To draw this cluster, first zoom in around the pile.

Select Zoom in from the toolbar. Click on (-2.0, -2.0) and hold the mouse button down. Now drag the mouse to (2.0, 2.0) and release the mouse button. This will zoom to the area directly around the pile.

Select the *Geometry line* tool.

- Draw a square cluster around the pile with coordinates (-1.0, -1.0) – (-1.0, 1.0) – (1.0, 1.0) – (1.0, -1.0) and click on (-1.0, -1.0) again to close the cluster. Right-click to end drawing new geometry lines.

Load

In order to simulate the load test, a point load must be added on top of the pile. As the pile is constructed to a level slightly below the soil surface, follow these steps to add the load to the correct level:

- First change the *Active work plane* to y = 20.0 m.
- Add an additional geometry line from (0, 0) to (-1.0, 0.0)

Select the *Point load* tool and add a load at the centre of the pile. Click once on (0, 0) to add a point load.

Borehole and material properties

In order to define the soil layers, a borehole needs to be added and material properties must be assigned.

Click the *Borehole* tool button and add a borehole to (-5.0, 0.0)

- The soil consists of four separate layers. It is therefore necessary to add 3 extra layer boundaries. Select the upper layer boundary (y = 0 m) and click *Insert* to add 3 extra layer boundaries.
- Click in the table in the column Y [m] on the y-coordinate of the top layer boundary. Enter a value of 20.6 m for this layer boundary.
- In the same manner, change the other layer boundaries to 14.30 m, 12.30 m, 9.6 m and 0.0 m. The borehole information is shown in Figure 4.3.
- Enter a value of y = 9.6 for the *Water level*.
- Click the material sets button and enter the properties for the layers as listed in Table 4.
- When entering the 'Pile' data set, click on the coloured rectangle in the lower left hand corner of the dialog, showing the current colour of the material set, to

open the colour dialog (see Figure 4.4). This dialog can be used to change the colour assigned to the material data set. Change the colour for the 'Pile' data set to grey by clicking on the indicated position in the coloured square at the lower left of the window.

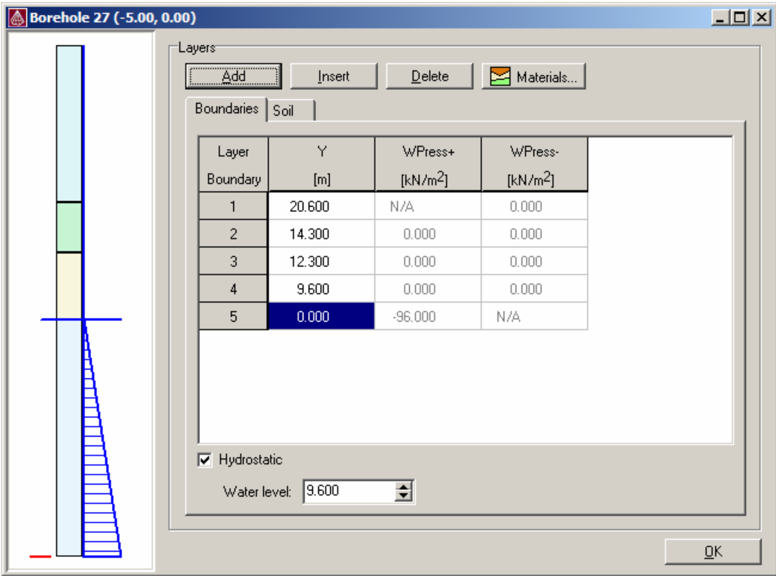


Figure 4.3 The Borehole window

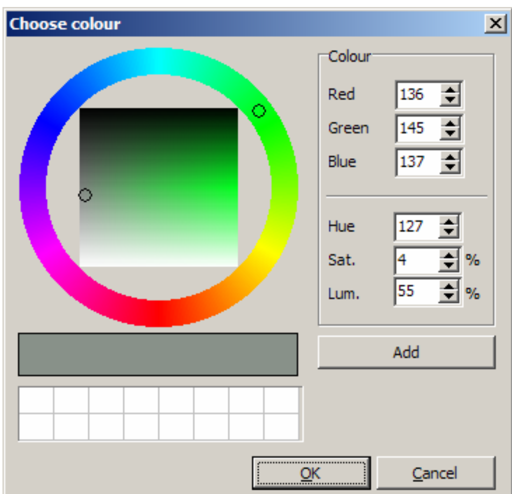


Figure 4.4 Changing colours of a material data set

Table 4 Material properties for the soil layers

Parameter	Name	Clayey Sand (Brown)	Silty Sand	Clayey Sand (Red)	Deep Sand	Pile	Unit
Material model	Model	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulumb	Linear-elastic	-
Material behaviour	Type	Drained	Drained	Drained	Drained	Non-porous	-
Unsaturated soil weight	γ_{unsat}	16.7	18.8	19.8	17.6	24	kN/m^3
Saturated soil weight	γ_{sat}	16.7	18.8	19.8	20.0	-	kN/m^3
Young's modulus	E	9150	13000	13500	19000	$29.2 \cdot 10^6$	kN/m^2
Poisson's constant	ν	0.3	0.2	0.3	0.3	0.15	-
Cohesion	c	13	12	14	17	-	kN/m^2
Friction angle	φ	26	23	23	23	-	°
Dilatancy angle	ψ	0	0	0	0	-	°
Interface reduction factor	R_{inter}	1	1	1	1	1	-
Lateral earth pressure coeff.	K_0	0.56	0.6	0.6	0.6	-	-

- Assign the material properties to the 4 soil layers by drag and drop in the borehole window. The Pile material set will be used later when defining the calculation stages.
- Switch to the *Soil* tab and enter the K_0 values for the respective soil layers according to Table 4.

2D Mesh generation

When a pile element is included in the geometry model, an automatic local refinement will be performed by the program around the pile contour. For this exercise, an additional local refinement is necessary. To generate an appropriate mesh, follow these steps:

Click the *Generate 2D mesh* button. A few seconds later the mesh is presented in the Output window. Inspect the mesh and click the <Update> button to return to the Input program.

- Back in the Input program, select the cluster around the pile, representing the excavation. From the *Mesh* menu, select *Refine cluster*. The Output window will re-appear, showing a mesh refinement around the pile. Return to the Input program and refine the excavation cluster once more.

3D Mesh generation

The generation of the 3D mesh is straightforward.

Click the *Generate 3D mesh* button. This will present the Output program once more, now showing a three-dimensional view of the generated mesh. Click the <Update> button to return to the geometry input mode.

- Click on the <Calculation> button to proceed to the calculation mode. Save the project under an appropriate name.

4.2 DEFINING CALCULATION STAGES

In the calculations, three stages will be considered. They are the generation of initial conditions, the construction of the pile and the loading of the pile. The calculation stage defining the initial conditions is already created automatically as a gravity loading phase. In order to change this phase to a K_0 procedure and generate initial stresses, follow these steps:

- Open the *Phases* window and change the calculation type for the initial phase to *K0 procedure* (see Figure 4.5).

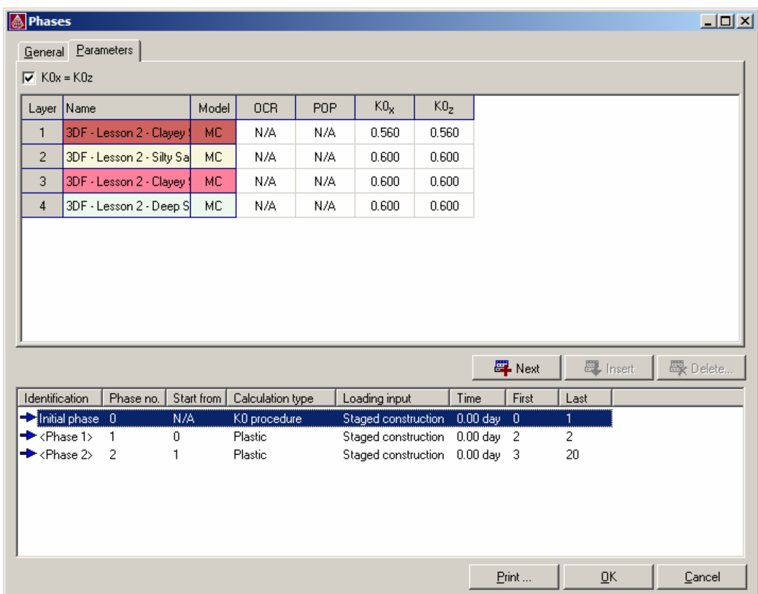


Figure 4.5 The *Phases* window with the *Parameters* tab sheet for the K_0 procedure

- On the Parameters tab, check the K_0 values given in Table 4 for the different layers.

To define the other two calculation stages, follow these steps:

Construction of the pile

- Click the <Next> button to add a new phase.
- Leave the settings for the *Iterative procedure* on their *Default* values.
- Click <OK> to close the window.
- Check that in the *Phase list* Phase 1 is selected.
- Select the work plane y = 20.0 m from the *Active work plane* list.

Click on the cluster representing the pile. It may be necessary to zoom in on the pile in order to properly select the cluster representing the pile. Use the Zoom in button on the toolbar for this purpose. When the pile cluster is selected, a *Select items* dialog will be shown, listing *Soil above* and *Soil below* as possible options.

- Click on the checkbox in front of *Soil above* and remove the tick mark, in order to switch off the soil cluster above the active work plane.
- Click on the text *Soil below* and then click the *Change* button to change the material set for the pile cluster. From the *Material sets* window, select the Pile material set and click <OK>.

Hint:	Instead of clicking on the *Soil above* or *Soil below* items and then clicking on the <Change> button, it is also possible to double-click the text *Soil above* or *Soil below*. This will immediately open the *Material sets* window.

- Click <OK> to close the *Select items* window.
- Click on the cluster around the pile, which will be excavated during pile Construction. In order to simulate the excavation, switch off both the Soil above and the Soil below by removing the tick mark in the *Select* window.
- Click <OK> to return to the Input program.

The pile has now been assigned the properties of the pile only between the work plane y = 20.0 and the first work plane below it. That is the work plane with y = 19.4 m. In order to assign pile properties also to the remainder of the pile:

- Change the active work plane to y = 19.4 m.
- Click on the pile cluster and select the *Soil below*. Click on *Change* and assign the pile properties also to this cluster.

- Now close the *Material sets* window and the *Select items* window to return to the input program.

> **Hint:** To check whether the calculation stage has been defined correctly, click the *Preview* button. This will present a 3D view of the geometry. This can be used to check which clusters are active and what material properties have been assigned to clusters.

Pile loading

The first calculation phase has now been defined. In order to define the second calculation phase:

Click the *Next phase* button to add a new phase. On the *Parameters* tab sheet, check that the option *Reset displacements to zero* is **not** checked. Close the window (see Figure 4.6).

- Select the work plane y = 20.0 as the active work plane.
- Double click the point load at the centre of the pile to activate it. A window will open, allowing you to change the direction and magnitude of the force. Enter a force of -800 kN in the y-direction.
- Click <OK> to close the window and return to the input program.

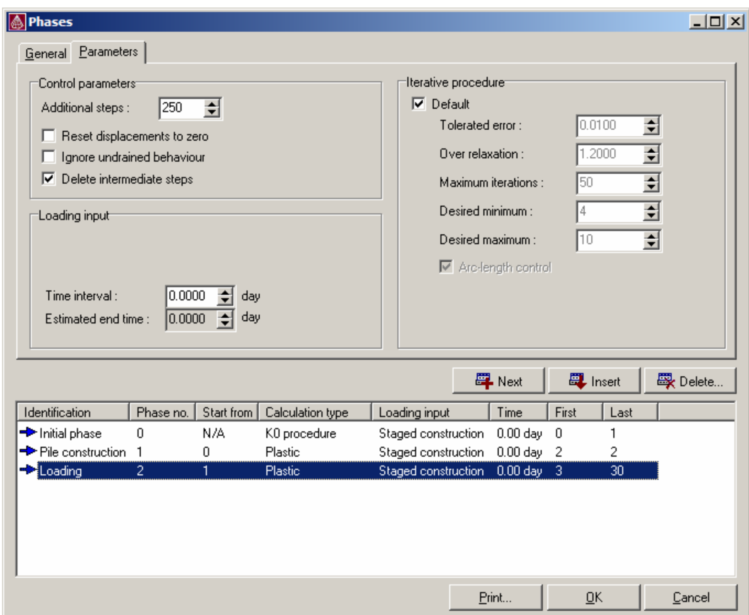


Figure 4.6 Iterative procedure settings window

This completes the definition of the calculation phases. Before starting the calculation, select the node at the top of the pile, in order to be able to plot the load displacement curve later on.

Click the *Select points for curves* button to open the Output program. All nodes present in the geometry are drawn. Select the node on top of the pile at (0, 0) in the work plane y = 20.0 m (see Figure 4.7). It may be necessary to zoom into the area around the pile in order to select the correct point. Close the output program to return to the input program.

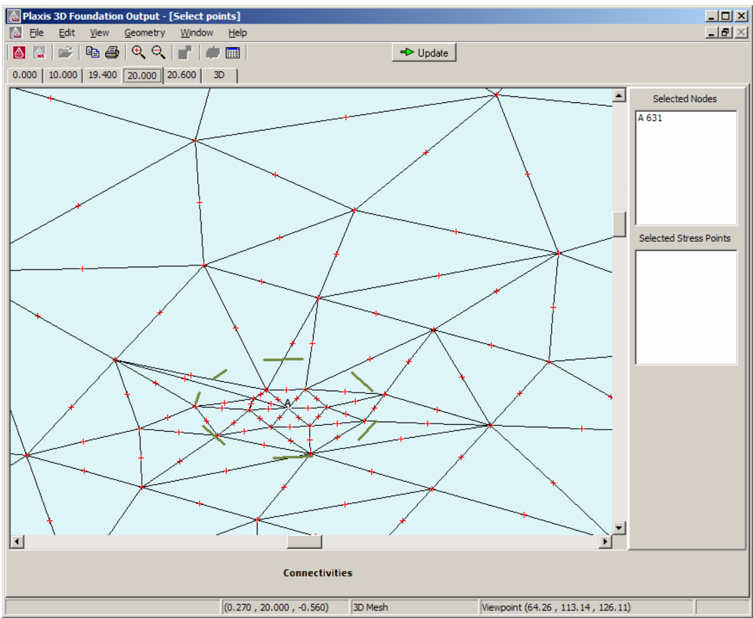
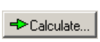


Figure 4.7 Select a point (A) on the pile for curves

4.3 CALCULATION

The geometry and calculation stages have now been defined and the calculation can be started.



Press the *Calculate* button to start the calculation.

The calculation process should now start. The program starts from the first calculation phase marked for calculation, which is the *Initial phase*. At the end of the calculation all phases should be indicated by a green tick mark, which indicates that they have been successfully calculated.

After the calculation, click the *Save* button to save the project. This is required for the creation of load-displacement curves.

4.4 VIEWING OUTPUT RESULTS

After the calculations, the results of the load test can be viewed by selecting a calculation phase from the Phase list and pressing the <Output> button. The Output button replaces the Calculate button after the calculation has finished.

- Select the final calculation phase and click the Output button. The Output program will open and show the deformed mesh at the end of the load test. The stresses and deformations in the work planes can be viewed by clicking on the corresponding tabs and selecting the desired output from the menu. For example select the work plane y = 10.0 to investigate the stresses around the pile tip (see Figure 4.8).

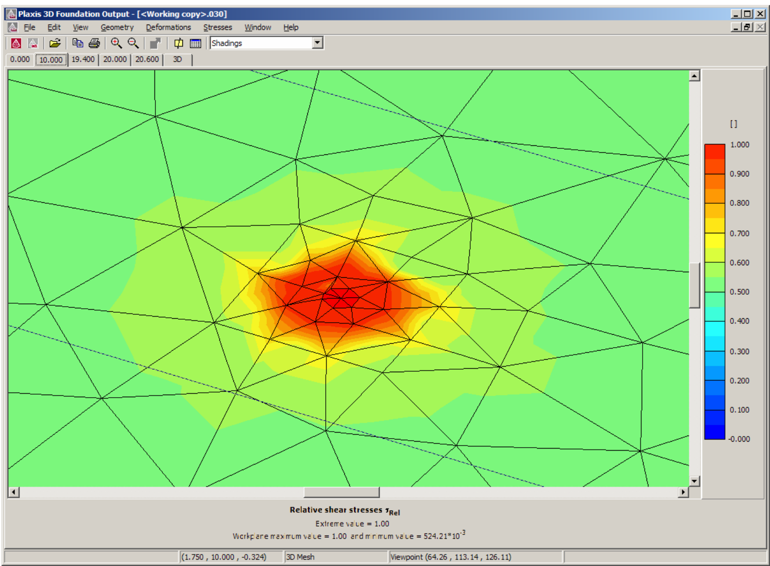


Figure 4.8 Relative shear stresses around the pile tip in work plane y = 10.0

In order to evaluate stresses and deformations inside the geometry, select the *Cross section* tool. A top view of the geometry is presented. For the load test a cross section through the pile is most interesting. Draw a cross section across the geometry through the origin. A vertical cross section is presented. This cross section can be rotated in the same way as a regular 3D view of the geometry. Select *Vertical displacements* from the *Deformations* menu (see Figure 4.9).

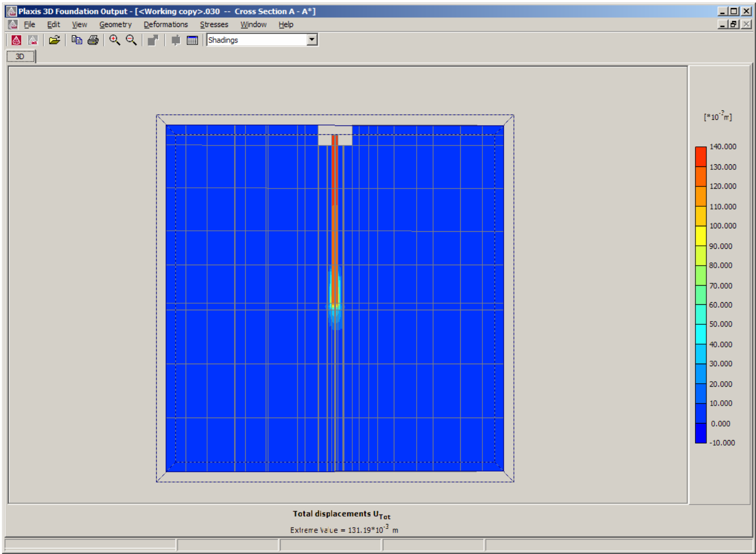


Figure 4.9 Cross section of vertical displacements

For the load test a load-displacement curve of the pile can be plotted in the Curves program. Follow these steps:

 Start the Curves program by clicking on the corresponding button on the upper left side of the toolbar.

- Click <OK> to create a New chart.
- Select the pile load project and press the <Open> button.
- In the Curve generation window, select for the x-axis the *Multiplier* option and From the *Type combo* box, select *Sum-Mstage*.
- For the y-axis, select *Displacement* and select the vertical displacement u_y as *Type* Check that the point at the top of the pile is selected in the Point combo box, this should be Point A (0.0 / 20.0 / 0.0).
- Click <OK> to generate the graph.

The graph will now show the vertical displacements against Σ-Mstage. Σ-Mstage is a parameter that signifies the amount of load unbalance in the current calculation phase already solved. It will increases from 0 at the beginning of a calculation phase to 1 at the end of the phase, if the calculation completes successfully. For the last phase, the value of Σ-Mstage corresponds to the amount of the applied point load of 800 kN actually being activated. To plot only this part of the load displacement curve, follow these steps:

- Right click in the graph and select *Format* from the pop-up menu. Select *Curve* from the submenu that appears. Alternatively select *Curve* from the *Format* menu.

- Click the Phases button on the top right of the *Curves settings* window to show the Select phases window.
- In the Select phases window, remove the tick mark before ‘Initial Phase’ and ‘Pile construction’. Click <OK> to close the window and click <OK> once more to update the graph.

The graph will now show only the displacement resulting from the loading of the pile (see Figure 4.10). The value Σ-Mstage = 1 in the graph corresponds to a load of 800 kN.

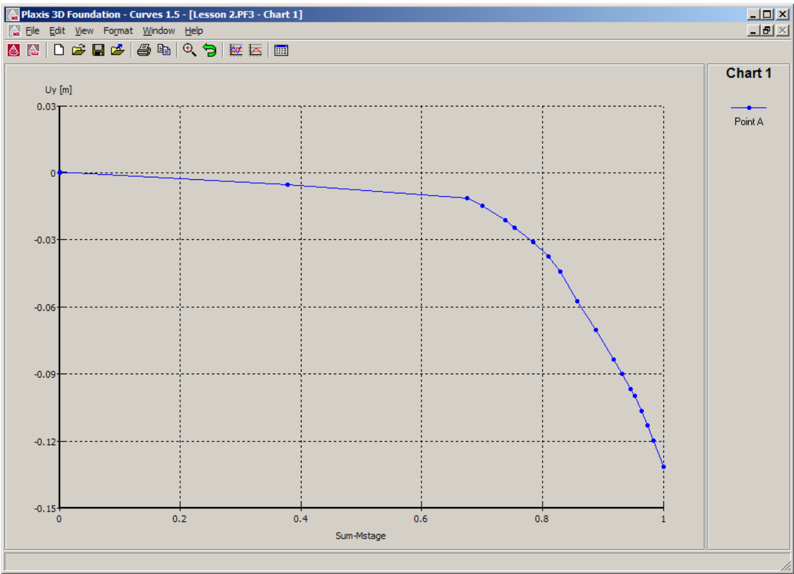


Figure 4.10 Load - displacement curve

5 A-SYMMETRIC RAFT FOUNDATION (LESSON 3)

This lesson describes the construction and subsequent loading of a T-shaped raft foundation on sand and soft clay layers. The geometry and dimensions of the raft foundation are presented in Figure 5.1. It consists of a 20 cm thick roughly T-shaped concrete floor on which 3 m high concrete walls have been cast.

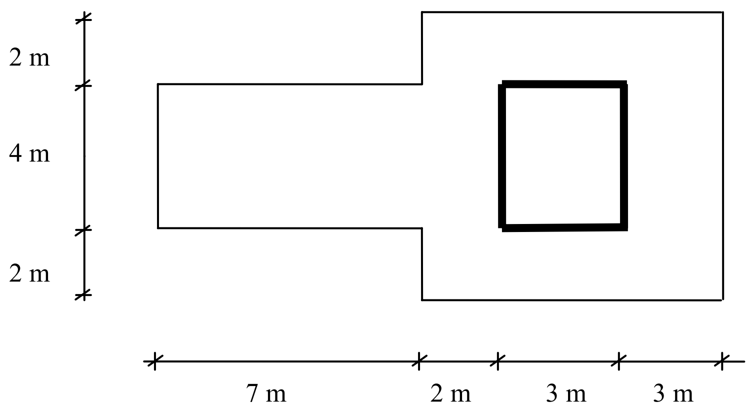


Figure 5.1 Geometry and dimensions of the T-shaped raft foundation

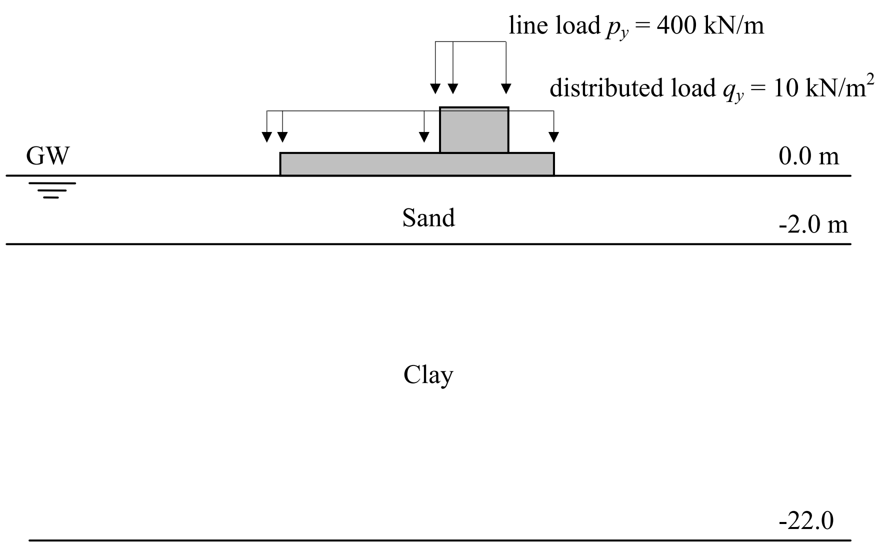


Figure 5.2 Cross-section of raft foundation: presentation of the soil layers

This raft foundation is constructed on a 2 m thick sand layer laying on top of a 20 m thick clay layer, as shown in Figure 5.2. Construction of the raft and wall is expected to take 30 days after which the structure is loaded with a distributed load q_y = -10 kN/m^2 distributed over the floor and a line load p_y = -400 kN/m acting along the top of the walls.

5.1 INPUT

The proposed geometry for this exercise is 50 m wide and 30 m long. The foundation is 15 m long and 8 m wide and is placed in the centre of the geometry.

General settings

To enter the geometry for this exercise, follow these steps:

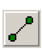
- Start the Input program and select *New project* from the *Create/Open project* dialog box.
- Enter an appropriate title for the exercise and set the model dimensions to x_{min} = 0 m, x_{max} = 50 m, z_{min} = 0 m z_{max} = 30 m. Keep the default settings for the units and the grid spacing.
- Click <OK> and the draw area should appear.

Work planes

- In addition to the predefined work plane at y = 0.0 m, add three work planes at respectively y = 3.0 m, y = -2.0 m and y = -22.0 m
- Click <OK> to close the *Work planes* window.

Geometry

In the work plane y = 0 m, we will first create the geometry of the concrete floor.



Select the *Geometry line* tool.

- Draw a cluster with the following corner points (22.0, 13.0), (22.0, 11.0), (30.0, 11.0), (30.0, 19.0), (22.0, 19.0), (22.0, 17.0), (15.0, 17.0), (15.0, 13.0). Click on (22.0, 13.0) again to close the cluster. Right-click to end drawing new geometry lines.

Select the *Floor* option from the geometry toolbar.

- Click anywhere within the area enclosed by the previously created lines. This area should now be coloured olive (green), indicating that a floor has been added.
- Change the work plane to y = 3 m.

Select the *Wall* option and add walls to the geometry with corner points at (24.0, 13.0), (27.0, 13.0), (27.0, 17.0), (24.0, 17.0). Once again right-click to end drawing new walls.

Table 5 Material properties for the soil layers

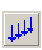
Parameter	Name	Sand	Clay	Unit
Top of Layer		0	-2	M
Material model	Model	MC	SSC	-
Type of material behaviour	Type	Drained	Undrained	-
Soil weight above phreatic level	γ_{unsat}	17	17	kN/m^3
Soil weight below phreatic level	γ_{sat}	19	18	kN/m^3
Permeability	k_x, k_y, k_z	1	$5 \cdot 10^{-5}$	m/day
Young's modulus	E_{ref}	$1.6 \cdot 10^4$	-	kN/m^2
Poisson's ratio	ν	0.3	0.15	-
Modified compression index	λ^*	-	0.012	-
Modified swelling index	κ^*	-	$2.5 \cdot 10^{-3}$	-
Modified creep index	μ^*	-	$6 \cdot 10^{-4}$	-
Cohesion	c_{ref}	1	6	kN/m^2
Friction angle	φ	32	26	°
Dilatancy angle	ψ	2	0	°
Interface reduction factor	R_{inter}	1	1	-
Lateral earth pressure coeff.	K_0	0.47	0.634	-
Overconsolidation ratio	*OCR*	-	1.5	-

Loads

After the geometry of structural elements has been defined, the loads acting on both the floor and the walls have to be defined:

Select the *Distributed load (horizontal planes)* option from the toolbar.

- Add a load to the two clusters representing the concrete floor at work plane $y = 0$ m.



Select the *Line load (horizontal)* option.

- Add a line load at the location of the 4 walls in the work plane at $y = 3$ m. By default the line load has a value of 1 kN/m and is acting perpendicularly to the walls. Leave these at their default value for now.

Borehole

As all soil layers for this exercise are horizontal, only a single borehole is needed (see Figure 5.3).

- Add a borehole to the geometry and insert 1 additional layer boundary. Enter the layer boundary levels according to Figure 5.2.
- Set the water level to y = -2.0 m.
- Open the *Material properties* window. Create the two data sets given in Table 5 and assign them to their respective soil layers.
- Enter the corresponding K_0 values and the overconsolidation ratio *OCR* in the *Soil* tab sheet of the *Borehole* window.
- Click <OK> to close the borehole.

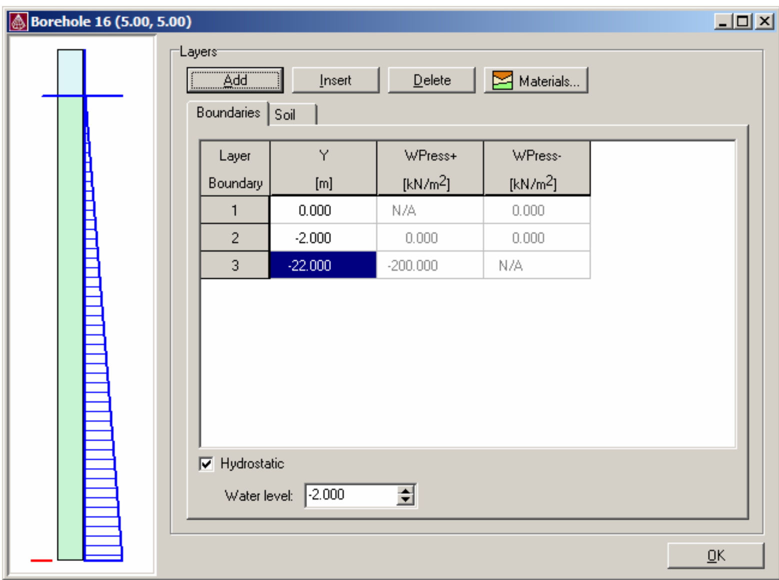


Figure 5.3 The *Borehole* Window

Floor and walls material properties

To enter the floor and walls material properties, follow these steps:

Open the material sets window and select *Floor* as the material type.

- Click <New> to create a new material data set.
- Name the data set Floor and enter the material parameters given in Table 6.

- Select the work plane y = 0.0 m and drag the Floor data set to the clusters representing the concrete floor and drop it there. It is necessary to repeat the operation twice in order to assign the data set to the entire floor.
- Change the material set *Type* to Walls.
- Click <New> to create another new material data set.
- Name the data set Walls and enter the material parameters given in Table 7.
- Change the work plane to y = 3 m and assign the Walls data set to the four wall elements visible in this work plane.

Table 6 Material properties for the floor

Parameter	Name	Value	Unit
Material model	*Model*	Linear, isotropic	-
Thickness	d	0.20	m
Volumetric weight	γ	24	kN/m^3
Young's modulus	E_i	$3.0 \cdot 10^7$	kN/m^2
Shear modulus	G_{ij}	$1.304 \cdot 10^7$	kN/m^2
Poisson's ratio	ν_{ij}	0.15	-

Table 7 Material properties for the walls

Parameter	Name	Value	Unit
Material model	*Model*	Linear, isotropic	-
Thickness	d	0.3	m
Volumetric weight	γ	24.0	kN/m^3
Young's modulus	E_i	$3.0 \cdot 10^7$	kN/m^2
Shear modulus	G_{ij}	$1.304 \cdot 10^7$	kN/m^2
Poisson's ratio	ν_{ij}	0.15	-

Mesh generation

The mesh for this example will be of medium coarseness, with a further refinement of the floors clusters. To generate this mesh, follow these steps:

- From the *Mesh* menu, select *Global settings* and set the element distribution to *Medium*. Select the two clusters of the concrete floor. Hold down the <Shift> key to select multiple clusters. Choose *Refine cluster* from the *Mesh* menu and the Output program will show the generated mesh.
- Return to the main draw area and generate the 3D mesh (see Figure 5.4). Click <Update> to return to the main draw area.

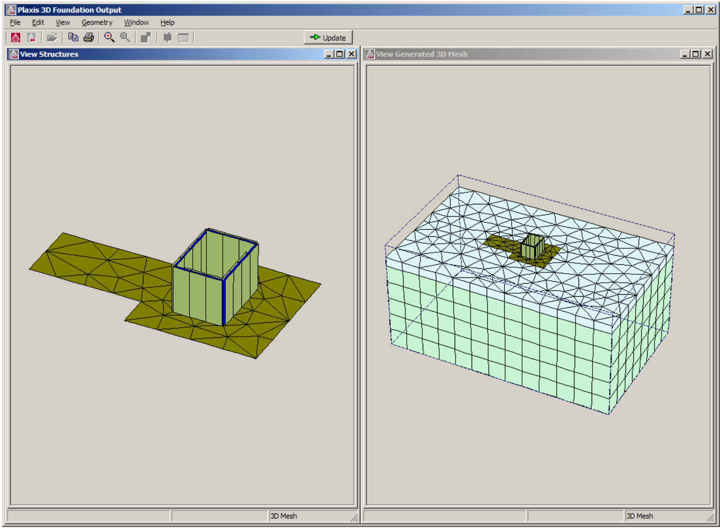


Figure 5.4 Preview of the generated 3D mesh

5.2 CALCULATIONS

The calculation consists of 4 phases. The initial phase will consist of the generation of initial stresses using the *K0 procedure*. The following phases will consist of the installation of the floor and walls (which is assumed to last 30 days) and the subsequent loading of the structural elements. The fourth phase will consider the consolidation of the soil layers until all excess pore pressures are dissipated.

- Open the *Phases* window and change the *Calculation type* for the first phase to *K0 procedure* and check the K_0 and *OCR* values as given in Table 5.
- Add a new *Plastic* calculation phase and enter a time period of 30 days in the *Time interval* edit box of the *Parameters* tab sheet.
- Close the *Phases* window and activate all walls along with the complete concrete floor in this phase. This second phase will simulate the casting of the structural elements over a time period of 30 days.
- Add a third phase and set the time period to 1 day.
- In the work plane y = 3 m, click the line load on one of the 4 wall elements. Activate the *Line load* in the *Select items* window that appears and click the *Change* button.
- In the *Line load* window, set the y-values to –400 and all x- and z-values to 0. This indicates a load of 400 kN/m, acting in the negative y-direction. Repeat the operation for each individual wall.

- In the work plane y = 0 m, click the distributed load. Select *Distributed load* from the *Select items* window that appears and click the *Change* button.
- In the *Distributed load* window, set the y-values to –10 and all x- and z-values to 0. This indicates a load of 10 kN/m^2, acting in the negative y-direction.
- Add a fourth calculation phase. Change the calculation type to *Consolidation* and set the L*oading type* to *Minimal excess pore pressure*. Set |P-stop| to 1 kN/m^2.
- Click <OK> to close the *Phases* window.

This completes the definition of the calculation phases. Before starting the calculation, select the node at one of the floor vertices in order to be able to plot the load displacement curve later on.

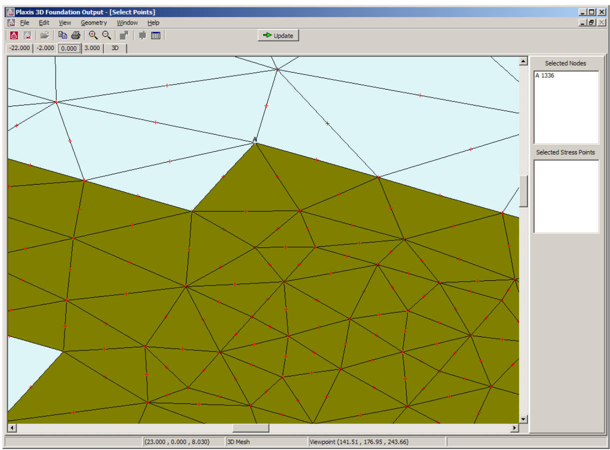


Figure 5.5 Location of selected corner point for curves

Click the *Select points for curves* button to open the Output program. All nodes of present in the geometry are drawn. Select the node on the corner of the floor at (22, 11) in the work plane y = 0.0 m. It may be necessary to zoom into the area around the raft in order to select the correct point (see Figure 5.5).

- Close the output program to return to the input program in order to start the calculation.
- Click on *Calculate* to run the calculation. After the calculation save the project

5.3 OUTPUT

The deformation of the soil after construction and loading of the structure, and before consolidation, can be investigated as follows:

- Select the phase in which the loads are applied, named <Phase 2> by default.

- Click <Output> to access results of this phase in the Output program (see Figure 5.7).

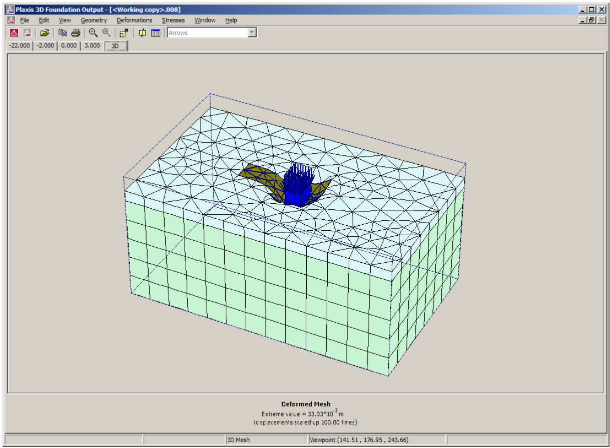


Figure 5.6 Deformed mesh after construction and loading of structural elements

It is also possible to assess the bending moments within the concrete floor at this stage. To display these results:

- Select the work plane y = 0.0 and double-click on the floor,
- In the Force menu, select the *Bending Moments M11* output item
- Select *Shadings* option in the combo box of the *Menu* bar (Figure 5.7).

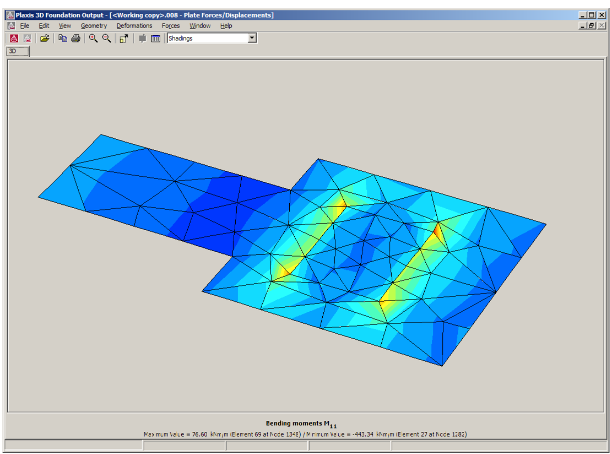


Figure 5.7 Bending moments in the concrete floor at the end of <Phase 2>

- Start the Curves program by clicking on the corresponding button on the upper left side of the toolbar.

- Click <OK> to create a *New chart*.
- Select the Lesson 3 project and press the <Open> button.
- In the Curve generation window (Figure 5.8), select for the *x*-axis the *Time* option.

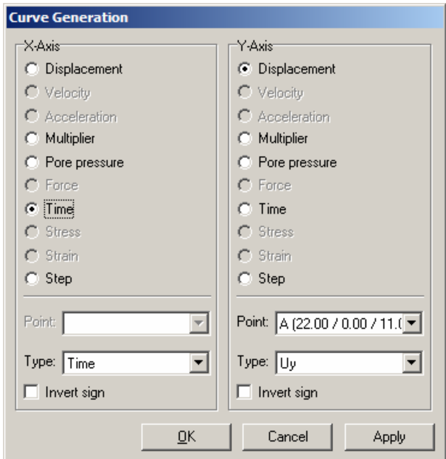


Figure 5.8 Setting up the time-displacement curve

- For the *y*-axis, select *Displacement* and select the vertical displacement u_y as *Type*. Check that the point at corner of the floor is selected in the *Point* combo box, this should be Point A (22.0 / 0.0 / 11.0).
- Click <OK> to generate the graph (Figure 5.9).

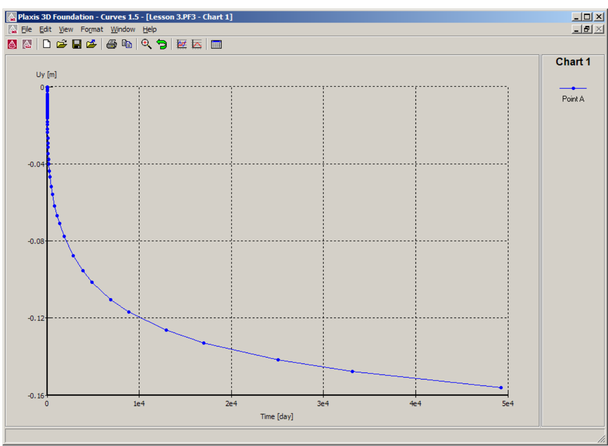


Figure 5.9 Time - displacement curve over calculation phases

6 LOAD CAPACITY OF A SUCTION PILE (LESSON 4)

In this lesson the load capacity of an off-shore foundation will be considered. A suction pile is a hollow steel pile of large diameter, with a closed top, that is installed in the seabed by pumping water from the inside. The resulting pressure difference between the outside and the inside is the driving force that installs the pile in the soil. The installation process itself will not be modelled. This exercise will investigate the load capacity of the anchor after installation. Three different angles of the pull out force will be considered. The geometry for the problem is sketched in Figure 6.1.

Many of the features used in the previous two exercises will be used here again. In addition, this exercise will show how multiple boreholes with different soil profiles can be used to model an inclined soil layer. Also, the use of structural elements above the soil surface is illustrated in this exercise. These new features will be described in full detail, whereas the features that were treated in the previous exercises will be described in less detail. Therefore it is suggested to complete the previous exercises before attempting this exercise.

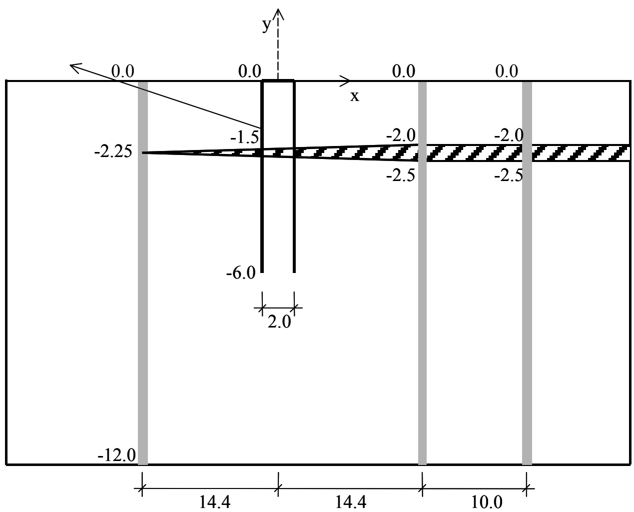


Figure 6.1 Geometry of the suction pile (not to scale)

6.1 GEOMETRY

The suction pile is a hollow steel tube, which is 6 m long and 2 m in diameter. The anchor line is attached on the side of the pile, 1.5 m from the top. It has been installed in a seabed, consisting of medium dense sand and a local silty layer. The local water depth is 50 m. An area 40 m wide and 40 m long surrounding the suction pile will be modelled. With these dimensions the model is sufficiently large to allow for any possible collapse mechanism to develop and to avoid any influence from the model boundaries.

The subsoil consists of three layers: an upper sand layer, the silt layer and a deep sand layer. The upper sand layer is approximately 2 m thick. The silt layer varies between 0 and 0.5 m locally and the sand layer below that extends to a large depth. Only the top 10 m of the deep sand layer will be included in the model. Hence the bottom of the model will be about 12 m below the seabed.

The installation of the suction pile will be modelled in a single calculation phase. The interaction between the steel pile and the soil will be modelled using an interface, allowing for the reduced friction between the steel surface and the soil. The pull out force from the anchor line is modelled by means of a point load on the suction pile surface, which is applied after the installation phase.

General settings

To enter the geometry for this exercise, follow these steps:

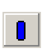
- Start the Input program and select *New project* from the *Create/Open project* dialog box.
- Enter an appropriate title for the exercise and set the model dimensions to x_{min} = 0 m, x_{max} = 40 m, z_{min} =0 m z_{max} = 40 m. Keep the default settings for the units and the grid spacing.
- Click <OK> and the draw area should appear.

Work planes and pile geometry

First create a work plane equal to the bottom level of the pile. To do this, follow these steps:


Select work planes from the toolbar and *Add* a work plane. Now select the lowest work plane in the list and change the *y*-level to -6.0 m.

- Close the work plane dialog.
- Select the work plane at y = 0.0 m.


Click the *Pile* button on the toolbar to open the *Piles* dialog.

- Accept the default setting for the pile type: *Circular tube* in order to create a hollow steel pile.
- Leave the thickness of the pile to 0.0, as the steel pile will be modelled as a shell element only. Change the diameter to 2 m.
- Make sure the options *Shell*, *Outside interface* and *Inside interface* are all checked.

- Select *Section 1* of the pile and change the *Angle* to 45°. Proceed to *Section 2* and also change the angle to 45°. Proceed to *Section 3*. Note that the *Angle* has automatically increased from the default angle of 60° to 90° as a result of the changes in sections 1 and 2. Change the angle for section 3 also to 45°. A new section is automatically added to complete the circle.
- Now all 4 (8) sections should have a 45° angle, and should all have a shell, an inside interface and an outside interface present. Click <OK> to close the pile designer.
- Select the work plane at y = 0.0 m. Move the cursor to (20.0, 20.0) and click once to place the pile in the geometry model.

Anchor Load

To introduce the anchor load 1.5 m below the top of the suction pile, another work plane is needed. To add the load, follow these steps:

- Open the work plane dialog and select the lowest work plane (y = -6.0 m)
- Click <Insert> to add a new work plane. A new work plane will appear at y = -3.0 m.
- Click on the work plane level and change it to y = -1.5 m.
- Close the work plane dialog.

Hint: The work plane at y = -1.5 m was not added earlier, as this would complicate adding the pile. A pile is always added between the current work plane and the one directly below it. Were the work plane at y = -1.5 already present when we added the pile, the pile would have been only 1.5 m high, instead of the desired 6 m, and a second pile should have been added between –1.5 and –6.0 m.

- Select the work plane at y = -1.5 m. Select point load on the tool bar and add a point load at (19.0, 20.0), the leftmost node of the pile.

Boreholes

To create the soil layers, three boreholes are needed. In order to add these boreholes, follow these steps:

Select the *Borehole* tool and add a borehole at (30.0, 10.0). The borehole window will automatically open. Insert 2 new layers and change their y-levels to –2.0 m and –2.5 m respectively. Change the bottom of the lowest layer boundary to y = -12 m. The correct values for the layer boundaries are given in Table 9.

Open the *Material* window and select *Soil & interfaces* as the *Set type*. Click the <New> button to create a new data set.

- For the sand layers, enter Sand for the *Identification* and select *Mohr-Coulomb* as the *Material* model. Enter the properties as listed in Table 8 and click <OK> to close the data set.
- Click <New> and create a second data set, named Silt. Enter the material properties from Table 8 and press <OK> to close the data set.
- Drag the Sand data set to the top and bottom soil layers in the borehole and drop it there. Assign the Silt data set to the middle layer. Interfaces are automatically assigned the data set of the corresponding cluster.
- Click <OK> to close the material sets dialog.

It is not necessary to enter K_0 values, as the initial stresses will be generated by means of gravity loading.

- Change the water level to 50.0 m and click <OK> to close the borehole window.
- While the borehole tool is still active, as indicated by the cursor, also add boreholes to (10.0, 30.0) and (40.0, 10.0).

Table 8 Material properties of the sand and silt layers and their interfaces.

Parameter	Name	Sand	Silt	Unit
Material model	Model	Mohr-Coulomb	Mohr-Coulomb	-
Type of material behaviour	Type	Drained	Undrained	-
Soil weight above phr. level	γ_{unsat}	17	16	kN/m^3
Soil weight below phr. level	γ_{sat}	20	18	kN/m^3
Young's modulus (constant)	E_{ref}	$1.5 \cdot 10^4$	$9 \cdot 10^3$	kN/m^2
Poisson's ratio	ν	0.3	0.35	-
Cohesion (constant)	c_{ref}	1.0	2.0	kN/m^2
Friction angle	φ	32.0	27.0	°
Dilatancy angle	ψ	2.0	0.0	°
Interface strength reduction	R_{inter}	0.7	0.9	-

Table 9 Y-coordinates for the layer boundary of the three boreholes

	Borehole location		
Layer boundary	(30.0, 10.0)	(10.0, 30.0)	(40.0, 10.0)
1	0	0	0
2	-2.0	-2.25	-2.0
3	-2.5	-2.25	-2.5
4	-12	-12	-12

In order to set the *y*-level of the soil layers in the boreholes at (10.0, 30.0) and (40.0, 10.0) to the correct values, follow these steps:

- Select the *Select* tool and double click the borehole at (10.0, 30.0). A dialog will open, asking whether you want to edit the geometry point or the borehole. Select the borehole and click <OK>.
- Change the *y*-level of layer boundary 2 to y = -2.25 m. Also change the *y*-level of layer boundary 3 to y = -2.25. As a result, the silt layer will have a zero thickness in this borehole.
- Click <OK> to close the borehole.
- Open the borehole at (40.0, 10.0) to check if the layer boundaries are correct. Double click the borehole and select borehole from the select dialog. The layer boundaries should be the same as in the first borehole created. Close the borehole.

This concludes the input of the soil layers.

To model the interaction between the soil and the suction pile, an interface (shown as a dotted line) was created inside and outside the suction pile wall. The interface is modelled in the mesh by *interface* elements.

The interface properties are defined by the parameter R_{inter} that can be set in the *Interfaces* tab sheet of a soil material set. When the *Interfaces* tab sheet is skipped, the R_{inter} parameter will have a default value of 1.0 (rigid). The R_{inter} parameter relates the strength of the interfaces to the strength of the soil, according to the equations:

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

With the default value R_{inter} = 1.0 (rigid), $c_{inter} = c_{soil}$ and $\varphi_{inter} = \varphi_{soil}$.

In general, strength properties in the interaction zone between soil and structures are lower than the adjacent soil. This reduction can be specified using the R_{inter} parameter. Hence, using R_{inter} < 1.0 gives a reduced interface friction and adhesion compared to the friction angle and the cohesion in the adjacent soil.

Suction pile material properties

To enter material properties for the suction pile, follow these steps:

Open the material sets window and select *Walls* as the material type.

- Click <New> to create a new material data set.
- Name the data set Suction pile and enter the material parameters given in Table 10.
- Select the work plane y = 0.0 and drag the Suction pile data set to the blue circle representing the suction pile and drop it there. It may be necessary to zoom in around the pile to drop the data set at the correct position.
- Change the work plane to y = -1.5 m and also assign the Suction pile data set to the wall elements visible in this work plane.

Table 10 Material properties for the suction pile

Parameter	Name	Value	Unit
Thickness	d	0.15	m
Volumetric weight	γ	78.5	kN/m^3
Young's modulus	E_i	$2.1 \cdot 10^8$	kN/m^2
Shear modulus	G_{ij}	$9.545 \cdot 10^7$	kN/m^2
Poisson's ratio	ν_{ij}	0.1	-

2D Mesh Generation

In order to limit the mesh refinement during mesh generation to the area immediately surrounding the suction pile, an additional soil cluster will be created. In order to do so, follow these steps:

- Select *Geometry line* from the toolbar.
- Draw a cluster with corner points (15.0, 15.0), (25.0, 15.0), (25.0, 25.0) and (15.0, 25.0). Close the cluster by clicking on the first corner and end drawing the cluster.

This completes the geometry definition and material parameters input. In order to generate the mesh:

- Select the cluster directly surrounding the pile.
- Select *Refine cluster* from the *Mesh* menu and click <Update> to return to the main window.
- Refine the cluster once more.
- Select *Generate 3D Mesh* from the toolbar to generate the 3D Mesh.
- Inspect the resulting mesh (Figure 6.2) and click <Update> to return to the main window.

- Click <Calculate> to switch to calculation mode. Save the problem when prompted.

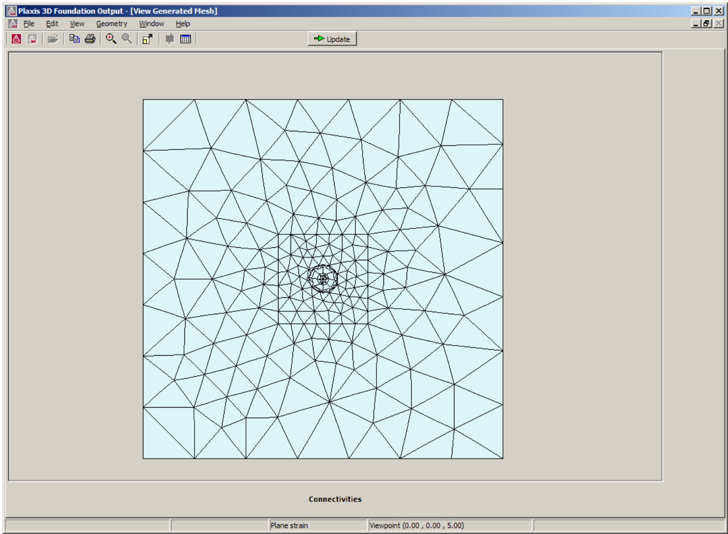


Figure 6.2 Refining the cluster surrounding the suction pile

6.2 CALCULATIONS

The calculation for this exercise will consist of five phases. These are the determination of initial conditions, the installation of the suction pile and three different load conditions.

Initial conditions and suction pile installation

As the soil layers are not completely horizontal, for this exercise the gravity loading option will be used to determine initial stresses. However, the default definition of calculation parameters is not completely suitable for this exercise. To change the settings for this phase, follow these steps:

Open the *Phases* window check that the calculation type is set to *Gravity loading*. Leave it at this setting and switch to the *Parameters* tab sheet. Make sure the *Ignore undrained behaviour* option is checked.

To model the installation of the suction pile, follow these steps:

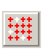
- Press <Next> to add a new calculation phase.
- Switch to the *Parameters* tab sheet and check the *Ignore undrained behaviour* option.

- Leave all other settings at their default values and click <OK> to return to the main window.
- Select the work plane $y = 0.0$ and click on the wall of the suction pile to activate it. It should be coloured blue, indicating the wall is active.
- Select the work plane $y = -1.5$ m and also activate the suction pile wall in this work plane. Leave the soil inside the suction pile active.
- Now add another calculation phase. On the *Parameters* tab sheet, check the option *Reset displacements to zero*.
- In the work plane $y = -1.5$ m, double click the point load at (19.0, 20.0). It is represented by a grey circle. Select *Point load* from the select window that appears.
- In the Point load window, set the X-value to –1000 and the Y-value to 0. This indicates a load of 1000 kN, acting in the negative x-direction.
- Click <OK> to close the window. The load should now be represented by a blue arrow. If it is not active (i.e. coloured grey), click once on the point (19.0, 20.0) in order to activate the load.

To add two more load situations, both starting directly after the installation of the suction pile, follow these steps:

- Add a new calculation phase. On the *General* tab sheet, set *Start from phase* to Phase 1. On the *Parameters* tab sheet, select *Reset displacements to zero*. Click <OK> to close the phase list.
- In the draw area, double click on the point load at (19.0, 20.0). Select the point load and enter a load of –866 kN in x-direction and 500 kN in y-direction. This represents a total load of 1000 kN at an angle of 30° with the horizontal.
- Add another calculation phase. Once more set *Start from phase* to Phase 1 and *Reset displacements to zero*.
- Double click the point load and set the force to –500 in x-direction and 866 in y-direction, representing a force of 1000kN at an angle of 60° with the horizontal.

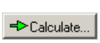
This completes the input of the calculation phases. Before starting the calculation, select some nodes in order to generate curves later on.



Click the *Select points for curves* button on the toolbar.

- Select the node where the load acts on the suction pile wall. Switch to the work plane $y = -1.5$ m and select the node at (19.0, 20.0).
- If desired, select more nodes and click <Update> to return to the draw area.

This concludes the input of the calculation phases. To start the calculation:



Click the <Calculate> button to start the calculation.

6.3 VIEWING OUTPUT RESULTS

After the calculation finishes, the results can be viewed by selecting the phase of interest. First, in order to compare the deformation of the suction pile under the different loads, follow these steps:

- Select the third calculation phase from the phase list and click <Output> to open the output program (see Figure 6.3).

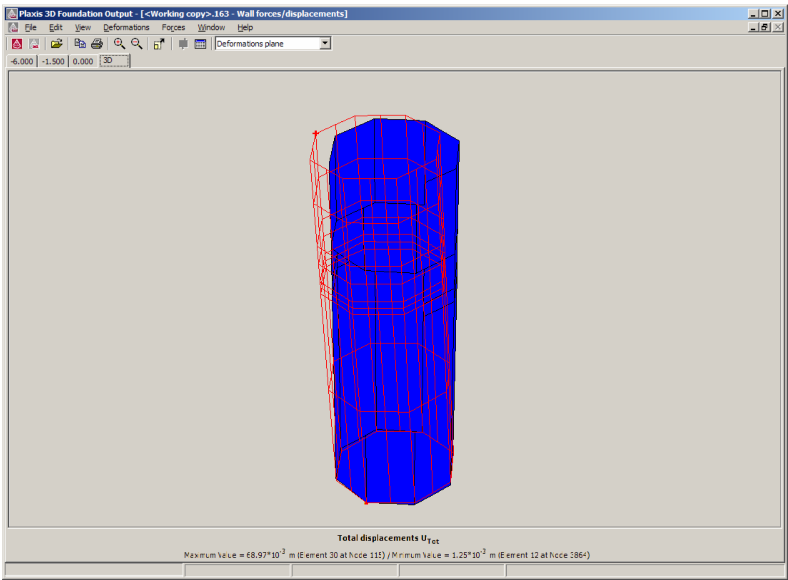


Figure 6.3 Total displacements at the end of Phase 3

Select *Open File* from the toolbar. A file open dialog is shown. From this dialog select the name of the current project. In the lower part of the dialog a list of available phases is shown. Select Phase 4 and click <Open> to open this phase.

- Repeat these actions for Phase 5.

The output for all three loading phases is now open simultaneously. In order to compare the results visually:

- Select *Tile vertically* from the *Window* menu. The results from the three phases will be shown next to each other (see Figure 6.4).

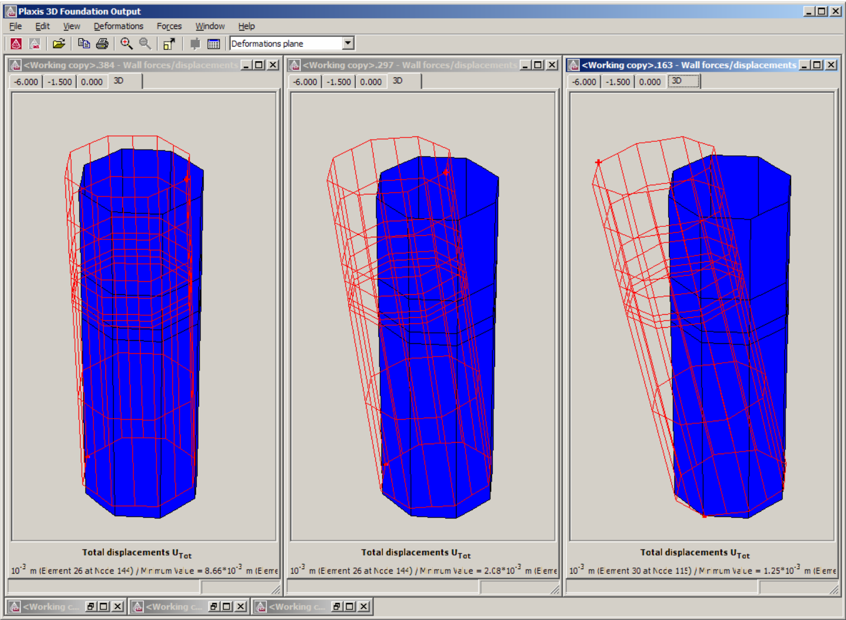


Figure 6.4 Output for three phases tiled vertically

7 EXCAVATION PIT (LESSON 5)

This lesson describes the construction of an excavation pit in soft clay and peat layers. The excavation pit was constructed as part of a research programme to study the behaviour of sheet pile walls in soft soil as described in detail in Kort (2002)†. A highly simplified case is presented here. The test pit is a relatively small excavation of 12 by 14 m, excavated to a depth of 7.5 m below the surface. After the full excavation, an additional surface load is added on one side of the pit.

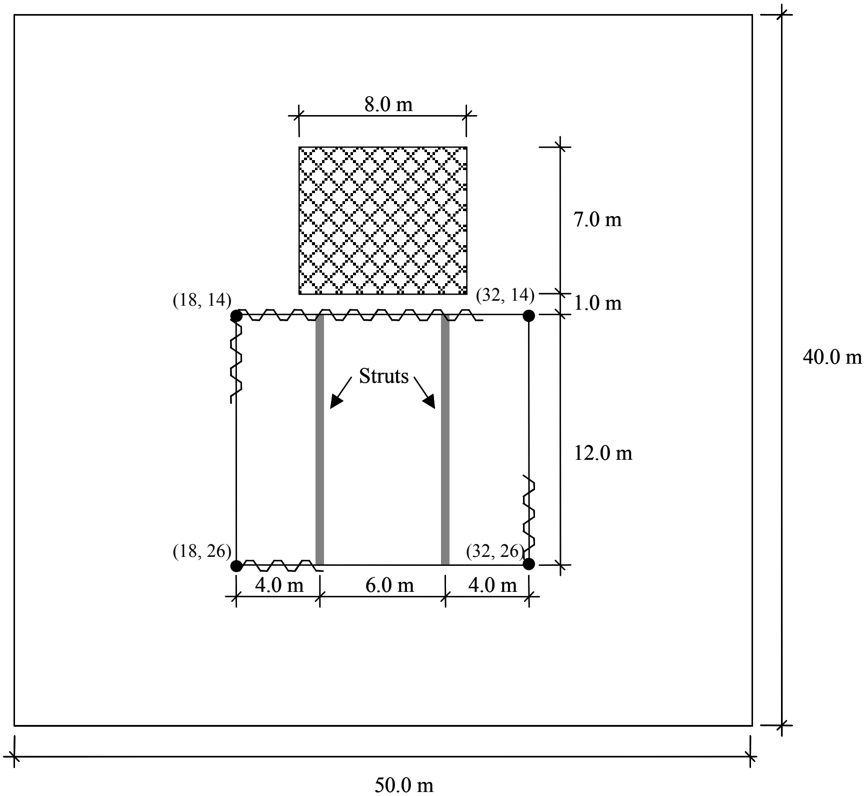


Figure 7.1 Top view of the excavation pit

Although the original test pit contained different types of sheet pile elements on the sides of the excavation, only a single type is modelled here.

† D.A. Kort . Steel Sheet Pile Walls in Soft Soil. Ph.D. Thesis Delft University of Technology, DUP Science, 2002

7.1 INPUT

The proposed geometry for this exercise is 50 m wide and 40 m long, as shown in Figure 7.1. The cross section of the excavation pit with soil layers is shown in Figure 7.3. The excavation pit, 12 m wide by 14 m long is placed in the center of the geometry. An additional cluster for the surface load is needed next to the pit.

- After entering the general settings for this exercise, add a work plane at *y* = -19.0 m first. Also change the level of the other work plane to *y* = 0.5 m.
- In the work plane *y* = 0.5 m, add the sheet pile walls to the geometry using a single chain of walls. After adding the sheet pile walls, add work planes at *y* = -0.5 m, *y* = -5.0 m and *y* = -7.0 m. Check that the sheet pile walls are present between *y* = 0.5 m and *y* = -19.0 m.

Two beams will be used to model the struts between the sheet pile walls. To add these beams, follow these steps:

Add a beam to the geometry at *y* = 0.5 m using the *Beam* option from the tool bar. Click on (22.0, 14.0) to start the beam. Click on (22.0, 26.0) to add a second point to the beam. Right click to end drawing a beam.

- Also add a beam between (28.0, 14.0) and (28.0, 26.0).

A further set of beams will be use to model the walings at the top of the sheet pile walls. In order to add this beam:

- Add a beam chain of 4 beams at the same location as the sheet pile walls in the work plane y = 0.5 m. Click once on the first corner of the sheet pile walls, click once on the second corner etc, until all four beams are drawn. Then right click to end drawing. Do not right click before, as this will end drawing the beam chain and start drawing a new beam chain.
- In the work plane *y* = -0.5 m add a cluster composed of geometry lines to the geometry with corner points (21.0, 6.0), (21.0, 13.0), (29.0, 13.0) and (29.0, 6.0).

Select the distributed load (horizontal planes) from the toolbar and add a load to the newly added cluster.

Material properties

- As all soil layers for this exercise are horizontal, only a single borehole is needed. Add a borehole to the geometry and insert 4 additional layer boundaries (see Figure 7.2). The bottom of the lowest layer is at *y* = -32 m. The level of the other layer boundaries is given in Table 11. Set the phreatic level to *y* = -2.0 m and open the material properties window.
- Add the five data sets given in Table 11 and assign them to their respective soil layers.

Table 11 Material properties for the soil layers

Parameter	Name	Sand	Clay	Peat	Deep Clay	Deep Sand	Unit
Top of Layer		-0.5	-1.5	-5.0	-10.5	-17.0	m
Material model	Model	MC	MC	MC	MC	MC	-
Type of material behaviour	Type	Drained	Undr.	Undr.	Undr.	Drained	-
Soil weight above phreatic level	γ_{unsat}	16	17	8	15	17	kN/m^3
Soil weight below phreatic level	γ_{sat}	20	18	10.5	16	20	kN/m^3
Permeability	k_i	1	$5 \cdot 10^{-3}$	$1 \cdot 10^{-4}$	$2 \cdot 10^{-3}$	1	m/d
Young's modulus	E_{ref}	$1.6 \cdot 10^4$	$1 \cdot 10^3$	$1.5 \cdot 10^3$	$3 \cdot 10^3$	$2.8 \cdot 10^4$	kN/m^2
Poisson's ratio	ν	0.3	0.35	0.35	0.35	0.3	-
Cohesion	c_{ref}	1	10	10	10	0.1	kN/m^2
Friction angle	φ	32	30	25	30	34	°
Dilatancy angle	ψ	2	0	0	0	4	°
Tension cut-off	-	0	5	5	5	0	kN/m^2
Interface reduction factor	R_{inter}	1	1	1	1	1	-
Lateral earth pressure coeff.	K_0	0.47	0.5	0.58	0.5	0.44	-

Hint: The *Tension cut-off* option is activated by default, at a value of 0 kPa. This option is found in the *Advanced* options on the *Parameters* tab sheet of the *Soil* material data set window. Here the Tension cut-off value can be changed or the option can be deactivated entirely.

- In the *Soil* tab sheet, enter the corresponding K_0 values.
- Uncheck the *Hydrostatic* check box and change the water pressure at layer boundary 5 to *WPress+* = -160 kPa and *WPress-* = -160 kPa.
- Similarly, change the water pressure at the bottom boundary 6 to –310 kPa.
- Close the borehole window and return the main draw area.
- Create material data sets also for the sheet pile walls, walings and the struts, and assign them to the structural elements. Take care to assign the material data set for the walls to the wall elements in each of the four work planes between $y = 0.5$ and $y = -19.0$ m.

In creating the material data set for the sheet pile wall, the geometrical orthotropy of the sheet piles and walings is converted into orthotropic Young's and shear moduli for the wall material data set (see Figure 7.4, Table 12, Table 13). Using $E_1 = 12\ E_{steel}\ I_1 / d^3$ the moment of inertia of the sheet pile wall as supplied by the manufacturer can be converted to a representative Young's modulus. Similar relations hold for the other directions. See Chapter 6 of the Material Models Manual for more details and an example for a sheet pile wall.

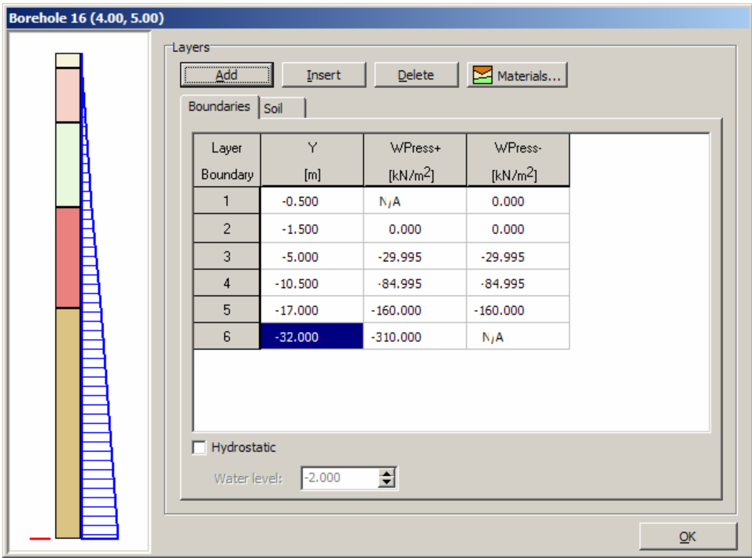


Figure 7.2 Borehole with the soil layers and pore pressure distribution

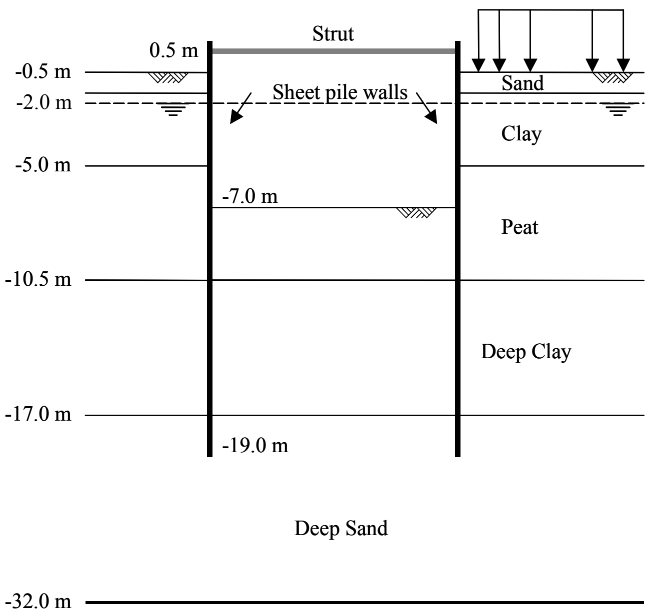


Figure 7.3 Cross section of the excavation pit with the soil layers

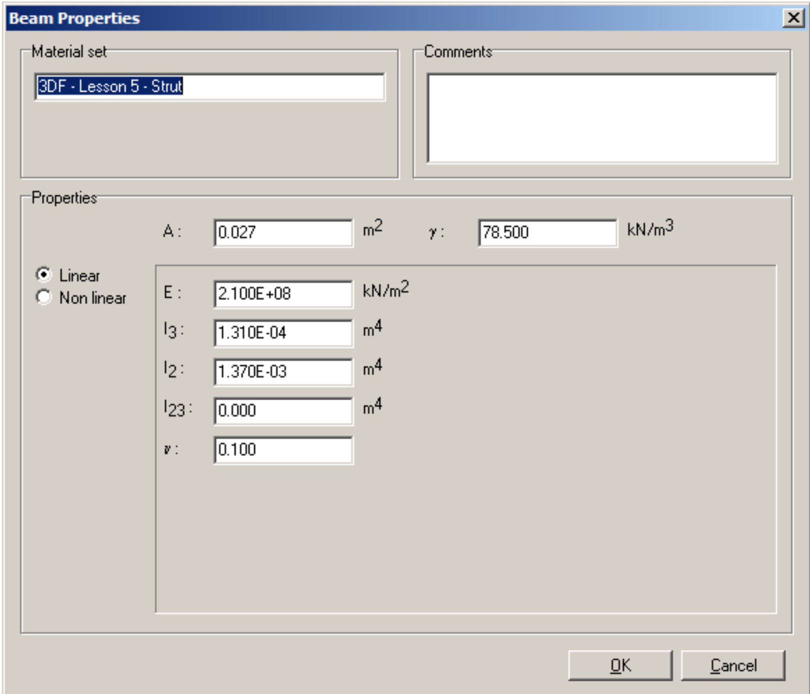


Figure 7.4 Material data set for struts

Table 12 Material properties for the sheet pile walls

Parameter	Name	Sheet pile wall	Unit
Material model	Model	Linear	-
Thickness	d	0.014	m
Volumetric weight	γ	78.5	kN/m^3
Young's modulus	E_1	$1.8 \cdot 10^{11}$	kN/m^2
	E_2	$1.8 \cdot 10^{7}$	kN/m^2
	E_3	$2.1 \cdot 10^{8}$	kN/m^2
Shear modulus	G_{12}	$6 \cdot 10^{9}$	kN/m^2
	G_{13}	$1 \cdot 10^{7}$	kN/m^2
	G_{23}	$3 \cdot 10^{7}$	kN/m^2
Poisson's ration	ν	0	-

Table 13 Material properties for the struts

Parameter	Name	Strut	Waling	Unit
Material model	-	Linear	Linear	
Cross section area	A	0.027	0.020	m^2
Volumetric weight	γ	78.5	78.5	kN/m^3
Young's modulus	E	$2.1 \cdot 10^8$	$2.1 \cdot 10^8$	kN/m^2
Moment of Inertia	I_2	$1.37 \cdot 10^{-3}$	$5.77 \cdot 10^{-4}$	m^4
	I_3	$1.31 \cdot 10^{-4}$	$1.08 \cdot 10^{-4}$	m^4
	I_{23}	0	0	m^4
Poisson's ratio	ν	0.1	0	-

Mesh generation

The mesh for this example will be of medium coarseness, with a further refinement of the excavated clusters. To generate this mesh, follow these steps:

- From the *Mesh* menu, select *Refine global*. This will globally refine the mesh and the Output program will be opened to present the results. Click <Update> to return to the main draw area and select the three clusters of the excavation pit and the cluster of the load area. Hold down the <Shift> key to select multiple clusters. Choose *Refine cluster* from the *Mesh* menu and the Output program will show the generated mesh (see Figure 7.5).

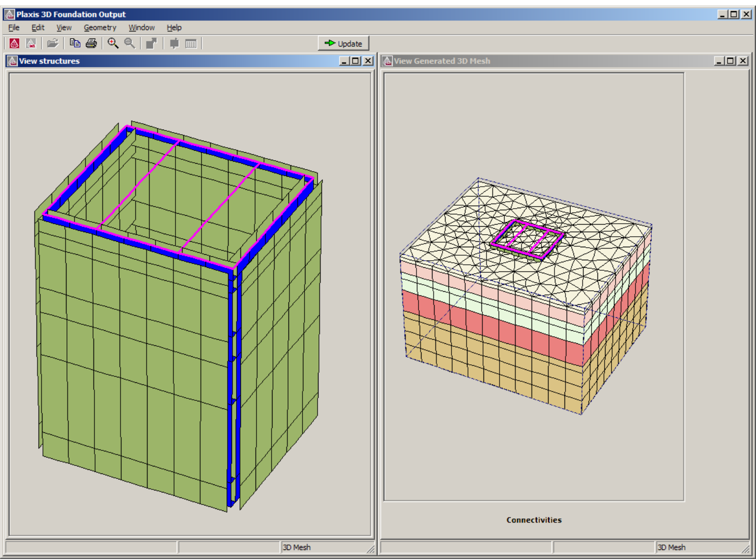


Figure 7.5 Preview of the generated 3D mesh

- Return to the main draw area and generate the 3D mesh. Click <Update> to return to the main draw area.

7.2 CALCULATIONS

The calculation consists of 7 phases. The initial phase will consist of the generation of initial stresses using the K_0 procedure. The following phases will consist of the installation of the sheet piles and the subsequent excavation of the pit. The sixth phase will be the application of the additional load next to the pit and the final phase will be a consolidation phase with a length of 300 days.

- Change the *Calculation type* for the first phase to *K0 procedure* and check the K_0 values as given in Table 11.
- Add a new phase and enter a time period of 7 days in the *Time* edit box of the *Parameters* tab sheet.
- Close the *Phases* window and activate all walls and beams in this phase. Take care to activate the walls in all four work planes between $y = 0.5$ and $y = -19.0$. This will simulate the installation of the sheet piles and the struts.
- Add a further phase and set the time period to 6 days. Deactivate the three soil clusters inside the excavation pit between $y = -0.5$ and $y = -7.0$. Leave the pore pressures in these clusters unchanged. This simulates the wet excavation of the pit.
- Add a fourth calculation phase and set the time period to 4 days. This phase will simulate the dewatering of the excavation pit.
- Select the work plane at $y = -0.5$ m. Select one of the three clusters inside the excavation and deactivate the *water below* for this cluster. Repeat this the other two clusters inside the excavation.
- Change the work plane to y = -5.0 m. Select one of the clusters inside the excavation and select water below. Click on the <Change> button to change the pore pressure distribution. In the *pore pressure distribution window*, select *cluster phreatic level* and set the *phreatic level* to $y = -5.0$ m. Repeat this for the other two clusters inside the excavation.
- Change the work plane to $y = -7.0$ m. For all three clusters inside the excavation, change the *cluster pore pressure distribution* for the water below to *Interpolate between adjacent lines or clusters*.
- Add a fifth calculation phase. Change the calculation type to *Consolidation* and set the time period to 25 days.
- Add a sixth calculation phase and set the time period to 2 days. Activate the surface load at $y = -0.5$. Set the load to -100 kPa acting in the downward y-direction.

- Add a seventh calculation phase and change the calculation type to *Consolidation*. Set the time period to 300 days.
- Start the calculation.

7.3 OUTPUT

The deformation and bending moments of the sheet pile wall in the final phase can be investigated by first opening the output for the final phase.

- Select the final phase and click <Output> to open this phase in the output program.
- Switch to one of the work planes and click on the wall and interfaces to open output for the walls in a separate window. From the selection box, choose output for the wall. Change to deformation shadings and afterwards to bending moments, shaded (see Figure 7.6).

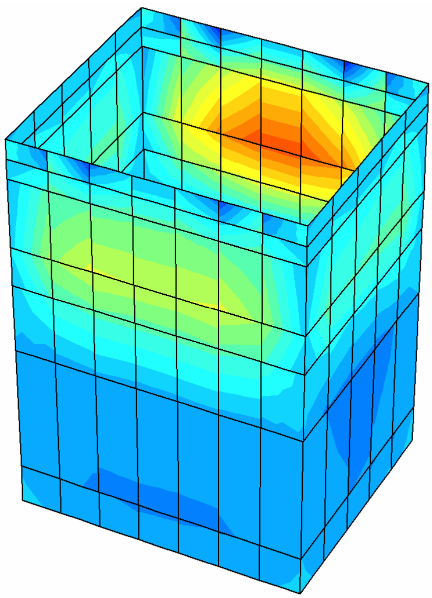


Figure 7.6 Bending moments in the sheet pile wall at the end of the final phase

- Similarly as for the wall, open an output window for the interfaces and display the excess pore pressures at the end of the final phase (see Figure 7.7).

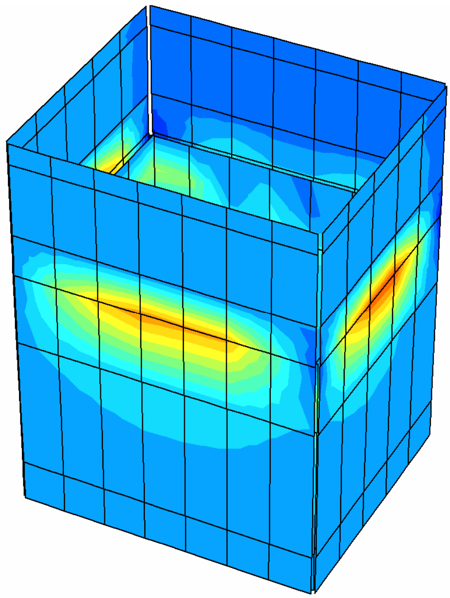


Figure 7.7 Excess pore pressures in the interfaces along the sheet pile wall at the end of the final phase

APPENDIX A - MENU TREE

A.1 INPUT MENU

File	**Edit**	**View**	**Geometry**	**Loads**	**Materials**	**Mesh**	**Help**
Go to Output Program	Undo	Zoom In	Geometry Line	Distr. Load Horz. Plane	Soils and Interfaces	Global Settings	Manuals
Go to Curves Program	Copy	Zoom Out	Pile	Distr. Load Vert. Plane	Beams	Triangulate	License Update
New		Reset View	Beam	Line Load	Walls	Refine Global	Disclaimer
Open		Table	Vert. Beam	Vert. Line Load	Floors	Refine Cluster	About
Save		Rulers	Wall	Point Load	Springs	Refine Line	
Save as		Axes	Floor			Refine Point	
Print		Cross Hair	Line Fixity			Reset All	
Work Directory		Grid	Vert. Line Fixity			Generate 2D Mesh	
General Settings		Snap to Grid	Spring			Generate 3D Mesh	
Pack Project		Point Numbers	Borehole				
(Recent Projects)		Chain Numbers	Work Planes				
Exit							

A.2 OUTPUT MENU

File	**Edit**	**View**	**Geometry**	**Deformations**	**Stresses**	**Window**	**Help**
Open	Copy	Zoon In	Structures	Deformed Mesh	Effective Stresses	Cascade	Manuals
Close	Scale	Zoom Out	Interfaces	Total Displacements	Cartesian Effective Stresses	Tile Horizontally	License Update
Close All	Scan Line	Reset View	Materials	Phase Displacements	Total Stresses	Tile Vertically	Disclaimer
Print	Interval	Viewpoint	Phreatic Level	Incremental Displacements	Cartesian Total Stresses	(Active Windows)	About
Work Directory		Cross Section	Loads	Total Strains	Equivalent Isotropic Stress		
(Recent Projects)		Table	Fixities	Cartesian Total Strains	Isotropic Preconsolidation Stress		
Exit		Title	Connectivity Plot	Incremental Strains	Isotropic Overconsolidation Ratio		
		legend	Nodes	Cartesian Incremental Strains	Plastic Points		
		Axes	Stress Points		Active Pore Pressures		
		Model Contour	Element/Chain Numbers		Excess Pore Pressures		
		Shadow	Node Numbers		Steady State Pore Pressures		
		Partial Geometry	Stress Point Numbers		Groundwater Head		
		Distance Measurement	Material Set Numbers				
		General Info	Cluster Numbers				
		Load Info					
		Material Info					
		Calculation Info					

Floors and Walls:

Deformations	**Forces**
Total Displacements	Axial N1
Phase Displacements	Axial N2
Incremental Displacements	Shear Q12
	Shear Q23
	Shear Q13
	Bending Moments M11
	Bending Moments M22
	Torsion Moments M12
	Groundwater Head

Cross sections:

Deformations	**Stresses**
Total Displacements	Cartesian Effective Stresses
Phase Displacements	Cartesian Total Stresses
Incremental Displacements	Cross Section Stresses
Total Strains	Effective Mean Stress (p’)
Cartesian Total Strains	Total Mean Stress (p)
Incremental Strains	Deviatoric Stresses (q’)
Cartesian Incremental Strains	Equivalent Isotropic Stress
	Isotropic Preconsolidation Stress
	Isotropic Overconsolidation Ratio
	Active Pore Pressure
	Excess Pore Pressure
	Steady State Pore Pressure
	Groundwater Head

Interfaces:

Deformations	Stresses
Total Displacements	Effective Normal Stresses
Phase Displacements	Vertical Shear Stresses
Incremental Displacements	Horizontal Shear Stresses
	Relative Shear Stresses
	Active Pore Pressures
	Excess Pore Pressures

A.3 CURVES MENU

File	**Edit**	**View**	**Format**	**Window**	**Help**
New	Copy	Zoom In	Curve	Cascade	Manuals
Open		Reset View	Table	Tile Horizontally	License Update
Save		Table		Tile Vertically	Disclaimer
Add Curve		Legend		(Active Windows)	About
Delete Chart		Value indication			
Close					
Close All					
Work Directory					
Print					
(Recent Projects)					
Exit					