# PLAXIS 3D FOUNDATION

# Reference Manual

# version 1.5

TABLE OF CONTENTS

# 1 INTRODUCTION

The PLAXIS 3D FOUNDATION program is a special purpose three-dimensional finite element computer program used to perform deformation analyses for various types of foundations in soil and rock. The program uses a convenient graphical user interface that enables users to quickly generate a true three-dimensional finite element mesh based on a composition of horizontal cross sections at different vertical levels. The program has special features to model a limited number of piles, but it can also be used for other types of geotechnical structures. Users need to be familiar with the Windows environment, and should preferably (but not necessarily) have some experience with the standard PLAXIS (2D) deformation program. To obtain a quick working knowledge of the main features of the 3D FOUNDATION program, users should work through the example problems contained in the Tutorial Manual.

The Reference Manual is intended for users who want more detailed information about the program features. The manual covers topics that are not covered exhaustively in the Tutorial Manual. It also contains practical details on how to use the 3D FOUNDATION program for a wide variety of problem types.

The user interface consists of three sub-programs: Input, Output and Curves. The Input program is used to define the problem geometry and calculation phases. The Output program is used to inspect the results of calculations in a three-dimensional view or in cross-sections. The Curves program is used to plot graphs of output quantities of pre-selected geometry points.

The contents of this Reference Manual are arranged according to the sub-programs and their respective options as listed in the corresponding menus. This manual does not contain detailed information about the constitutive models, the finite element formulations or the non-linear solution algorithms used in the program. For detailed information on these and other related subjects, users are referred to the various papers listed in the Scientific Manual and the Material Models Manual.

# 2 GENERAL INFORMATION

Before describing the specific features in the three parts of the PLAXIS 3D FOUNDATION user interface, this first chapter is devoted to some general information that applies to all parts of the program.

## 2.1 UNITS AND SIGN CONVENTIONS

### *Units*

It is important in any analysis to adopt a consistent system of units. At the start of the input of a geometry, a suitable set of basic units should be selected. The basic units comprise a unit for length, force and time. These basic units are defined in the *General settings* window of the Input program. The default units are metres [m] for length, kiloNewton [kN] for force and day [day] for time. However, the user is free to choose whichever system is most convenient, only the unit of time is limited to [s], [min], [hour] and [day]. All subsequent input data should conform to this system and the output data should be interpreted in terms of this same system. From the basic set of units, as defined by the user, the appropriate unit for the input of a particular parameter is generally listed directly behind the edit box or, when using input tables, above the input column. In all of the examples given in the PLAXIS 3D FOUNDATION manuals, the standard units are used.

For convenience, the units of commonly used quantities in a 3D FOUNDATION analysis are listed below:

| | | Standard | Alternative |
|---|---|---|---|
| Basic units: | Length | [m] | [in.] |
| | Force | [kN] | [lb] |
| | Time | [day] | [sec] |
| Geometry: | Coordinates | [m] | [in.] |
| | Displacements | [m] | [in.] |
| Material properties: | Young's modulus | [kN/m$^2$] = [kPa] | [psi] = [lb/in$^2$] |
| | Cohesion | [kN/m$^2$] | [psi] |
| | Friction angle | [deg.] | [deg.] |
| | Dilatancy angle | [deg.] | [deg.] |
| | Unit weight | [kN/m$^3$] | [lb/cu in.] |
| | Permeability | [m/day] | [in./sec] |

| | | | |
|---|---|---|---|
| Forces & stresses: | Point loads | [kN] | [lb] |
| | Line loads | [kN/m] | [lb/in.] |
| | Distributed loads | [kPa] | [psi] |
| | Stresses | [kPa] | [psi] |

**Hint:** Units are only used as a reference for the user. Note that changing the basic units in the *General settings* does not affect the input values.

> If it is the user's intention to use a different system of units on an existing set of input data, the user has to modify all parameters manually.

### *Sign convention*

The generation of a three-dimensional (3D) finite element model in the PLAXIS 3D FOUNDATION program is based on the creation of a geometry model. The geometry model involves a composition of work planes (*x-z* planes) and boreholes. A work plane is a horizontal cross section at a particular vertical level (*y*-level) in which structures and loads are defined (Figure 2.1).

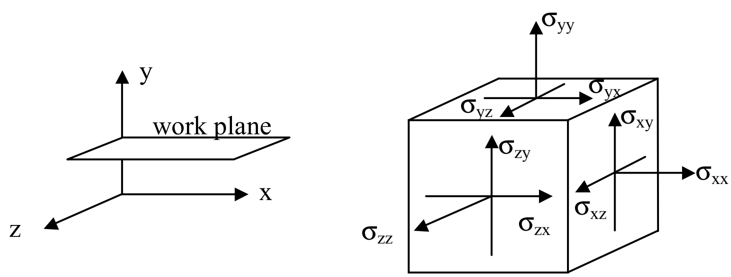



Figure 2.1 Coordinate system, example of work plane and indication of positive stress components.

In addition to the work planes, multiple vertical boreholes can be defined to determine the soil stratigraphy at different locations. In between the boreholes the soil layer positions are interpolated. Soil layers and ground surface may be non-horizontal. During the generation of a 3D mesh, all data from work planes and boreholes are properly taken into account.

Stresses computed in the PLAXIS 3D FOUNDATION program are based on the Cartesian coordinate system shown in Figure 2.1. In all of the output data, compressive stresses and forces, including pore pressures, are taken to be negative, whereas tensile stresses and forces are taken to be positive. Figure 2.1 shows the positive stress directions.

## 2.2 FILE HANDLING

The PLAXIS 3D FOUNDATION program handles all files with a modified version of the general Windows® file requester (Figure 2.2). With the file requester, it is possible to search for files in any admissible directory of the computer (and network) environment. The main file used to store information of a PLAXIS 3D FOUNDATION project has a structured format and is named <*project*>.PF3, where <*project*> is the project title. Besides this file, additional data is stored in multiple files in the sub-directory <*project*>.DF3. It is generally not necessary to enter such a directory because it is not possible to read individual files in this directory.

If a PLAXIS 3D FOUNDATION project file (*.PF3) is selected, a small bitmap of the corresponding project geometry is shown in the file requester to enable a quick and easy recognition of a project.

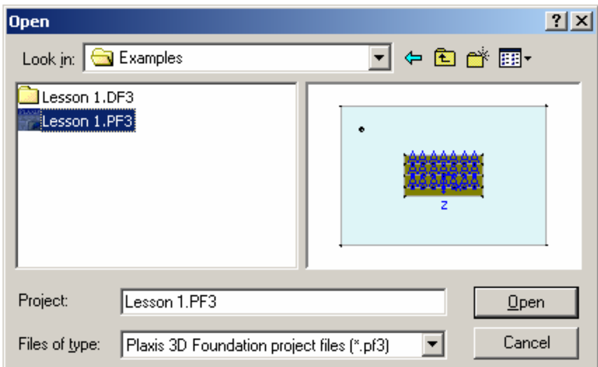



Figure 2.2 PLAXIS file requester

### 2.2.1 COMPRESSING PROJECT FILES

In order to save space and to facilitate moving projects to different computers, it is possible to archive a project using the *Pack project* option from the *File* menu. This will open the *PLAXIS project compression* window (Figure 2.3), where the user can opt to include the output for the initial phase only, for selected phases only, or for all phases (the default). After clicking <OK> the program will then archive all necessary input files and the selected output files into a single file named <*project*>.PF3ZIP. This file is located in the same directory as the <*project*>.PF3 file. At this point it is save to remove the original <*project*>.PF3 file and the <*project*>.DF3 directory.

If a compressed project file (*.PF3ZIP) is selected in the PLAXIS file requester, the project will automatically be uncompressed in the current directory and opened, as if the corresponding <*project*>.PF3 file had been opened.

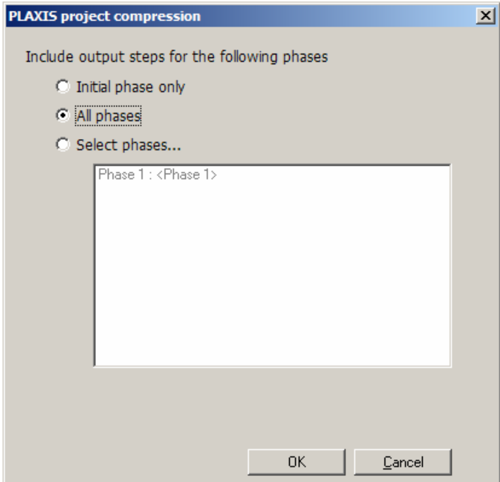



Figure 2.3 Pack project dialog

## 2.3 INPUT PROCEDURES

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

| | |
|---|---|
| Input of geometry objects | (e.g. drawing a wall) |
| 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. These input procedures are described in detail in Section 2.3 of the *Tutorial Manual*.

## 2.4 HELP FACILITIES

To inform the user about the various program options and features, in the Help menu a link has been created to a digital version of the Manual.

Many program features are available as buttons in a toolbar. When the mouse pointer is positioned on a button for more than a second, a short description ('hint') appears in a yellow flag, indicating the function of the button.

# 3 INPUT (PRE-PROCESSING)

## 3.1 THE INPUT PROGRAM

This icon represents the Input program. This program consists of two different modes: *Model* and *Calculation*. The *Model* mode contains all facilities to create and to modify a geometry model and to generate a 2D and 3D finite element mesh. The *Calculation* mode contains all facilities to define calculation phases representing different stages of loading or construction, including the initial situation. In this chapter the description is focused on the creation of a model and a finite element mesh (*Model* mode). The various options of the *Calculation* mode are described in Chapter 4.

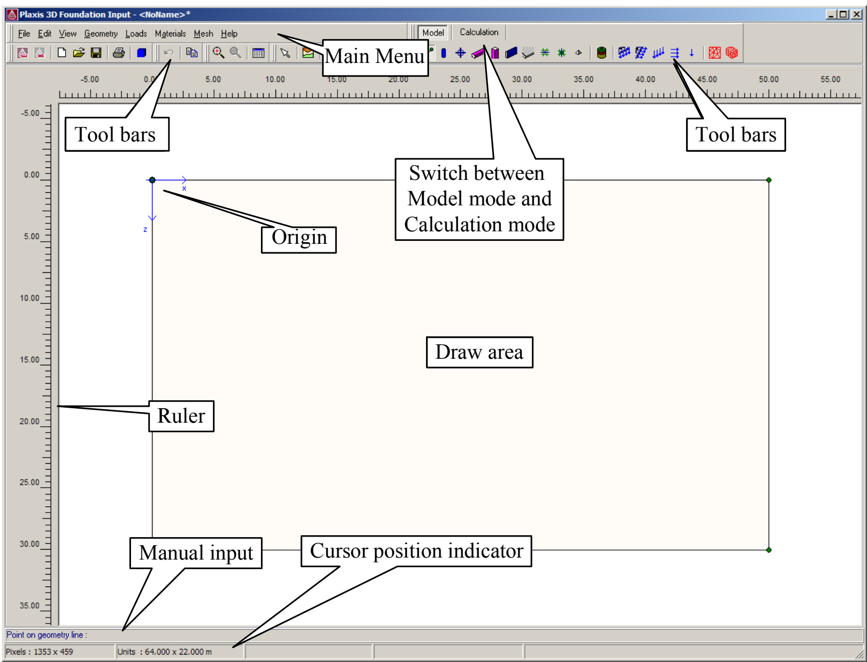



Figure 3.1 Main window of the Input program (*Model* mode)

At the start of the Input program a dialog box appears in which a choice must be made between the selection of an existing project and the creation of a new project. When selecting *New project* the *General settings* window appears in which the basic model parameters of the new project can be set (Section 3.2.2 General settings).

When selecting *Existing project,* the dialog box allows for a quick selection of one of the four most recent projects. If an existing project is to be selected that does not appear in the list, the option <<<*More files*>>> can be used. As a result, the file requester appears which enables the user to browse through all available directories and to select the desired PLAXIS 3D FOUNDATION project file (*.PF3). After selection of an existing project, the corresponding geometry is presented in the main window. The main window of the Input program contains the following items (Figure 3.1).

### *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 toolbar.

### *Toolbar (File)*

This toolbar 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).

### *Toolbar (Edit)*

This toolbar contains buttons for editing operations, corresponding with the options in the *Edit* menu.

### *Toolbar (View)*

This toolbar 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.

### *Toolbar (General)*

This toolbar 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.

### *Toolbar (Model)*

This toolbar contains buttons related to the creation of a geometry model, such as *Geometry lines*, *Piles*, *Beams*, *Walls*, *Floors*, *Line fixities*, *Springs*, *Boreholes* and *Loads*, as well as options for 2D and 3D mesh generation.

### *Toolbar (Calculation)*

This toolbar contains buttons related to the definition of calculation phases. Details about these options are given in Chapter 4.

### *Rulers*

At both the left and the top of the draw area, rulers indicate the physical $x$- and $z$-coordinates of the geometry model. This enables a direct view of the geometry dimensions, except for the vertical position. The rulers can be switched off in the *View* sub-menu.

### *Draw area*

The draw area is the drawing sheet on which the geometry model is created and modified. The creation and modification of objects in a geometry model is mainly done by means of the mouse, but for some options a direct keyboard input is available (see below, *Manual input*). 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.

### *Axes*

If the physical origin is within the range of given dimensions it is presented by a small circle in which the $x$- and $z$-axes are indicated by arrows. The indication of the axes can be switched off in the *View* sub-menu.

### *Manual input*

If drawing with the mouse does not give the desired accuracy, the *Manual input* line can be used. Values for the $x$- and $z$-coordinates can be entered here by typing the required values separated by a space ($x$-value <space> $z$-value). Manual input of coordinates can be given for all objects.

Instead of the input of absolute coordinates, increments with respect to the previous point can be given by typing an @ directly in front of the value (@$x$-value <space> @$z$-value).

In addition to the input of coordinates, existing geometry points may be selected by their number.

### *Cursor position indicator*

The cursor position indicator gives the current position of the mouse cursor both in physical units ($x$-, $z$-coordinates) and in screen pixels.

## 3.2 THE INPUT MENU

The main menu of the Input program contains pull-down sub-menus covering most options for handling files, transferring data, viewing graphs, defining work planes, creating geometry objects, generating finite element meshes and entering data in general. In the *Model* mode, the menu consists of the sub-menus *File*, *Edit*, *View*, *Geometry*, *Loads*, *Properties*, *Mesh* and *Help*.

### *The File sub-menu*

| | |
|---|---|
| *Go to Output program* | To open the Output program. |
| *Go to Curves program* | To open the Curves program. |
| *New* | To create a new project. The *General settings* window is presented. |
| *Open* | To open an existing project. The file requester is presented. |
| *Save* | To save the current project under the existing name. If a name has not been given before, the file requester is presented. |
| *Save as* | To save the current project under a new name. The file requester is presented. |
| *Print* | To print the content of the draw area on a selected printer. The print window is presented. |
| *Work directory* | To set the default directory where 3D FOUNDATION project files will be stored. |
| *General settings* | To set the basic parameters of the model (Section 3.2.2). |
| *Pack project* | To compress a project and pack it into a single file, to facilitate sending the project by e-mail. The file is named <project>.PF3ZIP and stored in the <project>.DF3 directory. |
| (*recent projects*) | Convenient way to open one of the four most recently edited projects. |
| *Exit* | To leave the Input program. |

### *The Edit sub-menu*

| | |
|---|---|
| *Undo* | To restore a previous status of the geometry model (after an input error). Repetitive use of the undo option is limited to the 10 most recent actions. |
| *Copy* | To copy the content of the draw area to the Windows clipboard. |

### *The View sub-menu*

| | |
|---|---|
| *Zoom in* | To zoom into a rectangular area for a more detailed view. After selection, the zoom area must be indicated using the mouse. Press the left mouse button at a corner of the zoom area; hold the mouse button down and move the mouse to the opposite corner of the zoom area; then release the button. The program will zoom into the selected area. The zoom option may be used repetitively. |
| *Zoom out* | To restore the view to before the most recent zoom action. |

| | |
|---|---|
| *Reset view* | To restore the full draw area. |
| *Table* | To view the table with the $x$- and $z$-coordinates of all geometry points in the geometry model. The table may be used to adjust existing coordinates. |
| *Rulers* | To show or hide the rulers along the draw area. |
| *Axes* | To show or hide the arrows indicating the $x$- and $z$-axes. |
| *Cross hair* | To show or hide the cross hair during the creation of objects in a geometry model. |
| *Grid* | To show or hide the grid in the draw area. |
| *Snap to grid* | To activate or deactivate the snapping into the regular grid points. |
| *Point numbers* | To view the geometry point numbering. |
| *Chain numbers* | To view the numbering of structural object chains. |

### *The Geometry sub-menu*

The *Geometry* sub-menu contains the basic options to create work planes, boreholes or geometry objects of a geometry model. In addition to a normal geometry line, the user may select *Piles*, *Beams*, *Walls*, *Floors*, *Line fixities* or *Springs*. The various options in this sub-menu are explained in detail in Section 3.3.

### *The Loads sub-menu*

The *Loads* sub-menu contains the options to add loads and boundary conditions to the geometry model. The various options in this sub-menu are explained in Section 3.4.

### *The Materials sub-menu*

The *Materials* sub-menu contains the options to define data sets of parameters for soil and structural objects. The various options in this sub-menu are explained in Section 3.5.

### *The Mesh sub-menu*

The *Mesh* sub-menu contains the options to generate a 2D finite element mesh, to apply local and global mesh refinement on the 2D mesh and to generate a 3D finite element mesh. The options in this sub-menu are explained in detail in Section 3.6.

### *The Help sub-menu*

The Help menu contains options to open the online version of the documentation, to verify and update the licence information stored in the hardlock and to view the *About* box and version information of the program. The options in this sub-menu are explained in Section 2.4

### 3.2.1 READING AN EXISTING PROJECT

An existing PLAXIS 3D FOUNDATION project can be read by selecting the *Open* option in the *File* sub-menu. The default directory that appears in the file requester is the directory where all program files are stored during installation. This default directory can be changed by means of the *Work directory* option in the *File* menu. In the file requester, the *Files of type* is, by default, set to 'PLAXIS 3D FOUNDATION project files (*.PF3)', which means that the program searches for files with the extension .PF3. After the selection of such a file and clicking on the *Open* button, the corresponding geometry is presented in the draw area.

#### *Save project under a new name*

If it is desired to keep an existing project as it is, while an attempt is made to elaborate on the project, the existing project can be saved under a new name. This can best be done immediately after the existing project is opened in the Input program.

Please note that as soon as the mesh of an existing project is regenerated, the project files are automatically updated and it is not possible any longer to cancel previous actions.

#### *Converting Version 1 project files*

Although the file structure of PLAXIS 3D FOUNDATION Version 1 and Version 1.5 files is slightly different, it is possible to open old projects, after which they are automatically converted to the new Version 1.5 file structure.

Please note that, after the files are converted and the converted project is saved, it is no longer possible to open the project in Version 1. If it is desired to keep a Version 1 project available so it can still be read in Version 1, the project should be copied or saved under a different name using the *Save as* option in the *File* sub-menu.

#### *Opening packed projects*

If a compressed project file (*.PF3ZIP) is selected in the Plaxis file requester, the project will automatically be uncompressed in the current directory and opened, as if the corresponding <*project*>.PF3 file had been opened.

### 3.2.2 GENERAL SETTINGS

The *General settings* window appears at the start of a new problem and may later be selected from the *File* sub-menu (see Figure 3.2). The *General settings* window contains the two tab sheets *Project* and *Dimensions*. The *Project* tab sheet contains the project name and location, project description, gravity acceleration and the unit weight of water. The *Dimensions* tab sheet contains the basic units for length, force and time (Section 2.1), the dimensions of the 3D model and the grid settings.

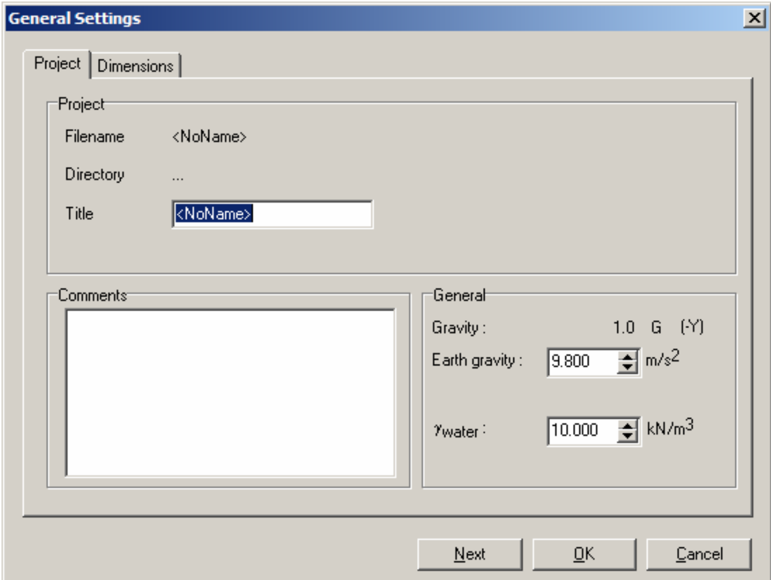



Figure 3.2 General settings window (*Project* tab sheet)

### *Gravity*

Earth gravity has been preset to 9.8 m/s$^2$ and the direction of gravity coincides with the negative *y*-axis, i.e. perpendicular to the draw area. Gravity is implicitly included in the unit weights given by the user (Section 3.5.2).

### *Unit weight of water*

In projects that involve pore pressures, the input of a unit weight of water is required to determine the effective stresses and pore pressures. The water weight ($\gamma_{water}$) can be entered in the *Project* tab sheet of the *General settings* window. By default, the unit weight of water is set to 10 kN/m$^3$, if the default basic units of kilo-Newton and metre are used.

### *Units*

Units for length, force and time to be used in the analysis are defined when the input data are specified. These basic units are entered in the *Dimensions* tab sheet of the *General settings* window (see Figure 3.3).

The default units, as suggested by the program, are m (metre) for length, kN (kiloNewton) for force and day for time. The corresponding units for stress and weights are listed in the box below the basic units.

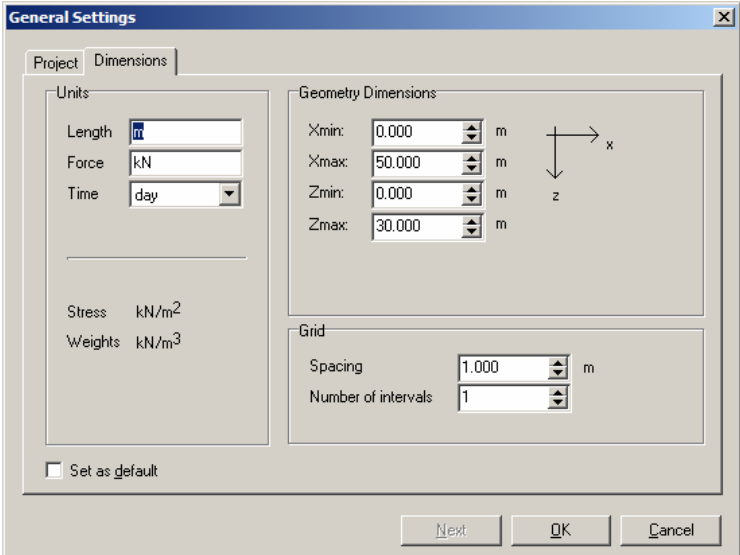



Figure 3.3 General settings window (*Dimensions* tab sheet)

All input values should be given in a consistent set of units (Section 2.1). The appropriate unit of a certain input value is usually given directly behind the edit box, based on the basic user-defined units.

### *Geometry dimensions*

At the start of a new project, the user needs to specify the dimensions of the draw area in such a way that the model that is to be created will fit within the dimensions. The initial setting of the $x_{min}$, $x_{max}$, $z_{min}$ and $z_{max}$ parameters set the outer horizontal boundaries of the geometry model. All these parameters are entered in the *Dimensions* tab sheet of the *General settings* window.

### *Grid*

To facilitate the creation of the geometry model, the user may define a grid for the draw area. This grid may be used to snap the pointer into certain 'regular' positions. The grid is defined by means of the parameters *Spacing* and *Number of intervals*. The *Spacing* is used to set up a coarse grid, indicated by the small dots on the draw area.

The actual grid is the coarse grid divided into the *Number of intervals*. The default number of intervals is 1, which gives a grid equal to the coarse grid. The grid specification is entered in the *Dimensions* tab sheet of the *General settings* window. The *View* sub-menu may be used to activate or deactivate the grid and snapping option.

## 3.3 GEOMETRY

The generation of a 3D finite element model begins with the creation of a geometry model. A geometry model is a composition of boreholes and horizontal work planes (*x-z* planes). The work planes are used to define geometry lines and structures. The boreholes are used to define the local soil stratigraphy, ground surface level and pore pressure distribution.

It is recommended to start the creation of a geometry model by defining all necessary work planes. In a work plane, geometry points, lines and area clusters can be created. Points and lines are entered by the user, whereas clusters are generated by the program.

In addition to these basic components, structural objects or special conditions can be assigned to the work planes to simulate beams, walls, floors, piles, soil-structure interaction or loadings. Work planes should not only include the initial situation, but also situations that arise in the various calculation phases.

After the geometry components in the various work planes have been defined, the user should create boreholes to define the soil stratigraphy, ground level and pore pressure distribution. During the definition of boreholes, data sets of model parameters for the various soil layers can be created and assigned to the borehole. In addition, data sets of model parameters for structural behaviour can be created and assigned to the structural objects in the work planes (Section 3.5).

When the full geometry model has been defined (including all objects appearing in any work plane at any construction stage) and all geometry components have their initial properties, the finite element mesh can be generated.

From the geometry model, a 2D mesh is generated first (Section 3.6). This 2D mesh can be optimised by global and local refinement, after which an extension into the third dimension (the *y*-direction) can be made. PLAXIS automatically generates this 3D mesh, taking into account the information from the work planes and the boreholes.

### *Selecting geometry components*

When the *Selection* tool (white arrow) is active, a geometry component may be selected by clicking once on that component in the work plane. Multiple components of the same type can be selected simultaneously by holding down the <Shift> key on the keyboard while selecting the desired components.

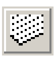
### *Properties of geometry components*

Most geometry components have certain properties, which can be viewed and altered in property windows. After double clicking a geometry component the corresponding property window appears. If more than one object is located on the indicated point, a selection dialog box appears from which the desired component can be selected.

### 3.3.1 WORK PLANES

Work planes are horizontal planes (*x-z* planes) at a certain vertical level (*y*-level) in which geometry points and lines and, in particular, structures and loads can be defined. At the start of a new project, a single initial work plane is automatically created at the level $y = 0$. The level of this work plane may be changed by the user and the user may also create additional work planes. If the geometry model includes piles, it is recommended to first create the work planes corresponding to the pile top and bottom level, then create the piles (Section 3.3.9) and then create other work planes and structural objects.

The outer boundaries of the work planes are based on the initial setting of the *xmin*, *xmax*, *zmin* and *zmax* parameters, as defined in the *General settings*. All work planes have the same outer boundaries. Moreover, if geometry points or lines are defined in any of the work planes, they will also appear in all other work planes. In this way, the structure of all work planes is similar.

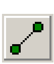


An overview of all work planes can be obtained by clicking on the *Work planes* button in the *General* toolbar. As a result, a *Work planes* window appears in which the *y*-levels of the existing work planes are listed in a table. The window also shows a simplified vertical cross section graph of the model in which the positions of all work planes are indicated. New work planes may be created using the *Add* or *Insert* buttons at the top of the window. Clicking the *Add* button introduces a new work plane above the upper work plane, taking into account an offset of 3 length units. This value may be changed by the user. Clicking the *Insert* button introduces a new work plane between the selected work plane and the one above by cutting the distance between them into two equal parts. This value may also be changed by the user. To change the position of a work plane, click in the table and type the required value. Existing work planes may be removed by selecting them (either in the table or in the graph) and clicking the *Delete* button. To leave the *Work planes* window, the *OK* button at the bottom of the window must be pressed.

When creating a wall (Section 3.3.6) at the lowest work plane, a new work plane is automatically introduced at a distance of 3 length units below the current work plane. This is because a wall can only be placed between two work planes in vertical direction.

All existing work planes are contained in the *Active work plane* combo box in the *General* toolbar. The combo box indicates the currently active work plane in blue. From the combo box any existing work plane may be selected (activated) for the purpose of creating or editing points or lines, defining structural objects or manipulating existing work plane settings. Selection of a work plane is done by clicking on the ▼ sign at the right side of the combo box and subsequently clicking on the desired work plane.

### 3.3.2 POINTS AND LINES



The basic input item for the creation of a geometry in a work plane is the *Geometry line*. This item can be selected from the *Geometry* sub-menu as well as from the *Model* toolbar. Geometry points or lines that are defined in a particular work plane are automatically copied to all other work planes.

When the *Geometry line* option is selected, the user may create points and lines in the active work plane by clicking with the mouse pointer (graphical input) or by typing coordinates at the command line (keyboard input). As soon as the left hand mouse button is clicked, a new point is created, provided that there is no existing point close to the pointer position. If there is an existing point close to the pointer, the pointer snaps into the existing point without generating a new point. After the first point is created, the user may draw a line by entering another point, etc.. The drawing of points and lines continues until the right hand mouse button is clicked at any position or the <Esc> key is pressed.

If a point is to be created on or close to an existing line, the pointer snaps onto the line and creates a new point exactly on that line. As a result, the line is split into two new lines. If a line crosses an existing line, a new point is created at the crossing of both lines. As a result, both lines are split into two new lines. If a line is drawn that partly coincides with an existing line, the program makes sure that over the range where the two lines coincide only one line is present. All these procedures guarantee that a consistent geometry is created without double points or lines.

Existing points or lines may be modified or deleted by first choosing the *Selection* tool from the toolbar. To move a point or line, select the point or the line in the work plane and drag it to the desired position. To delete a point or line, select the point or the line in the work plane and press the <Del> button on the keyboard. If more than one object is present at the selected position, a delete dialog box appears from which the object(s) to be deleted can be selected. If a point is deleted where two geometry lines come together that are in line with each other, then the two lines are combined to give one straight line between the outer points. If the two lines are not in line with each other or more than two geometry lines come together in the point to be deleted, then all these connected geometry lines will be deleted.

After each drawing action the program determines the area clusters that can be formed. A cluster is a closed loop of different geometry lines. In other words, a cluster is an area fully enclosed by geometry lines. The detected clusters are lightly shaded. The clusters are divided into soil elements during mesh generation (Section 3.6). Lines and clusters can be given certain properties to simulate structural behaviour (Section 3.3.3 and further) or loading conditions (Section 3.4).

At the start of a project, boundary lines and a single cluster are automatically generated based on the input of the $x_{min}$, $x_{max}$, $z_{min}$ and $z_{max}$ parameters in the General settings window. These boundary lines are simply geometry lines. When desired, the outer model boundary may be extended by moving these geometry lines or by creating new geometry lines outside the existing model boundary, provided that these lines form completed clusters and will thus form the new outer model boundary. It may be necessary to change the dimensions of the draw area in the *General settings* window to do this. Please note that it is not possible to create individual lines outside the outer model boundary.

If it is the user's intension to create a pile, beam, wall, line fixity, line load of distributed load on a vertical plane, it is not necessary to create a geometry line first, because when such an object is created using the corresponding button or menu option, a geometry line is automatically generated together with the object. If, on the other hand, the user wants to create a floor or distributed load on a horizontal plane, it is necessary to first create the corresponding cluster by means of geometry lines.

### 3.3.3 HORIZONTAL BEAMS

Horizontal beams are structural objects used to model slender (one-dimensional) structures in the ground with a significant flexural rigidity (bending stiffness) and an axial stiffness. Horizontal beams coincide with the active work plane. Hence, before the creation of a horizontal beam, the appropriate work plane needs to be selected from the *Active work plane* combo box.

Horizontal beams can be selected from the *Geometry* sub-menu or by clicking on the corresponding button in the *Model* toolbar. The creation of a horizontal beam in a work plane is similar to the creation of a geometry line (Section 3.3.2), but the cursor has a different shape. A horizontal beam is indicated by a purple line. Beams that do not have a material data set assigned have a light purple colour, whereas beams with an assigned data set have a dark purple colour. When creating horizontal beams, corresponding geometry lines are created simultaneously. These geometry lines appear in all work planes, whilst the horizontal beam only appears in the active work plane.

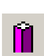
#### *Horizontal beam elements*

Horizontal beams are composed of 3-node line elements (beam elements) with six degrees of freedom per node: Three translational degrees of freedom ($u_x$, $u_y$ and $u_z$) and three rotational degrees of freedom ($\phi_x$, $\phi_y$, $\phi_z$). Beam forces are numerically integrated from the four Gaussian integration points (stress points). Details about the element formulation are given in the Scientific Manual. The beam elements are based on Mindlin's beam theory. This theory allows for beam deflections due to shearing as well as bending. In addition, the element can change length when an axial force is applied.

#### *Beam properties*

The material properties of beams are contained in material data sets and can be conveniently assigned using drag-and-drop (Section 3.5.6). The basic geometry parameters include the cross-section area $A$, and the unit weight of the beam material $\gamma$. Distinct moments of inertia can be specified for bending in horizontal and vertical direction. As an alternative for the linear elastic properties, non-linear elastic properties may be specified by means of ($M$-$\kappa$) and ($N$-$\varepsilon$) diagrams. Structural forces are evaluated from the stresses at the stress points (see Scientific manual) and extrapolated to the element nodes. These forces can be viewed graphically and tabulated in the Output program.

### 3.3.4 VERTICAL BEAMS



Vertical beams are structural objects used to model slender (one-dimensional) structures in the ground with a significant flexural rigidity (bending stiffness) and an axial stiffness. Vertical beams are located between the active work plane and the next work plane below the current one.

Hence, before the creation of a vertical beam, work planes need to be created corresponding with the top and the bottom of the beam. In addition, the work plane at the upper side of the beam needs to be selected from the *Active work plane* combo box. The vertical beam can then be created on this work plane. If a vertical beam is created on the lowest available work plane, a new work plane will automatically be introduced at a distance of 3 length units below this work plane.

Vertical beams can be selected from the *Geometry* sub-menu or by clicking on the corresponding button in the *Model* toolbar. The creation of a vertical beam in a work plane is similar to the creation of a geometry point (Section 3.3.2), but the cursor has a different shape. A vertical beam is indicated by a symbol in the shape of a capital I. Beams that do not have a material data set assigned have a light purple colour, whereas beams with an assigned data set have a dark purple colour.

The program does not permit the creation of single, unconnected geometry points. All geometry points should be part of a line. As a result, vertical beams can only be added to existing geometry points or be created on existing lines. When creating vertical beams on existing lines, corresponding geometry points are created simultaneously. These geometry points appear in all work planes, whilst the vertical beam only appears in the active work plane.

### *Vertical beam elements*

Vertical beams are composed of 3-node line elements (beam elements) with six degrees of freedom per node: Three translational degrees of freedom ($u_x$, $u_y$ and $u_z$) and three rotational degrees of freedom ($\phi_x$, $\phi_y$, $\phi_z$). Beam forces are numerically integrated from the four Gaussian integration points (stress points).

Details about the element formulation are given in the Scientific Manual. The beam elements are based on Mindlin's beam theory. This theory allows for beam deflections due to shearing as well as bending. In addition, the element can change length when an axial force is applied.

### *Beam properties*

The material properties of beams are contained in material data sets and can be conveniently assigned using drag-and-drop (Section 3.5.6), similar to horizontal beams. The basic geometry parameters include the cross-section area $A$, and the unit weight of the beam material $\gamma$. Distinct moments of inertia can be specified for bending in horizontal and vertical direction.

As an alternative for the linear elastic properties, non-linear elastic properties may be specified by means of ($M$-$\kappa$) and ($N$-$\varepsilon$) diagrams. Structural forces are evaluated from the stresses at the stress points (see Scientific manual) and extrapolated to the element nodes. These forces can be viewed graphically and tabulated in the Output program.

### 3.3.5 FLOORS

Floors are structural objects used to model thin horizontal (two-dimensional) structures in the ground with a significant flexural rigidity (bending stiffness). Floors coincide with the active work plane and extend over a full cluster. Before the creation of a floor, the corresponding contour needs to be created using geometry lines (Section 3.3.2). These geometry lines appear in all work planes. Hence, before the creation of the floor, the appropriate work plane needs to be selected from the *Active work plane* combo box.

To make the cluster into a floor at the active work plane, select the *Floor* option from the *Geometry* sub-menu or click on the corresponding button in the *Model* toolbar. Move the cursor (now indicating that a floor is being created) to the corresponding cluster and click once. As a result, the cluster changes into a floor, indicated by a green colour (olive). Floors that do not have a material data set assigned have a light green colour, whereas floors with an assigned data set have a dark green colour.

In contrast to walls, there are no interface elements generated along floors.

#### *Floor elements (plate elements)*

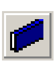
Floors are composed of 6-node triangular plate elements with six degrees of freedom per node: Three translational degrees of freedom ($u_x$, $u_y$ and $u_z$) and three rotational degrees of freedom ($\phi_x$, $\phi_y$, $\phi_z$). Element stiffness matrices and plate forces are numerically integrated from the 3 Gaussian integration points (stress points). Details about the element formulation are given in the Scientific Manual. The plate elements are based on Mindlin's plate theory. This theory allows for plate deflections due to shearing as well as bending. In addition, the element can change length when an axial force is applied.

#### *Floor properties*

The material properties of floors are contained in material data sets and can be conveniently assigned using drag-and-drop (Section 3.5.7). The basic geometry parameters include the thickness $d$, and the unit weight of the floor material $\gamma$. Distinct stiffnesses can be specified for the different floor directions. As an alternative for the linear elastic properties, non-linear elastic properties may be specified by means of ($M$-$\kappa$) and ($N$-$\varepsilon$) diagrams.

Structural forces are evaluated from the stresses at the stress points (see Scientific Manual) and extrapolated to the element nodes. These forces can be viewed graphically and tabulated in the Output program.

### 3.3.6 WALLS



Walls are structural objects used to model thin vertical (two-dimensional) structures in the ground with a significant flexural rigidity (bending stiffness). Walls are located between the active work plane and the next work plane below the current one. Hence, before the creation of a wall, work planes need to be created corresponding with the top and the bottom of the wall. In addition, the work plane at the upper side of the wall needs to be selected from the Active work plane combo box. The wall can then be created on this work plane. If a wall is created on the lowest available work plane, a new work plane will automatically be introduced at a distance of 3 length units below this work plane.

Walls can be selected from the *Geometry* sub-menu or by clicking on the corresponding button in the *Model* toolbar. The creation of a wall in a work plane is similar to the creation of a geometry line (Section 3.3.2), but the cursor has a different shape. A wall is indicated by a blue line. Walls that do not have a material data set assigned have a light blue colour, whereas walls with an assigned data set have a dark blue colour. When creating walls, corresponding geometry lines are created simultaneously. These geometry lines appear in all work planes, whilst the wall is only created and visualised in the active work plane (representing a wall between the active work plane and the work plane below). Moreover, when creating walls, corresponding interfaces are automatically generated at both sides of the wall to allow for proper soil-structure interaction.

### *Wall elements (plate elements)*

Walls are composed of 8-node quadrilateral plate elements with six degrees of freedom per node: Three translational degrees of freedom ($u_x$, $u_y$ and $u_z$) and three rotational degrees of freedom ($\phi_x$, $\phi_y$, $\phi_z$). Along degenerated soil elements, walls are composed of 6-node triangular plate elements, compatible with the triangular side of the degenerated soil element. Element stiffness matrices and plate forces are numerically integrated from the 2x2 (or 3 for triangular plate elements) Gaussian integration points (stress points). Details about the element formulation are given in the Scientific Manual.

The plate elements are based on Mindlin's plate theory. This theory allows for plate deflections due to shearing as well as bending. In addition, the element can change length when an axial force is applied.

### *Wall properties*

The material properties of walls are contained in material data sets and can be conveniently assigned using drag-and-drop (Section 3.5.7). The basic geometry parameters include the thickness *d*, and the unit weight of the wall material $\gamma$. Distinct stiffnesses can be specified for different wall directions. As an alternative for the linear elastic properties, non-linear elastic properties may be specified by means of ($M$-$\kappa$) and ($N$-$\varepsilon$) diagrams. Structural forces are evaluated from the stresses at the stress points (see Scientific Manual) and extrapolated to the element nodes. These forces can be viewed graphically and tabulated in the Output program.

## 3.3.7 INTERFACE ELEMENTS

Interfaces are composed of 16-node interface elements. Interface elements consist of eight pairs of nodes, compatible with the 8-noded quadrilateral side of a soil element.

Along degenerated soil elements, interface elements are composed of 6 node pairs, compatible with the triangular side of the degenerated soil element. In some 2D output plots, interface elements are shown to have a finite thickness, but in the finite element formulation the coordinates of each node pair are identical, which means that the element has zero thickness.

However, each interface is assigned a 'virtual thickness' which is an imaginary dimension used to calculate the stiffness properties of the interface. The virtual thickness is defined as the *Virtual thickness factor* times the average element size. The average element size is determined by the global coarseness setting for the 2D mesh generation (Section 3.6.3). The default value of the *Virtual thickness factor* is 0.1. This value cannot be changed by the user. Further details of the relevance of the virtual thickness are given in Section 3.5.4.

The stiffness matrix for quadrilateral interface elements is obtained by means of Gaussian integration using 3x3 integration points. The position of these integration points (or stress points) is chosen such that the numerical integration is exact for linear stress distributions. For more details about the element formulation reference is made to the Scientific Manual.

At wall ends (both in horizontal direction and in vertical direction) interface element node pairs are 'degenerated' to single nodes. There are no interface elements beyond the wall. Also when walls are connected to floors or horizontal beams, interface element node pairs are locally 'degenerated' to single nodes to avoid a disconnection between the wall and the floor or beam.

### *Interface properties*

A typical application of interfaces would be to model the interaction between a pile or basement wall and the soil, which is intermediate between smooth and fully rough. The roughness of the interaction is modelled by choosing a suitable value for the strength reduction factor in the interface ($R_{inter}$). This factor relates the interface strength (pile friction or wall friction, and adhesion) to the soil strength (friction angle and cohesion).

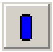
For detailed information on the interface properties, see Section 3.5.4.

### 3.3.8 CONNECTIONS OF STRUCTURAL ELEMENTS

Structural elements (beams, floors and walls) have rotational degrees of freedom ($\phi_x$, $\phi_y$, $\phi_z$) in addition to the translational degrees of freedom ($u_x$, $u_y$, $u_z$). When such elements are connected (i.e. when they share at least one geometry point), they will use the same degrees of freedom in these connection points. This applies to the translational degrees of freedom as well as the rotational degrees of freedom. As a result, the connection between these elements is rigid (moment connection).

When floors or horizontal beams are connected to walls, the node pairs of the interface element adjacent to the wall are locally 'degenerated' to a single node to avoid a disconnection between the wall and the floor or beam.

### 3.3.9 PILES



The pile option can be used to create volumetric piles with a circular, square or user-defined cross-section. A pile cross-section is composed of arcs and/or lines, optionally supplied with a tube (consisting of shell elements) and/or interfaces. The pile option is available from the *Geometry* sub-menu or from the *Model* toolbar.

Before the creation of a pile it is necessary that work planes corresponding to the top and bottom of the pile have been created (Section 3.3.1). When piles are present in the model, it is advised that these are created before other work planes or structural objects are created.

#### *Pile designer*

Once the pile option has been selected, the *Piles* window (pile designer) appears.

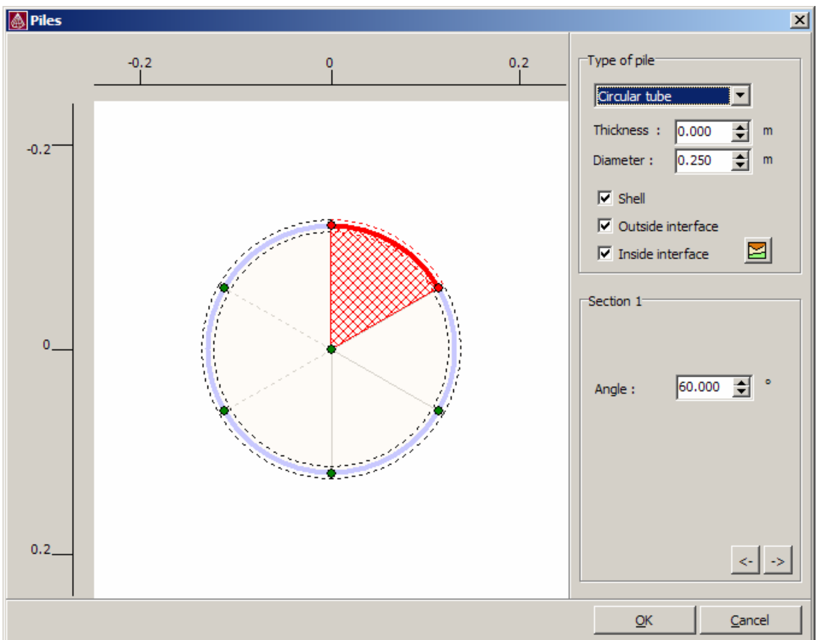



Figure 3.4 Pile designer with standard pile shape

The pile designer contains the following items (Figure 3.4):

*Display area:* Area in which the pile cross-section is plotted.

*Rulers:* The rulers indicate the dimension of the pile cross-section in local coordinates. The origin of the local ($x'$-$z'$) system of axes is used as a reference point for the positioning of the pile in the geometry model.

*Type of pile group box:* Box containing parameters to set the basic pile cross-section.

*Section group box:* Box containing shape parameters and attributes of individual pile sections.

*Material sets button:* To open the material database in which material data sets may be created and assigned to the pile.

*Standard buttons:* To accept (OK) or to cancel the created pile cross-section.

### *Type of pile*

There are five basic pile types that can be selected: Massive circular pile, Circular tube, Massive square pile, Square tube, User-defined pile.

*Massive circular pile:* This option can be used to create a massive circular pile composed of volume elements with an (optional) interface at the outside of the pile. A shell cannot be added. The pile diameter can be specified by means of the *Diameter* parameter.

*Circular tube:* This option can be used to create a cylindrical pile (tube) composed of shell (wall) elements with (optional) interfaces at both sides of the shell. Optionally, a non-zero thickness can be specified to create a thick shell composed of volume elements. The pile inner diameter can be specified by means of the *Diameter* parameter.

*Massive square pile:* This option can be used to create a massive square pile composed of volume elements with an (optional) interface at the outside of the pile. A shell cannot be added. The pile width can be specified by means of the *Width* parameter.

*Square tube:* This option can be used to create a hollow square pile composed of shell (wall) elements with (optional) interfaces at both sides of the shell. Optionally, a non-zero thickness can be specified to create a thick shell composed of volume elements. The pile inner width can be specified by means of the *Width* parameter.

*User-defined shape:* This option can be used to create foundation structures or geometry shapes that are composed of arcs and lines. A shell (wall) and interfaces may be added at individual sections.

In case of a circular or square tubular pile, if a positive value for the *Thickness* parameter is entered, the pile outline or shell will consist of two lines at a distance given by the *Thickness* parameter. The two lines will form separate clusters when inserting the pile in the geometry model. By default, a circular pile consists of six sections of 60 degrees and a square pile consists of eight sections of 45 degrees. The number of sections can only be changed for user-defined piles. This ensures that the pile cross section is composed of at least six elements for reasons of accuracy.

Please note that a further mesh refinement does not depend on the number of sections, but may be achieved by means of the 2D mesh refinement options.

### *User-defined shape*

A user-defined shape is supposed to be symmetric and composed of different sections. The right half can be defined by the user whereas the left half is the mirror image of the right half. Each section outline is either an *Arc* (part of a circle, defined by a *Centre point*, a *Radius* and an *Angle*), or a *Line* increment (defined by a *Start point* and a *Length*). In addition, sharp corners can be defined, i.e. a sudden transition in the tangent of two adjacent pile sections.

The first section starts with a 'horizontal' tangent at the 'upper' point in the graph, and runs in clockwise direction. The position of this first start point is determined by the *Centre* coordinates and the *Radius* (if the first section is an *Arc*) or by the start point coordinates (if the first section is a *Line*). The end point of the first section is determined by the *Angle* (in case of an arc) or by the *Length* (in case of a line). The start point of a next section coincides with the end point of the previous section.

The start tangent of the next section is equal to the end tangent of the previous section. In case both sections are arcs, the two sections have the same radial, but do not necessarily have the same radius (Figure 3.5).

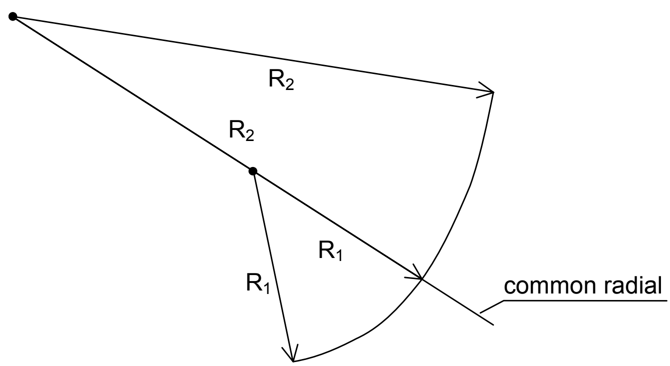



Figure 3.5  Detail of connection point between two pile sections

Hence, the centre point of the next section is located on this common radial and the exact position follows from the section radius. If the tangent of the pile outline in the connection point is discontinuous, a sharp corner may be introduced by selecting *Corner* for the next section. In this case a sudden change in the tangent can be specified by the *Angle* parameter.

The radius and the angle of the last section are automatically determined such that the end tangent is 'horizontal' at the 'lowest' point in the pile designer.

The number of sections follows from the sum of the section angles. Since the cross-section is assumed to be symmetric, the sum of all definable section angles is 180 degrees (half a cross-section). The maximum angle of a section is 90.0 degrees. The automatically calculated angle of the last section completes the cross-section. If this angle is decreased, a new section will be created. If the angle of an intermediate section is decreased, the angle of the last section is increased by the same amount, until the maximum angle is reached. Upon further reduction of the intermediate section angle a new section will be created. If the angle of one of the intermediate tunnel sections is increased, the angle of the last tunnel section is automatically decreased. This may result in elimination of the last section.

### *Assigning material properties*

After creating the pile cross section, material properties may directly be assigned to the pile. This can be done by pressing the *Material sets* button. As a result, the material database appears where material data sets may be created. From the material database, material sets may be assigned to the corresponding object in the pile designer (volume cluster or shell section).

To assign a data set, select the appropriate data set from the material database tree view (click on the data set and hold the right hand mouse button down), drag it to the graph in the pile designer (hold the mouse button down while moving) and drop it on the desired object (release the mouse button)

### *Including pile in geometry model*

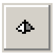
After pressing the <OK> button in the pile designer the window is closed and the main input window is displayed again. Before the pile is included in the geometry model, the work plane at the top of the pile must be selected from the *Active work plane* combo box. A pile symbol is attached to the cursor to emphasize that the reference point for the pile must be selected. The reference point will be the point where the origin of the local pile coordinate system is located in the active work plane. The reference point can entered by clicking with the mouse in the geometry model or by entering the coordinates in the manual input line.

As a result, the pile geometry is included in the work plane. If the pile cross-section includes a shell, the shell is modelled by wall elements. These wall elements are only present between the active work plane and the next work plane below. If the pile extends over more work planes, the same pile cross-section must be defined in all work planes, except for the lowest one. It is advised that the work planes corresponding to the top and bottom of the pile are created first, followed by the creation of the pile cross-section, before other work planes or structural objects are created.

### *Editing an existing pile*

An existing pile can be edited by double clicking its reference point or one of the other pile points. As a result, the pile designer reappears showing the existing pile cross-section. Desired modifications can now be made. On clicking the <OK> button, the 'old' pile is removed and the 'new' pile is directly included in the geometry model using the original reference point. Note that previously assigned material sets of a shell (wall) must be reassigned after modification of the pile. If a pile extends over more work planes, the pile must be edited in each of the corresponding work planes

### 3.3.10 SPRINGS



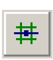
A *Spring* is a spring element that is attached to a structure at one side and fixed 'to the world' at the other side. Springs can be used to simulate piles in a simplified way, i.e. without taking into account pile-soil interaction. Alternatively, springs can be used to simulate anchors or props to support retaining walls. Springs can only be attached to structural objects in a work plane.

Springs coincide with the active work plane. Hence, before the creation of a spring, the appropriate work plane needs to be selected from the *Active work plane* combo box. When creating springs, corresponding geometry points are created simultaneously. These geometry points appear in all work planes, whilst the spring itself only appears in the active work plane.

Springs can be selected from the *Geometry* sub-menu or by clicking on the corresponding button in the *Model* toolbar. A spring can only be connected to existing geometry lines of structural objects (beams, floors, walls). The creation of a spring in a work plane is similar to the creation of a geometry point (Section 3.3.2), but the cursor has a different shape. A spring is indicated by a blue square.

After the spring has been created, a *Spring properties* window appears in which the spring material data set and the spring direction needs to be entered. The user may enter the spring direction by specifying the individual *x*-, *y*- and *z*- components. The default direction is (0, -1, 0), which is in downward direction. The components all together do not need to form a vector of unit length. The material set may be selected by pressing the *Change* button. As a result, the material database is opened in which material sets can be created and assigned to the spring. A spring material data set contains the spring stiffness divided by the effective length (Section 3.5.9). It is also possible to specify a non-linear elastic spring stiffness through a ($F$, $u$)-diagram to simulate non-linear spring behaviour.

### 3.3.11 HORIZONTAL LINE FIXITIES



Line fixities can be used to fix parts of the model in *x*-, *y*- and *z*-direction. Horizontal line fixities coincide with the active work plane. Hence, before the creation of a horizontal line fixity, the appropriate work plane needs to be selected from the *Active work plane* combo box.

Horizontal line fixities can be selected from the *Geometry* sub-menu or by clicking on the corresponding button in the *Model* toolbar. The creation of a horizontal line fixity in a work plane is similar to the creation of a geometry line (Section 3.3.2), but the cursor has a different shape. A horizontal line fixity is indicated by a green line, with two parallel lines perpendicular to each fixed direction. When creating horizontal line fixities, corresponding geometry lines are created simultaneously. These geometry lines appear in all work planes, whilst the horizontal line fixity only appears in the active work plane.

By default, line fixities will fix the corresponding geometry points in *x*-, *y*- and *z*-direction by imposing prescribed displacement components equal to zero. However, some of these components may be set free. By double-clicking the corresponding geometry line and selecting the horizontal line fixity from the *Select* window, the *Horizontal line fixity* window appears. In this window it can be indicated which direction has to be set free by clicking on the appropriate check box (*x*-, *y*- or *z*-direction). On a geometry line where fixities are used as a condition, the fixities have priority over loading conditions during the calculations.

### 3.3.12 VERTICAL LINE FIXITIES

Line fixities can be used to fix parts of the model in *x*-, *y*- and *z*-direction. Vertical line fixities are located between the active work plane and the next work plane below the current one. Hence, before the creation of a vertical line fixity, work planes need to be created corresponding with the top and the bottom of the line fixity. In addition, the work plane at the upper side of the line fixity needs to be selected from the *Active work plane* combo box. The vertical line fixity can then be created on this work plane. If a vertical line fixity is created on the lowest available work plane, a new work plane will automatically be introduced at a distance of 3 length units below this work plane.

Vertical line fixities can be selected from the *Geometry* sub-menu or by clicking on the corresponding button in the *Model* toolbar. The creation of a line fixity in a work plane is similar to the creation of a geometry point (Section 3.3.2), but the cursor has a different shape. A vertical line fixity is indicated by a green square, with two parallel red lines perpendicular to each fixed direction.

The program does not allow the creation of single, unconnected geometry points. All geometry points should minimally be part of a line. As a result, vertical line fixities can only be added to existing geometry points or be created on existing lines. When creating vertical line fixities on existing lines, corresponding geometry points are created simultaneously. These geometry points appear in all work planes, whilst the vertical line fixity only appears in the active work plane.

By default, line fixities will fix the corresponding geometry points in *x*-, *y*- and *z*-direction by imposing prescribed displacement components equal to zero. However, some of these components may be set free. By double-clicking the corresponding geometry line and selecting the vertical line fixity from the *Select* window, the *Vertical line fixity* window appears. In this window it can be indicated which direction has to be set free by clicking on the appropriate check box (*x*-, *y*- or *z*-direction). On a geometry line where fixities are used as a condition, the fixities have priority over loading conditions during the calculations.

### 3.3.13 STANDARD BOUNDARY FIXITIES

PLAXIS automatically imposes a set of general fixities to the boundaries of the geometry model. These conditions are generated according to the following rules:

- Vertical model boundaries with their normal in *x*-direction (i.e. parallel to the *y*-*z*-plane) are fixed in *x*-direction ($u_x = 0$) and free in *y*- and *z*-direction.
- Vertical model boundaries with their normal in *z*-direction (i.e. parallel to the *x*-*y*-plane) are fixed in *z*-direction ($u_z = 0$) and free in *x*- and *y*-direction.
- Vertical model boundaries with their normal neither in *x*- nor in *z*-direction (skew boundary lines in a work plane) are fixed in *x*- and *z*-direction ($u_x = u_z = 0$) and free in *y*-direction.
- The model bottom boundary is fixed in all directions ($u_x = u_y = u_z = 0$).
- The 'ground surface' of the model is free in all directions.

### 3.3.14 BOREHOLES

Boreholes are used to define the soil stratigraphy and ground surface level. Soil layers and ground surface may be non-horizontal by using several boreholes at different locations. Moreover, boreholes are used to define the pore pressure distribution in the sub-soil and can be used to define the initial stress conditions in the soil. The borehole option is available from the *Geometry* sub-menu or from the *Model* toolbar.

Once the borehole option has been selected and a borehole has been created in the geometry, the borehole input window appears. Existing boreholes can be entered by double clicking a borehole in the geometry using the selection tool (arrow).

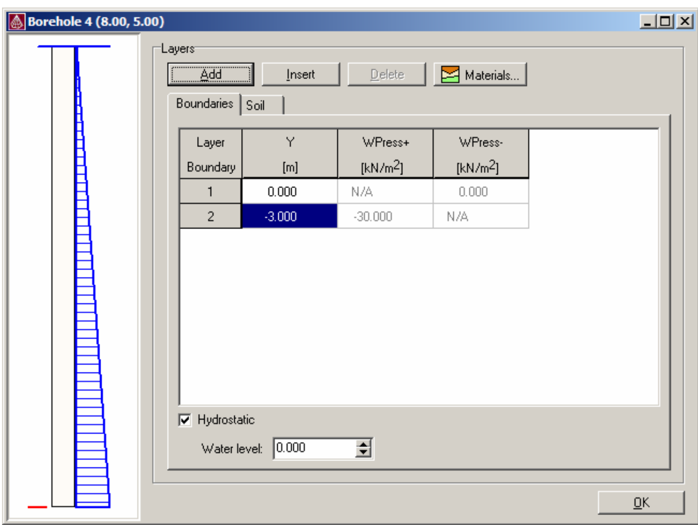



Figure 3.6 Borehole window

The borehole window contains the following items (Figure 3.6):

| | |
|---|---|
| *Soil column:* | Graph of the borehole with indication of soil surface, layer boundary levels, water level and pore pressure distribution. |
| *Upper buttons:* | Buttons to add, insert or delete soil layer levels or to open the material database. |
| *Boundaries tab sheet:* | Table with soil layer boundary levels ($y$) and pore pressures just above (+) and just below (-) these levels. |
| *Water section:* | Information about the type of pore pressure distribution and water level. |
| *Soil tab sheet:* | Table with soil layer *Name, Material model* and initial stress condition parameters, $K_{0,x}$ $K_{0,z}$, *POP* and *OCR*. |
| *Standard buttons:* | To accept (*OK*) or to cancel the created borehole. |

### *Soil layers*

The first borehole that is created at the start of a new project contains a single soil layer defined by two layer boundary levels. The upper layer boundary level corresponds to the ground surface and the lowest layer boundary level corresponds to the bottom of the model. The upper level is, by default, located at $y$=0 and the bottom level corresponds with the lowest work plane, provided that this work plane is below the upper level; otherwise it is located 3 length units under the upper level.

New layers may be created using the *Add* or *Insert* buttons. Clicking the *Add* button introduces a new layer below the lowest work plane and thus a new layer boundary below the lowest level. By default, this level is 3 length units below the current lowest layer boundary level. Clicking the *Insert* button introduces a new layer by cutting the layer between the selected level and the level above into two equal parts. Clicking the *Insert* button while the upper layer boundary is selected, introduces a new layer above the top work plane. By default this level is 3 length units above the top work plane level. The layer boundary levels (including top and bottom level) and thus the layer thickness can be changed by the user on the *Boundaries* tab sheet. Existing layer boundaries may be removed by selecting the corresponding level in the table and clicking the *Delete* button. However, it should be noted that this will not only delete the layer boundary at the current borehole but also the corresponding layer boundaries at all other boreholes. As an alternative to the *Delete* option, layers may be eliminated locally by setting the upper level equal to the lower level. The 3D mesh generator will recognize this and automatically eliminate element layers with a zero thickness.

Once a borehole has been defined, the next borehole that is created will be a copy of the previous borehole. In the new borehole, layer boundaries and pore pressures may be changed by the user to create a non-horizontal soil stratigraphy or phreatic level. If the new borehole should contain a soil layer that does not exist in previous boreholes, then this additional soil layer may be added using the *Add* or *Insert* button.

In principle, this action creates a new soil layer in all existing boreholes, but the new layer has a zero thickness in all boreholes that have been defined earlier to ensure that existing layer distributions are not influenced by this action. The thickness of the new layer in the current borehole is according to the description given in the previous paragraph. If a certain layer, appearing in one of the other boreholes, does not appear in the current borehole, it should be eliminated by setting the corresponding upper and lower boundary levels equal (zero layer thickness). The *Delete* option should NOT be used in this case to avoid elimination of this layer in other boreholes where it does exist.

During mesh generation, the position of the soil layers and ground surface in between the boreholes is interpolated.

### *Assigning soil properties*

Different soil layers will have different properties. Individual layer properties can be defined in material data sets. Material data sets are contained in the material database. The material database can be entered by clicking on the material sets button at the upper right hand side of the borehole window. As a result, the material database appears from which data sets can be selected and assigned to the borehole (see Section 3.5 for more information about the material database and the creation of material data sets). To assign a data set, select the desired data set from the material database tree view (click on the data set and hold the left hand mouse button down), drag it to the soil column in the borehole window (hold the mouse button down while moving) and drop it on the desired layer (release the mouse button). The layer should now show the corresponding material data set colour. The drag and drop procedure should be repeated until all layers have their appropriate data set. The names and colours of the material data sets for all layers are also shown in the *Soil* tab sheet.

When using multiple boreholes it should be noted that assigning a new data set to an existing layer in one particular borehole will also influence the other boreholes, since all layers appear in all boreholes, except for layers with a zero thickness.

### *Water conditions*

In addition to the definition of soil layers, the pore pressure distribution is to be specified in the borehole window. The table of layer boundary levels in the *Boundaries* tab sheet includes two additional colums, one containing the pore pressure distribution above the layer boundary (WPress+) and one containing the pore pressure distribution below the layer boundary (WPress−). Pore pressures as well as external water pressures (if the water level is above the ground surface) are generated on the basis of this information. Please note that pressures are considered to be negative.

If the pore pressure distribution is hydrostatic, it can simply be generated on the basis of a water level (phreatic level). Therefore, the *Hydrostatic* parameter must be checked and the appropriate *Water level* must be entered in the corresponding edit box of the *Boundaries* tab sheet. The water pressures at the layer boundaries are automatically calculated from the water level and the unit water weight as entered in the *General settings* window. The values are presented in the table, but these values cannot be edited as long as the Hydrostatic parameter is checked. In case the water level is above the top layer boundary (i.e. above the ground surface) external water pressures will generated.

If the pore pressure distribution is not hydrostatic, the *Hydrostatic* parameter must be unchecked. With this setting, the water pressures at the layer boundaries can be entered manually in the table. Distinction can be made between water pressures above the layer boundary (WPress+) and water pressures below the layer boundary (WPress–). When entering a non-zero value (pressure is negative!) for WPress+ at the top layer boundary (ground surface), this pressure is interpreted as an external water pressure.

***Initial stresses***

The initial stresses in a soil body are influenced by the weight of the soil, the water conditions and the history of its formation. This stress state can be generated using the *K0 procedure* or using *Gravity loading* (see Section 4.1.6). If the *K0 procedure* is used, proper $K_0$ values need to be specified for all layers. Moreover, when advanced soil models are used, information is needed about the preconsolidation stress level. The latter can be specified by means of *OCR* and/or *POP* values (see Section 2.8 of the Material Models manual). The *Soil* tab sheet offers a table where these parameters can be entered. The same parameters can also be entered in the *Parameters* tab sheet of the *Phases* window of the Calculation program.

The meaning of $K_0$, *OCR* and *POP*, as well as the steps needed to perform a *K0 procedure*, are described in Section 4.1.6. By default, $K_0$ is calculated from Jaky's formula $K_0 = 1 - \sin\varphi$, where $\varphi$ is the friction angle from the corresponding material data set. For overconsolidated stress states in advanced models the default value of $K_0$ is higher, see Section 4.1.6 for details.

Please note that material data sets must be assigned to all soil layers before the parameters in the *Soil* tab sheet can be entered.

## 3.4 LOADS

The *Loads* sub-menu contains the options to introduce distributed loads, line loads and point loads in the geometry model. Distributed loads can be divided into loads on a horizontal plane and loads on a vertical plane. For line loads distinction is made between horizontal and vertical line loads.

### 3.4.1 DISTRIBUTED LOADS ON HORIZONTAL PLANES

A distributed load on a horizontal plane can be used to model an equally distributed load that acts on a geometry cluster or a floor. Distributed loads on horizontal planes coincide with the active work plane and extend over a full cluster. Hence, before the creation of such a load, the appropriate work plane needs to be selected from the *Active work plane* combo box.

Prior to the actual creation of the load, a cluster has to be generated by drawing geometry lines along the area where the distributed load is to be put (Section 3.3.2). These geometry lines and clusters will appear in all work planes. Alternatively, existing clusters may be cut into separate clusters or fully used for distributed loads. Also clusters that have been created for the purpose of creating a floor may be loaded by a distributed load.

To make the cluster into a loaded area at the active work plane, select the *Distributed load (horiz. plane)* option from the *Geometry* sub-menu or click on the corresponding button in the *Model* toolbar. Move the cursor (now indicating that a distributed load is to be created) to the corresponding cluster and click once. As a result, the cluster will be indicated as a blue cross-hatched area.

The input values of a distributed load are given in force per area (for example kN/m$^2$). Distributed loads may consist of a *x*-, *y*- and/or *z*-component. By default, when applying a distributed load to a work plane, the load will be a unit pressure in vertical direction. The input value of a load may be changed by double clicking in the corresponding cluster and, if needed, selecting the distributed load from the selection dialog box. As a result, the distributed loads window appears in which the three components of the load can be specified (Figure 3.7).

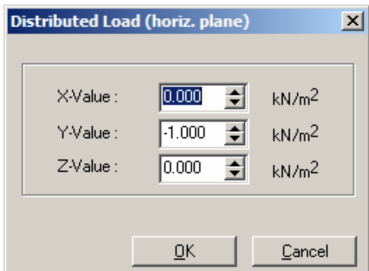
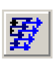



Figure 3.7 Input window for distributed loads on horizontal planes

Although the input values of distributed loads are specified in the geometry model, the activation, deactivation or change of loads may be considered in the framework of *Staged construction*.

### 3.4.2 DISTRIBUTED LOADS ON VERTICAL PLANES



This type of load can be used, for example, to model a wind load on a building facade. The distributed load acts on a vertical plane between the active work plane and the work plane below. Hence, before the creation of such a load, the appropriate work plane (at the upper side of the load) needs to be selected from the *Active work plane* combo box. Please note that it is not possible to create these distributed loads from the bottom work plane.

Distributed loads on a vertical plane can be selected from the *Loads* sub-menu or by clicking on the corresponding button in the *Model* toolbar. The creation of a distributed load in a work plane is similar to the creation of a geometry line (Section 3.3.2), but the cursor has a different shape. A distributed load is indicated by blue arrows. When creating distributed loads, corresponding geometry lines are created simultaneously (if they do not yet exist). These geometry lines appear in all work planes, whilst the distributed load is only visualised in the active work plane.

The input values of a distributed load are given in force per area (for example kN/m$^2$). Distributed loads may consist of an *x*-, *y*- and/or *z*-component. By default, the load will be a unit pressure perpendicular to the vertical plane on which the load is applied.

The input value of a load may be changed by double clicking on the corresponding geometry line and, if needed, selecting the distributed load from the selection dialog box. As a result, the distributed loads window appears in which the three components of the load can be specified in four different points, i.e. two points of the corresponding geometry line in the active work plane and two geometry points in the work plane below (Figure 3.8).

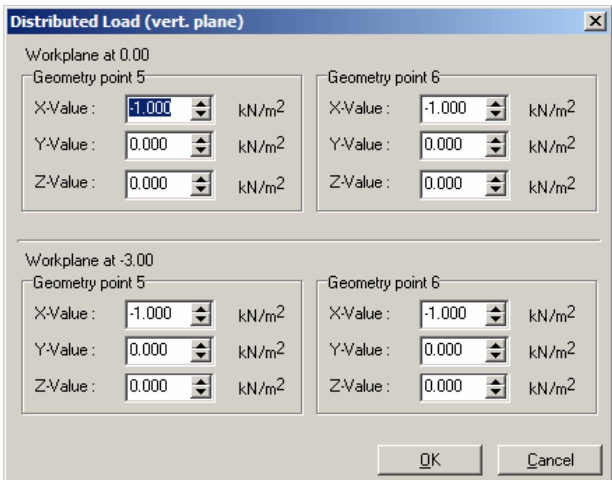
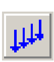



Figure 3.8 Input window for distributed loads on vertical planes

Although the input values of distributed loads are specified in the geometry model, the activation, deactivation or change of loads may be considered in the framework of *Staged construction*.

### 3.4.3 HORIZONTAL LINE LOADS



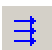
This option may be used to create line loads in a work plane. The creation of a horizontal line load is similar to the creation of a geometry line (Section 3.3.2), but the cursor has a different shape. Hence, before the creation of such a load, the appropriate work plane needs to be selected from the *Active work plane* combo box. Horizontal line loads can be selected from the *Loads* sub-menu or by clicking on the corresponding button in the toolbar.

The input values of a horizontal line load are given in a force per unit of length. Horizontal line loads may consist of an *x*-, *y*- and/or *z*-component. By default, when applying horizontal line loads, the load will be one unit in the negative *y*-direction. The input values of the load may be changed by double clicking the corresponding geometry line and, if needed, selecting the load from the selection dialog box. As a result, the line load window is opened, showing the three components of the load in two different points, i.e. the two points of the corresponding geometry line in the active work plane.

On a part of the geometry where both line fixities and line loads are applied, the fixities have priority over the line loads during the calculations. Hence, it is not useful to apply line loads on a fixed line. However, when only one displacement direction is fixed whilst the other direction is free, it is possible to apply a load in the free direction.

Although the input values of line loads are specified in the geometry model, the activation, deactivation or change of loads may be considered in the framework of *Staged construction*.

### 3.4.4 VERTICAL LINE LOADS



This option may be used to create line loads in a vertical direction. Vertical line loads are located between the active work plane and the next work plane below the current one. Hence, before the creation of a vertical line load, work planes need to be created corresponding with the top and the bottom of the load. In addition, the work plane at the upper side of the beam needs to be selected from the Active work plane combo box. The vertical load can then be created on this work plane. If a vertical line load is created on the lowest available work plane, a new work plane will automatically be introduced at a distance of 3 length units below this work plane.

Vertical line loads can be selected from the *Loads* sub-menu or by clicking on the corresponding button in the toolbar. The creation of a vertical line load is similar to the creation of a geometry point (Section 3.3.2), but the cursor has a different shape. A vertical line load is indicated by three blue lines in the shape of a capital H.

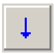
The program does not allow the creation of single, unconnected geometry points. All geometry points should minimally be part of a line. As a result, vertical line loads can only be added to existing geometry points or be created on existing lines. When creating vertical line loads, corresponding geometry points are created simultaneously. These geometry points appear in all work planes, whilst the vertical line load only appears in the active work plane.

The input values of a vertical line load are given in a force per unit of length. Vertical line loads may consist of an *x*-, *y*- and/or *z*-component. By default, when applying vertical line loads, the load will be one unit in the positive *x*-direction. The input values of the load may be changed by double clicking the corresponding geometry point and, if needed, selecting the load from the selection dialog box. As a result, the line load window is opened, showing the three components of the load in two different points, i.e. the geometry point in the active work plane and the corresponding geometry point in the work plane below.

On a part of the geometry where both line fixities and line loads are applied, the fixities have priority over the line loads during the calculations. Hence, it is not useful to apply line loads on a fixed line. However, when only one displacement direction is fixed whilst the other direction is free, it is possible to apply a load in the free direction.

Although the input values of line loads are specified in the geometry model, the activation, deactivation or change of loads may be considered in the framework of *Staged construction*.

### 3.4.5 POINT LOADS



This option may be used to create point loads. Point loads can only be applied to existing geometry lines or structural objects. The creation of a point load in a work plane is similar to the creation of a geometry point (Section 3.3.2). Point loads can be selected from the *Loads* sub-menu or by clicking on the corresponding button in the toolbar.

The input values of a point load is given in the unit of force. Point loads may consist of an *x*-, *y*- and/or *z*-component. By default, when applying point loads, the load will be one unit in the negative *y*-direction. The input value of a load may be changed by double clicking the corresponding geometry point and, if needed, selecting the load from the selection dialog box. As a result, the point loads window is opened in which the three components of the load can be specified (Figure 3.9).

On a part of the geometry where both line fixities and point loads are applied, the fixities have priority over the point loads during the calculations. Hence, it is not useful to apply point loads on a fixed line. However, when only one displacement direction is fixed whilst the other direction is free, it is possible to apply a load in the free direction.

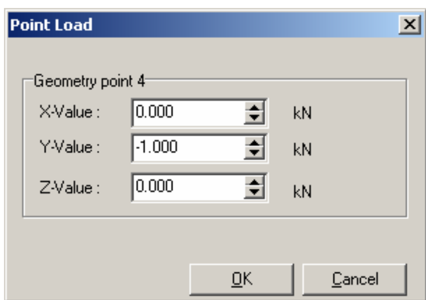



Figure 3.9 Input window for point loads

Although the input values of point loads are specified in the geometry model, the activation, deactivation or change of loads may be considered in the framework of *Staged construction*.

### 3.4.6 BOUNDARY CONDITIONS FOR CONSOLIDATION

PLAXIS automatically imposes a set of boundary conditions for consolidation on the geometry model. These conditions are generated according to the following rules:

- Vertical model boundaries are impermeable in the direction of their normal. Flow along these boundaries is allowed. This means that vertical model boundaries with their normal in *x*-direction (i.e. parallel to the *y*-*z*-plane) are impermeable in *x*-direction and vertical model boundaries with their normal in *z*-direction (i.e. parallel to the *x*-*y*-plane) are impermeable in *z*-direction.
- The model bottom boundary is impermeable in *y*-direction and flow along the bottom boundary is allowed in *x*- and *z*-direction.
- The 'ground surface' of the model is open.

These boundary conditions cannot be changed by the user.

## 3.5 MATERIAL PROPERTIES

In PLAXIS, soil properties and material properties of structures are stored in material data sets. There are five different types of material sets: Data sets for soil and interfaces, beams, walls, floors and springs. All data sets are stored in a material database. From the database, the data sets can be assigned to the soil clusters or to the corresponding structural objects in the geometry model.

### *Database with material data sets*

The material database can be activated by selecting one of the options from the *Materials* sub-menu or by clicking on the *Material sets* button in the toolbar. As a result, a material sets window appears showing the contents of the project database. The project database contains the material sets for the current project. For a new project the project database is empty. In addition to the project database, there is a global database. The global database can be used to store material data sets in a global directory and to exchange data sets between different projects. The global database can be viewed by clicking on the *Global* button in the upper part of the *Material set* window. When doing so, the window will be extended to the one as presented in Figure 3.10.

At both sides of the window (*Project database* and *Global database*) there are two combo boxes and a tree view. From the combo box on the left hand side, the *Set type* can be selected. The *Set type* parameter determines which type of material data set is displayed in the tree view (*Soil & Interfaces*, *Walls*, *Beams*, *Floors*, *Springs*). The data sets in the tree view are identified by a user-defined name. For data sets of the *Soil & Interfaces* type, the data sets can be ordered in groups according to the material model, the material type or the name of the data set. This order can be selected in the *Group order* combo box. The *None* option can be used to discard the group ordering.

The small buttons between the two tree views (> and <) can be used to copy individual data sets from the project database to the global database or vice versa. The >> button is used to copy all data sets of the project database to the global database.

Below the tree view of the global database there are two buttons. The *Open* button is used to open an existing global database with material data sets. The *Del* button can be used to delete a selected material data set from the global database. By default, the global database for soil and interface data contains the data sets of all the tutorial lessons and is contained in the file 'Soildata.MDB'. This file is compatible with other PLAXIS database files for soil and interfaces and is stored in the DB sub-directory of the 3D FOUNDATION program directory.

The buttons below the tree view of the project database are used to create, modify, copy or delete data sets. A new data set is created by clicking on the *New* button. As a result, a new window appears in which the material properties or model parameters can be entered. The first item to be entered is always the *Identification*, which is the user-defined name of the data set. After completing a data set it will appear in the tree view, indicated by its name as defined by the *Identification*.

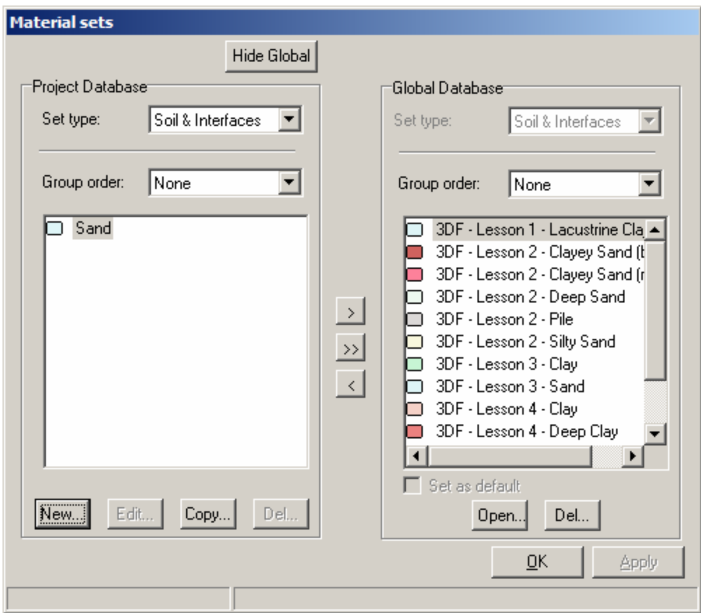


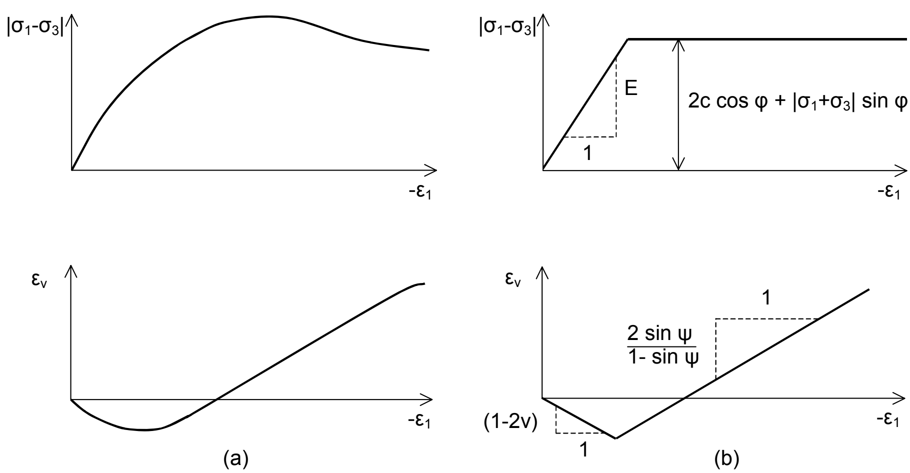

Figure 3.10 Material sets window showing the project and the global database

Existing data sets may be modified by selecting the corresponding name in the tree view and clicking on the *Edit* button. On selecting an existing data set and clicking on the *Copy* button a new data set is created of which all parameters are set equal to those of the selected (existing) data set. Finally, when a data set is no longer required, it may be deleted by first selecting it in the tree view and then clicking on the *Del* button.

### 3.5.1 MODELLING OF SOIL BEHAVIOUR

Soil and rock tend to behave in a highly non-linear way under load. This non-linear stress-strain behaviour can be modelled at several levels of sophistication. Clearly, the number of model parameters increases with the level of sophistication.

The well-known Mohr-Coulomb model can be considered as a first order approximation of real soil behaviour. This linear elastic perfectly-plastic model requires five basic input parameters, namely a Young's modulus $E$, a Poisson's ratio $\nu$, a cohesion $c$, a friction angle $\varphi$, and a dilatancy angle $\psi$. As geotechnical engineers tend to be familiar with the above five parameters and rarely have any data on other soil parameters, attention will be focused here on this basic soil model. PLAXIS also supports some advanced soil models. These models and their parameters are discussed in the Material Models Manual.




| | | | |
|---|---|---|---|
| $\sigma_1$ | Axial stress | $\varepsilon_1$ | Axial strain |
| $\sigma_3$ | Constant confining pressure | $\varepsilon_v$ | Volumetric strain |

Figure 3.11 Results from standard drained triaxial tests (a) and elastic-plastic model (b).

## *Basic model parameters in relation to real soil behaviour*

To understand the five basic model parameters, typical stress-strain curves as obtained from standard drained triaxial tests are considered (Figure 3.11). The material has been compressed isotropically up to some confining stress $\sigma_3$. After this, the axial pressure $\sigma_1$ is increased whilst the radial stress is kept constant. In this second stage of loading geomaterials tend to produce curves as shown in Figure 3.11a. The increase in the volume (or volumetric strain) is typical for dense sands and overconsolidated clays and is also frequently observed for rocks. Figure 3.11b shows the test results put into an idealised form using the Mohr-Coulomb model. The figure gives an indication of the meaning and influence of the five basic model parameters. Note that the dilatancy angle $\psi$ is needed to model the irreversible increase in volume.

| | |
|---|---|
| **Hint:** | From the figures it can also be seen that the behaviour as represented by the model is at best an approximation of the real soil behaviour. This also applies to other soil tests. The proper selection of model parameters is necessary to make the difference between the model and the real soil behaviour as small as possible. However, the model by itself will always have some inaccuracies and limitations in describing real soil behaviour. It is important that the user is aware of these inaccuracies and limitations. |

### 3.5.2 MATERIAL DATA SETS FOR SOIL AND INTERFACES

The material properties and model parameters for soil clusters are entered in material data sets (Figure 3.12). The material properties of interfaces are related to the soil properties and are entered in the same data sets as the soil properties. A data set for soil and interfaces generally represents a certain soil layer and can be assigned to the corresponding soil layer in the soil column of a borehole. The name of the data set is shown in the cluster properties window. Interfaces that are present in or along that cluster obtain the same material data set.

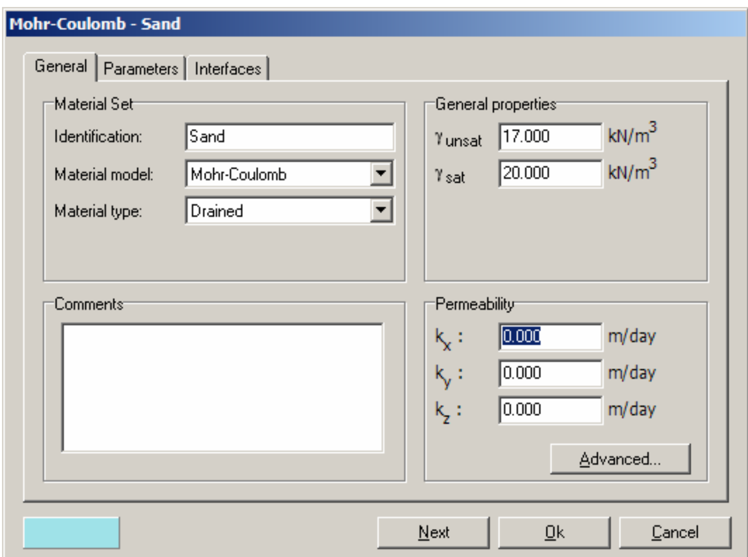



Figure 3.12 Soil and Interface material set window (*General* tab sheet)

Several data sets may be created to distinguish between different soil layers. A user may specify any identification title for a data set. It is advisable to use a meaningful name since the data set will appear in the database tree view by its identification. For easy recognition in the model, a colour is given to a certain data set.

This colour also appears in the database tree view. PLAXIS selects a unique default colour for a data set, but this colour may be changed by the user. Changing the colour can be done by clicking on the colour box in the lower left hand corner of the data set window.

The properties in the data sets are divided into three tab sheets: *General*, *Parameters* and *Interfaces*. The *General* tab sheet contains the type of soil model, the type of soil behaviour and the general soil properties such as unit weights and permeabilities. The *Parameters* tab sheet contains the stiffness and strength parameters of the selected soil model. The *Interfaces* tab sheet contains the parameters that relate the interface properties to the soil properties.

### *Material model*

PLAXIS supports different models to simulate the behaviour of soil and other continua. The models and their parameters are described in detail in the Material Models Manual. A short discussion of the available models is given below:

### *Linear elastic model*

This model represents Hooke's law of isotropic linear elasticity. The model involves two elastic stiffness parameters, namely Young's modulus $E$, and Poisson's ratio $\nu$. The linear elastic model is too limited for the simulation of soil behaviour. It is primarily used for stiff structures in the soil.

### *Mohr-Coulomb model*

This well-known model is used as a first approximation of soil behaviour in general. The model involves five parameters, namely Young's modulus $E$, Poisson's ratio $\nu$, the cohesion $c$, the friction angle $\varphi$, and the dilatancy angle $\psi$.

### *Hardening Soil model*

This is an elastoplastic type of hyperbolic model, formulated in the framework of friction hardening plasticity. Moreover, the model involves compression hardening to simulate irreversible compaction of soil under primary compression. This second-order model can be used to simulate the behaviour of sands and gravel as well as softer types of soil such as clays and silts.

### *Soft Soil creep model*

This is a second order model formulated in the framework of viscoplasticity. The model can be used to simulate the time-dependent behaviour of soft soils like normally consolidated clays and peat. The model includes logarithmic primary and secondary compression.

### *Type of material behaviour (Material type)*

In principle, all model parameters in PLAXIS are meant to represent the effective soil response, i.e. the relation between the stresses and strains associated with the soil skeleton. An important feature of soil is the presence of pore water. Pore pressures significantly influence the soil response. To enable incorporation of the water-skeleton interaction in the soil response PLAXIS offers for each soil model a choice of three types of behaviour:

### *Drained behaviour*

Using this setting no excess pore pressures are generated. This is clearly the case for dry soils and also for full drainage due to a high permeability (sands) and/or a low rate of loading. This option may also be used to simulate long-term soil behaviour without the need to model the precise history of undrained loading and consolidation.

### *Undrained behaviour*

This setting is used for a full development of excess pore pressures. Flow of pore water can sometimes be neglected due to a low permeability (clays) and/or a high rate of loading.

All clusters that are specified as undrained will indeed behave undrained, even if the cluster or a part of the cluster is located above the phreatic level. Note that effective model parameters should be entered, i.e. $E'$, $\nu'$, $c'$, $\varphi'$ and not $E_u$, $\nu_u$, $c_u$ ($s_u$), $\varphi_u$. In addition to the stiffness and strength of the soil skeleton, PLAXIS adds a bulk stiffness for the water and distinguishes between total stresses, effective stresses and excess pore pressures:

Total stress: $$\Delta p = K_u \Delta\varepsilon_v$$

Effective stress: $$\Delta p' = (1 - B)\Delta p = K'\Delta\varepsilon_v$$

Excess pore pressure: $$\Delta p_w = B\Delta p = \frac{K_w}{n}\Delta\varepsilon_v$$

Here $\Delta p$ is an increment of the total mean stress, $\Delta p'$ is an increment of the effective mean stress and $\Delta p_w$ is an increment of the excess pore pressure. $B$ is Skempton's $B$-factor, relating the proportion of the increment in total mean stress to the increment in excess pore pressure. $K_u$ is the undrained bulk modulus, $K'$ is the bulk modulus of the soil skeleton, $K_w$ is the bulk modulus of the pore fluid, $n$ is the porosity of the soil and $\Delta\varepsilon_v$ is an increment of volumetric strain.

For undrained behaviour PLAXIS does not use a realistic bulk modulus of water, because this may lead to ill-conditioning of the stiffness matrix and numerical problems. In fact, the total stiffness against isotropic compression of both soil and water is, by default, based on an implicit undrained bulk modulus:

$$K_u = \frac{2G(1+\nu_u)}{3(1-2\nu_u)} \quad \text{where} \quad G = \frac{E'}{2(1+\nu')} \quad \text{and} \quad \nu_u = 0.495$$

This results in pore water being slightly compressible and thus a *B*-factor that is slightly lower than 1.0. Hence, in isotropic loading a few percent of the load will therefore go into effective stresses, at least for small values of the effective Poisson's ratio $\nu'$. For undrained material behaviour the effective Poisson's ratio, $\nu'$, should be smaller than 0.35. Using higher values of Poisson's ratio would mean that the water would not be sufficiently stiff with respect to the soil skeleton. The default value of the undrained Poisson's ratio, $\nu_u$, can be overruled by a manual input of Skempton's *B*-factor in the Advanced Mohr-Coulomb parameters window (See for more details page 3-45).

### *Non-porous behaviour*

Using this setting neither initial nor excess pore pressures will be taken into account in clusters of this type. Applications may be found in the modelling of concrete or structural behaviour. *Non-porous* behaviour is often used in combination with the *Linear elastic* model. The input of a saturated weight is not relevant for non-porous materials.

### *Saturated and unsaturated weight ($\gamma_{sat}$ and $\gamma_{unsat}$)*

The saturated and the unsaturated weight refer to the total unit weight of the soil skeleton including the fluid in the pores. The unsaturated weight $\gamma_{unsat}$ applies to all material above the phreatic level and the saturated weight $\gamma_{sat}$ applies to all material below the phreatic level. The unit weights are entered as a force per unit volume. For non-porous material only the unsaturated weight is relevant, which is just the total unit weight. For porous soils the unsaturated weight is obviously smaller than the saturated weight. For sands, for example, the saturated weight is generally around 20 kN/m$^3$ whereas the unsaturated weight can be significantly lower, depending on the degree of saturation. Note that soils in practical situations are never completely dry. Hence, it is advisable not to enter the fully dry unit weight for $\gamma_{unsat}$. For example, clays above the phreatic level may be almost fully saturated due to capillary action. Other zones above the phreatic level may be partially saturated. However, the steady state pore pressures above the phreatic level are always set equal to zero. In this way tensile capillary stresses are disregarded. However, excess pore stresses (both pressure and suction) may occur above the phreatic line as a result of undrained behaviour. The latter does not affect the unit weight of the soil. Weights are activated by means of *Gravity loading* or *K0 procedure* in the *Calculation* mode, which is always the first calculation phase (Initial phase).

### *Permeabilities ($k_x$ , $k_y$ and $k_z$)*

Coefficients of permeability (hydraulic conductivity) have the dimension of velocity (unit of length per unit of time). The input of permeability parameters is required for consolidation calculations. In such calculations, it is necessary to specify the coefficient of permeability for all clusters including almost impermeable layers and fully impervious layers. PLAXIS 3D FOUNDATION allows for the anisotropic permeability of soils where the anisotropy directions coincide with the principal axes *x*, *y* and *z*.

To simulate an almost impermeable material (for example, concrete or uncracked rock), you should enter a permeability that is low in relation to the surrounding soil instead of entering the real permeability. In general, a factor of 1000 will be sufficient to obtain satisfactory results.

### *Advanced general properties*

The *Advanced* button on the *General* tab sheet may be clicked to enter some additional properties for advanced modelling features. As a result, an additional window appears, as shown in Figure 3.13. One of the advanced features is to account for the change of permeability during a consolidation analysis. This can be applied by entering a proper value for the $c_k$ parameter and the void ratio's.

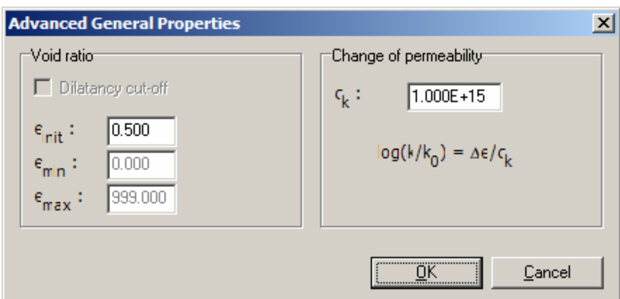



Figure 3.13 Advanced general properties window

### *Change of permeability ($c_k$)*

By default, the $c_k$-value in the *Change of permeability* box is equal to $10^{15}$, which means that a change of permeability is not taken into account. On entering a real value, the permeability will change according to the formula:

$$\log\left(\frac{k}{k_0}\right) = \frac{\Delta e}{c_k}$$

Where $\Delta e$ is the change in void ratio with respect to the initial void ratio $e_{init}$, $k$ is the permeability in the calculation and $k_0$ is the input value of the permeability in the data set (= $k_x$ , $k_y$ and $k_z$).

It is recommended to use a changing permeability only in combination with the Soft Soil Creep model. In that case the $c_k$-value is generally in the order of the compression index $C_c$. For all other models the $c_k$-value should be left to its default value of $10^{15}$.

### *Void ratio ($e_{init}$, $e_{min}$, $e_{max}$)*

The void ratio, $e$, is related to the porosity, $n$ ($e = n / (1-n)$). This quantity is used in some special options. The initial value $e_{init}$ is the value in the initial situation. The actual void ratio is calculated in each calculation step from the initial value and the volumetric strain $\Delta\varepsilon_v$. These parameters are used to calculate the change of permeability when input is given for the $c_k$ value (see above). In addition to $e_{init}$, a minimum value $e_{min}$ and a maximum value $e_{max}$ can be entered. These values are related to the maximum and minimum density that can be reached in the soil. When the Hardening Soil model is used with a certain (positive) value of dilatancy, the mobilised dilatancy is set to zero as soon as the maximum void ratio is reached (this is termed dilatancy cut-off). For other models this option is not available. To avoid the dilatancy cut-off in the Hardening Soil model, the option may be de-selected in the *Advanced parameters* window.

### 3.5.3 PARAMETERS OF THE MOHR-COULOMB MODEL

The Mohr-Coulomb model is a well-known model that can be used as a first approximation of soil behaviour in general. The model involves five parameters, namely Young's modulus $E$, Poisson's ratio $\nu$, the cohesion $c$, the friction angle $\varphi$, and the dilatancy angle $\psi$. When selecting Mohr-Coulomb as Model on the *General* tab sheet, the *Parameters* tab sheet displays the specific Mohr-Coulomb parameters and some alternatives (Figure 3.14).

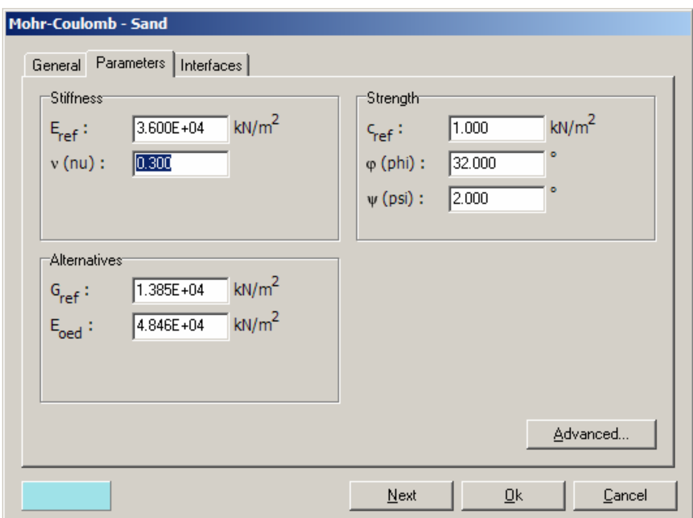



Figure 3.14 Soil and interface material set window (*Parameters* tab sheet of the Mohr-Coulomb model)

## *Young's modulus (E)*

PLAXIS uses the Young's modulus as the basic stiffness modulus in the elastic model and the Mohr-Coulomb model, but some alternative stiffness moduli are displayed as well. A stiffness modulus has the dimension of stress (force per unit of area). The values of the stiffness parameter adopted in a calculation require special attention as many geomaterials show a non-linear behaviour from the very beginning of loading.

In soil mechanics, the initial slope of the $|\sigma_1 - \sigma_3|$ vs. $\varepsilon_1$ curve resulting from a triaxial test is usually indicated as $E_0$ and the secant modulus at 50% strength is denoted as $E_{50}$ (Figure 3.15). For highly over-consolidated clays and some rocks with a large linear elastic range, it is realistic to use $E_0$ whereas for sands and near normally consolidated clays it is more appropriate to use $E_{50}$, at least for loading conditions.

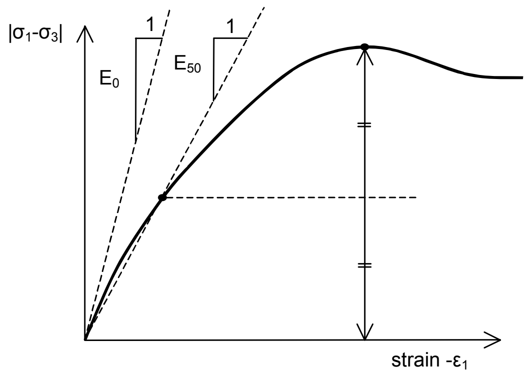



Figure 3.15 Definition of $E_0$ and $E_{50}$

For soils, both the initial modulus and the secant modulus tend to increase with the confining pressure. Hence, deep soil layers tend to have greater stiffness than shallow layers. Moreover, the observed stiffness depends on the stress path that is followed. The stiffness is much higher for unloading and reloading than for primary loading. Also, the observed soil stiffness in terms of a Young's modulus is generally lower for drained compression than for shearing. Furthermore, the observed stiffness depends on the amount of straining the soil undergoes.

For small vibrations (with strain levels below $10^{-5}$) the stiffness is much higher than for engineering strain levels (above $10^{-3}$). Hence, when using a constant stiffness modulus to represent soil behaviour one should choose a value that is consistent with the strain level, stress level and the expected stress path.

Note that some stress-dependency of soil behaviour is taken into account in the advanced models in PLAXIS, which are described in the Material Models manual. For the Mohr-Coulomb model, PLAXIS offers a special option for the input of a stiffness increasing with depth (see *Advanced Mohr-Coulomb parameters*). Note that for material data sets where the type of material behaviour is set to undrained, the Young's modulus has the meaning of an effective Young's modulus, whereas the *Undrained* setting takes care of the low compressibility.

### *Poisson's ratio ($\nu$)*

Standard drained triaxial tests may yield a significant rate of volume decrease at the very beginning of axial loading and, consequently, a low initial value of Poisson's ratio $\nu_0$.

For some cases, such as particular unloading problems, it may be realistic to use such a low initial value, but in general when using the Mohr-Coulomb model the use of a higher value is recommended.

The selection of a Poisson's ratio is particularly simple when the elastic model or Mohr-Coulomb model is used for gravity loading (initial calculation phase). For this type of loading the 3D FOUNDATION program should give realistic ratios of $K_0 = \sigma_h / \sigma_v$. As the Mohr-Coulomb model will give the well-known ratio of $\sigma_h / \sigma_v = \nu / (1-\nu)$ for one-dimensional compression it is easy to select a Poisson's ratio that gives a realistic value of $K_0$. Hence, $\nu$ is evaluated by matching $K_0$. This subject is treated more extensively in Section 4.1.6, which deals with initial stress distributions. In many cases one will obtain $\nu$ values in the range between 0.3 and 0.4. In general, such values can also be used for loading conditions other than one-dimensional compression. For unloading situations a lower Poisson's ratio (as low as 0.2) is generally more appropriate. Please note that in this way it is not possible to create $K_0$ values larger than 1, as may be observed in highly overconsolidated stress states.

Further note that for material data sets where the type of material behaviour is set to *Undrained*, the Poisson's ratio has the meaning of an effective Poisson's ratio, whereas the *Undrained* setting takes care of the low compressibility. To ensure that the soil skeleton is much more compressible than the pore water, the effective Poisson's ratio should be smaller than 0.35 for undrained behaviour.

### *Alternative stiffness parameters*

In addition to Young's modulus, PLAXIS allows for the input of alternative stiffness moduli, such as the shear modulus $G$, and the oedometer modulus $E_{oed}$. These stiffness moduli are related to the Young's modulus according to Hooke's law of isotropic elasticity, which involves Poisson's ratio $\nu$:

$$G = \frac{E}{2(1+\nu)} \qquad\qquad E_{oed} = \frac{(1-\nu)E}{(1-2\nu)(1+\nu)}$$

When entering one of the alternative stiffness parameters, PLAXIS will calculate the corresponding Young's modulus and retain the entered Poisson's ratio.

### *Cohesion (c)*

The cohesive strength has the dimension of stress. In the Mohr-Coulomb model, the cohesion parameter may be used to model the effective cohesion $c'$ of the soil, in combination with a realistic effective friction angle $\varphi'$ (see Figure 3.16a). This may not only be done for drained soil behaviour, but also if the type of material behaviour is set to *Undrained*, as in both cases PLAXIS will perform an effective stress analysis.

Alternatively, the cohesion parameter may be used to model the undrained shear strength $c_u$ (or $s_u$) of the soil, in combination with $\varphi = \varphi_u = 0$ (see Figure 3.16b).

The disadvantage of using effective strength parameters *c'* and *φ'* in combination with the material type being set to *Undrained* is that the undrained shear strength as obtained from the model may deviate from the undrained shear strength in reality because of differences in the actual stress path being followed. In this respect, advanced soil models generally perform better than the Mohr-Coulomb model, but in all cases it is recommended to compare the resulting stress state in all calculation phases with the present shear strength in reality ( $|\sigma_1 - \sigma_3| \leq 2\, c_u$ ).

On the other hand, the advantage of using effective strength parameters is that the change in shear strength with consolidation is obtained automatically, although it is still recommended to check the resulting stress state after consolidation.

The advantage of using the cohesion parameter to model undrained shear strength (in combination with $\varphi = 0$) is that the user has direct control over the shear strength, independent of the actual stress state and stress path followed. Please note that this option may not be appropriate when using advanced soil models.

PLAXIS can handle cohesionless sands ($c' = 0$), but some options will not perform well, particularly when the corresponding soil layer reaches the ground surface. To avoid complications, non-experienced users are advised to enter at least a small value (use $c' >$ 0.2 kPa). Please note that a positive value for the cohesion will lead to a tensile strength, which may be unrealistic for soils. The *Tension cut-off* option may be used to reduce the tensile strength. See *Tension cut-off* for more details.

### *Friction angle (φ)*

The friction angle $\varphi$ (phi) is entered in degrees. In general the friction angle is used to model the effective friction of the soil, in combination with an effective cohesion *c'* (Figure 3.16a). This may not only be done for drained soil behaviour, but also if the type of material behaviour is set to *Undrained*, since in both cases PLAXIS will perform an effective stress analysis. Alternatively, the soil strength is modelled by setting the cohesion parameter equal to the undrained shear strength of the soil, in combination with $\varphi = 0$ (Figure 3.16b).

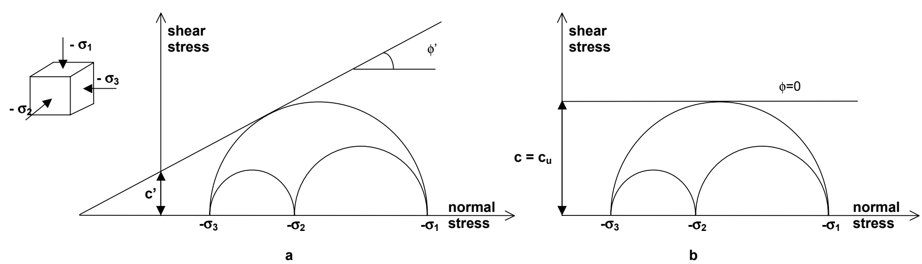



Figure 3.16 Stress circles at yield; one touches Coulomb's envelope. a) using effective strength parameters. b) using undrained strength parameters.

High friction angles, as sometimes obtained for dense sands, will substantially increase plastic computational effort. The computing time increases more or less exponentially with the friction angle. Hence, high friction angles should be avoided when performing preliminary computations for a particular project. Computing time tends to become large when friction angles in excess of 35 degrees are used.

The friction angle largely determines the shear strength as shown in Figure 3.16 by means of Mohr's stress circles. A more general representation of the yield criterion is shown in Figure 3.17.

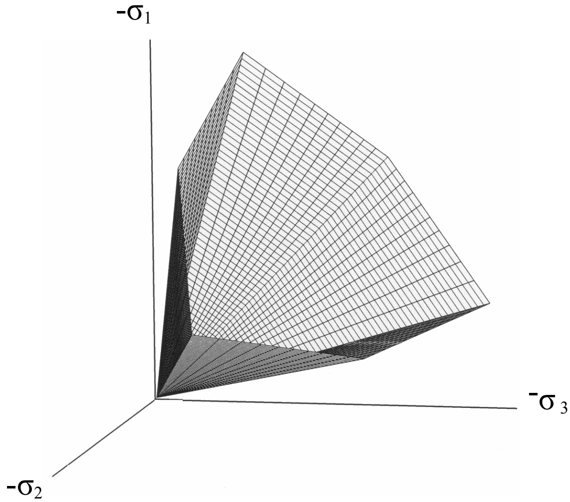



Figure 3.17 Failure surface in principal stress space for cohesionless soil

### *Dilatancy angle (ψ)*

The dilatancy angle $\psi$ (psi) is specified in degrees. Apart from heavily over-consolidated layers, clay soils tend to show no dilatancy at all (i.e. $\psi = 0$). The dilatancy of sand depends on both the density and on the friction angle. In general the dilatancy angle of soils is much smaller than the friction angle. For quartz sands the order of magnitude is $\psi \approx \varphi$ - 30°. In most cases, however, the angle of dilatancy is zero for $\varphi$-values of less than 30°. A small negative value for $\psi$ is only realistic for extremely loose sands. In the Hardening Soil model the end of dilatancy, as generally observed when the soil reaches the critical state, can be modelled using the *Dilatancy cut-off*. For details see the Material Models Manual.

When the soil strength is modelled as $c = c_u$ ($s_u$) and $\varphi = 0$, the dilatancy angle must be set to zero. Great care must be taken when using a positive value of dilatancy in combination with material type set to *Undrained*. In that case the model will show unlimited soil strength due to suction.

### *Advanced Mohr-Coulomb parameters*

When using the Mohr-Coulomb model, the *Advanced* button in the *Parameters* tab sheet may be clicked to enter some additional parameters for advanced modelling features. As a result, an additional window appears (Figure 3.18). The advanced features comprise the increase of stiffness, cohesive strength with depth and the use of a tension cut-off..

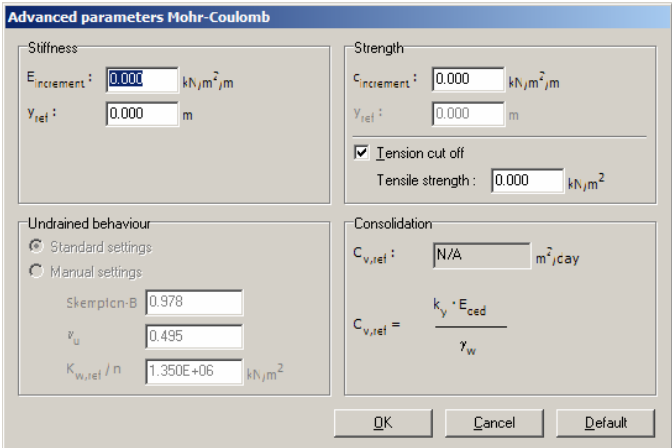



Figure 3.18 Advanced Mohr-Coulomb parameters window

### *Increase of stiffness ($E_{increment}$)*

In real soils, the stiffness depends significantly on the stress level, which means that the stiffness generally increases with depth. When using the Mohr-Coulomb model, the stiffness is a constant value. To account for the increase of the stiffness with depth the $E_{increment}$-value may be used, which is the increase of the Young's modulus per unit of depth (expressed in the unit of stress per unit depth). At the level given by the $y_{ref}$ parameter and above, the stiffness is equal to the reference Young's modulus $E_{ref}$, as entered in the *Parameters* tab sheet.

The actual value of Young's modulus in the stress points below $y_{ref}$ is obtained from the reference value and $E_{increment}$. Note that during calculations a stiffness increasing with depth does not change as a function of the stress state.

### *Increase of cohesion ($c_{increment}$)*

PLAXIS offers an advanced option for clay layers in which the cohesion increases with depth. To account for the increase of the cohesion with depth the $c_{increment}$ value may be used, which is the increase of the cohesion per unit of depth (expressed in the unit of stress per unit depth). At the level given by the $y_{ref}$ parameter and above, the cohesion is equal to the reference cohesion $c_{ref}$, as entered in the *Parameters* tab sheet. The actual value of cohesion in the stress points below $y_{ref}$ is obtained from the reference value and $c_{increment}$.

### *Skempton B-parameter*

When the *Material type* (type of material behaviour) is set to *Undrained*, PLAXIS automatically assumes an implicit undrained bulk modulus, $K_u$, for the soil as a whole (soil skeleton + water) and distinguishes between total stresses, effective stresses and excess pore pressures (see Undrained behaviour):

Total stress: $\Delta p = K_u \Delta \varepsilon_v$

Effective stress: $\Delta p' = (1 - B)\Delta p = K' \Delta \varepsilon_v$

Excess pore pressure: $\Delta p_w = B \Delta p = \dfrac{K_w}{n} \Delta \varepsilon_v$

Note that *effective* model parameters should be entered in the material data set, i.e. $E'$, $\nu'$, $c'$, $\varphi'$ and not $E_u$, $\nu_u$, $c_u$ ($s_u$), $\varphi_u$. The undrained bulk modulus is automatically calculated by PLAXIS using Hooke's law of elasticity:

$$K_u = \frac{2G(1+\nu_u)}{3(1-2\nu_u)} \quad \text{where} \quad G = \frac{E'}{2(1+\nu')}$$

and $\nu_u = 0.495$ (when using the *Standard setting*)

or $\nu_u = \dfrac{3\nu' + B(1-2\nu')}{3 - B(1-2\nu')}$ (when using the *Manual setting*)

A particular value of the undrained Poisson's ratio, $\nu_u$, implies a corresponding reference bulk stiffness of the pore fluid, $K_{w,ref} / n$:

$$\frac{K_{w,ref}}{n} = K_u - K' \quad \text{where} \quad K' = \frac{E'}{3(1-2\nu')}$$

This value of $K_{w,ref} / n$ is generally much smaller than the real bulk stiffness of pure water, $K_w^0$ (= $2 \cdot 10^6$ kN/m$^2$). If the value of Skempton's $B$-parameter is unknown, but the degree of saturation, $S$, and the porosity, $n$, are known instead, the bulk stiffness of the soil skeleton can be estimated from:

$$\frac{K_w}{n} = \frac{K_w^0 K_{air}}{SK_{air} + (1-S)K_w^0} \frac{1}{n}$$

and $K_{air}$ = 200 kN/m$^2$ for air under atmospheric pressure. The value of Skempton's $B$-parameter can now be calculated from the ratio of the bulk stiffnesses of the soil skeleton and the pore fluid:

$$B = \frac{1}{1 + \dfrac{nK'}{K_w}} \quad \text{where} \quad K' = \frac{E'}{3(1-2\nu')}$$

### *Tension cut-off*

In some practical problems, an area with tensile stresses may develop. According to the Coulomb envelope shown in Figure 3.16 this is allowed when the shear stress (given by the radius of Mohr circle) is sufficiently small. However, the soil surface near a trench in clay sometimes shows tensile cracks.

This indicates that soil may also fail in tension instead of in shear. This behaviour can be included in a PLAXIS 3D FOUNDATION analysis by selecting the tension cut-off. In this case Mohr circles with positive principal stresses are not allowed. When selecting the tension cut-off the allowable *Tensile strength* may be entered. For the Mohr-Coulomb model and the Hardening Soil model the tension cut-off is, by default, selected with a tensile strength of zero.

## 3.5.4 PARAMETERS FOR INTERFACE BEHAVIOUR

In addition to the soil properties, the data set also contains parameters to derive interface properties from the soil model parameters in the case that interface elements are located in the corresponding soil layer. The main interface parameter is the strength reduction factor $R_{inter}$, which can be found on the third tab sheet of the *Material data set* window (Figure 3.19).

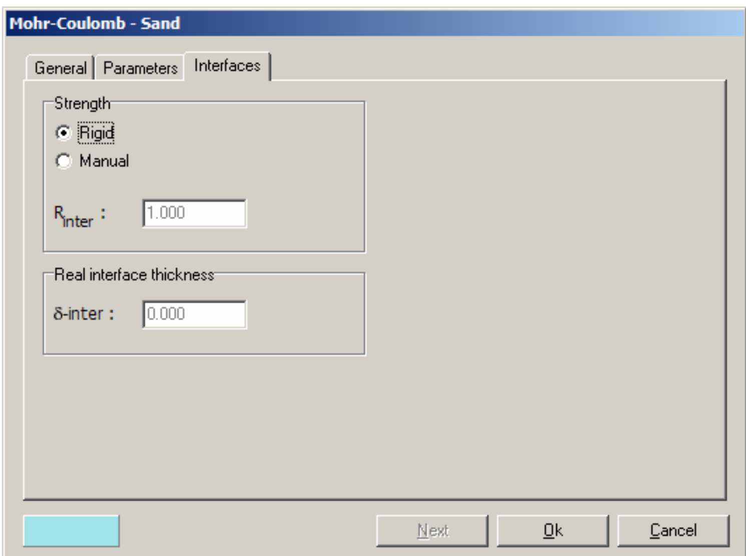



Figure 3.19 Soil and interface material set window (*Interfaces* tab sheet)

### *Interface strength ($R_{inter}$)*

An elastic-plastic model is used to describe the behaviour of interfaces for the modelling of soil-structure interaction. The Coulomb criterion is used to distinguish between elastic behaviour, where small displacements can occur within the interface, and plastic interface behaviour when permanent slip may occur.

For the interface to remain elastic the shear stress $\tau$ is given by:

$$|\tau| < \sigma_n \tan\varphi_i + c_i$$

where

$$|\tau| = \sqrt{\tau_{S1}^2 + \tau_{S2}^2}$$

where $\tau_{s1}$ and $\tau_{s2}$ are shear stresses in the two (perpendicular) shear directions and $\sigma_n$ is the effective normal stress.

For plastic behaviour $\tau$ is given by:

$$|\tau| = \sigma_n \tan\varphi_i + c_i$$

where $\varphi_i$ and $c_i$ are the friction angle and cohesion of the interface. The strength properties of interfaces are linked to the strength properties of a soil layer. Each data set has an associated strength reduction factor for interfaces $R_{inter}$. The interface properties are calculated from the soil properties in the associated data set and the strength reduction factor by applying the following rules:

$$c_i = R_{inter}\, c_{soil}$$

$$\tan\varphi_i = R_{inter} \tan\varphi_{soil} \leq \tan\varphi_{soil}$$

$$\psi_i = 0° \text{ for } R_{inter} < 1 \text{, otherwise } \psi_i = \psi_{soil}$$

The interface strength can be set using the following options:

### *Rigid*

This option is used when the interface should not not have a reduced strength with respect to the surrounding soil. These interfaces should be assigned the *Rigid* setting (which corresponds to $R_{inter}$ = 1.0). As a result, the interface properties, including the dilatancy angle $\psi_i$, are the same as the soil properties in the data set, except for Poisson's ratio ν.

### *Manual*

If the interface strength is set to *Manual*, the value of $R_{inter}$ can be entered manually. In general, for real soil-structure interaction the interface is weaker and more flexible than the associated soil layer, which means that the value of $R_{inter}$ should be less than 1. Suitable values for $R_{inter}$ for the case of the interaction between various types of soil and structures in the soil can be found in the literature. In the absence of detailed information it may be assumed that $R_{inter}$ is of the order of 2/3. A value of $R_{inter}$ greater than 1 would not normally be used.

When the interface is elastic then both slipping (relative movement parallel to the interface) and gapping or overlapping (i.e. relative displacements perpendicular to the interface) could be expected to occur.

The magnitudes of these displacements are:

$$\text{Elastic gap displacement } = \frac{\sigma_n \, t_i}{E_{oed,i}}$$

$$\text{Elastic slip displacement } = \frac{\tau \, t_i}{G_i}$$

where $G_i$ is the shear modulus of the interface, $E_{oed,i}$ is the one-dimensional compression modulus of the interface and $t_i$ is the virtual thickness of the interface, generated during the creation of interfaces in the geometry model. The shear and compression moduli are related by the expressions:

$$E_{oed,i} = 2\, G_i \frac{1 - \nu_i}{1 - 2\nu_i}$$

$$G_i = R_{inter}^2 G_{soil} \leq G_{soil}$$

$$\nu_i = 0.45$$

It is clear from these equations that, if the elastic parameters are set to low values, then the elastic displacements may be excessively large. If the values of the elastic parameters are too large, however, then numerical ill-conditioning can result. The key factor in the stiffness is the virtual thickness. This value is automatically chosen such that an adequate stiffness is obtained.

### *Real interface thickness ($\delta_{inter}$)*

The real interface thickness $\delta_{inter}$, is a parameter that represents the real thickness of a shear zone between a structure and the soil. The value of $\delta_{inter}$ is only of importance when interfaces are used in combination with the Hardening Soil model. The real interface thickness is expressed in the unit of length and is generally of the order of a few times the average grain size. This parameter is used to calculate the change in void ratio in interfaces for the dilatancy cut-off option. The dilatancy cut-off in interfaces can be of importance to calculate the correct bearing capacity of tension piles.

### 3.5.5 MODELLING UNDRAINED BEHAVIOUR

In order to model undrained soil behaviour, different modelling schemes are possible in PLAXIS. These methods are described here briefly. For a more detailed treatise the reader is referred to Section 2.4, Section 2.5, Section 2.6 and Section 2.7 of the Material Models Manual.

| | |
|---|---|
| **Hint:** | The modelling of undrained soil behaviour is even more complicated than the modelling of drained behaviour. Therefore, the user is adviced to take the utmost care with the modelling of undrained soil behaviour. |

#### *Undrained effective stress analysis with effective stiffness parameters*

The first option is to model undrained soil behaviour in an effective stress analysis using effective model parameters. This is achieved by identifying the *Type of material behaviour* of a soil layer as *Undrained*. PLAXIS will then automatically add a bulk modulus for water to the bulk modulus of the soil and thereby transform the effective stiffness parameters $E$ and $\nu$ into undrained parameters $E_u$ and $\nu_u$. Note that the index $u$ is used to indicate parameters for undrained soil and should not be confused with the index $ur$ used to denote unloading-reloading parameters.

Any volumetric strain occurring in an undrained material during a Plastic calculation phase will now give rise to excess pore pressures. PLAXIS differentiates between steady state pore pressures and excess pore pressures, the latter generated due to svolumetric strain occurring during plastic calculations. This enables the determination of effective stresses during undrained plastic calculations and allows undrained calculations to be performed with effective input parameters. This special option to model undrained material behaviour based on effective stiffness parameters is available for all material models available in the PLAXIS program. This enables undrained calculations to be executed with effective input parameters, with explicit distinction between effective stresses and (excess) pore pressures.

#### *Undrained effective stress analysis with effective strength parameters*

In general for soils, stress states at failure are quite well described by the Mohr-Coulomb failure criterion with effective strength parameters $\varphi$' and $c$'. This also applies to undrained conditions. In PLAXIS, effective strength parameters can be used quite well in combination with the *Material type* set to *Undrained*, since PLAXIS distinguishes between effective stresses and (excess) pore pressures (= effective stress analysis). The advantage of using effective strength parameters in undrained conditions is that the increase of shear strength with consolidation is automatically obtained.

However, especially for soft soils, effective strength parameters are not always available, and one has to deal with measured undrained shear strength ($c_u$ or $s_u$) as obtained from undrained tests. Undrained shear strength, however, cannot easily be used to determine the effective strength parameters $\varphi$' and $c$'. Moreover, even if one would have proper effective strength parameters, care has to be taken as to whether these effective strength parameters will provide the correct undrained shear strength in the analysis. This is because the effective stress path that is followed in an undrained analysis may not be the same as in reality, due to the limitations of the applied soil model. For example, when using the Mohr-Coulomb model with the *Material type* set to *Undrained*, the model will follow an effective stress path where the mean effective stress, $p$', remains constant all the way up to the failure.

It is known that especially soft soils, like normally consolidated clays and peat, will follow an effective stress path in undrained loading where $p$' reduces significantly. As a result, the maximum deviatoric stress that can be reached in the model is over-estimated. In other words, the mobilized shear strength in the model supersedes the available undrained shear strength. On the other hand, advanced models do include, to some extent, the reduction of mean effective stress in undrained loading, but even when using advanced models it is generally advised to check the mobilised shear strength against the available (undrained) shear strength.

### *Undrained effective stress analysis with undrained strength parameters*

As an alternative for undrained analyses with effective strength parameters, PLAXIS offers the possibility of an undrained effective stress analysis (*Material type* = *Undrained*) with direct input of the undrained shear strength, i.e. $\varphi = \varphi_u = 0$ and $c = c_u$. This option is only available for the Mohr-Coulomb model and the Hardening Soil model, but not for the Soft Soil Creep model.

Note that if the Hardening Soil model is used with $\varphi = 0$, the stiffness moduli in the model are no longer stress-dependent and the model exhibits no compression hardening, although the model retains its separate unloading-reloading modulus and shear hardening. For this reason the use of $\varphi = 0$ in the Hardening Soil model is discouraged.

Further note that whenever the *Material type* parameter is set to *Undrained*, effective values must be entered for the stiffness parameters (Young's modulus $E$ and Poisson ratio $\nu$ in case of the Mohr-Coulomb model or the respective stiffness parameters in the advanced models.)

### *Undrained total stress analysis with all parameters undrained*

If, for any reason, it is desired not to use the *Undrained* option in PLAXIS to perform an undrained analysis, one may simulate undrained behaviour using a total stress analysis with undrained parameters. In that case, stiffness is modelled using an undrained Young's modulus $E_u$ and an undrained Poisson ratio $\nu_u$, and strength is modelled using an undrained shear strength $c_u$ ($s_u$) and $\varphi = \varphi_u = 0°$. Typically, for the undrained Poisson ratio a valueclose to 0.5 is selected (between 0.495 and 0.499). A value of exactly 0.5 is not possible, since this would lead to singularity of the stiffness matrix.

In PLAXIS it is possible to perform a total stress analysis with undrained parameters if the Mohr-Coulomb is used. In this case, one should select *Non-porous* as the *Material type* (and not *Undrained*). The disadvantage of this approach is that no distinction is made between effective stresses and pore pressures. Hence, all output referring to effective stresses should now be interpreted as total stresses and all pore pressures are equal to zero.

Note that in graphical output of stresses the stresses in *Non-porous* clusters are not plotted. If one does want graphical output of stresses one should select *Drained* instead of *Non-porous* for the type of material behaviour and make sure that no pore pressures are generated in these clusters.

Also note that a direct input of undrained shear strength does not automatically give the increase of shear strength with consolidation.

This type of approach is not possible when using the Soft Soil Creep model. If the Hardening Soil model is used in a total stress analysis using undrained parameters, i.e. $\varphi = \varphi_u = 0°$, the stiffness moduli in the model are no longer stress-dependent and the model exhibits no compression hardening, although the model retains its separate unloading-reloading modulus and shear hardening.

### 3.5.6 MATERIAL DATA SETS FOR BEAMS

In addition to material data sets for soil and interfaces, the material properties and model parameters for beams are also entered in separate material data sets. A data set for beams generally represents a certain type of beam material or beam profile, and can be assigned to the corresponding (chain of) horizontal and/or vertical beam elements in the geometry model. The name of the data set is shown in the *Beam properties* window (Figure 3.20).

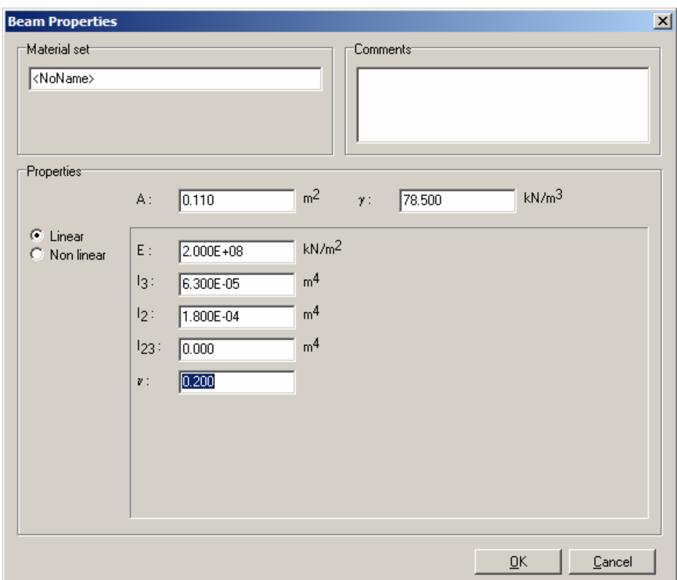



Figure 3.20 Beam properties window (linear behaviour)

Several data sets may be created to distinguish between different types of beams. A user may specify any identification title for a data set. It is advisable to use a meaningful name since the data set will appear in the database tree view by its identification.

## *General properties*

A beam has two general properties: The cross-section area $A$, and the unit weight $\gamma$. The cross-section area is the actual area (in the unit of length squared) perpendicular to the axial beam direction where beam material is present. For beams that have a certain profile (such as steel beams), the cross-section area can be found in tables that are provided by steel factories. The unit weight (in the unit of force per unit of volume) is the unit weight of the material from which the beam is composed. The product $\gamma A$ determines the distributed beam weight.

## *Stiffness properties*

Stiffnesses can be linear or non-linear. Linear beam stiffnesses involve a Young's modulus $E$, a Poisson's ratio $\nu$, and three moments of inertia $I_2$ (against bending around the second axis), $I_3$ (against bending around the third axis) and $I_{23}$ (against oblique bending; zero for symmetric beam profiles). The definition of various quantities according to the beam's local system of axes are visualised in Figure 3.21 for horizontal beams and in Figure 3.22 for vertical beams.

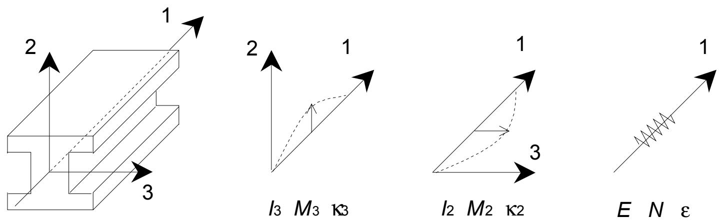



Figure 3.21 Definition of moment of inertia ($I$), positive bending moment ($M$) positive curvature ($\kappa$) and stiffness ($E$) for a horizontal beam based on local system of axes

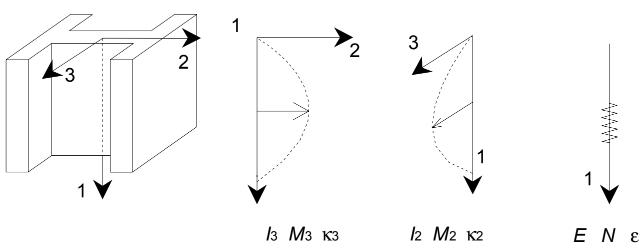



Figure 3.22 Definition of moment of inertia ($I$), positive bending moment ($M$) positive curvature ($\kappa$) and stiffness ($E$) for a vertical beam based on local system of axes

More information about the behaviour and structural forces in beams can be found in Chapter 6 of the Material Models Manual.

### *Non-linear behaviour*

When selecting the *Non-linear* radio button in the material data set window, tables with pairs of ($N$-$\varepsilon$), ($M_2$-$\kappa_2$) (bending around the second axis) and ($M_3$-$\kappa_3$) (bending around the third axis) can be defined (Figure 3.23). If elastic properties were defined before non-linear behaviour was selected, then three pairs equivalent to the elastic stiffness properties are automatically inserted in each table.

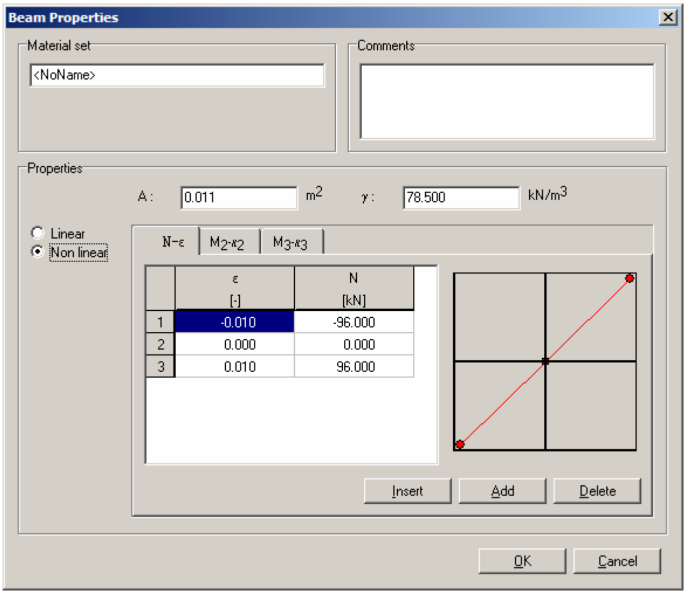



Figure 3.23 Beam properties window (non-linear behaviour)

The user may change existing values by selecting the desired value in the table and entering a new value. To insert, add or delete pairs, select a value in the table and use the desired button. The number of pairs in a table (points in the graph) is practically unlimited.

Tensile normal forces and extension are considered to be positive. For positive directions of bending moments and curvature in the local system of axes, see Figure 3.21 and Figure 3.22. It is advised to define both negative and positive values for strains and curvatures in the tables. Strains and curvatures must start with the 'lowest' (most negative) value at line 1, and must be entered in the right order towards the most positive value. If, during calculations, strains or curvatures occur outside the range between the minimum and maximum defined values, then it is assumed that the corresponding force is obtained by linear extrapolation from the last two pairs in the table.

Please note that each value in a table must be larger than its predecessor. Hence, it is not possible to emulate perfectly-plastic behaviour or softening. It should also be noted that the non-linear modelling of structural behaviour by means of these multi-linear diagrams does not lead to irreversible deformations or plasticity. Upon unloading, exactly the same curve is followed backwards.

### 3.5.7 MATERIAL DATA SETS FOR WALLS

Similarly walls have separate material data sets. A data set for walls generally represents a certain type of wall material or wall profile, and can be assigned to the corresponding (chain of) wall elements in the geometry model. The name of the data set is shown in the wall properties window (Figure 3.24).

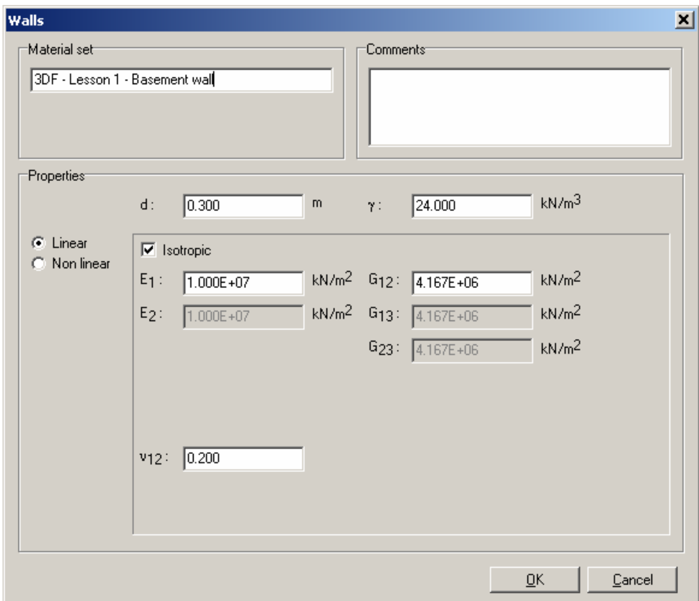



Figure 3.24 Wall properties window (linear behaviour)

Several data sets may be created to distinguish between different types of walls. A user may specify any identification title for a data set. It is advisable to use a meaningful name since the data set will appear in the database tree view by its identification.

## *General properties*

A wall has two general properties: The (equivalent) thickness $d$, and the unit weight $\gamma$. The (equivalent) thickness (in the unit of length) is the material cross-section area of the wall across its major axial direction per 1 m width. For massive walls without a particular profile this is just the wall thickness, but for walls that have a certain profile (such as sheet-pile walls or sandwich plates), the thickness is relatively small and should be properly calculated from the above definition. The unit weight is the unit weight of the material from which the wall is composed. The product $\gamma d$ determines the distributed weight of the wall.

### *Stiffness properties*

Wall stiffnesses can be linear or non-linear. The 3D FOUNDATION program allows for orthotropic material behaviour in walls, which is defined by the following parameters:

$E_1$: Young's modulus in first axial direction

$E_2$: Young's modulus in second axial direction

$G_{12}$: In-plane shear modulus

$G_{13}$: Out-of-plane shear modulus related to shear deformation over first direction

$G_{23}$: Out-of-plane shear modulus related to shear deformation over second direction

$\nu_{12}$: Poisson's ratio

These parameters appear in the following (approximate) relationships for structural forces:

$$\begin{bmatrix} N_1 \\ N_2 \end{bmatrix} = \begin{bmatrix} E_1 d & \nu_{12} E_2 d \\ \nu_{12} E_2 d & E_2 d \end{bmatrix} \begin{bmatrix} \varepsilon_1 \\ \varepsilon_2 \end{bmatrix}$$

$$\begin{bmatrix} Q_{12} \\ Q_{13} \\ Q_{23} \end{bmatrix} = \begin{bmatrix} G_{12} d & 0 & 0 \\ 0 & G_{13} d & 0 \\ 0 & 0 & G_{23} d \end{bmatrix} \begin{bmatrix} \gamma_{12} \\ \gamma_{13} \\ \gamma_{23} \end{bmatrix}$$

$$\begin{bmatrix} M_{11} \\ M_{22} \\ M_{12} \end{bmatrix} = \begin{bmatrix} \frac{E_1 d^3}{12} & \frac{\nu_{12} E_2 d^3}{12} & 0 \\ \frac{\nu_{12} E_2 d^3}{12} & \frac{E_2 d^3}{12} & 0 \\ 0 & 0 & \frac{G_{12} d^3}{12} \end{bmatrix} \begin{bmatrix} \kappa_{11} \\ \kappa_{22} \\ \kappa_{12} \end{bmatrix}$$

Figure 3.25 visualises the wall's local system of axes and the major quantities. The local system of axes in a wall element is such that the first and the second local axis lie in the plane of the wall whereas the third axis is perpendicular to the plane of the wall.

If the *Isotropic* option is checked the input is limited to $E_1$ and $\nu_{12}$, where as $E_2 = E_3 = E_1$, $G_{12} = G_{13} = G_{23} = E / 2(1+ \nu_{12})$ and $\nu_{13} = \nu_{23} = \nu_{12}$.

More information about the behaviour and structural forces in walls can be found in Chapter 6 of the Material Models Manual.

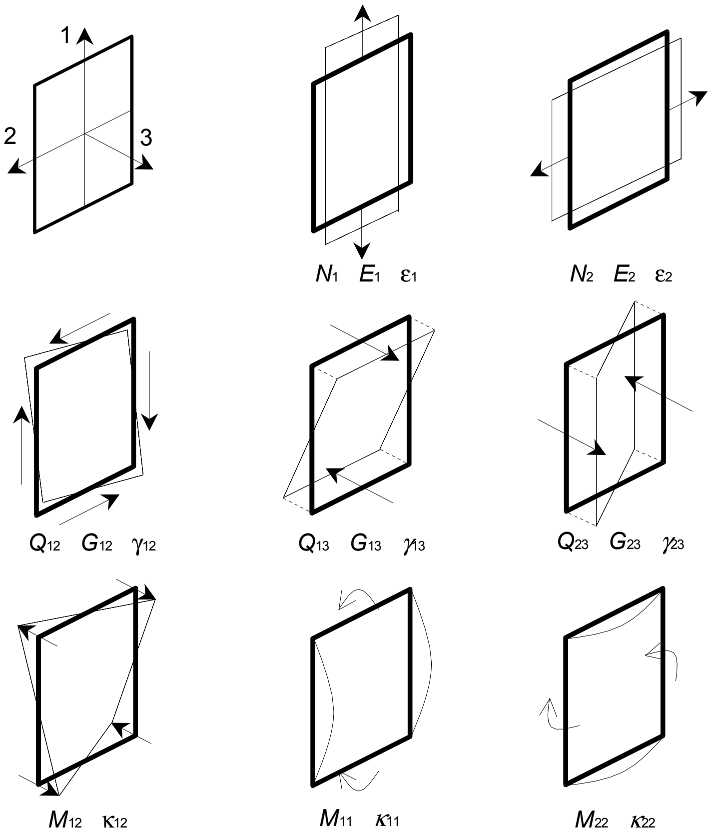



Figure 3.25 Definition of a wall's local system of axes and various quantities

### *Non-linear behaviour*

When selecting the *Non-linear* radio button in the material data set window, tables with pairs of ($N_1$-$\varepsilon_1$), ($N_2$-$\varepsilon_2$), ($Q_{12}$-$\gamma_{12}$), ($Q_{13}$-$\gamma_{13}$), ($Q_{23}$-$\gamma_{23}$), ($M_{11}$-$\kappa_{11}$), ($M_{22}$-$\kappa_{22}$) and ($M_{12}$-$\kappa_{12}$) can be defined (Figure 3.26). In this system, the different force quantities are fully de-coupled. If elastic properties were defined before non-linear behaviour was selected, then three pairs equivalent to the elastic stiffness properties are automatically inserted in each table. The user may change existing values by selecting the desired value in the table and entering a new value. To insert, add or delete pairs, select a value in the table and use the desired button. The number of pairs in a table (points in the graph) is practically unlimited. Tensile normal forces and extension are considered to be positive. For positive directions of bending moments and curvature in the local system of axes, see Figure 3.25. It is advised to define both negative and positive values for strains and curvatures in the tables.

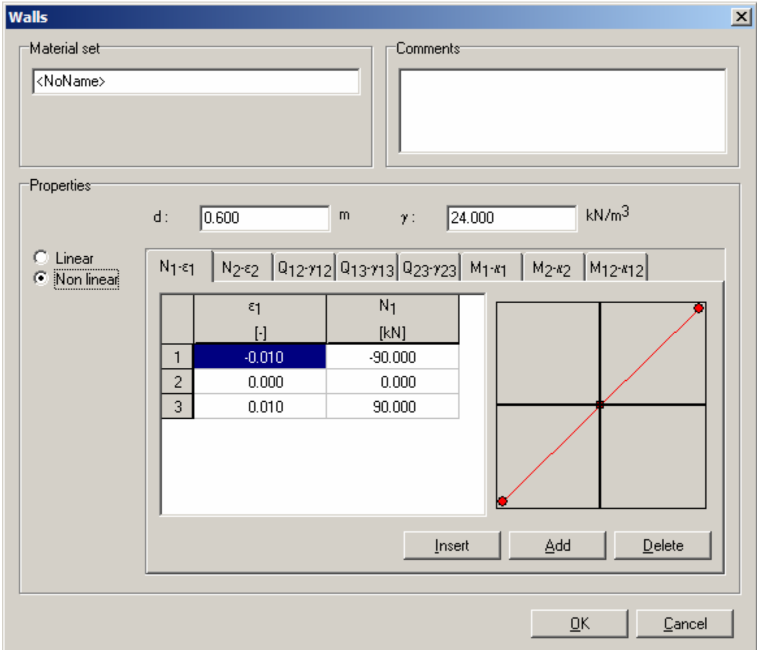



Figure 3.26 Wall properties window (non-linear behaviour)

Strains and curvatures must start with the 'lowest' (most negative) value at line 1, and must be entered in the right order towards the most positive value. If, during calculations, strains or curvatures occur outside the range between the minimum and maximum defined values, then it is assumed that the corresponding force is obtained by linear extrapolation from the last two pairs in the table.

Please note that each value in a table must be larger than its predecessor. Hence, it is not possible to emulate perfectly-plastic behaviour or softening. It should also be noted that the non-linear modelling of structural behaviour by means of these multi-linear diagrams does not lead to irreversible deformations or plasticity. Upon unloading, exactly the same curve is followed backwards.

### 3.5.8 MATERIAL DATA SETS FOR FLOORS

Similarly floors have separate material data sets. A data set for floors generally represents a certain floor material or floor profile, and can be assigned to the corresponding cluster of floor elements in the geometry model. The name of the data set is shown in the floor properties window (Figure 3.27).

Several data sets may be created to distinguish between different types of floors. A user may specify any identification title for a data set. It is advisable to use a meaningful name since the data set will appear in the database tree view by its identification.

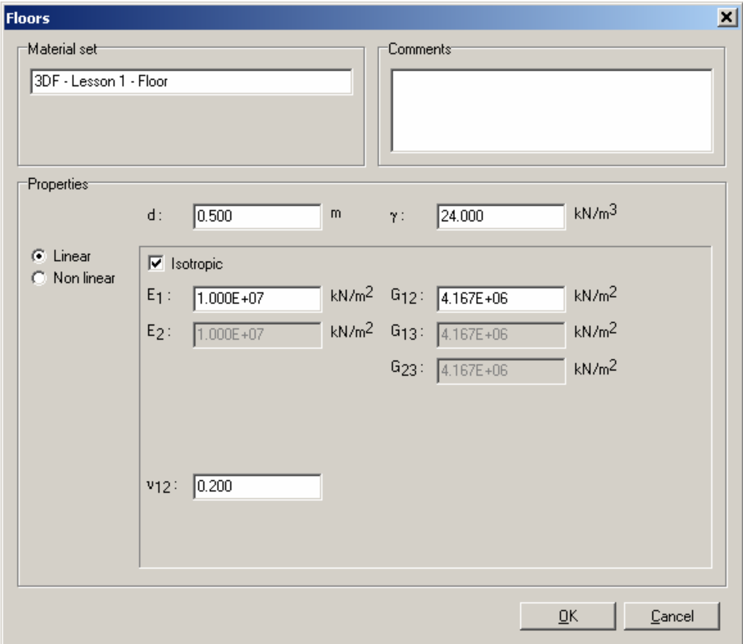



Figure 3.27 Floor properties window (linear behaviour)

## *General properties*

A floor has two general properties: The (equivalent) thickness $d$, and the unit weight $\gamma$. The (equivalent) thickness (in the unit of length) is the material cross-section area of the floor across its major axial direction per 1 m width. For massive floors without a particular profile this is just the floor thickness, but for floors that have a certain profile (such as a prefab concrete floor profile or sandwich floor), the thickness should be properly calculated from the above definition. The unit weight is the unit weight of the material from which the floor is composed. The product $\gamma \cdot d$ determines the distributed weight of the floor.

## *Stiffness properties*

Floor stiffnesses can be linear or non-linear. The 3D FOUNDATION program allows for orthotropic material behaviour in floors, which is defined by the following parameters:

$E_1$: Young's modulus in first axial direction

$E_2$: Young's modulus in second axial direction

$G_{12}$: In-plane shear modulus

$G_{13}$: Out-of-plane shear modulus related to shear deformation over first direction

$G_{23}$: Out-of-plane shear modulus related to shear deformation over second direction

$\nu_{12}$: Poisson's ratio

These parameters appear in the following (approximate) relationships for structural forces:

$$\begin{bmatrix} N_1 \\ N_2 \end{bmatrix} = \begin{bmatrix} E_1 d & \nu_{12} E_2 d \\ \nu_{12} E_2 d & E_2 d \end{bmatrix} \begin{bmatrix} \varepsilon_1 \\ \varepsilon_2 \end{bmatrix}$$

$$\begin{bmatrix} Q_{12} \\ Q_{13} \\ Q_{23} \end{bmatrix} = \begin{bmatrix} G_{12} d & 0 & 0 \\ 0 & G_{13} d & 0 \\ 0 & 0 & G_{23} d \end{bmatrix} \begin{bmatrix} \gamma_{12} \\ \gamma_{13} \\ \gamma_{23} \end{bmatrix}$$

$$\begin{bmatrix} M_{11} \\ M_{22} \\ M_{12} \end{bmatrix} = \begin{bmatrix} \frac{E_1 d^3}{12} & \frac{\nu_{12} E_2 d^3}{12} & 0 \\ \frac{\nu_{12} E_2 d^3}{12} & \frac{E_2 d^3}{12} & 0 \\ 0 & 0 & \frac{G_{12} d^3}{12} \end{bmatrix} \begin{bmatrix} \kappa_{11} \\ \kappa_{22} \\ \kappa_{12} \end{bmatrix}$$

Figure 3.28 visualises the floor's local system of axes and the major quantities. The local system of axes in a floor element is such that the first and the second local axis lie in the plane of the floor whereas the third axis is perpendicular to the plane of the floor. If the *Isotropic* option is checked the input is limited to $E_1$ and $\nu_{12}$, where as $E_2 = E_3 = E_1$, $G_{12} = G_{13} = G_{23} = E / 2(1+ \nu_{12})$ and $\nu_{13} = \nu_{23} = \nu_{12}$.

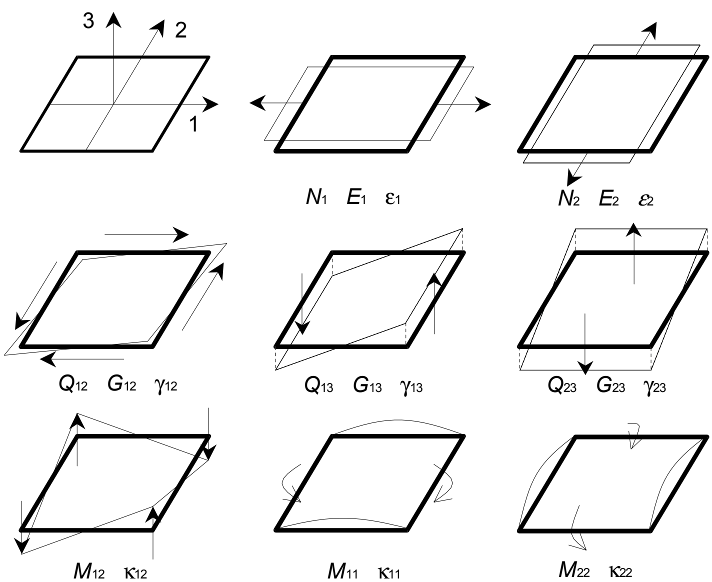



Figure 3.28 Definition of a floor's local system of axes and various quantities

More information about the behaviour and structural forces in floors can be found in Chapter 6 of the Material Models Manual.

### *Non-linear behaviour*

When selecting the *Non-linear* radio button in the material data set window, tables with pairs of ($N_1$-$\varepsilon_1$), ($N_2$-$\varepsilon_2$), ($Q_{12}$-$\gamma_{12}$), ($Q_{13}$-$\gamma_{13}$), ($Q_{23}$-$\gamma_{23}$), ($M_{11}$-$\kappa_{11}$), ($M_{22}$-$\kappa_{22}$) and ($M_{12}$-$\kappa_{12}$) can be defined (Figure 3.29). In this system, the different force quantities are fully de-coupled. If elastic properties were defined before non-linear behaviour was selected, then three pairs equivalent to the elastic stiffness properties are automatically inserted in each table. The user may change existing values by selecting the desired value in the table and entering a new value. To insert, add or delete pairs, select a value in the table and use the desired button. The number of pairs in a table (points in the graph) is practically unlimited. Tensile normal forces and extension are considered to be positive. For positive directions of bending moments and curvature in the local system of axes, see Figure 3.28.

It is advised to define both negative and positive values for strains and curvatures in the tables. Strains and curvatures must start with the 'lowest' (most negative) value at line 1, and must be entered in the right order towards the most positive value. If, during calculations, strains or curvatures occur outside the range between the minimum and maximum defined values, then it is assumed that the corresponding force is obtained by linear extrapolation from the last two pairs in the table.

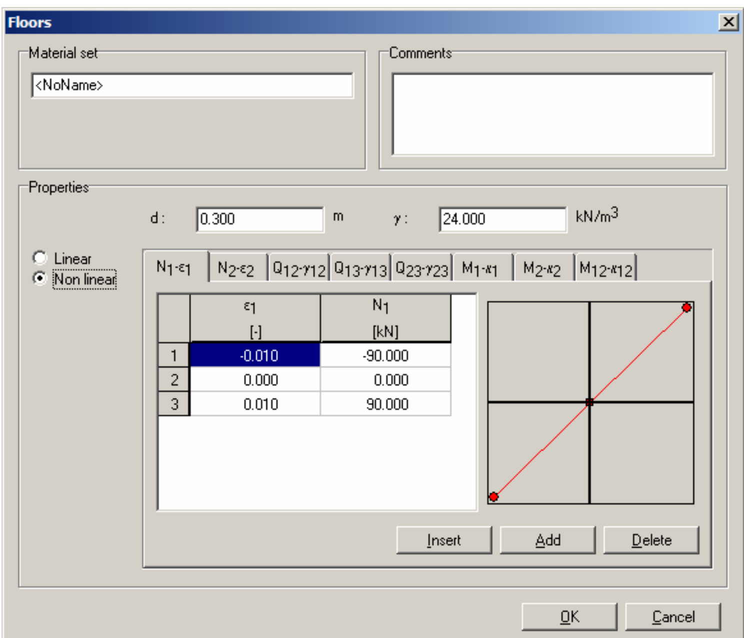



Figure 3.29 Floor properties window (non-linear behaviour)

Please note that each value in a table must be larger than its predecessor. Hence, it is not possible to emulate perfectly-plastic behaviour or softening. It should also be noted that the non-linear modelling of structural behaviour by means of these multi-linear diagrams does not lead to irreversible deformations or plasticity. Upon unloading, exactly the same curve is followed backwards.

### 3.5.9 MATERIAL DATA SETS FOR SPRINGS

Similarly springs have separate material data sets. A data set for springs generally represents a certain type of pile response or anchor or strut behaviour, and can be assigned to the corresponding spring elements in the geometry model. The name of the data set is shown in the spring properties window (Figure 3.30).

Several data sets may be created to distinguish between different types of piles or supports. A user may specify any identification title for a data set. It is advisable to use a meaningful name since the data set will appear in the database tree view by its identification.

#### *Stiffness properties*

Springs do not have a weight assigned to it. The only property is an axial stiffness *EA/L*, entered in the unit of force. The axial stiffness can be linear or non-linear.

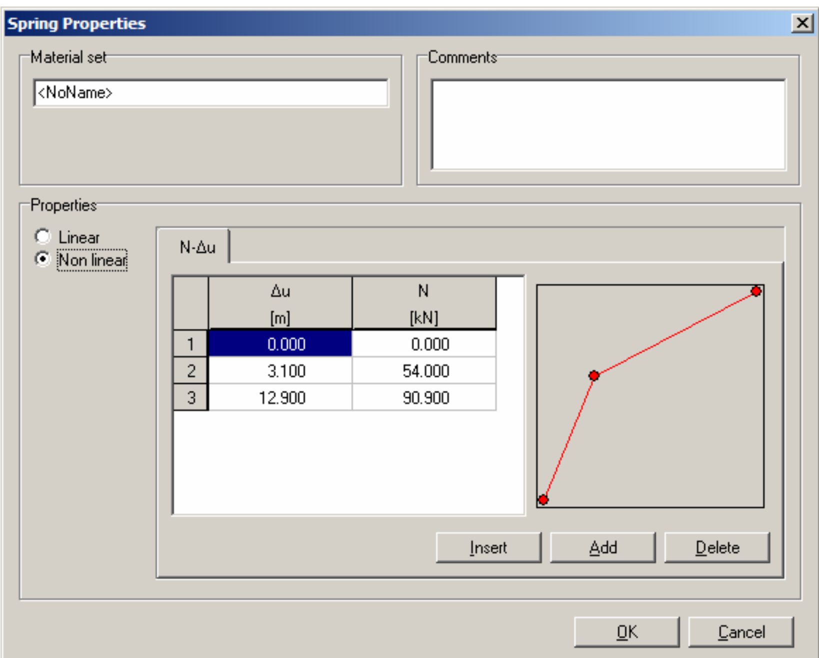



Figure 3.30 Spring properties window (non-linear behaviour)

When selecting the *Non-linear* radio button in the material data set window, a table with pairs of ($N$-$\Delta u$) can be defined (Figure 3.30). If an elastic stiffness was defined before non-linear behaviour was selected, then three pairs equivalent to the elastic stiffness properties are automatically inserted in the table. The user may change existing values by selecting the desired value in the table and entering a new value. To insert, add or delete pairs, select a value in the table and use the desired button. The number of pairs in a table (points in the graph) is practically unlimited.

Tensile normal forces and extension are considered to be positive. It is advised to define both negative and positive values for displacements in the table. Displacements must start with the 'lowest' (most negative) value at line 1, and must be entered in the right order towards the most positive value. If, during calculations, displacements occur outside the range between the minimum and maximum defined values, then it is assumed that the corresponding force is obtained by linear extrapolation from the last two pairs in the table.

Please note that each value in a table must be larger than its predecessor. Hence, it is not possible to emulate perfectly-plastic behaviour or softening. It should also be noted that the non-linear modelling of structural behaviour by means of these multi-linear diagrams does not lead to irreversible deformations or plasticity. Upon unloading, exactly the same curve is followed backwards.

### 3.5.10 ASSIGNING DATA SETS TO GEOMETRY COMPONENTS

After creating material data sets, the data sets must be assigned to the corresponding geometry components (soil layers and structures). This is done in different ways, which are explained below.

#### *Soil (boreholes)*

Regarding soil data, material data sets can be assigned to individual soil layers in the boreholes. Therefore a borehole should be double-clicked to open the corresponding *Borehole* window. In the *Borehole* window the material sets button at the upper right hand side of the window should be clicked to open the material database.

To assign a data set to a particular soil layer, select the desired data set from the material database tree view (click on the data set and hold the left hand mouse button down), drag it to the soil column in the borehole window (hold the mouse button down while moving) and drop it on the desired layer (release the mouse button). The layer should now show the corresponding material data set colour. The drag and drop procedure should be repeated until all layers have their appropriate data set. Note that material sets cannot be dragged directly from the global database tree view.

When multiple boreholes are used it should be noted that assigning a data set to a layer in one particular borehole will also influence the other boreholes, since all layers appear in all boreholes, except for layers with a zero thickness.

| | |
|---|---|
| **Hint:** | By default, the colours of data sets have a low intensity. To increase the intensity of all data set colours, the user may press <Ctrl><Alt><C> simultaneously on the keyboard. There are three levels of colour intensity that can be selected in this way. |

### *Structures*

Regarding structures (beams, floors, walls, springs), there are two different methods of assigning material data sets.

- The first method is based on an open *Material sets* window, showing the created material sets in the project database tree view. The desired material set can be dragged (select it and keep the left mouse button down) to the draw area and dropped on the desired component. It can be seen from the shape of the cursor whether or not it is valid to drop the material set. When assigning a material set in this way, the structure will blink red when the material properties were assigned successfully. Structures with a material data set assigned to them will have a darker colour than structures that do not have a material data set assigned.
- The second method is to double-click the desired structure and, if needed, subsequently select it from the *Select* dialog box. As a result, the properties window appears on which the material set is indicated. If no material set has been assigned, the material set box displays <Unassigned>. When clicking on the *Change* button the *Material sets* window appears from which the desired material set can be selected. The material set can be dragged from the project database tree view and dropped on the properties window. Alternatively, after the selection of the required material set it can be assigned to the selected structure by clicking on the *Apply* button in the *Material sets* window. In this case, the *Material sets* window remains open. When clicking on the <OK> button instead, the material set is also assigned to the selected structure and the *Material sets* window is subsequently closed.

Regarding piles, material data sets can be assigned through the pile designer. After double-clicking the pile, the pile designer is opened. In the pile designer, the material database can be opened by clicking the *Material sets* button. To assign a data set, select the desired data set in the material database tree view and then drag and drop it on the pile in the draw area. Alternatively, pile properties can be assigned in the framework of *Staged construction* (Section 4.3.3).

## 3.6 MESH GENERATION

To perform finite element calculations, the geometry has to be divided into elements. A composition of finite elements is called a finite element mesh. When the geometry model is fully defined and material properties have been assigned to all soil layers and structural objects, it is recommended to first generate a 2D mesh of work planes. The 2D mesh should be made fully satisfactory (including global and local refinements; Section 3.6.3 to 3.6.6) before proceeding to the 3D mesh generation. It is advisable to avoid very fine meshes, since this will lead to excessive calculation times. If the 2D mesh is satisfactory, 3D mesh generation can be performed. The 3D mesh generation process will take the information from the work planes at different levels as well the soil stratigraphy from the boreholes into account. By default, when using multiple boreholes, the 3D mesh generation process results in a smoothly curved ground surface and soil layer boundaries. If it is desired to introduce sharp transitions in the ground surface and soil layer boundaries (e.g. to model embankments), the *Triangulate* option should be used (Section 3.6.1). The 3D FOUNDATION program allows for a fully automatic generation of 2D and 3D finite element meshes.

However, meshes that are automatically generated by PLAXIS may not be accurate enough to produce acceptable numerical results. Please note that the user remains responsible to judge the accuracy of the finite element meshes and may need to consider global and local refinement options.

The basic soil elements of a 3D finite element mesh are the 15-node wedge elements (Figure 3.31). These elements are generated from the 6-node triangular elements as generated in the 2D mesh. Due to the presence of non-horizontal soil layers, some 15-node wedge elements may degenerate to 13-node pyramid elements or even to 10-node tetrahedral elements.

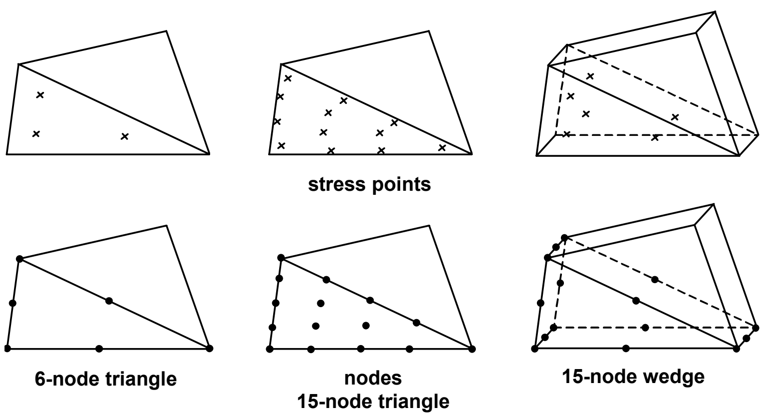



Figure 3.31 Comparison of 2D and 3D soil elements

The 15-node wedge element is composed of 6-node triangles in horizontal direction and 8-node quadrilaterals in vertical direction. The accuracy of the 15-node wedge element and the compatible structural elements are comparable with the 6-node triangular element and compatible structural elements in a 2D PLAXIS analysis. Higher order element types, for example comparable with the 15-node triangle in a 2D analysis, are not considered for a 3D FOUNDATION analysis because this will lead to large memory consumption and unacceptable calculation times.

In addition to the soil elements, special types of elements are used to model structural behaviour. For beams, 3-node line elements are used, which are compatible with the 3-noded sides of a soil element. In addition, 6-node and 8-node plate elements are used to simulate the behaviour of walls and floors. Moreover, 12-node and 16-node interface elements are used to simulate soil-structure interaction. The element formulations are given in the Scientific Manual.

### 3.6.1 TRIANGULATION

If the *Triangulate* option from the *Mesh* sub-menu is selected, the PLAXIS 3D FOUNDATION program performs a coarse triangulation of the geometry in the draw area before the mesh is generated. This triangulation is based on the outer model boundaries and the geometry points, including points where boreholes have been defined. Hence, the coarse triangulation is influenced by the particular composition of the boreholes. The purpose of this triangulation is to define planes between geometry points and the model boundaries, where the ground surface level and the soil layer boudaries of the generated 3D mesh are linearly interpolated. This may be useful in situations where the ground surface and soil layer boundaries show sharp transitions (for example at an embankment crest and toe.)

### 3.6.2 2D MESH GENERATION

The 2D mesh generator is a special version of the triangle mesh generator developed by Sepra[1]. The generation of the 2D mesh is based on a robust triangulation procedure, which results in 'unstructured' meshes. These meshes may look disorderly, but the numerical performance of such meshes is usually better than for regular (structured) meshes.

The required input for the 2D mesh generator is a plane geometry model composed of points, lines and clusters as present in the draw area of the input program, combined with additional lines automatically generated by the program. Points and lines are mainly entered by the user when creating the various work planes. Additional lines are automatically generated by the program based on the triangulation and crossings between inclined soil layers and work planes. When creating structural elements or loadings, corresponding geometry lines are automatically created in each work plane. In this way, all work planes have the same composition of points and lines. Points and lines may also be used to influence the position and (local) distribution of elements.

Clusters are areas that are fully enclosed by geometry lines. Clusters are automatically generated during the creation of the geometry model.

The generation of the 2D mesh is started by clicking on the mesh generation button in the toolbar or by selecting the *Generate* option from the *Mesh* sub-menu. The generation is also activated directly after the selection of a refinement option from the *Mesh* sub-menu.

---

1 Ingenieursbureau Sepra, Boomkwekerij 30, 2535 KD Den Hoorn (NL)

Before the actual 2D mesh generation is performed, possible crossings between soil layers and work planes are determined from the borehole data. If a soil layer crosses a work plane, additional geometry lines are introduced in the geometry model. This is to make sure that a consistent 3D mesh can be generated, taking into account both the work plane data as well as the borehole data.

After the 2D mesh generation the Output program is started and a plot of the 2D mesh is displayed. Although interface elements have a zero thickness, the interfaces in the mesh are drawn with a certain thickness to show the connections between soil elements, structural elements and interface elements. This so-called *Connectivity plot* is also available as a regular output option (Section 5.9.6). The scale factor (Section 5.4) can be used to reduce the graphical thickness of the interfaces. To return to the Input program, the *Update* button should be pressed.

### 3.6.3 GLOBAL COARSENESS

The 2D mesh generator requires a general meshing parameter which represents the average element size, $l_e$. In PLAXIS this parameter is calculated from the outer geometry dimensions ($x_{min}$, $x_{max}$, $z_{min}$, $z_{max}$ ) and a *Global coarseness* setting as defined in the *Mesh* sub-menu:

$$l_e = \sqrt{\frac{(x_{max} - x_{min})(z_{\max} - z_{\min})}{n_c}}$$

Distinction is made between five levels of global coarseness: *Very coarse*, *Coarse*, *Medium*, *Fine*, *Very fine*. By default, the global coarseness is set to *Coarse*. The average 2D element size and the number of generated triangular elements in the 2D mesh depends on this global coarseness setting. A rough estimate is given below (based on a generation without local refinement):

| | | | |
|---|---|---|---|
| *Very coarse* | : | Around 50 elements | $n_c = 25$ |
| *Coarse* | : | Around 100 elements | $n_c = 50$ |
| *Medium* | : | Around 250 elements | $n_c = 100$ |
| *Fine* | : | Around 500 elements | $n_c = 200$ |
| *Very fine* | : | Around 1000 elements | $n_c = 400$ |

The exact number of elements depends on the geometry details and local refinement settings.

### 3.6.4 GLOBAL REFINEMENT

A 2D finite element mesh can be refined globally by selecting the *Refine global* option from the *Mesh* sub-menu. When selecting this option, the global coarseness parameter is increased one level (for example from *Coarse* to *Medium*) and the mesh is automatically regenerated.

### 3.6.5 LOCAL COARSENESS

In areas where large stress concentrations or large deformation gradients are expected, it is desirable to have a more accurate (finer) finite element mesh, whereas other parts of the geometry might not require a fine mesh. Such a situation often occurs when the geometry model includes edges or corners of structural objects. For these cases PLAXIS uses local coarseness parameters in addition to the global coarseness parameter. The local coarseness parameter is the *Local element size* factor, which is contained in each geometry point. These factors give an indication of the relative element size with respect to the average element size as determined by the *Global coarseness* parameter. By default, the *Local element size* factor is set to 1.0 at all geometry points. To reduce the length of an element to half the average element size, the *Local element size* factor should be set to 0.5.

The local element size factor can be changed by double clicking the corresponding geometry point. Alternatively, when double clicking a geometry line, one can set the local element size factor for both points of the geometry line simultaneously. Values in the range from 0.05 to 5.0 are acceptable. Points that belong to a pile will automatically obtain a local element size factor of 0.2.

Note that the local element size factor only applies to the 2D finite element mesh and does not influence the 3D mesh generation.

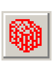
### 3.6.6 LOCAL REFINEMENT

Instead of specifying local element size factors, a local refinement can be achieved by selecting clusters, lines or points and selecting a local refinement option from the *Mesh* sub-menu.

When selecting one or more clusters, the *Mesh* sub-menu allows for the option *Refine cluster*. Similarly, when selecting one or more geometry lines, the *Mesh* sub-menu provides the option *Refine line*. When selecting one or more points, the option *Refine point* is available.

Using one of the options for the first time will give a local element size factor of 0.5 for all selected geometry points or all geometry points that are included in the selected clusters or lines. Repetitive use the local refinement option will result in a local element size factor which is half the current factor. However, the minimum and maximum value are restricted to the range 0.05 to 5.0. After selecting one of the local refinement options, the 2D mesh is automatically regenerated.

### 3.6.7 3D MESH GENERATION



When the 2D mesh is satisfactory, a fully 3D mesh can be generated. This can be done by clicking on the 3D mesh generation button or selecting the corresponding option from the *Mesh* sub-menu. In fact, it is possible to generate a 3D mesh directly, i.e. without generating a 2D mesh first. The 3D FOUNDATION program will then automatically generate a 2D mesh using default or existing coarseness settings and subsequently generate a 3D mesh. However, in this case the user has less control over the accuracy of the mesh.

The 3D mesh is based on a system of horizontal and pseudo-horizontal planes in which the 2D mesh is used. These planes are formed by the work planes and the soil layer boundaries as defined by the boreholes and the coarse triangulation. A single borehole leads to soil layer boundaries that are true horizontal planes. When multiple boreholes are used the soil layer boundaries may form non-horizontal planes. The precise vertical position of such planes in all mesh points is obtained by linear interpolation from the coarse triangulation. When an inclined soil layer boundary crosses a work plane, a mesh line will be available in the 2D mesh to guarantee consistency of the 3D mesh at such crossings.

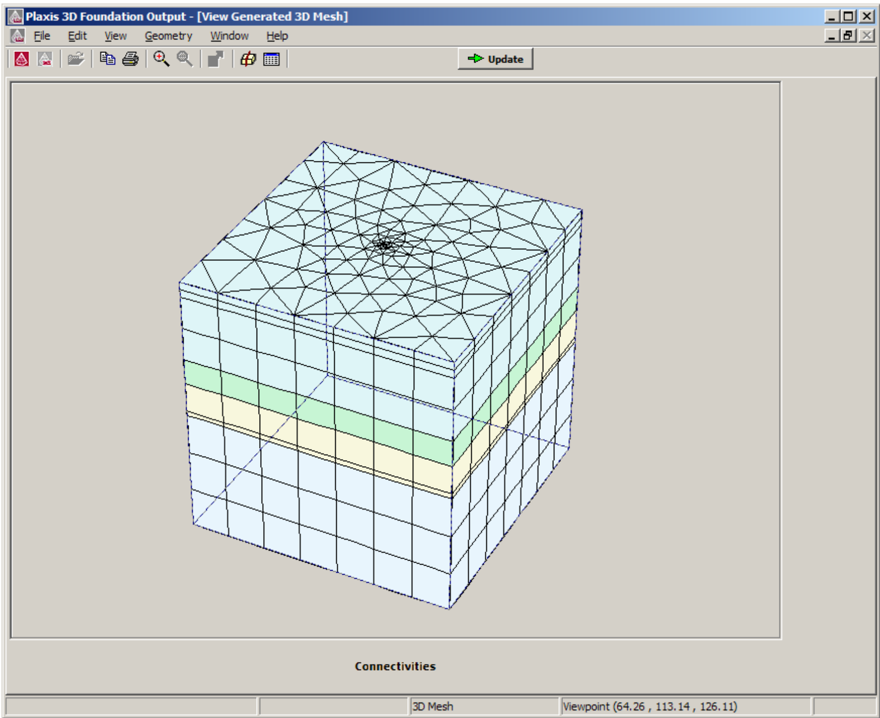



Figure 3.32  3D mesh in Output window.

In principle, the system of horizontal and pseudo-horizontal planes forms the element distribution in vertical direction. If the distance between two successive planes is (locally) significantly larger than the *Average element size*, $l_e$, as defined for the 2D mesh (Section 3.6.3), additional planes are introduced. This is done in such a way that the element size in vertical direction is approximately equal to the average element size, which reduces the possibility that badly shaped elements occur. If the soil layer thickness varies, the element distribution in vertical direction may also vary. As a result of different numbers of element layers in vertical direction, 15-node wedge elements may be degenerated to 13-node pyramid elements (single degeneration) or to 10-node tetrahedral elements (double degeneration) at the point where the number of elements in vertical direction changes.

After clicking the *Generate 3D mesh* button or selecting the corresponding item in the *Mesh* sub-menu, the 3D mesh generation procedure is started and the 3D mesh is displayed in the Output program within seconds. The full 3D mesh and mesh at predefined work planes can be viewed by moving through the tab sheets. The arrow keys of the keyboard allow the user to rotate the model and view it from any desired viewpoint (Figure 3.32). The total number of elements in the mesh can be seen when selecting *General project information* from the *View* submenu.

### 3.6.8 ADVISED MESH GENERATION PRACTICE

3D finite element calculations are very time-consuming. The time consumption highly depends on the number of elements used in the analysis. Moreover, when using a large number of 3D elements, the stiffness matrix may be too large to fit in physical memory. As a result virtual memory is used which significantly slows down the calculation speed. Hence, care should be taken when generating 3D finite element meshes.

Since the 3D mesh is generated from the 2D mesh, care must be taken to ensure that the number of elements in the 2D mesh does not become too large. In general, 2D meshes in the PLAXIS 3D FOUNDATION program will generally be coarser than meshes in 2D PLAXIS versions.

To perform efficient finite element calculations, a preliminary analysis can be performed using a relatively coarse mesh, based on a *Coarse* 2D mesh. This analysis can be used to check whether the model is large enough and to see where stress concentrations and large deformation gradients occur. This information should be used to create a more refined finite element model.

To create efficiently a more refined 2D finite element mesh, one should first select the required *Global coarseness* from the *Mesh* sub-menu. In addition, when local refinements are desired, one should start by refining clusters, then refining lines and finally refining points. If desired, points can be given a direct local element size factor. In the 3D mesh generation it is recommended to carefully check the resulting 3D mesh, especially when multiple work planes and non-horizontal soil layers are present. This may lead to very small or oddly shaped soil elements. If this is the case, the number of boreholes, the borehole positions or the soil layer boundaries in the boreholes could be slightly adapted until a satisfactory 3D mesh is obtained.

# 4 CALCULATIONS

With the 3D mesh generation, the geometry modelling process is complete. To proceed with the calculations, the *Calculation* mode should be entered. This is done by clicking on the *Calculate* button above the *Geometry* toolbar in the Input program. When doing so, the user is asked to first save the project under an appropriate name. It is also possible, after starting the Input program and reading an existing project, to proceed directly to the *Calculation* mode, provided that the input data of the project have been fully defined earlier.

Finite element calculations can be divided into several sequential calculation phases. Each calculation phase corresponds to a particular loading or construction stage. The first calculation phase (Initial phase) in the 3D FOUNDATION program is always a calculation of the initial stress field for the initial geometry configuration by means of *Gravity loading* or *K0 procedure*. After this initial phase, subsequent calculation phases may be defined by the user.

When entering the *Calculation* mode, the draw area shows a top view of the geometry model, similar as in the *Model* mode. The general toolbar has not changed and shows the same options as in the *Model* mode (Section 3.1). The *Geometry* toolbar has changed into a *Calculation* toolbar (Figure 4.1) which contains items to define, select and preview calculation phases, to select nodes for load-displacement curves and to execute calculations.

## 4.1 THE CALCULATION MENU

The main menu of the *Calculation* mode contains pull-down sub-menus covering the general options for handling files, viewing the draw area, opening the material database and defining calculation phases. The *Calculation* menu consists of the sub-menus *File*, *Edit*, *View*, *Materials*, *Calculation* and *Help*.

### *The File sub-menu*

| | |
|---|---|
| *Go to Output program* | To open the Output program. |
| *Go to Curves program* | To open the Curves program. |
| *Save* | To save the current status of the calculation settings. |
| *Save as* | To save the current project under a new name. The file requester is presented. |
| *Print* | To print the current content of the draw area. |
| *Work directory* | To set the directory where PLAXIS 3D FOUNDATION project files will be stored. |
| *General settings* | To view the basic parameters of the model (Section 3.2.2). |

| | |
|---|---|
| *Pack project* | To compress a project and pack it into a single file, to facilitate sending the project by e-mail. The file is named <project>.PF3ZIP and stored in the <project>.DF3 directory. |
| *Exit* | To leave the program. |

### *The Edit sub-menu*

| | |
|---|---|
| *Undo* | To restore a previous status of the geometry model (after an input error). Repetitive use of the undo option is limited to the 10 most recent actions. |
| *Copy* | To copy the content of the draw area to the Windows clipboard. |

### *The View sub-menu*

| | |
|---|---|
| *Zoom in* | To zoom into a rectangular area for a more detailed view. After selection, the zoom area must be indicated using the mouse. Press the left mouse button at a corner of the zoom area; hold the mouse button down and move the mouse to the opposite corner of the zoom area; then release the button. The program will zoom into the selected area. The zoom option may be used repetitively. |
| *Zoom out* | To restore the view to before the most recent zoom action. |
| *Reset view* | To restore the full draw area. |
| *Table* | To view the table with the *x*- and *z*-coordinates of all geometry points in the geometry model. |
| *Rulers* | To show or hide the rulers along the draw area. |
| *Axes* | To show or hide the arrows indicating the *x*- and *z*-axes. |
| *Cross hair* | To show or hide the cross hair during the selection of objects in a geometry model. |
| *Grid* | To show or hide the grid in the draw area. |
| *Snap to grid* | To activate or deactivate the snapping into the regular grid points. |
| *Point numbers* | To view the geometry point numbering. |
| *Chain numbers* | To view the numbering of structural object chains. |

### *The Materials sub-menu*

| | |
|---|---|
| *Soil and interfaces* | To view the material database with soil and interface data sets. |
| *Beams* | To view the material database with data sets for beams. |

| | |
|---|---|
| *Walls* | To view the material database with data sets for walls. |
| *Floors* | To view the material database with data sets for floors. |
| *Springs* | To view the material database with data sets for springs. |

### 4.1.1 THE CALCULATION TOOLBAR

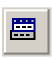
The *Calculation* toolbar (Figure 4.1) contains items to define, select and preview calculation phases, to select nodes for load-displacement curves and to execute calculations.


Figure 4.1 Calculation toolbar

#### *Next phase*

The *Next phase* button can be used to proceed to the next calculation phase. If the last phase was focused in the list of calculation phases, a new phase is introduced and the window with calculation phase settings is opened.

#### *Phases window*



The *Phases* button can be used to open the window with calculation phase settings (Figure 4.2). The *Phases* window is divided into an upper part and a lower part.

The upper part of the *Phases* window consists of a *General* tab sheet and a *Parameters* tab sheet. The *General* tab sheet is used to identify the calculation phase (*Number/ID*) and, more importantly, to determine the type of calculation and the ordering of calculation phases by selecting the calculation phase that is used as a starting point for the current calculation (*Start from phase*). For more information about these features, see Section 4.1.3 and 4.1.5. Furthermore, the *General* tab sheet contains a *Comments* box where the user can store any information related to a particular calculation phase. The *Log info* box displays messages generated during the finite element calculation and is used for logging purposes. The *Parameters* tab sheet is used to define numerical parameters for controlling the calculation process. More information on these parameters can be found in Section 4.2.1 and 4.2.2.

The lower part of the *Phases* window shows the list of calculation phases. Each line in the list corresponds to a separate phase. For each phase, the line shows the corresponding identification string, the phase number, a number referring to the phase to start from and the first and last load step number of the phase. If the phase has not yet been executed, the step numbers are set to N/A. The active calculation phase is indicated as a blue or a grey bar. A calculation phase that has been selected for execution is indicated by a blue arrow (→) in front of the line. Calculation phases that have been successfully finished are indicated by a green tick mark (√), whereas phases that did not finish successfully are indicated by a red cross (×).

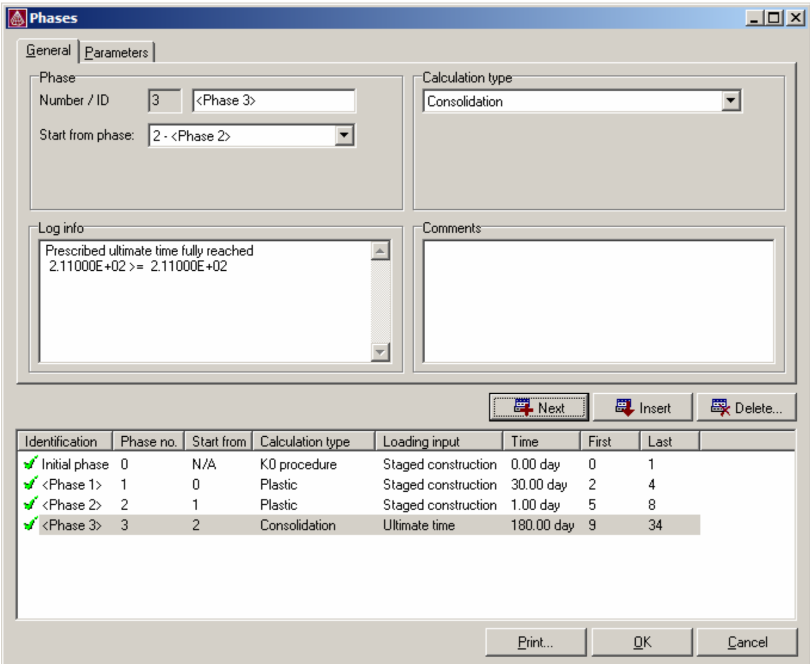


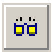

Figure 4.2  Phases window

In between the upper part and the lower part of the *Phases* window there are buttons to add a new calculation phase at the end of the list (*Next*), to insert a calculation phase above the selected phase (*Insert*) or to delete the selected calculation phase (*Delete*). The last two buttons are not available when the Initial phase is selected.

### *Calculation phases*

A copy of the list of calculation phases in the *Phases* window is included in the *Calculation* toolbar, presented as a combo box. The active calculation phase is indicated as a blue bar. The combo box may be used to select an individual calculation phase with the purpose to define or redefine geometry settings for that phase or to show its computational results. Changes in geometry settings only apply to the selected calculation phase and do not influence other phases. A blue arrow (→) in front of the phase name indicates that the phase is selected for calculation. A green tick mark (√) indicates that the phase was successfully finished. A red cross (×) indicates that the defined situation could not be reached during the calculation, which could mean that a calculation error or a failure situation has occurred.

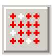
### *Preview*



The preview button can be used to show a 3D plot of the situation as defined in the current calculation phase. The 3D plot is presented in the Output program on the basis of the geometry settings for the current calculation phase. This option is particularly useful to check whether the geometry settings for the current calculation phase have been made correctly, before actually starting the calculation process.

In addition to the 3D plot, tab sheets are available in which the model can be viewed at the work plane levels.

### *Select points for curves*



The select points for curves button can be used to preselect points for which load-displacement curves or stress paths can be generated after the calculation using the Curves program (Chapter 6). The calculation kernel will store displacements, stresses and other data for the selected points in a separate file. For more information, see Section 4.5.

### *Calculate / Output*


The *Calculate* button can be used to start the calculation process. This button is only visible when the currently active calculation phase is selected for calculation, as indicated by the blue arrow (→). When the currently active calculation phase has already been calculated, as indicated by a green tick mark (√) or a red cross (×), an *Output* button is available instead. The *Output* button can be used to present the computational results in the Output program.

## 4.1.2 DEFINING CALCULATION PHASES

Consider a new project for which no calculation phase has been defined yet. In this case, the list with calculation phases contains only one line, indicated as *Initial phase*. This line represents the initial situation of the project, i.e. the initial geometry configuration and the corresponding initial stress field.

The initial stress state can be calculated by means of a real finite element calculation in which soil weight is applied by means of gravity loading or using a simplified procedure, which is called the K0 procedure. What type of calculation is performed in the initial phase depends on the selection of the *Calculation type* in the *General* tab sheet.

For the initial phase the options are limited to *Gravity loading* or *K0 procedure*. If not all geometry components are active in the initial situation, the user must deactivate these components in the draw area (Section 4.3 Staged Construction). The *Initial phase* is the starting point for further calculations. However, deformations calculated in the initial phase are not considered to be relevant for further calculations. Therefore, these displacements are, by default, reset to zero in the beginning of the next calculation phase.

To introduce a new calculation phase after the *Initial phase*, the *Next phase* button in the *Calculation* toolbar should be pressed, after which the *Phases* window is opened (Figure 4.2). As an alternative, when the *Phases* window is already open, the *Next* button just above the list of calculation phases can be pressed.

After the introduction of the new calculation phase, the phase settings have to be defined. This should be done using the tab sheets *General* and *Parameters* in the upper part of the *Phases* window. The definition starts with the selection of the *Calculation type* in the *General* tab sheet, which has two options: *Plastic* and *Consolidation.* On pressing the <Enter> key after each input, the user is guided through all parameters. Most parameters have a default setting, which simplifies the input. In general, only a few parameters have to be considered to define a calculation phase. More details on the various parameters are given in the following sections.

In addition to the parameter settings, the user has to define the geometry and load configuration that has to be considered in the calculation phase. This is done in the draw area. Therefore the *Phases* window has to be closed by clicking on the *OK* button. The *Phase list* combo box in the *Calculation* toolbar will indicate the currently active calculation phase. Now, the geometry and load configuration can be changed by activating / deactivating or double clicking geometry objects (Section 4.3 Staged Construction). If another phase is selected from the *Phase list* combo box and this phase has not yet been defined before, it will automatically adopt the settings from the phase where it starts from.

When all parameters have been set and the geometry settings have been made, the user can choose either to define another calculation phase or to start the calculation process. Introducing and defining another calculation phase can be done in the same way as described above. The calculation process can be started by clicking on the *Calculate* button in the *Calculation* toolbar or by selecting the *Start calculation* option in the *Calculation* menu. It is not necessary to define all calculation phases before starting the calculation process since the program allows for defining new calculation phases after previous phases have been calculated.

### 4.1.3 ORDER OF CALCULATION PHASES

The order of calculation phases is defined by means of the *Start from phase* parameter in the *General* tab sheet of the *Phases* window. This parameter refers to the phase from which the selected calculation phase should start (this is termed the reference phase). By default, the previous phase is selected here, but, if more calculation phases have already been defined, the reference phase may also be an earlier phase. A phase that appears later in the calculation list cannot be selected.

When defining only a single calculation phase in addition to the *Initial phase*, it is obvious that the additional calculation should start from the results of the *Initial phase*. On the other hand, later calculation phases may also start from the Initial phase. This could be the case if different loadings or loading sequences are to be considered separately for the same project. Another example where the phase order is not straightforward is in calculations where, for a certain situation, a load is increased until failure to determine the safety margin. When continuing the construction process, the next phase should start from the previous construction stage rather than from the failure situation.

### 4.1.4 INSERTING AND DELETING CALCULATION PHASES

When inserting and deleting calculation phases the user has to keep in mind that the start conditions for the subsequent phases will change and must again be specified manually.

In general, a new calculation phase is defined at the end of the calculation list using the *Next phase* option. It is possible, however, to insert a new phase between two existing phases. This can be done in the *Phases* window by pressing the *Insert* button while the line where the new phase is to be inserted is selected. By default, the new phase will start from the results of the previous phase in the list, as indicated by the *Start from* value. This means that the status of geometry objects, loads and water conditions is adopted from the previous phase.

The user has to define the new settings for the inserted phase in a similar way as defining a new phase at the end of the list.

The next phase, which originally started from a previous phase, will keep the existing *Start from phase* value and will thus not start automatically from the inserted phase. If it is desired that the next phase starts from the inserted phase then this should be specified manually by changing the *Start from phase* parameter in the *General* tab sheet of the *Phases* window (Section 4.1.3). In this case it is required that the next phase is fully redefined, since the start conditions have changed. This may also have consequences for the phases thereafter.

Besides inserting calculation phases it is also possible to delete phases. This is done in the *Phases* window by selecting the phase to be deleted and clicking on the *Delete* button. Before deleting a phase it should be checked which of the subsequent phases refer to the phase to be deleted in the *Start from* column. After confirmation of the delete operation, all phases of which the *Start from* value referred to the deleted phase will be modified automatically such that they now refer to the predecessor of the deleted phase. Nevertheless, it is required that the modified phases are redefined, since the start conditions have changed.

### 4.1.5 TYPES OF CALCULATIONS

The first parameter to be set when defining a calculation phase is the *Type of calculation*. This is done in the combo box at the upper right-hand side of the *General* tab sheet. Distinction is made between four basic types of calculation: *Plastic* calculation, *Consolidation* analysis, *Gravity loading* and *K0 procedure*. The latter two types are only available for the initial phase.

## *Plastic calculation*

A *Plastic* calculation is used to carry out an elastic-plastic deformation analysis according to small deformation theory. The stiffness matrix in a plastic calculation is based on the original undeformed geometry. This type of calculation is appropriate in most practical geotechnical applications. In general, a plastic calculation does not take time effects into account, except when the Soft Soil Creep model is used (see Material Models Manual).

Considering the quick loading of water-saturated clay-type soils, a *Plastic* calculation may be used for the limiting case of fully undrained behaviour using the *Undrained* option in the material data sets. On the other hand, performing a fully drained analysis can assess the settlements on the long term. This will give a reasonably accurate prediction of the final situation, although the precise loading history is not followed and the process of consolidation is not dealt with explicitly.

When changing the geometry configuration (Section 4.3) it is also possible (for each calculation phase) to redefine the water boundary conditions and recalculate the pore pressures. See Section 4.3.4 for details on generating pore pressures.

For more details on theoretical formulations of a plastic calculation reference should be made to the Scientific Manual.

### *Consolidation analysis*

A *Consolidation* analysis is usually conducted when it is necessary to analyse the development and dissipation of excess pore pressures in a saturated clay-type soil as a function of time. PLAXIS 3D FOUNDATION allows for true elastic-plastic consolidation analyses. In general, consolidation analysis without additional loading is performed after an undrained plastic calculation. It is also possible to apply loads during a consolidation analysis. However, care should be taken when a failure situation is approached, since the iteration process may not converge in such a situation.

For more details on theoretical formulations of a consolidation analysis, you are referred to the Scientific Manual.

### *Gravity loading*

*Gravity loading* is a type of *Plastic* calculation, in which initial stresses are generated based on the volumetric weight of the soil. All options that are available for a Plastic calculation are available. In a *Gravity loading* analysis the relative proportion of weight is raised from 0 to 1. In all phases after the initial phase, the full soil weight remains activated. A detailed description of *Gravity loading* is given in Section 4.1.6.

*Gravity loading* is only available for the initial calculation phase.

### *K0 procedure*

The *K0 procedure* is only available for the initial calculation phase. It is a special calculation method available in PLAXIS 3D FOUNDATION which can be used to define the initial stresses for the model, taking into account the loading history of the soil. A detailed description of the *K0 procedure* is given in Section 4.1.6.

### 4.1.6 INITIAL STRESS GENERATION

Many analysis problems in geotechnical engineering require the specification of a set of initial stresses. The initial stresses in a soil body are influenced by the weight of the material and the history of its formation. This stress state is usually characterised by an initial vertical stress $\sigma_{v,0}$. The initial horizontal stress $\sigma_{h,0}$ , is related to the initial vertical stress by the coefficient of lateral earth pressure, $K_0$ ($\sigma_{h,0} = K_0\ \sigma_{v,0}$).

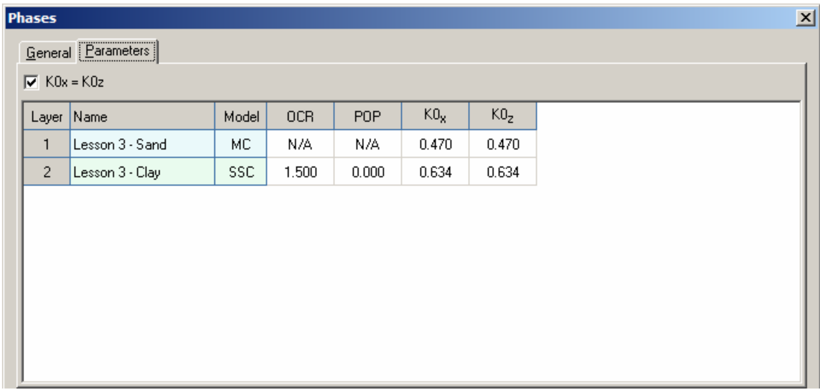



Figure 4.3 Initial stress generation window (*K0-procedure*)

In the PLAXIS 3D FOUNDATION program, initial stresses may be generated by specifying $K_0$ or by using *Gravity loading*. The *K0 procedure* option can be selected by selecting *K0 procedure* in the *Calculation type* combo box in the *General* tab sheet of the *Phases* window. Alternatively, *Gravity loading* can be selected. These options are the only available options for the initial phase. It is recommended to generate and inspect results from initial stresses first before defining and executing other calculation phases.

As a rule, one should use the *K0 procedure* only in cases with a horizontal surface and with all soil layers and phreatic levels parallel to the surface. For all other cases one should use *Gravity loading*. Further possibilities and limitations of both methods are described below.

To make sure that gravity loading results in initial effective stresses in situations where undrained materials are used, the option *Ignore undrained behaviour* should be selected in the *Parameters* tab sheet.

Once the initial stresses have been set up, then displacements should be reset to zero at the start of the next calculation phase. This removes the effect of the initial stress generation procedure on the displacements developed during subsequent calculations, whereas the stresses remain.

#### *K0 procedure*

If *K0 procedure* is selected, the *Parameters* tab sheet of the *Phases* window offers a table in which, with various other parameters, $K_0$ values can be entered (Figure 4.3).

Two $K_0$ values can be specified, one for the *x*-direction and one for the *z*-direction.

$$K_{0,x} = \sigma'_{xx} / \sigma'_{yy} \qquad K_{0,z} = \sigma'_{zz} / \sigma'_{yy}$$

In practice, the value of $K_0$ for a normally consolidated soil is often assumed to be related to the friction angle by Jaky's empirical expression :

$$K_0 = 1 - \sin\varphi$$

In an over-consolidated soil, $K_0$ would be expected to be larger than the value given by this expression.

The meaning of the various parameters in the tab sheet is described below.

### *Layer:*

The first column displays the layer number.

### *Name:*

The second column displays the name of the material data set that is used in the corresponding layer

### *OCR and POP:*

The third and the fourth column are used to enter either an overconsolidation ratio (*OCR*) or a pre-overburden pressure (*POP*). Either one of these values is utilised to generate the pre-consolidation pressures for the Soft Soil Creep model and the Hardening Soil model. When using other material models the input of *OCR* and *POP* is not applicable. See the Material Models Manual for more information.

### $K_{0,x}$ $K_{0,z}$:

The fifth and sixth columns are used to enter $K_0$-values. The default $K_0$-value is based on Jaky's formula (1 - sin$\varphi$), but this value may be changed by the user. Entering a negative value for $K_0$ will result in a recalculation of $K_0$ from 1-sin$\varphi$. Be careful with very low or very high $K_0$ values, since these values may cause initial plasticity. The checkbox *$K_{0z}$ is equal to $K_{0x}$* can be used to set all $K_{0z}$ values equal to the $K_{0x}$ values for all clusters.

After clicking the <Calculate> button, the initial stress generation starts. The *K0 procedure* considers only soil weight and pore pressures. External loads and weight of structural elements are not taken into account. Activating loads and structural objects in the initial configuration therefore has no effect.

Using very low or very high $K_0$-values in the *K0 procedure* may lead to stresses that violate the Mohr-Coulomb failure condition. In this case PLAXIS automatically reduces the lateral stresses such that the failure condition is obeyed. Hence, these stress points are in a plastic state and are thus indicated as plastic points. The plot of plastic points may be viewed after the presentation of the initial effective stresses in the Output program by selecting the *Plastic points* option from the *Stresses* sub-menu. Although the corrected stress state obeys the failure condition, it may result in a stress field which is not in equilibrium. It is generally preferable to generate an initial stress field that does not contain plastic points. For a cohesionless material it can easily be shown that to avoid soil plasticity the value of $K_0$ is bounded by:

$$\frac{1-\sin\varphi}{1+\sin\varphi} < K_0 < \frac{1+\sin\varphi}{1-\sin\varphi}$$

When the *K0 procedure* is adopted, PLAXIS will generate vertical stresses that are in equilibrium with the self-weight of the soil. Horizontal stresses, however, are calculated from the specified value of $K_0$. Even if $K_0$ is chosen such that plasticity does not occur, the *K0 procedure* does not ensure that the complete stress field is in equilibrium. Full equilibrium is only obtained for a horizontal soil surface with any soil layers parallel to this surface and a horizontal phreatic level. If the stress field requires only small equilibrium corrections, then these may be carried out using the calculation procedures described above. If the stresses are substantially out of equilibrium, then the *K0 procedure* should be abandoned in favour of the *Gravity loading* procedure.

At the end of the *K0 procedure*, the full soil weight is activated. In contrast to other PLAXIS programs, the soil weight can not be activated or de-activated in any other calculation phase.

### *Gravity loading*

If *Gravity loading* is adopted, then the initial stresses are set up by applying the soil self-weight in the first calculation phase.

In this case, when using an elastic perfectly-plastic soil model such as the Mohr-Coulomb model, the ratio of horizontal effective stress over vertical effective stress, $K_0$, depends strongly on the assumed values of Poisson's ratio. It is important to chose values of Poisson's ratio that give realistic values of $K_0$. If necessary, separate material data sets may be used with Poisson's ratio adjusted to provide the proper $K_0$-value during gravity loading. These sets may be changed by other material sets in subsequent calculations (Section 4.3.3). For one-dimensional compression an elastic computation will give:

$$\nu = \frac{K_0}{\left(1+K_0\right)}$$

If a value of $K_0$ of 0.5 is required, for example, then it is necessary to specify a value of Poisson's ratio of 0.333. As Poisson’s ratio must be lower than 0.5, it is not straightforward to generate $K_0$ values larger than 1 using *Gravity loading*. If $K_0$ values larger than 1 are desired, it is necessary to simulate the loading history or use the *K0 procedure*.

In some cases plastic points will be generated during the *Gravity loading* procedure. For cohesionless soils in one-dimensional compression, for example, plastic points will be generated unless the following inequality is satisfied:

$$\frac{1-\sin\varphi}{1+\sin\varphi} < \frac{\nu}{1-\nu} < 1$$

### *Results of initial stress generation*

If, after the generation of initial stresses, the Output program is started, a plot of the initial effective stresses is can be inspected.

Using $K_0$ values that differ substantially from unity may sometimes lead to an initial stress state that violates the Mohr-Coulomb criterion. The user can easily see if this is the case by inspecting the plot of *Plastic points*, which can be selected from the *Stresses* menu in the Output program. If this plot shows many red plastic points (Coulomb points), the value of $K_0$ should be chosen closer to 1.0. If there are a small number of plastic points, it is advisable to perform a plastic nil-step. When using the Hardening Soil model and defining a normally consolidated initial stress state (*OCR* = 1.0 and *POP* = 0.0), the plot of plastic points shows many blue cap points. Users need not be concerned about these plastic points as they just indicate a normally consolidated stress state.

### *Plastic nil-step*

If the *K0 procedure* generates an initial stress field that is not in equilibrium or where plastic points occur, then a plastic nil-step should be adopted. A plastic nil-step is a plastic calculation step in which no additional load is applied (Section 4.3.6). After this step has been completed, the stress field will be in equilibrium and all stresses will obey the failure condition.

If the original *K0 procedure* generates a stress field that is far from equilibrium, then the plastic nil-step may fail to converge. This happens, for example, when the *K0 procedure* is applied to problems with very steep slopes. For these problems the *Gravity loading* procedure should be adopted instead.

It is important to ensure that displacements calculated during a plastic nil-step (if one is used) do not affect later calculations. This may be achieved by using the *Reset displacements to zero* option in the subsequent calculation phase.

## 4.2 LOAD STEPPING PROCEDURES

When soil plasticity is involved in a finite element calculation the equations become non-linear, which means that the problem needs to be solved in a series of calculation steps. An important part of the non-linear solution procedure is the choice of step size and the solution algorithm to be used.

During each calculation step, the equilibrium errors in the solution are successively reduced using a series of iterations. The iteration procedure is based on an accelerated initial stress method. If the calculation step is of a suitable size then the number of iterations required for equilibrium will be relatively small, usually about five to ten. If the step size is too small, then many steps are required to reach the desired load level and computer time will be excessive. On the other hand, if the step size is too large then the number of iterations required for equilibrium may become excessive or the solution procedure may even diverge.

The 3D FOUNDATION program has an automatic load stepping procedure for the solution of non-linear plasticity problems. Users do not need to worry about the proper selection of load steps and numerical procedures, since the program will automatically use the most appropriate procedure to guarantee optimum performance.

The automatic load stepping procedure is controlled by a number of calculation control parameters (Section 4.2.1 and 4.2.2). There is a convenient default setting for most control parameters, which strikes a balance between robustness, accuracy and efficiency. For each calculation phase, the user can influence the automatic solution procedures by manually adjusting the control parameters. This can be done in the *Parameters* tab sheet of the *Phases* window. In this way it is possible to have a stricter control over step sizes and accuracy. Before proceeding to the description of the calculation control parameters, a detailed description is given of the solution procedures themselves.

### *Automatic step size procedure*

For each calculation phase the user specifies the new state or the total load that is to be applied at the end of this phase. The calculation program will compare the new situation (at the end of this phase) with the previous situation (at the end of the phase where it starts from) and will solve the difference during the current calculation phase by applying multiple load steps. In fact, the program will try to reach equilibrium for the new situation in the final load step of the current phase.

The size of the first load step in a calculation phase is automatically determined by performing trial calculations, taking into account the *Tolerated error* (Section 4.2.2). When a new load step is applied (first step or later steps), a series of iterations is carried out to reach equilibrium. There are three possible outcomes of this particular process. These outcomes are:

- Case 1: The solution reaches equilibrium within a number of iterations that is less than the *Desired minimum* control parameter. By default, the *Desired minimum* is 4, but this value may be changed in the *Iterative procedure* group of the *Parameters* tab sheet in the *Phases* window (Section 4.2.2). If fewer iterations than the desired minimum are required to reach the equilibrium state then the calculation step is assumed to be too small. In this case, the size of the load increment is multiplied by two and further iterations are applied to reach equilibrium.
- Case 2: The solution fails to converge within a *Desired maximum* number of iterations. By default, the *Desired maximum* is 10, but this value may be changed in the *Iterative procedure* group of the *Parameters* tab sheet in the *Phases* window

(Section 4.2.2). If the solution fails to converge within the desired maximum number of iterations then the calculation step is assumed to be too large. In this case, the size of the load increment is reduced by a factor of two and the iteration procedure is continued.

- Case 3: The number of required iterations lies between the *Desired minimum* and the *Desired maximum* in which case the size of the load increment is assumed to be satisfactory. After the iterations are complete the next calculation step begins. The initial size of this calculation step is made equal to the size of the previous successful step.

If the outcome corresponds to either case 1 or case 2 then the process of increasing or reducing the step size continues until case 3 is achieved.

The calculation will proceed until one of the following situations occurs:

1. The total specified load has been applied. In this case the calculation phase has successfully finished. In the list of calculation phases the phase is preceded by a green tick mark (√) and in the *General* tab sheet of the *Phases* window a message is displayed in the *Log info* box, indicating that the specified situation has been reached.

2. A collapse load has been reached. In this case the total specified load has not been applied. Collapse is assumed when the applied load reduces in magnitude in two successive calculation steps. In the list of calculation phases the phase is preceded by a red cross (×) and in the *General* tab sheet of the *Phases* window the following message is displayed in the *Log info* box: *Prescribed ultimate state not reached; Soil body collapses*.

3. The maximum specified number of additional load steps (*Additional steps*; Section 4.2.1) has been applied. In this case it is likely that the calculation stopped before the total specified load has been applied. In the list of calculation phases the phase is preceded by a red cross (×) and in the *General* tab sheet of the *Phases* window the following message is displayed in the *Log info* box: *Prescribed ultimate state not reached; Not enough load steps*. It is advised to recalculate the calculation phase with an increased number of *Additional steps*.

4. The *Cancel* button was pressed. In this case the total specified load has not been applied. In the list of calculation phases the phase is preceded by a red cross (×) and in the *General* tab sheet of the *Phases* window the following message is displayed in the *Log info* box: *Prescribed ultimate state not reached; Cancelled*.

5. A numerical error has occurred. In this case the total specified load has not been applied. There may be different causes for a numerical error. Most likely, it is related with an input error. It is suggested to carefully inspect the input data, the finite element mesh and the defined calculation phase. In the list of calculation phases the phase is preceded by a red cross (×) and in the *General* tab sheet of the *Phases* window the following message is displayed in the *Log info* box: *Prescribed ultimate state not reached; Numerical error*.

### 4.2.1 CALCULATION CONTROL PARAMETERS

The *Parameters* tab sheet in the *Phases* window is used to define the control parameters of the automatic load stepping procedure for each individual calculation phase (Figure 4.4).

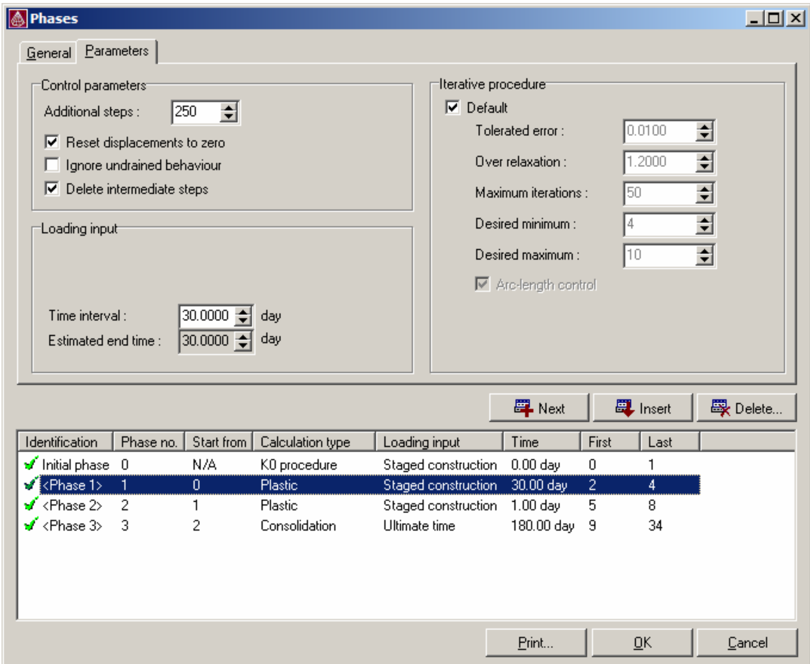



Figure 4.4 *Parameters* tab sheet of the *Phases* window

#### *Additional steps*

This parameter specifies the maximum number of calculation steps that is performed in a particular calculation phase. The number of additional steps should be set to an integer number representing a maximum of the required number of load steps for this calculation phase. In fact, the number of additional steps is an upper bound to the actual number of steps that will be executed. In general, it is desired that a calculation is completed within the number of additional steps and stops according to the first or second criterion as described in Section 4.2 (specified load reached or collapse load reached). If a calculation reaches the maximum number of additional steps, it usually means that the ultimate level has not been reached. In that case the user needs to increase the number of *Additional steps* and rerun the calculation phase.

By default, the *Additional steps* parameter is set to 250, which is generally sufficient to complete the calculation phase. However, this number may be changed within the range 1 to 1000.

### *Reset displacements to zero*

This option should be selected when irrelevant displacements of previous calculation steps are to be disregarded at the beginning of the current calculation phase, so that the new calculation starts from a zero displacement field. For example, deformations due to gravity loading are physically meaningless.

Hence, this option should be chosen after gravity loading to remove these displacements. If the option is not selected then incremental displacements occurring in the current calculation phase will be added to those of the previous phase. The selection of the *Reset displacements to zero* option does not influence the stress field.

### *Ignore undrained behaviour*

This option should be selected if it is desired to exclude temporarily the effects of undrained behaviour in situations where undrained material clusters are used. As a result, all undrained material clusters become temporarily drained. Existing excess pore pressures that were previously generated will remain, but no new excess pore pressures will be generated in that particular calculation phase.

*Gravity loading* of undrained materials will result in unrealistic excess pore pressures. Stresses due to the self-weight of the soil, for example, are based on a long-term process in which the development of excess pore pressures is irrelevant. The *Ignore undrained behaviour* option enables the user to specify the material type from the beginning as undrained for the main loading stages and to ignore the undrained behaviour during the gravity loading stage (Initial phase).

Hence, the behaviour of all undrained clusters is considered to be drained during this preliminary calculation.

### *Delete intermediate steps*

This option is by default selected to save disk space. As a result, all additional output steps within the calculation phase, except for the last one, are deleted when a calculation phase has finished successfully. In general the final output step contains the most relevant results of the calculation phase, whereas intermediate steps are less important.

If desired, the option can be de-selected to retain all individual output steps. If a calculation phase does not finish successfully then all calculation steps are retained, regardless of the selection of the *Delete intermediate steps* option. This enables a stepwise evaluation of the cause of the problem.

### *Time interval, Realised end time, Estimated end time:*

These time parameters control the progress of time in the calculations. All time parameters are expressed in the unit of time as defined in the *Dimensions* tab sheet of the *General settings* window. A non-zero value for the *Time interval* parameters is only relevant when a consolidation analysis is performed or when using time-dependent material models (such as the Soft Soil Creep model).

The meaning of the various time parameters is described below:

- *Time interval* is the total time period considered in the current calculation phase.
- *Realised end time* is the actual accumulated time at the end of a finished calculation phase.
- *Estimated end time* is an estimation of the accumulated time at the end of a phase that is to be calculated. This parameter is estimated from the *Time interval* of the current phase and the *Realised* or *Estimated end time* of the previous phase.

### 4.2.2 ITERATIVE PROCEDURE CONTROL PARAMETERS

The iterative procedures, in particular the automatic step size procedures, are influenced by some control parameters. These parameters can be set in the *Iterative procedure* group of the *Parameters* tab sheet in the *Phases* window. The 3D FOUNDATION program has an option to adopt a standard setting for these parameters, which gives in most cases good performance of the iterative procedures.

Users who are not familiar with the influence of the control parameters on the iterative procedures are advised to select the *Default* check box. In some situations, however, it might be desired or even necessary to change the default setting. In this case the user should de-select the *Default* check box and enter the parameters manually.

#### *Tolerated error*

In any non-linear analysis where a finite number of calculation steps are used there will be some drift from the exact solution, as shown in Figure 4.5.

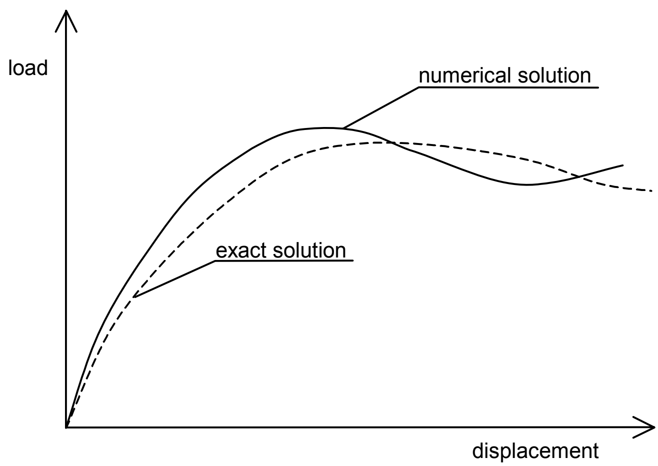



Figure 4.5 Computed solution versus exact solution

The purpose of a solution algorithm is to ensure that the equilibrium errors, both locally and globally, remain within acceptable bounds (Section 4.11). The error limits adopted in the 3D FOUNDATION program are linked closely to the specified value of the *Tolerated error*. Within each step, the calculation program continues to carry out iterations until the calculated errors are smaller than the specified value.

If the tolerated error is set to a high value then the calculation will be relatively quick but may be inaccurate. If a low tolerated error is adopted then computer time may become excessive. In general, the standard setting of 0.01 is suitable for most calculations.

If a calculation gives failure loads that tend to reduce unexpectedly with increasing displacement, then this is a possible indication of excessive drift of the finite element results from the exact solution. In these cases the calculation should be repeated using a lower value of the tolerated error. For further details of the error checking procedures used in the 3D FOUNDATION program see Section 4.11.

**Hint:** Be careful when using a tolerated error larger than the default value of 0.01, as this may give inaccurate results which are not in equilibrium.

### *Over-relaxation*

To reduce the number of iterations needed for convergence, the 3D FOUNDATION program makes use of an over-relaxation procedure as indicated in Figure 4.6. The parameter that controls the degree of over-relaxation is the over-relaxation factor. The theoretical upper bound value is 2.0, but this value should never be used. For low soil friction angles, for example $\varphi < 20°$, an over-relaxation factor of about 1.5 tends to optimise the iterative procedure.

If the problem contains soil with higher friction angles, however, then a lower value may be required. The standard setting of 1.2 is acceptable in most calculations.

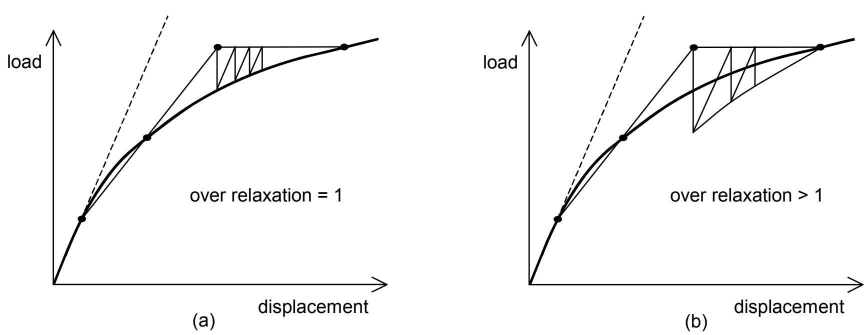



Figure 4.6 Iteration process without (a) and with (b) over-relaxation

### *Maximum iterations*

This value represents the maximum allowable number of iterations within any individual calculation step. In general, the solution procedure will restrict the number of iterations that take place. This parameter is required only to ensure that computer time does not become excessive due to errors in the specification of the calculation. The standard value of *Maximum iterations* is 50, but this number may be changed within the range 1 to 100.

### *Desired minimum and desired maximum*

The 3D FOUNDATION program makes use of an automatic step size algorithm. This procedure is controlled by the two parameters *Desired minimum* and *Desired maximum*, specifying the desired minimum and maximum number of iterations per step respectively. The standard values of these parameters are 4 and 10 respectively, but these numbers may be changed within the range 1 to 100. For details on the automatic step size procedures see Section 4.2.

It is occasionally necessary for the user to adjust the values of the desired minimum and maximum from their standard values. It is sometimes the case, for example, that the automatic step size procedure generates steps that are too large to give a smooth load-displacement curve. This is often the case where soils with very low friction angles are modelled. To generate a smoother load-displacement response in these cases, the calculations should be repeated with smaller values for these parameters, for example:

Desired minimum = 3　　　　　Desired maximum = 7

If the soil friction angles are relatively high, or if high-order soil models are used, then it may be appropriate to increase the desired minimum and maximum from their standard values to obtain a solution without the use of excessive computer time. In these cases the following values are suggested:

Desired minimum = 7　　　　　Desired maximum = 15

In this case it is recommended to increase the *Maximum iterations* to 75.

### *Arc-length control*

The *Arc-length* control procedure is a method that is by default selected in the 3D FOUNDATION program to obtain reliable collapse loads for load-controlled calculations. The iterative procedure adopted when arc-length control is not used is shown in Figure 4.7a for the case where a collapse load is being approached. In the case shown, the algorithm will not converge. If arc-length control is adopted, however, the program will automatically evaluate the portion of the external load that must be applied for collapse as shown in Figure 4.7b. Arc-length control is activated by selecting the corresponding check box in the *Iterative procedure* box of the *Parameter* tab sheet. Arc-length control will influence the size of load increments, but it does not influence the end result of a calculation phase.

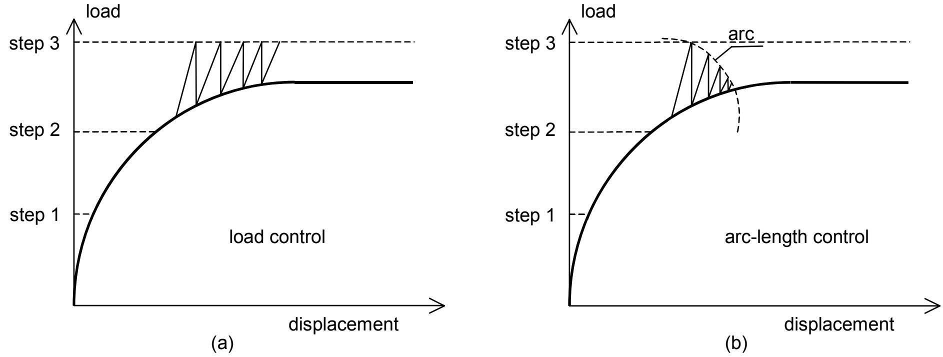



Figure 4.7 Iterative procedure for normal load control (a) and arc-length control (b)

| | |
|---|---|
| **Hint:** | The use of arc-length control occasionally causes spontaneous unloading to occur (i.e. sudden changes in sign of the displacement and load increments) when the soil body is far from collapse. If this occurs, then the user is advised to restart the calculation, de-selecting the *Default* check box and de-selecting *Arc-length control*. Note that if arc-length control is deselected and failure is approached, convergence problems may occur. |

## 4.3 STAGED CONSTRUCTION

*Staged construction* is a very useful approach to the specification of loads and construction stages. In this special PLAXIS feature it is possible to change the geometry and load configuration by activating or deactivating loads, soil volume clusters or structural objects as created in the geometry input. Staged construction enables an accurate and realistic simulation of various loading, construction and excavation processes. The option can also be used to reassign material data sets or to change the water pressure distribution in the geometry.

The staged construction facility is available in the *Calculation* mode of the Input program. To carry out a staged construction calculation, it is first necessary to properly create a geometry model that includes all of the objects that are to be used during the calculation. Objects that are not required at the start of the calculation should be deactivated in the *Initial phase* using the staged construction facility.

If the *Phases* window is open, it must first be closed by clicking on the <OK> button before changes can be made to the geometry configuration.

Most geometry changes can be made in the *Calculation* mode by clicking on the corresponding objects in the draw area. In most cases a selection window appears from which a further selection of the item or items to be activated or de-activated or changed must be made. Changes to the geometry configuration generally cause substantial out-of-balance forces. These out-of-balance forces are stepwise applied to the finite element mesh using the automatic load stepping procedures.

In each calculation phase, a parameter that controls the staged construction process (*ΣMstage*) is stepwise increased. At the beginning of the phase, *ΣMstage* is equal to zero. This situation actually represents the reference situation as defined by the *Start from phase* parameter (Section 4.1.3). At the end of the phase, *ΣMstage* is equal to unity, at least when the calculation was successful. The situation *ΣMstage* = 1 represents the situation as defined for the current calculation phase. In the initial phase, *ΣMstage* has the meaning of the proportion of the material self weight that has been applied, which is in other PLAXIS programs known as *ΣMweight*.

### 4.3.1 ACTIVATING AND DEACTIVATING CLUSTERS OR STRUCTURAL OBJECTS

Loads, soil volume clusters or structural objects may be activated or deactivated to simulate a process of construction or excavation. Hence, it is possible, for example, to first make an excavation with soil retaining walls, then install a basement floor and subsequently constructing the building above. In this way the three-dimensional effects around the excavation can be analysed realistically.

Before changing geometry objects, the desired work plane must be selected from the *Active work plane* combo box in the general toolbar. The draw area indicates which objects in or below the active work plane are active by showing their original colour. Active soil clusters below the active work plane are drawn in the material data set colour whereas inactive soil clusters below the active work plane are drawn in the background colour (white). Active loads or structural objects are drawn in their original colour, whereas inactive loads or structural objects are drawn in grey. When clicking once on an object, the object will change from active to passive, and vice versa. If more than one object is present on a cluster or geometry line (for example plates and distributed loads), a selection window appears from which the desired object can be selected.

For objects in between two work planes (vertical beams, walls or soil volume clusters), in general the work plane above the object should be selected, from where the object below can be activated or deactivated. For soil, the selection window also allows for activation or deactivation of soil clusters above the active work plane, and thus a selection window appears that allows for the option *soil above* and *soil below*. Interfaces are always activated and deactivated together with the adjacent soil clusters and cannot be activated or deactivated separately.

In addition to the selection of geometry objects from the selection window, this window also allows for a change of properties. In this way the input value of loads or the material properties of objects can be changed. A change of properties can be defined by first selecting the desired object and then pressing the *Change* button in the selection window.

The activation or deactivation of soil volume clusters or structural objects or a change of properties can introduce substantial out-of-balance forces. These out-of-balance forces are solved during the staged construction calculation process.

### 4.3.2 CHANGING LOADS

After selecting the corresponding work plane and clicking on a geometry line or cluster where a load is present, the selection window appears. When selecting the desired load, the input values of this load can be changed by pressing the *Change* button in the selection window.

During creation in the *Model* mode, a default value is given to the load which represents a unit load in vertical direction. These load values may be changed in each calculation phase to simulate changing loads in the various stages of construction. The change of loads can introduce substantial out-of-balance forces. These out-of-balance forces are solved during the staged construction calculation process.

#### *Distributed load on a horizontal work plane*

When clicking on a cluster in a work plane where a distributed load is present, and subsequently selecting *Distributed load* from the selection window, the distributed load window appears as presented in Figure 4.8.

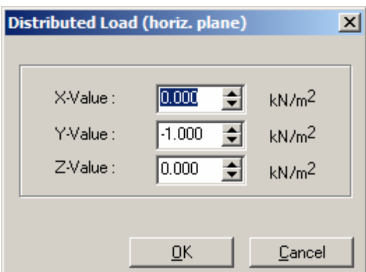



Figure 4.8 Input window for a distributed load on a horizontal plane

In this window, the *x*-, *y*- and *z*-components of the distributed load can be entered directly. Distributed loads on horizontal work plane clusters are always uniformly distributed.

#### *Line load*

When clicking on a point or a line in a work plane where a horizontal or vertical line load is created, and subsequently selecting *Line load* from the selection window, the line load window appears as presented in Figure 4.9. In this window, the *x*-, *y*- and *z*-components of the line load can be entered individually for the two corresponding geometry points. In this way, line loads can be distributed linearly over a geometry line.

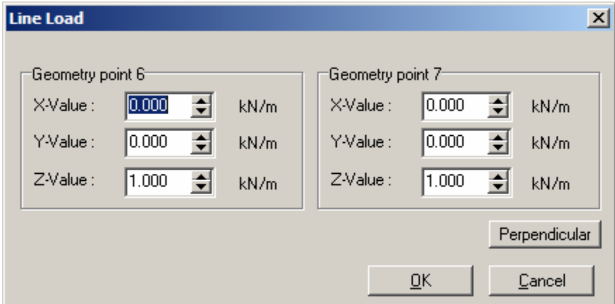



Figure 4.9 Input window for a line load

### *Distributed load on a vertical plane*

When clicking on a line in a work plane where a vertical distributed load (from the active work plane to the work plane below) is created, and subsequently selecting *Distributed load* from the selection window, the distributed loads window appears as presented in Figure 4.10.

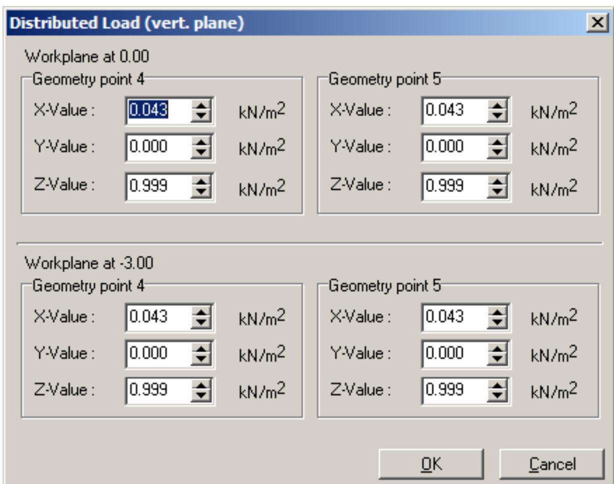



Figure 4.10 Input window for a distributed load on a vertical plane

In this window, the *x*-, *y*- and *z*-components of the distributed load can be entered individually for the corresponding geometry points, i.e. two points of the geometry line in the active work plane and two geometry points of the geometry line in the underlying work plane. Both in horizontal direction and in vertical direction a linear interpolation is made to obtain the actual values of the load in between the specified geometry points.

### *Point load*

When clicking on a point in a work plane where a point load is created, and subsequently selecting *Point load* from the selection window, the point load window appears as presented in Figure 4.11. In this window, the *x*-, *y*- and *z*-component of the point load can be entered individually.

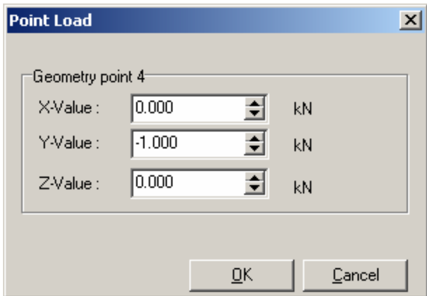



Figure 4.11 Input window for a point load

### 4.3.3 REASSIGNING MATERIAL DATA SETS

It is possible in a calculation phase to assign new material data sets to soil volume clusters or structural objects. This option may be used to simulate the change of material properties with time during the various stages of construction. The option may also be used to simulate soil improvement processes, e.g. removing poor quality soil and replacing it with soil of a better quality.

After selecting the corresponding work plane and clicking on a cluster or geometry line, the selection window appears. When selecting the desired object, the material properties of this object can be changed by pressing the *Change* button in the selection window. Instead of changing the data in the material data set itself, another data set should be assigned to the soil cluster or structural object. This ensures the consistency of data in the material database. Hence, if it is desired to change the properties of a soil volume cluster or structural object during a calculation, an additional data set should be created during the input of the model.

As an alternative for the change of material data sets for individual soil volume clusters, complete layers may be changed by reassigning material data sets in the boreholes. Therefore a borehole should be double-clicked and a new data set should be assigned to the desired soil layer by opening the material database and dragging/dropping the material data set on the corresponding layer in the borehole (Section 3.3.14). Please note that changing the data set of a soil layer in one particular borehole will also change the corresponding layer in all other boreholes. As a result, the whole soil layer will obtain the new properties, except for the soil volume clusters that were assigned other material data sets using the first option to change the material properties per cluster, as described above. A change of soil cluster properties using the selection window has priority over a change of soil properties using boreholes.

The change of soil cluster properties using the selection window can be undone using the *Reset* button in the selection window. This will reassign the soil properties from the borehole to the selected cluster. The change of soil properties using boreholes can be undone by reassigning the desired material in the borehole.

The change of certain properties, for example when replacing peat by dense sand, can introduce substantial out-of-balance forces due to a difference in unit weight. These out-of-balance forces are solved during the staged construction calculation process.

### 4.3.4 CHANGING WATER PRESSURE DISTRIBUTION

In addition to, or instead of, a change in the geometry configuration, the water pressure distribution in the geometry may also be changed. An example of a problem that may be analysed using this option is the calculation of additional settlements due to the lowering of the water table. Another example is the dry excavation and construction of a basement.

#### *Changing the water pressures in boreholes*

The distribution of the water pressure is specified by means of the boreholes. A borehole is accessible in the *Calculation* mode by double clicking on it. As a result, the *Borehole* window appears in which the actual water level is specified by the *Water level* parameter at the bottom of the window. The table of layer boundary levels includes two additional columns, one containing the pore pressure distribution above the layer boundary (WPress+) and one containing the pore pressure distribution below the layer boundary (WPress–). Pore pressures as well as external water pressures (if the water level is above the ground surface, and as a result acts as a distributed load on the soil surface) are generated on the basis of this information. Note that pressure is considered to be negative.

If the pore pressure distribution is hydrostatic, it can simply be generated on the basis of a water level (phreatic level). Therefore, the *Hydrostatic* parameter must be checked and the appropriate *Water level* must be entered in the corresponding edit box. The water pressures at the layer boundaries are automatically calculated from the water level and the unit water weight as entered in the *General settings* window. The values are presented in the table, but these values cannot be edited as long as the *Hydrostatic* parameter is checked. In the case that the water level is above the top layer boundary (i.e. above the ground surface) external water pressures will be generated.

If the pore pressure distribution is not hydrostatic, the *Hydrostatic* parameter must be unchecked. With this setting, the water pressures at the layer boundaries can be entered manually in the table. Distinction can be made between water pressures above the layer boundary (WPress+) and water pressures below the layer boundary (WPress–). When entering a non-zero value (pressure is negative!) for WPress+ at the top layer boundary (ground surface), this pressure is interpreted as an external water pressure. A change of the water level and/or pore pressure distribution in one borehole does not influence the other boreholes. Hence, when using multiple boreholes, the water pressure distribution must be changed in each individual borehole.

### *Cluster dry*

In addition to the global water pressure distribution it is possible to remove water pressures from individual volume clusters in order to make them 'dry', or to change the water pressures for an individual cluster using a prescribed pore pressure distribution of that cluster. This can be done by selecting the corresponding work plane and clicking on a cluster, after which the selection window appears. In addition to the soil volume cluster itself, the water in those clusters can be deactivated by deselecting the option *Water below* (or *Water above*) and removing the tick mark in front, which will result in zero pore pressures in these clusters. Deactivation of water can be done independent from the soil itself. Hence, if the soil is deactivated and the water level, as defined in the boreholes, is above the excavation level, then there is still water in the excavated area. If it is the user's intension to simulate a dry excavation, then the water must be explicitly deactivated.

A change of water pressures using the selection window (i.e. deactivating or changing *Water below* or *Water above*) has priority over a change of water pressures using boreholes.

After the new input, the water pressures will automatically be translated to nodal pore pressure values and external water pressures. The change of water pressures can introduce substantial out-of-balance forces. These out-of-balance forces are solved during the staged construction calculation process.

### *Cluster pore pressure distribution*

Alternatively, the user can select the option *Water above* (or *Water below*) and press the *Change* button. This will open the *Cluster pore pressure distribution* window (Figure 4.12) in which the pore pressure situation can be defined.

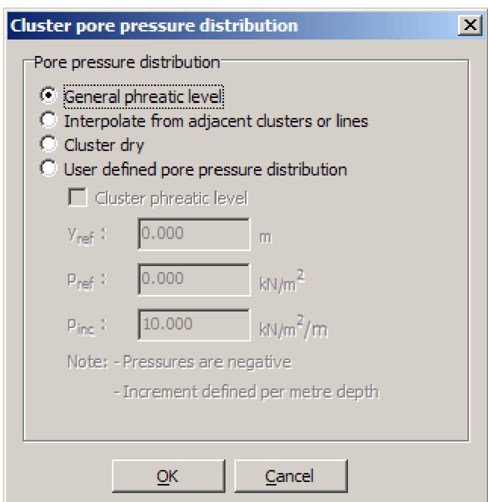



Figure 4.12 Cluster pore pressure distribution window

After selecting *Change*, four options will become available. The option *General phreatic level* can be used to revert back to the pore pressure distribution defined in the boreholes. Select *Cluster dry* if all pore pressures in the selected cluster are supposed to be zero. This has the same effect as deselecting the *Water above* or *Water below* tick mark in the selection window.

### *Interpolate pore pressure between clusters*

In the *Cluster pore pressure distribution* window it is also possible to generate pore pressures in a cluster based on the *Interpolate from adjacent clusters or lines* option. This option is, for example, used if a relatively impermeable layer is located between two permeable layers with a different groundwater head. The pore pressure distribution in the relatively impermeable layer will not be hydrostatic, so it cannot be defined by means of a phreatic level.

On selecting the option *Interpolate from adjacent clusters or lines,* the pore pressure in that cluster is interpolated linearly in a vertical direction, starting from the value at the bottom of the cluster above and ending at the value at the top of the cluster below, except if the pore pressure in the cluster above or below is defined by means of a user-defined pore pressure distribution. In the latter case the pore pressure is interpolated from the general phreatic level. The *Interpolate...* option can be used repetitively in two or more successive clusters (on top of each other). In the case that a starting value for the vertical interpolation of the pore pressure cannot be found, then the starting point will be based on the general phreatic level.

Please note that the *Interpolate* option does not take *User defined pore pressure distributions* into account.

### *User defined pore pressure distribution*

Alternatively, a *user defined pore pressure distribution* can be entered. When selecting this option, you can enter a reference level, $y_{ref}$, in the unit of length, a reference pressure, $p_{ref}$, in the unit of stress (i.e. the pore pressure at the reference level) and an increment of pressure, $p_{inc}$, in the unit of stress per unit of depth. In this way any linear pore pressure distribution can be defined. The reference level, $y_{ref}$, refers to the vertical level ($y$-coordinate) where the pore pressure is equal to the reference pressure, $p_{ref}$. If the cluster is (partly) located above the reference level, the pore pressure in that part of the cluster will also be equal to the reference pressure. Below the reference level, the pore pressure in the cluster is linearly increased, as set by the value of $p_{inc}$. Please note that the values of $p_{ref}$ and $p_{inc}$ are negative for pressure and pressure increase with depth, respectively. A user-defined pore pressure distribution cannot be used as a starting point to interpolate pore pressures in adjacent clusters. This should be taken into account when the *Interpolate pore pressures from adjacent clusters or lines* option is used in the cluster above or below.

If *User defined pore pressure distribution* is selected, the option *Cluster phreatic level* option becomes available. If this option is selected, only a phreatic level $y_{ref}$ needs to be entered to define the pore pressure distribution for the cluster. At the phreatic level pore pressures will be set equal to zero and will increase linearly below this level, based on the *Water weight* entered in the *General settings*. Actually, a cluster phreatic level is not necessarily a true phreatic level. In the case of an aquifer layer, the cluster phreatic level represents the pressure head, which is the height of water supported in a stand-pipe or piezometer tube.

A change of water pressures using the cluster pore pressure distribution window has priority over a change of water pressures using boreholes.

After the new input, the water pressures will automatically be translated to nodal pore pressure values and external water pressures. The change of water pressures can introduce substantial out-of-balance forces. These out-of-balance forces are solved during the staged construction calculation process.

### 4.3.5 APPLYING VOLUMETRIC STRAINS IN CLUSTERS

In addition to changing material properties for clusters, it is also possible to apply a volumetric strain in individual clusters. This can be done by selecting the corresponding work plane and clicking on a cluster, after which the selection window appears. If the soil volume cluster itself is activated, a volumetric strain in those clusters can be activated by selecting the option *Volumetric strain below* (or *Volumetric strain above*) and pressing the *Change* button. This will open the *Cluster volumetric strain* window (Figure 4.13), in which the volumetric strain can be specified. A positive value of the volumetric strain represents a volume increase (expansion), whereas a negative value represents a volume decrease. This option can be used to simulate mechanical processes that occur in the soil that result in volumetric strains, such as grouting.

Note that the imposed volumetric strain is not always fully applied, depending on the stiffness of the surrounding clusters and objects.

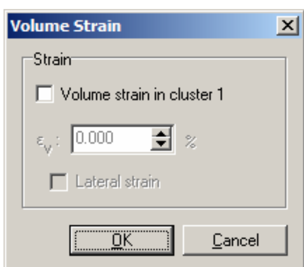



Figure 4.13 Volumetric strain window

#### *Applying lateral strains*

If the *Volumetric strain below* (or *Volumetric strain above*) option is selected for a soil cluster, it is possible to prescribe a lateral strain by selecting the *Lateral strain* option (Figure 4.13). Instead of applying a volumetric strain, the program now prescribes lateral strains for the soil cluster by specifying $\varepsilon_x = \varepsilon_z = ½\ \varepsilon_v$ and no axial strain, i.e. $\varepsilon_y =$

0. A positive value of the lateral strain represents an expansion, whereas a negative value represents a shrinkage.

Note that the imposed strain is not always fully applied, depending on the stiffness of the surrounding clusters and objects.

This option can be used to simulate the installation process of the pile in the soil and to increase the lateral stresses around the pile.

### 4.3.6 PLASTIC NIL-STEP

The staged construction calculation process may also be used to carry out a plastic nil-step. A plastic nil-step is a calculation phase in which no additional loading is applied. This may sometimes be required to solve large out-of-balance forces and to restore equilibrium. Such a situation can occur after a calculation phase in which large loadings were activated (for example gravity loading). In this case no changes should be made to the geometry configuration or to the water conditions. If necessary, such a calculation can be performed with a reduced *Tolerated error*.

### 4.3.7 STAGED CONSTRUCTION PROCEDURE IN CALCULATIONS

At the start of a staged construction calculation the information about active and inactive objects in the 3D model is transformed into information on element level. Hence, deactivating a soil cluster results in 'switching off' the corresponding soil elements during the calculation. The following rules apply for elements that have been switched off:

- All effective stresses and excess pore pressures are set to zero. If the element is reactivated then the stresses will be developed from zero. All inactive nodes have zero displacement.
- Boundaries that arise from the removal of elements are automatically taken to be free.
- Steady state pore pressures (not excess pore pressures) are always taken into account even for inactive soil elements. This means that PLAXIS will automatically generate suitable water pressures on submerged boundaries caused by the removal of elements. This may be checked by previewing the calculation phase. If, on the other hand, it is desired to remove the water from the excavated part of the soil, then a new water pressure distribution should be defined or the water should be explicitly de-activated (Section 4.3.4).

External loads or prescribed displacements that act on a part of the geometry that is inactive will not be taken into account.

### 4.3.8 UNFINISHED STAGED CONSTRUCTION CALCULATION

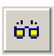
At the start of a staged construction calculation, the multiplier that controls the staged construction process, $\Sigma Mstage$, is zero and this multiplier is stepwise increased to 1.0.

When *ΣMstage* has reached the value of 1.0, the current phase is finished. However, if a staged construction calculation is not properly finished, i.e. the multiplier *ΣMstage* is less than 1.0 at the end of a staged construction analysis, then a warning appears in the *Log info* box.

There are two main reasons for an unfinished construction stage:

- Failure of the soil body has occurred during the calculation. This means that it is impossible to finish the construction stage. Note that the out-of-balance force is still partly unsolved so that further calculations starting from the last calculation phase are meaningless.
- The maximum number of loading steps was insufficient. In this case the construction stage should be continued by performing another staged construction calculation that is directly started without changing the geometry configuration or water pressures. Note that it is advised against applying any other type of loading as long as the multiplier *ΣMstage* has not reached the value 1.0.

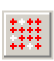
## 4.4 PREVIEWING A CONSTRUCTION STAGE



When a construction stage is fully defined, a 3D view of the situation can be presented by pressing the *Preview* button in the *Calculation* toolbar. This enables a direct visual check before the calculation is started. The 3D model is presented in the Output program. In addition to the 3D plot, tab sheets are presented in which the model can be viewed at the work plane levels. After the preview, press the *Update* button to return to the *Calculation* mode.

## 4.5 SELECTING POINTS FOR CURVES

After the calculation phases have been defined and before the calculation process is started, some points may be selected by the user for the generation of load-displacement curves or stress paths. During the calculation information for these selected points is stored in a separate file. After the calculation, the Curves program may be used to generate load-displacement curves or stress paths. The generation of these curves is based on the information stored in the separate file. It is therefore not possible to generate curves for points that have not been preselected. It is also not possible to add new nodes to the list after some calculation phases have been calculated without recalculating these phases. Hence, all nodes must be defined before any calculation phase is performed, or, alternatively, all calculation phases must be recalculated.



The points can be entered by selecting the *Select points for curves* option from the *View* menu or by clicking on the corresponding button in the toolbar. As a result, the Output program is opened showing the 3D finite element mesh. In addition, tab sheets are available in which the model can be viewed at the work plane levels. From the work planes, up to 10 nodes may be selected for the generation of load-displacement curves. Selection takes place by clicking the desired tab sheet, moving the mouse pointer to the desired node and clicking the left mouse button. When necessary the zoom option may be used. Selected nodes are indicated by characters in alphabetical order and are listed in the *Nodes* list box to the right. These characters will reappear in the Curves program to identify the points for which load-displacement curves are to be generated. A selected node can be deselected by clicking again on that node. To deselect all selected nodes, the *Deselect all nodes* button may be clicked.

In addition to the nodes, stress points may be selected for the generation of stress paths, strain paths and stress-strain diagrams. If the *3D* view of the geometry is selected, the *Stress points* option can be selected from the *Geometry* sub-menu. The resulting plot shows all stress points in the geometry The amount of visible stress points can be decreased using the *Partial geometry* option. Alternatively, the stress points directly below a particular work plane can be selected from the work plane tab sheets. Up to 10 stress points may be selected by clicking the left mouse button or double-clicking the stress point number in the lower *Stress points* list box to the right. As for the nodes, the stress points are indicated by characters in alphabetical order.

When all desired nodes and stress points have been selected, the *Update* button in the upper right corner should be pressed to store the information and return to the *Calculation* mode.

If the finite element mesh is regenerated (after being refined or modified) then the position of nodes and stress points will change. As a result, previously selected nodes and stress points may appear in completely different positions. Therefore nodes and stress points should be reselected after regeneration of the mesh.

When the calculations are started without the selection of nodes and stress points for curves, the user will be prompted to select such points. The user can then decide to select points or, alternatively, to start the calculations without selected points. In the latter case it will not be possible to generate load-displacement curves or stress-strain curves.

## 4.6 EXECUTION OF THE CALCULATION PROCESS

When calculation phases have been defined and points for curves have been selected, the calculation process can be executed. Before starting the process, however, it is useful to carefully check the list of calculation phases. In principle, all calculation phases indicated with a blue arrow (→) will be executed in the calculation process. By default, when defining a calculation phase, it is automatically selected for execution. A previously executed calculation phase is indicated by a green tick mark (√) if the calculation was successful, otherwise it is indicated by a red cross (×). To select or deselect a calculation phase for execution, the corresponding line should be double clicked in the *Phases* window.

The calculation process can be started by pressing the *Calculate* button in the *Calculation* toolbar. This button is only visible if a calculation phase is focused that is selected for execution, as indicated by the blue arrow. As a result, the program first performs a check on the ordering and consistency of the calculation phases. In addition, the first calculation phase to be executed is determined and all selected calculation phases in the list are subsequently executed, provided that failure does not occur. To inform the user about the progress of the calculation process, the active calculation phase will be focused in the list.

## 4.7 ABORTING A CALCULATION

If, for some reason, the user decides to abort a calculation, this can be done by pressing the *Cancel* button in the separate window that displays information about the iteration process of the current calculation phase.

If the *Cancel* button is pressed, the total specified load will not be applied. In the list of calculation phases the phase is preceded by a red cross (×) and in the *General* tab sheet of the *Phases* window the following message is displayed in the *Log info* box: *Prescribed ultimate state not reached; Cancelled*.

## 4.8 OUTPUT DURING CALCULATIONS

During a 3D finite element deformation analysis, information about the iteration process is presented in a separate window. The information comprises the current value of the *ΣMstage* parameter and other parameters for the particular calculation phase. The significance of the *ΣMstage* parameter is described in Section 4.3. In addition, the following information is presented in the window (Figure 4.14):

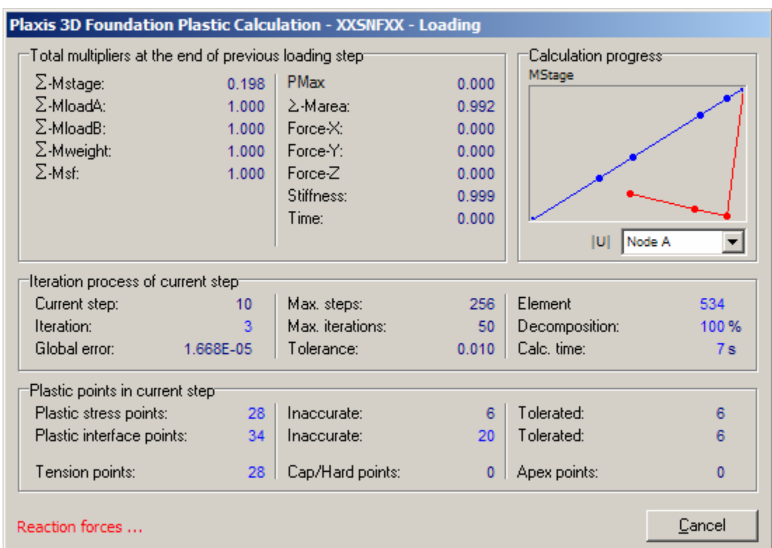



Figure 4.14 Calculation information window

### *Load-displacement curve*

A small load-displacement curve is shown in the upper right-hand side of the window. By default, the displacement of the first preselected node for curves (Section 4.5) is plotted here against the development of the *ΣMstage* parameter. This graph may be used to roughly evaluate the progress of the calculation. If desired, one of the other preselected nodes may be chosen from the combo box under the curve. If a consolidation analysis is being calculated, the graphs displays the development of the maximum excess pore pressure against the logarithm of time.

### *Step and iteration numbers*

The *Current step* and *Iteration* values indicate the current calculation step and iteration number. The *Maximum steps* value indicates the last step number of the current calculation phase according to the *Additional steps* parameter (Section 4.2.1). The *Maximum iterations* value corresponds to the *Maximum iterations* parameter in the settings for the iterative procedure (Section 4.2.2).

### *Global error*

The *Global error* is a measure of the global equilibrium errors within the calculation step. These errors tend to reduce as the number of iterations increases. For further details of this parameter see Section 4.11.

### *Stiffness*

The *Stiffness* parameter gives an indication of the amount of plasticity that occurs in the calculation. The *Stiffness* is defined as

$$Stiffness = \int \frac{\Delta\varepsilon \cdot \Delta\sigma}{\Delta\varepsilon D^e \Delta\varepsilon}$$

When the solution is fully elastic, the *Stiffness* is equal to unity, whereas at failure the stiffness approaches zero. The *Stiffness* is used in determining the *Global error*. See Section 4.11 for more details.

### *Tolerance*

The *Tolerance* is the maximum global equilibrium error that is allowed. The value of the *Tolerance* corresponds to the value of the *Tolerated error* in the settings for the iterative procedure (Section 4.2.2). The iteration process will at least continue as long as the *Global error* is larger than the *Tolerance*. For details see Section 4.11.

### *Plastic stress points*

This is the total number of stress points in soil elements that are in a plastic state. In addition to the points where Mohr's circle touches the Coulomb failure envelope, the points due to hardening plasticity are included.

### *Plastic interface points:*

This gives the total number of stress points in interface elements that have become plastic.

### *Inaccurate stress points:*

The *Inaccurate* values give the number of plastic stress points in soil elements and interface elements for which the local error exceeds the tolerated error. For further details see Section 4.11.

### *Tolerated number of inaccurate stress points:*

The *Tolerated* values are the maximum number of inaccurate stress points in soil elements and interface elements respectively that are allowed. The iteration process will at least continue as long as the number of inaccurate stress points is larger than the tolerated number. For further details see Section 4.11.

### *Tension points:*

A *Tension point* is a stress point that fails in tension. These points will develop when the *Tension cut-off* is used in some of the material sets, as explained in Section 3.5.3.

### *Cap/Hard points:*

A *Cap point* occurs if the *Hardening Soil model* or the *Soft Soil Creep model* is used and the stress state in a point is equivalent to the preconsolidation stress, i.e. the maximum stress level that has previously been reached (OCR ≤ 1.0). A *Hard(ening) point* occurs if the *Hardening Soil model* is used and the stress state in a point corresponds to the maximum mobilised friction angle that has previously been reached.

### *Apex points:*

These are special plastic points where the allowable shear stress is zero, i.e. $\tau_{max} = c + \sigma \tan\varphi = 0$. The iterative procedure tends to become slow when the number of plastic apex points is large.

Apex points can be avoided by selecting the *Tension cut-off* option in the material data sets for soil and interfaces.

### *Calculation status*

In addition to the calculation time, the status line (located at the bottom of the calculation window) indicates what part of the calculation process is currently being executed. The following processes are indicated:

- *Reading data ...*
  when reading calculation data from disk.

- *Renumbering ...*
  when optimising the node numbering and determining matrix properties.
- *Forming matrix ...*
  when forming the global stiffness matrix.
- *Solving equations ...*
  when solving the global system of equations to obtain the displacement increments.
- *Calculating stresses ...*
  when calculating the strain increments and constitutive stresses.
- *Reaction forces ...*
  when calculating the reaction forces and the out-of-balance forces.
- *Writing data ...*
  when writing output data to disk.

### *Cancel button*

If, for some reason, the user decides to abort a calculation, this can be done by pressing the *Cancel* button on the calculation window. By pressing this button, the calculation process is aborted and control is returned to *Calculation* mode of the user interface. Note that after pressing the button it may take a few seconds before the calculation process is actually stopped. In the calculations list, a red cross (×) appears in front of the aborted calculation phase, indicating that the phase was not successfully finished. Moreover, the execution of all further calculation phases is stopped.

## 4.9 SELECTING CALCULATION PHASES FOR OUTPUT

After the calculation process has finished, the calculation list is updated. Calculation phases that have been successfully finished are indicated by a green tick mark (√), whereas phases that did not finish successfully are indicated by a red cross (×). In addition, messages from the calculations are displayed in the *Log info* box of the *General* tab sheet in the *Phases* window.

When a calculation phase, which has been previously executed, is selected, then the toolbar shows an *Output* button. On selecting a finished calculation phase and clicking on the *Output* button, the results of the selected phase are directly displayed in the Output program.

## 4.10 ADJUSTMENTS TO INPUT DATA IN BETWEEN CALCULATIONS

Care should be taken with the change of input data (in the Input program) in between calculation phases. In general, this should not be done since it causes the input to cease to be consistent with the calculation data. In most cases there are other ways to change data in between calculation phases instead of changing the input data itself.

When changing the geometry (i.e. changing the position of points or lines or adding new objects), the program will reset all data related to construction stages to the initial configuration. This is done because, in general, after a change of the geometry the staged construction information ceases to be valid. When doing so, the mesh has to be regenerated. In the *Calculation* mode, the user has to redefine the construction stages and the calculation process must restart from the *Initial phase*.

When the finite element mesh is regenerated without changing the geometry (for example to refine the mesh), then all calculation information (including construction stages) is retained. Note that in this case it is still necessary to restart the calculation from the *Initial phase*.

When changing material properties in existing data sets without changing the geometry, then all calculation information is retained as well. In this case, clusters refer to the same data sets, but the properties as defined in these data sets have changed. This procedure is useful in the case that the same calculation is repeated with modified parameters to perform a sensitivity analysis. However, in that case it is recommended to save the modified project under a new name.

The same applies to a change in water pressures and a change in input values of existing loads. Please note that the change of material properties, water pressures and loads is also possible within the *Staged construction* facility (Section 4.3).

## 4.11 AUTOMATIC ERROR CHECKS

During each calculation step, the calculation kernel performs a series of iterations to reduce the out-of-balance errors in the solution. To terminate this iterative procedure when the errors are acceptable, it is necessary to establish the out-of-balance errors at any stage during the iterative process automatically. The out-of-balance error is based on a global equilibrium error. This value must be below a predetermined limit for the iterative procedure to terminate.

### *Global error check*

The global error checking parameter used in the calculation kernel is related to the sum of the magnitudes of the out-of-balance nodal forces. The term 'out-of-balance nodal forces' refers to the difference between the external loads and the forces that are in equilibrium with the current stresses. To obtain this parameter, the out-of-balance loads are made dimensionless as shown below:

$$Global\ error = \frac{\Sigma \left\| Out\ of\ balance\ nodal\ forces \right\|}{\Sigma \left\| Active\ loads \right\| + CSP \cdot \left\| Inactive\ loads \right\|}$$

Here CSP is the current value of the *Stiffness* parameter, defined as:

$$Stiffness = \int \frac{\Delta\varepsilon \cdot \Delta\sigma}{\Delta\varepsilon\, D^e \Delta\varepsilon}$$

which is a measure for the amount of plasticity that occurs during the calculation. See the Material Models Manual, Section 2.3, for more information on the stiffness parameters. When the solution is fully elastic, the *Stiffness* is equal to unity, whereas at failure the *Stiffness* approaches zero. This means that the solution becomes more accurate as more plasticity occurs, since the denominator in the *Global error* expression reduces.

### *Local error check*

Local errors refer to the errors at each individual stress point. To understand the local error checking procedure used in PLAXIS it is necessary to consider the stress changes that occur at a typical stress point during the iterative process.

The variation of one of the stress components during the iteration procedure is shown in Figure 4.15. At the end of each iteration, two important values of stress are calculated by PLAXIS. The first of these, the 'equilibrium stress', is the stress calculated directly from the stiffness matrix (e.g. point **A** on Figure 4.15). The second important stress, the 'constitutive stress', is the value of stress on the material stress-strain curve at the same strain as the equilibrium stress, i.e. point **B** on Figure 4.15.

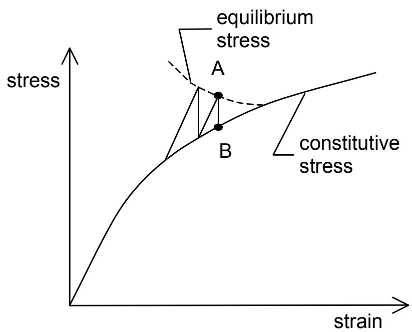



Figure 4.15 Equilibrium and constitutive stresses

The dashed line in Figure 4.15 indicates the path of the equilibrium stress. In general this equilibrium stress path depends on the nature of the stress field and the applied loading. For the case of a soil element obeying the Mohr-Coulomb criterion, the local error for the particular stress point at the end of the iteration is defined:

$$Local\ error = \frac{\left\| \sigma^{\mathrm{e}} - \sigma^{\mathrm{c}} \right\|}{T_{\max}}$$

In this equation the numerator is a norm of the difference between the equilibrium stress tensor, $\sigma^e$, and the constitutive stress tensor, $\sigma^c$. This norm is defined by:

$$\left\| \sigma^e - \sigma^c \right\| = \sqrt{\left(\sigma_{xx}^e - \sigma_{xx}^c\right)^2 + \left(\sigma_{yy}^e - \sigma_{yy}^c\right)^2 + \left(\sigma_{zz}^e - \sigma_{zz}^c\right)^2 + \left(\sigma_{xy}^e - \sigma_{xy}^c\right)^2 + \left(\sigma_{yz}^e - \sigma_{yz}^c\right)^2 + \left(\sigma_{zx}^e - \sigma_{zx}^c\right)^2}$$

The denominator of the equation for the local error is the maximum value of the shear stress as defined by the Coulomb failure criterion. In case of the Mohr Coulomb model, $T_{max}$ is defined as:

$$T_{\max} = \max\left(½\left(\sigma_3 - \sigma_1\right), c\cos\varphi\right)$$

When the stress point is located in an interface element the following expression is used:

$$Local\ error = \frac{\sqrt{\left(\sigma_n^e - \sigma_n^c\right)^2 + \left(\tau^e - \tau^c\right)^2}}{c_i - \sigma_n^c \tan\varphi_i}$$

where $\sigma_n$ and $\tau$ represent the normal and shear stresses respectively in the interface. To quantify the local accuracy, the concept of *inaccurate plastic points* is used. A plastic point is defined to be inaccurate if the local error exceeds the value of the user specified *tolerated error* (see Section 4.2.2).

### *Termination of iterations*

For PLAXIS to terminate the iterations in the current load step, all of the following three error checks must be satisfied. For further details of these error-checking procedures see Reference 18.

$$Global\ error \leq Tolerated\ error$$

$$No.of\ \ inaccurate\ soil\ \ points\ \leq\ 3 + \frac{No.of\ \ plastic\ soil\ \ points}{10}$$

$$No.of\ \ inaccurate\ interface\ \ points\ \leq\ 3 + \frac{No.of\ \ plastic\ interface\ \ points}{10}$$

# 5 OUTPUT DATA (POST PROCESSING)

The main output quantities of a finite element calculation are the displacements and the stresses. In addition, when a finite element model involves structural elements, structural forces are calculated in these elements. An extensive range of facilities exist within the PLAXIS 3D FOUNDATION program to display the results of a finite element analysis. The set of facilities that may be selected from the Output program are described in this chapter.

## 5.1 THE OUTPUT PROGRAM

This icon represents the Output program. The Output program contains all facilities to view and list the results of generated input data and 3D finite element calculations. At the start of the Output program, the user has to select the model and the appropriate calculation phase or step number for which the results are to be viewed. After this selection a first output window is opened, displaying the deformed mesh in 3D view. In addition to the 3D view, it is possible to view output in individual work planes by selecting corresponding tab sheets in the *Output* window. The main window of the Output program contains the following items (Figure 5.1).

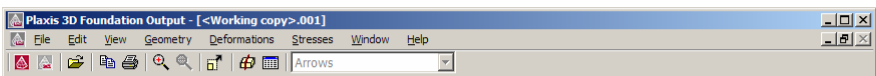



Figure 5.1 Main window of the Output program (without display area)

### *Output menu*

The *Output* menu contains all operation and output facilities of the Output program. The menu items may change, depending on the type of the active output form. Some options are also available as buttons in the toolbar.

### *Output forms*

These are windows on which particular output is displayed. Output forms may contain plots of the full model, plots of special objects of the model, or tables of output data. Multiple output forms may be opened simultaneously.

A graphical output form contains a series of tab sheets; one for the full 3D model and one for each individual work plane as defined by the user. By default, the full 3D model is presented (most right-hand tab sheet). To view results in a particular work plane, the user may select such a plane by clicking on the corresponding tab sheet. Planes are presented in the orientation as they appear in the 3D model.

### *Toolbar*

The toolbar contains buttons that may be used as a shortcut to menu facilities. In addition, a combo box is included that may be used to directly select the type of presentation of the displayed quantity. For example, displacements can be presented as *Arrows*, *Contours* or *Shadings*. Other quantities may be presented in other ways.

### *Status line*

The status bar may contain information about the viewing position and the position of the cursor (mouse) in the model. The cursor position is only given for a work plane.

## 5.2 THE OUTPUT MENU

The main menu of the Output program contains pull-down sub-menus covering most options for handling files, transferring data and viewing graphs and tables. The major type of results from a finite element calculation comprises deformations and stresses. Hence, these two aspects form the major part of the *Output* menu. When displaying a basic 3D geometry model, the total menu consists of the sub-menus *File*, *Edit*, *View*, *Geometry*, *Deformations*, *Stresses*, *Window* and *Help*. The menu depends on the type of data that is presented on the output form.

### *The File sub-menu*

| | |
|---|---|
| *Open* | To open a project for which output is to be viewed. The file requester is presented. |
| *Close* | To close the active output form. |
| *Close all* | To close all output forms. |
| *Print* | To print the active output on a selected printer. The print window is presented. |
| *Work directory* | To set the default directory where 3D FOUNDATION project files are stored. |
| (*recent projects*) | To quickly open one of the four most recent projects. |
| *Exit* | To leave the program. |

### *The Edit sub-menu*

| | |
|---|---|
| *Copy* | To copy the active output to the Windows clipboard. |
| *Scale* | To modify the scale of the presented quantity. |
| *Interval* | To modify the range of values of the presented quantity in contour line plots and plots with shadings. |

| | |
|---|---|
| *Scan line* | To change the scan line for displaying contour line labels. A scan line is only presented in work planes. After selection, the scan line must be indicated by the mouse. Press the left mouse button at one end of the line; hold the mouse button down and move the mouse to the other end. A contour line label will appear on each crossing of a contour line and the scan line. |

## *The View sub-menu*

| | |
|---|---|
| *Zoom in* | To zoom into a rectangular area on the screen for a more detailed view. After selection, the zoom area must be specified with the mouse.<br>Press the left mouse button at a corner of the zoom area; hold the mouse button down and move the mouse to the opposite corner of the zoom area; then release the button. The program will zoom into the selected area. The zoom option may be used repetitively. |
| *Zoom out* | To restore the view of before the most recent zoom action. |
| *Reset view* | To restore the original plot. |
| *Viewpoint* | To change the viewpoint of the 3D projection of the model. This allows rotation of the model or a selection from several predefined viewpoints. |
| *Cross-section* | To select a user-defined cross-section with a distribution of the presented quantity. The cross-section must be selected by the mouse. Press the left mouse button at one end of the cross-section; hold the mouse button down and move the mouse to the other end of the line. (Section 5.8). |
| *Table* | To view a table of numerical values of the presented quantity (Section 5.7). |
| *Title* | To show or hide the title of the active plot |
| *Legend* | To show or hide the legend of contours or shadings |
| *Axes* | To show or hide the *x*-, *y*- and *z*-axis in the active plot, provided the origin is in the range of model coordinates. |
| *Contour lines* | To show or hide the outer boundary contours of the geometry model. |
| *Shadow* | To activate or deactivate the option that object surfaces appear 'brighter' or 'darker', depending on their orientation with respect to the viewer. When this option is active, object surfaces appear most bright when the normal to the surface points in the direction of the viewer. The surface becomes darker the more it deviates from this direction. |

| | |
|---|---|
| *Partial geometry* | To show or hide a part of the geometry to view 'inside' the 3D model (Section 5.9.5). |
| *Distance measurement* | Allows the calculation of the distance between two nodes or stress points in the model. This option is only available when nodes or stress points are displayed (see the Geometry sub-menu below.) |
| *General info* | To view the general project information (Section 5.9.1). |
| *Load info* | To view tables of the active loads in the current step (Section 5.9.2). |
| *Material info* | To view the material data (Section 5.9.3). |
| *Calculation info* | To view the calculation information of the presented step (Section 5.9.4). |

## *The Geometry sub-menu*

| | |
|---|---|
| *Structures* | To display all structural objects in the model. |
| *Interfaces* | To display all interfaces in the model. |
| *Materials* | To display the material colours in the model. |
| *Phreatic level* | To display the general phreatic level in the model. |
| *Loads* | To display the external loads in the model. |
| *Fixities* | To display the user-defined fixities in the model. |
| *Connectivity plot* | To view the connectivity plot (Section 5.9.6) |
| *Nodes* | To display the nodes in the model (only for work planes). |
| *Stress points* | To display the stress points in the model |
| *Element /Chain numbers* | To display the soil element and chain numbers. Only possible when elements are displayed. |
| *Node numbers* | To display the node numbers. Only possible when nodes are displayed. |
| *Stress point numbers* | To display the stress point numbers. Only possible when stress points are displayed. |
| *Material set numbers* | To display the material set numbers in soil elements. |
| *Cluster numbers* | To display the cluster numbers in soil elements. |

## *The Deformations sub-menu*

The *Deformations* sub-menu contains various options to visualise the displacements, deformations and strains in the finite element model (Section 5.4). In principle, displacements are contained in the nodes of the finite element mesh, so displacement related output is presented on the basis of the nodes.

### *The Stresses sub-menu*

The *Stresses* sub-menu contains various options to visualise the stress state in the finite element model (Section 5.5). Stresses are contained in the integration points of the finite elements mesh, so stress related output is presented on the basis of the integration points.

## 5.3 SELECTING OUTPUT STEPS

Output may be selected by clicking on the *Open file* button in the toolbar or by selecting the *Open* option from the *File* sub-menu. As a result, a file requester is opened from which the desired PLAXIS 3D FOUNDATION project file (*.PF3) can be selected (Figure 5.2).

When the user selects a particular project, the file requester displays the corresponding list of calculation phases from which a further selection should be made. On selecting a calculation phase, a new output form is opened in which the results of the final calculation step of the selected phase are presented. If it is desired to select an intermediate calculation step, then a single mouse click should be given on the *Phase* column above the list with calculation phases in the file requester. As a result, the calculation list changes into a list with all available step numbers, from which the desired step number can be selected. Please note that all individual steps are only available if the option *Delete intermediate steps* has been deselected in the *Parameters* tab sheet of the *Phases* window.

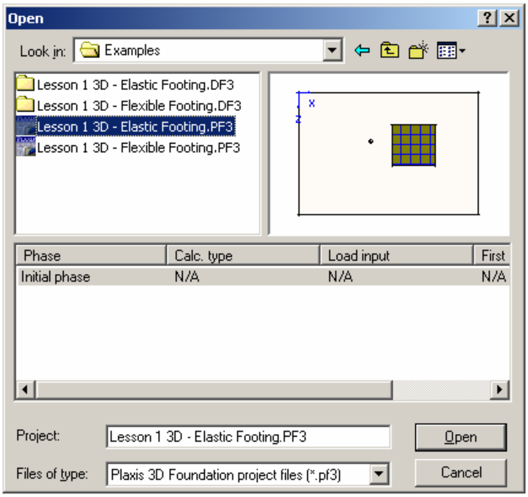
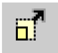



Figure 5.2 File requester for the selection of an output step

In addition to this general selection of output data, an alternative option is provided in the *Calculation* mode of the Input program, as described in Section 4.9.

## 5.4 DEFORMATIONS

The *Deformations* sub-menu contains various options to visualise the deformations and the strains in the finite element model. By default, the displayed quantities are scaled automatically by a factor (1, 2 or 5) $\cdot 10^{n}$ to give a diagram that may be read conveniently.



The scale factor may be changed by clicking on the *Scale factor* button in the toolbar or by selecting the *Scale* option from the *Edit* sub-menu. The scale factor for strains refers to a reference value of strain that is drawn as a certain percentage of the geometry dimensions. To compare strain plots of different calculation phases, the scale factors in the different plots must be made equal.

If *Contours* or *Shadings* are selected from the presentation box in the toolbar, then the range of values of the displayed quantity may be changed either by selecting the *Interval* option from the *Edit* sub-menu or by clicking on the legend. The maximum value of the particular quantity is included in the title underneath the plot and may be viewed by selecting the *Title* option from the *View* sub-menu.

### 5.4.1 DEFORMED MESH

The *Deformed mesh* is a plot of the finite element mesh in the deformed shape. By default, the deformations are scaled up to give a plot that may be read conveniently. If it is desired to view the deformations on the true scale (i.e. the geometry scale), then the *Scale* option may be used. The deformed mesh plot may be selected from the *Deformations* sub-menu.

### 5.4.2 TOTAL, HORIZONTAL AND VERTICAL DISPLACEMENTS

The *Total displacements* are the accumulated displacements at all nodes at the end of the current calculation step, displayed in a plot of the geometry. This option may be selected from the *Deformations* sub-menu. A further selection can be made among absolute (vectorial) total displacements, $|u|$, and the individual total displacement components, $u_x$, $u_y$ and $u_z$. The total displacements may be presented as *Arrows* or *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the toolbar.

### 5.4.3 PHASE DISPLACEMENTS

The *Phase displacements* are the accumulated displacement increments in the whole calculation phase at all nodes as calculated at the end of the current calculation step, displayed on a plot of the geometry. In other words, the phase displacements are the differential displacements between the end of the current calculation phase and the end of the previous calculation phase. This option may be selected from the *Deformations* sub-menu.

A further selection can be made among absolute (vectorial) phase displacements, $|\Sigma\Delta u|$, and the individual phase displacement components, $\Sigma\Delta u_x$, $\Sigma\Delta u_y$ and $\Sigma\Delta u_z$. The phase displacements may be presented as *Arrows* or *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the toolbar.

### 5.4.4 INCREMENTAL DISPLACEMENTS

The *Incremental displacements* are the displacement increments at all nodes as calculated for the current calculation step, displayed on a plot of the geometry. This option may be selected from the *Deformations* sub-menu. A further selection can be made among absolute (vectorial) displacement increments, $|\Delta u|$, and the individual displacement increment components, $\Delta u_x$, $\Delta u_y$ and $\Delta u_z$. The displacement increments may be presented as *Arrows* or *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the toolbar. The contours of displacement increment are particularly useful for the observation of localisation of deformations within the soil when failure occurs.

### 5.4.5 TOTAL STRAINS

The *Total strains* are the accumulated strains in the geometry at the end of the current calculation step, displayed in a plot of the geometry. This option may be selected from the *Deformations* sub-menu.

When this option is selected, a further selection can be made between volumetric strains or shear strains, which can be represented as *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the toolbar.

When a work plane is selected, the *Deformations* sub-menu also allows for the presentation of principal strain directions. These are, in fact, principal strains projected on the selected plane. Only one point per element (triangle) is displayed, which represents the average of all strains in the triangle. The length of each line represents the magnitude of the principal strain and the direction indicates the principal direction. Strain components that represent extension are indicated by an arrow rather than a line. Note that compression is considered to be negative.

### 5.4.6 CARTESIAN TOTAL STRAINS

When selecting *Cartesian total strains* from the *Deformations* sub-menu, a further selection can be made between the individual total strain components $\varepsilon_{xx}$ , $\varepsilon_{yy}$ , $\varepsilon_{zz}$ , $\gamma_{xy}$ , $\gamma_{yz}$ and $\gamma_{zx}$. Cartesian strain components can be represented as *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the toolbar.

### 5.4.7 INCREMENTAL STRAINS

The *Incremental strains* are the strain increments in the geometry as calculated for the current calculation step, displayed in a plot of the geometry. This option may be selected from the *Deformations* sub-menu. When this option is selected, a further selection can be made between volumetric strain increments or shear strain increments, which can be represented as *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the toolbar.

When a work plane is selected, the *Deformations* sub-menu also allows for the presentation of principal strain increment directions. These are, in fact, principal strain increments projected on the selected plane. Only one point per triangular element is displayed, which represents the average of the strain increment in the triangle. The length of each line represents the magnitude of the principal strain increment and the direction indicates the principal direction. Strain increments that represent extension are indicated by an arrow rather than a line. Note that compression is considered to be negative.

### 5.4.8 CARTESIAN INCREMENTAL STRAINS

When selecting *Cartesian incremental strains* from the *Deformations* sub-menu, a further selection can be made between the individual strain increment components $\Delta\varepsilon_{xx}$, $\Delta\varepsilon_{yy}$, $\Delta\varepsilon_{zz}$, $\Delta\gamma_{xy}$, $\Delta\gamma_{yz}$ and $\Delta\gamma_{zx}$. Cartesian strain increment components can be represented as *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the toolbar.

## 5.5 STRESSES

The *Stresses* sub-menu contains various options to visualise the stress state in the finite element model. When selecting *Contours* or *Shadings* from the presentation box in the toolbar, then selecting the *Interval* option from the *Edit* sub-menu may change the range of values of the displayed quantity. The maximum value of the particular quantity is included in the title underneath the plot and may be viewed by selecting the *Title* option from the *View* sub-menu.

### 5.5.1 EFFECTIVE STRESSES

The *Effective stresses* are the effective stresses (i.e. stresses in the soil skeleton) in the geometry at the end of the current calculation step, displayed in a plot of the geometry. This option may be selected from the *Stresses* sub-menu.

When this option is selected, a further selection can be made between effective mean stresses ($p'$), relative shear stresses ($\tau_{rel}$) or deviatoric stresses ($q$), which can be represented as *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the toolbar.

The relative shear stress option gives an indication of the proximity of the stress point to the failure envelope. The relative shear stress, $\tau_{rel}$, is defined as:

$$\tau_{rel} = \frac{\tau^*}{\tau_{max}}$$

where $\tau^*$ is the maximum value of shear stress (i.e. the radius of the Mohr stress circle). The parameter $\tau_{max}$ is the maximum value of shear stress for the case where the Mohr's circle is expanded to touch the Coulomb failure envelope keeping the centre of Mohr’s circle constant.

When a work plane is selected, the *Stresses* sub-menu box also allows for the presentation of principal stress directions. These are, in fact, principal stresses projected on the selected plane. Only one point per triangular element is displayed, which represents the average of the effective principal stress in the triangle. The length of each line represents the magnitude of the effective principal stress and the direction indicates the principal direction. Stresses that represent tension are indicated by an arrow rather than a line. Note that pressure is considered to be negative.

### 5.5.2 CARTESIAN EFFECTIVE STRESSES

When selecting *Cartesian effective stresses* from the *Stresses* sub-menu, a further selection can be made between the individual effective stress components $\sigma'_{xx}$, $\sigma'_{yy}$, $\sigma'_{zz}$, $\sigma_{xy}$, $\sigma_{yz}$ and $\sigma_{zx}$. Cartesian stress components can be represented as *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the toolbar. Figure 5.3 shows the sign convention adopted for Cartesian stresses. Note that pressure is considered to be negative.

### 5.5.3 TOTAL STRESSES

The *Total stresses* are the total stresses (i.e. effective stresses + active pore pressures) in the geometry at the end of the current calculation step, displayed in a plot of the geometry. This option may be selected from the *Stresses* sub-menu.

When a 3D plot is selected, only mean stresses ($p$) can be displayed, which can be represented as *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the toolbar.

When a work plane is selected, the *Stresses* sub-menualso allows for the presentation of principal stress directions. These are, in fact, principal stresses projected on the selected plane. Only one point per triangular element is displayed, which represents the average of the total principal stress in the triangle. The length of each line represents the magnitude of the total principal stress and the direction indicates the principal direction. Stresses that represent tension are indicated by an arrow rather than a line. Note that pressure is considered to be negative.

### 5.5.4 CARTESIAN TOTAL STRESSES

When selecting *Cartesian total stresses* from the *Stresses* sub-menu, a further selection can be made between the individual total stress components $\sigma_{xx}$, $\sigma_{yy}$, $\sigma_{zz}$, $\sigma_{xy}$, $\sigma_{yz}$ and $\sigma_{zx}$. Cartesian stress components can be represented as *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the toolbar. Figure 5.3 shows the sign convention adopted for Cartesian stresses. Note that pressure is considered to be negative.

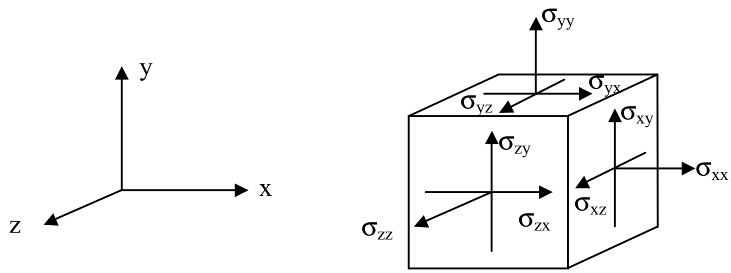



Figure 5.3 Sign convention for stresses

### 5.5.5 EQUIVALENT ISOTROPIC STRESS

The *Equivalent isotropic stress* is only available in the Hardening Soil model and the Soft Soil Creep model. The equivalent isotropic stress level is defined as

$$p^{eq} = \sqrt{p^2 + \frac{\tilde{q}^2}{\alpha^2}} \qquad \text{for the Hardening Soil model}$$

$$p^{eq} = p' - \frac{q^2}{M^2 (p' - c\cot(\varphi))} \qquad \text{for the Soft Soil Creep model}$$

The equivalent isotropic stress may be selected from the stress sub-menu. The distribution of equivalent isotropic stress can only be presented as contours or shadings.

### 5.5.6 ISOTROPIC PRECONSOLIDATION STRESS

The *Isotropic preconsolidation stress* is only available in the Hardening Soil model and the Soft Soil Creep model. The isotropic preconsolidation stress represents the maximum equivalent isotropic stress level that a stress point has experienced up to the current load step.

The isotropic preconsolidation stress may be selected from the stress sub-menu. The distribution of isotropic preconsolidation stress can only be presented as contours or shadings.

### 5.5.7 ISOTROPIC OVERCONSOLIDATION RATIO

The *Isotropic overconsolidation ratio* is only available in the Hardening Soil model and the Soft Soil Creep model. The isotropic overconsolidation ratio is the ratio between the isotropic preconsolidation stress and the current equivalent isotropic stress.

The isotropic overconsolidation ratio may be selected from the stress sub-menu. The distribution of isotropic overconsolidation ratio can only be presented as contours or shadings.

### 5.5.8 PLASTIC POINTS

The *Plastic points* are the stress points in a plastic state, displayed in a plot of the undeformed geometry. Plastic points can be shown in the 3D mesh or in the work planes. The plastic stress points are indicated by small symbols that can have different shapes and colours, depending on the type of plasticity that has occurred. A red open square (Mohr-Coulomb point) indicates that the stresses lie on the surface of the Coulomb failure envelope. A white solid square (tension cut-off point) indicates that the tension cut-off criterion was applied. A blue crossed square (cap point) represents a state of normal consolidation where the preconsolidation stress is equivalent to the current stress state. The latter type of plastic points only occurs if the Soft Soil Creep model or the Hardening Soil model is used. Additionally, for the Hardening Soil model a green square (hardening point) with a plus sign represents points on the shear hardening envelope, while a green crossed square is simultaneous a cap point and a hardening point. The Coulomb plastic points are particularly useful to check whether the size of the mesh is sufficient. If the zone of Coulomb plasticity reaches a mesh boundary (excluding the centre-line in a symmetric model) then this suggests that the size of the mesh may be too small. In this case the calculation should be repeated with a larger model.

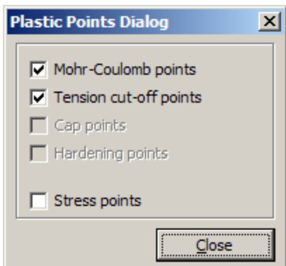



Figure 5.4 Plastic Points Dialog

When *Plastic points* is selected the *Plastic points Dialog* is shown (Figure 5.4). Here the user can select which types of plastic points are displayed. For details of the use of advanced soil models, the user is referred to the Material Models manual.

### 5.5.9 ACTIVE PORE PRESSURES

The *Active pore pressures* are the total water pressures $p_w$ (i.e. steady-state pore pressures + excess pore pressures) in the geometry at the end of the current calculation step, displayed in a plot of the undeformed geometry. This plot may be selected from the *Stresses* sub-menu. By default, active pore pressures are presented as shadings in a 3D plot, but they may also be viewed as contours by selecting the corresponding option from the presentation combo box.

In work planes, active pore pressures may also be viewed as principal stresses, although they are isotropic and do not have any principal directions. The length of lines represents the magnitude of the pore pressure and the directions coincide with the *x*- and *z*-axis.

Pore pressures that are tensile are indicated by an arrow rather than a line. Note that pressure is considered to be negative. In a work plane, only one point per element is displayed, which represents the average of the active pore pressure in that element in the plane.

### 5.5.10 EXCESS PORE PRESSURES

The *Excess pore pressures* are the water pressures due to loading of undrained clusters or by using the consolidation analysis at the end of the current calculation step, displayed in a plot of the undeformed geometry. This plot may be selected from the *Stresses* sub-menu. By default, excess pore pressures are presented as shadings in a 3D plot, but they may also be viewed as contours by selecting the corresponding option form the presentation combo box.

In work planes, excess pore pressures may also be viewed as principal stresses, although they are isotropic and do not have any principal directions.

The length of lines represents the magnitude of the pore pressure and the directions coincide with the *x*- and *z*-axis. Excess pore pressures that are tensile are indicated by an arrow rather than a line. Note that pressure is considered to be negative. In a work plane only one point per element is displayed, which represents the average of the excess pore pressure in that element in the plane.

### 5.5.11 STEADY STATE PORE PRESSURES

The Steady state pore pressures are the result of the water pressure generation based on the phreatic level and the cluster pore pressure distribution of the individual clusters. The definition of the pore pressure distribution is described in Section 4.3.4.

The steady state pore pressures are displayed in a plot of the undeformed geometry. This plot may be selected from the *Stresses* sub-menu. By default, excess pore pressures are presented as shadings in a 3D plot, but they may also be viewed as contours by selecting the corresponding option form the presentation combo box.

In work planes, excess pore pressures may also be viewed as principal stresses, although they are isotropic and do not have any principal directions.

The length of lines represents the magnitude of the pore pressure and the directions coincide with the *x*- and *z*-axis. Excess pore pressures that are tensile are indicated by an arrow rather than a line. Note that pressure is considered to be negative. In a work plane only one point per element is displayed, which represents the average of the excess pore pressure in that element in the plane.

### 5.5.12 GROUNDWATER HEAD

The *Groundwater head* is defined as:

$$h = y - \frac{p_w}{\gamma_w}$$

where $y$ is the vertical coordinate, $p_w$ is the active pore pressure and $\gamma_w$ is the unit weight of water. The plot of the *Groundwater head* may be selected from the *Stresses* sub-menu. The distribution of the groundwater head can only be presented as contours or shadings. The plot is particularly interesting when excess pore pressures occur.

## 5.6 STRUCTURES AND INTERFACES

By default, structures (i.e. beams, walls, floors) and interfaces are displayed in the geometry. Otherwise, these objects may be displayed by selecting the *Structures* option from the *Geometry* sub-menu. Output for these types of elements can be obtained by double clicking the desired object in a work plane. As a result, a new form is opened on which all structural elements of the selected type appear. At the same time the menu changes to provide the particular type of output for the selected object.

If it is desired to display output of several but not all structures of the same type in a single form, then the <Shift> and <Ctrl> keys on the keyboard can be used to modify the selection. If the <Ctrl> key is held down and an object is a single clicked, this will add the clicked object to the selection. Holding down the <Shift> key instead will add the entire chain to which the clicked object belongs to the selection. When all desired structures of a type have been selected in this manner, the user should double click anywhere in the geometry, while holding down the <Shift> or <Ctrl> key on the keyboard.

If multiple structural elements are selected, the selection remains active until all elements are deselected or the user double-clicks next to structural elements.

Please note that only structural elements of the same type can be selected at the same time. For example, if a beam is selected, only other beams can be added to the selection and no. floors or walls.

### 5.6.1 BEAMS

Output data for beams comprises deformations and forces. From the *Deformations* sub-menu the user may select the absolute (vectorial) displacements, $|u|$, the individual displacement components, $u_x$, $u_y$ and $u_z$ as well as the corresponding incremental displacements. From the *Forces* sub-menu the options *Axial force N*, *Shear force Q12*, *Shear force Q13*, *Bending moment M2* and *Bending moment M3* are available.

The numbers behind the particular forces refer to the beam's local system of axes (1,2,3) (Figure 5.6a and Figure 5.7a), which is indicated in any plot of the beam in a work plane. The first direction is the axial direction. For horizontal beams the second direction is vertical and always corresponds with the global $y$-axis and the third direction is horizontal and perpendicular to the beam axis. For vertical beams the second direction always corresponds with the global $x$-direction and the third direction corresponds with the global negative $z$-direction.

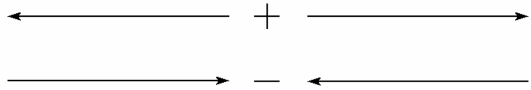
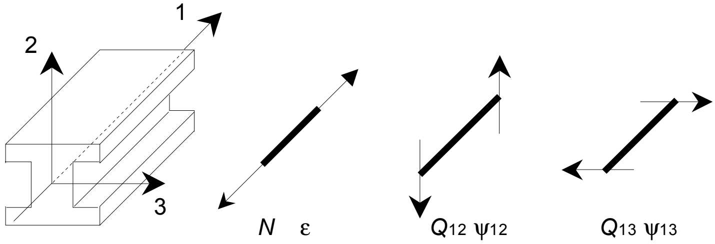


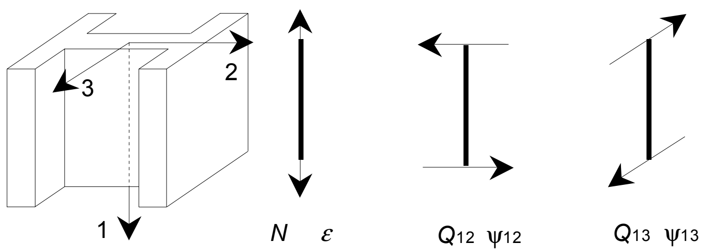

Figure 5.5  Sign convention for axial forces in beams, walls, and floors

The *Axial force N* is the force in the axial beam direction (Figure 5.6b and Figure 5.7b). An axial force is positive when it generates tensile stresses, as indicated in Figure 5.5.



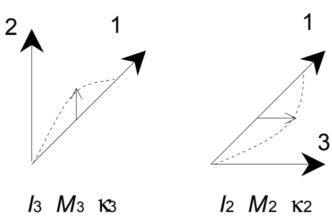

a. Local axes    b. Axial force *N*    c. Shear force *Q*12    d. Shear force *Q*13

Figure 5.6  Axial force and shear forces horizontal beams



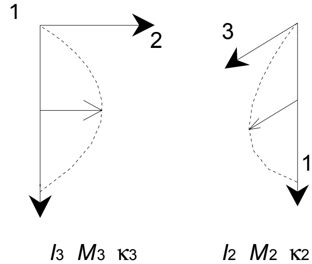

a. Local axes    b. Axial force *N*    c. Shear force *Q*12    d. Shear force *Q*13

Figure 5.7  Axial force and shear forces vertical beams




a. Bending moment *M*3    b. Bending moment *M*2

Figure 5.8  Bending moments horizontal beams




a. Bending moment *M*3 b. Bending moment *M*2

Figure 5.9 Bending moments vertical beams

The *Shear force Q12* is the shear force over the second beam axis (Figure 5.6c and Figure 5.7c), whereas the *Shear force Q13* is the shear force over the third beam axis (Figure 5.6d and Figure 5.7d).

The *Bending moment M3* is the bending moment due to bending around the third axis (Figure 5.8a and Figure 5.9a), whereas the *Bending moment M2* is the bending moment due to bending around the second axis (Figure 5.8b and Figure 5.9b).

### 5.6.2 WALLS

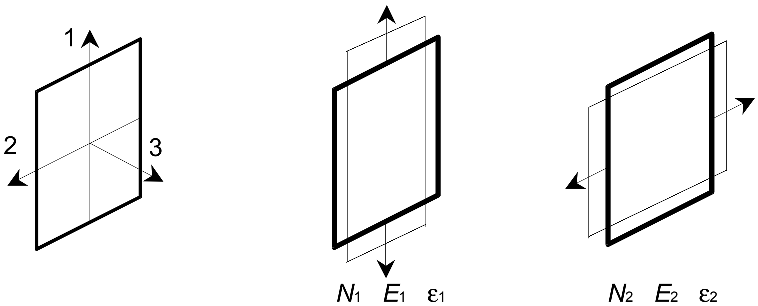
Output data for walls comprises deformations and forces. From the *Deformations* sub-menu the user may select the absolute (vectorial) displacements, $|u|$, the individual displacement components, $u_x$, $u_y$ and $u_z$ as well as the corresponding incremental displacements. From the *Forces* sub-menu the options *Axial force N1*, *Axial force N2*, *Shear force Q12*, *Shear force Q13*, *Shear force Q23*, *Bending moment M11*, *Bending moment M22* and *Torsion moment M12* are available. The numbers behind the particular forces refer to the wall's local system of axes (1,2,3) (Figure 5.7a), which is indicated in any plot of the wall in a work plane. The first direction is the vertical in-plane direction of the wall, the second direction is the horizontal in-plane direction of the wall and the third direction is perpendicular to the wall.

The *Axial force N1* is the axial force in the first direction (Figure 5.10b). The *Axial force N2* is the axial force in the second direction (Figure 5.10c).

The *Shear force Q12* is the in-plane shear force (Figure 5.11a). The *Shear force Q13* is the shear force perpendicular to the plate over the horizontal direction (Figure 5.11b), whereas the *The Shear force Q23* is the shear force perpendicular to the plate over the horizontal direction (Figure 5.11c).

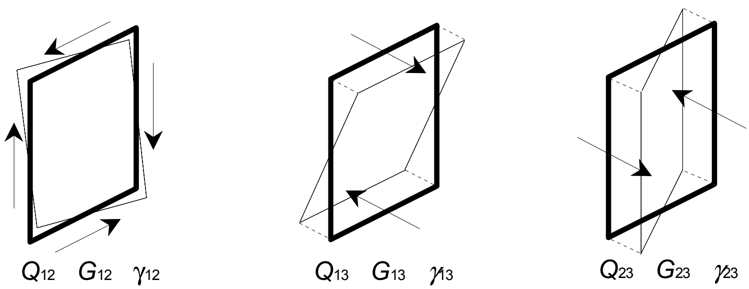
The *Bending moment M11* is the bending moment due to bending over the horizontal axis (around the horizontal axis), which is generally the direction where the major bending moments occur (Figure 5.12b). The *Bending moment M22* is the bending moment due to bending over the vertical axis (around the vertical axis) (Figure 5.12c).

The *Torsion moment M12* is the moment according to transverse shear forces (Figure 5.12a).



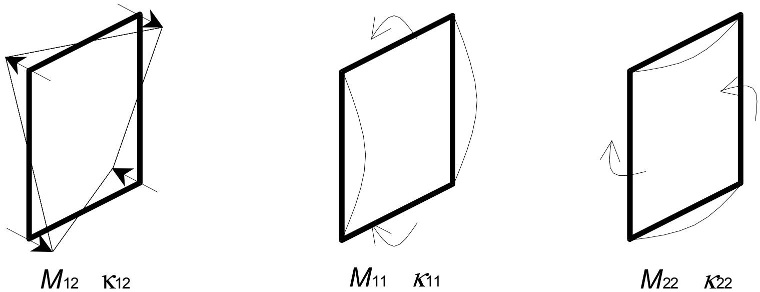

a. Local wall directions b. Axial force $N_1$ c. Axial force $N_2$

Figure 5.10




a. Shear force $Q_{12}$ b. Shear force $Q_{13}$ c. Shear force $Q_{23}$

Figure 5.11

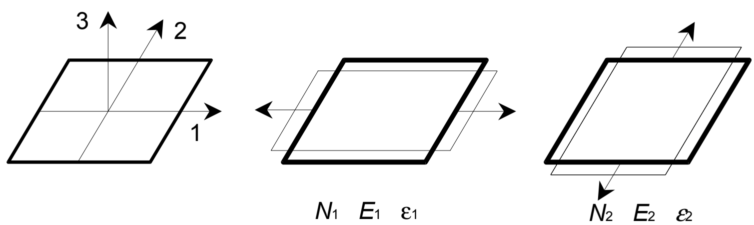



a. Torsion moment $M_{12}$ b. Bending moment $M_{11}$ c. Bending moment $M_{22}$

Figure 5.12

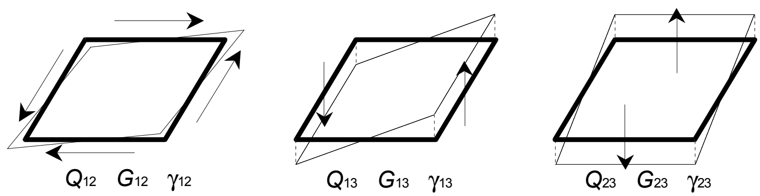
### 5.6.3 FLOORS

Output data for floors comprises deformations and forces. From the *Deformations* sub-menu the user may select the absolute (vectorial) displacements, |$u$|, the individual displacement components, $u_x$, $u_y$ and $u_z$ as well as the corresponding incremental displacements. From the *Forces* sub-menu the options *Axial force N1*, *Axial force N2*, *Shear force Q12*, *Shear force Q13*, *Shear force Q23*, *Bending moment M11*, *Bending moment M22* and *Torsion moment M12* are available. The numbers behind the particular forces refer to the floor's local system of axes (1,2,3) (Figure 5.13a). The first and second direction are in the floor plane (horizontal) whereas the third direction is perpendicular to the floor (vertical). In fact, the first direction always coincides with the global $x$-direction, the second direction corresponds with the global negative $z$-direction and the third direction is equal to the global $y$-direction.



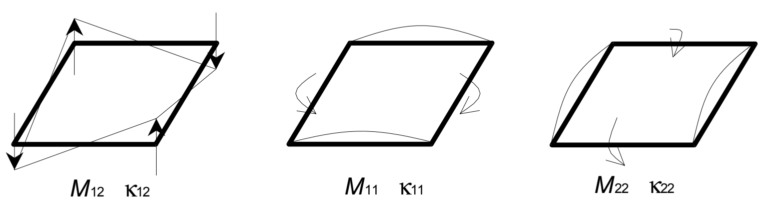

a. Local floor directions b. Axial force $N_1$ c. Axial force $N_2$

Figure 5.13 Axial forces




a. Shear force $Q_{12}$ b. Shear force $Q_{13}$ c. Shear force $Q_{23}$

Figure 5.14 Shear forces




a. Torsion moment $M_{12}$ b. Bending moment $M_{11}$ c. Bending moment $M_{22}$

Figure 5.15 Moments

The *Axial force N1* is the axial force in the first direction (Figure 5.13b). The *Axial force N2* is the axial force in the second direction (Figure 5.13c).

The *Shear force Q12* is the in-plane shear force (Figure 5.14a). The *Shear force Q13* is the shear force perpendicular to the plate over the first direction (Figure 5.14b), whereas the *Shear force Q23* is the shear force perpendicular to the plate over the second direction (Figure 5.14c).

The *Bending moment M11* is the bending moment due to bending over the first axis (around the second axis) (Figure 5.15b). The *Bending moment M22* is the bending moment due to bending over the second axis (around the first axis) (Figure 5.15c). The *Torsion moment M12* is the moment according to transverse shear forces (Figure 5.15a).

### 5.6.4 INTERFACES

Output for interfaces can be obtained by double clicking on the corresponding geometry lines in a work plane. The output for an interface comprises deformations and stresses. From the *Deformations* sub-menu the user may select *Total displacements*, *Phase displacements* and *Incremental displacements*. From the *Stresses* sub-menu the options *Effective normal stresses*, *Vertical* shear *stresses*, *Horizontal* shear *stresses,*, *Relative shear stresses*, *Active pore pressures* and *Excess pore pressures* are available. The *Effective normal stresses* are effective stresses perpendicular to the interface. The *Shear stresses* are the shear stresses in the first or second direction. Note that pressure is considered to be negative.

### 5.6.5 VOLUME PILES

Output for the volume elements which are part of volume piles is obtained along with the output for all volume elements on the main form of the output program. However, when double clicking the purple reference point that marks the top of a volume pile, a separate form is opened in which only output for the centre line of the volume pile is displayed. In this form the output of deformations of the centre line is available, and additionally the structural forces in the pile are calculated by integrating the stresses over the pile cross section. Apart from the regular *Deformations* sub-menu, a *Forces* sub-menu is available. From the *Forces* sub-menu the options *Axial force N*, *Shear force Q12*, *Shear force Q13*, *Bending moment M2* and *Bending moment M3* are available.

The numbers behind the particular forces refer to the pile's local system of axes (1,2,3) (Figure 5.7a), which is indicated in any plot of the pile in a work plane. The first direction is the axial $y$-direction, the second direction always corresponds with the global $x$-direction and the third direction corresponds with the global negative $z$-direction.

The *Axial force N* is the force in the axial beam direction (Figure 5.7b). An axial force is positive when it generates tensile stresses, as indicated in Figure 5.5.

The *Shear force Q12* is the horizontal shear force over the pile's $x$-axis (Figure 5.7c), whereas the *Shear force Q13* is the horizontal shear force over the pile's $z$-axis (Figure 5.7d).

The *Bending moment M3* is the bending moment due to bending around the $z$-axis (Figure 5.9a), whereas the *Bending moment M2* is the bending moment due to bending around the $x$-axis (Figure 5.9b).

## 5.7 VIEWING OUTPUT TABLES

For all types of plots the numerical data can be viewed in output tables by clicking on the *Table* button in the toolbar or by selecting the *Table* option from the *View* menu. As a result, a new form is opened in which the corresponding quantities are presented in tables. At the same time the menu changes to allow for the selection of other quantities that may be viewed in tables.

### *Tables of displacements*

When selecting the *Table* option when a displacement plot is displayed, a table form appears in which the coordinates and displacement components at all nodes are presented. The total displacements $u_x$ , $u_y$ and $u_z$ are the accumulated displacements from all previous calculation phases, whereas the incremental displacements $\Delta u_x$ , $\Delta u_y$ and $\Delta u_z$ are the incremental displacements in the current step and the phase displacements $Pu_x$, $Pu_y$ and $Pu_z$ are the accumulated displacements in the whole calculation phase.

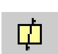
### *Tables of stresses and strains*

When viewing tables of stresses or strains in soil elements, the tables display the Cartesian components at the local nodes of all elements in addition to the nodal coordinates. Note that compression (pressure) is considered to be negative.

### *Stresses and forces in interfaces and structures*

When viewing tables of interface stresses, the table presents the effective normal stresses ($\sigma'_n$), the shear stresses ($\tau_y$, $\tau_{x\text{-}2}$ and $\tau_{rel}$), the active pore pressure ($p_{active}$) and the excess pore pressure ($p_{excess}$) at all interface stress points. When viewing tables of beam forces, the table presents the axial force *N*, the shear forces *Q12* and *Q13* and the bending moments *M3* and *M2* at the nodes. When viewing tables of plate forces (walls or floors), the table presents the axial forces (*N1* and *N2*), the shear forces (*Q12, Q13* and *Q23)*, the bending moments (*M11* and *M22)* and the torsion moment *M12* at the nodes.

## 5.8 VIEWING OUTPUT IN A CROSS-SECTION



To gain insight in the distribution of a certain quantity in the soil it is often useful to view the distribution of that quantity in a particular cross-section of the model. To this end the work planes as defined in the geometry model are always available to the user by means of tab sheets in the output form.

In addition to these predefined planes the user may define vertical cross-sections by clicking on the *Cross-section* button in the toolbar or by selecting the corresponding option from the *View* menu. Upon selection of this option, the 3D model is presented in such a way that the *y*-axis is pointing towards to the user so that the screen coincides with the *x*- and *z*-direction. The cross-section can now be specified precisely by clicking on one end of the cross-section line in the plot and moving the cursor to the other end while holding down the mouse button.

Cross-sections exactly in the *x*- or *z*-direction may be drawn by simultaneously holding down the <Shift> key on the keyboard. After releasing the mouse button, a new form is opened in which the distribution of the quantity is presented along the indicated cross-section. At the same time, the menu changes to allow for the selection of all other quantities that may be viewed along the indicated cross-section.

Multiple cross-sections may be drawn in the same geometry. Each cross-section will appear on a different output form. To identify different cross-sections, the end points of a cross-section are indicated with characters in alphabetical order.

The distribution of quantities in cross-sections is obtained from interpolation of nodal data.

### *Deformations*

In addition to the horizontal and vertical displacement and strain components, as available for the full geometry, the cross-section option allows for *Normal strains* and *Shear strains*. The *Normal strain* is defined as the strain perpendicular to the vertical cross-section plane, and the *Shear strain* is defined as the shear strain along the cross-section plane.

### *Stresses*

Different options are available to draw the effective and total stresses in the cross-section. Stresses may be viewed as *Principal stresses* projected on the cross-section plane, as *Normal stresses* or as *Shear stresses*. The *Normal stresses* are defined as the stress perpendicular to the vertical cross-section plane, and the *Shear stress* is defined as the shear stress along the cross-section plane.

Note that pressure is considered to be negative.

### *Tables*

For all types of plots that can be made for cross-sections, the numerical data can also be viewed in output tables by clicking on the *Table* button in the toolbar or by selecting the *Table* option from the *View* menu. As a result, a new form is opened in which the corresponding quantities are presented for only those elements that are intersected by the cross-section.

## 5.9 VIEWING OTHER DATA

The *View* menu provides facilities to enhance the graphical presentation of the 3D model. Moreover, this menu includes options to view general model data (*General info*), information about the current loadings (*Load info*) and material data (*Material info*). In addition, some general output data relating to the calculation process (*Calculation info*) is available from this sub-menu.

### 5.9.1 GENERAL PROJECT INFORMATION

The *General info* option of the *View* sub-menu contains some general information about the project.

### 5.9.2 LOAD INFORMATION

The *Load info* option of the *View* sub-menu provides information on the load that has been defined for the calculation phase to which the current step belongs. Distinction is made between distributed loads, line loads and point loads. For each type of loading a separate tab sheet is available in the load info window.

When an intermediate step of a calculation phase is opened or when the calculation phase did not finish successfully, the magnitude of the load, as presented in the tables, does not correspond to the load that has actually been appplied at the end of the step. In that case, the applied value of the load, $q$, can be determined in the following way:

$$q = q^{i-1} + \Sigma Mstage\ (q^{i} - q^{i-1})$$

where

$q^{i-1}$ = Total load applied in previous phase

$q^{i}$ = Total load defined for current phase

### 5.9.3 MATERIAL DATA

Material properties and model parameters can be viewed with the *Material info* option of the *View* sub-menu. Within this option a selection can be made from the following types of data sets: Soil and interfaces, beams, floors and walls and springs. Within the *Soil and interfaces* option the data sets are arranged in tab sheets according to the material models. The data may be send to the printer by clicking on the *Print* button.

### 5.9.4 CALCULATION PARAMETERS

If the option *Calculation info* is selected from the *View* menu, then a window appears presenting the *ΣMstage* parameter and various other calculation parameters corresponding to the end of the calculation step.

### 5.9.5 PARTIAL GEOMETRY

To enable the inspection of certain internal parts of the geometry (for example individual layers or volume clusters) it is possible to make other parts of the geometry invisible by using the *Partial Geometry* option of the *View* menu. As a result, a window is presented where a soil cluster, a group of soil clusters, or individual elements, can be set visible or invisible.

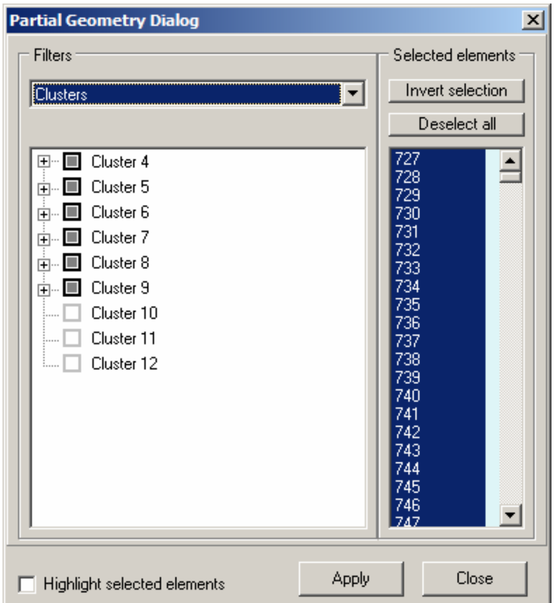



Figure 5.16 Selection window for partial geometry option

To enable a quick selection, clusters can be grouped in three ways: according to *Clusters* (x-z area grouping) or *Material sets* (material grouping). The type of grouping can be selected by choosing the appropriate option in the *Partial geometry* window.

Visible clusters are indicated by a filled black square, whereas invisible soil clusters are indicated by an open square. By clicking on a square, groups (and individual clusters) can be toggled from being visible to being invisible and vice versa. By clicking on the +sign in front of the group, the subclusters in the group can be selected individually. Clusters that have been set inactive in the framework of staged construction are always invisible and cannot be made visible.

It may be necessary to view the cluster numbers. This can be done by activating the *Cluster numbers* option in the *Geometry* menu.

Apart from selecting clusters, it is also possible to select individual elements in order to set them visible or invisible. To this end a list of element numbers is presented on the right side of the dialog. Clicking on a element number will change the element from visible to invisible and vice versa.

Two buttons allow the user to adapt the selection of visible elements. The *Invert selection* button will toggle all visible elements invisible and all invisible elements visible. The *Deselect all* button will set all elements to invisible.

The *Highlight selected elements* option allows you to preview which elements are included in the current selection, in order to check the selection before it is actually applied. After pressing the <Apply> button, the 3D model is presented according to the visible/invisible setting as specified in the partial geometry option. On pressing the <Close> button the *Partial geometry* window is closed without further changes.

### 5.9.6 CONNECTIVITY PLOT

A *Connectivity plot* is a plot of the mesh in which the element connections are clearly visualised. This plot is particularly of interest when interface elements are included in the mesh. Interface elements are composed of pairs of nodes in which the nodes in a pair have the same coordinates. In the *Connectivity plot*, however, the nodes in a pair are drawn with a certain distance in between so that it is made clear how the nodes are connected to adjacent elements. The connectivity plot is only presented in the work planes and not in the full 3D model.

In the *Connectivity plot* it can, for example, be seen that when an interface is present between two soil elements, that the two soil elements do not have common nodes and that the connection is formed by the interface. In a situation where interfaces are placed along both sides of a plate, the plate and the adjacent soil elements do not have nodes in common. The connection between the plate and the soil is formed by the interface. This can also be viewed in the *Connectivity plot*.

### 5.9.7 OVERVIEW OF PLOT VIEWING FACILITIES

To enhance the interpretation of output results, the PLAXIS 3D FOUNDATION program has several options to view the 3D model. An overview of these options is given below:

#### *Perspective view*

By default, a 3D model in the Output program is presented in perspective view. This facility makes the appearance of the 3D model realistic and natural, although it is presented on a flat screen. Note that lines that are parallel by definition may be plotted non-parallel. The perspective view cannot be deactivated.

#### *Shadowing*

To make the appearance of the 3D model even more realistic, the *Shadow* option in the *View* sub-menu may be used. When this option is active, then object surfaces that have the same colour by definition (such as soil elements with the same material data set) appear 'brighter' or 'darker', depending on their orientation with respect to the viewer. Object surfaces appear most bright when the normal to the surface points in the direction of the viewer. The surfaces become darker the more the normal deviates from this direction.

### *Changing the orientation of a 3D model*

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 more or less to the right, the positive *y*-direction is upwards and the positive *z*-direction points more or less to 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.

Alternatively, the orientation of the 3D model can be changed using the *Viewpoint* option in the *View* sub-menu. This option allows the direct input of the rotation angles around the view axes, or the selection of several predefined view points (Figure 5.17)

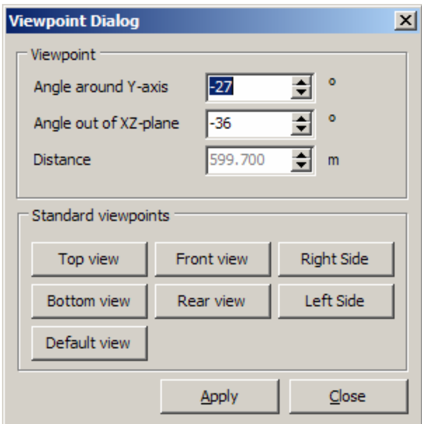



Figure 5.17 Viewpoint dialog

### *Zooming*

To enlarge a part of the model for viewing a particular detail, the zoom option of the *View* sub-menu may be used. After selection of the zoom option, the zoom area (a rectangular area on the screen) must be selected with the mouse. The zoom option may be used repetitively.

### *Viewing structural objects*

Output of structural objects can be viewed in more detail by double clicking the desired structural object in a work plane.

### *Viewing cross-section*

Apart from the predefined work planes, a user may define vertical cross-sections to view output. This can be done by selecting the *Cross-section* option of the *View* sub-menu.

### *Changing the intensity of material data set colours*

Apart from the *Shadow* option, material data set colours can appear in three different intensities. To globally increase the intensity of all data set colours, the user may press <Ctrl><Alt><C> simultaneously on the keyboard. There are three levels of colour intensity that can be selected in this way.

## 5.10 EXPORTING DATA

Data as displayed in output forms may be exported to other programs using the Windows clipboard function. This function can be activated by clicking on the *Copy to clipboard* button in the toolbar or by selecting the *Copy* option from the *Edit* menu. Plots are exported such that they appear, for example, as figures in a drawing package or in a word processor when pasting the clipboard data. Data in tables are exported such that they appear in different cells in a spreadsheet program when pasting the clipboard data.

In addition to the clipboard function, hardcopies of graphs and tables can be produced by sending the output to an external printer. When clicking on the *Print* button or selecting the corresponding option from the *File* menu, the print window appears in which selections can be made of the various plot components that are to be included in the hardcopy. In addition, basic information is presented in a frame around the plot. For this purpose, a project title and a project description may be entered, which are presented on the hardcopy. When pressing the *Set-up* button, the standard printer set-up window is presented in which specific printer settings can be changed.

When clicking on the *Print* button, the plot is send to the printer. This process is fully carried out by the Windows® operating system. For more information on the installation of printers or other output devices reference is made to the corresponding manuals.

When the *Copy to clipboard* option or the *Print* option is used on a plot that shows a zoomed part of the model, only the part that is currently visible will be exported to the clipboard or the printer.

# 6 LOAD-DISPLACEMENT CURVES

The Curves program can be used to draw load- or time-displacement curves, stress-strain diagrams and stress or strain paths of pre-selected points in the geometry. These curves visualise the development of certain quantities during the various calculation phases, and this gives an insight into the global and local behaviour of the soil. The points at which curves may be generated must be selected using the *Select points for curves* option in the *Calculation* mode of the Input program before starting the calculation process (Section 4.5). Distinction is made between nodes and stress points. In general, nodes are used for the generation of load-displacement curves, whereas stress points are used for stress-strain diagrams and stress paths. A maximum of 10 nodes and 10 stress points may be selected. During the calculation process, information related to these points is stored in curves data files. The information in these files is then used for the generation of the curves. It is not possible to generate curves for points that have not been pre-selected, since the required information is not available in the curves data files.

## 6.1 THE CURVES PROGRAM

This icon represents the Curves program. The Curves program contains all facilities to generate load-displacement curves, stress paths and stress-strain diagrams. At the start of the Curves program, a choice must be made between the selection of an existing chart and the creation of a new chart. When selecting *New chart*, the *Curve generation* window appears in which parameters for the generation of a curve can be set (Section 6.3). When selecting *Existing chart*, the selection window allows for a quick selection of one of the four most recent charts. If an existing chart is to be selected that does not appear in the list, the option <<<*More files*>>> can be used. As a result, the general file requester appears which enables the user to browse through all available directories and to select the desired PLAXIS chart file (*.G## where ## is any number between 00 and 99). After the selection of an existing project, the corresponding chart is presented in the main window. The main window of the Curves program contains the following items (Figure 6.1):

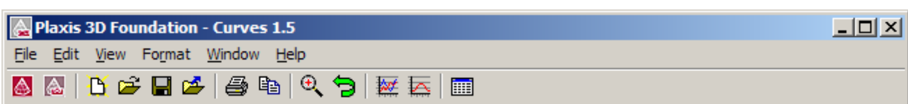



Figure 6.1 Toolbar in main window of the Curves program (without display area)

### *Curves menu*

The *Curves* menu contains all options and operation facilities of the Curves program. Some options are also available as buttons in the toolbar.

### *Toolbar*

This bar contains buttons that may be used as a shortcut to menu facilities.

### *Chart windows*

These are windows on which charts are displayed. Multiple chart forms may be opened simultaneously and each chart can contain a maximum of eight curves.

## 6.2 THE CURVES MENU

The *Curves* menu consists of the following sub-menus:

### *The File sub-menu*

| | |
|---|---|
| *New* | To create a new chart. The file requester is presented. |
| *Open* | To open a chart. The file requester is presented. |
| *Save* | To save the current chart under the existing name. If a name has not been given before, the file requester is presented. |
| *Add curve* | To add a new curve to the current chart (Section 6.4). |
| *Delete chart* | To delete a chart from the current project. |
| *Close* | To close the active chart window. |
| *Close all* | To close all chart windows. |
| *Print* | To print the active chart on a selected printer. The print window is presented. |
| *Work directory* | To set the directory where curve files will be stored. |
| (*recent charts*) | To quickly open one of the four most recently edited charts. |
| *Exit* | To leave the program. |

### *The Edit sub-menu*

| | |
|---|---|
| *Copy* | To copy the current chart to the Windows® clipboard. |

### *The View sub-menu*

| | |
|---|---|
| *Zoom in* | To zoom into a rectangular area for a more detailed view. The zoom area must be selected using the mouse. Press the left mouse button at a corner of the zoom area; hold the mouse button down and move the mouse to the opposite corner of the zoom area; then release the button. The program will modify the range on the axes according to the selected area. The zoom option may be used repetitively. |
| *Reset view* | To restore the full chart. |
| *Table* | To view the table with the values of all the curve points. |

| | |
|---|---|
| *Legend* | To view the legend of the current chart. The symbols and colours of the lines in the legend correspond to the symbols and colours of the curves. |
| *Value indication* | To view detailed curve data when the cursor (mouse pointer) is positioned on a curve. |

### *The Format sub-menu*

| | |
|---|---|
| *Curve ...* | To change the presentation or regenerate the curves in the current chart window (Section 6.6.1). |
| *Chart ...* | To change the presentation of the frame (axes and grid) in the current chart window (Section 6.6.2). |

## 6.3 CURVE GENERATION

A new curve can be generated by starting up the Curves program or by selecting the *New* option in the *File* menu. As a result, the file requester appears and the project for which the curve has to be generated for must be selected. After selection of the project, the *Curve generation* window appears, as presented in Figure 6.2.

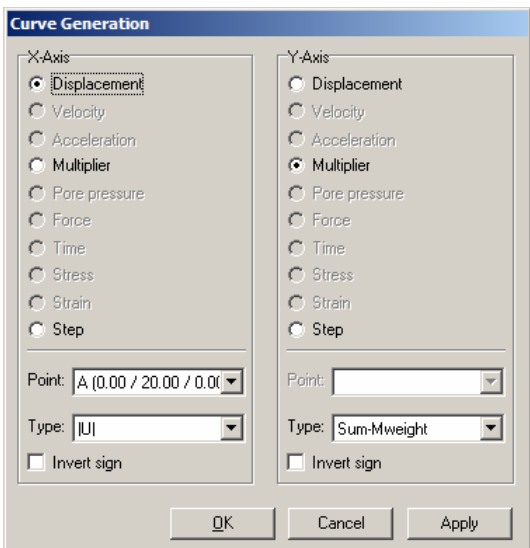



Figure 6.2 Curve generation window

Two similar groups with various items are shown, one for the *x*-axis and one for the *y*-axis. In general, the *x*-axis corresponds to the horizontal axis and the *y*-axis corresponds to the vertical axis. However, this convention may be changed using the *Exchange axes* facility in the *Chart settings* window (Section 6.6.2). For each axis, a combination of selections should be made to define which quantity is plotted on that axis. The *Invert sign* option may be selected to multiply all values of the *x*-quantity or the *y*-quantity by -1.

The combination of the step-dependent values of the *x*-quantity and the *y*-quantity form the points of the curve to be plotted. The curve point numbers correspond to the calculation step numbers plus one. The first curve point (corresponding to step 0) is numbered 1. When both quantities have been defined and the *OK* button is pressed, the curve is generated and presented in a chart window.

### *Load-displacement curves*

Load-displacement curves can be used to visualise the relationship between the applied loading and the resulting displacement of a certain point in the geometry. In general, the *x*-axis relates to the displacement of a particular node (*Displacement*), and the *y*-axis contains data relating to load level. The latter is related with the value of *ΣMstage* in the following way: Applied load = Total load applied in previous phase + *ΣMstage* times (Total load applied in current phase – Total load applied in previous phase). Also other types of curves can be generated.

The selection of *Displacement* must be completed with the selection of a pre-selected node in the *Point* combo box and the selection of a displacement component in the *Type* combo box. The type of displacement can be either the length of the displacement vector ($|u|$) or one of the individual displacement components ($u_x$, $u_y$ or $u_z$). The displacements are expressed in the unit of length, as specified in the *General settings* window of the Input program.

Instead of selecting a displacement component for the horizontal axis, it is also possible to select the calculation step numer (*Step*). When interpreting the curve it should be noted that during the calculation the step sizes might change due to the automatic load stepping procedures.

The selection of *Multiplier* must be completed with the selection of the desired load system, represented by the corresponding multiplier in the *Type* combo box. As the activation of a load system is not related to a particular point in the geometry, the selection of a *Point* is not relevant in this case. Note that the 'load' is not expressed in units of stress or force. To obtain the actual load, the presented value should be multiplied by the input load as specified by means of staged construction.

Another quantity that can be presented in a curve is the *Pore pressure*. The selection of *Pore pressure* must be completed with the selection of a pre-selected node in the *Point* combo box. The *Type* combo box is not relevant in this case. Pore pressures are expressed in the unit of stress.

### *Time-displacement curves*

Time-displacement curves can be useful to interpret the results of calculations in which the time-dependent behaviour of the soil plays an important role (e.g. consolidation and creep). In this case the *Time* option is generally selected for the $x$-axis, and the $y$-axis contains data for the displacements of a particular node. The selection of *Time* does not require additional selections in the *Point* and *Type* combo boxes. *Time* is expressed in the unit of time, as specified in the *General settings* window of the input program.

### *Stress-strain diagrams*

Stress-strain diagrams can be used to visualise the local stress-strain behaviour of the soil. In fact, stress-strain diagrams represent the idealised behaviour of the soil according to the selected soil model. The selections *Stress* or *Strain* must be completed with the selection of a pre-selected stress point from the *Point* combo box and the selection of a certain component in the *Type* combo box. The following stress and strain components are available:

### *Stresses:*

| | |
|---|---|
| $\sigma'_{xx}$ | effective stress in $x$-direction |
| $\sigma'_{yy}$ | effective vertical stress ($y$-direction) |
| $\sigma'_{zz}$ | effective stress in $z$-direction |
| $\sigma_{xy}$ | shear stress in the $x$-$y$-plane |
| $\sigma_{yz}$ | shear stress in the $y$-$z$-plane |
| $\sigma_{zx}$ | shear stress in the $x$-$z$-plane |
| $\sigma'_1$ | in absolute sense the largest effective principal stress |
| $\sigma'_2$ | the intermediate effective principal stress |
| $\sigma'_3$ | in absolute sense the smallest effective principal stress |
| $p'$ | isotropic effective stress (mean effective stress) |
| $q$ | deviatoric stress (equivalent shear stress) |
| $p_{excess}$ | excess pore pressure |

### *Strains:*

| | |
|---|---|
| $\varepsilon_{xx}$ | strain in $x$-direction |
| $\varepsilon_{yy}$ | vertical strain ($y$-direction) |
| $\varepsilon_{zz}$ | strain in $z$-direction |
| $\gamma_{xy}$ | shear strain in $x$-$y$-plane |
| $\gamma_{yz}$ | shear strain in $y$-$z$-plane |
| $\gamma_{zx}$ | shear strain in $x$-$z$-plane |
| $\varepsilon_1$ | in absolute sense the largest principal strain |

| | |
|---|---|
| $\varepsilon_2$ | intermediate principal strain |
| $\varepsilon_3$ | in ablolute sense the smallest principal strain |
| $\varepsilon_v$ | volumetric strain |
| $\varepsilon_q$ | deviatoric strain (equivalent shear strain) |

See the *Scientific Manual* for a definition of the stress and strain components. The phrase 'in absolute sense' in the description of the principal components is added because, in general, the normal stress and strain components are negative (compression is negative). Note that the deviatoric stress and strain components are always positive. Stress components are expressed in the units of stress; strains are dimensionless.

### *Stress paths and strain paths*

A stress path represents the development of the stress state at a local point of the geometry. Similarly, a strain path represents the development of strain. These types of curves are useful to analyse the local behaviour of the soil. Since soil behaviour is stress-dependent and soil models do not take all aspects of stress-dependency into account, stress paths are useful to validate previously selected model parameters. For the generation of stress paths and strain paths, a selection can be made from the available stress and strain components as listed above.

## 6.4 MULTIPLE CURVES IN ONE CHART

It is often useful to compare the development of displacements or stresses at different points in a geometry, or even in different geometries or projects. Therefore PLAXIS allows for the generation of a maximum of eight curves on the same chart. Once a single curve has been generated, the *Add curve* option can be used to generate a new curve in the current chart.

This option can be selected by clicking on the *Add curve* button in the toolbar or by selecting the corresponding option from the *File* sub-menu. In addition, a selection must be made to specify whether the curve is based on the *Current project* or on *Another project*. In the latter case, the project can be selected using the file requester.

The *Add curve* procedure is similar to the *New* option (Section 6.3). However, when it comes to the actual generation of the curve, the program imposes some restrictions on the selection of data to be presented on the *x*- and the *y*-axis. This is to ensure that the new data are consistent with the data of any existing curve or curves.

## 6.5 REGENERATION OF CURVES

If, for any reason, a calculation process is repeated or extended with new calculation phases, it is generally desirable to update existing curves to comply with the new data. This can be done by means of the *Regenerate* facility. This facility is available in the

*Curve settings* window (Section 6.6.1), which can be selected by clicking on the *Change curve settings* button in the toolbar or by selecting the *Curves* option in the *Format* menu. When clicking on the *Regenerate* button, the *Curve generation* window appears, showing the existing setting for *x*- and *y*-axis. Pressing the *OK* button is sufficient to regenerate the curve to include the new data. Another <OK> closes the *Curve settings* window and displays the newly generated curve.

When multiple curves are used in one chart, the *Regenerate* facility should be used for each curve individually. The *Regenerate* facility may also be used to change the quantity that is plotted on the *x*- or *y*-axis.

## 6.6 FORMATTING OPTIONS

The layout and presentation of curves and charts may be customised by selecting the options in the *Format* menu. Distinction is made between the *Curve* settings and the *Chart* settings. The *Curves* option is used to modify the presentation of curves, and the *Chart* option is used to set the frame and axes in which the curves appear.

### 6.6.1 CURVE SETTINGS

The *Curve* settings can be selected from the *Format* menu. Alternatively, the *Curve settings* button in the toolbar may be clicked. As a result, the *Curve settings* window appears, as presented in Figure 6.3.

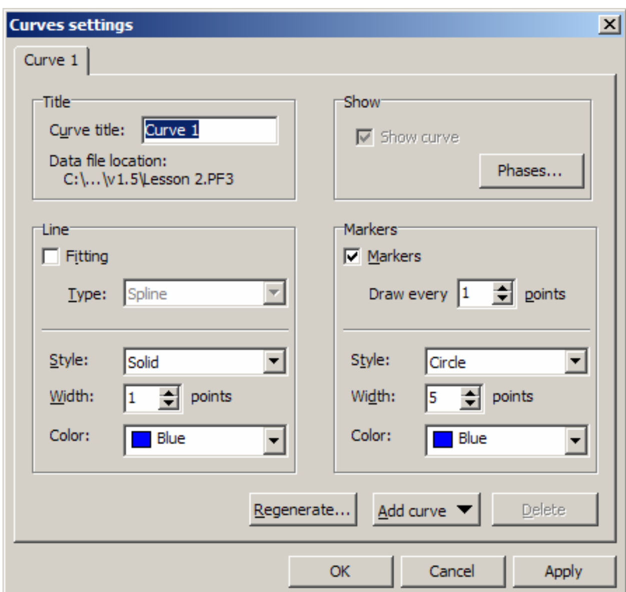



Figure 6.3 Curve settings window

The *Curve settings* window contains for each of the curves in the current chart a tab sheet with the same options. If the correct settings are made, the *OK* button may be pressed to activate the settings and to close the window.

Alternatively, the *Apply* button may be pressed to activate the settings, but the window is not closed in this case. When pressing the *Cancel* button the changes to the settings are ignored.

***Title***

A default title is given to any curve during its generation. This title may be changed in the *Curve title* edit box. When a legend is presented for the active chart in the main window, the *Curve title* appears in the legend.

***Show curve***

When multiple curves are present within one chart, it may be useful to hide temporarily one or more curves to focus attention on the others. The *Show curve* option may be deselected for this purpose.

***Phases***

The *Phases* button may be used to select for which calculation phases the curve has to be generated. This option is useful when not all calculation phases should be included in the curve.

***Line and marker presentation***

Various options are available to customise the appearance of the curve lines and markers.

***Fitting***

To draw a smooth curve, the user can select the *Fitting* item. When doing so, the type of fitting can be selected from the *Type* combo box. The *Spline* fitting generally gives the most satisfactory results, but, as an alternative, a curve can be fitted to a polynomial using the least squares method.

***Regenerate***

The *Regenerate* button may be used to regenerate a previously generated curve to comply with new data (Section 6.5).

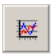
***Add curve***

The *Add curve* button may be used to add new curves to the current chart (Section 6.4).

### *Delete*

When multiple curves are present within one chart, the *Delete* button may be used to erase a curve.

## 6.6.2 CHART SETTINGS



The *Chart* settings relate to the presentation of the frame and axes in the chart. These settings can be selected from the *Format* menu. Alternatively, the *Chart settings* button in the toolbar may be clicked. As a result, the *Chart settings* window appears, as shown in Figure 6.4.

If the correct settings are made, the *OK* button may be pressed to activate the settings and to close the window.

Alternatively, the *Apply* button may be pressed to activate the settings, but the window is not closed in this case. When pressing the *Cancel* button, the changes to the settings are ignored.

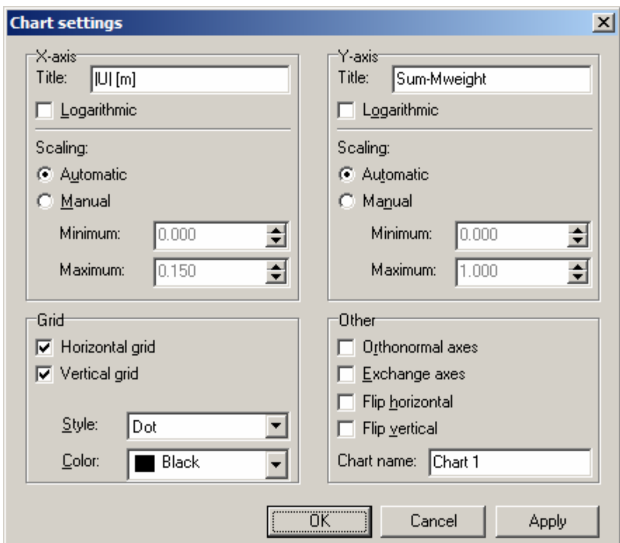



Figure 6.4 *Chart settings* window

### *Titles*

By default, a title is given to the *x*-axis and the *y*-axis, based on the quantity that is selected for the curve generation. However, this title may be changed in the *Title* edit boxes of the corresponding axis group. In addition, a title may be given to the full chart, which can be entered in the *Graph title* edit box. This title should not be confused with the *Curve title* as described in Section 6.6.1.

### *Scaling of x- and y-axis*

By default, the range of values indicated on the *x*- and *y*-axis is scaled automatically, but the user can select the *Manual* option and enter the desired range in the *Minimum* and *Maximum* edit boxes. As a result, data outside this range will not appear in the plot. In addition, it is possible to plot the *x*- and/or *y*-axis on a logarithmic scale using the *Logarithmic* check box. The use of a logarithmic scale is only valid if the full range of values along an axis is strictly positive.

### *Grid*

Grid lines can be added to the plot by selecting items *Horizontal grid* or *Vertical grid*. The grid lines may be customised by means of the *Style* and *Colour* options.

### *Orthonormal axes*

The option *Orthonormal axes* can be used to ensure that the scale used for the *x*-axis and the *y*-axis is the same. This option is particularly useful when values of similar quantities are plotted on the *x*-axis and *y*-axis, for example when making diagrams of different displacement components.

### *Exchange axes*

The option *Exchange axes* can be used to interchange the *x*-axis and the *y*-axis and their corresponding quantities. As a result of this setting, the *x*-axis will become the vertical axis and the *y*-axis will become the horizontal axis.

### *Flip horizontal or vertical*

Selecting the option *Flip horizontal* or *Flip vertical* will respectively reverse the horizontal or the vertical axis.

## 6.7 VIEWING A LEGEND

By default, a legend is presented at the right hand side of each curves window. The legend gives a short description of the data presented in the corresponding curve. The description appearing in the legend is actually the *Curve title*, which is automatically generated based on the selection of quantities for the *x*- and *y*-axis. The *Curve title* can be changed in the *Curve settings* window (Section 6.6.1). The legend can be activated or deactivated in the *View* menu. The size of the legend can be changed with the mouse., by dragging the double line between the chart and the legend.

## 6.8 VIEWING A TABLE

To view the numerical data presented in the curves, a table may be opened. The *Table* option can be selected by clicking on the *Table* button in the toolbar or by selecting the corresponding option in the *View* menu. As a result, a table appears showing the numerical values of all points on the curve or curves in the current chart. The desired curve to be displayed can be selected in the *Curve* combo box above the table. There are options available in the *Table* window for printing or copying all the data, or a selected part of it, to the Windows clipboard.

### *Editing curve data*

In contrast to the Output program, the Curves program allows for editing of the table by the user. The curve points can be modified by clicking in the table of *x*- and *y*-coordinates and typing a new value for the currently selected coordinate. These changes are made permanent once *OK* is selected or can be undone selecting *Cancel*.

# Index

# Appendix A - Program and Data File Structure

## A.1 Program structure

The full PLAXIS 3D FOUNDATION program consists of various sub-programs, modules and other files which are copied to various directories during the installation procedure (Section *Installation* in the *General information* part). The most important files are located in the PLAXIS 3D FOUNDATION program directory. Some of these files and their functions are listed below:

| | |
|---|---|
| FNS.EXE | Input program (pre-processor) (Chapter 3 and 4) |
| FNSOUT.EXE | Output program (post-processor) (Chapter 5) |
| CURVES3D.EXE | Curves program (Chapter 6) |
| PLXMESHW.EXE | 2D mesh generator |
| PLX3DMSH.EXE | 3D mesh generator |
| TMESH.DLL | Mesh topology |
| PLX3DFK0.EXE | Initial conditions generation program |
| PLASW3DF.EXE | Deformation analysis program |
| PLXSSCR3DF.DLL | Module presenting the PLAXIS 3D FOUNDATION logo's |
| PLXCLC3DF.DLL | Module presenting the screen output during a deformation analysis (Section 4.8) |
| PLXREQ.DLL | PLAXIS file requester (Section 2.2) |

The material data sets in the global database (Section 3.4.6) are, by default, stored in the DB sub-directory of the program directory. The sub-directory EMPTYDB contains an empty material database structure which may be used to 'repair' a project of which, for any reason, the material database structure was damaged. This can be done by copying the appropriate files to the project directory (Section A.2). The precise material data have to be re-entered in the Input program.

## A.2 Project data files

The main file used to store information for a PLAXIS 3D FOUNDATION project has a structured format and is named <*project*>.PF3, where <*project*> is the project title. Besides this file, additional data is stored in multiple files in the sub-directory <*project*>.DF3. The files in this directory may include:

| | |
|---|---|
| CALC.INF | (contains criterion how calculation was finished; Section 4.8) |
| PLAXMESH.ERR | (error message file) |
| PLAXIS.* | (.MSI; .MSO) |
| ANCHORS.MDB | (PLAXIS material database for anchors) |
| BEAMS.MDB | (PLAXIS material database for plates) |
| GEOTEX.MDB | (PLAXIS material database for geogrids) |
| SOILDATA.MDB | (PLAXIS material database for soil and interfaces) |
| FNSBEAMS.MDB | (PLAXIS 3D FOUNDATION material database for beams) |
| FNSPLATES.MDB | (PLAXIS 3D FOUNDATION material database for walls and floors) |
| FNSSPRINGS.MDB | (PLAXIS 3D FOUNDATION material database for springs) |
| <*project*>.* | (.CXX; .H00; .HIS; .HXX; .INP; .L## [1]; .MSH; .SF4; .SIS; .SXX; .W00; .W## [1]; .ZIN; .ZMS; .000; .### [2]) |

[1] = Two digit calculation phase number (01, 02, …). Above 99 gives an additional digit in the file extension.

[2] = Three digit calculation step number (001, 002, …). Above 999 gives an additional digit in the file extension.

When it is desired to copy a PLAXIS 3D FOUNDATION project under a different name or to a different directory, it is recommended to open the project that is to be copied in the Input program and to save it under a different name using the *Save as* option in the *File* menu. In this way the required file and data structure is properly created. However, calculation steps (<*project*>.### where ### is a calculation step number) are not copied in this way. If it is desired to copy the calculation steps or to copy a full project manually, then the user must take the above file and data structure exactly into account, otherwise PLAXIS will not be able to read the data and may produce an error.

During the creation of a project, before the project is explicitly saved under a specific name, intermediately generated information is stored in the TEMP directory as specified in the Windows® operating system using the project name XXSNFXX. The TEMP directory also contains some backup files ($FNS$.# where # is a number) as used for the repetitive undo option (Section 3.2). The structure of the $FNS$.# files is the same as the project file (.PF3). Hence, these files may also be used to 'repair' a project of which, for any reason, the project file was damaged. This can be done by copying the most recent backup file to <*project*>.PF3 in the work directory.