PLAXIS 3D TUNNEL
Reference Manual

version 2

TABLE OF CONTENTS

1 INTRODUCTION

The PLAXIS 3D Tunnel program is a special purpose three-dimensional finite element computer program used to perform deformation and stability analyses for various types of tunnels 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 repetitive geometrical cross-section. The program has special features for NATM and shield tunnels, 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 Tunnel 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 Tunnel program for a wide variety of problem types.

The user interface consists of four sub-programs (Input, Calculations, Output and Curves). The contents of this Reference Manual are arranged according to these four 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 Chapter 7, the Scientific Manual and the Material Models Manual.

2 GENERAL INFORMATION

Before describing the specific features in the four parts of the PLAXIS 3D Tunnel 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. 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 Tunnel manuals, the default units are used.

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

Basic units:	Length	[m]	[in.]
	Force	[kN]	[lb]
	Time	[day]	[sec]
Geometry:	Coordinates	[m]	[in.]
	Displacements	[m]	[in.]
Material properties:	Young's modulus	[kPa] = [kN/m^2]	[psi] = [lb/in^2]
	Cohesion	[kPa]	[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]

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

For the creation of a three-dimensional (3D) finite element model in the PLAXIS 3D Tunnel program, a vertical cross-section model has to be created first. This vertical cross-section model is a two-dimensional (2D) model created in the *x*-y plane of the global coordinate system (Figure 2.1). In order to obtain a 3D model, the 2D model is extended into the third dimension (*z*-direction). During the creation of a vertical cross-section model the positive *z*-direction is pointing towards the user.

Stresses computed in the PLAXIS 3D Tunnel 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.

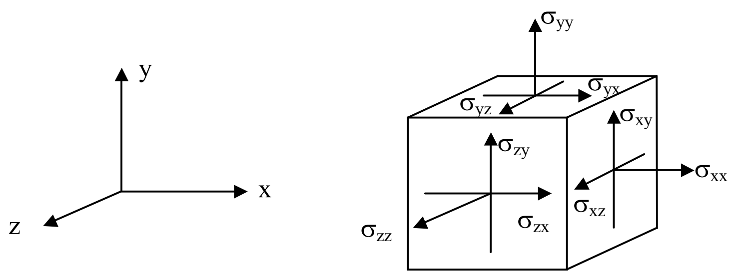


Figure 2.1 Coordinate system and indication of positive stress components.

2.2 FILE HANDLING

The PLAXIS 3D Tunnel 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 for a PLAXIS 3D Tunnel project has a structured format and is named <*project*>.PL3, where <*project*> is the project title. Besides this file, additional data is stored in multiple files in the sub-directory <*project*>.DT3. 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 Tunnel project file (*.PL3) 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.

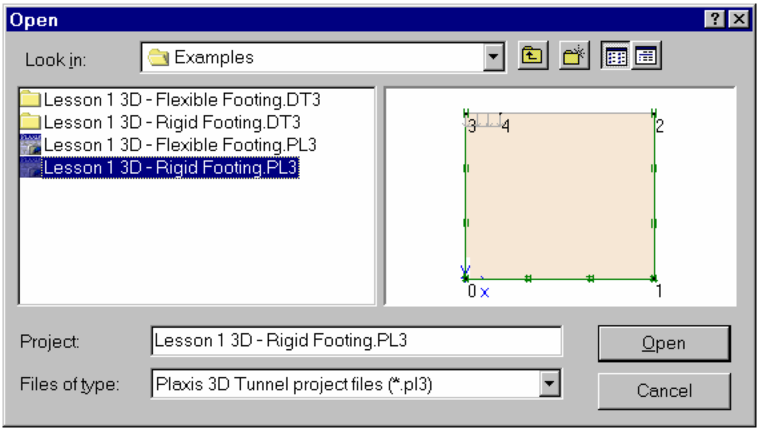


Figure 2.2 PLAXIS file requester

2.3 INPUT PROCEDURES

In PLAXIS, 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 soil layer)
Input of text	(e.g. entering a project name)
Input of values	(e.g. entering the soil 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, the user interface is equipped with on-line help facilities. The general help facility can be activated by selecting the options from the *Help* menu. Pressing the <Help> button in a window or pressing the <F1> key on the keyboard activates context-sensitive help. On pressing the

<Help> button, general information about a particular window or feature is provided, whereas pressing the <F1> key provides specific information about a particular parameter.

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)

To carry out a three-dimensional finite element analysis using the PLAXIS 3D Tunnel program, the user has to create a three-dimensional (3D) model and specify the material properties and boundary conditions. The model is created in the Input program. To set up a 3D model, the user must first create a vertical cross-section model in the *x-y*-plane. The vertical cross-section model is composed of points, lines and other components. From this cross-section model a two-dimensional (2D) finite element mesh is generated first. Subsequently, a 3D model is created by specifying all relevant *z*-coordinates to which the vertical cross-section model and the 2D mesh are to be copied. The resulting 3D model consists of equal parallel planes (*z-planes*) and *slices*. A *slice* is defined as a volume between two successive *z*-planes. From the 3D model with the 2D meshes in the *z*-planes a fully 3D mesh is generated. When generating a 3D mesh in this way, the mesh does not allow for any geometry variation in *z*-direction. However, when defining calculation phases, loads and geometry objects may be activated or deactivated in individual *z*-planes or slices (Section 4.6 Staged construction). In this way, truly three-dimensional models can be created.

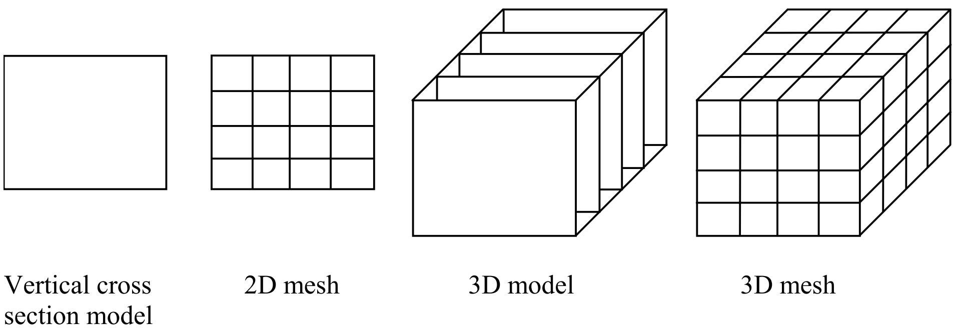


Figure 3.1 Creating a 3D model and finite element mesh

When a vertical cross-section model is created in the Input program it is suggested that the different input items are selected in the order given by the second toolbar (from left to right). In principle, first draw the cross-section contour, then add the soil layers, then structural objects, then construction layers, then boundary conditions and then loadings. The general cross-section should include all objects appearing somewhere in any *x*-y cross-section of the full 3D model. The second toolbar acts as a guide through the Input program and ensures that all necessary input items are dealt with. Of course, not all input options are generally required for any particular analysis. For example, some structural objects or loading types might not be used when only soil loading is considered, or the generation of water pressures may be omitted if the problem is completely dry, or the initial stress generation may be omitted if the initial stress field is calculated by means of gravity loading. Nevertheless, by following the toolbar the user is reminded of the various input items and will select the ones that are of interest. PLAXIS will also give warning messages if some necessary input has not been specified.

When changing an existing model, it is important to realise that the finite element mesh and, if applicable, the initial conditions must be regenerated to make them in agreement with the updated model. This is also checked by PLAXIS. On following these procedures the user can be confident that a consistent finite element model is obtained.

3.1 THE INPUT PROGRAM

This icon represents the Input program. The Input program contains all facilities to create and to modify a vertical cross-section model, to generate a 2D and 3D finite element mesh and to generate initial conditions. The generation of the initial conditions is done in a separate mode of the Input program (Initial conditions mode). The description is first focused on the creation of a cross-section model and a finite element mesh (Geometry creation mode).

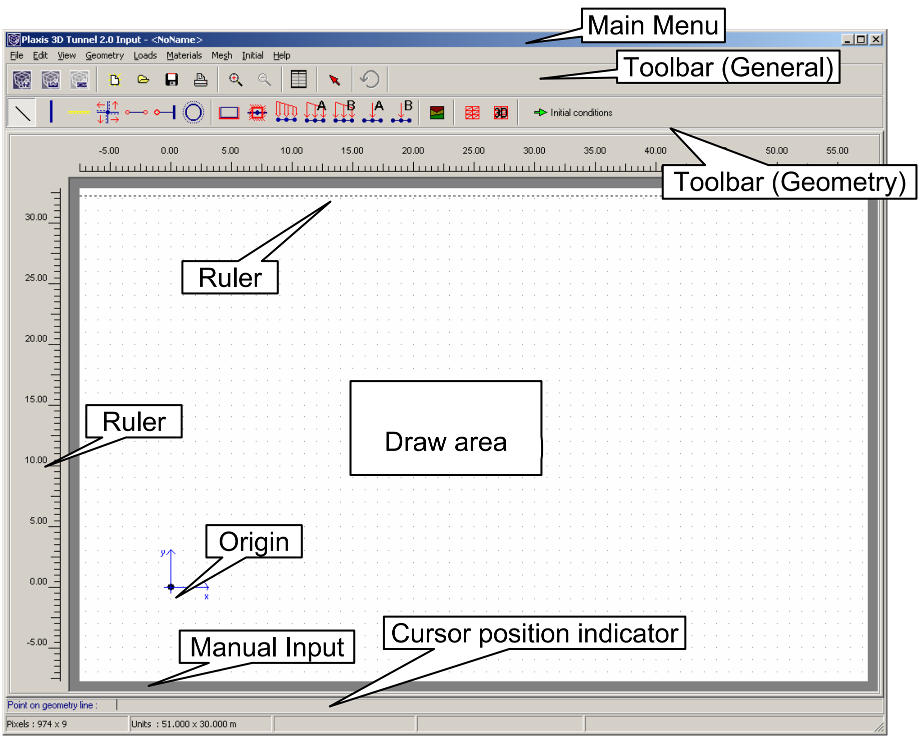


Figure 3.2 Main window of the Input program (Geometry creation mode)

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 Tunnel project file (*.PL3). After the 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 3.2)

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 (General):

This toolbar contains buttons for general actions such as disk operations, printing, zooming or selecting objects. It also contains buttons to start the other sub-programs of the PLAXIS 3D Tunnel package (Calculations, Output, Curves).

Toolbar (Geometry):

This toolbar contains buttons for actions that are related to the creation of a vertical cross-section model. The buttons are ordered in such a way that, in general, following the buttons on the toolbar from the left to the right results in a fully defined model.

Rulers:

At both the left and the top of the draw area, rulers indicate the physical x- and y-coordinates of the vertical cross-section model. This enables a direct view of the geometry dimensions. The rulers can be switched off in the View sub-menu.

Draw area:

The draw area is the drawing sheet on which the cross-section model is created and modified. The creation and modification of a cross-section 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 *y*-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 *y*-coordinates can be entered here by typing the required values separated by a space (*x*-value <space> *y*-value). Manual input of coordinates can be given for all objects, except for *Rotation fixities*.

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> @*y*-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,y*-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, creating a cross-section model, generating finite element meshes and entering data in general. Distinction can be made between the menu of the Geometry creation mode and the menu of the Initial conditions mode. In the Geometry creation mode, the menu consists of the sub-menus *File*, *Edit*, *View*, *Geometry*, *Loads*, *Materials*, *Mesh*, *Initial* and *Help*. In the Initial conditions mode the menu shows the sub-menus *File*, *Edit*, *View*, *Geometry*, *Generate* and *Help*.

The File sub-menu:

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 cross-section model on a selected printer. The print window is presented.
Work directory	To set the default directory where 3D Tunnel project files will be stored.
Import	To import geometry data from other file types (Section 3.2.1).
General settings	To set the basic parameters of the model (Section 3.2.2).
(*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 cross-section model (after an input error). Repetitive use of the undo option is limited to the 10 most recent actions.
Copy	To copy the cross-section model to the Windows clipboard.
Clear selections	To undo all current selections.

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 y-coordinates of all geometry points. The table may be used to adjust existing coordinates.
Rulers	To show or hide the rulers along the draw area.
Cross hair	To show or hide the cross hair during the creation of a cross-section model.
Grid	To show or hide the grid in the draw area.
Axes	To show or hide the arrows indicating the x- and y-axes.
Snap to grid	To activate or deactivate the snapping into the regular grid points.
Point numbers	To show or hide the numbers of the geometry points

Chain numbers To show or hide the labels numbering the chains of plates and geo-grids.

The Geometry sub-menu:

The *Geometry* sub-menu contains the basic options to compose a vertical cross-section model. In addition to a normal geometry line, the user may select plates, geogrids, interfaces, anchors or tunnels. 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 cross-section model. The various options in this sub-menu are explained in Section 3.4.

The Materials sub-menu:

The *Materials* sub-menu is used to activate the database engine for the creation and modification of material data sets for soil and interfaces, plates, geogrids and anchors. The use of the database and the parameters contained in the data sets are described in detail 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 Initial sub-menu:

The *Initial* sub-menu contains the option to proceed to the Initial conditions mode of the Input program. The options in this sub-menu are explained in detail in Section 3.7.

The Geometry sub-menu of the Initial conditions mode:

This sub-menu contains the options to enter the unit weight of water and to draw a phreatic level. The options in this sub-menu are explained in detail in Section 3.8.

The Generate sub-menu of the Initial conditions mode:

This sub-menu contains options to generate initial water pressures or initial effective stresses. The options in this sub-menu are explained in detail in Section 3.8 and 3.9.

3.2.1 READING AN EXISTING PROJECT

An existing PLAXIS 3D Tunnel project can be read by selecting the *Open* option in the *File* 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 Tunnel project files (*.PL3)', which means that the program searches for files with the extension .PL3. After the selection of such a file and clicking on the <Open> button, the corresponding geometry is presented in the draw area.

3.2.2 GENERAL SETTINGS

The *General settings* window appears at the start of new problem and may later be selected from the *File* sub-menu. The *General settings* window contains the two tab sheets *Project* and *Dimensions*. The *Project* tab sheet contains the project name and description, the type of model, the type of elements and the orientation of the model. The *Dimensions* tab sheet contains the basic units for length, force and time (Section 2.1) and the dimensions of the draw area.

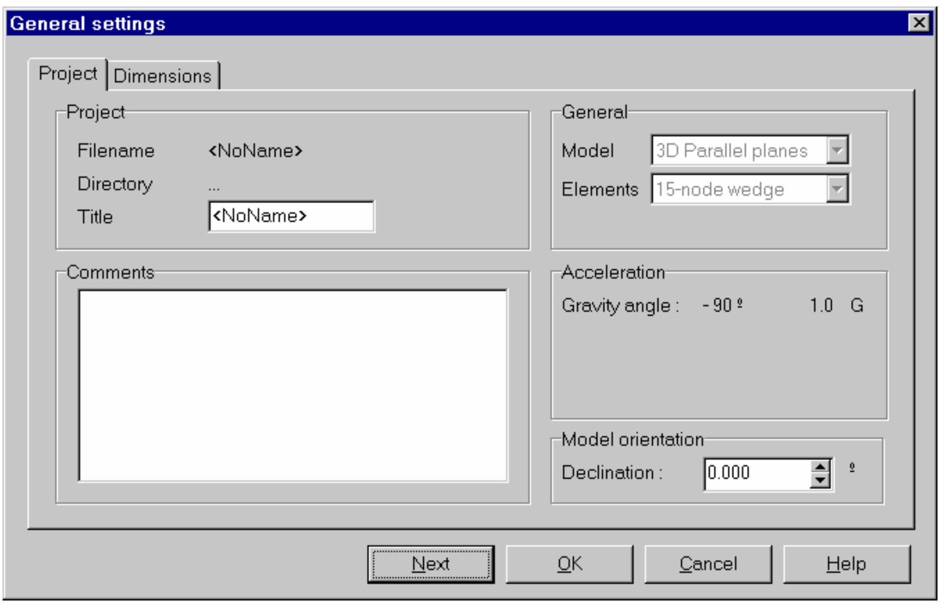


Figure 3.3 General settings window (*Project* tab sheet)

Model:

The *Model* parameter has been preset to *3D parallel planes* and cannot be changed by the user.

Elements:

The *Elements* parameter has been preset to *15-node wedge (3D)* and cannot be changed by the user. This type of volume element for soil behaviour gives a second order interpolation for displacements and the integration involves six stress points (Figure 3.4).

The accuracy of the 15-node wedge element in a 3D analysis is comparable with the 6-node triangular element 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 Tunnel analysis because this will lead to large memory consumption and unacceptable calculation times.

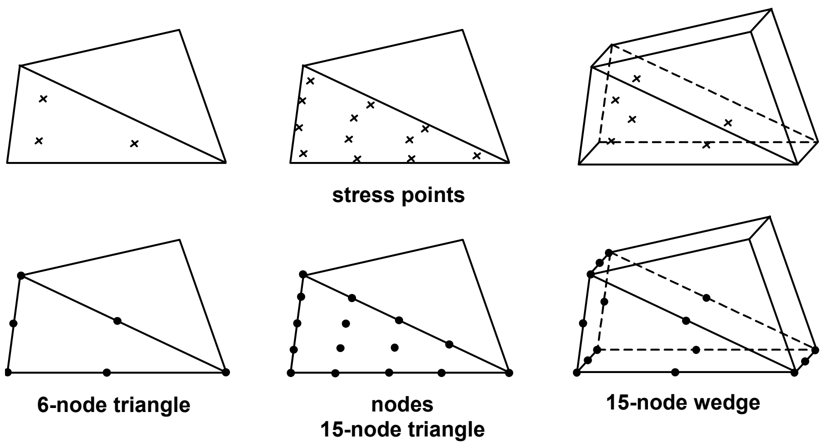


Figure 3.4 Position of nodes and stress points in soil elements

The 15-node wedge element is composed of 6-node triangles in *x*-*y*-direction and 8-node quadrilaterals in *z*-direction. In addition to the soil elements, 8-node plate elements, are used to simulate the behaviour of walls, plates and shells (Section 3.3.2) and 8-node geogrid elements are used to simulate the behaviour of geogrids and wovens. Moreover, 16-node interface elements are used to simulate soil-structure interaction (Section 3.3.4). The plate elements, geogrids and interface elements are compatible with the 8-node quadrilateral sides of a 15-node wedge element. Finally, the geometry creation mode allows for the input of fixed-end anchors and node-to-node anchors (Section 3.3.5 and 3.3.6).

Gravity:

Gravity has been preset to 1 G and the direction of gravity coincides with the negative *y*-axis, i.e. an orientation of –90° in the *x*-*y*-plane. Gravity is implicitly included in the unit weights given by the user (Section 3.5.2). In this way, the gravity is controlled by the total load multiplier for weights of materials, *ΣMweight* (Section 4.7.1).

Declination:

The vertical cross-section model is always created in the *x*-*y*-plane, whereas the 3D model extension is performed in the *z*-direction. As a result, when creating tunnels, the longitudinal direction of the tunnel in the PLAXIS 3D Tunnel program is always in *z*-direction. For some applications (see description of the Jointed Rock model in the Material Models Manual) it is necessary to define the orientation of the *z*-axis with respect to the geographical North-South direction. This is done by means of the *Declination* parameter. The *Declination* is the positive angle from the North direction to the positive *z*-direction of the model (Figure 3.5).

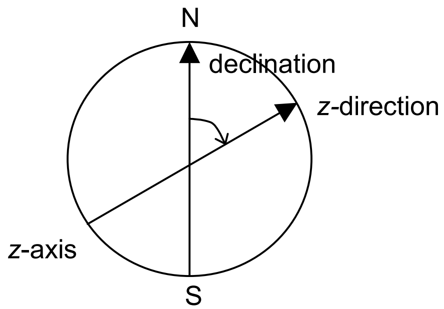


Figure 3.5 Definition of the *Declination* parameter

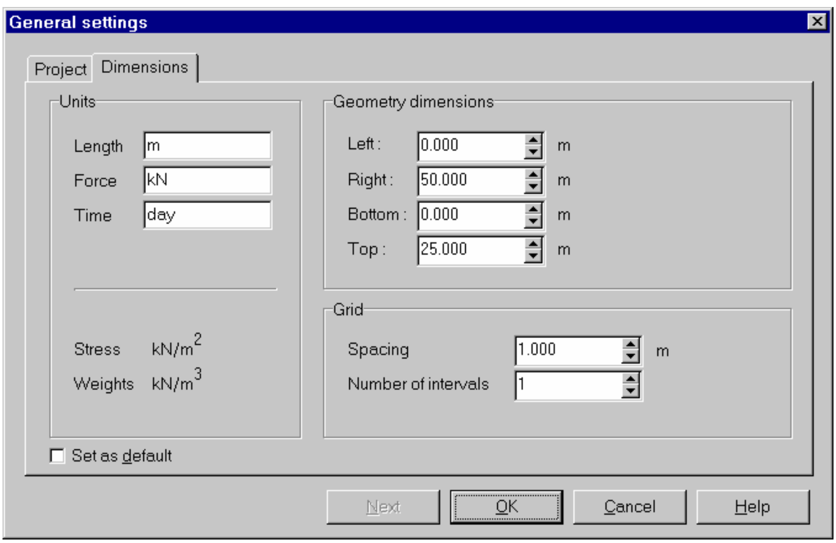


Figure 3.6 General settings window (*Dimensions* tab sheet)

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.

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.

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.

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 cross-section model that is to be created will fit within the dimensions. The dimensions are entered in the *Dimensions* tab sheet of the *General settings* window. The dimensions of the draw area do not influence the geometry itself and may be changed when modifying an existing project, provided that the existing geometry fits within the modified dimensions.

Grid:

To facilitate the creation of the cross-section 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 vertical cross-section model. The vertical cross-section model is a representation of the main vertical cross-section of the problem of interest, including all objects that are present in any vertical cross-section of the full 3D model. For example, if a wall is present only in a part of the 3D model, it must be included in the cross-section model.

A vertical cross-section model consists of points, lines and area clusters. 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 cross-section model to simulate tunnel linings, walls, plates, soil-structure interaction or loadings.

It is recommended to start the creation of a cross-section model by drawing the cross-section outline. In addition, the user may specify material layers, structural objects, lines used for construction phases, loads and boundary conditions. As mentioned before, the cross-section model must include all objects that are present in any cross-section of the full 3D model. Moreover, the cross-section model should not only include the initial situation, but also situations that arise in the various calculation phases.

After the geometry components of the cross-section model have been created, the user should compose data sets of material parameters and assign the data sets to the corresponding geometry components (Section 3.5). When the full cross-section model has been defined (including all objects appearing in any cross-section or in any construction stage) and all geometry components have their initial properties, the finite element mesh can be generated. From the cross-section model, a 2D mesh must be generated first (Section 3.6). If the 2D mesh is satisfactory, an extension into the third dimension (the *z*-direction) can be defined. This is done by specifying a single *z*-coordinate for each vertical plane that is required to define the 3D model. In the 3D model, vertical planes at specified *z*-coordinates are referred to as *z-planes*, whereas volumes between two *z*-planes are denoted as *slices* (Figure 3.7).

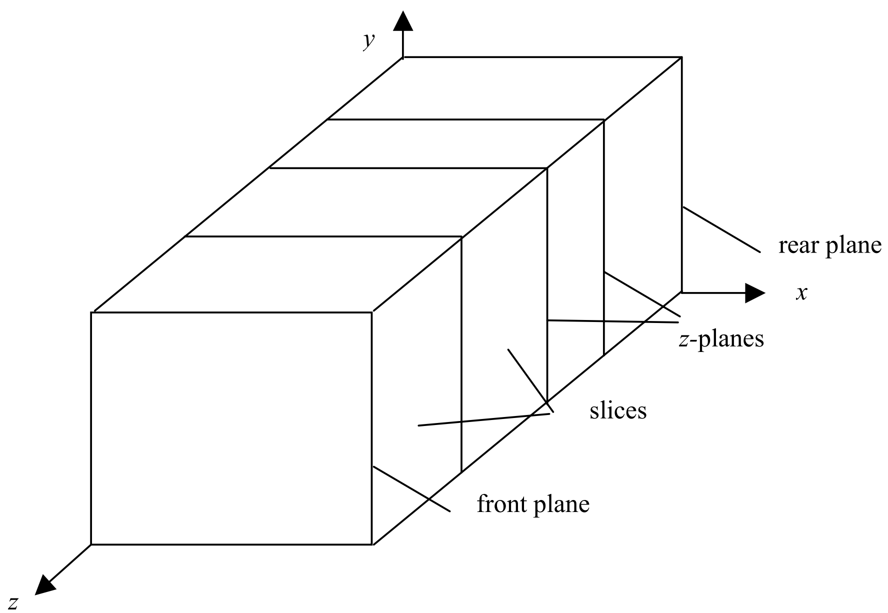
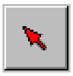


Figure 3.7 Definition of *z*-planes and slices

PLAXIS will generate a fully 3D finite element mesh based on the 2D meshes in each of the specified *z*-planes. In the 3D model, each *z*-plane is similar and includes all objects that are created in the cross-section model. Although the analysis is fully 3D, the model is essentially 2D, since there is no variation in *z*-direction. Later on it is possible to activate or deactivate objects that are not supposed to be present in a certain *z*-plane or a slice at a certain calculation phase (see *Staged construction,* Section 4.6). In this procedure, activation or deactivation of objects can be done for each *z*-plane or slice individually. In this way it is possible to create a true 3D situation.

Selecting geometry components

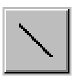


When the *Selection* tool (red arrow) is active, a geometry component may be selected by clicking once on that component in the cross-section model. Multiple components of the same type can be selected simultaneously by holding down the <Shift> key on the keyboard while selecting the desired components.

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 POINTS AND LINES



The basic input item for the creation of a cross-section model is the *Geometry line*. This item can be selected from the *Geometry* sub-menu as well as from the second toolbar.

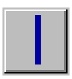
When the *Geometry line* option is selected, the user may create points and lines in the draw area by clicking with the mouse pointer (graphical input) or by typing coordinates at the command line (keyboard input). As soon as the left hand mouse button is clicked in the draw area a new point is created, 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 cross-section and drag it to the desired position. To delete a point or line, select the point or the line in the cross-section 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 only two geometry lines come together, then the two lines are combined to give one straight line between the outer points. If more than two geometry lines come together in the point to be deleted, then all these connected geometry lines will be deleted as well.

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. Each cluster can be given certain material properties to simulate the behaviour of the soil in that part of the geometry (Section 3.5.2). The clusters are divided into soil elements during mesh generation (Section 3.6).

3.3.2 PLATES



Plates are structural objects used to model slender quasi- two-dimensional structures with a significant flexural rigidity (or bending stiffness) and a normal stiffness in the three-dimensional model. Plates in the PLAXIS 3D Tunnel program can be used to simulate the influence of walls, plates, shells or linings extending in *z*-direction. In a vertical cross-section model, plates appear as 'blue lines'. If no material data set has been assigned, plates appear as light blue lines. Once a material set has been assigned they appear as dark blue lines. Examples of geotechnical structures involving plates are shown in Figure 3.8.

Plates can be selected from the *Geometry* sub-menu or by clicking on the corresponding button in the toolbar. The creation of plates in the cross-section model is similar to the creation of geometry lines (Section 3.3.1). When creating plates, corresponding geometry lines are created simultaneously. Hence, it is not necessary to create first a geometry line at the position of a plate.

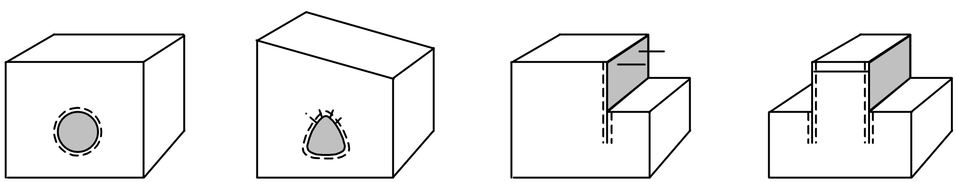


Figure 3.8 Applications in which plates, anchors and interfaces are used

By extending a vertical cross-section model into a 3D finite element model, plates are generated for all slices in the full *z*-range of the model. In the *Initial conditions*, plates are automatically deactivated and they may be activated in individual slices in the framework of *Staged construction*. Hence, it is possible to have plates in individual slices, but not along *z*-planes.

Plate elements

Plates in the 3D finite element model are composed of two-dimensional 8-node plate elements with six degrees of freedom per node: Three translational degrees of freedom (u_x, u_y, u_z) and three rotational degrees of freedom (ϕ_x, ϕ_y, ϕ_z) (Figure 3.9). The 8-node plate elements are compatible with the 8-noded quadrilateral face (in *z*-direction) of a soil element. The plate elements are based on Mindlin's beam theory (Reference 2). This theory allows for beam deflections due to shearing as well as bending. The Mindlin beam theory has been extended for plates. In addition to deformation perpendicular to the plate, the plate element can change length in each longitudinal direction when a corresponding axial force is applied. Plate elements can become plastic if a prescribed maximum bending moment or maximum axial force is reached.

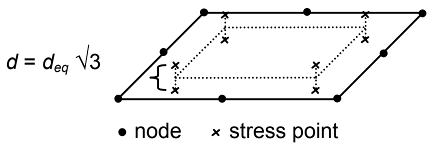


Figure 3.9 Position of nodes and stress points in an 8-node plate element

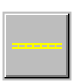
The material properties of plates are contained in material data sets (Section 3.5.3). The most important parameters are the flexural rigidity (bending stiffness) *EI* and the axial stiffness *EA*. From these two parameters an equivalent plate thickness d_{eq} is calculated from the equation:

$$d_{eq} = \sqrt{12\frac{EI}{EA}}$$

Bending moments and axial forces are evaluated from the stresses at the stress points. A plate element contains four pairs of Gaussian stress points. Within each pair, stress points are located at a distance $\frac{1}{2}\, d_{eq}\sqrt{3}$ above and below the plate centre-line. Figure 3.9 shows a single plate element with an indication of the nodes and stress points.

It is important to note that a change in the ratio EI / EA will change the equivalent thickness d_{eq} and thus the distance separating the stress points. If this is done when existing forces are present in the plate element, it would change the distribution of bending moments, which is unacceptable. For this reason, if material properties of plate elements are changed during an analysis (for example in the framework of Staged Construction) it should be noted that the ratio EI / EA must remain unchanged.

3.3.3 GEOGRIDS



Geogrids are slender quasi- two-dimensional structures with a normal stiffness but with no bending stiffness. Geogrids can only sustain tensile forces and no compression. These objects are generally used to model soil reinforcements. In a vertical cross-section model geogrids appear as ‘pink lines’ as long as no material data set has been assigned, and as 'yellow lines' as soon as a material data set has been assigned. Geogrids can be selected from the *Geometry* sub-menu or by clicking on the corresponding button in the toolbar. The creation of geogrids in the cross-section model is similar to the creation of geometry lines (Section 3.3.1). When creating geogrids, corresponding geometry lines are created simultaneously. The only material property of a geogrid is an elastic normal (axial) stiffness *EA*, which can be specified in the material database (Section 3.5.4).

By extending a vertical cross-section model into a 3D finite element model, geogrids are generated for all slices in the full z-range of the model. In the *Initial conditions*, geogrids are automatically deactivated and they may be activated in individual slices in the framework of *Staged construction*. Hence, it is possible to have geogrids in individual slices, but not along z-planes.

Geogrid elements

Geogrids are composed of two-dimensional 8-node geogrid elements with three degrees of freedom in each node (u_x, u_y, u_z). The 8-node geogrid elements are compatible with the 8-noded quadrilateral face (in the z-direction) of a soil element. Axial forces are evaluated at the Gaussian stress points. The location of these stress points is indicated in Figure 3.10.

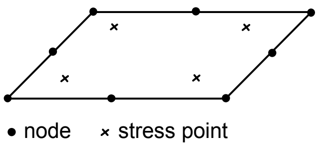
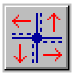


Figure 3.10 Position of nodes and stress points in a 8-node geogrid element

3.3.4 INTERFACES



Interfaces are used to model the interaction between structures and the soil. The interfaces can also be used, in combination with geometry lines, sheet piles or tunnels, for the simulation of impermeable screens. Examples of geotechnical structures involving interfaces are presented in Figure 3.8. Interfaces can be selected from the *Geometry* sub-menu or by clicking on the corresponding button in the toolbar.

The creation of an interface in the cross-section model is similar to the creation of a geometry line. The interface appears as a dashed line at the right hand side of the geometry line (considering the direction of drawing) to indicate at which side of the geometry line the interaction with the soil takes place. The side at which the interface will appear is also indicated by the arrow on the cursor pointing in the direction of drawing. In order to place an interface at the other side, it should be drawn in the opposite direction. Note that, interfaces can be placed at both sides of a geometry line. This enables a full interaction between structural objects (walls, plates, geogrids, etc.)

and the surrounding soil. To be able to distinguish between the two possible interfaces along a geometry line, the interfaces are indicated by a plus-sign (+) or a minus-sign (-). This sign is just for identification purposes; it does not have a physical meaning and it has no influence on the results.

A typical application of interfaces would be to model the interaction between a tunnel lining 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 ('wall' friction and adhesion) to the soil strength (friction angle and cohesion). For detailed information about the interface properties, see Section 3.5.2.

By extending a vertical cross-section model into a 3D finite element model, interfaces are generated for all slices in the full *z*-range of the model. It is not possible to have interfaces along *z*-planes. The activation or deactivation of interfaces in the *Initial conditions* and in *Staged Construction* is automatically done according to the activation or deactivation of the adjacent soil element (Section 3.9.1 and 4.6.3).

Interface elements

Interfaces are composed of 16-node interface elements. Figure 3.11 shows how interface elements are connected to soil elements. Interface elements consist of eight pairs of nodes, compatible with the 8-noded quadrilateral face (in the *z*-direction) of a soil element.

In the figure, the 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 a zero thickness.

Each interface has assigned to it a 'virtual thickness' which is an imaginary dimension used to define the material properties of the interface. The virtual thickness is calculated 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 3.6.1). The default value of the *Virtual thickness factor* is 0.1. This value can be changed by double clicking on the geometry line and selecting the interface from the selection dialog box. However, care should be taken when changing the default factor. Further details of the significance of the virtual thickness are given in Section 3.5.2.

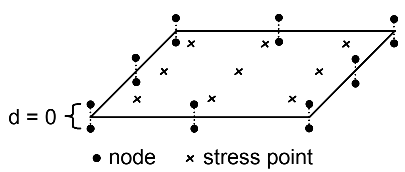


Figure 3.11 Distribution of nodes and stress points in interface elements

The stiffness matrix for interface elements is obtained by means of Gaussian integration using nine integration points. The position of these integration points (or stress points) is chosen such that the numerical integration is exact for linear stress distributions. Note that the position of the stress points is different from the position of the node pairs.

Figure 3.12 and Figure 3.13 show that problems of soil-structure interaction may involve points that require special attention. Corners in stiff structures and an abrupt change in boundary condition may lead to high peaks in the stresses and strains. Volume elements are not capable of reproducing these sharp peaks and will, as a result, produce non-physical stress oscillations. This problem can be solved by making use of interface elements as shown below.

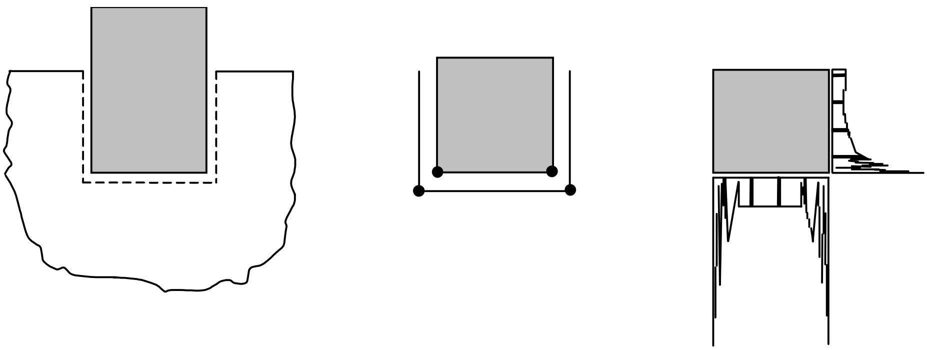


Figure 3.12 Inflexible corner point, causing poor quality stress results

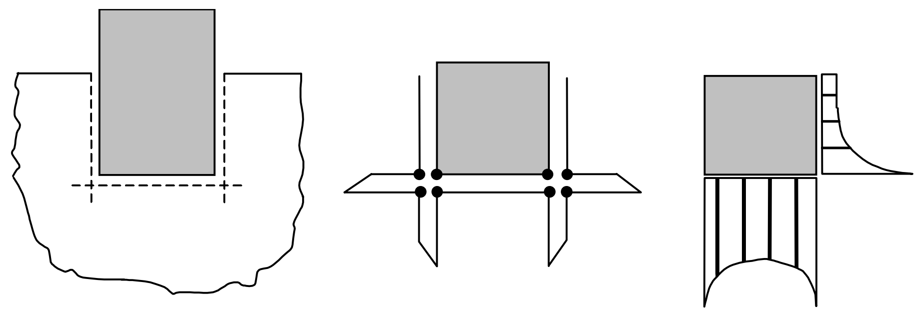
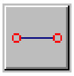


Figure 3.13 Flexible corner point with improved stress results

This figure shows that the problem of stress oscillation may be prevented by specifying additional interface elements inside the soil body. These elements will enhance the flexibility of the finite element mesh and will thus prevent non-physical stress results. Reference 22 provides additional theoretical details on this special use of interface elements.

When using interfaces in consolidation or groundwater flow calculations, they represent a fully impermeable screen by default. For inactive interfaces, node pairs are fully coupled, whereas for active interfaces they are fully separated. As a result, an active interface acts as a fully impermeable screen (separation of head degrees-of-freedom of node pairs) and an inactive interface is fully permeable (coupling of head degrees-of-freedom of node pairs).

3.3.5 NODE-TO-NODE ANCHORS



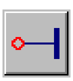
Node-to-node anchors are springs that are used to model ties between two points. This type of anchors can be selected from the *Geometry* sub-menu or by clicking on the corresponding button in the toolbar. Typical applications include the modelling of a cofferdam as shown in Figure 3.8. It is not recommended to draw a geometry line at the position where a node-to-node anchor is to be placed. However, the end points of node-to-node anchors must always be connected to geometry lines, but not necessarily to existing geometry points. In the latter case a new geometry point is automatically introduced. The creation of node-to-node anchors is similar to the creation of geometry lines (Section 3.3.1) but, in contrast to other types of structural objects, geometry lines are not simultaneously created with the anchors. Hence, node-to-node anchors will not divide clusters nor create new ones.

By extending a vertical cross-section model into a 3D finite element model, node-to-node anchors are generated in each plane for which a user has specified the z-coordinate (the z-planes). In the *Initial conditions*, anchors are automatically deactivated and they may be activated in individual z-planes in the framework of *Staged construction*. Hence, it is possible to have node-to-node anchors in individual z-planes, but not in the z-direction.

A node-to-node anchor is a two-node elastic spring element with a constant spring stiffness (normal stiffness). This element can be subjected to tensile forces (for anchors) as well as compressive forces (for struts). The absolute force can be limited to allow for the simulation of anchor failure. The properties can be entered in the material database for anchors (Section 3.5.5). Anchors are represented by a grey line as long as no material data set has been assigned to them, and as a black line as soon as a material data set has been assigned.

Node-to-node anchors can be pre-stressed during a plastic calculation using *Staged construction* as *Loading input* (Section 4.7.2)

3.3.6 FIXED-END ANCHORS



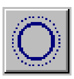
Fixed-end anchors are springs that are used to model a tying of a single point. This type of anchor can be selected from the *Geometry* sub-menu or by clicking on the corresponding button in the toolbar. An example of the use of fixed-end anchors is the modelling of struts (or props) to sheet-pile walls, as shown in Figure 3.8. Fixed-end anchors must always be connected to existing geometry lines, but not necessarily to existing geometry points. A fixed-end anchor is visualised as a rotated T (—|). The length of the plotted T is arbitrary and does not have any particular physical meaning. By default, a fixed-end anchor is pointing in the positive x-direction, i.e. the angle in the x,y-plane is zero. By double clicking in the middle of the T the anchor properties window appears in which the angle can be changed. Fixed-end anchors can be placed either parallel to the x,y-plane or parallel to the y,z-plane, which can be selected from the combo box in the properties window. The angle is defined in the anticlockwise direction, starting from the positive x-direction or from the negative z-direction towards the y-direction. In addition to the angle, the equivalent length of the anchor may be entered in the properties window. The equivalent length is defined as the distance between the anchor connection point and the fictitious point in the longitudinal direction of the anchor where the displacement is assumed to be zero.

By extending a vertical cross-section model into a 3D finite element model, fixed-end anchors are generated in each plane for which a user has specified the z-coordinate (the z-planes). In the *Initial conditions*, anchors are automatically deactivated and they may be activated in individual z-planes in the framework of *Staged construction*. Hence, it is possible to have fixed-end anchors in individual z-planes.

A fixed-end anchor is a one-node elastic spring element with a constant spring stiffness (or normal stiffness). The other end of the spring (defined by the equivalent length and the direction) is fixed.

The properties can be entered in the material database for anchors (Section 3.5.5). Anchors are represented by a grey line as long as no material data set has been assigned to them, and as a black line as soon as a material data set has been assigned. Fixed-end anchors can be pre-stressed during a plastic calculation using *Staged construction* as *Loading input* (Section 4.7.2).

3.3.7 TUNNELS



The tunnel option can be used to create circular and non-circular tunnel cross-sections which are to be included in the cross-section model. A tunnel cross-section is composed of arcs and lines, optionally supplied with a lining and an interface. A tunnel cross-section can be stored as an object on the hard disk and included in other projects. The tunnel option is available from the *Geometry* sub-menu or from the toolbar.

Tunnel designer

Once the tunnel option has been selected, the Tunnel designer input window appears. The tunnel designer contains the following items (Figure 3.14):

Tunnel menu:	Menu with options to open and save a tunnel object and to set tunnel attributes.
Toolbar:	Bar with buttons as shortcuts to set tunnel attributes.
Display area:	Area in which the tunnel cross-section is plotted.
Rulers:	The rulers indicate the dimension of the tunnel cross-section in local coordinates. The origin of the local system of axes is used as a reference point for the positioning of the tunnel in the cross-section model.
Section group box:	Box containing shape parameters and attributes of individual tunnel sections.

Other parameters: See further.

Standard buttons: To accept (OK) or to cancel the created tunnel.

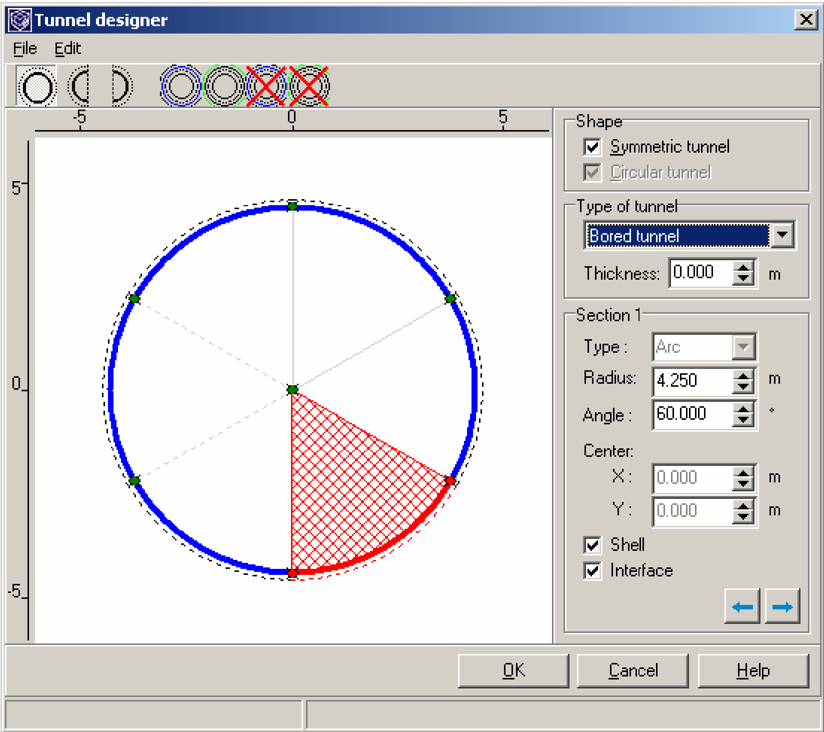
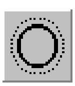


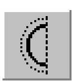
Figure 3.14 Tunnel designer with standard tunnel shape

Basic tunnel shape

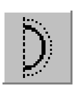
Once the tunnel option has been selected, use one of the following toolbar buttons to select a basic tunnel shape:



Whole tunnel



Half a tunnel - Left half



Half a tunnel - Right half

A *Whole tunnel* should be used if the full tunnel cross-section is included in the cross-section model. A half tunnel should be used if the cross-section model includes only one symmetric half of the problem where the symmetry line of the cross-section model corresponds to the symmetry line of the tunnel. Depending on the side of the symmetry line that is used in the cross-section model the user should select the right half of a tunnel or the left half. A half tunnel can also be used to define curved sides of a larger structure, such as an underground storage tank. The remaining linear parts of the structure can be added in the draw area using geometry lines or plates.

Type of tunnel:

You select the type of tunnel that is considered before creating the tunnel cross-section. The available options are: *None* , *Bored tunnel* or *NATM tunnel*.

None: Select this option when you want to create an internal geometry contour composed of different sections and have no intention to create a tunnel. Each section is defined by a line, an arc or a corner. The outline consists of two lines if you enter a positive value for the *Thickness* parameter. The two lines will form separate clusters with a corresponding thickness when inserting the outline in the cross-section model. In the tunnel designer these lines are drawn at an arbitrary distance.
A lining (shell) and an interface may be added to individual sections of the outside surface of the lining.

Bored tunnel: Select this option to create a circular tunnel that includes a homogeneous tunnel lining (composed of a circular shell) and an interface at the outside. The tunnel shape consists of different sections that can be defined with arcs. Since the tunnel lining is circular, each section has the radius that is defined in the first section. The tunnel outline consists of two lines if you enter a positive value for the *Thickness* parameter. This way a thick tunnel lining can be created that is composed of volume elements, for example to simulate the tunnel-boring machine (TBM).
The tunnel lining (shell) is considered to be homogeneous and continuous. As a result, assigning material data and the activation or deactivation of the shell can only be done for the lining as a whole (and not individually for each section). If the shell is active, a contraction of the tunnel lining (shrinkage) can be specified for each individual cross-section (z-plane) to simulate the volume loss due to the tunnel boring process (Section 4.6.10).

NATM tunnel: Select this option to create a tunnel that includes a tunnel lining (composed of plates) and an interface at the outside. The tunnel outline consists of different sections that can be defined with arcs. The outline consists of two lines if you enter a positive value for the *Thickness* parameter. This way a thick tunnel lining can be created that is composed of volume elements. It is possible to apply a shell to the outer contour line, for example to simulate a combination of an outer lining (sprayed concrete as plane) and an inner lining (final lining as volume).
The tunnel lining (shell) is considered to be discontinuous. As a result, assigning material data and the activation or deactivation of lining parts is done for each section individually. It is not possible to apply a contraction of the

shell (shrinkage) for NATM tunnels. To simulate the deformations due to the excavation and construction in NATM tunnels other calculation methods are available (see Section 4.6.8).

Tunnel sections:

The creation of a tunnel cross-section starts with the definition of the inner tunnel boundary, which is composed of sections. Each section 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 inclination angle of two adjacent tunnel sections. When entering the tunnel designer, a standard circular tunnel is presented composed of 6 sections (3 sections for half a tunnel). The first section starts with a horizontal tangent at the lowest point on the local *y*-axis (highest point for a left half), and runs in the anti-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 the case of an arc) or by the *Length* (in the 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. If both sections are arcs, the two sections have the same radial (normal of the tunnel section), but not necessarily the same radius (Figure 3.15). 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 tunnel 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 tunnel section are automatically determined such that the end radial coincides again with the *y*-axis.

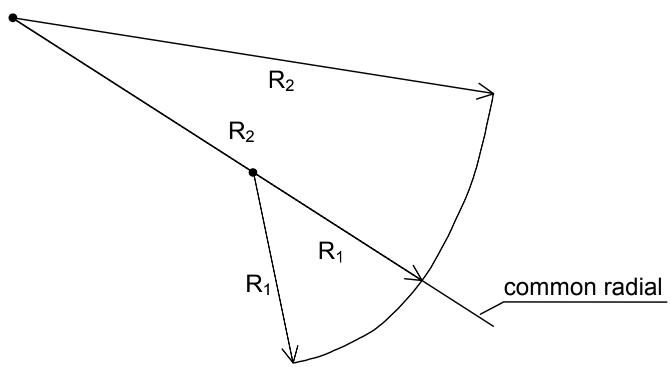


Figure 3.15 Detail of connection point between two tunnel sections

For a whole tunnel the start point of the first section should coincide with the end point of the last section. This is not automatically guaranteed. The distance between the start point and the end point (in units of length) is defined as the closing error. An eventual closing error is indicated on the status line of the tunnel designer. When a significant closing error exists, it is advisable to carefully check the section data.

The number of sections follows from the sum of the section angles. For whole tunnels the sum of the angles is 360 degrees and for half tunnels this sum is 180 degrees. The maximum angle of a section is 90.0 degrees. The automatically calculated angle of the last section completes the tunnel cross-section and it cannot be changed. 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.

Symmetric tunnel:

The option *Symmetric* is only relevant for whole tunnels. When this option is selected, the tunnel is made fully symmetric. In this case the input procedures are similar to those used when entering half a tunnel (right half). The left half of the tunnel is automatically made equal to the right half.

Circular tunnel:

When changing the radius of one of the tunnel sections, the tunnel ceases to be circular. To enforce the tunnel to be circular, the *Circular* option may be selected. If this option is selected, all tunnel sections will be arcs with the same radius. In this case the radius can only be entered for the first tunnel section. This option is automatically selected when the type of tunnel is a bored tunnel.

Including tunnel in cross-section model

After clicking on the <OK> button in the tunnel designer the window is closed and the main input window is displayed again. A tunnel symbol is attached to the cursor to emphasize that the reference point for the tunnel must be selected. The reference point will be the point where the origin of the local tunnel axes is located. When the reference point is entered by clicking with the mouse in the cross-section model or by entering the coordinates in the manual input line, the tunnel is included in the cross-section model, taking into account eventual crossings with existing geometry lines or objects.

By extending a vertical cross-section model into a 3D finite element model, tunnels are generated for all slices in the full *z*-range of the model. In the *Initial conditions*, tunnel linings (shells) are automatically deactivated. The excavation of a tunnel (or part of a tunnel) and the activation of a lining (or part of a lining) may be considered in individual slices in the framework of *Staged construction*. In this way it is possible to model the tunnel front and the excavation process of a tunnel in detail.

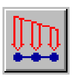
Editing an existing tunnel

An existing tunnel can be edited by double clicking its reference point or one of the other tunnel points. As a result, the tunnel designer window reappears showing the existing tunnel cross-section. Desired modifications can now be made. On clicking the <OK> button the 'old' tunnel is removed and the 'new' tunnel is directly included in the cross-section model using the original reference point. Note that previously assigned material sets of a lining must be reassigned after modification of the tunnel.

3.4 LOADS AND BOUNDARY CONDITIONS

The *Loads* sub-menu contains the options to introduce distributed loads, point (or line) loads and prescribed displacements in the cross-section model. Loads and prescribed displacements can be applied at the model boundaries as well as inside the model.

3.4.1 PRESCRIBED DISPLACEMENTS



Prescribed displacements are special conditions that can be imposed on the model to control the displacements of certain points. Prescribed displacements can be selected from the *Loads* sub-menu or by clicking on the corresponding button in the toolbar. The input of *Prescribed displacements* in the cross-section model is similar to the creation of geometry lines (Section 3.3.1). By default, the input values of prescribed displacements are set such that the vertical displacement component is one unit in the negative vertical direction (u_y = -1) and the horizontal displacement components are free.

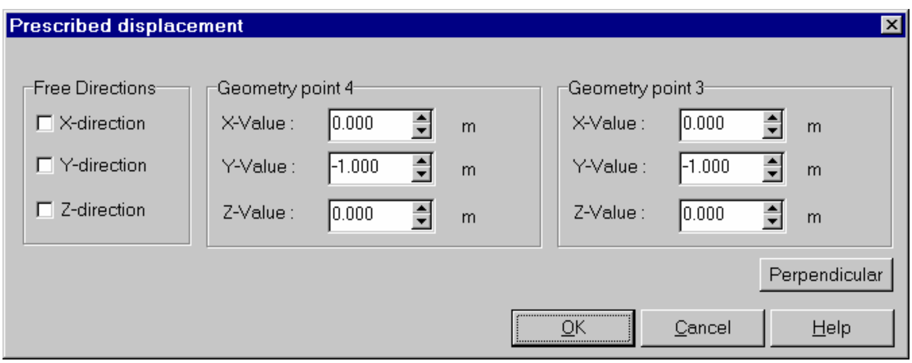


Figure 3.16 Input window for prescribed displacements

The input values of prescribed displacements can be changed by double clicking the corresponding geometry line and selecting *Prescribed displacements* from the selection dialog box. As a result, a prescribed displacements window appears in which the input values of the prescribed displacements of both end points of the geometry line can be changed. The distribution is always linear along the line. The input value must be in the range [-9999, 9999]. In the case that one of the displacement directions is prescribed whilst the other direction is free, one can use the check boxes in the *Free directions* group to indicate which direction is free. The <Perpendicular> button can be used to impose a prescribed displacement of one unit perpendicular to the corresponding geometry line and $u_z = 0$. For internal geometry lines, the displacement is perpendicular to the right side of the geometry line (considering that the line goes from the first point to the second point). For geometry lines at a model boundary, the displacement direction is towards the inside of the model.

On a geometry line where both prescribed displacements and loads are applied, the prescribed displacements have priority over the loads during the calculations, except if the prescribed displacements are not activated. On the other hand, when prescribed displacements are applied on a line with full fixities, the fixities have priority over the prescribed displacements, which means that the displacements on this line remain zero. Hence, it is not useful to apply prescribed displacements on a line with full fixity.

Prescribed displacements appear as line displacements in the cross-section model, but in the full 3D model they can be used both as line displacements on individual vertical cross-sections (*z*-planes) as well as surface displacements on volume sections in *z*-direction (slices). Although the global input values of prescribed displacements can be specified in the cross-section model, the precise distribution and activation or deactivation over the various planes and slices in *z*-direction may be changed in the framework of *Staged construction*. Moreover, an existing composition of prescribed displacements may be increased globally by means of the load multipliers *Mdisp* and *ΣMdisp* (Section 4.7.1).

During calculations, the reaction forces corresponding to prescribed displacements in *x*-, *y*- and *z*-direction are calculated and stored as output parameters (*Force-X*, *Force-Y*, *Force-Z*).

3.4.2 FIXITIES

Fixities are prescribed displacements equal to zero. These conditions can be applied to geometry lines as well as to geometry points. Fixities can be selected from the *Loads* sub-menu. In the 2D cross-section model, distinction can be made between *Horizontal fixity* ($u_x = 0$) and *Vertical fixity* ($u_y=0$). In addition, one can select *Total fixity*, which is a combination of both ($u_x=u_y=0$).

In the full 3D model, fixities in *z*-direction are, in principle, derived from the fixities in the other directions. If a point is fixed in the *x*- and *y*-direction it will automatically be fixed in the *z*-direction. If a point is free in *x*- or *y*-direction it is free in *z*-direction. The front plane and the rear plane in a 3D model are always fixed in the *z*-direction. On a geometry where fixities are used as a condition, the fixities have priority over other types of loading conditions (prescribed displacements or loads) during the calculations.

Prescribed displacements and interfaces

To introduce a sharp transition in different prescribed displacements or between prescribed displacements and fixities (for example to model a trap-door problem; Figure 3.17), it is necessary to introduce an interface at the point of transition perpendicular to the geometry line. As a result, the thickness of the transition zone between the two different displacements is zero. If no interface is used then the transition will occur within one of the elements connected to the transition point. Hence, the transition zone will be determined by the size of the element. The transition zone will therefore be unrealistically wide.

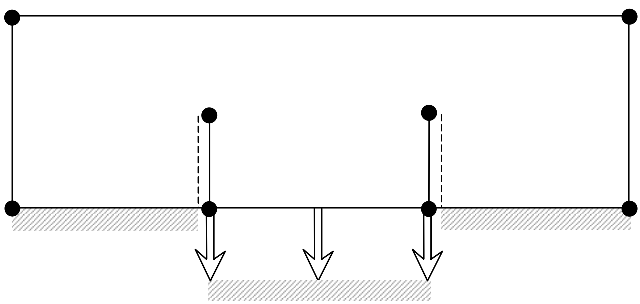
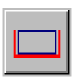


Figure 3.17 Modelling of a trap-door problem using interfaces

3.4.3 STANDARD FIXITIES



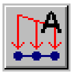
On selecting *Standard fixities* from the *Loads* sub-menu or by clicking on the corresponding button in the toolbar PLAXIS automatically imposes a set of general boundary conditions to the cross-section model. These boundary conditions are generated according to the following rules:

- Vertical geometry lines for which the x-coordinate is equal to the lowest or highest x-coordinate in the model obtain a horizontal fixity ($u_x = 0$).
- Horizontal geometry lines for which the y-coordinate is equal to the lowest y-coordinate in the model obtain a full fixity ($u_x = u_y = 0$).
- Plates that extend to the boundary of the cross-section model obtain a fixed rotation (around the z-axis) in the point at the boundary ($\phi_z = 0$) if at least one of the displacement directions of that point is fixed.

In the full 3D model, the front plane and the rear plane are fixed in the z-direction. Moreover, points that are fixed in x- and y-direction are automatically fixed in z-direction as well. Plates that extend to the front plane or the rear plane obtain a rotation fixity around the x-axis and a rotation fixity around the y-axis at these planes.

Standard fixities can be used as a convenient and fast input option for most practical applications.

3.4.4 DISTRIBUTED LOADS



The creation of a distributed load in the cross-section model is similar to the creation of a geometry line (Section 3.3.1). Two load systems (A and B) are available for a combination of distributed loads, line loads and point loads. The load systems A and B can be activated independently. Distributed loads for load system A or B can be selected from the *Loads* sub-menu or by clicking on the corresponding button in the toolbar.

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*- a *y*- and/or a *z*-component. By default, when applying loads to the cross-section boundary, the load will be a unit pressure perpendicular to the boundary. The input value of a load may be changed by double clicking the corresponding geometry line and selecting the corresponding load system from the selection dialog box. As a result, the distributed loads window is opened in which the three components of the load can be specified for both end points of the geometry line in the cross-section model. The distribution is always linear along the line.

Distributed loads appear as line loads in the cross-section model, but in the full 3D model they can be used both as line loads on individual vertical cross-sections (*z*-planes) as well as real distributed loads on volume sections in *z*-direction (slices). In the former case, the input value should be regarded as a force per unit of length, whereas in the latter case the input value should be regarded as a force per unit of area. Although the global input values of distributed loads can be specified in the cross-section model, the precise distribution and activation or deactivation of the loads over the various planes and slices may be changed in the framework of *Staged construction*. Moreover, an existing composition of loads may be increased globally by means of the load multipliers *MloadA* (or *ΣMloadA*) for load system A and *MloadB* (or *ΣMloadB*) for load system B (Section 4.7.1).

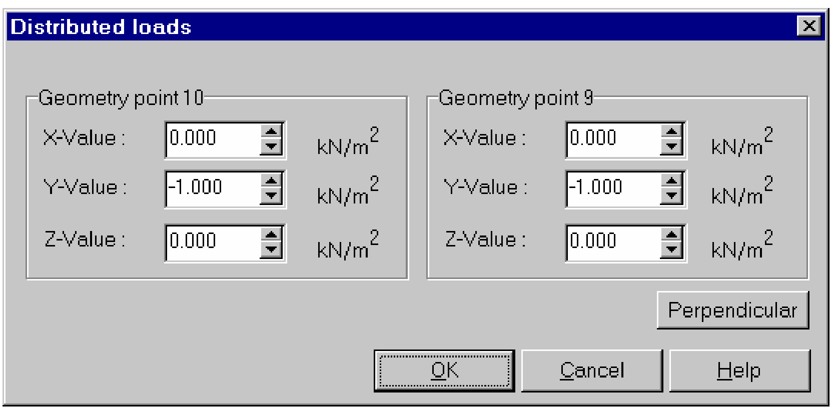
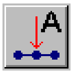


Figure 3.18 Input window for distributed loads

On a part of the geometry where both prescribed displacements and distributed loads are applied, the prescribed displacements have priority over the distributed loads during the calculations, provided that the prescribed displacements are active. Hence, it is not useful to apply distributed loads on a line with fully prescribed displacements. When only one displacement direction is prescribed whilst the other direction is free or inactive, it is possible to apply a distributed load in the free direction.

Although the distributed loads option does not allow for normal loads in the *z*-direction on vertical cross-section clusters, it is possible to create this type of load in individual *z*-planes of the 3D model in the framework of *Staged construction* (see *z*-loads). This special feature is described in Section 4.6.5.

3.4.5 POINT LOADS AND LINE LOADS



This option may be used to create point loads or line loads on a line extending in *z*-direction. The creation of a point load in the cross-section model is similar to the creation of a geometry point (Section 3.3.1). Two load systems (A and B) are available for a combination of distributed loads, line loads and point loads. The load systems A and B can be activated independently. Point loads for load system A or B 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 (or line load) are given in force per unit of length (for example kN/m). Point loads may consist of a *x*- a *y*- and/or a *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 point and selecting the corresponding load system 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.

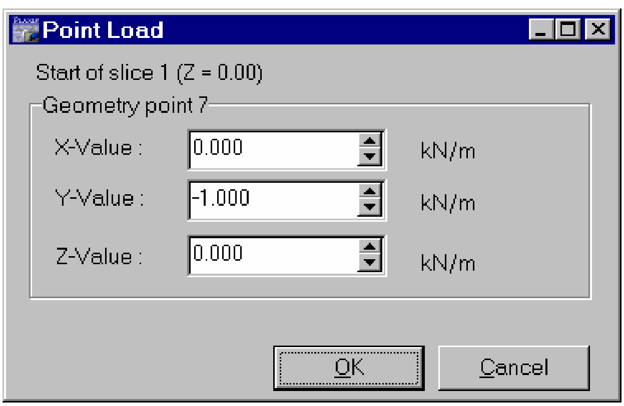
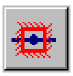


Figure 3.19 Input window for point loads

Point loads appear as point forces in the cross-section model, but in the full 3D model they can be used both as point loads on individual vertical cross-sections (*z*-planes) as well as line loads on volume sections in *z*-direction (slices). In the former case, the input value should be regarded as a force, whereas in the latter case the input value should be regarded as a force per unit of length. Although the global input values of point loads can be specified in the cross-section model, the precise distribution and activation or deactivation of the loads over the various planes and slices may be changed in the framework of *Staged construction*. Moreover, an existing composition of loads may be increased globally by means of the load multipliers *MloadA* (or *ΣMloadA*) for load system A and *MloadB* (or *ΣMloadB*) for load system B (Section 4.7.1).

On a part of the geometry where both prescribed displacements and point loads are applied, the prescribed displacements have priority over the point loads during the calculations, provided that the prescribed displacements are active. Hence, it is not useful to apply point loads on a line with fully prescribed displacements. When only one displacement direction is prescribed whilst the other direction is free, it is possible to apply a point load in the free direction.

3.4.6 ROTATION FIXITIES



Rotations fixities are used to fix the rotational degree of freedom of a plate around the z-axis. After selection of the option *Rotation fixities* from the *Loads* sub-menu or by clicking on the corresponding button in the toolbar, the geometry point (s) should be entered (using the mouse) where the rotation fixity is to be applied. This can only be done on plates, but not necessarily on existing geometry points. If a point in the middle of a plate is selected, a new geometry point will be introduced.

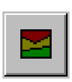
Existing rotation fixities can be eliminated by selecting the rotation fixity in the cross-section model and pressing the <Del> key on the keyboard.

When using the standard fixities option, plates that extend to the vertical boundaries automatically obtain a rotation fixity (around the z-axis). Plates that extend to the front plane or the rear plane in the full 3D model obtain a rotation fixity around the x-axis and a rotation fixity around the y-axis at these planes.

3.5 MATERIAL PROPERTIES

In PLAXIS, soil properties and material properties of structures are stored in material data sets. There are four different types of material sets: Data sets for soil & interfaces, plates, geogrids and anchors. 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 cross-section 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 window. When doing so, the window will be extended to the one as presented in Figure 3.20.

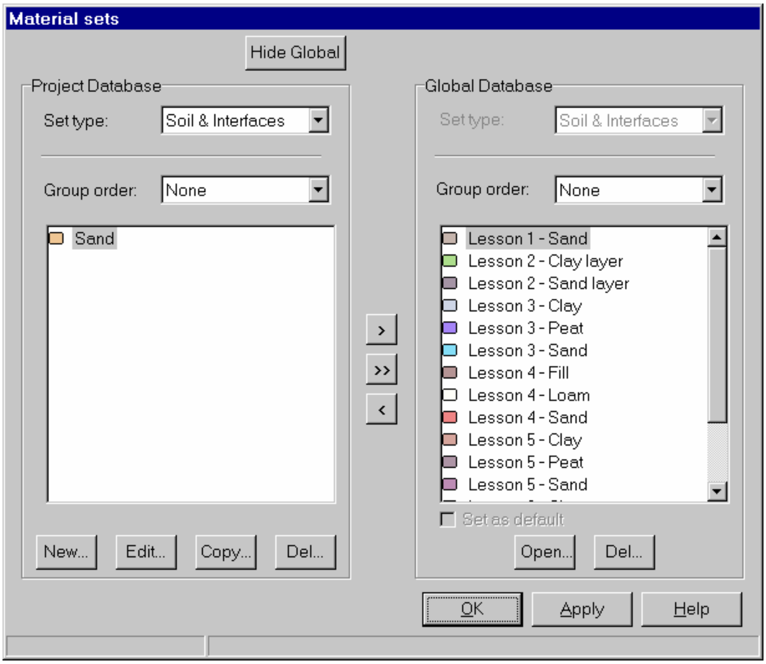


Figure 3.20 Material sets window showing the project and the global database

At both sides of the window (*Project data base* and *Global data base*) there are two list boxes and a tree view. From the list 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*, *Plates*, *Geogrids*, *Anchors*). 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* list 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 data base 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 data base files for soil and interfaces. Similarly, the global databases for plates (or beams), geogrids (or geotextiles) and anchors are contained in the files 'Beams.MDB', 'Geotex.MDB' and 'Anchors.MDB' respectively. These compatible PLAXIS files are stored in the DB sub-directory of the 3D Tunnel 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*.

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 elastic perfectl*y*-plastic model requires five basic input parameters, namely a Young's modulus, E, a Poisson's ratio, ν, a cohesion, c, a friction angle, φ, and a dilatancy angle, ψ. 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.

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 3.21). The material has been compressed isotropically up to some confining stress σ_3. After this, the axial pressure σ_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 3.21a. The increase in the volume (or volumetric strain) is typical for sands and is also frequently observed for rocks. Figure 3.21b 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 ψ is needed to model the irreversible increase in volume.

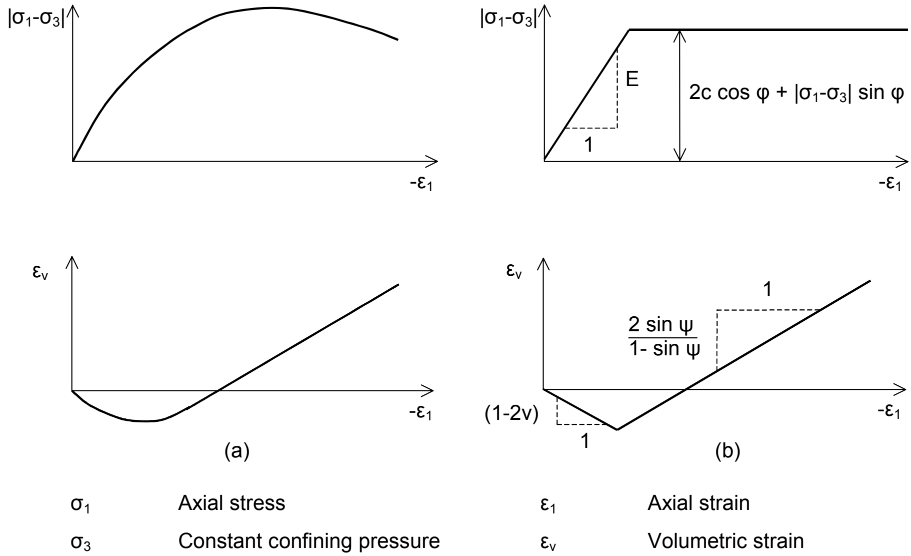


Figure 3.21 Results from standard drained triaxial tests (a) and elastic-plastic model (b)

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. 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 cluster(s) in the cross-section model. 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. This is indicated in the interface properties window as *<Cluster material>*.

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 data base 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 data base 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. 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.

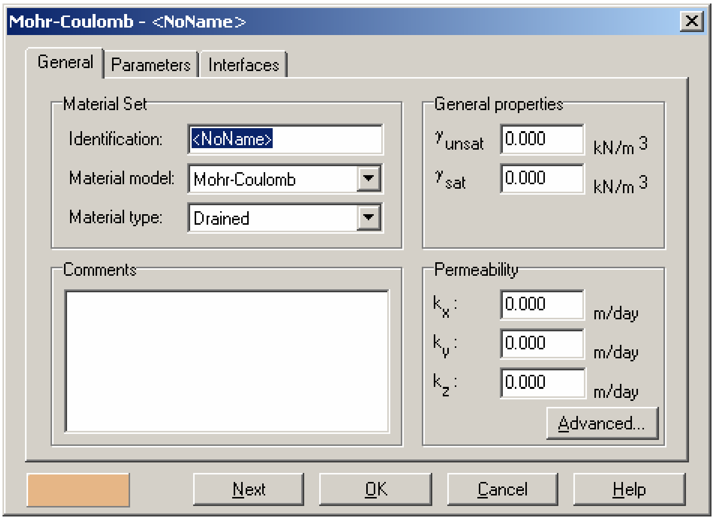


Figure 3.22 Soil and Interface material set window (*General* tab sheet)

Material model

PLAXIS supports various 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, ν.

The linear elastic model is very 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, ν, the cohesion, c, the friction angle, φ, and the dilatancy angle, ψ.

Jointed Rock model:

This is an anisotropic elastic-plastic model where plastic shearing can only occur in a limited number of shearing directions. This model can be used to simulate the behaviour of stratified or jointed rock.

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 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', ν', c', φ' and not E_u, ν_u, c_u (s_u), φ_u. In addition to the stiffness and strength of the soil skeleton, PLAXIS automatically adds a bulk stiffness for the water and distinguishes between effective stresses and excess pore pressures:

$$\text{Effective stress:} \qquad \Delta p' = K'\Delta\varepsilon_v$$

$$\text{Excess pore pressure :} \qquad \Delta p_w = \frac{K_w}{n} \Delta \varepsilon_v$$

Here $\Delta p'$ is an increment of the effective mean stress, n is the porosity of the soil, K_w is the bulk modulus of the pore fluid and $\Delta\varepsilon_v$ is an increment of volumetric strain. For the bulk modulus of the soil skeleton the theory of elasticity yields the well-known expression:

$$K' = \frac{E'}{3(1-2\nu')}$$

PLAXIS does not use a high 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 based on the above formula, assuming an undrained Poisson's ratio of 0.495. This results in a relatively low bulk modulus of water, K_w, namely:

$$\frac{K_w}{n} \approx 100\ G \qquad \text{where} \qquad G = \frac{E'}{2(1+\nu')}$$

Hence, the pore water is taken to be slightly compressible. 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. For undrained material behaviour the effective Poisson's ratio 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.

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.

The *Non-porous* material type may also be applied to interfaces. To completely block the flow through sheet pile walls or through other impervious structures, the surrounding interfaces may have a separate material data set in which the material type is set to *Non-porous*.

Saturated and unsaturated weight (γ_{sat} and γ_{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 γ_{unsat} applies to all material above the phreatic level and the saturated weight γ_{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 γ_{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 pore pressures above the phreatic level are always set equal to zero. In this way tensile capillary stresses are disregarded.

Weights are activated by means of the *ΣMweight* parameter in the initial stress generation (*K_0-procedure*) (Section 3.9.3) or by means of *Gravity loading* in the Calculation program.

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 and groundwater flow 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 TUNNEL allows for the anisotropic permeability of soils where the anisotropy directions coincide with the principal axis, 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 3.23.

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 $\Delta\varepsilon_v$. 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, option may be de-selected in the advanced general properties window.

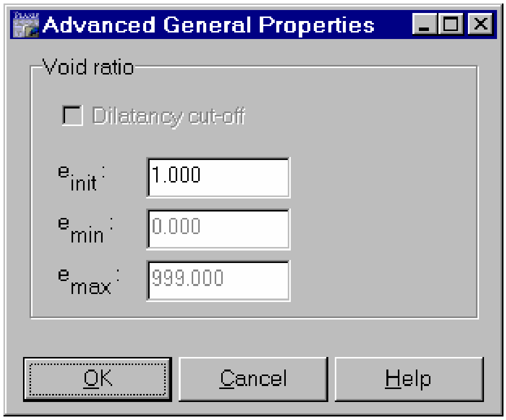


Figure 3.23 Advanced general properties window

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.

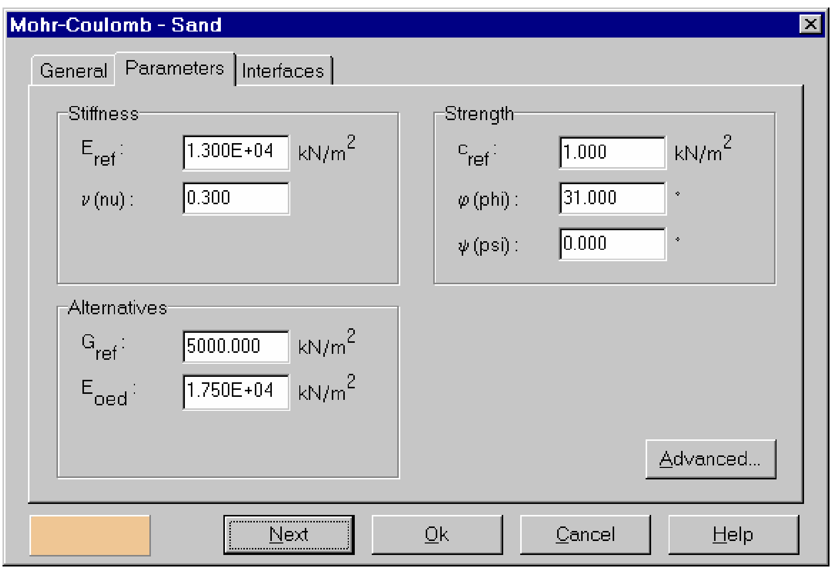


Figure 3.24 Soil and Interface material set window (*Parameters* tab sheet of the Mohr-Coulomb model)

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 is usually indicated as E_0 and the secant modulus at 50% strength is denoted as E_{50} (Figure 3.25). 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}.

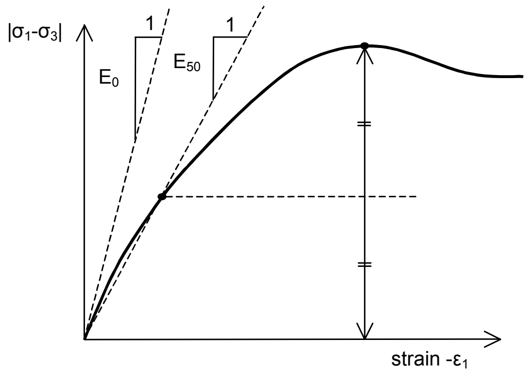


Figure 3.25 Definition of E_0 and E_{50}

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. Hence, when using a constant stiffness modulus to represent soil behaviour one should choose a value that is consistent with the 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 parameters*).

Poisson's ratio (ν)

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 (ν_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 (increasing *ΣMweight* from 0 to 1 in a plastic calculation). For this type of loading PLAXIS should give realistic ratios of $K_0 = \sigma_h / \sigma_v$. As both models 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, ν is evaluated by matching K_0. This subject is treated more extensively in Appendix A, which deals with initial stress distributions. In many cases one will obtain ν 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.

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, ν:

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

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. PLAXIS can handle cohesionless sands ($c = 0$), but some options will not perform well. To avoid complications, non-experienced users are advised to enter at least a small value (use $c > 0.2$ kPa). PLAXIS offers a special option for the input of layers in which the cohesion increases with depth (see *Advanced parameters*).

Friction angle (φ)

The friction angle, φ (phi), is entered in degrees. 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 3.26 by means of Mohr's stress circles.

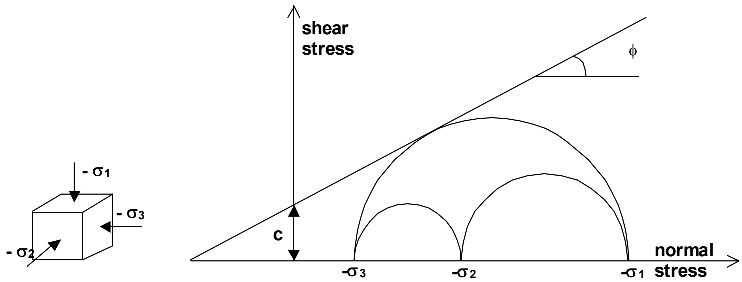


Figure 3.26 Stress circles at yield; one touches Coulomb's envelope

A more general representation of the yield criterion is shown in Figure 3.27. The Mohr-Coulomb failure criterion proves to be better for describing soil behaviour than the Drucker-Prager approximation, as the latter failure surface tends to be inaccurate for axisymmetric configurations.

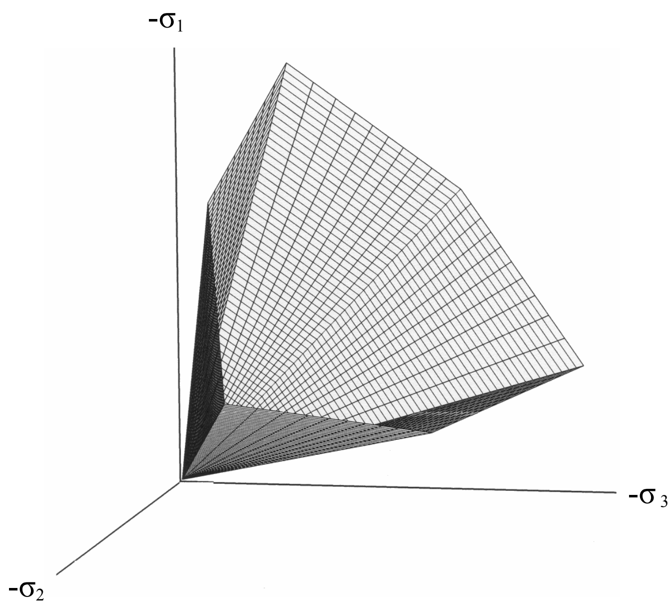


Figure 3.27 Failure surface in principal stress space for cohesionless soil

Dilatancy angle (ψ)

The dilatancy angle, ψ (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. For quartz sands the order of magnitude is $\psi \approx \varphi - 30°$. In most cases, however, the angle of dilatancy is zero for φ-

values of less than 30°. A small negative value for ψ is only realistic for extremely loose sands. For further information about the link between the friction angle and dilatancy, see Reference 3.

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 as shown in Figure 3.28. The advanced features comprise the increase of stiffness and cohesive strength with depth and the use of a tension cut-off. In fact, the latter option is used by default, but it may be deactivated here, if desired.

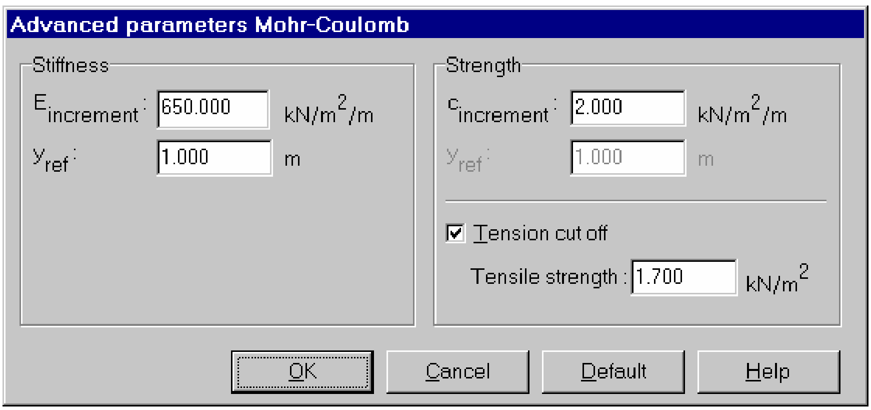


Figure 3.28 Advanced Mohr-Coulomb parameters window

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 the input of 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 $c_{increment}$.

Tension cut-off:

In some practical problems, an area with tensile stresses may develop. According to the Coulomb envelope shown in Figure 3.26 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 3D Tunnel 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.

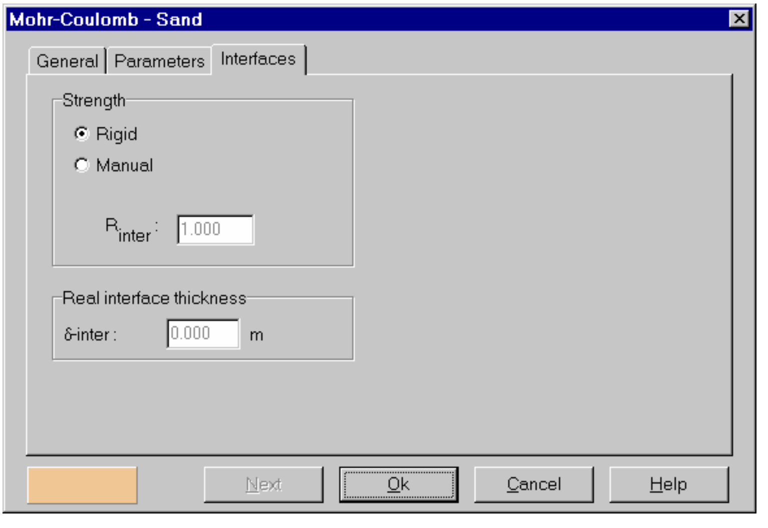


Figure 3.29 Soil and Interface material set window (*Interfaces* tab sheet)

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 τ is given by:

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

where

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

where τ_{s1} and τ_{s2} are shear stresses in the two (perpendicular) shear directions and σ_n is the effective normal stress.

For plastic behaviour τ is given by:

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

where φ_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:

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

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

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

In addition to Coulomb's shear stress criterion, the tension cut-off criterion, as described before, also applies to interfaces (if not deactivated):

$$\sigma_n < \sigma_{t,i} = R_{inter}\, \sigma_{t,soil}$$

where $\sigma_{t,soil}$ is the tensile strength of the soil.

The interface strength can be set using the following options:

Rigid:

This option is used when the interface should not influence the strength of the surrounding soil. For example, extended interfaces around corners of structural objects (Figure 3.13) are not intended for soil-structure interaction and should not have reduced strength properties. 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 ψ_i, are the same as the soil properties in the data set, except for Poisson's ratio ν_i.

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:

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

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

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 cross-section model. K_n is the interface elastic normal stiffness and K_s is the interface elastic shear stiffness. The shear and compression moduli are related by the expressions:

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

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

$$\nu_i = 0.45$$

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. The user may change the virtual thickness. This can be done in the properties window that appears after double clicking an interface (Section 3.3.4).

Real interface thickness (δ_{inter})

The real interface thickness, δ_{inter}, is a parameter that represents the real thickness of a shear zone between a structure and the soil. The value of δ_{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.

3.5.3 MATERIAL DATA SETS FOR PLATES

Plates are used to model the behaviour of slender walls, plates or thin shells. Distinction can be made between elastic and elastoplastic behaviour.

Stiffness properties

For elastic behaviour an axial stiffness, *EA*, and a flexural rigidity, *EI*, should be specified as material properties. The plate is homogeneous, in the sense that the axial stiffness is similar for all in-plane directions and the flexural rigidity is similar for all types of out-of-plane bending. The values of *EA* and *EI* relate to stiffness per unit width. Hence, the axial stiffness, *EA*, is given in force per unit width and the flexural rigidity, *EI*, is given in force length squared per unit width. From the ratio of *EI* and *EA* an equivalent thickness for an equivalent plate (d_{eq}) is automatically calculated from the equation:

$$d_{eq} = \sqrt{12 \frac{EI}{EA}}$$

For the modelling of plates, PLAXIS uses the Mindlin beam theory as described in Reference 2. This means that in addition to out-of-plane bending, shear deformation is taken into account. The shear stiffness of the plate is determined from:

$$\text{Shear stiffness} \quad = \quad \frac{5EA}{12\left(1+\nu\right)} = \frac{5E\left(d_{eq} \bullet 1\,m\right)}{12\left(1+\nu\right)}$$

This implies that the shear stiffness is determined from the assumption that the plate has a rectangular cross-section. In the case of modelling a solid wall, this will give the correct shear deformation. However, in the case of steel profile elements, like sheet-pile walls, the computed shear deformation may be too large. You can check this by judging the value of d_{eq}. For steel profile elements, d_{eq} should be at least of the order of a factor 10 times smaller than the length of the plate to ensure negligible shear deformations.

Poisson's ratio

In addition to the above stiffness parameters, a Poisson's ratio, ν, is required. For thin structures with a certain profile (like sheet-pile walls), it is advisable to set Poisson's ratio to zero. For real massive structures (like concrete walls) it is more realistic to enter a true Poisson's ratio of the order of 0.15.

If the plate is fixed in axial or longitudinal direction (e.g. when the plate goes from the front *z*-plane to the rear *z*-plane, which are fixed in the *z*-direction), the value of Poisson's ratio will influence the flexural rigidity of the plate as follows:

$$\text{Input value of flexural rigidity} \qquad EI$$

$$\text{Observed value of flexural rigidity} \qquad \frac{EI}{1-\nu^2}$$

The stiffening effect of Poisson's ratio is, in that case, caused by the stresses in the *z*-direction (σ_{zz}) and the fact that strains are prevented in *z*-direction.

Weight

In a material set for plates a specific weight can be specified, which is entered as a force per unit area. For relatively massive structures this force is, in principle, obtained by multiplying the unit weight of the plate material by the thickness of the plate. Note that in a finite element model, plates are superimposed on a continuum and therefore 'overlap' the soil. To calculate accurately the total weight of soil and structures in the model, the unit weight of the soil should be subtracted from the unit weight of the plate material. The manufacturer generally provides the specific weight (force per unit area) for sheet-pile walls.

The weight of plates is activated together with the soil weight by means of the *ΣMweight* parameter.

Strength parameters (plasticity)

Plasticity may be taken into account by specifying a maximum bending moment, M_p. The maximum bending moment is given in units of force times length per unit width. In addition to the maximum bending moment, the axial force is limited to N_p. The maximum axial force, N_p, is specified in units of force per unit width. The maximum axial force is calculated from the maximum bending moment, based on the implicit plate profile. This value cannot be changed independently by the user. The relationship between M_p and N_p is visualised in Figure 3.30. The diamond shape represents the ultimate combination of forces for which plasticity will occur. Force combinations inside the diamond will result in elastic deformations only. The Scientific Manual describes in more detail how PLAXIS 3D Tunnel deals with plasticity in plates. By default the maximum moment is set to $1 \cdot 10^{15}$ units if the material type is set to elastic (the default setting).

Bending moments and axial forces are calculated at the stress points of the plate elements (Figure 3.8). If M_p or N_p is exceeded, stresses are redistributed according to the theory of plasticity, so that the maxima are complied with. This will result in irreversible deformations. Output of bending moments and axial forces is given in the nodes, which requires extrapolation of the values at the stress points. Due to the position of the stress points in a plate element, it is possible that the nodal values of the bending moment may slightly exceed M_p.

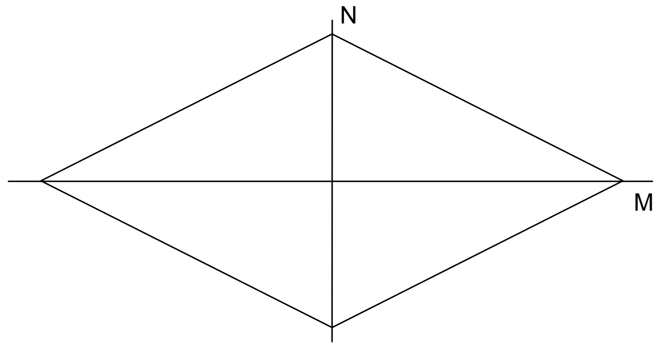


Figure 3.30 Combinations of maximum bending moment and axial force

It is possible to change the material data set of a plate in the framework of staged construction. However, it is very important that the ratio of *EI* / *EA* is not changed, since this will introduce an out-of-balance force (Section 3.3.2).

3.5.4 MATERIAL DATA SETS FOR GEOGRIDS

Geogrids are flexible elements that represent a grid or sheet of fabric. Distinction can be made between elastic and elastoplastic behaviour. Geogrids cannot sustain compressive forces. The only property in a geogrid data set is the elastic axial stiffness, *EA*, entered in units of force per unit width. The axial stiffness, *EA*, is usually provided by the geogrid manufacturer and can be determined from diagrams in which the elongation of the geogrid is plotted against the applied force in a longitudinal direction. The axial stiffness is the ratio of the force (in kN) and the elongation (in m). For elastoplastic behaviour the maximum tensile force, N_p , can be specified in force per unit width.

3.5.5 MATERIAL DATA SETS FOR ANCHORS

A material data set for anchors may contain the properties of node-to-node anchors as well as fixed-end anchors. In both cases the anchor is just a spring element. The major anchor property is the axial stiffness, *EA*, entered in the unit of force. If the material type is selected as elastoplastic, a maximum anchor force, F_{max}, can be entered in the unit of force. If the material type is set to elastic (the default setting), the maximum force is set to $1 \cdot 10^{15}$ units.

Anchors can be pre-stressed in a *Staged construction* calculation. In such a calculation the pre-stress force for a certain calculation phase can directly be given in the anchor property window. The pre-stress force is not considered to be a material property and is therefore not included in an anchor data set.

3.5.6 ASSIGNING DATA SETS TO GEOMETRY COMPONENTS

After creating all material data sets for the various soil layers and structures, the data sets must be assigned to the corresponding components. This can be done in different ways.

The first method is based on an opened material sets window, showing the created material sets in the project data base 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. Note that material sets cannot be dragged directly from the global data base tree view.

The second method is to double click the desired component. 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 required material set can be selected. The material set can be dragged from the project data base tree view and dropped on the properties window. Alternatively, after the selection of the required material set it can be assigned to the selected geometry component by clicking on the <Apply> button in the material sets window. In this case, the material sets window remains opened. When clicking on the <OK> button instead, the material set is also assigned to the selected geometry component and the material sets window is subsequently closed.

The third method is to move the cursor to a geometry component and to click the right hand mouse button. Through the cursor menu (*properties*) one can select the desired geometry component. As a result, the properties window appears. From here the selection of the proper material set is the same as for the second method.

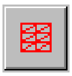
After assigning a material data set to a soil cluster, the cluster obtains the colour of the corresponding data set. By default, the colours of data sets have a low intensity. To increase the intensity of all data set colours, the user may press <Ctrl><Alt><C> simultaneously on the keyboard. There are three levels of colour intensity that can be selected in this way.

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 cross-section model is fully defined and material properties are appointed to all clusters and structural objects, a 2D mesh must be generated first. The 2D mesh should be made fully satisfactory (including global and local refinements; see Section 3.6.1 to 3.6.5) before proceeding to the 3D mesh extension. It is advisable to avoid fine meshes, since this will lead to excessive calculation times.

The basic volume elements of the 3D finite element mesh are the 15-node wedge elements, as described in Section 3.2.2. These elements are generated from the 6-node triangular elements as generated in the 2D mesh. In addition to the basic volume elements, there are special elements for structural behaviour (plates, geogrids and anchors), as described in Section 3.3.2 to 3.3.7. PLAXIS allows for a fully automatic generation of 2D finite element meshes and a semi-automatic generation of 3D meshes.

3.6.1 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 mesh generator is a cross-section model composed of points, lines and clusters, of which the clusters (areas enclosed by lines) are automatically generated during the creation of the cross-section model. Geometry lines and points may also be used to influence the position and distribution of elements.

The generation of the 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.

After the 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 and interfaces. This so-called *Connectivity plot* is also available as a regular output option (Section 5.9.5). 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.

3.6.2 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 cross-section dimensions (x_{min}, x_{max}, y_{min}, y_{max}) and a *Global coarseness* setting as defined in the *Mesh* sub-menu:

1 Ingenieursbureau Sepra, Park Nabij 3, 2267 AX Leidschendam (NL)

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

Distinction is made between five levels of global coarseness: *Very coarse*, *Coarse*, *Medium*, *Fine*, *Very fine*. By default, the global coarseness is set to *Very Coarse*. The average 2D element size and the number of generated triangular elements 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 exact geometry and eventual local refinement settings.

3.6.3 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.

3.6.4 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 cross-section model includes edges or corners or 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.

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

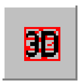
3.6.5 LOCAL REFINEMENT

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 around 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 , 5.0]. After selecting one of the local refinement options, the 2D mesh is automatically regenerated.

3.6.6 3D MESH GENERATION



After generation of the 2D mesh, the mesh must be extended to a fully 3D mesh. This can be done by clicking on the 3D mesh generation button or selecting the corresponding option from the *Mesh* sub-menu. As a result, a new window will appear in which the positions (*z*-coordinates) of particular cross-section planes (*z*-planes) can be specified (Figure 3.31). The previously generated 2D mesh is repeated at each *z*-plane. The 3D mesh is created by connecting the corners of the 2D triangular elements to the corresponding points of the corresponding elements in the next *z*-plane. In this way, a 3D mesh composed of 15-node wedge elements is created.

The *z*-planes are not only used to generate 3D elements, but also to distinguish between parts of the geometry in *z*-direction. Hence, these planes divide the geometry in slices. The 2D clusters in the *z*-planes and the slices define three-dimensional clusters of elements, i.e. volumes that can be activated or deactivated simultaneously. All operations in the framework of Staged Construction are defined by *z*-planes and slices. Hence, a *z*-plane is needed wherever a discontinuity in the geometry or in the loading occurs or should occur in the initial situation or in the construction process. At least 2 planes (corresponding to one slice) are required for a 3D mesh. If necessary, slices are automatically divided by sub-planes, so that the size of the elements in *z*-direction is about equal to the average element size as defined for the 2D mesh generation.

The table in Figure 3.31 gives the *z*-coordinates of each *z*-plane. The positive *z*-direction points towards the user, as indicated in the top view at the right hand side of the window. The user's viewpoint is shown at the bottom. The top view shows the active plane in red.

The original cross-section model is taken at z = 0. New planes can be created by using the <Add> button or the <Insert> button. When using the <Add> button, a new plane is created in the positive *z*-direction, i.e. in the direction of the user. By default, the *z*-

coordinate of the new plane is chosen such that the thickness of the section between the new plane and the previous plane is about equal to the average element size, but this value may be changed by the user, provided that its *z*-coordinate is larger than the previous plane. In fact, the numbering of planes starts with the highest *z*-coordinate and increases with decreasing *z*-coordinate. When using the <Insert> button, a new plane is created half way between the active plane and the next plane, but the *z*-coordinate may be changed by the user, provided that it remains between the two planes. If the <Insert> button is pressed when the last plane (z=0) is active, a new plane is introduced with a negative *z*-coordinate. Existing planes may be deleted by using the <Delete> button.

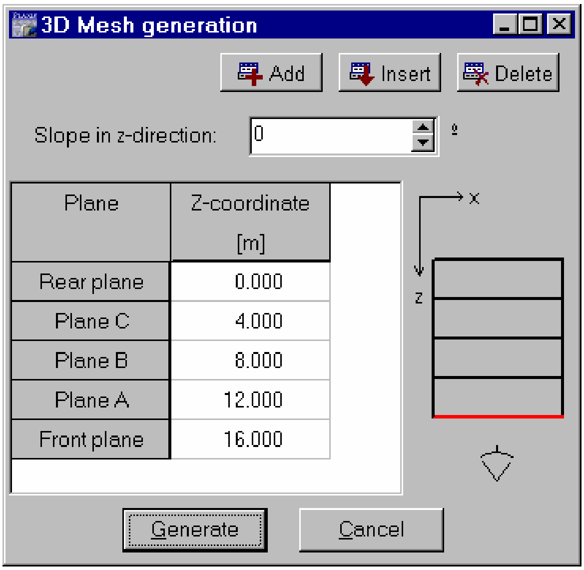


Figure 3.31 3D mesh generation window.

In principle, the size of volume elements in the *z*-direction is determined by the distance between two successive planes. However, if the distance between two successive planes is significantly larger than the average element size, as defined for the 2D mesh, the 3D mesh generator will automatically generate sub-slices to avoid badly shaped elements. There is, however, a way to create elements with larger or smaller size in *z*-direction by assigning a local element size factor to a plane. This can be achieved by clicking on the active plane (horizontal red line) in the top view. As a result, a new window appears in which the local element size factor can be entered. This local element size factor determines the position and number of sub-planes and thus the size of the 3D elements in the *z*-direction in the 3D mesh generation. The 2D mesh is not influenced by this local element size factor.

By default, all planes obtain the same *x*- and *y*-coordinates as defined by the cross-section model in the geometry creation mode. Hence, a default 3D extension results in a 'horizontal' 3D model. However, it is possible to increase or decrease the vertical position of planes in *z*-direction. This can be done by specifying a value for the *Slope in z-direction* parameter, which represents the positive inclination angle (in degrees) of the full model in the negative *z*-direction. Note that all *z*-planes remain vertical. The vertical position of the front plane (i.e. the plane with the highest *z*-coordinate) is according to the cross-section model, but subsequent *z*-planes are shifted in *y*-direction according to the specified slope. A positive slope gives a positive shift in *y*-direction for successive planes (with decreasing *z*-coordinate). The input value of the slope must be in the range [-89, 89] degrees. However, care should be taken with large slopes, since the model may deviate from what the user may expect. For example, when a circular tunnel is created and a large slope is specified for the 3D mesh extension, the cross-section area perpendicular to the longitudinal direction of the tunnel becomes non-circular.

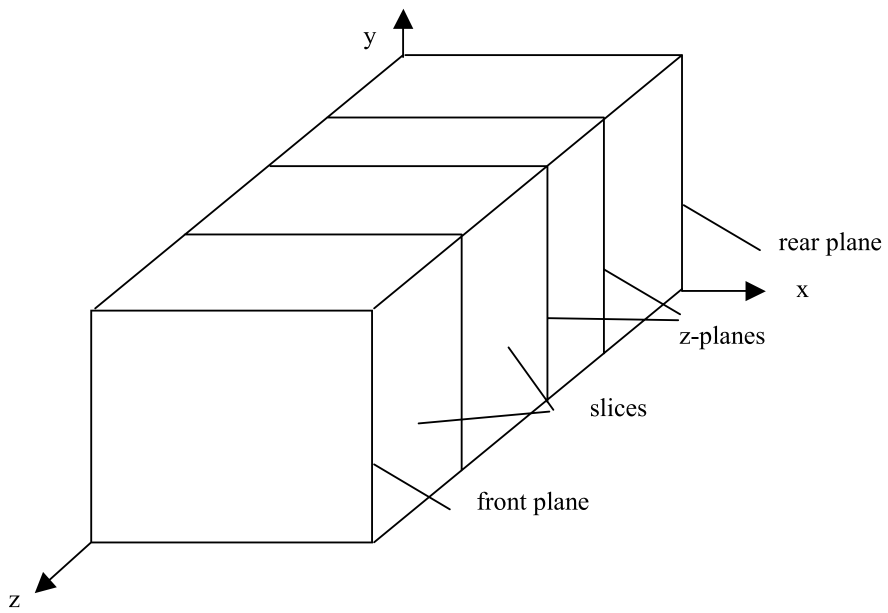


Figure 3.32 Definition of planes and slices.

After clicking the <Generate> button, the 3D mesh extension procedure is started and the 3D mesh is displayed in the Output window. The total number of 3D wedge elements is equal to the product of the number of 2D triangular elements in the 2D mesh and the total number of slices. The 3D mesh and its various *z*-planes can be viewed by moving through the tab sheets. The arrow keys of the keyboard allow the user to rotate the model so that it can be viewed from different angles (Figure 3.33).

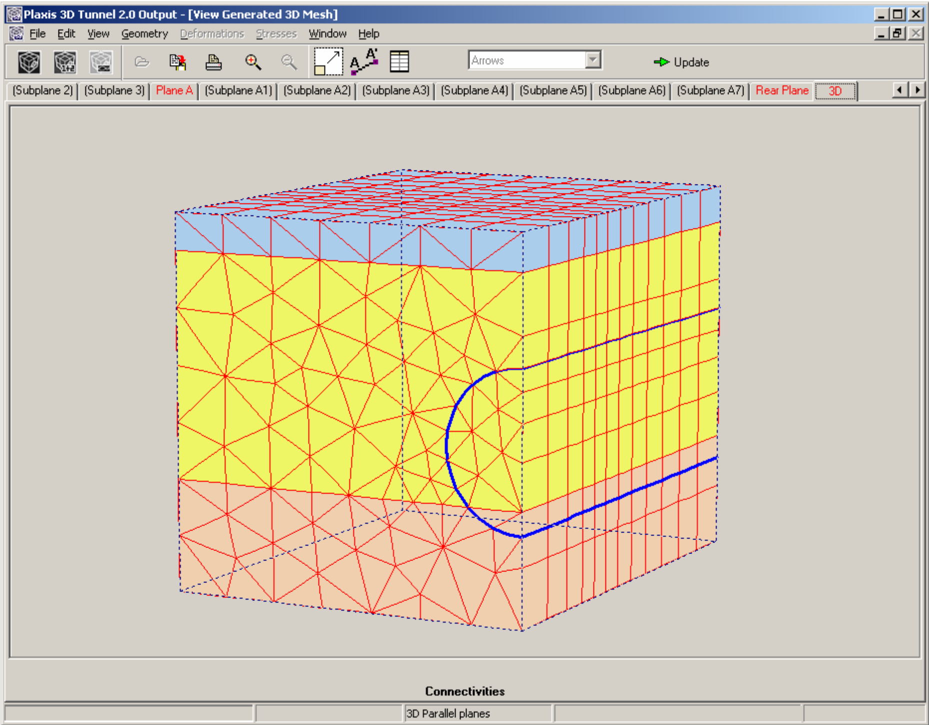


Figure 3.33 3D mesh (extension of the 2D mesh) with 15-node wedge elements.

3.6.7 ADVISED MESH GENERATION PRACTICE

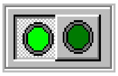
3D finite element calculations are very time-consuming. The time consumption highly depends on the number of elements used in the analysis. Moreover, when using a large number of 3D elements, the stiffness matrix may be too big to fit in memory. Hence, care should be taken when generating 3D finite element 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 Tunnel program must generally be coarser than meshes in the 2D PLAXIS version.

To perform efficient finite element calculations, a preliminary analysis can be performed using a very coarse mesh, starting with a very 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 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 that the front plane and the rear plane be created first. In addition, intermediate *z*-planes should be created at the required *z*-positions in the model. Finally, additional planes may be inserted where local refinement is desired, or, alternatively, the 3D local refinement option may be used.

3.7 INITIAL CONDITIONS



Once the 3D finite element mesh has been generated, the initial stress state and the initial configuration must be specified on the basis of the vertical cross-section model. This is done in the initial conditions part of the input program. The initial conditions consist of two different modes: One mode for the generation of initial water pressures (water conditions mode) and one mode for the specification of the initial geometry configuration and the generation of the initial effective stress field (geometry configuration mode).

Switching between these two modes is done by means of the 'switch' in the toolbar. The initial conditions allow for a return to the geometry creation mode, but this should generally not be done, since some information regarding the initial conditions will be lost.

3.8 WATER CONDITIONS

PLAXIS 3D TUNNEL is generally used for deformation analysis where the determination of accurate effective stresses is an important aspect. In such an analysis, a clear distinction is made between active pore pressures, p_{active}, and effective stresses, σ'. In the active pore pressures, a further distinction is made between steady-state pore pressures, p_{steady}, and excess pore pressures, p_{excess}:

$$p_{\text{active}} = p_{\text{steady}} + p_{\text{excess}}$$

Excess pore pressures are pore pressures that occur due to loading of clusters for which the type of material behaviour in the material data set is specified as *Undrained*. In a *Plastic* calculation, excess pore pressures can be created only in these *Undrained* clusters. A consolidation analysis may be used to calculate the time-dependent generation or dissipation of excess pore pressures. In this type of calculation the development of excess pore pressures is determined by the Permeability parameters rather than by the type of material behaviour.

Steady-state pore pressures are pore pressures that represent a stable hydraulic situation. Such a situation is obtained when external water conditions remain constant over a long period. To reach a steady-state, it is not necessary that pore pressures, by themselves, are in static equilibrium (i.e. a horizontal phreatic surface), since situations in which permanent groundwater flow or seepage occur may also lead to a stable state.

Steady-state pore pressures and external water pressures, (referred to as 'water pressures'), are generated in the water conditions mode. Water pressures can easily be generated on the basis of phreatic levels. Alternatively, water pressures may be generated by means of a steady-state groundwater flow calculation. The latter requires the input of boundary conditions on the groundwater head, which are taken, by default, from the general phreatic level.

The *Water conditions* mode may be skipped in projects that do not involve water pressures. In such a case, a general phreatic level is taken at the bottom of the geometry model and all pore pressures and external water pressures are taken to be zero.

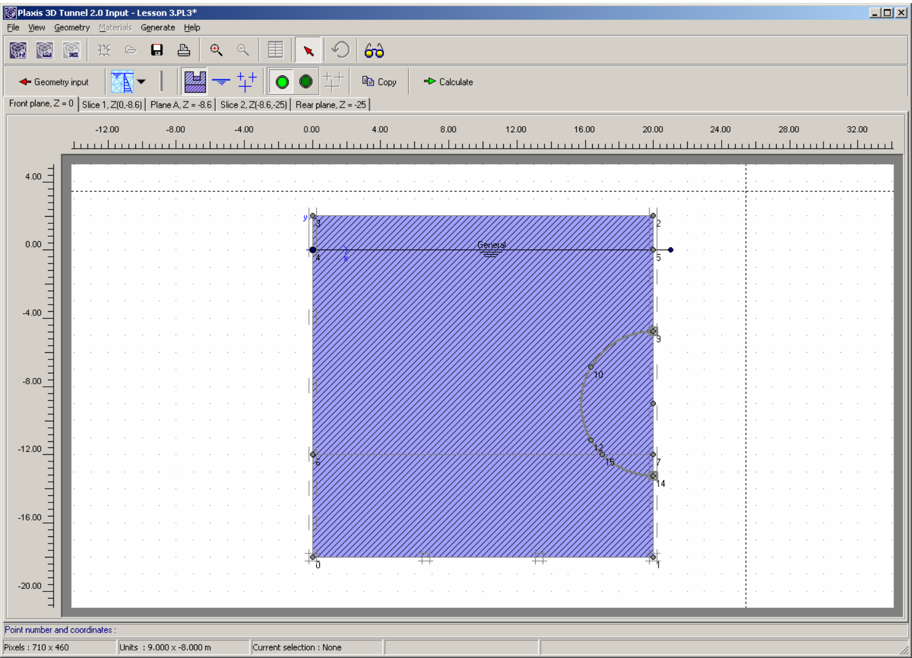


Figure 3.34 Initial conditions window (water conditions mode)

Water weight

In projects involving pore pressures, the unit weight of water, γ_w, is required. When you enter the *Water conditions* mode for the first time, a window is displayed in which you can enter the water weight. You can also enter the water weight by choosing the *Water weight* option from the *Geometry* submenu.

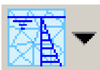
By default, the unit weight of water is set to 10 kN/m^3.

Cavitation cut-off

In the same window where the water weight is entered, it is also possible to activate a cavitation cut-off pressure. In case of unloading of undrained materials tensile excess pore stresses may be generated. These excess pore stresses might give rise to tensile active pore stresses. In case the cavitation cut-off option is activated, excess pore stresses are limited so that the tensile active pore stress is never larger than the cavitation cut-off stress.

By default, the cavitation cut-off option is not activated. If it is activated, the default cavitation cut-off stress is set to 100 kN/m^2.

3.8.1 SELECTION OF PORE PRESSURE GENERATION METHOD



Steady state pore pressures can be generated based either on phreatic levels or on groundwater flow. The selection between these two methods is made using the *Select pore pressure generation method* button on the toolbar. After a method is selected, the button changes its appearance to reflect the method chosen. This choice limits the options that can be choosen in planes and slices to those relevant for the selected method.

3.8.2 BY PHREATIC LEVEL

Phreatic levels

Pore pressures and external water pressures can be generated on the basis of phreatic levels. A phreatic level represents points where the water pressure is zero. Using the phreatic level as input, the generated water pressure will increase linearly with depth (i.e. the pressure variation is hydrostatic). Before entering a phreatic level, you must enter the correct unit weight of water by selecting the *water weight* option from the *Geometry* menu. You can choose the option to enter phreatic levels from the *Geometry* submenu or by clicking on the corresponding button in the toolbar. The input of a phreatic level is similar to the creation of a geometry line. You can only enter phreatic levels when a *Plane* tab is active. Note that a phreatic level specified for any plane applies to the whole z-range.

On a plane, phreatic levels are defined by two or more points. Points may be entered from 'left' to 'right' (increasing *x*-coordinate) or vice versa (decreasing *x*-coordinate). The points and lines are superimposed on the geometry model, but they do not interact with the model. Intersections of phreatic levels and existing geometry lines do not introduce additional geometry points.

If a phreatic level does not cover the full *x*-range of the geometry model, the phreatic level is considered to extend horizontally from the point furthest to the left to minus infinity and from the point furthest to the right to plus infinity. Above the phreatic level the pore pressures will be zero, whereas below the phreatic level there will be a hydrostatic pore pressure distribution, at least when the water pressure is generated on the basis of phreatic levels.

General phreatic level

For any *z*-plane, if none of the clusters is selected and a phreatic level is drawn, this phreatic level is considered to be the *General phreatic level*. By default, the general phreatic level is located at the bottom of the geometry model and applies to the full *z*-range of the model. When you enter a new line, the old general phreatic level is replaced. The general phreatic level can be used to generate a simple hydrostatic pore pressure distribution for the full geometry. By default, the general phreatic level is assigned to all clusters in the geometry.

If the general phreatic level is outside the geometry model and the corresponding boundary is a free boundary, external water pressures will be generated on the basis of this surface. This also applies to free boundaries that arise due to the excavation (de-activation) of soil clusters in the framework of Staged construction. The calculation program will treat external water pressures as distributed loads and they are taken into account together with the soil weight and the pore pressures as controlled by the *ΣMweight* parameter. The external water pressures are calculated in such a way that equilibrium of water pressures is achieved across the boundary.

Cluster phreatic level

To allow for a discontinuous hydrostatic pore pressure distribution, each cluster can be given a separate *Cluster phreatic level*. 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.

To enter a cluster phreatic level, first select the cluster for which you want to specify a separate water pressure level. Then choose the *Phreatic level* option on the toolbar or from the *Geometry* submenu and enter the level while the cluster remains selected. When you select multiple clusters at the same time (by holding down the *Shift* key) and entering a phreatic level, this line will be assigned to all selected clusters as a cluster phreatic level. The clusters for which no specific cluster phreatic level was entered retain the general phreatic level. To identify which phreatic level belongs to a particular cluster, you can select the cluster and see which phreatic level is indicated in red. If no phreatic level is indicated in red, then another option was chosen for that cluster (see below).

When you double-click a cluster in a plane (in the *Water conditions* mode), the *Cluster pore pressure distribution* window is displayed, in which you can activate or deactivate radio buttons to define how the pore pressures will be generated for that soil cluster (Figure 3.35). If a cluster phreatic level was assigned to the cluster by mistake, you can reset it to the general phreatic level by activating *General phreatic level* in this window. The cluster phreatic level is then deleted. You can also delete a cluster phreatic level by selecting the corresponding cluster and pressing the <Delete> key on the keyboard.

A cluster phreatic level is applied to all corresponding clusters over the full *z*-range of the model.

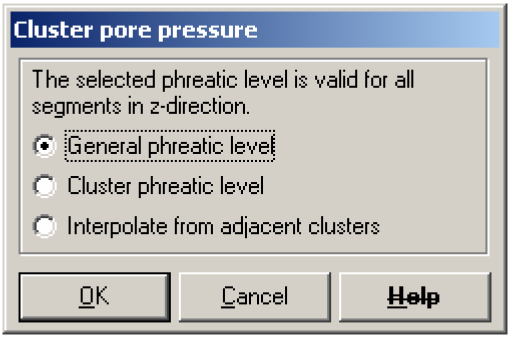


Figure 3.35 Cluster pore pressure distribution window

Interpolation of pore pressures from adjacent clusters

A third way to generate pore pressures in a soil cluster (in the full *z*-range) is to use the *Interpolate from adjacent clusters* option.

The *Interpolate from adjacent clusters* option can be useful if, for example, 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.

When you choose the *Interpolate from adjacent clusters* option, 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 (see below). In the latter case, the pore pressure is interpolated from the general phreatic level. You can use the *Interpolate from adjacent clusters* option repetitively in two or more successive clusters (on top of each other). If a starting value for the vertical interpolation of the pore pressure cannot be found, the starting point will be based on the general phreatic level.

The *Interpolate from adjacent clusters* option applies to all corresponding clusters over the full *z*-range of the model.

User-defined pore pressure distribution

If the pore pressure distribution in a particular soil cluster is very specific and cannot be defined using one of the above options, you may specify it as a user-defined pore pressure distribution. This option can be useful for specifying water pressure in front of a tunnel boring machine. This option is available for each volume cluster and can, in contrast to the previous options, be defined for each individual slice in the 3D model. The option can be selected by selecting the desired slice (in the water conditions mode) and double clicking the desired cluster. As a result the user-defined pore pressure distribution window appears in which the pore pressure situation can be defined (Figure 3.36)

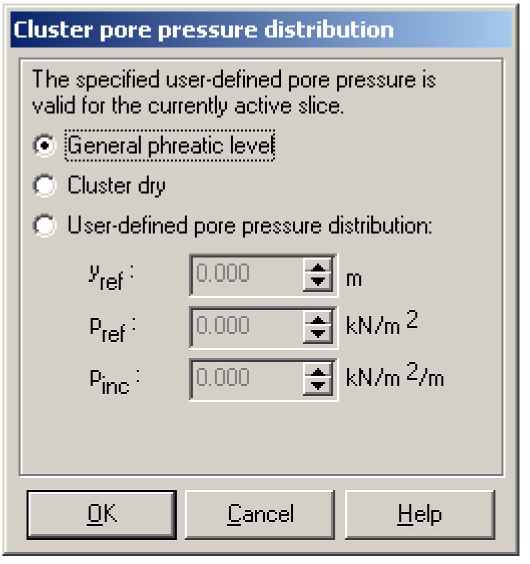


Figure 3.36 User-defined pore pressure distribution window

When using 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 as:

$$p = p_{ref} + \left(y - y_{ref}\right) p_{inc}$$

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}. 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 to interpolate pore pressures in other clusters. You should take this into account when using the *Interpolate pore pressures from adjacent clusters* option in the cluster above or below.

The *Cluster dry* option can be used to set pore pressures to zero for a specific cluster. The steady-state pore pressures in that cluster are then set to zero and the soil weight is considered to be the *unsaturated weight*. Note that clusters representing solid (concrete) structures where pore pressures should be excluded permanently (such as diaphragm walls or caissons) can be specified as *Non-porous* in the corresponding material data set. It is not necessary to set such non-porous clusters to *Cluster dry* in the *Water conditions* mode. Also note that in undrained clusters, excess pore pressures can still be generated whilst the *Cluster dry* option is being used.

Any user-defined pore pressure distribution applies to the selected volume cluster in the selected slice only.

Water pressures in inactive clusters

When generating water pressures on the basis of phreatic levels, and when some clusters are inactive in the initial geometry configuration, no distinction is made between active clusters and inactive clusters. This means that steady pore pressures are generated both for active clusters and inactive clusters according to the corresponding phreatic level. If you want to exclude water pressures in certain clusters, you should use the *Cluster dry* option or define a cluster phreatic level below the corresponding cluster.

3.8.3 GROUNDWATER FLOW

In cases where the total groundwater head difference in a soil structure is pronounced, it becomes necessary to calculate the groundwater flow. Groundwater flow calculations require the input of boundary conditions for the groundwater heads.

Prescribed Groundwater heads

At a specific z-plane, you can prescribe a groundwater head by double-clicking the geometry line where the boundary condition is needed. The *Groundwater head* window is then displayed, Figure 3.37. You can then enter the groundwater head at both end points of the geometry line. Note that if the head is specified, the program automatically calculates the corresponding pore pressure.

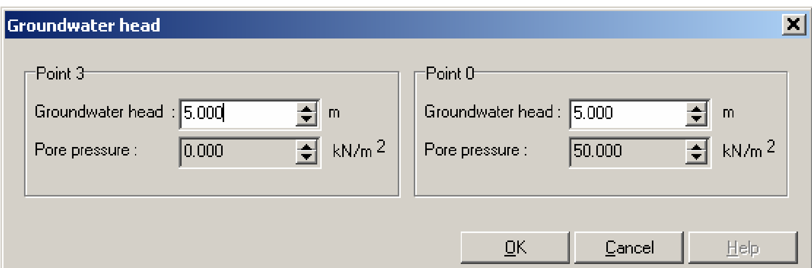


Figure 3.37 Groundwater head input window

You must follow this procedure at least for the front plane and the rear plane. The program will automatically calculate the heads at the intermediate nodes using linear interpolation between z-planes where no heads were defined.

You can remove a prescribed groundwater head by selecting the corresponding geometry line and pressing the <Delete> key on the keyboard.

If a groundwater head is prescribed at an outer geometry boundary, external water pressures will be generated for that boundary. The deformation analysis program will treat external water pressures as traction loads, which are taken into account together with the soil weight and the pore pressures.

External water table

In cases where the heads are constant along the z-direction, you can specify the heads by simply drawing the external water table using the same button as for the phreatic level. Figure 3.38 displays prescription of the heads using the phreatic level option. In the case of an excavation, the boundary of the excavation will be assigned automatically as a possible seepage surface. When pore pressures are generated by means of a groundwater flow calculation, the input phreatic level by itself does not have any meaning inside the geometry. In fact a more realistic phreatic level will result from the groundwater flow calculation.

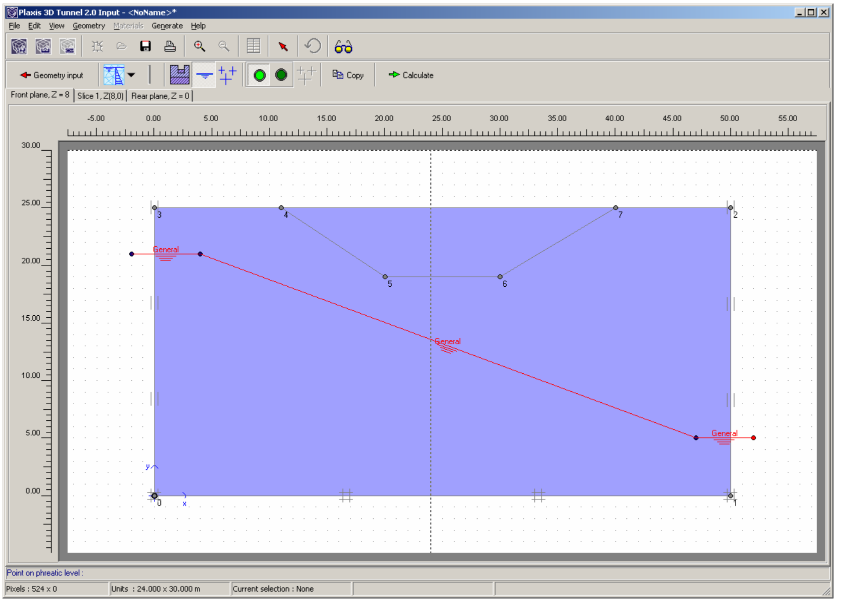


Figure 3.38 Example showing the use of an external water table for the input of groundwater head boundary conditions

In the case of a dry natural ditch as shown in Figure 3.39, the water table should not cross the boundaries of the ditch (as shown in the figure). Otherwise the program will specify a prescribed head along the ditch boundaries. However, you may prescribe heads at the ditch boundaries or specify them as closed.

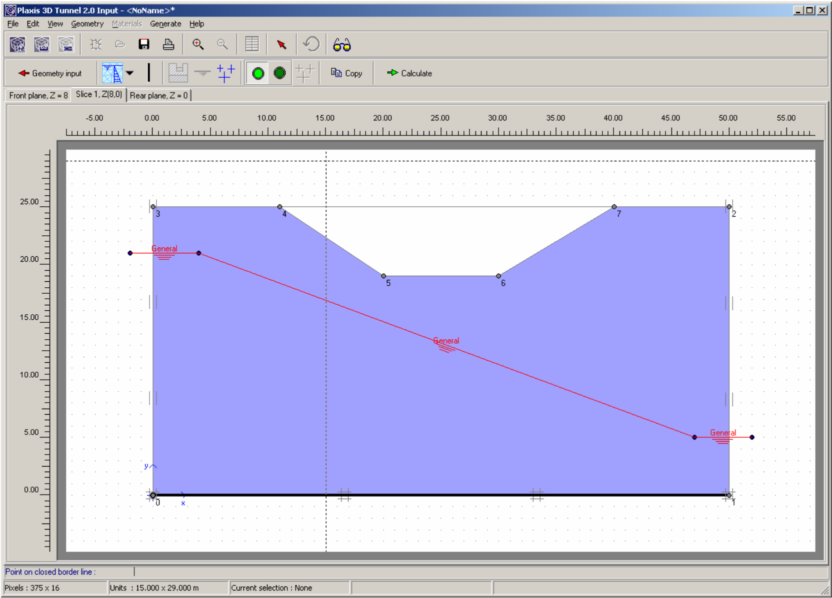


Figure 3.39 Example showing the right use of an external water table in the situation of a dry ditch

User-defined groundwater head in z-planes

In cases where a cluster in a *z*-plane acts as a boundary between active and inactive elements (i.e. between soil and excavation), boundary conditions may be specified for these clusters. This can be done by selecting the appropriate *z*-plane and double-clicking the desired cluster in that plane. As a result, the user-defined groundwater head window appears in which the option *User-defined groundwater head* can be selected (Figure 3.40).

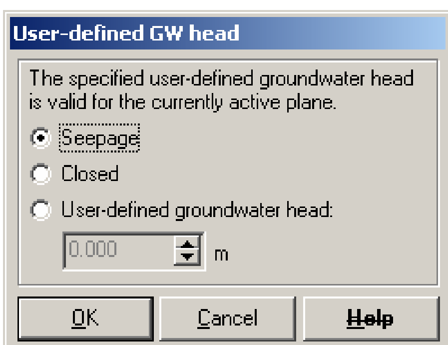


Figure 3.40 User-defined groundwater head input window

By default, the cluster in a z-plane is considered to be a seepage boundary. When selecting *user-defined groundwater head*, the groundwater head can be entered in units of length. Alternatively, the selected cluster can be set to *Closed*, to avoid groundwater flow through this plane.

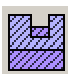
Closed flow boundary

A closed flow boundary is a geometry boundary where flow across this boundary does not occur. You can choose this option by clicking the *Closed flow boundary* button in the toolbar or by choosing the corresponding option from the *Conditions* submenu. This option is only available in *Slice* tab sheets.

The input of a closed flow boundary is similar to the creation of a geometry line. However, a closed flow boundary can only be placed over an existing geometry line at the outer boundary of the geometry model.

Closed flow boundaries can be specified for each slice individually. When entering closed flow boundaries for the first time at the first slice, they are automatically copied to all other slices.

Closed front and rear plane



Similar to closed flow boundaries defined on the outer boundaries of the model, the front and rear plane of the model can also be closed for groundwater flow. No groundwater flow occurs through closed front or rear planes.

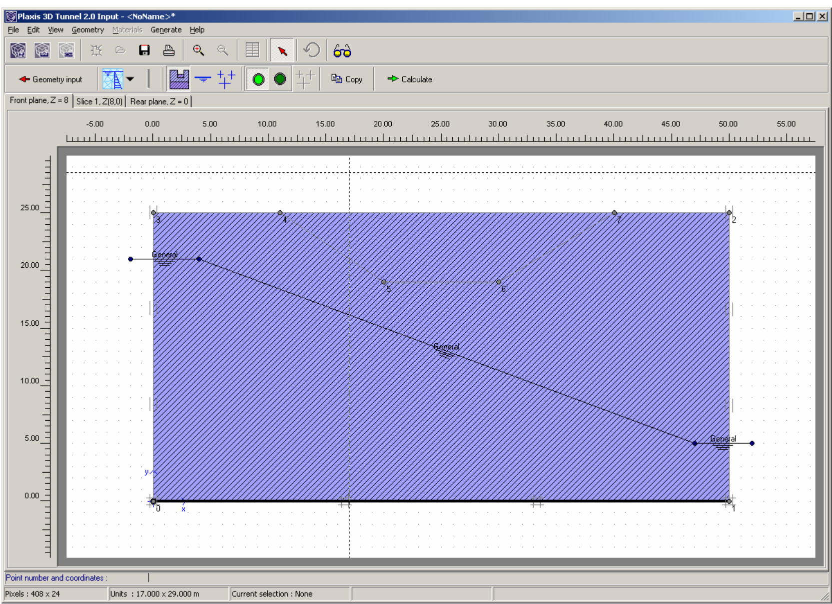


Figure 3.41 A closed front plane, indicated by black cross-hatching on the plane

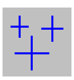
Both the front and the rear plane are closed by default. You can switch between the open or closed state using the *Click to open/close this plane* button on the toolbar. This option is available only in the front and rear plane. A closed plane is indicated by a black cross-hatch drawn on the plane.

Water pressures in inactive clusters

When you deactivate clusters in the *Geometry configuration* mode and perform a groundwater flow calculation, the inactive clusters are not included in the groundwater flow calculation itself. Instead, the pore pressure at stress points within the deactivated clusters is determined afterwards from the general phreatic level. Therefore, if inactive clusters are located (partly) below the water level, there will be a hydrostatic water pressure distribution below the water level, whereas the water pressure above the water level will be zero in these clusters.

The boundary between active and inactive clusters is considered to be a seepage boundary so that water can flow across such a boundary. In order to make such a boundary impermeable, it must be closed. To do this, add *interface elements* to the geometry and activate these elements in water conditions mode to make them impermeable (see Section 3.8.5).

3.8.4 WATER PRESSURE GENERATION



When you are specifying the phreatic levels or the hydraulic boundary conditions, you can easily calculate the pore pressure distribution. To do this, click on the *Generate water pressures* button (blue crosses) on the toolbar or choose the *Water pressures* option from the *Generate* submenu.

Generate by phreatic levels

Water pressure generation by *Phreatic levels* is based on the input of a general phreatic level, cluster phreatic levels and other options as described in Section 3.8.2.

When generating water pressures on the basis of phreatic levels, no distinction is made between active clusters and inactive clusters. This means that hydrostatic pore pressures are generated both for active and inactive clusters. If you want to exclude water pressures in certain clusters, use the *Cluster dry* option or define a cluster phreatic level below the cluster.

Generate by groundwater flow calculation

Water pressure generation by *Groundwater calculation* is based on the non-linear calculation of steady-state groundwater flow in confined and unconfined soil layers.

PLAXIS 3D TUNNEL includes an option for performing 3D groundwater flow calculations. When clusters have been de-activated in the *Geometry configuration* mode the inactive clusters are not included in the groundwater flow calculation itself. Instead, the pore pressure at stress points within the inactive clusters is determined afterwards from the *general phreatic level*. Therefore, if inactive clusters are located (partly) below the general phreatic level, there will be a hydrostatic water pressure distribution below the general phreatic level, whereas the water pressure above the general phreatic level is zero in these clusters.

When pore pressures are generated on the basis of a *Groundwater calculation*, you must select the settings for the control parameters of the iterative procedure. In general, the *Standard setting* can be used. For more details on groundwater flow calculations see Section 3.8.5.

When performing a 3D groundwater flow calculation, a window appears in which the groundwater flow calculation settings can be specified.

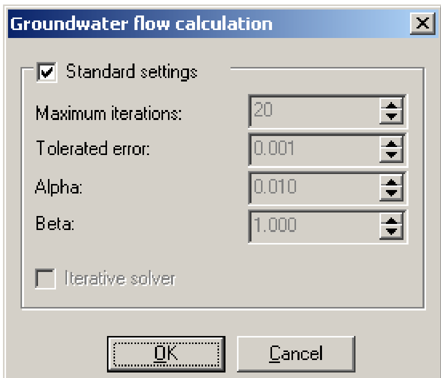


Figure 3.42 Groundwater flow calculation window

Results of water pressure generation

When you click *OK* in the *Water pressure generation* window, the water pressures are calculated according to one of the two options. After the generation of water pressures, the *Output* program starts and a plot of the water pressures and the general phreatic level is displayed. To return to the *Input* program, press the *Update* button.

Generated water pressures may be used as input data for deformation analysis. The water pressures are not active until they are actually applied in the calculation. Activation of water pressures is associated with activation of the soil weight using the *ΣMweight* parameter. In principle, stress points in elements with a zero steady pore pressure are considered to be unsaturated whereas stress points that have a non-zero steady pore pressure are considered to be saturated. Therefore, the value of the pore pressure determines whether the saturated soil weight (γ_{sat}) or the unsaturated soil weight (γ_{unsat}) is applied in a deformation analysis.

3.8.5 STEADY-STATE GROUNDWATER FLOW CALCULATION

In many practical problems, the pore pressures prevailing in a soil mass are not those associated with hydrostatic water conditions but with hydraulic boundary conditions – for example, in the presence of different water levels upstream and downstream across a dam. Such a difference in the water level (and therefore in the total heads) will cause seepage flow to occur from the upstream side to the downstream side. In this case, the resulting pore water pressures will have a distribution different to that generated due to hydrostatic conditions. As such, the effective stress distribution in the soil mass will also be different.

In view of this, PLAXIS 3D TUNNEL Version 2 includes a new feature for the calculation of pore water pressures due to steady-state groundwater flow. The calculation involves determining the total head distribution, the phreatic surface and the size of the seepage surface.

A steady-state groundwater flow calculation may be used for confined as well as unconfined flow problems. The confined flow problems result in a linear set of equations, which can be solved directly. The unconfined flow calculation involves determining the position of the free phreatic surface and the associated size of the seepage surface. This results in a system of non-linear equations, which requires an iterative solution procedure. In both cases, when performing a groundwater flow calculation in PLAXIS 3D TUNNEL, the iterative procedure for solving non-linear systems is used since it is not clear beforehand whether the flow is confined or unconfined. However, after the first iteration, the program will realize whether the problem is confined or not; two iterations will be sufficient for the confined problem.

Interface elements

When using interfaces in a groundwater flow calculation, the interfaces are fully impermeable by default. In this way interfaces may be used to block the flow perpendicular to the interface – for example to simulate the presence of an impermeable screen. Plates are fully permeable. In fact, it is only possible to simulate impermeable walls or plates when interface elements are included between the plate elements and the surrounding soil elements. On the other hand, if interfaces are present in the mesh, you may want to explicitly avoid any influence of the interface on the flow and the pore pressure distribution – for example, in interfaces around corner points of structures (Section 3.3.4). In such a case, the interface should be deactivated in the *Water conditions* mode. You can do this separately for a consolidation analysis and a groundwater flow calculation. For inactive interfaces, the head degrees-of-freedom of the interface node pairs are fully coupled, whereas for active interfaces these degrees-of-freedom are fully separated.

To conclude:

- An active interface is fully impermeable (separation of head degrees-of-freedom of node pairs).
- An inactive interface is fully permeable (coupling of head degrees-of-freedom of node pairs).

Seepage surfaces

Flow problems with a free water level may involve a seepage surface on the downstream boundary, as shown in Figure 3.43. A seepage surface occurs when the water level touches an open downstream boundary. The seepage surface is a line on which the groundwater head, h, equals the elevation head y (= vertical position). This condition arises from the fact that the water pressure is zero on the seepage surface, which is the same condition as that at the phreatic level.

If no specific condition is prescribed at a particular boundary line, the program assumes that this boundary is free and sets the possible seepage condition accordingly. The size of the seepage surface is calculated by the program using the iterative procedure for solving the non-linear system of equations.

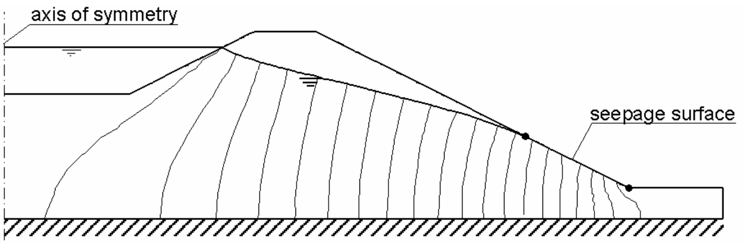


Figure 3.43 Flow through an embankment with indication of a seepage surface

Groundwater flow Calculation settings

In general, the implemented *Standard settings* may be used for groundwater flow calculations, which will normally lead to an acceptable solution. Alternatively, you may specify the control parameters manually.

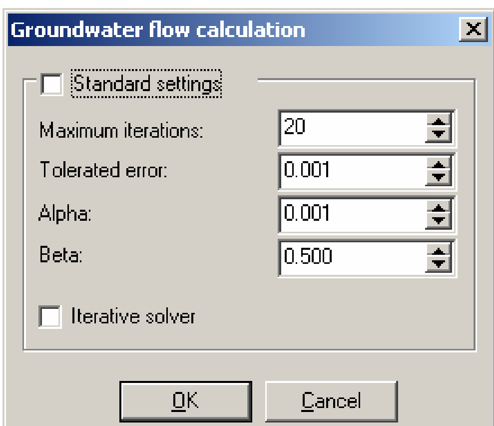


Figure 3.44 Groundwater flow calculation control parameters window

When you choose the *Manual settings* option in the *Water pressures generation* window and click the *Define* button, a new window appears in which the settings of the groundwater flow calculation control parameters are displayed in Figure 3.44. A description of these parameters is given below.

Tolerated error:

This parameter represents the tolerated global error in the total head distribution between two successive iterations. When using the standard setting, the *Tolerated error* is set to 0.001.

Residual relative permeability α :

This parameter specifies the residual value for the relative permeability k^r, which describes the relative magnitude of the coefficient of permeability in the dry region. In principle, α must be given zero value to prevent water from flowing in the dry zone (Figure 3.45). However, for numerical purposes, it is given a small value. In the *Standard* settings, α is assumed to be equal to 0.3. This value results, for the prism elements used in PLAXIS 3D TUNNEL with 6 Gauss points, in $K_e^d = 0.3^6 \, K_e^w = 0.0007 \, K_e^w$, in which K_e^d represents the element conductivity matrix in the dry region and K_e^w is its corresponding matrix in the wet region. Such a large difference in the conductivity matrix gives a rather good approximation for the simulation of a system consisting of wet and dry regions, and yet numerically stable.

Transition between wet and dry zones (unsaturated zone) β :

This parameter specifies the size of the transition zone between wet and dry regions, where k^r is varying linearly between 1 and α, (see Figure 3.45).

In essence, β represents the size of the unsaturated zone in a system, although the degree –of saturation is not taken into consideration in steady-state calculations. The variation of the degree –of saturation in the unsaturated zone will be considered in future developments when the transit groundwater flow is implemented.

In the *Standard* settings, β is defined as equal to 0.3 m. In principle, the program can calculate with any value of β. However, calculation with β less than 0.3 (depending on the size of the geometry, elements and boundary conditions) may require more iterations for convergence. For large geometries, with more than 10 m in all directions, β of more than 0.3 m can be used for possible reduction of the mesh size and the number of iterations.

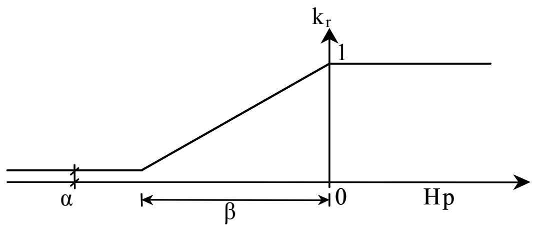
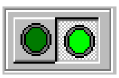


Figure 3.45 Meaning of the groundwater flow calculation parameters

Maximum iterations:

This parameter puts a limit on the number of iterations needed for achieving convergence. When using the standard setting, the maximum number of iterations is equal to 20, which is generally sufficient.

3.9 INITIAL GEOMETRY CONFIGURATION



The initial geometry configuration enables you to deselect the geometry clusters that are not active in the initial situation. In addition, initial effective stresses may be generated using the K_0-*procedure*. When the *Water conditions* mode is active, the *Geometry configuration* mode can be entered by clicking on the right hand side of the 'switch' in the toolbar.

3.9.1 DEACTIVATING LOADS AND GEOMETRY OBJECTS

In projects where structures are to be constructed (like tunnel shell elements) the model will contain some components (such as loads, plates, anchors or soil clusters above the initial ground surface) that are initially not active. Soil clusters above the initial ground surface must be deactivated by the user. The 3D Tunnel program will automatically deactivate all loads and structural objects in the initial geometry configuration, since, in general, these objects are to be applied in a later stage and are not present in the initial situation. Moreover, the K_0-procedure for the generation of initial stresses (Section 3.9.3) does not take external loads and weights of structural elements into account.

In the initial configuration, no distinction can be made between individual planes or slices in the z-direction. Hence, activation or deactivation of components is based on the cross-section model. activation or deactivation of components can be done by single clicking on the component in the cross-section model. Interfaces are always activated and deactivated together with the adjacent soil clusters and cannot be activated or deactivated separately. Deactivated clusters are drawn in the background colour (white) and deactivated structural objects are drawn in grey. Clicking once again on a deactivated component will reactivate that component.

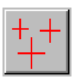
Anchors may only be active if the soil or structures to which they are connected are also active. Otherwise the calculation program deactivates them automatically. If loads or prescribed displacements act on a part of the geometry that is inactive, then these conditions are not applied during the analysis.

Although the external loads can be 'activated' in the initial configuration, they are not considered in the initial stress generation (K_0-procedure). It should also be noted that weights of structural elements are disregarded in the initial stress generation. External loads or structural objects in the initial configuration therefore have no effect.

3.9.2 CHANGING MATERIAL DATA SETS

On double clicking a cluster or structural object in the *Geometry configuration* mode the properties window appears in which the material data set of that component can be changed. This option is usually not considered in the initial conditions because the initial material setting is directly entered during the creation of the cross-section model. The option is more useful as a calculation option in the framework of *Staged construction* (Section 4.6.3).

3.9.3 INITIAL STRESS GENERATION (K_0-PROCEDURE)



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 ($\sigma_{h,0} = K_0\ \sigma_{v,0}$).

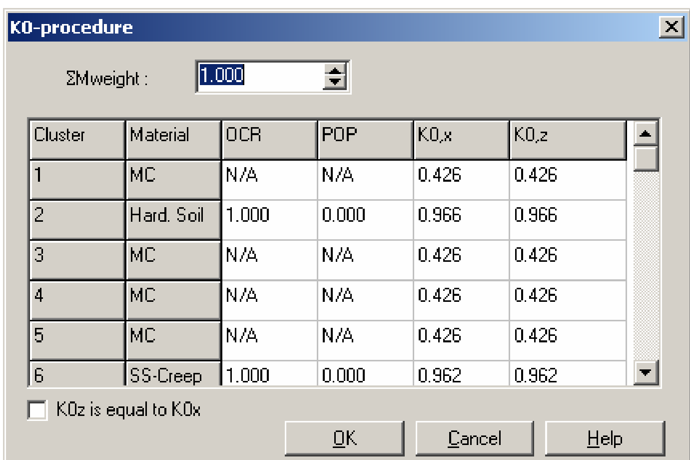


Figure 3.46 Initial stress generation window (K_0-*procedure*)

In the PLAXIS 3D Tunnel program, initial stresses may be generated by specifying K_0 or by using *Gravity loading*. Two K_0-values can be specified, one for the x-direction and one for the *z*-direction. The possibilities and limitations of both methods are further described in Appendix A.

When the generation of initial stresses is based on the K_0-procedure, it is not possible to allow for a variation of stresses in the *z*-direction. Hence, the initial stress situation is defined on the basis of the cross-section model, but the stresses are generated for the full 3D model. The K_0-procedure option can be selected by clicking on the *Generate initial stresses* button (red crosses) in the toolbar or by selecting *Initial stresses* from the *Generate* sub-menu.

As a result, a window appears with a table in which, with various other parameters, K_0-values can be entered (Figure 3.46). The meaning of the various parameters in the window is described below.

ΣMweight:

Before entering the values in the table a value for the *ΣMweight* parameter should be given. This parameter represents the proportion of gravity that is applied. In general, the default value of 1.0 can be accepted, which implies that the full soil weight is activated. To reset previously generated initial stresses to zero, *ΣMweight* should be set to zero and the initial stresses must be regenerated.

Cluster:

The first column displays the cluster number. When entering a value in the table the corresponding cluster is indicated in the main window on the background. If necessary, the initial stress generation window may be moved to another position to view the indicated cluster.

Model:

The second column displays the material model that is used in the particular cluster (Elastic = Elastic model; MC = Mohr-Coulomb model; Hard Soil = Hardening Soil model; SS-Creep = Soft Soil Creep model). See the Material models manual for more information.

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-sinφ), 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-sinφ. Be careful with very low or very high K_0-values, since these values may cause initial plasticity (see Appendix A). 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.

Upon pressing the <OK> button, the initial stress generation starts. The K_0-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.

Results of initial stress generation

After the generation of initial stresses the Output program is started and a plot of the initial effective stresses is presented. In general, the initial stresses at a stress point follow from the weight of the material above this point and the value of *ΣMweight*:

$$\sigma_{v,0} = \sum Mweight \left(\sum_i \gamma_i \bullet h_i - p_i \right) \qquad ; \qquad \sigma_{h,0} = K_0 \sigma_{v,0}$$

where γ_i is the unit weight of individual layers, h_i is the layer depth and p_w is the initial pore pressure in the stress point.

Using K_0-values that differ substantially from unity may sometimes lead to an initial stress state that violates Coulomb's 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.

To return to the Input program, the <Update> button should be pressed.

3.10 STARTING CALCULATIONS

With the generation of the initial stresses, the generation of the initial situation of the finite element model is complete. By clicking on the <Calculate> button in the toolbar, a dialog box appears in which the user is prompted to save the data. This may be done using an existing file name (just press <Yes>) or using a new name (press <Save as>). The latter option may also be used to create a copy of a previously generated model. As a result, the file requester appears in which the file name can be specified. When a new model was created which was not saved before, a file name must be given in both save options. On pressing the <No> button, the data will not be saved.

Pressing the <Cancel> button will close the dialog box after which the initial conditions mode of the Input program is re-entered. In all other cases (<Save>, <Save as> and <No>) the Input program is closed and the Calculation program is started.

4 CALCULATIONS

After the generation of a 3D finite element model, the actual finite element calculations can be executed. Therefore it is necessary to define which types of loadings or construction stages are to be activated during the calculations. This is done in the Calculations program.

The PLAXIS 3D Tunnel program allows for three different types of finite element calculations to calculate elastoplastic deformations: a *3D Plastic* calculation, a *3D Consolidation* analysis and a *3D Phi-c reduction* (safety analysis). These calculation types are indicated in the Calculations program. All three types of calculations (Plastic, Consolidation, Phi-c reduction) optionally allow for the effects of large displacements being taken into account. This is termed *Updated mesh*, which is available as an advanced option. The different types of calculations are explained in Section 4.4.2. In engineering practice, a project can be divided into project phases; a calculation process in the 3D Tunnel program is also divided into calculation phases. Examples of calculation phases are the activation of a certain load system or the simulation of a construction stage.

Each calculation phase is generally divided into a number of calculation steps. This is necessary because the non-linear behaviour of the soil requires loadings to be applied in small proportions (called load increments). In most cases, however, it is sufficient to specify the situation that has to be reached at the end of a calculation phase. Robust and automatic load stepping procedures in PLAXIS will take care of the sub-division into appropriate load increments.

4.1 THE CALCULATIONS PROGRAM

This icon represents the Calculations program. The Calculations program contains all facilities to define and start up finite element calculations. At the start of the Calculations program, the user has to select the project for which calculations are to be defined. The selection window allows for a quick selection of one of the four most recent projects. If a project 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 3D Tunnel project file (*.PL3). The selection of a project is not necessary when clicking on the <Calculate> button in the initial conditions mode of the Input program. In this case, the current project is automatically selected in the Calculations program. After the selection of a project, the main window of the Calculations program appears, which contains the following items (Figure 4.1).

Calculations menu:

The Calculations menu contains all operation facilities of the Calculations program. Most options are also available as buttons in the toolbar.

Toolbar:

The toolbar contains buttons that may be used as a shortcut to menu facilities. The meaning of a particular button is presented after the pointer is positioned above the button.

Tab sheets (upper part):

The tab sheets are used to define a calculation phase (Section 4.3 and further).

List of calculation phases (lower part):

This list gives an overview of all calculation phases of a project. Each line 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, the calculation type, the type of loading, the first and the last step number and the water pressure situation to be used.

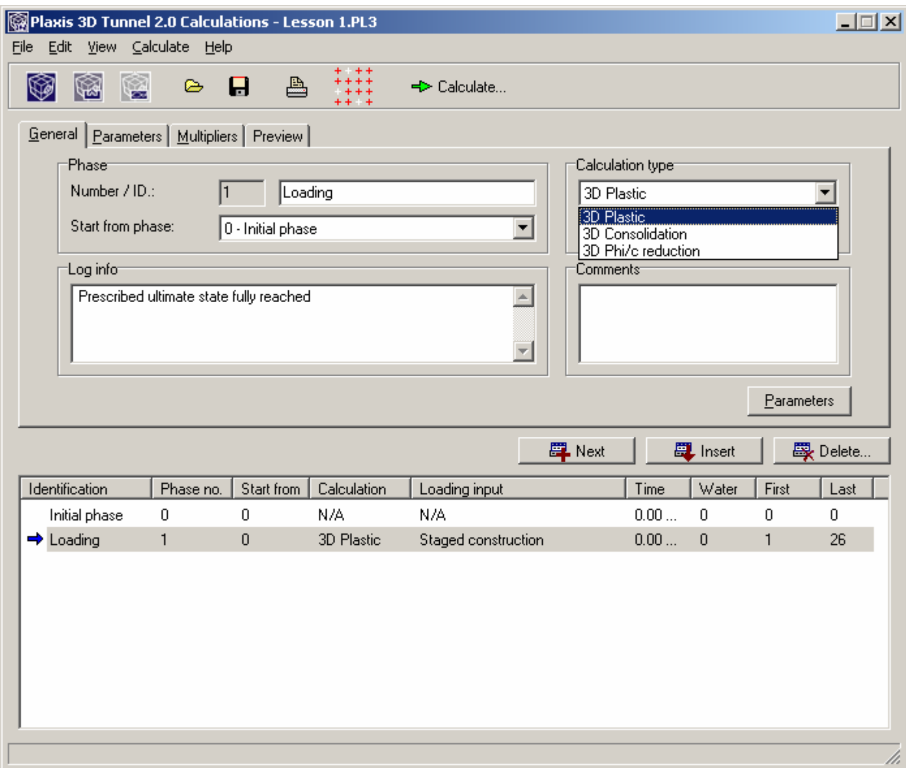


Figure 4.1 Main window of the Calculations program

If the phase has not yet been executed, the step numbers are blank. 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 (×).

Hint: If the list is not visible or too short, it may be enlarged by dragging down the bottom of the main window of the Calculations program.

4.2 THE CALCULATIONS MENU

The main menu of the Calculations program contains pull-down sub-menus covering most options for handling files, defining calculation phases and executing calculations. The Calculations menu consists of the sub-menus *File*, *Edit*, *View*, *Calculate* and *Help*.

The File sub-menu:

Open	To open a project for which calculation phases have to be defined. The file requester is presented.
Save	To save the current status of the calculations list.
Work directory	To set the directory where PLAXIS 3D Tunnel project files will be stored.
(*recent projects*)	To quickly open one of the four most recent projects.
Exit	To leave the program.

The Edit sub-menu:

Next phase	To focus on the next phase in the calculations list. If the next phase does not exist, a new calculation phase is introduced.
Insert phase	To insert a new calculation phase at the position of the currently focused phase.
Delete phase	To erase the selected calculation phase or phases.
Select all	To select all calculation phases.

The View sub-menu:

Calculation manager	To view the calculation manager window, from which all active calculation processes are controlled.
Select points for curves	To select nodes and stress points for the generation of load-displacement curves and stress paths.

The Calculate sub-menu:

Current project	To start up the calculation process of the current project.
Multiple projects	To select a project for which the calculation process has to be started. The file requester is presented.

4.3 DEFINING A CALCULATION PHASE

Consider a new project for which no calculation phase has yet been defined. In this case, the calculations list contains only one line, indicated as 'Initial phase'. This line represents the initial situation of the project as defined in the initial conditions mode of the Input program. The 'Initial phase' is the starting point for further calculations. To introduce the first calculation phase for the current project, the <Next> button just above the calculations list should be pressed after which a new line appears. Alternatively, the *Next phase* option may be selected from the *Edit* menu. When the Calculations program was started by clicking on the <Calculate> button in the initial conditions mode of the Input program, then a first undefined calculation phase is automatically introduced.

After the introduction of the new calculation phase, the phase has to be defined. This should be done using the tab sheets *General*, *Parameters* and *Multipliers* in the upper part of the main window. The definition starts with the selection of the *Calculation type* in the *General* tab sheet, which has three options: *3D Plastic, 3D Consolidation* and *3D Phi/c reduction*. 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.

When all parameters have been set, 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 toolbar or, alternatively, by selecting the *Current project* option in the *Calculate* 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.

4.3.1 INSERTING AND DELETING CALCULATION PHASES

When inserting and deleting calculation phases you have 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> button. It is possible, however, to insert a new phase between two existing phases. This is done by pressing the <Insert> button while the line where the new phase is to be inserted is focused. 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 multipliers, water conditions, active clusters and structural objects 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* 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 (Section 4.4.1). 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 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.

4.4 GENERAL CALCULATION SETTINGS

The *General* tab sheet is used to define the general settings of a particular calculation phase (Figure 4.2):

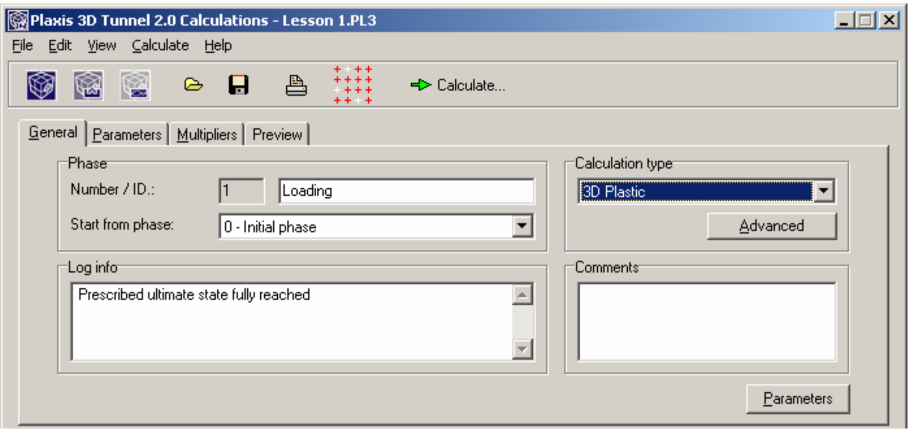


Figure 4.2 *General* tab sheet of the Calculations window

Calculation type:

The selections made in the two combo boxes of the *Calculation type* group determine the type of calculation and the load stepping procedure that is used (Section 4.4.2 and later sections).

Phase:

The items in the *Phase* group can be used to identify the calculation phase and, more importantly, to determine the ordering of calculation phases by selecting the calculation phase that is used as a starting point for the current calculation (Section 4.4.1).

Comments and log info:

The *Comments* box can be used to 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.

4.4.1 PHASE IDENTIFICATION AND ORDERING

The *Phase* box in the *General* tab sheet shows the phase number and an identification string of the current calculation phase. By default, the identification string is set to <Phase #>, where # is the current phase number, but this string may be changed by the user to give it a more appropriate name. The identification string and phase number appear in the calculation list at the lower part of the window.

In addition, the *Start from phase* parameter must be selected in the *Phase* box. This parameter refers to the phase from which the current 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, it is obvious that the calculation should start from the situation as generated in the Initial conditions of the Input program. However, 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 ordering is not straightforward is in calculations where safety analyses for intermediate construction stages are considered. Safety analyses in PLAXIS are based on the method of *Phi-c reduction* (Section 4.8), which results in a state of failure. When continuing the construction process, the next stage should start from the previous construction stage rather than from the results of the safety analysis. Alternatively, the safety analyses for the various construction stages can be performed at the end of the calculation process. In that case the *Start from phase* parameter in the safety calculations should refer to the corresponding construction stages.

4.4.2 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 three basic types of calculations: A *3D Plastic* calculation, a *3D Consolidation* analysis and *3D Phi-c reduction* (safety analysis).

3D Plastic calculation

A *3D Plastic* calculation is used to carry out an elastic-plastic deformation analysis according to small deformation theory. The stiffness matrix in a 3D 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 *3D 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 4.6.2) it is also possible (for each calculation phase) to redefine the water boundary conditions and recalculate the pore pressures due to either the hydrostatic phreatic level or groundwater flow. See Section 3.8 for details on generating pore pressures.

For more details on theoretical formulations reference should be made to the Scientific Manual.

3D Consolidation analysis

3D 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 TUNNEL 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. Consolidation analyses can be performed in the context of large deformations.

For more details on theoretical formulations, you are referred to the Scientific Manual.

3D Phi-c reduction (safety analysis)

A safety analysis in PLAXIS 3D TUNNEL can be executed by reducing the strength parameters of the soil. This process is termed *Phi-c reduction* and is available as a separate type of calculation. *Phi-c reduction* should be selected when it is desired to calculate a global safety factor for the situation at hand. A safety analysis can be performed after each individual calculation phase and thus for each construction stage. However, note that a phi-c reduction phase cannot be used as a starting condition for another calculation phase because it ends in a state of failure. Therefore it is advised to define all safety analyses at the end of the list of calculation phases and to use the *Start from phase* parameter as a reference to the calculation phase for which a safety factor is calculated.

When performing a safety analysis, no loads can be increased simultaneously. In fact, *Phi-c reduction* is a special type of plastic calculation. The input of a time increment is generally not relevant in this case.

When using *Phi-c reduction* in combination with advanced soil models, these models will actually behave as a standard Mohr-Coulomb model, since stress-dependent stiffness behaviour and hardening effects are excluded from the analysis. In that case, the stiffness is calculated at the beginning of the calculation phase and kept constant until the calculation phase is completed. For further details on *Phi-c reduction*, see Section 4.8.

Updated Mesh analysis

The three basic types of calculations (Plastic calculation, Consolidation analysis, Phi-c reduction) can optionally be performed as an *Updated Mesh* analysis, taking into account the effects of large deformations. You can choose this option by clicking the *Advanced* button in the *Calculation type* group on the *General* tab, Figure 4.3.

An *Updated Mesh* option is a calculation in which effects of large deformations are taken into account. This type of calculation should be considered where deformations are expected to significantly affect the shape of the geometry. The stiffness matrix in an updated mesh analysis is based on the deformed geometry. In addition, a special definition of stress rates is adopted that includes rotation terms. These calculation procedures are based on an approach known as an Updated Lagrange formulation (Ref. 2).

For most applications, the effect of large deformations are negligible. However, there are circumstances under which you may need to take these effects into account. Typical applications are the analysis of reinforced soil structures (tension stiffening effect), the analysis of collapse loads of large offshore footings, and the study of projects involving soft soils where large deformations can occur.

Note that an updated mesh calculation cannot be followed by a 'normal' calculation. On the other hand, a normal calculation can be followed by an updated mesh calculation, provided that you use the *Reset displacements to zero* option (Section 4.5). It should be noted that an updated mesh analysis takes much longer and is less robust than a normal calculation. Therefore, you should only use this option in special cases.

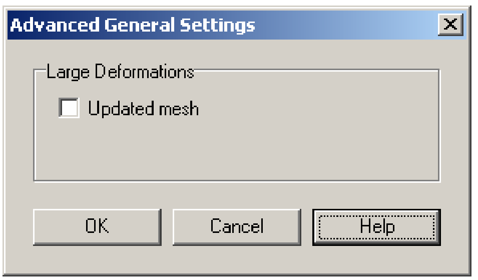


Figure 4.3 Advanced general settings window

4.4.3 LOAD STEPPING PROCEDURES

When soil plasticity is involved in a finite element calculation the equations become non-linear. This means that each calculation phase needs to be solved in a series of calculations steps (load steps). An important part of the non-linear solution procedure is the choice of step sizes and the solution algorithm to be used.

During each load 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 to reach equilibrium may become excessive or the solution procedure may even diverge.

In PLAXIS 3D TUNNEL there are various procedures available for the solution of non-linear plasticity problems. All procedures are based on an automatic step size selection. The following procedures are available: *Load advancement ultimate level*, *Load advancement number of steps* and *Automatic time stepping*. Users do not need to worry about the proper selection of these procedures, since PLAXIS 3D TUNNEL will automatically use the most appropriate procedure by itself to guarantee optimum performance.

The automatic load stepping procedures are controlled by a number of calculation control parameters (see 4.6). There is a convenient default setting for most control parameters, which strikes a balance between robustness, accuracy and efficiency. Users can influence the automatic solution procedures by manually adjusting the control parameters. In this way it is possible to have a stricter control over step sizes and accuracy. Before proceeding to the calculation control parameters, a detailed description is given of the solution procedures themselves.

4.4.4 AUTOMATIC STEP SIZE PROCEDURES

Both of the *Load advancement* procedures (*Ultimate level* and *Number of steps*) make use of an automatic step size algorithm (Reference 17). The size of the first load step is either chosen automatically (see 4.5.2) or manually by the user (see 4.5.3), depending on the applied algorithm. The automatic step size procedure for subsequent computations is described below.

When a new load step is applied, a series of iterations are 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 6, but this value may be changed in the *Manual setting* of the *Iterative procedure* in the *Parameters* tab sheet. 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 15, but this value may be changed in the *Manual setting* of the *Iterative procedure* in the *Parameters* tab sheet. 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.

4.4.5 LOAD ADVANCEMENT ULTIMATE LEVEL

This automatic step size procedure is primarily used for calculation phases where a certain 'state' or load level (the 'ultimate state' or 'ultimate level') has to be reached. The procedure terminates the calculation when the specified state or load level is reached or when soil failure is detected. By default, the number of *Additional steps* is set to 250, but this parameter does not play an important role, since in most cases the calculation stops before the number of additional steps is reached.

An important property of this calculation procedure is that the user specifies the state or the values of the total load that is to be applied. A *Plastic* calculation where the *Loading input* is set to *Staged construction* or *Total multipliers* uses this *Load advancement ultimate level* procedure.

The size of the first load step is obtained automatically using one of the two following methods:

- PLAXIS 3D TUNNEL performs a trial calculation step and determines a suitable step size on the basis of this trial.
- PLAXIS 3D TUNNEL sets the initial load step size to be equal to the final load step size of any previous calculation.

The first method is generally adopted. The second method would only be used if the loading applied during the current load step is similar to that applied during the previous load step, for example if the number of load steps applied in the previous calculation proved to be insufficient. The calculation will proceed until one of the three following criteria has been satisfied:

- The total specified load has been applied. In this case the calculation phase has succesfully finished and the following message is displayed in the *Log info* box of the *General* tab sheet: *Prescribed ultimate state fully reached*.

- The maximum specified number of additional load steps has been applied. In this case it is likely that the calculation stopped before the total specified load has been applied. 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*.
- 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. The following message is displayed in the *Log info* box: *Prescribed ultimate state not reached; Soil body collapses*.

4.4.6 LOAD ADVANCEMENT NUMBER OF STEPS

This automatic step size procedure always performs the number of *Additional steps* that has been specified. This algorithm is, in general, used for calculation phases where a complete failure mechanism should be developed during the analysis. A safety analysis by means of *Phi-c-reduction* or a *Plastic* calculation where the *Loading input* is set to *Incremental multipliers* uses this *Load advancement number of steps* procedure.

If this option is selected then it is necessary for the user to specify the initial step size. After the first step has been completed then the program uses the standard automatic step size algorithm to determine the sizes of subsequent steps. It cannot be determined beforehand what the load level will be at the end of such a calculation. The calculation will proceed until the number of *Additional steps* has been applied. In contrast to the *Ultimate level* procedure the calculation will not stop when failure is reached.

4.4.7 AUTOMATIC TIME STEPPING (CONSOLIDATION)

When the *Calculation type* is set to *Consolidation* then the *Automatic time stepping* procedure is used. This procedure will automatically choose appropriate time steps for a consolidation analysis. When the calculation runs smoothly, resulting in very few iterations per step, then the program will choose a larger time step. When the calculation uses many iterations due to an increasing amount of plasticity, then the program will take smaller time steps.

The first time step in a consolidation analysis is generally based on the *First time step* parameter. This parameter is, by default, based on the advised minimum time step (overall critical time step) as described in 4.6.1. The *First time step* parameter can be changed in the *Manual setting* of the *Iterative procedure*. However, care should be taken with time steps that are smaller than the advised minimum time step.

In a consolidation analysis where the *Loading input* is set to *Incremental multipliers*, the applied first time step is based on the *Time increment* parameter rather than on the *First time step* parameter. In this case, the specified number of *Additional steps* is always performed. In a consolidation analysis where the *Loading input* is set to *Staged construction* or *Minimum pore pressure*, the specified number of *Additional steps* is just an upper bound. In that case, the calculation is generally stopped earlier, when other conditions are met.

4.5 CALCULATION CONTROL PARAMETERS

The *Parameters* tab sheet is used to define the control parameters of a particular calculation phase (Figure 4.7). This tab sheet contains the following items:

Additional steps

This parameter specifies the maximum number of calculation steps (load steps) performed during a particular calculation phase.

If you select a *Plastic* calculation or a *Consolidation* analysis as the calculation type and set the loading input to *Staged construction*, *Total multipliers* or *Minimum pore pressure*, the number of additional steps is an upper limit to the actual number of steps that will be executed. In general, such a calculation should be completed within the number of additional steps and stops according to the first or third criterion as described in Section 4.4.5).

If such a calculation reaches the maximum number of additional steps, it usually means that the ultimate level has not been reached. The default *Additional steps* parameter is 250, which is generally sufficient to complete the calculation phase. However, you may change this number within the range 1 to 1000.

If you select a *Plastic* calculation or a *Consolidation* analysis and set the loading input to *Incremental multipliers*, you should set the number of additional steps to an integer number representing the required number of steps for this calculation phase. In this case, the number of additional steps is always exactly executed. The default *Additional steps* parameter is 250, but you can change this number within the range 1 to 1000. The same applies to a *Phi-c reduction* calculation, except that the default *Additional steps* parameter is 100.

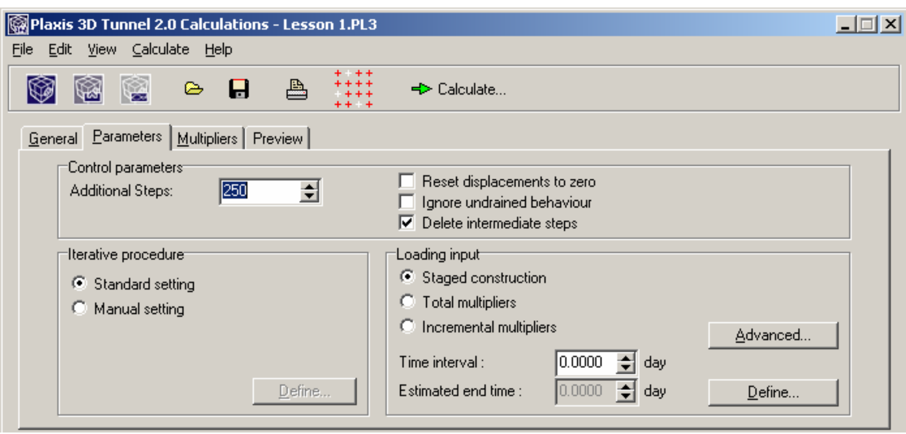


Figure 4.4 *Parameters* tab in the *Calculations* window

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 may 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.

The use of the *Reset displacements to zero* option may not be used in a sequence of calculations where the Updated Mesh option is used. However, if a single or sequence of Updated Mesh calculations starts from a calculation where the Updated Mesh option is not used, then the *Reset displacements to zero* option must be used in the first Updated Mesh calculation.

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 stage and to ignore the undrained behaviour during the gravity loading stage. Hence, the behaviour of all undrained clusters is considered to be drained during this preliminary calculation.

The *Ignore undrained behaviour* option is not available for consolidation analyses, since a consolidation analysis does not consider the *Material type* (drained or undrained) as specified in the material data sets.

Delete intermediate steps

This option may be selected for calculations of the *Load Adv. Ultimate level* type to save disk space. When selecting this option all additional output steps within the calculation phase, except for the last one, are deleted when a calculation phase has been finished successfully. In general the final output step contains the most relevant result of the calculation phase, whereas intermediate steps are less important.

4.5.1 ITERATIVE PROCEDURE CONTROL PARAMETERS

The iterative procedures, in particular the load advancement procedures, are influenced by some control parameters. These parameters can be set in the *Iterative procedure* group. The PLAXIS 3D Tunnel 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 *Standard setting*. In some situations, however, it might be desired or even necessary to change the standard setting. In this case the user should select the *Manual setting* and click on the <Define> button in the *Iterative procedure* group. As a result, a window is opened in which the control parameters are displayed with their current values (Figure 4.5).

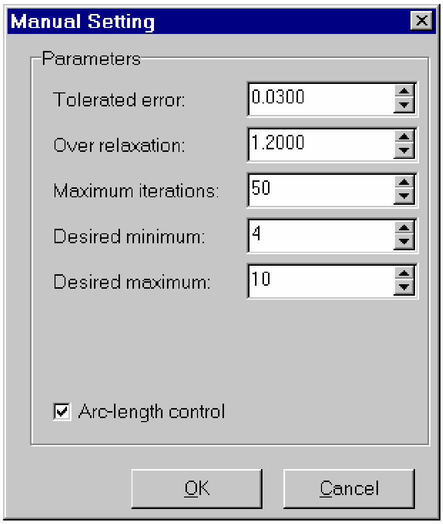


Figure 4.5 Iterative procedure control parameters window

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 4.6. The purpose of a solution algorithm is to ensure that the equilibrium errors, both locally and globally, remain within acceptable bounds (Section 4.15). The error limits adopted in the PLAXIS 3D Tunnel 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 plastic 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 PLAXIS 3D TUNNEL see Section 4.15.

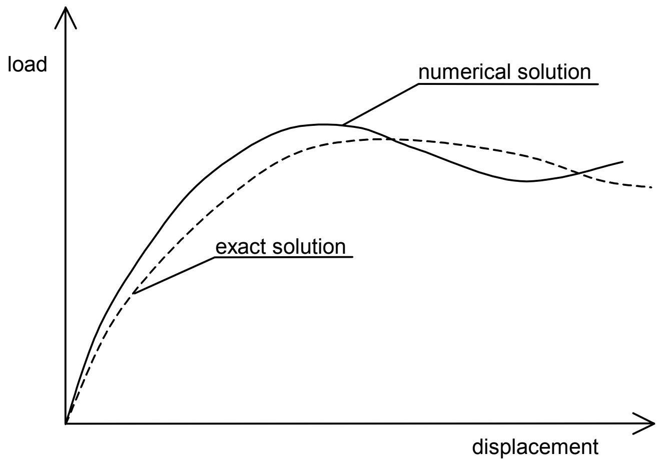


Figure 4.6 Computed solution versus exact solution

Over-relaxation

To reduce the number of iterations needed for convergence, the PLAXIS 3D Tunnel program makes use of an over-relaxation procedure as indicated in Figure 4.7. 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 φ<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.

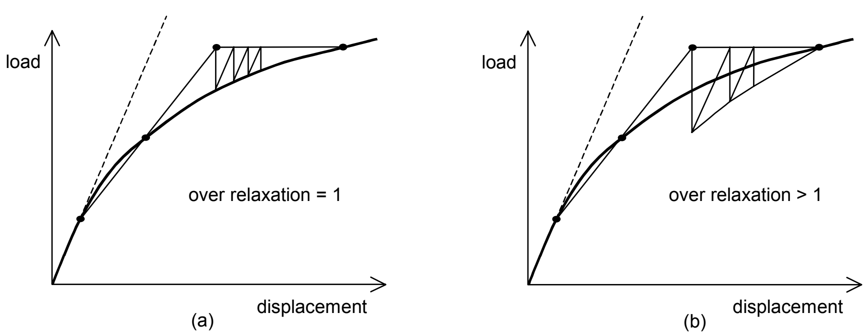


Figure 4.7 Iteration process with (a) and without (b) over-relaxation

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 60, but this number may be changed within the range 1 to 100.

If the maximum allowable number of iterations is reached in the final step of a calculation phase, then the final result may be inaccurate. If this is the case then the message '*Maximum iterations reached in final step*' is displayed in the *Log info* box of the *General* tab sheet. Such a situation occasionally occurs when the solution process does not converge. This may have various causes, but it mostly indicates an input error.

Desired minimum and desired maximum

If either of the two *Load advancement* options (*Ultimate level* or *Number of steps*) is adopted then PLAXIS 3D TUNNEL 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 6 and 15 respectively, but these numbers may be changed within the range 1 to 100. For details on the automatic step size procedures see Section 4.4.4 to 4.4.6.

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 = 4 Desired maximum = 10

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 = 8 Desired maximum = 20

In this case it is recommended to increase the *Maximum iterations* to 100.

Arc-length control

The *Arc-length* control procedure is a method that is by default selected in PLAXIS 3D TUNNEL (except in *Consolidation* phases) to obtain reliable collapse loads for load-controlled calculations (Reference 9). The iterative procedure adopted when arc-length control is not used is shown in Figure 4.8a 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 4.8b.

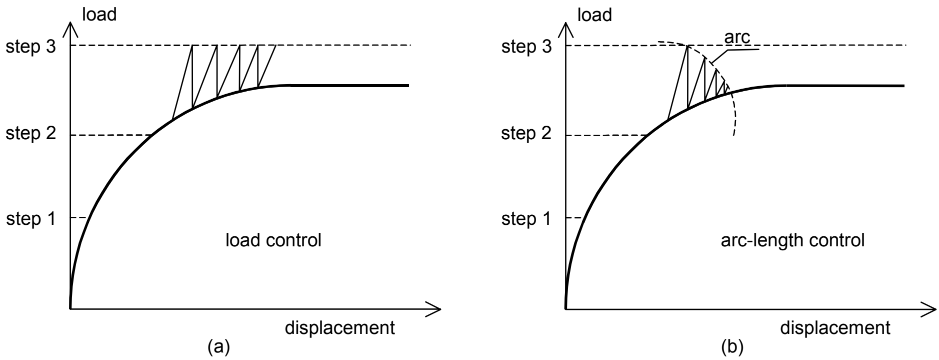


Figure 4.8 Iterative procedure for normal load control (a) and arc-length control (b)

Arc-length control is activated by selecting the corresponding check box in the iterative procedure control parameters window. The arc-length control procedure should be used for load-controlled calculations, but it may be deactivated, if desired, for displacement-controlled calculations. When using incremental load multipliers, arc-length control will influence the resulting load increments. As a result, the load increments applied during the calculation will generally be smaller than prescribed at the start of the analysis.

Hint: The use of arc-length control may occasionally cause 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, selecting the *Manual setting* and deselecting *Arc-length control*. Note that if arc-length control is deselected and failure is approached, convergence problems may occur.

First time step

The *First time step* is the increment of time used in the first step of a consolidation analysis, except when using *Incremental multipliers* as *Loading input*. By default, the first time step is equal to the overall critical time step, as described below.

You should be careful when using time steps that are smaller than the recommended minimum time step. As for most numerical integration procedures, accuracy increases when the time step is reduced, but there is a threshold value for consolidation. The level of accuracy rapidly decreases below a particular time increment (critical time step).

For one-dimensional consolidation (vertical flow), this critical time step is calculated as:

$$\Delta t_{critical} = \frac{H^2 \gamma_w (1-2\nu)(1+\nu)}{40 \quad k_y E(1-\nu)}$$

where γ_w is the unit weight of the pore fluid, ν is Poisson's ratio, k_y is the vertical permeability, E is the elastic Young's modulus, and H is the height of the element used. Fine meshes allow for smaller time steps than coarse meshes. For unstructured meshes with different element sizes or when dealing with different soil layers and thus different values of k, E and ν, the above formula yields different values for the critical time step. To be on the safe side, the time step should not be smaller than the maximum value of the critical time steps of all individual elements. This overall critical time step is automatically adopted as the *First time step* in a consolidation analysis.

For an introduction to the critical time step concept, see Reference 30. For detailed information for various types of finite elements, see Reference 28.

Extrapolation

Extrapolation is a numerical procedure, which is automatically used in PLAXIS 3D TUNNEL if applicable, when a certain loading that was applied in the previous calculation step is continued in the next step. In this case, the displacement solution to the previous load increment can be used as a first estimate of the solution to the new load increment. Although this first estimate is generally not exact (because of the non-linear soil behaviour), the solution is usually better than the solution according to the initial stress method (based on the use of the elastic stiffness matrix) (Figure 4.9). After the first iteration, subsequent iterations are based on the elastic stiffness matrix, as in the initial stress method (Reference 20). Nevertheless, using *Extrapolation* the total number of iterations needed to reach equilibrium is less than without extrapolation. The extrapolation procedure is particularly useful when the soil is highly plastic.

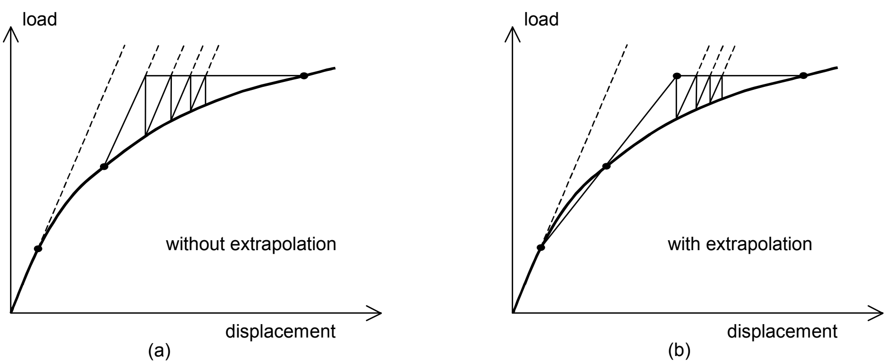


Figure 4.9 Difference between elastic prediction (a) and extrapolation from previous step (b)

4.5.2 LOADING INPUT

The *Loading input* group box is used to specify which type of loading is considered in a particular calculation phase. Only one of the described loading types can be activated in any one calculation phase.

In *Plastic* calculations, distinction is made between the following types of *Loading input*:

- Loading in the sense of changing the load combination, stress state, weight, strength or stiffness of elements, activated by changing the load and geometry configuration or pore pressure distribution by means of *Staged construction*. In this case, the total load level that is to be reached at the end of the calculation phase is defined by specifying a new geometry and load configuration, and/or pore pressure distribution, in the *Staged construction* mode (Section 4.6).
- Loading in the sense of increasing or decreasing a predefined combination of external forces, activated by changing *Total multipliers*. In this case, the total load level that is to be reached at the end of the calculation phase is defined by entering values for the *Total multipliers* in the *Multipliers* tab sheet.
- Loading in the sense of increasing or decreasing a predefined combination of external forces, activated by changing *Incremental multipliers*. In this case, the first increment of load is defined by entering values for the *Incremental multipliers* in the *Multipliers* tab sheet, and this loading is continued in subsequent steps.

When selecting *Phi-c reduction* only the *Incremental multipliers* option is available.

In a *Consolidation* analysis, the following options are available:

- Consolidation and simultaneous loading in the sense of changing the load combination, stress state, weight, strength or stiffness of elements, activated by changing the load and geometry configuration or pore pressure distribution by means of *Staged construction*. It is necessary to specify a value for the *Time interval* parameter, which has in this case the meaning of the total consolidation period applied in the current calculation phase. The applied first time increment is based on the *First time step* parameter in the Calculations control parameters window. The *Staged construction* option should also be selected if it is desired to allow for a certain consolidation period without additional loading.
- Consolidation without additional loading, until all excess pore pressures have decreased below a certain minimum value, specified by the *P-stop* parameter. By default, *P-stop* is set to 1 stress unit, but this value may be changed by the user. Please note that the *P-stop* parameter is an absolute value, which applies to pressure as well as tensile stress. The input of a *Time interval* is not applicable in this case, since it cannot be determined beforehand how much time is needed to fulfill the minimum pore pressure requirement. The applied first time increment is based on the *First time step* parameter in the Calculations control parameters window.
- Consolidation and simultaneous loading in the sense of increasing or decreasing a predefined combination of external forces, activated by changing *Incremental*

multipliers. It is necessary to specify a value for the *Time increment* parameter in the unit of time. The *Time increment* sets in this case the applied first time step and determines the loading rate, together with the incremental multipliers in the *Multipliers* tab sheet.

Staged construction

If *Staged construction* is selected from the *Loading input* box, then the user can specify a new state that is to be reached at the end of the calculation phase. This new stage can be defined by pressing the <Define> button and changing the geometry, the input values of loads, the load configuration and the water pressure distribution in the Staged Construction mode. The *Staged construction* option may also be used to perform plastic nil-steps to solve existing out-of-balance forces. In this case, no changes in the geometry, load level, load configuration and water pressure distribution should be made.

Before specifying the construction stage, the *Time interval* of the calculation phase should be considered. The *Time interval* is expressed in the unit of time. A non-zero value is only relevant in the case of a Consolidation analysis or if the Soft Soil Creep model is used (see Material Models manual). The appropriate value can be entered in the *Loading input* group of the *Parameters* tab sheet.

Since staged construction is performed using the *Load advancement ultimate level* procedure (Section 4.4.5), it is controlled by a total multiplier ($\Sigma Mstage$). This multiplier generally starts at zero and is expected to reach the ultimate level of 1.0 at the end of the calculation phase. In some special situations, however, it might be necessary to split the staged construction process into more than one calculation phase and to specify an intermediate value of $\Sigma Mstage$. This can be done by clicking on the <Advanced> button, which is only available for a *Plastic* calculation. As a result, a window appears in which the desired ultimate level of $\Sigma Mstage$ can be specified. However, care must be taken with an ultimate level smaller than 1.0, since this is associated with a resulting out-of-balance force. Such calculations must always be followed by another staged construction calculation. Before starting any other type of calculation the $\Sigma Mstage$ parameter must first have reached the value 1.0. This can be verified after a calculation by selecting the *Reached values* option in the *Multipliers* tab sheet (Section 4.7.2).

Total multipliers

If the *Total multipliers* option is selected in the *Loading input* box, then the user may specify the multipliers that are applied to current configuration of the external loads. The actual applied load at the end of the calculation phase is the product of the input value of the load and the corresponding load multiplier, provided a collapse mechanism or unloading does not occur earlier. Before specifying the external loads, the *Time interval* of the calculation may be specified in the *Loading input* box of the *Parameters* tab sheet. The time interval is the time involved in the current calculation phase, expressed in the unit of time as specified in the *General settings* window of the Input program. A non-zero value is only relevant if the Soft Soil Creep model is used. The combination of the total multipliers and the time interval determine the loading rate that is applied in the calculation. In addition to the time interval, an estimate is given of the total time at the end of the calculation phase (*Estimated end time*), which is a summation of all time intervals of preceding calculation phases including the current one. If the calculation phase has been executed, the *Realised end time* is given instead, which is the total time that has actually been reached at the end of the calculation phase.

Incremental multipliers

Selecting *Incremental multipliers* in the *Loading input* box enables the user to specify incremental load multipliers that are applied to current configuration of the external loads. The initially applied load increment in the first step of the calculation phase is the product of the input value of the load and the corresponding incremental multiplier. Note that the resulting increments of load in the first calculation step will be influenced by the *Arc-length control* procedure if it is active.

Before entering an increment of external load, a *Time increment* can be entered in the *Loading input* box of the *Parameters* tab sheet. This is only relevant for a *Consolidation* analysis or if the Soft Soil Creep model is used. The combination of the incremental multipliers and the time increment determine the loading rate that is applied in the calculation. The time increment is expressed in the unit of time as entered in the *General settings* window of the Input program.

Minimum pore pressure (consolidation)

This consolidation option involves an extra criterion for terminating the analysis. Here, the number of Additional steps is a maximum number, which will not be reached if the other criterion prevails. In this case, the other criterion is a prescribed minimum excess pore pressure *P-stop*. The calculation stops when the maximum absolute excess pore pressure is below the prescribed value of *P-stop*. For example, when the maximum excess pore pressure has reached a certain value during the application of load, you can make sure that the consolidation process is continued until all nodal values of excess pore pressure are less than *P-stop*.

The degree of consolidation is an important indication of the consolidation state. Strictly, the degree of consolidation, x, is defined in terms of the proportion of the final settlement although the term is often used to describe the proportion of pore pressures that have dissipated to at least (100-x)% of their values immediately after loading. You can use the *Minimum pore pressure* option to specify the final degree of consolidation in any analysis. In order to specify an appropriate value of minimum pore pressure, *P-stop*, you must determine the maximum absolute excess pore pressure immediately after loading. This parameter, *Pmax*, is displayed on the *Multipliers* tab of the previous calculation phase when you choose the *Reached values* option (Section 4.7.1). A suitable value for *P-stop* may be determined from the expression:

$$P\text{-}stop = Pmax\ (100\text{-}x)\%$$

For example, in order to consolidate to 90%, the appropriate value of *P-stop* is one tenth of *Pmax*.

Time increment, 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 on the *Dimensions* tab in the *General settings* window. A non-zero value for the *Time increment* or *Time interval* parameters is only relevant when a consolidation analysis is being 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 increment* is the increment of time considered in a single step (first step) in the current calculation phase.
- *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.

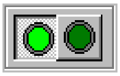
4.6 STAGED CONSTRUCTION

Staged construction is the most important type of *loading input*. With this special PLAXIS feature, you can change the geometry and load configuration by deactivating or reactivating loads, 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. You can also use the option to reassign material data sets or to change the water pressure distribution in the geometry.

To perform a staged construction calculation, you must first create a geometry model that includes all 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 geometry configuration at the end of the Input program (Section 4.6.2).

A staged construction analysis can be executed in a *Plastic* calculation as well as a *Consolidation* analysis. On the *Parameters* tab, you can choose the *Staged construction* option in the *Loading input* box. When you then click the *Define* button, the Input program starts and the staged construction window is displayed. This window is similar to the initial conditions window, except that you cannot activate options that are only relevant for the initial conditions (such as the K_0-procedure). It is also not possible to enter the *Geometry creation* mode of the Input program from the staged construction window. You can, however, use the specific staged construction options.

In a similar way to the *Initial conditions* window, the *Staged construction* window consists of two different modes: The *Geometry configuration* mode and the *Water conditions* mode. You can use the *Geometry configuration* mode to activate or deactivate loadings, soil clusters and structural objects and to reassign material data sets to clusters and structural objects. In addition to these facilities, staged construction allows for the pre-stressing of anchors. You can use the *Water conditions* mode to generate a new water pressure distribution based on the input of a new set of phreatic levels or on a groundwater flow calculation using a new set of boundary conditions.



You can switch between the *Water conditions* mode and the *Geometry configuration* mode using the toggle button on the toolbar. After defining the new situation, click the *Update* button to store the information and return to the *Calculations* program. You can also define the next calculation phase or start the calculation process.

Changes to the geometry configuration or the water conditions generally cause substantial out-of-balance forces. You can apply these out-of-balance forces applied step-by-step to the finite element mesh using a *Load advancement ultimate level* procedure. During a staged construction calculation, a multiplier that controls the staged construction process (*ΣMstage*) is increased from zero to the ultimate level (generally 1.0). In addition, a parameter representing the active proportion of the geometry (*ΣMarea*) is updated.

4.6.1 COPY OPTION

This option enables easy copying of staged construction settings from a slice or a plane to another slice or plane. You can also copy data for a group of slices or planes. When you choose the *Copy* option, PLAXIS opens a *Copy* dialog window (*Copy slice or plane*), as shown in Figure 4.10 and Figure 4.11.

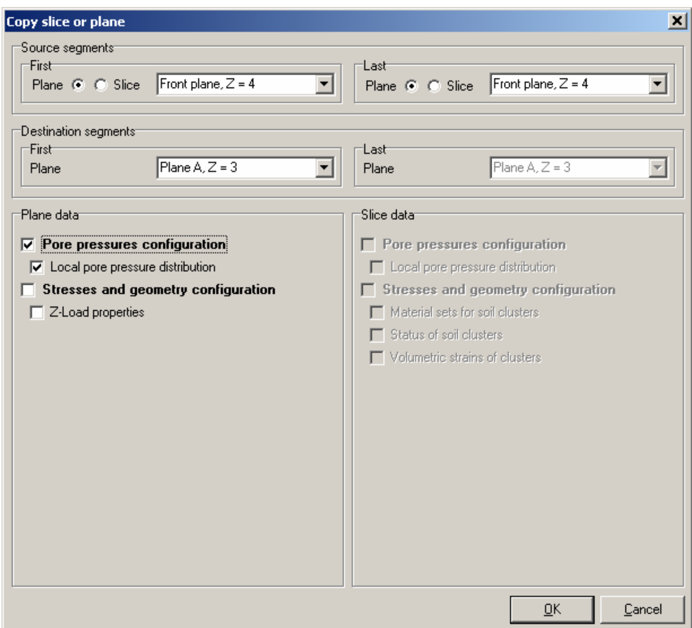


Figure 4.10 Copy options for plane settings

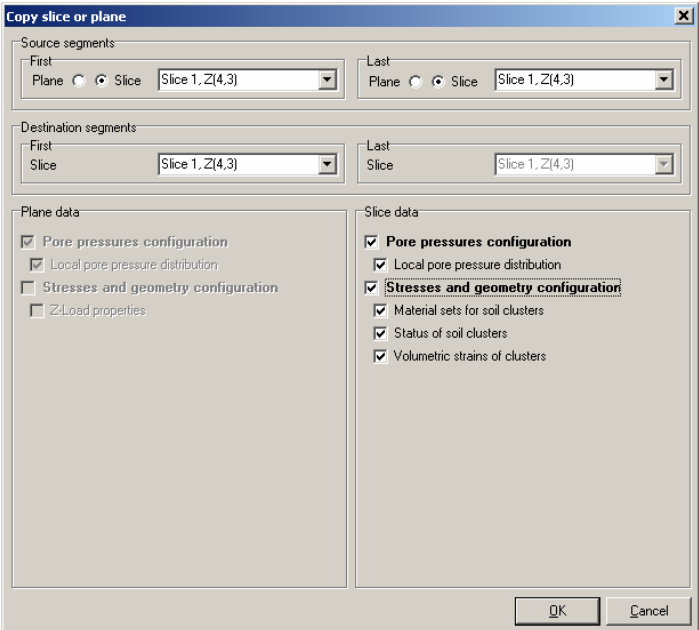


Figure 4.11 Copy options for slice settings

This window contains four main group boxes: *Source segments*, *Destination segments*, *Plane data* and *Slice data*.

Source segments

In this box, you can choose from which plane or slice you want to copy data to another plane or slice. You can copy data of one plane or slice to more than one plane or slice.

In the *First* segment box, you can select either *Plane* or *Slice* by clicking on the relevant radio button. Then, in the list displayed in the combo box, you can select the plane or slice whose data you want to copy to another plane or slice.

In the *Last* segment box, you must specify the last segment. If you only want to copy one plane or slice, you must choose the same segment as specified in the *First* segment. However, if you want to copy more than one plane or slice, you can choose any plane or slice one from the list displayed in the combo box on the right. When you select a range of slices to be copied to another range of slices, the planes between these slices will be copied automatically.

Destination segments

In this box, you can choose to where you want to copy the chosen plane(s) or slice(s). In the *First* segment box, you can choose the slice or plane to which you want to copy the settings. The *Last* segment will be set automatically according to the range specified in *Source segments*.

Slice data

In this box, you can choose the kind of input data you want to copy to the new slice(s). Two types of data can be copied: *Pore pressures configuration* and *Stresses and geometry configuration.*

In the *Pore pressures configuration*, you can make a copy of the *User defined pore-pressure distribution* (Section 3.8.2).

In the *Stresses and geometry configuration*, the following data can be copied:

- *Material sets* for: soils, plates, geogrids, anchors, interface material sets (Section 3.5).
- *Status of soil clusters*: this option enables you to copy the cluster status switched on or switched off (Section 4.6)
- *Volume strains of clusters*: this option allows you to copy prescribed volumetric strains in a soil volume cluster. The option is useful for simulating mechanical processes such as grouting of a material (see Section 4.6.7).

Plane data

In this box, you can choose the kind of input data you want to copy to the new plane(s). Two types of data can be copied: *Pore pressures configuration* and *Stresses and geometry configuration.*

In the *Pore pressures configuration*, you can choose the *User defined pore-pressure distribution* (Section 4.5) to be copied.

In the *Stresses and geometry configuration*, a copy can be made to the *Z-load properties* (see Section 4.6.4).

4.6.2 CHANGING GEOMETRY CONFIGURATION

Just as in the initial geometry configuration, the clusters or structural objects may be reactivated or deactivated to simulate a process of construction or excavation. In the framework of staged construction, this can be done for individual slices or planes. Hence, it is possible, for example, to excavate a tunnel and install the lining in the slices z=0 to 10 m, whereas in the slices z=10 to 20 m the original soil profile is retained without the tunnel being active. In this way the three-dimensional effects around the tunnel front can be analysed realistically.

Tab sheets are introduced to distinguish between the various slices and *z*-planes. A *z*-plane is a vertical cross-section with a unique *z*-coordinate, as specified in the 3D mesh generation (Section 3.6.6). *Z*-planes (or, referred to more briefly below as ‘planes’) are indicated by 'Front plane' (highest *z*-coordinate), 'Plane A', 'Plane B', ..., 'Rear plane' (lowest *z*-coordinate). Planes are used to activate point loads, line loads (line in *x*- and/or *y*-direction for which z is constant) or *z*-loads (distributed loads in the *z*-direction, acting perpendicular to the face of a soil cluster), or to apply a contraction to a tunnel cross-section, or to activate or deactivate anchors or pre-stress them. A slice is a volume between two planes, i.e. a vertical slice of the model over a small range of *z*-coordinates within the full range of *x*- and *y*-coordinates. Slices are used to activate distributed loads or line loads (loads on single points representing lines in *z*-direction), or to activate or deactivate soil volume clusters and plates, or to reassign material data sets to soil or structural objects, or to apply an internal pressure or volume strain in a volume cluster. To view the position of the selected plane or slice in the model, the *Top view* option may be selected from the *View* menu. The top view is interactive, which means that individual slices or planes may also be selected by clicking on them in the top view.

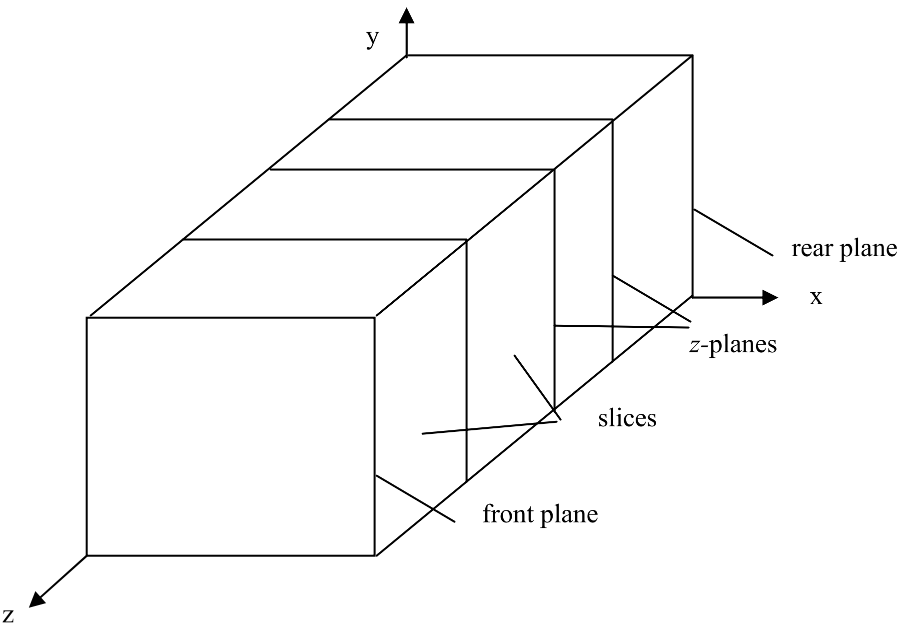


Figure 4.12 Definition of *z*-planes and slices

The geometry configuration can be changed by selecting the desired tab sheet and clicking on the object in the geometry. 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 geometry line (for example plates and distributed loads), a selection window appears from which the desired object can be selected.

Active soil clusters are drawn in the material data set colour whereas deactivated clusters are drawn in the background colour (white). Active structural objects are drawn in their original colour, whereas deactivated structures are drawn in grey. Interfaces are always activated and deactivated together with the adjacent soil clusters and cannot be activated or deactivated separately.

When double clicking a structural object, the corresponding properties window appears and the properties can be changed.

In the selection window that appears after double clicking a volume cluster, you can either change the material properties or apply a volumetric strain to the selected cluster.

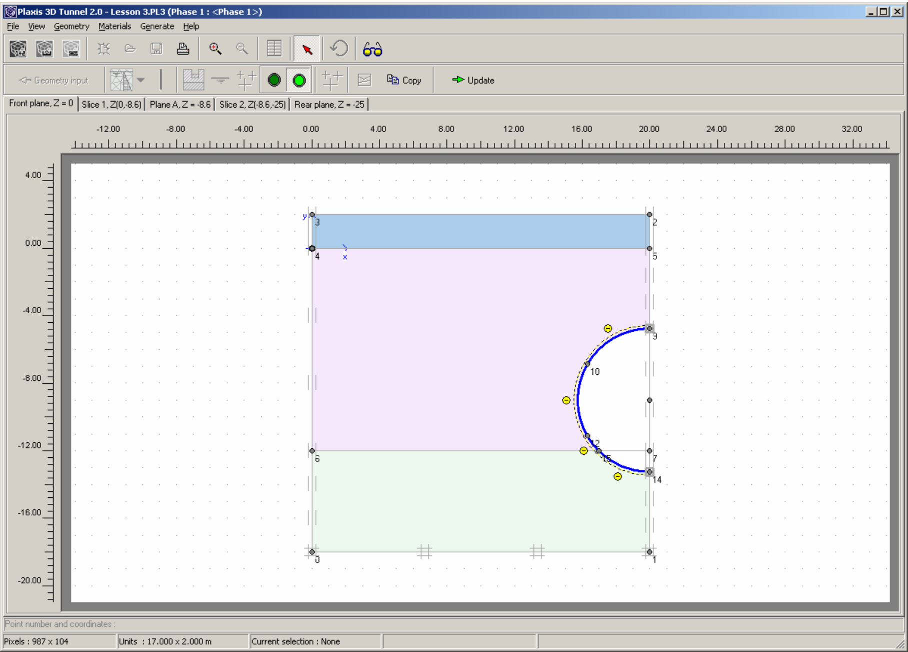


Figure 4.13 Geometry configuration setting in staged construction window

Interfaces can be activated or deactivated individually. Deactivation of interfaces may be considered in the following situations:

- To avoid soil-structure interaction (slipping and gapping) e.g. before a sheet pile wall or tunnel is installed in the soil (when corresponding plate elements are inactive).
- To avoid blocking of flow before a structure composed of plate elements is active.,

In any case, interface elements are present in the finite element mesh from the very beginning. However, the following special conditions are applied to inactive interfaces:

- Purely elastic behaviour (no slipping or gapping)
- Fully coupled pore pressure degrees-of-freedom in node pairs (no influence on flow in consolidation or groundwater calculations).

4.6.3 ACTIVATING AND DEACTIVATING CLUSTERS OR STRUCTURAL OBJECTS

Soil volume clusters and structural objects can be activated or deactivated by selecting the desired tab sheet and clicking once on the cluster or structural object in the geometry. Soil volumes, plates and geogrids can only be activated or deactivated in slices, whereas anchors can only be activated or deactivated in planes. Anchors may only be active if at least one of the soil clusters or plates to which they are connected is also active; otherwise the calculations program deactivates them automatically.

At the start of a staged construction calculation the information about active and inactive objects in the 3D model is transformed into information on an 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 stresses 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 elements. This means that PLAXIS will automatically generate suitable water pressures on submerged boundaries caused by the removal of elements. This may be checked when entering the water conditions mode. On 'excavating' (i.e. deactivating) clusters below the general phreatic level, the excavation remains filled with water. 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 in the water conditions mode. This feature is demonstrated in chapter 5 of the Tutorial Manual.
- External loads or prescribed displacements that act on a part of the geometry that is inactive will not be taken into account.

For elements that have been inactive and that are (re)activated in a particular calculation, the following rules apply:

- The stresses will develop from zero.
- When a node becomes active, an initial displacement is estimated by stressless predeforming the newly activated elements such that they fit within the deformed mesh as obtained from the previous step. Further increments of displacement are added to this initial value. As an example, one may consider the construction of a block in several layers, allowing only for vertical displacements (one-dimensional compression). Starting with a single layer and adding one layer on top of the first will give settlements of the top surface. If a third layer is subsequently added to the second layer, it will be given an initial deformation corresponding to the settlements of the surface.

- If an element is (re)activated and the *Material type* of the corresponding material data set has been set to *Undrained*, then the element will temporarily behave drained in the phase where the element was activated. This is to allow for the development of effective stresses due to the self weight in the newly activated soil. If the element remains active in later calculation phases, then the original type of material behaviour is retained in those phases.

4.6.4 ACTIVATING OR CHANGING LOADS

Loads that were created in the geometry input are deactivated in the initial situation, but they can be reactivated using a staged construction process. As is the case for structural objects, loads can be activated or deactivated by selecting the desired slice or plane and clicking once on the load in the geometry. Active loads are drawn in their original colour, whereas deactivated loads are drawn in grey.

When activating loads, the actual value of the load that is applied during a calculation is determined by the input value of the load and the corresponding load multiplier (*ΣMloadA* or *ΣMloadB*).

Input value of a load

By default, the input value of a load is the value as given during the geometry creation. The input value of the load may be changed in each calculation phase in the framework of Staged construction. This can be done by selecting the desired slice or plane and double clicking the load in the geometry. As a result, a load window appears in which the input values of the loads can be changed.

If a plane is selected and a point load is double clicked, the load is a true point force and the input window looks as follows:

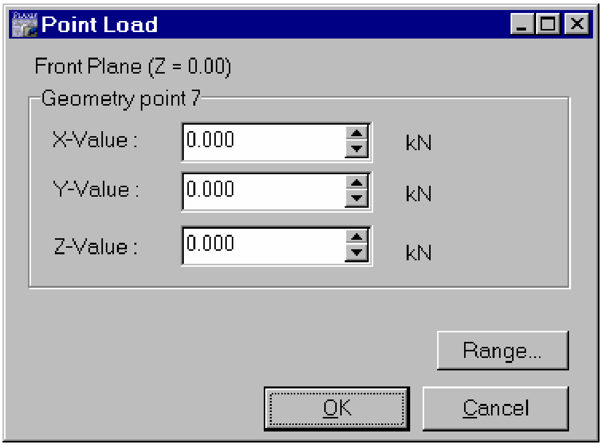


Figure 4.14 Input window for a point load on a *z*-plane

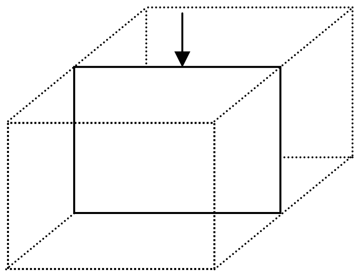


Figure 4.15 Point load on a *z*-plane

In this window, the *x*-, *y*- and *z*-components of the point force can be entered directly. The <Range> button may be used as a quick option to select a range of planes for which the currently entered input values of the load should apply. If a plane is selected and a distributed load is double clicked, the load is actually a line load and the input window looks as follows:

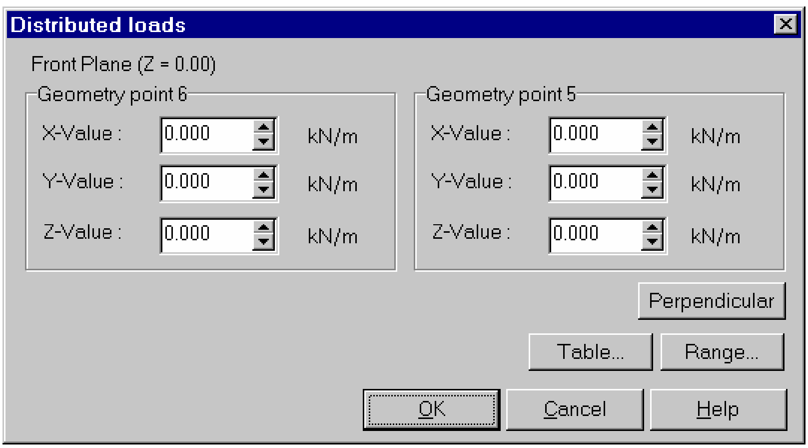


Figure 4.16 Input window for a line load on a *z*-plane

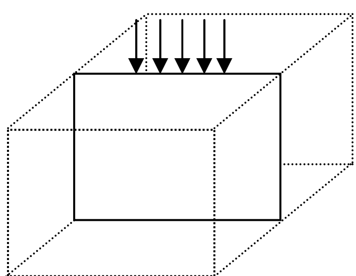


Figure 4.17 Line load on a *z*-plane

In this window, the *x*-, *y*- and *z*-components of the line load in the two respective geometry points can be entered directly. The <Perpendicular> button may be used to make sure that the line load is perpendicular to the corresponding geometry line in the plane, and the *z*-component is zero. If the line load is to be changed to the same input values in more than just the current plane, a range of planes can be selected by using the <Range> button. The <Table> button can be used to get an overview of the corresponding line loads on all planes or to change individual input values. If a slice is selected and a point load is double clicked, the load becomes a line load and the input window looks as follows:

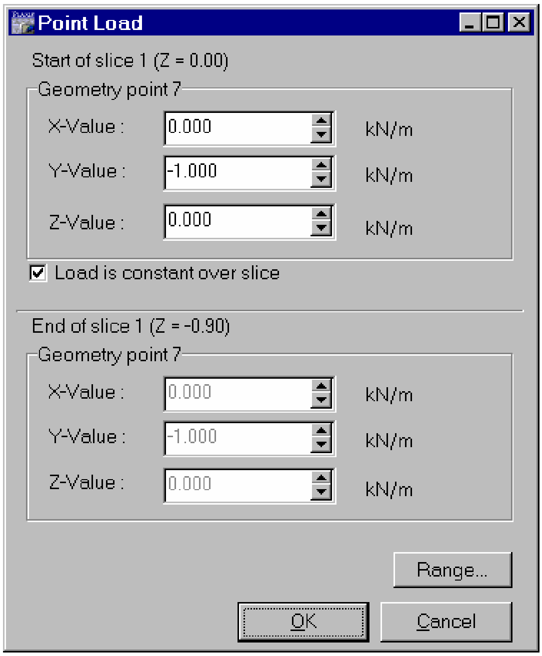


Figure 4.18 Input window for a point load on a slice for the case where the data are used to define a line load

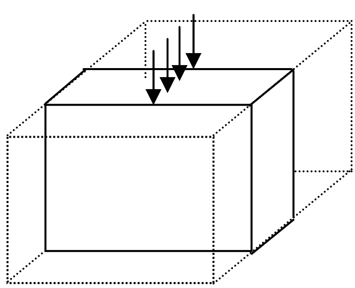


Figure 4.19 Line load on a slice

In this window the *x*-, *y*- and *z*-components of the line load in the two respective geometry points can be entered directly. The *load is constant over slice* check box may be used to make the input values of the second point equal to the first point. The <Range> button may be used as a quick option to select a range of slices for which the currently entered input values of the load should apply. If a slice is selected and a distributed load is double clicked, the load is distributed load on an area, and the input window looks as follows:

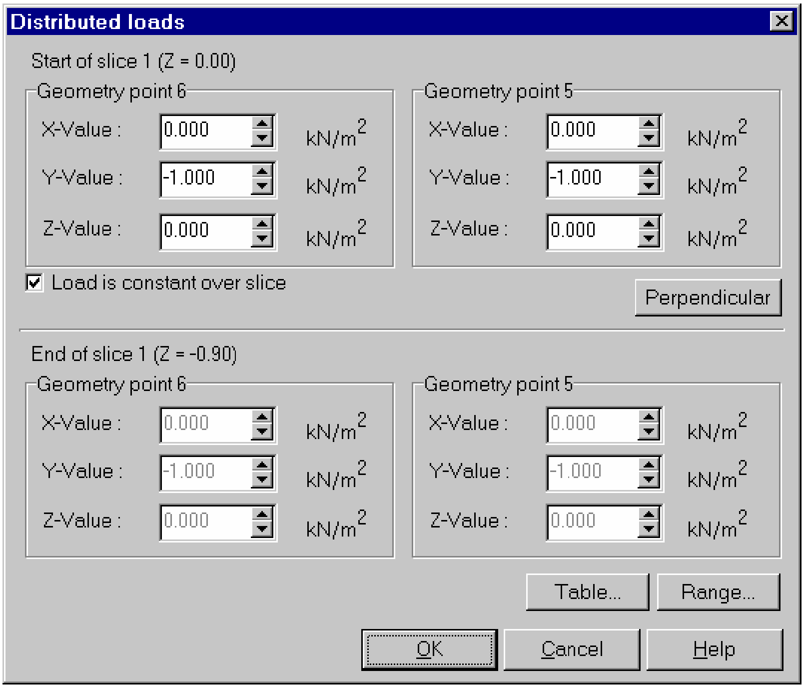


Figure 4.20 Input window for a distributed load on a slice

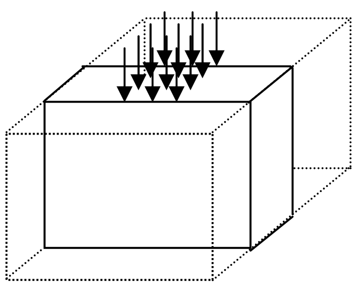


Figure 4.21 Distributed load on a slice

In this window the *x*-, *y*- and *z*-components of the distributed load at the four respective geometry points can be entered directly. The *load is constant over slice* check box may be used to make the input values of the lower two points equal to the upper two points. The <Perpendicular> button may be used to make sure that the distributed load is perpendicular to the corresponding geometry line in the slice, and the *z*-component is zero. If the distributed load is to be changed to the same set of input values in more than just the current slice, a range of slices can be selected by using the <Range> button. The <Table> button can be used to obtain an overview of the corresponding distributed loads on all slices or to change individual input values.

Load multiplier

The actual value of the load that is applied during a calculation is determined by the product of the input value of the load and the corresponding load multiplier (*ΣMloadA* or *ΣMloadB*). The multiplier *ΣMloadA* is used to globally increase (or decrease) all loads of load system A (point loads, line loads, distributed loads and *z*-loads), whereas *ΣMloadB* is used to change all loads of load system B (Section 4.7). However, in general it is not necessary to change the load multipliers when applying or changing loads by means of staged construction since the program will automatically set the corresponding multiplier to unity if it is zero. Note that if the existing value of the multiplier is not equal to zero and not equal to unity, the existing multiplier is retained and the load that is actually applied in the calculation is different from the input value of the load as entered in the staged construction mode.

4.6.5 APPLYING *Z*-LOADS

A *z*-load is a distributed load, with only a *z*-component, that acts on a whole soil cluster face in a *z*-plane. These loads can, for example, be used to model the support of the excavation face at the front of a tunnel.

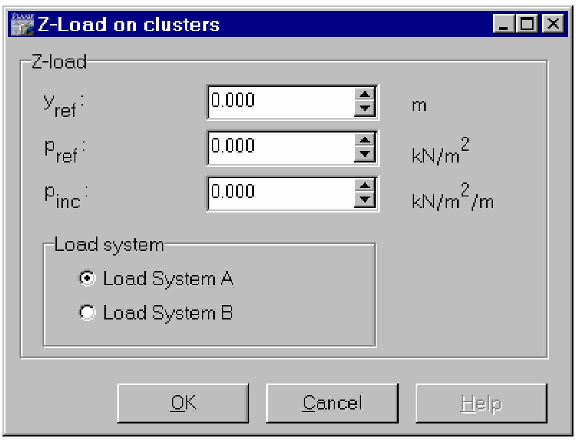


Figure 4.22 Input window for a *z*-load on a cluster

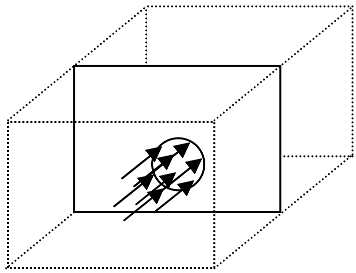


Figure 4.23 *Z*-load on a circular cluster

In contrast to other types of external loads, *z*-loads are not defined during the creation of a geometry model. To activate a *z*-load, the desired plane should be selected and the desired cluster on which the load is to be applied should be double clicked. As a result, the *z*-load input window appears in which the input values of the load can be entered.

Z-loads may linearly increase with depth. Therefore three input values are requested: a reference level (y_{ref}), a reference load at the reference level (p_{ref}) and an increase of the load per unit of depth (p_{inc}). Note that the sign of the load depends on the direction in which it acts: positive when acting in the positive *z*-direction (i.e. towards the user) and negative when acting in the negative *z*-direction. By default, the *z*-load is included in load system A, but the user may instead choose load system B. The actual value of the *z*-load that is applied during a calculation is determined by the product of the input value of the load and the corresponding load multiplier (*ΣMloadA* or *ΣMloadB*). However, in general it is not necessary to change the load multipliers when applying or changing *z*-loads by means of staged construction since the program will automatically set the corresponding multiplier to unity if it is zero. Note that if the existing value of the multiplier is not equal to zero and not equal to unity, the existing multiplier is retained and the load that is actually applied in the calculation is different from the input value of the load as entered in the staged construction mode.

4.6.6 APPLYING PRESCRIBED DISPLACEMENTS

Prescribed displacements that were created in the geometry input are not automatically applied during calculations, but they can be activated by means of a staged construction process. As long as prescribed displacements are not active, they do not impose any condition on the model. Hence, at parts of the model where prescribed displacements have been defined that are currently inactive, the nodes are fully free. Similar as for loads, prescribed displacements can be activated or deactivated by selecting the desired slice or plane and clicking once on the prescribed displacement in the geometry. Active prescribed displacements are drawn in their original colour, whereas inactive prescribed displacements are drawn in grey.

If it is desired to temporarily 'fix' the nodes where prescribed displacements are created, the input value of the prescribed displacement should be set to 0.0 rather than deactivating the prescribed displacement. In the former case a prescribed displacement of zero is applied to the nodes, whereas if the prescribed displacement is deactivated the nodes are left free.

When activating prescribed displacements, the actual value of the prescribed displacement that is applied during a calculation is determined by the input value of the prescribed displacement and the corresponding load multiplier ($\Sigma Mdisp$).

Input value of prescribed displacement

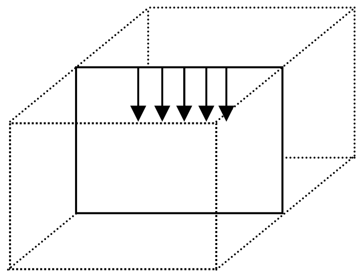


Figure 4.24 Prescribed displacements on a plane

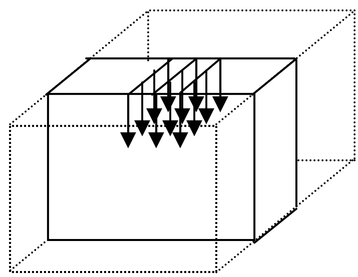


Figure 4.25 Prescribed displacements on a slice

By default, the input value of a prescribed displacement is the value given during the geometry creation. The input value of the load may be changed in each calculation phase using a staged construction procedure. This can be done by selecting the desired slice or plane and double clicking the prescribed displacement in the geometry. As a result, a prescribed displacement window appears in which the input values of the prescribed displacement can be changed.

When a prescribed displacement is applied on a slice, the prescribed displacement will act on an area, in the same way as a distributed load. When a prescribed displacement is applied on a plane, the prescribed displacement will act on a line in the 3D model, which is similar to the case of a line load.

Corresponding multiplier

The actual value of the prescribed displacement that is applied during a calculation is determined by the product of the input value of the prescribed displacement and the corresponding load multiplier ($\Sigma Mdisp$). The multiplier $\Sigma Mdisp$ is used to globally increase (or decrease) all prescribed displacements (Section 4.7). However, in general it is not necessary to change the multiplier when applying or changing prescribed displacements by means of a staged construction process since the program will automatically set the corresponding multiplier to unity if it is zero. Note that if the existing value of the multiplier is not equal to zero and not equal to unity, the prescribed displacement that is actually applied in the calculation is different from the input value of the prescribed displacement as entered in the staged construction mode.

4.6.7 REASSIGNING MATERIAL DATA SETS

The option to reassign material data sets 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.

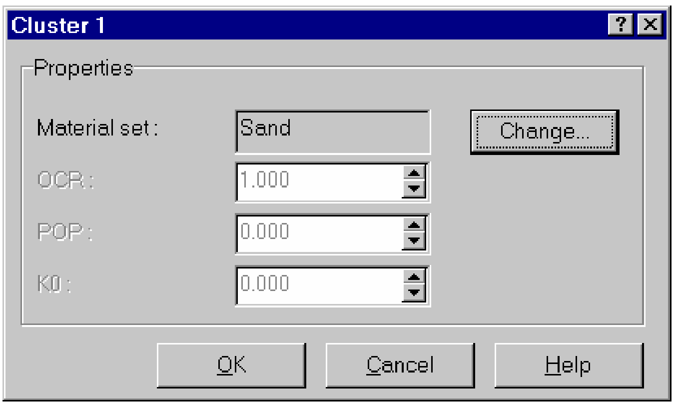


Figure 4.26 Soil properties window

On selecting the desired slice and double clicking a soil cluster or structural object in the geometry, the properties window appears in which the material data set of that object can be changed. Instead of changing the data in the material data set itself, another data set should be assigned to the cluster or object. This ensures the consistency of data in the material database. Hence, if it is desired to change the properties of a cluster during a calculation, an additional data set should be created during the input of the cross-section model.

The change of certain properties, for example when replacing peat by dense sand, can introduce substantial out-of-balance forces. These out-of-balance forces are solved during the staged construction calculation. This is the most important reason why the reassignment of material data sets is considered to be a part of a staged construction process.

If a change in the data set of a plate is considered it is important to note that a change the ratio *EI/EA* will change the equivalent thickness d_{eq} and thus the distance separating the stress points. If this is done when existing forces are present in the plate element, it would change the distribution of bending moments, which is unacceptable. For this reason, if material properties of a plate are changed during an analysis, it should be noted that the ratio *EI/EA* must remain unchanged.

4.6.8 APPLYING A VOLUMETRIC STRAIN IN VOLUME CLUSTERS

In the PLAXIS Tunnel program you can impose an internal volumetric strain in volume clusters. This option may be used to simulate mechanical processes that result in volumetric strains in the soil, such as grouting. In the selection window that appears after selecting the desired slice and double-clicking a soil cluster, you can select *Volumetric strain*.

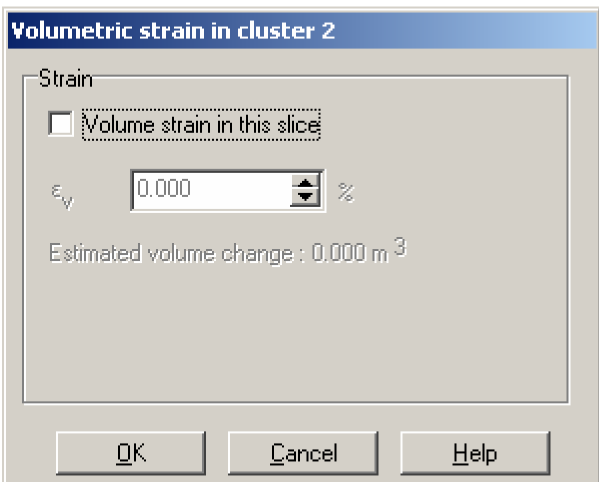


Figure 4.27 Volume strain window

The value of the volumetric strain that you want to apply to the full volume cluster can be entered in the resulting volume strain window. In contrast to other types of loading, volume strains are not activated with a separate multiplier.

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

A positive value of the volume strain represents a volume increase, whereas a negative value represents a volume decrease.

4.6.9 PRE-STRESSING OF ANCHORS

Pre-stressing of anchors can be activated from the geometry configuration mode of the staged construction window. Therefore the desired plane should be selected and the desired anchor should be double-clicked. As a result, the anchor properties window appears, which indicates by default a pre-stress force of zero. On selecting the *Adjust pre-stress force* check box it is possible to enter a value for the pre-stress force in the corresponding edit box. Note that tension is considered to be positive and compression is considered negative.

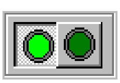
To deactivate a previously entered pre-stress force, the *Adjust pre-stress force* parameter must be deselected rather than setting the pre-stress force to zero. In the former case the anchor force will further develop based on the changes of stresses and forces in the geometry. In the latter case the anchor force will remain at zero, which is generally not correct. After the input of the pre-stress force the <OK> button should be pressed. As a result, the anchor properties window is closed and the geometry configuration mode is presented, where the pre-stressed anchor is indicated with a 'p'. During the staged construction calculation the pre-stressed anchor is automatically deactivated and a force equal to the pre-stress force is applied instead. At the end of the calculation the anchor is reactivated and the anchor force is initialised to match the pre-stress force exactly, provided that failure had not occurred. In subsequent calculations the anchor is treated as a spring element with a certain stiffness, unless a new pre-stress force is entered.

4.6.10 APPLYING CONTRACTION OF A TUNNEL LINING

To simulate soil volume loss due to the construction of a shield tunnel, the contraction method may be used. In this method a contraction is applied to all, or part, of the tunnel lining to simulate a reduction of the tunnel cross-section area. The contraction is expressed as a percentage, representing the ratio of the area reduction and the original outer tunnel cross-section area. Contraction can only be applied to circular tunnels (bored tunnels) with an active continuous homogeneous lining (see 3.3.7). Different values of contraction may be entered for individual *z*-planes. Contraction can be activated in the staged construction mode by selecting the desired plane and double clicking the centre point of a tunnel for which a contraction is to be specified. As a result, the contraction window appears, in which an input value of contraction can be entered. In contrast to other types of loading, contraction is not activated with a separate multiplier.

Note that the entered value of contraction is not always fully applied, depending on the stiffness of the surrounding clusters and objects. The computed contraction can be viewed in the output program (Section 5.9.4)

4.6.11 CHANGING WATER PRESSURE DISTRIBUTION



In addition to, or instead of, a change in the geometry configuration, the water pressure distribution in the geometry may be changed. An example of a problem that may be analysed using this option includes the deformation and force development of a tunnel lining due to excavation and dewatering of the tunnel.

Click the 'switch' in the toolbar to display the water conditions mode and to change the water pressure distribution. The window displays the current situation with an indication of the phreatic level or boundary conditions for a groundwater flow calculation. Phreatic levels (i.e. the general phreatic level and user-defined phreatic levels) can be changed when a plane is selected, and their setting applies to all planes in the 3D model. A new set of phreatic levels may be created when any plane is selected. For a description of the input of phreatic levels, reference is made to Section 3.8.

Instead of allowing for a different set of phreatic levels for individual slices, the 3D Tunnel program allows for the pore pressure distribution to be defined for individual volume clusters. This can be done by selecting the desired slice (in the water conditions mode) and double-clicking on the desired cluster. As a result, the user-defined pore pressure distribution window appears in which the pore pressure situation can be defined.

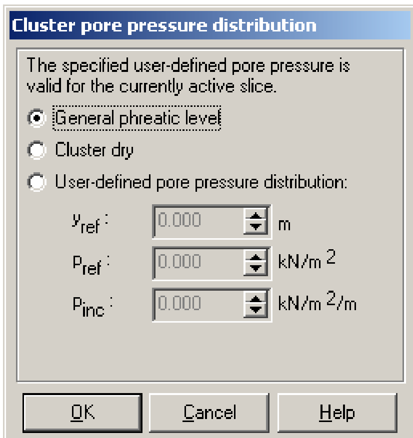


Figure 4.28 User-defined pore pressure distribution window

After selecting *User defined pore pressure distribution* in the Pore pressure window two options will come available. Select *Cluster dry* if all pore pressures in the selected cluster are supposed to be zero. To define a user-defined pore pressure distribution the user should enter the appropriate values for y_{ref}, p_{ref}, and p_{inc}. The meaning of these parameters is the same as with *z*-loads (Section 4.6.5), except that positive values for p_{ref}, and p_{inc} represent tension, whereas negative values represent compression.

After the new input, the water pressures must first be generated before pressing the <Update> button. The water pressures can be generated by clicking on the *Generate water pressures* button in the toolbar or by selecting the *Water pressures* option from the *Generate* menu. After the generation, the new water pressure distribution is presented in the Output program. By clicking on the <Update> button in the Output program, this program is closed and the Input program reappears. By subsequently clicking on the <Update> button in the Input program, the staged construction window is closed and the Calculation program reappears. The *Water* column of the calculation list now shows the number of the current phase to indicate that the water conditions have changed in this phase.

PLAXIS 3D TUNNEL can also make changes in the water pressure distribution as a consequence of a change in the hydrodynamic boundary conditions, which results in steady-state groundwater flow, see Section 3.8.3.

4.6.12 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 Staged Construction window. This enables a direct visual check before the calculation is started. After the preview, press the Update button to return to the Staged Construction mode. If the situation is satisfactory, press the Update button to return to the Calculation program.

4.6.13 PLASTIC NIL-STEP

Staged construction 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. 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. Hence, after selecting Staged Construction as the loading input in the Parameters tab sheet of the Calculations program, the next calculation phase can be defined or the calculation can be started.

4.6.14 STAGED CONSTRUCTION WITH ΣMSTAGE<1

In general, the total multiplier associated with the staged construction process, *ΣMstage*, goes from zero to unity in each calculation phase where staged construction has been selected as the loading input. In some very special situations it may be useful to perform only a part of a construction stage. This can be done by clicking on the <Advanced> button in the *Parameters* tab sheet and specifying an ultimate level of *ΣMstage* smaller than 1.0. The lowest allowed input value is 0.001. If *ΣMstage* is lower than this value, the load is considered to be negligible and no calculations take place. A value larger than 1.0 is not normally used. By entering the default value of 1.0, the staged construction procedure is performed in the normal way.

In general, care must be taken with an ultimate level of *ΣMstage* smaller than 1.0, since this leads to a resulting out-of-balance force at the end of the calculation phase. Such a calculation phase must always be followed by another staged construction calculation. If *ΣMstage* is not specified by the user, the default value of 1.0 is always adopted, even if a smaller value was entered in the previous calculation phase.

4.6.15 UNFINISHED STAGED CONSTRUCTION CALCULATION

At the start of a staged construction calculation, the multiplier that controls the staged construction process, *ΣMstage*, is zero and this multiplier is stepwise increased to the ultimate level (generally 1.0). When *ΣMstage* has reached the ultimate level, the current phase is finished. However, if a staged construction calculation is not properly finished, i.e. the multiplier *ΣMstage* is less than the desired ultimate level at the end of a staged construction analysis, then a warning appears in the *Log info* box.

There are two possible 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.

4.7 LOAD MULTIPLIERS

During a deformation analysis, it is necessary to control the magnitude of all types of loading. In general, loadings are controlled using a system of multipliers. The loadings to be applied are calculated from the product of the input value of the load and the corresponding multiplier. Distinction is made between *Incremental multipliers* and *Total multipliers*.

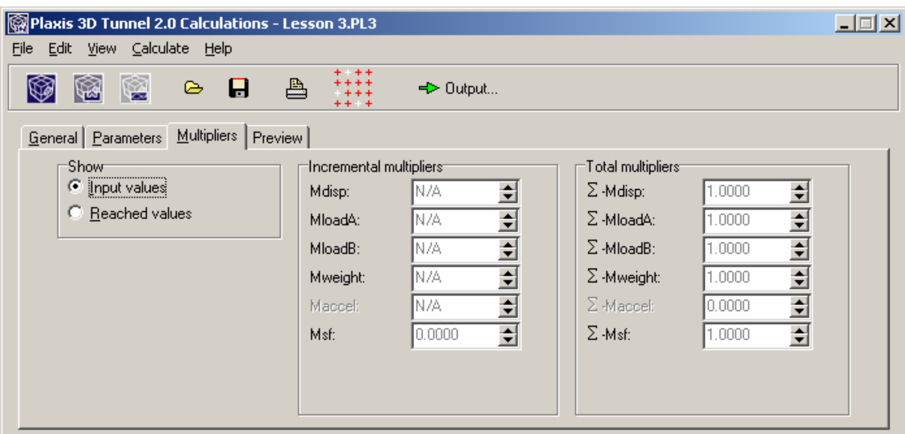


Figure 4.29 *Multipliers* tab sheet of the Calculations window

Incremental multipliers represent the increment of load for an individual calculation step, whereas total multipliers represent the total level of the load in a particular calculation step or phase. The way in which the various multipliers are used depends on the type of calculation being performed (4.7.2). Both the incremental multipliers and the total multipliers for a particular calculation phase are displayed in the *Multipliers* tab sheet (Figure 4.29). All incremental multipliers are denoted by *M*... whereas all total multipliers are denoted by *ΣM*.... A multiplier does not have a unit associated with it, since it is just a factor. Descriptions of the various load multipliers are given below.

4.7.1 STANDARD LOAD MULTIPLIERS

Mdisp, ΣMdisp:

These multipliers control the magnitude of prescribed displacements as entered in the staged construction mode (see 4.6.6). The total value of the prescribed displacement applied in a calculation is the product of the corresponding input values as entered in the staged construction mode and the parameter *ΣMdisp*. When applying prescribed displacements by entering an input value of prescribed displacement in the staged construction mode, and the value of *ΣMdisp* is still zero, *ΣMdisp* is automatically set to unity. The value of *ΣMdisp* may be used to globally increase or decrease the applied prescribed displacement. In calculations where the *Loading input* was set to *Incremental multipliers*, *Mdisp* is used to specify a global increment of the prescribed displacement in the first calculation step.

MloadA, ΣMloadA, MloadB, ΣMloadB:

These multipliers control the magnitude of the distributed loads, line loads, point loads and *z*-loads as entered in the load systems A and B (Section 3.4.4, 3.4.5, 4.6.4 and 4.6.5). The total value of the loads of either load system applied in a calculation is the product of the corresponding input values as entered in the staged construction mode and the parameter *ΣMloadA* or *ΣMloadB* respectively. When applying loads by entering an input value of load in the staged construction mode, and the value of the corresponding multiplier is still zero, this multiplier is automatically set to 1.0. The values of *ΣMloadA* and *ΣMloadB* may be used to globally increase or decrease the applied load. In calculations where the *Loading input* was set to *Incremental multipliers*, *MloadA* and/or *MloadB* are used to specify a global increment of the corresponding load systems of the first calculation step.

Mweight, ΣMweight:

It is possible in PLAXIS to carry out calculations in which gravity loading is applied to the problem. The multipliers *Mweight* and *ΣMweight* control the proportion of standard gravity applied in the analysis and thus the portion of the material weights (soil, water and structures) as specified in the Input program. The total proportion of the material weights applied in a calculation is given by the parameter *ΣMweight*. In calculations where the *Loading input* was set to *Incremental multipliers*, *Mweight* is used to specify the increment of weight in the first calculation step.

The multiplier is applied to the material weights as well as to the water weight. Hence, if *ΣMweight* is zero then the soil weight is not taken into account and all water pressures (excluding eventual excess pore pressures generated during undrained loading) will also be zero. If *ΣMweight* is set to 1.0 then the full soil weight and water pressures will be applied. A value of *ΣMweight* larger than 1.0 is generally not used, except for the simulation of a centrifuge test.

In contrast to most other total multipliers, *ΣMweight* can have a value larger than zero at the start of the Calculations program. This is the case if the K_0-procedure for the generation of the initial stress field is used in the initial conditions mode of the Input program. In this case, by default, the value of *ΣMweight* is set to 1.0 at the start of the analysis to account for the full soil weight and water pressures.

Msf, ΣMsf:

These multipliers are associated with the *Phi-c reduction* option in PLAXIS for the computation of safety factors (Section 4.8). The total multiplier *ΣMsf* is defined as the quotient of the original strength parameters and the reduced strength parameters and controls the reduction of the $\tan\varphi$ and c at a given stage in the analysis. In contrast to most other total multipliers, *ΣMsf* is set to 1.0 at the start of a calculation to set all material strengths to their unreduced values. *Msf* is used to specify the increment of the strength reduction of the first calculation step. This increment is by default set to 0.1, which is generally found to be a good starting value.

Input values and reached values:

The input values of the multipliers might differ from the values that were actually reached after the calculation. The radio buttons in the *Show* group can be used to display either the *Input values* or the *Reached values*.

If the *Reached values* option is selected another group box appears in which some other multipliers and calculation parameters are displayed.

Extrapolation

If the prescribed values of the multipliers are such that a combination of loads, as activated in the previous calculation phase, is continued in the current calculation phase, then the calculation will recognise this situation and will automatically use the extrapolation procedure, as described in Section 4.5.1.

4.7.2 OTHER MULTIPLIERS AND CALCULATION PARAMETERS

Time increment, Time interval, Realised end time, Estimated end time:

These parameters control the time in the calculations. The *Time interval* represents an increment of time and the *Realised end time* represents the actual time. In contrast to the multipliers, *Time interval* and the *Realised end time* are expressed in the unit of time. Time increments are only relevant when using time dependent material models (such as the creep model).

- Time increment is the increment of time considered in the current calculation phase.
- 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.

ΣMstage:

The *ΣMstage* parameter is associated with the *Staged construction* option in PLAXIS (Section 4.6). This total multiplier gives the proportion of a construction stage that has been completed. Without input from the user, the value of *ΣMstage* is always zero at the start of a staged construction analysis and at the end it will generally be 1.0. It is possible to specify a lower ultimate level of *ΣMstage* using the <Advanced> option of the *Parameters* tab sheet. However, care should be taken with this option. In calculations where the loading input is not specified as *Staged construction*, the value of *ΣMstage* remains zero.

ΣMarea:

The *ΣMarea* parameter is also associated with the *Staged construction* option. This parameter gives the proportion of the total volume of soil clusters in the 3D model that is currently active. If all soil clusters are active then *ΣMarea* has a value of 1.0.

Stiffness:

As a structure is loaded and plasticity develops then the overall stiffness of the structure will decrease. The *Stiffness* parameter gives an indication of the loss of stiffness that occurs due to material plasticity. The parameter is a single number that is 1.0 when the structure is fully elastic and reduces in magnitude as plasticity develops.

At failure the value is approximately zero. It is possible for this parameter to have negative values if softening occurs.

Force-X, Force-Y, Force-Z:

These parameters indicate the forces corresponding to the non-zero prescribed displacements (Section 3.4.1), expressed in the unit of force. *Force-X, Force-Y* and *Force-Z* are the value of the total force in the *x*-, *y*- and *z*-directions respectively, applied to non-zero prescribed displacements.

Pmax:

The *Pmax* parameter is associated with undrained material behaviour and represents the maximum absolute excess pore pressure in the mesh, expressed in the unit of stress. During undrained loading in a plastic calculation *Pmax* generally increases.

4.8 PHI-C-REDUCTION

Phi-c reduction is an option available in PLAXIS to compute safety factors. This option is only available for *Plastic* calculations using the *Manual control* or the *Load advancement number of steps* procedure. In the *Phi-c reduction* approach the strength parameters $\tan\varphi$ and *c* of the soil are successively reduced until failure of the structure occurs. The strength of interfaces, if used, is reduced in the same way. The strength of structural objects like beams and anchors is not influenced by *Phi-c reduction*.

The total multiplier ΣMsf is used to define the value of the soil strength parameters at a given stage in the analysis:

$$\sum Msf = \frac{\tan\varphi_{input}}{\tan\varphi_{reduced}} = \frac{c_{input}}{c_{reduced}}$$

where the strength parameters with the subscript 'input' refer to the properties entered in the material sets and parameters with the subscript 'reduced' refer to the reduced values used in the analysis. In contrast to most other total multipliers, ΣMsf is set to 1.0 at the start of a calculation to set all material strengths to their unreduced values. The strength parameters are successively reduced automatically until failure of the structure occurs. At this point the factor of safety is given by:

$$SF = \frac{available\ \ strength}{strength\ \ at\ \ failure} = \text{value of} \sum Msf \text{ at failure}$$

This approach resembles the method of calculation of safety factors conventionally adopted in slip-circle analyses. When using *Phi-c reduction* in combination with advanced soil models, these models will actually behave as a standard Mohr-Coulomb model, since stress-dependent stiffness behaviour and hardening effects are excluded.

The stress-dependent stiffness modulus (where this is specified in the advanced model) at the end of the previous step is used as a constant stiffness modulus during the phi-c reduction calculation.

To capture the failure of the structure accurately, the use of *Arc-length control* in the iteration procedure is recommended. The use of a *Tolerated error* of no more than 3% is also recommended. Both recommendations are complied with when using the *Standard setting* of the *Iterative procedure*. For a detailed description of the method of *Phi-c reduction* see Reference 4.

4.9 UPDATED MESH ANALYSIS

In conventional finite element analysis, the effect of a geometry change of the mesh on the equilibrium conditions is assumed to be negligible. This approximation is usually appropriate when the deformations are relatively small, as is the case for most engineering applications. However, there are circumstances under which it is necessary to take this effect into account. Typical applications in which updated mesh analyses may be necessary include the analysis of reinforced soil structures, the analysis of large offshore footing collapse problems and the study of problems where soils are soft and large deformations occur.

When large deformation theory is included in a finite element program, some special features need to be considered. First, it is necessary to include additional terms in the structure stiffness matrix to model the effects of large structural distortions on the finite element equations.

Second, it is necessary to include a procedure to correctly model the stress changes that occur when finite material rotations occur. This particular feature of large displacement theory is usually solved by adopting a definition of stress rate that includes rotation rate terms. Several stress rate definitions have been proposed by researchers working in this field, although none of these are completely satisfactory. In PLAXIS the co-rotational rate of Kirchhoff stress (otherwise known as the Hill stress rate) is adopted. This stress rate would be expected to give accurate results provided that the shear strains do not become excessive.

Third, it is necessary to update the finite element mesh as the calculation proceeds. This is done automatically in PLAXIS when you choose the *Updated mesh* option.

It should emphasised that the updated mesh procedures used in PLAXIS involve considerably more than simply updating nodal coordinates as the calculation proceeds. These calculation procedures are in fact based on an approach known as an Updated Lagrangian formulation (Reference 24). Implementation of this formulation in PLAXIS is based on the use of various advanced techniques that are beyond the scope of this manual (Reference 30).

Calculation procedures

To perform an updated mesh analysis, click the *Advanced* button in the *Calculation type* box on the *General* tab. The *Advanced general settings* window is then displayed, in which you choose the *Updated mesh* option. Updated mesh calculations are performed using iteration procedures similar to those used for plasticity calculations. Therefore, an updated mesh analysis uses the same parameters as those used for plasticity analysis in PLAXIS. However, because of the large deformation effect, the stiffness matrix is always updated at the beginning of a load step. Due to this updating, and also due to the additional terms involve in the mathematical formulations, the iterative procedure in an updated mesh analysis is considerably slower than that for conventional plasticity analysis.

Practical considerations

Updated mesh analysis tends to require more computer time than an equivalent, conventional, plasticity calculation. It is recommended, therefore, that when a new project is under study, a conventional plasticity calculation is performed before an updated mesh analysis is attempted.

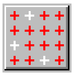
It is not possible to give simple guidelines to indicate when an updated mesh analysis is necessary and where a conventional analysis is sufficient. One simple approach would be to inspect the deformed mesh at the end of a conventional calculation using the *Deformed mesh* option in the Output program. If the geometry changes are relatively large (on a real scale!), it may be necessary to repeat the calculation using the updated mesh option. It cannot definitely be decided from the general magnitudes of the deformations obtained from a conventional plasticity calculation whether geometric effects are important or not. If you are in any doubt about whether updated mesh analysis is necessary, you can only resolve the issue by performing the updated mesh analysis and comparing the results with the equivalent conventional analysis.

In general, it is not appropriate to use an updated mesh calculation for gravity loading to set up the initial stress field. Displacements resulting from gravity loading are physically meaningless and should therefore be reset to zero. It is not possible to reset displacements to zero after an updated mesh analysis. Therefore, gravity loading should be applied in a normal plastic calculation.

Changing from a 'normal' plastic calculation or consolidation analysis to an updated mesh analysis is only valid when displacements are reset to zero, because the updated mesh analysis must start from an un-deformed geometry. Changing from an updated mesh calculation to a 'normal' plastic calculation or consolidation analysis is not valid, because then all large deformation effects will suddenly be disregarded.

4.10 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 calculations 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.



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 front plane of the 3D finite element mesh with all nodes. Other planes of the 3D mesh may be selected by clicking on the tabs above the output window. 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. Selected nodes are indicated by characters in alphabetical order. 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.

In addition to the nodes, stress points may be selected for the generation of stress paths, strain path and stress-strain diagrams. On clicking on the *Select stress points for stress/strain curves* button in the upper right corner, the plot shows all stress points near the plane corresponding to the active tab sheet. Stress points in front of the plane (higher *z*-coordinate than the plane) are given light purple colour, whereas stress points behind the plane (lower *z*-coordinate) are given a dark purple colour. Up to 10 stress points may be selected for the generation of curves of stresses and strains. As for the nodes, the stress points are indicated by characters in alphabetical order.

If it is desired to select additional nodes, then the *Select nodes for load-displacement curves* button may be selected, after which the plot with nodes reappears and additional nodes can be selected. However, when additional nodes are selected, the calculation process must be executed again from the first calculation phase in the list. To deselect all selected nodes, the *Deselect all nodes or stress points* button may be clicked. If this button is clicked when the plot of the nodes is presented, then only the nodes will be deselected whereas the selected stress points remain. On the other hand, if this button is clicked when the plot of the stress points is presented, then only the stress points will be deselected whereas the selected nodes remain.

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 Calculations program.

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, then 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.

4.11 EXECUTION OF THE CALCULATION PROCESS

When calculation phases have been defined and points for curves have been selected, then the calculation process can be executed. Before starting the process, however, it is useful to check the calculation list. 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 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. Alternatively, the right hand mouse button may be pressed on the corresponding line and the option *Mark calculate* or *Unmark calculate* should be selected from the cursor menu.

Starting the calculation process

The calculation process can be started by pressing the <Calculate> button in the toolbar. This button is only visible if a calculation phase is focused that is selected for execution, as indicated by the blue arrow. Alternatively, the *Current project* option can be selected from the *Calculate* menu. 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.

Multiple projects

In addition to the execution of the calculation process of the current project it is possible to select more projects for which calculations have to be executed subsequently. This can be done by selecting the *Multiple projects* option from the *Calculate* menu.

As a result the file requester appears from which the desired project can be selected. All projects for which calculations are to be executed appear in the *Calculation manager* window.

The calculation manager

The *Calculation manager* window can be opened by selecting the corresponding option from the *View* menu. The window shows the status of all projects for which calculations have been executed or are to be executed. An example is presented in Figure 4.30.

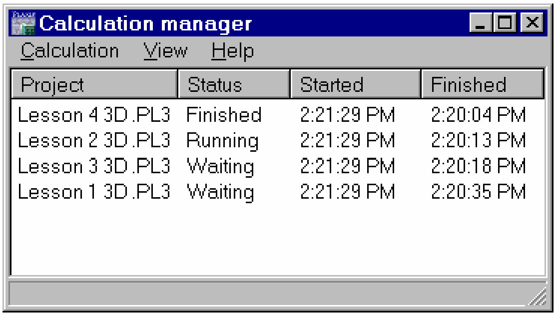


Figure 4.30 Calculation manager window

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.

4.12 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 values of the total load multipliers and other parameters for the particular calculation phase. The significance of load multipliers and some other parameters was described in Section 4.7. In addition, the following information is presented in the window:

Load-displacement curve:

During a calculation phase a small load-displacement curve is presented from which the status of the geometry (between fully elastic and failure) can be estimated. By default, the displacement of the first preselected node is plotted against the total multiplier of the activated load system. In the case that prescribed displacements are activated, the major force parameter (*Force-X, Force-Y* or *Force-Z*) is displayed instead of the *ΣMdisp* multiplier.

If desired, one of the other preselected nodes may be chosen from the combo box under the curve.

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. The

Maximum iterations value corresponds to the *Maximum iterations* parameter in the settings for the iterative procedure.

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 4.15.

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. The iteration process will at least continue as long as the global error is larger than the tolerance. For details see Section 4.15.

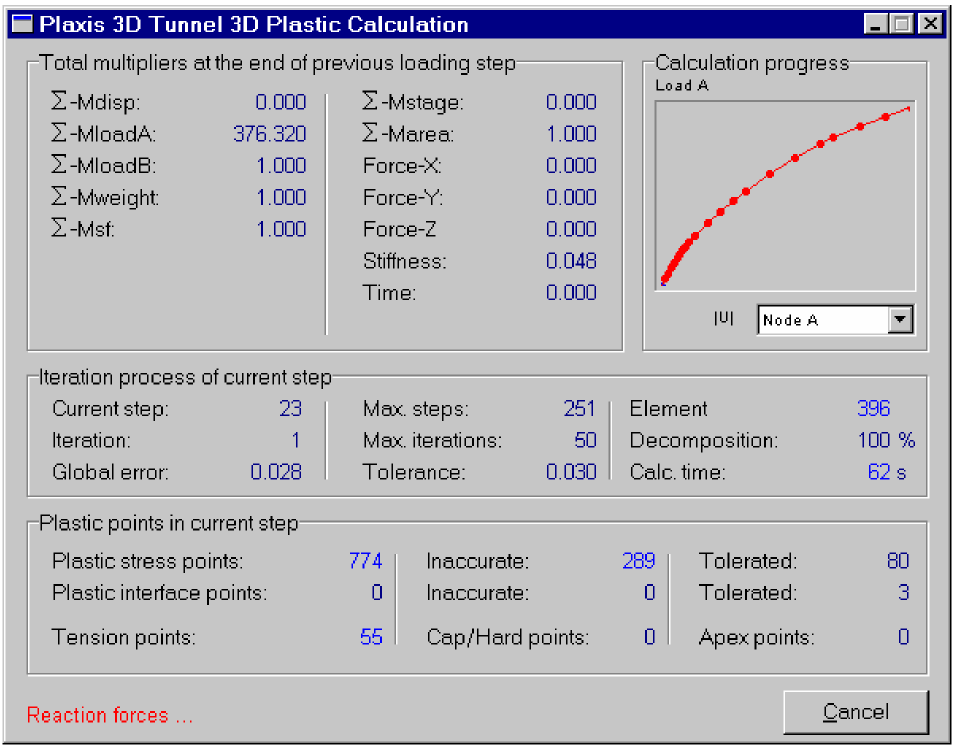


Figure 4.31 Calculation window.

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 4.15.

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 4.15.

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 3.5.2.

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 $\leq$ 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 (# MB) ...*
  when forming the global stiffness matrix. The matrix size is indicated.
- *Decomposing matrix (# m # s) ...*
  when decomposing the global stiffness matrix. This process generally takes quite some time. An estimate of this time is given.
- *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 calculations window. By pressing this button, the calculation process is aborted and control is returned to calculations part 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.

4.13 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.

When a calculation phase is focused that has been executed, 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. The user can select multiple calculation phases at the same time by holding down the <Shift> key on the keyboard while selecting phases. When subsequently clicking on the <Output> button, the results of all selected phases are displayed in separate windows in the Output program. In this way, results of different calculation phases can easily be compared.

4.14 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 some 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 and the initial conditions (i.e. water pressures and initial stresses) have to be regenerated. In the Calculation program, the user has to redefine the construction stages and the calculation process must restart from the first 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 regenerate the initial conditions and to restart the calculation from the first 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. However, this procedure is not very useful, since PLAXIS allows for a change of data sets within the *Staged construction* calculation option. Hence, it is better to create the data sets that will be used in later calculation phases beforehand and to use the *Staged construction* option to change data sets during calculations. The same applies to a change in water pressures and a change in input values of existing loads, since the latter is also possible using the *Staged construction* option.

4.15 AUTOMATIC ERROR CHECKS

During each calculation step, the PLAXIS Tunnel program 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-equilibrium errors at any stage during the iterative process automatically. Two separate error indicators are used for this purpose. One of these is based on a measure of the global equilibrium error and the other is a local error check. The values of both of these indicators must be below predetermined limits for the iterative procedure to terminate. These two error indicators and the associated error checking procedures are described below.

Global error check

The global error checking parameter used in PLAXIS 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 non-dimensionalised as shown below:

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

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 4.32. 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 4.32). 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 4.32.

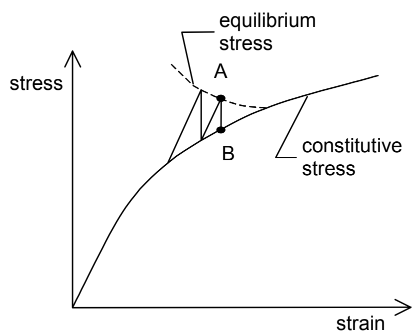


Figure 4.32 Equilibrium and constitutive stresses

The dashed line in Figure 4.32 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:

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

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

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

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:

$$Local\ error = \frac{\sqrt{\left(\sigma_{n}^{e} - \sigma_{n}^{c}\right)^{2} + \left(\tau_{1}^{e} - \tau_{1}^{c}\right)^{2} + \left(\tau_{2}^{e} - \tau_{2}^{c}\right)^{2}}}{c_{i} - \sigma_{n}^{c}\tan\varphi_{i}}$$

where $\sigma_{n,}$ τ_1 and τ_2 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 4.5.1).

Termination of iterations

For PLAXIS to terminate the iterations in the current calculation 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 at the nodes and the stresses at the stress points. 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 Tunnel 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 cross-section planes (or *z*-planes) by selecting corresponding tab sheets in the Output window.

The main window of the Output program contains the following items (Figure 5.1)

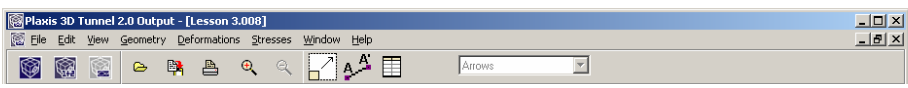


Figure 5.1 Toolbar in main window of the Output program

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 *z*-plane as generated by the 3D mesh extension. By default, the full 3D model is presented (most right-hand tab sheet). To view results in a particular *z*-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 *z*-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.
(*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 *z*-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. 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.
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
Grid	To show or hide the grid in the active plot.
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.
Shadow	To activate or deactivate the option that object surfaces appear 'lighter' or 'darker', depending on their orientation with respect to the viewer. When this option is active, object surfaces appear most light 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 5.9.1).
General info	To view the general project information (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.
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 fixities in the model.
Presc. displacements	To display the prescribed displacements in the model.
Connectivity plot	To view the connectivity plot (Section 5.9.5)
Elements	To display the soil elements in the model (only for *z*-planes).
Nodes	To display the nodes in the model (only for *z*-planes).
Stress points	To display the stress points in the model (only for *z*-planes).
Element numbers	To display the soil element 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 deformations and strains in the finite element model (Section 5.4).

The Stresses sub-menu:

The *Stresses* sub-menu contains various options to visualise the stress state in the finite element model (Section 5.5).

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 Tunnel project file (*.PL3) can be selected (Figure 5.2).

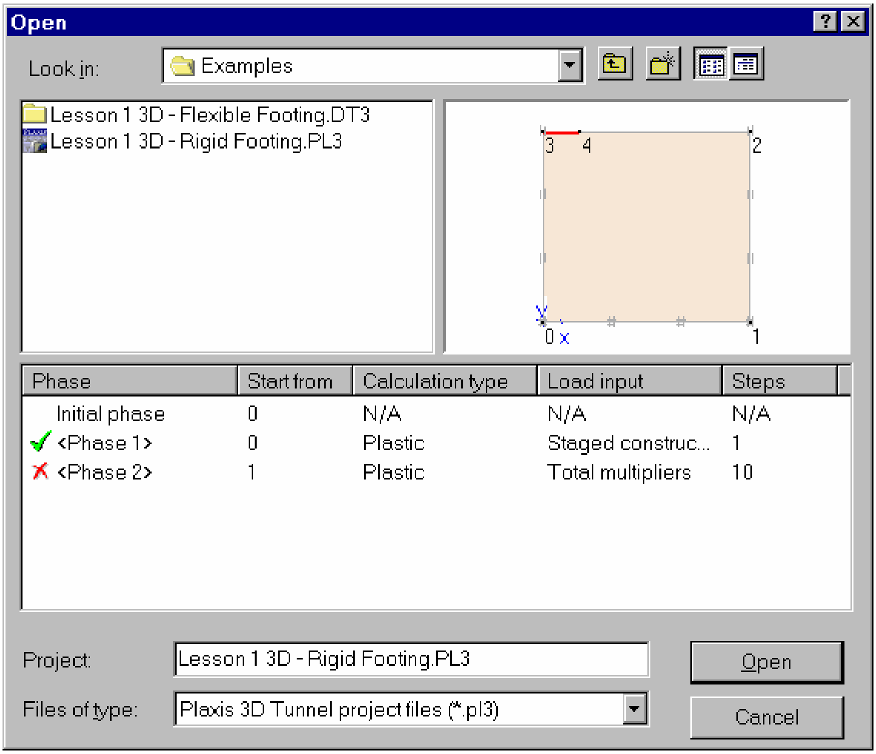


Figure 5.2 File requester for the selection of an output step

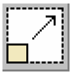
When the user selects a particular project, the file requester displays the corresponding list of calculation phases from which a further selection should be made. On selecting a calculation phase, a new output form is opened in which the results of the final calculation step of the selected phase are 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 step numbers, from which the desired step number can be selected.

In addition to this general selection of output data, an alternative option is provided by the Calculation program, as described in Section 4.13.

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. This plot may be selected from the *Deformations* sub-menu. If it is desired to view the deformations on the true scale (i.e. the geometry scale), then the *Scale* option may be used.

5.4.2 TOTAL DISPLACEMENTS

The *Total displacements* are the total 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 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 as *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the toolbar. When selecting *Contours* or *Shadings*, the geometry is plotted on the deformed shape.

5.4.3 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 displacement increments, |Δu|, and the individual displacement increment components, Δu_x, Δu_y and Δu_z. The displacement increments may be presented as *Arrows* or as *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the toolbar. When selecting *Contours* or *Shadings*, the geometry is plotted on the deformed shape. The contours of displacement increment are particularly useful for the observation of localisation of deformations within the soil when plastic failure occurs.

5.4.4 TOTAL STRAINS

The *Total strains* are the 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 a 3D plot 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 cross-section plane is selected, the presentation box 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 the total strain in the triangle. The length of each line represents the magnitude of the principal strain and the direction indicates the principal direction. Strains that represent extension are indicated by an arrow rather than a line. Note that compression is considered to be negative.

5.4.5 CARTESIAN STRAINS

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

5.4.6 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 a 3D plot 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.

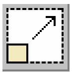
When a cross-section plane is selected, the presentation box also allows for the presentation of principal strain increment directions. These are, in fact, principal strain increments projected on the selected plane. Only one point per triangular element is displayed, which represents the average of 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.

5.4.7 CARTESIAN STRAIN INCREMENTS

When selecting *Cartesian strain increments* 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. 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 scaling 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 stresses refers to a reference value of stress that is drawn as a certain percentage of the geometry dimensions. To compare stress plots of different calculation phases, the scale factors in the different plots must be made equal.

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 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, a further selection can be made between effective mean stresses (p') or relative shear stresses (τ_{rel}), 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, τ_{rel}, is defined as:

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

where τ is the maximum value of shear stress (i.e. the radius of the Mohr stress circle). The parameter τ_{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 intermediate principal stress constant.

When a cross-section plane is selected, the presentation 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 principal stress and the direction indicates the principal direction. Stresses that represent tension are indicated by an arrow rather than a line. Note that compression is considered to be negative.

5.5.2 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, a further selection can be made between mean stresses (p) or deviatoric stresses (q), which can be represented as *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the toolbar.

When a cross-section plane is selected, the presentation 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 total principal stress in the triangle. The length of each line represents the magnitude of the principal stress and the direction indicates the principal direction. Stresses that represent tension are indicated by an arrow rather than a line. Note that compression is considered to be negative.

5.5.3 CARTESIAN STRESSES

When selecting *Cartesian stresses* from the *Stresses* sub-menu, a further selection can be made between the individual effective stress components σ'_{xx} , σ'_{yy} , σ'_{zz}, σ'_{xy}, σ'_{yz} and σ'_{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.

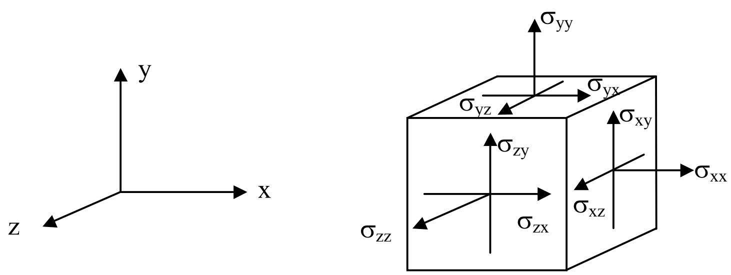


Figure 5.3 Sign convention for stresses

5.5.4 OVERCONSOLIDATION RATIO

The *Overconsolidation ratio* is only displayed if the Hardening Soil model or the Soft Soil Creep model is used.

The overconsolidation ratio, *OCR*, as defined in this option, is the ratio between the isotropic preconsolidation stress, p_p, and the current equivalent isotropic stress p^{eq}.

$$OCR = \frac{p_p}{p^{eq}}$$

where

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

$$p^{eq} = \sqrt{\left(p'\right)^2 + q^2/M^2} \qquad \text{(Hardening Soil model)}$$

The overconsolidation ratio may be displayed as *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the toolbar.

5.5.5 PLASTIC POINTS

The *Plastic points* are the stress points in a plastic state, displayed in a plot of the undeformed geometry. Plastic points are only shown in the *z*-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 indicates that the stresses lie on the surface of the Coulomb failure envelope. A white solid square indicates that the tension cut-off criterion was applied. A blue crossed square 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. For details of the use of advanced soil models, the user is referred to the Material Models manual.

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.

5.5.6 ACTIVE PORE PRESSURES

The *Active pore pressure* is the total water pressure p_w which is equal to the steady-state pore pressure + the excess pore pressure. The calculation results are plotted on an undeformed mesh at the end of the current calculation step. You can select this plot from the *Stresses* submenu.

In a 3D mesh, the default display of the active pore pressure uses *Shadings*. As an alternative, you can select *Contours* of active pore pressures from the *Presentation* combo box.

In a 2D plane, the active pore pressure is presented as that for the principal stresses. The length of lines represents the magnitude of the active pore pressure and the directions coincide with the *x*-axis and *y*-axis. Active pore pressures that are tensile are indicated by an arrow rather than a line. Note that pressure is considered to be negative. As an alternative to the *Principal directions*, you can select *Contours* or *Shadings* of active pore pressures from the *Presentation* combo box.

5.5.7 EXCESS PORE PRESSURES

When consolidation analysis is performed, *Excess pore pressure* will be calculated in the undrained soil layers. The calculation results are plotted on an un-deformed mesh at the end of the current calculation step. You can select the plot of the excess pore pressure from the *Stresses* submenu.

By default, the excess pore pressures are displayed as those for the principal stresses, although they do not have any principal direction. The length of the lines represents the magnitude of the excess pore pressure and the directions coincide with the *x*-axis and *y*-axis. Excess pore pressures that are tensile are indicated by an arrow rather than a line. Note that pressure is considered to be negative.

As an alternative to the *Principal directions*, you can select *Contours* or *Shadings* of excess pore pressures from the *Presentation* combo box.

5.5.8 GROUNDWATER HEAD

The groundwater head is defined as:

$$h = y + \frac{p}{\gamma_w}$$

in which y is the vertical coordinate, p is the active pore pressure and γ_w is the unit weight of water.

The *Groundwater head* option is one of the options in the *Stresses* submenu. This option is relevant in projects where a groundwater flow calculation has been performed to generate the pore pressure distribution, but also in situations where excess pore pressures are generated in undrained soil clusters.

To display the distribution of the groundwater head as *Contours* or *Shadings*, choose the appropriate option from the presentation combo box.

5.5.9 FLOW FIELD

When a groundwater flow calculation is performed, the specific discharge at element stress points will also be calculated and made available in the *Output* program. You can view the specific discharges by choosing the *Flow field* option from the *Stresses* submenu. You can view the flow field as *Arrows*, *Contours* or *Shadings* by choosing the appropriate option from the presentation box on the toolbar.

When the specific discharges are displayed as arrows, the length of the arrow indicates the magnitude of the specific discharge and the direction of the arrow indicates the flow direction.

5.6 STRUCTURES AND INTERFACES

By default, structures (i.e. plates, geogrids, anchors) 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 plane. As a result, a new form is opened on which the selected object appears. 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 multiple structures of the same type in a single form, then all these objects, except for the last one, should be selected with a single click while holding down the <Shift> key on the keyboard, and the last one should be double clicked.

5.6.1 PLATES

Output data for a plate comprises deformations and forces. From the *Deformations* sub-menu the user may select the absolute displacements, |u|, or the individual displacement components, u_x, u_y and u_z. From the *Forces* sub-menu the options *Axial force N1*, *Axial force N2*, *Shear force Q23*, *Shear force Q13*, *Shear force Q12*, *Bending moment M11*, *Bending moment M22* and *Torsion moment* are available. Forces are calculated at the plate stress points, which do not coincide with the nodes. Moreover, forces can be discontinuous across element boundaries. To show force discontinuities in the results you can display the forces either in front (+) or behind (-) a plane using the P+ or P- button on the toolbar, where P is the actual plane character. An axial force is positive when it generates tensile stresses, as indicated in Figure 5.4. The precise definition of forces is given below.

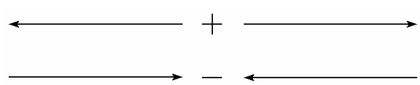


Figure 5.4 Sign convention for axial forces in plates, geogrids and anchors

The *Axial force N1* is the axial force in the major direction of the plate (Figure 5.5a), which is the direction as entered by the user in the geometry creation (a combination of *x*- and *y*-direction). The *Axial force N2* is the axial force in *z*-direction (Figure 5.5b).

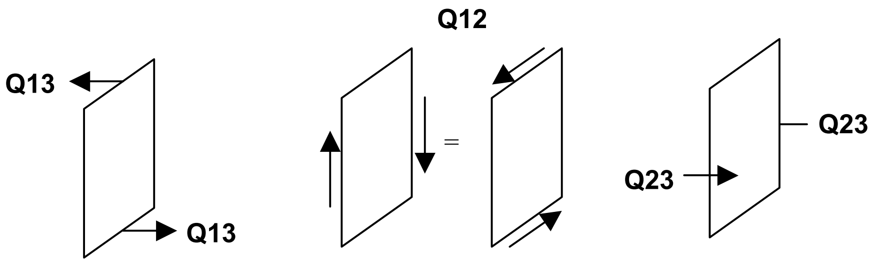
The *Shear force Q13* is the shear force perpendicular to the plate over its major direction (Figure 5.6a), whereas the *Shear force Q12* is the in-plane shear force (Figure 5.6b). *The Shear force Q23* is the shear force perpendicular to the plate over its *z*-direction (Figure 5.6c).

The *Bending moment M11* is the bending moment due to bending around the *z*-axis, which is generally the direction where the major bending moments occur (Figure 5.7a). The *Bending moment M22* is the bending moment due to bending around the other plate axis in the x,*y*-plane (Figure 5.7b). The *Torsion moment M12* is the bending moment according to transverse shear forces (Figure 5.7c).

a: Axial force N1

b: Axial force N2

Figure 5.5

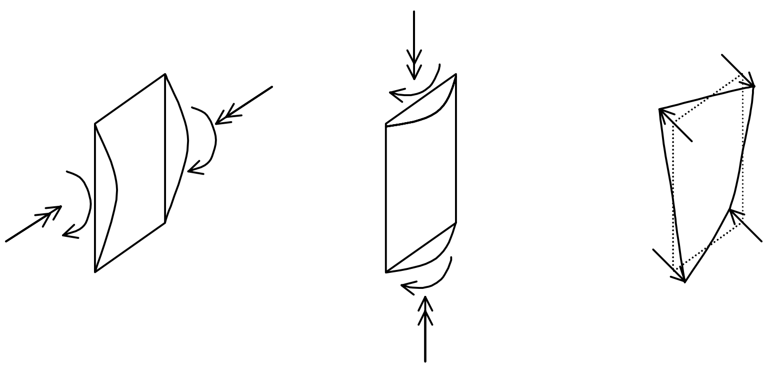


a: Shear force Q13

b: Shear force Q12

c: Shear force Q23

Figure 5.6



a: Bending moment M11

b: Bending moment M22

c: Torsion moment M12

Figure 5.7

5.6.2 GEOGRIDS

Output for geogrids can be obtained by double clicking on the corresponding yellow lines in a *z*-plane. The output for a geogrid comprises deformations and forces. From the *Deformations* sub-menu the user may select the absolute displacements, |u|, or the individual displacement components, u_x, u_y and u_z. From the *Forces* sub-menu the options *Axial force N1* and *Axial force N2* are available (Figure 5.5). Tensile forces in geogrids are always positive. Compressive forces are not allowed in these elements.

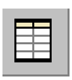
5.6.3 INTERFACES

Output for interfaces can be obtained by double clicking on the corresponding dashed lines in a *z*-plane. The output for an interface comprises deformations and stresses. From the *Deformations* sub-menu the user may select *Total displacements*, *Total increments*, *Relative displacements* and *Relative increments*. From the *Stresses* sub-menu the options *Effective normal stresses*, *Shear stresses*, *Shear stresses (z)*, *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 *x*- or *y*- (or combined *x-y*-) direction, whereas the *Shear stresses (z)* are the shear stresses in *z*-direction. Note that pressure is considered to be negative.

5.6.4 ANCHORS

When double clicking an anchor (either a node-to-node anchor or a fixed-end anchor) in a *z*-plane, a small window is presented in which the anchor force is displayed. Also the maximum force and the anchor stiffness are displayed in this window. If the absolute value of the anchor force is equal to the maximum force, then the anchor is in a plastic state. Tensile forces are defined to be positive, as indicated in Figure 5.4.

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 displacement components at all nodes are presented. The total displacements u_x , u_y and u_z are the total displacements from all previous calculation phases, whereas the incremental displacements Δu_x , Δu_y and Δu_z are the incremental displacements in the current step.

Tables of stresses and strains

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

The *Status* column in the table of the stresses indicates whether a stress point is an *Elastic* point, a *Plastic* point, a *Tension* point, an *Apex* point, a *Hardening* point, or a *Cap* point. An *Elastic* point is a stress point that is currently not in a state of yielding. A *Plastic* point is a stress point where the Mohr's stress circle touches the Coulomb failure envelope. A *Tension* point is a stress point that has failed in tension according to the tension cut-off criterion. An *Apex* point is a stress point at the apex of the failure envelope. A *Hardening* point is a stress point where stress state corresponds to the maximum mobilised friction angle that has previously been reached (only Hardening Soil model). A *Cap* point is a stress point where the stress state is equivalent to the preconsolidation stress, i.e. the maximum stress level that has previously been reached.

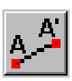
Tables of nodes and stress points

When tables of stresses or strains are shown, the menu includes the sub-menu *Geometry*. This sub-menu contains options to view the position and numbering of the element nodes and stress points. The *Element stress points* option also displays the actual elastic stiffness modulus, *E*, the actual cohesion, *c*, and the actual overconsolidation ratio, *OCR*. This facility is particularly interesting when using models where the stiffness or cohesion increases with depth or when using stress-dependent stiffness models. The table shows which stiffness and cohesion have actually been applied in all stress points in the current calculation step.

Stresses and forces in interfaces and structures

When viewing tables of interface stresses, the table presents the effective normal stresses (σ'_n), the shear stresses (σ_s and $\sigma_{s,z}$), 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 forces (*N1* and *N2*), the shear forces (*Q12, Q23* and *Q13)*, the bending moments (*M11* and *M22)* and the torsion moment *M12* at the nodes. For geogrids, the table only presents the forces in the axial directions of the geogrid (*N1* and *N2*). For anchors there is no other table available than the one that is presented after double clicking the anchor in the geometry.

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 predefined *z*-planes as generated in the 3D mesh extension 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 other 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 model is presented in such a way that the *z*-axis is exactly perpendicular to the user so that the screen coincides with the *x*- and *y*-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. Exact horizontal or vertical cross-sections 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 (for displacements) or extrapolation from the stress points (for stresses). Note that in the latter case, the results might be less accurate than the values at the stress points.

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 cross-section line, and the *Shear strain* is defined as the shear strain along the cross-section line.

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 cross-section, and the *Shear stress* is defined as the shear stress along the cross-section in the (x,y)-plane. Note that pressure is considered to be negative.

5.9 VIEWING OTHER DATA

The *View* menu provides facilities to enhance the graphical presentation of the 3D model, such as the *Partial geometry* option. Moreover, this menu includes options to view general model data (*General 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 PARTIALLY INVISIBLE 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, or a group of soil clusters, can be set visible or invisible.

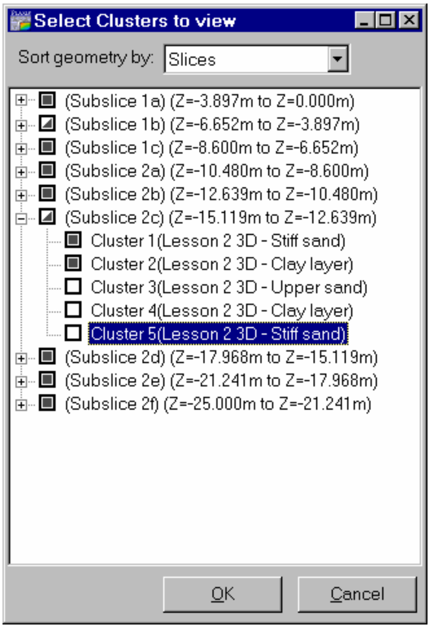


Figure 5.8 Selection window for partial geometry option

To enable a quick selection, clusters can be grouped in three ways: According to *Slices* (slice grouping), *Clusters* (*x*-y area grouping) or *Material sets* (material grouping). The type of grouping can be selected by choosing the appropriate option from the grouping combo box.

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 clusters 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.

After pressing the <OK> button, the 3D model is presented according to the visible/invisible setting as specified in the partial geometry option.

5.9.2 GENERAL PROJECT INFORMATION

The *General info* option of the *View* sub-menu contains some general information about the project.

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 four types of data sets: Soil and interfaces, plates, geogrids, anchors. 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 MULTIPLIERS AND CALCULATION PARAMETERS

If the option *Calculation info* is selected from the *View* menu, then a window appears presenting the load multipliers and various calculation parameters corresponding to the end of the calculation step.

In the *Multipliers* tab sheet, the status of the loading process is given including the values of the incremental and total multipliers. The incremental multipliers give the increase of load in the current step; the total multipliers give the total load that is present at the end of the current step. The significance of the individual multipliers is discussed in Section 4.7. The screen also shows the *Extrapolation factor* and the *Relative stiffness*. The extrapolation factor gives the factor relating the current loading step to the previous one in the case of a continuation of the same load (Section 4.5.1). The relative stiffness gives an indication of the significance of plasticity in the soil body. When loading a body to failure, the relative stiffness gradually reduces from 1.0 (elasticity) to zero (failure).

The *Additional info* tab sheet displays the status of a construction stage and the forces on boundaries with a prescribed non-zero displacement. In the *Staged construction* box the parameter *ΣMarea* gives the proportion of the total volume of soil elements that are currently active, whereas the incremental parameter *Marea* gives the proportional increment of volume that has been applied in the current step. The parameter *ΣMstage* gives the proportion of the construction stage that has been completed, and the incremental parameter *Mstage* gives the proportional increment that has been applied in the current step (see also Section 4.6.4 and 4.7).

The *Forces* box gives the values of the parameters *Force-X*, *Force-Y* and *Force-Z* (the force component in the *x*-, *y*- or *z*-direction respectively due to non-zero prescribed boundary displacements). In addition, when undrained soil clusters are used, the *Consolidation* box shows the maximum value of excess pore pressure in the current step.

The *Step info* tab sheet gives information about the iteration process in the current step. The significance of the data shown is discussed in Section 4.5.1.

5.9.5 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 predefined *z*-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*.

5.9.6 CONTRACTION

When contraction is applied to a circular tunnel lining, the actual (or realised) contraction developed in the finite element analysis may differ slightly from the input value specified in the staged construction. After double-clicking a circular tunnel lining, the corresponding tab sheet shows the *Realised value* of the contraction in a particular cross-section. The *Realised value* is defined as:

$$realised\ value = \frac{original\ \ tunnel\ \ area\ \ \text{minus}\ tunnel\ \ area\ \ at\ \ current\ \ step}{original\ \ area\ \ of\ \ tunnel}$$

Note that the *Realised value* is usually slightly smaller than the *Input value*. This is caused by the fact that contraction of the lining is reduced by the stiffness of the surrounding soil skeleton. For relatively stiff linings with respect to the surrounding soil, the *Realised value* will only be slightly smaller. For linings that are relatively flexible, however, the difference may become more significant. If the *Realised value* turns out to be too low, it is necessary to slightly increase the *Input value* in the corresponding calculation phase and then repeat the calculation.

5.9.7 OVERVIEW OF PLOT VIEWING FACILITIES

To enhance the interpretation of output results, the PLAXIS 3D Tunnel program has several facilities to view the 3D model. An overview of these facilities 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, or plates) appear 'brighter' or 'darker', depending on their orientation with respect to the viewer. Object surfaces appear most brightest when the normal to the surface points in the direction of the viewer. The surfaces become darker the more the normal deviates fom 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 to the right, the positive y-direction is upwards and the positive z-direction points towards the user. The ← and → keys may be used to rotate the model around the y-axis whereas the ↑ and ↓ keys may be used to rotate the model in its current orientation around the horizontal screen axis.

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.

Partial geometry

It may be desired to view 'inside' the 3D model or to present output data of individual soil layers or volume clusters. This is possible by using the *Partial geometry* option of the *View* sub-menu. After selection of this option, the user can select which volume clusters are shown and which are hidden.

Viewing structural objects

Output of structural objects can be viewed in more detail by double clicking the desired structural object in a z-plane.

Viewing cross-section

Apart from the predefined z-planes, a user may define other cross-sections to view output. This can be done by selecting the *Cross-section* option of the *View* sub-menu. A cross-section appears as a plane in the 3D view and as a line in the z-planes.

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 AND STRESS PATHS

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 Calculations program before starting the calculation process (Section 4.11).

Distinction is made between nodes and stress points (Figure 3.4). 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):

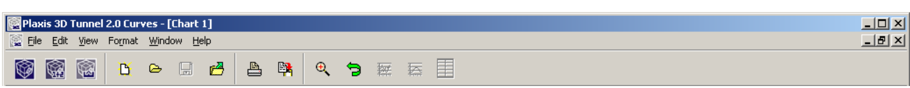


Figure 6.1 Toolbar in main window of the Curves program

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.

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.

Toolbar:

This bar contains buttons that may be used as a shortcut to menu facilities.

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.
Close	To close the active chart window.
Add curve	To add a new curve to the current chart (Section 6.4).
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 Format sub-menu:

Curves	To change the presentation or regenerate the curves in the current chart window (Section 6.6.1).
Frame	To change the presentation of the frame (axes and grid) in the current chart window (Section 0).

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.

Zoom out — To restore the view of before the most recent zoom action.

Reset view — To restore the original draw area.

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.

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.

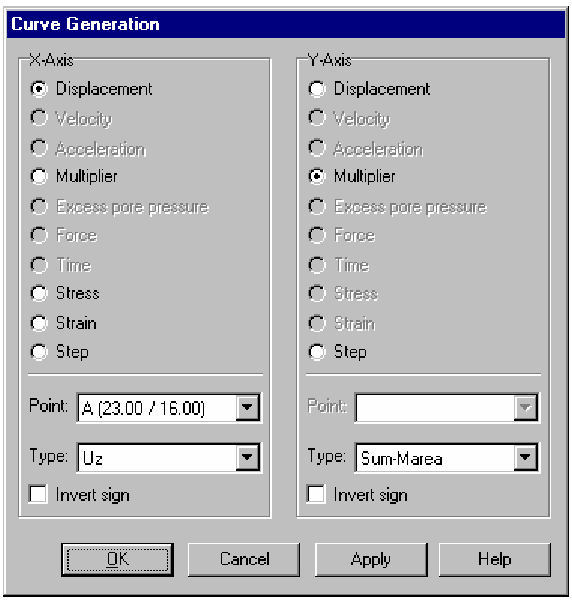


Figure 6.2 Curve generation window

Two similar boxes 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 *Frame settings* window (Section 0). 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. This option may, for example, be used to plot stresses (which are generally negative) as positive values.

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 problem. In general, the x-axis relates to the displacement of a particular node (*Displacement*), and the y-axis contains data relating to load level (*Multipliers*). Other types of curves can also 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.

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. For a description of the multipliers the user is referred to Section 4.7. 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 in the Staged Construction mode.

Another quantity that can be presented in a curve is the *Excess pore pres*sure. The selection of *Excess pore pres.* 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. Excess pore pressures are expressed in the units of stress.

When non-zero prescribed displacements are activated in a calculation, the reaction forces against the prescribed displacements in the x-, y- and z-direction are calculated and stored as output parameters. These force components can also be used in load-displacement curves by selecting the *Force* option.

The selection of the *Force* option must be completed with the selection of the desired component (*Force-X*, *Force-Y*, or *Force-Z*) in the *Type* combo box. The *Force* is expressed in the units of force, as specified in the *General settings* window of the input program.

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. 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:

σ'_{xx}	effective stress in *x*-direction
σ'_{yy}	effective vertical stress (*y*-direction)
σ'_{zz}	effective stress in *z*-direction
σ_{xy}	shear stress in the *x*-*y*-plane
σ_{yz}	shear stress in the *y*-*z*-plane
σ_{zx}	shear stress in the *x*-*z*-plane
σ'_1	in absolute sense the largest effective principal stress
σ'_2	the intermediate effective principal stress
σ'_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:

ε_{xx}	strain in *x*-direction
ε_{yy}	vertical strain (*y*-direction)
ε_{zz}	strain in *z*-direction
γ_{xy}	shear strain in *x*-*y*-plane
γ_{yz}	shear strain in *y*-*z*-plane
γ_{zx}	shear strain in *x*-*z*-plane
ε_1	in absolute sense the largest principal strain
ε_2	the second principal strain
ε_3	the third principal strain
ε_v	volumetric strain
ε_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

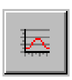
If, for any reason, a calculation process is repeated or extended with new calculation phases, it is generally desirable to update existing curves to comply with the new data. This can be done by means of the *Regenerate* facility. This facility is available in the *Curve settings* window (Section 6.6.1), 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 *Frame* settings. The *Curves* option is used to modify the presentation of curves, and the *Frame* 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. 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 *Graph title* edit box. When a legend is presented for the active chart in the main window, the *Curve title* appears in the legend.

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.

Add curve:

The <Add curve> button may be used to add new curves to the current chart (Section 6.4).

Regenerate:

The <Regenerate> button may be used to regenerate a previously generated curve to comply with new data (Section 6.5).

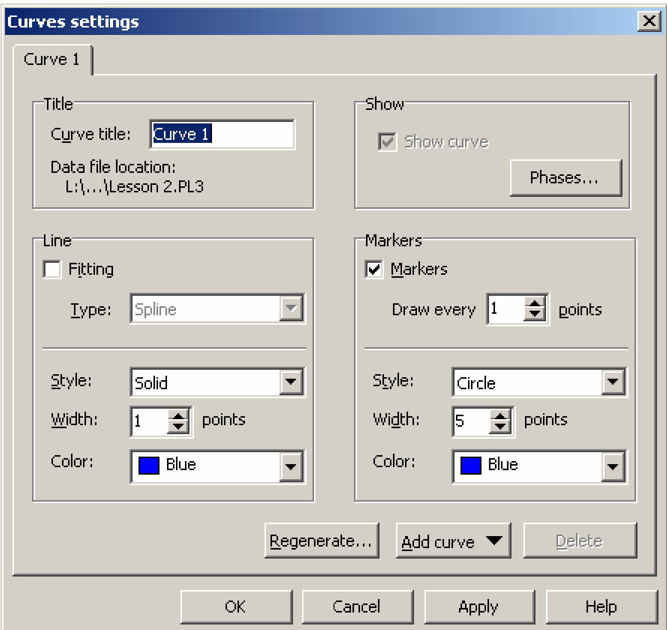


Figure 6.3 Curve settings window

Show:

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> option may be deselected for this purpose.

Delete:

When multiple curves are present within one chart, the <Delete> button may be used to erase a curve.

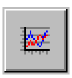
Line and marker presentation:

Various options are available to customise the appearance of the curve lines and markers.

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. For example, when the development of the *ΣMsf* multiplier is plotted against a displacement component to determine safety factors, then only phi-c reduction calculation phases are relevant. The *Phases* option may then be used to de-select the other calculation phases.

6.6.2 FRAME SETTINGS



The *Frame* settings relate to the presentation of the frame and axes in the chart. These settings can be selected from the *Format* menu. Alternatively, the *Frame settings* button in the toolbar may be clicked. As a result, the *Frame settings* window appears, as shown in Figure 6.4.

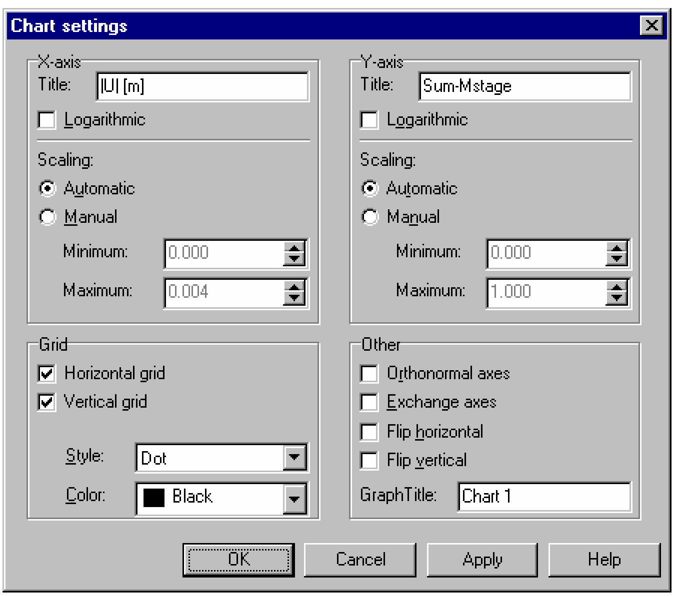


Figure 6.4 Frame settings window

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.

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 *Chart 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 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 plotting stress paths or strain diagrams.

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. This option is particularly useful when plotting stress paths or stress-strain diagrams, since stresses and strains are generally negative.

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. The legend can be activated or deactivated in the *View* menu. The size of the legend can be changed with the mouse.

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 menu 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. Curve points can be inserted or deleted and existing values can be modified. These options are available by clicking on the right hand mouse button, or from the *Edit* menu when the *Table* window is active. Using the *Insert* option results in a new zero *x*- and *y*-value at the position of the cursor. The values may be edited using the *Edit* option. Using the *Delete* option results in a deletion of both the *x*- and *y*-value, so that the point disappears from the curve. When the chart contains more than one curve, deletion and insertion of points is restricted to a single curve.

Editing load-displacement curves is often needed when gravity loading is used to generate the initial stresses for a project. As an example of the procedures involved, consider the embankment project indicated in Figure 6.5. In this example project soil is to be added to an existing embankment to increase its height. The purpose of this example analysis is to calculate the displacement of point A as the embankment is raised. One approach to this problem is to generate a mesh for the final embankment and then deactivate the clusters corresponding to the additional soil layer by using the *Initial geometry configuration* item of the Input program.

An alternative procedure would be to generate the initial stresses for the project, i.e. the stresses for the case where the original embankment has been constructed but the new material has not yet been placed. This should be done using the gravity loading procedure. In this procedure the soil self-weight is applied by increasing *ΣMweight* from zero to 1.0 in a Plastic calculation of the Load Advancement Ultimate Level type.

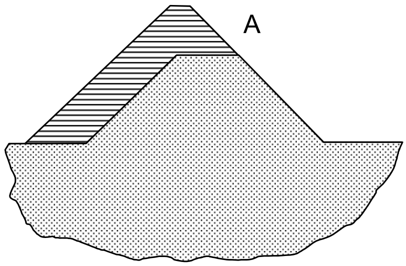


Figure 6.5 Raising an embankment

The settlement behaviour of point A when gravity loading is applied is shown by the initial horizontal line in Figure 6.6a. This line will, in general, consist of several plastic calculation steps, all with the same value of *ΣMarea*.

To model the behaviour of the soil structure as a whole as the additional material is placed, then the cluster of the additional material should be activated using a staged construction calculation. At the start of this staged construction calculation, all displacements should be reset to zero by the user. This removes the effect of the physically meaningless displacements that occur during gravity loading.

The load-displacement curve obtained at the end of the complete calculation for point A is shown in Figure 6.6a. To display the settlement behaviour without the initial gravity loading response it is necessary to edit the corresponding load-displacement data. The unwanted initial portion, with the exception of point 1, should be deleted. The displacement value for point 1 should then be set to zero. The resulting curve is shown in Figure 6.6b.

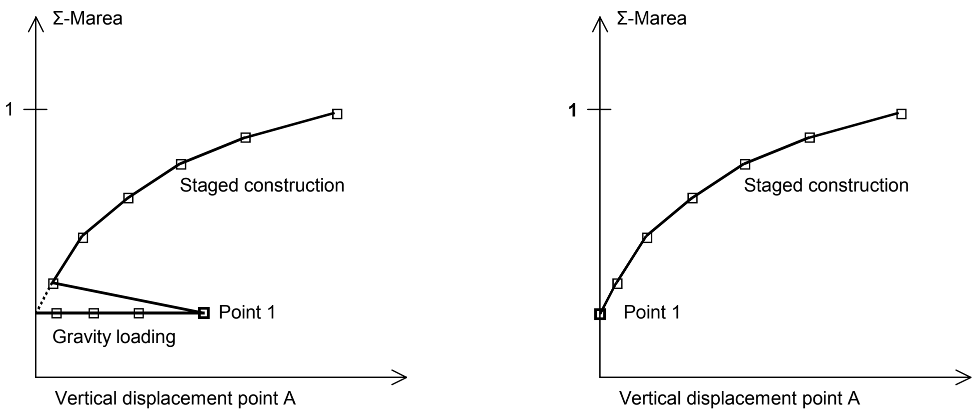


Figure 6.6 Load-displacement curves of the embankment project.

7 REFERENCES

[1] Bakker, K.J. and Brinkgreve, R.B.J. (1990). The use of hybrid beam elements to model sheet-pile behaviour in two dimensional deformation analysis. Proc. 2nd European Specialty Conference on Numerical Methods in Geotechnical Engineering. Santander, Spain, 559-572.

[2] Bathe, K.J. (1982). Finite element analysis in engineering analysis. Prentice-Hall, New Jersey.

[3] Bolton, M.D. (1986). The strength and dilatancy of sands. Geotechnique 36(1), 65-78.

[4] Brinkgreve, R.B.J. and Bakker, H.L. (1991). Non-linear finite element analysis of safety factors. Proc. 7th Int. Conf. on Comp. Methods and Advances in Geomechanics, Cairns, Australia, 1117-1122.

[5] Burd, H.J. and Houlsby, G.T. (1989). Numerical modelling of reinforced unpaved roads. Proc. 3rd Int. Symp. on Numerical Models in Geomechanics, Canada, 699-706.

[6] De Borst, R. and Vermeer, P.A. (1984). Possibilities and limitations of finite elements for limit analysis. Geotechnique 34(20), 199-210.

[7] Hird, C.C. and Kwok, C.M. (1989). Finite element studies of interface behaviour in reinforced embankments on soft grounds. Computers and Geotechnics, 8, 111-131.

[8] Nagtegaal, J.C., Parks, D.M. and Rice, J.R. (1974). On numerically accurate finite element solutions in the fully plastic range. Comp. Meth. Appl. Mech. Engng. 4, 153-177.

[9] Rheinholdt, W.C. and Riks, E. (1986). Solution techniques for non-linear finite element equations. State-of-the-art Surveys on Finite Element Techniques, eds. Noor, A.K. and Pilkey, W.D. Chapter 7.

[10] Rowe, R.K. and Ho, S.K. (1988). Application of finite element techniques to the analysis of reinforced soil walls. The Application of Polymeric Reinforcement in Soil Retaining Structures, eds. Jarett, P.M. and McGown, A. 541-553.

[11] Schikora K., Fink T. (1982). Berechnungsmethoden moderner bergmännischer Bauweisen beim U-Bahn-Bau. Bauingenieur, 57, 193-198.

[12] Sloan, S.W. (1981). Numerical analysis of incompressible and plastic solids using finite elements. Ph.D. Thesis, University of Cambridge, U.K.

[13] Sloan, S.W. and Randolph, M.F. (1982). Numerical prediction of collapse loads using finite element methods. Int. J. Num. Analyt. Meth. in Geomech. 6, 47-76.

[14] Smith I.M. (1982). Programming the finite element method with application to geomechanics. John Wiley & Sons, Chichester.

[15] Song E.X. (1990). Elasto-plastic consolidation under steady and cyclic loads. Ph.D. Thesis, Delft University of Technology, The Netherlands.

[16] Van Langen, H. (1991). Numerical analysis of soil structure interaction. Ph.D. Thesis, Delft University of Technology, The Netherlands.

[17] Van Langen, H. and Vermeer, P.A. (1990). Automatic step size correction for non-associated plasticity problems. Int. J. Num. Meth. Eng. 29, 579-598.

[18] Vermeer, P.A. and Van Langen, H. (1989). Soil collapse computations with finite elements. Ingenieur-Archive 59, 221-236.

[19] Vermeer P.A. and Verruijt A. (1981). An accuracy condition for consolidation by finite elements. Int. J. for Num. Anal. Met. in Geom., Vol. 5, 1-14.

[20] Zienkiewicz, O.C. (1977). The Finite Element Method. McGraw-Hill, London.

[21] Owen D.R.J. and Hinton E. (1982). Finite Elements in Plasticity. Pineridge Press Limited, Swansea.

[22] Van Langen, H. and Vermeer, P.A. (1991). Interface elements for singular plasticity points. Int. J. Num. Analyt. Meth. in Geomech. 15, 301-315.

[23] Bathe K.J., Koshgoftaar M.R. (1979), Finite element free surface seepage analysis without mesh iteration. Int. J. Num. An. Meth Geo, Vol. 3, pp. 13-22.

[24] Desai C.S. (1976), Finite element residual schemes for unconfined flow. Int. J. Num. Meth. Eng., Vol. 10, pp. 1415-1418.

[25] Li G.C., Desai C.S. (1983), Stress and seepage analysis of earth dams. J. Geotechnical Eng., Vol. 109, No. 7, pp. 946-960.

[26] Riks E. (1979), An incremental approach to the solution of snapping and buckling problems. Int. J. Solids & Struct. Vol. 15, pp. 529-551.

[27] Song E.X. (1990). Elasto-plastic consolidation under steady and cyclic loads. Ph.D. Thesis, Delft University of Technology, The Netherlands.

[28] Van Genuchten. M. Th. (1980), A closed form equation for predicting the hydraulic function of unsaturated soils, Journal Soil Science Society of America, 44.

[29] Van Langen, H. (1991). Numerical analysis of soil structure interaction. Ph.D. Thesis, Delft University of Technology, The Netherlands.

[30] Vermeer P.A. and Verruijt A. (1981). An accuracy condition for consolidation by finite elements. Int. J. for Num. Anal. Met. in Geom., Vol. 5, 1-14.

[31] Zienkiewicz O.C. (1967), The finite element method in structural and continuum mechanics. McGraw-Hill, London, UK.

INDEX

A

B

C

D

E

F

V

W

X

Y

Z

APPENDIX A - GENERATION OF INITIAL STRESSES

Many analysis problems in geotechnical engineering require the specification of a set of initial stresses. These stresses, caused by gravity, represent the equilibrium state of the undisturbed soil or rock body.

In a PLAXIS analysis these initial stresses need to be specified by the user. Two possibilities exist for the specification of these stresses:

K_0 -procedure

Gravity loading

As a rule, one should only use the *K_0 -procedure* in cases with a horizontal surface and with any soil layers and phreatic lines parallel to the surface. For all other cases one should use *Gravity loading*.

A.1 THE K_0-PROCEDURE

If this approach is chosen, the user should select the *Initial stresses* option from the *Generate* sub-menu in the *Initial conditions* mode. When selecting this option, then it is possible to enter values for the coefficient of lateral earth pressure for each individual soil cluster. In addition to the parameter K_0, one has to enter a value for *ΣMweight.* For *ΣMweight* = 1.0 gravity will be fully activated. The coefficient, K_0, represents the ratio of the horizontal and vertical effective stresses. Actually, two separate K_0-values can be used, one in x-direction and one in z-direction:

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

In practice, the value of K_0 for a normally consolidated soil is often assumed to be related to the friction angle by the 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.

Using very low or very high K_0-values in the K_0 *-procedure* may lead to stesses that violate the 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*. 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 K_0 *-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 K_0 *-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 below. If the stresses are substantially out of equilibrium, then the K_0 *-procedure* should be abandoned in favour of the *Gravity loading* procedure.

Plastic nil-step

If the K_0 *-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 4.6.13). After this step has been completed, the stress field will be in equilibrium and all stresses will obey the failure condition.

Divergence

If the original K_0*-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 K_0 *-procedure* is applied to problems with very steep slopes. For these problems the *Gravity loading* procedure should be adopted instead.

Initial displacements

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.

A.2 GRAVITY LOADING

If *Gravity loading* is adopted, then the initial stresses (i.e, those corresponding to the 'Initial phase') are zero. The initial stresses are then 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 final value of 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 4.6.7). 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.

It is often the case that plastic points are generated during the *Gravity loading* procedure. For cohesionless soils, for example, plastic points will be generated unless the following inequality is satisfied:

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

The generation of a small number of plastic points during *Gravity loading* is quite acceptable.

Plastic calculation

Gravity loading may be applied, if desired, in a single calculation phase. This should be carried out using the *Load advancement ultimate level* calculation with *ΣMweight* set to 1.0.

Initial displacements

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 *Gravity loading* procedure on the displacements developed during subsequent calculations.

APPENDIX B - PROGRAM AND DATA FILE STRUCTURE

B.1 PROGRAM STRUCTURE

The full PLAXIS program consists of various sub-programs, modules and other files which are copied to various directories during the installation procedure (see *Installation* in the *General information* part). The most important files are located in the PLAXIS program directory. Some of these files and their functions are listed below:

GEO3D.EXE	Input program (pre-processor) (see Chapter 3)
BATCH3DT.EXE	Calculations program (see Chapter 4)
PLX3DOUT.EXE	Output program (post-processor) (see Chapter 5)
CURVES3D.EXE	Curves program (see Chapter 6)
PLXMESHW.EXE	Mesh generator
GWFLOW.EXE	Groundwater flow analysis program
PLASW3D.EXE	Deformation analysis program (plastic calculation, consolidation, updated mesh)
PLXSCR3D.DLL	Module presenting the PLAXIS 3D Tunnel logo's
PLXCLC3D.DLL	Module presenting the screen output during a deformation analysis (Section 4.12)
PLXREQ.DLL	PLAXIS file requester (Section 2.2)

The material data sets in the global database (Section 3.5) are, by default, stored in the DB sub-directory of the PLAXIS 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 (see B.2). The precise material data have to be re-entered in the Input program.

B.2 PROJECT DATA FILES

The main file used to store information for a PLAXIS 3D Tunnel project has a structured format and is named <*project*>.PL3, where <*project*> is the project title. Besides this file, additional data is stored in multiple files in the sub-directory <*project*>.DT3. The files in this directory may include:

CALC.INF	
PLAXMESH.ERR	
PLX3DERR	
PLAXIS.*	(.MSI; .MSO)
ANCHORS.*	(.MDB; .LDB)
BEAMS.*	(.MDB; .LDB)
GEOTEX.*	(.MDB; .LDB)
SOILDATA.*	(.MDB; .LDB)
<*project*>.*	(.3DA; .CXX; .H00; .HXX; .INP; .L## [1]; .MSH; .S; .SF2; .SF4; .SIS; .SXX; .W00; .W## [1]; .000; .### [2]; .ZIN; .ZMS)

[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 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 XXOEGXX. The TEMP directory also contains some backup files (GEO.# where # is a number) as used for the repetitive undo option (Section 3.2). The structure of the GEO.# files is the same as the PLAXIS project files. 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*>.PL3 in the PLAXIS work directory.