PLAXIS Version 8
Reference Manual

TABLE OF CONTENTS

1 INTRODUCTION

PLAXIS is a special purpose two-dimensional finite element computer program used to perform deformation and stability analyses for various types of geotechnical applications. Real situations may be modelled either by a plane strain or an axisymmetric model. The program uses a convenient graphical user interface that enables users to quickly generate a geometry model and finite element mesh based on a representative vertical cross-section of the situation at hand. Users need to be familiar with the Windows environment. To obtain a quick working knowledge of the main features of PLAXIS, users should work through the example problems contained in the *Tutorial Manual*.

The *Reference Manual* is intended for users who wish to obtain 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 PLAXIS 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 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 from a list of standard units. 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. Table 2.1 gives an overview of all available units, the [default] settings and conversion factors to the default units. All subsequent input data should conform to the selected system of units and the output data should be interpreted in terms of this same system. From the basic set of units 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 this way input errors due to wrong units are reduced. In all of the examples given in the PLAXIS manuals, the default units are used.

Table 2.1 Available units and their conversion factor to the default units

Length	Conversion	Force	Conversion	Time	Conversion
mm	= 0.001 m	N	= 0.001 kN	s (sec)	= 1/86400 day
[m]	= 1 m	[kN]	= 1 kN	min	= 1/1440 day
in (inch)	= 0.0254 m	MN	= 1000 kN	hr	= 1/24 day
ft (feet)	= 0.3048 m	lb (pound)	= 0.0044482 kN	[day]	= 1 day
		klb (kilopound)	= 4.4482 kN		

For convenience, the units of commonly used quantities in two different sets of units are listed below:

	Standard			Different	
Basic units:	Length	metre	[m]	feet	[ft]
	Force	kilonewton	[kN]	kilo pound	[klb]
	Time	day	[day]	second	[sec]
Geometry:	Coordinates		[m]	[ft]	
	Displacements		[m]	[ft]	

Material properties:	Young's modulus	[kN/m^2] = [kPa]	[kips] = [klb/sq ft]
	Cohesion	[kN/m^2] = [kPa]	[kips]
	Friction angle	[deg.]	[deg.]
	Dilatancy angle	[deg.]	[deg.]
	Unit weight	[kN/m^3]	[klb/cu ft]
	Permeability	[m/day]	[ft/sec]
Forces & stresses:	Point loads	[kN]	[klb]
	Line loads	[kN/m]	[klb/ft]
	Distributed loads	[kPa]	[kips]
	Stresses	[kPa]	[kips]

Units are generally only used as a reference for the user but, to some extent, changing the basic units in the *General settings* will automatically convert existing input values to the new units. This applies to parameters in material data sets and other material properties in the Input program. It does not apply to geometry related input values like geometry data, loads, prescribed displacements or phreatic levels or to any value outside the Input program. If it is the user's intention to use a different system of units in an existing project, the user has to modify all geometrical data manually and redo all calculations.

In a plane strain analysis, the calculated forces resulting from prescribed displacements represent forces per unit length in the out of plane direction (z-direction; see Figure 2.1). In an axisymmetric analysis, the calculated forces (*Force-X*, *Force-Y*) are those that act on the boundary of a circle subtending an angle of 1 radian. In order to obtain the forces corresponding to the complete problem therefore, these forces should be multiplied by a factor of 2π. All other output for axisymmetric problems is given per unit width and not per radian.

Sign convention

The generation of a two-dimensional finite element model in PLAXIS is based on a geometry model. This geometry model is created in the *x-y* plane of the global coordinate system (Figure 2.1), whereas the *z*-direction is the out-of-plane direction. In the global coordinate system the positive *z*-direction is pointing towards the user.

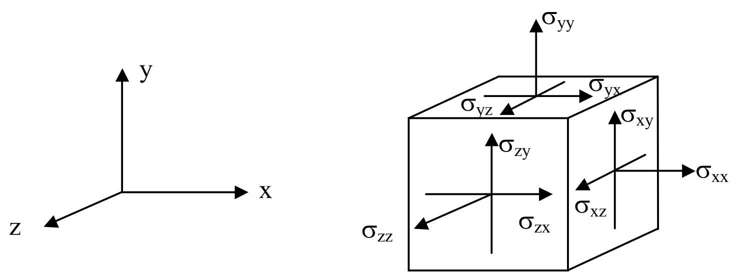


Figure 2.1 Coordinate system and indication of positive stress components.

Although PLAXIS version 8 is a 2D program, stresses are based on the 3D Cartesian coordinate system shown in Figure 2.1. In a plane strain analysis σ_{zz} is the out-of-plane stress. In an axisymmetric analysis, x represents the radial coordinate, y represents the axial coordinate and z represents the tangential direction. In this case, σ_{xx} represents the radial stress and σ_{zz} represents the hoop stress.

In all of the output data, compressive stresses and forces, including pore pressures, are taken to be negative, whereas tensile stresses and forces are taken to be positive. Figure 2.1 shows the positive stress directions.

2.2 FILE HANDLING

All file handling in PLAXIS is done using a modified version of the general Windows® file requester (Figure 2.2).

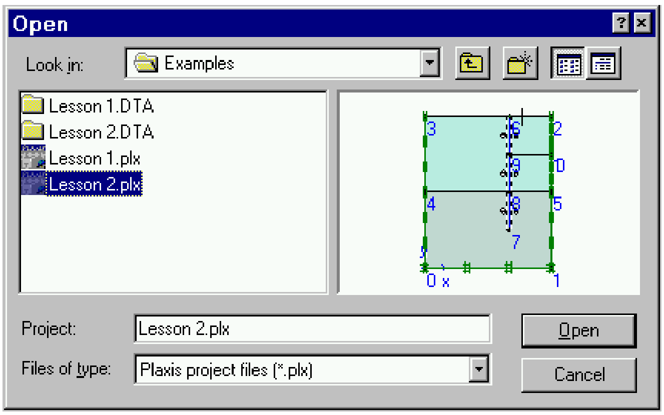


Figure 2.2 PLAXIS file requester

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 project has a structured format and is named <*project*>.PLX, where <*project*> is the project title. Besides this file, additional data is stored in multiple files in the sub-directory <*project*>.DTA. 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 project file (*.PLX) 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.

2.2.1 COMPRESSING PROJECT FILES

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

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

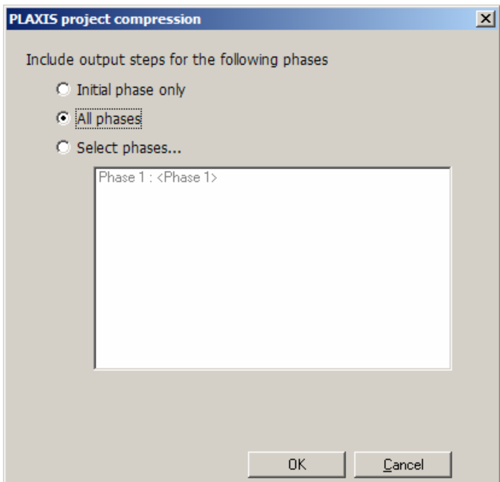


Figure 2.3 Pack project dialog

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 model parameters)
Input of selections	(e.g. choosing a soil model)

The mouse is generally used for drawing and selection purposes, whereas the keyboard is used to enter text and values. These input procedures are described in detail in Section 2.3 of the *Tutorial Manual*.

2.4 HELP FACILITIES

To inform the user about the various program options and features, in the Help menu a link has been created to a digital version of the Manual. Many program features are available as buttons in a tool bar. 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 finite element analysis using PLAXIS, the user has to create a finite element model and specify the material properties and boundary conditions. This is done in the Input program. To set up a finite element model, the user must create a two-dimensional geometry model composed of points, lines and other components, in the *x*-*y*-plane. The generation of an appropriate finite element mesh and the generation of properties and boundary conditions on an element level is automatically performed by the PLAXIS mesh generator based on the input of the geometry model. Users may also customise the finite element mesh in order to gain optimum performance. The final part of the input comprises the generation of water pressures and initial effective stresses to set the initial state.

When a geometry model is created in the Input program it is suggested that the different input items are selected in the order given by the second tool bar (from left to right). In principle, first draw the geometry contour, then add the soil layers, then structural objects, then construction layers, then boundary conditions and then loadings. Using this procedure, the second tool bar 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 situation 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 tool bar 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 geometry model, to generate a corresponding 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 geometry model and a finite element mesh (*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 project file (*.PLX). After the selection of an existing project, the corresponding geometry is presented in the main window.

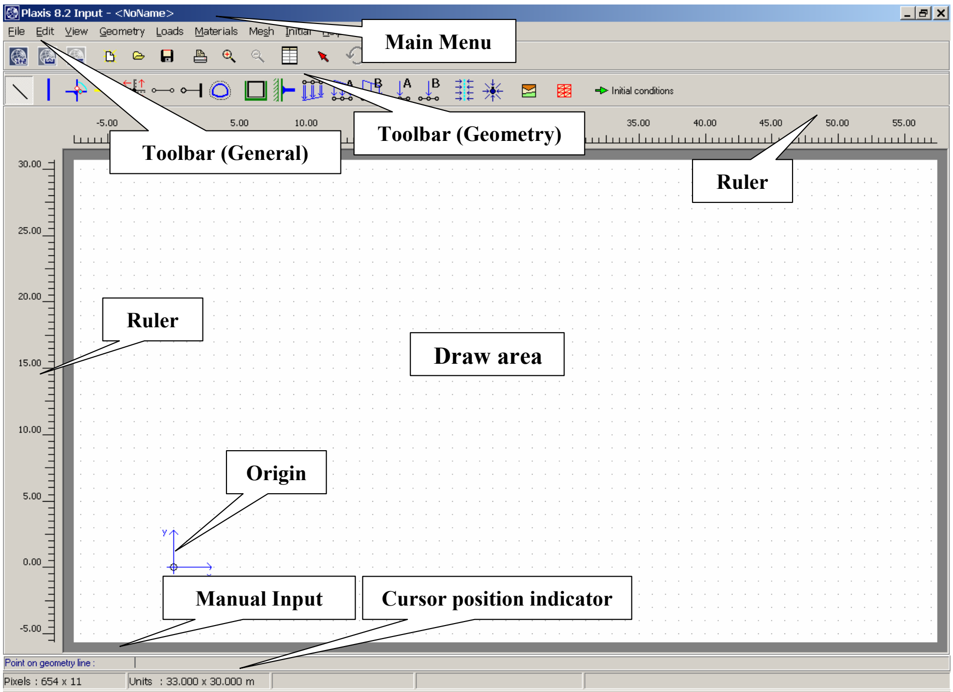


Figure 3.1 Main window of the Input program (*Geometry* creation mode)

The main window of the Input program contains the following items (Figure 3.1):

Input menu

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

Tool bar (General)

This tool bar contains buttons for general actions such as disk operations, printing, zooming or selecting objects. It also contains buttons to start the other sub-programs (Calculations, Output, Curves).

Tool bar (Geometry)

This tool bar contains buttons for actions that are related to the creation of a geometry model. The buttons are ordered in such a way that, in general, following the buttons on the tool bar from the left to the right results in a 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 geometry model. This enables a direct view of the geometry dimensions. The rulers can be switched off in the *View* sub-menu. When clicking on the rulers the *General settings* window appears in which the geometry dimensions can be changed.

Draw area

The draw area is the drawing sheet on which the geometry model is created and modified. The creation and modification of a geometry model is mainly done by means of the mouse, but for some options a direct keyboard input is available (see below, Manual input). The draw area can be used in the same way as a conventional drawing program. The grid of small dots in the draw area can be used to snap to regular positions.

Axes

If the physical origin is within the range of given dimensions it is presented by a small circle in which the x- and 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 <Enter>). Manual input of coordinates can be given for all objects, except for *Hinges* and *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 <Enter>).

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 geometry 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*, *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 geometry model on a selected printer. The print window is presented.
Work directory	To set the default directory where PLAXIS 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 geometry model (after an input error). Repetitive use of the undo option is limited to the 10 most recent actions.
Copy	To copy the geometry model to the Windows clipboard.

The View sub-menu

Zoom in	To zoom into a rectangular area for a more detailed view. After selection, the zoom area must be indicated using the mouse. Click 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 geometry 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 geometry point numbers.
Chain numbers	To show or hide the 'chain' numbers of geometry objects. 'Chains' are clusters of similar geometry objects that are drawn in one drawing action without intermediately clicking the right hand mouse button or the <Esc> key.

The Geometry sub-menu

The *Geometry* sub-menu contains the basic options to compose a geometry model. In addition to a normal geometry line, the user may select plates, geogrids, interfaces, anchors, tunnels, hinges/rotation springs, drains and wells. The various options in this sub-menu are explained in detail in Section 3.3.

The Loads sub-menu

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

The Materials sub-menu

The *Materials* sub-menu is used to activate the data base engine for the creation and modification of material data sets for soil and interfaces, plates, geogrids and anchors. The use of the data base 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 set the basic element type, to generate a finite element mesh and to apply a local or global mesh refinement. 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 Geometry sub-menu of the Initial conditions mode

This sub-menu contains the options to enter the unit weight of water, to draw a phreatic level or to create additional boundary conditions for groundwater flow or consolidation analyses. 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 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 (2D) project files (*.PLX)', which means that the program searches for files with the extension .PLX. After the selection of such a file and clicking on the *Open* button, the corresponding geometry is presented in the draw area.

Although the file structure of PLAXIS Version 8 projects is slightly different from Version 7, it is possible to select 'old' projects, after which they are automatically converted to Version 8.

It is also possible to read geometry files of the DOS-version of the GeoDelft M-series using the *Import* option. In this case the *Files of type* should be set to 'M-series geometry files (*.GEO)'. This option can only be used to read geometry data; soil data is not imported. If such a file is selected and the *Open* button is clicked, the corresponding data is read and the corresponding geometry is presented in the draw area. This geometry is considered to be a new geometry model and not an extension to an existing model. If the number of geometry points is very large, the option may not work properly.

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 (Figure 3.2). 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 and acceleration data. The *Dimensions* tab sheet contains the basic units for length, force and time (Section 2.1) and the dimensions of the draw area.

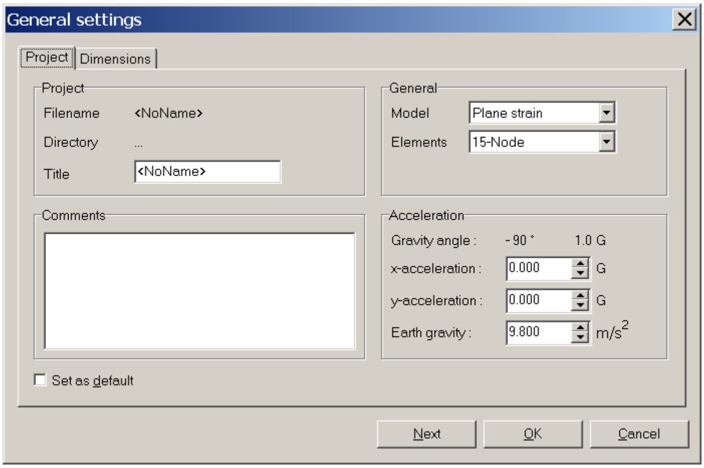


Figure 3.2 General settings window (*Project* tab sheet)

Model

PLAXIS Version 8 may be used to carry out two-dimensional finite element analyses. Finite element models may be either *Plane strain* or *Axisymmetric*. Separate PLAXIS programs are available for 3D analyses. The default setting of the *Model* parameter is *Plane strain*.

A *Plane strain* model is used for geometries with a (more or less) uniform cross section and corresponding stress state and loading scheme over a certain length perpendicular to the cross section (*z*-direction). Displacements and strains in *z*-direction are assumed to be zero. However, normal stresses in *z*-direction are fully taken into account.

An *Axisymmetric* model is used for circular structures with a (more or less) uniform radial cross section and loading scheme around the central axis, where the deformation and stress state are assumed to be identical in any radial direction. Note that for axisymmetric problems the *x*-coordinate represents the radius and the *y*-coordinate corresponds to the axial line of symmetry. Negative *x*-coordinates cannot be used.

The selection of *Plane strain* or *Axisymmetric* results in a two dimensional finite element model with only two translational degrees of freedom per node (x- and y-direction).

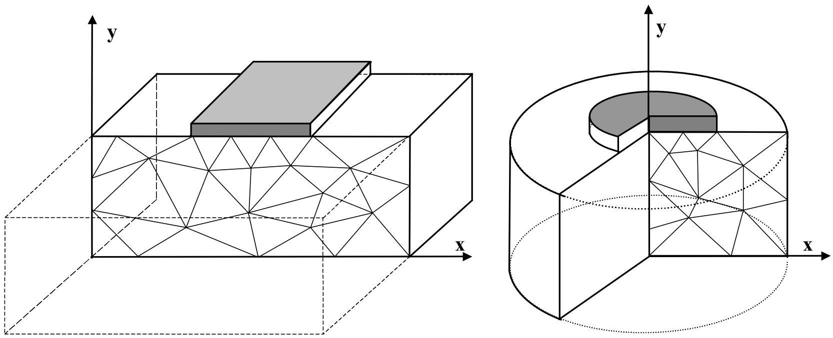


Figure 3.3 Example of a plane strain and axisymmetric problem

Elements

The user may select either 6-node or 15-node triangular elements (Figure 3.2) to model soil layers and other volume clusters. The 15-node triangle is the default element. It provides a fourth order interpolation for displacements and the numerical integration involves twelve Gauss points (stress points). For the 6-node triangle the order of interpolation is two and the numerical integration involves three Gauss points. The type of element for structural elements and interfaces is automatically taken to be compatible with the soil element type as selected here.

The 15-node triangle is a very accurate element that has produced high quality stress results for difficult problems, as for example in collapse calculations for incompressible soils (Refs. 8, 12, 13). The use of 15-node triangles leads to relatively high memory consumption and relatively slow calculation and operation performance. Therefore a more simple type of elements is also available.

The 6-node triangle is a fairly accurate element that gives good results in standard deformation analyses, provided that a sufficient number of elements are used. However, care should be taken with axisymmetric models or in situations where (possible) failure plays a role, such as a bearing capacity calculation or a safety analysis by means of *phi-c reduction*. Failure loads or safety factors are generally overpredicted using 6-noded elements. In those cases the use of 15-node elements is preferred.

One 15-node element can be thought of a composition of four 6-node elements, since the total number of nodes and stress points is equal. Nevertheless, one 15-node element is more powerful than four 6-node elements.

In addition to the soil elements, compatible plate elements are used to simulate the behaviour of walls, plates and shells (Section 3.3.2) and geogrid elements are used to simulate the behaviour of geogrids and wovens (Section 3.3.4). Moreover, compatible interface elements are used to simulate soil-structure interaction (Section 3.3.5). Finally, the geometry creation mode allows for the input of fixed-end anchors and node-to-node anchors (Section 3.3.6 and 3.3.7).

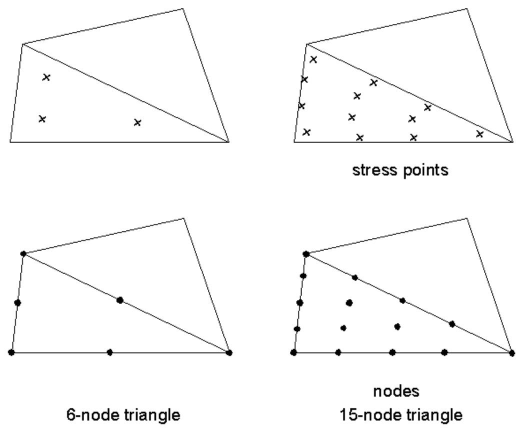


Figure 3.4 Position of nodes and stress points in soil elements

Gravity and acceleration:

By default, the earth gravity acceleration, *g*, is set to 9.8 m/s^2 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 4.8.1).

In addition to the normal gravity the user may prescribe an independent acceleration to model dynamic forces in a pseudo-static way. The input values of the *x*- and *y*-acceleration components are expressed in terms of the normal gravity acceleration *g* and entered in the *Project* tab sheet of the *General settings* window. The activation of the additional acceleration in calculations is controlled by the load multipliers *Maccel* and *ΣMaccel* (Section 4.8.1).

In real dynamic calculations (available as a separate PLAXIS module) the value of the gravity acceleration, *g*, is used to calculate the material density, ρ, from the unit of weight, γ ($\rho = \gamma/g$).

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 (Figure 3.3).

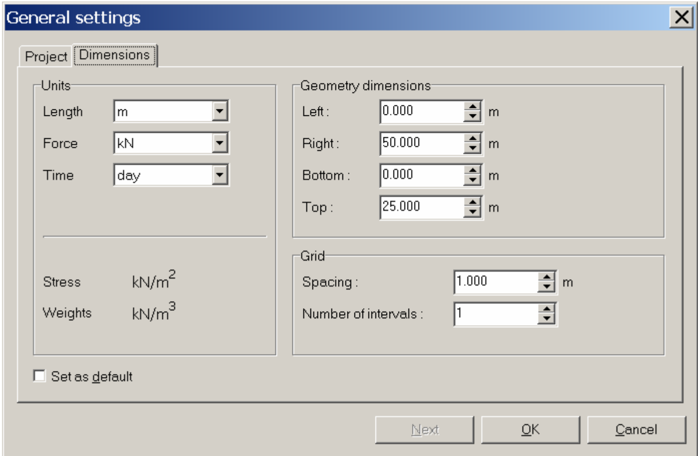


Figure 3.5 General settings window (*Dimensions* tab sheet)

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 unit weights are listed in the box below the basic units.

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 set of 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 geometry 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. Clicking on the rulers in the geometry creation mode may be used as a shortcut to proceed to the input of the geometry dimensions in the *General settings* window.

Grid

To facilitate the creation of the geometry model, the user may define a grid for the draw area. This grid may be used to snap the pointer into certain 'regular' positions. The grid is defined by means of the parameters *Spacing* and *Number of intervals*. The *Spacing* is used to set up a coarse grid, indicated by the small dots on the draw area. The actual grid is the coarse grid divided into the *Number of intervals*. The default number of intervals is 1, which gives a grid equal to the coarse grid. The grid specification is entered in the *Dimensions* tab sheet of the *General settings* window. The *View* sub-menu may be used to activate or deactivate the grid and snapping option.

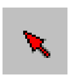
3.3 GEOMETRY

The generation of a finite element model begins with the creation of a geometry model, which is a representation of the problem of interest. A geometry model consists of points, lines and 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 geometry model to simulate tunnel linings, walls, plates, soil-structure interaction or loadings.

It is recommended to start the creation of a geometry model by drawing the full geometry contour. In addition, the user may specify material layers, structural objects, lines used for construction phases, loads and boundary conditions. The geometry model should not only include the initial situation, but also situations that occur in the various calculation phases.

After the geometry components of the geometry 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 geometry model has been defined and all geometry components have their initial properties, the finite element mesh can be generated (Section 3.6).

Selecting geometry components

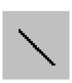


When the *Selection* tool (red arrow) is active, a geometry component may be selected by clicking once on that component in the geometry model. Multiple components of the same type can be selected simultaneously by holding down the <Shift> key on the keyboard while selecting the desired components.

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 geometry model is the *Geometry line*. This item can be selected from the *Geometry* sub-menu as well as from the second tool bar.

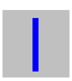
When the *Geometry line* option is selected, the user may create points and lines in the draw area by clicking with the mouse pointer (graphical input) or by typing coordinates at the command line (keyboard input). As soon as the left hand mouse button is clicked in the draw area a new point is created, 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 tool bar. 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 <Delete> 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 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 structures in the ground with a significant flexural rigidity (or bending stiffness) and a normal stiffness. Plates can be used to simulate the influence of walls, plates, shells or linings extending in *z*-direction. In a geometry model, plates appear as 'blue lines'. Examples of geotechnical structures involving plates are shown in Figure 3.6.

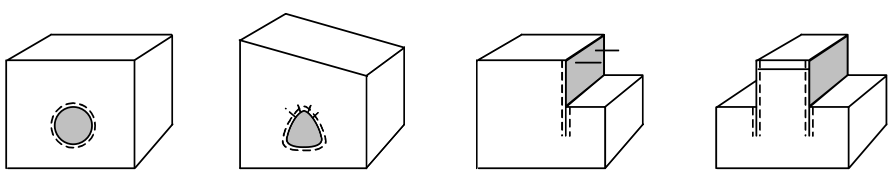


Figure 3.6 Applications in which plates, anchors and interfaces are used

Plates can be selected from the *Geometry* sub-menu or by clicking on the corresponding button in the tool bar. The creation of plates in the geometry 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. Plates can be erased by selecting them in the geometry and pressing the <Delete> key.

The material properties of plates are contained in material data sets (Section 3.5.6). 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:

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

Plates can be activated or de-activated in calculation phases using *Staged construction* as *Loading input*.

Beam elements

Plates in the 2D finite element model are composed of beam elements (line elements) with three degrees of freedom per node: Two translational degrees of freedom (u_x, u_y) and one rotational degrees of freedom (rotation in the x-y plane: ϕ_z). When 6-node soil elements are employed then each beam element is defined by three nodes whereas 5-node beam elements are used together with the 15-node soil elements (Figure 3.7). The beam elements are based on Mindlin's beam theory (Reference 2). This theory allows for beam deflections due to shearing as well as bending. In addition, the element can change length when an axial force is applied. Beam elements can become plastic if a prescribed maximum bending moment or maximum axial force is reached.

Bending moments and axial forces are evaluated from the stresses at the stress points. A 3-node beam element contains two pairs of Gaussian stress points whereas a 5-node beam element contains four pairs of stress points. Within each pair, stress points are located at a distance ½ $d_{eq}\sqrt{3}$ above and below the plate centre-line.

Figure 3.7 shows a single 3-node and 5-node beam element with an indication of the nodes and stress points.

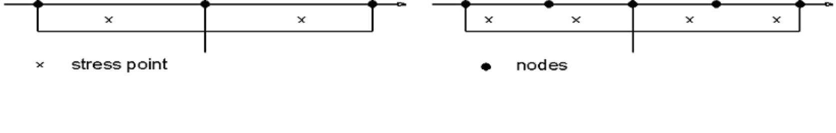
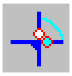


Figure 3.7 Position of nodes and stress points in a 3-node and a 5-node beam element

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 beam 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 (for example in the framework of Staged Construction) it should be noted that the ratio *EI / EA* must remain unchanged.

3.3.3 HINGES AND ROTATION SPRINGS



A hinge is a plate connection that allows for a discontinuous rotation in the point of connection (joint). By default, in a geometry point where plate ends come together, the rotation is continuous and the point contains only one rotational degree of freedom. In other words, the default plate connection is rigid (clamped). If it is desired to create a hinge connection (a joint where plate ends can rotate freely with respect to each other) or a rotation spring (a joint where the rotation of plate ends with respect to each other requires a finite torque), the option *Hinges and rotation springs* can be selected from the *Geometry* sub-menu or by clicking on the corresponding button in the tool bar.

When this option is selected and an existing geometry point is clicked where at least two plates come together, the hinges and rotation springs window appears presenting a detailed view of the joint with all connected plates. For each individual plate end it can be indicated whether the connection is a hinge or a clamp. A hinge is indicated by an open circle whereas a clamp is indicated by a solid circle.

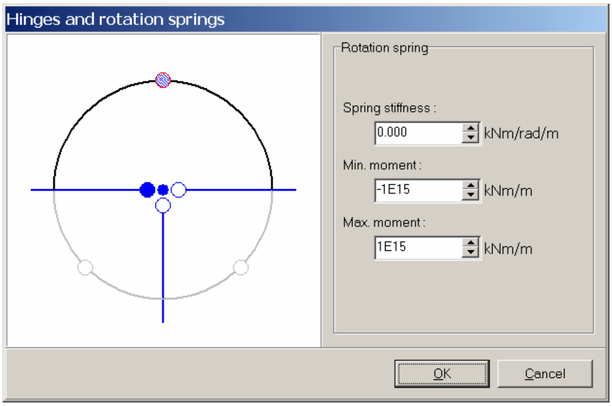


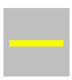
Figure 3.8 Example of a joint in the hinges and rotation springs window

After selecting a particular plate connection by clicking on the corresponding circle, the connection can be toggled from a clamp into a hinge or vice versa by clicking again on the circle. For each hinge, an additional rotational degree of freedom is introduced in order to allow for an independent rotation.

In reality, plate connections may allow for rotations, but this generally requires a torque. To simulate such a situation, PLAXIS enables the input of rotation springs and corresponding relative rotation spring stiffnesses between two plates. This option is only useful if at least one of the two individual plate connections is a hinge (otherwise the connection between the two plates is rigid). To define rotation springs in a joint, the joint is surrounded by large circle sections in which rotation springs can be activated. Possible locations of rotation springs are indicated by small circles (comparable with the hinges) on the large circle sections. In the case of a straight plate there are no large circles around the joint. In that case the central circle represents the rotation spring. After selecting a particular rotation spring by clicking on the corresponding circle, the rotation spring can be toggled on and off by clicking again on the circle.

When a rotation spring is created, the properties of the rotation spring must be entered directly in the right part of the window. The properties of a rotation spring include the spring stiffness and the maximum torque that it can sustain. The spring stiffness is defined as the torque per radian (in the unit of Force times Length per Radian per Length out of plane).

3.3.4 GEOGRIDS



Geogrids are slender 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. Examples of geotechnical structures involving geotextiles are presented in Figure 3.9.

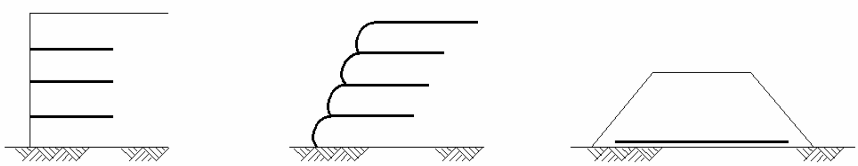


Figure 3.9 Applications in which geogrids are used

Geogrids can be selected from the *Geometry* sub-menu or by clicking on the corresponding button in the tool bar. The creation of geogrids in the geometry model is similar to the creation of geometry lines (Section 3.3.1). In a geometry model geogrids appear as 'yellow lines'. 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 data base (Section 3.5.8). Geogrids can be erased by selecting them in the geometry and pressing the <Delete> key. Geogrids can be activated or de-activated in calculation phases using *Staged construction* as *Loading input*.

Geogrid elements

Geogrids are composed of geogrid elements (line elements) with two translational degrees of freedom in each node (u_x, u_y). When 15-node soil elements are employed then each geogrid element is defined by five nodes whereas 3-node geogrid elements are used in combination with 6-node soil elements. Axial forces are evaluated at the Newton-Cotes stress points.

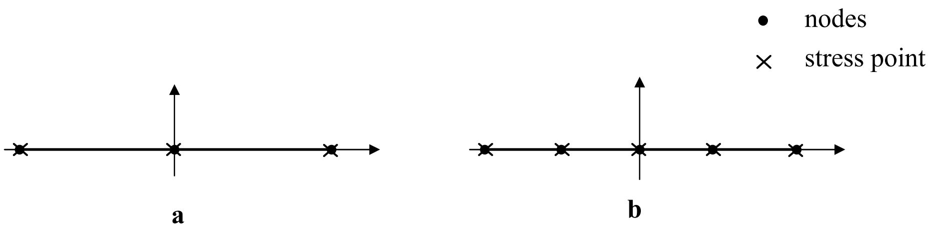


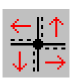
Figure 3.10 Position of nodes and stress points in 3-node and 5-node geogrid elements

These stress points coincide with the nodes. The locations of the nodes and stress points in geogrid elements are indicated in Figure 3.10.

Modelling ground anchors

Geogrids may be used in combination with node-to-node anchors to simulate a ground anchor. In this case the geogrid is used to model the grout body and the node-to-node anchor is used to model the anchor rod (Section 3.3.6)

3.3.5 INTERFACES



Each interface has assigned to it a 'virtual thickness' which is an imaginary dimension used to define the material properties of the interface. The higher the virtual thickness is, the more elastic deformations are generated. In general, interface elements are supposed to generate very little elastic deformations and therefore the virtual thickness should be small. On the other hand, if the virtual thickness is too small, numerical ill-conditioning may occur. 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 mesh generation (Section 3.6.2). This value is also provided in the General info window in the Output program. 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. In general, care should be taken when changing the default factor. However, if interface elements are subjected to very large normal stresses, it may be required to reduce the *Virtual thickness factor*. Further details of the significance of the virtual thickness are given in Section 3.5.2.

The creation of an interface in the geometry 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. 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. Interfaces can be erased by selecting them in the geometry and pressing the <Del> key.

A typical application of interfaces would be to model the interaction between a sheet pile wall and the soil, which is intermediate between smooth and fully rough. The roughness of the interaction is modelled by choosing a suitable value for the strength reduction factor in the interface (R_{inter}). This factor relates the interface strength (wall friction and adhesion) to the soil strength (friction angle and cohesion). Instead of entering R_{inter} as a direct interface property, this parameter is specified together with the soil strength parameters in a material data set for soil and interfaces. For detailed information about the interface material properties, see Section 3.5.33.5.2.

Interfaces can be activated or de-activated in calculation phases using *Staged construction* as *Loading input*.

Interface elements

Interfaces are composed of interface elements. Figure 3.11 shows how interface elements are connected to soil elements. When using 15-node soil elements, the corresponding interface elements are defined by five pairs of nodes, whereas for 6-node soil elements the corresponding interface elements are defined by three pairs of nodes. 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.

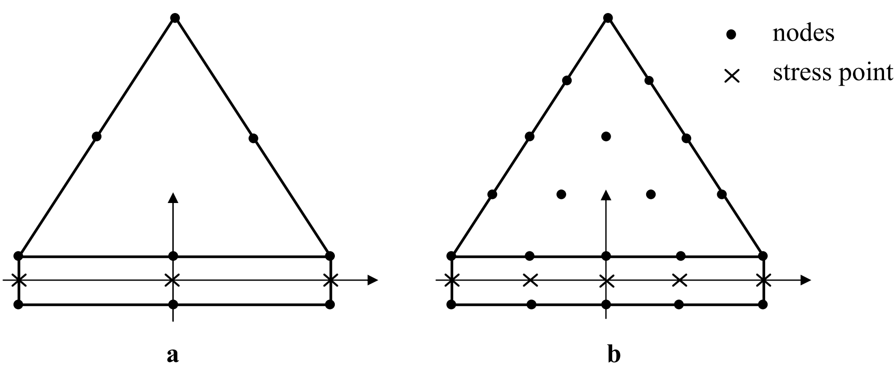


Figure 3.11 Distribution of nodes and stress points in interface elements and their connection to soil elements

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 3.6.2). 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 3.5.3.

The stiffness matrix for interface elements is obtained by means of Newton Cotes integration. The position of the Newton Cotes stress points coincides with the node pairs. Hence, five stress points are used for a 10-node interface element whereas three stress points are used for a 6-node interface element.

Interface properties

The basic property of an interface element is the associated material data set for soil and interfaces. This property is contained in the interface properties window, which can be entered by double-clicking an interface in the geometry model and selecting the positive or negative interface element or interface chain from the selection window. Alternatively, the right-hand mouse button may be clicked, then the *Properties* option should be selected and finally the positive or negative interface element or interface chain may be selected from the right-hand mouse button menu. As a result, the interface properties window appears showing the associated *Material set*, which can be changed using the *Change* button.

In addition, the interface properties window shows the *Virtual thickness factor*. This factor is used to calculate the Virtual thickness of interface elements (see page 3-17 Interface elements). The standard value of the *Virtual thickness factor* is 0.1. Care should be taken when changing the standard value. The standard value can be restored using the *Standard* button.

In a consolidation analysis or a groundwater flow analysis, interface elements can be used to block the flow perpendicular to the interface, for example to simulate an impermeable screen. In fact, when interfaces are used in combination with plates, the interface is used to block the flow since plate elements are fully permeable. In situations where interfaces are used in a mesh where they should be fully permeable, it is possible to de-activate the interface (see Sections 3.8.3, 3.8.6, 3.9.1).

Interfaces around corner points

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.

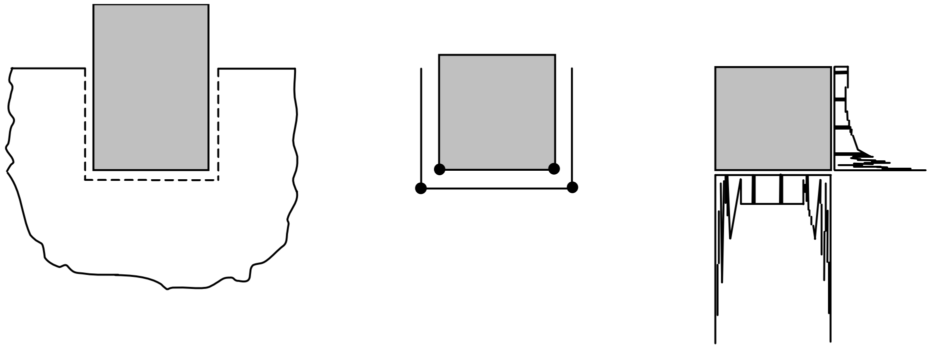


Figure 3.12 Inflexible corner point, causing poor quality stress results

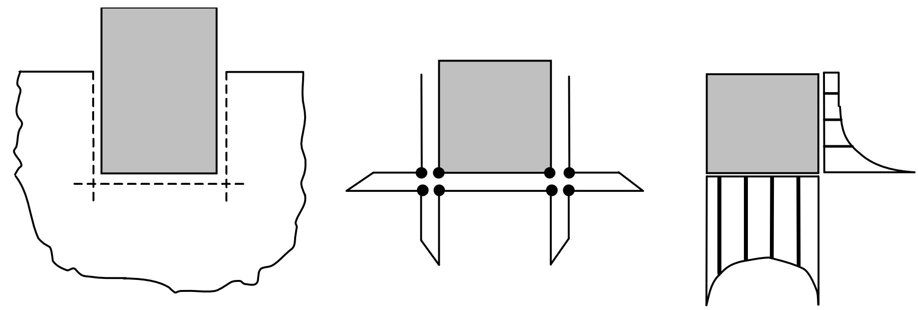
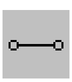


Figure 3.13 Flexible corner point with improved stress results

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. However, these elements should not introduce an unrealistic weakness in the soil. Therefore special attention should be made to the properties of these interface elements (Figure 3.29). Reference 22 provides additional theoretical details on this special use of interface elements.

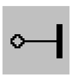
3.3.6 NODE-TO-NODE ANCHORS



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 tool bar. Typical applications include the modelling of a cofferdam as shown in Figure 3.6. 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.

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). Both the tensile force and the compressive force can be limited to allow for the simulation of anchor or strut failure. The properties can be entered in the material data base for anchors (Section 3.5.9). Node-to-node anchors can be activated, de-activated or pre-stressed in a calculation phase using *Staged construction* as *Loading input*.

3.3.7 FIXED-END ANCHORS

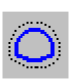


Fixed-end anchors are springs that are used to model a tying of a single point. This type of anchor can be selected from the *Geometry* sub-menu or by clicking on the corresponding button in the tool bar. An example of the use of fixed-end anchors is the modelling of struts (or props) to sheet-pile walls, as shown in Figure 3.6. 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. The angle is defined in the anticlockwise direction, starting from the positive *x*-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.

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 3.5.9).

Fixed-end anchors can be activated, de-activated or pre-stressed in a calculation phase using *Staged construction* as *Loading input*.

3.3.8 TUNNELS



The tunnel option can be used to create circular and non-circular tunnel cross-sections which are to be included in the geometry 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 (i.e. as a file with the extension .TNL) and included in other projects. The tunnel option is available from the *Geometry* sub-menu or from the tool bar.

Tunnel designer

Once the tunnel option has been selected, the Tunnel designer input window appears.

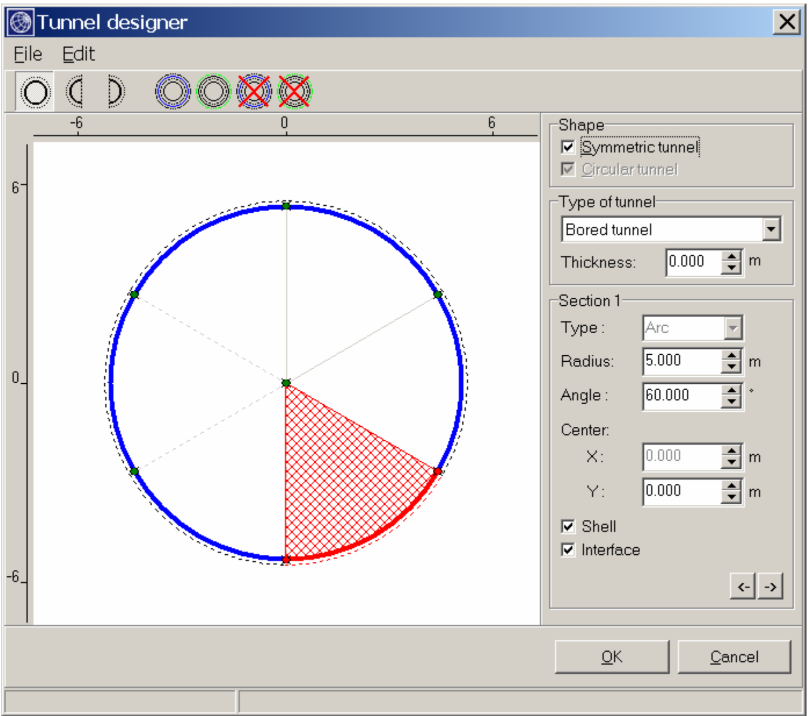


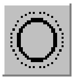
Figure 3.14 Tunnel designer with standard tunnel shape

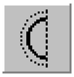
The tunnel designer contains the following items (Figure 3.14):

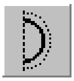
Tunnel menu:	Menu with options to open and save a tunnel object and to set tunnel attributes.
Tool bar:	Bar with buttons as shortcuts to set tunnel attributes.
Display area:	Area in which the tunnel cross-section is plotted.
Rulers:	The rulers indicate the dimension of the tunnel cross-section in local coordinates. The origin of the local coordinate system is used as a reference point for the positioning of the tunnel in the geometry model.
Section group box:	Box containing shape parameters and attributes of individual tunnel sections. Use the buttons <- -> to select other sections.
Other parameters:	See further.
Standard buttons:	To accept (*OK*) or to cancel the created tunnel.

Basic tunnel shape

Once the tunnel option has been selected, the following toolbar buttons can be used 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 geometry model. A half tunnel should be used if the geometry model includes only one symmetric half of the problem where the symmetry line of the geometry model corresponds to the symmetry line of the tunnel. Depending on the side of the symmetry line that is used in the geometry 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

Before creating the tunnel cross-section the type of tunnel must be selected. 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 geometry model. 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.

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 in the framework of staged construction 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 to simulate the volume loss due to the tunnel boring process (Section 4.7.8).

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 plate) 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 in the framework of staged construction 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 (Section 4.7.6 and 4.7.11).

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.

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. The closing error is indicated on the status line of the tunnel designer. A well-defined tunnel cross-section must have a zero closing error. When a significant closing error exists, it is advisable to carefully check the section data.

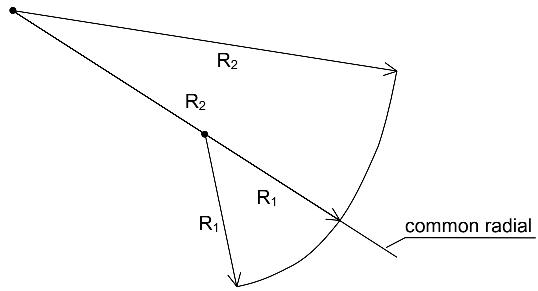


Figure 3.15 Detail of connection point between two tunnel sections

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 or by reducing the last 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.

When the creation of the tunnel cross-section is finished, it can be saved as a tunnel object on the hard disk by using the *Save* option from the *File* menu in the Tunnel designer window.

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 geometry 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 coordinate system is located. When the reference point is entered by clicking with the mouse in the geometry model or by entering the coordinates in the manual input line, the tunnel is included in the geometry model, taking into account eventual crossings with existing geometry lines or objects.

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 geometry model using the original reference point. Note that previously assigned material sets of a lining must be reassigned after modification of the tunnel.

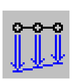
Moving an existing tunnel

An existing tunnel can be moved in the geometry by dragging the tunnel reference point. Note that this is only possible if the tunnel reference point does not coincide with another point.

3.4 LOADS AND BOUNDARY CONDITIONS

The *Loads* sub-menu contains the options to introduce distributed loads, line loads or point loads and prescribed displacements in the geometry 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 tool bar. The input of *Prescribed displacements* in the geometry 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 component is free.

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. 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.

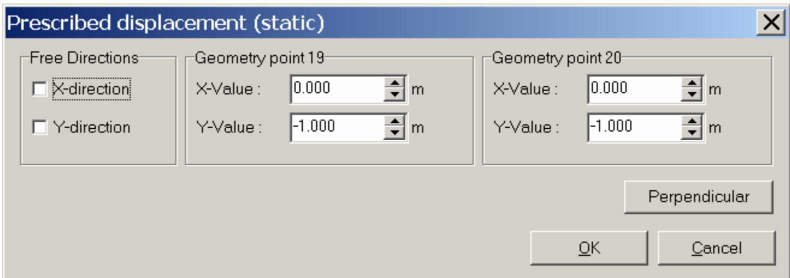


Figure 3.16 Input window for prescribed displacements

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.

Although the input values of prescribed displacements can be specified in the geometry model, the actual values that are applied during a calculation may be changed in the framework of *Staged construction* (Section 4.7.4). Moreover, an existing composition of prescribed displacements may be increased globally by means of the load multipliers *Mdisp* and *ΣMdisp* (Section 4.8.1).

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

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 geometry model, distinction can be made between *Horizontal fixities* (u_x = 0) and *Vertical fixities* (u_y=0). In addition, one can select *Total fixities*, which is a combination of both (u_x=u_y=0).

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.

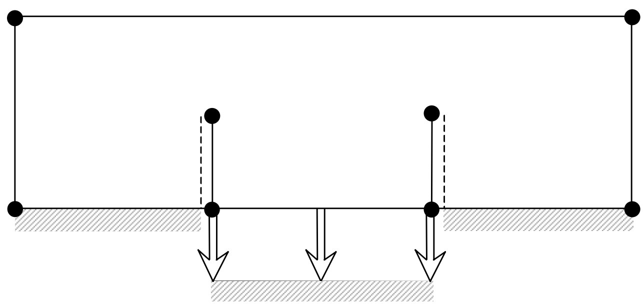
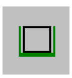


Figure 3.17 Modelling of a trap-door problem using interfaces

3.4.3 STANDARD FIXITIES

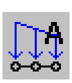


On selecting *Standard fixities* from the *Loads* sub-menu or by clicking on the corresponding button in the tool bar PLAXIS automatically imposes a set of general boundary conditions to the geometry model. These boundary conditions are generated according to the following rules:

- 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 geometry model obtain a fixed rotation in the point at the boundary ($\phi_z = 0$) if at least one of the displacement directions of that point is fixed.

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

3.4.4 DISTRIBUTED LOADS



The creation of a distributed load in the geometry 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 or 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 tool bar.

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*- and/or *y*-component. By default, when applying loads to the geometry 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 two components of the load can be specified for both end points of the geometry line in the geometry model. The distribution is always linear along the line.

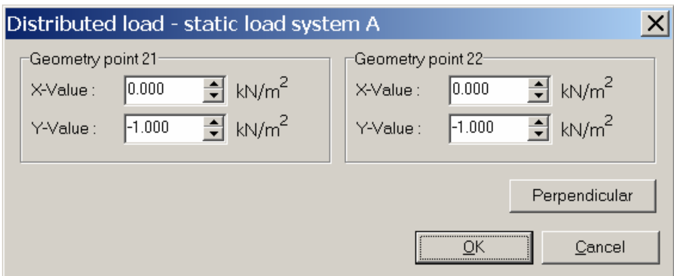
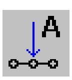


Figure 3.18 Input window for distributed loads

Although the global input values of distributed loads can be specified in the geometry model, the actual value that is applied in a calculation may be changed in the framework of *Staged construction* (Section 4.7.3). 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 4.8.1).

On a geometry line 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, it is possible to apply a distributed load in the free direction.

3.4.5 POINT LOADS



This option may be used to create point loads, which are actually line loads in the out-of-plane direction. The input values of point loads are given in force per unit of width (for example kN/m). In axisymmetric models, point loads are in fact line loads on a circle section of 1 radian. In that case the input value of is still given in force per unit of width, except when the point load is located at x = 0. In the latter case (axisymmetry; point load in x = 0) the point load is a real point load and the input value is given in the unit of force (for example kN, though the input window still shows kN/m). Note that this force is acting on a circle section of 1 radian only. To derive the input value from a real situation, the real point force must be divided by 2π to get the input value of the point force at the centre of the axisymmetric model.

The creation of a point or line load in the geometry 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 and line loads or 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 tool bar.

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*- and/or *y*-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 two components of the load can be specified.

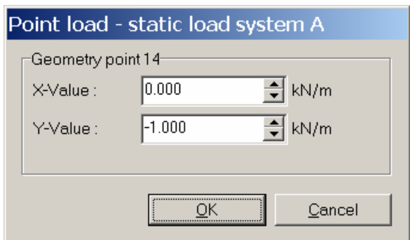
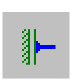


Figure 3.19 Input window for point loads

Although the input values of point loads can be specified in the geometry model, the actual value that is applied in a calculation 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 4.8.1).

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.

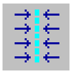
3.4.6 ROTATION FIXITIES



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 tool bar, 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 geometry model and pressing the <Delete> key on the keyboard.

3.4.7 DRAINS

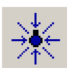


Drains are used to prescribe lines inside the geometry model where (excess) pore pressures are set to zero. This option is only relevant for consolidation analyses or groundwater flow calculations. The *Drain* option can be selected from the *Geometry* sub-menu or by clicking on the corresponding button in the tool bar. The creation of a drain in the geometry model is similar to the creation of a geometry line (Section 3.3.1).

In a consolidation analysis, *excess* pore pressures are set to zero in all nodes that belong to a drain, whereas in a groundwater flow analysis, *active* pore pressures are set to zero, provided that the drain is active.

Drains can be activated or de-activated in calculation phases using *Staged construction* as *Loading input*.

3.4.8 WELLS



Wells are used to prescribe points inside the geometry model where a specific discharge is subtracted from (source) or added to (sink) the soil. This option is only relevant for groundwater flow calculations. The *Well* option can be selected from the *Geometry* sub-menu or by clicking on the corresponding button in the tool bar. The creation of a well in the geometry model is similar to the creation of a fixed-end anchor, but it is not restricted to existing geometry lines.

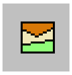
After creating a well, the discharge of the well can be specified by double-clicking the well in the geometry model. This may require zooming into the area where the well is located. As a result, a well window appears. In this window the discharge can be specified as a positive value in the unit of volume per unit time per unit of width out of plane. In addition, it can be selected whether the well is used to apply *Extraction* from the soil (positive discharge) or to apply *Infiltration* in the soil (negative discharge).

Before performing a groundwater flow calculation, wells can be activated or de-activated (Section 3.9.1).

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 data base. From the data base, the data sets can be assigned to the soil clusters or to the corresponding structural objects in the geometry model.

Data base with material data sets



The material data base can be activated by selecting one of the options from the *Materials* sub-menu or by clicking on the *Material sets* button in the tool bar. As a result, a material sets window appears showing the contents of the project data base. The project data base contains the material sets for the current project. For a new project the project data base is empty. In addition to the project data base, there is a global data base. The global data base can be used to store material data sets in a global directory and to exchange data sets between different projects. The global data base 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.

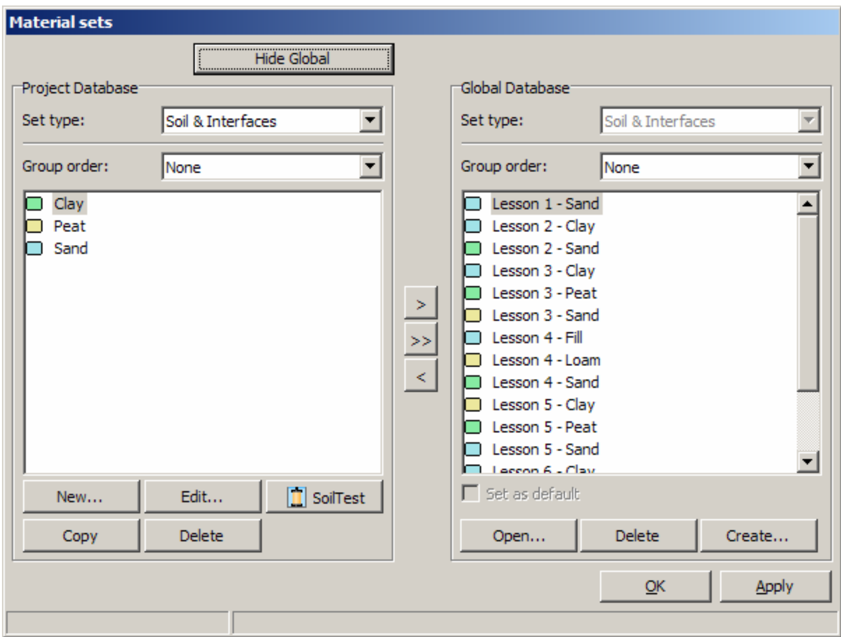


Figure 3.20 Material sets window showing the project and the global data base

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 data base to the global data base or vice versa. The >> button is used to copy all data sets of the project data base to the global data base.

Below the tree view of the global data base there are three buttons. The *Open* button is used to open an existing data base with material data sets (i.e. a file with the extension .MDB), which is then used as the global data base. The *Delete* button can be used to delete a selected material data set from the global data base. The *Create* button is used to store the global data base with material data sets as a separate data base.

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', which is stored in the DB sub-directory of the PLAXIS program directory. This file is compatible with similar data base files of other PLAXIS products. Similarly, the global data bases 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 also stored in the DB sub-directory of the PLAXIS program directory.

The buttons below the tree view of the project data base are used to view, 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 project data base 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. When a data set is no longer required, it may be deleted by first selecting it and clicking on the *Delete* button. In situations where it is not possible to change the project data base (i.e. in the initial conditions or staged construction mode), the *Edit* button is replaced by a *View* button. Clicking on this button enables existing data sets to be viewed.

The Soil Test button will open a separate window for the Soil Tests option. This offers a convenient way to simulate several basic soil tests and check the behaviour of the selected soil material model with the given material parameters. See Section 3.5.11 for details.

3.5.1 MODELLING OF SOIL BEHAVIOUR

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

The well-known Mohr-Coulomb model can be considered as a first order approximation of real soil behaviour. This linear elastic perfectly-plastic model requires five basic input parameters, namely a Young's modulus, E, a Poisson's ratio, 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.

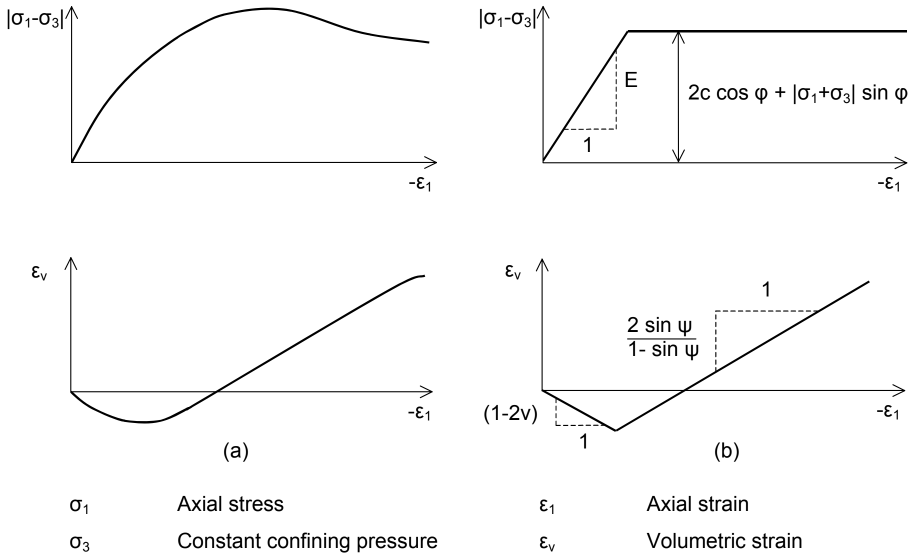


Figure 3.21 Results from standard drained triaxial tests (a) and elastic-plastic model (b).

Basic model parameters in relation to real soil behaviour

To understand the five basic model parameters, typical stress-strain curves as obtained from standard drained triaxial tests are considered (Figure 3.21). 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.

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

3.5.2 MATERIAL DATA SETS FOR SOIL AND INTERFACES

The material properties and model parameters for soil clusters are entered in material data sets (Figure 3.22). 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 geometry 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>*.

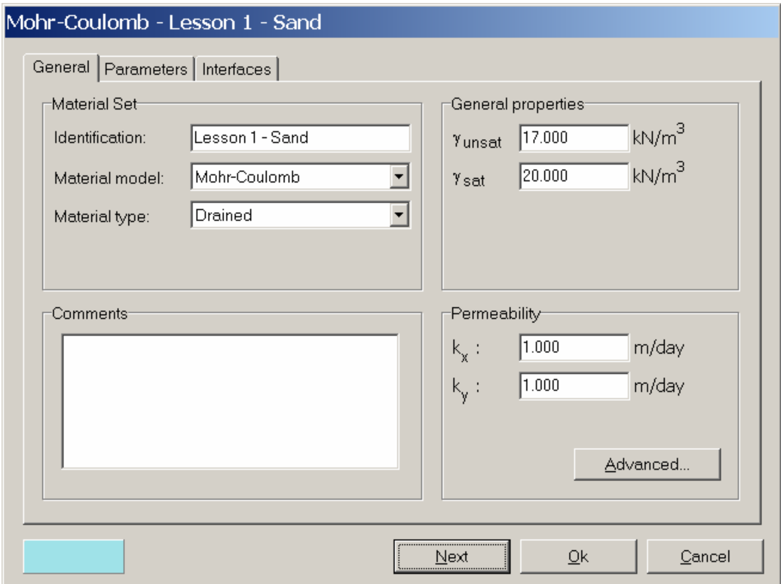


Figure 3.22 *Soil and interface* material set window (*General* tab sheet)

Several data sets may be created to distinguish between different soil layers. A user may specify any identification title for a data set. It is advisable to use a meaningful name since the data set will appear in the 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.

3.5.3 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 too limited for the simulation of soil behaviour. It is primarily used for stiff structures in the soil.

Mohr-Coulomb model

This well-known model is used as a first approximation of soil behaviour in general. The model involves five parameters, namely Young's modulus, E, Poisson's ratio, 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.

HS small model

This is an elastoplastic type of hyperbolic model, similar to the Hardening Soil model. Moreover, this model incorporates strain dependent stiffness moduli, simulating the different reaction of soils to small strains (for example vibrations with strain levels below 10^{-5}) and large strains (engineering strain levels above 10^{-3}).

Modified Cam-Clay model

This is a rather simple critical state model that can be used to simulate the behaviour of normally consolidated soft soils. The model assumes a logarithmic relationship between the volumetric strain and the mean effective stress.

Soft Soil model

This is a Cam-Clay type model that can be used to simulate the behaviour of soft soils like normally consolidated clays and peat. The model performs best in situations of primary compression.

Soft Soil creep model

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

User-defined Soil model

With this option it is possible to use other constitutive models than the standard PLAXIS models. For a detailed description of this facility, reference is made to the Material Models manual.

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 stiffness parameters should be entered, i.e. E' and ν' and not E_u and ν_u. In addition to the stiffness of the soil skeleton, PLAXIS automatically adds a bulk stiffness for the water and distinguishes between effective stresses and excess pore pressures:

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

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

Here $\Delta p'$ is an increment of the effective mean stress and Δp_w is an increment of the excess pore pressure. B is Skempton's B-factor, relating the proportion of the increment in total mean stress to the increment in excess pore pressure. K' is the bulk modulus of the soil skeleton, K_w is the bulk modulus of the pore fluid, n is the porosity of the soil and $\Delta\varepsilon_v$ is an increment of volumetric strain. For the bulk modulus of the soil skeleton the theory of elasticity yields the well-known expression:

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

For undrained behaviour 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, by default, an undrained Poisson's ratio ν_u of 0.495. This results in a relatively low bulk modulus of water, K_w, namely:

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

This results in pore water being slightly compressible and thus a B-factor that is slightly lower than 1.0. Hence, in isotropic loading a few percent of the load will therefore go into effective stresses, at least for small values of the effective Poisson's ratio ν'. For undrained material behaviour the effective Poisson's ratio ν' should be smaller than 0.35. Using higher values of Poisson's ratio would mean that the water would not be sufficiently stiff with respect to the soil skeleton.

The default value of the undrained Poisson's ratio, ν_u, can be overruled by a manual input of Skempton's B-factor in the Advanced Mohr-Coulomb parameters window (See for more details page 3-46).

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 and permeability is not relevant for non-porous materials.

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. However, excess pore stresses (both pressure and suction) may occur above the phreatic line as a result of undrained behaviour. The latter does not affect the unit weight of the soil.

Weights are activated by means of 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 and k_y)

Permeabilities have the dimension of discharge per area, which simplifies to unit of length per unit of time. This is also known as the coefficient of permeability. The input of permeability parameters is required for consolidation analyses and groundwater flow calculations.

In this case it is necessary to specify permeabilities for all clusters, including almost impermeable layers that are considered to be fully impervious. PLAXIS distinguishes between a horizontal permeability, k_x, and a vertical permeability, k_y, since in some types of soil (for example peat) there can be a significant difference between horizontal and vertical permeability.

In real soils the difference in permeabilities between the various layers can be quite large. However, care should be taken when very high and very low permeabilities occur simultaneously in a finite element model, as this could lead to ill-conditioning of the flow matrix. In order to obtain accurate results, the ratio between the highest and lowest permeability value in the geometry should not exceed 10^5.

To simulate an almost impermeable material (for example concrete or uncracked rock) the user should enter a permeability that is low relative 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. One of the advanced features is to account for the change of permeability during a consolidation analysis. This can be applied by entering a proper value for the c_k-parameter and the void ratio's.

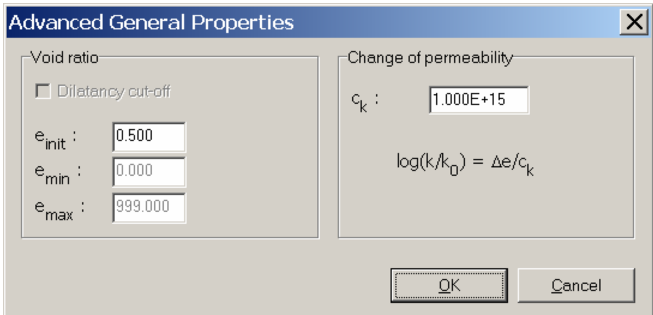


Figure 3.23 Advanced general properties window

Change of permeability (c_k)

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

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

Where Δe is the change in void ratio, k is the permeability in the calculation and k_0 is the input value of the permeability in the data set (= k_x and k_y). It is recommended to use a changing permeability only in combination with the Soft Soil (Creep) model. In that case the c_k-value is generally in the order of the compression index C_c. For all other models the c_k-value should be left to its default value of 10^{15}.

Void ratio (e_{init}, e_{min}, e_{max})

The void ratio, e, is related to the porosity, n ($e = n$ / (1-n)). This quantity is used in some special options. The initial value, e_{init}, is the value in the initial situation. The actual void ratio is calculated in each calculation step from the initial value and the volumetric strain $\Delta\varepsilon_v$. 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.

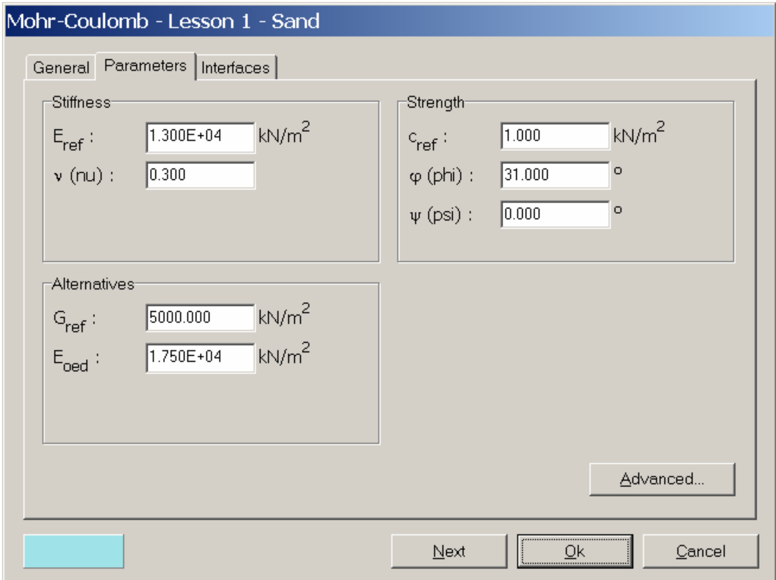


Figure 3.24 Soil and interface material set window (*Parameters* tab sheet of the Mohr-Coulomb model)

3.5.4 PARAMETERS OF THE MOHR-COULOMB MODEL

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

Young's modulus (E)

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

In soil mechanics, the initial slope 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 subjected to loading it is more appropriate to use E_{50}, at least for loading conditions.

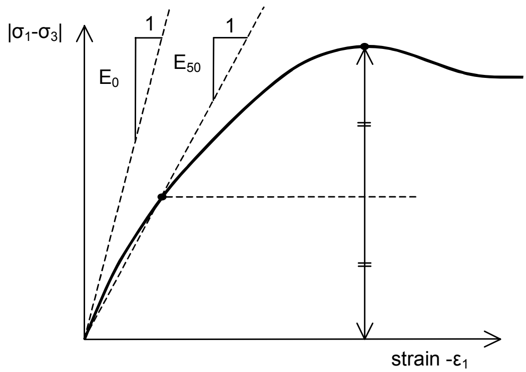


Figure 3.25 Definition of E_0 and E_{50}

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. Furthermore, the observed stiffness depends on the amount of straining the soil undergoes. For small vibrations (with strain levels below 10^{-5}) the stiffness is much higher than for engineering strain levels (above 10^{-3}). Hence, when using a constant stiffness modulus to represent soil behaviour one should choose a value that is consistent with the 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*).

Note that for material data sets where the type of material behaviour is set to undrained, the Young's modulus has the meaning of an effective Young's modulus, whereas the *Undrained* setting takes care of the low compressibility.

Poisson's ratio (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 $\sigma_h \,/\, \sigma_v = \nu \,/\, (1\text{-}\nu)$ 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. For unloading situations a lower Poisson’s ratio (as low as 0.2) is generally more appropriate.

In the case of undrained behaviour, it is advised to enter an effective value of Poisson's ratio and to select *Undrained* as the type of material behaviour. By doing so PLAXIS will automatically add a bulk stiffness for the pore fluid based on an implicit undrained Poisson's ratio of 0.495 (See page 3-36, Undrained behaviour). In this case the effective Poisson's ratio, as entered here, 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 to simulate undrained behaviour.

Alternative stiffness parameters

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

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

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

Cohesion (c)

The cohesive strength has the dimension of stress. In the Mohr-Coulomb model, the cohesion parameter may be used to model the effective cohesion c' of the soil, in combination with a realistic effective friction angle φ' (see Figure 3.26a). This may not only be done for drained soil behaviour, but also if the type of material behaviour is set to *Undrained*, as in both cases PLAXIS will perform an effective stress analysis. Alternatively, the cohesion parameter may be used to model the undrained shear strength c_u (or s_u) of the soil, in combination with $\varphi = \varphi_u = 0$ (see Figure 3.26b).

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

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

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

Friction angle (φ)

The friction angle, φ (phi), is entered in degrees. In general the friction angle is used to model the effective friction of the soil, in combination with an effective cohesion c' (Figure 3.26). This may not only be done for drained soil behaviour, but also if the type of material behaviour is set to *Undrained*, since in both cases PLAXIS will perform an effective stress analysis. Alternatively, the soil strength is modelled by setting the cohesion parameter equal to the undrained shear strength of the soil, in combination with φ = 0 (Figure 3.26).

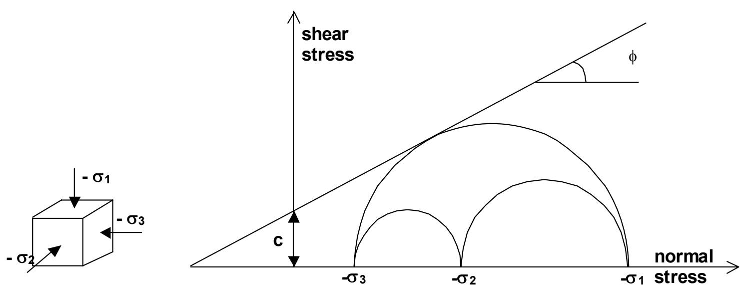


Figure 3.26 Stress circles at yield; one touches Coulomb's envelope

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. A more general representation of the yield criterion is shown in Figure 3.27.

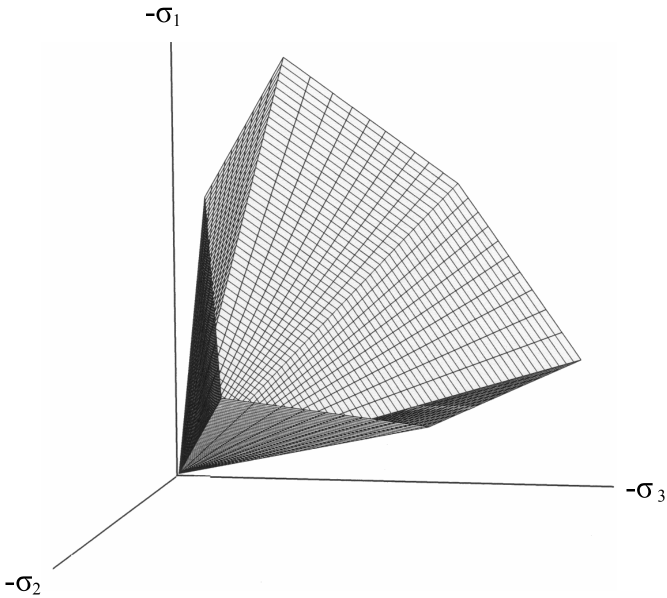


Figure 3.27 Failure surface in principal stress space for cohesionless soil

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. In the Hardening Soil model the end of dilatancy, as generally observed when the soil reaches the critical state, can be modelled using the *Dilatancy cut-off*. For details see the Material Models manual.

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

Advanced Mohr-Coulomb parameters

When using the Mohr-Coulomb model, the *Advanced* button in the *Parameters* tab sheet may be clicked to enter some additional parameters for advanced modelling features. As a result, an additional window appears 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.

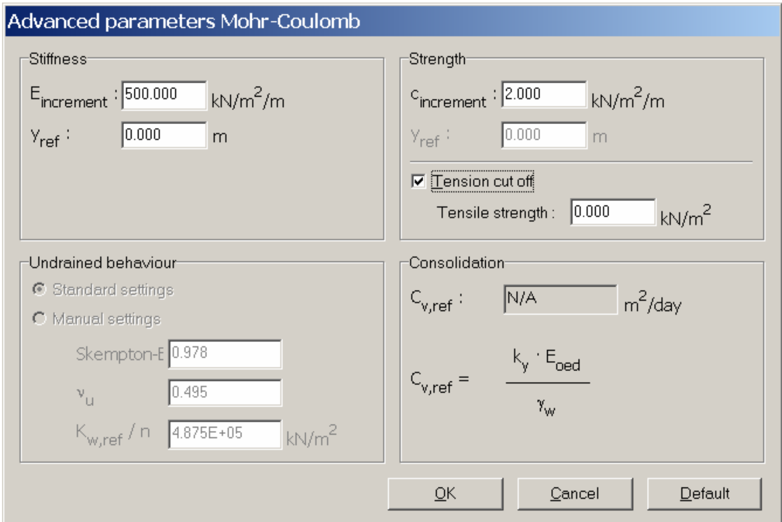


Figure 3.28 Advanced Mohr-Coulomb parameters window

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 $c_{increment}$.

Skempton B-parameter

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

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

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

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

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

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

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

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

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

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

This value of $K_{w,ref} / n$ is generally much smaller than the real bulk stiffness of pure water, K_w^0 (= $2 \cdot 10^6$ kN/m^2).

If the value of Skempton's B-parameter is unknown, but the degree of saturation, S, and the porosity, n, are known instead, the bulk stiffness of the soil skeleton can be estimated from:

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

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

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

Tension cut-off

In some practical problems, an area with tensile stresses may develop. According to the Coulomb envelope shown in Figure 3.26 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 PLAXIS analysis by selecting the tension cut-off. In this case Mohr circles with positive principal stresses are not allowed. When selecting the tension cut-off the allowable *Tensile strength* may be entered. For the Mohr-Coulomb model and the Hardening Soil model the tension cut-off is, by default, selected with a tensile strength of zero.

3.5.5 PARAMETERS FOR INTERFACE BEHAVIOUR

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

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 (adhesion) 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.

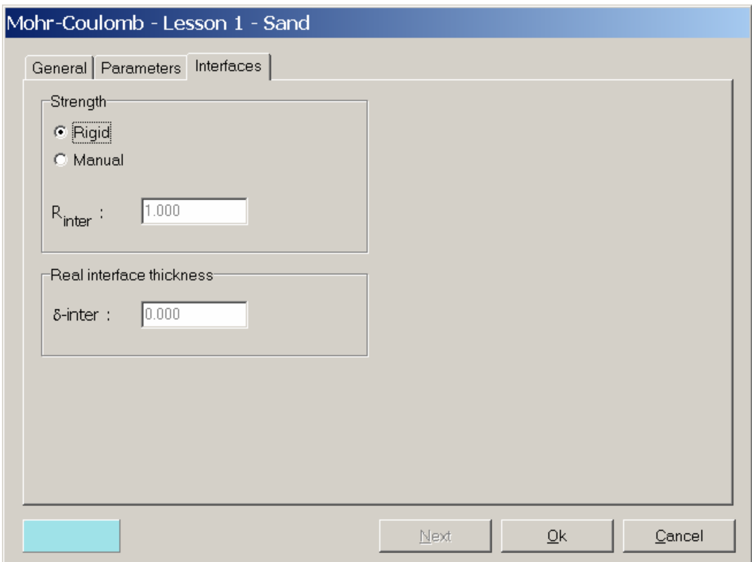


Figure 3.29 Soil and Interface material set window (*Interfaces* tab sheet)

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 geometry model. K_N is the elastic interface normal stiffness and K_S is the elastic interface shear stiffness. The shear and compression moduli are related by the expressions:

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

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

$$\nu_i = 0.45$$

It is clear from these equations that, if the elastic parameters are set to low values, then the elastic displacements may be excessively large. If the values of the elastic parameters are too large, however, then numerical ill-conditioning can result. The key factor in the stiffness is the virtual thickness. This value is automatically chosen such that an adequate stiffness is obtained. The user may change the virtual thickness. This can be done in the properties window that appears after double-clicking an interface (Section 3.3.5).

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.

Interfaces below or around corners of structures

When interfaces are extended below or around corners of structures to avoid stress oscillations (Section 3.3.5), these extended interfaces are not meant to model soil-structure interaction behaviour, but just to allow for sufficient flexibility. Hence, when using R_{inter} < 1 for these interface elements an unrealistic strength reduction is introduced in the ground, which may lead to unrealistic soil behaviour or even failure. Therefore it is advised to create a separate data set with R_{inter} = 1 and to assign this data set only to these particular interface elements. This can be done by dropping the appropriate data set on the individual interfaces (dashed lines) rather than dropping it on the associated soil cluster (the dashed lines should blink red; the associated soil cluster may not change colour). Alternatively, you can click the right-hand mouse button on these particular interface elements and select *Properties* and subsequently *Positive interface element* or *Negative interface element*. In the interface properties window click the *Change* button, after which the appropriate data set can be assigned to the interface element.

Interface permeability

Interfaces do not have a permeability assigned to them, but they are, by default, fully impermeable. In this way interfaces may be used to block the flow perpendicular to the interface in a consolidation analysis or a groundwater flow calculation, for example to simulate the presence of an impermeable screen. This is achieved by a full separation of the pore pressure degrees-of-freedom of the interface node pairs. On the other hand, if interfaces are present in the mesh it may be the user's intension to explicitly avoid any influence of the interface on the flow and the distribution of (excess) pore pressures, for example in interfaces around corner points of structures (Section 3.3.5). In such a case the interface should be de-activated in the water conditions mode. This can be done separately for a consolidation analysis and a groundwater flow calculation. For inactive interfaces the pore pressure degrees-of-freedom of the interface node pairs are fully coupled.

In conclusion:

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

In previous PLAXIS versions, interfaces were given a physical permeability perpendicular to the interface, k_n, and a permeability in longitudinal direction, k_s, whereas factors were used to make the interfaces relatively permeable or relatively impermeable. This approach could lead to unsatisfactory results (significant flow through impermeable interfaces or numerical solution problems). Considering the fact that the permeability in interfaces is a pure numerical property and not a physical property, we have decided to adopt a new approach as described above. The option in previous PLAXIS versions to set the interface permeability to *Drain* has disappeared, since a special drain element is now available (Section 3.4.7).

3.5.6 MODELLING UNDRAINED BEHAVIOUR

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

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

Undrained effective stress analysis with effective stiffness parameters

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

Any volumetric strain occurring in an undrained material during a Plastic calculation phase will now give rise to excess pore pressures. PLAXIS differentiates between steady state pore pressures and excess pore pressures, the latter generated due to svolumetric strain occurring during plastic calculations. This enables the determination of effective stresses during undrained plastic calculations and allows undrained calculations to be performed with effective input parameters.

This special option to model undrained material behaviour based on effective stiffness parameters is available for all material models available in the PLAXIS program. This enables undrained calculations to be executed with effective input parameters, with explicit distinction between effective stresses and (excess) pore pressures.

Undrained effective stress analysis with effective strength parameters

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

However, especially for soft soils, effective strength parameters are not always available, and one has to deal with measured undrained shear strength (c_u or s_u) as obtained from undrained tests. Undrained shear strength, however, cannot easily be used to determine the effective strength parameters φ' and c'. Moreover, even if one would have proper effective strength parameters, care has to be taken as to whether these effective strength parameters will provide the correct undrained shear strength in the analysis. This is because the effective stress path that is followed in an undrained analysis may not be the same as in reality, due to the limitations of the applied soil model.

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

Undrained effective stress analysis with undrained strength parameters

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

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

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

Undrained total stress analysis with all parameters undrained

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

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

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

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

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

3.5.7 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. For both axisymmetric and plane strain models the values of *EA* and *EI* relate to a stiffness per unit width in the out-of-plane direction. 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 bending, shear deformation is taken into account. The shear stiffness of the plate is determined from:

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

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 or structures that are relatively flexible in the out-of-plane direction (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.

Since PLAXIS considers plates (extending in the out-of-plane direction) rather than beams (one-dimensional structures), 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 caused by the stresses in the out-of-plane direction (σ_{zz}) and the fact that strains are prevented in this 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. For sheet-pile walls the weight (force per unit area) is generally provided by the manufacturer. This value can be adopted directly since sheet-pile walls usually occupy relatively little volume.

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. When the combination of a bending moment and an axial force occur in a plate, then the actual bending moment or axial force at which plasticity occurs is lower than respectively M_p or N_p. 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 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 beam elements (Figure 3.7). 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 beam element, it is possible that the nodal values of the bending moment may slightly exceed M_p.

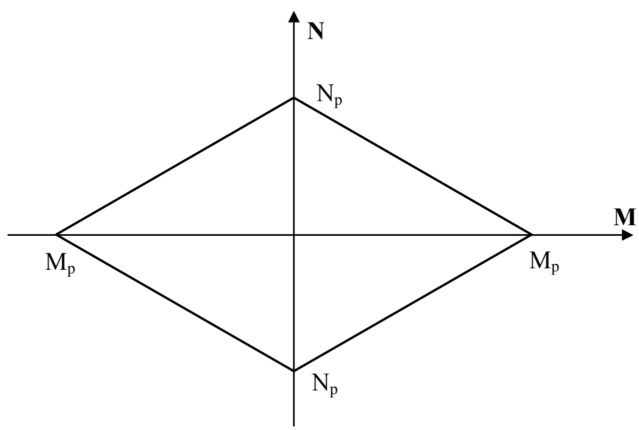


Figure 3.30 Combinations of maximum bending moment and axial force

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).

3.5.8 MATERIAL DATA SETS FOR GEOGRIDS

Geogrids are flexible elastic elements that represent a grid or sheet of fabric. Geogrids cannot sustain compressive forces.

Stiffness properties

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 axial force per unit width and the axial strain ($\Delta l/l$ where Δl is the elongation and l is the length).

$$EA = \frac{F}{\Delta l / l}$$

Strength parameters (plasticity)

Plasticity may be taken into account by specifying a maximum axial tension force, N_p. The maximum axial tension force is specified in units of force per unit width. If 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 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 geogrid element, it is possible that the nodal values of the axial force may slightly exceed N_p.

3.5.9 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 per anchor in the unit of force and not per unit width in the out-of-plane direction. To calculate an equivalent stiffness per unit width, the out-of-plane spacing, L_s, must be entered. If the material type is selected as elastoplastic, two maximum anchor forces, $F_{max,tens}$ (maximum tension force) and $F_{max,comp}$ (maximum compression force) can be entered in the unit of force (also per anchor). In the same way as the stiffness, the maximum anchor forces are divided by the out-of-plane spacing in order to obtain the proper maximum force in a plane strain analysis. If the material type is set to elastic (the default setting), the maximum forces are 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.

3.5.10 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.

When data sets are assigned to structural objects, these objects will blink red for about half a second to confirm the correct data set assignment.

3.5.11 SIMULATION OF SOIL TESTS

The *Soil test* option is a quick and convenient procedure to simulate basic soil tests on the basis of a single point algorithm, i.e. without the need to create a complete finite element model. This option can be used to compare the behaviour as defined by the soil model and the parameters of a soil data set with the results of laboratory test data obtained from a site investigation. The *Soil test* option is available from the *Material sets* window if a soil data set is selected. Alternatively, the *Soil test* option can be reached from the *Soil and interface material set* dialog. In that case the open soil material set must first be saved before the soil test option can be started. Once the *Soil test* option has been selected, a seperate window will open (Figure 3.32).

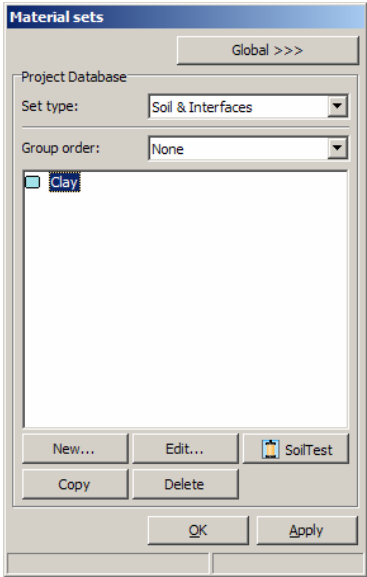


Figure 3.31 Material sets window

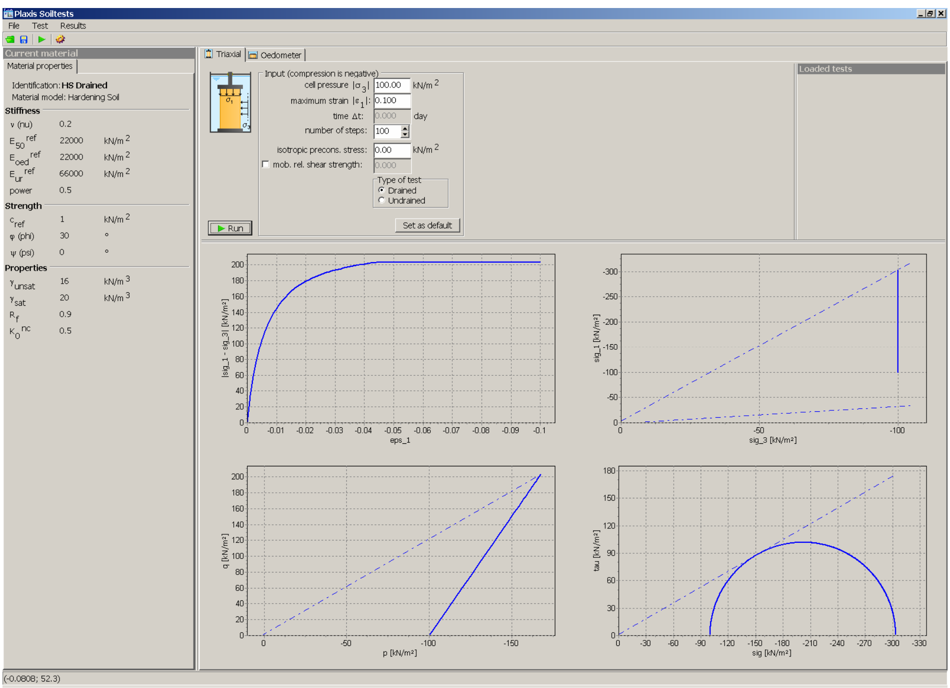


Figure 3.32 Soil test window showing drained triaxial test input

This window contains a menu, a toolbar and several smaller sections. The menu and toolbar allow for saving and loading of soil test results. The main section is the *Test type* window, which shows an overview of the current test settings. It contains tab sheets for different types of tests, as well as a list of previously run tests that have been opened from the menu. Below the *Test type* window is the *Results* section where several predefined diagrams are shown with the results of the latest soil test. Next to the *Test type* window an overview of the material model and corresponding parameters of the current data set is shown. The various items are described in more detail below.

Menu

The menu contains the following items:

File – Open:	Open a soil test data file (*.vlt)
File – Save:	Save a test in a soil test data file (*.vlt)
File – Exit:	Close the *Soil test* window
Test – Triaxial:	Set the test type to triaxial test
Test – Oedometer:	Set the test type to oedometer test
Results – Settings:	Select the configuration of diagrams to display

Material data set properties

The *Material data set properties* section displays the name, material model and parameters of the currently selected data set model. The parameters cannot be edited directly. In order to change the model parameters, close the *Soil test* window and edit the material data set from the *Material sets* window.

Test type

The *Test type* window contains tab sheets for different types of soil tests. Currently, two tests are available, namely *Triaxial* and *Oedometer*.

Triaxial

The *Triaxial* tab sheet contains facilities to define an isotropically consolidated standard drained or undrained triaxial loading test. The following settings can be defined:

Cell pressure (σ_3)*:*	The absolute value of the isotropic cell pressure at which the sample is consolidated, entered in units of stress. This sets the initial isotropic stress state.
Number of steps:	The number of steps that will be used in the calculation.
Maximum strain (ε_1)*:*	The absolute value of the axial strain that will be reached in the last calculation step.

Isotropic preconsolidation pressure:	The isotropic preconsolidation pressure to which the soil has been subjected. If the soil is normally consolidated this value should be set equal to the *Cell pressure*. This option is only available for the advanced soil models.
Mobilized relative shear strength:	This option is only available for the Hardening Soil and HSsmall models to set the initial shear hardening contour. This value must be between 0 (= isotropic stress state) and 1 (=failure state).
Type of test:	Determines whether a consolidated *Drained* or consolidated *Undrained* triaxial test will be simulated.

Oedometer

The *Oedometer* tab sheet contains facilities to define a one-dimensional compression (oedometer) test. The following settings can be defined:

Phases:	Lists the different phases of the oedometer test. Each phase is defined by *Duration* (in units of time), a vertical *Stress increment* (in units of stress) and a *Number of steps*. The initial state is always assumed to be stress free. The given stress increment will be reached at the end of the given duration in the given number of steps. The input values can be changed by clicking in the table. A negative stress increment implies additional compression, whereas a positive stress increment implies unloading or tension. If a period of constant load is desired, enter the desired duration with a zero stress increment.
Add:	Adds a new phase to the end of the *Phases* list.
Insert:	Inserts a new phase before the currently selected phase.
Remove:	Removes the currently selected phase from the *Phases* list.
Isotropic preconsolidation pressure:	The isotropic preconsolidation pressure to which the soil has been subjected. If the soil is normally consolidated this value should be set equal to the initial stress state, i.e. zero. This option is only available for the advanced soil models.
Mobilized relative shear strength:	This option is only available for the Hardening Soil and HSsmall models to set the initial shear hardening contour. This value must be between 0 (= isotropic stress state) and 1 (=failure state).

General

The *Test type* window also contains the following items:

Run: The *Run* button starts the currently selected test. Once the calculation has finished, the results will be shown in the *Results* window.

Set as default: The *Set as default* button saves the current input parameters as the default parameters. These will be loaded the next time the *Soil test* window is opened.

Loaded tests: Within each tab sheet the *Loaded tests* window lists all previously run tests of the current type which have been opened from the *File* menu.

Delete: The *Delete* button removes the currently selected test from the list of loaded tests. It does not remove the soil test file (*.vlt) from disk.

Results

The *Results* window shows several predefined diagrams which are typical to display the results of the current test. Double-clicking one of the graphs opens the selected diagram in a larger window (Figure 3.33).

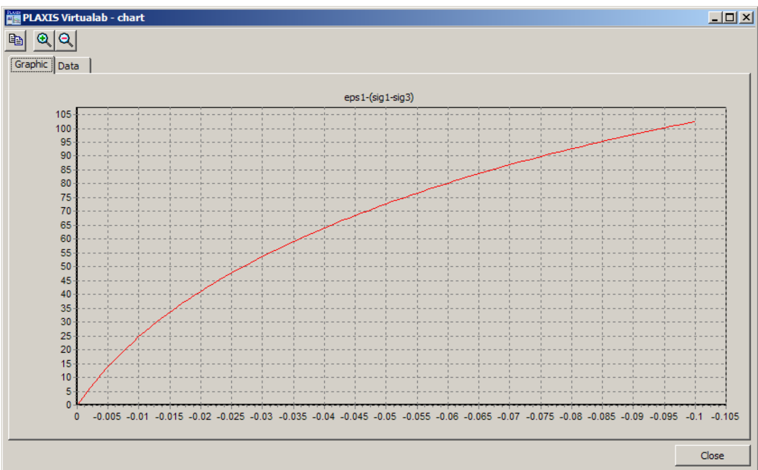
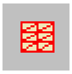


Figure 3.33 Results diagram window

This window shows the selected diagram on the *Graphic* tab sheet. The *Data* tab sheet lists the data points that are used to plot this diagram. Both the diagram and the data can be copied to the clipboard using the *Copy* button on the toolbar. The diagram can be zoomed using the mouse. Click and hold the left mouse button in the diagram area. Move the mouse to a second location and release the mouse button. This will zoom the diagram to the selected area. The zoom action can be undone using the *Zoom out* option on the toolbar. The right hand mouse button can be used for panning. Click and hold the right mouse button and move the diagram to the desired position.

3.6 MESH GENERATION



When the geometry model is fully defined and material properties are assigned to all clusters and structural objects, the geometry has to be divided into finite elements in order to perform finite element calculations. A composition of finite elements is called a mesh. The basic type of element in a mesh is the 15-node triangular element or the 6-node triangular element, as described in Section 3.2.2. In addition to these 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 mesh generation of finite element meshes. The mesh generator is a special version of the Triangle mesh generator developed by Sepra[1]. The generation of the 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.

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

The required input for the mesh generator is a geometry model composed of points, lines and clusters, of which the clusters (areas enclosed by lines) are automatically generated during the creation of the geometry 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 tool bar 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 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) may be used to reduce the graphical thickness of the interfaces. To return to the Input program, the *Update* button must be clicked.

3.6.1 BASIC ELEMENT TYPE

The basic type of element is entered in the *Project* tab sheet of the *General Settings* window in the *File* sub-menu. On selecting *Basic element type* from the *Mesh* sub-menu, the General Settings window is opened and the cursor is positioned at the *Elements* parameter. The user may select either 15-node or 6-node triangular elements (Figure 3.4) as the basic type of element to model soil layers and other volume clusters. The

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

type of element for structures and interfaces is automatically taken to be compatible with the basic type of soil element.

3.6.2 GLOBAL COARSENESS

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

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

Distinction is made between five levels of global coarseness: *Very coarse*, *Coarse*, *Medium*, *Fine*, *Very fine*. By default, the global coarseness is set to *Coarse*. The average 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:	$n_c = 25$	Around 50 elements
Coarse:	$n_c = 50$	Around 100 elements
Medium:	$n_c = 100$	Around 250 elements
Fine:	$n_c = 200$	Around 500 elements
Very fine:	$n_c = 400$	Around 1000 elements

The exact number of elements depends on the shape of the geometry and eventual local refinement settings. The number of elements is not influenced by the *Type of elements* parameter, as set in the *General settings.* Note that a mesh composed of 15-node elements gives a much finer distribution of nodes and thus much more accurate results than a similar mesh composed of an equal number of 6-node elements. On the other hand, the use of 15-node elements is more time consuming than using 6-node elements.

3.6.3 GLOBAL REFINEMENT

A 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 geometry 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.

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 mesh is automatically regenerated.

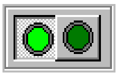
3.6.6 ADVISED MESH GENERATION PRACTICE

To perform efficient finite element calculations, a preliminary analysis can be performed using a relatively coarse 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 detailed 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.

3.7 INITIAL CONDITIONS

Once the geometry model has been created and the finite element mesh has been generated, the initial stress state and the initial configuration must be specified. 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 tool bar. 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 is generally used for effective stress analyses in which 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}:

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

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. Water pressures may also be obtained from the separate PLAXIS program for transient and unsaturated groundwater flow. This program is available as an extension to Version 8. Although transient flow does not generally give steady-state pore pressures, the pore pressures obtained from this program are treated in a deformation analysis as if they are steady.

The water conditions mode may be skipped in projects that do not involve water pressures. In this 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.

3.8.1 WATER WEIGHT AND CAVITATION CUT-OFF

Water weight

In projects that involve pore pressures, the input of a unit weight of water is required to distinguish between effective stresses and pore pressures.

On entering the water conditions mode for the first time, a window appears in which the water weight can be entered. The water weight can also be entered by selecting the *Water weight* option from the *Geometry* sub-menu.

By default, the unit weight of water is set to 10 kN/m^3 or its equivalent value when other units of force or length have been chosen.

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 pressures may be generated. These excess pore pressures might give rise to tensile active pore pressures. In case the cavitation cut-off option is activated, excess pore pressures are limited so that the tensile active pore pressure 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.

3.8.2 PHREATIC LEVELS

Pore pressures and external water pressures can be generated on the basis of phreatic levels. A phreatic level represents a series of points where the water pressure is just zero. Using the input of a phreatic level, the water pressure will increase linearly with depth according to the specified water weight (i.e. the pressure variation is assumed to be hydrostatic). Before entering a phreatic level the user must enter the correct water weight. The option to enter phreatic levels can be selected from the *Geometry* sub-menu or by clicking on the corresponding button in the tool bar. The input of a phreatic level is similar to the creation of a geometry line (Section 3.3.1).

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. Crossings of a 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 most left point to minus infinity and from the most right point 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. The generation of water pressures is actually performed when selecting the *Generate water pressures* option (Section 3.8.4)

General phreatic level

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; on entering 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. The general phreatic level is, by default, 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 such that equilibrium of water pressures is achieved across the boundary. However, if the phreatic level crosses the boundary in a non-existing geometry point, the external water pressures cannot be calculated accurately (Figure 3.34).

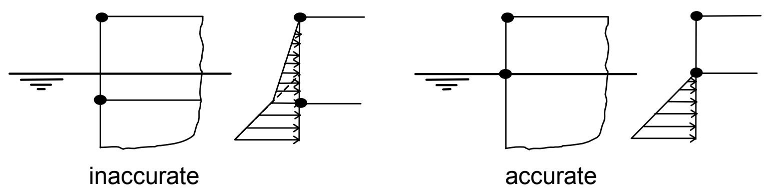


Figure 3.34 Inaccurate and accurate modelling of external water pressures

This is because the value of the external water pressure is only defined at the two end points of the geometry line and the pressure can only vary linearly along a geometry line. Hence, to calculate external water pressures accurately, the general phreatic level should preferably cross the model boundary at existing geometry points. This condition should be taken into account when creating the geometry model. If necessary, an additional geometry point should be introduced for this purpose at the geometry boundary.

The general phreatic level can also be used to create boundary conditions for the groundwater head in the case that pore pressures are calculated on the basis of a groundwater flow calculation (Section 3.8.3).

Cluster phreatic level

To allow for a discontinuous pore pressure distribution, each cluster can be given a separate *Cluster phreatic level*. In fact, 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 height, i.e. the virtual zero-level of the pore pressures in that layer.

A cluster phreatic level can be entered by first selecting the cluster for which a separate phreatic level has to be specified and subsequently selecting the *Phreatic level* option from the tool bar or the *Geometry* sub-menu and entering the phreatic level while the cluster remains selected. When selecting multiple clusters at the same time (by holding the *Shift* key down) 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, one 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).

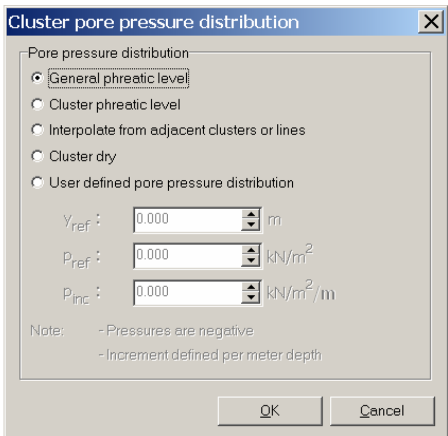


Figure 3.35 Cluster pore pressure distribution window

After double-clicking on a cluster in the water conditions mode the *Cluster pore pressure distribution* window appears in which it is indicated by means of radio buttons how the pore pressures will be generated for that soil cluster. If a cluster phreatic level was assigned to the cluster by mistake, it can be reset to the general phreatic level by selecting *General phreatic level* in this window. As a result, the cluster phreatic level is deleted.

In addition to the general phreatic level and cluster phreatic level option there are some other options available, which are explained below.

Interpolation of pore pressures from adjacent clusters or lines

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

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

In addition to the values in the layers above or below the cluster where the pore pressure is to be interpolated from, one can also directly enter the groundwater head at geometry lines for interpolation purposes. This can be done by double-clicking the corresponding geometry line. As a result, a groundwater head window appears in which the desired groundwater head at both points of the line can be entered. On entering the groundwater head at a point, the program will also display the corresponding pore pressure (pore pressure = water weight times [groundwater head minus vertical position]).

If for the a cluster the option *Interpolate from adjacent clusters or lines* is selected and a groundwater head is defined on an adjacent line, the interpolation will start from the pore pressure on that line rather than from the pore pressure value in the adjacent cluster. In other words, the interpolation procedure gives priority to an eventual direct input of pore pressures in adjacent geometry lines over the pore pressure values in adjacent clusters.

A direct input of a groundwater head in geometry lines is only relevant if the adjacent soil cluster is set to *Interpolate...* or if pore pressures are generated by means of a groundwater flow calculation. Note that when pore pressures are generated on the basis of phreatic levels, the interpolation of pore pressures is used in vertical direction only and not in horizontal direction. Hence, the direct input of a groundwater head on vertical geometry lines does not have any effect in this case.

A direct input of a groundwater head in geometry lines can be erased by selecting the corresponding geometry line and pressing the *Delete* key on the keyboard.

Cluster dry

A fast and convenient option is available for drained and undrained clusters that should be made dry or, in other words, that should have zero pore pressures. This can be done by selecting the *Cluster dry* option. As a result, the steady state pore pressures in that cluster are set to zero and the soil weight is considered to be the *unsaturated weight*.

Note that clusters representing massive (concrete) structures where pore pressures should be excluded permanently (like 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. It should also be noted that in undrained clusters excess pore pressures can still be generated whilst the *Cluster dry* option is used.

User-defined pore pressure distribution

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

Water pressures in inactive clusters

When generating water pressures on the basis of phreatic levels when some clusters are inactive in the initial geometry configuration (Section 3.9.1), 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 it is desired to exclude water pressures in certain clusters, the *Cluster dry* option should be used or a cluster phreatic level should be defined below the corresponding cluster.

3.8.3 BOUNDARY CONDITIONS FOR GROUNDWATER FLOW CALCULATIONS

In addition to the generation of water pressures on the basis of phreatic levels, water pressures can also be generated on the basis of a groundwater flow calculation. This requires the input of boundary conditions on the groundwater head. In principle, two types of boundary conditions exist: A prescribed groundwater head and a prescribed specific discharge normal to the boundary. The latter can only be specified as a specific discharge equal to zero, which is named a *Closed flow boundary*.

Prescribed groundwater head

The prescribed groundwater head on external geometry boundaries is, by default, derived from the position of the general phreatic level, at least when the general phreatic level is outside the active geometry. Also internal geometry lines that have become external boundaries due to a de-activation of soil clusters are considered to be external geometry boundaries and are therefore treated similarly.

In addition to the automatic setting of boundary conditions based on the general phreatic level, a prescribed groundwater head may be entered manually. This procedure is similar to the direct input of a groundwater head for geometry lines. After double-clicking an existing geometry line, a window appears in which the groundwater head at the two points of that line can be entered. On entering the groundwater head at a point, the program will display the corresponding pore pressure (pore pressure = water weight times [groundwater head minus vertical position]).

A prescribed groundwater head can be removed 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 and they are taken into account together with the soil weight and the pore pressures.

Closed flow boundary

A closed flow boundary is an object that can be placed at the boundary of the geometry model to ensure that flow across that boundary will not occur. This option can be selected by clicking on the *Closed flow boundary* button in the tool bar or by selecting the corresponding option from the *Geometry* sub-menu. 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 exactly over existing geometry lines at the outer boundary of the geometry model.

When a boundary geometry line is indicated as a closed flow boundary, it is still possible to prescribe a groundwater head on that boundary. Although the groundwater head is then not used as a boundary condition in the groundwater flow calculation, it will be used to generate external water pressures that are applied in any subsequent deformation analysis.

Seepage surfaces

Flow problems with a free phreatic level may involve a seepage surface on the downstream boundary, as shown in Figure 3.36.

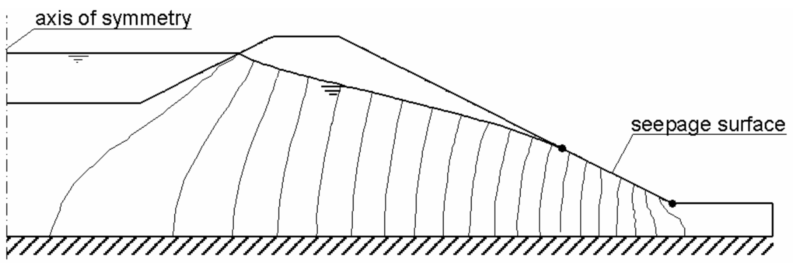


Figure 3.36 Flow through an embankment with indication of a seepage surface

A seepage surface will always occur when the phreatic level touches an open downstream boundary. The seepage surface is not a streamline (in contrast to the phreatic level) or an equipotential line. It 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 that exists at the phreatic level.

For seepage boundaries the hydraulic head, h, needs to be equal to the vertical position, y, which is the default condition used in PLAXIS. It is not necessary to know the exact length of the seepage surface before the calculation begins, since the same boundary conditions ($h = y$) may be used both above and below the phreatic level. 'Open' boundaries with $h = y$ may therefore be specified for all boundaries where the hydraulic head is unknown. Alternatively, for boundaries well above the phreatic level where it is obvious that a seepage surface does not occur, it may also be appropriate to prescribe those boundaries as closed flow boundaries. If no specific condition is prescribed for a particular boundary line, PLAXIS assumes that this boundary is 'open' and sets the seepage condition here.

Inactive clusters in groundwater flow calculation

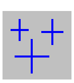
Please note that this option has changed in Version 8 compared to previous PLAXIS versions.

On de-activating clusters in the geometry configuration mode (Section 3.9.1) and performing a groundwater flow calculation for that situation, the inactive clusters do not take part in the groundwater flow calculation itself, but the pore pressure at stress points within the de-activated clusters is determined afterwards from the *general phreatic level*. Hence, 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.

The boundary between active and inactive clusters is considered to be an 'open' boundary so that water can flow across such a boundary. If it is desired to make such a boundary impermeable, then an interface must be created at the 'active' side of the boundary. This interface must be set impermeable (Section 3.3.5) and should also be active itself.

In a deformation analysis, water pressures in inactive soil clusters act as external water pressures on active geometry boundaries.

3.8.4 WATER PRESSURE GENERATION



After the input of phreatic levels or the input of boundary conditions for a groundwater flow calculation, the water pressures can be generated. This can be done by clicking on the *Generate water pressures* button (blue crosses) in the tool bar or by selecting the *Water pressures* option from the *Generate* sub-menu. As a result, a window appears in which it can be indicated whether the water pressures should be generated on the basis of phreatic levels or by means of a groundwater flow calculation. The former option is quick and straightforward whereas the latter option (groundwater flow calculation) can be more realistic but it requires more input parameters and it takes more time.

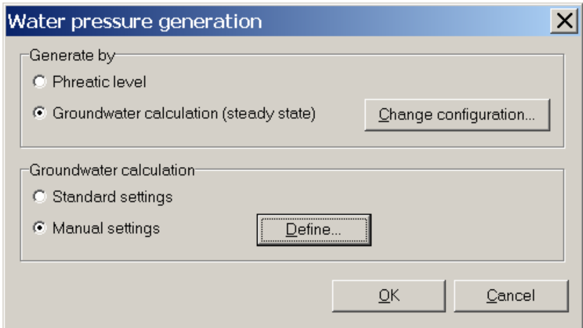


Figure 3.37 Water pressure generation window

Generate by phreatic levels

The 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. This generation is quick and straightforward.

When generating water pressures on the basis of phreatic levels when some clusters are inactive in the initial geometry configuration (Section 3.9.1), 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 it is desired to exclude water pressures in certain clusters, the *Cluster is dry* option should be used or a cluster phreatic level should be defined below the cluster.

Generate by groundwater calculation

PLAXIS includes a steady state groundwater flow calculation module. The water pressure generation by *Groundwater calculation* is based on a finite element calculation using the generated mesh, the permeabilities of the soil clusters and the flow boundary conditions (prescribed groundwater head and closed flow boundaries; Section 3.8.3). This generation is more complex and therefore more time consuming than a generation by means of phreatic levels, but the results can be more realistic, provided that the additional input parameters are properly selected.

When clusters have been de-activated in the geometry configuration mode (Section 3.9.1), the inactive clusters do not take part in the groundwater flow calculation itself, but the pore pressure at stress points within the inactive clusters is determined afterwards from the *general phreatic level*. Hence, 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. The water pressure generation window allows for a direct switch to the geometry configuration mode to activate or de-activate clusters. This can be done by clicking on the *Change configuration* button. After the desired selection has been made, you can return to the water pressure generation window by clicking on the *Continue* button in the tool bar.

When selecting *Groundwater calculation* it is necessary to 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, Section 3.8.5.

Transient groundwater flow

In addition to steady-state groundwater flow, PLAXIS allows for a time-dependent calculation of pore water pressures in saturated and unsaturated conditions due to changing boundary conditions on the groundwater head with time. The results of such a transient flow calculation, i.e. the time-dependent distribution of pore pressures, can be used as input data for a deformation analysis. This option requires the presence of the PLAXIS Groundwater flow module, which is available as an extension to Version 8. The definition of a time-dependent distribution of pore pressures can only be done during the definition of calculation stages. It is not available during the definition of the initial conditions.

Results of water pressure generation

When clicking the *OK* button in the water pressure generation window, the water pressures are calculated according to the selected option. After the generation of water pressures the Output program is started and a plot of the water pressures and the general phreatic level is displayed. To return to the Input program, the *Update* button should be clicked.

Generated water pressures may be used as input data for a deformation analysis. The water pressures are not active until they are actually applied in a calculation. Activation of water pressures is associated with the 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. Hence, 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

Geotechnical engineers regularly need to deal with pore pressures and groundwater flow when solving geotechnical problems. Many situations involve permanent flow or seepage. Dams and embankments are subject to permanent seepage of groundwater. Similarly, permanent flow occurs around retaining walls which separate different groundwater levels. Flow of this sort is governed by pore pressures that are more or less independent of time. Hence, these pore pressures can be considered to be steady-state pore pressures. The PLAXIS Professional Version 8 includes a steady-state groundwater flow calculation module to analyse such situations. This feature is described in the current section. A separate but interactive PLAXIS module for time-dependent flow is available as an extension to Version 8.

The distribution of steady-state pore pressures in a groundwater flow calculation is determined by the boundary conditions, the geometry and the permeabilities of the different soil clusters. For a detailed description of the governing differential equations of a steady-state groundwater flow problem, reference is made to the Scientific Manual.

When using interfaces in a groundwater flow calculation, the interfaces are, by default, fully impermeable. 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 it may be the user's intention 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 3.3.5). In such a case the interface should be de-activated in the water conditions mode. This can be done separately for a consolidation analysis and a groundwater flow calculation. For inactive interfaces the pore pressure degrees-of-freedom of the interface node pairs are fully coupled whereas for active interfaces the pore pressure degrees-of-freedom are fully separated.

In conclusion:

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

A steady-state groundwater flow calculation may be used for confined as well as for unconfined flow problems. The determination of the position of the free phreatic surface and the associated length of the seepage surface is one of the main objectives of an unconfined groundwater flow calculation. In this case it is necessary to use an iterative solution procedure. For confined flow problems, however, an iterative solution procedure is not strictly necessary, since a direct solution can be obtained. Nevertheless, when performing a groundwater flow calculation in PLAXIS the user must select the settings for the control parameters of the iterative procedure, since it is not clear beforehand whether the flow is confined or unconfined. In general, the implemented *Standard settings* may be used, which will normally lead to an acceptable solution. Alternatively, the user may specify the control parameters manually.

Manual setting of groundwater calculation control parameters

On selecting the *Manual settings* option in the *Water pressures generation* window and clicking the *Define* button, a new window is opened in which the current setting of the groundwater calculation control parameters is displayed (see Figure 3.38). A description of the meaning of these parameters is given below.

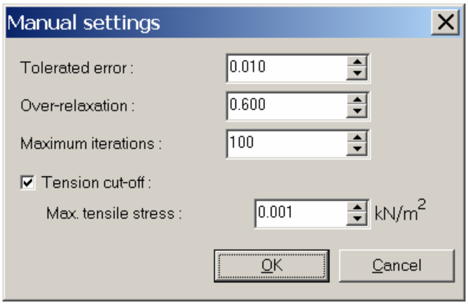


Figure 3.38 Groundwater calculation control parameters window

Tolerated error:

This is the tolerated global (relative) error in the water mass balance. When using the standard setting, the *Tolerated error* is set to 0.01.

Over-relaxation:

This is the over-relaxation factor in the iterative solution procedure. When using the standard setting, the over-relaxation factor is set to 0.6, i.e. in fact under-relaxation is used. An over-relaxation factor larger than 1.0 may be used to speed up the iteration process, but it may also lead to divergence. The theoretical upper bound value of the over-relaxation factor is 2.0.

Maximum iterations:

This parameter puts a restriction on the number of iterations used in unconfined groundwater flow calculations. When using the standard setting, the maximum number of iterations is equal to 100, which is generally sufficient. In some cases, however, a larger number of iterations is needed to obtain a converged solution. The program allows for any value up to 999.

Tension cut-off:

In situations where unconfined flow occurs, tensile pore stresses will be generated. In some such situations these tensile pore stresses can become unrealistically large. The use of these tensile pore stresses in a deformation analysis, while effective strength parameters are used for the soil, will lead to an overestimation of the shear strength. In order to avoid such a situation, tensile pore stresses can be cut off by selecting the *Tension cut-off* option. Subsequently, the *Max. tensile stress* parameter can be set to the maximum allowable tensile stress (in the unit of stress). When using the standard setting, the *Tension cut-off* option is selected and the *Max. tensile stress* parameter is set to zero.

Limitations

Although concepts of partially saturated soil are used in the iterative solution procedure for a free phreatic surface, the steady-state groundwater flow calculation kernel in PLAXIS Version 8 is not designed for the analysis of flow in partially saturated soil. Analysis of flow in partially saturated soil requires more complex relationships between soil permeability, degree of saturation and tensile pore stresses. Such relationships are included in the separate PLAXIS groundwater flow module, which is available as an extension to Version 8.

3.8.6 CLOSED CONSOLIDATION BOUNDARIES

A consolidation analysis can be performed in PLAXIS to calculate the development of excess pore pressures with time. A consolidation analysis involves additional boundary conditions for excess pore pressures. By default, all geometry boundaries are 'open', which means that water can flow in or out at the boundaries. In other words, the excess pore pressure is zero at the boundary.

At some boundaries, however, this condition may not be correct, for example at vertical boundaries representing a line of symmetry or if the bottom of the geometry model is located in an impervious layer. In these cases there is no flow across these boundaries. For these situations you can use the *Closed consolidation boundary* option. This option can be selected by clicking on the *Closed consolidation boundary* button in the tool bar or by selecting the corresponding option from the *Geometry* sub-menu. The input of a closed consolidation boundary is similar to the creation of a closed flow boundary (Section 3.8.3).

A closed consolidation boundary does not automatically imply a closed flow boundary or vice versa. If a project involves a groundwater flow calculation as well as a consolidation analysis and a part of the boundary is considered impermeable, then, in principle, both the *Closed flow boundary* and the *Closed consolidation boundary* conditions must be applied to this boundary. There can be situations where different conditions for groundwater flow and consolidation are to be considered at a certain boundary, therefore distinction is made between closed flow boundaries and closed consolidation boundaries.

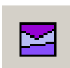
When using interfaces in a consolidation analysis, the interfaces are, by default, fully impermeable, which means that no consolidation takes place across the interface. In this way interfaces have a similar functionality as a *Closed consolidation boundary*, except that interfaces can be used at the inside of a geometry whereas closed consolidation boundaries can only be used at the geometry boundary. If interfaces are present in the mesh it may also be the user's intension to explicitly avoid any influence of the interface on the consolidation process, for example in interfaces around corner points of structures (Section 3.3.5). In such a case the interface should be de-activated in the water conditions mode. This can be done separately for a consolidation analysis and a groundwater flow calculation. For inactive interfaces the excess pore pressure degrees-of-freedom of the interface node pairs are fully coupled whereas for active interfaces the excess pore pressure degrees-of-freedom are fully separated.

In conclusion:

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

It is not possible to prescribe excess pore pressures as a boundary condition for a consolidation analysis. Excess pore pressures at the beginning of a consolidation analysis can only be the result of earlier calculations where undrained clusters were used, i.e. clusters where the *Material type* in the corresponding material data set was set to *Undrained*. For more information on consolidation analyses see Section 4.4.2, 4.5.4 and the Scientific Manual.

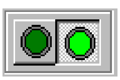
3.8.7 MATERIAL PROPERTIES FOR UNSATURATED FLOW



If the Plaxis groundwater flow module has been installed an additional Material Sets button is available. This button offers access to the Soil and Interfaces material sets that have been previously defined. Instead of defining strength and stiffness parameters for the soil it is possible here to define the material parameters for unsaturated flow.

For a full listing of the available models and their meaning one is referred to the separate manual for the Plaxis groundwater flow module.

3.9 INITIAL GEOMETRY CONFIGURATION



To proceed from the water conditions mode to the geometry configuration mode, click on the right-hand side of the 'switch' in the tool bar.

The geometry configuration mode is, a.o., used to set the initial geometry configuration and enables you to de-select the geometry clusters that are not active in the initial situation. In addition, initial effective stresses may be generated using the K_0-*procedure*.

3.9.1 DE-ACTIVATING LOADS AND GEOMETRY OBJECTS

In projects where embankments or structures are to be constructed the geometry model will contain some components (such as loads, plates, geotextiles, anchors, interfaces or soil clusters above the initial ground surface) that are initially not active. Soil clusters above the initial ground surface must be de-activated by the user. PLAXIS will automatically de-activate 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. Note that the *K0 procedure* for the generation of initial stresses (Section 3.9.3) does not take external loads and weights of structural elements into account.

Activation or de-activation of geometry components can be done by single clicking on the component in the geometry model. Note that, in contrast to previous PLAXIS versions, interfaces can also be activated or de-activated individually. When an interface is inactive in a deformation analysis it behaves purely elastic (no slipping or gapping). In a groundwater flow calculation or a consolidation analysis, inactive interfaces are fully permeable. In fact, the (excess) pore pressure degrees-of-freedom of the corresponding node pairs are fully coupled.

De-activated clusters are drawn in the background colour (white) and de-activated structural objects or interfaces are drawn in grey. Clicking once again on a de-activated component will re-activate that component.

Anchors may only be active if the soil or structures to which they are connected are also active. Otherwise the calculation program de-activates 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.

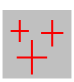
Although the external loads can be 'activated' in the initial configuration, they are not considered in the initial stress generation (*K0 procedure*). It should also be noted that weights of structural elements are disregarded in the initial stress generation. External loads or structural objects in the initial configuration therefore have no effect.

3.9.2 VIEWING OR RE-ASSIGNING 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 viewed. In contrast to the creation of material data sets in the geometry creation mode, the soil properties and model parameters can only be viewed, but cannot be changed in the data set.

It is possible to re-assign material data sets to clusters or structural objects. However, this option is usually not considered in the initial conditions because the initial material setting is directly entered during the creation of the geometry model. The option is more useful as a calculation option in the framework of *Staged construction* (Section 4.7.5).

3.9.3 INITIAL STRESS GENERATION (K0 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 effective stress $\sigma'_{v,0}$. The initial horizontal effective stress $\sigma'_{h,0}$, is related to the initial vertical effective stress by the coefficient of lateral earth pressure, K_0 ($\sigma'_{h,0} = K_0\ \sigma'_{v,0}$).

In PLAXIS initial stresses may be generated by specifying K_0 or by using *Gravity loading*. The possibilities and limitations of both methods are further described in Appendix A.

The generation of initial stresses based on the *K0 procedure* can be selected by clicking on the *Generate initial stresses* button (red crosses) in the tool bar 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 3.39).

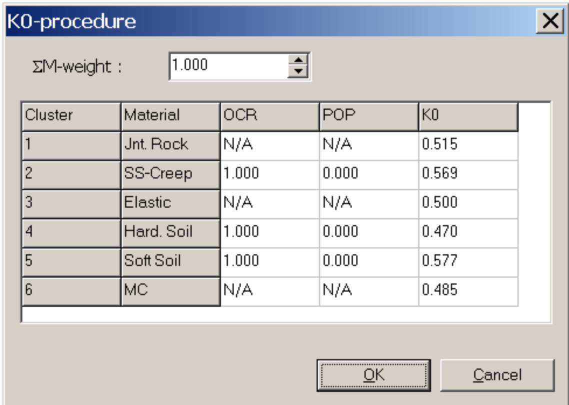


Figure 3.39 Initial stress generation window (*K0 procedure*)

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 (cross-hatched in red). 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; MCC = Modified Cam-Clay; SS = Soft Soil model; SS-Creep = Soft Soil Creep model; Jnt.Rock = Jointed Rock 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, the Hardening Soil model and the HS small 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:

The fifth column is used to enter K_0-value for all individual clusters. The default K_0-value is based on Jaky's formula (1-sinφ), but this value may be overruled by the user. Entering a negative value for K_0 will restore the default value. Be careful with very low or very high K_0-values, since these values may cause initial plasticity (see Appendix A).

Upon clicking the *OK* button, the initial stress generation starts. The K_0-procedure considers only soil weight and calculates only effective stresses and pore pressures in soil elements and interfaces. 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*:

$$\sigma'_{v,0} = \sum Mweight\left(\sum_i \gamma_i \cdot h_i - p_w\right) \qquad \sigma'_{h,0} = K_0 \sigma'_{v,0}$$

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. Although PLAXIS corrects such stress states to comply with Coulomb’s criterion, the resulting stress state may be different than expected. 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 or the Soft 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 after viewing the results of the initial stress generation, the *Update* button should be clicked.

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 tool bar, a dialog box appears in which the user is prompted to save the data. This may be done using an existing file name (just click *Yes*) or using a new name (click *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 clicking the *No* button, the data will not be saved; as a result, all data entered after a former save action is lost.

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

PLAXIS allows for a different types of finite element calculations. Groundwater flow was discussed in the previous chapter on the Input program, since a groundwater flow calculation is generally used to generate a water pressure distribution for use as input data for a deformation analysis. The Calculations program considers only deformation analyses and distinguishes between a *Plastic* calculation, a *Consolidation* analysis, *Phi-c reduction* (safety analysis) and a *Dynamic* calculation. The latter option requires the presence of the PLAXIS Dynamics module, which is available as an extension to Version 8. The first 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 the engineering practice, a project is divided into project phases. Similarly, a calculation process in PLAXIS is also divided into calculation phases. Examples of calculation phases are the activation of a particular loading at a certain time, the simulation of a construction stage, the introduction of a consolidation period, the calculation of a safety factor, etc. 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 steps). 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 procedures in PLAXIS will take care of the sub-division into appropriate load steps.

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 project file (*.PLX). 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)

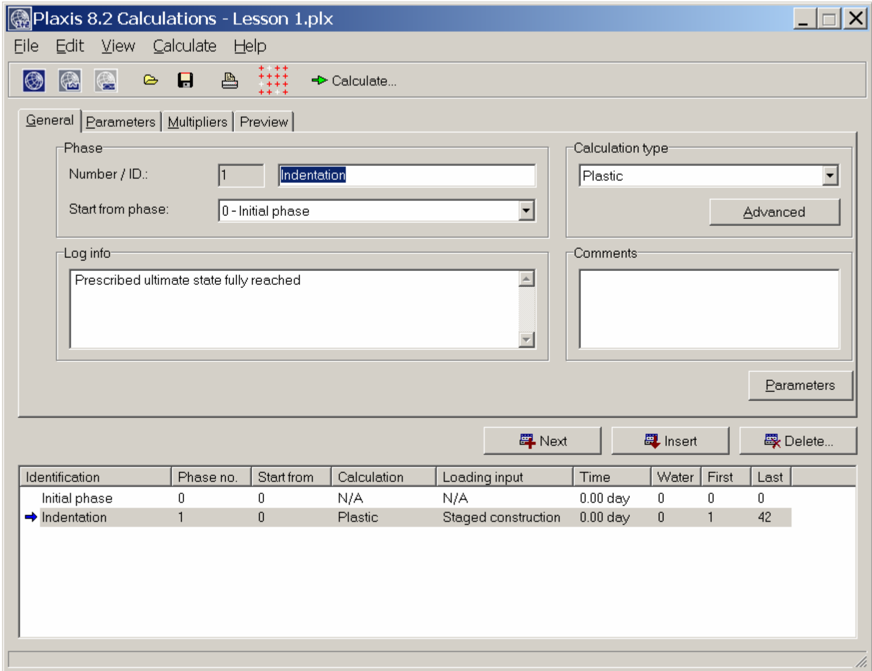


Figure 4.1 Main window of the Calculations program

Calculations menu:

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

Tool bar:

The tool bar 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 and preview 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 time interval, the water pressure situation to be used and the first and last step number.

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 of calculation phases 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.
Print	To print the list of calculations phases.
Work directory	To set the directory where PLAXIS 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.
Copy to clipboard	To copy the list of calculation phases to the clipboard.
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. After selection of a project, the project is added to the calculation manager window.

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' with phase number 0. 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 clicked 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. On pressing the *Enter* or *Tab* key after each input parameter, 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 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 tool bar 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 clicking 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 active clusters, structural objects, loads, water conditions and multipliers is adopted from the previous phase.

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

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).

Calculation type:

The selections made in the *Calculation type* group determine the type of calculation that is used (Section 4.4.2).

Log info and Comments:

The *Log info* box displays messages generated during the finite element calculation and is used for logging purposes. The *Comments* box can be used to store any information related to a particular calculation phase.

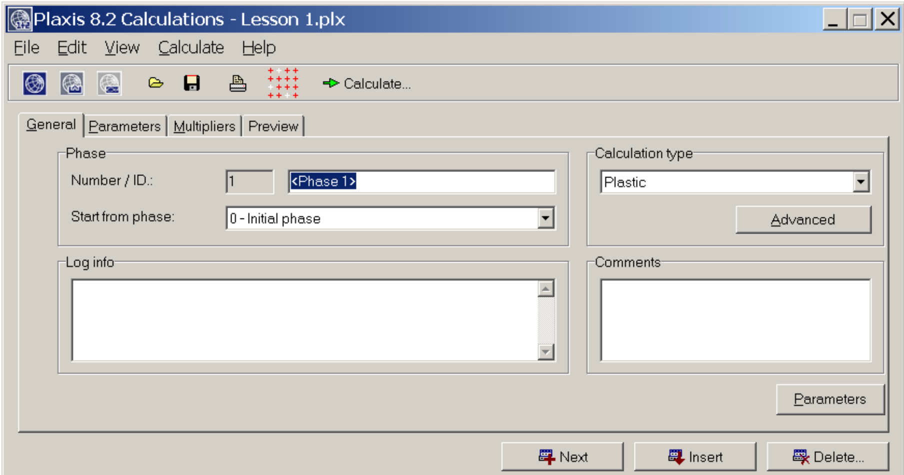


Figure 4.2 *General* tab sheet of the Calculations window

4.4.1 PHASE IDENTIFICATION AND ORDERING

The *Phase* group in the *General* tab sheet shows the phase number and an identification string of the current calculation phase. PLAXIS automatically assigns a number to each calculation phase which cannot be changed by the user. The identification string is, by default, set to <Phase #>, where # is the 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 list of calculation phases at the lower part of the window.

In addition, the *Start from phase* parameter must be selected from the combo box in the *Phase* group. 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 4.9), 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, 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 CALCULATION TYPES

The type of calculation (or shorter *Calculation type*) of a phase is primarily defined 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 *Plastic* calculation, a *Consolidation* analysis and *Phi-c reduction* (safety analysis). Optionally, a *Dynamic* calculation is available in the combo box, but this requires the presence of the PLAXIS Dynamics module, which is available as an extension to Version 8.

Plastic calculation

A *Plastic* calculation should be selected to carry out an elastic-plastic deformation analysis in which it is not necessary to take the decay of excess pore pressures with time into account. If the Updated Mesh option in the advanced general settings window has not been selected, the calculation is performed according to the small deformation theory. The stiffness matrix in a normal plastic calculation is based on the original undeformed geometry. This type of calculations is appropriate in most practical geotechnical applications.

Although a time interval can be specified, a plastic calculation does not take time effects into account, except when the Soft Soil Creep model is used (see Material Models Manual). Considering the quick loading of water-saturated clay-type soils, a plastic calculation may be used for the limiting case of fully undrained behaviour using the *Undrained* option in the material data sets. On the other hand, performing a fully drained analysis can assess the settlements on the long term. This will give a reasonably accurate prediction of the final situation, although the precise loading history is not followed and the process of consolidation is not dealt with explicitly.

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

Consolidation analysis

A *Consolidation* analysis should be selected when it is necessary to analyse the development or dissipation of excess pore pressures in water-saturated clay-type soils as a function of time. PLAXIS allows for true elastic-plastic consolidation analyses. In general, a 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 situations. Please note that some of the limitations of PLAXIS Version 7 regarding consolidation analyses have been improved in this version. For example, it is now possible to apply construction stages in time using a consolidation analysis. Moreover, consolidation analyses can be performed in the framework of large deformations. For more details on theoretical formulations, reference should be made to the Scientific Manual.

Phi-c reduction (safety analysis)

A safety analysis in PLAXIS 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 4.9.

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. This option can be selected using the *Advanced* button in the *Calculation type* group of the *General* tab sheet. It can also be selected whether water pressures should be continuously recalculated according to the updated position of the stress points. This option is termed *Updated water pressures* and is meant to take into account the effects of soil settling (partly) below a constant phreatic level.

An *Updated Mesh* analysis is a calculation where effects of large deformations are taken into account. This type of calculation should be considered when deformations are to be expected that significantly influence 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 include rotation terms. These calculation procedures are based on an approach known as an Updated Lagrange formulation (Ref. 2). More information on this subject can be found in Section 4.11.

For most applications the effects of large deformations are negligible so that it is not necessary to select this advanced option, but there are circumstances under which it may be necessary 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.

Please note that an updated mesh calculation cannot be followed by a 'normal' calculation. Reversely, a normal calculation can be followed by an updated mesh calculation, provided that the option *Reset displacements to zero* is used (Section 4.6).

It should be noted that an updated mesh analysis takes much more time and is less robust than a normal calculation. Hence, this option should only be used in special cases.

4.5 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 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 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 (Section 4.6.1). 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.

4.5.1 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 (Section 4.5.2) or manually by the user (Section 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. The following three outcomes of this process are possible:

- Case 1: The solution reaches equilibrium within a number of iterations that is less than the *Desired minimum* control parameter. By default, the *Desired minimum* is 4, but this value may be changed in the *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 10, 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*: the load increment size is then 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.

4.5.2 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 performs a trial calculation step and determines a suitable step size on the basis of this trial.
- PLAXIS 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 four following criteria has been satisfied:

- The total specified load has been applied. In this case the calculation phase has successfully 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 and the current stiffness parameter CSP is less than 0.02 (see Section 4.18 for the definition of CSP). The following message is displayed in the *Log info* box: *Prescribed ultimate state not reached; Soil body collapses*.
- The load advancement procedure is unable to further increase the applied load, but the current stiffness parameter CSP is larger than 0.02. The following message is displayed in the Log info box: *Prescribed ultimate state not reached: load advancement procedure fails. Try manual control.* In this case the total load specified has not been applied. In this case the user can attempt to rerun the calculation with slight changes to the Iterative procedure settings, in particular deselecting the *Arc-length control* option.

4.5.3 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.

4.5.4 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 Section 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.

4.6 CALCULATION CONTROL PARAMETERS

The *Parameters* tab sheet is used to define the control parameters of a particular calculation phase and the corresponding solution procedure (Figure 4.3).

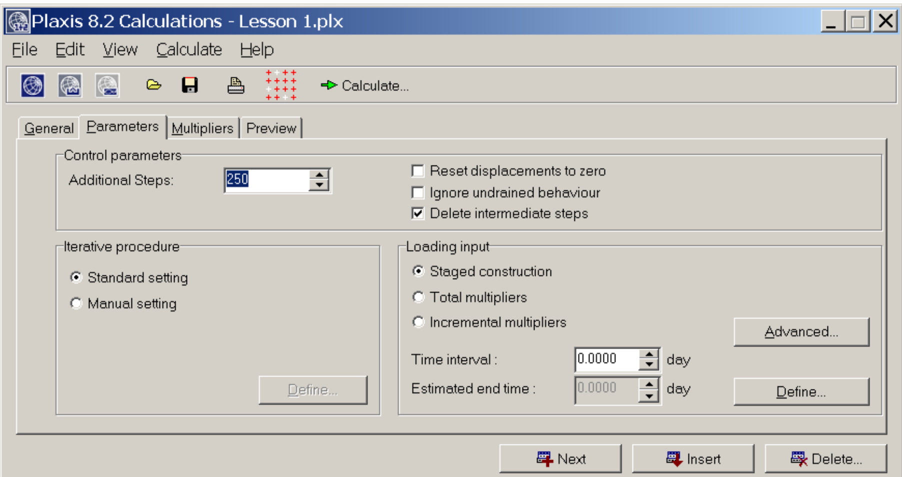


Figure 4.3 *Parameters* tab sheet of the Calculations window

The *Parameters* tab sheet contains the following items:

Additional steps

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

If a *Plastic* calculation or a *Consolidation* analysis is selected as the calculation type and the loading input is set to *Staged construction*, *Total multipliers* or *Minimum pore* *pressure*, then the number of additional steps is an upper bound to the actual number of steps that will be executed. In general, it is desired that such a calculation is completed within the number of additional steps and stops according to the first or third criterion as described in Section 4.5.2 (*Prescribed ultimate state reached* or *soil body collapses*). If such a calculation reaches the maximum number of additional steps, it usually means that the ultimate level has not been reached. By default, the *Additional steps* parameter is set to 250, which is generally sufficient to complete the calculation phase. However, this number may be changed within the range 1 to 1000.

If a *Plastic* calculation or a *Consolidation* analysis is selected and the loading input is set to *Incremental multipliers*, then the number of additional steps should be set 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. By default, the *Additional steps* parameter is set to 250, but this number can be changed within the range 1 to 1000. The same applies to a *Phi-c reduction* calculation, except that the *Additional steps* parameter is, by default, set to 30.

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 an Updated Mesh calculation starts from a calculation where the Updated Mesh option is not used, then the *Reset displacements to zero* option **must** be used in this 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 material data sets are used in which the *Material type* is set to *Undrained*. As a result, all undrained material clusters become temporarily drained. Existing excess pore pressures that were previously generated will remain, but no new excess pore pressures will be generated in that particular calculation phase.

Gravity loading of undrained materials will result in unrealistic excess pore pressures. Stresses due to the self-weight of the soil, for example, are based on a long-term process in which the development of excess pore pressures is irrelevant. The *Ignore undrained behaviour* option enables the user to specify the material type from the beginning as undrained for the main loading stages and to ignore the undrained behaviour during the gravity loading stage. 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 is by default selected to save disk space. As a result, all additional output steps within the calculation phase, except for the last one, are deleted when a calculation phase has finished successfully. In general the final output step contains the most relevant result of the calculation phase, whereas intermediate steps are less important. If desired, the option can be de-selected to retain all individual output steps. If a calculation phase does not finish successfully then all calculation steps are retained, regardless of the selection of the Delete intermediate steps option. This enables a stepwise evaluation of the cause of the problem.

4.6.1 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. PLAXIS has an option to adopt a *Standard setting* for these parameters, which in most cases leads to 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*.

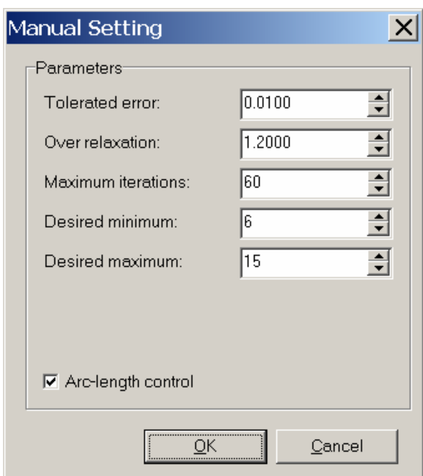


Figure 4.4 Iterative procedure control parameters window

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 4.4).

Tolerated error

In any non-linear analysis where a finite number of calculation steps are used there will be some drift from the exact solution, as shown in Figure 4.5. The purpose of a solution algorithm is to ensure that the equilibrium errors, both locally and globally, remain within acceptable bounds (Section 4.19). The error limits adopted in PLAXIS 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 but for failure load calculations it may be more practical to use an increased value of 0.03 or even 0.05.

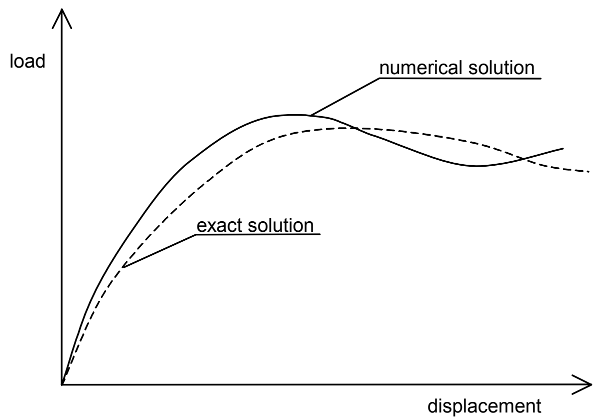


Figure 4.5 Computed solution versus exact solution

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 see Section 4.19.

Over-relaxation

To reduce the number of iterations needed for convergence, PLAXIS makes use of an over-relaxation procedure as indicated in Figure 4.6. The parameter that controls the degree of over-relaxation is the over-relaxation factor. The theoretical upper bound value is 2.0, but this value should never be used. For low soil friction angles, for example 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.

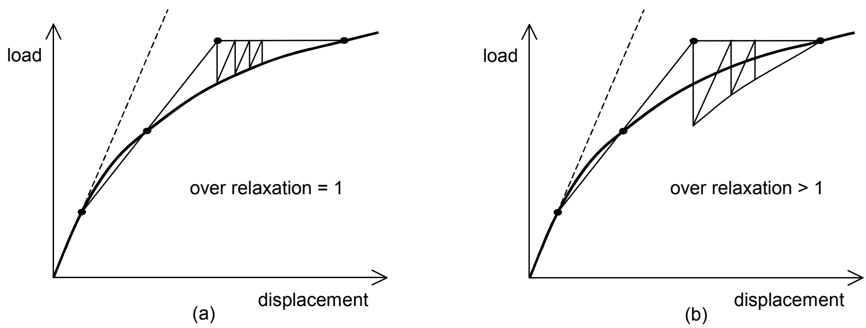


Figure 4.6 Iteration process with (a) and without (b) over-relaxation

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 a *Plastic* calculation type or a *Phi-c reduction* analysis is selected as calculation type then PLAXIS makes use of an automatic step size algorithm (*Load advancement ultimate level* or *Number of steps*). 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 4.5.1 to 4.5.3.

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

Desired minimum = 3 *Desired maximum* = 7

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

Desired minimum = 7 *Desired maximum* = 15

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

Arc-length control

The *Arc-length* control procedure is a method that is by default selected in a *Plastic* calculation or a *Phi-c reduction* calculation to obtain reliable collapse loads for load-controlled calculations (Reference 9). Arc-length control is not available for *Consolidation* analyses.

The iterative procedure adopted when arc-length control is not used is shown in Figure 4.7a for the case where a collapse load is being approached. In the case shown, the algorithm will not converge. If arc-length control is adopted, however, the program will automatically evaluate the portion of the external load that must be applied for collapse as shown in Figure 4.7b.

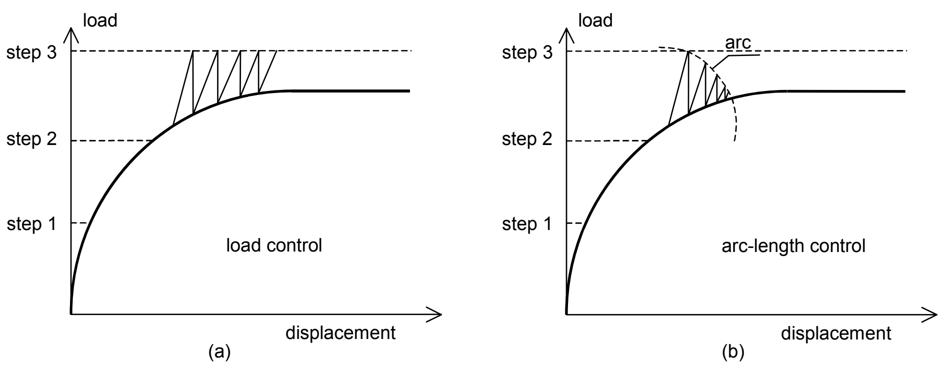


Figure 4.7 Iterative procedure for normal load control (a) and arc-length control (b)

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 multipliers* as loading input, 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 occasionally causes spontaneous unloading to occur (i.e. sudden changes in sign of the displacement and load increments) when the soil body is far from collapse. If this occurs, then the user is advised to de-select *Arc-length control* and restart the calculation. 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.

Care should be taken with time steps that are smaller than the advised minimum time step. As for most numerical integration procedures, accuracy increases when the time step is reduced, but for consolidation there is a threshold value. Below a particular time increment (critical time step) the accuracy rapidly decreases. For one-dimensional consolidation (vertical flow) this critical time step is calculated as:

$$\Delta t_{critical} = \frac{H^2 \gamma_w (1-2\nu)(1+\nu)}{80\ k_y E(1-\nu)} \qquad \text{(15-node triangles)}$$

$$\Delta t_{critical} = \frac{H^2 \gamma_w (1-2\nu)(1+\nu)}{40\ k_y E(1-\nu)} \qquad \text{(6-node triangles)}$$

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, the reader is referred to Reference 19. Detailed information for various types of finite elements is given by Reference 15.

Extrapolation

Extrapolation is a numerical procedure, which is automatically used in PLAXIS 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 4.8). 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.

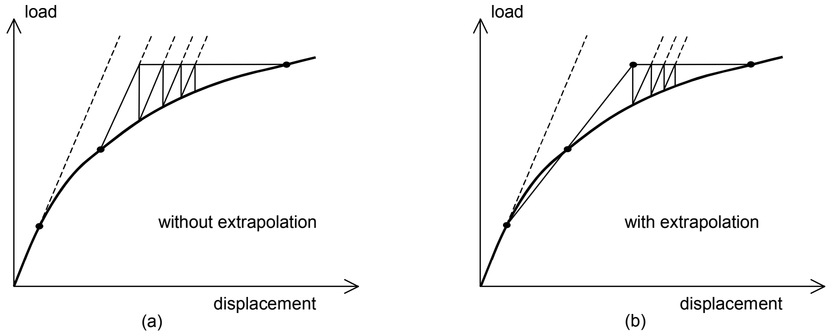


Figure 4.8 Difference between elastic prediction (a) and extrapolation from previous step (b)

4.6.2 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 4.7).
- 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 current configuration of external loads and 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 4.5.2), 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 *ΣMstage*. 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 *ΣMstage* 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 *ΣMstage* 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 4.8.2).

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, the user 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. The *Minimum pore pressure* option may be used to specify the final degree of consolidation in any analysis. In order to specify an appropriate value of minimum pore pressure, *P-stop*, it is necessary to determine the maximum absolute excess pore pressure immediately after loading. This parameter, *Pmax*, is displayed in the *Multipliers* tab sheet of the previous calculation phase when selecting the *Reached values* option (Section 4.8.2). A suitable value of *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 in the *Dimensions* tab sheet of the *General settings* window. A non-zero value for the *Time increment* or *Time interval* parameters is only relevant when a consolidation analysis is performed, when transient groundwater flow is considered 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.

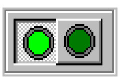
4.7 STAGED CONSTRUCTION

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

To carry out a staged construction calculation, it is first necessary to create a geometry model that includes all of the objects that are to be used during the calculation. Objects that are not required at the start of the calculation should be deactivated in the initial geometry configuration at the end of the Input program (Section 3.9.1).

A staged construction analysis can be executed in a *Plastic* calculation as well as a *Consolidation* analysis. In the *Parameters* tab sheet, the *Staged construction* option can be selected in the *Loading input* box. On subsequently clicking on the *Define* button, the Input program is started and the staged construction window appears. This window is similar to the initial conditions window, except that options that are only relevant for the initial conditions (such as the *K0 procedure*) cannot be activated. It is also not possible to enter the geometry creation mode of the Input program from the staged construction window. On the other hand, specific staged construction options are available.

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. The geometry configuration mode can be used 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. The water conditions mode can be used 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.



Switching between the water conditions mode and the geometry configuration mode can be achieved using the 'switch' in the tool bar. After the new situation has been defined, the *Update* button should be clicked to store the information and return to the Calculations program. In addition, the next calculation phase may be defined or the calculation process may be started.

Changes to the geometry configuration or the water conditions generally cause substantial out-of-balance forces. These out-of-balance forces are stepwise applied 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 ($\Sigma Mstage$) is increased from zero to the ultimate level (generally 1.0). In addition, a parameter representing the active proportion of the geometry ($\Sigma Marea$) is updated.

4.7.1 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. This can be done by clicking on the object in the geometry model. When clicking once on an object, the object will change from active to inactive, 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.

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 soil cluster, you can either change the material properties (Section 4.7.5) or apply a volume strain to the selected cluster (Section 4.7.6).

In contrast to previous PLAXIS versions, 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).

4.7.2 ACTIVATING AND DEACTIVATING CLUSTERS OR STRUCTURAL OBJECTS

Soil clusters and structural objects can be activated or deactivated by clicking once on the cluster or structural object in the geometry model. 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 geometry 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:

- Properties, such as weight, stiffness and strength, are not taken into account.
- All stresses are set to zero.
- All inactive nodes will have zero displacements.

- 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 Lesson 4 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:

- Stiffness and strength will be fully taken into account from the beginning (i.e. the first step) of the calculation phase.
- Weight will, in principle, be fully taken into account from the beginning of the calculation phase. However, in general, a large out-of-balance force will occur at the beginning of a staged construction calculation. This out-of-balance force is stepwise solved in subsequent calculation steps.
- 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.

4.7.3 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 clicking once on the load in the geometry model. 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 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.

- After double-clicking a point load a window appears in which the *x*- and *y*-components can be entered directly (Figure 4.9).
- After double-clicking a distributed load a window appears in which the *x*- and *y*-components can be entered directly at the two respective geometry points (Figure 4.10).

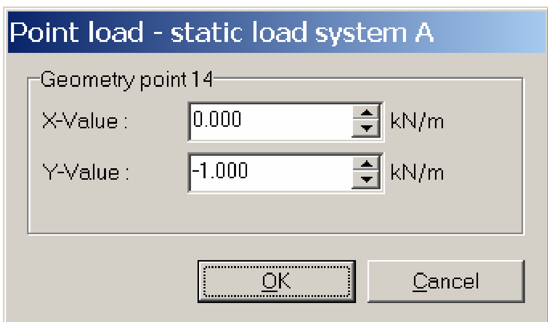


Figure 4.9 Input window for a point load

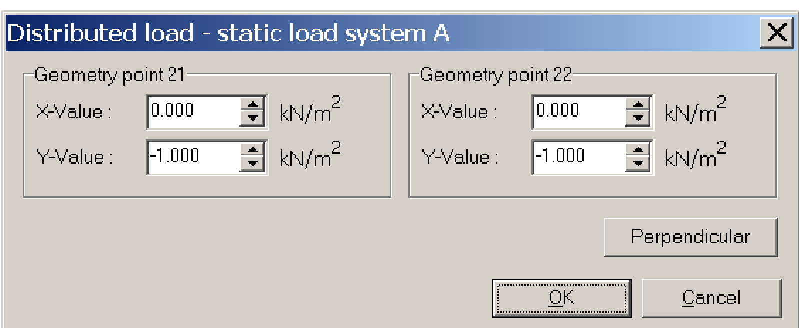


Figure 4.10 Input window for a distributed load

The *Perpendicular* button may be used to make sure that the distributed load is perpendicular to the corresponding geometry line.

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 ($\Sigma MloadA$ or $\Sigma MloadB$). The multiplier $\Sigma MloadA$ is used to globally increase (or decrease) all loads of load system A (point loads and distributed loads), whereas $\Sigma MloadB$ is used to change all loads of load system B (Section 4.8.1). 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 initially set the corresponding multiplier to unity.

4.7.4 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 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 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

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 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.

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 (*ΣMdisp*). The multiplier *ΣMdisp* is used to globally increase (or decrease) all prescribed displacements (Section 4.8.1). 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 initially set the corresponding multiplier to unity.

4.7.5 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.

On double-clicking a soil cluster or structural object in the geometry model, the properties window appears in which the material data set of that object can be changed.

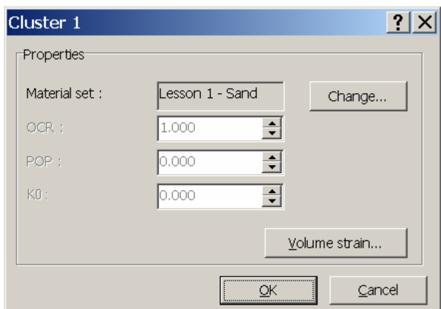


Figure 4.11 Soil properties window

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 data base. 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 geometry model. The material data set of the cluster can be changed by clicking the *Change* button. As a result, the material data base is presented with all existing material data sets. The parameters of existing data sets can be viewed (not changed) by selecting the desired data set and clicking the *View* button.

After selecting the appropriate material data set from the data base tree view and clicking the *OK* button the data set is assigned to the soil cluster or structural object.

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 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 beam 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.

4.7.6 APPLYING A VOLUMETRIC STRAIN IN VOLUME CLUSTERS

In PLAXIS you can impose an internal volumetric strain in soil clusters. This option may be used to simulate mechanical processes that result in volumetric strains in the soil, such as grouting. In the properties window that appears after double-clicking a soil cluster, you can click the *Volumetric strain* button.

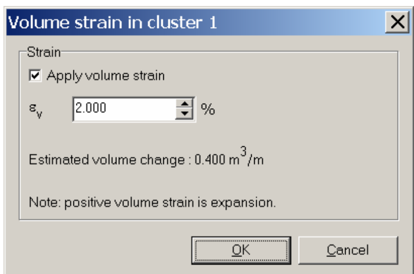


Figure 4.12 Volume strain window

In the Volume strain window that appears you can specify the volumetric strain. In addition, an estimation of the total volume change is given in the unit of volume per unit of width in the out-of-plane direction.

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 expansion, whereas a negative value represents a volume decrease.

4.7.7 PRE-STRESSING OF ANCHORS

Pre-stressing of anchors can be activated from the geometry configuration mode of the staged construction window. Therefore 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. A pre-stress force should be given as a force per unit of width in the out-of-plane direction. 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 clicked. 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.

4.7.8 APPLYING CONTRACTION OF A TUNNEL LINING

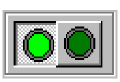
To simulate soil volume loss due to the construction of a shield tunnel, the contraction method may be used. In this method a contraction is applied to the tunnel lining to simulate a reduction of the tunnel cross-section area. The contraction is expressed as a percentage, representing the ratio of the 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 (Section 3.3.8).

Contraction can be activated in the staged construction mode by 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.

As the contraction is applied to the tunnel lining (shell elements) these must be present and active during the phase a contraction is applied. Note that no contraction can be applied to a tunnel lining represented by volume elements.

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)

4.7.9 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. Examples of problems that may be analysed using this option include the settlement of soft soil layers due to a lowering of the water table, the deformation and force development of walls or tunnel linings due to excavation and dewatering, and the stability of a river embankment after an increase of the external water level.

Click on the left-hand side of the ‘switch’ in the tool bar 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 levels and eventual boundary conditions for a groundwater flow calculation. A new set of phreatic levels or boundary conditions for a groundwater flow calculation may now be created. For a description of the input of phreatic levels and boundary conditions, reference is made to Section 3.8.2 and 3.8.3 respectively.

After the new input, the water pressures must first be generated before clicking the *Update* button. The water pressures can be generated by clicking on the *Generate water pressures* button in the tool bar or by selecting the *Water pressures* option from the *Generate* menu. In the *Water pressure generation* window it must be indicated whether the water pressures should be generated on the basis of phreatic levels or by means of a groundwater flow calculation. On clicking the *OK* button, the calculation will start (Section 3.8.4).

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.

Excavation and dewatering

Special attention is paid here to the simulation of staged excavation and dewatering, as considered in Lesson 4 (Chapter 6) of the Tutorial Manual. If dewatering of an excavation is considered and the bottom of the excavation is not fully 'closed' by means of an injected or impermeable layer, then groundwater flow will occur. This process can be simulated by means of a groundwater flow calculation in PLAXIS. Groundwater flow influences the pore pressure distribution in the surrounding soil.

The boundary conditions for a groundwater flow calculation for such a situation can conveniently be set by changing the general phreatic level such that it represents the initial phreatic level at the outer geometry boundary and that it represents the lowered water table within the excavation. It is likely that the general phreatic level is then composed of multiple points. Based on this general phreatic level PLAXIS automatically sets a prescribed groundwater head to permeable geometry boundaries, including the new 'internal' boundaries that arise due to the excavation, i.e. geometry lines separating active and inactive clusters. Walls can be made impermeable by activating the adjacent interface elements, provided that the interface permeability is set to impermeable (Section 3.3.5). If only one symmetric half of the excavation is modelled, then the 'centre line' must be made impermeable by using the *Closed flow boundary* option. This may also apply to the bottom boundary, if in reality the soil is impermeable here.

After the boundary conditions have been set, the water pressures can be generated by clicking on the *Generate water pressures* button in the tool bar. In the *Water pressure generation* window the *Groundwater calculation* option must be selected.

When water pressures are generated by means of a groundwater flow calculation, then the general phreatic line is used for convenience to set the boundary conditions for the groundwater flow calculation, to generate the external water pressures for a deformation analysis. The general phreatic line does not have a meaning inside active soil clusters, since the pore pressure distribution inside active soil clusters is computed in the groundwater flow calculation from the boundary conditions and the permeability of the soil.

4.7.10 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. If desired, the *Tolerated error* may be reduced by selecting the *Manual setting* of the *Iterative* procedure in the *Parameters* tab sheet.

When creating a new calculation phase using the *Next* or *Insert* button in the Calculations window, the default setting is such that this phase can directly serve as a plastic nil-step.

4.7.11 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.

Tunnel construction with ΣMstage<1

In addition to the simulation of the construction of shield tunnels using the contraction method (Section 4.7.8), it is possible with PLAXIS to simulate the construction process of tunnels with a sprayed concrete lining (NATM). The major point in such an analysis is to account for the three-dimensional arching effect that occurs within the soil and the deformations that occur around the unsupported tunnel face. A method that takes these effects into account is described below.

There are various methods described in the literature for the analysis of tunnels constructed according to the New Austrian Tunnelling Method. One of these is the so-called β-method (Reference 11), but other have presented similar methods under different names. The idea is that the initial stresses p_k acting around the location where the tunnel is to be constructed are divided into a part $(1-\beta)\ p_k$ that is applied to the unsupported tunnel and a part $\beta\, p_k$ that is applied to the supported tunnel (Figure 4.13). The β-value is an 'experience value', which, among other things, depends on the ratio of the unsupported tunnel length and the equivalent tunnel diameter. Suggestions for this value can be found in literature (Reference 11).

Instead of entering a β-value in PLAXIS, one can use the staged construction option with a reduced ultimate level of *ΣMstage*. In fact, when deactivating the tunnel clusters an initial out-of-balance force occurs that is comparable with p_k. In the beginning of the staged construction calculation, when *ΣMstage* is zero, this force is fully applied to the active mesh and it will be stepwise decreased to zero with the simultaneous increase of *ΣMstage* towards unity. Hence, the value of *ΣMstage* can be compared with $1-\beta$. In order to allow for the second step in the β-method, the ultimate level of *ΣMstage* should be limited to a value of $1-\beta$ while deactivating the tunnel clusters. This can be done by clicking on the *Advanced* button while the *Staged construction* option has been selected from the *Loading input* group of the *Parameters* tab sheet. In general, care must be taken with an ultimate level of *ΣMstage* smaller than 1.0, since this is associated with a resulting out-of-balance force at the end of the calculation phase. In this case the next calculation phase is a staged construction calculation in which the tunnel construction is completed by activating the tunnel lining. By default, the ultimate level of *ΣMstage* is 1.0. Hence, the remaining out-of-balance force will be applied to the geometry including the tunnel lining.

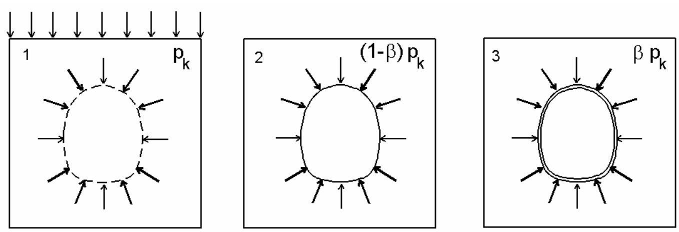


Figure 4.13 Schematic representation of the β-method for the analysis of NATM tunnels

The process is summarised below:

1. Generate the initial stress field and apply eventual external loads that are present before the tunnel is constructed.
2. De-activate the tunnel clusters without activation of the tunnel lining and apply an ultimate level of *ΣMstage* equal to $1-\beta$.
3. Activate the tunnel lining.

4.7.12 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 has 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. The reached value of the *ΣMstage* multiplier may be viewed by selecting the *Reached values* option in the *Show* group on the *Multipliers* tab sheet (4.8.2).

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 not possible 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. Alternatively, the phase may be recalculated using a larger number of *Additional steps*. 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.

In the case of an unfinished staged construction calculation, the load that has actually been applied differs from the defined load configuration. The reached value of the *ΣMstage* multiplier may be used in the following way to estimate the load that has actually been applied:

$$f_{applied} = f_0 + \Sigma Mstage\ (f_{defined} - f_0)$$

where $f_{applied}$ is the load that has actually been applied, f_0 is the load at the beginning of the calculation phase (i.e. the load that has been reached at the end of the previous calculation phase) and $f_{defined}$ is the defined load configuration.

4.8 LOAD MULTIPLIERS

During a deformation analysis, it is necessary to control the magnitude of all types of loading. In general, loads are activated in the framework of staged construction by entering an appropriate input value. Nevertheless, the loadings to be applied are calculated from the product of the input value of the load and the corresponding multiplier. Hence, as an alternative to staged construction, loads can globally be increased by changing the corresponding multiplier. Distinction is made between *Incremental multipliers* and *Total multipliers*. 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 *Loading input* as selected in the *Parameters* tab sheet. Both the incremental multipliers and the total multipliers for a particular calculation phase are displayed in the *Multipliers* tab sheet (Figure 4.14). 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.

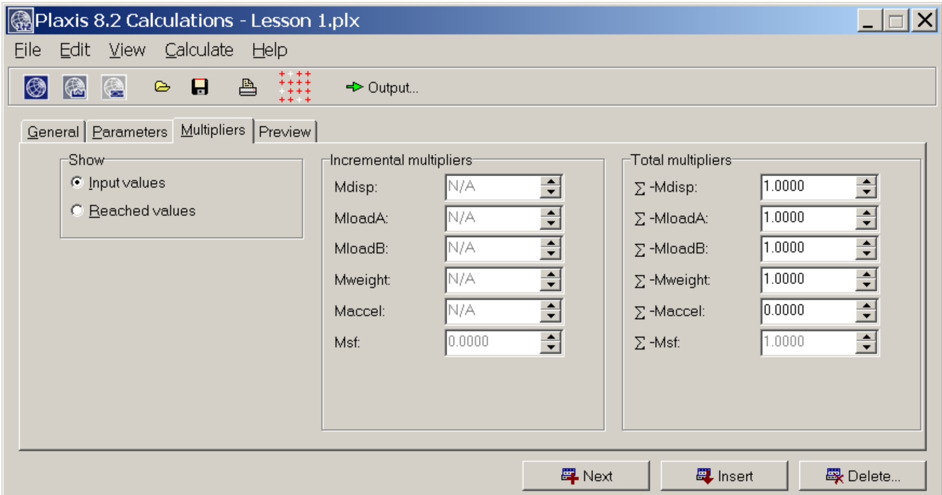


Figure 4.14 *Multipliers* tab sheet of the Calculations window

4.8.1 STANDARD LOAD MULTIPLIERS

Mdisp, ΣMdisp:

These multipliers control the magnitude of prescribed displacements as entered in the staged construction mode (Section 4.7.4). 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 and point loads as entered in the load systems A and B (Section 4.7.3). 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.

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.

Maccel, ΣMaccel

These multipliers control the magnitude of the pseudo-static forces as a result of the acceleration components as entered in the *General settings* window of the Input program (Section 3.2.2). The total magnitude of the acceleration applied during the calculation is the product of the input values of the acceleration components and the parameter *ΣMaccel.* Initially, the value of *ΣMaccel* is set to zero. In calculations where the *Loading input* was set to *Incremental multipliers*, *Maccel* can be used to specify the increment of acceleration of the first calculation step.

Pseudo-static forces can only be activated if the weight of the material is already active (*ΣMweight* = 1). For *ΣMweight* = 1 and *ΣMaccel* = 1 both gravity forces and pseudo-gravity forces are active. The figure below gives an overview of different combinations of soil weight and acceleration. Note that the activation of an acceleration component in a particular direction results in a pseudo-static force in the opposite direction. When increasing *ΣMweight* without increasing *ΣMaccel* the resulting force will be increased without a change of the resulting direction.

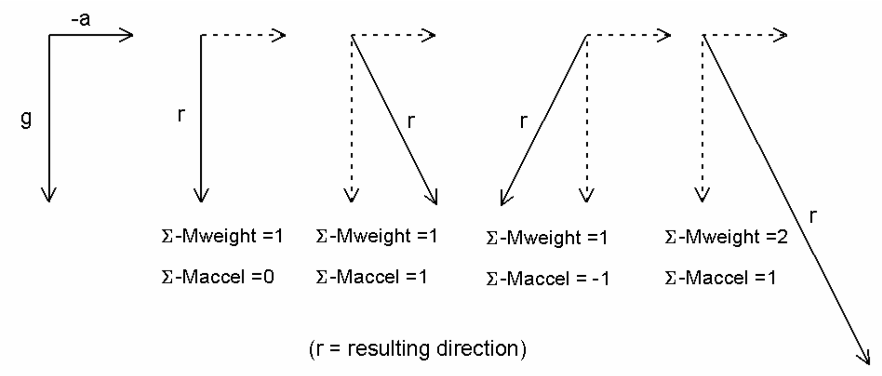


Figure 4.15 Resulting force direction *r* due to combinations of gravity and acceleration *a*

Msf, ΣMsf:

These multipliers are associated with the *Phi-c reduction* option in PLAXIS for the computation of safety factors (Section 4.9). The total multiplier *ΣMsf* is defined as the quotient of the original strength parameters and the reduced strength parameters and controls the reduction of tanφ and *c* at a given stage in the analysis. *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. This may be the case if failure of the soil body occurs. 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.

4.8.2 OTHER MULTIPLIERS AND CALCULATION PARAMETERS

ΣMstage:

The *ΣMstage* parameter is associated with the *Staged construction* option in PLAXIS (Section 4.7). 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.

Σ*Marea:*

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 geometry 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:

These parameters indicate the forces corresponding to the non-zero prescribed displacements (Section 3.4.1). In plane strain models, *Force-X* and *Force-Y* are expressed in the unit of force per unit of width in the out-of-plane direction. In axisymmetric models, *Force-X* and *Force-Y* are expressed in the unit of force per radian. In order to calculate the total reaction force under a circular footing simulated by prescribed displacements, *Force-Y* should be multiplied by 2π. *Force-X* and *Force-Y* are the value of the total force in the *x*- and *y*-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, whereas *Pmax* generally decreases during a consolidation analysis.

4.9 PHI-C-REDUCTION

Phi-c reduction is an option available in PLAXIS to compute safety factors. This option can be selected as a separate *Calculation type* in the *General* tab sheet. In the *Phi-c reduction* approach the strength parameters tanφ 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 plates 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. ΣMsf is set to 1.0 at the start of a calculation to set all material strengths to their unreduced values.

A *Phi-c reduction* calculation is performed using the *Load advancement number of steps* procedure. The incremental multiplier *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. The strength parameters are successively reduced automatically until all *Additional steps* have been performed. By default, the number of additional steps is set to 100, but a larger value up to 1000 may be given here, if necessary. It must always be checked whether the final step has resulted in a fully developed failure mechanism. If that is the case, the factor of safety is given by:

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

If a failure mechanism has not fully developed, then the calculation must be repeated with a larger number of additional steps.

To capture the failure of the structure accurately, the use of *Arc-length control* in the iteration procedure is required. The use of a *Tolerated error* of no more than 3% is also required. Both requirements are complied with when using the *Standard setting* of the *Iterative procedure*.

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.

The *Phi-c reduction* approach resembles the method of calculating safety factors as conventionally adopted in slip-circle analyses. For a detailed description of the method of *Phi-c reduction* see Reference 4.

4.10 SENSITIVITY ANALYSIS & PARAMETER VARIATION

After a project has been completely defined, the *Calculations* program allows for an analysis of the influence of variations of parameters on the computational results. Preferably, the project should have been calculated and the user should have verified that the project is consistent and the results are useable. Variations that can be considered include mainly model parameters of material data sets for *Soil & Interfaces*, *Plates*, *Geogrids* and *Anchors*. Moreover, small geometric variations that use the same finite element mesh may also be considered, such as a variation in water pressures or a variation in the magnitude of a load, provided that exactly the same finite element mesh is used. Variations of model parameters, named *Material variations*, can be directly defined in a separate window that appears after selecting the *Sensitivity* option or the *Parameter variation* option from the *Calculate* sub-menu. Small geometric variations and combinations of geometric variations must be made manually on a copy of the original project.

4.10.1 PARAMETER VARIATION

The *Parameter variation* option is used to analyse the upper and lower bounds of results by performing complete calculations for all combinations of the upper and lower bound values of the parameters to be varied. In this respect, a complete calculation involves all defined calculation phases after the initial phase. If *n* is the number of parameters to be varied, the total number of complete calculations is 2^n+1 (where +1 is a copy of the original project). Hence, if *n* is a large number, the complete analysis may take hours or even days to perform. Some parameters will have a larger influence on the variation of results than others, and there may be even parameters whose influence on the variation of results is negligible. Therefore, it may be useful to analyse the influence of individual parameter variations first (this is called a *Sensitivity* analysis) and then perform the *Parameter variation* analysis with only those parameters that have a significant influence.

4.10.2 SENSITIVITY ANALYSIS

The *Sensitivity* option is used to analyse the influence of individual parameter variations on the results with the purpose to evaluate the relative influence of those parameters. The relative influence (sensitivity) is evaluated on the basis of a user-defined criterion;

for example the horizontal displacement of a particular node. Please note that nodes or stress points used in these criteria can only be taken from the set of nodes and stress points that have been selected for load-displacement curves or stress-strain curves (see Section 4.13).

In a *Sensitivity* analysis the upper and lower bound values of parameters are varied individually. If *n* is the number of parameters to be varied, the total number of complete calculations is 2*n*+1 (where +1 is a copy of the original project). Note that for *n*>2 the number of calculations required for a sensitivity analysis is less than the number of calculations required for *Parameter variation*. Therefore, it may be efficient to first perform a sensitivity analysis in order to identify the parameters with the largest influence on the results, and then perform a parameter variation analysis with a reduced number of parameters to be varied.

4.10.3 DEFINING PARAMETER VARIATIONS

Both the *Sensitivity* and the *Parameter variation* option can be selected from the *Calculate* sub-menu. To define the variations of parameters, select the *Define & Run* option behind the corresponding menu item. As a result, a new window appears showing an empty list of material variations and also an empty list of geometric variations (Figure 4.16).

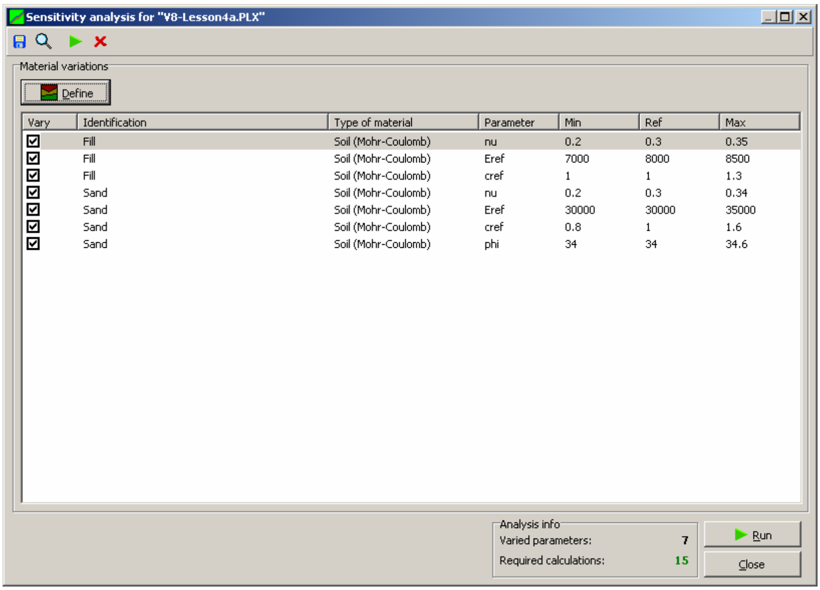


Figure 4.16 *Sensitivity* and *Parameter variation* analyses window

To define variations of model parameters (*Material variations*), click on the *Define* button in the *Material variations* group box. As a result, a new window opens with tab sheets for the different types of material data sets, i.e. *Soil & Interfaces*, *Plates*, *Geogrids* and *Anchors*. Each tab sheet shows the corresponding predefined material data sets. After selecting a data set from the list of predefined sets, the lower part of the window shows the corresponding parameters with their input value.

A parameter for which a variation is considered can be selected by clicking on the corresponding check box. The lower and upper bound values of the parameter can be specified in the *Min* and *Max* boxes behind the original input value. The *Min*-value must be smaller or equal to the input value. The *Max*-value must be larger or equal to the input value. If not all of the model parameters fit in the window, a scroll bar is available at the right-hand side, which may be used to reach the model parameters below the visible ones. After all desired parameters have been selected and their upper and lower bound values have been specified, press the *OK* button to return to the *Sensitivity* or *Parameter variations* window. The selected parameters are now listed in this window. The check box before each parameter can be used to select whether or not to take the variation of the parameter into account.

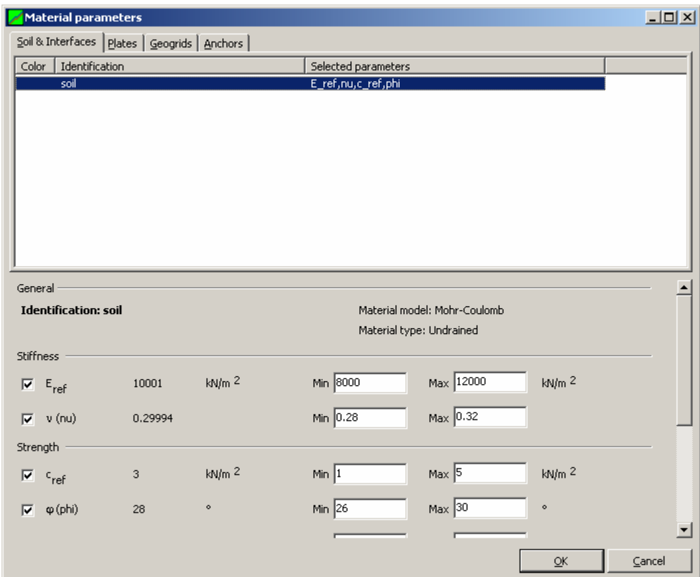


Figure 4.17 *Material parameters* window

4.10.4 DEFINING GEOMETRIC VARIATIONS

In addition to model parameters, the *Parameter variation* option allows for small geometric variations such as a variation in water pressures or a variation in the magnitude of a load, provided that exactly the same finite element mesh is used as for the original project. Before such a geometric variation can be selected in the *Geometric variations* group box, it has to be created manually in the *Input* program. This can best be done by opening the original project, save it under a different name, make the small geometric change and save the project again. For each geometric variation both a 'minimum' project and a 'maximum' project need to be created.

To include a small geometric variation in the *Sensitivity* or *Parameter variation* window, click on the *Add* button in *Geometric variations* group box and select the new line in the list. The standard description of a geometric variation is named *Variation #*, where # is the number of the variation. This name may be changed in the *Description* box at the right-hand side of the window. Underneath the *Description*, the name of the project can be entered in which the 'minimum' of the geometric variation has been defined. Alternatively, this project may be selected using a file requester by clicking on the ... button behind the edit box. Subsequently, the 'maximum' project must be entered or selected. In addition, you must specify a *Reference* value of the geometric variation (i.e. the value used in the original project), a minimum value (*Min*; corresponding to the 'minimum' project) and a maximum value (*Max*; corresponding to the 'maximum' project). Please note that these values are not automatically generated, but they are necessary in a sensitivity analysis to properly calculate the sensitivity scores (see Section 4.10.6).

> **Hint:** Significant geometric changes that involve a different finite element mesh cannot be considered, because results of parameter variations are compared on the basis of corresponding node and stress point numbers.
>
> > Combinations of geometric variations cannot be generated automatically and need to be taken into account by manually creating additional geometric variations.

4.10.5 STARTING THE ANALYSIS

After all parameter variations have been defined and the desired parameters have been selected in the *Sensitivity* or *Parameter variation* window, the analysis can be started by pressing the *Run* button. The *Analysis info* group box shows the total number of calculations that is required, which is $2(n+m)+1$ for a sensitivity analysis or $2m(2^n)+1$ (or 2^n+1 for $m=0$) for a parameter variation analysis, where n is the number of model parameters to be varied and m is the number of geometric variations. The calculation program creates copies of the original project (SA#) inside the <*Project*>.DTS folder for a *Sensitivity* analysis or the <*Project*>.DTP folder for *Parameter variation*. SA1 is always a copy of the original project with its original model parameters, whereas SA# (with #>2) are copies in which the material data in the SA#.MAT file is changed according to the defined material variations. A list of all copies with their corresponding parameter values can be viewed by selecting the *Overview* option behind *Sensitivity and Parameter variaion* in the *Calculate* menu.

All projects are passed on to the PLAXIS *Calculation manager* (see Section 4.14.3). The *Calculation manager* controls the execution of the calculations and shows the status. When all calculations have finished, the calculation manager window can be closed and the results can be evaluated.

4.10.6 SENSITIVITY – VIEW RESULTS

The result of a *Sensitivity analysis* is an overview of the relative influence (sensitivity) of the parameter variations. This sensitivity is evaluated on the basis of user-defined criteria. Criteria can be based on nodal displacements, stress or strain components. The points used for these criteria can be selected from the set of points as defined for load-displacement curves or stress-strain curves. These points have to be predefined for the original project, before the sensitivity analysis is started. The results of the SA1 project are used as reference values for the calculation of the parameter sensitivity.

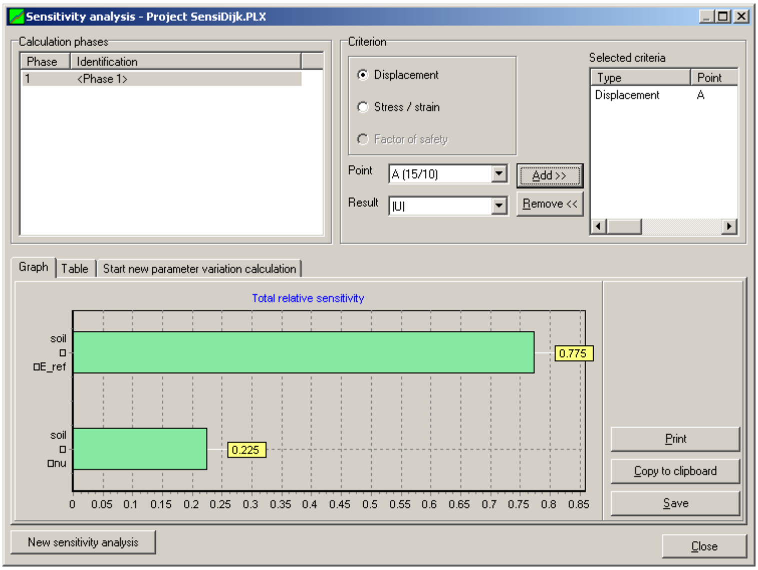


Figure 4.18 Sensitivity analysis results window

To view the result of a sensitivity analysis, select *View results* behind *Sensitivity* in the *Calculate* sub-menu. When doing so, a new window appears. The upper part of the window is used to define the criteria on the basis of which the sensitivity is evaluated.

To add a criterion, the following steps need to be performed:

1. Select the *Calculation phase* from which the results are considered.
2. Select *Displacement* or *Stress/Strain* from the *Criterion* group box
3. Select the desired point (node or stress point) from the *Point* combo box
4. Select the desired displacement, stress or strain component from the *Result* combo box
5. Click *Add* to add the criterion to the list of selected criteria
6. If necessary, select and add more criteria. Each of the defined criteria has the same 'weight'. To give a criterion a 'double weight', it should be added twice. A falsely added criterion can be removed by selecting it from the list of selected criteria and pressing the Remove button.

The lower part of the window is used to show the sensitivity of all varied parameters on the basis of the selected criteria, both in graphical and in tabulated form. Moreover, another tab sheet is available to directly select the parameters that will be taken into account in a parameter variation analysis. On the *Graph* tab sheet the sensitivity of a parameter is indicated by the size of the corresponding green bar. You can select the parameters that will be taken into account in a subsequent parameter variation analysis by clicking on the corresponding bar. By doing so, the bar colour changes from green to red. Clicking once again will deselect the corresponding parameter. The selected parameters are also shown in the *Parameter variation* tab sheet. On the *Table* tab sheet an overview is given of the different copies of the original project, their parameter values, the absolute results of the different criteria and the parameter sensitivity score (see *Theory of sensitivity analysis* in the Scientific Manual). On the basis of the sensitivity scores, the total sensitivity of a particular parameter is defined as the sum of the two corresponding parameter sensitivity scores divided by the total sum of sensitivity scores.

If it is desired to modify the upper and lower values of model parameters and to perform a new sensitivity analysis, the *New sensitivity analysis* button at the bottom of the window should be pressed. By doing so, the sensitivity window is opened, where new upper and lower values can be defined.

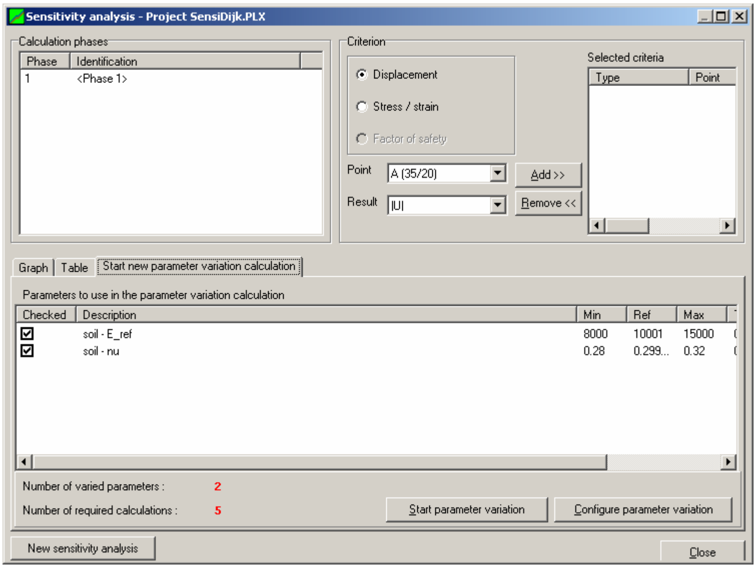


Figure 4.19 The *Sensitivity analysis* window showing the *Start new parameter variation calculation* tab sheet

From the *Sensitivity results* window you can directly start a *Parameter variation* analysis using the *Parameter variation* tab sheet. On this tab sheet you can select the parameters that should be taken into account in the *Parameter variation* analysis, and start the analysis by pressing the *Run* button. When parameters have been selected on the *Graph* tab sheet ('red' bar), they will automatically have a check mark in the *Parameter variation* tab sheet.

4.10.7 PARAMETER VARIATION – UPPER & LOWER VALUES

Before the results of a *Parameter variation* can be viewed, upper and lower values of load multipliers, displacements and structural forces from all parameter variation results need to be collected and stored in separate project files. This is not automatically done after parameter variation, but you can perform this action by selecting the *Upper & lower values* option behind *Parameter variation* in the *Calculate* sub-menu. As a result, a small window appears in which the original project is shown. To start the collection of upper and lower values, press the *Start* button. The program will now create another folder named <*Project*>.DTM, in which copies of the original project are created, named <*Project*>_Min and <*Project*>_Max. In the corresponding project data directories (<*Project*>_Min.DTA and <*Project*>_Max.DTA), the output files corresponding to the last step of each calculation phase are modified such that they will contain the minimum (maximum) values of load multipliers, the minimum (maximum) nodal displacements and the minimum (maximum) structural forces per node of structural elements from all parameter variation results. After this action has been finished, the upper and lower values can be viewed and processed in a similar way as the results of a 'normal' calculation using the *Output* program.

4.10.8 VIEWING UPPER AND LOWER VALUES

To view the upper (maximum) or lower (minimum) values of load multipliers, displacements, and structural forces, resulting from the parameter variation analysis, select the *Open* option from the *File* menu. In the file requester, select the <*Project*>.DTM directory and subsequently the <*Project*>_Max.plx or <*Project*>_Min.plx project. Select the desired calculation phase and press the *Output* button. Alternatively, the *Output* program can be started and the desired calculation phase of the *Minimum* or *Maximum* project can be opened. Results can be viewed as for any PLAXIS project. *Minimum* and *Maximum* values may be compared by opening the *Minimum* and *Maximum* project simultaneously.

> **Hint:** Results collected in the *Minimum* or *Maximum* project may come from different parameter variations and may therefore show discontinuities or present a situation that is not in equilibrium.

4.10.9 VIEWING RESULTS OF VARIATIONS

To view the results of individual parameter variations or combinations of parameter variations, the corresponding copy of the original project (SA#) can be opened in the *Calculation* window or the *Output* window using the same procedures as for a normal PLAXIS project. The results of parameter variations are stored in the <*Project*>.DTS or

<*Project*>.DTP folder. In order to see which of the copies contain which parameter variations, the *Overview* option behind *Sensitivity* or *Parameter variation* in the *Calculate* menu may be used.

4.10.10 SENSITIVITY – DELETE RESULTS

Results from a *Sensitivity analysis* can be removed by selecting the *Delete results* option behind the corresponding item in the *Calculate* sub-menu. By doing so, the <*Project*>.DTS folder, including all data in this folder, will be deleted.

4.10.11 PARAMETER VARIATION – DELETE RESULTS

Results from a *Parameter variation* analysis can be removed by selecting the *Delete results* option behind the corresponding item in the *Calculate* sub-menu. By doing so, the <*Project*>.DTP folder and the <*Project*>.DTM folder, including all data in these folders, will be deleted.

4.11 UPDATED MESH ANALYSIS

In conventional finite element analysis, the influence of the geometry change of the mesh on the equilibrium conditions is neglected. This is usually a good approximation when the deformations are relatively small as is the case for most engineering structures. However, there are circumstances under which it is necessary to take this influence into account. Typical applications where 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. Firstly 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.

Secondly, it is necessary to include a procedure to model correctly the stress changes that occur when finite material rotations occur. This particular feature of large displacement theory is usually dealt with 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 wholly 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.

Thirdly, it is necessary to update the finite element mesh as the calculation proceeds. This is done automatically within PLAXIS when the *Updated mesh* option is selected.

It should be clear from the descriptions given above 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 2). Implementation of this formulation within PLAXIS is based on the use of various advanced techniques that are beyond the scope of this manual (Reference 16).

Calculation procedures

In order to carry out an updated mesh analysis the *Advanced* button should be clicked in the *Calculation type* box in the *General* tab sheet. As a result, an *Advanced general settings* window appears in which the *Updated mesh* option can be selected. Updated mesh calculations are carried out using iteration procedures similar to the conventional plasticity options within PLAXIS, as described in preceding sections. Therefore an updated mesh analysis uses the same parameters. However, because of the large deformation effect, the stiffness matrix is always updated at the beginning of a load step. Due to this procedure and to the additional terms and more complex 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 carried out before an updated mesh analysis is attempted.

It is not possible to give simple guidelines that may be used 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 large (on a real scale!) then significant importance of geometric effects might be suspected. In this case the calculation should be repeated 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 the user is in any doubt about whether updated mesh analysis is necessary then the issue can only be resolved by carrying out 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. Resetting displacements to zero is not possible after an updated mesh analysis. Hence, gravity loading should be applied in a normal plastic calculation.

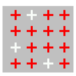
Changing from a 'normal' plastic calculation or consolidation analysis to an updated mesh analysis is only valid when displacements are reset to zero, because a series of updated mesh analyses must start from an undeformed geometry. Changing from an updated mesh calculation to a 'normal' plastic calculation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

4.12 PREVIEWING A CONSTRUCTION STAGE

When a construction staged is fully defined, a preview of the situation is presented on the *Preview* tab sheet of the Calculations window. This option is only available if the calculation phase has been defined in the Staged Construction mode. It enables a direct visual check of construction stages before the calculation process is started.

4.13 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 tool bar. As a result, the Output program is opened showing the finite element mesh with all nodes. Up to 10 nodes may be selected for the generation of load-displacement curves. Selection takes place by 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 in the finite element mesh. 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 after calculations have been performed, 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 clicked 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.

4.14 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 list of calculation phases. In principle, all calculation phases indicated with a blue arrow (→) will be executed in the calculation process. By default, when defining a calculation phase, it is automatically selected for execution. A previously executed calculation phase is indicated by 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 clicked on the corresponding line and the option *Mark calculate* or *Unmark calculate* should be selected from the cursor menu.

4.14.1 STARTING THE CALCULATION PROCESS

The calculation process can be started by clicking the *Calculate* button in the tool bar. 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.

4.14.2 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.

4.14.3 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 4.20.

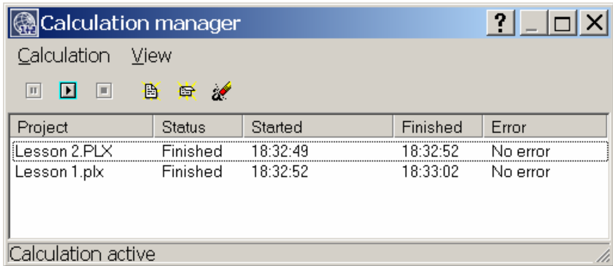


Figure 4.20 Calculation manager window

4.14.4 ABORTING A CALCULATION

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

4.15 OUTPUT DURING CALCULATIONS

During a 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 running calculation phase. The significance of load multipliers and some other parameters is described in Section 4.8. 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. If no nodes have been preselected the displacement of the lowest-leftmost node in the mesh will be plotted. In the case that prescribed displacements are activated, the major force parameter (*Force-X* or *Force-Y*) 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.

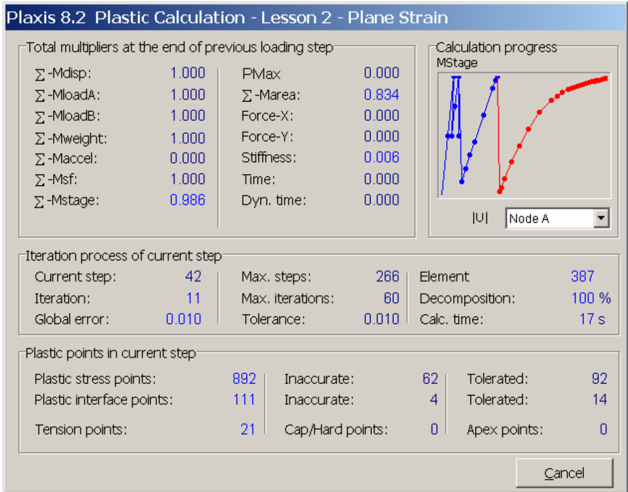


Figure 4.21 Calculation window.

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 4.19.

Stiffness

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

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

When the solution is fully elastic, the *Stiffness* is equal to unity, whereas at failure the stiffness approaches zero. The *Stiffness* is used in determining the *Global error*. See Section 4.19 for more details.

Tolerance:

The *Tolerance* is the maximum global equilibrium error that is allowed. The value of the tolerance corresponds to the value of the *Tolerated error* in the settings for the iterative procedure. The iteration process will at least continue as long as the *Global error* is larger than the *Tolerance*. For details see Section 4.19.

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 4.19.

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 4.19.

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 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.

Cancel button

If, for some reason, the user decides to abort a calculation, this can be done by clicking the *Cancel* button in the calculations window. By clicking this button, the calculation process is aborted and the control is returned to the calculations part of the user interface. Note that after clicking 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.

4.16 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 tool bar 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.

4.17 RESET STAGED CONSTRUCTION SETTINGS

If a calculation phase has been defined using the *Staged construction* option and subsequently changes are made to previous phases these changes are not carried out into the subsequent phases as is the case when a staged construction phase is defined for the first time. It is of course possible to also make these changes manually in all subsequent staged construction phases but sometimes it is more practical to clear the *Staged construction* settings and start over. It is possible to clear the staged construction settings and set them to those of the preceeding staged construction phase by selecting the corresponding line in the list of calculation phases and click the right mouse button. Select *Reset staged construction* from the popup menu. Similarly the water conditions can be reset to those of the preceeding staged construction phase by selecting *Reset water conditions* from the popup menu.

4.18 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), all calculation information (including construction stages) is retained on a geometry level but not on a mesh level. In this case it is necessary to regenerate the initial conditions and redefine all calculation phases including the generation of pore pressures, if applicable, in all calculation phases before restarting 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 (Section 4.7.5). 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 (Section 4.7.3 and 4.7.9).

4.19 AUTOMATIC ERROR CHECKS

During each calculation step, PLAXIS 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:

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

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

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

which is a measure for the amount of plasticity that occurs during the calculation. $\Delta\varepsilon$, $\Delta\sigma$ and D^e are defined in the Material Models Manual, section 2.3. When the solution is fully elastic, the *Stiffness* is equal to unity, whereas at failure the *Stiffness* approaches zero.

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 4.22.

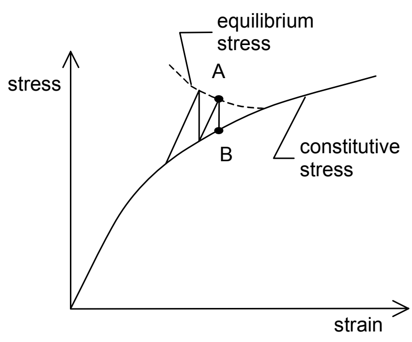


Figure 4.22 Equilibrium and constitutive stresses

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 4.22). 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 4.22.

The dashed line in Figure 4.22 indicates the path of the equilibrium stress. In general this equilibrium stress path depends on the nature of the stress field and the applied loading. For the case of a soil element obeying the Mohr-Coulomb criterion, the local error for the particular stress point at the end of the iteration is defined:

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

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

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

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

$$T_{\max} = \max\left(\tfrac{1}{2}\left(\sigma_3 - \sigma_1\right), c\cos\varphi\right)$$

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

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

where σ_n and τ 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 4.6.1).

Termination of iterations

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

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

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

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

5 OUTPUT DATA POST PROCESSING

The main output quantities of a finite element calculation are the displacements 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 PLAXIS 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 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. The main window of the Output program contains the following items (Figure 5.1)

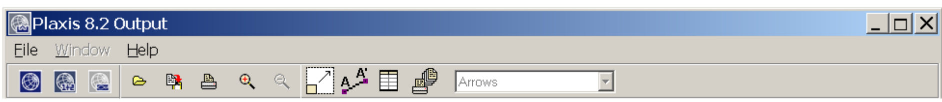


Figure 5.1 Toolbar in main window of the Output program

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 tool bar.

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.

Tool bar:

The tool bar 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 contains information about the type of analysis (plane strain or axisymmetry) and the position of the cursor (mouse) in the model.

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 full geometry model, the total menu consists of the sub-menus *File*, *Edit*, *View*, *Geometry*, *Deformations*, *Stresses*, *Window* and *Help*. The menu depends on the type of data that is presented on the output form.

The File sub-menu:

Open	To open a project for which output is to be viewed. The file requester is presented.
Close	To close the active output form.
Close all	To close all output forms.
Print	To print the active output on a selected printer. The print window is presented.
Work directory	To set the default directory where PLAXIS project files will be stored.
Report generation	To generate a project report with input data and computational results.
(*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. After selection, the scan line must be indicated by the mouse. Click 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. Click 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. Click 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).
Rulers	To show or hide the rulers along the active plot.
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.
General info	To view the general project information (Section 5.9.1).
Load info	To view the active loads in the presented step (Section 5.9.2).
Material info	To view the material data (Section 5.9.3).
Calculation info	To view the calculation information of the presented step (Section 5.9.4).
Create animation	To create an animation (movie) of a series of calculation steps.

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.
Nodes	To display the nodes in the model.
Stress points	To display the stress points in the model.
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 tool bar or by selecting the *Open* option from the *File* sub-menu. As a result, a file requester is opened from which the desired PLAXIS project file (*.PLX) can be selected (Figure 5.2).

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

If it is desired to select an intermediate calculation step, then a single mouse click should be given on the *Phase* column above the list with calculation phases in the file requester. As a result, the calculation list changes into a list with all 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 4.16.

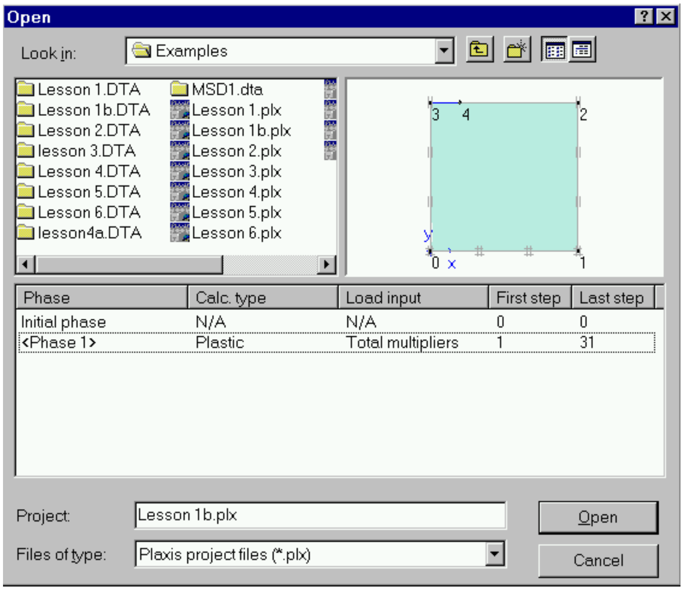
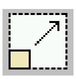


Figure 5.2 File requester for the selection of an output step

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 tool bar 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 tool bar, 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, HORIZONTAL AND VERTICAL DISPLACEMENTS

The *Total displacements* are the absolute accumulated displacements $|u|$, combined from the horizontal (x) and vertical (y) displacement components at all nodes at the end of the current calculation step, displayed on a plot of the geometry. Similarly, the *Horizontal displacements* and *Vertical displacements* are, respectively, the accumulated horizontal (x) and vertical (y) displacement components at all nodes at the end of the current calculation step. These options may be selected from the *Deformations* sub-menu. The displacements may be presented as *Arrows* or as *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the tool bar.

5.4.3 PHASE DISPLACEMENTS

The *Total phase displacements* are the absolute accumulated displacements $|u|$, combined from the horizontal (x) and vertical (y) displacement components at all nodes for the current calculation phase, displayed on a plot of the geometry. Similarly, the *Horizontal phase displacements* and *Vertical phase displacements* are, respectively, the accumulated horizontal (x) and vertical (y) displacement components at all nodes for the current calculation phase. These options may be selected from the *Deformations* sub-menu. The phase displacements may be presented as *Arrows* or as *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the tool bar.

5.4.4 INCREMENTAL DISPLACEMENTS

The *Total increments* are the absolute displacement increments of the current step $|\Delta u|$, combined from the horizontal and vertical displacement increments at all nodes as calculated for the current step, displayed on a plot of the geometry. Similarly, the *Horizontal increments* and *Vertical increments* are, respectively, the horizontal (x) and vertical (y) displacement increments at all nodes as calculated for the current step. These options may be selected from the *Deformations* sub-menu. The displacement increments may be presented as *Arrows* or as *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the tool bar. The contours of total increments are particularly useful for the observation of localisation of deformations within the soil when plastic failure occurs.

5.4.5 TOTAL STRAINS

The *Total strains* are the accumulated strains in the geometry at stress points 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.

Total strains can be represented as *Principal directions* (principal strains), *Volumetric strains* (ε_v) or equivalent *Shear strains* (ε_s) by selecting the appropriate option from the presentation box in the tool bar. Volumetric strains and shear strains can be displayed as *Contours* or *Shadings*.

Principal strains are plotted as crosses in the element stress points. When using 15-node elements, three stress points per element are displayed. When using 6-node elements only one point per element is displayed, which represents the average of the total principal strains in the element. 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.

5.4.6 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} and γ_{xy}. Cartesian strain components can be represented as *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the tool bar. Note that compression is considered to be negative.

5.4.7 INCREMENTAL STRAINS

The *Incremental strains* are the strain increments in the geometry at stress points as calculated for the current calculation step, displayed in a plot of the geometry. This option may be selected from the *Deformations* sub-menu.

Incremental strains can be represented as *Principal directions* (principal strain increments), *Volumetric strains* (incremental volumetric strains $\Delta\varepsilon_v$) or equivalent *Shear strains* (incremental equivalent shear strains $\Delta\varepsilon_s$) by selecting the appropriate option from the presentation box in the tool bar. Volumetric strains and shear strains can be displayed as *Contours* or *Shadings*.

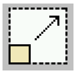
Principal strain increments are plotted as crosses in the element stress points. When using 15-node elements, three stress points per element are displayed. When using 6-node elements only one point per element is displayed, which represents the average of the principal strain increments in the element. The length of each line represents the magnitude of the principal strain and the direction indicates the principal direction. Strain increments that represent extension are indicated by an arrow rather than a line. Note that compression is considered to be negative.

5.4.8 CARTESIAN STRAIN INCREMENTS

When selecting *Cartesian strain increments* from the *Deformations* sub-menu, a further selection can be made between the individual strain increment components $\Delta\varepsilon_{xx}$, $\Delta\varepsilon_{yy}$ and $\Delta\gamma_{xy}$. Cartesian strain increment components can be represented as *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the tool bar. Note that compression is considered to be negative.

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)·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 tool bar 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 tool bar, 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.

Effective stresses can be represented as *Principal directions* (principal stresses), *Mean* stresses (*p*') or *Relative shear* stresses (τ_{rel}) by selecting the appropriate option from the presentation box in the tool bar. Mean stresses and relative shear stresses can be displayed as *Contours* or *Shadings*.

Principal effective stresses are plotted as crosses in the element stress points. When using 15-node elements, three stress points per element are displayed. When using 6-node elements only one point per element is displayed, which represents the average of the principal effective stress in the element. 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 pressure is considered to be negative.

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.

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.

Total stresses can be represented as *Principal directions* (principal stresses), *Mean* stresses (p) or *Deviatoric* stresses (q) by selecting the appropriate option from the presentation box in the tool bar. Mean stresses and deviatoric stresses can be displayed as *Contours* or *Shadings*.

Principal total stresses are plotted as crosses in the element stress points. When using 15-node elements, three stress points per element are displayed. When using 6-node elements only one point per element is displayed, which represents the average of the total principal stress in the element. 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 pressure is considered to be negative.

5.5.3 CARTESIAN EFFECTIVE STRESSES

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

5.5.4 CARTESIAN TOTAL STRESSES

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

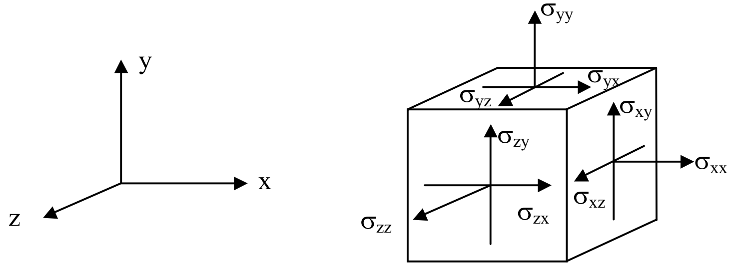


Figure 5.3 Sign convention for stresses

5.5.5 OVERCONSOLIDATION RATIO

The *Overconsolidation ratio* is only displayed if the Hardening Soil model, HS small 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

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

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

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

5.5.6 PLASTIC POINTS

The *Plastic points* are the stress points in a plastic state, displayed in a plot of the undeformed geometry. The plastic stress points are indicated by small symbols that can have different shapes and colours, depending on the type of plasticity that has occurred. A red open square (Mohr-Coulomb point) indicates that the stresses lie on the surface of the Coulomb failure envelope. A white solid square (tension cut-off point) indicates that the tension cut-off criterion was applied. A blue crossed square (cap point) represents a state of normal consolidation where the preconsolidation stress is equivalent to the current stress state. The latter type of plastic points only occurs if the Soft Soil (Creep) model or the Hardening Soil model is used. Additionally, for the Hardening Soil model and HS small model a green square with a plus sign (hardening point) represents points on the shear hardening envelop (mobilised friction envelop) while a green crossed square is simultaneously a hardening point and a cap point.

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.

5.5.7 ACTIVE PORE PRESSURES

The *Active pore pressures* are the total water pressures p_w (i.e. steady-state pore pressures + excess pore pressures) in the geometry at the end of the current calculation step, displayed in a plot of the undeformed geometry. This plot may be selected from the *Stresses* sub-menu. By default, active pore pressures are presented just like principal stresses, although they are isotropic and do not have any principal direction. The length of lines represents the magnitude of the active pore pressure and the directions coincide with the *x*- 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.

When using 15-node elements, three stress points per element are displayed. When using 6-node elements only one point per element is displayed, which represents the average of the active pore pressure in the element.

As an alternative for the *Principal directions*, the user may select *Contours* or *Shadings* of active pore pressures from the presentation combo box.

5.5.8 EXCESS PORE PRESSURES

The *Excess pore pressures* are the water pressures due to loading of undrained clusters or by using the consolidation analysis at the end of the current calculation step, displayed in a plot of the undeformed geometry. This plot may be selected from the *Stresses* sub-menu. By default, excess pore pressures are presented just like principal stresses, although they do not have any principal direction. The length of lines represents the magnitude of the excess pore pressure and the directions coincide with the *x*- 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.

When using 15-node elements, three stress points per element are displayed. When using 6-node elements only one point per element is displayed, which represents the average of the excess pore pressure in the element.

As an alternative for the *Principal directions*, the user may select *Contours* or *Shadings* of excess pore pressures from the presentation combo box.

5.5.9 GROUNDWATER HEAD

The groundwater head is an alternative quantity for the active pore pressure. The *Groundwater head* is defined as:

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

where 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 available from the *Stresses* sub-menu. This option is most 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 occur. The distribution of the groundwater head can only be presented as *Contours* or *Shadings* by selecting the appropriate option from the presentation combo box.

5.5.10 FLOW FIELD

When a groundwater flow calculation has been performed to generate the pore pressure distribution, then the specific discharges at the element stress points are available in the Output program in addition to the pore pressure distribution. The specific discharges can be viewed by selecting the *Flow field* option from the *Stresses* sub-menu. The flow field may be viewed as *Arrows* or as *Contours* or *Shadings* by selecting the appropriate option from the presentation box in the tool bar.

When the specific discharges are presented as arrows, then the length of the arrow indicates the magnitude of the specific discharge whereas the arrow direction indicates the flow direction.

5.5.11 DEGREE OF SATURATION

The groundwater flow module within PLAXIS may be used to calculate a pore pressure distribution for confined as well as for unconfined flow problems. The determination of the position of the free phreatic surface and the associated length of the seepage surface is one of the main objectives of an unconfined groundwater flow calculation. In this case a relationship is used between the pore pressure and the degree of saturation. Both quantities are calculated in a groundwater flow calculation and are made available in the Output program.

If it is desired to view the *Degree of saturation*, the corresponding option may be selected from the *Stresses* sub-menu. In fact, the degree of saturation is only relevant if a groundwater flow calculation has been performed. The degree of saturation is generally 100% below the phreatic level and it reduces to zero within a finite zone above the phreatic level.

5.6 STRUCTURES AND INTERFACES

Structures (i.e. plates, geogrids, anchors) and interfaces are, by default, displayed in the geometry. Optionally, these objects may be hidden or shown again using the *Structures* option from the *Geometry* sub-menu. Output for these types of elements can be obtained by double-clicking the desired object in the geometry. 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

The output data for a plate comprises deformations and forces. From the *Deformations* sub-menu the user may select the accumulated absolute displacements, |u|, at the end of the calculation step, or the individual accumulated displacement components u_x and u_y. From the *Forces* sub-menu the options *Axial forces, Shear forces* and *Bending moments* are available. For axisymmetric models the *Force* sub-menu also includes the forces in the out-of-plane direction (*Hoop forces*). These forces represent the actual forces at the end of the calculation step.

In addition to the actual forces, PLAXIS keeps track of the historical maximum and minimum forces in all subsequent calculation phases. These maximum and minimum values up to the current calculation step may be viewed after selecting the *Force envelopes* option in the *Forces* sub-menu and subsequently selecting the desired force option (*Axial forces*, *Shear forces*, *Bending moments* or *Hoop forces*).

Note that axial forces or hoop forces are positive when they generates tensile stresses, as indicated in Figure 5.4.

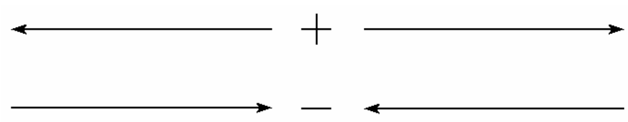


Figure 5.4 Sign convention for axial forces and hoop forces in plates, geogrids and anchors

If a circular tunnel (bored tunnel) is modelled and a contraction is applied to the tunnel lining, then the *Total realised contraction* and the *Realised contraction increment* are displayed in the plot title.

5.6.2 GEOGRIDS

The output data for geogrids can be obtained by double-clicking on the corresponding yellow line in the geometry. The output for a geogrid comprises deformations and forces. From the *Deformations* sub-menu the user may select the accumulated absolute displacements, |u|, or the individual displacement components, u_x and u_y. From the *Forces* sub-menu the option *Axial force* is available. 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 the geometry. The output for an interface comprises deformations and stresses.

From the *Deformations* sub-menu the user may select *Total, Horizontal and Vertical displacements*, *Total, Horizontal and Vertical increments*, *Relative displacements* and *Relative increments*. Relative displacements are differential displacements between the node pairs. These options may be used to view if plastic shearing has occurred in the interface.

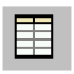
From the *Stresses* sub-menu the options *Effective normal stresses*, *Shear stresses*, *Relative shear stresses*, *Active pore pressures* and *Excess pore pressures* are available. The *Effective normal stresses* are the effective stresses perpendicular to the interface, whereas the *Shear stresses* are the shear stresses in the interface. The relative shear stress is defined as the ratio between the shear stress and the maximum value of shear stress according to the Coulomb failure criterion, while keeping the effective normal stress constant.

Note that pressure is considered to be negative for the normal stresses and pore pressures.

5.6.4 ANCHORS

When double-clicking an anchor (either a node-to-node anchor or a fixed-end anchor), a small window is presented in which the anchor force is displayed. Also the maximum forces and the anchor stiffness are displayed in this window. If the absolute value of the anchor force is equal to a maximum force, then the anchor is in a plastic state. Tensile forces are defined 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 tool bar 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 and u_y are the accumulated displacements from all previous calculation phases, whereas the incremental displacements Δu_x and Δu_y 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 and HS small 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.

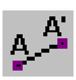
Tables of nodes and stress points

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. Note that for advanced models with different stiffness moduli the actual elastic stiffness modulus is equal to the unloading stiffness modulus.

Stresses and forces in interfaces and structures

When viewing tables of interface stresses, the table presents the effective normal stresses (σ'_n), the shear stress (τ), the active pore pressure (p_{active}) and the excess pore pressure (p_{excess}) at all interface stress points. When viewing tables of plate forces, the table presents the axial force (*N*), the shear force (*Q*) and the bending moment (*M*) at the nodes. For geogrids, the table only presents the force in the axial direction of the geogrid (*N*). 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. This option is available in PLAXIS for all types of stresses and displacements in soil elements. It can be selected by clicking on the *Cross-section* button in the tool bar or by selecting the corresponding option from the *View* menu. After selection of this option the user has to specify the cross section on one end of the cross-section line in the geometry 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 currently displayed 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 and strains). 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 Cartesian 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. In addition to the Cartesian effective and total stress components, active and excess pore pressures, as available for the full geometry, the cross-section option allows for *Normal stresses* and *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 line. Note that pressure is considered to be negative.

Integration of stresses: Equivalent force

When normal stress components are plotted in a cross-section, PLAXIS automatically calculates and displays an equivalent force that represents the integral of the normal stress over the cross-section. The value and position of the equivalent force are displayed in the plot title.

5.9 VIEWING OTHER DATA

The *View* 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 GENERAL PROJECT INFORMATION

The *General info* option of the *View* sub-menu contains some general information about the project (file name, directory, title), the model (plane strain or axisymmetry) and the generated finite element mesh (basic element type, number of elements, nodes, stress points, average element size l_e).

5.9.2 LOAD INFORMATION

The *Load info* option of the *View* sub-menu shows the realised values of the active loads and prescribed displacements in the current calculation step.

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 4.8. 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 4.6.1). 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 4.7 and 4.8.2).

The *Forces* box gives the values of the parameters *Force-X* and *Force-Y* (the force component in the x- and y -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 4.6.1.

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.

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 *Total realised contraction* and the *Realised contraction increment* are displayed in the plot title. The *Total realised contraction* is defined as:

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

Note that the *Total realised contraction* 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 *Total realised contraction* will only be slightly smaller. For linings that are relatively flexible, however, the difference may become more significant. If the *Total realised contraction* 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, PLAXIS has several facilities to view the finite element model. An overview of these facilities is given below:

Zooming

To enlarge a part of the model for viewing a particular detail, the zoom option of the *View* sub-menu may be used. After selection of the zoom option, the zoom area (a rectangular area on the screen) must be selected with the mouse. The zoom option may be used repetitively.

Viewing structural objects

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

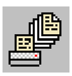
Viewing cross-section

Users may define cross-sections to view output. This can be done by selecting the *Cross-section* option of the *View* sub-menu.

Changing the intensity of material data set colours

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 this way.

5.10 REPORT GENERATION



To document project input data and computational results, a *Report generation* facility is available in the PLAXIS Output program. This option requires the presence of the Microsoft® Word software. The *Report generation* option can be selected from the *File* sub-menu or by clicking on the corresponding button in the tool bar. By doing so, the report generation window appears in which a selection can be made of the project data that is to be included in the report.

By default, the various groups of items that can be selected are presented in a tree view. Groups of items and individual items that are selected to be included in the report are indicated by a filled black square, whereas unselected (groups of) items are indicated by an open square. By clicking on a square, (groups of) items can be toggled from being selected to being unselected and vice versa. By clicking on the +sign in front of a group, the items in the group can be selected individually.

The *Select* group box allows for a further selection of *Phases*, *Curves*, *Structures* and *Cross sections*. The *Phases* option enables a further selection of all available and performed calculation phases. The *Curves* option enables a selection of existing (saved) curves. The *Structures* option enables a selection of plates, geogrids or interfaces that are currently displayed in separate output windows. The *Cross sections* option enables a selection of existing cross sections. Hence, to include output data of structures or cross-sections in the report it is necessary to first display the desired structures or cross-sections in separate output windows.

After the selection of all desired items, the *Contents view* button may be clicked to view the contents of the report to be created. The *Tree view* button may be used to restore the tree view of selected and unselected items, providing the possibility to change the current selections.

If the selections are satisfactory, then the *Start* button should be clicked to generate the report. As a result, the Microsoft® Word program is started and the report is created in a new document. From there it may be printed or included in other documentation. For details on the use of the Microsoft® Word program, reference is made to the corresponding manual.

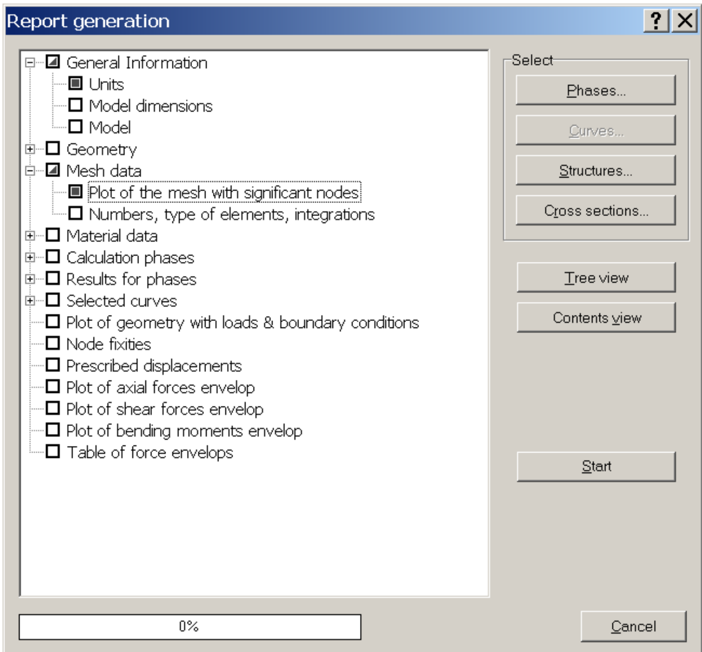


Figure 5.5 Report generation window

5.11 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 tool bar 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 clicking 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 4.13). 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 6.2). 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 project. After the selection of one of the available charts within this project, the corresponding chart is presented in the main window. The main window of the Curves program contains the following items (Figure 6.1):

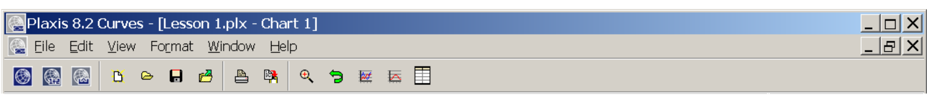


Figure 6.1 Tool bar in main window of the Curves program

Curves menu:

The Curves menu contains all options and operation facilities of the Curves program. Some options are also available as buttons in the tool bar.

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 ten curves.

Tool bar:

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 6.6.2).

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. Click 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.
Value indication	To view detailed curve data when the cursor (mouse pointer) is positioned on a curve.

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.

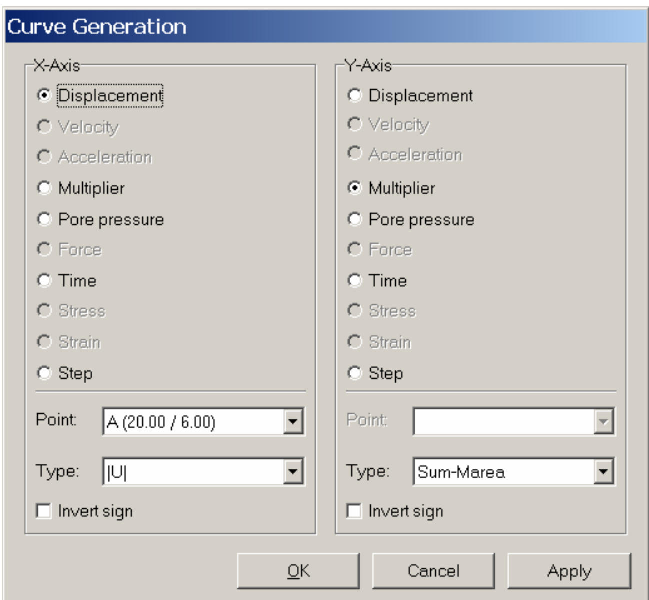


Figure 6.2 Curve generation window

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 6.6.2). For each axis, a combination of selections should be made to define which quantity is plotted on that axis. The *Invert* *sign* option may be selected to multiply all values of the *x*-quantity or the *y*-quantity by -1. 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 clicked, the curve is generated and presented in a chart window.

Load-displacement curves

Load-displacement curves can be used to visualise the relationship between the applied loading and the resulting displacement of a certain point in the geometry. In general, the *x*-axis relates to the displacement of a particular node (*Displacement*), and the *y*-axis contains data relating to load level (*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 or u_y). 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 4.8. 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*- and *y*-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* or *Force-Y*) in the *Type* combo box. In plain strain models the *Force* is expressed in the units of force per unit of width in the out-of-plane direction. In axisymmetric models the *Force* is expressed in the unit of force per radian. Hence, to calculate the total reaction force under a circular footing that is simulated by means of prescribed displacements, the *Force-Y* value should be multiplied by 2π.

Time-displacement curves

Time-displacement curves can be useful to interpret the results of calculations in which the time-dependent behaviour of the soil plays an important role (e.g. consolidation and creep). In this case the *Time* option is generally selected for the x-axis, and the y-axis contains data for the displacements of a particular node. The selection of *Time* does not require additional selections in the *Point* and *Type* combo boxes. *Time* is expressed in the unit of time, as specified in the *General settings* window of the input program.

Stress-strain diagrams

Stress-strain diagrams can be used to visualise the local stress-strain behaviour of the soil. In fact, stress-strain diagrams represent the idealised behaviour of the soil according to the selected soil model. The selections *Stress* or *Strain* must be completed with the selection of a pre-selected stress point from the *Point* combo box and the selection of a certain component in the *Type* combo box. The following stress and strain components are available:

Stresses:

σ'_{xx}	effective horizontal stress (x-direction)
σ'_{yy}	effective vertical stress (y-direction)
σ'_{zz}	effective stress in the out-of-plane direction (z-direction)
σ_{xy}	shear stress
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)
τ^*	maximum shear stress (radius of Mohr's circle)
p_{excess}	excess pore pressure

Strains:

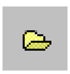
ε_{xx}	horizontal strain (x-direction)
ε_{yy}	vertical strain (y-direction)
γ_{xy}	shear strain
ε_1	in absolute sense the largest principal strain
ε_2	the intermediate principal strain
ε_3	in absolute sense the smallest principal strain
ε_v	volumetric strain
ε_q	deviatoric strain (equivalent shear strain)
γ^*	radius of Mohr's strain circle in the x-y plane

See the Material Models 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 ten 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 tool bar 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

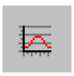
If, for any reason, a calculation process is repeated or extended with new calculation phases, it is generally desirable to update existing curves to comply with the new data. This can be done by means of the *Regenerate* facility. This facility is available in the *Curve settings* window (Section 6.6.1), which can be selected by clicking on the *Change curve settings* button in the tool bar 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. Clicking 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 tool bar 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 clicked to activate the settings and to close the window. Alternatively, the *Apply* button may be clicked to activate the settings, but the window is not closed in this case. When clicking the *Cancel* button the changes to the settings are ignored.

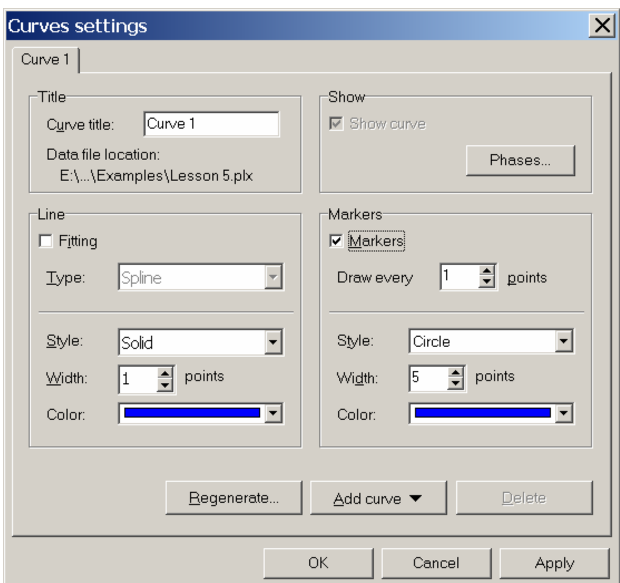


Figure 6.3 Curve settings window

Title:

A default title is given to any curve during its generation. This title may be changed in the *Curve title* edit box. When a legend is presented for the active chart in the main window, the *Curve title* appears in the legend.

Show curve:

When multiple curves are present within one chart, it may be useful to hide temporarily one or more curves to focus attention on the others. The *Show curve* option may be deselected for this purpose.

Phases:

The *Phases* button may be used to select for which calculation phases the curve has to be generated. This option is useful when not all calculation phases should be included in the curve. 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.

Line and marker presentation:

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

Fitting:

To draw a smooth curve, the user can select the *Fitting* item. When doing so, the type of fitting can be selected from the *Type* combo box. The *Spline* fitting generally gives the most satisfactory results, but, as an alternative, a curve can be fitted to a polynomial using the least squares method.

Regenerate:

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

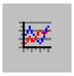
Add curve:

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

Delete:

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

6.6.2 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 tool bar may be clicked. As a result, the *Frame settings* window appears, as shown in Figure 6.4.

If the correct settings are made, the *OK* button may be clicked to activate the settings and to close the window. Alternatively, the *Apply* button may be clicked to activate the settings, but the window is not closed in this case. When clicking the *Cancel* button, the changes to the settings are ignored.

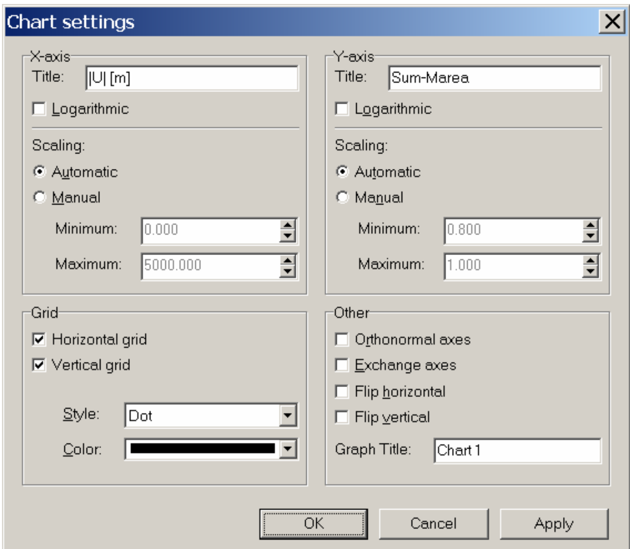


Figure 6.4 Frame settings window

Titles:

By default, a title is given to the *x*-axis and the *y*-axis, based on the quantity that is selected for the curve generation. However, this title may be changed in the *Title* edit boxes of the corresponding axis group. In addition, a title may be given to the full chart, which can be entered in the *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 greater than zero.

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 tool bar or by selecting the corresponding option in the *View* menu. As a result, a table appears showing the numerical values of all points on a curve 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. Copied data can be pasted in a spreadsheet program for further elaboration.

Editing curve data

In contrast to the Output program, the Curves program allows for editing of the table by the user. After selection of the appropriate curve from the combo box, the value of curve points can be can be modified. The values may be edited by typing a new value at the position of an existing value.

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.

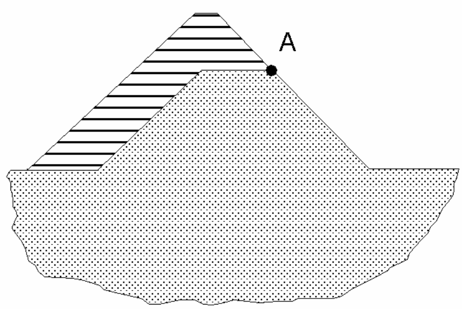


Figure 6.5 Raising an embankment

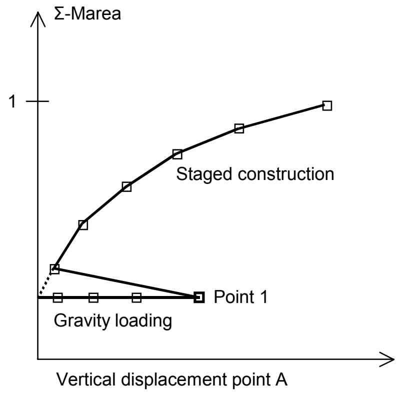
In this example project soil is to be added to an existing embankment to increase its height. The purpose of this example analysis is to calculate the displacement of point A as the embankment is raised. One approach to this problem is to generate a mesh for the final embankment and then deactivate the 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 using *Total multipliers* as *Loading input*.

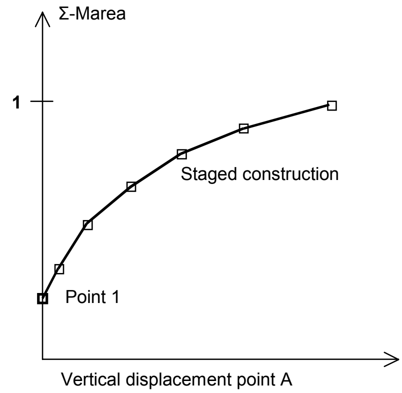
The settlement behaviour of point A when gravity loading is applied is shown by the initial horizontal line in Figure 6.6a. This line will, in general, consist of several plastic calculation steps, all with the same value of *ΣMarea*.

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.



a. Before editing



b. After editing

Figure 6.6 Load-displacement curves of the embankment project.

As an alternative to the above editing procedure, the gravity loading phase can be excluded form the list of calculation phases that are included in the curve (Section 6.6.1).

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.

INDEX

G

H

I

J

L

M

APPENDIX A - GENERATION OF INITIAL STRESSES

Many 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.*

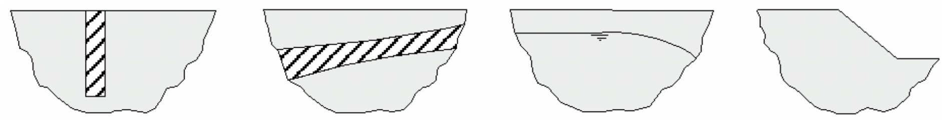


Figure A.1 Examples of non-horizontal surfaces and non-horizontal weight stratifications

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:

$$K_0 = \sigma'_{xx} / \sigma'_{yy}$$

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 stresses that violate the Coulomb failure condition. In this case PLAXIS automatically reduces the lateral stresses such that the failure condition is obeyed. Nevertheless, care should be taken because the stresses might be different form what the user expects. Anyhow, 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 that 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 calculation phase in which no additional load is applied (Section 4.7.10). After this phase 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 perfectl*y*-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 4.7.5). For one-dimensional compression an elastic computation will give:

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

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 a *Plastic* calculation in which the *Loading input* is set to *Total multipliers* and *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:

GEO.EXE	Input program (pre-processor) (see Chapter 3)
BATCH.EXE	Calculations program (see Chapter 4)
PLAXOUT.EXE	Output program (post-processor) (see Chapter 5)
CURVES.EXE	Curves program (see Chapter 6)
PLXMSHW.EXE	Mesh generator
GEOFLOW.EXE	Groundwater flow analysis program
PLASW.EXE	Deformation analysis program (plastic calculation, consolidation, updated mesh)
PLXSSCR.DLL	Module presenting the PLAXIS logo's
PLXCALC.DLL	Module presenting the screen output during a deformation analysis (Section 4.15)
PLXREQ.DLL	PLAXIS file requester (Section 2.2)

The material data sets in the global data base (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 data base structure which may be used to 'repair' a project of which, for any reason, the material data base 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 project has a structured format and is named <*project*>.PLX, where <*project*> is the project title. Besides this file, additional data is stored in multiple files in the sub-directory <*project*>.DTA. The files in this directory may include:

CALC.INF	
DBDWORK.INI	
PLAXMESH.ERR	
PLAXIS.*	(.MSI; .MSO)
ANCHORS.*	(.LDB; .MDB)
BEAMS.*	(.LDB; .MDB)
GEOTEX.*	(.LDB; .MDB)
SOILDATA.*	(.LDB; .MDB)
<*project*>.*	(.INP; .L## [1]; .MSH; .SF4; .HIS; .SIS; .CXX; .W00; .W## [1]; .N## [2]; .000; .### [3])

[1] = Two digit calculation phase number (01, 02, …). Above 99 gives an additional digit in the file extension.

[2] = Three digit groundwater flow calculation step number (001, 002, …). Above 999 gives an additional digit in the file extension.

[3] = Three digit deformation 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*>.PLX in the PLAXIS work directory.