Oasys
YOUR IDEAS BROUGHT TO LIFE

Slope

Slope

Help Guide

Oasys

YOUR IDEAS BROUGHT TO LIFE

13 Fitzroy Street
London
W1T 4BQ
Telephone: +44 (0) 20 7755 3302
Facsimile: +44 (0) 20 7755 3720

Central Square
Forth Street
Newcastle Upon Tyne
NE1 3PL
Telephone: +44 (0) 191 238 7559
Facsimile: +44 (0) 191 238 7555

e-mail: oasys@arup.com
Website: http://www.oasys-software.com

Slope

Printed: December 2022

Table of Contents

Part I

1   About Slope 21.0

Slope 21.0 is a development of Oasys Slope which includes the option of finite element steady state seepage analysis or defined pore pressure distribution, with analysis of slope stability by traditional limit equilibrium methods. Soil reinforcement can be included.

The main features are:

- The ground section is built up by drawing lines to form polygons representing soil strata. Data files from previous versions of Oasys Slope, in which the strata were represented by polylines from left to right of the problem, can also be imported and the program will generate the 2D polygons. A new feature for importing 3D TIN files from GIS systems, and defining a plane across which to cut the 3D surface, is also available.

- Pore pressures can be specified by the coordinates of one or more water tables, levels of water in piezometers, or an Ru value (pore pressure as a proportion of overburden). Pore pressures may optionally be generated instead, using groundwater boundary conditions specified on the mesh boundary lines. For this option, material permeabilities are required as well as material strength data. The 2D problem domain is used to create the finite element mesh for steady state seepage analysis .

- The strength of the materials is represented by specifying cohesion and an angle of shearing resistance. Linear variations of cohesion with depth can also be entered.

- Slopes which are submerged or partially submerged can be analysed.

- The location of circular slip surfaces is defined using a rectangular grid of centres (optionally inclined) and then either a number of different radii, a common point through which all circles must pass or a tangential surface which the circle almost touches.

- Any combination of reinforcement, consisting of horizontal geotextiles or horizontal or inclined soil nails, rock bolts or ground anchors, can be specified. The restoring moment contributed by the reinforcement is calculated according to BS8006.

- External forces can be applied to the ground surface to represent building loads or strut forces in excavations.

- Analysis first carries out a steady state seepage analysis or calculates pore pressures from the specified water data, then uses the result as input to a method of slices limit equilibrium slope stability analysis.

- Output includes plotting of pore pressures and other water data, and display of the analysed slip circles with their factors of safety against failure.

Part II

2 What's New

The program introduces a new user interface with a ribbon menu and windows which can be docked, floated on screen or minimised to the edge of the screen.
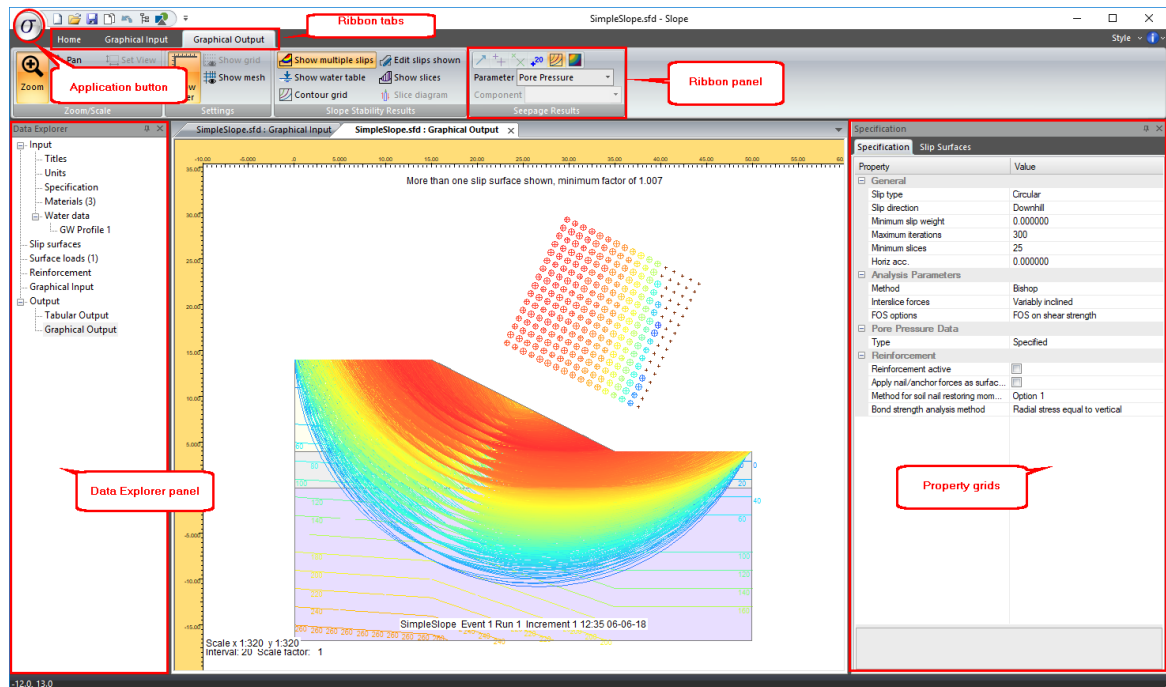
The ribbon contains all the commands available in the program, organised into tabbed groups. The most frequently used commands are on the Home tab. Opening some views automatically switches to show the tab most applicable to that view, but the tab can always be manually changed by the user. Some overall problem data (general specification such as analysis method, slip circle definition) is shown on "property grids" to the right of the main screen. Again, these views can be floated and docked in different positions if required.

The Data Explorer navigation view has links to the various data entry screens and output views. It is "docked" to the left of the main view, but can be moved to another location, closed, or set to auto-hide if required. Setting auto-hide is done using the push-pin symbol at the top of the Data Explorer. It will then minimise to the side of the main window unless you hover over the minimised view title bar. The Data Explorer can also be closed or re-opened using a check box on the Home tab of the ribbon.

The "application button" at the top left of the main window takes the place of the File menu. Clicking this button opens a menu with access to file operation commands (New, Open, Save etc.), some view commands (Export, Print etc.) and quick access to recently opened files.

This version of Slope introduces a new file extension for Slope files which is .sbd. Files with the suffix .sld, from previous versions of Slope, may also be opened.

Part III

3 Theory

3.1 Method of Slices

Slope analyses slope stability using the method of slices, in which the slipping mass is divided into a number of vertical slices and various equations of static equilibrium are applied either to each slice or to the slipping mass as a whole.

The annotation and sign convention for the method of slices is as follows:
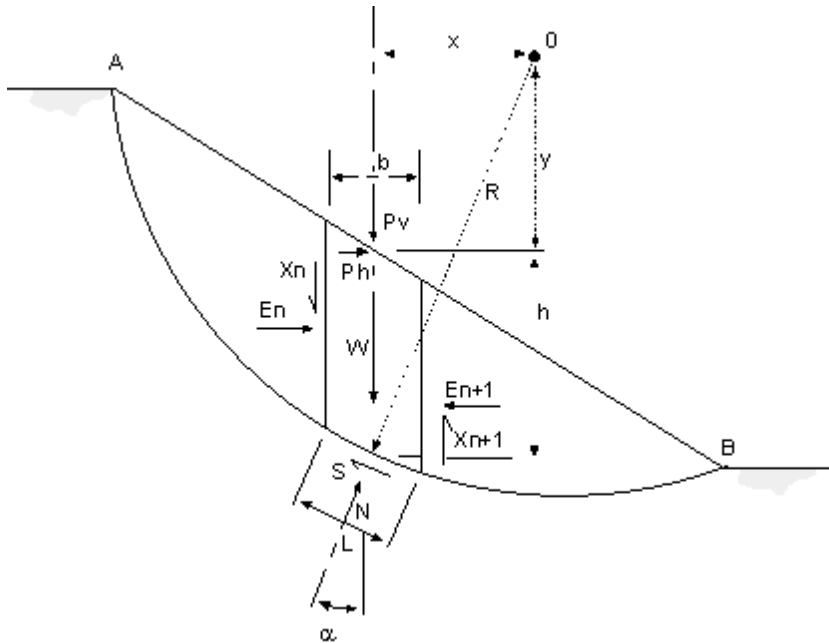
All forces are given as total forces (i.e. including water pressure).

F - Factor of Safety

Ph - Horizontal component of external loads

Pv - Vertical component of external loads

E - Horizontal Interslice Force

X - Vertical Interslice Force

W - Total weight of soil = γbh

N - Total normal force acting along slice base

R - Distance from slice base to moment centre

S - Shear force acting along slice base

h - Mean height of slice

b - Width of slice

L - Slice base length = b/cosα

u - Pore pressure at slice base

α - Slice base angle to horizontal

x - Horizontal distance of slice from moment centre

y - Vertical distance of slice surface from moment centre

γ - Unit weight of soil

c - Cohesion at base

? - Angle of friction at base

More:

[General Equations](#)

3.2   Governing Equations

The general expression to calculate the average overall factor of safety for a circular slip circle is:

$$F = \frac{\sum\limits_{A}^{B} S.R}{\sum\limits_{A}^{B}\left[(W + P_v)x + P_h y\right]} = \frac{\text{Restoring moment}}{\text{Disturbing moment}}$$

Where

S = cL + (N - uL) tan ?

and

N = (W + Pv + Xn - Xn+1) cos α - (En - En-1 + Ph) sin α

Note : As the factor of safety (F) is directly related to c and tan ? , it is a factor of safety on material shear strength.

For models which include soil reinforcement, the additional restoring moment contributed by the reinforcement is added to the soil strength restoring moment. For details of the method

of calculation, see [Reinforcement Calculations](#).

In addition other expressions for equilibrium are as follows:

For vertical equilibrium:

$$N \cos \alpha = W + P_v + (X_n - X_{n+1}) - (S \sin \alpha) / F$$

For horizontal equilibrium:

$$N \sin \alpha = (E_{n+1} - E_n) - P_h + (S \cos \alpha) / F$$

For full details of notation see [Method of Slices](#).

3.3 Bishop's Methods

Bishop's method ([Bishop AW, 1955](#)) is applicable to circular slip surfaces.

There are three methods of Bishop solution available in Slope. These differ only in the direction of the interslice forces.

Horizontal interslice forces

In this method, the interslice shear forces are assumed to sum to zero. This satisfies vertical equilibrium, but not horizontal equilibrium. This leads to errors in the calculated factors of safety, but these are usually small and on the safe side ([Spencer 1967](#)). The method satisfies overall moment equilibrium.

The limitations of the method have been investigated by [Whitman and Bailey (1967)](#). They concluded that the method can occasionally give misleading answers particularly in the case of [Interlock](#). If it is suspected that this may be a problem then the user should select the method with variably inclined interslice forces.

Parallel inclined interslice forces

This method (also known as Spencer's Method) is a refinement of the horizontal forces method, and satisfies conditions of horizontal, vertical and moment equilibrium for the slip as a whole. The program assumes that all the interslice forces are parallel and at a constant inclination throughout the slope.

This method was assessed by [Spencer (1967)](#). He has shown that in most cases the results differ only slightly from those obtained by the simplified method, which assumes only horizontal interslice forces. The differences between the two methods increase with slope angle. For steep slopes Spencer's method is more accurate, but some slip circles may have problems with interlock and so the final method with variably inclined interslice forces is the program default.

Variably inclined interslice forces

In this method the program calculates the interslice forces to maintain horizontal and vertical equilibrium of each slice. The inclinations of the interslice forces are then varied in each iteration until overall horizontal, vertical and moment equilibrium is also achieved.

3.3.1   Iteration procedure

Slope uses iteration to reach convergence for each of the Bishop methods as follows:

Factors of safety

For each iteration i, Slope calculates a new factor of safety F_i using the ratio of restoring moment to disturbing moment (which is a function of F_{i-1}). When the difference between F_i and F_{i-1} is within the specified tolerance, the calculation is complete.

The factor of safety, F, is the ratio of restoring moment to disturbing moment. However, this ratio is itself a function of F, (except in the Swedish circle method) so an iterative solution is necessary.

Horizontal interslice forces

1.   Slope starts at slice 1 (Note : Slices are numbered from left to right) and, by maintaining vertical equilibrium it calculates the resultant horizontal force.

2.   The program then uses this as the interslice force with slice 2. The process continues until the last slice which ends up with a resultant horizontal force.

In this method each slice and the slope as a whole is in vertical equilibrium, with zero vertical interslice forces. Horizontal equilibrium is not achieved within each slice or the slope as a whole. Therefore the only force check within each slice is for vertical equilibrium.

Constant inclined interslice forces

In this method Slope varies the ratio (which is constant), between the vertical and horizontal interslice forces, until the resultant of each is reduced to zero.

For this method each slice is not in equilibrium, only the slope as a whole. In the calculation equilibrium is effectively maintained for each slice in the direction normal to the interslice forces.

Variably inclined interslice forces

The variably inclined method is usually preferable, as it keeps every slice in horizontal and

vertical equilibrium at all times. However, it can exceed the soil strength along the slice interface as it does not check the vertical interslice forces against the shear strength of the material. The results should therefore be checked for this criterion.

The interslice force is adjusted separately, for both the vertical and horizontal direction, by adding the fraction of the residual values from the previous iteration. The fraction is determined by the horizontal length of the slip surface represented by that slice. The interslice force direction can vary by this method, but each slice is in equilibrium at all times as is the slope as a whole.
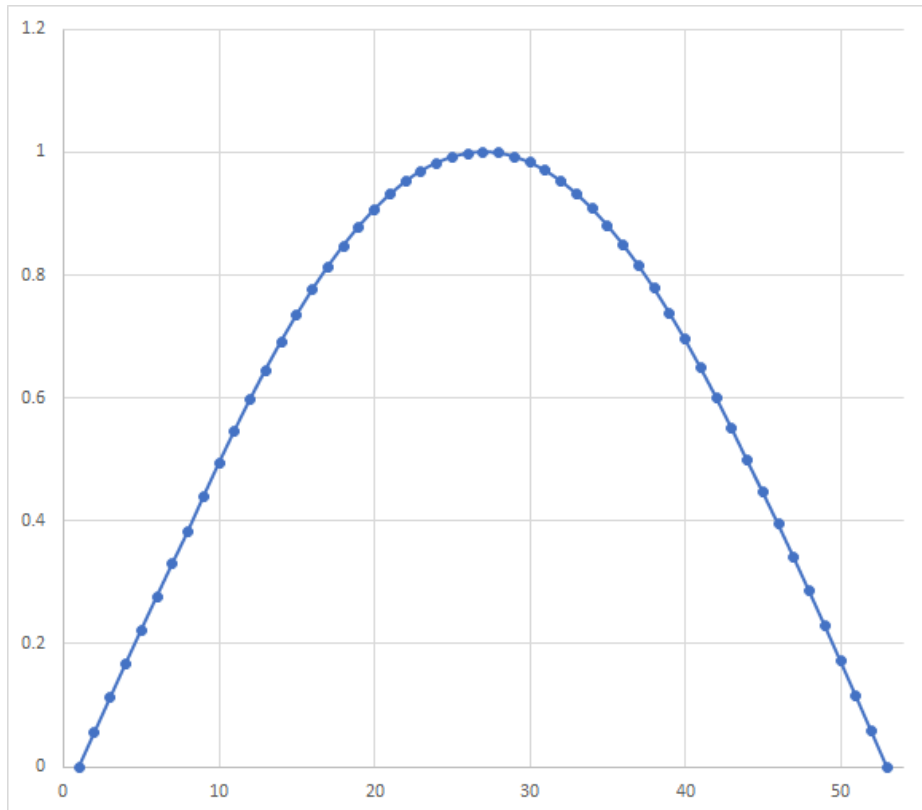
3.4 Morgenstern-Price Method

Morgenstern and Price's method (1965) is similar to the Bishop's method with parallel inclined interslice forces, but uses a function to express a varying ratio between the normal and shear interslice forces from left to right across the slip circle. (If this function is constant, the method produces the same result as the Bishop method with parallel inclined interslice forces.)

The choice of interslice function can be a constant ratio or a half-sine function, scaled by a factor λ, which is printed in the detailed results output. The shear interslice force at a slice boundary is calculated as:

$X = E \, \lambda \, f(x)$

where $f(x)$ is either constant or the half-sine function, which has the same shape as a sine curve and varies from 0 at the left end of the slip to 1 halfway across the slip, reducing to 0 at the right end of the slip:

This method satisfies both moment and force equilibrium for each slice and for the slip as a whole.

3.4.1 Iteration procedure

The iterative procedure used in the Morgenstern-Price method in Slope was developed by Zhu et al (2005).

For each iteration i, Slope calculates a new factor of safety F_i using the ratio of resisting forces to disturbing forces (which is a function of F_{i-1}).

The scaling factor λ is then calculated from moment equilibrium for all slices. The updated value is used to calculate a new set of interslice forces for the next iteration. When the difference between F_i and F_{i-1} is within the specified tolerance, and the change in the computed value of the scaling factor λ is within the specified tolerance (0.001), the calculation is complete.

For the first iteration, the program assumes that $F = 1$ and $\lambda = 0$.

This method generally requires fewer iterations than the Bishop methods, typically fewer than 10.

3.5   Janbu's Methods

Janbu's methods are applicable to **non-circular** slip surfaces. The method reduces to the Bishop solution for circular slip surfaces.

Like the Bishop's methods, three variants are available, which differ only in the direction of the interslice forces (horizontal, parallel inclined, or variably inclined).

Horizontal interslice forces

This method is taken from Janbu, 1957 and is applicable to **non-circular** slip surfaces. The assumed force distribution satisfies overall vertical and horizontal equilibrium, but **not moment equilibrium.**

The above leads to errors in the calculated factors of safety. These are on the safe side, but can be up to 15%. The more refined Janbu methods using inclined interslice forces are therefore recommended.

Parallel inclined interslice forces

This method is applicable to both circular and non-circular slip surfaces. Horizontal and vertical equilibrium are satisfied for each slice, and moment equilibrium for the slipped mass as a whole. This is achieved by taking moments about a point near to an equivalent centre of a circle.

When applied to circular slip surfaces the equations become identical to Bishop's method with parallel inclined forces and the calculated factor of safety is the same.

The benefits and limitations of the method are similar to those of Bishop's Method. The method is capable of giving misleading results due to the problem of interlock, (see Interlock). The program prints a warning message if the calculated factor of safety is likely to be in error.

Variably inclined interslice forces

This method is applicable to both circular and non-circular slip surfaces. It is designed to overcome the problem of interlock.

When applied to circular slip surfaces the equation becomes identical to Bishop's method with variably inclined interslice forces and the calculated factor of safety is the same.

3.6 Interlock

Bishop (1955) pointed out that there are a variety of force distributions which will satisfy the conditions of equilibrium. In many cases the assumption of horizontal or parallel inclined interslice forces is reasonable and leads to sensible results.

A case where errors can occur for horizontal or parallel interslice forces, is that of 'interlock'. This arises in the case of a deep slip with a low factor of safety, where the toe of the slip surface passes through a dense granular material.
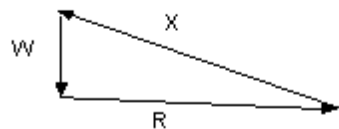
If the deep slip emerges at a steep angle and has a high mobilised angle of friction ? m where:

tan ? m = (tan ?) / F

Then the direction of the resultant force, R, on the base of the slice may be almost horizontal or even pointing downwards.
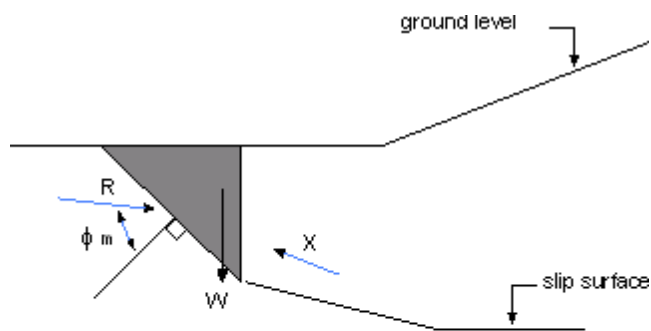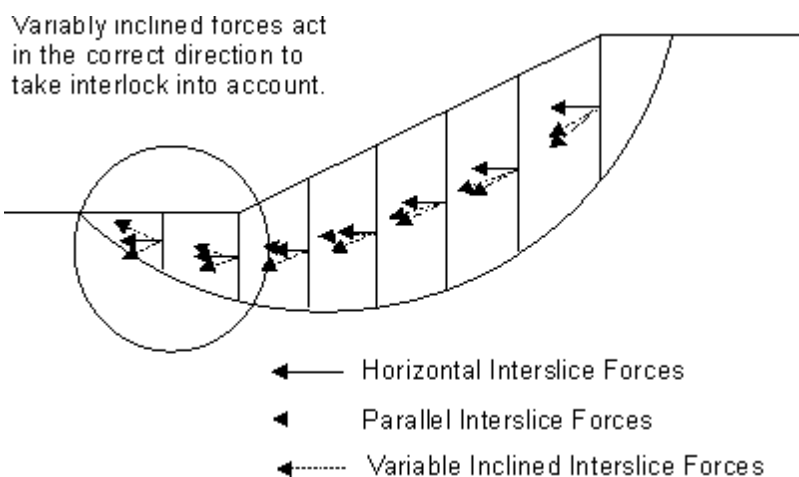

a. Section

b. Force Diagram

In order to satisfy equilibrium of this slice the interslice force, X, must point upwards. This direction is not consistent with the assumption of either horizontal or parallel inclined interslice forces.

Variably inclined forces act in the correct direction to take interlock into account.

→ Horizontal Interslice Forces
◄ Parallel Interslice Forces
◄------ Variable Inclined Interslice Forces

In such cases the method of variably inclined interslice forces should be used.

Note : Slope does not provide a warning when this problem may occur.

3.7   Positioning of Slices

Slope divides each slip mass into a number of slices. The resulting slice boundaries are located at the following points:

* at the left and right extents of the slip surface.
* at any change in gradient of a stratum.
* at each slip surface/stratum intersection.
* at each slip surface/phreatic surface intersection.
* at the mid point of a slice whose width is greater than the average slice width given by:

(Xright – Xleft) / Minimum number of slices

3.8   Reinforcement Calculations

If reinforcement is specified and active, the forces in the reinforcement are calculated and applied in the analysis in slightly different ways depending on the selected analysis method.

For the **Bishop** analysis method, reinforcement can be either specified as contributing additional restoring moment (hence increasing the factor of safety) or as surface loads, where the surface load applied equals the capacity of the reinforcement derived for the current slip surface.

If the reinforcement is used to contribute additional restoring moment, the soil restoring moment is calculated as usual (but with any partial factors taken into account), then divided by the moment correction factor. The reinforcement restoring moment is then added and the

factor of safety calculated.

For the **Morgenstern-Price** method, the calculated forces are either applied through the base of the slices where the reinforcing elements cross the slip surface, or as surface loads. The difference to the factor of safety in many cases will be small, but the normal and shear forces at the base will tend to show more "spikes" in the former method than in the surface load method.

Calculation of design capacity of reinforcing elements where they cross the slip surface

For end anchored elements (rockbolts Type B):

Tj = T/S

For ground anchors without pre-stress or soil nails, capacity is the minimum of design pullout force, tensile force and stripping force, so

Tj = min{T/S, BLO/S,(P+BLi)/S}

For ground anchors with prestress, the applicable prestress cannot exceed this value. The input prestress is reduced in proportion to the amount of fixed length outside the slip surface. In the output, the applicable prestress and any additional capacity are shown separately. The applicable prestress per m run of slope is:

Tpj = min{Tj, (Tp/S x LO/L)}

and the additional capacity is (Tj - Tpj).

For geotextiles, capacity is the minimum of design tensile force and pullout force, so

Tj =min{T, 2LOτ}

where
T is design tensile capacity per m run of slope (Tult x fcr/(fm11 x fm12 x fm21 x fm22 x fn x fs))
where
    fcr is the partial factor for creep reduction
    fm11 is the partial factor for manufacture
    fm12 is the partial factor for extrapolation of test data
    fm21 is the partial factor for damage
    fm22 is the partial factor for environment
    fn is the partial factor for economic ramification of failure
    fs is the partial factor for material strength
Tp is input prestress per anchor
S is out-of-plane spacing
B is bond strength (force per unit length of anchor/nail)
P is design surface plate capacity
Li is bonded length within the slip circle
LO is bonded length or length outside the slip circle

L is total bonded length

The calculation of pullout and stripping forces are mentioned above. To calculate them the shear / bond strength of the appropriate soil strength model has to be applied to the material the reinforcement is in (linear, hyperbolic etc).

B is bond strength (force per unit length of anchor/nail), which can be calculated or specified by the user. If calculated, the value is based on equation 12 from BS8081 or section 4.3.2 of BS8006-2.

For BS8081 the equation to calculate bond strength is:

$\pi D(\sigma v' \tan\delta + ca)/(fp \times fn)$

where

$\sigma v' = \gamma h + w vertical$

h is vertical distance between reinforcement and slope surface

For BS8006-2 the equation to calculate bond strength is:

$\pi D(\sigma r' \tan\delta + ca)/(fp)$

where

$\sigma r' = \sigma v'(1 + KL)/2$

and $KL = (1 + Ka)/2$

Shear strength of soil = $\tau = (\sigma v' .a. \tan\delta + ca)/(fp \times fn)$ (for drained linear strength model). It should be noted that a reduced pullout factor is adopted in this analysis as the factored strength and friction angle are used.

δ   is factored soil friction angle (tan-1(tan ? '/fmsphi)) where fmsphi is factor on friction angle
ca   is factored soil cohesion ((ac c)/fmsc)
γh   is weight of soil above the reinforcement behind the slip surface - soil unit weight is multiplied by the applicable partial factor
w   is surcharge on the surface above reinforcement behind the slip surface - with factors on dead and live load applied, so w = (dead load x dead load factor)+(live load x live load factor)
a   is coefficient of interaction between reinforcement and soil relating to the φ' of soil
ac   is coefficient of interaction between reinforcement and soil relating to the c' of soil
Ka   is the Rankine active earth pressure coefficient
fp   is partial factor on pull-out (BS8006 =1.3)
fn   is partial factor for structure importance (BS8006 =1-1.1)

If the bond strength is specified, the value B used in the calculations is the user's specified value divided by
(fp x fn).

The only partial factor not used at the moment is fs – sliding along reinforcement, which will be added in a later stage of development. This would apply if the slip surface is within a certain distance from the reinforcement, to reduce strength on slip surface.

Surcharges are excluded from the pullout calculation by default, but can be included by setting the field "Use in pullout calc" in the Materials table to Yes.

Calculation of additional restoring moment due to reinforcing elements (Bishop method only)

The additional restoring moment due to the reinforcement is defined in BS8006-2:2011 as

MRR = MRT + MRV

where MRT is the sum of moments due to tension in the nails/anchors and MRV is the sum of moments due to shear in soil nails. Calculation of shear developed in soil nails is not included, so the equation reduces to:

$$M_{RR} = M_{RT} = \sum_{j=1}^{m} \left[sin(\theta_j - \omega_j) + X \right] \frac{T_{dj} R_j}{S_j}$$

$$\text{Option 1} \quad X = 0$$

$$\text{Option 2} \quad X = \frac{sin\omega_j tan\varphi_d}{sin\theta_j + cos\theta_j tan\varphi_d}$$

$$\text{Option 3} \quad X = cos(\theta_j - \omega_j) tan\varphi_d$$

The component 'X' represents the nail tension increasing the normal force on the slip surface, these are adapted from Figure 18 of BS8006-2:2011.

For anchors with prestress, BS 8081 applies and an additional restoring moment due to prestress is :

$$M_{RF} = \sum_{j=1}^{m} \left[T_{pj} R \tan\phi' cos(\theta_j - \omega_j) \right]$$
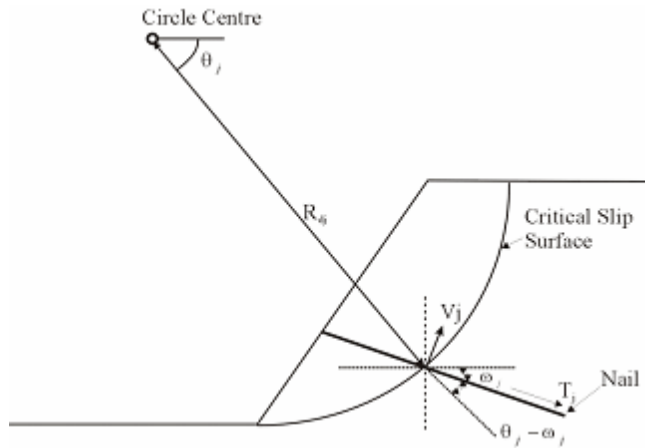
where Tpj is the applicable prestress as defined above.

Figure 1.

Tj = design tensile capacity of the reinforcing element
Sj = the horizontal spacing of reinforcement
Vj = design available shear resistance
Rdj = radius of the slip circle
θj = angle of the radius from the horizontal
ωj = angle of the reinforcing element from the horizontal.

**Application of reinforcement forces as surface loads or at slice bases**

For the Bishop method, if the "Apply as Surface Loads" box is ticked (currently available for soil nails and ground anchors) then the design capacities Tj and Tpj will be resolved into horizontal and vertical load components and applied at the point where the reinforcing element intersects the ground surface. The load applied to each slice will be shown in the "Point Loads" columns of the detailed results output table.

For the Morgenstern-Price method, the design capacities will be resolved into horizontal and vertical load components and either applied at the intersection of the reinforcing element and the ground surface (if "Apply as Surface Loads" is ticked) or at the intersection of the reinforcing element and the slip surface. The load applied to each slice will be shown in the "Point Loads" columns of the detailed results output table.
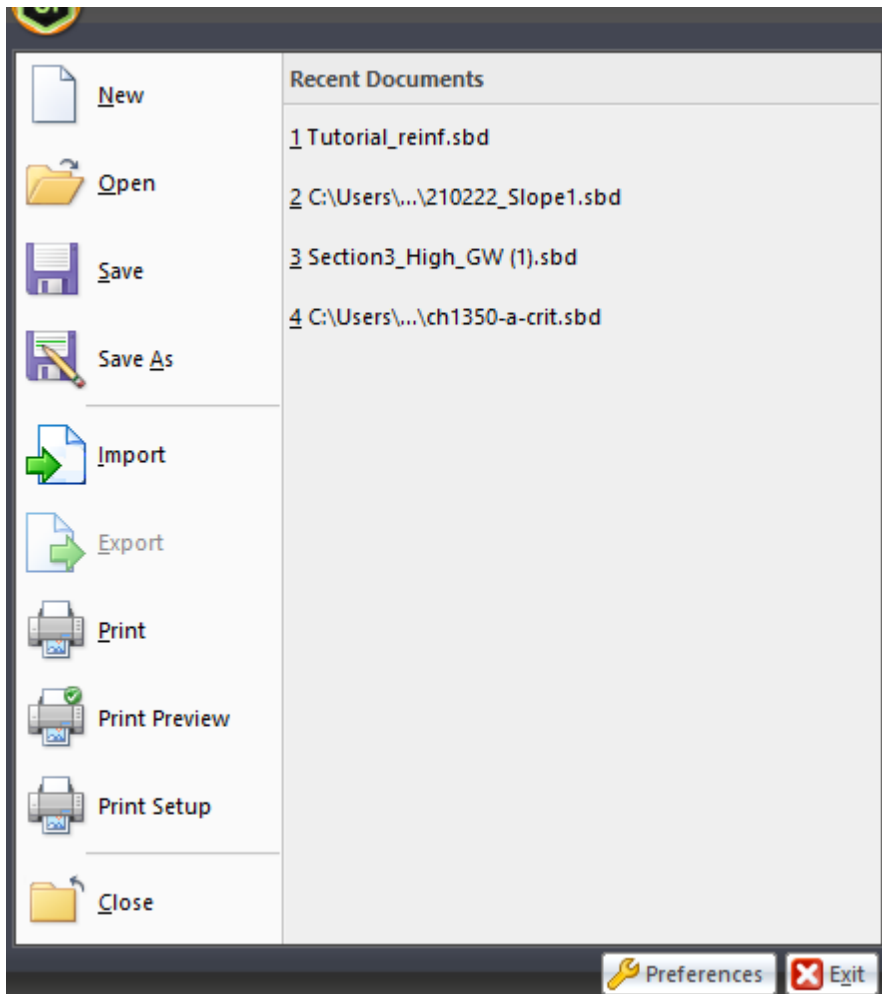
Part IV

4 Getting Started

On first opening the program, a welcome screen is shown (unless this option has been switched off by the user in the Preferences settings dialog).
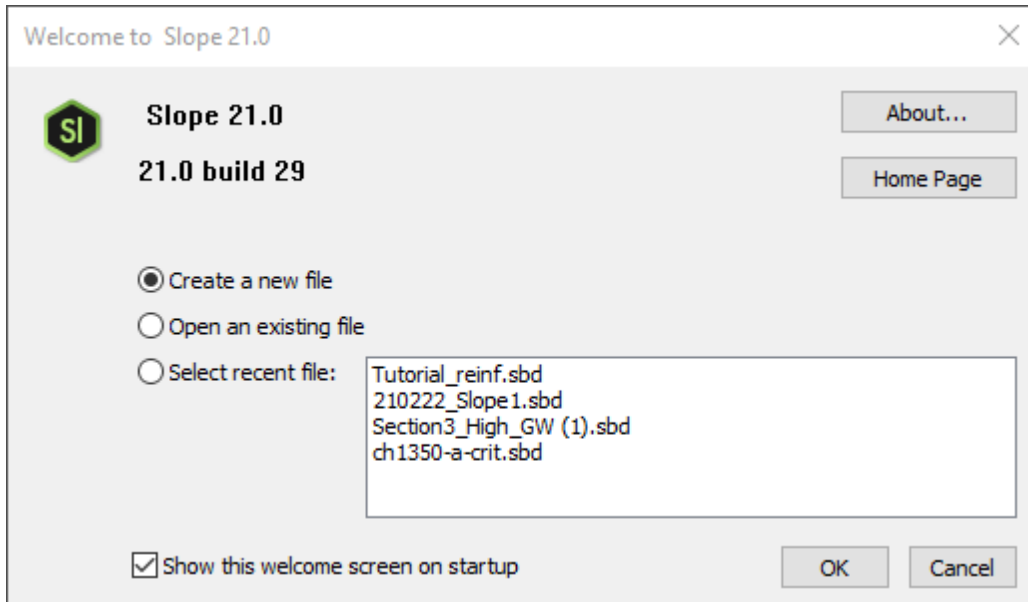
File commands (New, Open, Save, Save As etc.) are available from the application button menu at the top left corner of the program main window:

The application menu available from the Application button at top left of the ribbon

Navigation around the program is done using the commands on the ribbon toolbar and the left navigation panel (the Data Explorer).

The basic data consists of material parameters, ground model geometry and slip circle definition. Surface loads and reinforcement are optional. Water data is either entered as specified pore pressure distributions, which can be set per soil layer, or as groundwater boundary conditions in the graphical input view if a pore pressure calculation is required. The data can either be entered directly into the program, or imported from a data file from previous versions of Slope (filed with suffix .sld). Ground model geometry can also be imported from either a DXF file containing 2D polylines, or a "triangulated irregular network" TIN file, which contains one or more 3D surfaces exported from a GIS program. For a TIN import, a vertical plane along which to cut the 3D surfaces must be specified, so that the program can generate a 2D soil profile.

Once all required data has been entered, selecting Analyse on the ribbon toolbar then carries out a steady state seepage analysis or calculates pore pressures at all relevant points from the specified data, and the resulting pore pressures are used in a limit equilibrium slope stability analysis. The progress of the analysis, including how many slip circles have been analysed or
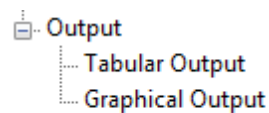
rejected, is shown on screen:
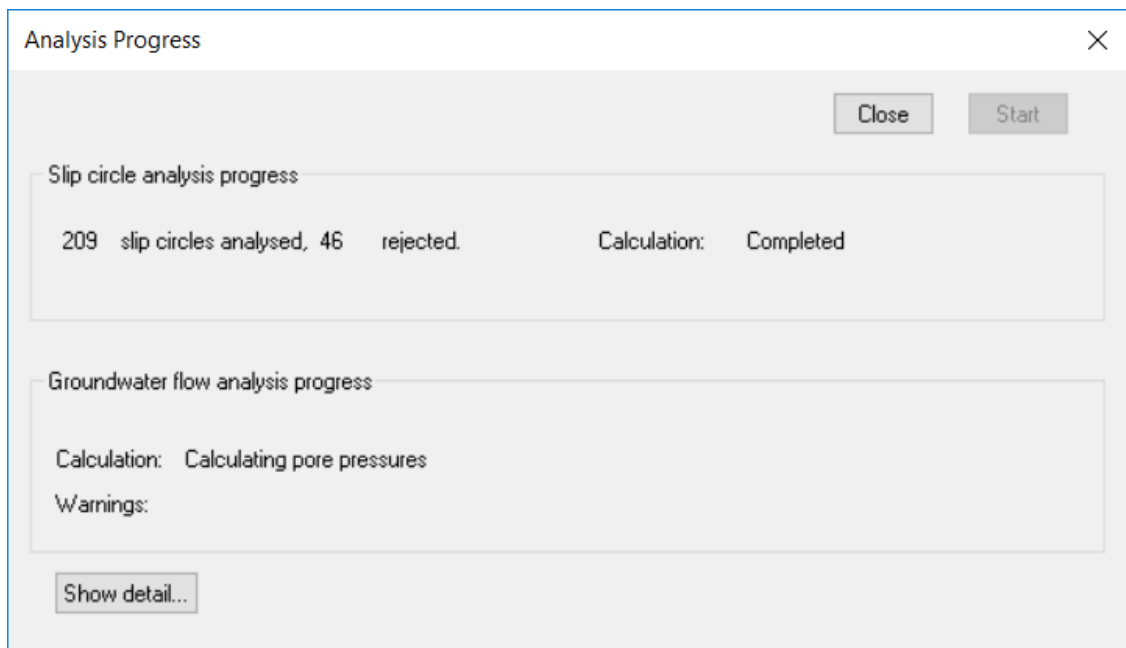
The Analysis Progress dialog

On completion of analysis and closing the analysis progress dialog, the Data Explorer and ribbon toolbar will be updated to allow access to the tabular and graphical output options.
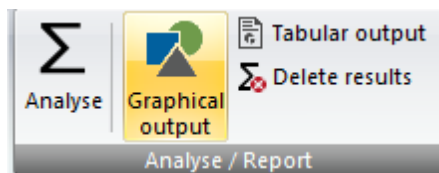
Analyse / Report panel on the ribbon   **Output options on the Data Explorer**

Part V

5 Entering Data

5.1 Defining Analysis Options

5.1.1 Titles

The Titles dialog allows entry of identification data for each program file.

The following fields are available:

Job Number	allows entry of an identifying job number*.
Initials	for entry of the users initials*.
Job Title	allows a single line for entry of the job title*.
Subtitle	allows a single line of additional job or calculation information*.
Calculation Heading	allows a single line for the main calculation heading*.
Notes	Allows detailed description of an individual calculation. This can be added to tabular output.
Location information	May be used for example to link a data file to a GIS location.
Checked	Initials of checker, if entered, will be added to the file history record

*These fields will be reproduced in the title block at the head of printed information using the "Calculation Sheet" layout.

5.1.2 Units

The Units dialog is accessible via the Data Explorer view or by clicking Units on the Home tab. It allows the user to specify the units for entering the data and reporting the results of the calculations. These choices are stored in individual data files.

Default options are the Système Internationale (SI) units - kN and m. The drop down menus provide alternative units with their respective conversion factors to Newtons and metres.

Standard sets of units may be set by selecting any of the buttons: SI, kN-m, kip-ft kip-in.

Once the correct units have been selected then click 'OK' to continue.

SI units have been used as the default standard throughout this document.

5.1.3 Specification

The Specification command, on the Data Explorer or ribbon Home tab, either switches focus to the Specification property grid or re-opens it if it has been closed or minimised. If results already exist, they must be deleted using the Delete Results button on the ribbon to enable editing of these settings.
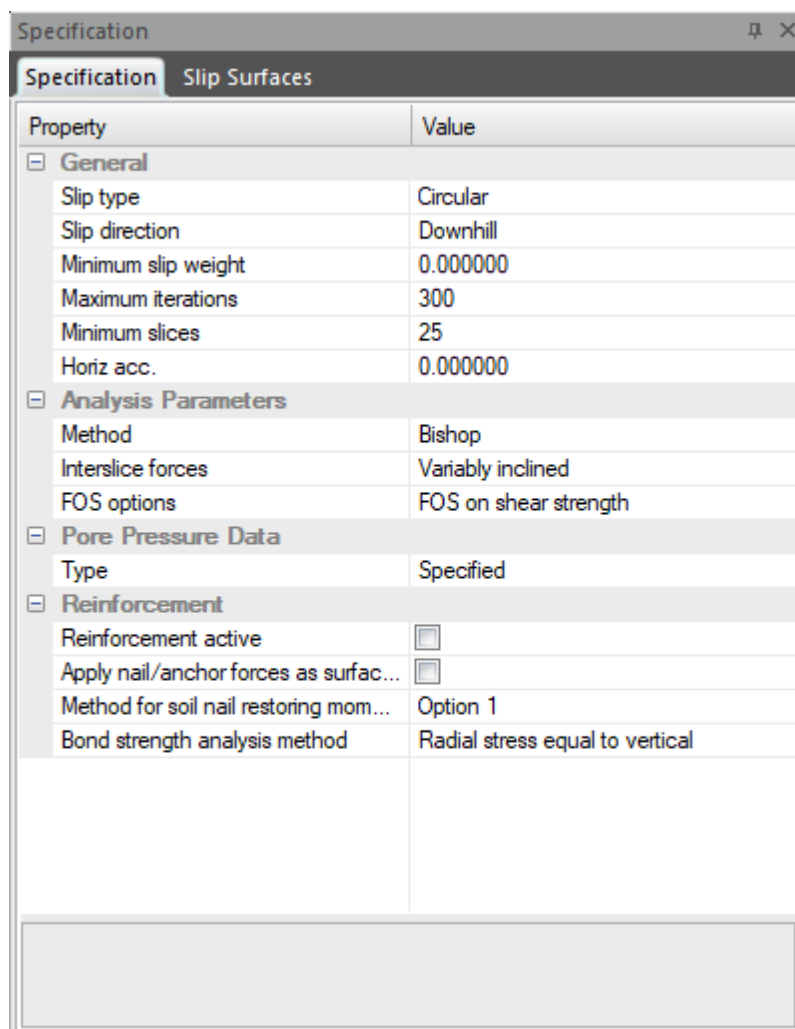
The fields in the Specification grid set the options for overall analysis of the problem.

General

Slip type - circular or non-circular

Slip direction - downhill, increasing x or decreasing x. It may be useful to specify the direction explicitly if there are multiple slope directions within the geometry.

Minimum slip weight - setting a non-zero value here will discard slip circles with lower weights - this can be useful to exclude small surface slips from the calculations.

Maximum Number of Iterations - all the methods of solution iterate to reach a solution. The

program defaults to 300 iterations, but the user can specify any number.

Minimum Number of Slices - the program requires the minimum number of slices for each slip surface to be specified. The default value is 25.

Analysis Parameters

For the Bishop or Janbu analysis methods, the direction of interslice forces (horizontal, parallel or variably inclined) must also be selected. The default is variably inclined forces. For the Morgenstern-Price method, there are two options for the interslice force function, constant or a half-sine function. For circular slips, the constant interslice force function is equivalent to the Bishop method with parallel inclined interslice forces.

For all methods including Bishop, Janbu and Morgenstern-Price, choose between FoS on shear strength (the default), FoS on disturbing or restoring loads, or a partial factor analysis. If partial factor analysis is specified, the visible fields will expand to allow selection of a partial factor set to apply to the input data. The results will be reported as an "over-design factor" rather than a factor of safety.

Pore Pressure Data

Choose between specified pore pressures for each soil layer, which can be hydrostatic, defined by piezometers or a single Ru value (see Water Data), or generated pore pressures. If Generated is selected, then permeabilities should be added for each soil material and groundwater boundary conditions must be set to enable a steady state seepage analysis to be run. See Generated pore pressures for details.

Reinforcement

Check boxes are available to set whether reinforcement is active and whether to apply nail/ anchor forces as surface loads. These apply to all reinforcement defined in the data file. The remaining options allow the user to specify the method for calculating the bond stress and restoring moment attributable to soil nails. Details of these methods are included in the Reinforcement Calculations section.

5.2   Defining the Ground Model

The ground model to be analysed consists of a 2D section through the slope, divided into connected polygons of different soil strata.

The material parameters for each stratum are entered in a table (see Material Properties), but the geometry of the slope and many of the other properties are entered and accessed via the Graphical Input view.

The following sections give details of the data input for the ground model and a summary of the features available in the graphical input. Detailed tutorials and videos on the methods described are available on the web site.

5.2.1   Material Properties

The Materials command opens this table from either the ribbon or the Data Explorer.

| SLOPEman-seepagelines.sfd : Graphics | SLOPEman-seepagelines.sfd : Material properties × | | | | | | | | | |

	A	B	C	D	E	F	G	H	I	J
		Unit Wt kN/m³			Shear strength parameters					
Material	Description	Above GWL	Below GWL	Condition	phi' OR phi0	Cohesion c' OR c0 [kN/m²]	k [kN/m²/m]	y0 [m] OR b	cu/p' ratio	Colour
Defaults	Material #	0.00	0.00	Drained - linear strength	0.00	0.00	0.00	0.00	0.00	
1	Made Ground	17.00	17.00	Drained - linear strength	20.00	2.00				
2	Sand1	18.00	18.00	Drained - linear strength	29.00	0.00				
3	Clay	21.00	21.00	Undrained		75.00	5.00	0.50	0.00	
4	Sand Lens	18.00	18.00	Drained - linear strength	30.00	0.00				
5										

Description : The name of the material. The default name will be Material 1.

Unit weight : The bulk unit weight of the material above and below the ground water table.

Condition : The condition of the material i.e. undrained, drained with linear strength, or drained with strength calculated using a power or hyperbolic function. Choose the required option from the drop-down list. Depending on the selection, the fields for the relevant shear strength parameters will be activated.

Shear strength parameters :
- ? ' (deg) : angle of friction
- c' : drained cohesion
- c0 : undrained shear strength at a specified elevation y0
- k : the rate of increase of shear strength with depth
- cu/p' : A ratio for normally consolidated soils, where p' is the effective vertical stress which is calculated by the program at the point on the slip surface for each slice
- a and b : constants used in the models for drained materials with strength expressed as a power or hyperbolic curve (see below).

Colour : colour to shade geometry areas of this material in the Graphical Input.

Material models available

Undrained   Enter values for either:
                         1. A single value of undrained shear strength c

                         2. A value of undrained shear strength c0, which varies linearly with
                              elevation y, where:

$$c = c0 + k(y0 - y)$$

c = undrained shear strength at any elevation y
c0 = undrained shear strength at a specified elevation y0
k = rate of increase of shear strength with depth

3. A ratio of cu/p' for normally consolidated soils, where p' is the effective vertical stress calculated at the point on the slip surface for each slice

4. A combination of 2 and 3. If both are selected, the higher value of strength is used.

Drained, linear strength	Enter the angle of friction in degrees and a value for drained cohesion c'.
Drained, power curve strength	Enter the angle of friction at which a linear relationship takes over, plus the two constant a and b. Slope calculates the material strength using the relationship $\tau = a\sigma_n b$.

Then $(\delta\tau / \delta\sigma) = ab\sigma_n^{b-1}$, which is equal to tan(?') at σ_n. The associated c' is given by $c' = ab\sigma_n^b(1-b)$.

The linear relationship $t = c' + \sigma_n \tan(?')$ takes over at some predetermined ?', say $?'_0$. When σ_n exceeds the stress at which this transition takes place, the strength relationship reverts to Mohr-Coulomb.

Drained, hyperbolic strength	Enter values for c' and $?_0$ as follows. We assume a relationship of

$$\tau = c_\infty \sigma_n \tan(?_0) / (c_\infty + \sigma_n \tan(?_0))$$

Then

$$(\delta\tau / \delta\sigma) = \tan\varphi = [(c_\infty \tan(?_0)) / (c_\infty + \sigma_n \tan(?_0))] - [(c_\infty \sigma_n \tan^2(?_0)) / (c_\infty + \sigma_n \tan(?_0))^2]$$

$?_0$ is the angle at $\sigma_n = 0$ and c_∞ = value of c when $\sigma_n = \infty$. Both φ_0 and c_∞ are constants.
c can be calculated from $c = \tau - \sigma_n \tan ?$.

5.2.1.1 Permeabilities

If the pore pressures are to be generated from steady state seepage analysis, a Permeability tab will be added to the Materials table. Information is required for every material and will be automatically created with default values.

Material	Description	Drainage condition	Unit wt of pore fluid	Bulk modulus K_w	Permeabilities		Incl.	Y_0	Gradient of k	Desens factor	Min pwp	Perm'y factor
					k1	k2						
			[kN/m³]	[kN/m²]	[m/s]		[°]	[m]			[kN/m²]	
Defaults	Material #	No water		2.2e+06						1.00	-1000.00	1e-06
1	Clay01	Consolidating	10	2.2e+06	1e-05	1e-05	0.00	0.00	0.00	1.00	-1000.00	1e-06
2	FIRM	Consolidating	10	2.2e+06	1e-05	1e-05	0.00	0.00	0.00	1.00	-1000.00	1e-06
3												

\ Materials λ Permeabilities /

Drainage condition	"Drained" means constant pore pressure.

Drainage condition

"Drained" means constant pore pressure.
"Undrained" means that no volume change will occur and pore pressures are computed.
"Consolidating" means that time-dependent consolidation is to be computed.
"No water" means that there is no water pressure and no flow of water. Thus it could be used to model, for example, an impermeable concrete wall. The behaviour is essentially drained, i.e. volume change can occur, but unlike other drained zones, a no water zone does not interact with consolidation or seepage. That is, its nodes do not become fixed head nodes forming a drainage boundary.

Drained, undrained and consolidating materials may be used together in the same run. Drained elements provide additional fixed heads at their nodes.

Unit weight of pore fluid γw

The unit weight of pore fluid with depth, which is used to calculate pore water pressures from specified data. Only in hydrostatic conditions is this equal to the weight density of the water.
γw should be set to zero for layers of soil above the water table which are unable to sustain suction.

Bulk mod Kw

Used for undrained materials. Repeated from the general materials table for convenience.
If Kw is positive, it represents the bulk modulus of pore water, usually taken to be 2200MN/m². For Water or Void materials, an appropriate value must always be quoted.

Permeabilities k1, k2

Orthogonal principal permeabilities.
k1 - horizontal, if inclination is zero
k2 - permeability at right angles to k1

Inclination

Inclination of the first principal permeability, measured anticlockwise from the x-axis (specified in degrees).

Level y0, Beta

Set these values if the permeability varies with depth within the material. y0 is the reference level at which k1 and k2 have been defined, and Beta describes the relationship between that and permeability at other depths:

$$\beta = -\,(dk / dy) / K(y0) \quad \text{(where } K(y0) \text{ is } K \text{ at } y0)$$

Desensitising factor F   This must be set to 1.0 for all materials for which an accurate consolidation analysis is required. However, it will sometimes be the case that the problem involves low cv materials, for which accurate consolidation is required, and high cv materials which provide boundary conditions to the others, and for which some error in pore pressure could be tolerated. In this case, it is desirable that the time steps are suited to the low permeability materials, and the desensitising factor is used to keep the high permeability materials in a stable state.

In effect, for each iteration, the coefficient of consolidation of the high cv materials is decreased by this factor; however the overall effect is made less severe as iterations proceed within each time increment. Users must inspect the results for materials with values of F greater than 1.0, and satisfy themselves that they are appropriate for the problem.

Min pwp   Minimum pore water pressure attainable by the material. This is the pressure at which the material becomes unsaturated; it will generally be zero or negative, representing suction. For unlimited suction, leave blank.

Permeability factor   If the pore pressure in the element tends to fall even lower than that specified above, it will be held at the specified minimum, and the permeability of the element will also be reduced by this user-specified factor. This represents conditions in a non-saturated zone. In simple cases, the factor should be set very low: the default is 10E-6 meaning that the permeability of the unsaturated material is one millionth of the saturated value. Other values could be used, given a proper understanding of unsaturated flow.

5.2.2   Soil Layers

Soil layers are created in the Graphical Input view. The 2D section through the ground to be analysed should be defined as a set of 2D polygons.

Lines to construct the polygons can be imported from DXF or 3D TIN files, or drawn directly in the 2D graphical input view using the drawing tools.
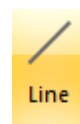
To begin drawing a polygon, select the Line button on the ribbon Graphical Input tab. Moving the cursor on to the input view will show a tooltip with the current snap coordinates of the cursor. Left-click will add a point at those coordinates and begin drawing a line. Subsequent left-clicks will add more points until the Escape key is pressed to end line drawing.

If a point at exact coordinates is required, including a point off the current screen limits, right-click and the tooltip coordinates will become editable. Enter the required coordinates with the x and y coordinates separated by a comma, and press the Enter key. The next point will be added at the coordinates typed in, and the screen limits reset to include it in the drawing area.

To join a line to an existing endpoint, ensure the cursor has changed to a small square. This indicates the program will "snap" the new point to an existing point. If this closes a polygon, a dialog box will appear asking for confirmation to create a new 2D area:
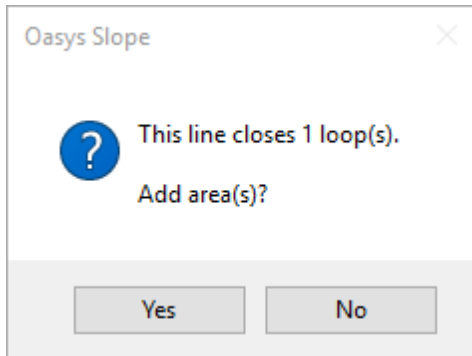
Clicking No leaves the lines as a closed polygon, but without defining it as a soil layer. This may be a suitable option if for example you have several layers to draw and want to get the geometry right before creating the areas at the end. You can create layers later by clicking the

Find layer button and clicking within a closed polygon. This method has the disadvantage that the layer will be created with the material undefined.

Clicking Yes shows the Layer Properties dialog in which a material number may be assigned to the area. Note the material data does not yet need to exist. The area will be drawn shaded in a default colour or the colour specified in the Material table for that number.
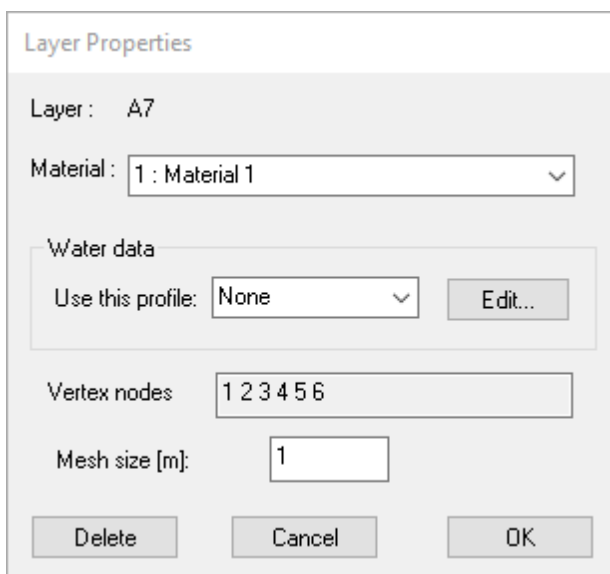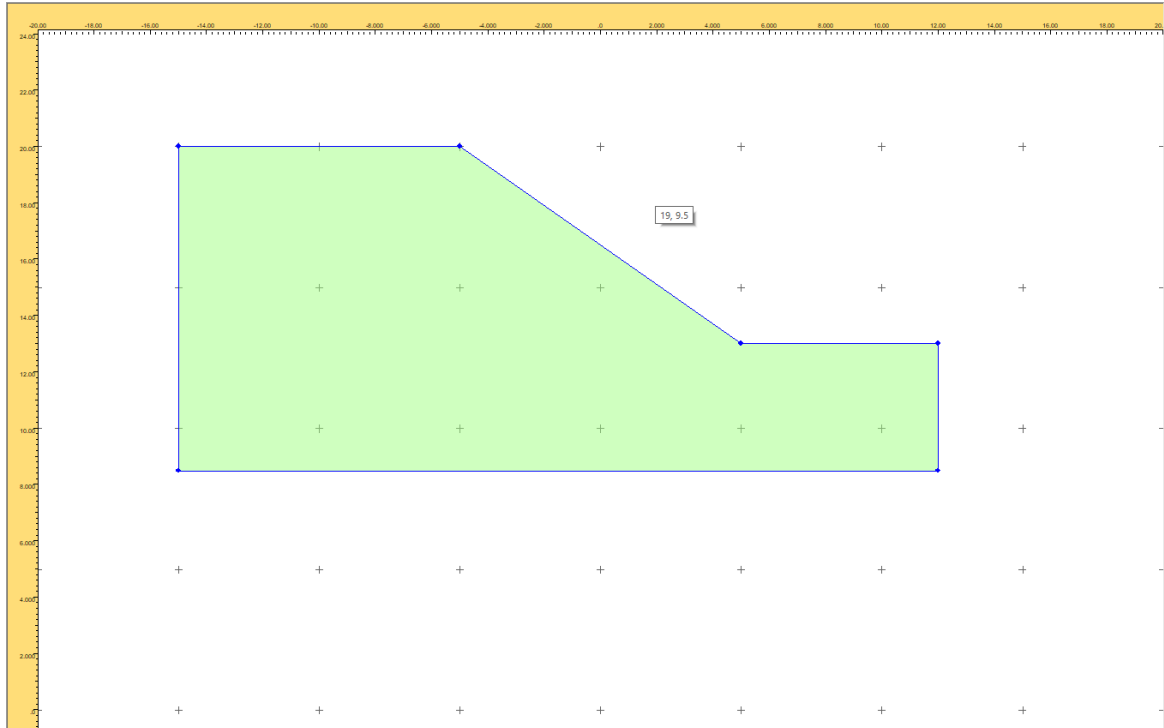
Further layers can be added by drawing new lines attached to the existing line endpoints until all the required layers are complete.

For details of importing soil layer data from other sources, see the following sections.

Importing DXF data
Importing TIN data
Generated pore pressures
Mesh generation

5.2.2.1 Importing DXF data

A DXF file can be imported into the graphical input view and used as the basis for stratum coordinates if required. Note that polyline data is not saved with the data file until selected and added to the model data.

Selecting Import from the application menu opens a file open dialog with two import options, DXF or LandXML files. For DXF, set the file type to DXF and navigate to the location of the required file. Select the file and click Open. If the coordinates in the DXF file need to be translated, enter the required translation coordinates on the dialog which appears.
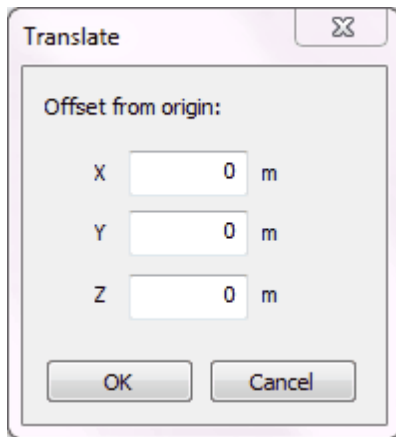
The graphical input updates to show any lines generated from the DXF file. These are added to the model data on import, and may need additional lines to convert them to polygons to form "soil layers" for analysis. See the Soil Layers topic for more details.

5.2.2.2 Importing TIN data

"TIN" (Triangulated irregular network) data can be exported from many GIS or BIM applications. They save a 3D surface as a set of connected triangles. Slope can now read LandXML files containing TIN data and cut a 2D section through the triangles to create a polyline representing the upper surface of a soil layer. Note that polyline data is not saved with the data file until selected and added to the model data.

To import TIN data, select Import from the application menu. This opens a file open dialog with two import options, DXF or LandXML files. For TIN, set the file type to LandXML and navigate to the location of the required file. Select the file and click Open. The program will read in all the TIN surfaces defined in the file and show a dialog for input of the required 2D section to cut. For convenience, the extents of the imported data are also shown.
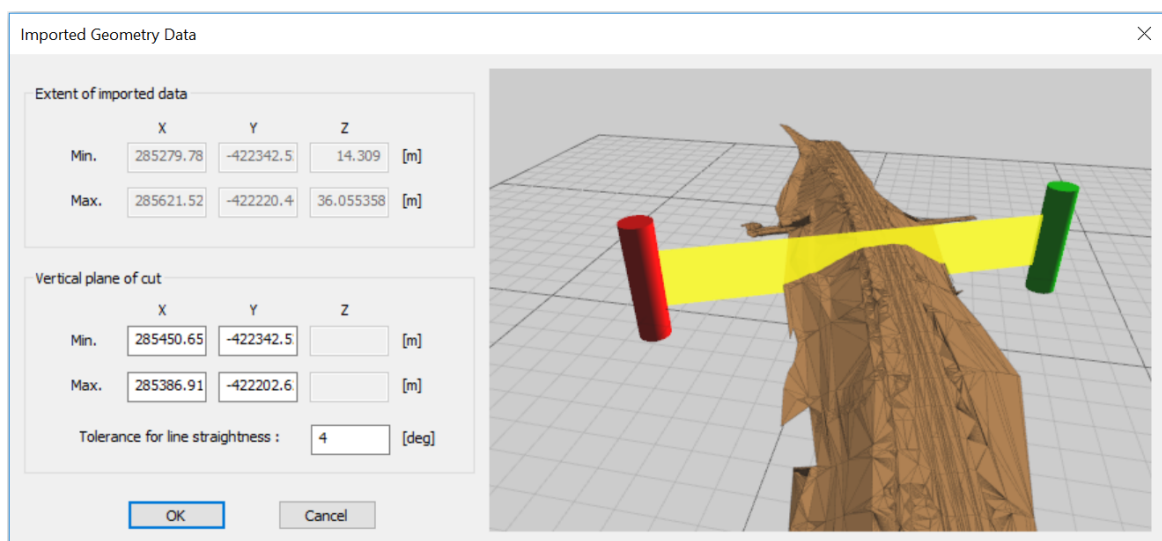
The "Tolerance for line straightness" setting will merge adjacent polyline segments if the
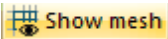
angle between them is less than this value in degrees, i.e. if the two line segments are almost parallel.

The graphical pane shows an image of the 3D surface, and the highlighted section the proposed cutting plane. This can be moved to the required position by dragging the end "goalposts" shown, or by revising the coordinates in the edit boxes.

Clicking OK on this dialog will add any imported data within the range of the specified vertical plane to the model data as lines. These may need additional lines to be drawn, to specify the polygons to form "soil layers" for analysis. See the Soil Layers topic for more details.

5.2.2.3 Mesh generation

Slope will generate nodes and elements before carrying out the pore pressure calculations, if generated pore pressures are required. This usually requires no intervention from the user.

If mesh information is required, the ⊞ Show mesh button can be used to view the mesh in either graphical view.
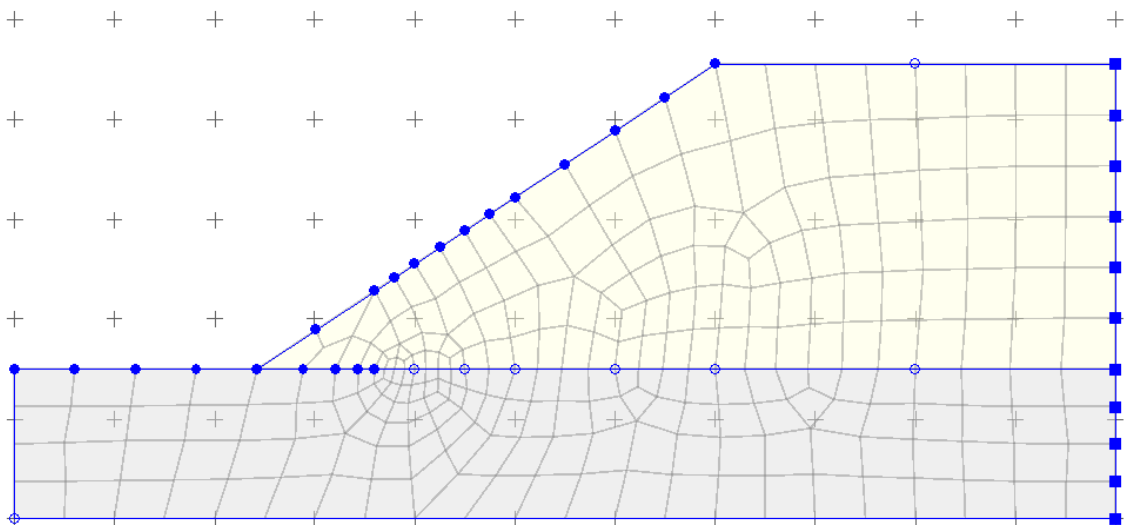
If a finer mesh is required, setting the element size on individual nodes or at line endpoints will create this for the next analysis.

5.2.3 Water Data

The method for entering water data for the slope depends on the setting for Specified or Generated pore pressures in the Specification pane.
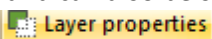
If the pore pressures are Specified, Water data is entered via a groundwater dialog. For
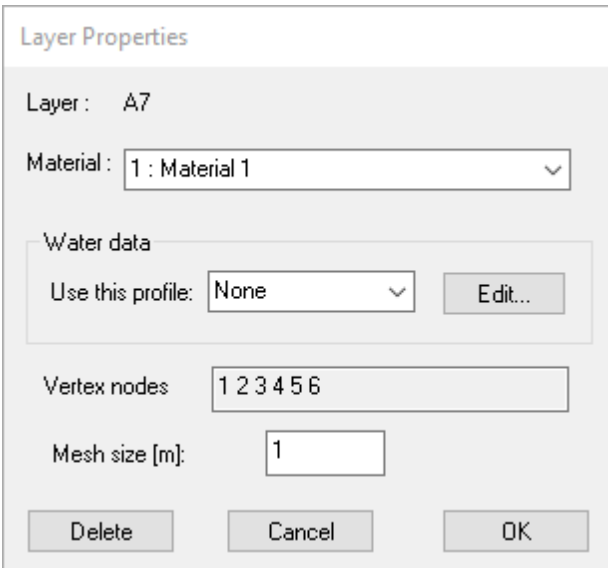
Generated pore pressures, material permeabilities must be defined (see [Permeabilities](#)) , and groundwater boundary conditions added to the model in the Graphical Input view.

5.2.3.1   Specified Pore Pressures

To enter or edit specified pore pressure, select Water Data on the ribbon Home tab or double click the Water Data item in the Data Explorer, or one of its sub-items if any water profiles already exist

The water data dialog shows a single water data specification at a time, so the available specifications are expanded in the Data Explorer so that any individual one may be edited.

The command can also be selected by clicking a soil layer area in the Graphical input and selecting **Layer properties** , then clicking the Edit button next to the water data field:

Either method opens the following dialog:

This data is only relevant if the pore pressures are set to Specified, so the command is disabled if pore pressures are to be generated from seepage calculations.

The water data types and required data for each are:

Hydrostatic: pore pressures are defined by a water table with hydrostatic pore pressures below. Enter the unit weight of water and maximum suction (negative pore pressure). Pore pressures lower than this value will be ignored. Enter the coordinates of the water table from left to right.

Piezometric: pore pressures are defined by a water table, with pore pressures below interpolated between values at piezometers within the soil mass.

Specified Ru: pore pressure at any point is proportional to the overburden at that point. The Ru value is the proportion of overburden to use in the calculation, so an Ru value of 0.3 will ...
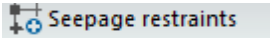
None: no pore pressures - this option can be used to clear pore pressure specification from a selected soil layer.
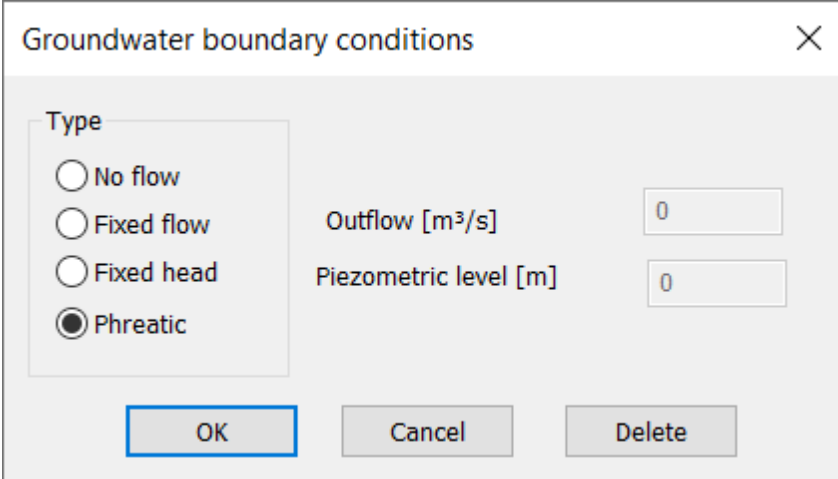
5.2.3.2 Generated Pore Pressures

Groundwater boundary conditions are required if pore pressures are to be generated from steady state seepage analysis. This setting is controlled from the Specification property grid. Boundary conditions can be one of four types: no flow, fixed head, fixed flow or phreatic.

Seepage boundary conditions can be applied to lines or to individual nodes. For a fixed head boundary, the piezometric water level must be entered. For a fixed flow boundary, the flow rate must be entered (SI units: m3/s/m). Phreatic boundaries are defined as those at which water may leave the mesh and the water pressure at these nodes will be maintained at atmospheric pressure. This would typically be at the ground surface, for example. No flow boundaries allow no net flow of water across them.

An interface between a "consolidating" and a "drained" soil layer is normally treated as a fixed head boundary. if this is not required, it may be specified to be "no flow" in the same way as any other user-defined groundwater boundary.
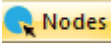
To apply boundary conditions, select the lines by clicking the button on the Graphics ribbon tab, then drawing a selection box around the required line(s) centres. Select from the "Edit data properties" panel on the ribbon and the boundary conditions dialog will appear:

When the mesh is generated, the required boundary condition will be applied to nodes generated along the line(s) with groundwater boundaries specified.

Sometimes a boundary condition may be required at a point, for example to model a "sink" or drainage point within the soil mass. This can be done by defining a node at that point and adding a seepage condition. To apply to individual nodes, select the nodes by clicking the button on the Graphics ribbon tab, then drawing a selection box or clicking on the

nodes. Select the seepage restraints command as before and enter the required details.

5.2.4 Polylines

Polylines tabular input can accessed from Polylines button on the Home tab of the ribbon.
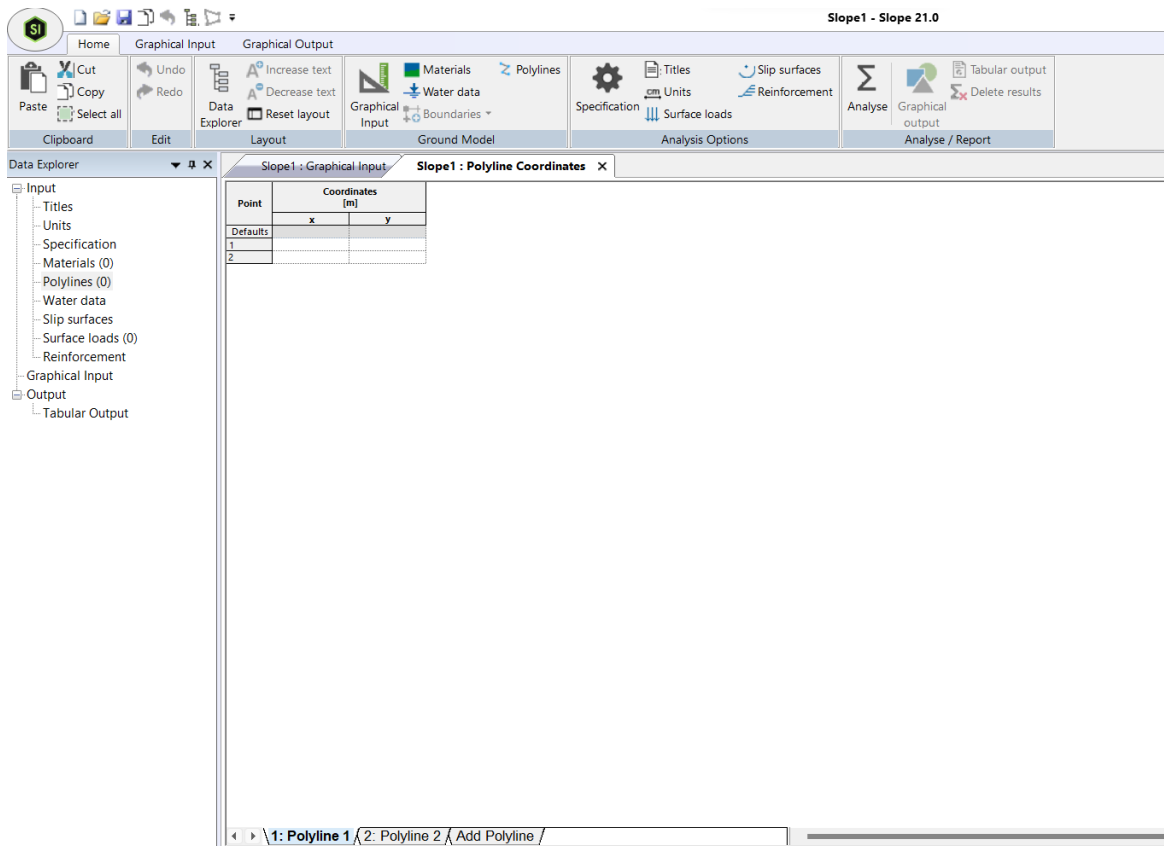
x and y coordinates of points on a polyline can be entered in the polylines tabular input. Polylines will be created by generating line segments between consecutive points of a polyline. These polylines can be viewed in the Graphical Input view for reference. New polylines can be added by clicking Add Polyline button at the bottom, similarly existing polylines can be deleted or renamed by right-clicking the respective tabs at the bottom.

All the polylines entered should follow certain rules:
- A polyline must have at least two points.
- A polyline should not form a closed loop or intersect itself.
- Avoid duplicate coordinates in a polyline.
- x coordinate of all the points in a polyline should be either ascending or descending order.
- Two polyline may touch or coincide but they should not intersect.

Strata enclosed by the polylines can be generated by clicking the Polylines -> Strata button on the Ribbon under the Graphical Input tab. Polylines are sorted according to their level and then extended in x direction to the maximum extents if possible. Maximum extents are determined by minimum and maximum value of x coordinate across all the points in all the poylines. Strata are then generated between the lowest polyline and second lowest and then

between the second lowest and the third lowest and so on. If needed, vertical members are automatically generated to join two polylines at the ends.

Note:
- Reference polylines which are in the Graphical Input view cannot be edited from Graphical Input.
- At least two polylines are required to generate strata.
- Polylines which are completely vertical are ignored while generating strata.
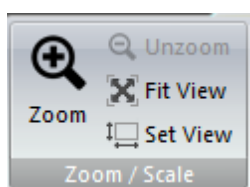
5.2.5 Graphical Input

There are many data entry and editing options available in the Graphical Input view. For details specific to each type of data entry, see the main topic for that data.

The ribbon tab contains the following command groups:

| Group | Commands available |

Zoom / Scale

Zoom - draw a box around the desired zoom area

Unzoom - return to previous zoom level

Fit view - fits the view extents to the current data

Set view - allows setting of the x or y extent of the view and the other dimension will change to keep an equal scale

Settings
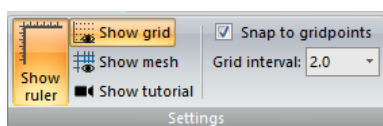
Show ruler - toggles the border ruler on or off

Show grid - toggles the grid crosses on or off

Show mesh - if pore pressures are being calculated, toggles the display of the finite elements on or off

Show tutorial - plays a demo video of the graphical input operations

Snap to gridpoints - new points / line endpoints will be snapped to the nearest grid interval

Grid interval - defaults to a suitable interval for the data, but may be changed using the dropdown list

Create

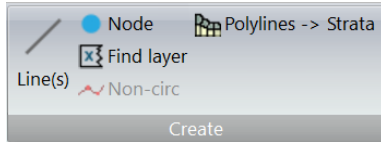Line(s) - allows line drawing by left-clicking to add line endpoints. Press ESC to terminate line drawing.

Node - shows a dialog to enter a new node's coordinates

Find layer - select this, then click within a closed polygon to automatically find and add the soil layer polygon

Non-circ - allows line drawing by left-clicking to add non-circular slip coordinates. Pressing the ESC key terminates drawing and clips the slip surface to the section boundaries. This will only be enabled if the non-circular slip type is selected.

Polylines -> Strata - generates strata enclosed by polylines from the polylines tabular input.

Select
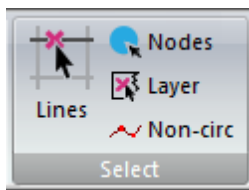
For these actions, click on the relevant button to begin, then:

Lines - draw a box around the centre of a line to select it. Click on the selected line to de-select it.

Nodes - click individual nodes to select or de-select them, or drag a box around a group of nodes.

Layer - click inside a layer to select or de-select it.

Selected lines, nodes or layers may be deleted by pressing the DELETE key. If they cannot be deleted (for example, a node which is part of a layer boundary) a warning dialog will be shown.

Press the ESC key to de-select all items when the required edit or action is complete.

Non-circ - click once near an existing non-circular slip. The points on the slip surface will be indicated by circles. Any point can be clicked and dragged to a different location to edit the position. Press the ESC key to finish and the slip surface will be clipped to the section boundaries.

Edit data properties

These buttons will only be enabled when existing data items are selected.
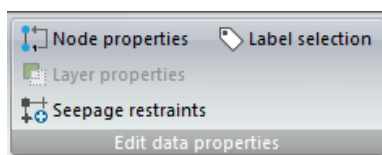
Node properties - if a single node is selected, this allows editing of the node coordinates and the finite element length adjacent to the node. (This is only relevant if pore pressures are being generated.) If multiple nodes are selected, their coordinates cannot be edited.

Layer properties - only available if a single layer is selected. Allows editing the layer properties as described in Soil Layers.

Seepage restraints - only available if pore pressures are being generated and if a single node, or one or more lines, is selected. Allows entry and editing of groundwater boundaries as described in Generated Pore Pressures.

5.3   Slip Surface Definition

Selecting the Slip Surfaces command in either the Data Explorer or the ribbon home tab will either:

- open the Slip Surfaces property grid, for circular slips
- open the Slip Surfaces tabular input, for non-circular slips

5.3.1   Circular Slip Definition

For circular slips, the Slip Surfaces command, on the Data Explorer or ribbon Home tab, either switches focus to the Slip Surface property grid or re-opens it if it has been closed or minimised. If results already exist, they must be deleted using the Delete Results button on the ribbon to enable editing of these settings. The fields in the property grid will appear as required by other settings, i.e. changing the radius type from "Defined radii" to "Tangent to stratum" will cause the remaining fields in that section to be re-drawn to show the relevant options for the tangent stratum.

A circular slip surface is defined by the x and y co-ordinates of the centre of the circle and the specification of the circle radius.

The centre of the circle is specified in terms of a single point, by x,y coordinates, or a grid, using the x,y coordinates of the bottom left corner of the grid and the inclination of the grid about this point, positive in the anticlockwise direction. If "Grid" is selected in the dropdown list, the other fields relevant to a grid of centres (number and spacing in the x and y directions, and rotation of the grid) will become visible.
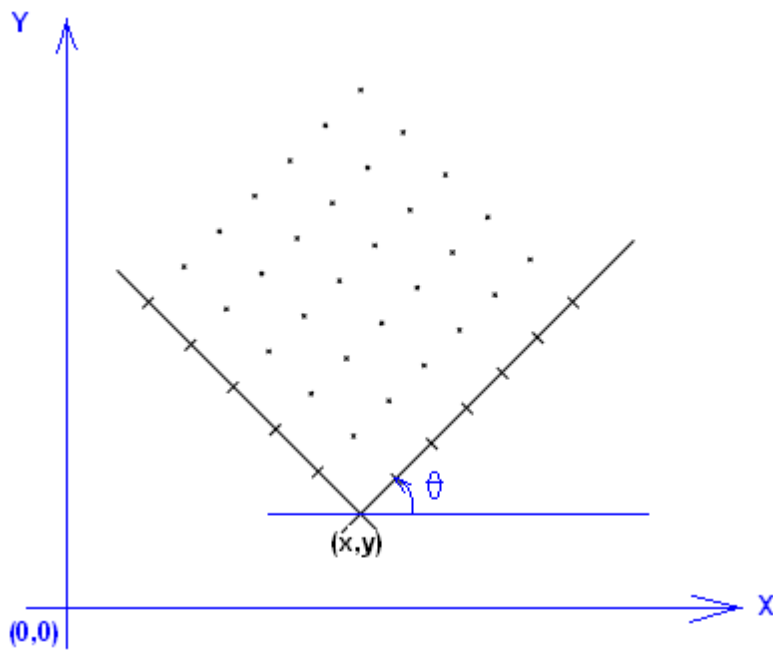
The extent of a grid is given by specifying the number of columns and the spacing of each grid line in the x and y directions.

There is an option to let the program extend the grid (at the same grid spacing and inclination) to find the minimum factor of safety. If this option is used the program will extend the grid (in any direction) if it is found that the centre of the slip surface with the minimum factor of safety is on the edge of the grid. This process is repeated until the minimum centre is no longer on the edge of the grid..

The radius of the slip circle(s) must be specified as one of the following. Selecting one of the options in this field will update the view to show the other relevant fields:

- The co-ordinates of a common point through which all circles must pass.
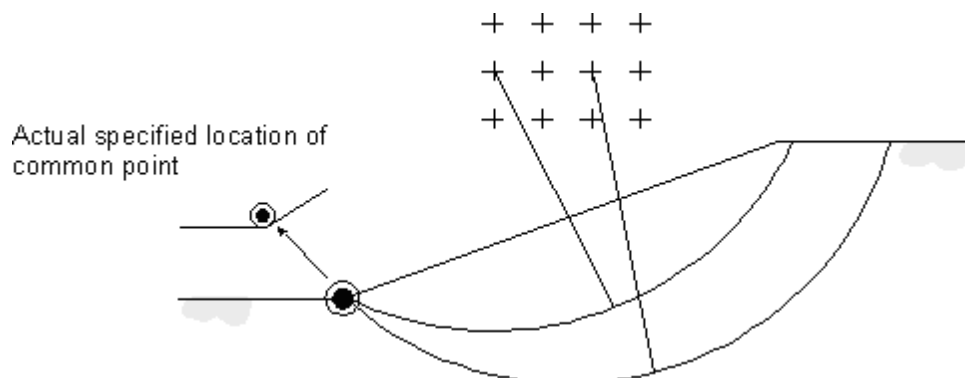
- Defined radii of the circles.For this case user can limit the radius value beyond which slips are not generated. Slips which do not intersect the ground (e.g. A and B in the diagram) will

be ignored.
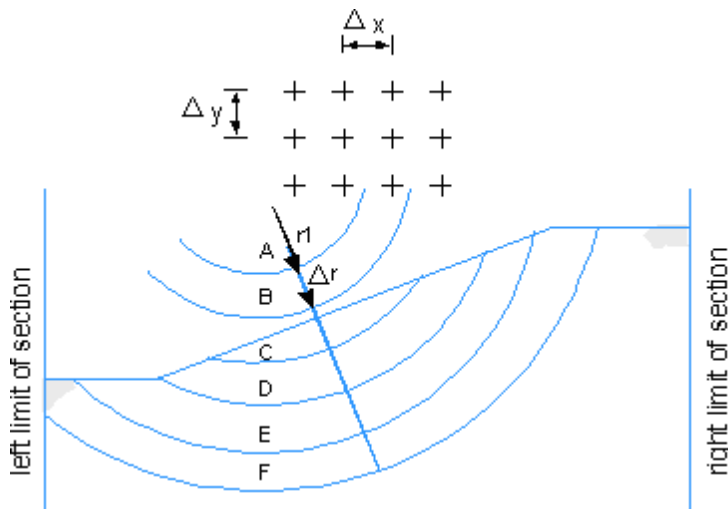
- A tangent surface, defined as a stratum boundary. In this case the circle stays just above the boundary. For sloping strata boundaries, as shown in the diagram, Slope will calculate the shortest radius to the boundary for each centre and take this as the location of the tangent. The calculated circle can therefore never cross the strata boundary line.
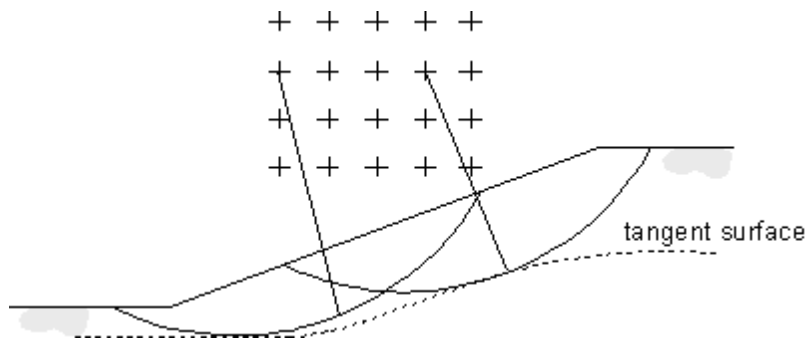
5.3.2   Non-circular Slip Definition

For non-circular slips, the Slip Surfaces command on the Data Explorer or ribbon Home tab opens a tabular view of slip surface coordinates. Multiple non-circular slips can be added by adding new tabs to this view.

Non-circular slips can also be added in the Graphical Input view via the non-circular command in the Create panel on the ribbon. After selecting the command button, left-click to add points, and press ESC to complete drawing. The slip surface will be checked and extended to intersect the ground surface if required.
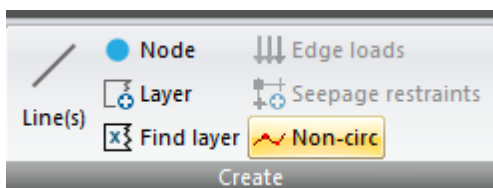
To edit existing non-circular slips in the graphical view, click the non-circular command button in the Select panel on the ribbon, and left-click on the slip surface. The coordinates of the slip surface will be highlighted with circular symbols. Click on a point and drag to move it around to a new position. Again, to complete drawing, press the ESC key and the slip will be checked and extended if required.

Note: for general operation of the Graphical Input view including setting the snap interval, see Graphical Input.

5.4 Surface Loads

The Surface Loads command opens this table either from the ribbon or the Data Explorer. Surface loads can be added by defining the left and right x coordinates of the loaded area.

Load	A	B	C	D	E	F	G
	Limits of Loaded Area		Distributed Load [kN/m hor/m]		Permanent or Variable load	Favourable or Unfavourable	Use in pull-
	Left [m]	Right [m]	Vertical	Horizontal			
Defaults	0.00	0.00	0.00	0.00	Permanent	Unfavourable	No
1	0.00	5.00	15.00	0.00	Variable		Yes
2							

Loads can be horizontal, vertical or inclined.
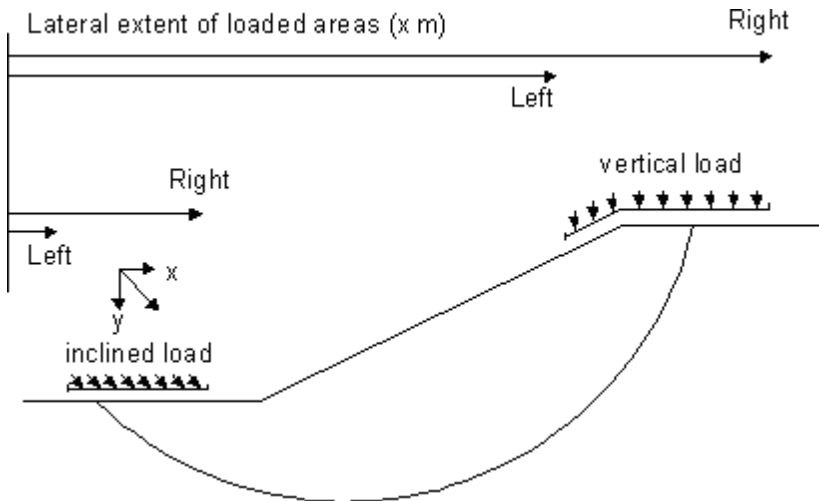
Vertical loads are expressed as the vertical force per unit horizontal width of the loaded area. For level ground this is equal to the normal stress on the ground surface. Vertical loads are positive when they act downwards.

Horizontal loads are expressed as the horizontal force also per unit horizontal width of the loaded area. For level ground this is equal to the shear stress on the ground surface, but for steeply inclined surfaces the 'pressure' specified is much greater than the actual pressure acting on the ground.

Horizontal loads are positive when they act in the direction of increasing x.

Inclined loads are defined by using a combination of the horizontal and vertical components.

Concentrated loads, in the form of anchors or struts, can be modelled by specifying surface loads of high intensity over short lengths of the ground surface.

5.5   Reinforcement

The Reinforcement command opens this dialog from either the ribbon or the Data Explorer. Four types of reinforcement are available:

- Ground anchors
- Rock bolts

- Soil nails
- Geotextile

The data items which are not applicable to each particular type of reinforcement are greyed out when that type is selected from the drop-down list.

Each set of reinforcing elements is given a name which is used to distinguish forces in the reinforcement in the tabular output table. Each set is drawn in a different colour on graphical input and output. NB If the reinforcement is marked inactive in the Specification dialog, it is drawn in grey on the graphical input, and omitted from the output, because it has no effect on the results.

Geometry
The uppermost level, number of layers and horizontal spacing are entered. The length of the top and bottom layers of reinforcement are entered. The lengths of intermediate layers are interpolated between these two values. The angle from horizontal is entered, except for geotextiles which are always assumed to be horizontal.

Capacity
Out-of-plane spacing, tensile capacity and plate capacity (if applicable) are entered. Plate capacity must be at least 50% of tensile capacity. The tensile capacity should represent the allowable capacity if BS8081 is used or ultimate capacity if EC3 is used.

Bond details and prestress
Bond length can be entered for ground anchors and rock bolts Type B. Soil nails are assumed to be 100% bonded along their length.
Bond strength can be specified or calculated from effective stress.
Prestress can be entered for ground anchors and can not exceed the tensile capacity.

Material partial factors
Click the Select button to set material partial factors for each set of reinforcing elements. This is optional - all partial factors will be set to 1.0 if no selection is made.

See Reinforcement Calculations for details of the calculations used to derive the forces and moments from reinforcing elements.

5.6   Importing previous format Slope files

The Open command on the application menu or the Quick Access Toolbar (see Getting Started), opens the File Open dialog.
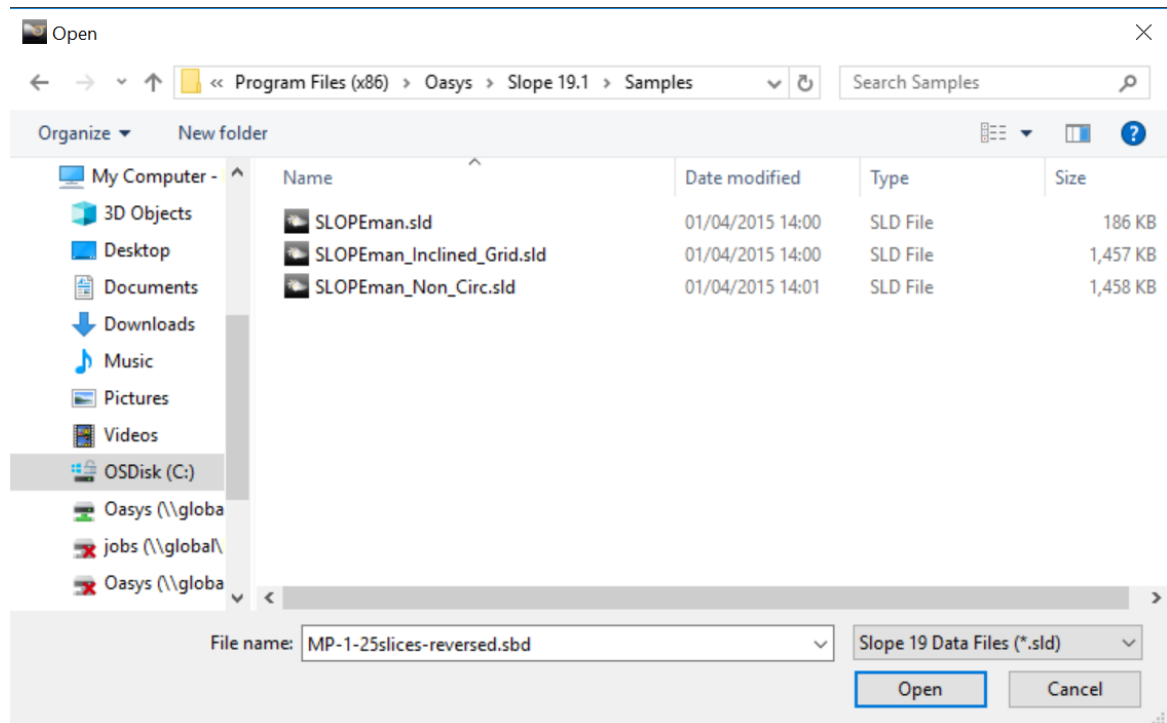
This offers various file types in a dropdown list. To import an existing Slope file, select the "Slope Data Files (*.sld)" option. This will allow you to browse to the required file location

and select the Slope file to be imported.

Another dialog describing the data which will be imported and explaining how to complete it for the seepage analysis is then shown. The base level of the problem is entered in this dialog to provide a lower level for the geometry data generation. (If this level is above any of the existing Slope strata, the program will calculate it's own base level.)
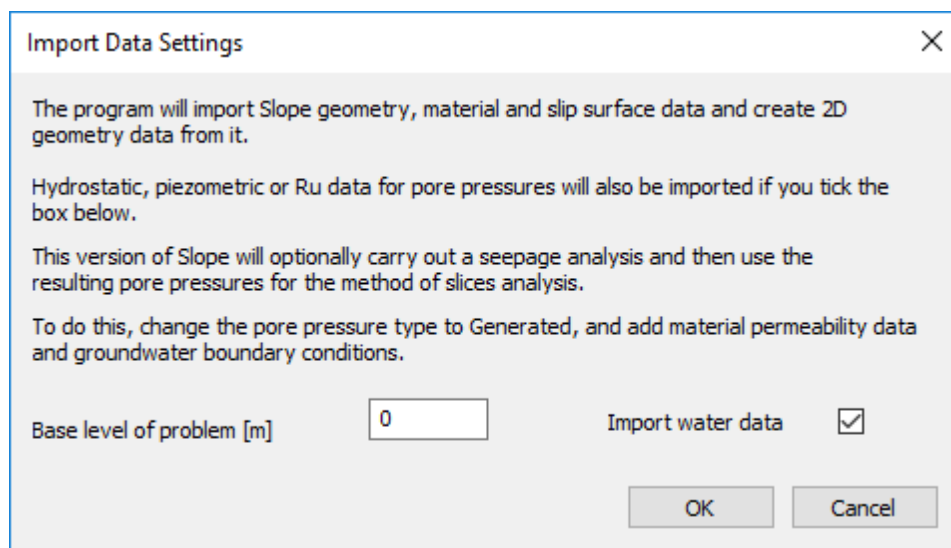
The program generates geometry data for mesh generation using the Slope strata coordinates, and imports material parameters, surface loads, slip surface data and reinforcement data. If the "Import water data" option is ticked, the program will import the water data defined in the Slope file, and set the pore pressure type to Specified. On completion of the import, the Graphical Input view will open and show the generated geometry data. The example below shows a file with a surface load and a grid of slip circle centres.
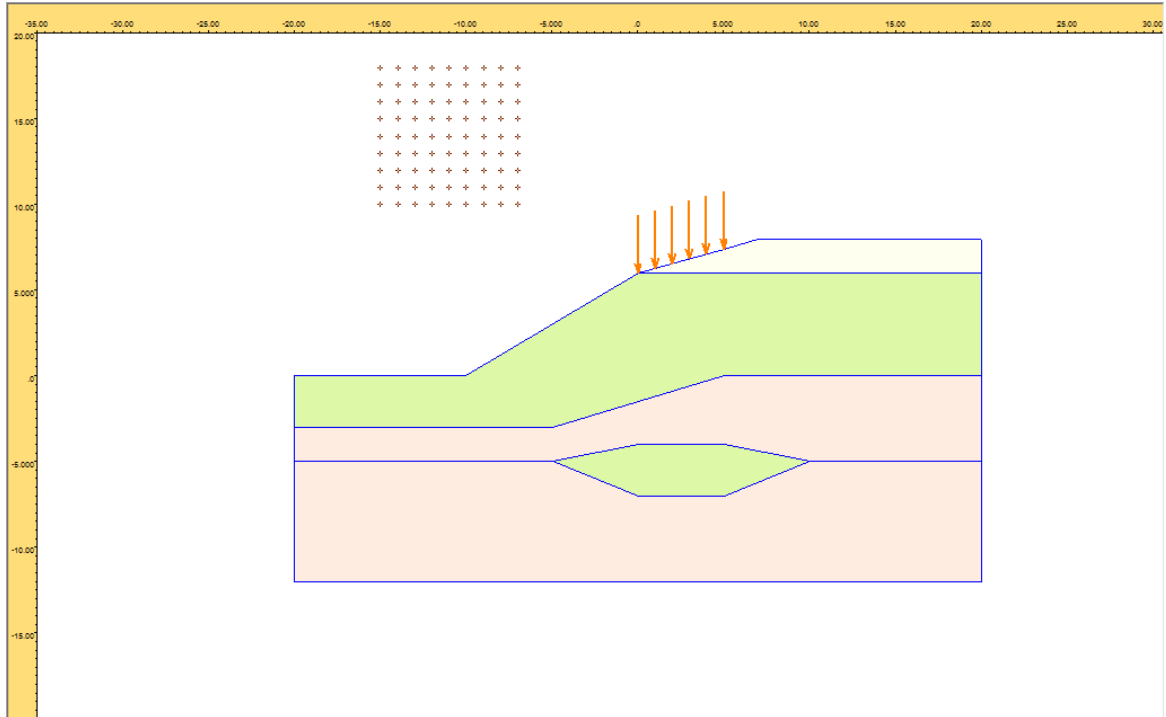
Part VI

6   Analysis and Results

6.1   Analysis and Data Checking
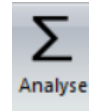
To run the seepage and slope stability analysis, click the Analyse button [Σ Analyse] on the ribbon Home tab.

Prior to analysis the program carries out some data checks and will report, for example, if groundwater boundary conditions are required but not specified.   If no data check errors are found, pressing the Start button will begin the analysis.
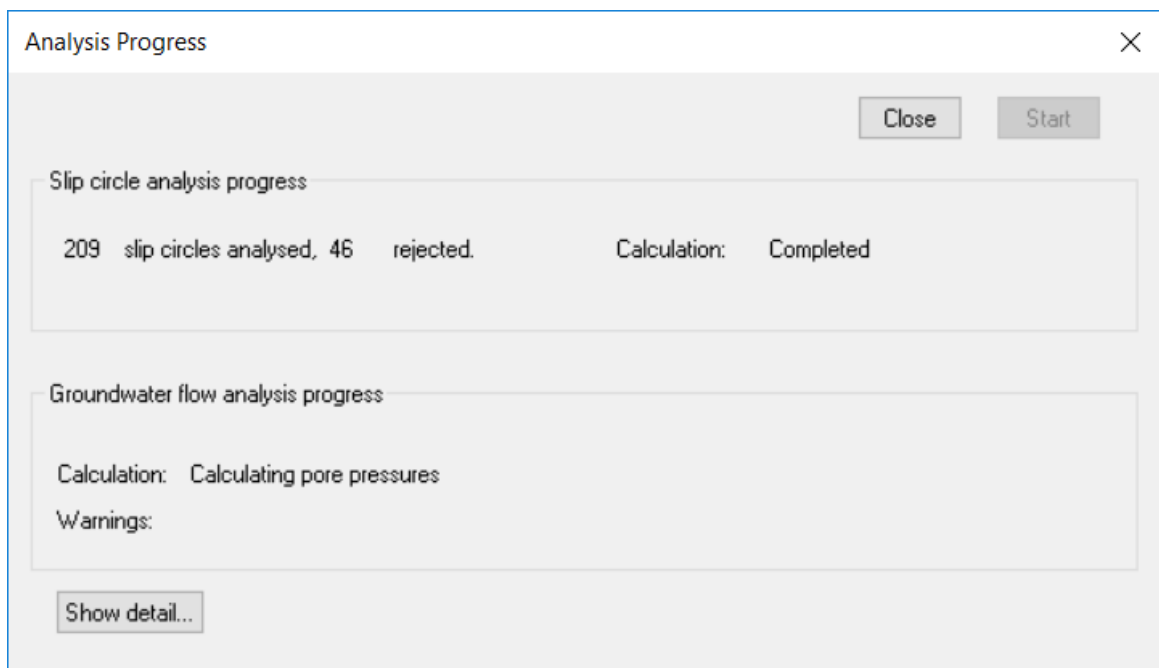
If the program is calculating pore pressures, the groundwater flow analysis is done first, and the resulting pore pressures are used as input to the slope stability analysis. The Analysis Progress dialog reports how many slip circles have been analysed and also if any have been rejected. The reasons for rejecting individual slip circles will be shown on the summary output.

Once the analysis is complete, clicking Close closes the progress dialog. Tabular and Graphical output options will then be available in the gateway and on the ribbon Home tab.

6.2   Tabular Output

Selecting Tabular Output from the gateway or the ribbon Home tab opens a text view showing the data and results. The tabulated output can be highlighted and then copied to the clipboard and pasted into many Windows applications. The output can also be directly

exported to various text or HTML formats by selecting Export from the Application button menu.

Slice No.	Strength Parameters		Average Pore Pressure [kN/m²]	Slice Weight [kN/m]	Forces on base [kN/m]		
	c' [kN/m²]	Tan phi			Normal	Shear (capacity)	Shear (mobilised)
1	0.0	0.5543	0.0	2.275	2.451	1.359	1.450
2	0.0	0.5543	0.0	6.380	6.307	3.496	3.730
3	0.0	0.5543	0.0	9.652	9.277	5.142	5.487
4	0.0	0.5543	0.0	12.10	11.39	6.314	6.737
5	0.0	0.5543	0.0	13.75	12.69	7.035	7.506
6	0.0	0.5543	0.0	14.63	13.24	7.337	7.829
7	0.0	0.5543	0.0	14.76	13.10	7.259	7.745
8	0.0	0.5543	0.0	14.19	12.35	6.844	7.303
9	0.0	0.5543	0.0	12.98	11.08	6.140	6.552
10	0.0	0.5543	0.0	11.18	9.380	5.200	5.548
11	0.0	0.5543	0.0	8.850	7.352	4.075	4.348
12	0.0	0.5543	0.0	4.329	9.340	5.177	5.524
13	2.000	0.3640	0.0	0.4393	2.950	1.846	1.969

The data is printed, followed by a summary of the results for all slip circles analysed and a full report of the results for the worst case slip circle with the lowest factor of safety.

More:

[Summary of Results](#)

[Full Results](#)

6.2.1   Summary of Results

This output summarises the results for all the slips analysed. The following items are tabulated for each slip circle.

- The x and y co-ordinates of the centre of rotation about which moments are taken.
- The radius of the circle for circular slips.
- The slip weight.
- The Factor of Safety or Over-Design Factor.
- The disturbing and restoring force or moment of the slip.

A column for comments provides the following information for circles which were rejected for analysis:

Comment	Suggested action
Radius too large	Increase lateral extent of ground profile in ±X direction (if required); but note that this message will always be shown where the initial radius and increment method is used to define the circles to be

analysed, unless a limiting radius is also defined.

Horizontal ground	Where the location of the slip surface is entirely within an area of horizontal ground.

Top of circle

Centre of circle

Radius too small.	Occurs if the circle radius is too small to reach the specified ground surface.
Center embedded	Where the center of the circle is below the level of the top of the slip.
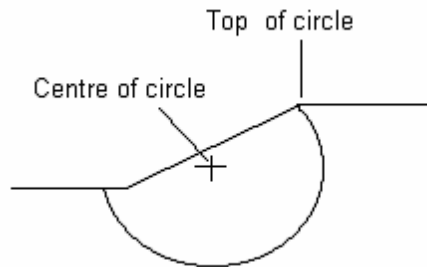
Weight too small.	Decrease the minimum slip weight.
Failed to converge.	Increase the maximum number of iterations.
Analysis Error.	Other calculation errors. Check input data.

6.2.2   Full Results

Detailed results are provided for the slip circle with the lowest factor of safety.
The output provides details of the interslice and base forces in addition to the overall reporting of force and moment equilibrium.

```
WORST CASE
Centre at (-16.000m,24.000m)      Radius 24.515m
Iterations: 88                    Horiz acceleration [%g]: 0.0
Net vertical force [kN/m]: -0.19692  Slip weight [kN/m] 125.52
Net horiz force [kN/m]: -0.34472     Disturbing moment [kN/m]: 1758.4
                                     Restoring moment [kNm/m]: 1648.0
                                     Reinf.Rest.Moment [kNm/m]: 0.0
                                     Factor of Safety: 0.93722

The system of interslice and base forces are in equilibrium
when the strengths available at the bases are divided by the computed
factor of safety.  The interslice forces shown in the following table
are in equilibrium with the factored strengths of the soil at the bases of slices.

Slip surface coordinates Pore Pressure   Interslice forces [kN/m]
Point x [m]    y [m]     L        R       T       E           E(u)
                      [kN/m²]  [kN/m²]
    1  -9.3063   0.41623 -        -4.5681    0.0         0.0   0.0
    2  -8.3801   0.69898 -6.7507  -6.7507 0.49650     0.67177  0.0
    3  -7.4659   1.0181  -9.3051  -9.3051 1.3049      2.1161   0.0
    4  -6.5649   1.3730  -12.227  -12.227 2.2986      3.8210   0.0
    5  -5.6787   1.7633  -15.513  -15.513 3.3374      5.3961   0.0
    6  -4.8086   2.1883  -19.157  -19.157 4.2854      6.5715   0.0
    7  -3.9559   2.6473  -23.153  -23.153 5.0280      7.1908   0.0
    8  -3.1220   3.1396  -27.495  -27.495 5.4857      7.2027   0.0
    9  -2.3083   3.6644  -32.177  -32.177 5.6267      6.6473   0.0
   10  -1.5158   4.2210  -37.191  -37.191 5.4751      5.6417   0.0
   11  -0.74602  4.8084  -42.530  -42.530 5.1159      4.3620   0.0
   12   0.0      5.4258  -48.184  -48.184 4.6971      3.0253   0.0
   13   0.64332  6.0000  -53.478  -53.478 1.3584      0.92847  0.0
   14   0.92453  6.2642  -55.924  -        0.19692    0.34472  0.0
```

Full output comprises the following:

Method of Analysis :	Number of iterations.
	Horizontal Acceleration (%g).
Location of slip surface :	x and y co-ordinates of the centre of rotation about which moment is taken.
	Radius(for circular slips).
Overall Results :	Net vertical force
Includes net vertical and horizontal forces to help provide some idea of the possible error in the calculated factor of safety.	Net horizontal force
	Slip weight
	Disturbing moment or force
	Restoring moment or force
	Factor of Safety or Over-Design Factor
Slip Surface Location :	Pore water pressure u
x and y co-ordinates (m, m OD) of	Interslice Forces

the base of the LEFT side of each slice are used to define the location of the slip surface.

Vertical Shear T

Horizontal Normal E

Horizontal Water Pressure E(u)

Slices :   Strength Parameters

Slices are numbered from left to right.

Cohesion c' tan ? ' (degrees)

Pore Pressure

Slice weight

Forces on the base

Normal N

Shear S

General Slice Information :   Surface Loads - vertical and horizontal

Water pressure on ground surface - Vertical and horizontal

6.3   Graphical Output

The ribbon command or the Gateway item "Graphical output" open the graphical output view. The program defaults to show all slip circles with a factor of safety below 3, or, if there are no circles with FoS less than 3, any with less than 10. The slip circles will be coloured in accordance with their factor of safety.

To change the currently plotted circle when a grid of centres has been analysed, move the cursor into the grid and right-click on the required centre. The circle with the lowest factor of safety for that centre will be plotted.
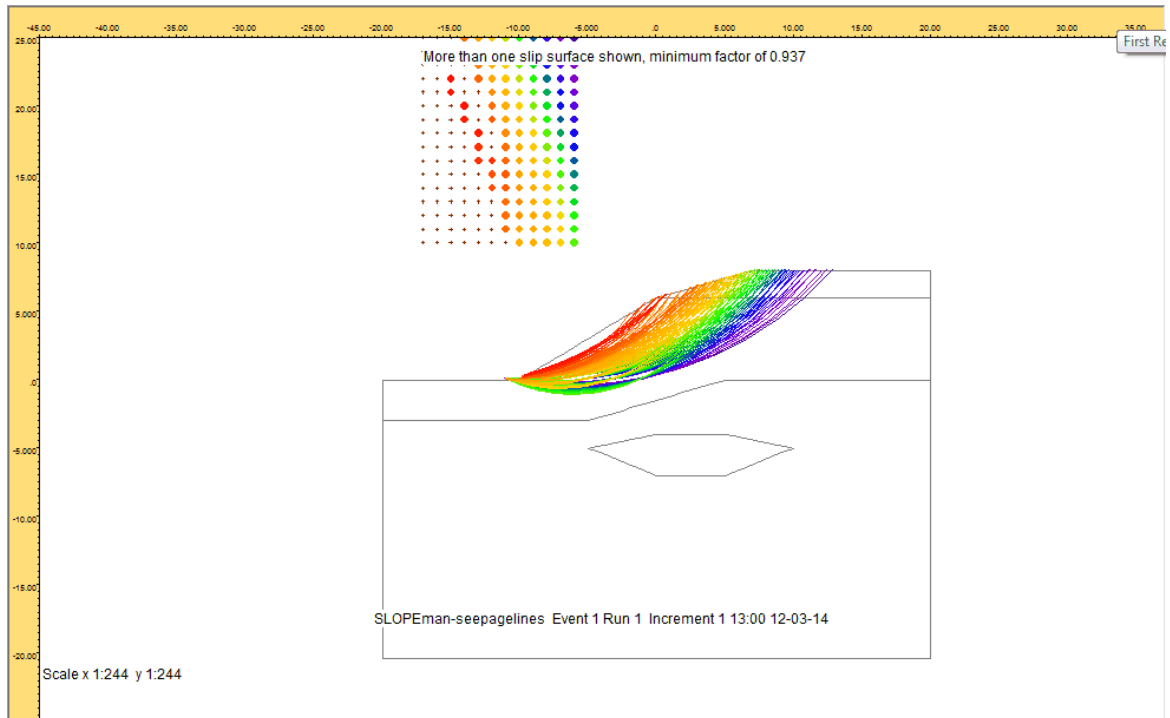
A number of other output options and commands are available on the Graphical Output tab:
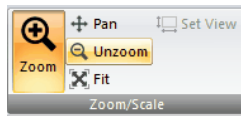
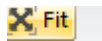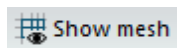**Zoom** : Select an area to 'zoom in' to by using the mouse to click on a point on the drawing and then dragging the box outwards to select the area to be viewed. The program will automatically scale the new view. The original area can be restored by clicking on the Unzoom' button.

Fit : re-size the view to fit in whole problem area.

Show mesh : Adds the elements to the graphical view.
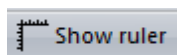
Show ruler : toggles between showing a ruler at the edges of the view, or the view contained within a rectangular border.

Slope Stability Results
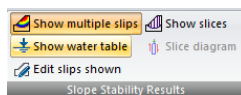
Show multiple slips : toggles between all the circles and only the worst case circle.

Show water table: clicking this button will toggle display of the hydrostatic pore pressure line (zero pore pressure).

Edit slips shown : opens a dialog to allow input of a minimum and maximum factor of safety to be plotted. Other circles will be filtered out and the graphics legend updated. See Edit Slips Shown for more details.

Show slices : draws the slices within the slipping mass of soil. If only one circle is plotted, the **Slice diagram** command then becomes available, and clicking it followed by any slice on the view opens a separate window showing the slice force polygon.

Seepage Results

If pore pressures were generated via a seepage analysis, the Parameter dropdown list allows selection from pore pressure, piezometric head, nodal flow, seepage velocity or hydraulic gradient. These parameters can be plotted as line contours, filled contours or numbers.

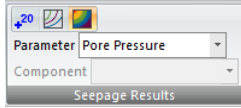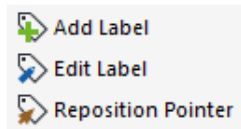**Labeling Functionality**

Add Label : This allows entering of text labels on the view. To add a label, click the Add Label button, enter the details, then click at the required label position.

Edit Label: To edit or delete an existing label, click the Edit Label button, right-click near the label, and edit/delete the label as required.To move an existing label left-click near the label, and drag the label as required.

Reposition Pointer : To reposition an existing pointer, left click the Reposition Pointer button and move the head of the pointer to reposition it.

6.4   Edit Slips Shown

If more than one slip surface is being plotted on the graphical output, a legend showing colour intervals corresponding to the plotted range of factors of safety will be shown. If there are many analysed circles, by default only those with factors of safety less than 3 will be shown, or (for example) up to 10 if there are no circles with FoS less than 3.   To amend this, or for more detail within a specific range of factors of safety, select the "Edit slips shown" button on the Graphical Output tab. A dialog box will be shown which allows the minimum and maximum factor of safety to be edited.

The contour interval used in plotting contours of factors of safety is also editable from this dialog. When any limited range is plotted, a note will be added to the graphical output to indicate that not all the available results are being shown (see example below). The full range can be re-displayed by clicking the Reset button on the Edit Graphics Settings dialog.

Factors of Safety

2.8 - 3.0

2.6 - 2.8

2.4 - 2.6

2.2 - 2.4

2.0 - 2.2

1.8 - 2.0

1.6 - 1.8

1.4 - 1.6

1.2 - 1.4

<1.2
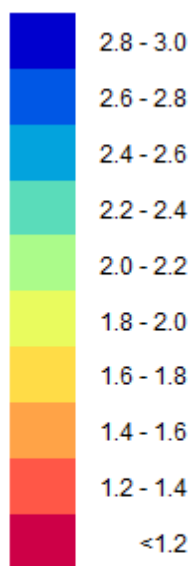
NOTE:
Plotted range is
min FoS to 3.0

Part VII

7   Programming Interface

Slope now includes a COM (Component Object Model) interface. This enables external programs to pass information and instructions to and from the program, via a "data object". COM objects can be used by many other programs, including Excel, MATLAB and Python. Slope 21.0 has a lot of code in common with the Oasys Geotechnical finite element program Safe, so the COM interface library is shared between them, and the COM object name and references are based on Safe.
Excel(VBA) and Python sample files to demonstrate the use of the COM API is provided with the program. They are installed in the folder C:\Program Files\Oasys\Slope\Samples.

Using the COM interface in Excel

Click the Developer tab and click on the "Visual Basic" button to open the VBA window. In the VBA window, click on "Tools" in the menu bar and select "References". Check the list for the entry "safeLib" and make sure it is ticked.
If the type library is not listed, add it as a new reference. The file is "safe.tlb" and is available in the program installation folder.

Using the COM interface in Python

In a Python script, or the interactive window, create the COM object as follows:
Import win32com.client
slp = win32com.client.Dispatch("safeLib.SafeAuto")
The first line imports the Python library which supports COM. The second line creates the COM object and associates it with the published COM interface, which has the name SafeAuto, contained in the type library safeLib.

7.1   COM function listing

The following functions are available in the COM interface. Note that the first function that should be used is 'Open'. Functions return either a number value (e.g. node level or displacement) or an integer return code so that success or failure can be tested if required.

The Analyse function needs to be run before using any of the functions which retrieve results, to ensure that there are results available.

The units of the various parameters are as defined in the current data file. The default unit system is kN for force, metres for length and millimetres for displacement.

Show

This function has no input/output variables. It opens Slope and shows the main program window.

Open(String sPathName) - returns an integer

This function opens a saved file. sPathName is a string giving the file path. If the file is opened successfully the function will return 0, or if it was unsuccessful it will return -1.

Analyse(Integer iStage) – returns an integer

This is a function instructing Slope to analyse the currently open file, up to the stage number iStage.

Save() – returns an integer

Saves the currently open file.

SaveAs(String sPathName) – returns an integer

Saves the currently open file with a different name.

DeleteResults() – returns an integer

Deletes the existing results.

Close – returns an integer

Closes the current file.

Material functions

GetNumMaterial() – returns the number of materials as an integer

CreateMaterial(String name) – creates a blank material record with the name specified

SetUnitWtAbove(integer imat, float value) – sets material "imat"'s unit weight above the water table to the value specified

SetUnitWtBelow(integer imat, float value) – sets material "imat"'s unit weight below the water table to the value specified

SetDrainageType(integer imat, integer type) – sets the material imat to be undrained, drained with linear strength, drained with a power curve or hyperbolic strength relationship (values 0 to 3 respectively)

SetPhi(integer imat, float value) – sets the friction angle of the material

SetCohesion(integer imat, float value) – sets the cohesion value for the material

SetCohesionRefLevel(integer imat, float value) – sets the level at which the cohesion value applies. This need not be specified unless the material has varying cohesion with depth.

SetCohesionGradient(integer imat, float value) – sets the gradient of cohesion. Need not be specified if the material has constant cohesion.

SetPowerCurveParams(integer imat, float value1, float value2) – sets the values of the two parameters for use in drained materials with a power curve relationship (see [Material Properties](#) for details)

SetCupRatio(integer imat, float value) – sets the Cu/p ratio for an undrained material. If this is set as well as the cohesion gradient, the lower of the two values will be used.

GetUnitWtAbove(integer imat) – returns the unit weight above the water table

GetUnitWtBelow(integer imat) – returns the unit weight below the water table

GetDrainageType(integer imat) – returns the material drainage setting

GetPhi(integer imat) – returns the friction angle of the material

GetCohesion(integer imat) – returns the cohesion value for the material

GetCohesionRefLevel(integer imat) – returns the level at which the cohesion value applies

GetCohesionGradient(integer imat) – returns the gradient of cohesion.

GetPowerCurveParamA(integer imat) – returns the first parameter for use in drained materials with a power curve relationship (see manual for details)

GetPowerCurveParamB(integer imat) – returns the second parameter for use in drained materials with a power curve relationship (see manual for details)

GetCupRatio(integer imat) – returns the Cu/p ratio for an undrained material

Geometry functions

DefineCutPlane(float xmin, float ymin, float xmax, float ymax) – define a vertical plane from (xmin, ymin) to (xmax, ymax) along which to cut a 3D surface. The 3D surface can be imported from a TIN surface contained in an XML file.

SetStraightnessTol(float tol) – sets a straightness tolerance, in degrees. Two adjacent lines on a polyline will be treated as one line (middle point will be omitted) if their angles are within this tolerance of each other, i.e. if the lines are near-parallel.

ImportXml(String path) – reads the specified XML file, imports the 3D data contained in it, and generates polylines where the 3D surface cuts the previously specified cutting plane

SetStratum(integer is, string name, string matname) – creates a "blank" stratum (no points yet) with the specified name. The integer should be the stratum number. The first stratum should have the number 1.

SetStratumPoint(integer is, integer ip, float xp, float yp) – set point 'ip' in stratum 'is' to the value (xp, yp)

NumStrata() – returns the number of strata as an integer

GetStratumPointX(integer is, integer ip) - returns the X coordinate of point 'ip' in stratum 'is' (zero-based point counter)

GetStratumPointY(integer is, integer ip) - returns the Y coordinate of point 'ip' in stratum 'is' (zero-based point counter)

Create2DSection() - creates a 2D section from the existing strata information

NumNodes() - returns the number of "nodes" (vertices of the soil layer polygons)

NodeX(integer inode) - returns the X coordinate of node 'inode' (1-based node counter)

NodeY(integer inode) - returns the Y coordinate of node 'inode' (1-based node counter)

SetNodeX(integer inode, float xnode) - sets the X coordinate of node inode. Caution required not to set invalid data.

SetNodeY(integer inode, float ynode) - sets the Y coordinate of node inode.

Water data functions

SetWaterTable(integer iref, float dX1, float fY1, float dX2, float dY2) - sets water table "iref" to have 2 points at (dX1, dY1) and (dX2, dY2). "iref" is a 1-based counter. If "iref" equals zero or is greater than the current number of water tables, a new one will be added.

SetWaterPoint(integer iref, float dX, float dY) - inserts the point (dX, dY) in x-coordinate order into the water table "iref" (1-based counter)

SetStratumWater(integer istratum, integer iref) - sets stratum "istratum" to have the water table "iref"

Slip circle functions

SetSingleOrGrid(integer i) – set 0 for a single circle or 1 for a grid of circle centres

SetPosition(float x, float y) – Coordinates of lower left corner of grid

SetNumXCentresAndSpacing(integer nx, float dx) – set the grid to have nx centres in the x direction at dx distance apart

SetNumYCentresAndSpacing(integer ny, float dy) – set the grid to have ny centres in the y direction at dy distance apart

SetGridAngle(float angle) – set the grid to an angle of inclination (positive in the anticlockwise direction from the bottom left of the grid)

SetRadSpec(integer itype) – which radius option - set

SetCommonPoint(float x, float y) – set the coordinates of the common point through which all slip circles will pass

SetRadius(float rad) – set the initial radius

SetRadInc(float rinc) – set the radius increment. Slip circles will be increased from the initial radius by this increment, until the maximum possible radius has been analysed

Results functions

CriticalCircleRef() – returns an integer corresponding to the position, in the results set, of the circle with the lowest factor of safety

GetMinFoS(integer iref) – returns the factor of safety. Iref should be the previously obtained critical circle reference.

GetCriticalCentreX(integer iref) – returns the x coordinate of the circle centre iref

GetCriticalCentreY(integer iref) – returns the y coordinate of the circle centre iref

GetCriticalCircleRadius(integer iref) – returns the radius of the slip circle iref

Load functions

NumLoads() - returns the number of loads

GetXminLoad(integer iload) - returns the minimum X coordinate of load 'iload' (1-based load counter)

GetXmaxLoad(integer iload) - returns the maximum X coordinate of load 'iload'

SetXminLoad(integer iload, float xmin) - sets the minimum X coordinate of load 'iload'

SetXmaxLoad(integer iload, float xmax) - sets the maximum X coordinate of load 'iload'

SetVertLoadIntensity(integer iload, float vP) - sets the vertical load intensity of load 'iload' to 'vP'

Reinforcement functions

SetReinfActive(bool b) - set reinforcement active if b is non-zero, or inactive if b is zero

Part VIII

8 References

BSI (2010 / 2011). Code of practice for strengthened/reinforced soils and other fills. BS 8006-1:2010 and BS 8006-2:2011.

Bishop A W (1955). The use of the Slip Circle in the Stability Analysis of Earth Slopes. Géotechnique Vol.5 No.1 pp 7-17.

Fredlund D G and Krahn J (1977). Comparison of slope stability methods of analysis. Canadian Geotechnical Journal Vol 14 p.429-439.

Janbu N (1957). Earth Pressures and Bearing Capacity Calculations by Generalized Procedure of Slices. Proc. 4th International Conference Soil Mech. Fdn. Engng. Vol.2 pp 207-212.

Morgenstern N R and Price V E (1965) The Analysis of the Stability of General Slip Surfaces. Geotechnique, Vol. 15, pp. 79-93.

Nash D (1987). A comparative review of limit equilibrium methods of stability analysis in slope stability. Anderson and Richards (eds), John Wiley & Sons.

Spencer E (1967). A Method of Analysis of Embankments ensuring Parallel Interslice Forces. Géotechnique Vol.77 pp 11.26.

Whitman R V, and Bailey W A (1967). Use of Computers for Slope Stability Analysis. International Soil Mech. Fdns. Div. Am. Soc, Civ. Engrs Vol.93 SM4 pp 475 - 498.

Zhu D Y et al (2005) A concise algorithm for computing the factor of safety using the Morgenstern-Price method. Canadian Geotechnical Journal Vol. 42 p.272-278.

Endnotes 2... (after index)

Back Cover