

Oasys

YOUR IDEAS BROUGHT TO LIFE



Slope Analysis Tutorial Manual



Oasys Slope

Copyright © Oasys 2020

All rights reserved. No parts of this work may be reproduced in any form or by any means - graphic, electronic, or mechanical, including photocopying, recording, taping, or information storage and retrieval systems - without the written permission of the publisher.

Products that are referred to in this document may be either trademarks and/or registered trademarks of the respective owners. The publisher and the author make no claim to these trademarks.

While every precaution has been taken in the preparation of this document, the publisher and the author assume no responsibility for errors or omissions, or for damages resulting from the use of information contained in this document or from the use of programs and source code that may accompany it. In no event shall the publisher and the author be liable for any loss of profit or any other commercial damage caused or alleged to have been caused directly or indirectly by this document.

This document has been created to provide a guide for the use of the software. It does not provide engineering advice, nor is it a substitute for the use of standard references. The user is deemed to be conversant with standard engineering terms and codes of practice. It is the users responsibility to validate the program for the proposed design use and to select suitable input data.

Printed: September 2020

Contents

1. Introduction	1
2. Slope analysis methods	2
3. Slope User Interface	3
4. Simple Slope Analysis.....	3
4.1 Geometry and Input Values.....	4
4.2 Creating the Input	5
4.3 Viewing the Results	14
4.4 Test Yourself	19
5. Partial Factor Analysis and Reinforcement Design	20
5.1 Partial Factor Analysis	21
5.2 Reinforcement Design.....	23
6. Customised Input and Output.....	27
6.1 Importing DXF Drawing	28
6.2 Limiting the Minimum Slip Circle Size	30
7. Generating Pore Pressures.....	31

1. Introduction

Slope is a program which is used for analysing the stability of slopes. The program is also applicable to earth pressure and bearing capacity problems, as well as to rock slopes and waste heaps.

The program offers a variety of established methods for calculating interslice forces. For circular slips, the Bishop method with either horizontal or constant inclined interslice forces and the Morgenstern-Price method with a constant or half-sine interslice force function are available. Slope also includes a variant of the Bishop method with variably inclined interslice forces which ensures that each slice is in equilibrium, both vertically and horizontally. For non-circular slip surfaces, the equivalent Janbu methods and the Morgenstern-Price method can be used. Slope allows the user to apply partial factors, including Eurocode 7, to run Ultimate Limit State (ULS) analysis.

The software analyses reinforcing elements such as soil nails, rock bolts, ground anchors and geotextiles to BS8006/BS8081. The user can specify water pressure, material shear strength, surface loads and horizontal ground acceleration. Slope checks a range of slip circles and gives the user the option to force slips through a point or keep them tangential to a strata. Note that the Janbu method is not suitable for analysing reinforced slopes.

The calculation itself is based on verified and robust numerical methods. The simplicity of calculation enables new users to set up and run a slope stability problem within hours of training.

The tutorial examples deal with simple slope stability analyses, intended to familiarise the user with Slope. The examples should therefore not be used as a basis for practical projects.

Users are expected to have an understanding of soil mechanics and geotechnical theory, and should be able to work in a Windows environment. The tutorial lessons are also available in the examples folder and can be used to check your results.

It is important to realise that Slope is a specialist program analysing a complex problem and the user must be fully aware of the various methods of analysis, requirements and limitations discussed in the User Manual before use. The Tutorial Manual will not provide theoretical background information on the analysis method, nor does it explain the details of various methods of analysis available in the program. These details can be found in the User Manual for Slope. This also contains detailed information on the available program features.

2. Slope analysis methods

Slope offers the following analysis methods:

- Bishop's methods (for circular slip surfaces)
- Janbu's methods (for non-circular slip surfaces)
- Morgenstern-Price's method (for either circular or non-circular slips)

All these methods of analysis use the method of slices to determine the factors of safety for slope stability. The detailed derivation for each solution is given in the User Manual.

The use of these methods allows analysis of both circular and non-circular slip surfaces to be carried out. The location of circular surfaces is defined using a rectangular grid of centres and then a number of different radii, a common point through which all circles must pass or a tangential surface which the circle almost touches. Non-circular slip surfaces are defined individually as a series of x and y coordinates.

The **ground section** is built up by specifying each layer of material as a closed polygon. The polygon data may be directly drawn or entered into Slope, or imported. Import options are from dxf files, from which Slope will read polylines to specify the upper surface of soil layers, or from LandXML files containing a "triangulated irregular network" or TIN surface. To generate a 2D soil layer from the TIN surface, the user specifies the vertical plane through which to cut the TIN surface.

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.

The **ground water profile** and pore water pressure distribution can be set individually for each soil stratum, using either:

- A phreatic surface with hydrostatic pore pressure distribution.
- A phreatic surface with a user-defined "piezometric" pore pressure distribution.
- An overall value of the pore pressure coefficient R_u .

A maximum soil suction can also be specified for each stratum.

Alternatively, **generated pore pressures** from a steady state finite element analysis can be used for the slope stability calculation. If this option is selected, material permeabilities and groundwater boundary conditions must be specified. The program will then run a steady state seepage analysis and use the resulting pore pressures as input to the slope stability analysis. This option is described in more detail in the User Manual.

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 BS8006-2:2010.

Slopes which are submerged or partially submerged can be analysed.

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

Horizontal acceleration of the slip mass can be included to represent **earthquake loading** .

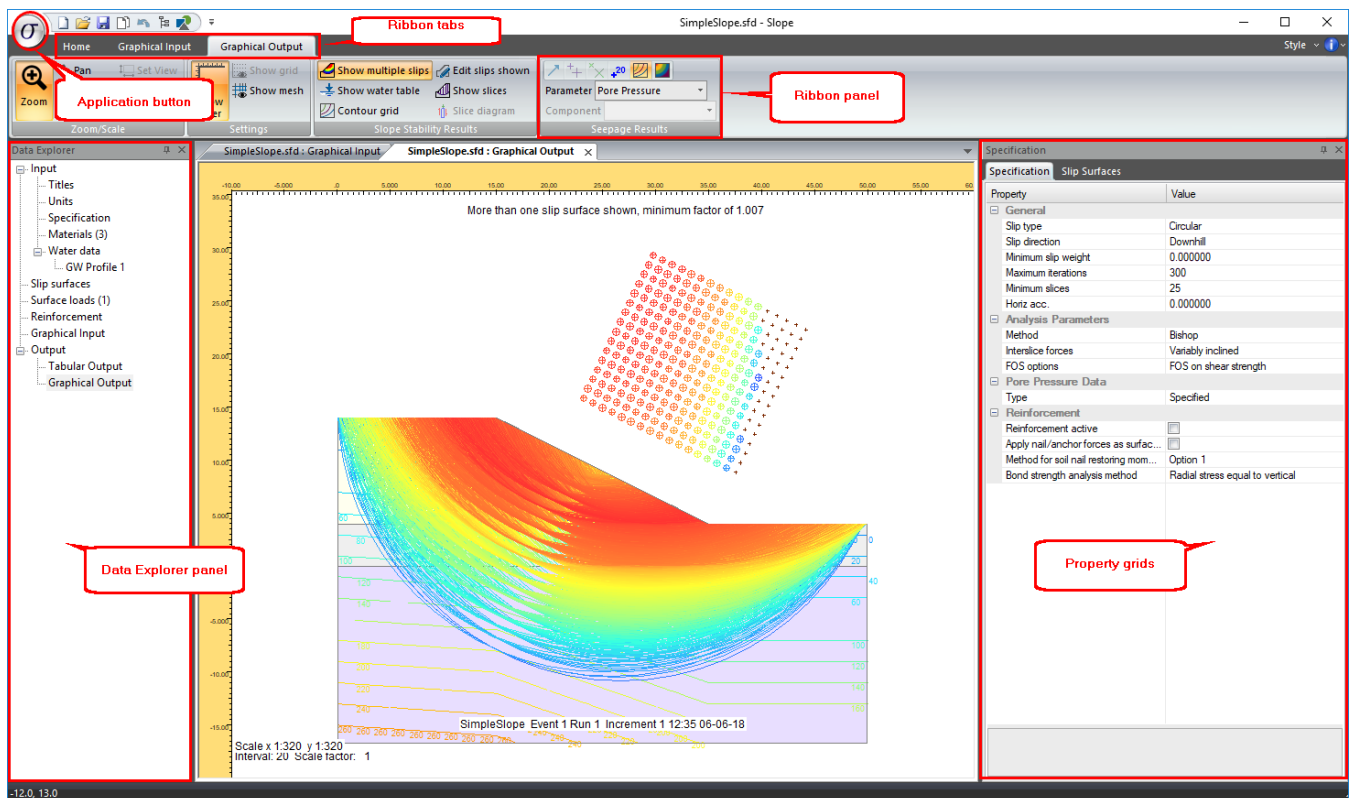
The calculated factor of safety can be applied to soil strength or the magnitude of the applied loads, by either

- causing failure - to represent bearing capacity problems, or
- preventing failure - for anchor forces.

Over-Design Factor can be calculated for an ultimate limit state factor set using Partial Factor Analysis.

3. Slope User Interface

The principal components of the user interface are illustrated below and these will be referred to in the tutorial manual.



4. Simple Slope Analysis

To cover the features available, the stability of a slope with varied stratigraphy and water table and a vertical load will be covered in this tutorial. This serves as a quick introduction to the Slope program.

Objectives

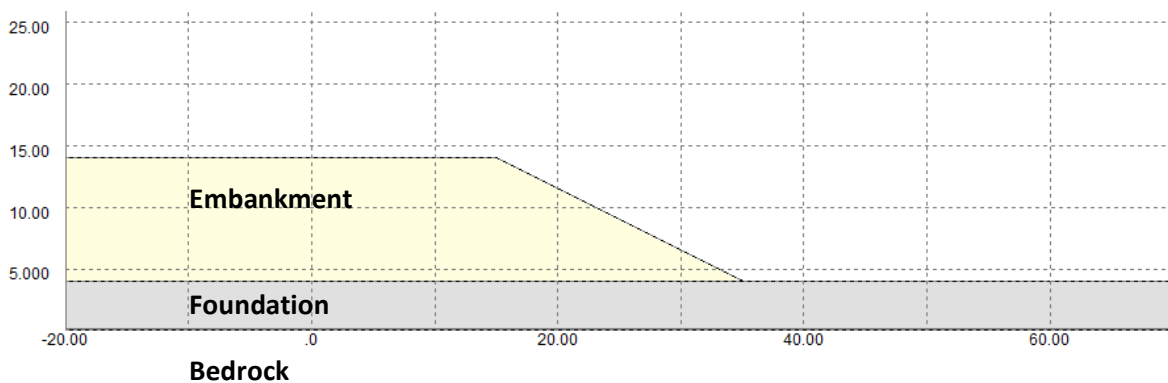
- Start a new project

- Input material properties using the table view
- Use the *Graphical Input* interface to create the soil stratigraphy
- Apply water data and surcharges
- Specify a range of slip circles to analyse
- View calculation results
- Export tabular outputs
- Print and adapt graphical outputs

4.1 Geometry and Input Values

For a design problem, the site investigation data and testing should be used to determine the stratigraphy data and the material properties.

For this slope stability analysis, the following geometry will need to be plotted.



Material Properties

Description	Unit Weight (kN/m ²)		Condition	Phi (deg)	Cohesion c' (kN/m ²)
	Above GWL	Below GWL			
Embankment Soil	15	19.5	Drained - linear strength	20	5
Foundation Soil	18	21.5	Drained - linear strength	25	10
Bedrock	22	24	Drained - linear strength	40	15

Strata Co-ordinates

Embankment

x	y
0	14
15	14
35	4
50	4

Foundation

x	y
0	4
35	4
50	4

Bedrock

x	y
0	0
50	0

Groundwater Profiles

Original

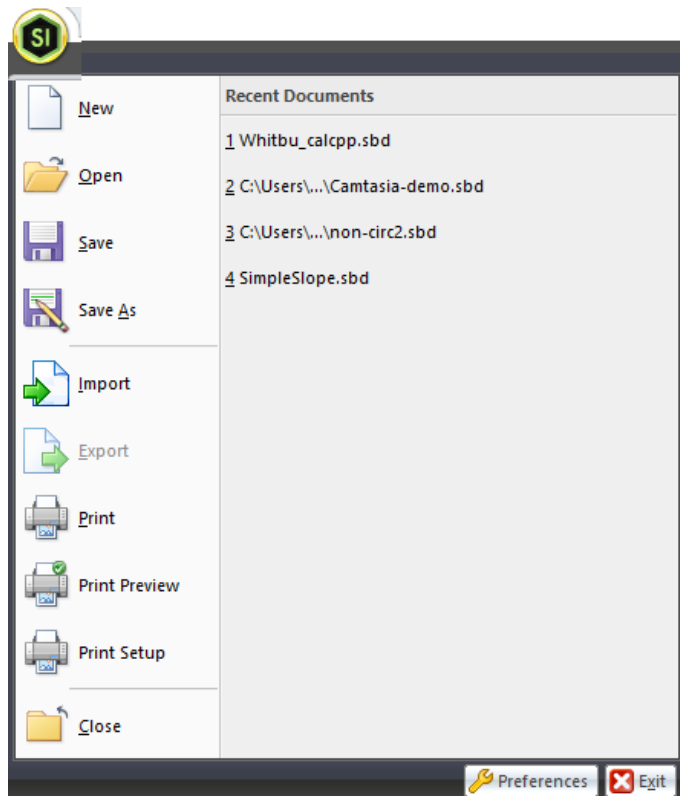
x	y
0	11
15	10
35	3
50	3

Raised

x	y
0	14
15	14
35	4
50	4

4.2 Creating the Input

1. Open the Oasys Slope Program.
2. Select *New file* on the Quick Access Toolbar at the top of the screen, or *New* from the Application menu.



3. A new untitled file will be opened. Double click on *Titles* in the Data Explorer, or click *Titles* on the ribbon, to open the **Titles** dialog and enter identification details for the analysis. Choosing **Units** allows you to change the units from the standard SI option.

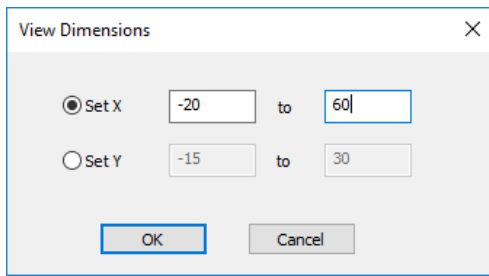
4. Double click on *Materials* in the Data Explorer, or click *Materials* on the ribbon, to open the **Materials** table. Enter the material parameters given in Section 4.1. Choose *Save* from the Quick Access Toolbar or the application menu and save the file.

5. Click the *Graphical Input* view tab. The ribbon tab should change to the **Graphical Input** tab.

6. Change the grid interval in the ribbon to 1.0m, if it is not showing that value already. The selection is made via a dropdown list.

7. Click on the **Line** button on the *Create* panel. If you hover over the button, a tool tip describing how to use it will appear.

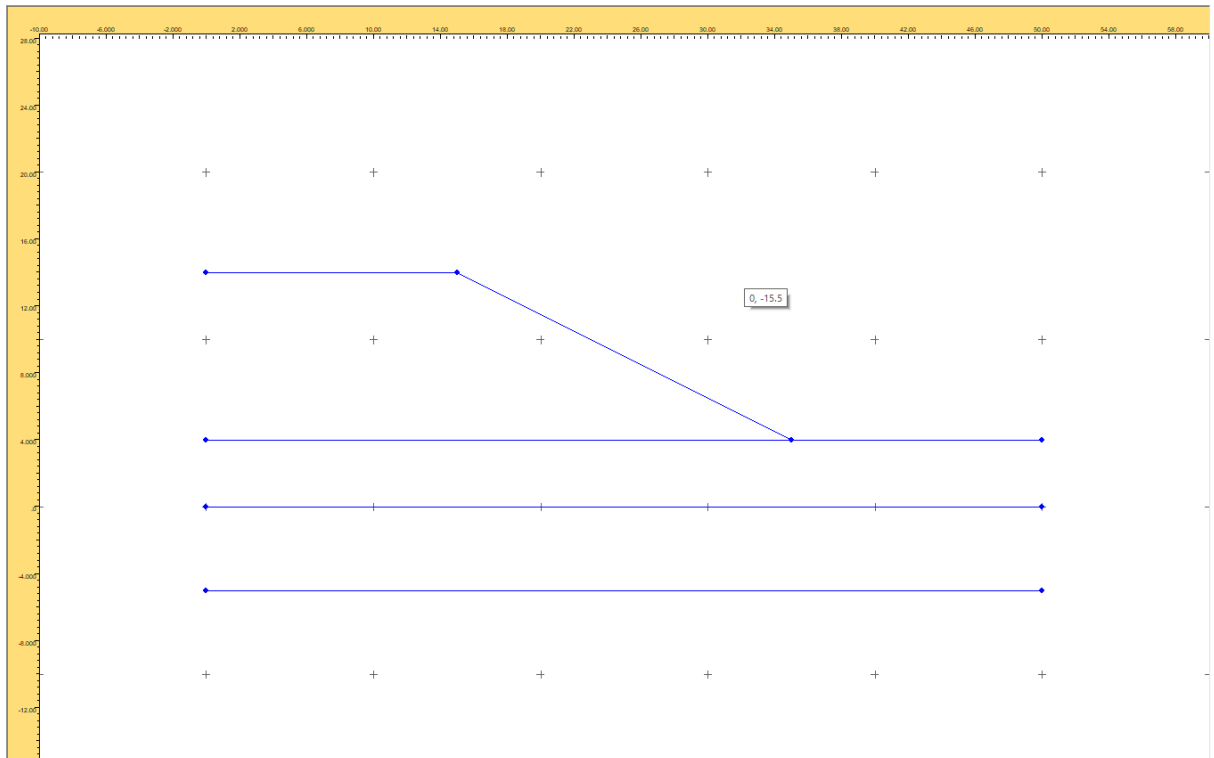
8. Left click to add points at the co-ordinates for the Embankment given in Section 4.1. As you move the cursor, a tool tip showing the current point coordinates will be shown. If a required point is outside the limits of the window, you can right-click and then enter the point coordinates into an edit box. You can also set the extents of the view by clicking the **Set View** button. This shows a **View Dimensions** dialog which allows setting of either the x or y dimension. The window is then redrawn with the dimensions recalculated.



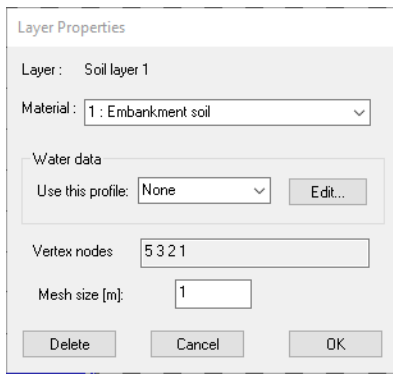
Press the *Escape* key to terminate drawing of the Embankment polyline. Continue to draw a line from (0,4) to (35,4) and another from (0,0) to (50,0), and a final line from (0,-5) to (50,-5) to represent the base of the problem.

Hint: as the cursor hovers over an existing point, when it changes to a square symbol, clicking will snap the current line to the existing point.

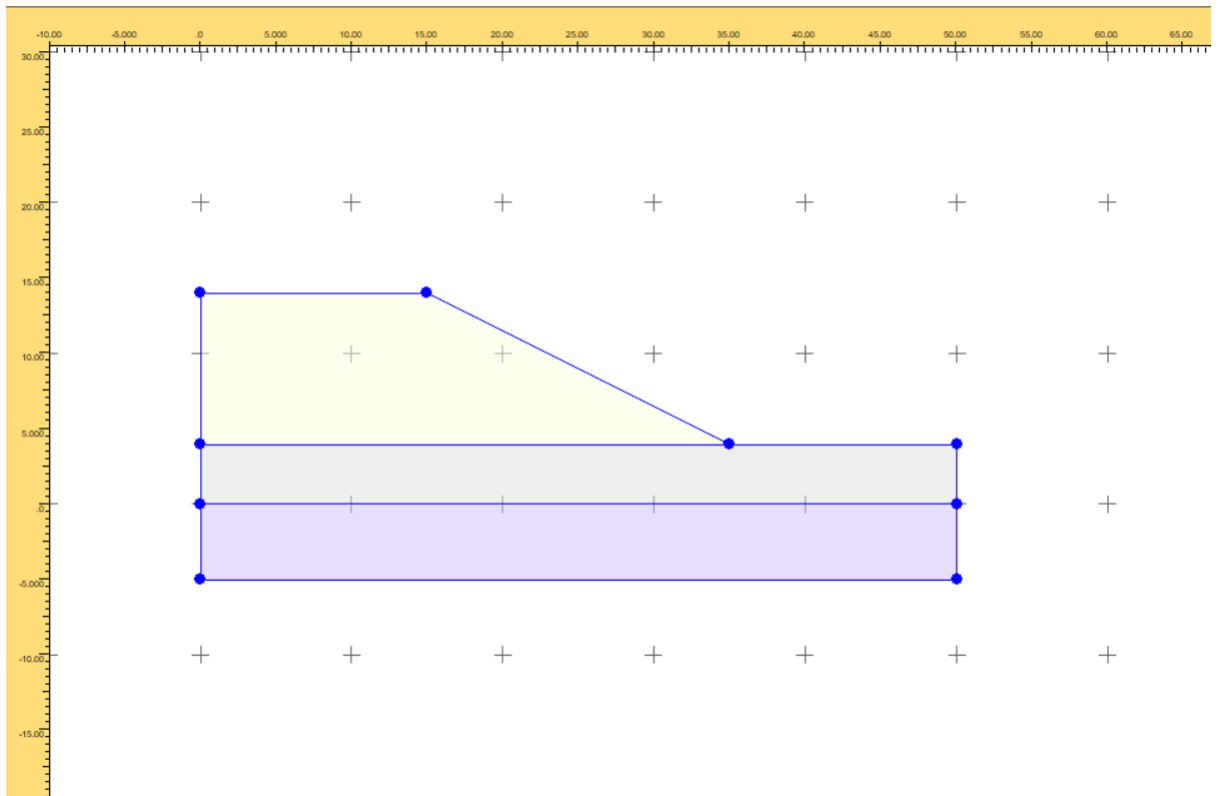
The screen should now look like this:



Complete the definition of the soil stratigraphy by adding vertical lines to join the left and rightmost points of each polyline. As polygons are closed, a popup dialog will ask if you want to create an 'area' representing a soil layer. Click Yes and choose the material to assign to the layer in the next dialog:

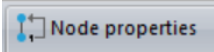


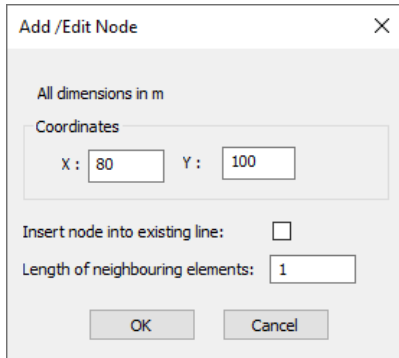
Click **OK** and the graphical input view will update to shade the new area in the material colour. Continue to close the 2D polygons and assign materials until the view looks like this:




Correcting errors

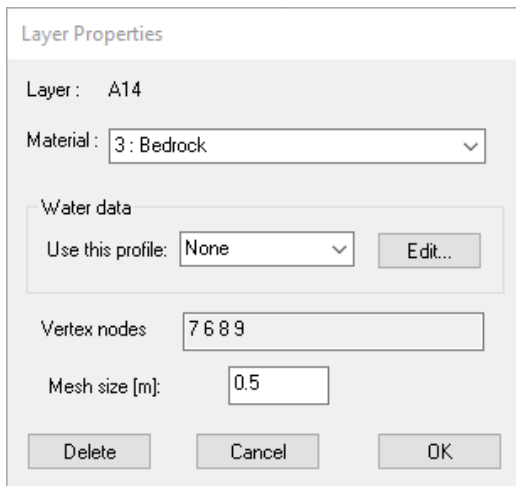
If you make a mistake in graphical input, you can either:

- Click the **Esc** key to stop drawing, then click the **Undo** button (on the Home tab or the Quick Access Toolbar) until the errors have been removed, or:
- Select the incorrect line or node(s) and press **Delete**, or:
- Select any node which is in the wrong position, click **Node properties** (this button  on the “Edit data properties” panel on the ribbon), and revise the node’s coordinates in the popup dialog.

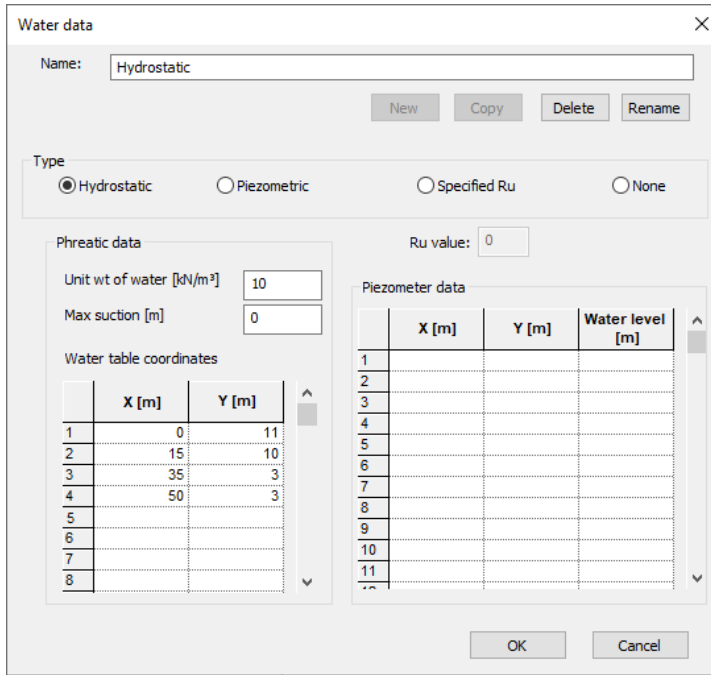


To assign pore pressures to the soil layers, click **Layers** on the Select panel of the ribbon. Left-click in any of the soil layers and it will be highlighted with a cross. Click **Layer properties** ( **Layer properties** button on the “Edit data properties” panel on the ribbon) for the Layer Properties popup dialog.

The Layer Properties dialog shows the current pore pressure specification as None:



Click the Edit button next to that field to open the Water Data dialog and enter the coordinates of the Original hydrostatic water table from Section 4.1.



Click OK and the water table will be saved and assigned to the currently selected layer. Select all the layers in turn, and assign the same water table to them using the **Layer Properties** dialog. Note that the water table will be drawn as a dashed line if it passes through layers where it has not been assigned.

The Water Data dialog is also available from the **Ground Model** panel on the Home ribbon tab, or via double-clicking on an existing water data item in the Data Explorer. The ground model data is now complete, so it's a good idea to save the revised file at this point.

- On the right side of the program window, note the settings on the **Specification** tab. These are default settings for circular slip surfaces, which are all as required for this analysis. Switch tabs to **Slip Surfaces** and change the Type from Single to Grid. This updates the tab with the fields to enter the bottom left grid coordinates, number of centres in the x and y directions with their spacing, and an optional angle of rotation. Enter a grid with the bottom left at (25, 15), with 15 centres at 1m spacing in both the x and y directions. Add an angle of rotation of -26 degrees.

(Hint: Choosing a finer grid may give more accuracy, but the analysis will take longer to carry out as more calculation points have been chosen. It is advisable to refine a large grid around the worst case slip circle centre after the initial analysis has been carried out.)

Check Extend Grid to find minimum factor of safety. This will enlarge the grid if the minimum factor of safety is outside the specified grid.

For the Circle Radius Specification, choose the Defined radii method with an initial radius of 1m and an increment of 1m. Leaving the "Limit radius" button unchecked will continue analysis of 1m circle radius increments until the circles reach the extents of the defined

ground model.

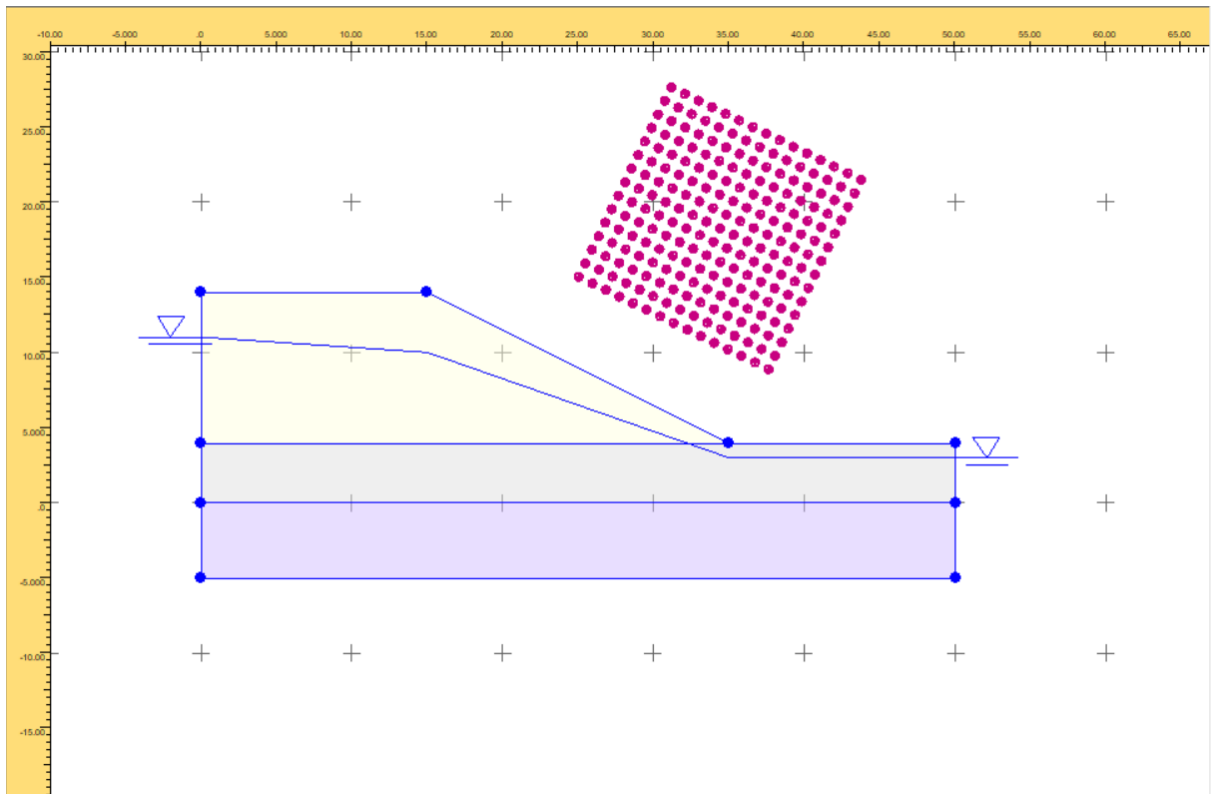
(Note: For a discussion of the different methods, please refer to the User Manual, Section 4.3.10.)

The image shows a software dialog box titled "Slip Surfaces". It has a "Specification" tab and a "Slip Surfaces" sub-tab. The main area is a table with two columns: "Property" and "Value".

Property	Value
<input type="checkbox"/> Circle centre(s)	
Type	Grid
<input type="checkbox"/> Coordinates	
x	25.000000
y	15.000000
Angle of rotation	-26.000000
<input type="checkbox"/> Centres on grid	
Number in x dim	15
x spacing	1.000000
Number in y dim	15
y spacing	1.000000
Extend grid	<input checked="" type="checkbox"/>
<input type="checkbox"/> Radius	
Type	Defined radii
Initial radius	1.000000
Increment	1.000000
<input type="checkbox"/> Limit radius to:	0.000000

At the bottom of the dialog box, there is a section titled "Extend grid" with the text "Tick to extend grid to find minimum factor of safety".

The Graphical Input Screen should show the following:



If the Strata, Groundwater or Grid are showing incorrectly, they can be amended to the correct values.

10. Double click on **Surface Loads** in the Data Explorer.

Enter a vertical live load of 10 kN/m per horizontal m.

This load should act between (0,14) and (15,14) along the crest of the slope.

(Note: A default of No is selected for the “Use in-pullout calc”. This excludes the surcharge in the pull-out calculation for any reinforcement. Section 3.6 of the User Manual gives more details about this calculation.)

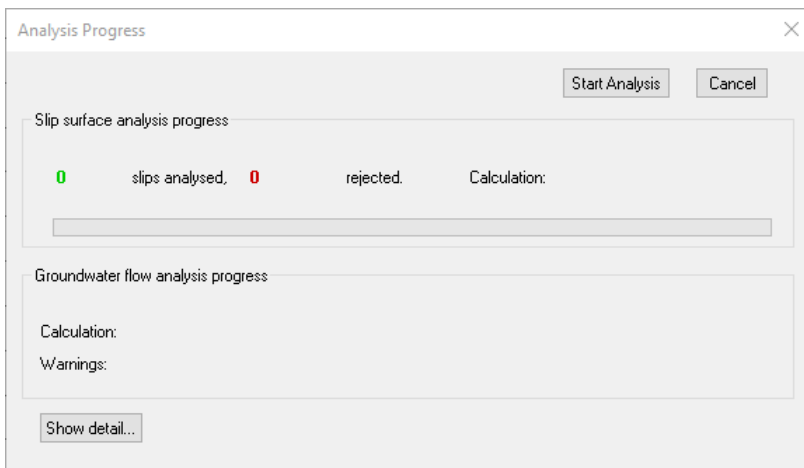
Load	Limits of Loaded Area		Distributed Load [kN/m hor/m]		Permanent or Variable load	Favourable or Unfavourable load	Use in pull-out calc.
	Left [m]	Right [m]	Vertical	Horizontal			
Defaults	0.00	0.00	0.00	0.00	Permanent	Unfavourable	No
1	0.00	15.00	10.00	0.00	Permanent		No
2							
3							

11. Check the **Graphical Input**. This should now show the surcharge.

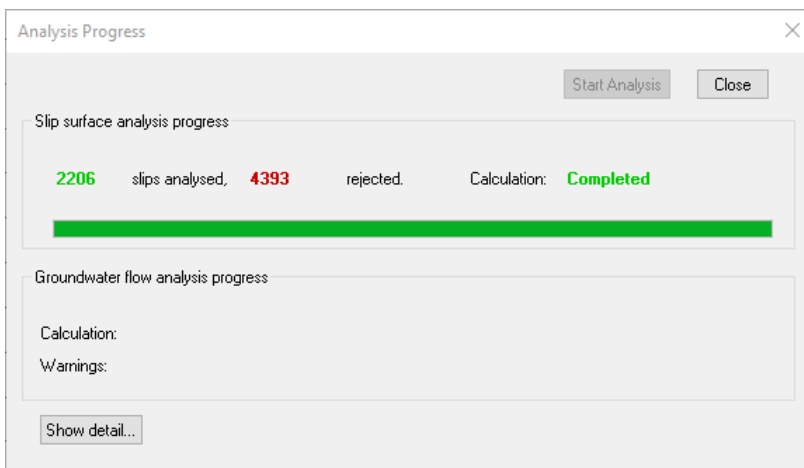
12. Save the file prior to analysis by clicking **Save** on the Quick Access Toolbar.



Click on the analyse button on the Home ribbon tab to open the Analysis Progress dialog. The Start button will carry out some data checks and start the analysis if there are no data errors.



The dialog updates to show a calculation status “In Progress” and how many circles have been analysed and how many rejected. On completion of the analysis, the calculation status is updated to “Completed” and the Start button greyed out. Click Close to dismiss the dialog.



The fields in the **Specification** and **Slip Surfaces** are also greyed out. If any data edits are required, the results need to be deleted to re-enable the fields. In other data views, making edits will prompt for deletion of the results before proceeding to accept the changes.

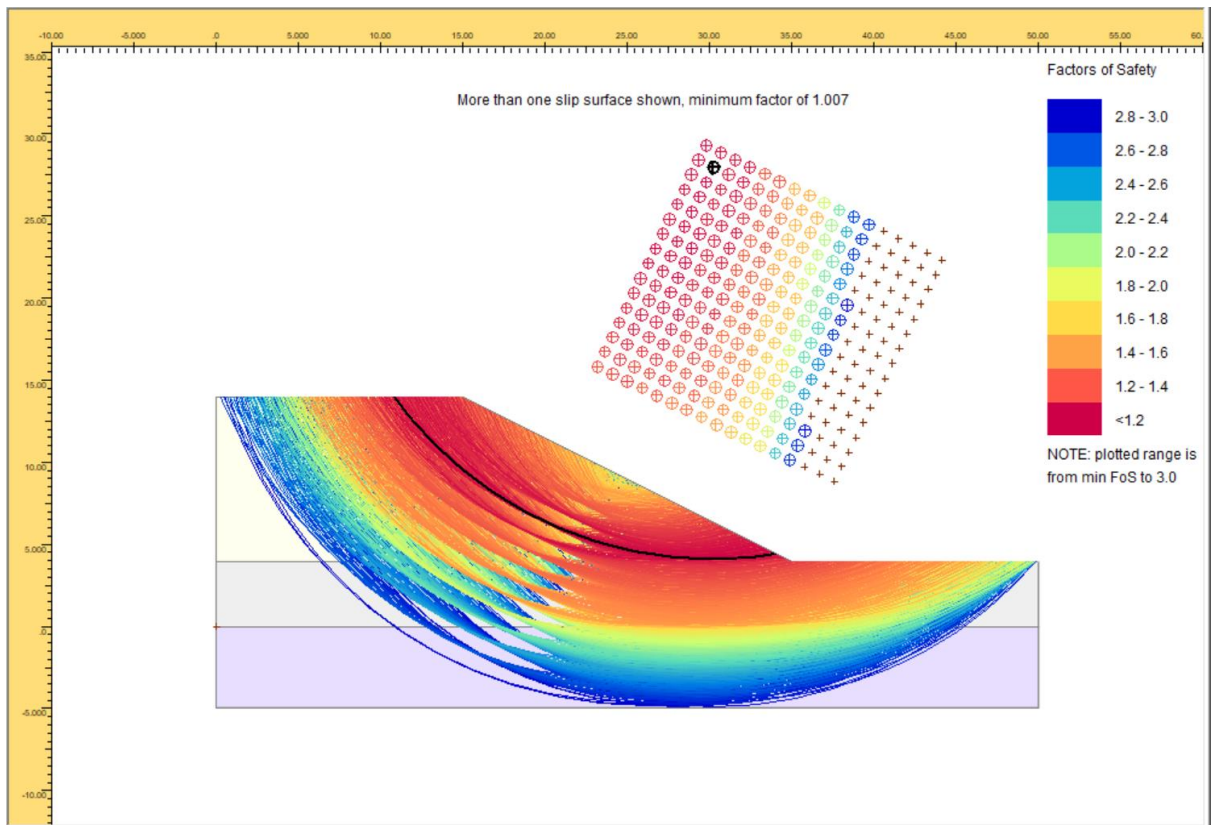
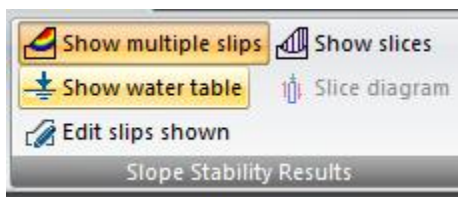
4.3 Viewing the Results

4.3.1 Graphical Output

Viewing a number of Slip Circles

Double click **Graphical Output** on the Data Explorer or click **Graphical Output** on the Home ribbon tab. The Graphical Output window will open showing the lowest factor of safety circle for each grid centre and the overall worst case slip circle.

Options available for this view are shown on a separate ribbon tab. The display options for slip circles are on a *Slope Stability Results* panel:



The initial range of factors of safety shown will default to 3 or lower, but this can be revised by clicking **Edit slips shown** and selecting a different Maximum FOS. The view and legend will be

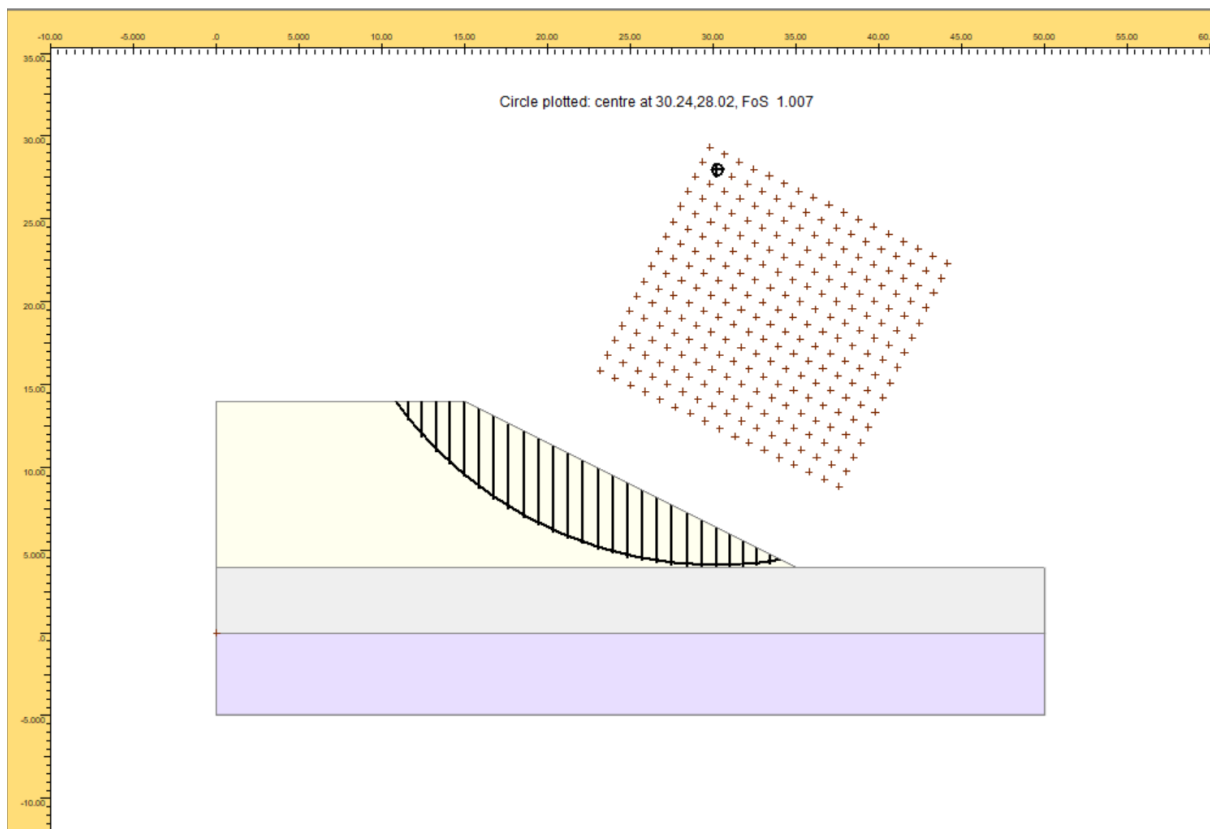
updated.

Viewing a Single Slip Circle

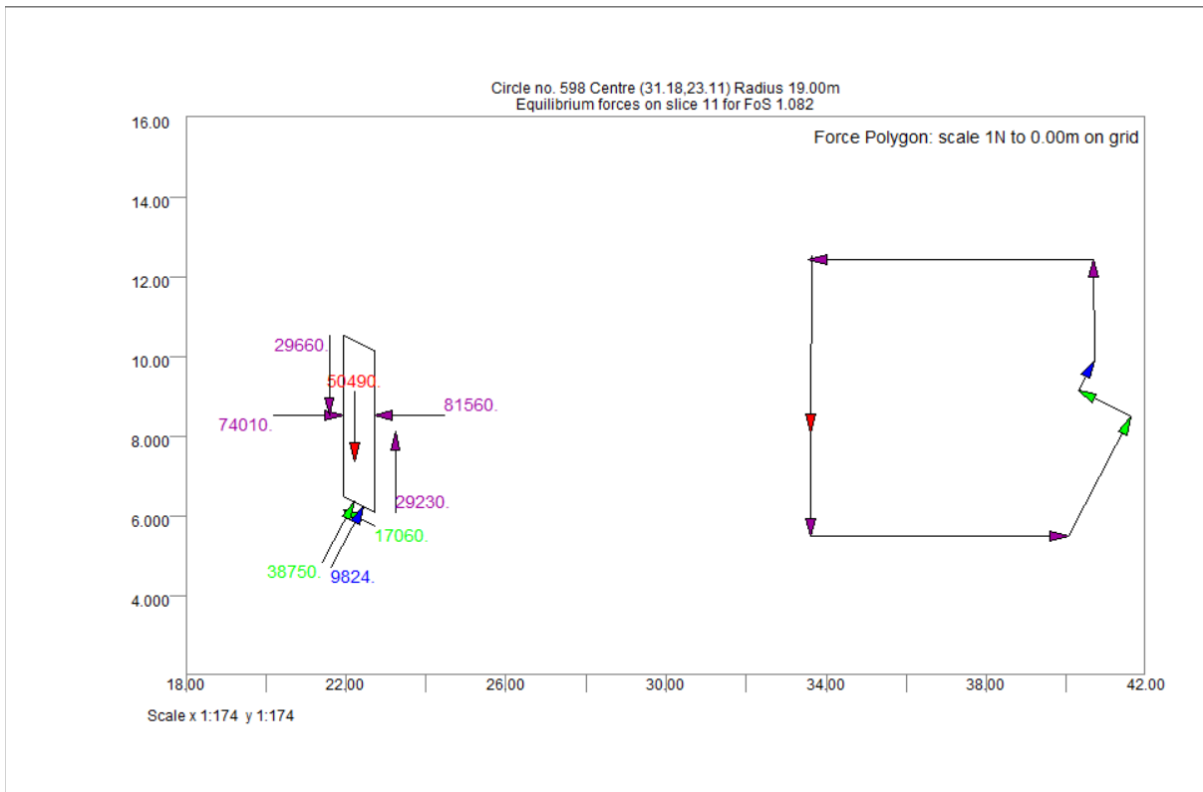
To view the worst case slip circle, switch off *Show multiple slips*.

Right click on the circles in the grid to view or switch off different circles. If only one circle is plotted, a tool tip with the restoring and disturbing moment, slip weight and other details for that slip circle will be shown.

Click on *Show slices* to view the slices analysed for the specific Slip Circle.

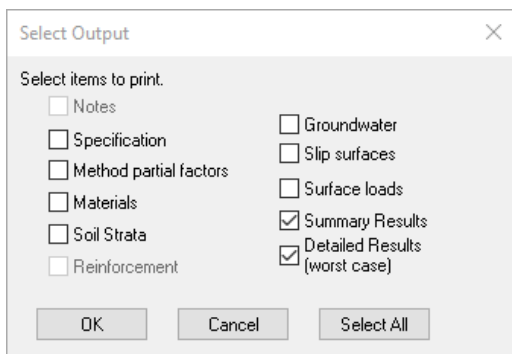


To view the slice diagram for a particular slice within the interslice analysis, click *Slice Diagram*. Then click on a slice of interest in the Graphical Output screen.



4.3.1 Tabular Output

1. Double click on **Tabular Output** in the Data Explorer
2. Select to view the Summary Results and Detailed Results (Worst Case). Note that when there are many slip circle centres, as in this problem with a large grid which may be extended, and multiple circles per centre, the summary results may take some time to be generated.



3. Review the results for each slip centre:

SimpleSlope.sbd : Slope Output x SimpleSlope.sbd : Graphical Input SimpleSlope.sbd : Surface Loads SimpleSlope.sbd : Graphical Output

RESULTS OF ANALYSIS

In the following tables, values in italics are factored values using the currently selected Method Partial Factors.

Slip Centre	Radius	Slip Weight	Comment/ FoS	Disturbing Moment	Soil Strength Restoring Moment	Reinforcement Restoring Moment
x [m]	y [m]	[m]	[kN/m]	[kNm/m]	[kNm/m]	[kNm/m]
25.000	15.000	6.0000	<i>34.348</i>	2.7309	86.336	235.77
25.000	15.000	7.0000	<i>150.32</i>	1.7467	404.91	707.26
25.000	15.000	8.0000	<i>323.67</i>	1.5726	930.39	1463.1
25.000	15.000	9.0000	<i>573.16</i>	1.4064	1744.0	2452.7
25.000	15.000	10.000	<i>896.09</i>	1.2867	2873.5	3697.2
25.000	15.000	11.000	<i>1294.8</i>	1.1911	4422.3	5267.5
25.000	15.000	12.000	<i>1741.1</i>	1.4090	6236.3	8787.0
25.000	15.000	13.000	<i>2263.7</i>	1.4732	8331.8	12274.
25.000	15.000	14.000	<i>2849.6</i>	1.5247	10716.	16339.
25.000	15.000	15.000	<i>3496.7</i>	1.5689	13406.	21033.
25.000	15.000	16.000	<i>4231.5</i>	2.1993	16353.	35964.
25.000	15.000	17.000	<i>5056.5</i>	2.4918	19523.	48649.
25.000	15.000	18.000	<i>5960.2</i>	2.7441	22822.	62626.
25.000	15.000	19.000	<i>6951.2</i>	2.9923	26386.	78953.
25.000	15.000	20.000	<i>8021.4</i>	3.2296	30146.	97360.
25.000	15.000	21.000		Only 1 intersection with ground level		
25.000	15.000	21.000		Only 1 intersection with ground level		
25.438	15.899	7.0000	<i>37.151</i>	2.6972	109.99	296.67
25.438	15.899	8.0000	<i>161.49</i>	1.6995	507.18	861.43
25.438	15.899	9.0000	<i>345.44</i>	1.5160	1147.3	1739.3
25.438	15.899	10.000	<i>608.60</i>	1.3489	2120.8	2860.6
25.438	15.899	11.000	<i>947.88</i>	1.2160	3499.6	4255.7
25.438	15.899	12.000	<i>1347.9</i>	1.2277	5235.4	6427.7
25.438	15.899	13.000	<i>1821.1</i>	1.3650	7285.6	9944.5
25.438	15.899	14.000	<i>2358.9</i>	1.4242	9613.8	13692.
25.438	15.899	15.000	<i>2960.9</i>	1.4740	12257.	18067.
25.438	15.899	16.000	<i>3628.4</i>	1.7052	16187.	25896.
25.438	15.899	17.000	<i>4393.0</i>	2.1747	18355.	39917.
25.438	15.899	18.000	<i>5243.1</i>	2.4612	21697.	53400.
25.438	15.899	19.000	<i>6180.4</i>	2.7046	25259.	68317.
25.438	15.899	20.000	<i>7194.3</i>	2.9574	28959.	85642.
25.438	15.899	21.000		Only 1 intersection with ground level		
25.438	15.899	21.000		Only 1 intersection with ground level		
25.877	16.798	8.0000	<i>39.800</i>	2.6699	135.63	362.12

(Hint: Review Section 5.2.1 of the User Manual to view the definition of the Column Headings and the definition of any error messages, such as the “Only 1 intersection” comment shown.)

4. Scroll down to view the detailed results for the worst case slip:

WORST CASE

Centre at (30.238m,28.021m) Radius 24.000m
 Iterations: 43 Horiz acceleration [g]: 0.0
 Net vertical force [kN/m]: 2.0750 Slip weight [kN/m] 1356.6
 Net horiz force [kN/m]: 5.0935 Disturbing moment [kN/m]: 12477.
 Restoring moment [kNm/m]: 12570.
 Reinf.Rest.Moment [kNm/m]: 0.0
 Factor of Safety: 1.0074

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.

Point	Slip surface coordinates		Pore Pressure		Interslice forces [kN/m]		E(u)
	x [m]	y [m]	L	R	T	E	
			[kN/m ²]				
			[kN/m ²]				
1	10.760	14.000	-	0.0	0.0	0.0	0.0
2	11.608	12.892	0.0	0.0	0.28547	2.0472	0.0
3	12.456	11.904	0.0	0.0	4.8217	8.8224	0.0
4	13.304	11.015	0.0	0.0	11.775	19.279	0.0
5	14.152	10.211	0.0	0.0	19.754	32.376	0.0
6	15.000	9.4900	5.2000	5.2000	27.833	48.549	1.3520
7	15.896	8.7780	9.0827	9.0827	33.755	65.408	4.1248
8	16.793	8.1410	12.315	12.315	38.218	82.383	7.5834
9	17.689	7.5640	14.948	14.948	41.149	98.419	11.172
10	18.586	7.0400	17.051	17.051	42.670	112.85	14.536
11	19.482	6.5670	18.643	18.643	42.825	125.00	17.379
12	20.378	6.1410	19.766	19.766	41.797	134.48	19.535
13	21.275	5.7580	20.459	20.459	39.806	141.12	20.928
14	22.171	5.4180	20.721	20.721	37.015	144.60	21.469
15	23.067	5.1180	20.584	20.584	33.673	144.98	21.185
16	23.964	4.8560	20.067	20.067	30.013	142.34	20.134
17	24.860	4.6320	19.170	19.170	26.257	136.75	18.373
18	25.757	4.4440	17.912	17.912	22.628	128.54	16.042
19	26.653	4.2910	16.305	16.305	19.325	118.04	13.292
20	27.549	4.1730	14.348	14.348	16.538	105.61	10.293
21	28.446	4.0890	12.050	12.050	14.416	91.785	7.2604
22	29.342	4.0380	9.4229	9.4229	13.065	77.146	4.4396
23	30.238	4.0210	6.4556	6.4556	12.601	62.260	2.0837
24	31.046	4.0350	3.4907	3.4907	12.194	48.921	0.60924
25	31.853	4.0760	0.25576	0.25576	11.036	35.874	0.0032706
26	32.660	4.1440	0.0	0.0	9.0856	23.814	0.0

5. The **Export** command on the application menu allows export of this output to various text file and HTML formats for further post-processing if required.

4.4 Test Yourself

Using the process outlined in the previous sections, apply the Raised Groundwater Profile outlined in Section 4.1.

Compare the impact of the new groundwater profile on the factor of safety.

Print out your new results.

(Hint: Remember to change the Titles section to give appropriate headings and save this file under a different name)

5. Partial Factor Analysis and Reinforcement Design

Following setting up a Slope Stability analysis, the user may wish to carry out an Ultimate Limit State (ULS) analysis. To follow the design process, this tutorial will show the user how to apply EC7 partial factors to the analysis set up in Section 4. The user will then be guided through reinforcement design to obtain an appropriate slope design.

Objectives

- Apply partial factors
- Apply reinforcements


5.1 Partial Factor Analysis

1. Open the file created in the previous section with the Original Groundwater Profile (Simple Slope.sbd). Delete the existing results.
2. Save as EC7 Analysis.sbd
3. Amend the **Titles** section to show that this file is carrying out a factored analysis
4. In the **Specification** pane, change FOS options to Partial factor analysis and choose “BS EN 1997-1:2011 DA1-1” as the Code. You can view the partial factors for this Code by double-clicking on the field, which will open the Method Partial Factors dialog:

External surcharge		Soil factors	
	Favourable	Unfavourable	
Permanent	1	1.35	Unit weight
Variable	0	1.5	Drained cohesion
			Undrained cohesion
			Tan of friction angle
Moment correction factor	1	Economic ramification of failure	1
Reinforcement pull-out	1.5	Sliding along reinforcement	1.5

It can be seen that this primarily factors actions (i.e. surcharges and reinforcement).

(Hint: To better understand the Eurocode 7 design approach, the user can download information from the Oasys Eurocode 7 seminar (https://www.oasys-software.com/index.php/webinars/webinar/ec7_and_geotechnical_analysis/) or review the Eurocode 7 design code.)

5. Prior to analysing, save the data file. As before, click on  to carry out the Slope Stability Analysis.
6. View the overdesign factor. Print the output showing the worst case slip circle.

Click on the *Delete Results* icon.

7. Apply EC7 DA1-2 in the **Method Partial Factors** input.
8. Re-analyse and compare the overdress factor for the worst case slip for Design Approach 1, Combinations 1 and 2.

The results for Combination 2 demonstrate the Slope is failing in the Ultimate Limit State.

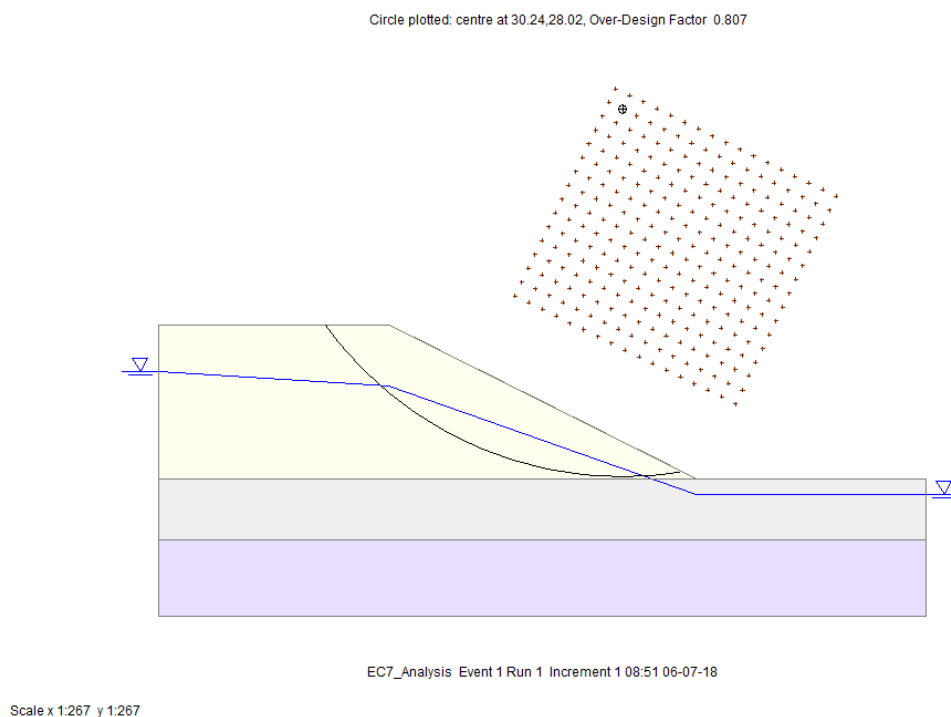
Save this analysis.

(Note: The user should find that Combination 2 gives a lower overdress factor. This is as expected, as Combination 2 generally factors materials. However, if reinforcement is added or extra surcharges are applied, Combination 1 may become more critical as this generally factors actions.)

5.2 Reinforcement Design

Based on the results in the Eurocode 7 analysis in Section 5.1, the user may wish to amend the slope geometry design or add reinforcement. This section considers designing reinforcement to increase the overdesign factor above 1.

1. Open the file created in the previous section (EC7 Analysis.sbd)
2. Double click on the **Graphical Output** in the Data Explorer.
The Ultimate Limit State analysis shows an Over design factor below 1.



3. Double click on **Reinforcement** on the Ribbon.

Input data for Geotextiles from the crest to the base of the slope with an Ultimate Capacity of 100kN.

An example input is shown on the following page.

Analyse and print the output with the new Overdesign Factor.

Is the slope with new reinforcement design acceptable?

Reinforced Slope.sld : Reinforcement Data

Reinforcement 1 Add reinforcement

Name: Reinforcement 1 Type: Geotextile

Layers
 Single Multiple Number: 9
Uppermost Level [m] 13 Spacing [m] 1
Offset from slope surface [m] 0

Bond Length b
 (%) (m): 0

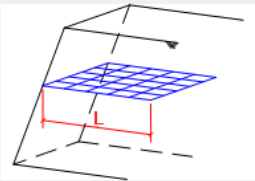
Bond strength (kN/m):
 Specify 0
 Calculate from effective stress

Length L [m]
Top layer: 15 Bottom layer: 15

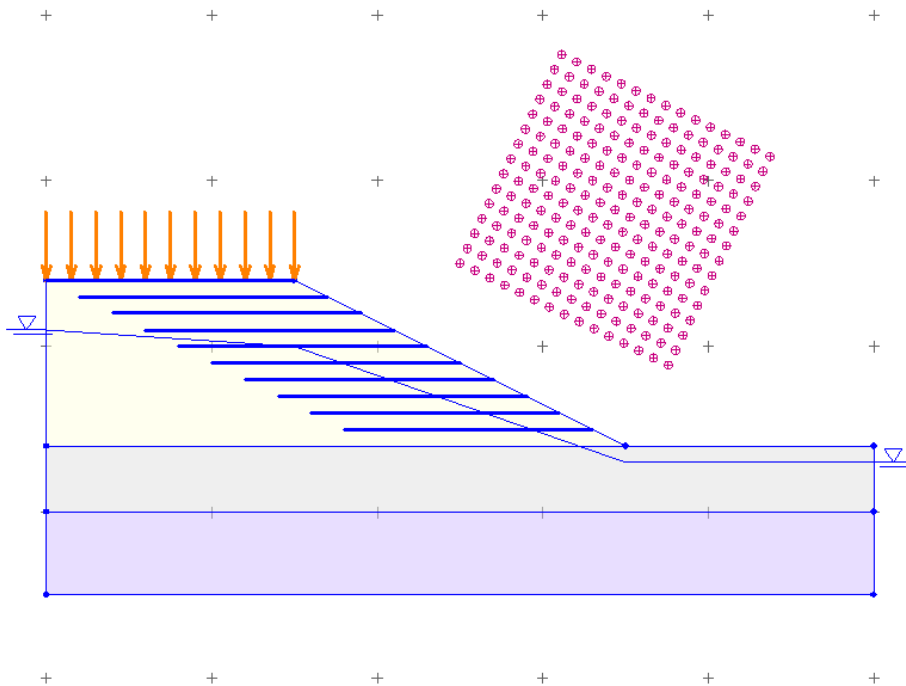
Capacity and Spacing
Out-of-plane spacing [m] 0
Ultimate tensile capacity (kN/m): 100
Plate capacity (kN/m): 0

Material partial factors
Select

Angle from horizontal w (°): 0
Prestress (kN): 0
Effective width (%) 100
Include stripping calculation

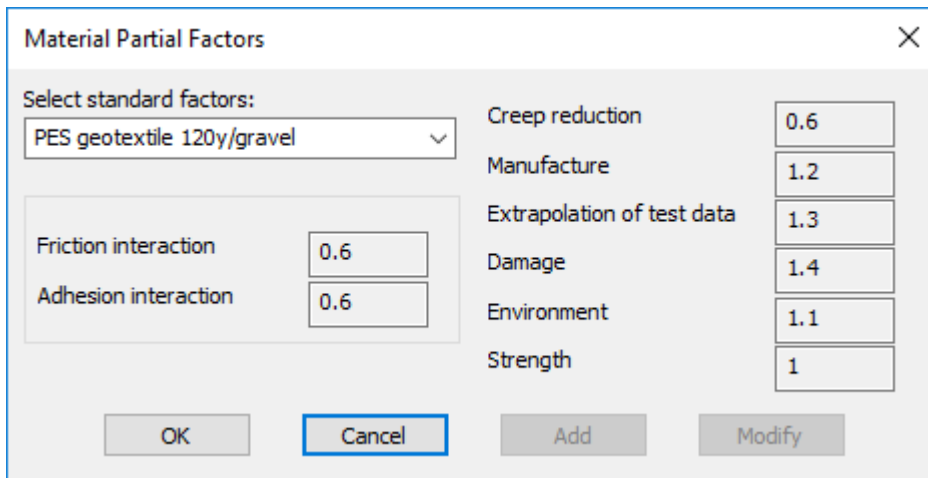


Apply Undo Add Copy Delete



Hint: If the reinforcement is not showing as blue, it is not active. Select Specification in the right property pane and tick Reinforcement Active (if there are existing results, you will need to delete them to make the data editable).

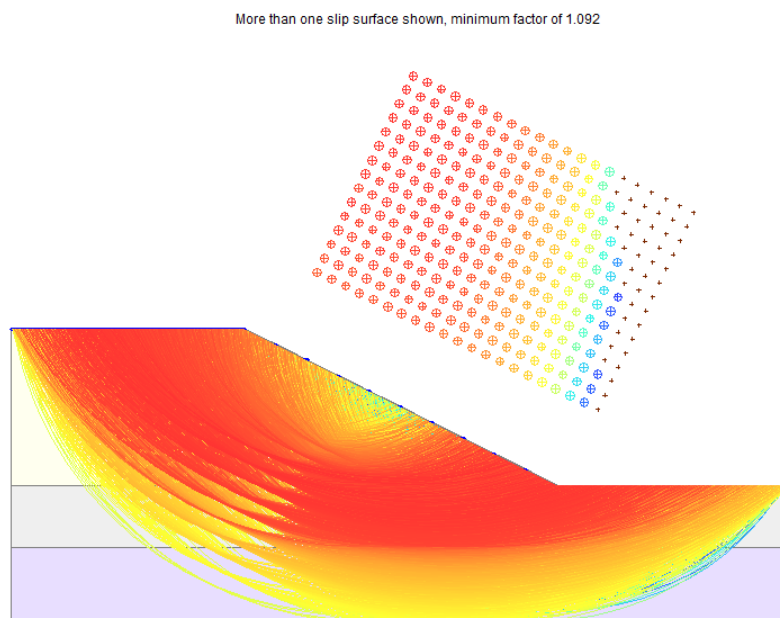
- In the Reinforcement Section of the Gateway, click on the Select button in the Materials Partial Factors section.



This section has a number of predefined partial factors for reinforcement.

Select *PES geotextile 120year/gravel* and click OK.

- Reanalyse and view the impact on the final Overdesign Factor.



6. Customised Input and Output

For more complex slope profiles and stratigraphies, drawing all the soil strata in the graphical input view can be time consuming. Oasys Slope allows the user to import dxf drawings and apply a strata and material or groundwater table to the different surfaces.

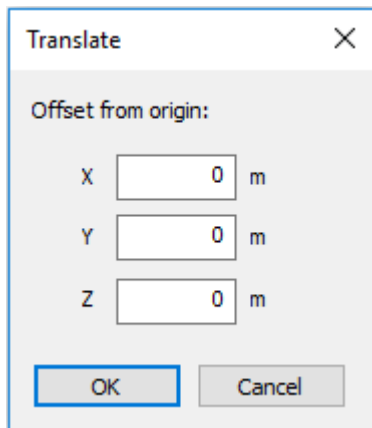
Objectives

- Import dxf drawings
- Limit the minimum slip circle size

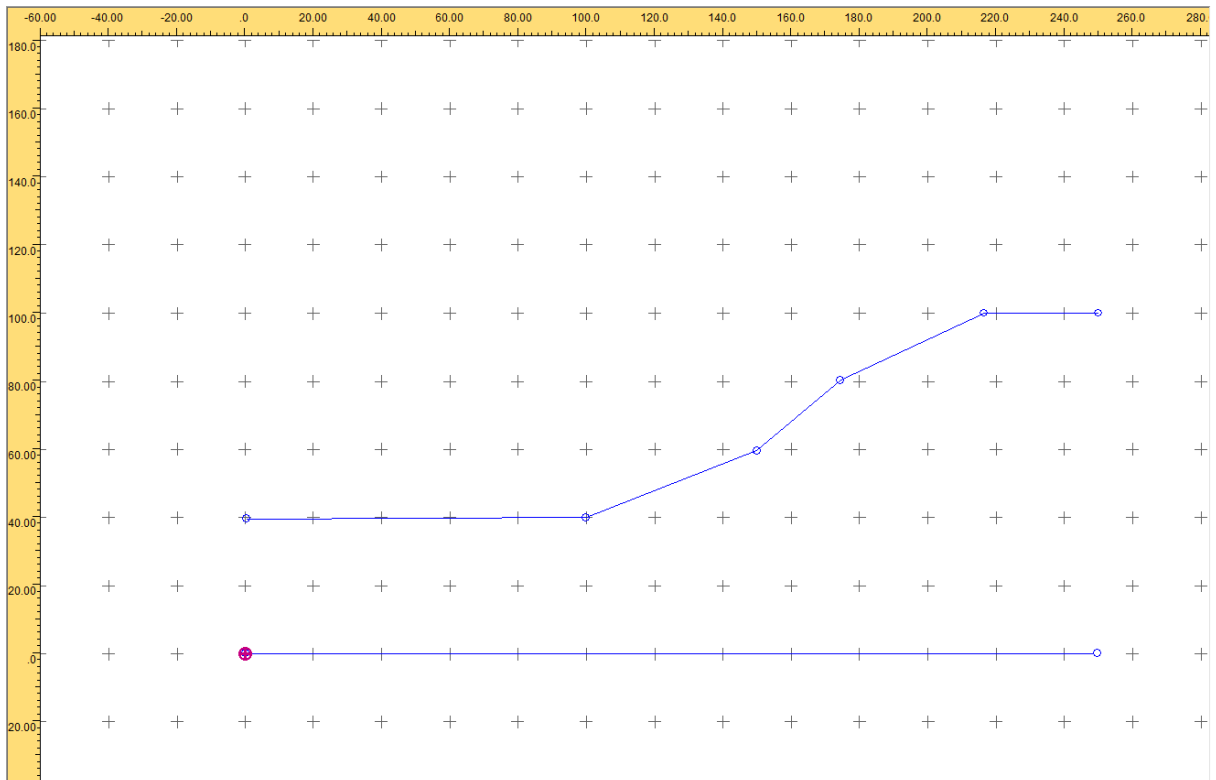
6.1 Importing DXF Drawing

1. Open a new Slope analysis file.
2. Input information into the Titles and Materials section.
3. In the Materials section, input the properties of the Embankment Soil (properties in Section 4.1).
4. Click the application button, then **Import**. Change the file type in the “Import XML or DXF” dialog to **DXF File**.
5. Import the file supplied with this tutorial (Slope Tutorial.dxf)

6. Place the origin co-ordinates at (0,0)



7. The lines imported from the dxf file will be converted to polylines representing soil strata, just as if they had been drawn using the Graphical Input tools:



8. You can now add new lines to complete the soil layer polygons and assign materials etc. as in the previous examples.

6.2 Limiting the Minimum Slip Circle Size

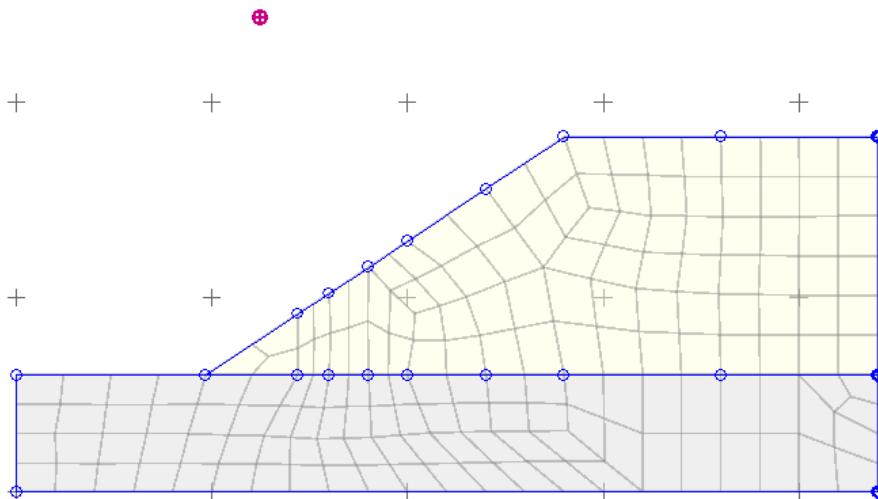
On the **Specification** pane, a minimum slip weight can be defined. This avoids reporting of small surface slips as the worst case. To try this, choose 1000kN as the minimum slip weight and compare the slip surface to the previous analysis.

7. Generating Pore Pressures

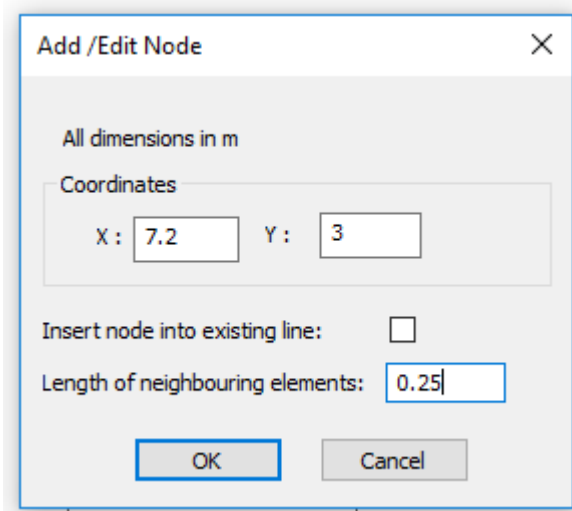
Slope can use either simplified pore pressure distributions entered by the user, or generate pore pressures based on material permeabilities and seepage boundary conditions. Some example files are provided with the program to show the difference between the two.

To replicate this comparison:

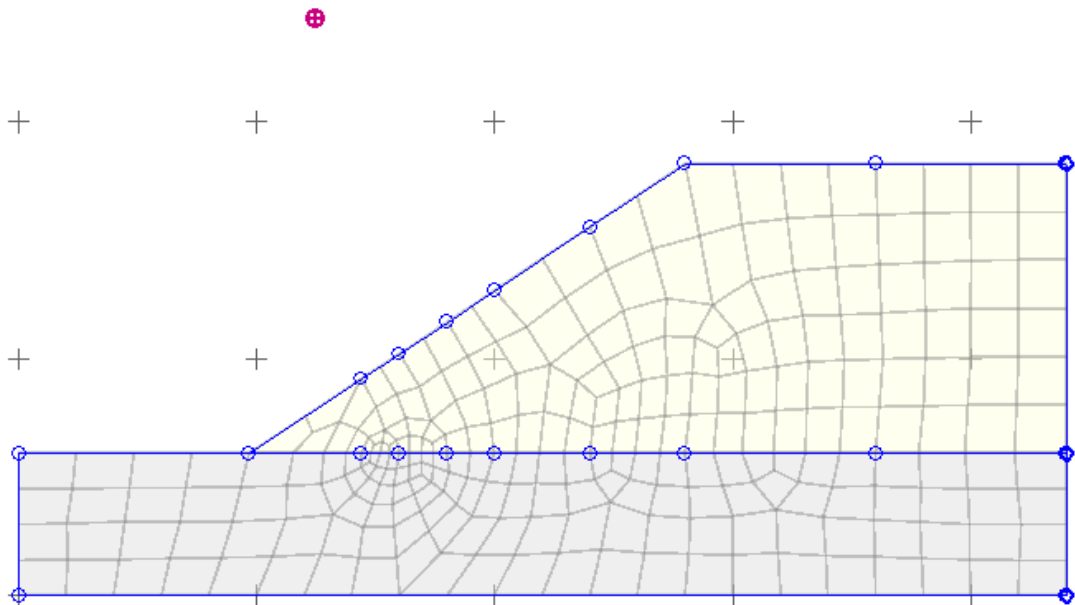
1. Open the provided example “Whitbu_revpp.sbd”. This has a specified water table with a set of piezometers to define the pore pressures.
2. Analyse the file. The calculated Factor of Safety for the Bishop method, with horizontal interslice forces, is 1.313.
3. Delete the results so that the data becomes editable, and on the **Specification** pane, change the “Pore Pressure Data” Type to **Calculated**.
4. The “Water data” item in the Data Explorer will disappear.
5. Save the data file with a different name.
6. Open the Materials table and you will see that a Permeabilities tab is now shown. For convenience, default values are set for the parameters. The different parameters are explained in the Help file and manual. For this exercise, the default values can be used.
7. Open the **Graphical Input** view. Click the “Show mesh” button on the ribbon. The view now shows an indicative mesh for the calculation of pore pressures.



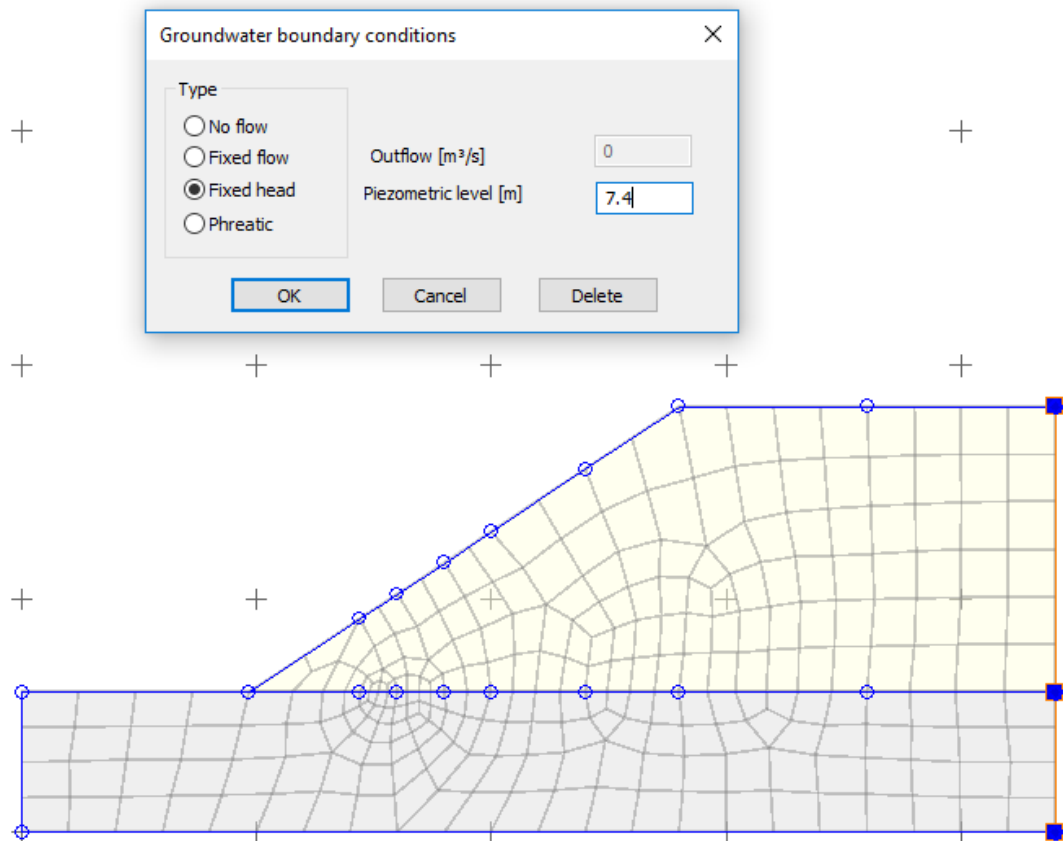
8. It would be better to refine the mesh around the areas where pore pressure may change rapidly. Individual nodes can be selected and a “mesh size” applied, which will immediately update the display. To do this:
 - a. Click the **Nodes** button in the Select panel of the ribbon
 - b. Select the node at (7.2, 3.0)
 - c. Choose **Node properties** from the “Edit data properties” panel of the ribbon
 - d. Add a value in the “Length of neighbouring elements” field. 0.25m is suggested:



9. Click OK and the view updates to show the new mesh. This looks like a better distribution of elements in the area where the water table is likely to change rapidly.



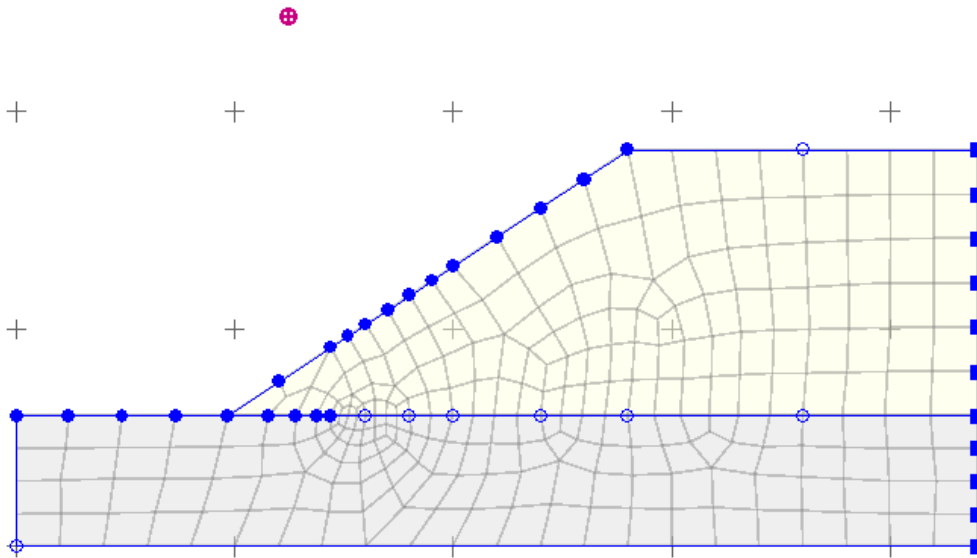
10. In the **Graphical Input** view, apply seepage boundary conditions, as follows:
 - a. Fixed head boundary of 7.4m at the right edge of the model (at $x = 22\text{m}$)
 - b. Phreatic boundaries on the slope face and between $x = 0\text{m}$ and $x = 7.2\text{m}$.
11. To apply a seepage boundary condition to lines:
 - a. Select the lines affected
 - b. Click the **Seepage restraints** button in the “Edit data properties” panel of the ribbon
 - c. Enter the required restraint type and the related data (level for a fixed head boundary, flow rate for a fixed flow boundary).



12. To apply a seepage restraint at a single node (not required in this example):
 - a. Select the required node
 - b. Click the **Node properties** button on the “Edit data properties” panel of the ribbon.
 - c. Enter the required restraint type and the related data in the dialog.

Note: if the same type of seepage restraint is set at both ends of a line, with the same settings for fixed head or flow if appropriate, the data will be updated to generate seepage restraints at intermediate nodes along the line. If this is not desired, it would be better to create an additional node off the line, and set the individual node seepage restraint on that.

13. Once all the seepage boundaries have been added, the graphical input should look like this:



14. Save the revised data and analyse the file. The analysis progress dialog will show when the analysis is completed. The “Show detail” button will expand the dialog to give details of the pore pressure calculations:

Analysis Progress ✕

Slip surface analysis progress

1 slips analysed, **0** rejected. Calculation: **Completed**

Groundwater flow analysis progress

Calculation: Completed

Warnings:

Evt	Inc	Itr	Max Inc Disp	At Node	IFAIL	Stress Errors				
						Max error	Elem	RMS error	Pwp error	Elem
1	1	1	0.0000	0	0	0.0000	0	0.00000	0.00000	0
1	1	2	0.0000	0	4955	0.0000	0	0.00000	71.94547	44
1	1	3	0.0000	0	0	0.0000	0	0.00000	-0.00000	248

Results for run 1 increment 1 saved

15. Close the Analysis Progress dialog and open the **Graphical Output** view. The new factor of safety is now 1.462.
16. In the Graphical Output ribbon tab, you can switch on the display of the water table and also plot line or filled contours of pore pressure or piezometric head (shown below).

