TALREN 4 – V 1.x

D. Tutorials and examples

1. DETAILED APPLICATION EXAMPLES (TUTORIALS)

1.1. Tutorial 1: typical soil-nailed wall

The purpose of this example is to show how to check the stability of a typical soil-nailing project (illustrated on the following figure).

The geometry of the project is defined on Figure 2.
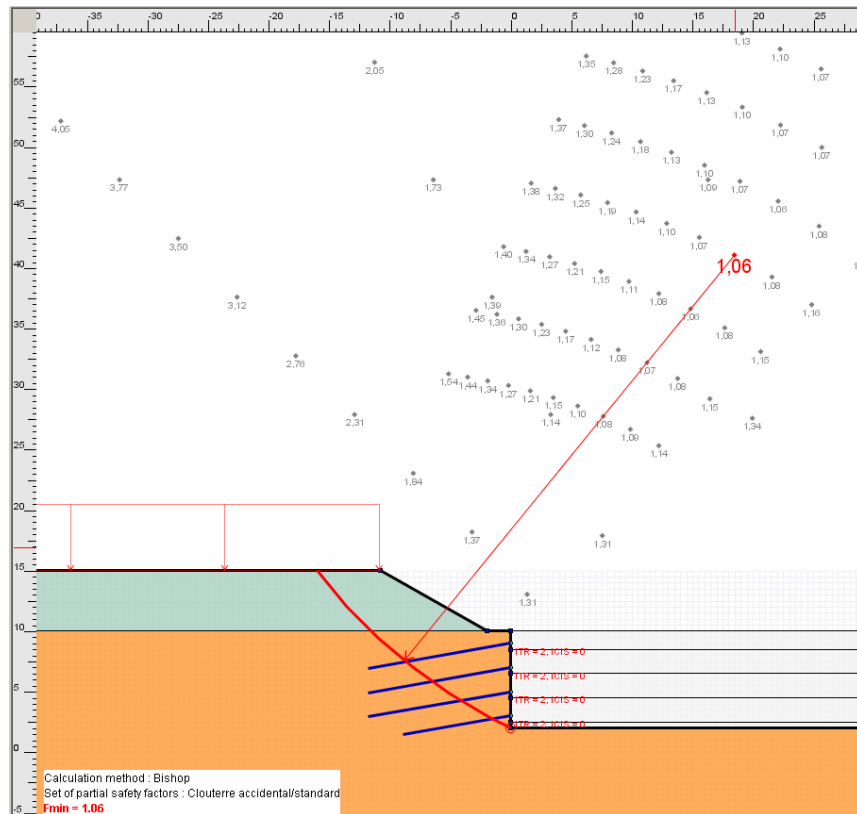
Figure 1: form project of a nailed wall
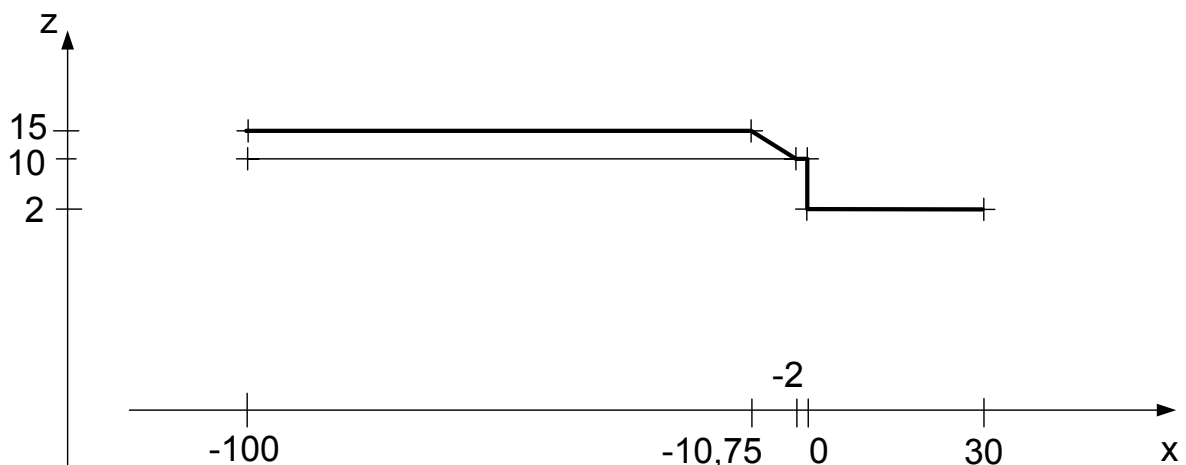
Figure 2: project geometry

1.1.1. Step 1: general settings

- Run TALREN: use Windows [Start] menu or click on the desktop icon.

- Select menu [File] then option [New] or click on the toolbar button [ ].

- Select menu [Project data] and option [General Settings]. Fill in the dialogue box with the requested information (Figure 4).

  - Concerning Xmin and Xmax: they are used for more than just graphical display: **Xmin and Xmax define the width on which the project has to be defined**. The slope boundary will be for example automatically calculated between Xmin et Xmax: if no segment of the project geometry reaches these boundaries, the slope boundary will not be identified, and the calculation will be impossible.

  - Units: the selected units will be valid for the whole project. However, it is possible to change units later. You will have the choice between: conversion all the data already defined, or no conversion of the data already defined (in case of an inconsistency between the units display and the data values themselves).

  - Calculation method and default partial safety factors: these are choices which will be saved as default values for this new project (but it will be possible to change thses choices for each situation individually). For example, if you define a project with multiple stages and multiple situations, and that you would like to perform all the calculations for the project using one single calculation method and one single set of partial safety factors, define these values as default values in the general settings: they will then be proposed for each new situation. If you don't define these choices as default choices for your project, you will have to define them manually for each situation.

    Note: It is not compulsory to define default partial safety factors. This choice can remain "empty". In that case, you will have to be select them manually for each situation.

  - Default partial safety factors: click on the button [Safety factors wizard].

    Select the set you want from the global list on the right (1) and click on the button [Transfer to local database] (2). The last set transferred is selected (red frame in the left list). Complete, if necessary, the missing values (3). Then validate with [OK] (4).
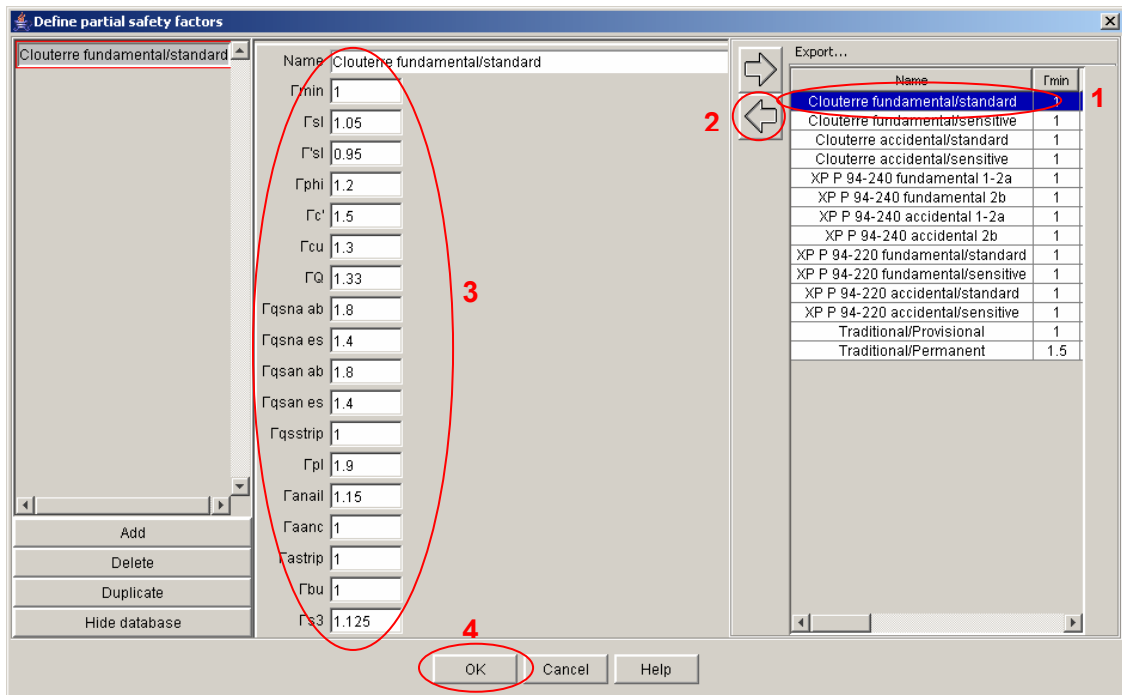
Figure 3: Selection of a partial safety factors

Important note: in the above window, the partial safety factors for anchors and struts have been completed in order to be able to validate the window, but will not be considered for the calculation in our example (no anchors or struts will be used). Thus, these values should not be considered as an example or reference to copy: each project should refer to relevant standards and recommendations, and to the corresponding partial safety factors.
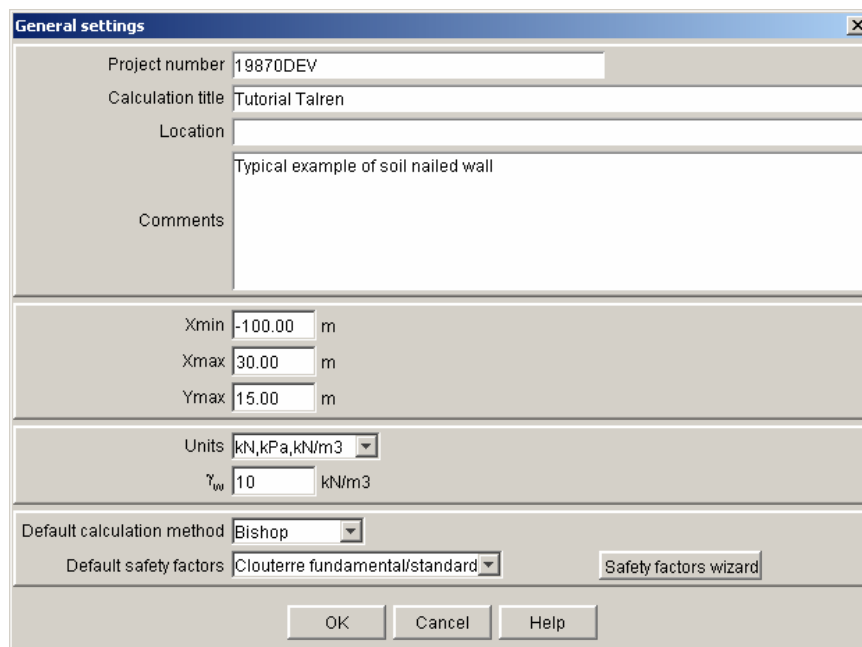
Figure 4: general description

- Select menu [File] then option [Save]. Select a destination directory and define a filename.

1.1.2. Step 2: geometry

The coordinates of the points defining the slope profile and the limits of the soil layers are indicated in Figure 2.

Important: it is also necessary to define in the Data mode all the earthwork levels which will be used in the stages/construction mode.

In order to define this geometry with TALREN 4, you have several choices: drawing with the mouse, typing in the coordinates in the main window, or using the geometry dialogue box. We will detail in this example successively all 3 methods.

- First of all, to facilitate the mouse drawing, it is advised to activate the grid display and attraction. To do so, click on menu [View] then on option [Grid]. You will then be able to define the grid spacing (choose 0.25 m for example), then activate both checkboxes and validate. The mouse will move on the screen by increments of 0.25 m. The smaller the grid spacing is, the more accurate the drawing will be.

- In order to draw the geometry with the mouse, select the toolbar button [pencil icon]. Click on the first point of the geometry to draw (using the horizontal and vertical rulers, and the mouse coordinates displayed in the status bar), then move the mouse, without releasing the button, to a 2nd point. When the button of the mouse is released, the 2nd point and the segment between both points are created. To define points without connecting them with segments, simply click on the points and release the mouse button after each click. The purpose is to obtain the same layout as in Figure 8.

  Drawing tip:

  o *If segments are drawn while maintaining the "Shift" button pressed, Talren 4 will draw either horizontal, or vertical lines.*

  o *To delete a point: click on it to select it, then click on the "Del" key, or right click on it then select [Delete] from the pop up menu. The same method can be applied to segments or any other object in the graphical view.*

  o *If you placed a point at a wrong position, click on the toolbar button [arrow icon], then double-click on the misplaced point: you can change and validate its coordinates.*

  o *The mouse coordinates are always displayed in the status bar.*

  o *You can display the numbers of the points and segments on the graphical view using menu [View], option "Points numbers" and "Segment numbers".*

  o *Many tools allow you to zoom in the graphical view: options [Zoom] in menu [View], toolbar buttons [icon], [icon] and [icon], options in the pop up menu (right click with the mouse on the graphical view). One of the zoom options ("View whole model") allows to adapt the zoom (scale) to display the whole model.*

IMPORTANT:

When the groundlevel definition has been completed, the slope boundary is displayed with a bold line:
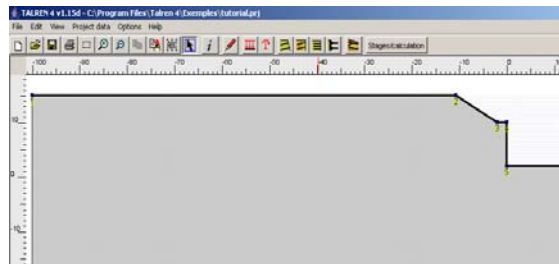
Figure 5: automatic identification (calculation) of the slope boundary

- In order to type in the points coordinates in the main window, select toolbar button ![pen icon]. Type in the coordinates of each point with the keyboard: click in the cell that appears in drawing mode (in the status bar at the bottom of the screen): type in the X and Y coordinates of the point (X and Y values should be separated with a blank space). Then press the "Enter" key. The background of the cell will then become blue: this means that the next point typed in will be connected to the previous one with a segment. To "pull up the pen", press the keyboard "Esc" button. The cell background turns white again, meaning that the next point typed in will not be connected to the previous one with a segment. One can at any time change from the "mouse drawing mode" to the "coordinates drawing mode".
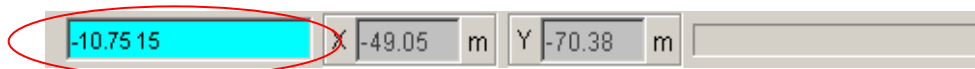
- In order to define the geometry using the geometry dialogue box (most similar way to previous version Talren 97), or check points coordinates: select menu [Project data] then option [Geometry]. The window of Figure 6 is displayed ("Points" tab). If you already defined geometry points with the mouse, they will be displayed in the table, otherwise, the table is empty.

  To change coordinates, simply click in the corresponding cells. To add new points, click on the "Add" button.
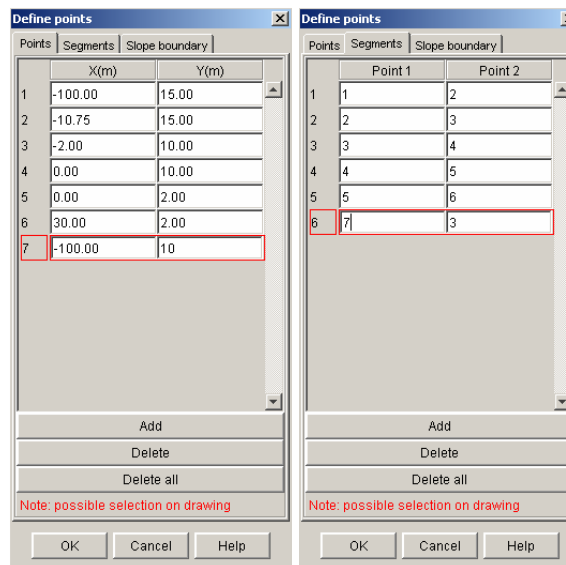
Figure 6: Geometry dialogue box, "Points" and "Segments" tabs

Tip: To go from a cell to the other (left to right on a same line, or change to the line below), the user can use the "Tab" key. "Shift"+"Tab" allow moving from one cell to the other but backwards.

When all points have been defined, click on the "Segments" tab (Figure 6): the table containing all the defined segments appears. It is possible to modify or to add segments.

When all segments have been defined, click on the "Slope boundary" tab (Figure 7). By default, Talren 4 automatically defines the slope boundary considering the project geometry. It will appear with a bold line on the graphical view.

If necessary, it is possible to deactivate the checkbox "Automatic slope boundary" and to modify manually the slope boundary (change the list of segments composing the slope boundary). However, it is recommended to keep to the automatic mode (so that the slope boundary can be changed automatically for each stage during the staged construction).
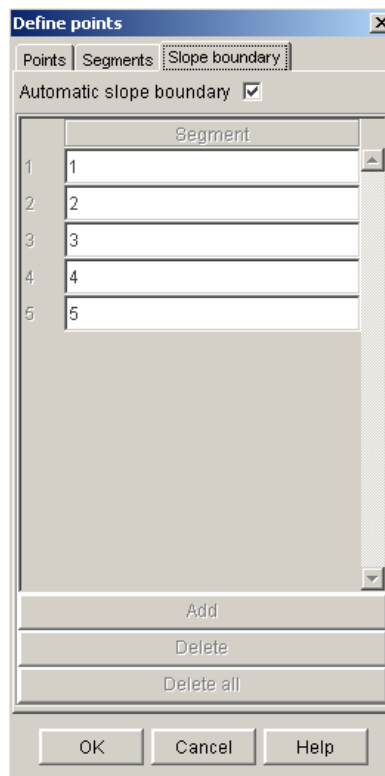
Figure 7: Geometry dialogue box, "Slope Boundary" tab
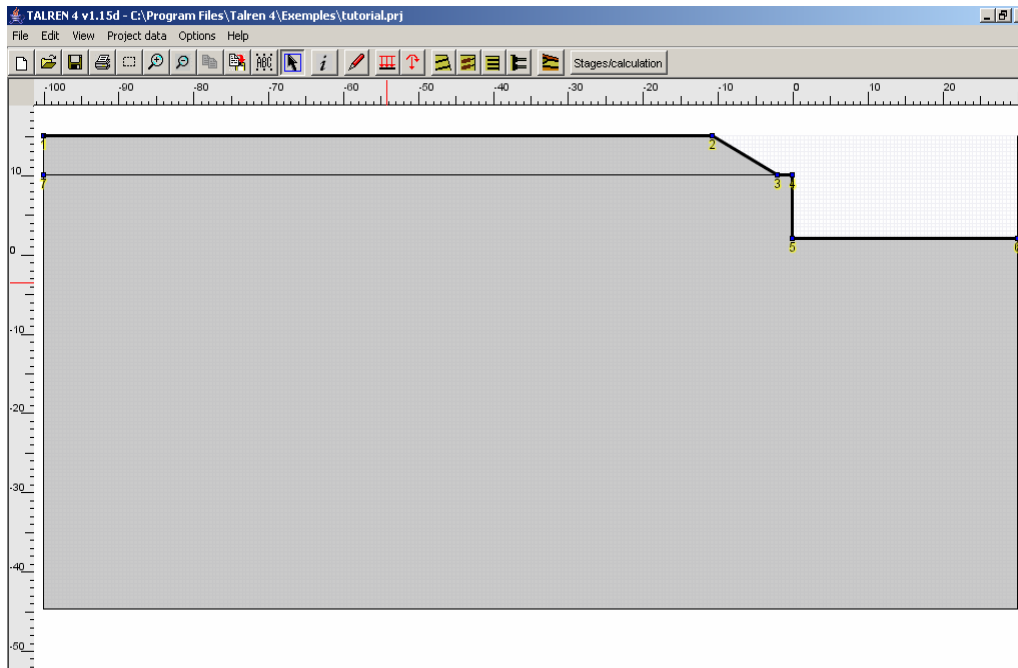
Figure 8: graphical view after geometry definition

Important note:

- The projet should always be defined with the uphill on the left of the model (the failure must happen from the left to the right, from X_{min} to X_{max}).
- If you defined your model in the wrong direction, you can reverse it automatically with menu "Options", item "Reverse model".
- The order of the points of a segment (left and right) is not important.

1.1.3. Step 3: loads definition

To define loads in TALREN 4, you have many choices: mouse drawing, input of coordinates directly in the main window, or dialogue box for loads definition. We will describe here successively all 3 methods. For this example, one load should be defined at groundlevel, between X = -100 and X = -10.75.

- In order to draw the load with the mouse, click on the toolbar button ⎯. Click on the left point of the load and then, without releasing the mouse button, move the mouse to the right point of the load (be careful, with a grid spacing of 0.5 m for example, you cannot draw the coordinate -10.75: you should get close to it, then change manually the coordinate in the dialogue box for load properties). The load is displayed on the graphical view (default orientation is vertical downwards). The load dialogue box must be activated to define the load value (right click on the load base line, then select [Edit] option from the pop up menu, or select menu [Project data] then [Loads]).

- In order to draw the load with data input in the main window: click on the toolbar button ⎯. Click in the cell that appears in the drawing mode (in the status bar at the bottom of the screen): type in the X and Y coordinates of the first point (X and Y values should be separated with a blank space: for example, type in: 10.75 15). Then press the "Enter" key, and type in the coordinates of the second load point and press "Enter" again. The load is displayed on the graphical view (default orientation is vertical downwards). The load dialogue box must be activated to complete the load value (right click on the load base line, then select [Edit] option from the pop up menu, or select menu [Project data] then [Loads]).

- In order to input the load using the corresponding dialogue box (most similar way to previous Talren 97 version), or to complete the load definition with the load value: choose menu [Project Data] then option [Loads]. The window of Figure 9 is displayed ("Distributed loads" tab). If you already defined a load with the mouse, it will be displayed in the table.

  To change some values, simply click in the corresponding cells. You can request display of load labels on the graphical view using menu [View]: option "Loads identification".
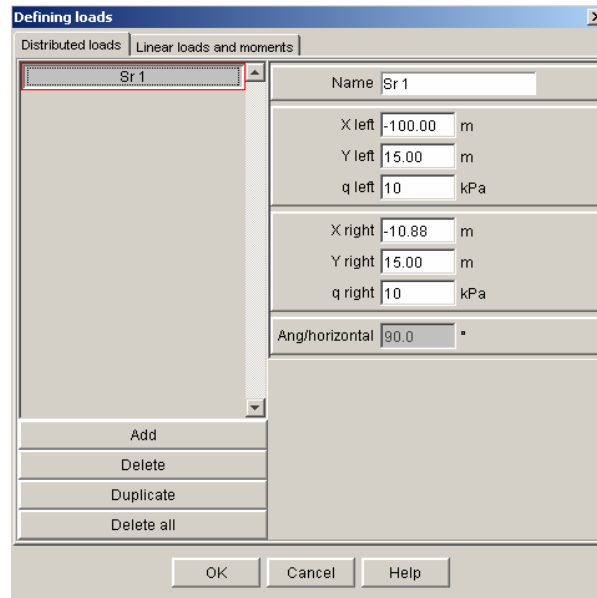
Figure 9: loads dialogue box, "Distributed loads" tab

1.1.4. Step 4: definition of the nails

Four nail levels have to be defined. They should all be defined in the "Project Data" mode. Then, they will be activated progressively in the successive construction stages.

Nails all have the same properties except for their length. Their properties are provided in Table 1:

Nail	$\varnothing$bar (m)	σ_e (MPa)	Sp. (m)	X	Y	L (m)	Angle (°)	LB (m)	ALB (°)	Re	Ind (*)
1	0.04	500	2	0	9	12	10	2	10	0.065	1
2	0.04	500	2	0	7	12	10	2	10	0.065	1
3	0.04	500	2	0	5	12	10	2	10	0.065	1
4	0.04	500	2	0	3	9	10	2	10	0.065	1

(*): The value of 1 means "calculated traction/imposed shear". In the event, we want to calculate the traction and use no shear forces.

Table 1: nail properties

For the 4 nail levels, we will also define:

- Rsc or qs nails from: curves (that choice decides the partial safety factor applied to pull-out resistance);
- Rsc (pull-out resistance) calculated based on qs;
- No variable shear force;

- Calculation of the traction on the external part of the failure surface only.

Here again, you have many choices to define reinforcements in TALREN 4: mouse drawing, or reinforcements dialogue box. We will describe here successively both these methods.

- In order to draw the nails with the mouse, click on the toolbar button ![icon]. Click on the position of the first nail head, and without releasing the mouse left button, move the mouse to the nail end. Release the mouse button. The nail is displayed on the graphical view.

- In order to complete the data (or to define the nail if you did not define it on the graphical view), choose menu [Project data] then option [Reinforcements] (or right click on the nail, then select "Edit" from the pop up menu). The window of Figure 10 appears ("Nails" tab). If the nail geometry has already been defined with the mouse, it will be displayed in the left list. Complete the missing data (simple click in the corresponding cells or "Tab" key to switch from one cell to the other).

- Since the other nail levels have almost the same properties, it is advised to click on the "Duplicate" button 3 times. 3 new nails are then created in the left list. Select each of them (click on each label in the left list), and change only what is necessary: level of the nail head and nail length. Then validate.

  It's possible to display the nails labels on the graphical view using menu [View]: option "Reinforcements identification".

  The nail properties can be edited with a right click on a nail in the graphical view (then select [Edit]).
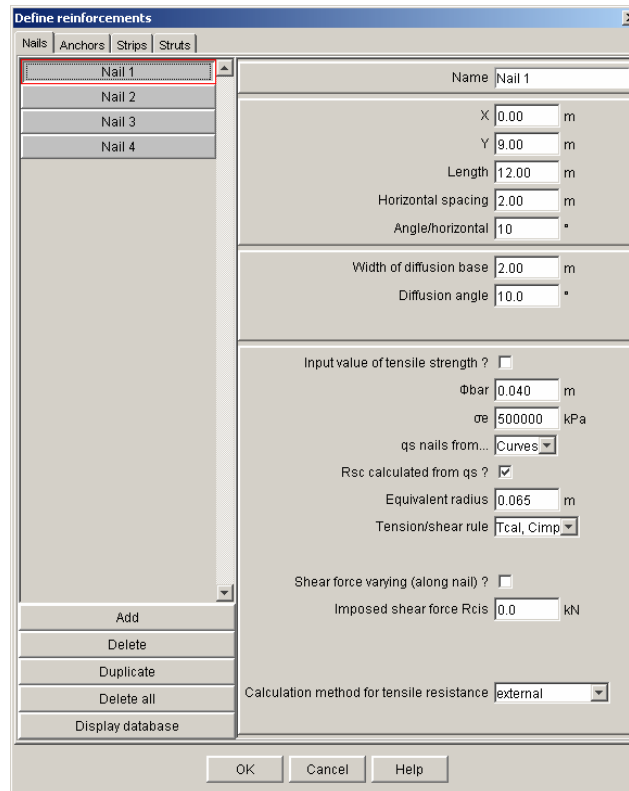
Figure 10: Reinforcements dialogue box,"Nails" tab

After definition of the 4 nail levels, the graphical view looks like Figure 11.
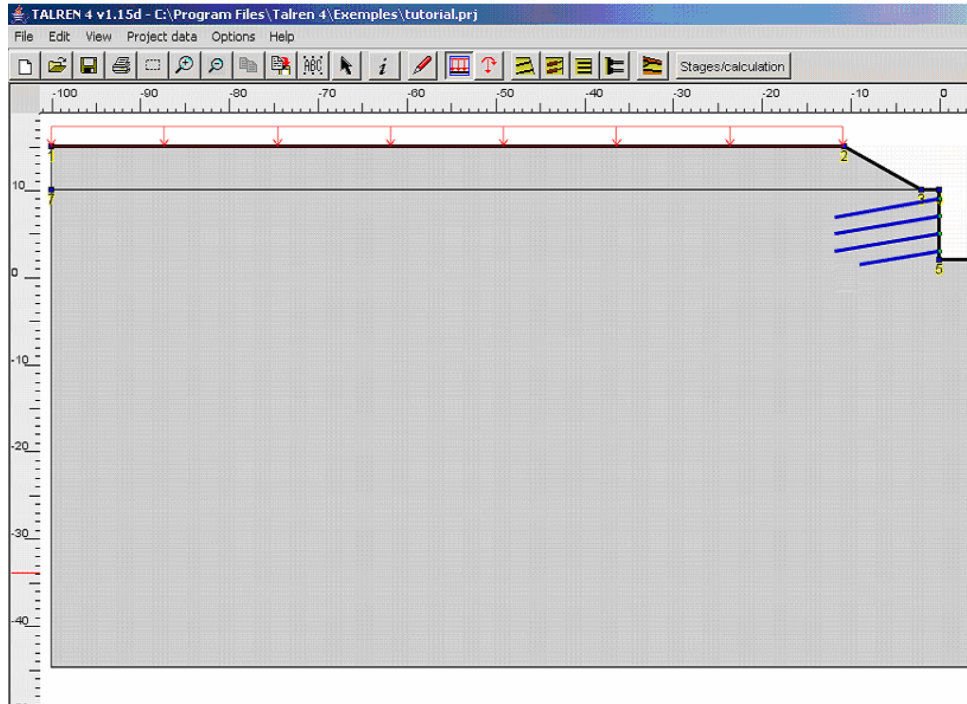
Figure 11: graphical view after nails definition

1.1.5. Step 5: definition of soil properties:
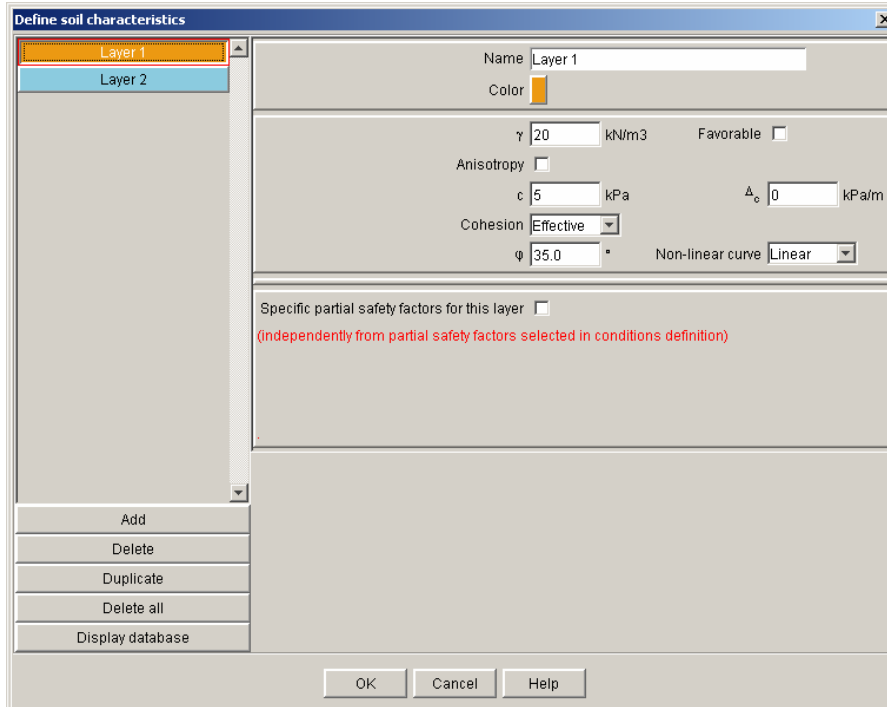
Two soil layers should be defined. Their characteristics are given in Table 2.

Layer	γ (kN/m^3)	φ (°)	c (kPa)	qs (kPa)
1	20	35	5	-
2	20	30	10	130

Table 2: soil properties

- Select menu [Project data] then option [Soil characteristics]. Click on the [Add] button, then fill in the data for the first layer (Figure 12). Click again on "Add", and define the data for the 2nd soil layer (Figure 13). To switch from a soil layer to the other; click on the labels in the left list of the dialogue box.

  The units to use are those defined in the project settings. They are repeated in all the dialogue boxes (for our example, units are, kN, m and degrees).
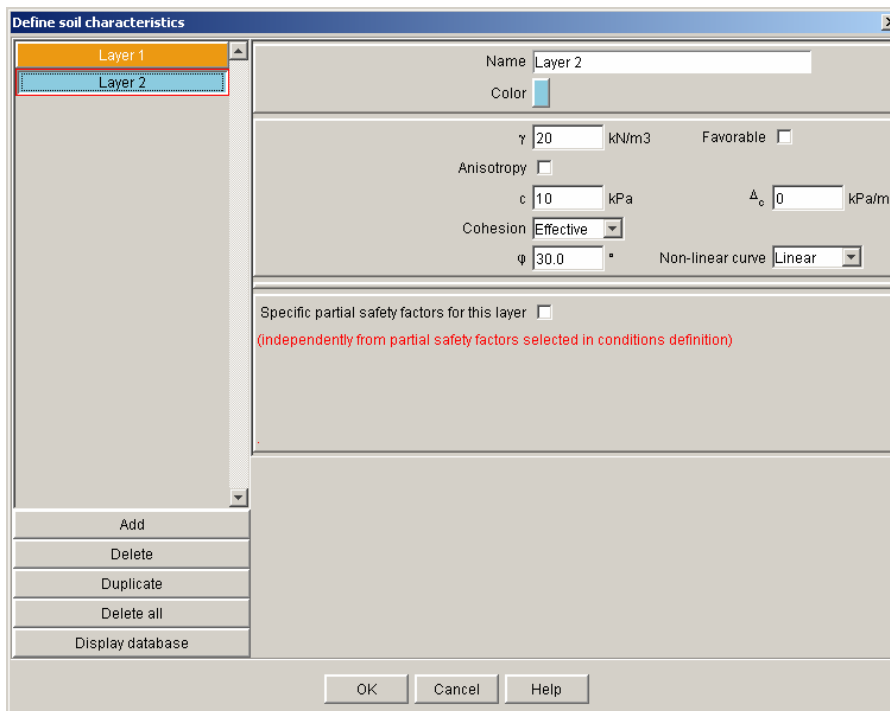
Figure 12: soil properties dialogue box (layer 1)

Figure 13: soil properties dialogue box (layer 2)

Tips:

- o *You can change the default name for each layer.*
- o *You can also change the colour for each layer: click on the coloured cell below the layer name.*

IMPORTANT

The nails cross layer 2 only and we requested the calculation of pull-out resistance based on qs values (in the nails definition). A qs value for layer 2 (130 kPa) should thus be defined.

However, Talren displays only the strictly necessary data, and the layers (soil properties) haven't been assigned to soil clusters yet. So Talren does not "know" that the nails go through layer 2. The qs data is thus not available yet on Figure 13.

We will have to define this qs value for layer 2 after layers will have been assigned.

- To assign soil properties to a soil cluster, let's use drag & drop: click with the left mouse button on the soil layer that you want to assign (in the left list of soil properties dialogue box), then, keeping the mouse button pressed, move the mouse onto the soil cluster you want to assign the layer to. Release the mouse button: the soil cluster will be painted with the same colour as the soil layer. Repeat the same operation to assign soil properties to all soil clusters. Close the dialogue box. The layout now looks like Figure 14.
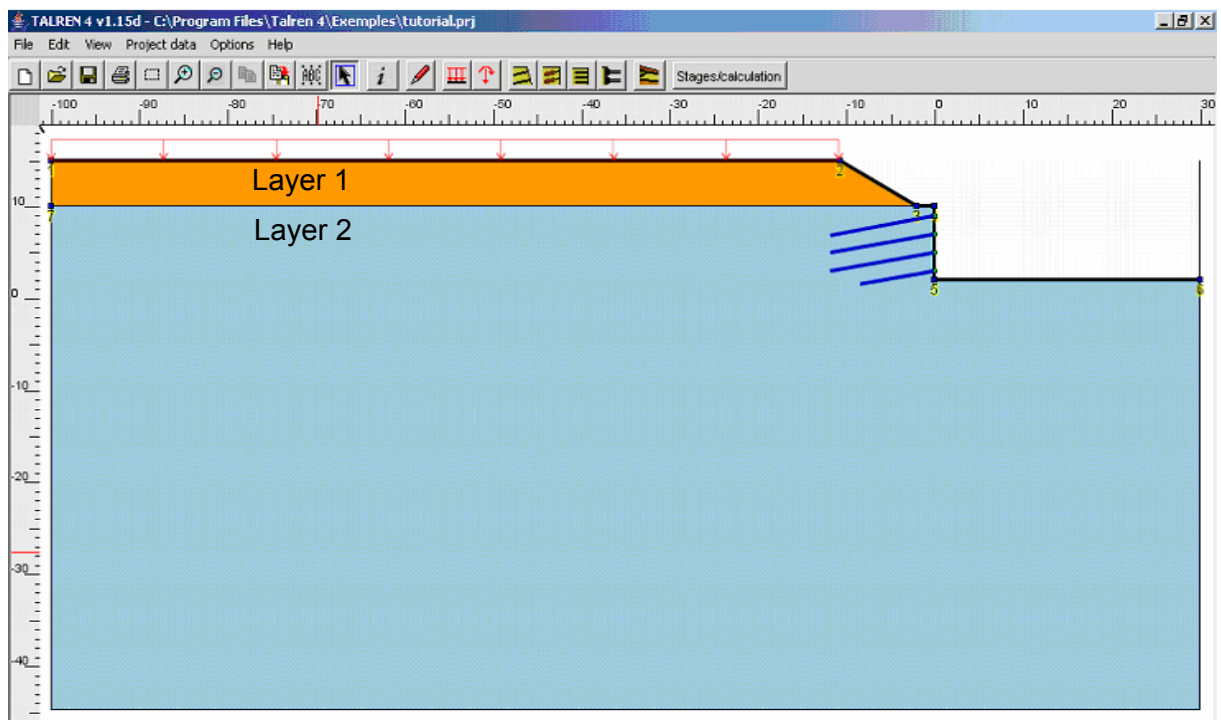
Figure 14: graphical view after soil properties assignment

To change the soil properties assigned to a soil cluster, another solution consists in right clicking on the soil cluster and selecting "Edit" from the pop up menu. You can then select another set of soil properties.
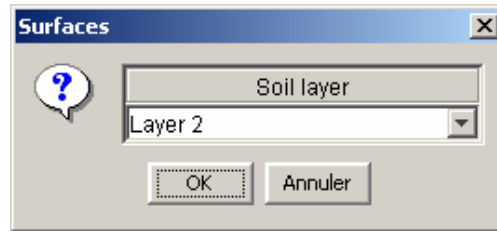
Figure 15: "Edit" dialogue box for a soil cluster

- After assigning the soil layers, open again the soil properties dialogue box if you closed it, or click on layer 1 and layer 2 again in the left list (the layer should be changed to update the display).

  Now the qs cell for layer 2 is displayed (Figure 16), but it is not for layer 1: qs is only displayed if nails go through the layer (and nails don't go through layer 1 for this project).
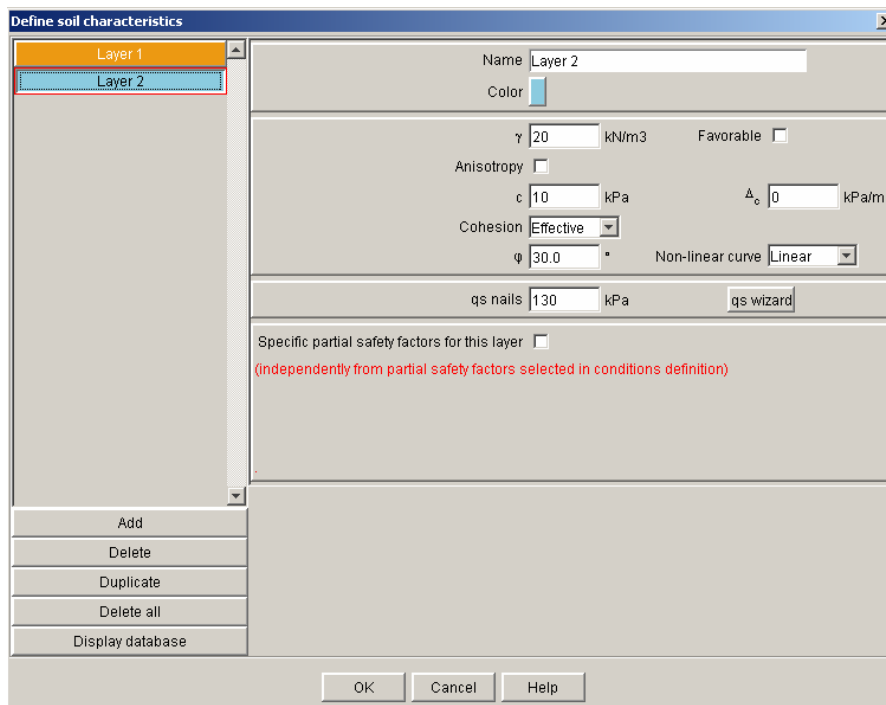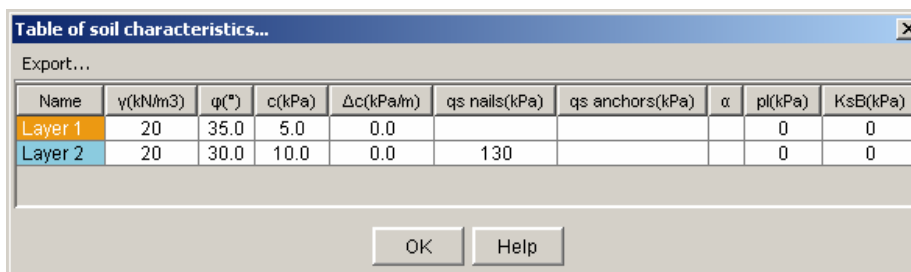
Figure 16: soil properties dialogue box (layer 2)

Note: at any moment, you can display tables summarising soil properties, reinforcements and loads data: select menu [View], and one of the 3 options [Table of…] (example below). Be careful: these summarising tables allow to view, but not to edit the data.

Name	γ(kN/m3)	φ(°)	c(kPa)	Δc(kPa/m)	qs nails(kPa)	qs anchors(kPa)	α	pl(kPa)	KsB(kPa)
Layer 1	20	35.0	5.0	0.0				0	0
Layer 2	20	30.0	10.0	0.0	130			0	0

Figure 17: table of soil characteristics

1.1.6. Step 6: definition of the intermediate excavation levels

We just finished the definition of the geometry for the final project configuration. But we will also have to check the stability for the temporary construction stages. It is thus necessary to define additional points and segments on the excavation side: excavation levels 10, 8.5, 6.5, 4.5 and 2.5 (Figure 18). The drawing of these additional points and segments can be performed either with the mouse on the graphical view, or with the geometry dialogue box (step 2).

The slope boundary is automatically updated according to the current groundlevel definition.

It is necessary to assign layer 2 to all these new soil clusters.
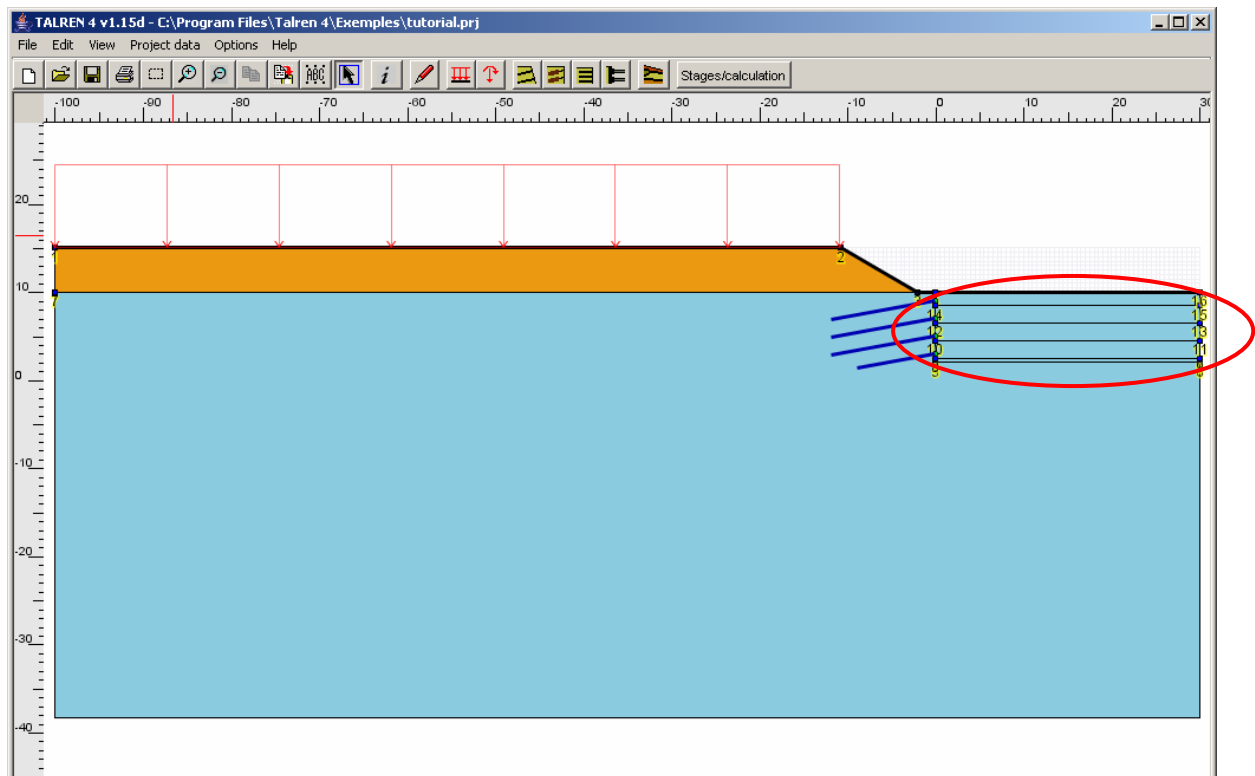
Figure 18: graphical view with intermediate excavation levels

IMPORTANT:

We could have defined directly the whole geometry in step 2.

Definition of all project data is now finished.

We can switch to the stages definition, using menu [Project data], option [Stages/Calculations], or clicking on the toolbar button [Stages/Calculations]: the stages/calculations mode is activated, and the first stage is automatically created (with a default geometry identical to the one defined in the Data mode, and all loads and reinforcement deactivated by default).

1.1.7. Step 7: definition of stage 1

This construction stage corresponds to the following operations:
- Excavation to level 8.50 (which will allow to install the first nail level).
- Installation of the first nail level.
- Excavation to level 6.50.

By default, the first stage is generated automatically with all soil clusters activated and with all loads/reinforcements deactivated.

In order to change the geometry for a stage, it's possible to activate or deactivate soil clusters: click the clusters with the left mouse button (they will be alternatively activated/deactivated). The same procedure applies to loads and reinforcements.

To create the first excavation level, click on the top soil cluster (Figure 19). To activate the nail, click on it. Then click the second soil cluster corresponding to the second excavation level (these 3 operations can be defined in any order).

In this example, there are no hydraulic conditions to define (no phreatic line for example).

Note: this operation is not necessary for our example here, but it would also be possible to change the soil properties assigned to soil clusters (to simulate a ground improvement for example): to do that, double-click on a soil cluster (or right-click and select "Edit" from the pop up menu), and select the soil layer to be assigned (it must have been defined in the data mode).

Calculation conditions should now be defined for the first stage: we will consider here 2 different situations (i.e. set of conditions).
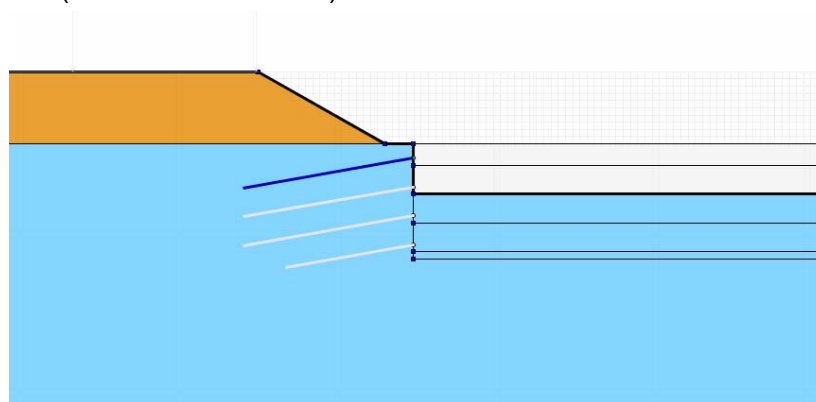
Figure 19: definition of the 1st construction stage

Tip:
- o *To easily identify stages, it is possible to change their name: select the first stage, then click again with the left mouse button on the stage name: the name of the stage is edited, and you can change it. You can also right click on the stage name, then choose "Rename", and type in the new name for the stage.*

1.1.7.1 Definition and calculation of situation 1

The first situation (i.e. set of conditions) for each stage is created automatically. It is however compulsory to edit and define its properties (for example partial safety factors and failure surfaces to be explored).

To access the situation properties, select menu [Stages and conditions], option [Define selected set of conditions] or right-click on the situation title, and choose option [Define selected set of conditions] from the pop up menu. The dialogue box of Figure 20 appears.

- Calculation method: Bishop (project default option as indicated in the General Settings).

- Partial safety factors: Clouterre Fundamental/standard (project default option as indicated in the General Settings).

- Seismic: we will not define seismic conditions for this situation.

- Definition of failure surfaces: we will choose for example for this first situation circular failure surfaces passing at the excavation bottom, with an automatic search. The requested data appear on Figure 20 and Figure 21 (the dialogue box of Figure 21 is displayed after selection of "Circular surfaces" and "Automatic search", and click on button "Define failure surfaces").

The value "Min abscissa for emergence" equal to -1.00 guarantees that Talren will not search for circles which a right emergence point "before" X = -1.00. For our example, we don't want to check the stability of the upper slope in layer 1: we want to check the general stability for the whole project. Validate.

Note: The number of slices is by default equal to 100. The project below has been defined with 150 slices to show that the number of slices can be increased.
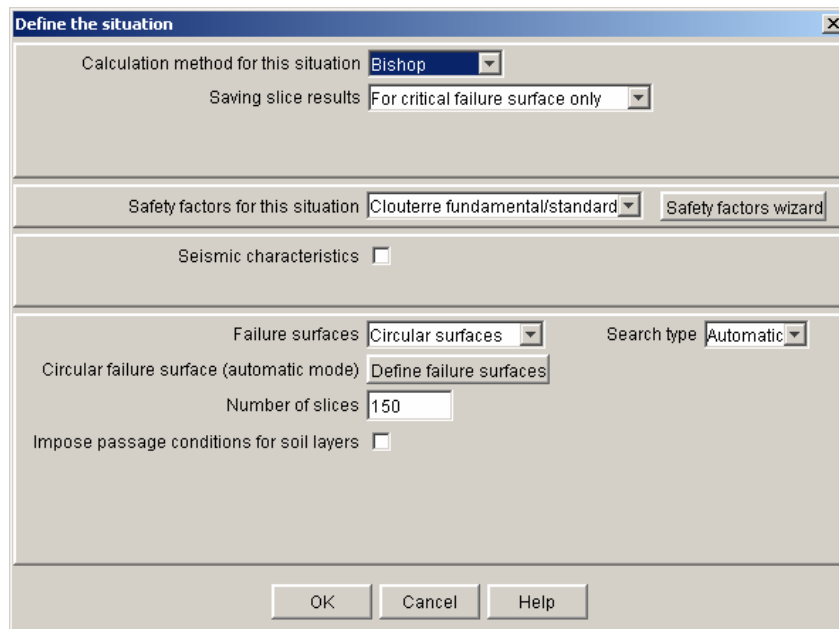
Figure 20: definition of the 1st situation for the 1st stage
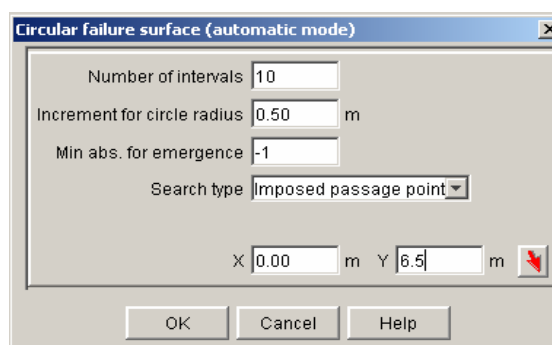
Figure 21: definition of failure surfaces for the 1st situation of the 1st stage

Tips:

o *To easily identify the situations, it is possible to change their names: select the situation, then left click with the mouse again on the situation name, and change its name, or right-click on the situation name, then choose "Rename", and define the new name for the situation.*

o *The button ![icon] enables to click the passage point on the graphical view (rather than typing in its coordinates).*

Representation of the imposed passage point
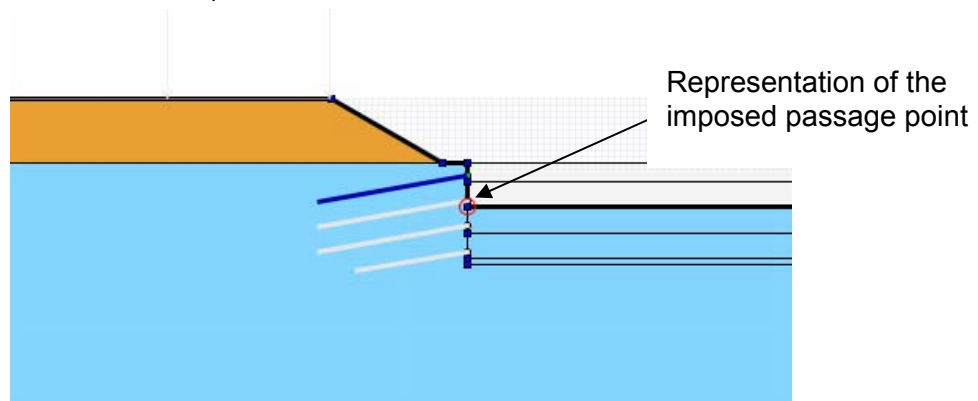
Figure 22: graphical view after definition of the 1st situation for the 1st stage

Calculation and results

• To calculate the situation, choose menu [Calculation and results] then option [Calculate the selected set of conditions], or click on the toolbar button ![Σ icon]. The calculation is performed and the result is displayed (Figure 23): the red circle(s) are those corresponding to the critical failure surfaces (smallest calculated safety factor).
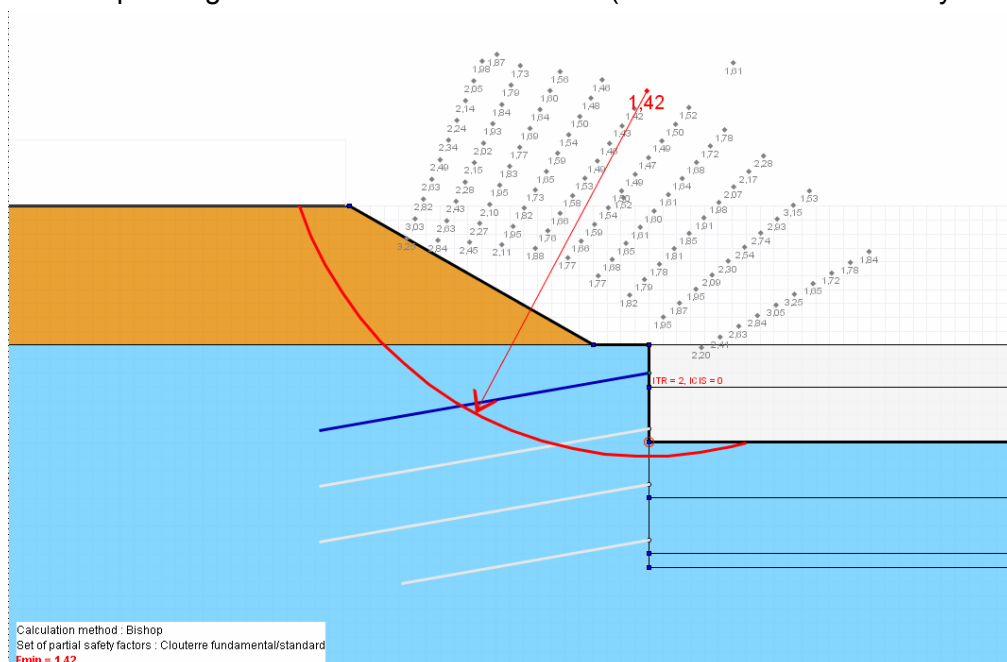
Figure 23: default output for situation 1 of stage 1

• The safety factor value displayed next to each centre is the minimal value obtained for all circles calculated with the same centre (if you defined a radius increment). Click on a centre to view the circle corresponding with the minimal safety factor obtained for this centre.

- The button ![icon] enables to copy the graphical view to the clipboard in order to paste it in another application (in a report for example).

- In order to change the settings for the output display, choose menu [Calculations and results] then option [Graphical display settings for output], or click on the toolbar button ![icon]. You can also right-click in the graphical view, and select this same option. The dialogue box shown on Figure 24 is displayed. It enables to request the display of:

  o All calculated circles;
  o All circles corresponding to a certain range of safety factor values;
  o Isovalues (colour shadings): here also, a range of safety factor values can be defined;
  o Indications on the design criterion for the reinforcements: this display, if requested, consists in texts "ITR = xxx; ICIS = xxx". These codes and their legends (usual reinforcements codes in Talren) can be viewed from the dialogue box for detailed reinforcements results).
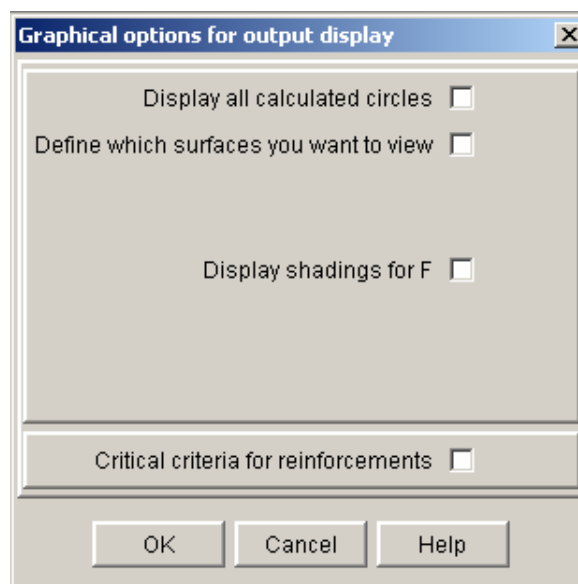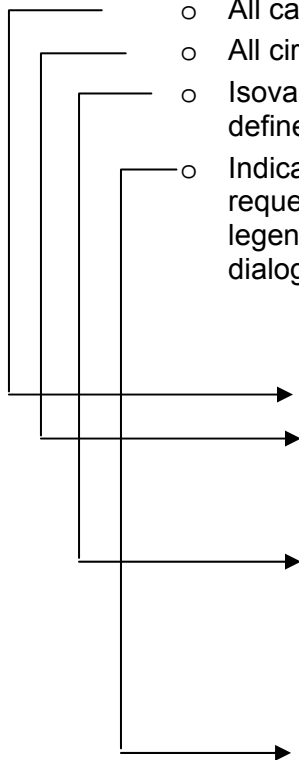
Figure 24: graphical output display settings
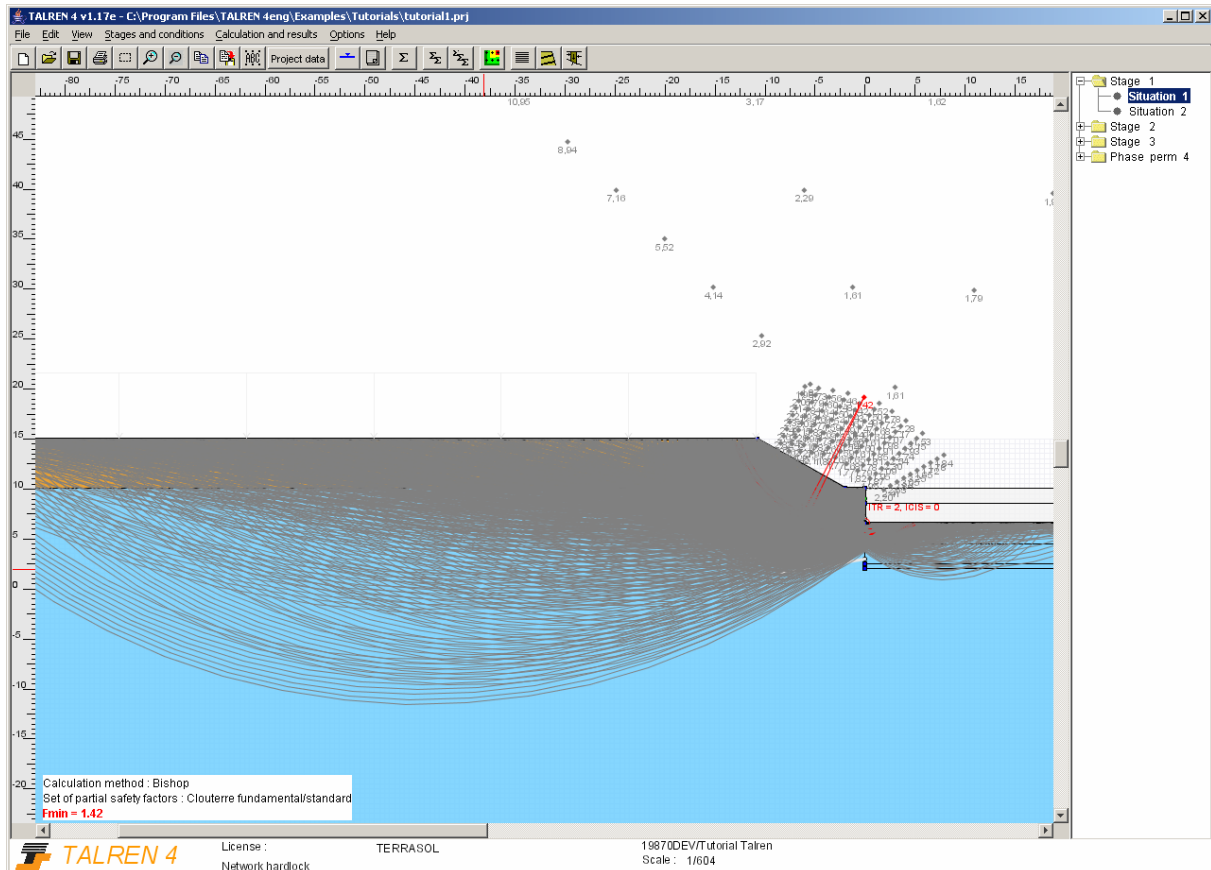
Figure 25: display of all the calculated circles (option "Display all…")

- It is possible to display other types of results, essentially as tables:

  o The table of detailed results for failure surfaces can be displayed with menu [Calculation and results], option [Detailed output], or with the toolbar button ☰: Figure 26. The green highlighted line corresponding to the critical failure surface is selected when opening the dialogue box. The table contains only the most critical failure surface obtained for each centre: if there are radius increments, the program calculates the minimum safety factor obtained for all circles with the same centre, and displays only the circle associated with this minimum. The checkbox "Display all results" below the table enables to request the display of all calculated circles.

  o The table of detailed results for reinforcements can be displayed with menu [Calculations and results], option [Forces in reinforcements], or with the toolbar button ⬛: Figure 27. The button "Legend for reinforcements" on this screen enables to view the legend for the different columns of the table.

  o The table of detailed results for slices can be displayed with menu [Calculation and results], option [Slices results], or with the toolbar button ⬛. This dialogue box enables to view for the critical circle the table of detailed results for slices, and the curve of normal stresses and pore pressures along the failure surface.

Detailed output (circular failure surfaces)

Export...

X0(m)	Y0(m)	Radius(m)	MMOT(kN)	F-SOL	F-SURCH	F-TOTAL
-1.94	14.34	9.07	4673.563	1.426	1.426	1.587
-1.45	14.94	10.06	5895.107	1.417	1.417	1.545
-0.97	15.54	10.08	5165.872	1.310	1.310	1.500
-0.48	16.14	10.64	5421.16	1.283	1.283	1.486
0	16.74	11.23	5686.034	1.258	1.258	1.474
0.48	17.34	11.84	5957.264	1.256	1.256	1.487
0.97	17.94	12.47	6229.332	1.255	1.255	1.499
1.45	18.54	13.12	6500.556	1.264	1.264	1.520
-4.07	13.48	9.07	5548.748	1.835	1.835	1.883
-3.62	14.11	9.92	6799.489	1.720	1.720	1.758
-3.18	14.74	9.82	6087.945	1.582	1.582	1.663
-2.73	15.37	10.27	6386.662	1.489	1.489	1.584
-2.29	16	10.76	6682.17	1.422	1.422	1.530
-1.84	16.62	11.29	6980.622	1.373	1.373	1.493
-1.4	17.25	11.84	7301.656	1.327	1.327	1.457
-0.95	17.88	12.42	7621.185	1.294	1.294	1.434
-0.51	18.51	13.02	7941.572	1.272	1.272	1.422
-0.06	19.14	13.14	6920.607	1.200	1.200	1.418
-5.27	13.61	9.34	5773.022	2.105	2.105	2.106
-4.87	14.27	10.16	7035.947	1.948	1.948	1.948
-4.47	14.93	10.53	7375.062	1.810	1.810	1.824
-4.07	15.58	10.45	6652.192	1.669	1.669	1.728
-3.67	16.24	10.91	6958.814	1.575	1.575	1.650
-3.27	16.9	11.4	7262.647	1.502	1.502	1.591
-2.87	17.56	12.42	8803.028	1.476	1.476	1.542
-2.47	18.22	12.47	7894.095	1.388	1.388	1.502
-2.07	18.88	13.05	8193.71	1.359	1.359	1.484
-1.67	19.54	13.64	8516.504	1.326	1.326	1.462
-6.43	13.7	10.65	7856.82	2.450	2.450	2.450

☐ Display all results

| OK | Help | Legend for F values |

Figure 26: detailed results for failure surfaces

Forces in reinforcements (circular failure surfaces)

X0(m)	Y0(m)	Radius(m)
-0.95	17.88	9.92
-0.95	17.88	9.42
-0.95	17.88	8.92
-0.51	18.51	12.02
-0.51	18.51	12.52
-0.51	18.51	13.02
-0.51	18.51	13.52
-0.51	18.51	14.02
-0.51	18.51	11.52
-0.51	18.51	11.02
-0.51	18.51	10.52
-0.51	18.51	10.02
-0.51	18.51	9.52
-0.06	19.14	12.64
-0.06	19.14	13.14
-0.06	19.14	13.64

Nails

Name	LU(m)	TR(kN)	ITR	IPTR	Tc(kN)	ICIS	IPCI
Nail 1	5.18	152.77	2	1	0	0	0

Forces in reinforcements should be considered for all failure surfaces and are not always maximum for the critical failure surface.

| OK | Help | Legend for F values | Legend for reinforcements |

Figure 27: forces in reinforcements

1.1.7.2 Definition and calculation of situation 2

First of all, a new situation should be added to stage 1. Several methods are available: menu [Stages and conditions] then option [Add a new set of conditions], or right-click on the first situation of the first stage then select [Add a new set of conditions]. It would have also been possible, in a similar way, to duplicate or copy/paste situation n° 1.

It is then possible to change the name of the new situation or to define its properties as for situation n° 1.
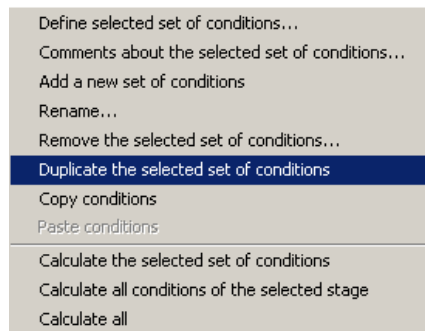
Figure 28: example of pop up menu (right-click on a situation)
Access to options "Duplicate" or "Copy/Paste"

Note: the duplication of conditions only works "inside" one same stage. However, the copy/paste option enables to copy a set of conditions in one stage and to paste it into another stage.

- We will define the same properties as for situation 1, except for failure surfaces: we will use manual search.
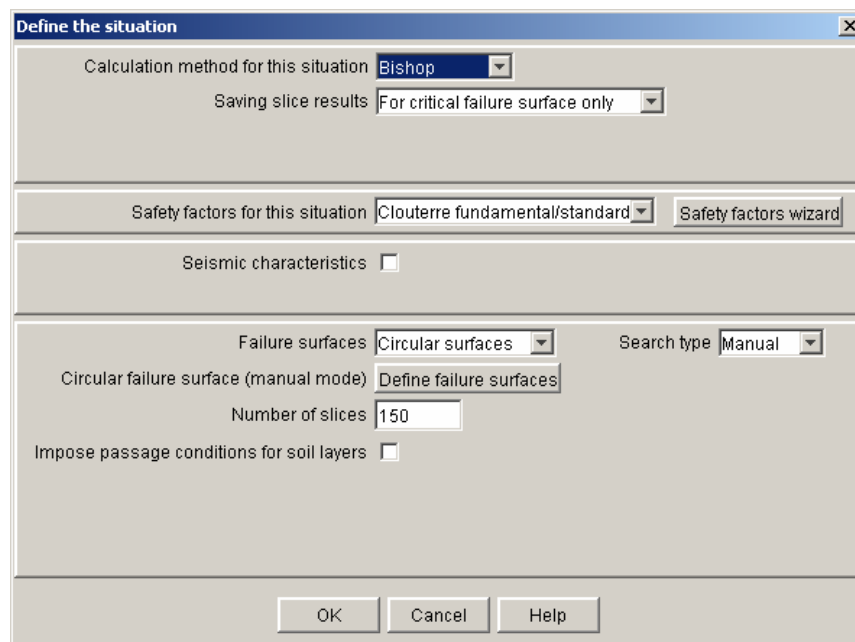
Figure 29: definition of situation 2 for the 1st stage

- When clicking on the button "Define surfaces", the dialogue box of Figure 30 is displayed. Fill in the settings with the values indicated on this figure (this will allow for a more accurate scan of the circles around the minimum safety factor located with the automatic search of situation 1).
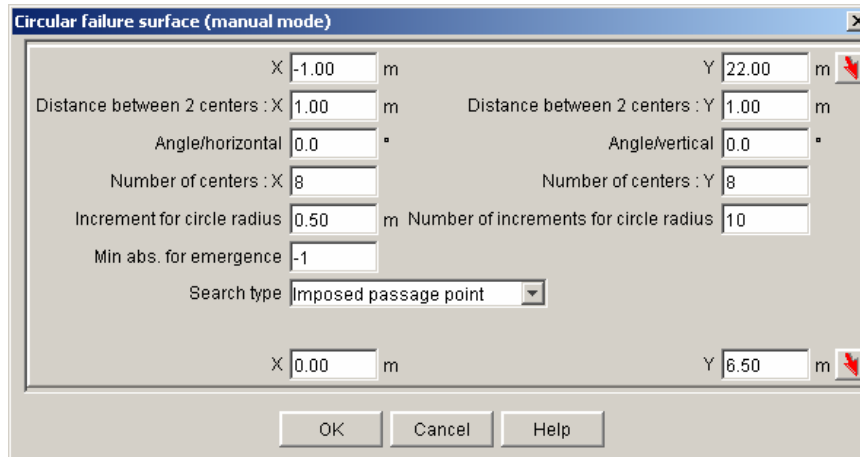
Figure 30: definition of failure surfaces for situation 2 of the 1st stage

Calculation and results

Proceed similarly as for situation 1. The output for situation 2 is displayed on Figure 31.

Figure 31: default ouput for situation 2 of stage 1

1.1.8. Step 8: definition of stage 2

This construction stage corresponds to the following operations:
- Installation of the 2nd nail level;
- Excavation to level 4.50.

To add a stage, select menu [Stages and conditions] then choose item [Add a new stage], or right click on stage 1, then select [Add a new stage]. By default, the new stage is created with the same configuration as the last stage (here as stage 1).

The definition of this stage configuration is done similarly as for stage 1: activation/deactivation of soil clusters and nails. The layout you should define is displayed on Figure 32.
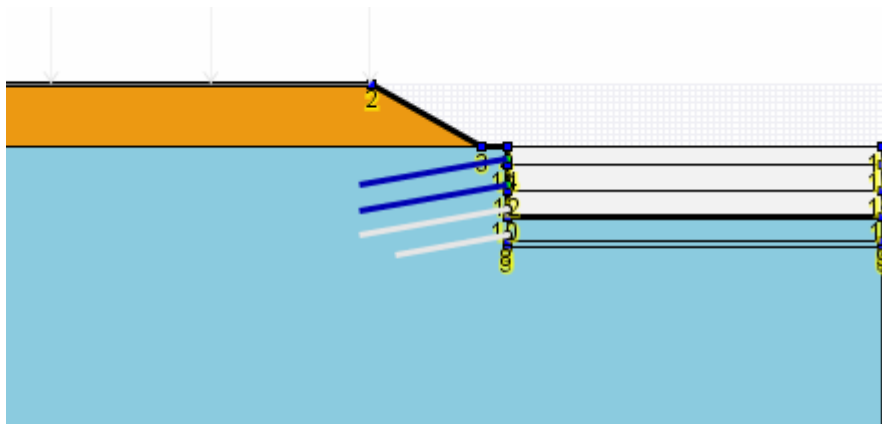
Figure 32: definition of 2nd calculation stage

We will consider one single situation for this stage.
The properties for the default situation (situation 1) of stage 2 should be defined:
- Partial safety factors: Clouterre fundamental/standard
- Seismic: we will not consider seismic conditions for this project.
- Definition of failure surfaces: we will choose here circular failure surfaces passing at the bottom of the excavation, with an automatic search. The requested data are provided below: the position of the passage point is Y = 4.50 instead of Y = 6.50. The number of intervals and the radius increment have been changed for the example.
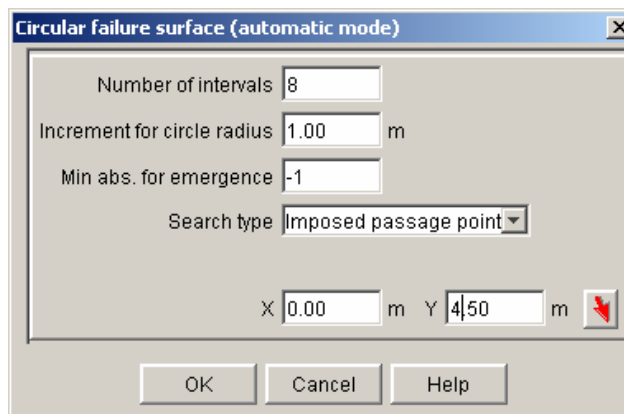
Figure 33: definition of situation 1 for the 2nd calculation stage

Calculation and results

Proceed similarly as for situation 1 of stage 1. The output is displayed on Figure 34.
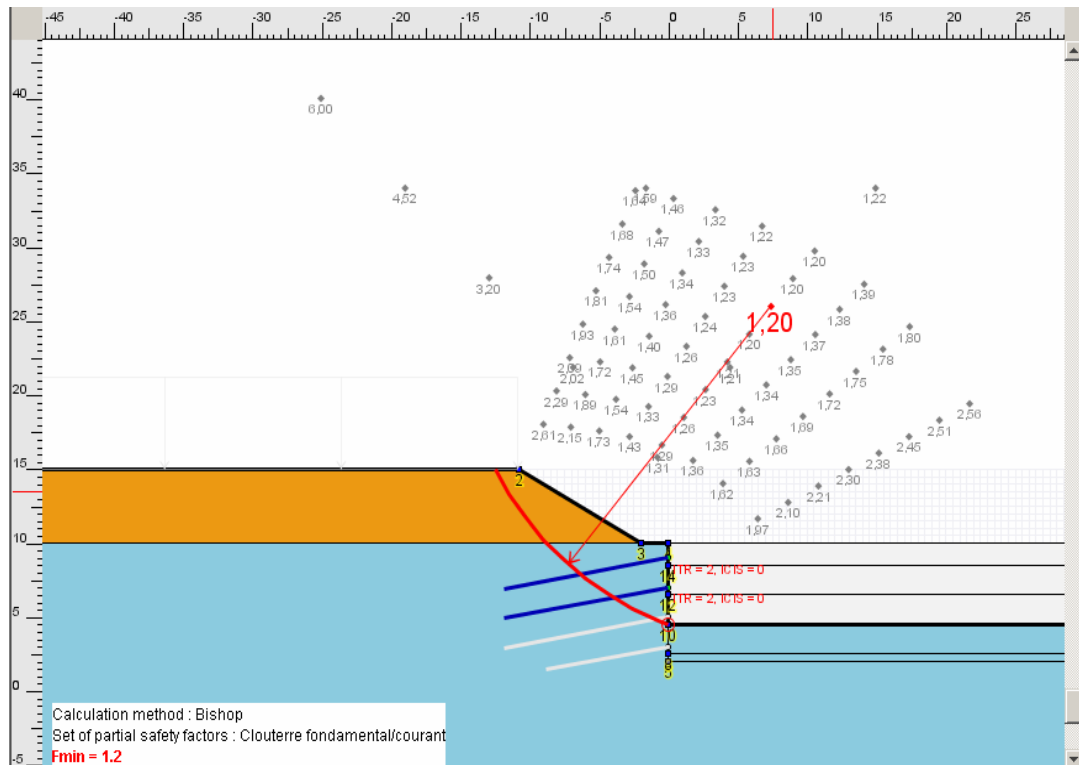
Figure 34: default output for situation 1 of stage 2

1.1.9. Step 9: definition of stage 3

This construction stage corresponds to the following operations:

- Installation of the 3rd nail level;
- Excavation to level 2.50.

The definition of this stage can be done similarly to that of the previous stages: activation/deactivation of soil clusters and nails. The configuration you should define is displayed on Figure 35.
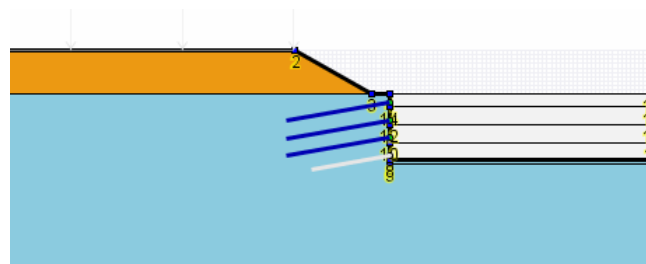
Figure 35: definition of 3rd calculation stage

We will first consider a first calculation situation, using automatic search.

Define the properties for the default first situation, with the same settings as for situation 1 of stage 2, except for the position of the passage point (level equal to 2.50), and for the number of intervals, chosen here with value of 10 in order to reduce the calculation time.

Calculation and results

Proceed the same way as before. The output is displayed on Figure 36.
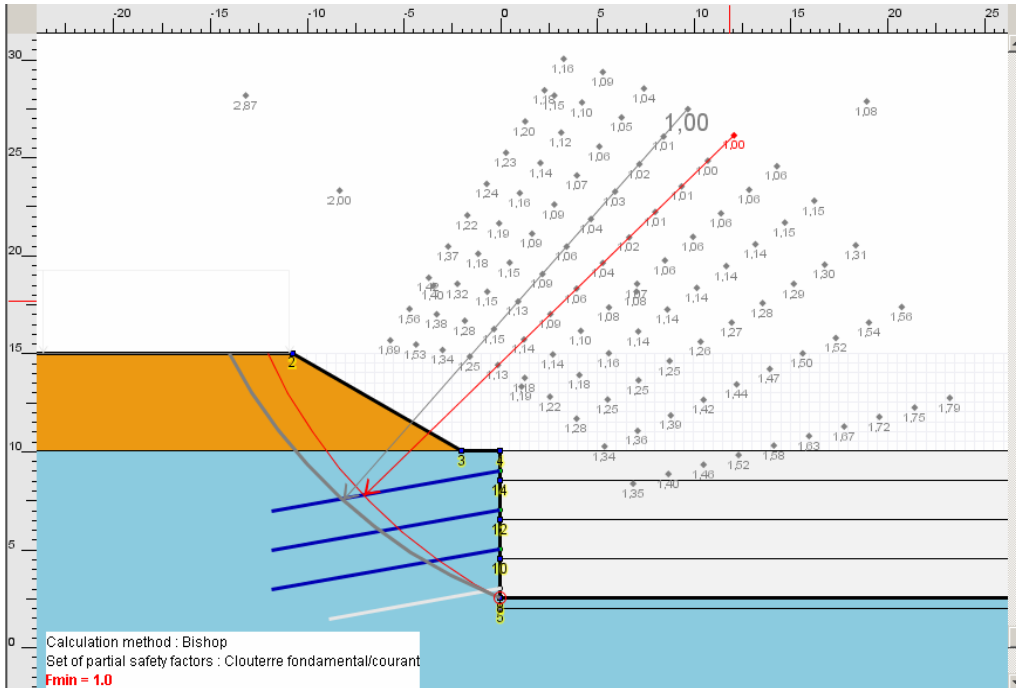
Figure 36: default ouput for situation 1 of stage 3

The scan above doesn't look accurate enough, and we have to examine a 2nd situation with a manual search: we will define a grid for the centres around the critical centre located with the automatic search. The result of this 2nd situation is provided on the figure below: the safety factor is less than 1.00.
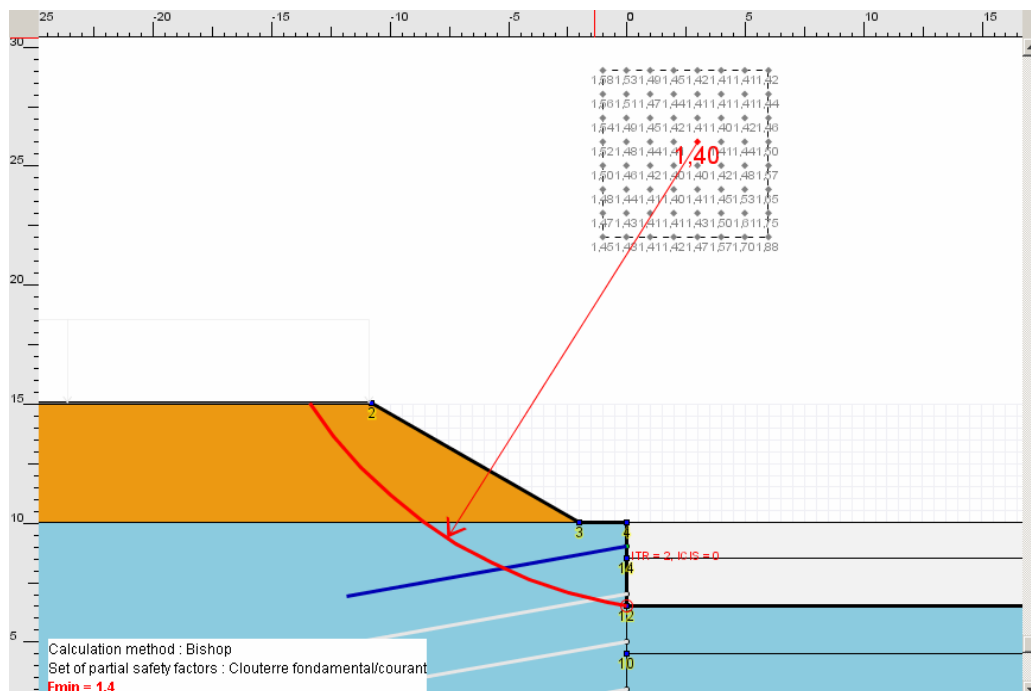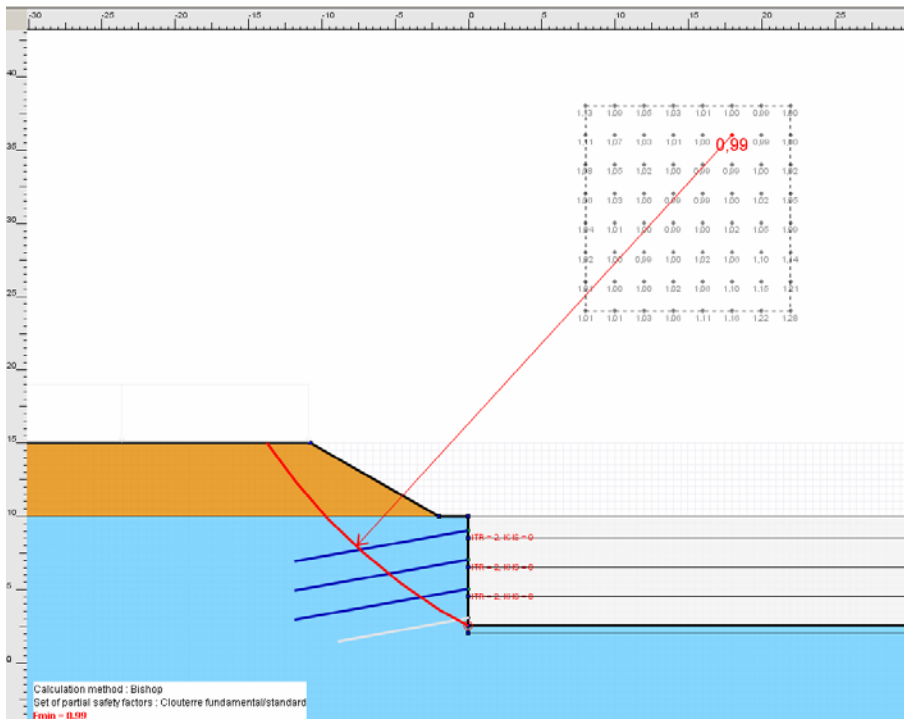

Figure 37: default output for situation 2 of stage 3

1.1.10. Step 10: definition of stage 4 (final stage)

This construction stage corresponds to the following operations:

- Installation of the 4th nail layer;
- Excavation to level 2.00.

The definition of this stage is done similarly to that of the previous stages: activation/deactivation of soil clusters and nails. The final layout is provided on Figure 38.
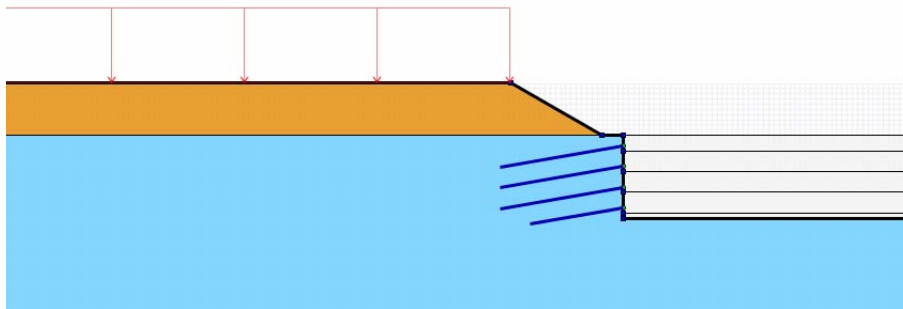
Figure 38: definition of the 4th calculation stage (final stage)

1.1.10.1 Definition of situation 1

We first consider a calculation situation with automatic search.

Properties for this first stage should be defined, with the same settings as for situation 1 of stage 1, except for the position of the passage point (level at 2.00).
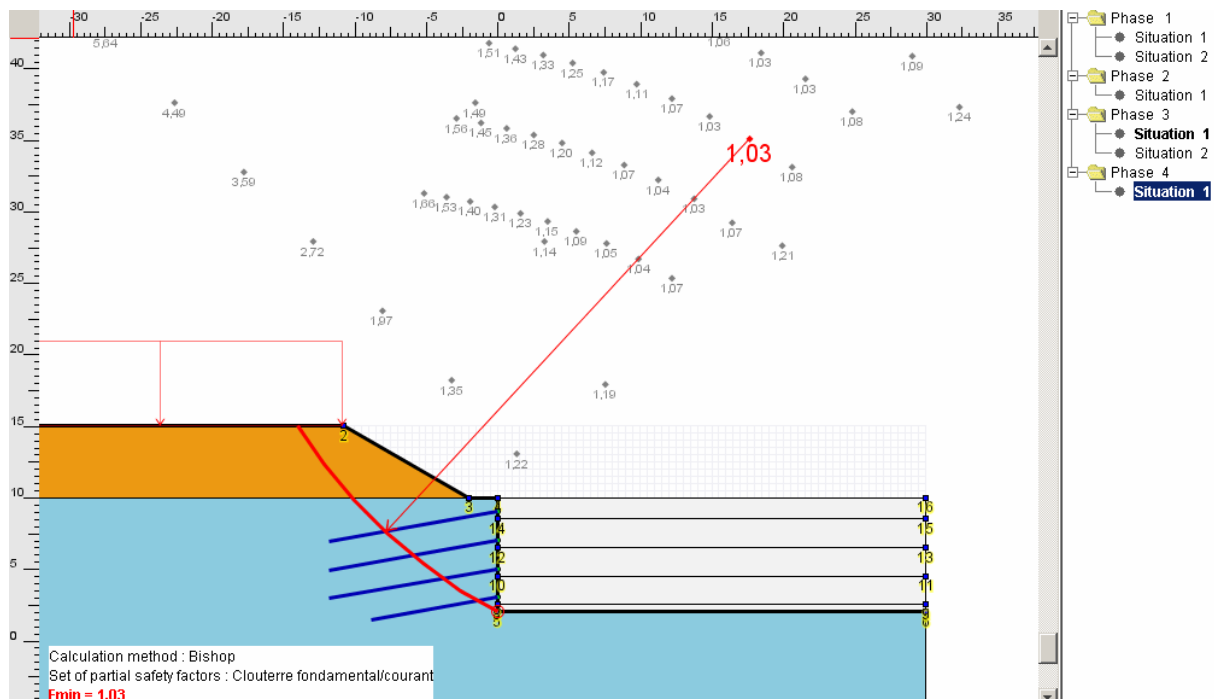
Figure 39: default output for situation 1 of stage 4

1.1.10.2 Definition of the situation 2

We will add a situation and define its properties:

- Partial safety factors: Clouterre Accidental/Standard. To change the set of partial safety factors, click on the button [Safety factor wizard], then follow the steps indicated on Figure 40.

  If the set "Clouterre Accidental/standard" wasn't defined in the Data mode, you will first have to switch to the Data Mode, add it to the Partial safety factors local database (use general settings or option "Partial Safety factors").
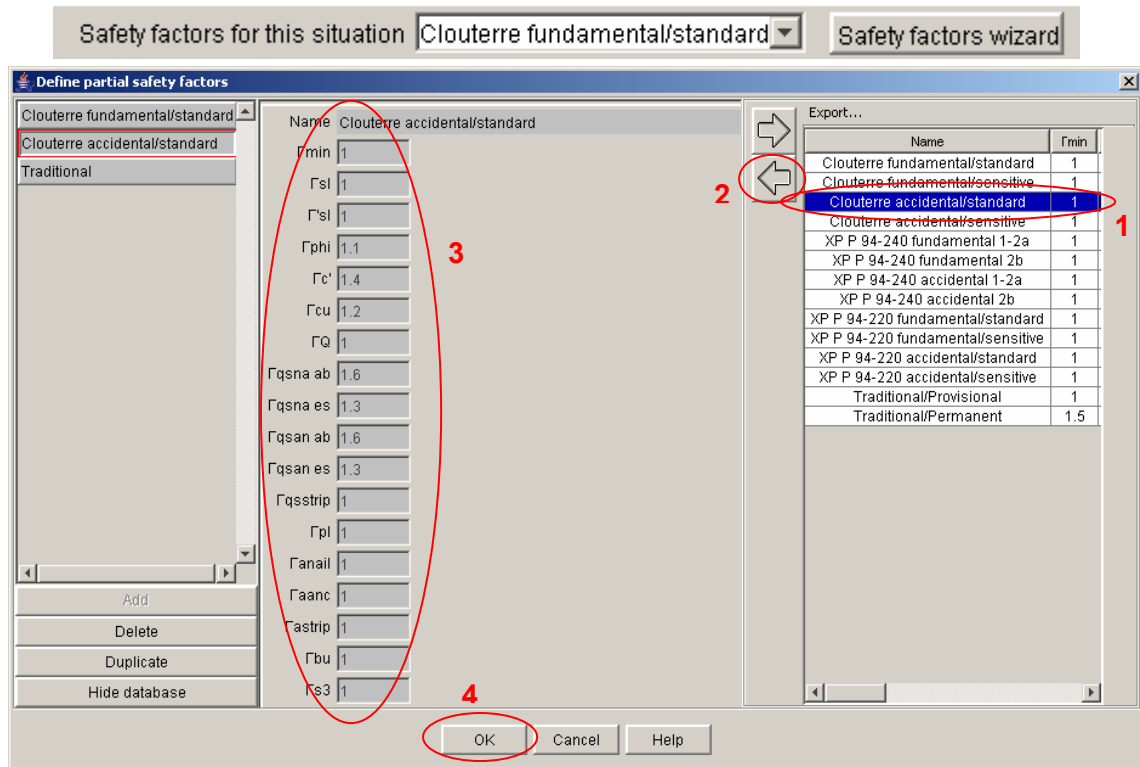
Figure 40: selection of a different set of partial safety factorsy

Important note: in the dialogue box above, the safety factors concerning the anchors, the strips and the braces have been defined in order to be able to validate the screen, but will not be used in the calculations for our example (only the factors for the nails are defined). These defined values are not in any way an example or a reference to use again: you should refer for each project to the relevant standards or recommendations, and thus to the corresponding values for the partial safety factors.

- Seismic conditions: we will consider here the accidental situation with earthquake. Therefore the checkbox "Seismic characteristics" should be activated, and the following values should be taken into account: $a_h/g = 0.12$ et $a_v/g = 0.06$.

  Important note: in the case of seismic sollicitations, it is advised to check the stability with the 4 possible sign combinations for the couple (a_h/g, a_v/g). Indeed, depending on the geometry, it is not always the same combination that leads to the most unfavourable results.

- Definition of failure surfaces: we will choose here the same failure surfaces as for situation 1.

Figure 41: definition of situation 2 for the 4th stage
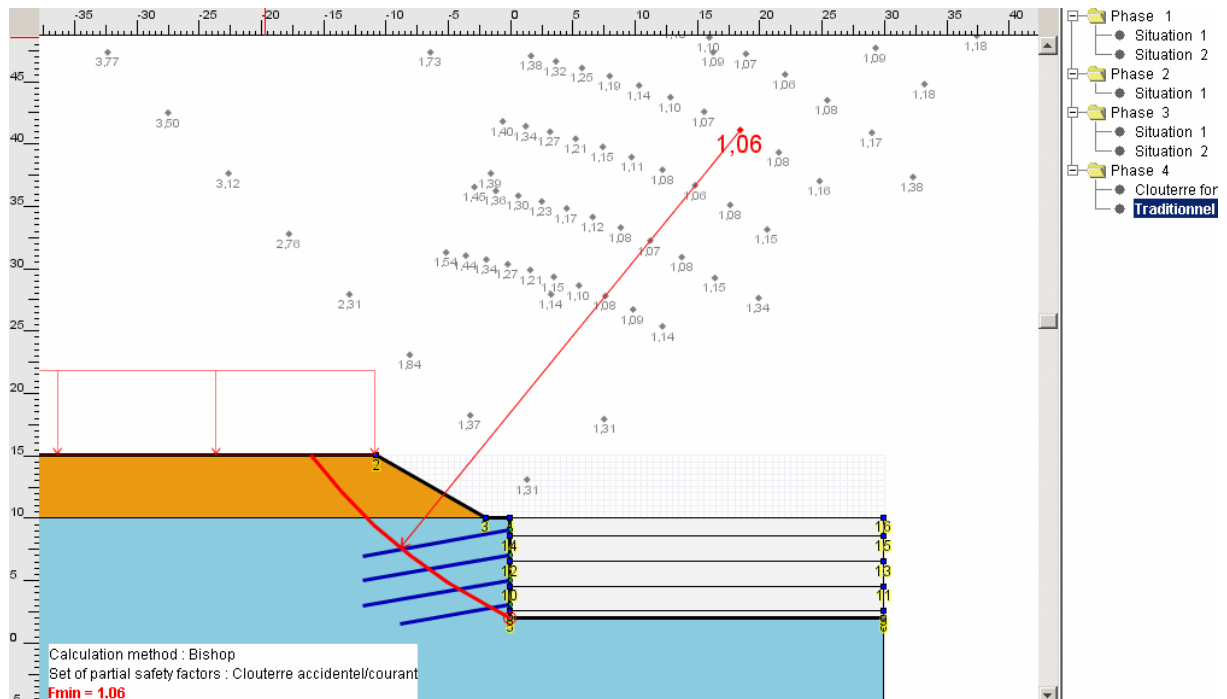
Figure 42: default ouput for situation 2 of stage 4

1.1.10.3 Definition of situation 3

Duplicate situation 1 (of stage 4), and change the set of partial safety factors to "traditional method".

This set is not complete in the database provided with Talren 4: you should go back to data mode, choose menu [Options], then [Partial safety factors], add a new set of factors (Figure 43) then validate. Switch again to the stages mode, select situation 3 of stage 4, and edit the situation properties again: select this new set of partial factors.

Name	Traditional
Γmin	1
Γsl	1
Γ'sl	1
Γphi	1
Γc'	1
Γcu	1
ΓQ	1
Γqsna ab	2
Γqsna es	1.5
Γqsan ab	2
Γqsan es	1.5
Γqsstrip	1
Γpl	2
Γanail	1.5
Γaanc	1.5
Γastrip	1
Γbu	1
Γs3	1

Figure 43: example of partial factors for the traditional method

Important note: again, these values do not constitute in any way an example or a reference to use again: you should refer for each project to the relevant standards or recommendations and thus to the corresponding values of partial safety factors.
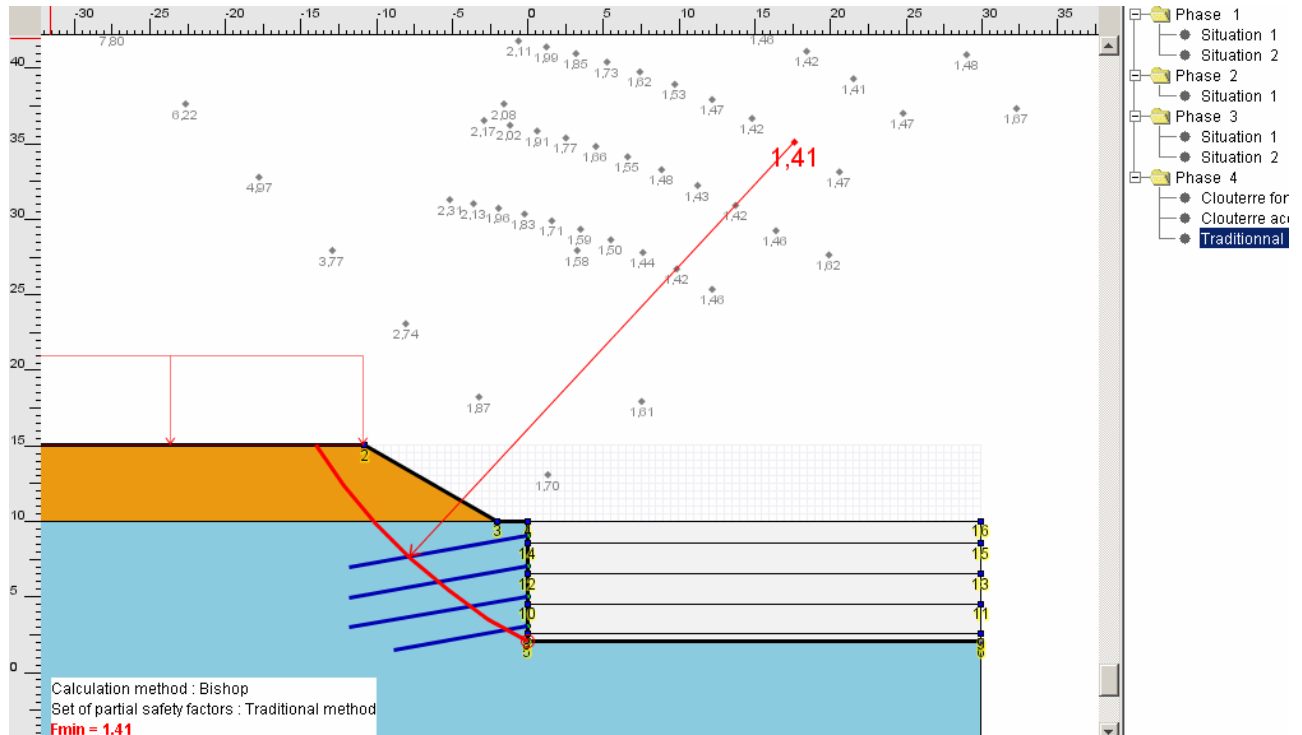
Figure 44: default output for the situation 3 of stage 4

1.1.10.4 Calculation and results for the 3 situations of stage 4

We will calculate the 3 situations at the same time:

- Select stage 4 (click on its name).

- Choose menu [Calculation and results], then option [Calculate all conditions of the selected stage], or click the toolbar button ![Sigma Sigma].

- The 3 situations are then calculated successively. To view the results, click successively on each situation.

- Please note that the names of the situations that have been calculated (i.e. for which results are available) are displayed with a bold font.

- If you change the properties of a situation, the corresponding results will be automatically erased. In the same way, if you change the properties of a stage (for example activation/deactivation of a soil cluster), the results of all situations for this stage will be erased, etc.

- Select menu [File] then [Save].

1.2. Tutorial 2: stabilisation of a slope with piles (loading of a .tal file)

1.2.1. Step 1: presentation of the project

We will check the stability of a slope:

- In its natural state;
- Considering 2 rows of piles.

This example was already provided with version TALREN 97, as an example file (exampl22e.tal).

We will load for this tutorial this .tal file, and work from this file with TALREN 4.

1.2.2. Step 2: loading file examp22e.tal

- Run TALREN: use Windows [Start] menu or click on the desktop icon.
- Select menu [File] then option [Open] or click on the button in the toolbar.
- Select the file extension: TALREN 97 (.tal). Then select file examp22e.tal (provided in the folder "/Examples/Examples tal" in the Talren 4 installation directory.
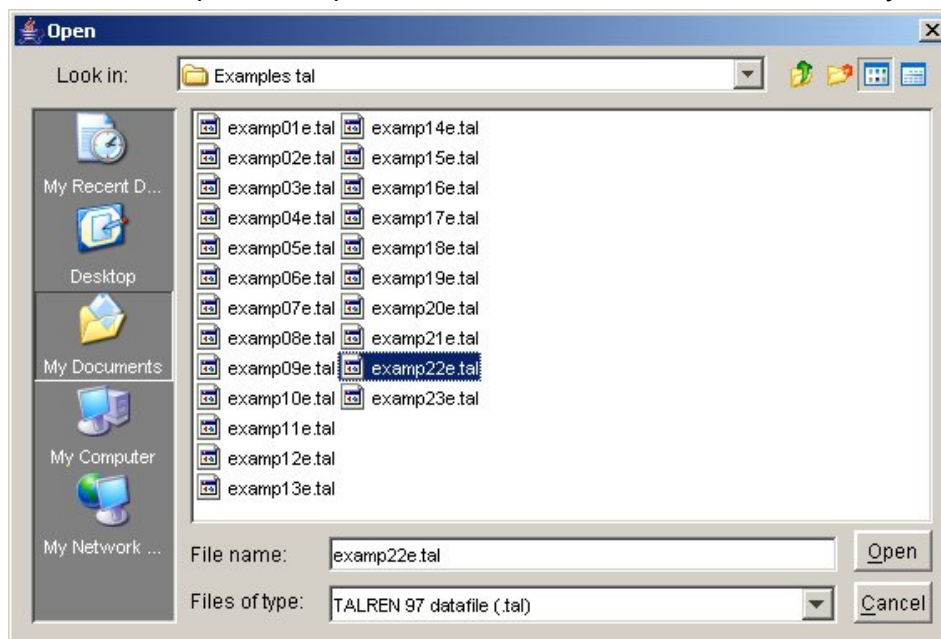
Figure 45: opening file examp22e.tal

- Select menu [File] then option [Save] or click on the toolbar button . This file has to be saved with the prj format (it's not possible to save it with the former format .tal). Select the directory and the filename (by default, Talren 4 proposes filename examp22e.prj. We chould change it here for example to tutorial2.prj).

Tip: It is possible using menu [File] to define or change the default working directory (used when opening ans saving files).

1.2.3. Step 3: data changes to carry out

The graphical view enables to check that the geometry was loaded and that the geometrical data are consistent with the Talren 97 file. Both figures below compare graphical views displayed by TALREN 97 and TALREN 4: geometry, soil layers, reinforcements, etc. indeed are the same.
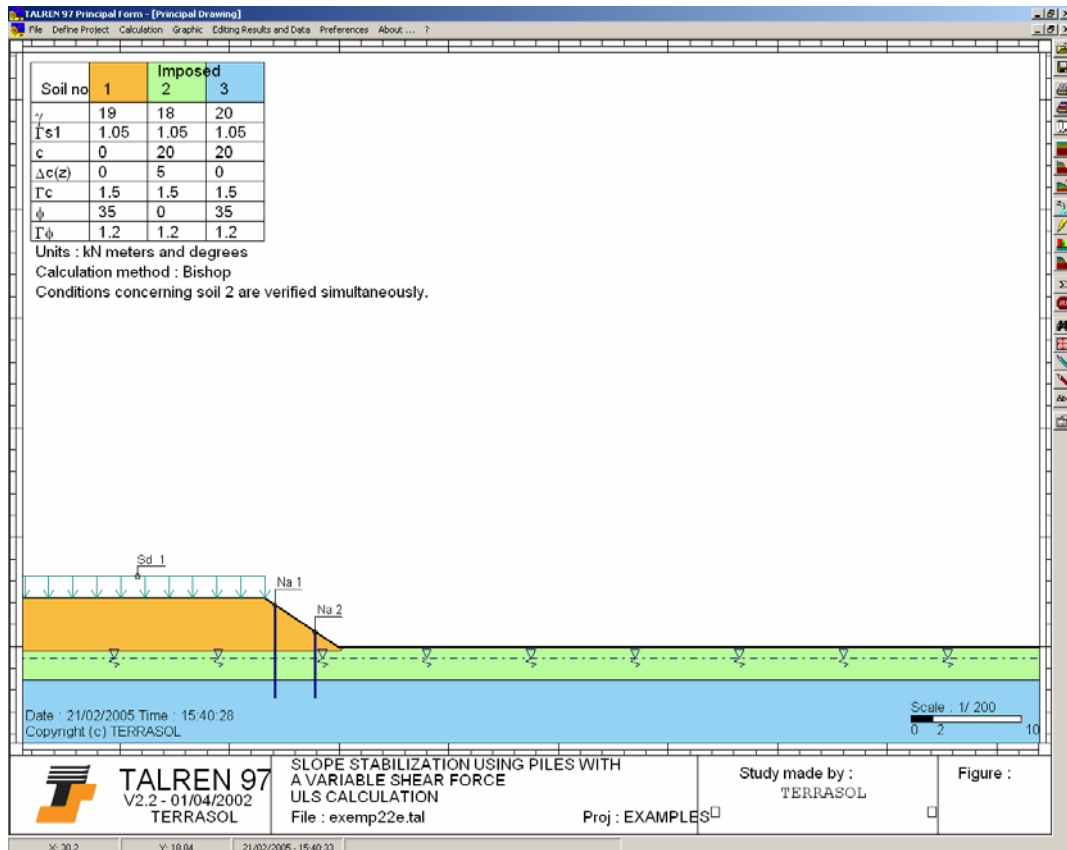
Figure 46: examp22e.tal file, displayed with Talren 97
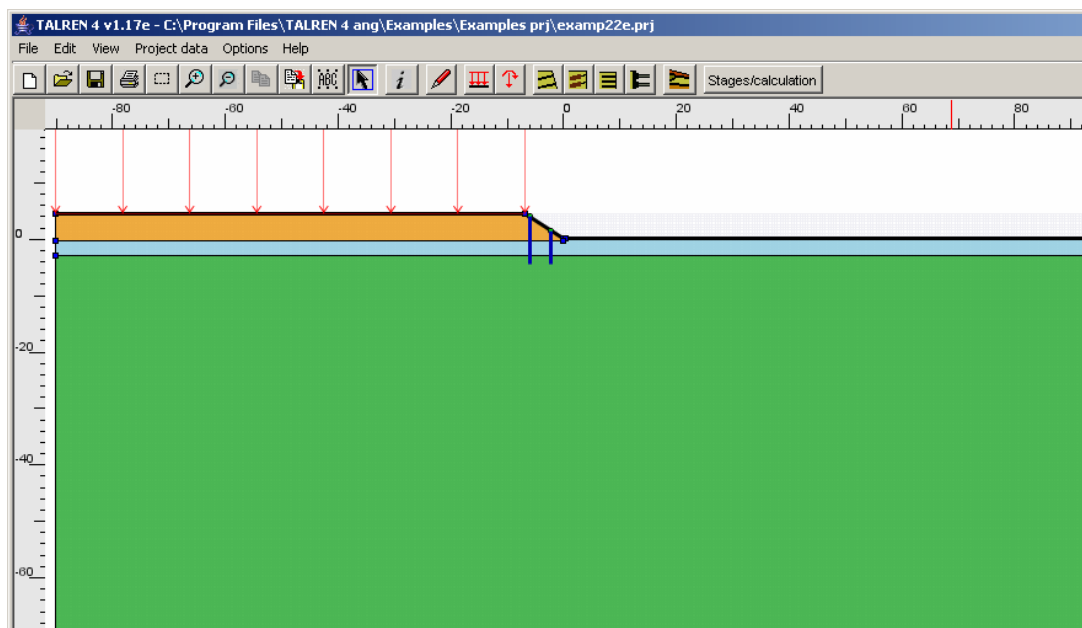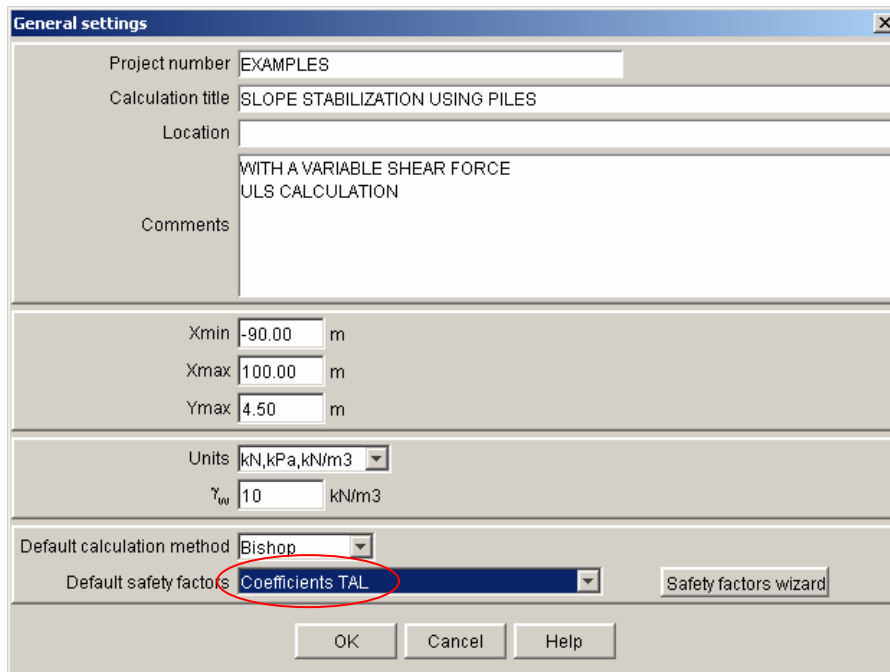
Figure 47: examp22e.prj file, displayed with Talren 4 (Project data mode)

We will browse a few data dialogue boxes to draw your attention on some specific aspects of .tal file loading.

- *General settings*: the titles, comments and units are those defined in the .tal file. The model dimensions are automatically initialised depending on the .tal file geometry, as well as the default calculation method and the default partial safety factors (figure below).

Figure 48: examp22e.prj file, general settings

For each .tal file loaded with Talren 4, a set of partial safety factors is automatically created: it includes all the partial factor factors defined in the .tal file.

Figure 49: examp22e.prj file, partial safety factors "Coefficients TAL"

- *Geometry*: click on menu [Project data] then on option [Geometry]; you can view the project points and segments. Click on the [Slope boundary] tab. When loading a .tal file, the slope boundary is never set to "automatic" mode (Talren 4 reads the slope boundary defined in the Talren 97 file). Except for special cases (slope boundary on purpose different from the groundlevel), it is recommended to activate the automatic mode (as shown below).

Figure 50: examp22e.prj file, slope boundary definition

- *Loads*: there is no special comment about loads here: a distributed load is defined in the .tal file, and is read without any problem by TALREN 4.

- *Reinforcements*: the project includes nails (simulating piles), for which shear only is to be taken into account (no traction). Shear is defined as variable along the nail.
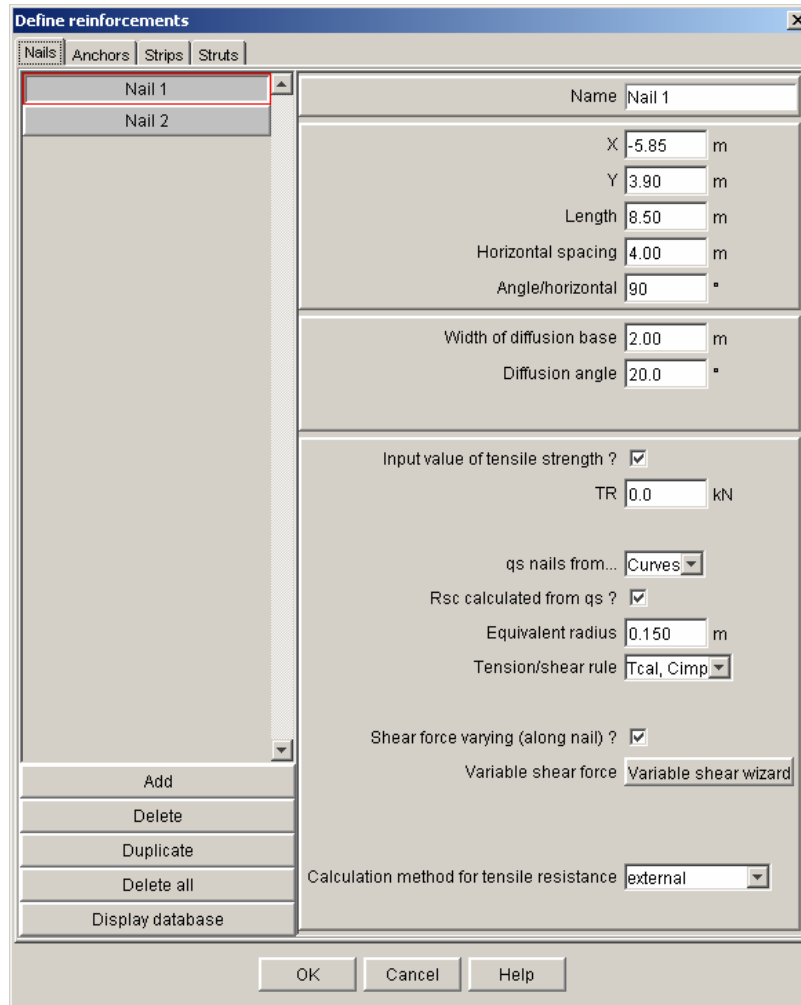
Figure 51: examp22e.prj file, properties of nail 1
(1st pile)
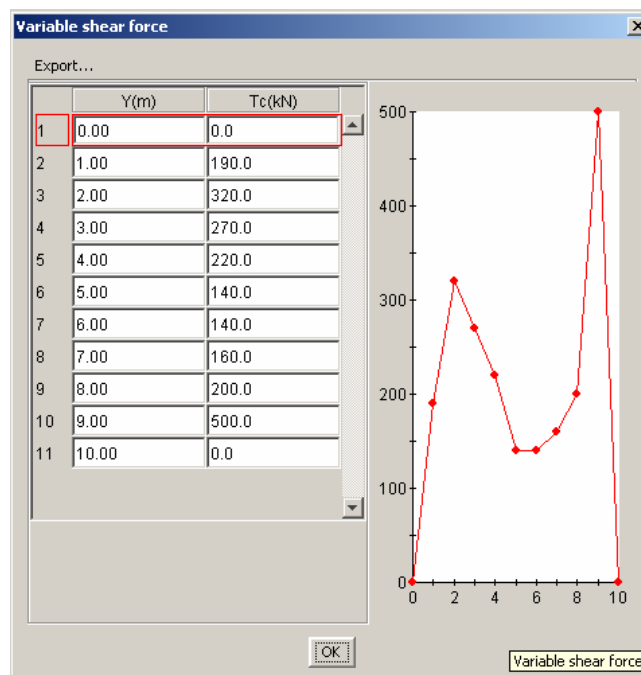
Figure 52: examp22e.prj file, variable shear along nail 1
(1st pile)

- *Soil properties*: the important point here is the use of the "specific partial safety factors" when reading a .tal file. Indeed, Talren 97 allowed you to define partial safety factors, but individually for each soil layer (not as full sets of factors associated with a given standard). Talren 4, when reading .tal files, can not therefore associate a set of partial safety factors (for example Clouterre) wirh the .tal file data. Moreover, it was possible in the .tal files to define different safety factors for each soil layer. Hence the use of these specific safety factors for each layer in Talren 4: they will replace during the calculation, for each layer, the partial factor set chosen in each situation, and thus guarantee that the calculation will give the same result with Talren 4 as previously with Talren 97.

Nevertheless, if the .tal file is consistent for example with a Clouterre calculation, fundamental/standard (i.e. if all soil layers use these same partial safety factors), it is possible to deactivate the checkbox "specific partial safety factors" for each layer, and then use for each situation the set of factors "Coefficients TAL" (automatically created), or even the predefined set Clouterre fundamental/standard for example).

Be careful: in this case, you should select for each layer the relevant cohesion type (effective or undrained); this choice decides which safety factor will be applied to the cohesion value. In the same way, you should check for each layer if the weight is considered as a favourable or unfavourable action. Finally, in the nail properties window, you should indicate whether the qs_{nails} values are issued from curves or from tests (all these elements also decide choices for partial safety factors to apply).
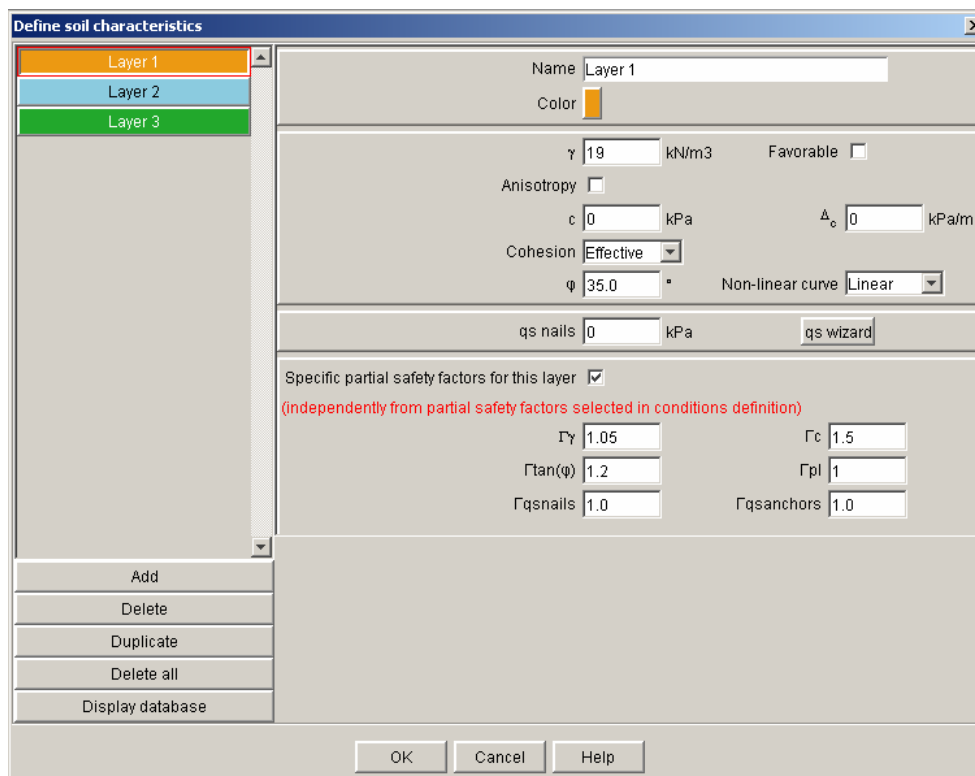
Figure 53: examp22e.prj file, soil properties

1.2.4. Step 4: definition of the stages

When reading a .tal file, Talren 4 automatically creates a stage and a situation, which are consistent with the calculation options of the .tal file: in our case, the piles and load are activated, the failure surfaces are circular, with a manual search.

But we want to consider 2 different stages: one without the piles and the other with the piles. It is therefore necessary to either add or insert a second stage. The option we chose for this example is to insert a stage, in order to define stage 1 without piles, and stage 2 with the piles.

It will be necessary for each stage to complete the hydraulic conditions.

- Switch to the Stages/calculation mode, using the menu [Project data] or the toolbar button.

- Select stage 1 (left click once on the stage label).

- Click on the button ⬚ to access the hydraulic conditions definition. The option "Phreatic level" is selected. Click on the button "Define phreatic level" to view the definition of the phreatic level. A dialogue box displaying the table of the points describing the phreatic level pops-up, but it contains only one point: a unique point was enough in Talren 97 to define a horizontal phreatic level. However, in Talren 4, it is necessary to define the phreatic level from X_{min} to X_{max}. So it is necessary to add a point with coordinates (100, -1,0). The dialogue box is shown in Figure 54.
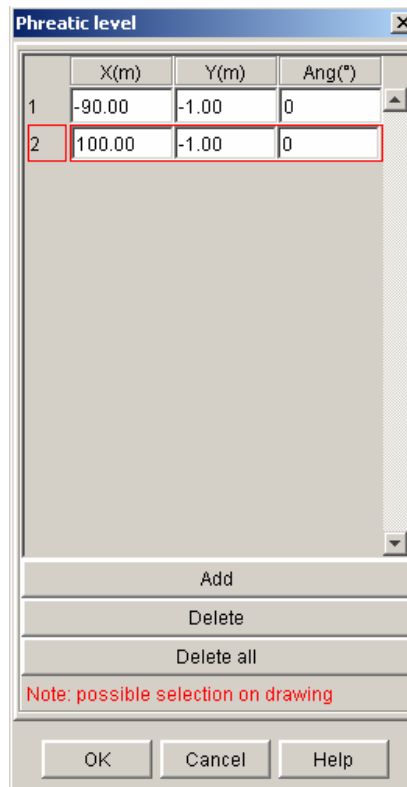
Figure 54: examp22e.prj file, definition of the phreatic level

- Then, to insert a stage, select menu [Stages and conditions], option [Insert a new stage], or right click on stage 1, and choose [Insert a new stage] from the pop up menu.

- Talren 4 inserts a new stage in position 1. It is initialised identically to the project data configuration (because it is inserted before stage 1), i.e. with all soil clusters activated, reinforcements and loads deactivated, and without any hydraulic conditions. It is therefore necessary to activate the load (click on the load base line),

  and define the hydraulic conditions (phreatic level): click on the toolbar button ⬚, select option "Phreatic level" then click on the button "Define phreatic level", and define a water table at level -1.00, from X_{min} to X_{max} (like before). The definition of stage 1 is displayed on Figure 55.

- The other stage (now stage 2) remains unchanged after insertion of stage 1: select it to display the corresponding graphical view: the piles and load are active (Figure 56), and the water level is still defined.
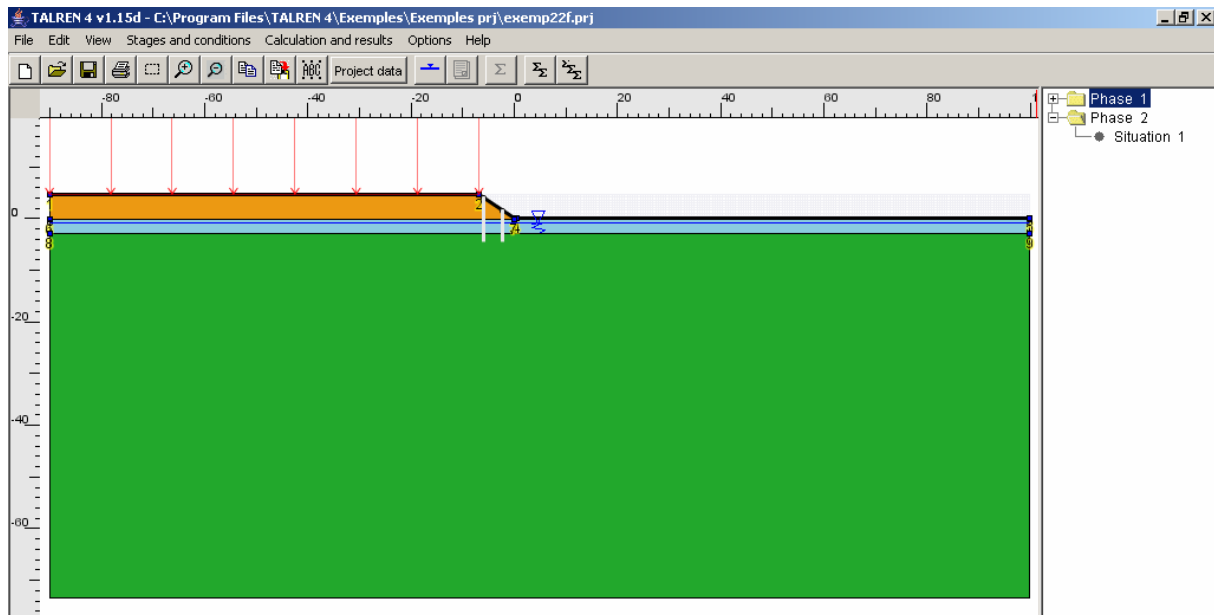
Figure 55: examp22e.prj file, definition of stage 1 (without piles)
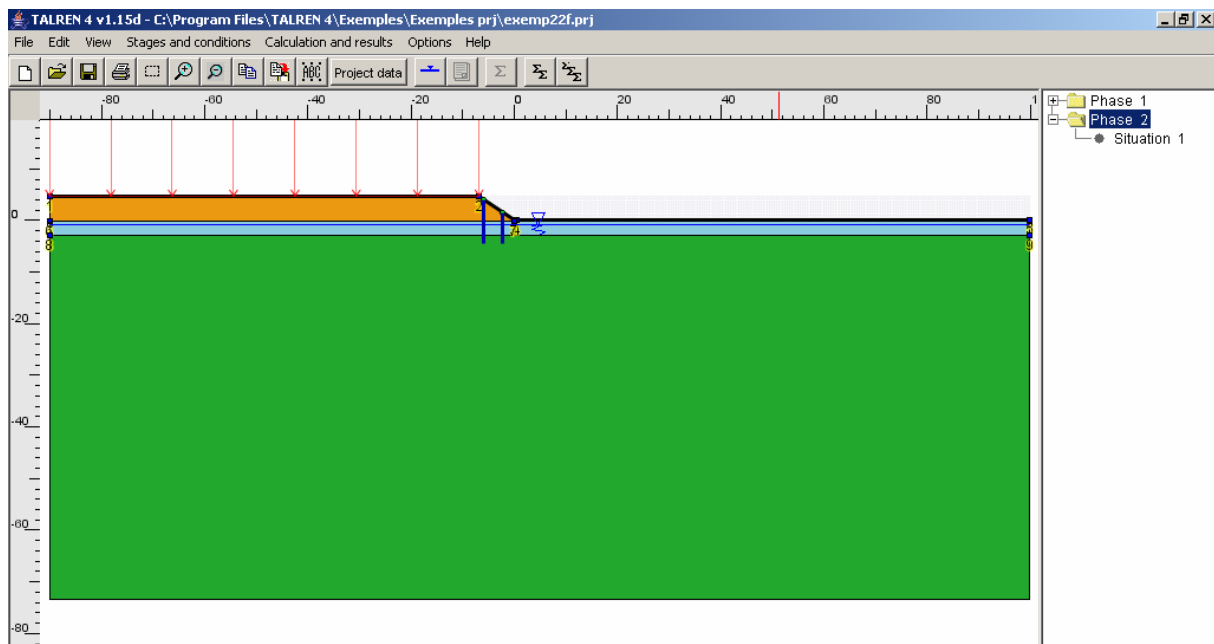
Figure 56: examp22e.prj file, definition of stage 2 (with piles)

1.2.5. Step 5: calculations for stage 1

We have to define at least a set of conditions (i.e. a situation) to allow calculations for stage 1.

The situation created by default is "empty".

But the situation 1 of stage 2 is fully defined (settings loaded from the .tal file). We will therefore copy/paste this situation:

- Click on situation 1 of stage 2.

- Right-click on this situation, and choose the option [Copy conditions].
- Click on stage 1, then right click on stage 1, and select [Paste conditions]: the pasted situation is added in second position.
- Delete situation 1 ("empty") of stage 1: click on it, then right click and choose [Remove the selected set of conditions] (or click on the [Del] button on the keyboard).
- To view the settings of situation 1, right click on it and choose [Define the selected set of conditions] in the menu. All the settings were initialised from the .tal file (Figure 57). To view failure surfaces, click on the button [Define failure surfaces]. Here too, all the settings are initialised from the .tal file (Figure 58).
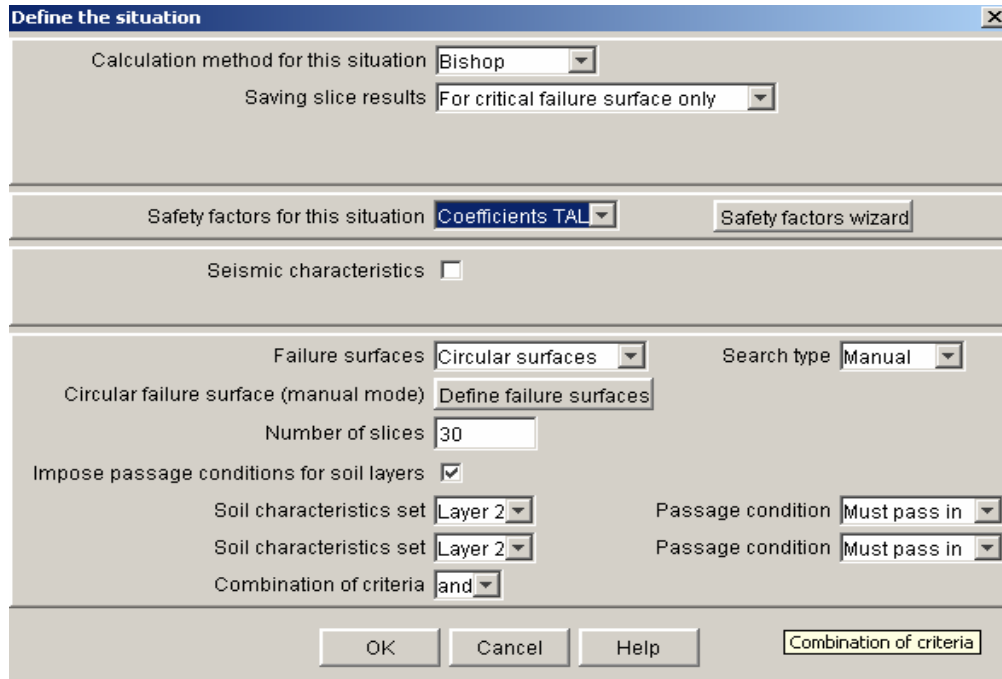
Figure 57: examp22e.prj file, definition of situation 1 for stage 1
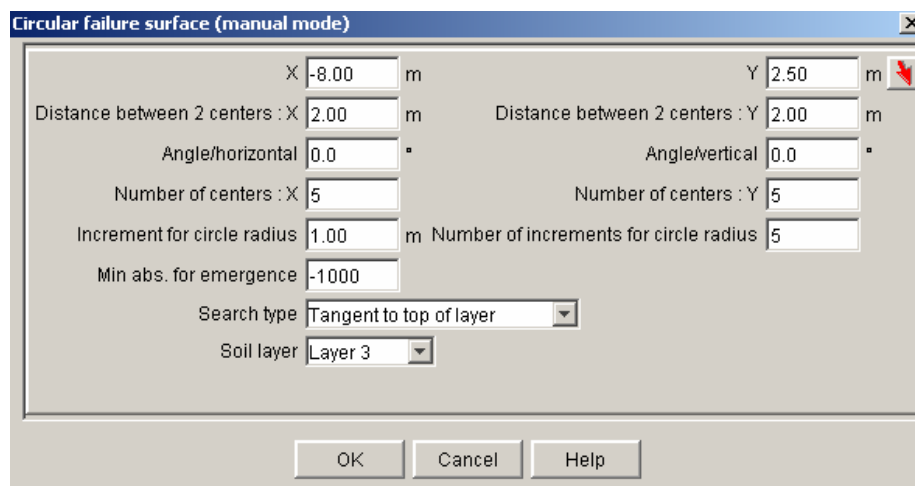
Figure 58: examp22e.prj file, definition of situation 1 for stage 1 (failure surfaces)

- Calculate the situation of stage 1: click for example on the toolbar button Σ. The results are displayed on Figure 59. The calculation provides F_{min} = 0.85 (unstable slope).
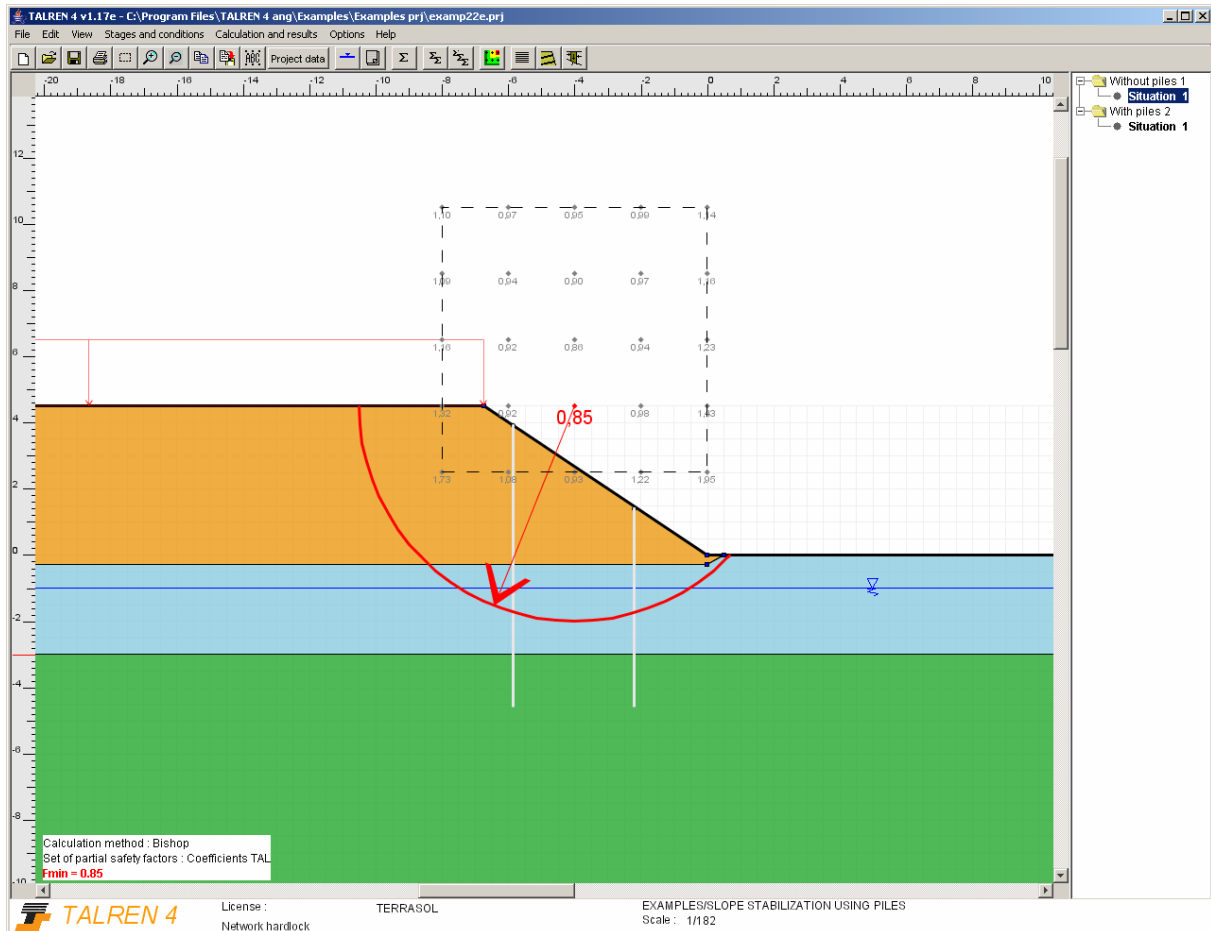
Figure 59: examp22e.prj file, default ouput, stage 1 / situation 1

You can define additional situations (copy/paste this first situation): for example, you can define a situation with an automatic search of circles tangent to soil layer 3, or an automatic search with an imposed passage point.

1.2.6. Step 6: calculations for stage 2

The set of conditions is already completed (loaded from .tal file).

- Calculate the single situation of stage 1: click for example on the toolbar button Σ. The result is displayed on Figure 60. The calculation provides F_{min} = 1.00: same result as the result obtained with Talren 97 for the examp22e.tal file.
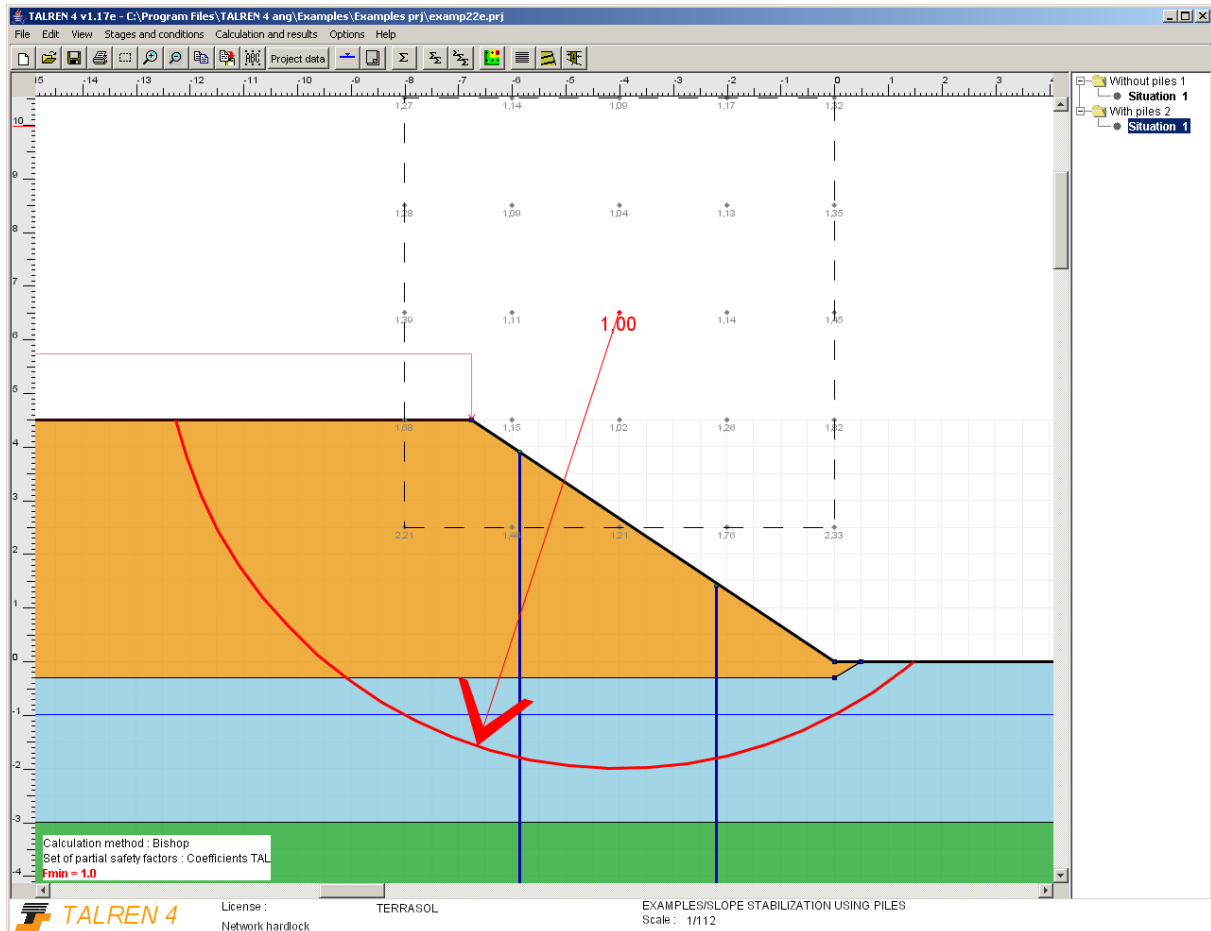
Figure 60: examp22e.prj file, default output, stage 2 / situation 1

You can define additional situations (copy/paste this first situation): for example, you can define a situation with an automatic search of circles tangent to soil layer 3, or an automatic search with an imposed passage point.

1.2.6.1 Other output types

As indicated for tutorial 1 (refer to this tutorial for further details), it is possible to change the settings for ouput display, as well as to display additional results:

- In order to change the settings for the output display, choose menu [Calculations and results] then the option [Graphical display settings for output], or click on the toolbar

  button ▉. You can also right-click in the graphical view, and select this same option. The dialogue box enables to request the display of:
  - o All calculated circles;
  - o All circles corresponding to a certain range of safety factor values;
  - o Isovalues (colour shadings): here also, a range of safety factor values can be defined;
  - o Indications on the design criterion for the reinforcements.

- It is also possible to access other types of results (mainly in tables):
  - o The table of detailed results for failure surfaces can be displayed with menu [Calculation and results], option [Detailed output], or with the toolbar button ▤. The green highlighted line corresponding to the critical failure surface is selected when opening the dialogue box. The table contains only the most

critical failure surface obtained for each centre: if there are radius increments, the program calculates the minimum safety factor obtained for all circles with the same centre, and displays only the circle associated to this minimum. The checkbox "Display all results" below the table enables to request the display of all calculated circles.

o The table of detailed results for reinforcements can be displayed with the menu [Calculations and results], option [Forces in reinforcements], or with the toolbar button ![button]: Figure 61. The button "Legend for reinforcements" on this screen enables to view the legend for the different columns of the table.

o The table of detailed results for slices can be displayed with menu [Calculation and results] menu, option [Slices results], or with the toolbar button ![button]. This dialogue box enables to view for the critical circle the table of detailed results for slices, and the curve of normal stresses and pore pressures along the failure surface.
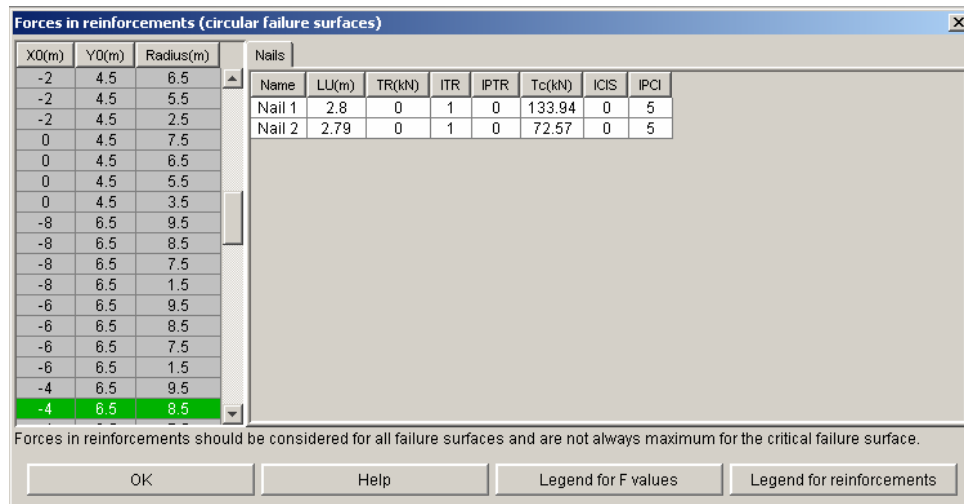
Figure 61: forces in the piles

1.3. Tutorial 3: stability of a slope with various hydraulic conditions

1.3.1. Step 1: presentation of the project and general settings

This example deals with the stability of a dike, for different hydraulic conditions: permanent groundwater level, short-term drawdown and long-term drawdown.

We will define a stage for each of these hydraulic conditions.

Note 1: in this example, we will not detail the operations as much as in tutorial 1: for further details, please refer either to tutorial 1, or to part B of the manual.

Note 2: for stages 2 and 3, this example is based on the importation of pore pressure meshes calculated with Plaxis v8. The Plaxis files to import are provided with the example files.

- Run TALREN: use Windows [Start] menu or click on the desktop icon.

- Select menu [File] then option [New] or click on the toolbar button ☐.

- Click on the button *i* or select menu [Project data] then option [General settings], and fill in the data for this dialogue box as shown on the figure below.

- For the default partial safety factors, click on the button [Safety factor wizard]. In the window that appears, click on [Add], then define the set of partial safety factors as shown on Figure 63.

  Important note: these partial safety factor values do not constitute in any way an example or a reference to use again: it is necessary for each project to refer to the relevant standards or recommendations, and to define the corresponding partial safety factors.
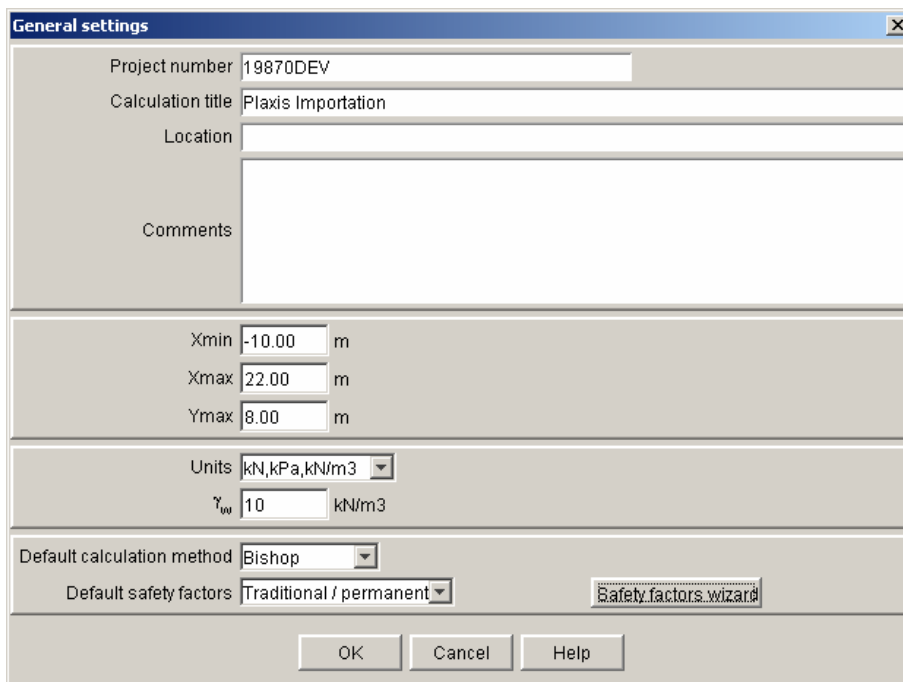
Figure 62: general settings
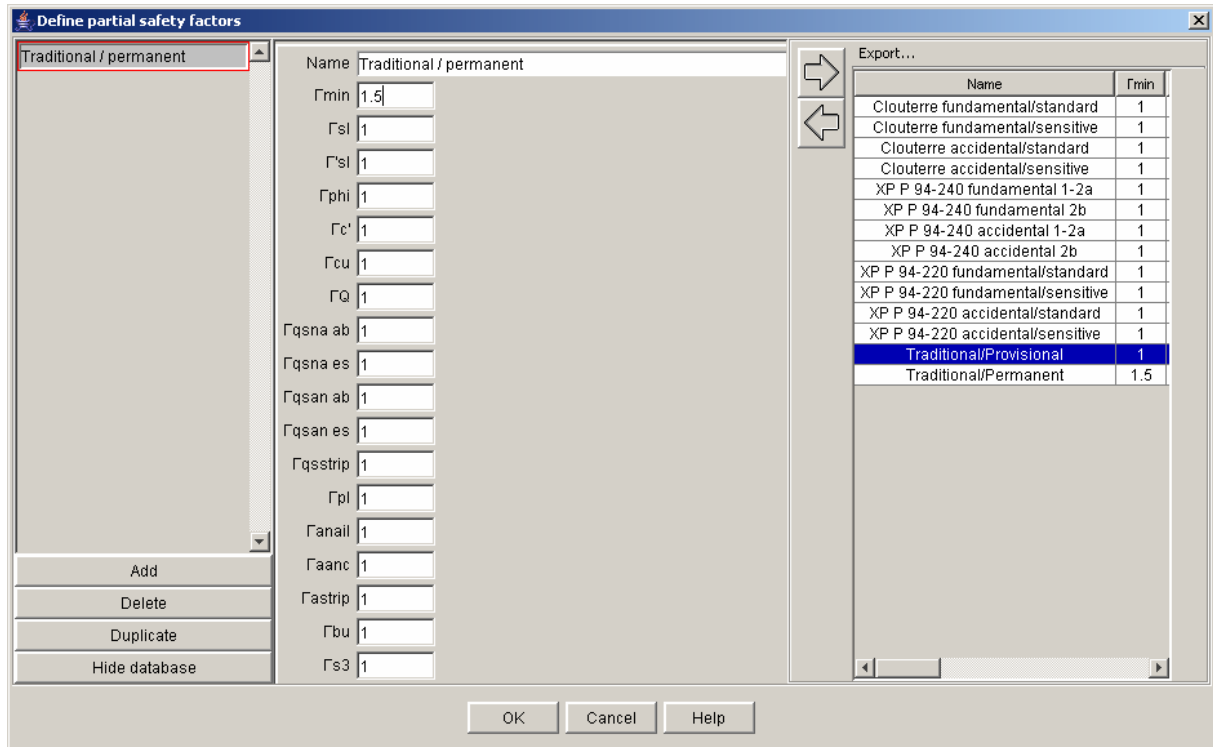
Figure 63: definition of partial safety factors

1.3.2. Step 2: geometry definition

The model dimensions and geometry are consistent with the figure below.

- To define this geometry in TALREN 4. use one of the 3 following methods: draw with the mouse, input the points coordinates directly in the main window, or use the geometry dialogue box.
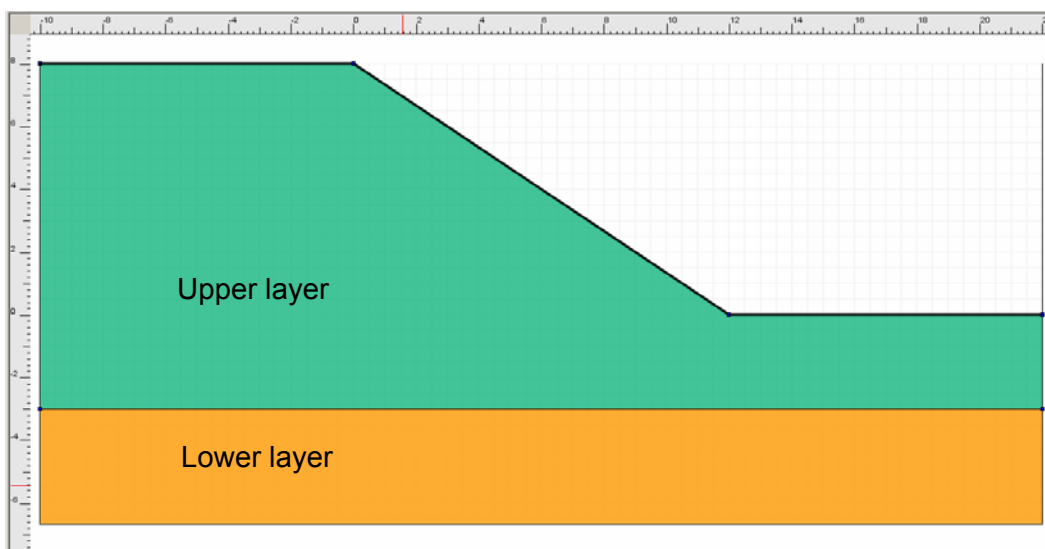
Figure 64: model geometry

1.3.3. Step 3: definition of the soil properties

Two soil layers should be defined. Their characteristics are given in Table 3.

Soil	γ (kN/m^3)	φ' (°)	c' (kPa)
1 (upper)	18	30	10
2 (lower)	22	35	1

Table 3: soil properties

- In order to define these characteristics, select for example menu [Project data] then [Soil characteristics]. Click on the button "Add", and fill in the data for the 1st layer (Figure 65). Then click again on "Add" and fill in the data for the 2nd soil layer. To switch from a soil layer to the other, click on the layer labels in the left list of the dialogue box.
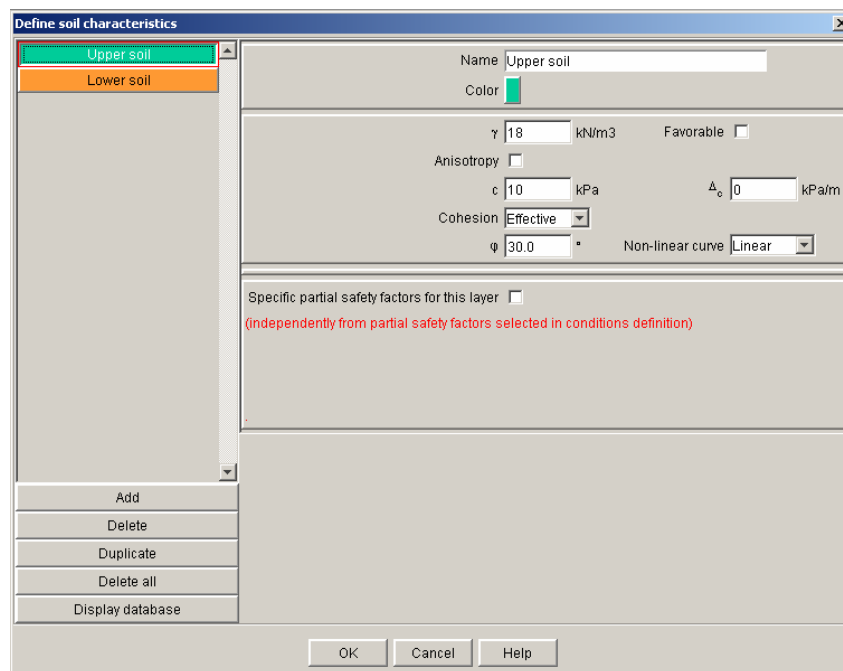
Figure 65: properties of the upper layer

- To assign these properties to soil clusters, use the drag & drop method (from the left list onto the soil clusters). Layer 1 is the upper layer, and layer 2 is the lower layer.
- The data input is now finished for this model.

1.3.4. Alternative method for steps 1, 2 and 3

Rather than defining the geometry and the soil characteristics, it would have been possible to open the Plaxis file corresponding to the same model. This could have been possible this way:

- Run TALREN: use Windows [Start] menu or click on the desktop icon.

- Select menu [File] and option [Open], or click on the toolbar button .

- Choose file extension .plx (Plaxis v8 files), then select the Plaxis file you want to open.

- The Plaxis model is then converted into a Talren 4 file, and a warning is displayed about the limits of this conversion (refer also to chapter B of this manual).

- The geometry is complete. When viewing the soil properties, you can check that they are also complete.

- Nevertheless, it is necessary to complete the project general settings (except for the model dimensions, automatically calculated with the model geometry). Click on the toolbar button $\boxed{i}$ and complete the missing data (Figure 62).

1.3.5. Step 4: definition of stage 1 (permanent groundwater level)

For this first stage, we will consider the permanent groundwater level at +6.00.

- To define it, switch to the Stages/calculation mode (using menu [Project data] or the toolbar button). The first stage is automatically created with all soil clusters activated.

- Click on the toolbar button $\boxed{}$ to open the hydraulic conditions dialogue box.

- Choose option "Phreatic level", and click on the button "Define phreatic level". A dialogue box with the table of phreatic level points is displayed. Define the groundwater level: either click on "Add" and input the coordinates of both points defining the horizontal phreatic level, or click on the graphical view at the position of both points. The data are displayed in Figure 66.

	X(m)	Y(m)	Ang(°)
1	-10.00	6.00	0
2	22.00	6.00	0

Add

Delete

Delete all

Note: possible selection on drawing

OK   Cancel   Help
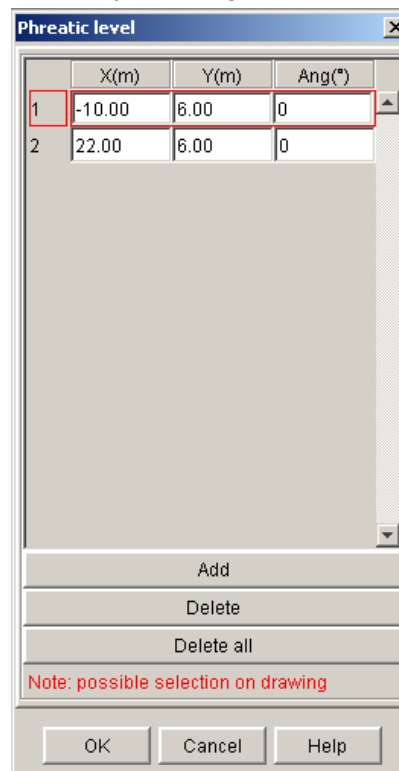
Figure 66: definition of the phreatic level

- An external water level should also be defined here. Activate the checkbox "Define an external water level", then click on the button [Define an external water level]. In the dialogue box that pops-up (Figure 67), activate the checkbox "Automatic calculation" (the external phreatic line will be calculated depending on the relative positions of the groundwater level and of the groundlevel).
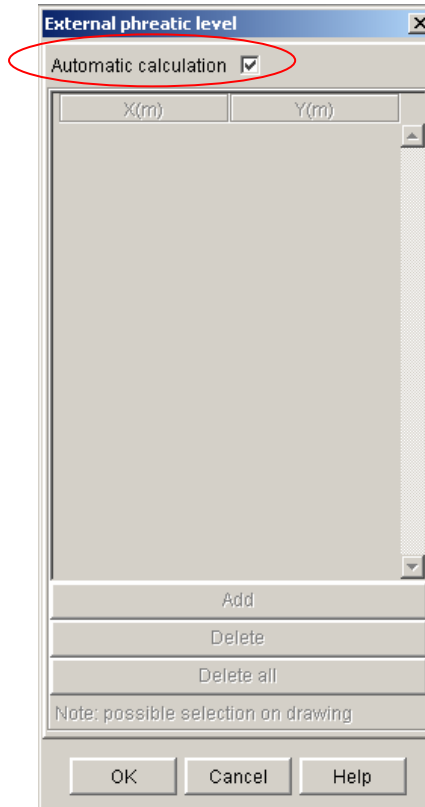
Figure 67: definition of the external preatic line

- The groundwater levels are displayed on the graphical view (the external preatic line is displayed with a bold line).
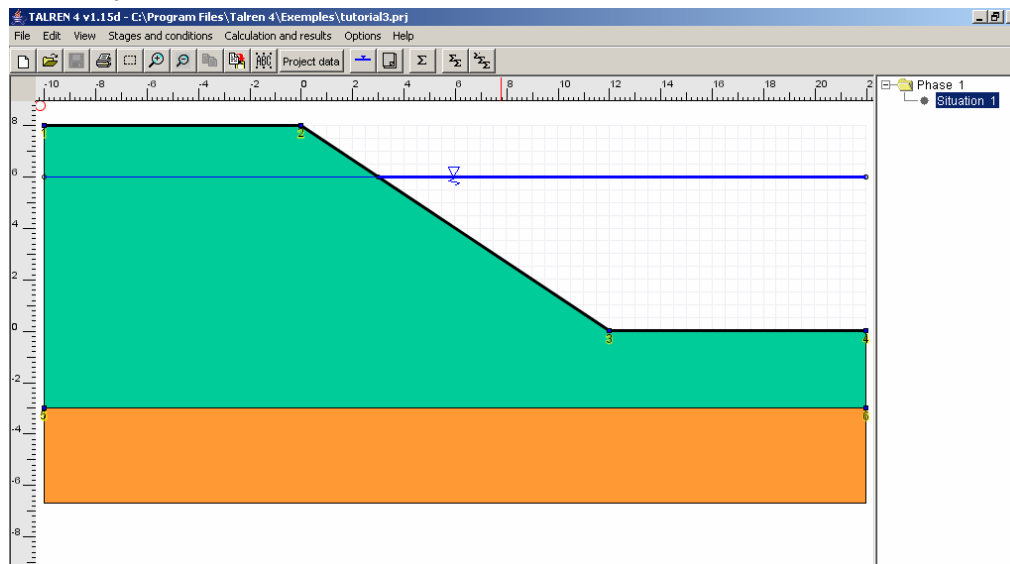
Figure 68: display of hydraulic conditions for stage 1
(phreatic level and external phreatic level)

Definition of situation 1

The first situation has been automatically created. It's however necessary to define its properties (for instance: partial safety factors and failure surfaces).

To access the situation properties, choose menu [Stages and conditions], option [Define the selected set of conditions], or right click on the situation name, and choose option [Define the

selected set of conditions], or click on the toolbar button [⊡]. Complete the dialogue box that appears with the following data (Figure 69 and Figure 70):

- Calculation method: Bishop (project default setting).
- Partial safety factors: traditional method / permament situation (project default setting).
- Seismic: we will not consider seismic conditions for this project.
- Definition of failure surfaces: we will choose circular failure surfaces passing at the bottom of the excavation, witn an automatic search. The corresponding data appear on Figure 69 and Figure 70 (we get to the screen of Figure 70 after selecting "Circular surfaces" and "Automatic search", and then clicking on the button [Define failure surfaces]).
- Please remember: the imposed passage point can be selected on the graphical view after clicking on the button [↘] in the dialogue box displayed on Figure 70).

The value of the parameter "Min abscissa for emergence" (equal to 12.00) guarantees that Talren will not search for circles which a right emergence point at X smaller than X = 12.00. Validate.
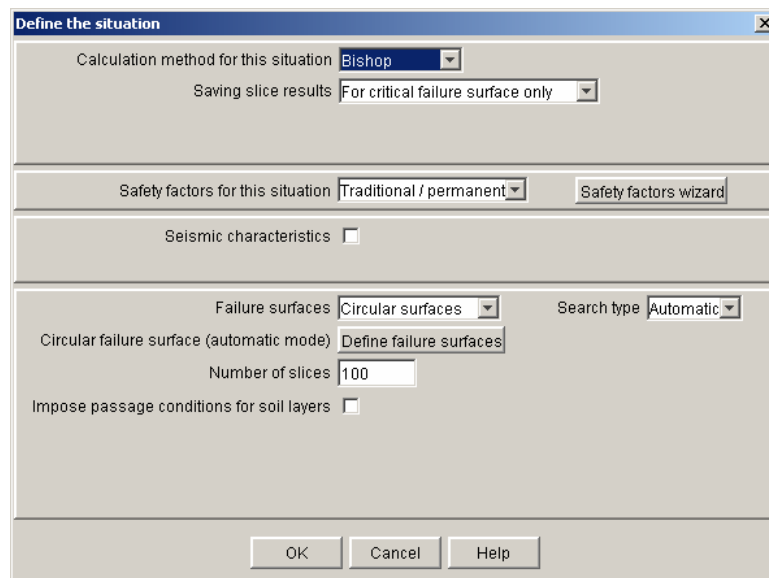
Figure 69: definition of the 1st situation for the 1st stage
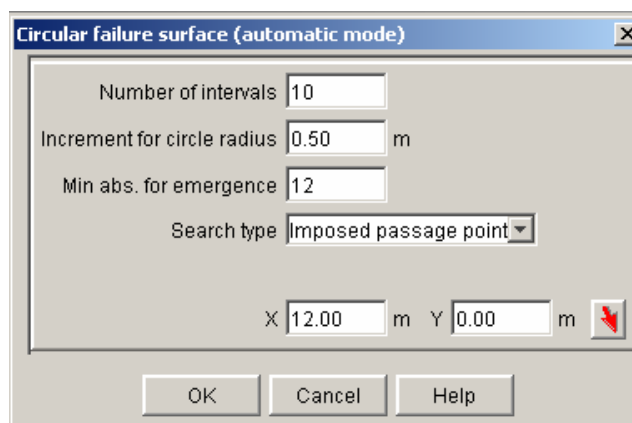
Figure 70: definition of failure surfaces for the 1st situation of the 1st stage

Calculation and results for situation 1

- To carry out the calculation for this situation, choose menu [Calculation] and option [Calculate the selected set of conditions] or click on the toolbar button Σ. The calculation is performed, then the default output is displayed (Figure 71): the circles are those corresponding to the minimal value for the safety factor.
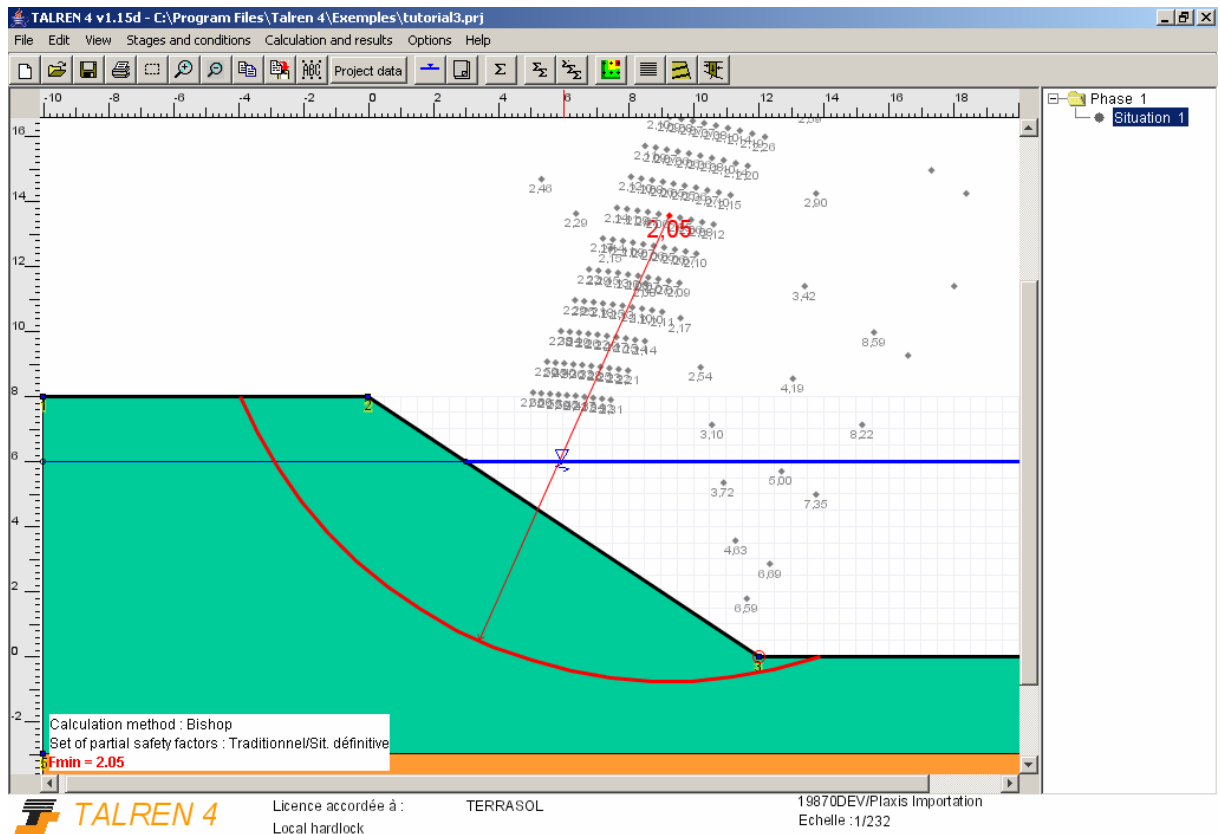
Figure 71: defaut output for situation 1 of stage 1

1.3.6. Step 5: definition of stage 2 (short-term drawdown)

Let's consider, in a 2nd stage, the stability of the dike with short term drawdown conditions (undrained Plaxis calculation).

- Select menu [Stages and conditions] then click on [Add a new stage], or right click on stage 1, then select [Add a new stage]. By default, the added stage has the same configuration as the first stage (same phreatic level for example).

- Click on the toolbar button ⊥ to display the dialogue box for hydraulic conditions.

- Instead of "Phreatic level", select "Triangular mesh of pore pressures", then click on the button [Import a Plaxis mesh].

- Choose the file "vidange tal fin.plx' (Figure 72), then the calculation step n° 34 (Figure 73).
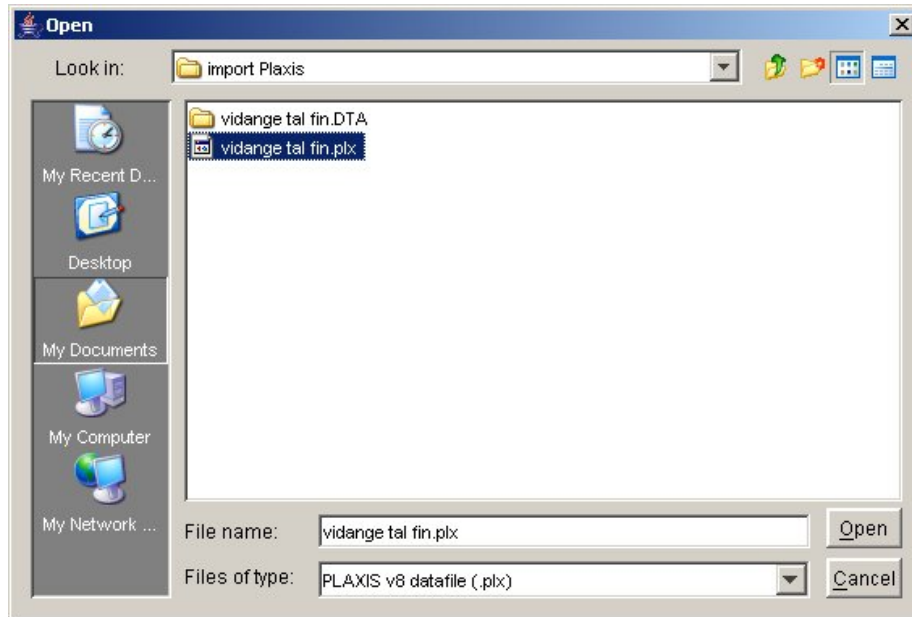
Figure 72: selection of the Plaxis file for pore pressures importation
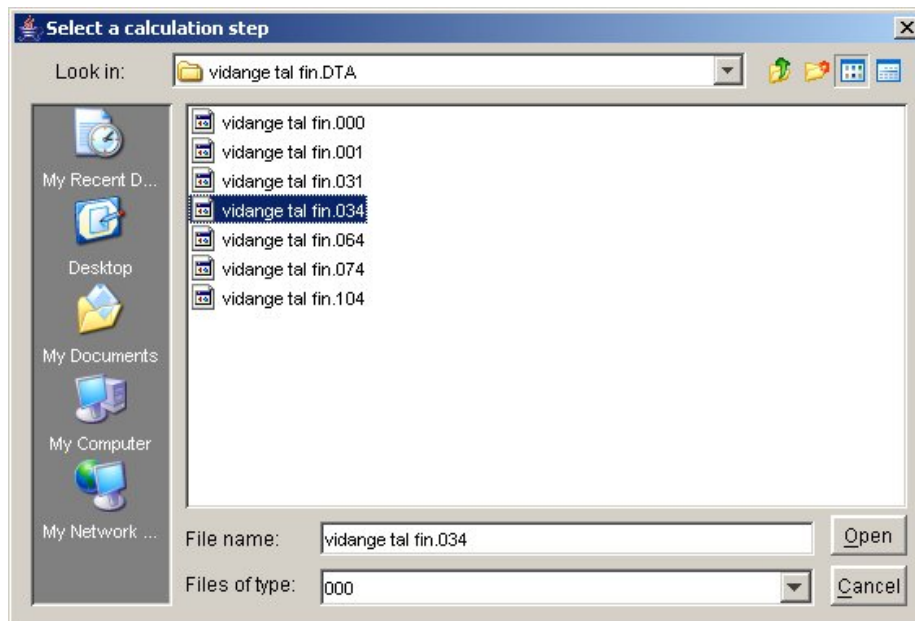
Figure 73: choice of the calculation step for pore pressures importation

- In order to view the nodes and triangles of the pore pressures mesh, as well as the pressure values, click on the button [Define nodes and triangles], then use tabs [Nodes] and [Triangles] (Figure 74). The imported mesh in our case contains 265 nodes and 647 triangles. Validate this dialogue box.

	X(m)	Y(m)	u(kN)
1	22.00	-6.00	108.556
2	22.00	-5.00	98.54
3	20.93	-6.00	108.575
4	22.00	-4.00	88.493
5	21.16	-4.15	89.977
6	22.00	-3.00	53.355
7	20.93	-3.00	53.334
8	-6.80	-6.00	118.451
9	-5.73	-6.00	117.936
10	-6.90	-5.31	111.604
11	-4.67	-6.00	117.421
12	-4.66	-5.19	109.288
13	3.60	6.00	116.008

	Point 1	Point 2	Point 3
634	247	239	154
635	259	206	154
636	288	305	251
637	305	280	251
638	300	335	306
639	335	312	306
640	340	187	259
641	329	86	174
642	205	364	199
643	364	365	199
644	322	332	174
645	332	329	174
646	321	340	249
647	340	259	249

Figure 74: dialogue box displaying nodes and triangles of the pore pressures mesh

- After defining the hydraulic mesh, it is necessary to complete the hydraulic conditions: we need to define the external phreatic line, which is now at the drawdown level (level 2.00 m). In the hydraulic conditions dialogue box, keep the checkbox "external phreatic line" activated, then click on the button [Define external phreatic line].

  This external phreatic line can be automatically determined from a phreatic level (stage 1), but with a pore pressure mesh, it cannot be calculated automatically, and it is necessary to define it manually (either in the table, or by drawing on the graphical view), after deactivating the checkbox "automatic calculation" (Figure 75).

Figure 75: definition of the external phreatic line for stage 2

- The hydraulic conditions for stage 2 are displayed on the graphical view below.
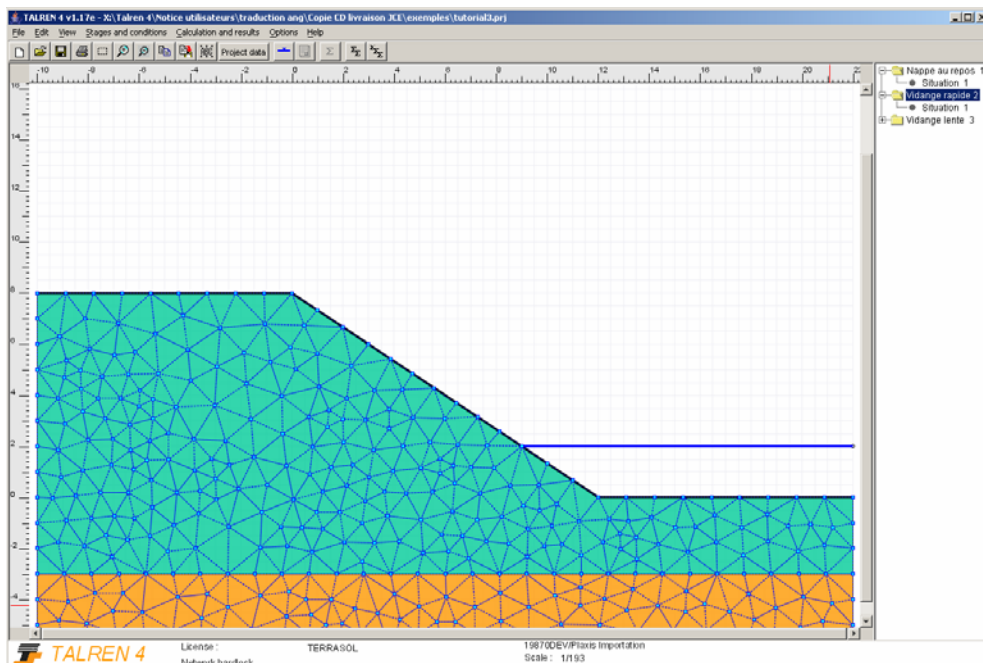
Figure 76: hydraulic conditions for stage 2
(triangular pore pressures mesh and external phreatic line)

Definition of situation 1

The first situation was automatically created. However, rather than defining its settings, we will use again the settings defined for the situation of stage 1 (the calculation settings are the same). To do so, we will copy/paste the situation 1 from stage 1 into stage 2.

- Click on situation 1 of stage 1.

- Right click and select option [Copy conditions] from the pop up menu.

- Click on stage 2.

- Right click and select option [Paste conditions] from the pop up menu.

- Then click on situation 1 of stage 2 to delete it (it is not possible to delete it first, because it is impossible to delete the single situation of a stage).

Calculation and results for situation 1

- Calculate the situation of stage 2 (the calculation is longer than for stage 1, because of the hydraulic mesh). The result is displayed on Figure 77.
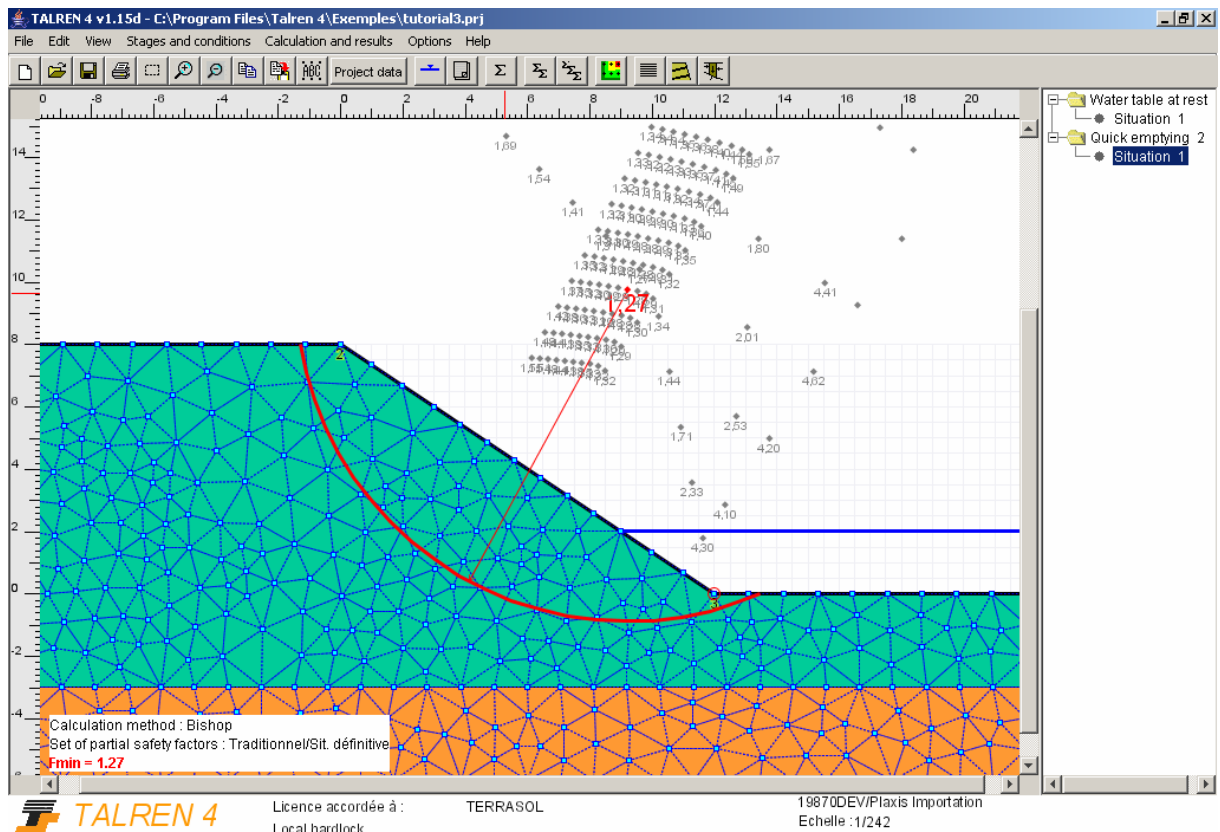
Figure 77: default output for situation 1 of stage 2

1.3.7. Step 6: definition of stage 3 (long-term drawdown)

Let's consider, in a 3rd stage, the stability of the dike with long term drawdown conditions (drained Plaxis calculation).

- Proceed as in the previous stage to add a stage, then click on the button [icon] to access the hydraulic conditions definition.

- Click on the button [Import Plaxis mesh], and choose the same Plaxis file as for the previous stage, but calculation step n° 74, corresponding with the Plaxis calculation for the long term drawdown.

- Confirm that you wish to delete the previous mesh (to replace it with the new mesh). Validate.

- The new mesh is geometrically identical to the previous one (since it is the same Plaxis file and hence the same Plaxis mesh), but the pore pressure values are different.

- The external phreatic line remains unchanged compared to the previous stage (same drawdown level).

Definition, calculation an results for situation 1

- Proceed as for previous stage to copy/paste a situation.

- Calculate this situation. The result is shown on Figure 78.
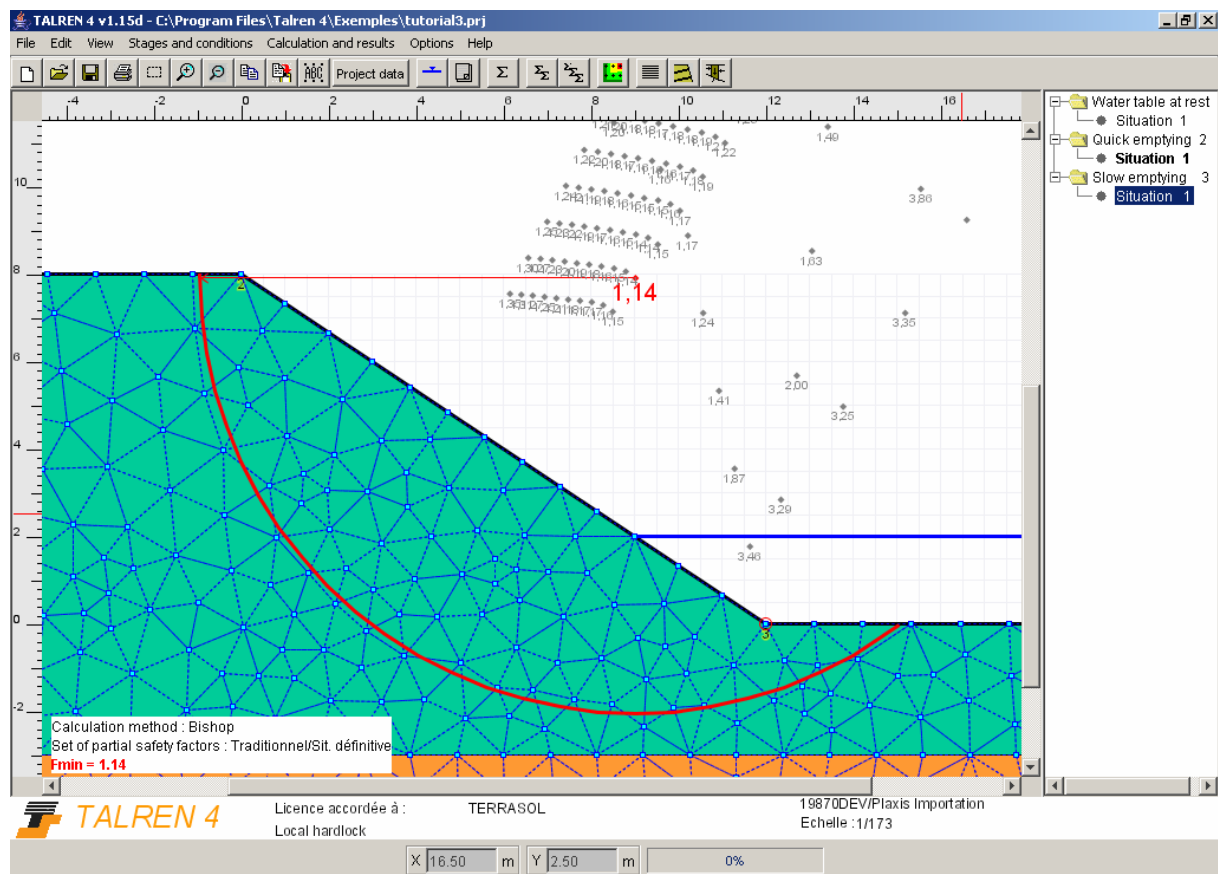
Figure 78: default output for situation 1 of stage 3

1.3.8. Summary of the results

The 3 calculations carried-out (all of them according to "traditional method") gave the following results.

Hydraulic conditions	Result: safety factor
Permanent water table	F = 2,05
Short term drawdown	F = 1,27
Long term drawdown	F = 1,14

Table 4: summary of results for tutorial 3
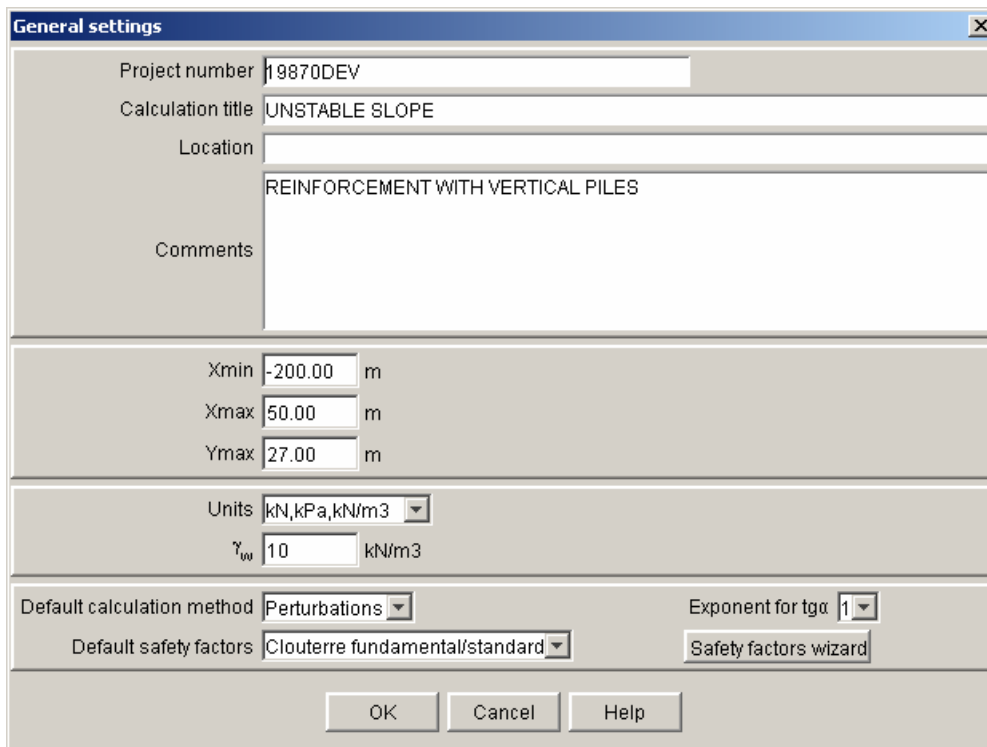
1.4. Tutorial 4: stability calculation with polygonal failure surfaces

1.4.1. Step 1: presentation of the project and general settings

This example deals with a natural slope that was stabilised with piles. We will consider polygonal failure surfaces, and compare the stability of the slope with and without piles, for several different failure surfaces.

Note: in this example we will not detail the operations as much as in tutorial 1: for further details, please refer to tutorial 1, or to part B of the manual.

- Run TALREN: use Windows [Start] menu or click on the desktop icon.
- Select menu [File] then option [New] or click on the toolbar button.
- Click on the button *i* or select menu [Project data] then option [General settings], and fill in the data for this dialogue box as shown on the figure below.

Figure 79: general settings

1.4.2. Step 2: geometry definition

The model dimensions and geometry are defined on the figures below.

- To define this geometry in TALREN 4, use one of the following methods: draw with the mouse, input of data directly in the main screen, or use of the geometry dialogue box.
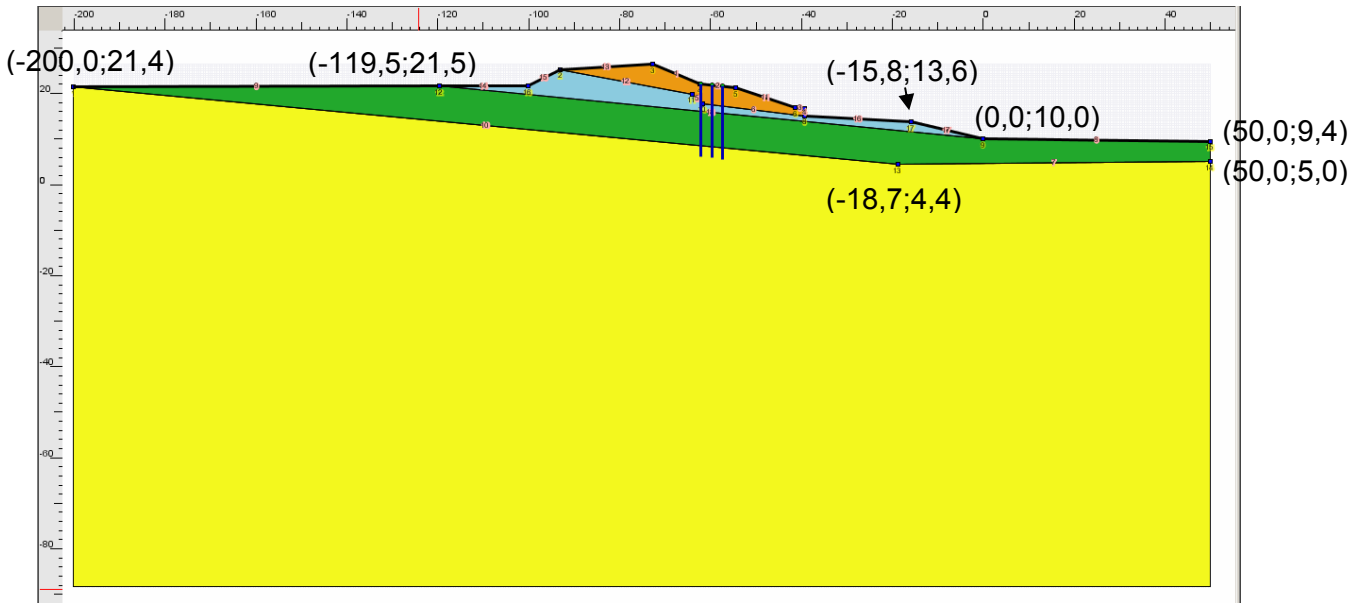
Figure 80: model geometry
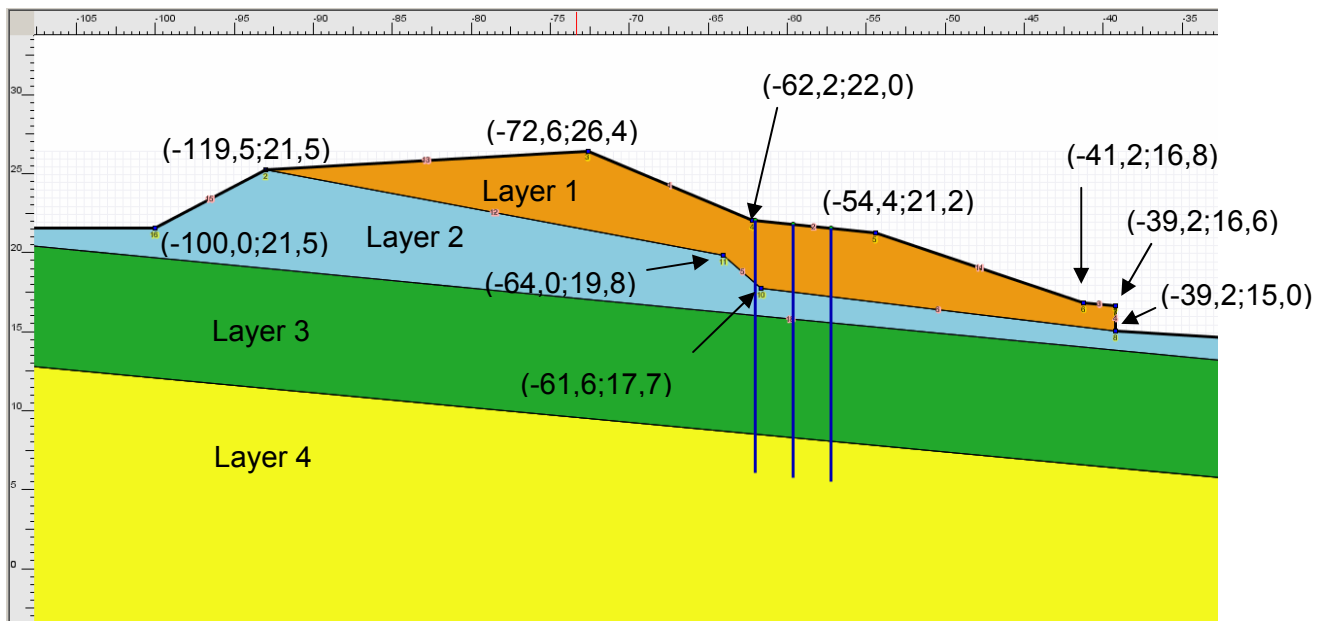
Figure 81: model geometry: zoom on the central part

1.4.3. Step 3: reinforcements definition

3 rows of piles are to be defined. The piles are modelled in this example with nails working in shear only.

- To define these 3 rows of nails in TALREN 4, use one of the 3 methods described in tutorial 1.
- The nail characteristics are defined in the table below.

Name	X(m)	Y(m)	L(m)	Ang(°)	eh(m)	Diffusion width Lb (m)	Diffusion angle ANGb (°)
Pile 1	-62.0	22.0	16.0	90.0	2.4	1.0	20.0
Pile 2	-59.6	21.75	16.0	90.0	2.4	1.0	20.0
Pile 3	-57.2	21.5	16.0	90.0	2.4	1.0	20.0

Name	Calculation mode	TR (kN)	Rsc calculated with qs ?	Equivalent grout radius re (m)	Rcis(kN)
Pile 1	T_{cal}, C_{imp}	0	Yes	1,0	540.0
Pile 2	T_{cal}, C_{imp}	0	Yes	1,0	540.0
Pile 3	T_{cal}, C_{imp}	0	Yes	1,0	540.0

Table 5: nails properties (to simulate the piles)
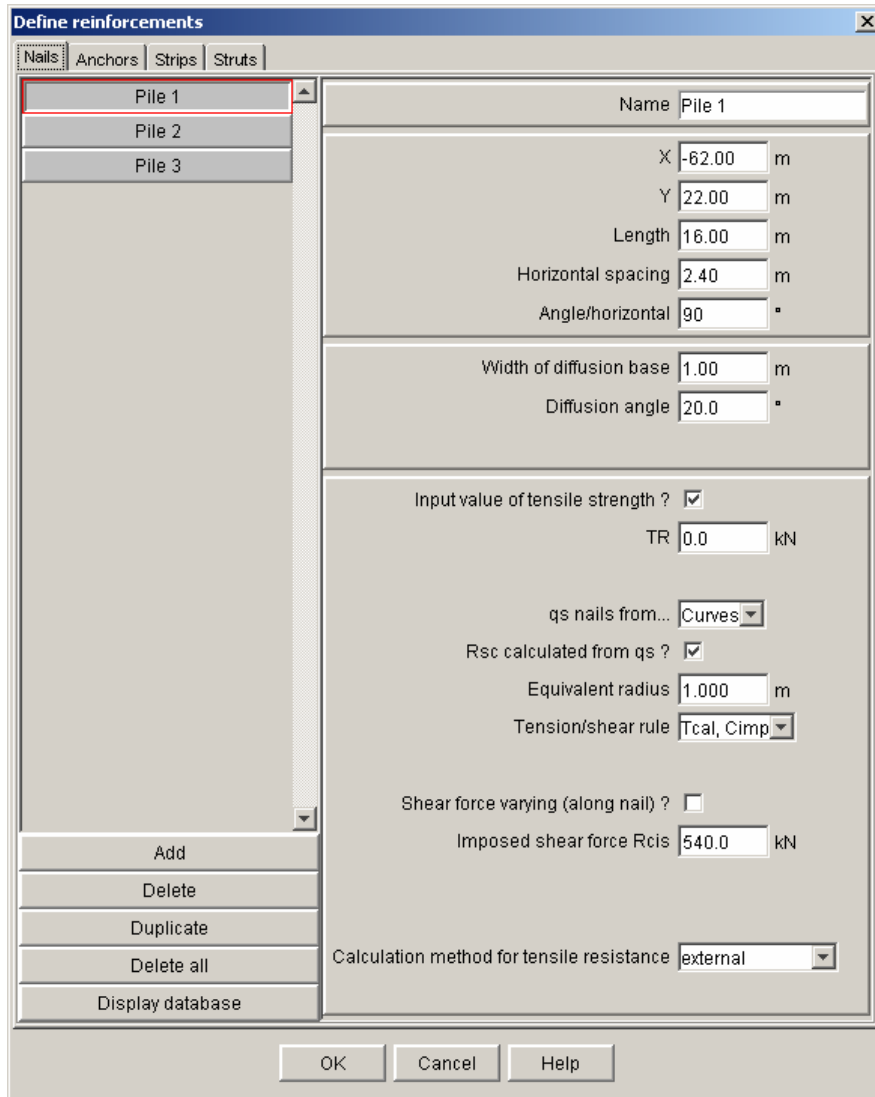
Figure 82: example of properties for nail 1 (pile 1)

1.4.4. Step 4: soil properties

4 soil layers are to be defined. Their characteristics are given in Table 6.

Layer	γ (kN/m^3)	φ' (°)	c' (kPa)
Layer 1	19.0	25.0	0.0
Layer 2	19.0	13.0	0.0
Layer 3	19.0	10.0	0.0
Layer 4	20.0	35.0	20.0

Table 6: soil characteristics

- To define these characteristics, select for example menu [Project data] then option [Soil characteristics]. Click on the [Add] button, and complete the data for the first layer. Click again on [Add] then define the data for the second soil layer and so on. To switch from one soil layer to the other, click on the layer names in the left list. The input for layer 4 is illustrated on the figure below.
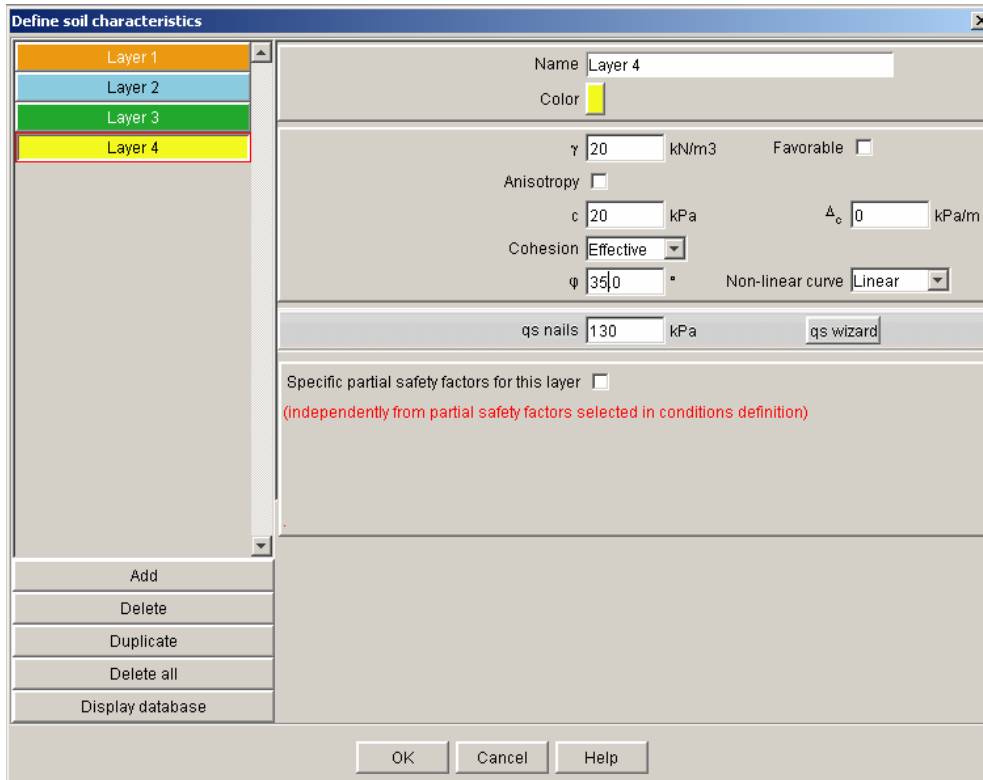
Figure 83: characteristics for layer 4

- To assign these soil properties to soil clusters, proceed for example with drag & drop from the left list onto soil clusters (refer to Figure 80 for the position of the layers).

1.4.5. Step 5: definition of stage 1

In this first stage, we will check the stability of the slope without piles (nails are deactivated by default when stage 1 is created).

A groundwater level should be taken into account for this project:

- Switch to the Stages/calculation mode, with the [Project data] menu or the toolbar button. The first stage is automatically created.

- Click on the button [⌐] to open the hydraulic conditions dialogue box.

- Choose the "Phreatic level" option, then click on the button [Define phreatic level]. A dialogue box containing the table of points describing the phreatic level is displayed. Define the phreatic level either by clicking on [Add] (and then typing in the coordinates of the points defining the phreatic level), or by clicking on the position of the phreatic level points on the graphical view. The data are illustrated on Figure 84.
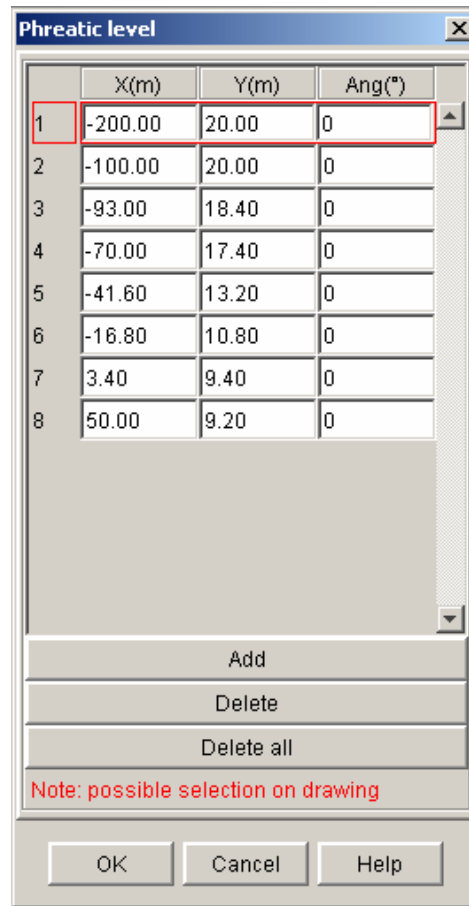
Figure 84: definition of the phreatic level
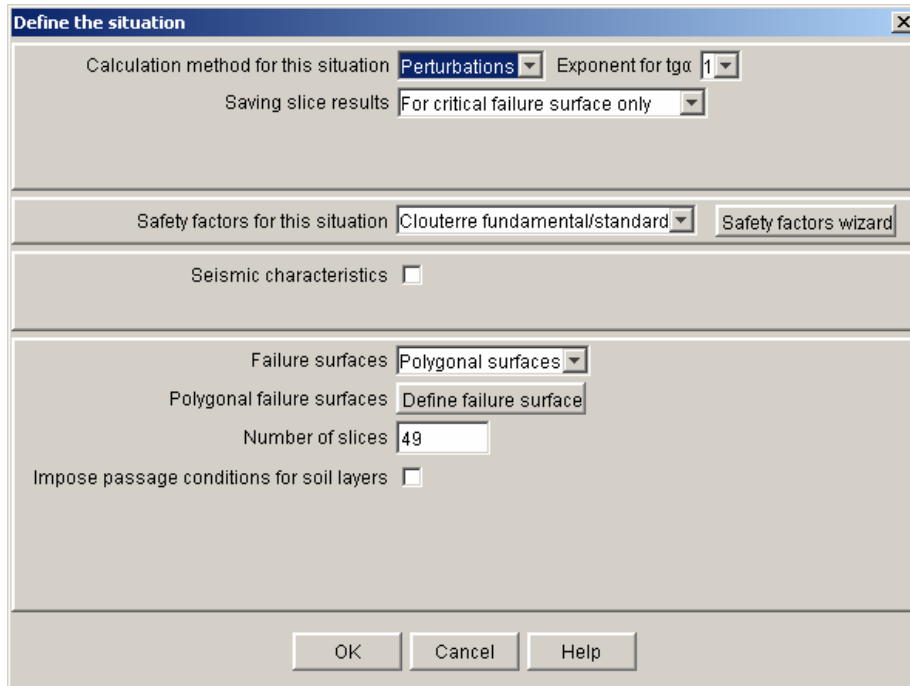
We are going to define now 4 polygonal failure surfaces. One single polygonal failure surface can be defined for each situation. 4 situations are therefore going to be defined for stage 1.

1.4.5.1 Definition of situation 1

The first situation is automatically created. It is however necessary to define its properties, and particularly: partial safety factors and failure surfaces.

To display the properties of the situation, choose menu [Stages and conditions], and option [Define selected set of conditions], or click with the right mouse button on the description of the situation, and choose option [Define selected set of conditions]. Fill in the data in the dialogue box according to the following indications (Figure 85 and Figure 86):

- Calculation method: Perturbations (project default option in the general settings).
- Partial safety factors: Clouterre fundamental/standard (project defaut option in the general settings).
- Seismic accelerations: we will not consider seismic conditions for this project.
- Definition of failure surfaces: we will choose polygonal failure surfaces. The corresponding data appear on Figure 85 and Figure 86 (the dialogue box of Figure 86 is displayed when clicking on button "Define failure surface" after selecting "Polygonal surfaces"). The values of u are zero in this example because the hydraulic conditions have been defined with a phreatic level (and not by pressures given along the polygonal failure surface).
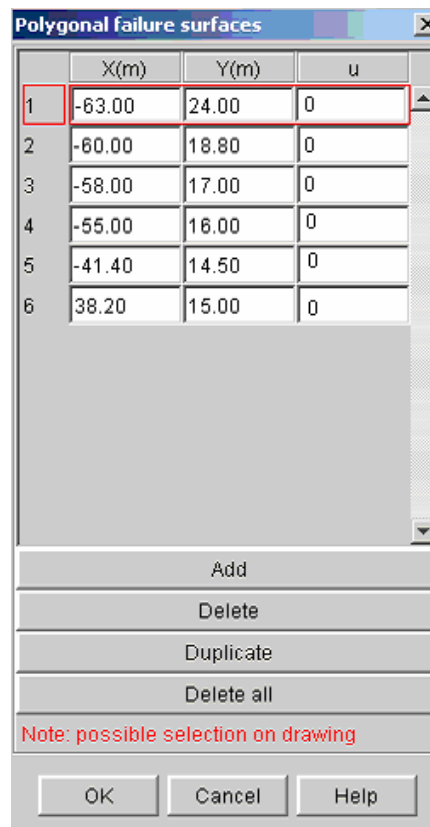
Figure 85: definition of the 1ˢᵗ situation for the 1ˢᵗ stage

Figure 86: definition of the polygonal failure surface for the 1ˢᵗ situation of the 1ˢᵗ stage
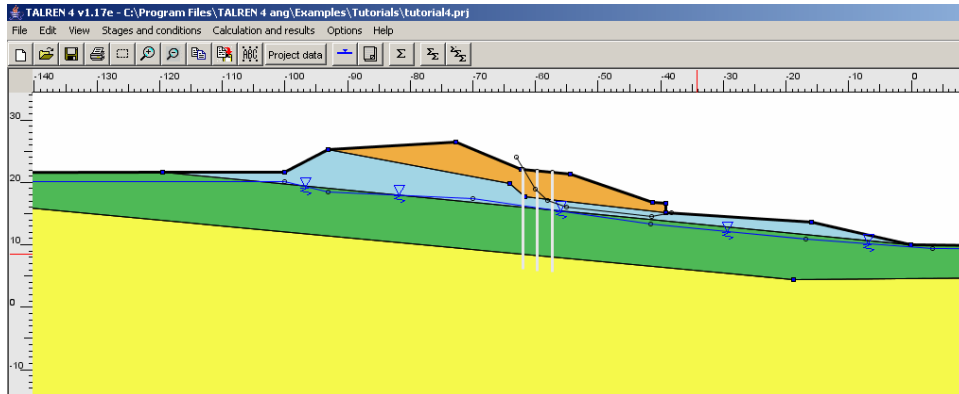
Figure 87: graphical view for the 1st situation of the 1st stage

Calculation and results for situation 1

- To calculate this situation, choose menu [Calculation and results] then option [Calculate the selected set of conditions] or click on the toolbar button Σ . The calculation is performed and the result is displayed (Figure 88): the circles represented on the graphical view are those that correspond to the minimum safety factor value.
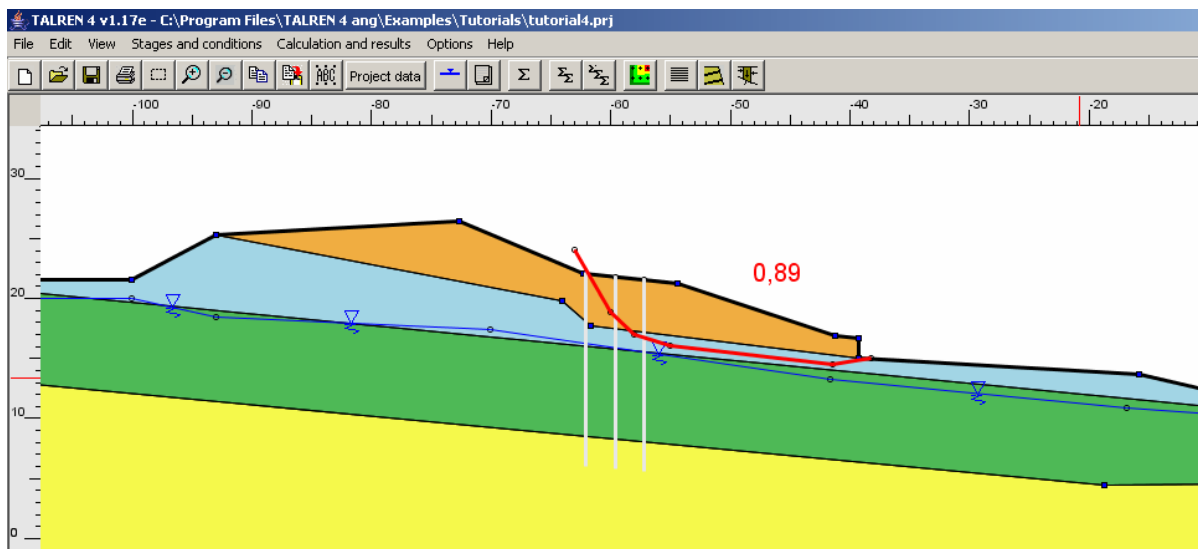
Figure 88: default output for situation 1 of stage 1

1.4.5.2 Definition of situation 2

Instead of redefining the settings for this situation, we will use the settings defined for situation 1, and change only the definition of the failure surface.

- Click on situation 1 to select it.

- Click with the right mouse button on situation 1 and select option [Duplicate the selected set of conditions].

- Other possible operations are: click with the right mouse button and select option [Copy conditions] from the pop up menu, then click again with the right mouse button and select option [Paste conditions].

- Display the situation properties and edit the failure surface definition: change it with the data below (Figure 89).

Figure 89: definition of the failure surface for the 2nd situation of the 1st stage

Calculation and results for situation 2

- Calculate situation 2. The result is displayed on Figure 90.
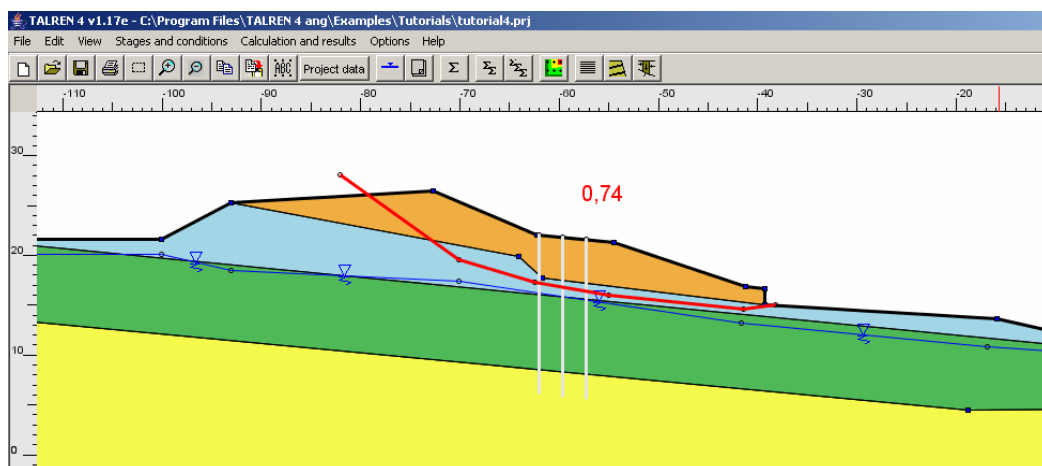
Figure 90: default output for situation 2 of stage 1

1.4.5.3   Definition of situation 3

The operations are the same as for situation 2. Definition of the failure surface for this situation is presented on Figure 91.

Figure 91: definition of the failure surface for the 3rd situation of the 1st stage

Calculation and results for situation 3

- Calculate situation 3. The result is presented on Figure 92.
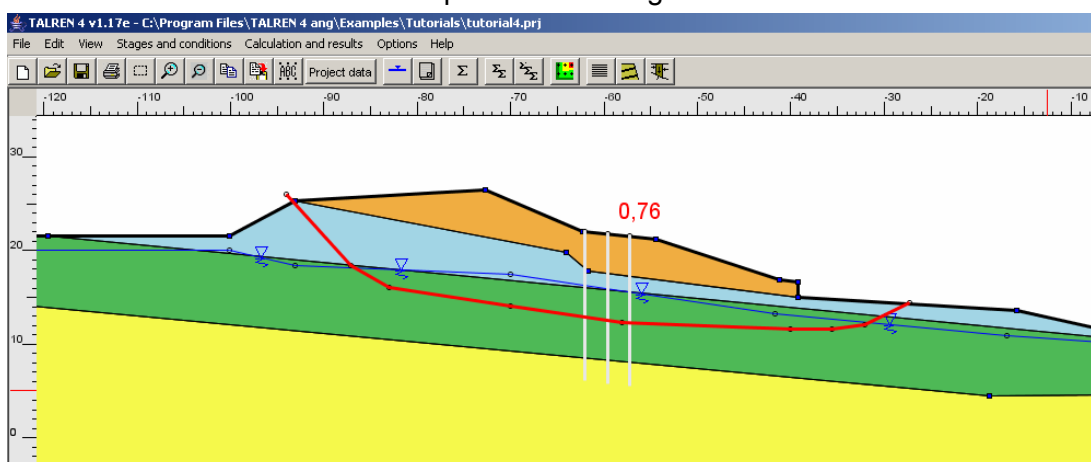
Figure 92: default output for situation 3 of stage 1

1.4.5.4 Definition of situation 4

The operations are the same as for situation 2. Definition of the failure surface for this situation is presented on Figure 93.

Figure 93: definition of the failure surface for the 4th situation of the 1st stage

Calculation and results for situation 4

- Calculate situation 4. The result is presented on Figure 94.
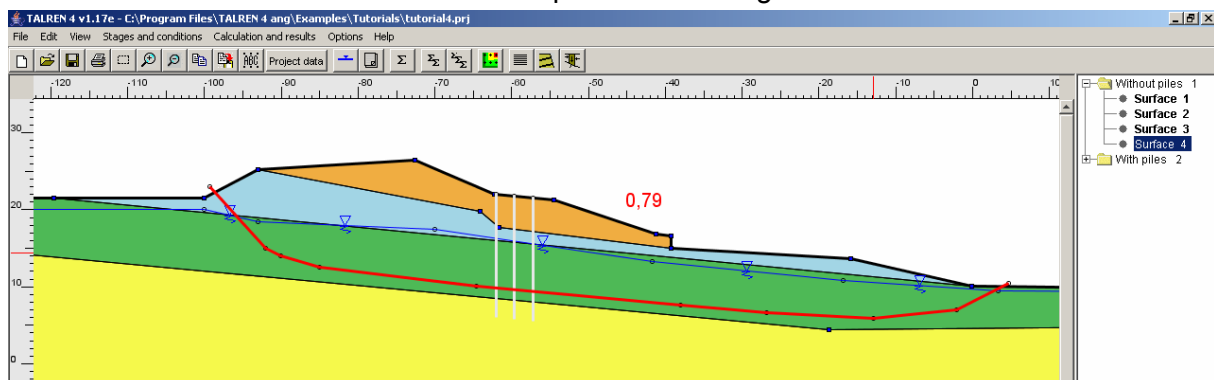
Figure 94: default output for situation 4 of stage 1

1.4.6. Step 6: definition of stage 2

In stage 2, we will check the slope stability with piles. The 3 rows of piles should therefore be activated. The groundwater conditions remain unchanged: it is therefore unnecessary to define them again.

- Select menu [Stages and conditions] then click on [Add a new stage], or click with the right mouse button on stage 1, then select [Add a new stage]. By default, the added stage is defined with the same geometrical configuration as the previous stage (groundwater conditions are the same for example).

- Click on each nail (pile) to activate it.

- Definition of stage 2 is finished.

1.4.6.1 Definition of situations 1 to 4

The same conditions are going to be defined (same failure surfaces) as in stage 1.

Instead of defining all the situations again, the situations defined in stage 1 are going to be copied and pasted into stage 2.

- Click on situation 1 of stage 1 to select it.

- Click with the right mouse button on this same situation and select option [Copy conditions] from the pop up menu.

- Click on stage 2.

- Click with the right mouse button on this same stage and select option [Paste conditions] from the pop up menu.

- Repeat these operations for situations 2, 3 and 4.

- Then click on situation 1 of stage 2 (the "empty" one) to delete it (it is not possible to delete it at first, because it is impossible to delete a single situation in a stage).

1.4.6.2 Safety factors obtained for situations 1 to 4

Table 7 summarises the results obtained for stage 1, and completes them with the results obtained for stage 2.

	Stage 1 (without piles)	Stage 2 (with the piles)
Situation 1	0,89	22,50
Situation 2	0,74	2,63
Situation 3	0,76	1,25
Situation 4	0,79	1,05

Table 7: summary of the results for tutorial 4
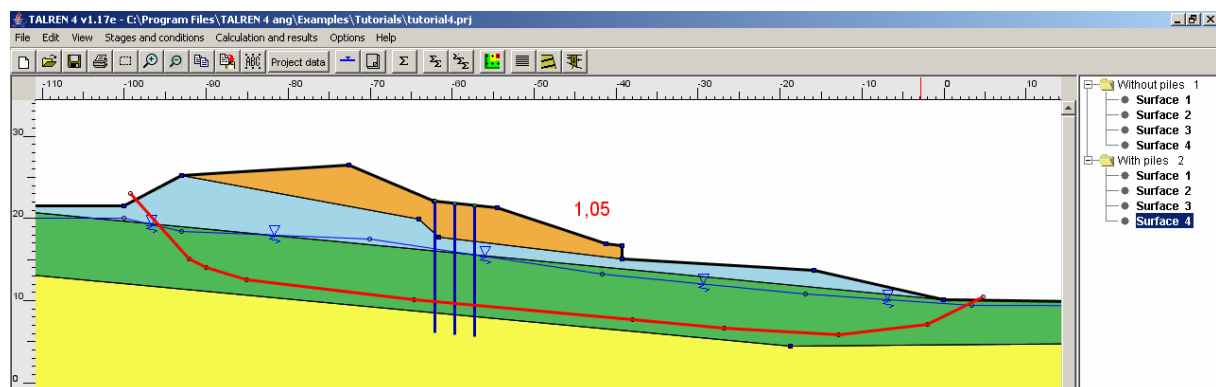
Figure 95: default output for situation 4 of stage 2

2. ADDITIONAL APPLICATION EXAMPLES

This section briefly presents some application examples: graphical views and comments.

The following table indicates for each example its most important characteristics.

WARNINGS:

- For most examples, only one family of failure surfaces is tested. It is obvious that in most of the examples presented, other families of failure surfaces should be calculated such as those corresponding to: temporary stages, passage of the circles in different layers, different passage points or search conditions, etc.
- For some examples, the calculations presented correspond to a stability analysis for structures previously designed with another calculation method such as: retaining wall design, bearing capacity of a foundation, etc.
- Some geotechnical structures designed with TALREN must be submitted to additional verifications. Example: nailed structure with small slenderness ratio (length of nails relatively short compared to support height) which can lead to important deformations of the soil nailed structure and high stresses at the bottom of the structure. These additional verifications should be decided by the design engineer.
- Partial safety factors: the choice "sensitive" or "standard" along with the action combinations (fundamental or accidental) are specific to each project. They cannot be extended to cases similar to the examples.
- The values provided in the examples for all parameters (geometry, material characteristics, safety factors, etc) are only example values, and cannot be used as references.

The files corresponding to these examples are provided both in .tal and .prj formats in the Talren 4 installation directory.

Table 8 : Summary of examples presented in chapter D of Talren 4 manual

PROJECT DATA MODE

FILE TYPE

	Tutorial 1	Tutorial 2	Tutorial 3	Vidange talus Tutorial 4	Highway wall	Example 1	Example 2	Example 3	Example 4	Example 5	Example 6	Example 7	Example 8	Example 9	Example 10	Example 11	Example 12	Example 13	Example 14	Exaple 15	Example 16	Example 17	Example 18	Example 19	Example 20	Example 21	Example 22	Example 23
TALREN 4 (.prj)	x	x		x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x
TALREN 97 (.tal)	x			x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x
PLAXIS v8 (.plx)			x																									

LOADS

	T1	T2	T3	Vid	HW	E1	E2	E3	E4	E5	E6	E7	E8	E9	E10	E11	E12	E13	E14	E15	E16	E17	E18	E19	E20	E21	E22	E23
DISTRIBUTED LOADS	1				3	1	3	x	1	1	1	2		5		3		3	1	3		1		4	1	1		x
LINEAR LOADS / ADDITIONAL MOMENTS	1				3	1			1							2		1	1							1		x

REINFORCEMENTS

	T1	T2	T3	Vid	HW	E1	E2	E3	E4	E5	E6	E7	E8	E9	E10	E11	E12	E13	E14	E15	E16	E17	E18	E19	E20	E21	E22	E23
NAILS	4			7				4	4										4				3					
ANCHORS (non proportional method for pull-out resistance)					1			4	1									1										
ANCHORS (proportional method for pull-out resistance)				5							5													5				
STRIPS																4												
STRUTS														5														
GEOTEXTILES							15																					
PILES OR MICROPILES	2			3	2					1	1										3	5				2		
SHEAR VARIABLE ALONG NAIL	x				x																					x	x	

SOIL PROPERTIES

	T1	T2	T3	Vid	HW	E1	E2	E3	E4	E5	E6	E7	E8	E9	E10	E11	E12	E13	E14	E15	E16	E17	E18	E19	E20	E21	E22	E23
NUMBER OF SOIL LAYERS	2	2	2	4	3	2	6	5	3	7	3	3	3	3	4	5	3	3	2	4	4	5	2	7	3	3	2	
SUBSTRATUM	x										x				x													
COHESION VARIABLE WITH DEPTH		x																							x	x	x	
COHESION ANISOTROPY												x																
NON LINEAR SHEAR STRENGTH CURVE													x															

STAGES/CALCULATIONS MODE

STAGES

	T1	T2	T3	Vid	HW	E1	E2	E3	E4	E5	E6	E7	E8	E9	E10	E11	E12	E13	E14	E15	E16	E17	E18	E19	E20	E21	E22	E23
NUMBER OF STAGES	4	3	3	2	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
MAX. NUMBER OF SITUATIONS/STAGE	3	1	1	4	1	1	1	3	1	1	1	1	1	1	1	1	1	1	1	1	4	1	1	1	1	1	1	1

HYDRAULIC CONDITIONS

	T1	T2	T3	Vid	HW	E1	E2	E3	E4	E5	E6	E7	E8	E9	E10	E11	E12	E13	E14	E15	E16	E17	E18	E19	E20	E21	E22	E23
PHREATIC LINE	x(*)			x	x	x	x	x	x	x(*)					x	x				x	x	x(*)		x	x(*)	x(*)		
WATER TABLE BASE															x													
U ALONG POLYGONAL FAILURE SURFACE																	x											
PORE PRESSURES MESH		x																x										
RU COEFFICIENT		x							x										x			x			x	x		
EXTERNAL WATER LEVEL									x						x					x							x	

CALCULATION METHOD

CIRCULAR FAILURE SURFACES

	T1	T2	T3	Vid	HW	E1	E2	E3	E4	E5	E6	E7	E8	E9	E10	E11	E12	E13	E14	E15	E16	E17	E18	E19	E20	E21	E22	E23
AUTOMATIC SEARCH / IMPOSED PASSAGE POINT	x	x	x	x																					x			
AUTOMATIC SEARCH / 1st CIRCLE TANGENT TO A GIVEN SOIL LAYER				x																								
MANUAL SEARCH / IMPOSED PASSAGE POINT						x	x		x							x			x	x			x	x		x		
MANUAL SEARCH / 1st CIRCLE TANGENT TO A GIVEN SOIL LAYER	x				x		x			x		x		x				x				x				x		
MANUAL SEARCH / 1st CIRCLE INTERCEPTING THE SLOPE																				x						x		
IMPOSE PASSAGE CONDITIONS IN SOME LAYERS								x	x																			

POLYGONAL FAILURE SURFACES

	T1	T2	T3	Vid	HW	E1	E2	E3	E4	E5	E6	E7	E8	E9	E10	E11	E12	E13	E14	E15	E16	E17	E18	E19	E20	E21	E22	E23
FELLENIUS																												
BISHOP	x	x		x		x		x	x	x	x	x	x	x	x	x		x	x	x		x	x			x	x	x
PERTURBATIONS		x	x	x	x							x		x	x		x				x			x	x			
YIELD DESIGN CALCULATION																								x	x			

SPIRAL FAILURE SURFACES

	T1	T2	T3	Vid	HW	E1	E2	E3	E4	E5	E6	E7	E8	E9	E10	E11	E12	E13	E14	E15	E16	E17	E18	E19	E20	E21	E22	E23
POLYGONAL FAILURE SURFACES				x				x	x												x							

SEISMIC CONDITIONS

	T1	T2	T3	Vid	HW	E1	E2	E3	E4	E5	E6	E7	E8	E9	E10	E11	E12	E13	E14	E15	E16	E17	E18	E19	E20	E21	E22	E23
LOGARITHMIC SPIRALS																			x									
SEISMIC CONDITIONS			x	x															x									

(*) : in the .tal file, one single point was enough to define a horizontal water level: this isn't the case in Talren 4 anymore: you should define at least 2 points

2.1. Example 1: underpinning of a building

This example presents the temporary underpinning of a building exerting high loads on the underlying soil. The reinforcement consists in two rows of piles placed by jet-grouting with a centre to centre spacing between rows of 1 m, and a row of anchors spaced horizontally 2.5 m centre to centre.

The design of the inclusions included a preliminary analysis to determine the internal forces in the piles (normal forces, shear forces, and bending moments) and to estimate displacements. The aim of the study, whose results are shown on the figure below, is to check the stability of the excavation with respect to circular failure surfaces.

Table of soil characteristics...

Export...

Name	γ(kN/m3)	φ(°)	c(kPa)	Δc(kPa/m)	qs nails(kPa)	qs anchors(kPa)	α	pl(kPa)	KsB(kPa)
Layer 1	19	39.0	0.0	0.0	250			3000	67500
Layer 2	19	0.0	999.0	0.0	300			5000	10000

OK   Help

Table of loads...

Distributed loads

Export...

Name	X left(m)	Y left(m)	q left(kPa)	X right(m)	Y right(m)	q right(kPa)	Ang/horizontal(°)	Width of diffusion base(m)	Diffusion angle(°)
Ld 1	-100	4.5	155	-10.2	4.5	155	90	0	0
Ld 2	-10.2	4.5	375	-4.2	4.5	375	90	0	0
Ld 3	-4.2	4.5	155	0	4.5	155	90	0	0

OK   Help
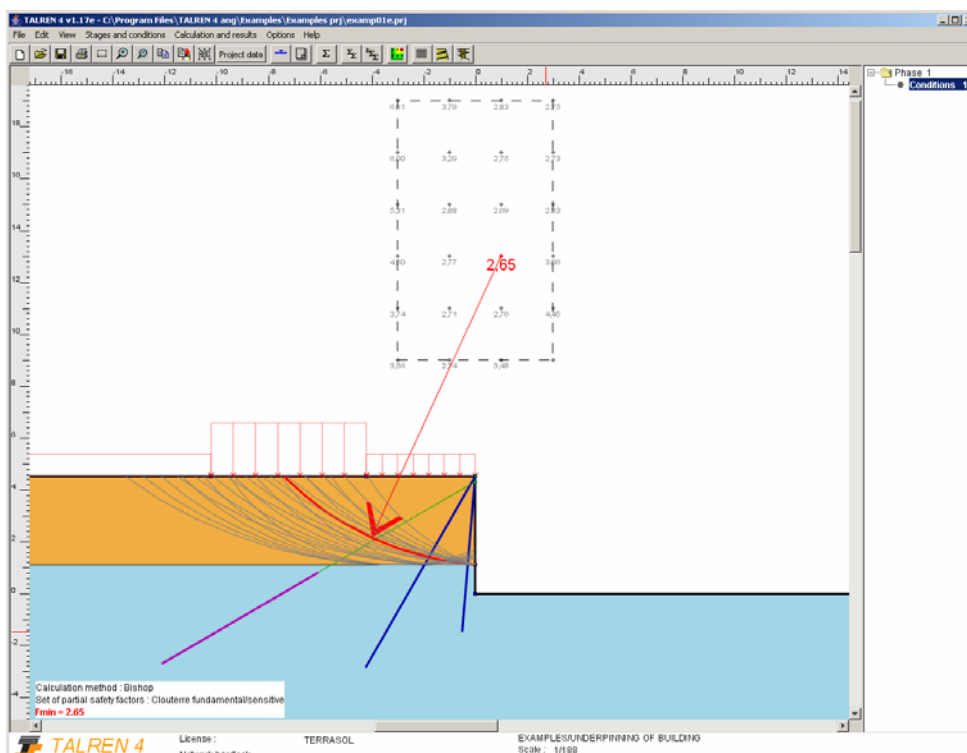
2.2. Example 2: stability of a temporary slope

This example presents the analysis of a temporary slope which appears unstable (considering Clouterre recommendations).
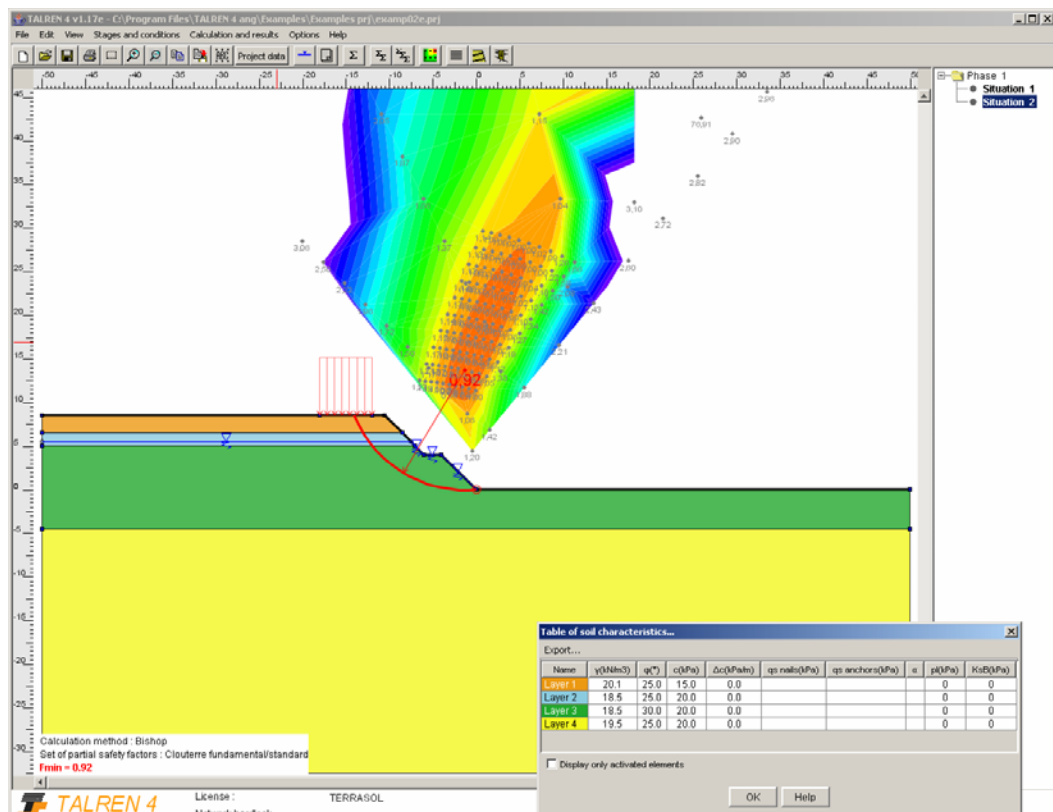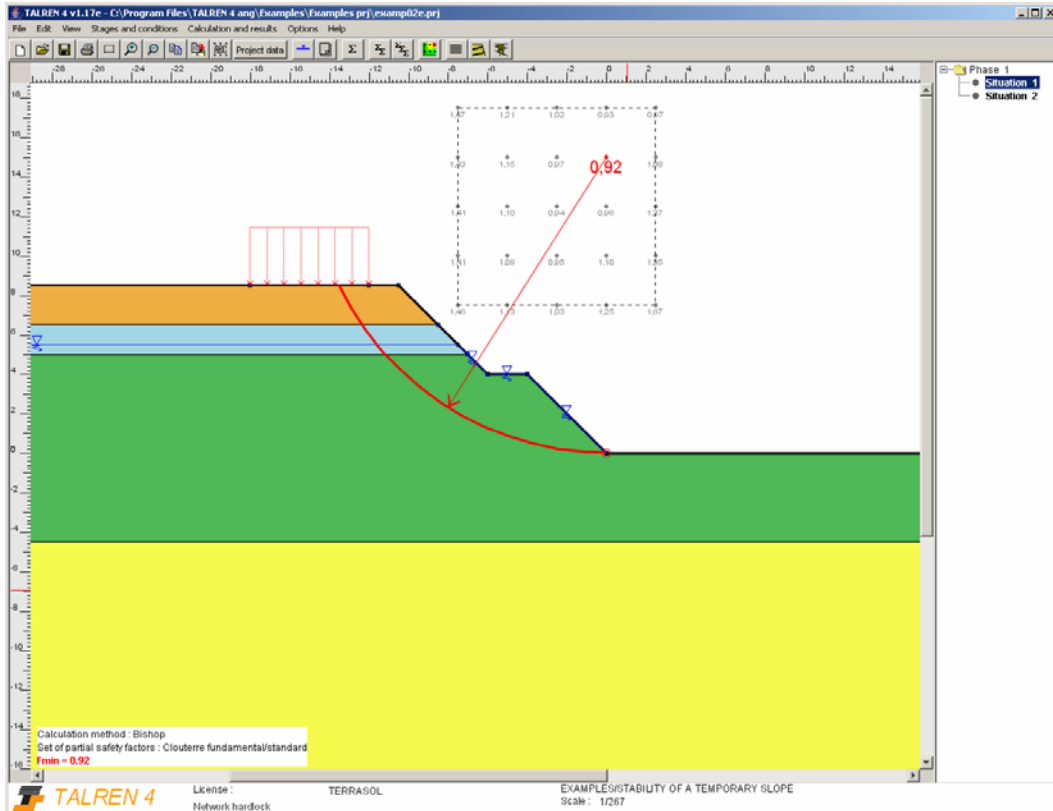
2.3. Example 3: stability of a support by geogrids

This example deals with the case of the extension of a parking lot near a slope. The solution calls for placing a crib wall retaining system reinforced by geogrid layers.

The stability analysis was accompanied by an analysis of the local equilibrium of the crib wall, along with a traditional stability analysis of the geogrid reinforced structure.

In Talren, the geogrids are defined as reinforcing strips (similar behaviour). The geogrid characteristics were obtained from the charts provided by the manufacturer.

| Name	Level	Width (m)	Horiz. sp. (m)	Length (m)	TR (kN)	μ0*	μ1*	γ_{fill} (kN/m^3)	Load factor on fill weight
Band 1	30.0	1.0	1.0	5.0	16.9	0.32	0.32	19.0	0.95
Band 2	29.5	1.0	1.0	5.0	16.9	0.32	0.32	19.0	0.95
Band 3	29.0	1.0	1.0	5.0	16.9	0.32	0.32	19.0	0.95
Band 4	28.5	1.0	1.0	5.0	16.9	0.32	0.32	19.0	0.95
Band 5	28.0	1.0	1.0	5.0	16.9	0.32	0.32	19.0	0.95
Band 6	27.5	1.0	1.0	5.0	16.9	0.32	0.32	19.0	0.95
Band 7	27.0	1.0	1.0	5.0	16.9	0.32	0.32	19.0	0.95
Band 8	26.5	1.0	1.0	5.0	16.9	0.32	0.32	19.0	0.95
Band 9	26.0	1.0	1.0	5.0	16.9	0.32	0.32	19.0	0.95
Band 10	25.5	1.0	1.0	5.0	16.9	0.32	0.32	19.0	0.95
Band 11	25.0	1.0	1.0	5.0	16.9	0.32	0.32	19.0	0.95
Band 12	24.5	1.0	1.0	5.0	16.9	0.32	0.32	19.0	0.95
Band 13	24.1	1.0	1.0	8.5	36.47	0.32	0.32	19.0	0.95
Band 14	23.75	1.0	1.0	9.0	36.47	0.32	0.32	19.0	0.95
Band 15	23.4	1.0	1.0	9.5	36.47	0.32	0.32	19.0	0.95
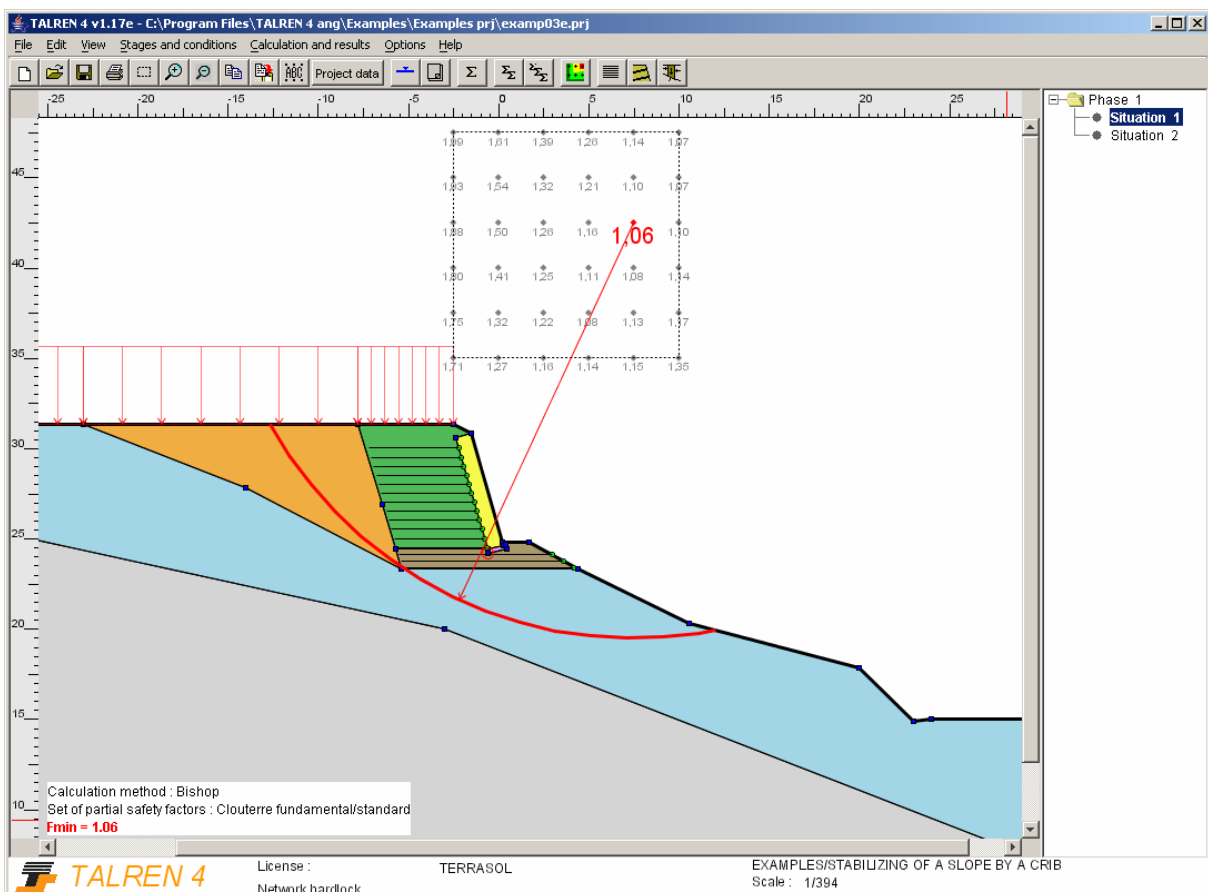
## 2.4. Example 4: reconstruction of a road following a slide

This example deals with the repair of a road damaged by a slide. The uphill part of the slide was reinforced by an anchoring system. The downhill part was stabilised by a drainage system.

Prior to the calculation, the shear strength properties of the soil leading to failure were determined along the known failure surface, considering the hydraulic conditions existing at the moment of the slide.

It was considered that the structure was non-sensitive, even though it was located in a sliding zone. This choice was based on the fact that the shear strength properties were relatively well known, since they were determined from actual failure.

Name	γ(kN/m3)	φ(°)	c(kPa)	Δc(kPa/m)
Layer 1	19	23.0	0.0	0.0
Layer 2	19	23.0	0.0	0.0
Layer 3	21	35.0	30.0	0.0
Layer 4	22	35.0	40.0	0.0
Layer 5	20	30.0	0.0	0.0

| Name	Y (m)	Horizontal spacing (m)	Inclination /horizontal (°)	Free length (m)	Grouted length (m)	TR (kN)	RQS (kN)
Anchor 1	435.0	5.0	25.0	9.0	10.0	1110.0	1500.0
Anchor 2	433.0	5.0	25.0	5.0	10.0	1110.0	1500.0
Anchor 3	431.5	5.0	25.0	5.75	8.5	1110.0	1500.0
Anchor 4	429.5	5.0	25.0	5.75	8.5	1110.0	1500.0
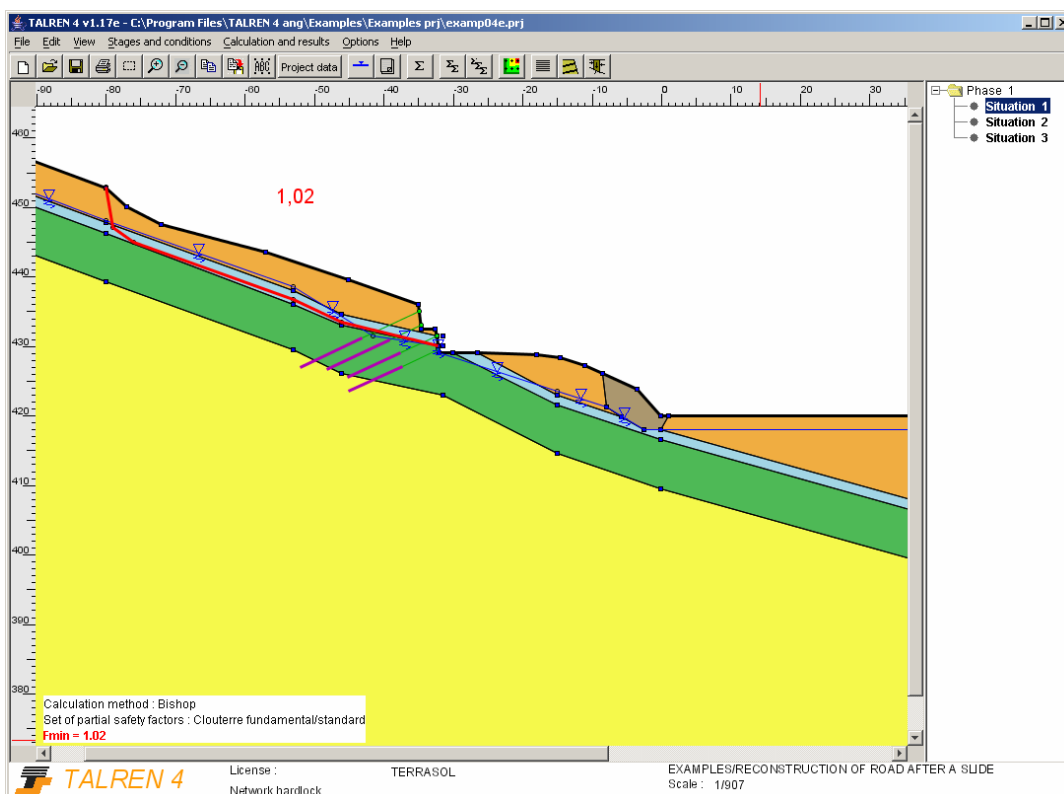
2.5. Example 5: retaining wall (soldier piles)

This example deals with the case of a "soldier pile" or "Berlin wall" retaining structure composed of temporary vertical micropiles fixed by a row of anchors at the top, and by 3 nail rows.

Name	γ(kN/m3)	φ(°)	c(kPa)	Δc(kPa/m)	qs nails(kPa)	qs anchors(kPa)	α	pl(kPa)	KsB(kPa)
Layer 1	20	30.0	0.0	0.0	0			0	0.1
Layer 2	21	35.0	20.0	0.0	130			800	7500
Layer 3	23	0.0	1000.0	0.0	0			0	0.1

Name	Level (m)	Length (m)	Horiz. spacing (m)	Inclination/ horiz. (°)	Equivalent radius (m)	Calcul. mode	Rcis (kN)	EI (kN.m²)	Mmax (kN.m)
Nail 1	157.25	9.0	2.5	10	0.047	T_{cal}, C_{imp}	0	0	0.0
Nail 2	155.0	9.0	2.5	10	0.047	T_{cal}, C_{imp}	0	0	0.0
Nail 3	152.75	7.0	2.5	10	0.047	T_{cal}, C_{imp}	0	0	0.0
Nail 4	160.0	13.0	1.25	90	0.1	T_{imp}, C_{cal}	271	2074	0.0

| Name	Level (m)	Ungrouted length (m)	Grouted length (m)	Horizontal spacing (m)	Inclination /horizontal (°)	RQS (kN)	TR (kN)
Anchor 1	159.5	6.0	8.0	2.5	10.0	800.0	900.0
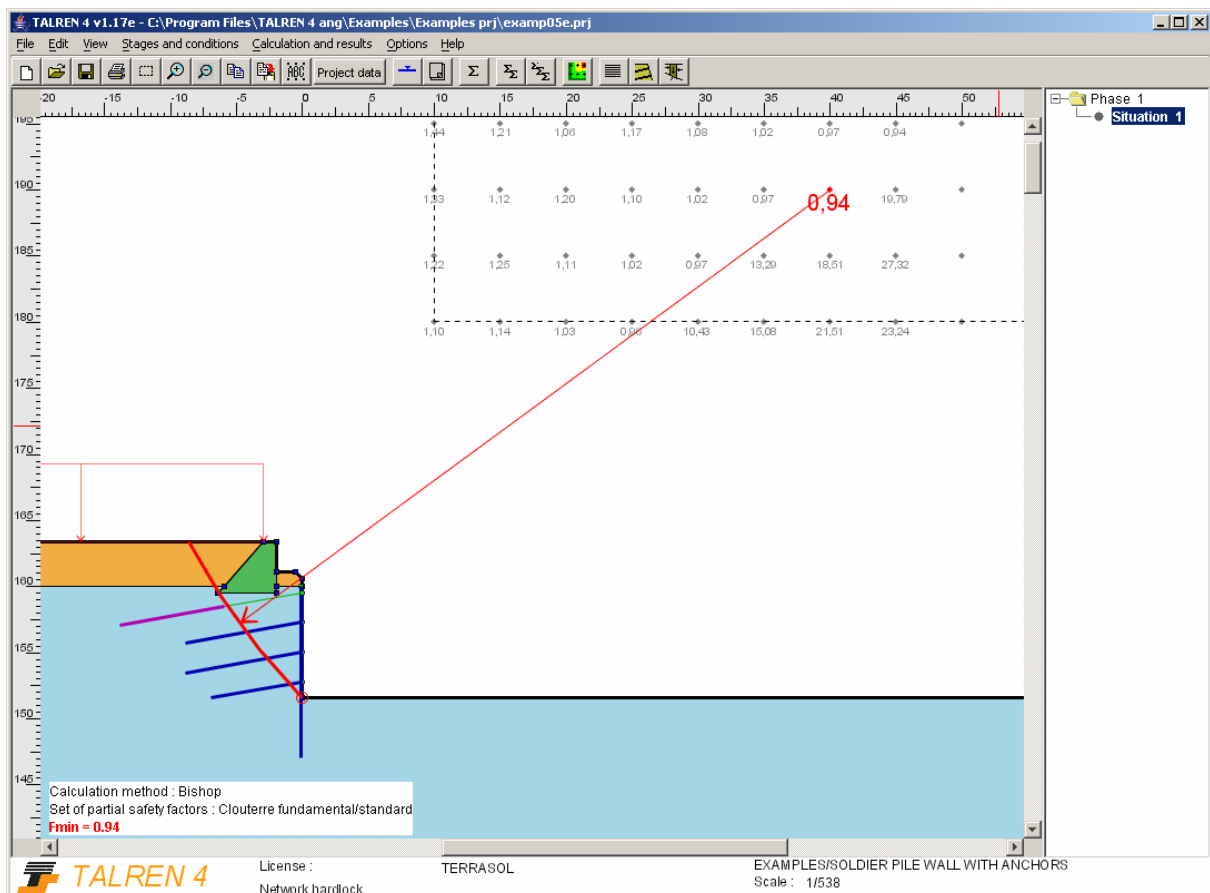
2.6. Example 6: shallow stabilisation of a slope

This example presents the stabilisation of a slope where the shallow layer is deteriorating with time.

The characteristics of the upper layer (composed actually of 2 layers) have been determined at the limit state of the existing slope.

Name	γ(kN/m3)	φ(°)	c(kPa)	Δc(kPa/m)	qs nails(kPa)	qs anchors(kPa)	α	pl(kPa)	KsB(kPa)
Layer 1	20	30.0	23.0	0.0	80			1000	15000
Layer 2	21	30.0	45.0	0.0	120			2000	22200
Layer 3	21	30.0	150.0	0.0	160			4000	186000
Layer 4	20	17.0	25.0	0.0	0			1000	15000
Layer 5	20	30.0	0.0	0.0	0			750	19250
Layer 6	18	25.0	0.0	0.0	0			0	0.1
Layer 7	20	45.0	0.0	0.0	0			750	19250
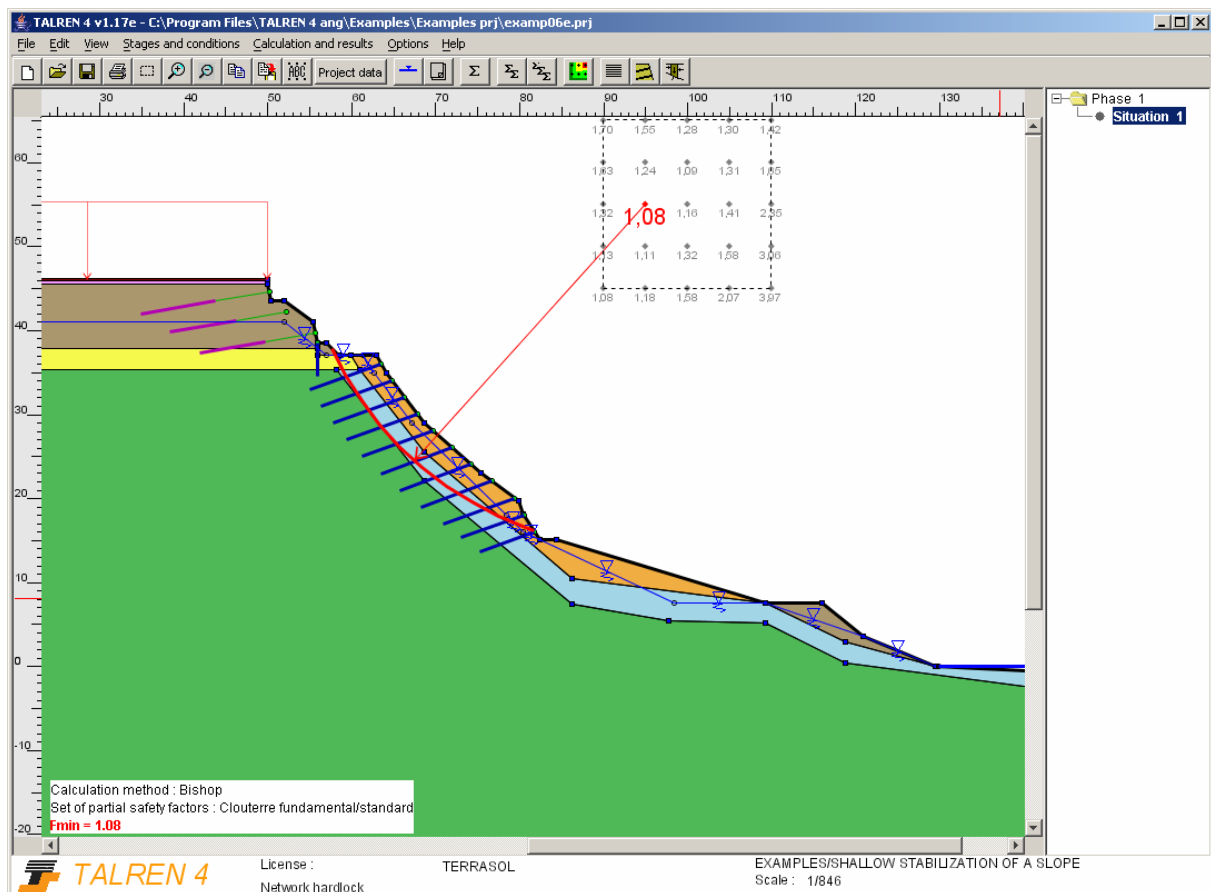
## *2.7. Example 7: geotextile at the base of an embankemnt on soft soil*

This example deals with the stability of an embankment placed on soft soil. The upper 50 centimeters of soft soil were replaced with good fill and the stability was ensured with a geotextile.

In Talren, the geotextile is equivalent to a reinforcing strip, since its behaviour is similar.

The soil/inclusion friction coefficient was considered to be equal to tan(φ).

Name	γ(kN/m3)	φ(°)	c(kPa)	Δc(kPa/m)
Layer 1	18	35.0	0.0	0.0
Layer 2	15	0.0	23.0	0.0
Layer 3	20	0.0	50.0	0.0

Name	Width (m)	Horizontal spacing (m)	Length (m)	TR (kN)	$\mu 0^*$	$\mu 1^*$	γ_{fill} (kN/m^3)	$\Gamma_{s1\ fill}$
Strip 1	1.0	1.0	17.0	100.0	0.7	0.7	18.0	0.95
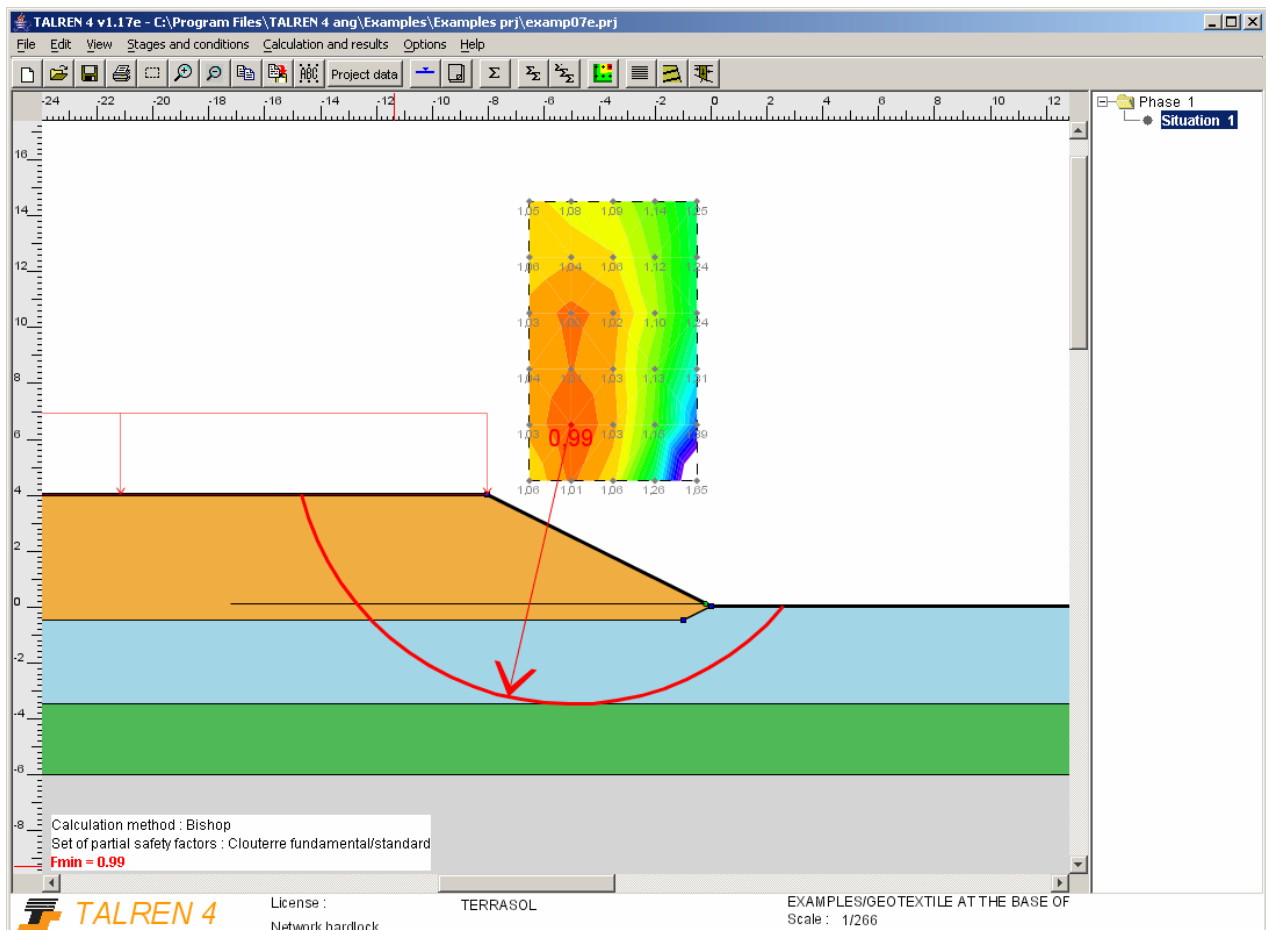
2.8. Example 8: cohesion anisotropy

This example deals with the case of a retaining wall composed of a material with cohesion anisotropy. The curve for cohesion anisotropy is given below.


Soil 3 : Anisotropic cohesion

N°	α	c
1	-90	370
2	-65	375
3	-50	260
4	-25	85
5	5	0
6	45	140
7	65	300
8	90	370
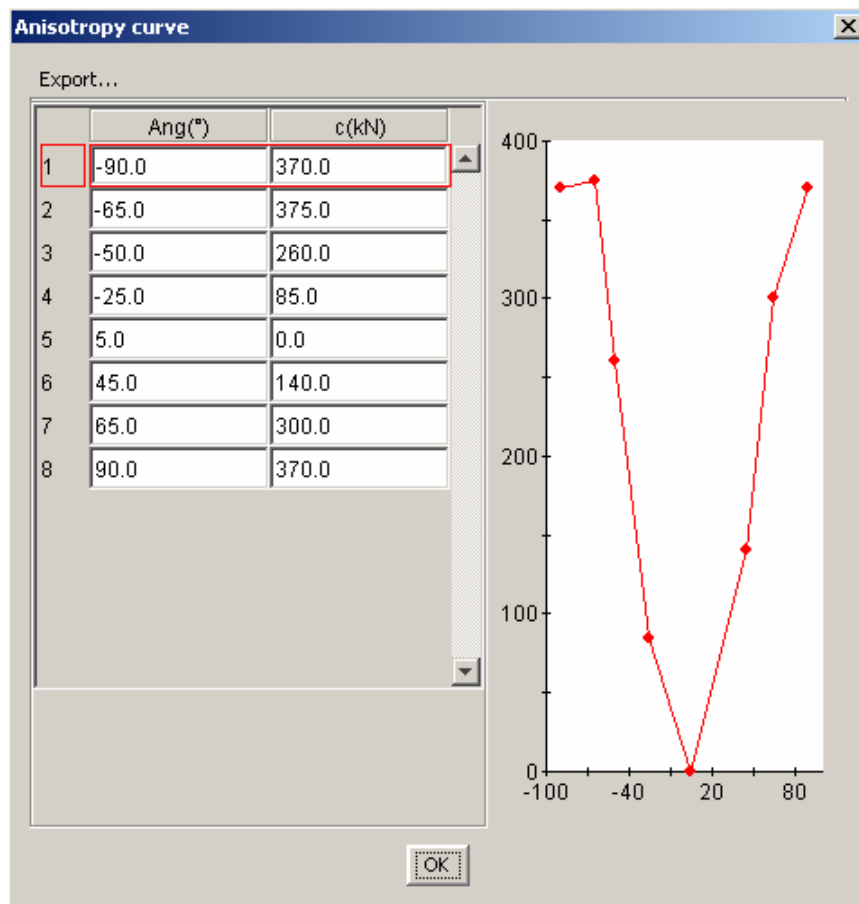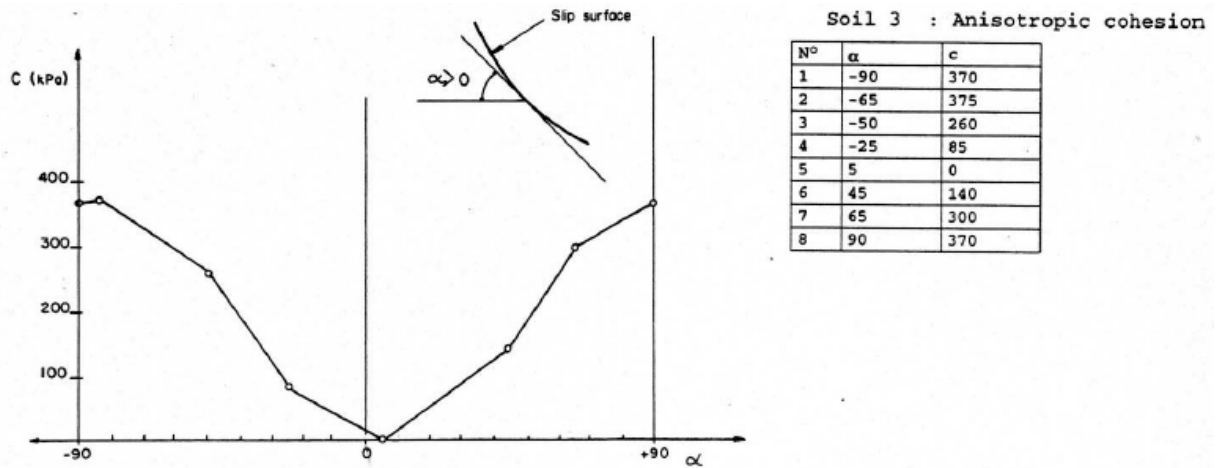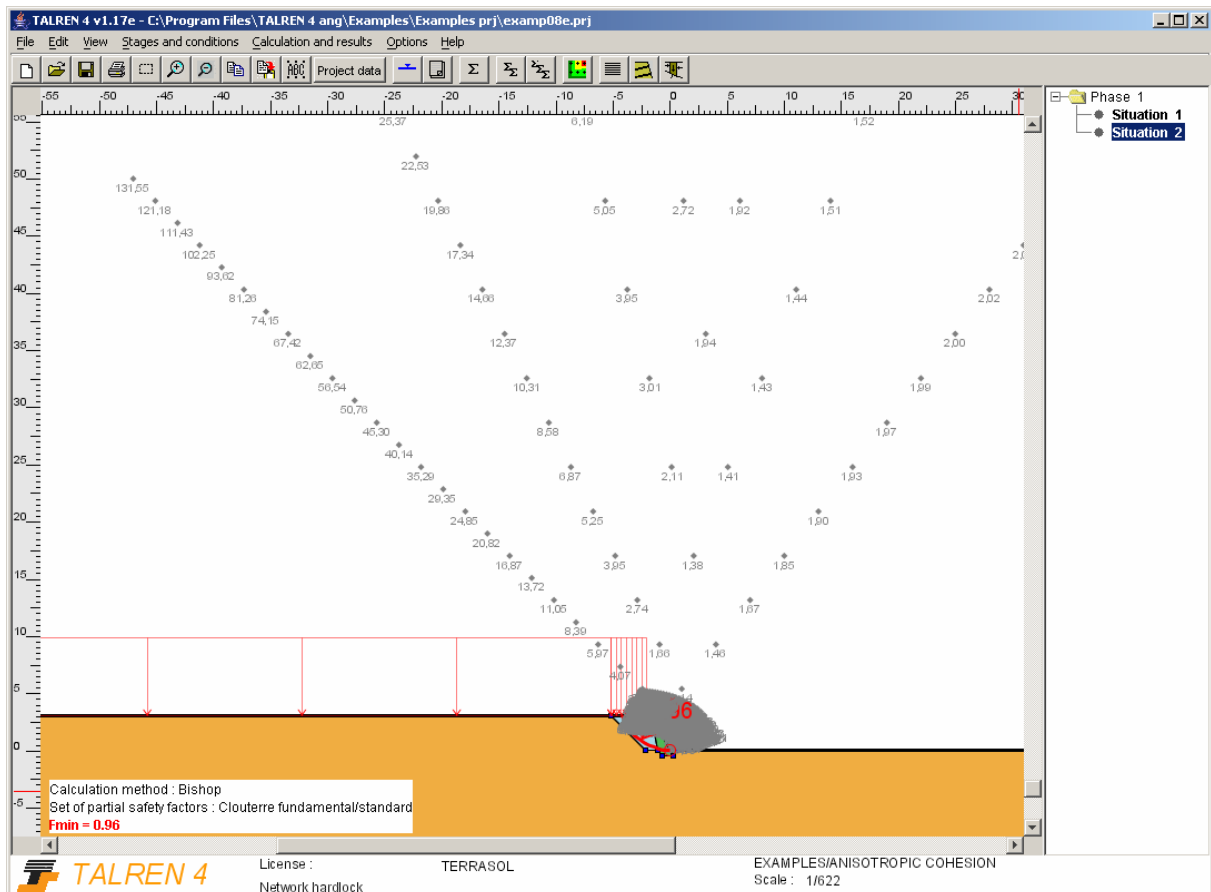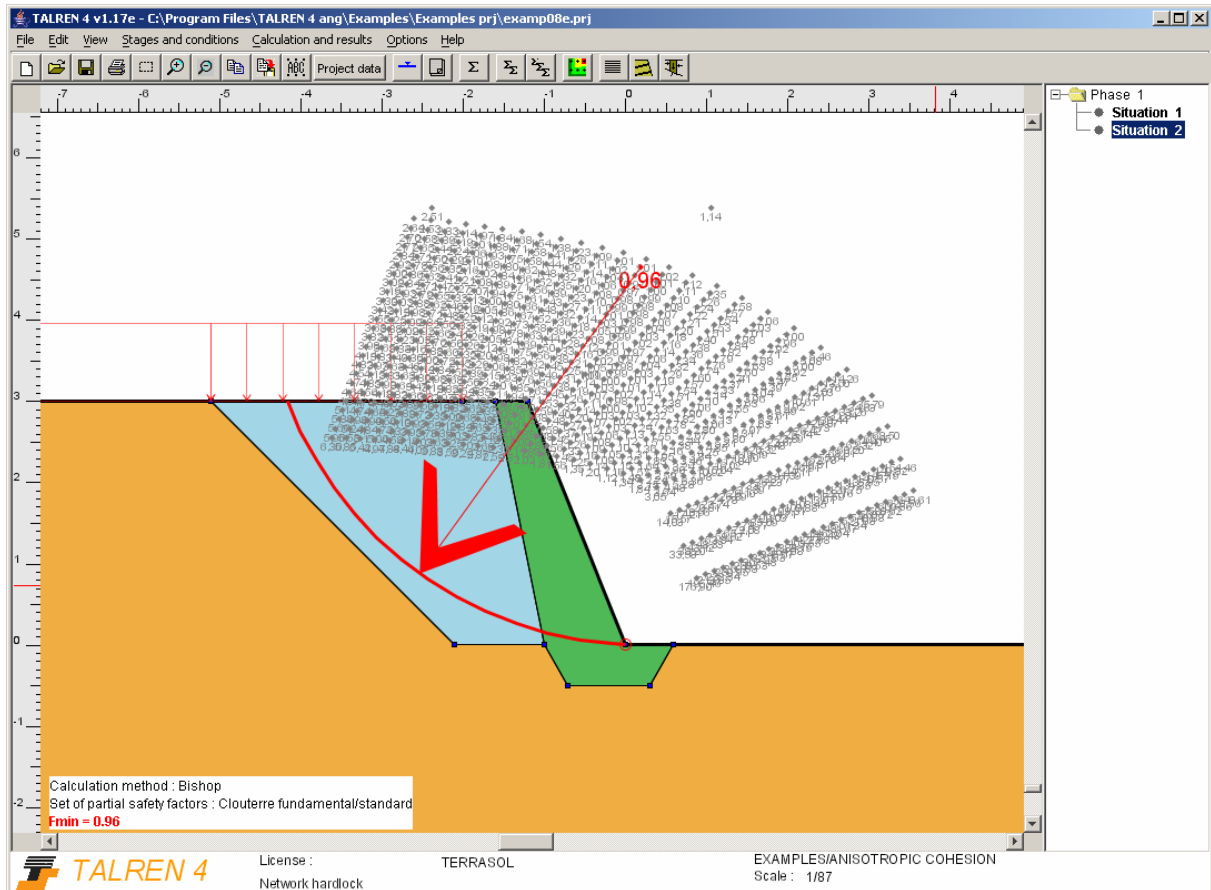
2.9. Example 9: non linear shear-strength curve

This example presents the stability calculation for an embankment built up on soft soil. The characteristics for soft soil layer (non-linear shear-strength curve) are presented below.

| Name	γ(kN/m3)	φ(°)	c(kPa)	Δc(kPa/m)
Layer 1	20	35.0	0.0	0.0
Layer 2	16	Non Lineaire	0.0	0.0
Layer 3	21	35.0	20.0	0.0
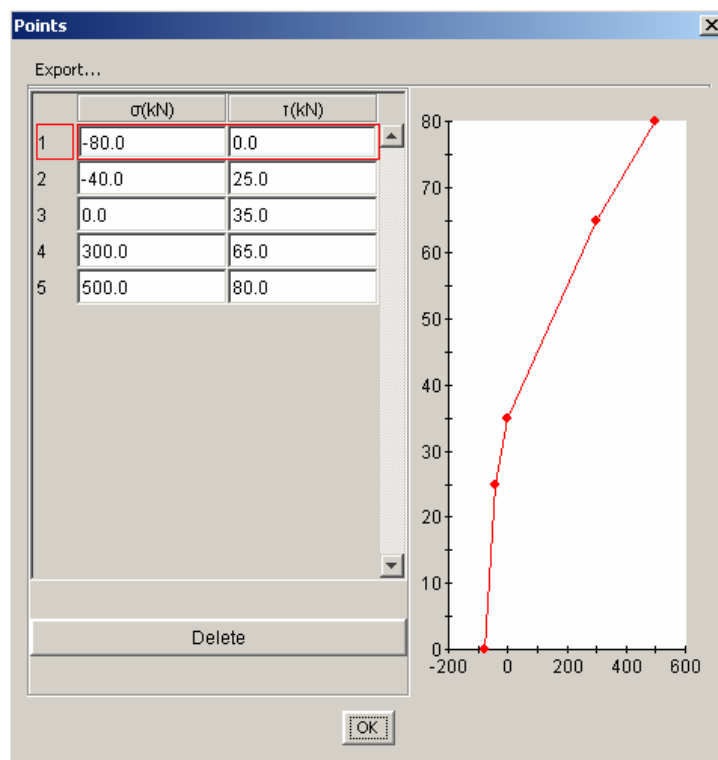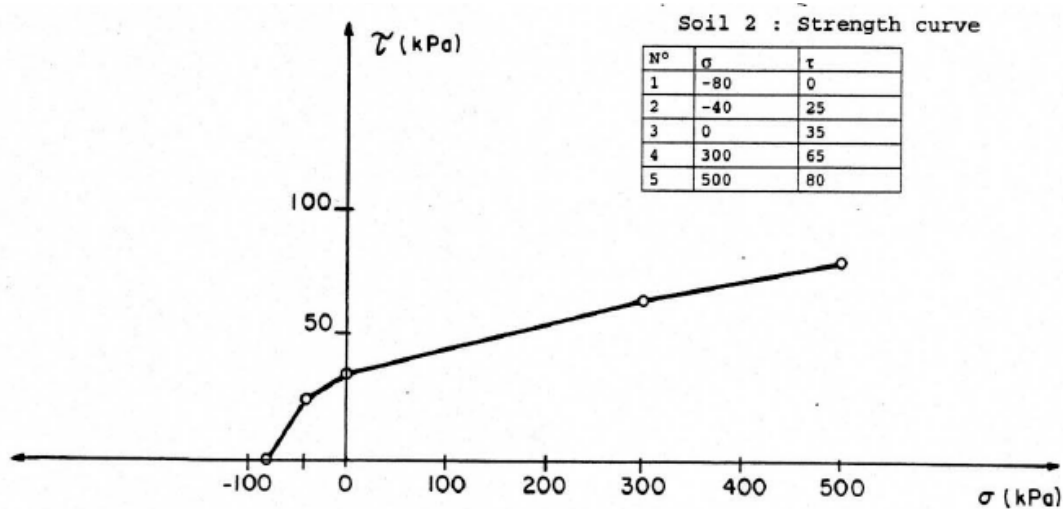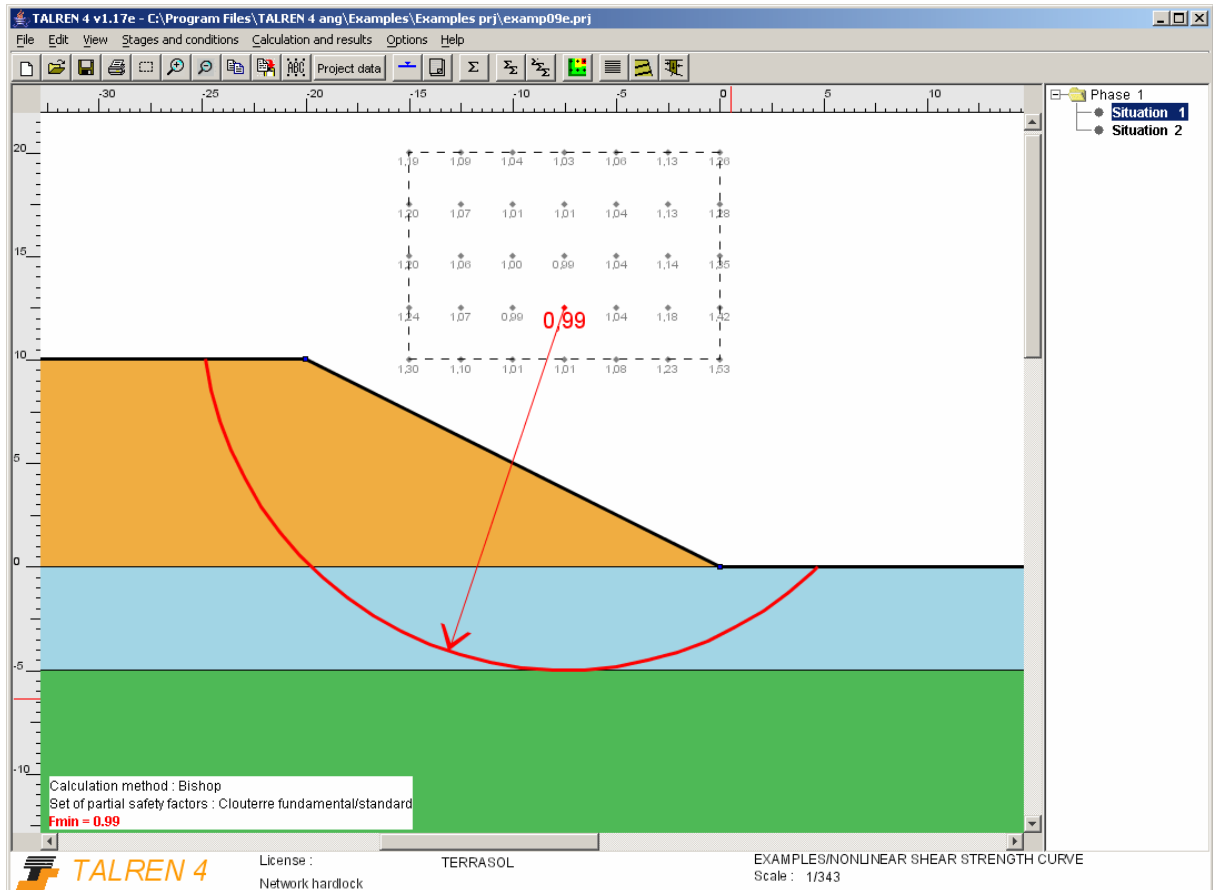
2.10. Example 10: excavation supported with struts in a sensitive area

This example examines the case of an excavation located near an existing building. The excavation walls were supported by 4 rows of temporary struts.

Name	γ(kN/m3)	φ(°)	c(kPa)	Δc(kPa/m)
Layer 1	19	30.0	0.0	0.0
Layer 2	20	20.0	15.0	0.0
Layer 3	21	25.0	20.0	0.0

| Name	X (m)	Y (m)	Horizontal spacing (m)	Inclination /horizontal (°)	TR (kN)
Strut 1	0.0	9.0	2.5	0.0	500.0
Strut 2	0.0	7.0	2.5	0.0	500.0
Strut 3	0.0	5.0	2.5	0.0	500.0
Strut 4	0.0	3.0	2.5	0.0	500.0
Strut 5	0.0	1.0	2.5	0.0	500.0
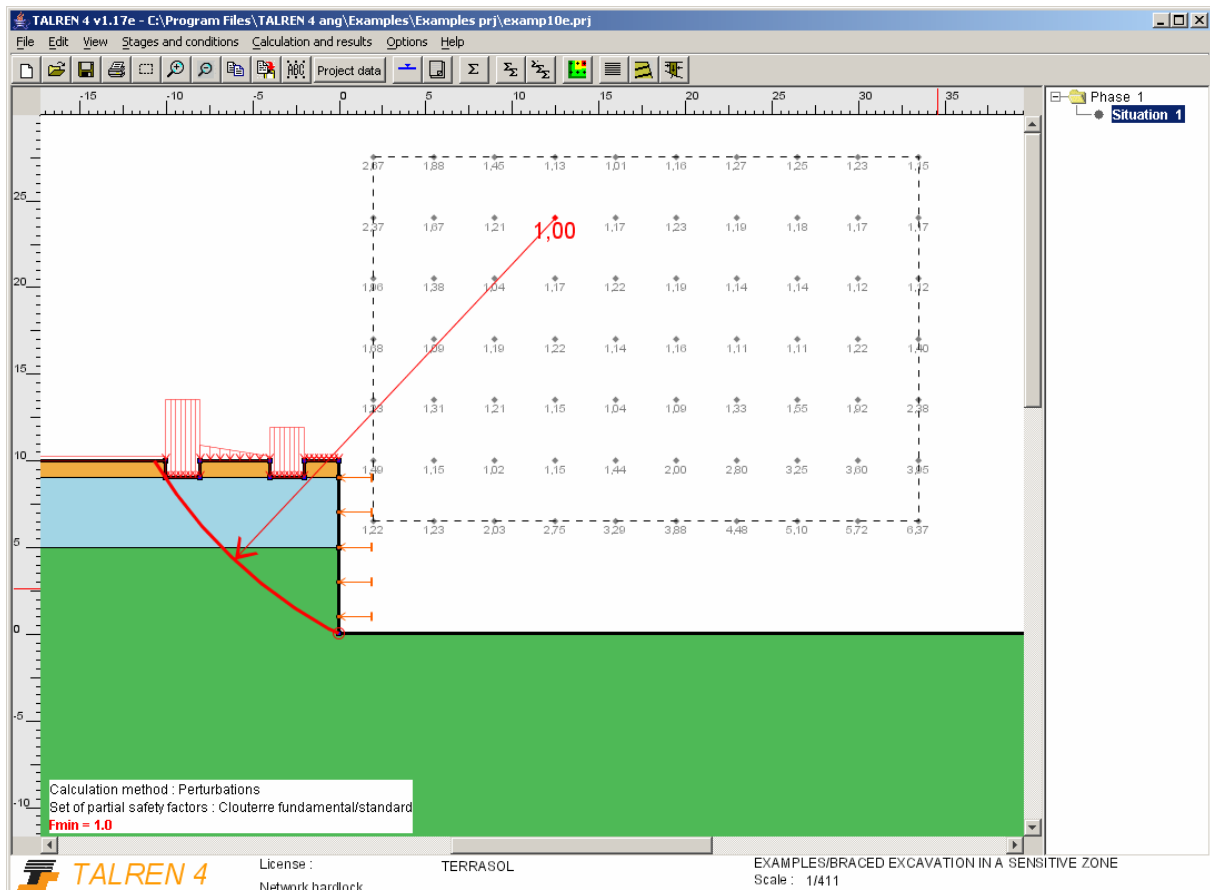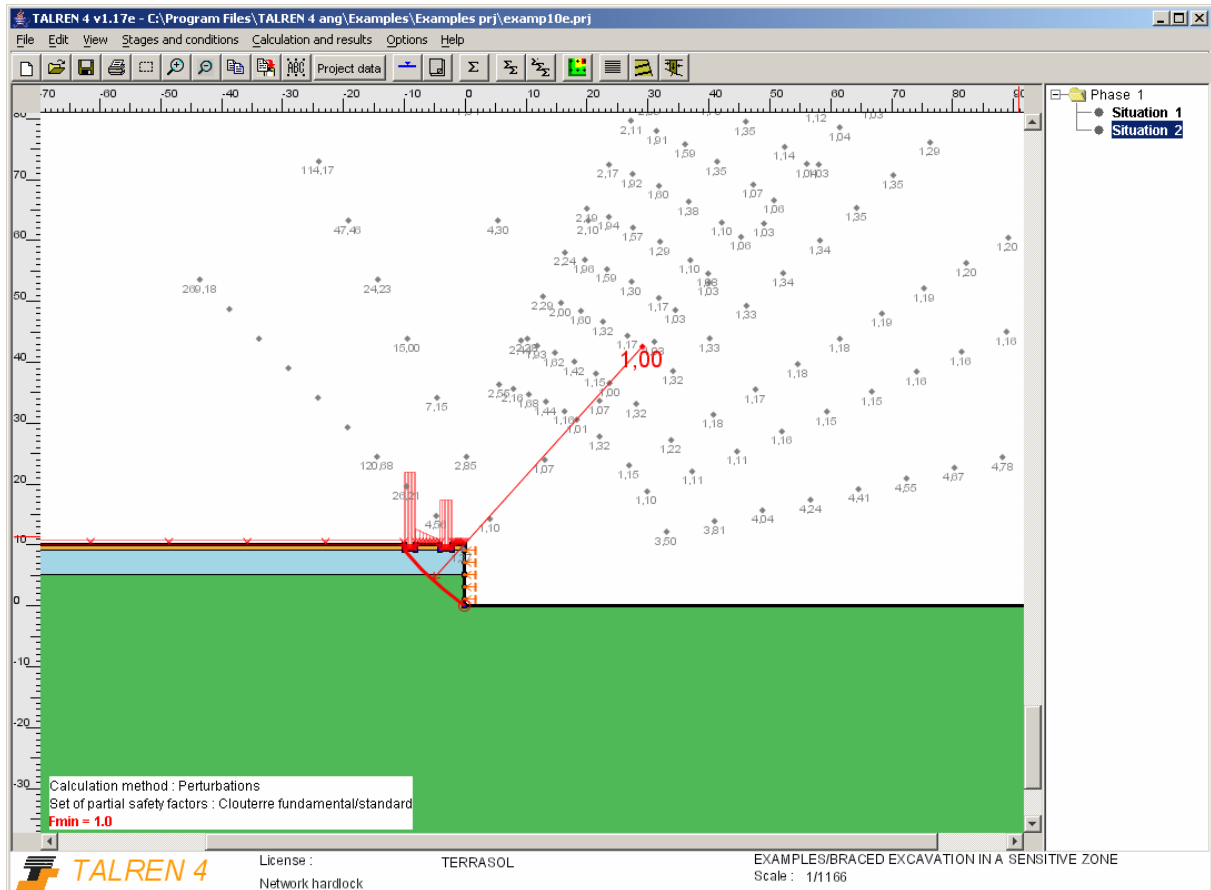
2.11. Example 11: dam with seepage

This example presents the stability analysis for an earth dam in which the seepage was simulated by approximating the equipotential lines by straight lines.

Name	γ(kN/m3)	φ(°)	c(kPa)	Δc(kPa/m)
Layer 1	20	35.0	10.0	0.0
Layer 2	20	50.0	0.0	0.0
Layer 3	20	40.0	0.0	0.0
Layer 4	19	20.0	12.0	0.0
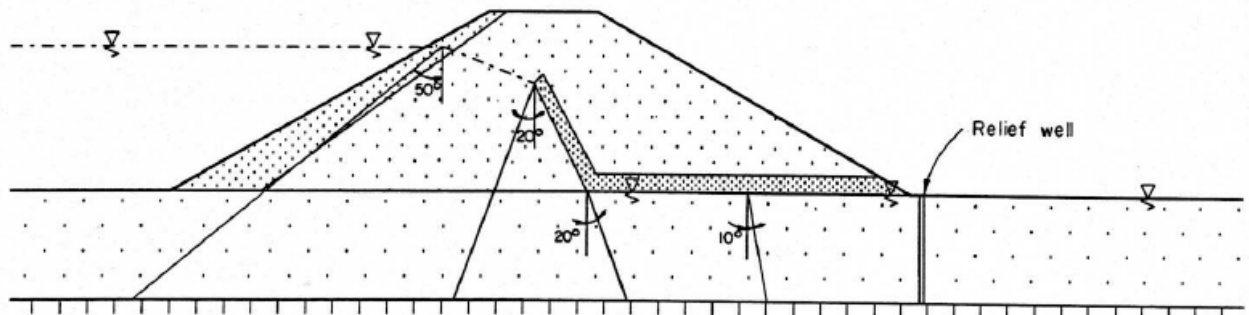
Defining hydraulic conditions

Hydraulic conditions Phreatic level

Phreatic line Define phreatic level

Define a water table base ? ☑
Definition of the water table base Define base of water table
Define an external phreatic line ? ☑
External phreatic line Define external phreatic line
ru for layers ☐

Definition of the water table base

OK   Cancel   Help

Phreatic level

	X(m)	Y(m)	Ang(°)
1	-100.00	8.00	0
2	-26.00	8.00	50
3	-21.00	6.00	20
4	-18.00	0.00	-20
5	-9.00	0.00	-10
6	0.00	0.00	0
7	100.00	0.00	0

Base of water table

	X(m)	Y(m)
1	-100.00	-5.90
2	100.00	-5.90

External phreatic level

Automatic calculation ☐

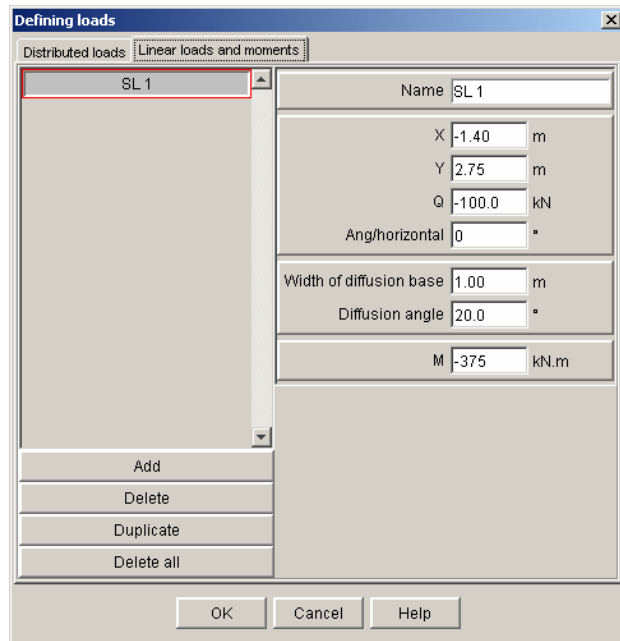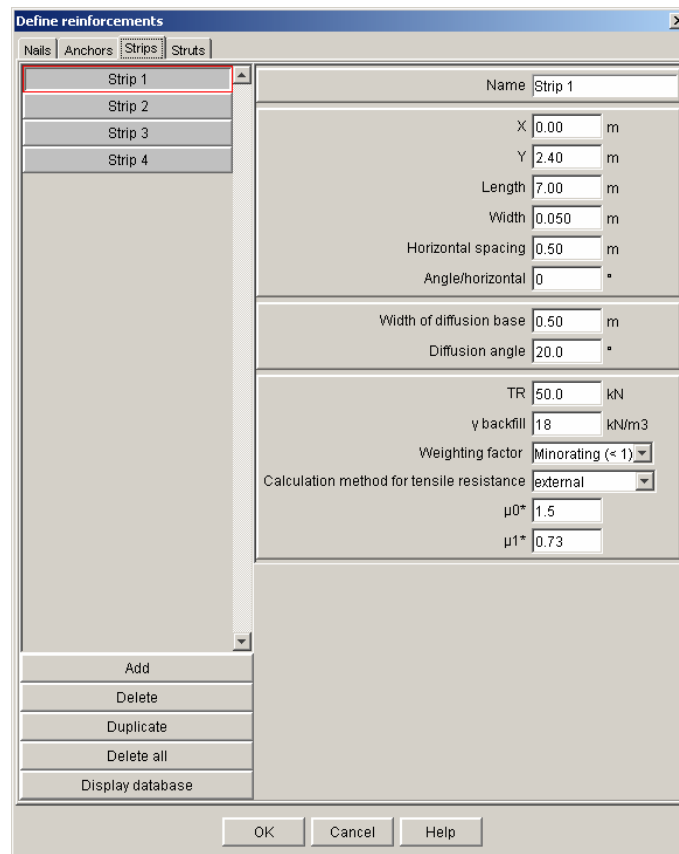|	X(m)	Y(m)
1	-100.00	8.00
2	-27.00	8.00
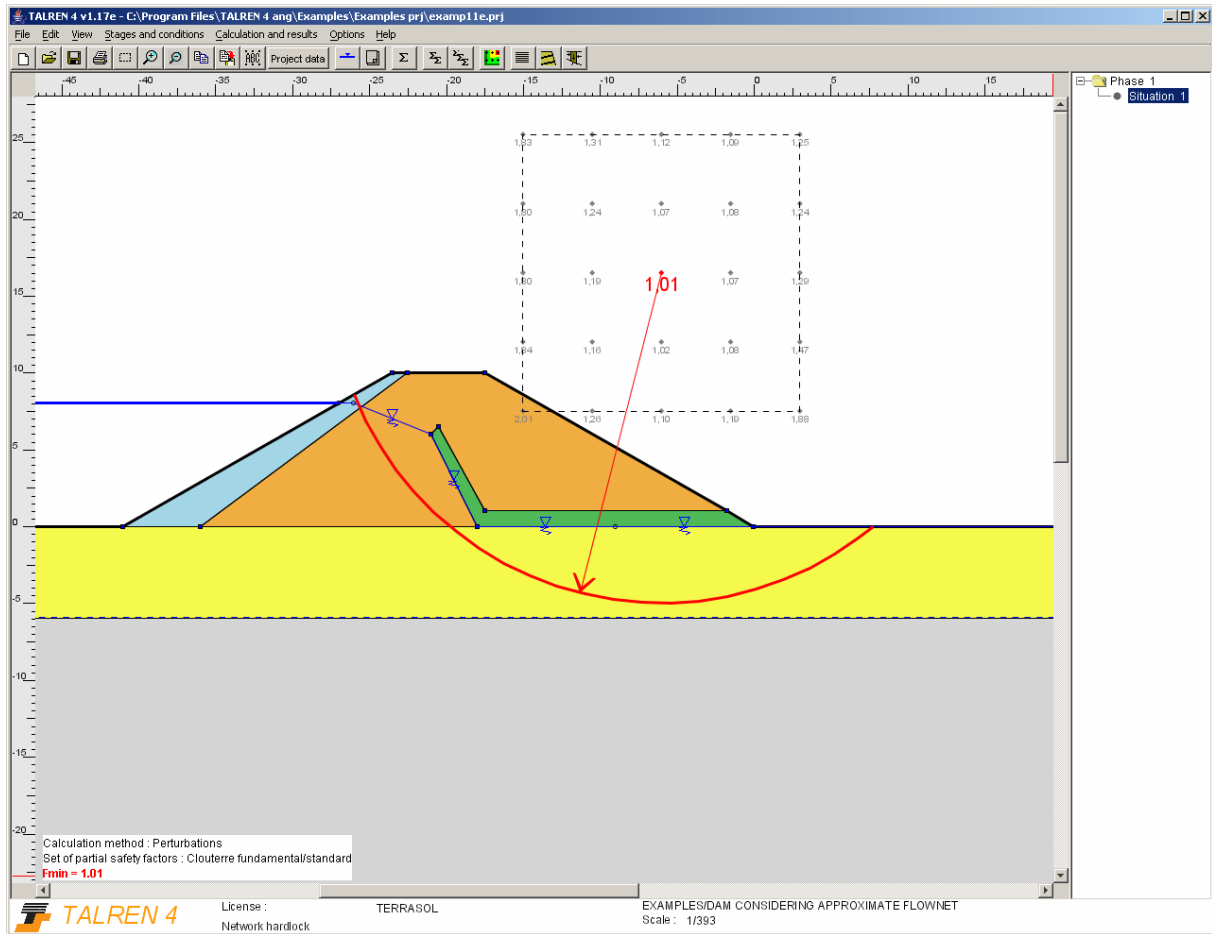
2.12. Example 12: stability of a Reinforced Earth abutment

This example presents the stability calculation for a Reinforced Earth (R.E.) abutment in which the failure circles pass below the foundation of the R.E. facing panels. The internal stability of the R.E. mass was determined according to the specific recommendations for Reinforced Earth.

The soil/reinforcement friction coefficients are those proposed in the recommendations for Reinforced Earth.

Name	γ(kN/m3)	φ(°)	c(kPa)	Δc(kPa/m)
Layer 1	20	36.0	0.0	0.0
LAyer 2	20	36.0	0.0	0.0
Layer 3	18	25.0	15.0	0.0
Layer 4	20	35.0	10.0	0.0
Layer 5	20	36.0	0.0	0.0

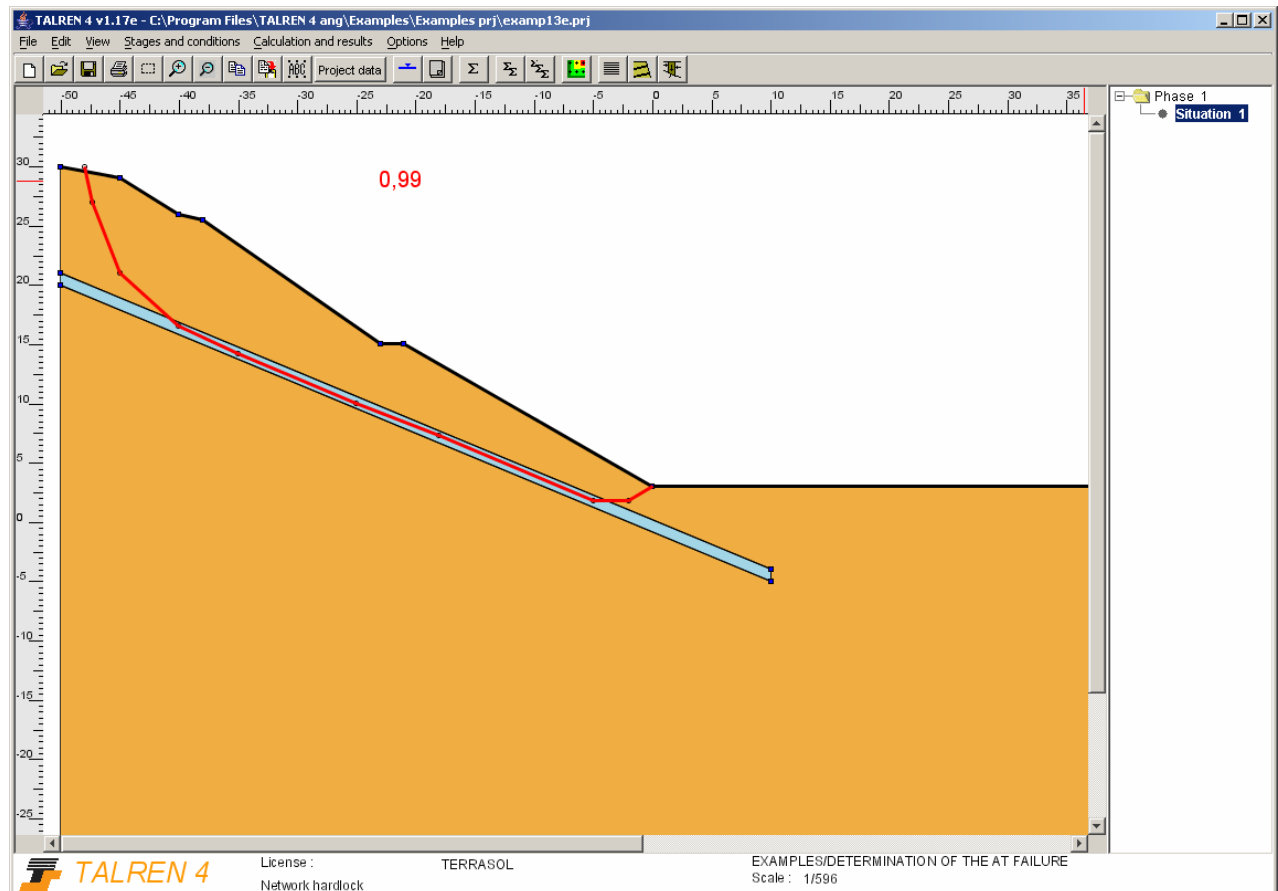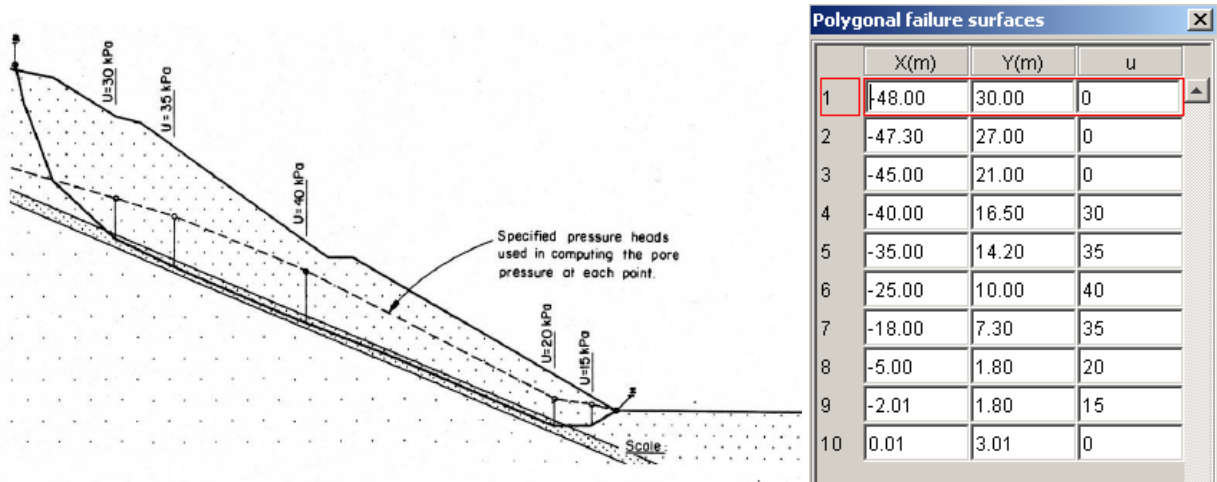## 2.13. Example 13: setting of the characteristics on a failure

This example presents the failure analysis of a slide for which the geometry and hydraulic conditions during failure are known, and for which we need to determine the shear strength properties that lead to failure from back analysis.

The pore pressures were defined along the polygonal failure surface in this example.
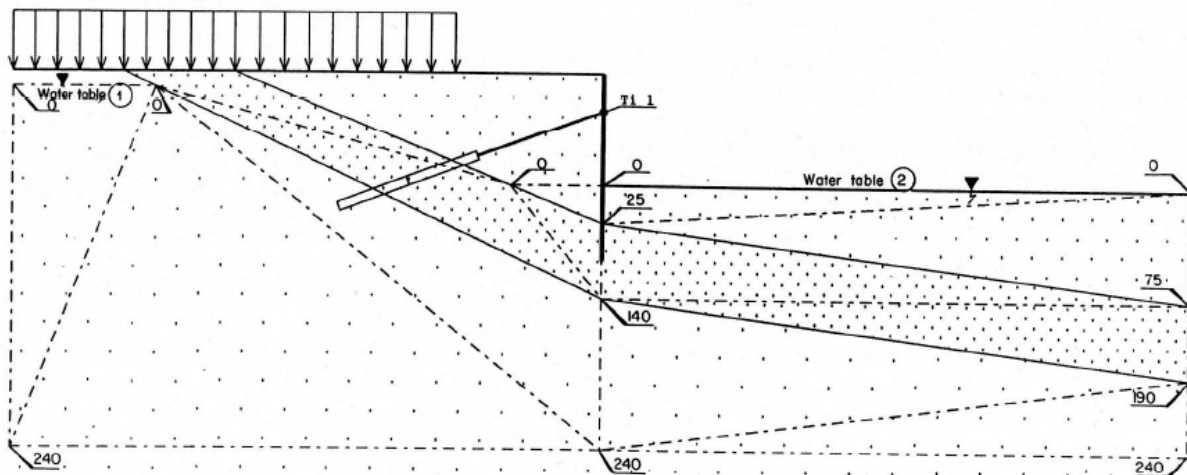

	X(m)	Y(m)	u
1	-48.00	30.00	0
2	-47.30	27.00	0
3	-45.00	21.00	0
4	-40.00	16.50	30
5	-35.00	14.20	35
6	-25.00	10.00	40
7	-18.00	7.30	35
8	-5.00	1.80	20
9	-2.01	1.80	15
10	0.01	3.01	0

2.14. Example 14: diaphragm wall with two water levels

This example deals with the case of an anchored diaphragm wall. The design of the wall and the active anchors was predetermined from an elasto-plastic analysis. We imposed that the circles pass below the bottom of the wall.

This circular failure analysis was used to determine the anchor lengths necessary to ensure global stability. The lengths could not be determined directly from an elasto-plastic analysis. Some cases could require the verification of non-circular failure surfaces passing beyond the fictitious anchor point, which in TALREN corresponds to the mid-grouted length.

This example, in which the pore pressures are defined at the nodes of a triangular manual mesh, presents the particular case of 2 water tables having different pressures heads and in which a head loss was caused by the presence of a low permeability soil layer separating them.

Name	γ(kN/m3)	φ(°)	c(kPa)	Δc(kPa/m)
Layer 1	20	30.0	10.0	0.0
Layer 2	20	25.0	15.0	0.0
Layer 3	20	35.0	10.0	0.0

Define a triangular mesh of pore pressur...

Nodes | Triangles

	X(m)	Y(m)	u(kN)
1	-40.00	19.00	0.0
2	-30.33	19.00	0.0
3	-6.25	12.50	0.0
4	0.00	12.50	0.0
5	40.00	12.50	0.0
6	0.00	10.00	25.0
7	40.00	5.00	75.0
8	0.00	5.00	140.0
9	40.00	0.00	190.0
10	-40.00	-5.00	240.0
11	0.00	-5.00	240.0
12	40.00	-5.00	240.0

Add

Delete

Delete all

Note: possible selection on drawing

OK   Cancel   Help

Calculation method : Bishop
Set of partial safety factors : Clouterre fundamental/standard
Fmin = 1.02
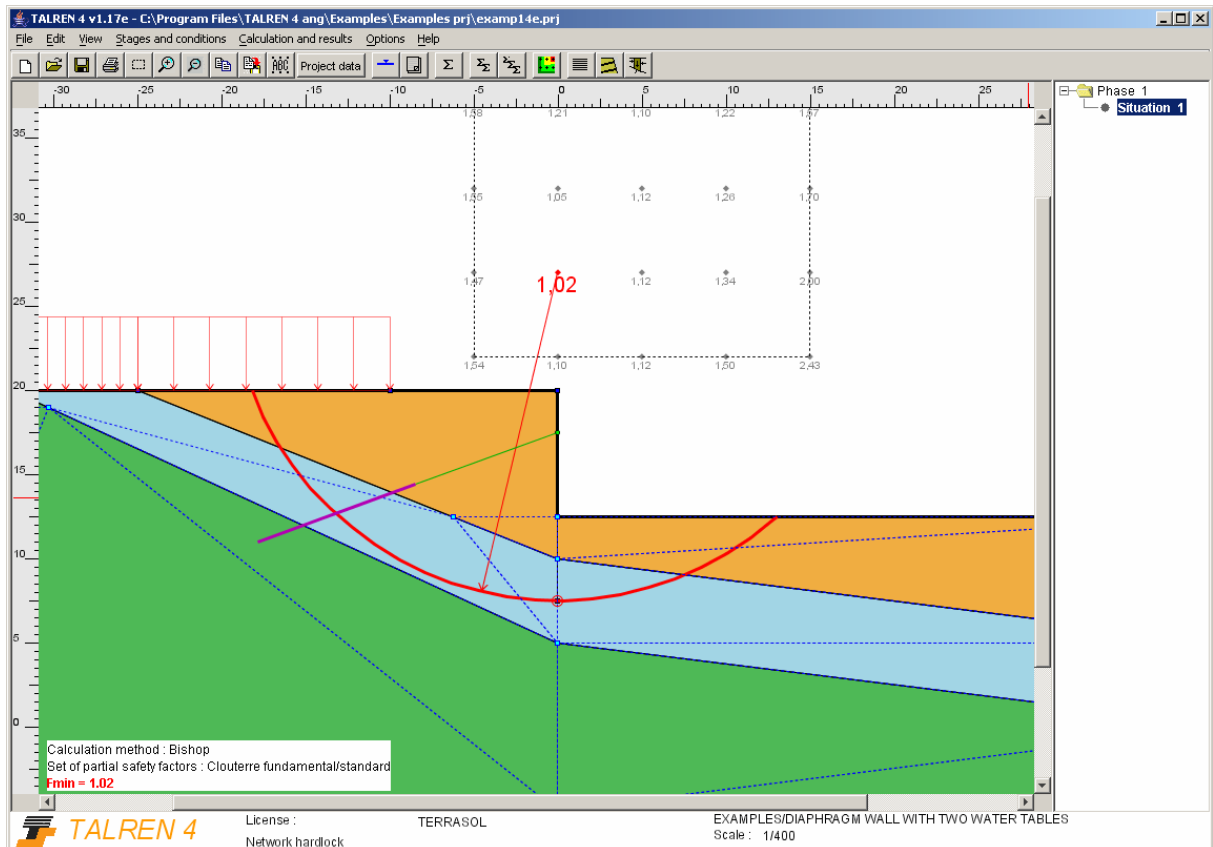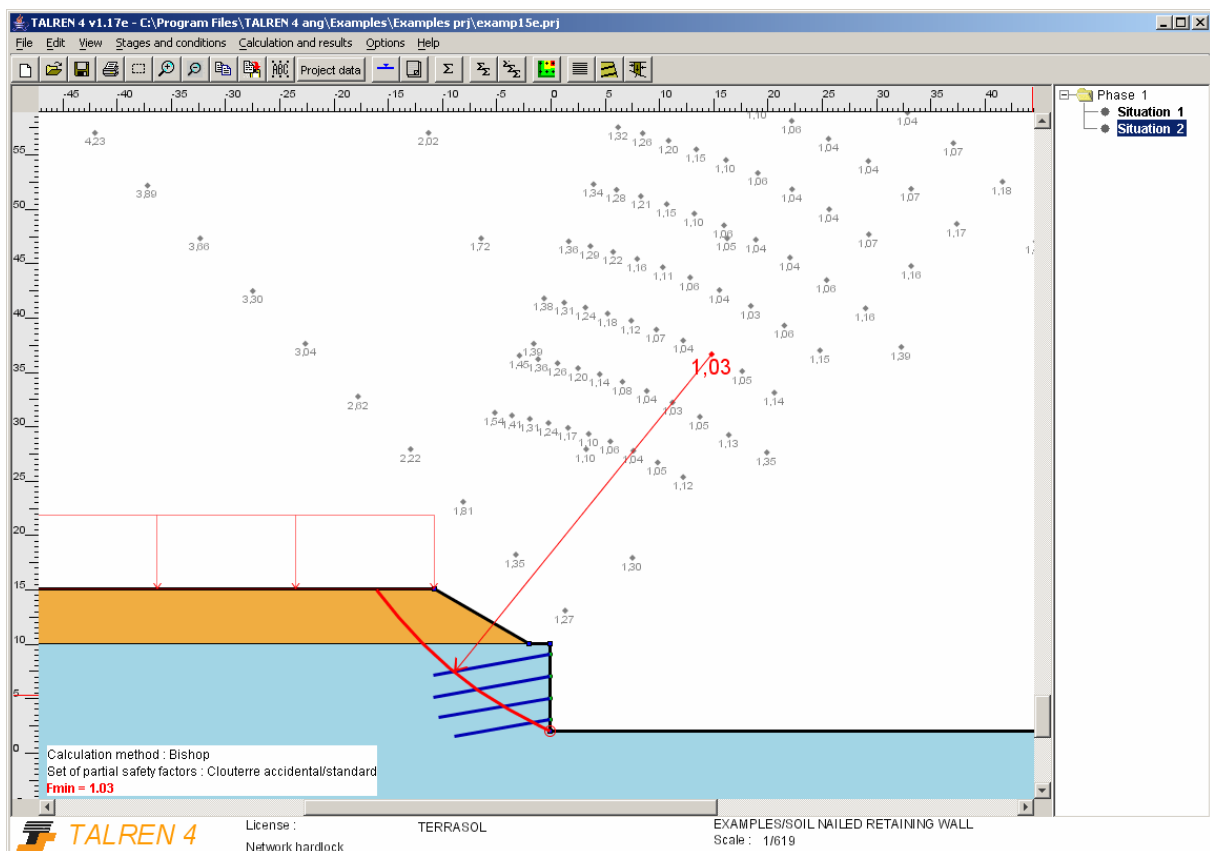
2.15. Example 15: nailed retaining wall with seismic conditions

This example presents the case of a soil nailed structure where seismic loading was simulated by the introduction of horizontal and vertical acceleration coefficients.

Name	γ(kN/m3)	φ(°)	c(kPa)	Δc(kPa/m)
Layer 1	20	35.0	5.0	0.0
Layer 2	20	30.0	10.0	0.0

Seismic characteristics ☑

ah/g 0.12   av/g 0.06

2.16. Example 16: stability of a quay wall (concrete cylinders)

This example deals with the stability of a quay wall composed of concrete cylinders and fill.

The critical state in which the internal water level is higher than the external water level was examined.
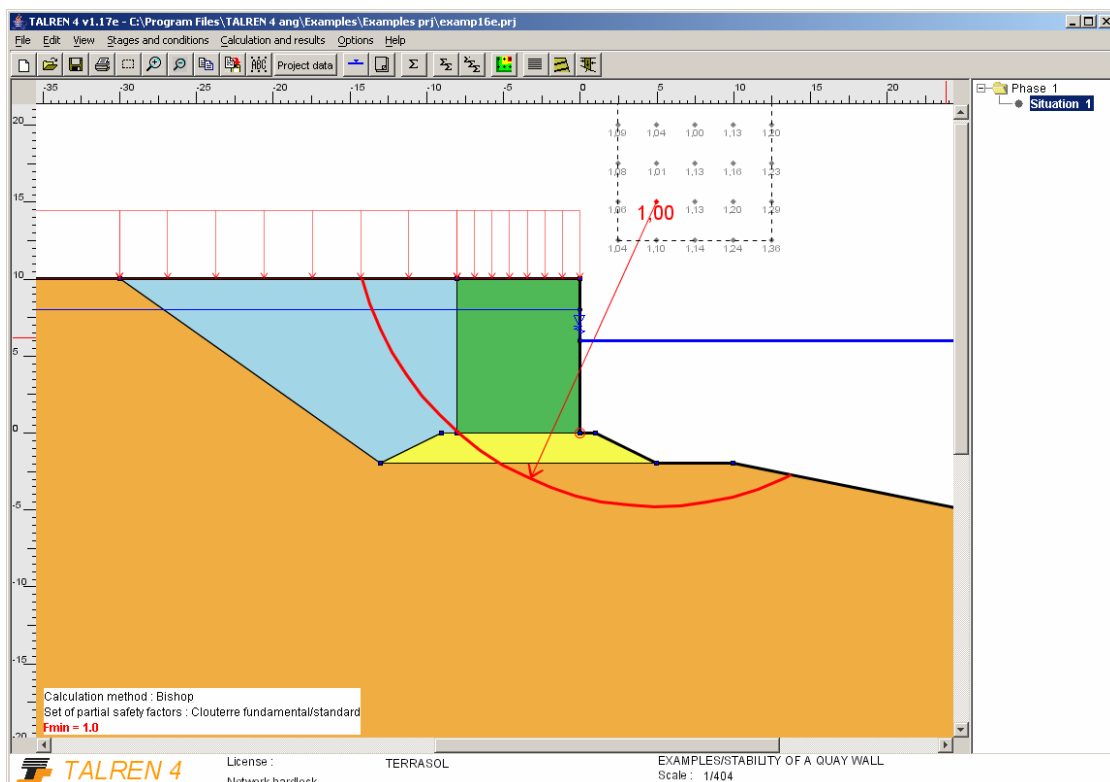

N°	σ1	σ2	Γq
Sd1	20	20	1.33
Sd2	20	20	1.33
Sd3	20	20	1.33

DISTRIBUTED LOADS
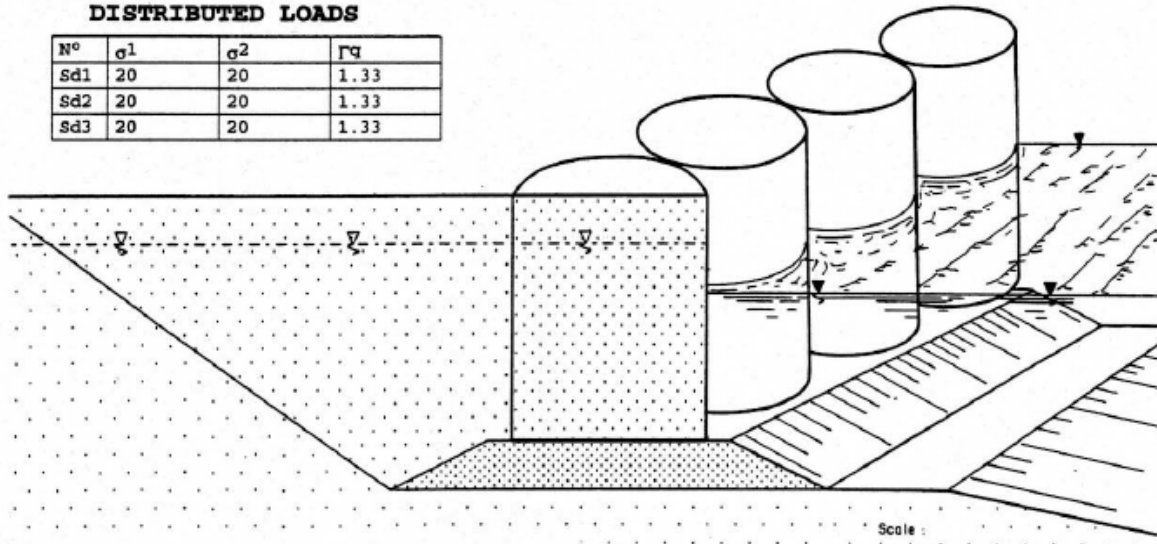
Name	γ(kN/m3)	φ(°)	c(kPa)	Δc(kPa/m)
Layer 1	20	30.0	15.0	0.0
Layer 2	20	30.0	0.0	0.0
Layer 3	20	0.0	1000.0	0.0
Layer 4	20	45.0	0.0	0.0

External phreatic level

Automatic calculation ☑

## 2.17. Example 17: stabilisation of a sliding by piles

This example presents the case of an embankment on an unstable slope reinforced with 3 rows of piles, equivalent in Talren to vertical nails working in shear.

The effective shear strength characteristics of the soil leading to failure were determined using back calculations.

The purpose was to increase stability by 20 to 30 %.

Name	γ(kN/m3)	φ(°)	c(kPa)
Layer 1	19	25.0	0.0
Layer 2	19	13.0	0.0
Layer 3	19	10.0	0.0
Layer 4	20	35.0	20.0

Name	Length (m)	Horizontal spacing (m)	Inclination /horizontal (°)	Calculation mode	Rsc (kN)	EI (kN.m2)	Mmax (kN.m)
Nail 1	16.0	1.0	2.4	T_{cal}, C_{imp}	540	0	0
Nail 2	16.0	1.0	2.4	T_{cal}, C_{imp}	540	0	0
Nail 3	16.0	1.0	2.4	T_{cal}, C_{imp}	540	0	0
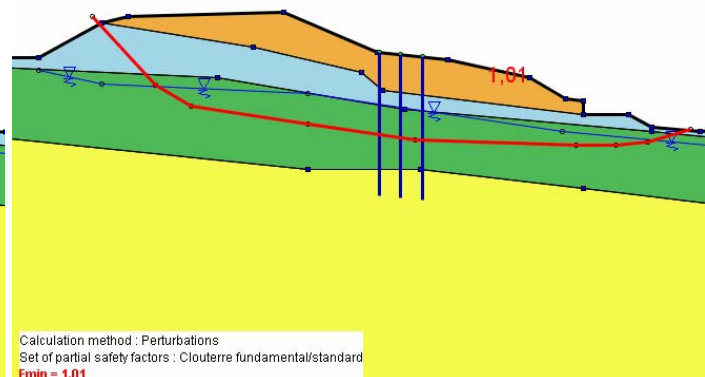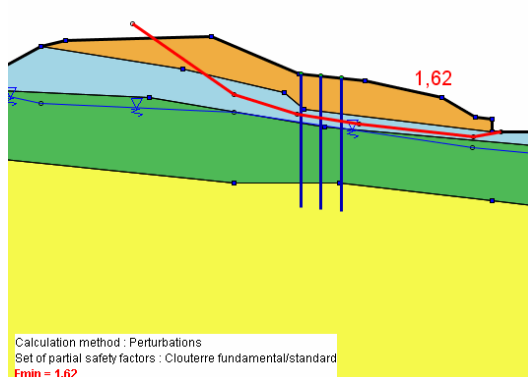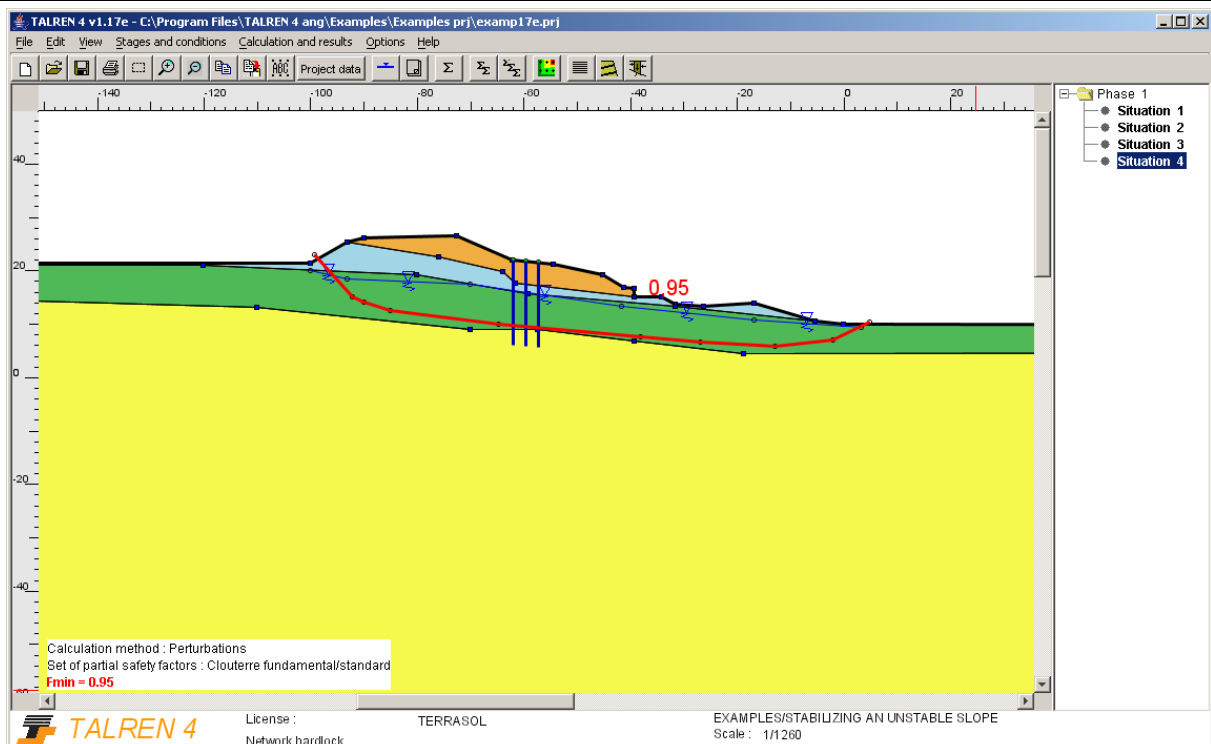
## 2.18. Example 18: stabilisation of an embankment on a soft soil with micro-piles

This example deals with the reinforcement of an embankment built up on a highly compressible soil with 5 rows of micropiles. In Talren, the inclusions are equivalent to nails working in shear.

The effective shear strength characteristics of the soil resulting in failure were determined from back calculations.

Name	γ(kN/m3)	φ(°)	c(kPa)	Δc(kPa/m)	qs nails(kPa)	qs anchors(kPa)	α	pl(kPa)	KsB(kPa)
Layer 1	18	30.0	0.0	0.0	40			400	8800
Layer 2	16	0.0	6.0	0.0	20			40	180
Layer 3	16.5	10.0	12.0	0.0	20			80	200
Layer 4	17	10.0	20.0	0.0	20			140	240
Layer 5	20	35.0	0.0	0.0	80			800	17000

Soil weight = favourable action for layer 2

| Name	Length (m)	Horizontal spacing (m)	Inclination /horizontal (°)	Equivalent grout radius (m)	Calculation mode	Rcis (kN)	TR (kN)	Mmax (kN.m)	EI (kN.m2)
Nail 1	13.0	3.0	110.0	0.166	T_{cal}, C_{cal}	-1110	-2220	183	7644
Nail 2	13.0	3.0	95.0	0.083	T_{cal}, C_{cal}	-1110	-2220	183	7644
Nail 3	13.0	3.0	90.0	0.083	T_{cal}, C_{cal}	-1110	-2220	183	7644
Nail 4	13.0	3.0	85.0	0.083	T_{cal}, C_{cal}	-1110	-2220	183	7644
Nail 5	13.0	3.0	70.0	0.166	T_{cal}, C_{cal}	-1110	-2220	183	7644
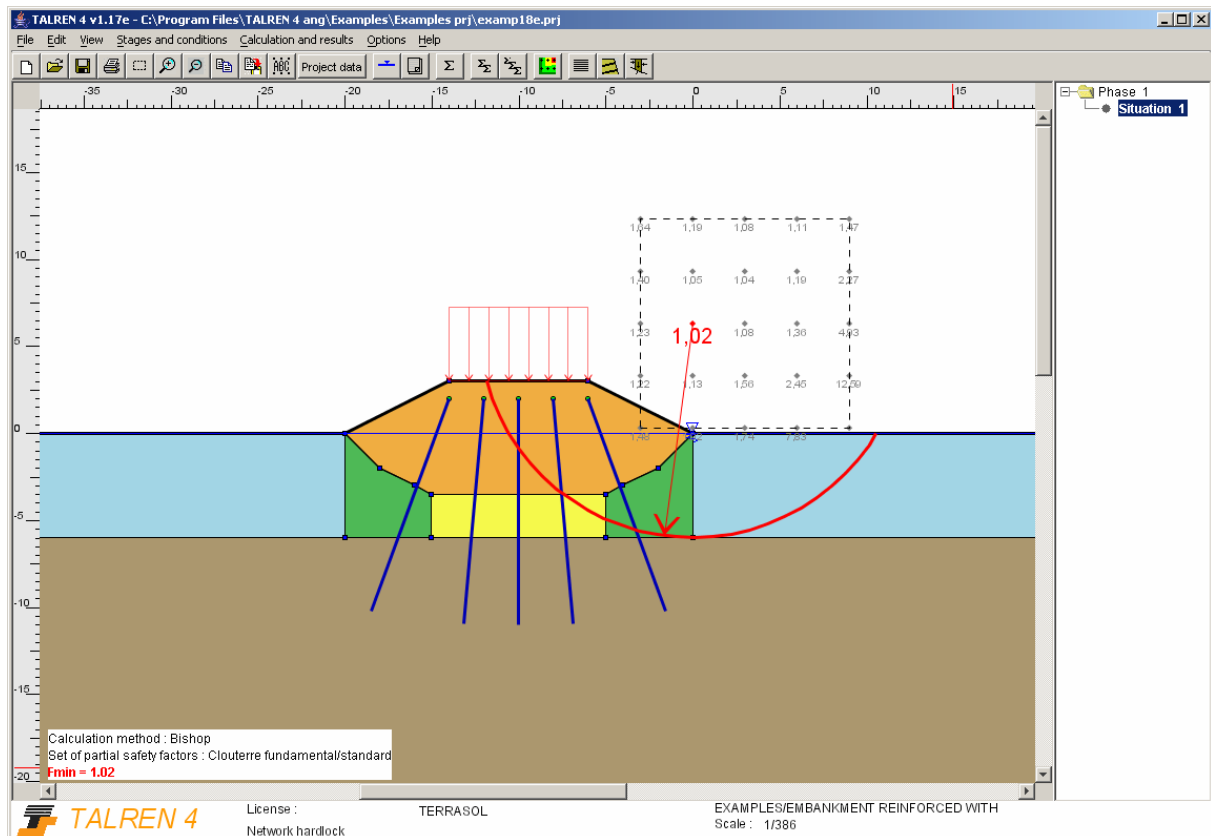
2.19. Example 19: retaining structure for a sloped embankment

This example deals with the case of a retaining structure for an embankment whose slope angle is close to the soil friction angle. Under these conditions, the retaining structure is generally not adequate to provide an acceptable safety factor for large radius failure surfaces. The traditional approach consists in calculating the safety factor for the slip surfaces in which the horizontal distance between the bottom of the retaining structure and the upper or lower emergence point of the failure surface is limited to 3H (H being the excavation height near the structure).

A second approach is to apply different partial safety factors to the soil properties (ULS method). This approach consists in applying the classical safety factors to the soils located within the reinforced zone. Outside the reinforced zone, the partial safety factor applied to the soil properties is equal to the safety factor of the original slope before construction of the retaining structure. This value in then adjusted by the partial safety factor related to the calculation method Γ_{s3}. The design of the retaining structure will thereby ensure that its construction will not affect the stability for large failure surfaces. For this example, the slope angle is equal to 30° with respect to the horizontal and the internal friction angle is 35°. This results in a safety factor of 1.213 for the original slope. Consequently, we imposed a partial safety factor Γ_φ for the soil located outside the reinforced zone equal to 1,21/Γ_{s3}, which leads to a value of 1.078.
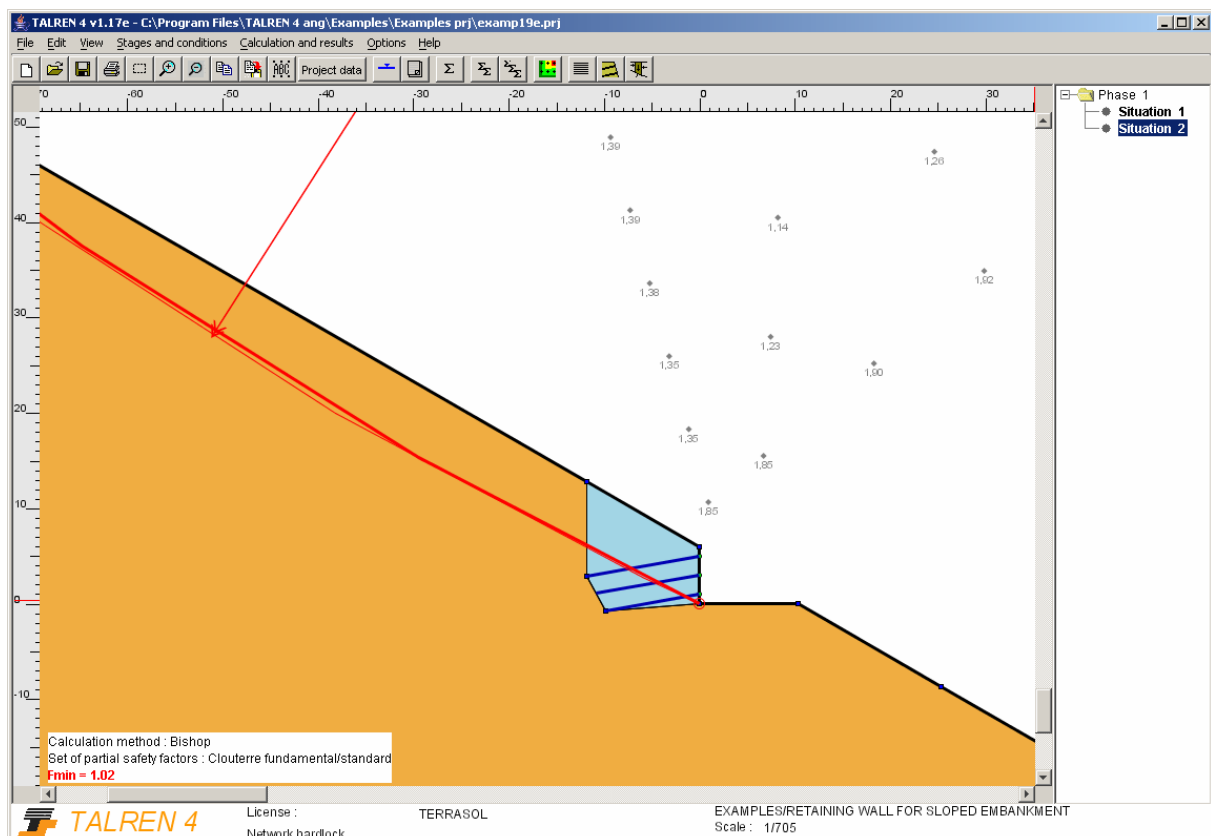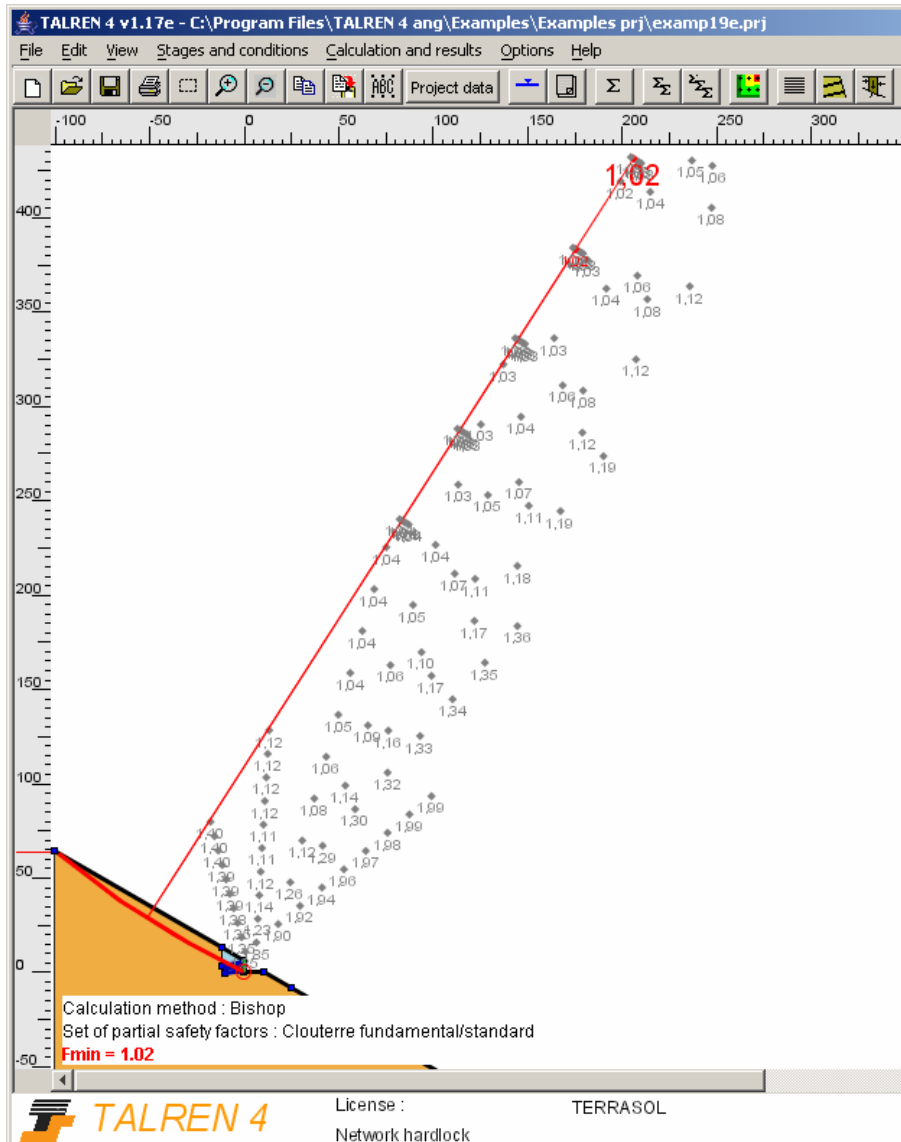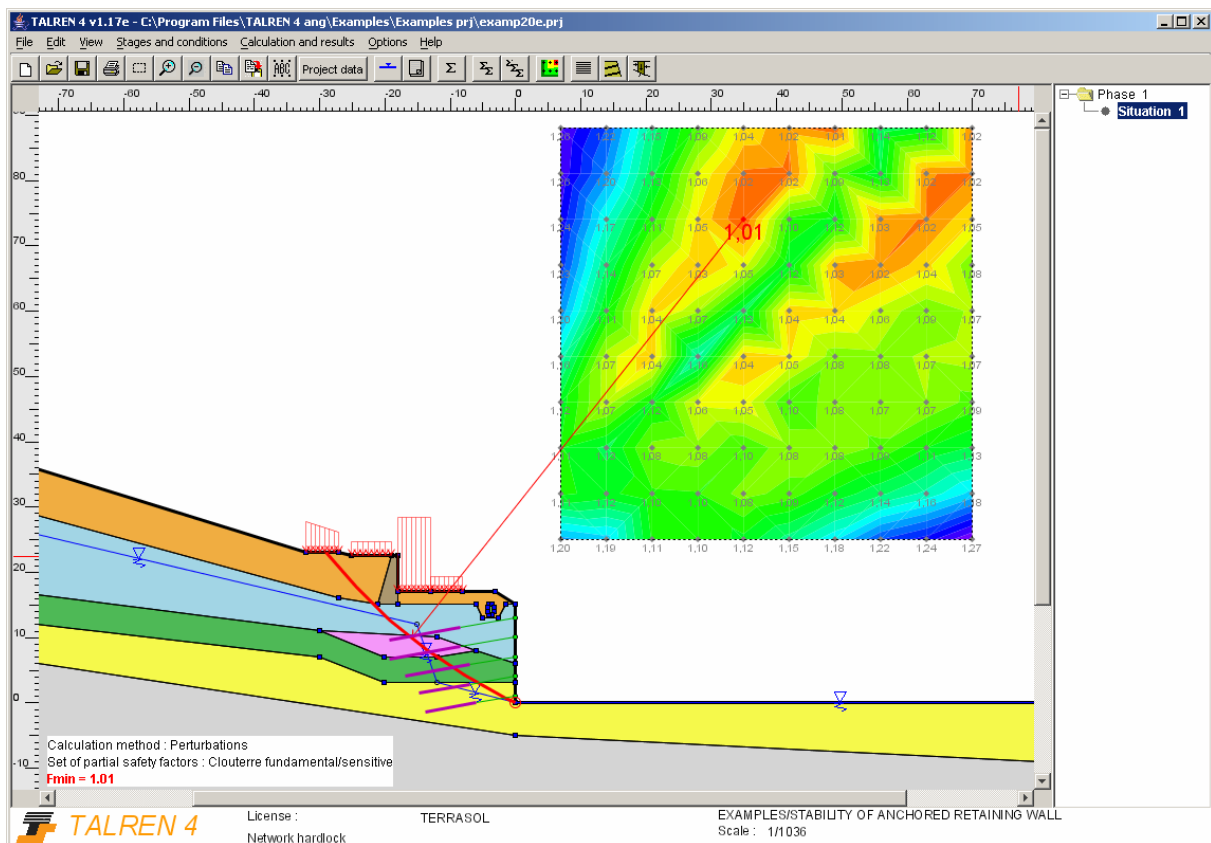
2.20. Example 20: anchored retaining structure

This example presents a calculation in which the pull-out resistance is determined as a function of the grouted length beyond the failure surface.

Name	γ(kN/m3)	φ(°)	c(kPa)	Δc(kPa/m)
Layer 1	19	25.0	10.0	0.0
Layer 2	19	35.0	10.0	0.0
Layer 3	19	35.0	20.0	0.0
Layer 4	19	36.0	30.0	0.0
Layer 5	20	40.0	200.0	0.0
Layer 6	17	30.0	20.0	0.0
Layer 7	0	0.0	0.0	0.0

2.21. Exemple 21: backfill on a soil with a variable cohesion with depth

This example deals with the case of a backfill constructed on a soil (layer 2) which has a cohesion varying with depth.

Name	γ(kN/m3)	φ(°)	c(kPa)	Δc(kPa/m)
Layer 1	19	35.0	0.0	0.0
Layer 2	18	0.0	60.0	5.0
Layer 3	20	35.0	20.0	0.0
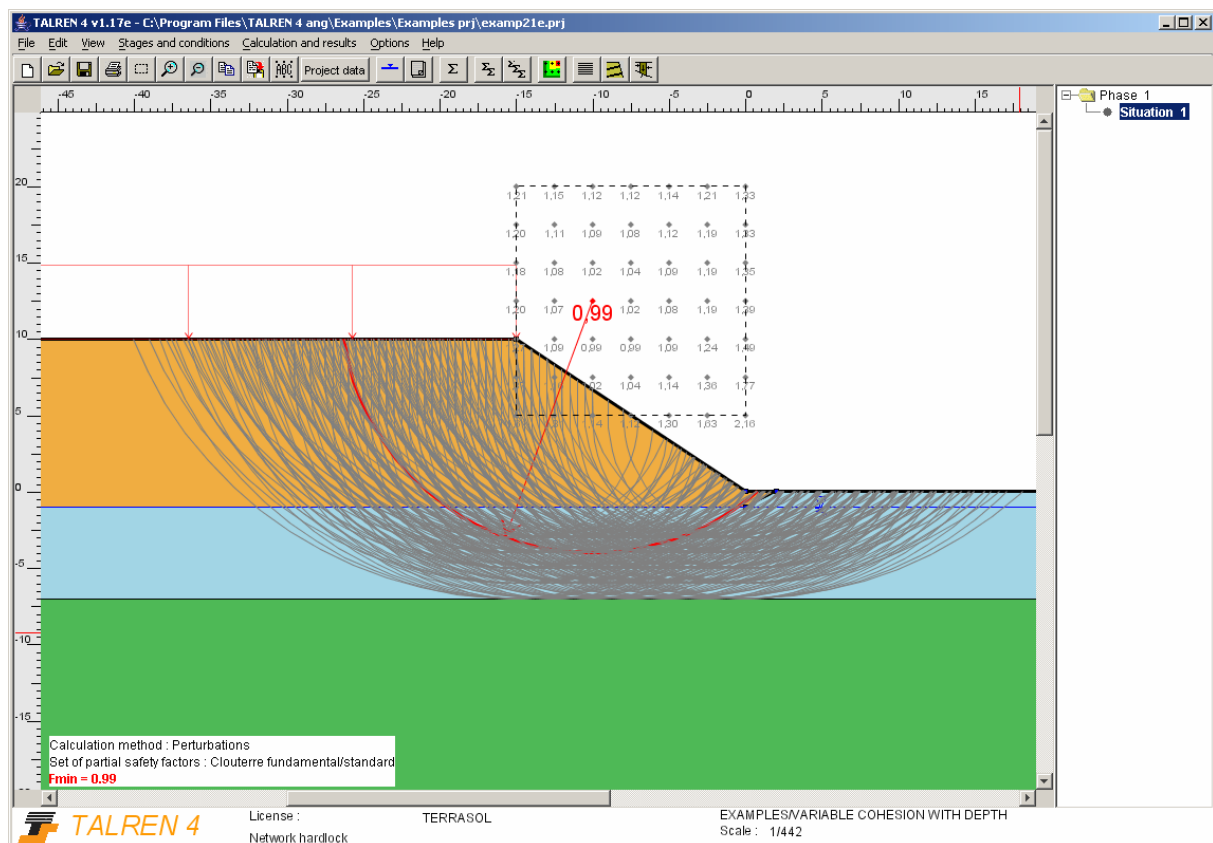
2.22. Example 22: slope stabilisation with piles(using variable shear option)

For this example, the shear forces in the piles were determined using the PILATE program (LCPC), imposing a soil displacement to the portion of the inclusion located within the active zone. PILATE allows for the determination of the shear force mobilised in an inclusion embedded in a multi-layered medium for each node in which the shear force will be imposed in Talren.
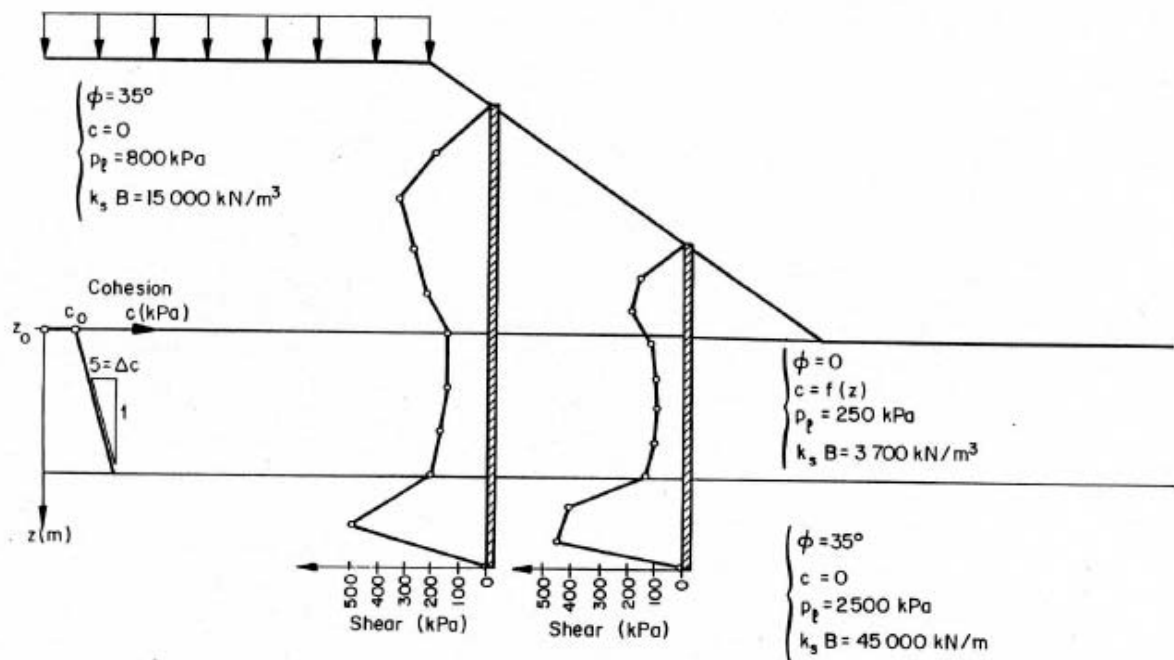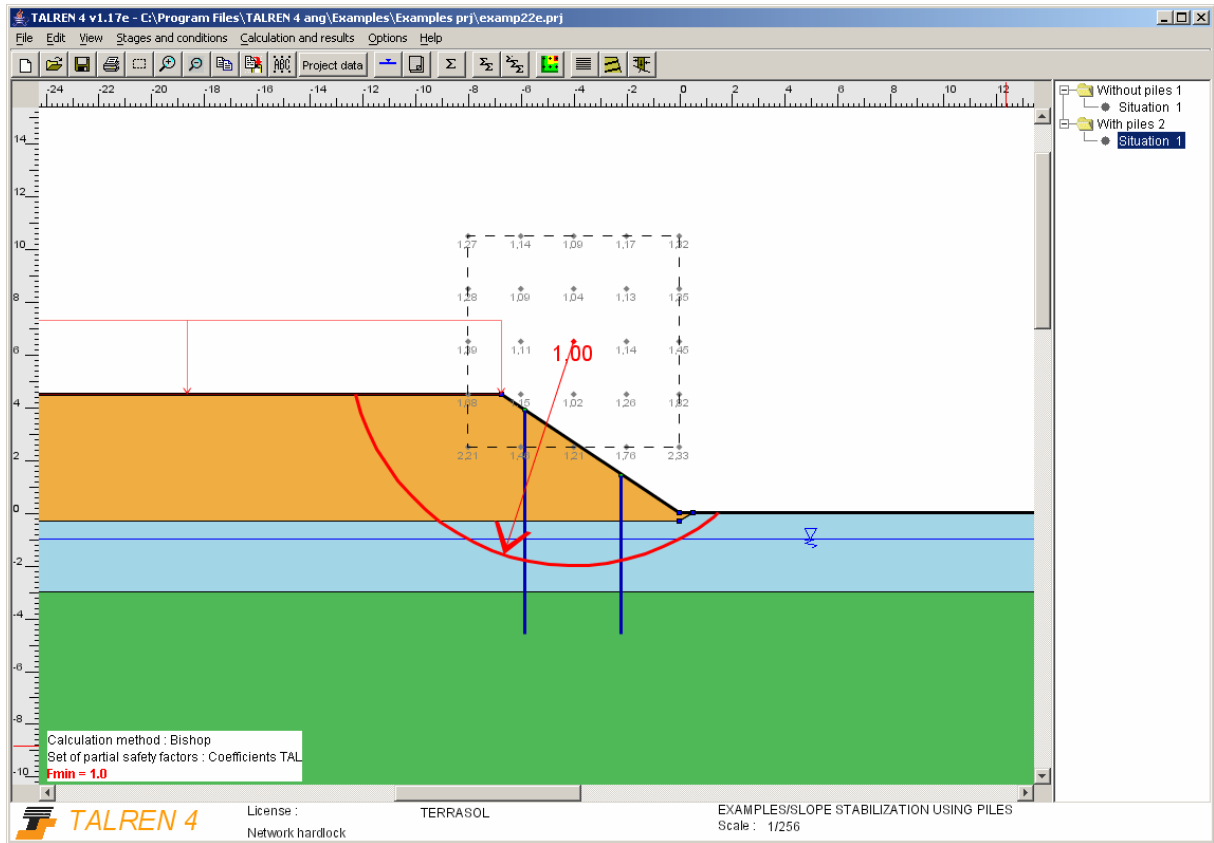

Sections : H – beam type B (h = 300 mm) 2 rows – T_e = 240 MPa – Horz. spacing = 2 m
(HEB 300)

B = 300 mm

EI = 52 850 kN.m^2

M_{plast} = 400 kN.m (maximum bending moment)

Shear for nail 1

	Y(m)	Tc(kN)
1	0.00	0.0
2	1.00	190.0
3	2.00	320.0
4	3.00	270.0
5	4.00	220.0
6	5.00	140.0
7	6.00	140.0
8	7.00	160.0
9	8.00	200.0
10	9.00	500.0
11	10.00	0.0

Shear for nail 2

	Y(m)	Tc(kN)
1	0.00	0.0
2	1.00	140.0
3	2.00	175.0
4	3.00	105.0
5	4.00	85.0
6	5.00	80.0
7	6.00	90.0
8	7.00	115.0
9	8.00	410.0
10	9.00	450.0
11	10.00	0.0
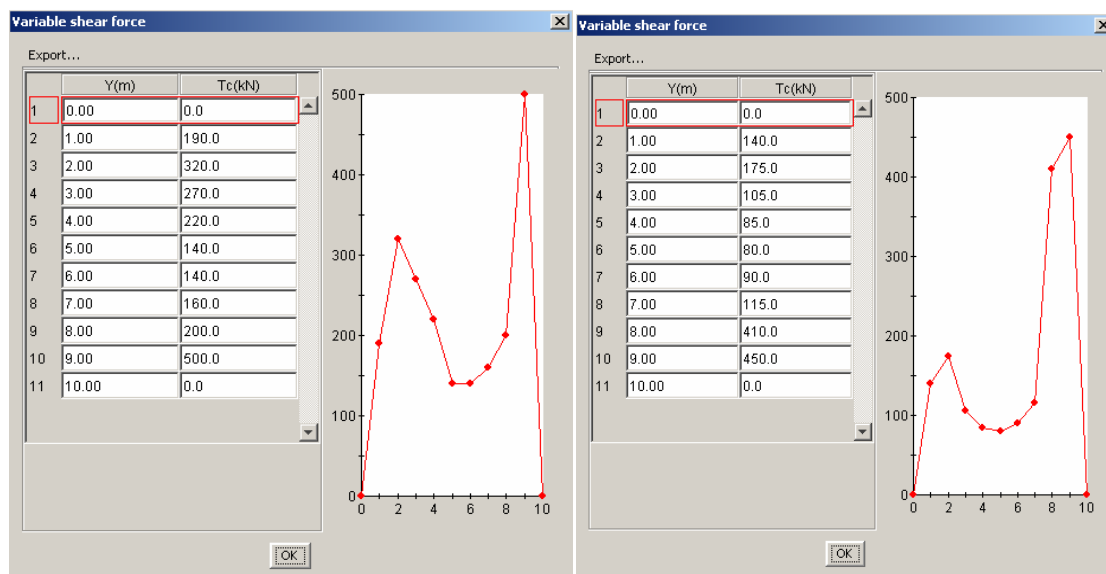
2.23. Example 23: ru coefficient

For this example, the pore pressures are calculated using the ru coefficients: $u = r_u.\gamma h$.
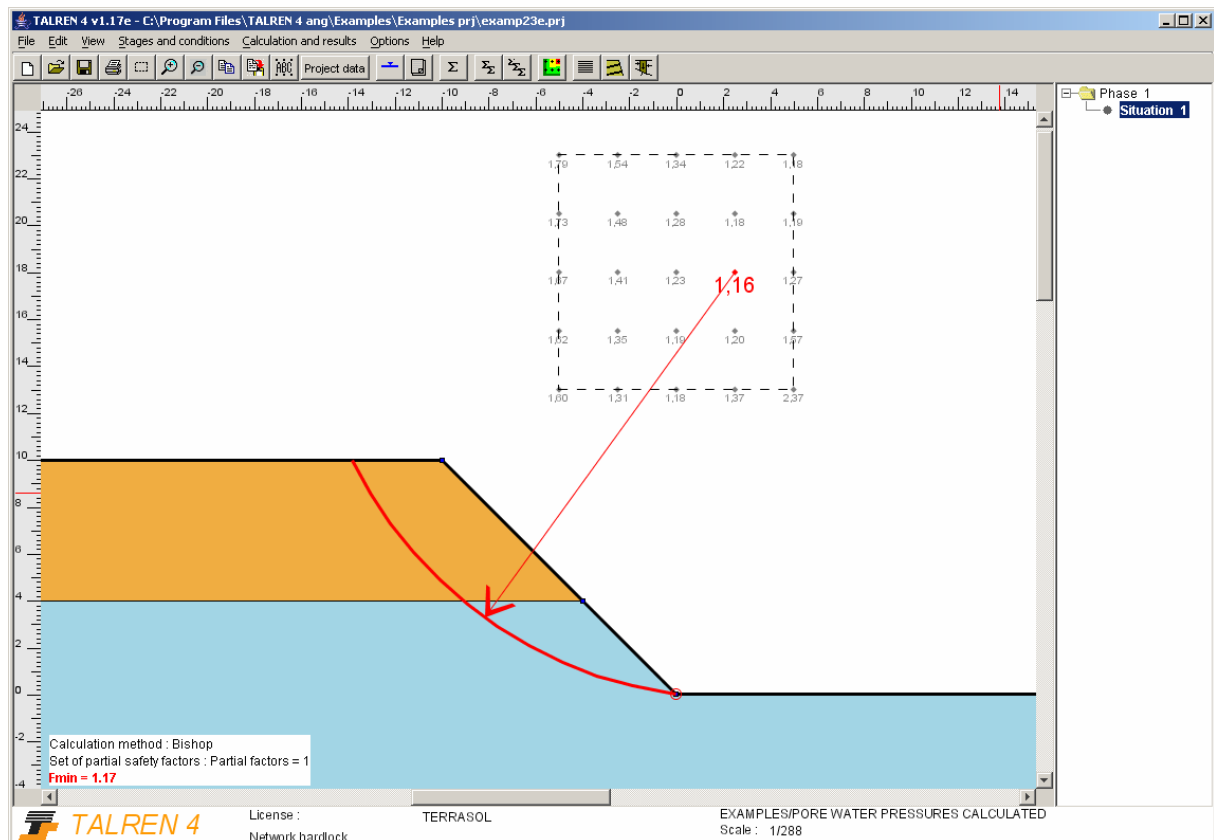
2.24. Exemple 24: construction of a highway retaining wall

This example illustrates a staged calculation: 6 temporary stages and 1 final stage.
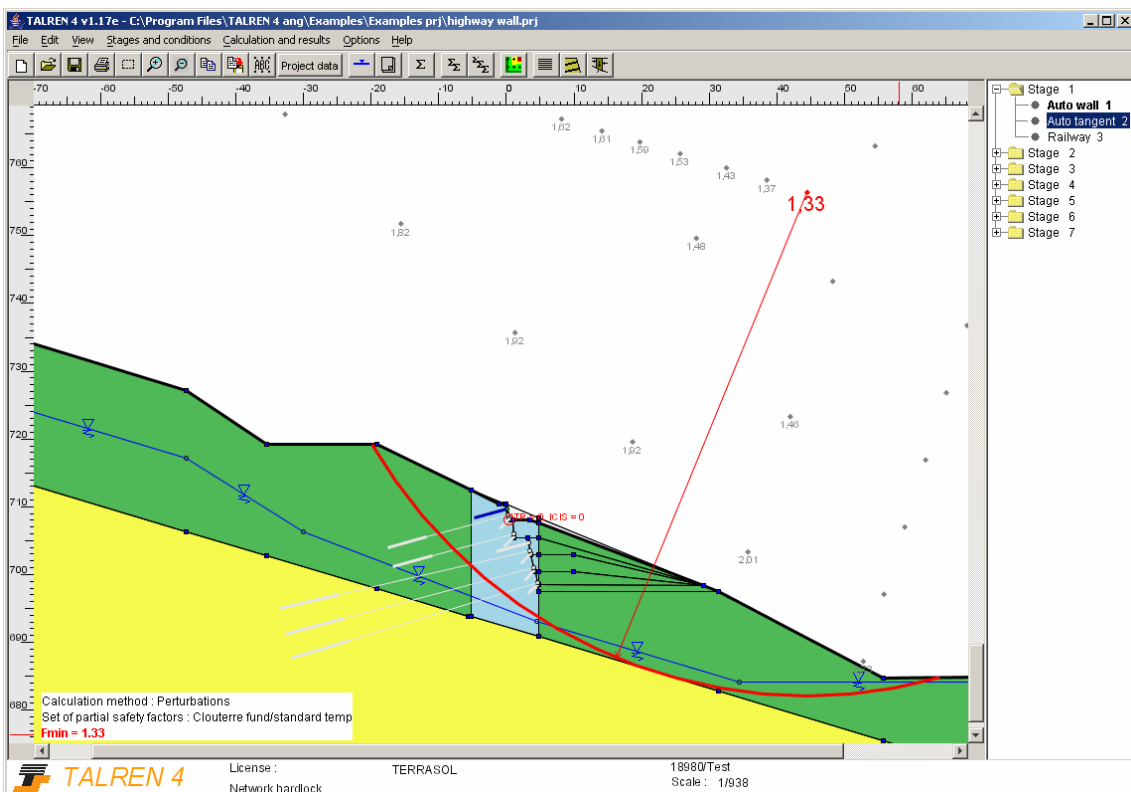Several types of failure surfaces have been analysed for each stage.

Stage 1

Stage 2

Stage 3

Stage 4

Stage 5

Stage 6

Stage 7: final stage
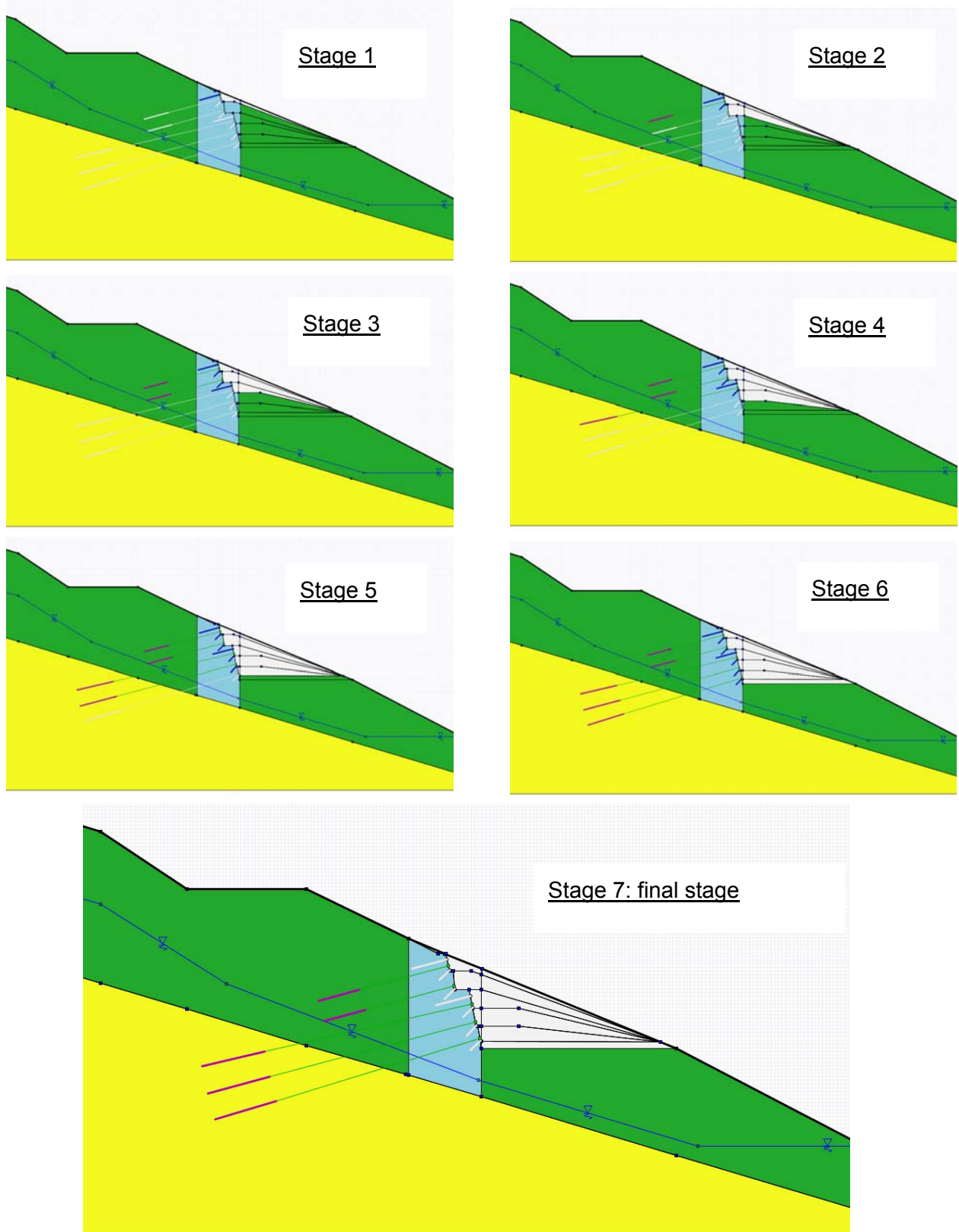
Here are some output examples:

Stage 1 / Situation 1 (automatic search, passage point at the bottom of the wall)
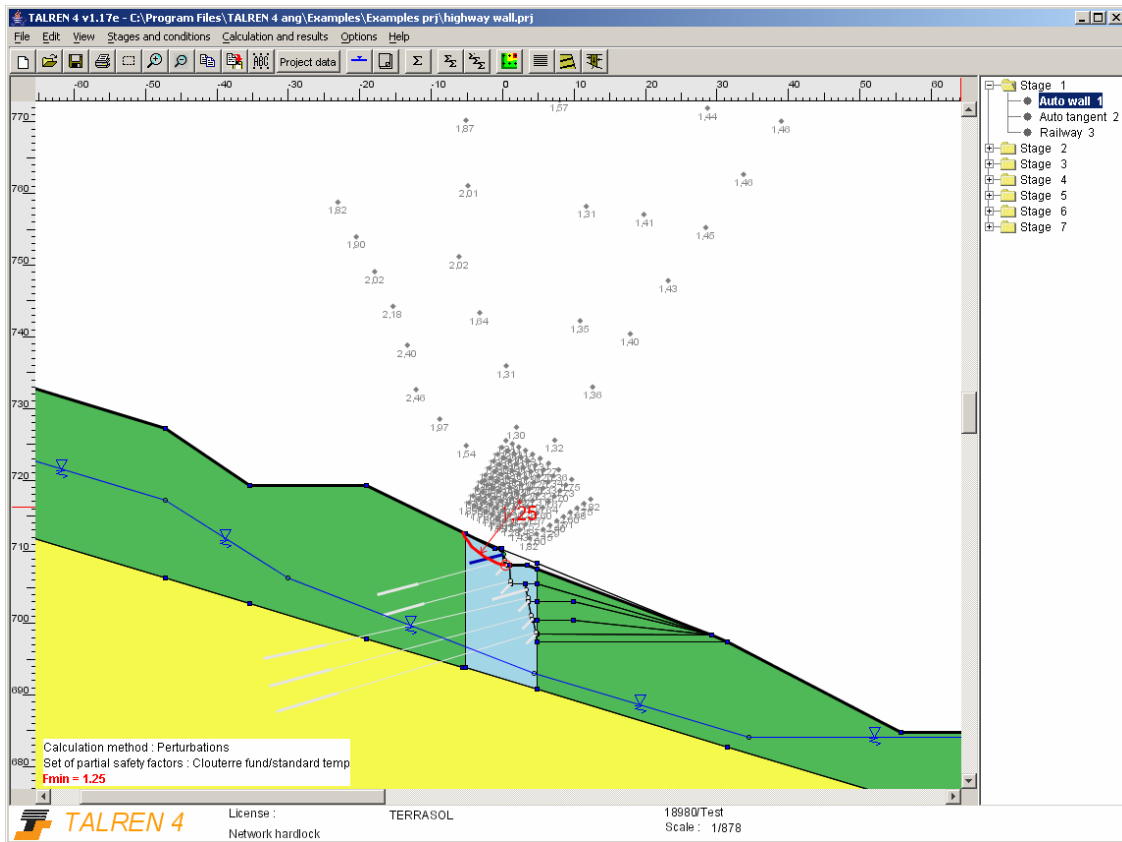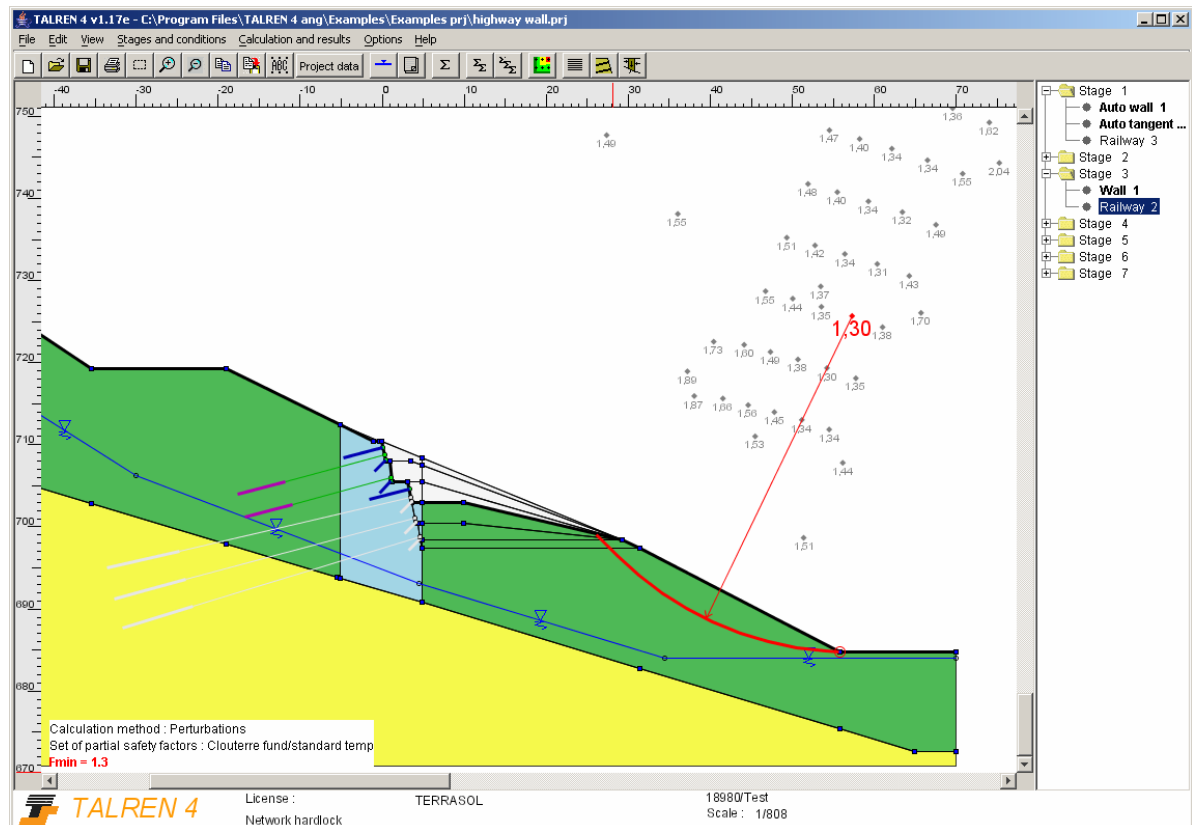

Stage 1 / Situation 2 (automatic search, first circle tangent to layer 3)
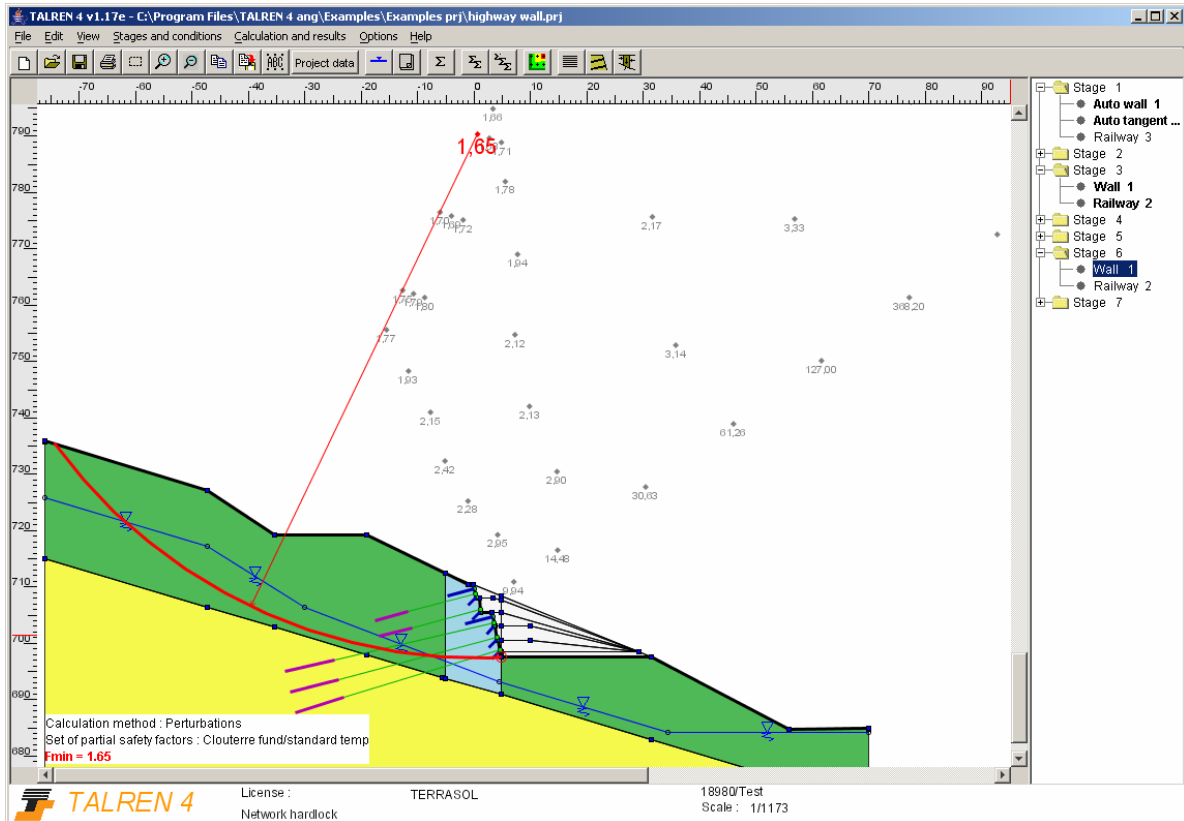
Stage 3 / Situation 1 (automatic search, passage point at the bottom of the wall)
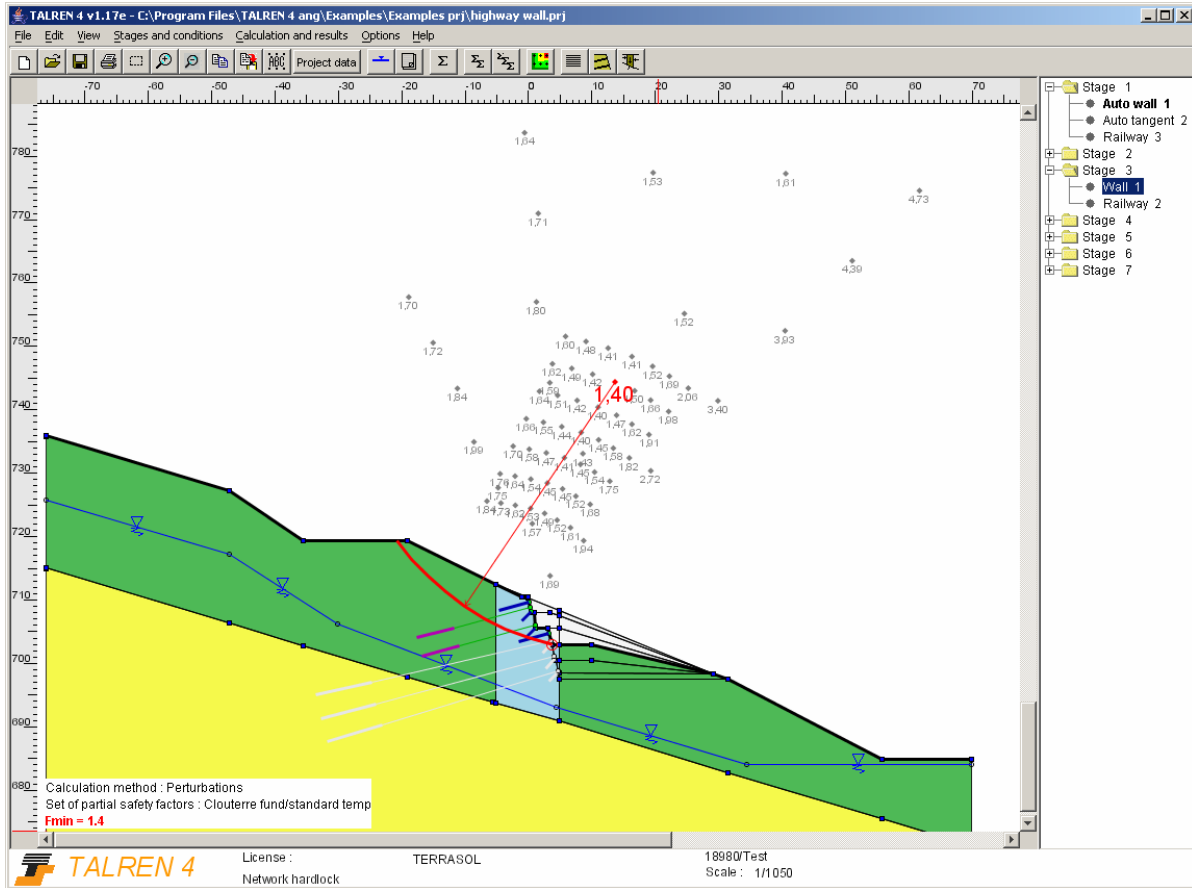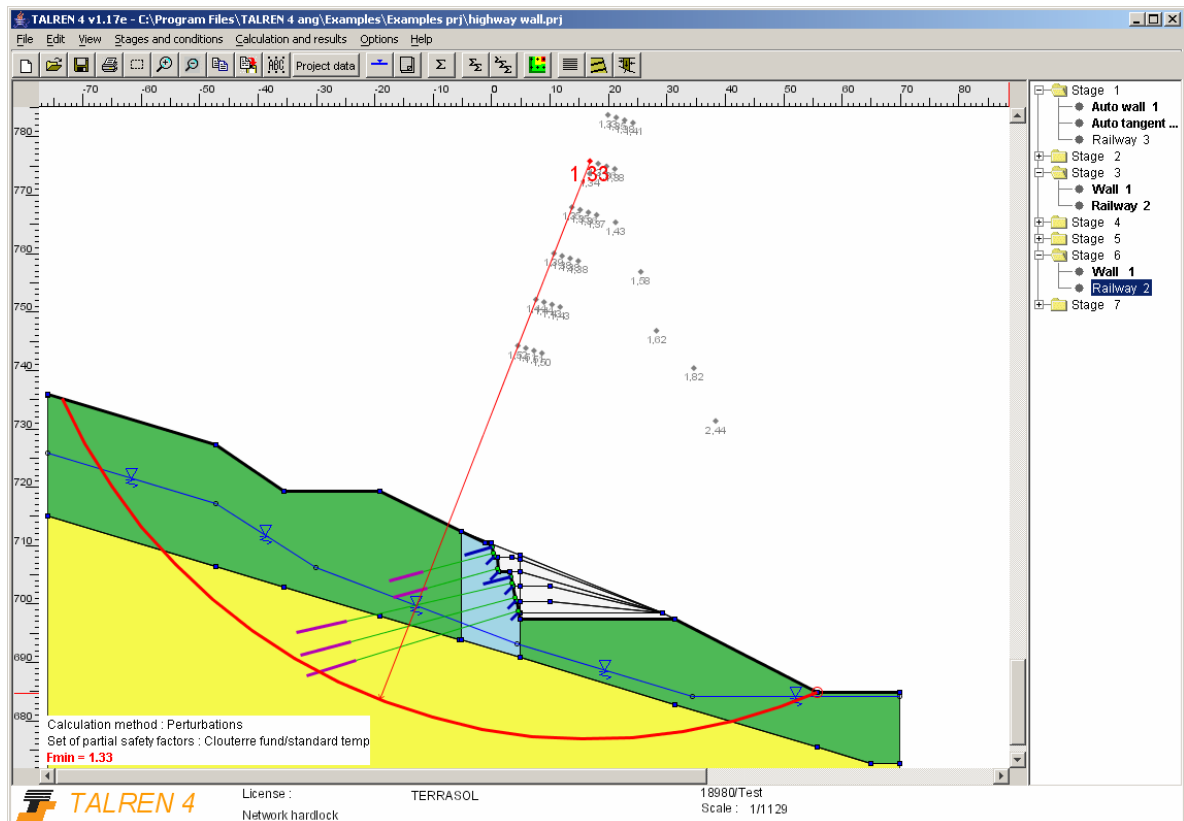

Stage 3 / Situation 2 (automatic search, passage point at the base of the SNCF slope)
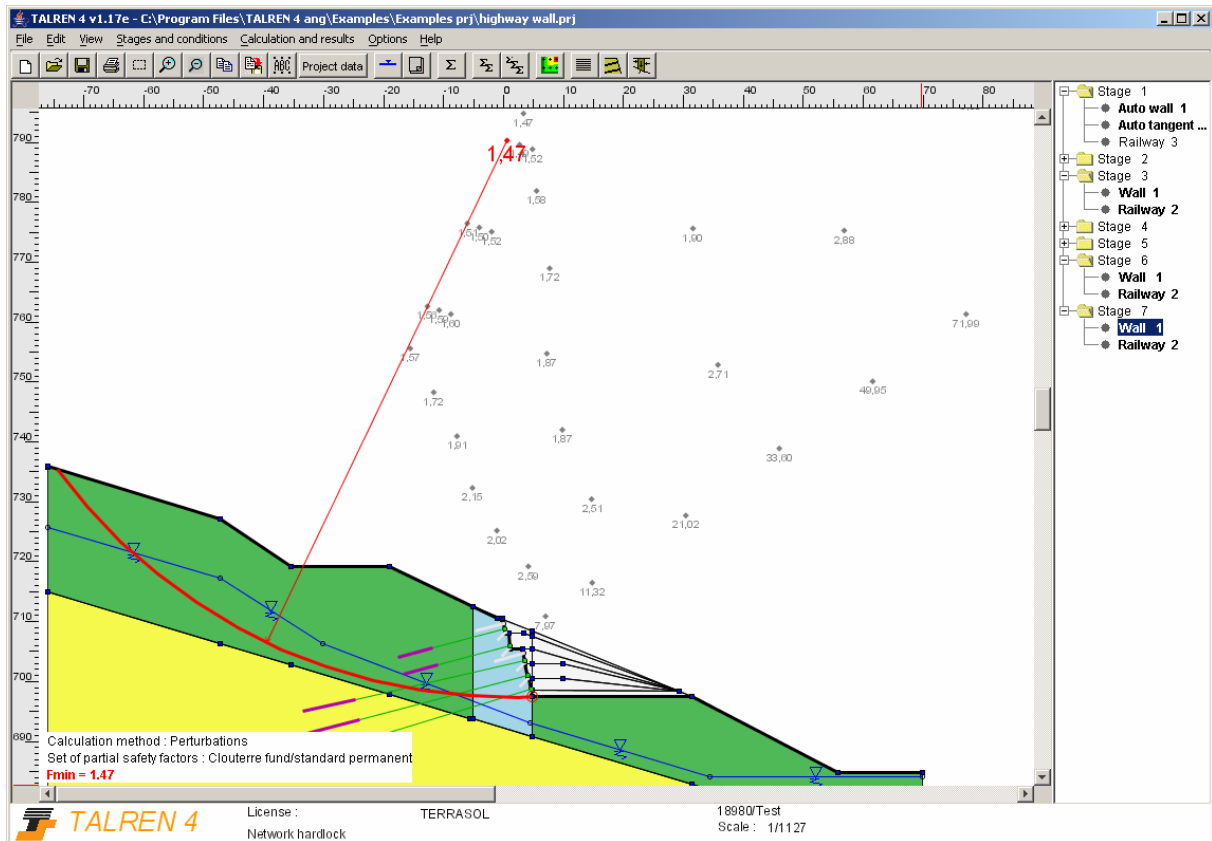
Stage 6 / Situation 1 (automatic search, passage point at the bottom of the wall)
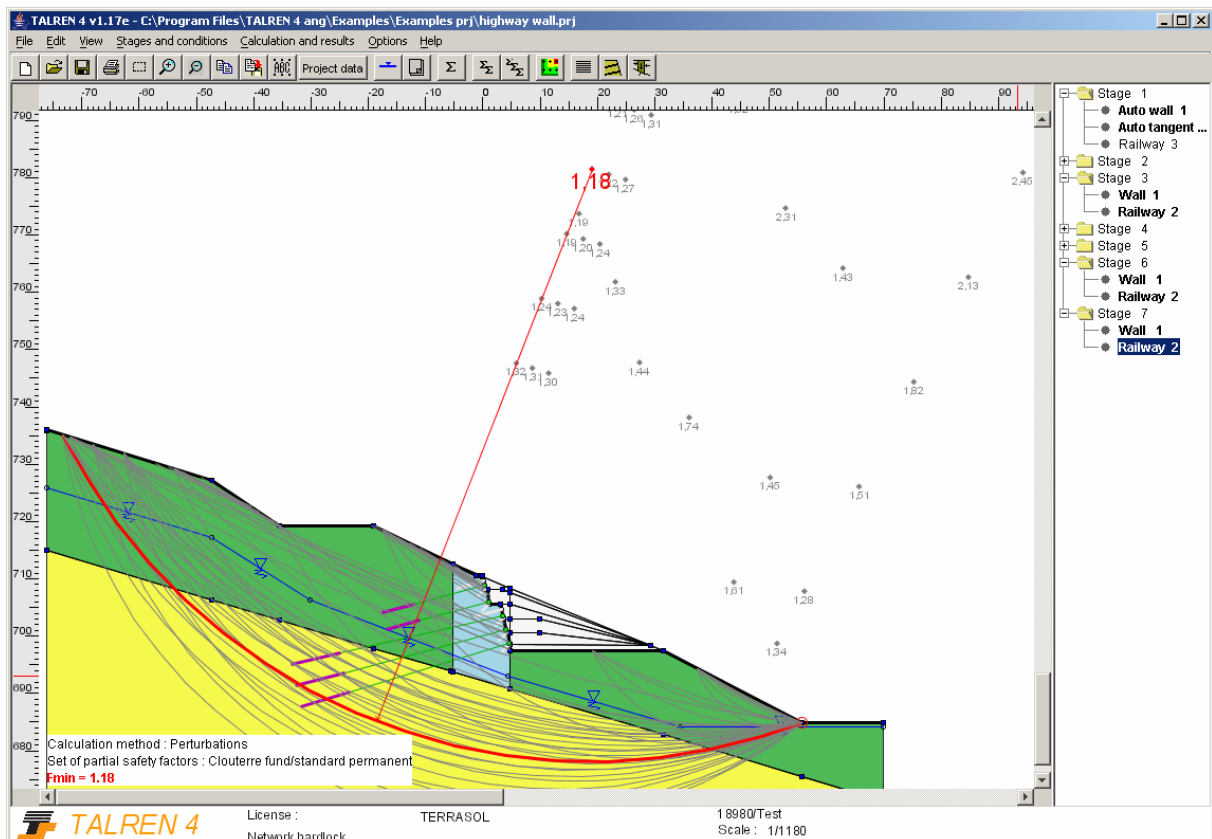

Stage 6 / Situation 2 (automatic search, passage point at the base of the SNCF slope)

Stage 7 (final) / Situation 1 (automatic search, passage point at the bottom of the wall)

Stage 7 (final) / Situation 2
(automatic search, passage point at the base of the SNCF slope)

Stage 7 (final) / Situation 2
(automatic search, passage point at the base of the SNCF slope, shadings display)