TALREN 4 – V 1.x

B. User Manual

LIST OF FIGURES

LIST OF TABLES

1. <u>INTRODUCTION</u>

TALREN 4 is a major evolution compared to previous versions of TALREN, and notably compared to TALREN 97.

TALREN 4 is a new user-friendly software running under Windows®, and TALREN 97 users as well as new users of TALREN will find it very easy to use.

The main characteristics of TALREN 4 are presented in chapter 2.

The detail of available functionalities is then presented in the following chapters.

We draw the attention of the reader particularly to the paragraphs designated by "***IMPORTANT***": they include important indications for a good use of the software.

2. GENERAL PREVIEW

2.1. Main features of TALREN 4

2.1.1. The interactive graphical interface

The TALREN 4 user interface is an interactive graphical interface, developed according to the following principles:

- Most operations, in particular the drawing of the profile, can be performed with the mouse. It is also possible to view and modify properties of the various objects (data and results) with the right mouse button.

- TALREN 4 proposes a graphical view whenever possible: curves for cohesion anisotropy, or stresses along the failure surface are 2 examples of graphical display.

- The non-necessary data are not displayed: for example, if no nail is defined, the value of qs_{nails} is not necessary: this datum is therefore "hidden" in this case.

- Each datum is defined only once, to guarantee a better liability and to save time when the user wants to change a value (only one change required).

- Many wizards (partial safety factors) and databases (soils, reinforcements) are available.

The different ways of using this graphical interface are detailed in the following chapters.

2.1.2. Handling the staged construction

TALREN 4 enables to analyse geotechnical projects, and not only individual cross-sections, which means that the management of construction stages is integrated in the program: from initial conditions, it is possible to define in the same file many successive construction stages (each stage corresponding to a cross-section of the project):

- Activation/deactivation of soil elements, loads, or reinforcements,

- Modification of soil properties,

- Modification of hydraulic conditions.

It is therefore not necessary to create several files to analyse the successive construction stages of a project.

Moreover, it is possible to define many calculation conditions (also called situations here) for a same stage (i.e. physical cross-section). The conditions properties include:

- Different calculation methods,

- Different partial safety factors,

- Different seismic conditions,

- Different failure surfaces.

It is therefore possible, for a given calculation stage, to check the stability of different types of failure surfaces (different passage points for circular failure surfaces for example), or to compare the fundamental situation with an accidental situation.

The "tree" organisation of stages and situations is illustrated with an example in Figure 1.
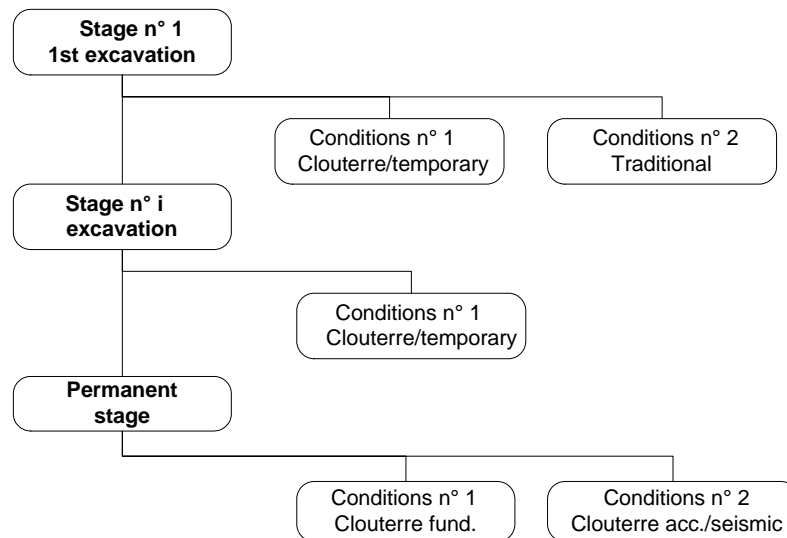
Figure 1: stages and situations

Each calculation is performed for a given situation of a given construction stage.

Each calculation is independent from the calculations for other stages and situations. For example, the result of the calculation for situation m of phase n does not depend neither on the results of phase n-1 nor on the results of situation m-1 nor on any other results.

One must perform as many calculations for a project file as situations defined for all stages of this project file.

2.1.3. The automatic search for circular surfaces

In the case of circular failure surfaces, an automatic search option is available as a complement to the manual search (definition of a mesh for centres of circles).

This automatic search is compatible with options "imposed passage point" and "first circle tangent to a soil layer", and enables an automatic scan of all the possible space for the centres of the circles. The following figures illustrate the principle of the automatic scan. The settings for the automatic search are detailed in section 5.2.5.4, and the scan method is detailed in section C.3.2.1.2 of the technical manual.

Number of directions = number of intervals

Number of distances = number of intervals (further distances are not represented here)

L

L

L/2

L/2

Left model boundary

Imposed passage point

Number of radii = number of intervalss

Right model boundary

L
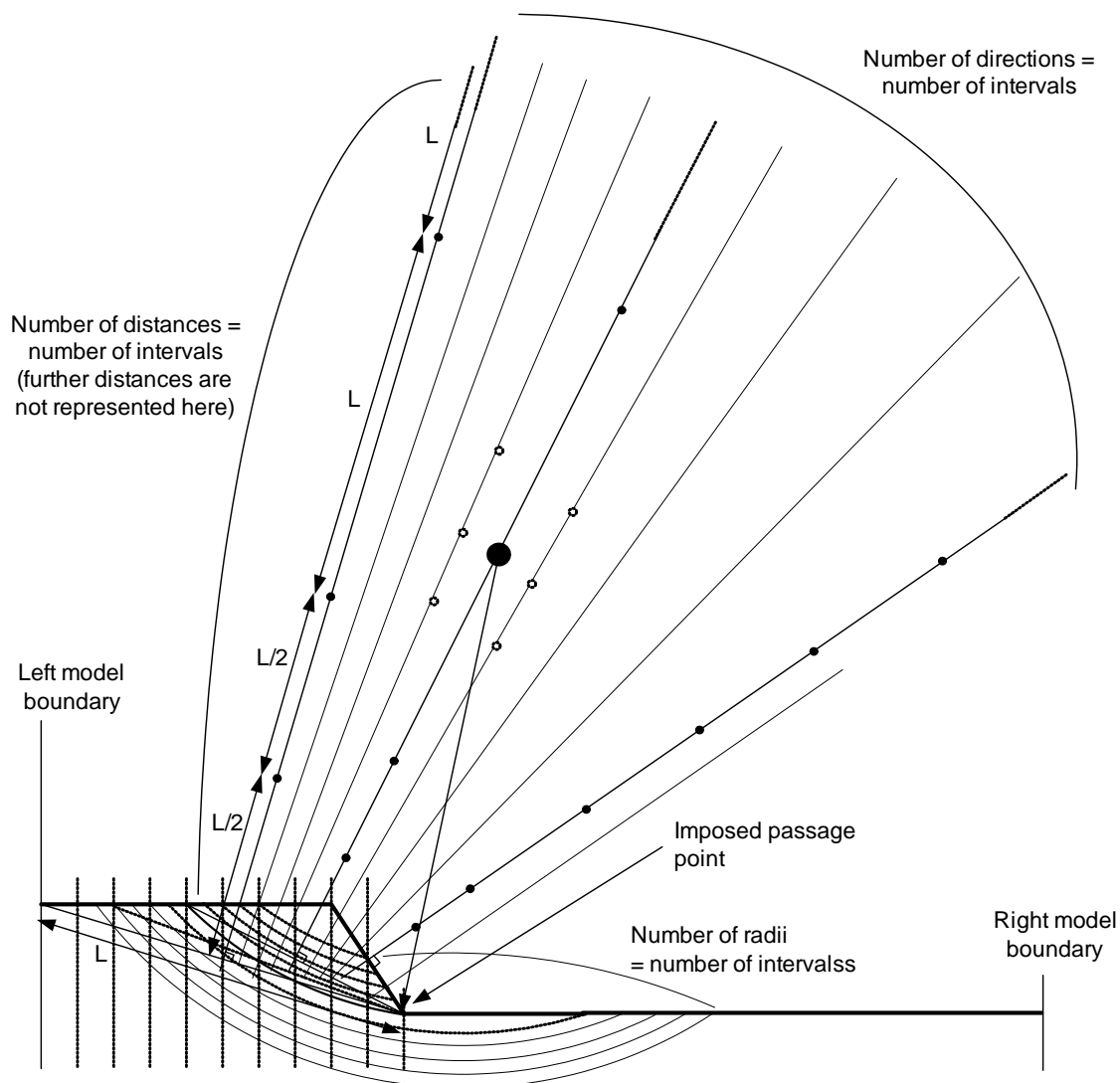
Figure 2: scan principle for the automatic search of the critical circle,
with imposed passage point for the first circle:
First scan
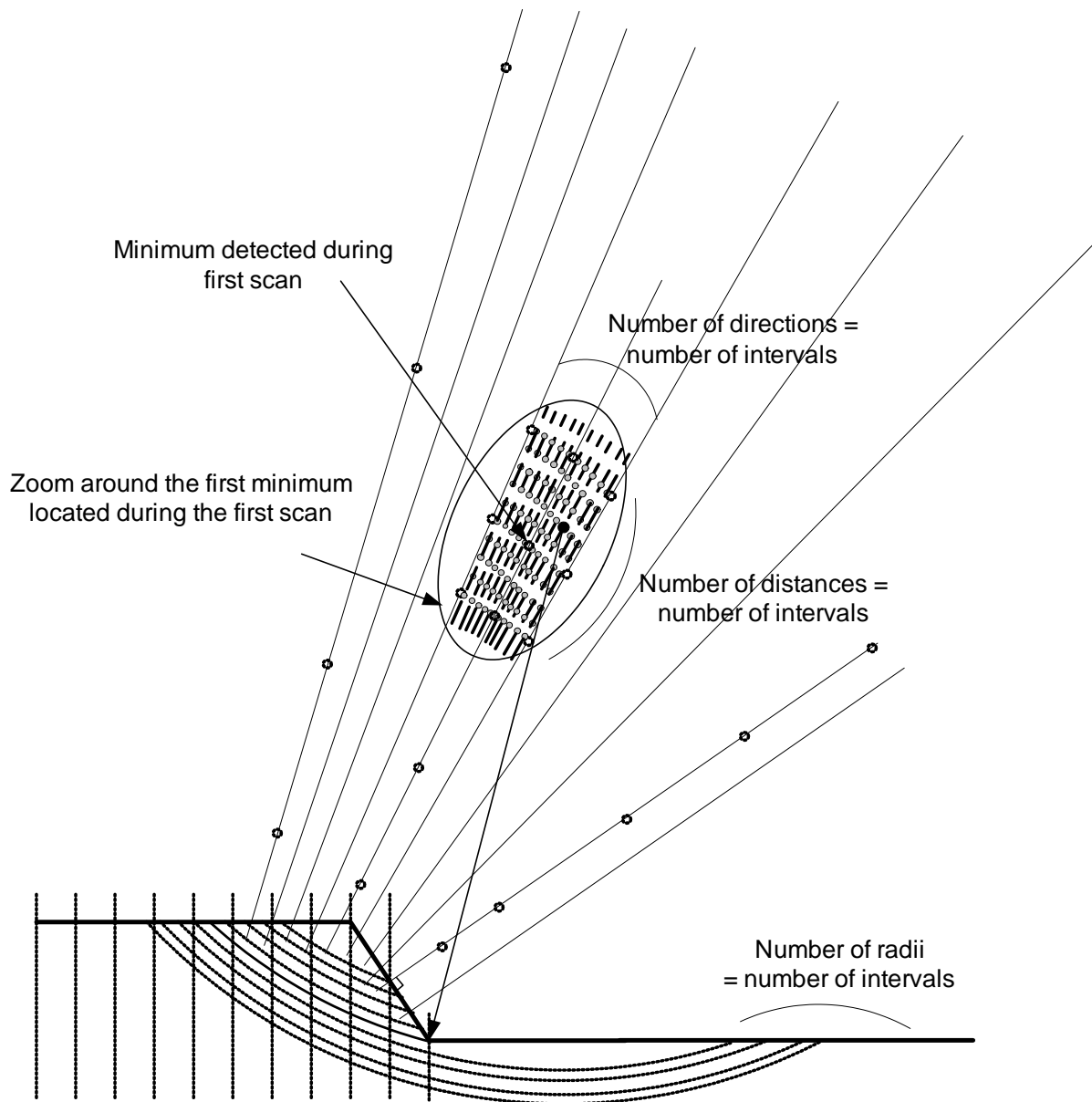
*Figure 3: scan principle for the automatic search of the critical circle,
with imposed passage point for the first circle:
Second scan ("zoom")*
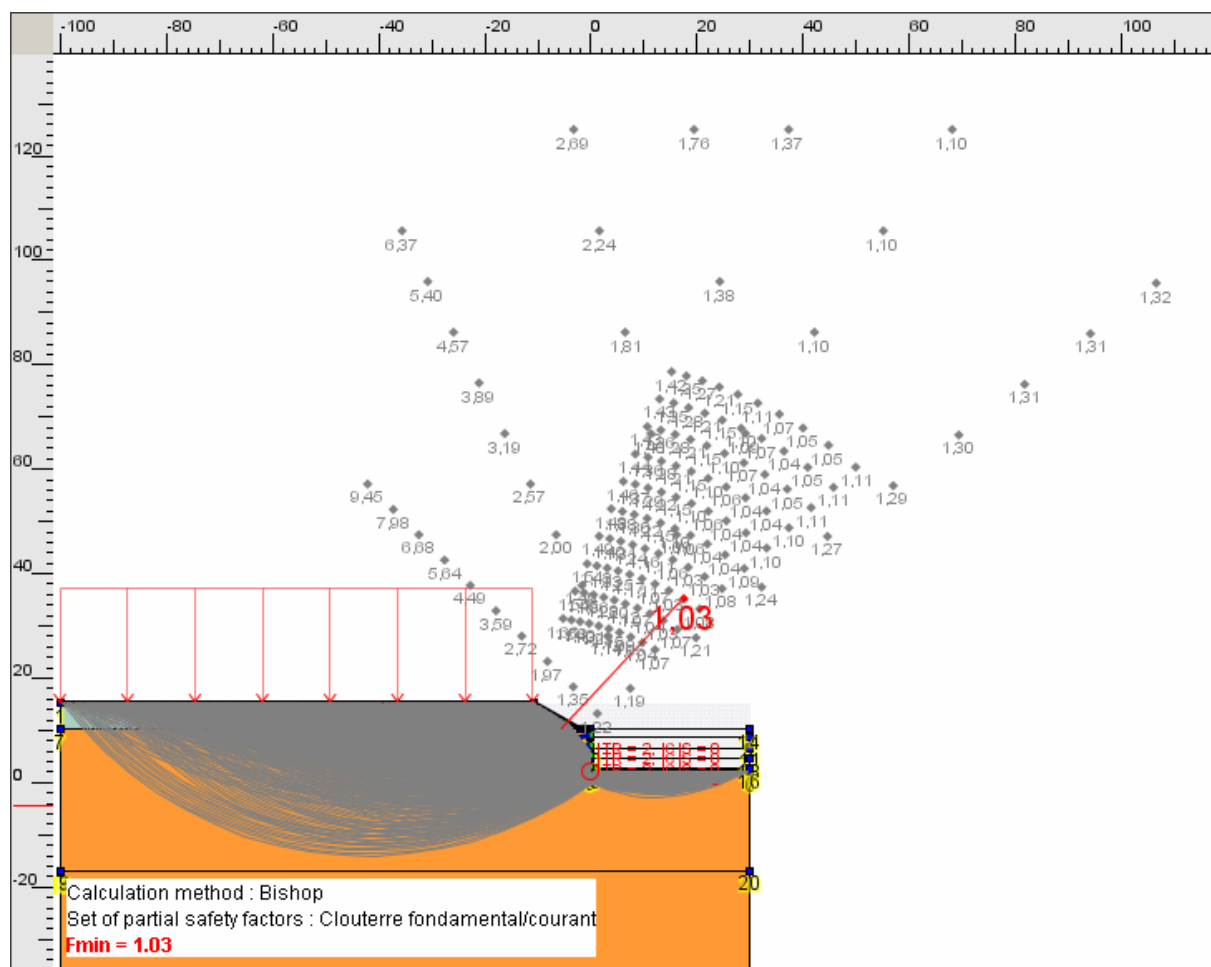
Figure 4: example of an automatic search with imposed passage point
(display of all calculated circles)

Moreover, in the case of a calculation with yield design method associated to logarithmic spirals, the user defines entry and exit intervals for the spirals, and a few additional parameters (indicated in section 5.2.5.4). TALREN 4 performs an automatic search of all the spirals corresponding to these entry/exit intervals, as illustrated in Figure 5 and Figure 6.
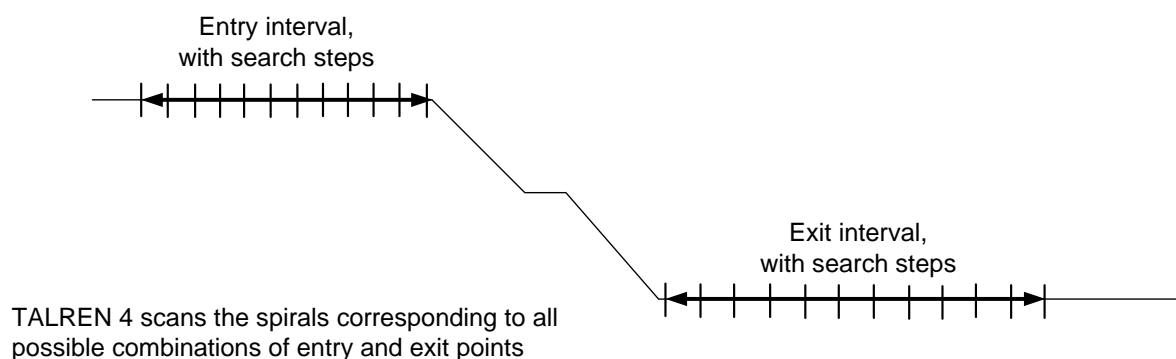
Figure 5: principle of the automatic search
for yield design calculation (logarithmic spirals)
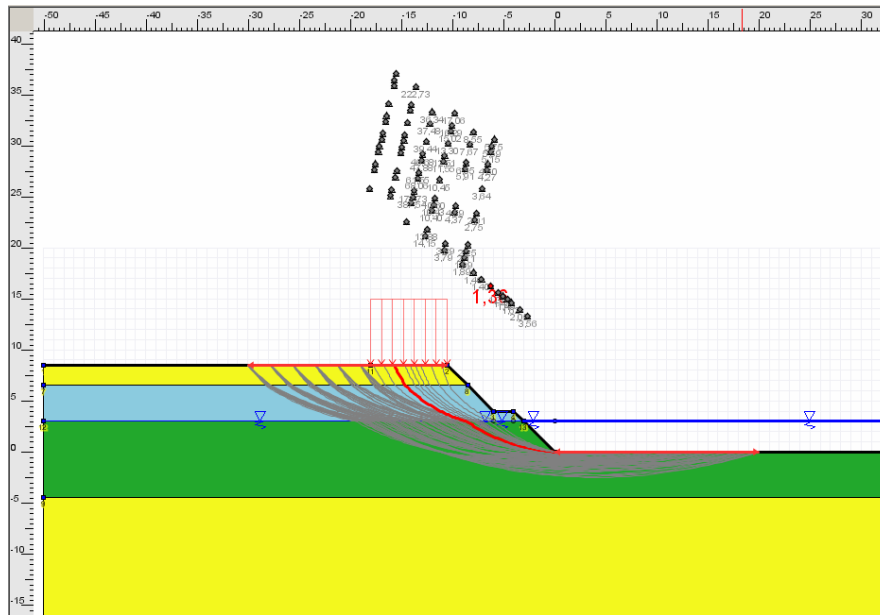
*Figure 6: example of an automatic search for logarithmic spirals
(display of all calculated spirals)*

2.1.4. Yield design calculation (logarithmic spirals)

TALREN 4 proposes, in addition to the 3 calculation methods with limit equilibrium (Fellenius, Bishop, perturbations), the yield design calculation method (with logarithmic spiral failure surfaces, Figure 6). For more information concerning this new calculation method, please refer to the technical manual (chapter C of this manual).

2.1.5. Compatibility with previous versions of TALREN, and with PLAXIS

The ascendant compatibility of Talren 4 is ensured with Talren 97, except for a few exceptions (section 7.1.1).

Moreover, Talren 4 can load Plaxis v8 files.

For more details, please refer to section 7.1.

2.2.   General steps when creating a new project with Talren 4

TALREN 4 is composed of two modes: the "Data" mode on the one hand, and the "Stages/calculations" mode on the other hand.

The classical steps to follow when creating a new project are the following:

- Run Talren 4: the default active mode is the "Data" mode;
- Create a new project file (or open an existing project file);
- Define or modify the general settings (not compulsory if opening an existing file);
- Shift to the "Stages/Calculations" mode with the button of the contextual toolbar;
- The first stage is generated automatically: define or modify its properties if necessary;
- The first situation is generated automatically: define or modify its properties if necessary;
- If necessary, create additional conditions (situations) for the first stage, with the pop up menu or with the "Stages and conditions" menu;

- If necessary, create additional stages, with the pop up menu or with the "Stages and conditions" menu;

- Perform the calculations, either for the selected situation, or for all situations of the selected stage, or for all situations of all stages: the calculation can be run either with one of the 3 calculation options of the "Calculations and results" menu, or with one of the 3 calculation buttons of the contextual toolbar.

- View the different output types (either on the graphical display, or in the detailed tables).

Please refer to chapter D to learn more about the practical use of Talren 4 (detailed application examples).

3. THE VARIOUS OPTIONS TO USE THE INTERFACE

3.1. The different zones on the screen

The different zones of the TALREN 4 interface are the following:

- The main menu (completed by the pop up menu);
- The general toolbar and the contextual toolbar;
- The explorer for stages and conditions (visible only in Stages/Calculations mode);
- The graphical view;
- The status bar;
- The dialogue boxes.

These different objects are described in the following chapters.
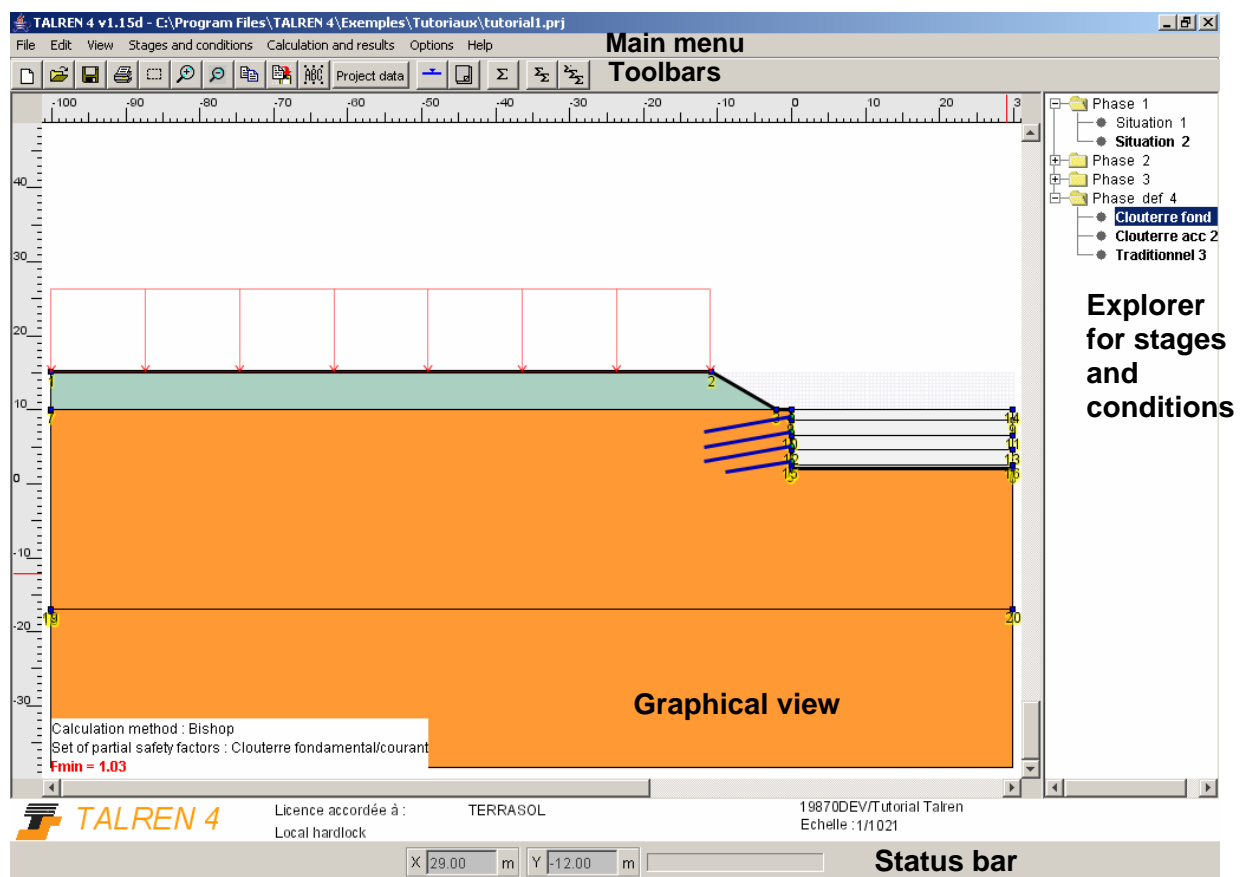
Figure 7: identification of the zones on the screen (in Stages/Calculations mode)

3.2. Menus, toolbars and stages explorer

3.2.1. Main menu

The main menu (schematic view in Figure 8), includes the following main headings:

- File
- Edit
- View
- Project data
- Stages and conditions
- Calculation and results
- Options
- Help

The "Data" menu on the one hand, the "Stages and conditions" and "Calculation and results" menus on the other, are not displayed simultaneously. Only the menus corresponding to the active mode are displayed.

Anytime, the menu items that are not relevant are disabled. For example, the "Table of reinforcements" option of the "View" menu is disabled if no reinforcement has been defined.

The menu items can all be reached with a keyboard shortcut like "Alt + F + N" for example to access the "File" menu, "New" item. The letters to be used for the shortcuts appear underlined in the menus (in accordance with Windows standards), as soon as you push the "Alt" button of the keyboard.

The menu options are not detailed here: they are mentioned along with the description of the functionalities in the following chapters.

TERRASOL

| File | Edit | View | Project data | Stages and conditions | Calculation and results | Options | Help |

File:
New
Open
Save
Save as
Close

Print

Working directory

Last file opened 1
Last file opened 2
Last file opened 3
Last file opened 4

Exit

Edit:
Cut
Copy
Paste
Delete

Project data:
General settings
Geometry
Loads
Reinforcements
Soil characteristics

Stages/calculation

Distributed loads
Linear loads and moments

Nails
Anchors
Strips
Struts

Calculation and results:
Calculate the selected set of cond.
Calculate all conditions of the selected stage
Calculate all

Graphical display settings for output

Detailed output
Forces in reinforcements
Slices results

Help:
Help topics
About

View:
Rulers
Grid

Soil layers painting
Point numbers
Segment numbers
Loads identification
Reinforcements identification

Background drawing
General comments

Zoom

View all stages

Table of soil characteristics
Table of loads
Table of reinforcements

Colors
Patterns
None

Loading picture
Adjusting to scale
Deleting picture

Zoom in
Zoom out

Define scale
Define a zoom with mouse
View whole model

Options:
Partial safety factors
Reinforcements database
Soils database

Reverse model

Stages and conditions:
Hydraulic conditions for selected stage
Comments for selected stage

Insert a new stage
Add a new stage
Remove selected stage

Define selected set of conditions
Comments about selected set of conditions

Add a new set of conditions
Remove the selected set of conditions
Duplicate the seletec set of conditions

Project data

Figure 8: main menu

Figure 9: project Data menu

3.2.2. Pop up menu

It can be reached by a right click of the mouse.

It enables to reach at any time:

- The most common display options (some zoom options: for example, "View whole model" is very useful);

- The operations available for the object selected before right-clicking: for example properties, deletion, edition, activation, deactivation. The available options depend on the active mode (Project Data or Stages/Calculation);

- All other relevant options depending on the position of the mouse and on the selected element. For example on Figure 10, it is possible to edit graphical display settings, since the Stages/Calculations mode is active, and the calculation has already been performed.
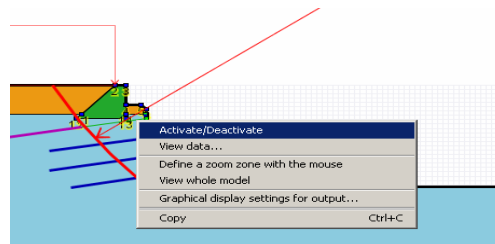
Figure 10: pop up menu (stages mode, after a right click on a soil layer)

3.2.3. Main toolbar

Figure 11: main toolbar

The buttons hereabove correspond respectively to the following functions:

- New file;

- Open an existing file;

- Save a file;

- Print;

- Zoom in a user-defined window;

- Zoom in;

- Zoom out;

- Copy to clipboard the selected element;

- Screenshot;

• General comments;

The buttons all have a legend, which is displayed when the mouse is placed on each of them.

3.2.4. Contextual toolbars

The "project data" toolbar on the one hand, and the "Stages/Calculations" toolbar on the other hand, are not displayed simultaneously. The one displayed is the one corresponding to the active mode.

The buttons all have a legend, which is displayed when the mouse is placed on each of them.

3.2.4.1. "Project data" contextual toolbar

This one is equivalent to the "Project data" menu:

Figure 12: "Project data" toolbar

The buttons hereabove correspond respectively to the following functions:

• Selection tool;
• General settings for the project;
• Drawing of geometrical lines;
• Drawing of loads (distributed loads and linear loads/moments);
• Drawing of reinforcements (nails, anchors, strips and struts);
• Definition of soil properties (dialogue box);
• Shifting to "Stages/calculations" mode.

3.2.4.2. "Stages/calculations" contextual toolbar

This one corresponds to the items of the "Stages and conditions" and "Calculations and results" menus:

Figure 13: "Stages/calculations" toolbar

The buttons hereabove correspond respectively to the following functions:

• Shifting to "Project data" mode;
• Definition of the hydraulic conditions (dialogue box);
• Properties of the selected situation (dialogue box);
• Calculation of the selected situation;
• Calculation of all situations of the selected stage;
• Calculation of all situations of all stages;
• Settings for the graphical output display (dialogue box, available if results are available for the selected situation).
• Detailed output for failure surfaces (dialogue box, available if results are available for the selected situation);
• Detailed ouput for reinforcements (dialogue box, available if results are available for the selected situation);
• Detailed output for slices (dialogue box, available if results are available for the selected situation).

3.2.5. Explorer for stages and conditions

In the "Stages/calculations" mode, an explorer is displayed in the right part of the screen (Figure 14). It enables to change from a stage or situation to another, and also to insert, add, delete or rename stages and situations (either with the "Stages and conditions" menu, or with the pop up menu that appears when right-clicking on a stage or situation).
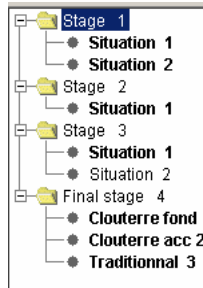
Figure 14: explorer for stages and conditions

3.3.   *Graphical view and rulers*

The graphical view includes different objects:

- A horizontal ruler, placed at the top of the graphical window. Its settings (scale, origin, graduation step, etc) are determined automatically as a function of the project width, or of the current zoom level.

- The same type of ruler, with the same characteristics, but vertical on the left of the graphical window;
- Vertical and horizontal scrollbars depending on the zoom level;
- The graphical view itself, for the drawing of the cross-section. At the bottom of this part, a white bar is displayed with the name of the company owning the program (information included in the hardlock), the user mode (local, network or demo), the title of the project and the drawing scale.
- A pop up menu, accessible by a right mouse click (section 3.2.2).

3.3.1. Conventions about the graphical drawing

3.3.1.1.   Scale

It is automatically defined as a function of the zoom level, or it can be decided by the user (section 3.3.2.7).

In all cases, the scale in x and y directions is the same.

3.3.1.2.   Directions

IMPORTANT: the cross-section must always be defined with the "uphill on the left" (example on Figure 15b), because the calculation is correct only if "the failure direction goes from the left to the right".

However, you may not want to reverse all coordinates manually if the cross-section you want to work from is defined in the other direction (uphill on the right).

Moreover, in the case of dams, you wish to define the whole cross-section (both sides of the dam), and then analyse each side of the dam.

Therefore the "Reverse model" item is available in the "Options" menu.
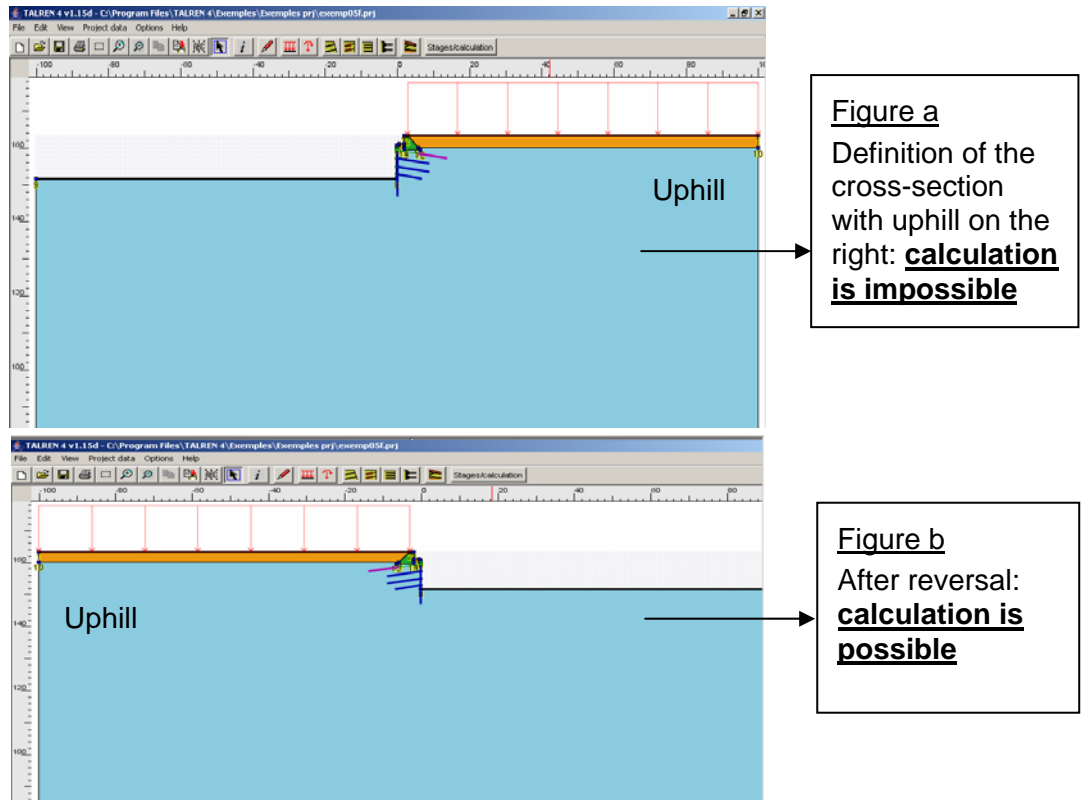
Figure 15: example of a model reversal

This item opens a dialogue box, which proposes:

- either to reverse the model in the current file. In this case, all the x values and the angles (load inclinations, reinforcements angles, etc) will have their sign changed (from positive values to negative values and vice versa). The project file will keep the same name. This case is relevant when defining a cross-section based on a document with the uphill on the right: you have to reverse the model to be able to perform the calculation "in the right direction" (case of Figure 15).

- or to reverse the model in a new project file. In this case too, all the x values and the angles will have their sign changed (from positive values to negative values and vice versa). The user will be asked for a new filename (the original file will not be altered). This case is relevant when defining a dam with both sides: 2 projects file should be created to analyse the stability of each side of the dam.

In all cases, when running the calculations, the sliding must happen "towards the right".

3.3.2. Display settings for the graphical view

These general options can be reached with the "View" menu (either in the "Project Data" mode or in the "Stages/calculations" mode).

3.3.2.1.   Rulers

This option activates or deactivates both rulers (horizontal and vertical).

The rulers settings (scale and position) are calculated automatically depending on the current zoom level.

3.3.2.2.   Grid

This option opens the dialogue box of Figure 16.
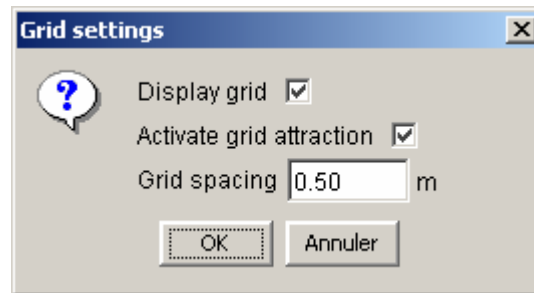
Figure 16: grid settings

- If "display the grid" is checked, the grid will be displayed on the drawing, with the indicated spacing;

- If "activate grid attraction" is checked, the "magnetism" is activated, and the mouse will move with the indicated spacing. When drawing an element, this element (geometry point, load, reinforcements, etc) will be automatically connected to the closest grid point. If this option is not activated, however, the elements will be drawn at the exact coordinates of each mouse click.

3.3.2.3. Soil layers painting

3 options are available:

- Painting with colours (default choice);
- Painting with patterns;
- No painting.

The last option is useful for printings that will be copied or faxed.

In the case of colours or patterns: the choice of the colour or of the pattern associated to each layer is proposed in the dialogue box for soil properties (section 4.6).

- With the "Colours" option: the user has to choose a colour (with a Windows® standard colour choice).

- With the "Patterns" option: the user has to choose a pattern, from a predefined list.

If the user changes the painting option in an existing project, he should open the soil properties dialogue box to define the new patterns or colours (the old ones are not erased, because the user might come back later to the previous choice).

3.3.2.4. Numbers of points and segments, labels of loads and reinforcements

Each one of these 4 options is available in the "View" menu, and can be activated/deactivated independently from the others.

3.3.2.5. Background picture

This option is usually used before starting to define the project geometry, but it is also possible to import a background picture for an existing file.

- Loading a picture: this option opens a standard dialogue box to select the picture file. The supported files are the gif and jpeg formats.
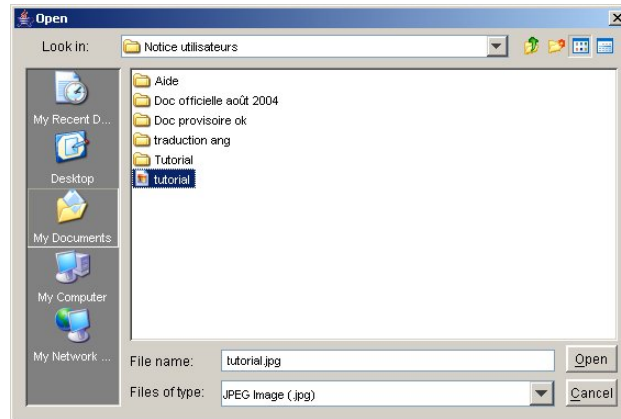
Figure 17: selecting a background picture

After validation, the background picture is displayed in the graphical view (Figure 18).
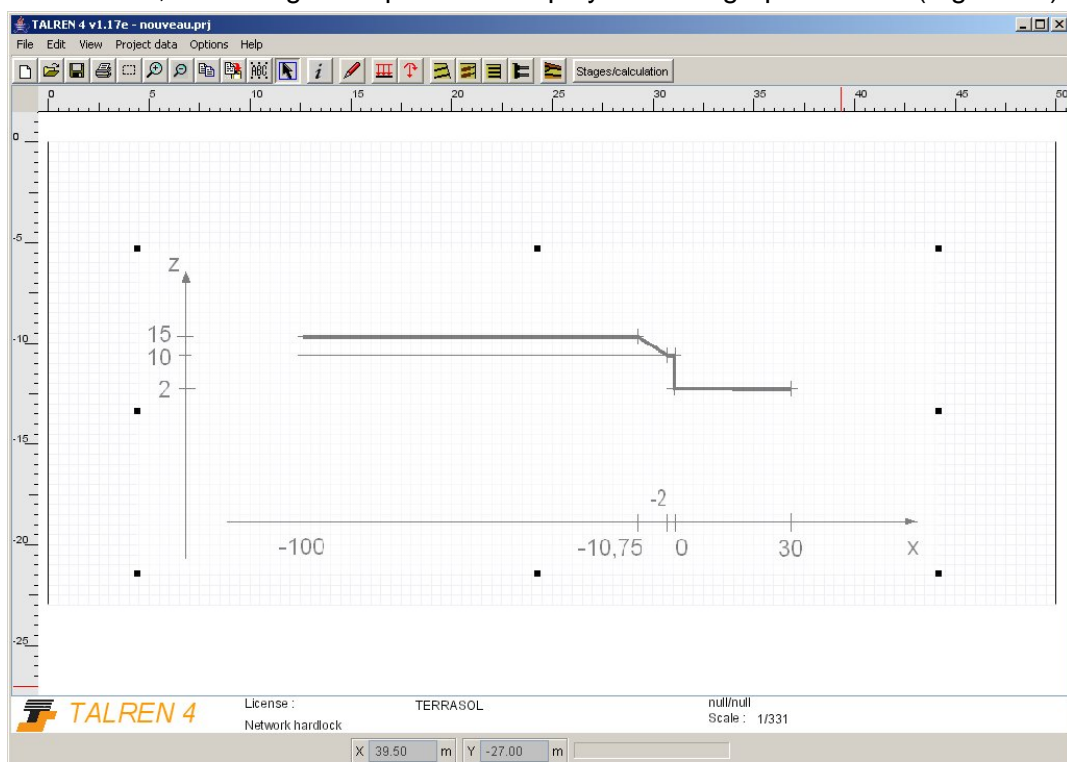
Figure 18: example of a background picture imported into a new project

The user can then move the background (drag & drop with the mouse): left click on the background (where no other element is displayed), then move the mouse keeping the left button pressed. The background can be moved partially "outside" the graphical view.

In the same way, the user can change the size of the background picture: when clicking on the background with the left mouse button (where no other element is displayed), "handles" appear on the picture contour. The user can move these handles to change the picture size.

- Adjusting the picture to scale: this option enables to adjust the scale of the background picture to the one of the graphical view (it is available only if a background picture is active). After selecting this option, you are requested to draw a reference segment on the picture (horizontal, vertical or any direction). A dialogue box then opens (Figure 19) and requests the "real" length of this segment. The size of the picture is then adjusted, and it can be placed wherever wanted.
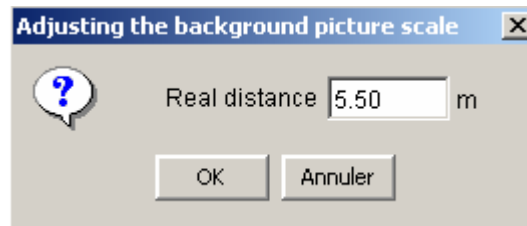
Figure 19: adjusting background picture to scale

- • Removing the background picture: this option enables to remove the background picture when it is not needed anymore (it is therefore available only if a background picture is active).

3.3.2.6. General comments

It is possible to define general comments (texts) that will appear on the graphical view, whatever mode is active.

This option, available in the "View" menu but also with the ABC button of the general toolbar, opens a dialogue box (Figure 20) allowing to define or modify any number of general comments.
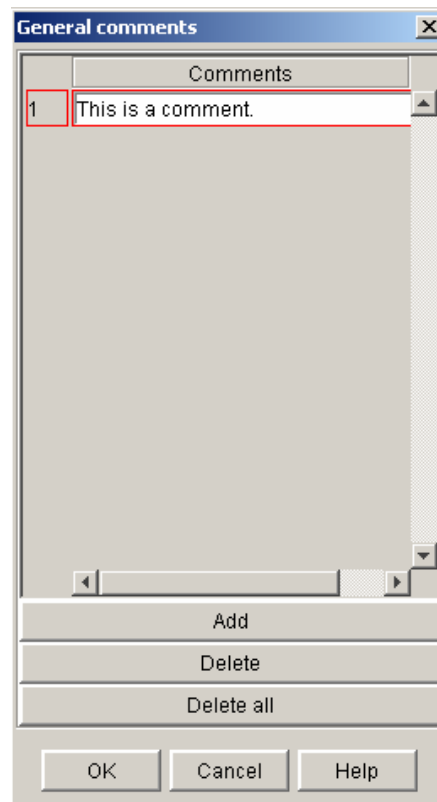
Figure 20: general comments dialogue box

To add a new comment, click on the "Add" button, then type in the text of the comment in the new line created in the table. To modify an existing comment, click on the corresponding line and change the text.

The "Delete" and "Delete all" buttons enable to delete one or all comments.

After validation of the dialogue box, the general comments appear with a black font on the graphical view.

You can move them (drag&drop with the mouse): click with the left button on the comment you want to move, move the mouse and stop pressing th left button). It is also possible to

delete a comment directly from the graphical view: select it with the mouse, then click on the "Del" button on the keyboard.

Comments specific to each stage and/or each situation can also be defined in the same way (menu "Stages and conditions", item "Comments for selected phase" and "Comments for select set of conditions").

The 3 types of comments (general, stages and conditions) are displayed with different colours.

3.3.2.7. Zoom

Five zoom options are available:

- Zoom in and Zoom out: these two options are associated. They are available from the "View" menu, but also from two buttons of the general toolbar [🔍] (zoom in) and [🔍] (zoom out). They enable to change the zoom level.

  Horizontal and vertical scrollbars enable to "move" the model, when only part of it is displayed (depending on the zoom level).

- Define scale: this option is available only in the "View" menu. It opens a dialogue box (Figure 21) that enables to define a given scale (the default value is the current scale of the opened project). When the dialogue box is closed, the drawing is adjusted to the scale that was typed in. Again, scrollbars are displayed if necessary.
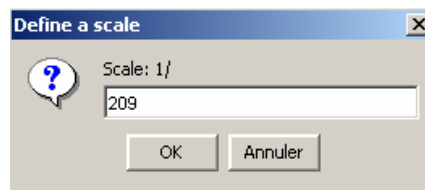
Figure 21: defining a scale

- Defining a zone with the mouse: this option is available from the "View" menu and with the [⬚] button of the general toolbar. It enables to draw the zone that should be zoomed in: click on the left corner of the zone to zoom in, then move the mouse and release the left button in the opposite corner of the zone to zoom in. The scale is then adjusted to the zone that was defined (always keeping the same scale in both directions). Again, scrollbars are displayed if necessary.

- View whole model: this option is available from the "View" menu, but also with the pop up menu in the graphical view. It displays automatically the zoom level that allows to view the whole project. It is the default option when opening a project.

It is possible to use different successive zoom options: for example, first zoom into a zone defined by the mouse, then zoom out or define a scale.

The scale value (always the same in both directions x and y) is always displayed in the white bar at the bottom of the graphical view.

As indicated previously, the default active option is "View whole model". When creating a new project, the scale is initialised after the maximum and minimum x-coordinates of the project have been typed in (in the "General settings" dialogue box, section 4.2).

As the user adds new objects to the model geometry, the scale is adapted if necessary.

The scale value is never saved: when opening a file, the zoom level is always reinitialised to "View whole project".

3.3.2.8. Table of soil characteristics

This option opens a window in which a summary table of with all sets of soil properties for the project is displayed. All characteristics are in the table, except for the complex characteristics

like cohesion anisotropy, for which it is simply indicated that they are defined by a specific table.
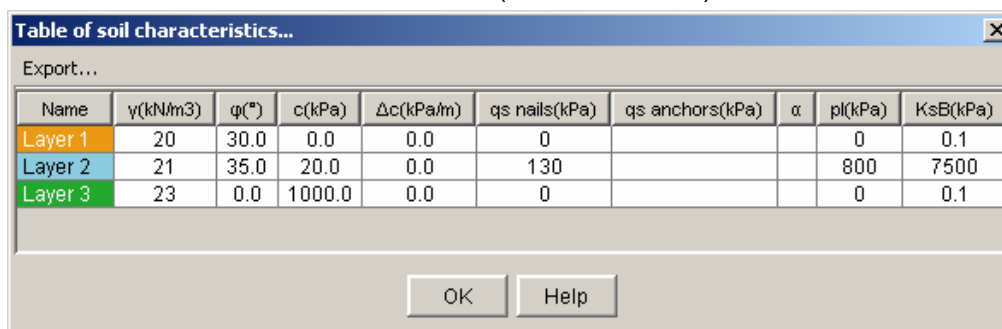
The table can be moved, and can be kept opened while working in Talren 4.

This option enables to view properties of all the project soil layers at once, and thus to quickly check data.

Thanks to this option, you can also copy/paste the soils table in a report (Microsoft Word® document for example), either as a picture, or as a table.

A checkbox allows to choose if you want to display all soil layers, or only those that are active in the current stage. In the "Project data" mode, this checkbox is not displayed.

This table does not allow for data change (no edit mode). Values can only be changed in the dialogue box for soil characteristics definition (menu or button).

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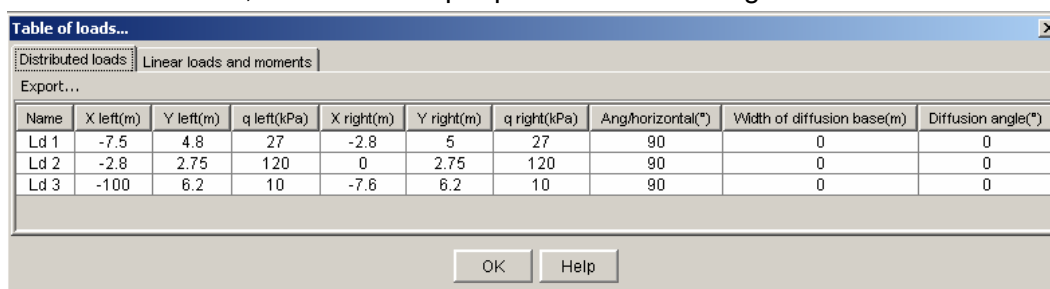
Name	γ(kN/m3)	φ(°)	c(kPa)	Δc(kPa/m)	qs nails(kPa)	qs anchors(kPa)	α	pl(kPa)	KsB(kPa)
Layer 1	20	30.0	0.0	0.0	0			0	0.1
Layer 2	21	35.0	20.0	0.0	130			800	7500
Layer 3	23	0.0	1000.0	0.0	0			0	0.1

Figure 22: table of soil characteristics

3.3.2.9. Table of loads

This option opens a window in which 2 summary tables are available through tabs: one table per load type (distributed and linear/moments). Only the load types for which elements have been defined in the project are available. If no load is defined in the project, the item for table of loads is disabled in the "View" menu.

The table can be moved, and can be kept opened while working in Talren 4.

Table of loads...

Distributed loads | Linear loads and moments

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	-7.5	4.8	27	-2.8	5	27	90	0	0
Ld 2	-2.8	2.75	120	0	2.75	120	90	0	0
Ld 3	-100	6.2	10	-7.6	6.2	10	90	0	0

Figure 23: table of loads

This option enables to view all the project loads at once, and thus to quickly check data.

Thanks to this option, you can also copy/paste the table in a report (Microsoft Word® document for example), either as a picture, or as a table.

A checkbox allows to choose if you want to display all loads, or only those that are active in the current stage. In the "Project data" mode, this checkbox is not displayed.

This table does not allow for data change (no edit mode). Values can only be changed in the dialogue box for loads definition (menu or button).
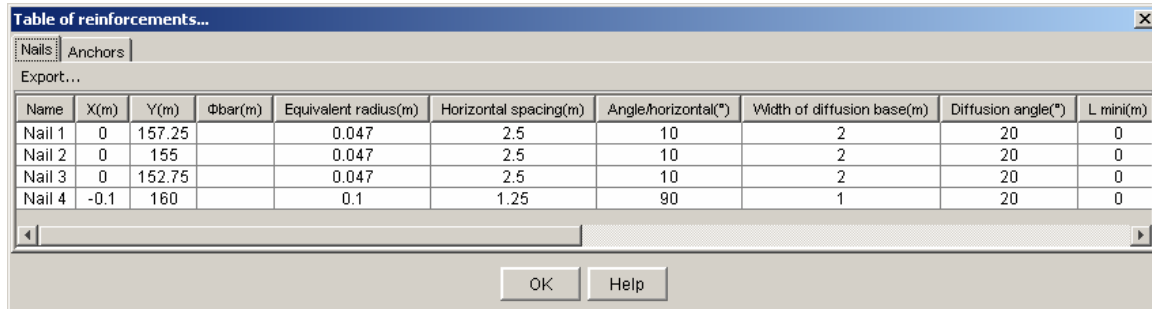
3.3.2.10. Table of reinforcements

This option opens a window in which 4 summary tables are available through tabs: one table per reinforcement type: nails, anchors, strips and braces. Only the reinforcement types for

which elements have been defined in the project are available. If no reinforcement is defined in the project, the item for table of reinforcements is disabled in the "View" menu.

The table can be moved, and can be kept opened while working in Talren 4.

This option enables to view all the project reinforcements at once, and thus to quickly check data.

Name	X(m)	Y(m)	Φbar(m)	Equivalent radius(m)	Horizontal spacing(m)	Angle/horizontal(°)	Width of diffusion base(m)	Diffusion angle(°)	L mini(m)
Nail 1	0	157.25		0.047	2.5	10	2	20	0
Nail 2	0	155		0.047	2.5	10	2	20	0
Nail 3	0	152.75		0.047	2.5	10	2	20	0
Nail 4	-0.1	160		0.1	1.25	90	1	20	0

Figure 24: table of reinforcements

Thanks to this option, you can also copy/paste the table in a report (Microsoft Word® document for example), either as a picture, or as a table.

A checkbox allows to choose if you want to display all reinforcements, or only those that are active in the current stage. In the "Project data" mode, this checkbox is not displayed.

This table does not allow for data change (no edit mode). Values can only be changed in the dialogue box for reinforcements definition (menu or button).

3.4. Status bar

It displays the following information:

- Coordinates (true scale) of the mouse on the graphical view;
- Gauge indicating the calculation progress (during calculation only);

X -13.00 m   Y 91.00 m   0%
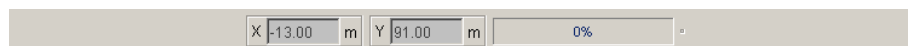
Figure 25: status bar

3.5. Standard functionnalities of the Windows® environment

3.5.1. File: New / Open / Save (as) / Working directory / Close / Recent files / Exit

These functionalities are standard.

They display usual screens of Windows® applications. For example, the "open" or "save" dialogue boxes propose an explorer allowing to choose a directory (and if necessary to create it), etc.
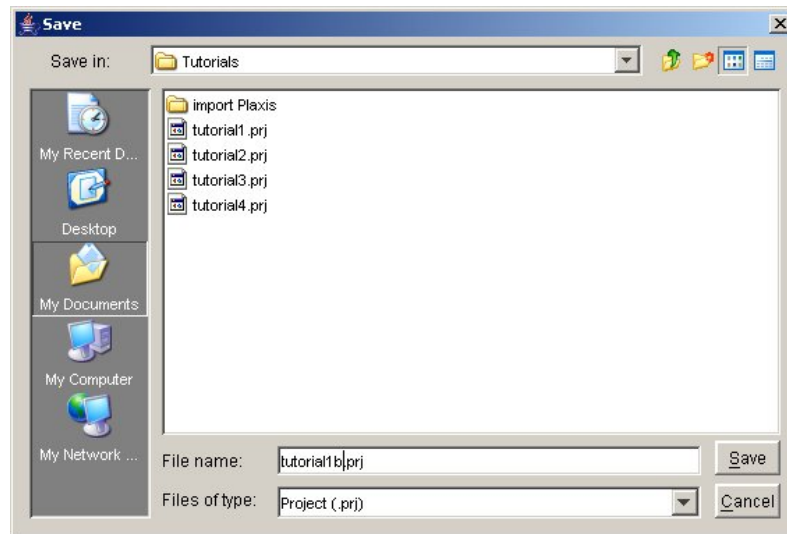
Figure 26: "save" dialogue box

The menu options and corresponding buttons are activated/deactivated with respect to relevant options (for example, the "Save" option will be active if data changes have been made since the last save, if not, the "save as" option alone is available).

The 4 most recent files are accessible directly from the "File" menu.

When opening or saving files, the default directory is the working directory. After installing Talren 4, the default working directory is the examples directory. It can be changed at any time using the option "Working directory" of the "File" menu (Figure 27).
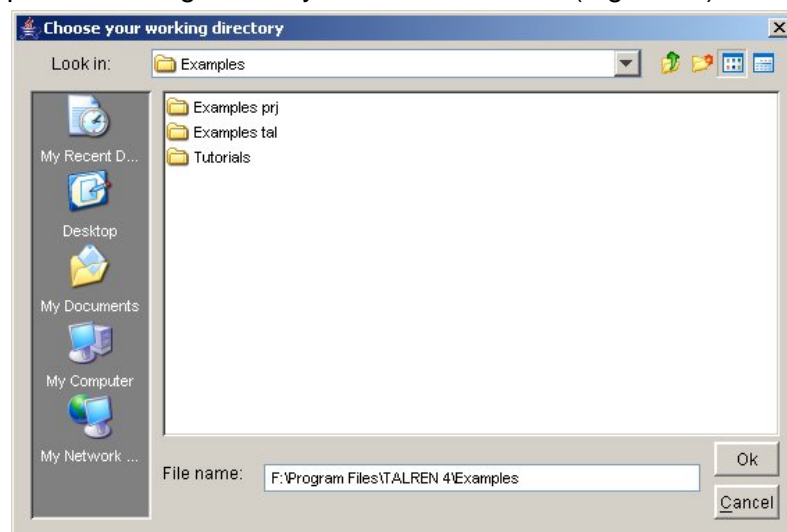
Figure 27: changing the working directory

3.5.2. Printing

It is managed by the Windows® version of each computer.

Printer settings can be changed before printing: choice of the printer and printer settings available when validating printing choices. Refer to section 6.5.

3.5.3. Copy/Paste and copy to clipboard

The usual Copy/Paste Windows® functionality is available in many forms, depending on the element to which it applies.

3.5.3.1.   Copy/paste texts and values

The texts and values typed in the cells (individual or table cells) can be copied and pasted into another cell with usual keyboard shortcuts: "Ctrl + Ins" or "Ctrl + C" to Copy, then "Shift + Ins" or "Ctrl + V" to paste.

Note: in the tables, it's possible to copy only the content of each cell individually. It's not possible to copy a line or column, or the whole table.

3.5.3.2.   Copy/Paste soil characteristics, loads, and reinforcements

There are many ways in Talren 4 to copy/paste these elements:

- For loads and reinforcements only: select an element with the mouse, copy it with the "Edit/Copy" menu item (or with the shortcut "Ctrl+C"), paste it with the "Edition/Paste" menu item (or with the shortcut "Ctrl+V"). The new element is slightly moved compared to the original element, so that you can see both elements. In the list of loads or reinforcements, the new element appears in last position.

- For loads and reinforcements only: same operation as hereabove, but the copy operation can be reached by the pop up menu (right click with the mouse on the element), or with the [icon] button of the main toolbar.

- For soil characteristic sets, loads and reinforcements: in the corresponding dialogue boxes, the "Duplicate" button enables to copy/paste an element with one single click. In the list on the left of the dialogue box, the new duplicated element appears in last position. Moreover, here again (for loads and reinforcements), the new element is slightly moved compared to the original element.

3.5.3.3.   Copy graphical view to clipboard

The [icon] button of the main toolbar enables at any time to copy the graphical view to the clipboard (same zoom level, same display parameters, etc. as on the screen) The screenshot includes the horizontal and vertical rulers.

3.5.3.4.   Copy tables to clipboard

Any table can be copied to the clipboard, with the "Export" menu of the active dialogue box (Figure 28):

- Either as a table of values;

- Or as a picture. Be careful, in this case, only the visible part of the table is copied (in the case of large tables, with the horizontal scrollbar, a part of the table will be missing on the picture).
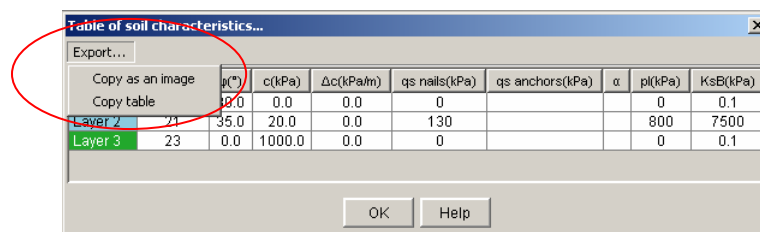
Figure 28: table copy options

3.5.3.5.   Copy graphics to clipboard

Any graphic displayed on the screen can be copied to the clipboard (as a picture), with the "Export" menu of the active dialogue box. In the example of Figure 29, only the first menu option concerns graphics. The two other options concern tables.
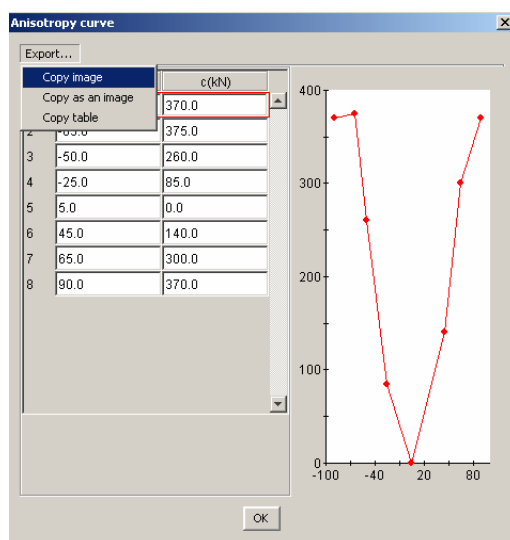
Figure 29: graphics copy options

3.5.4. Help

An online help is available ("Help" menu). Help buttons are also available in most dialogue boxes.

3.5.5. About

This standard option enables to display an information screen: version of Talren 4, username (from the hardlock), Terrasol coordinates, etc.

4. "PROJECT DATA" MODE

4.1. Principles

All elements necessary for the definition of the different construction stages of a project should be defined in the "Project data" mode: geometrical lines delimiting the layers, soil profile, excavation or embankment levels, all soil layer properties that will be used, loads and reinforcements.

These elements will then be activated and/or deactivated in the successive construction stages.

In the Data mode, it is therefore necessary to define all the data necessary for the description of the whole project.

Those data are the following:

- General settings: title, units (and water unit weight, which is changed automatically depending on the units, but can also be changed manually), calculation method and partial safety factors to be used by default in the project, etc;
- Geometry: slope boundaries, limit of soil layers, earthwork levels;
- Soil layer properties: all layers which will be used during the construction stages;
- Loads: all loads that will be activated during the construction stages;
- Reinforcements: all reinforcements that will be activated during the construction stages.

The way to define stage properties is detailed in the different following sections.
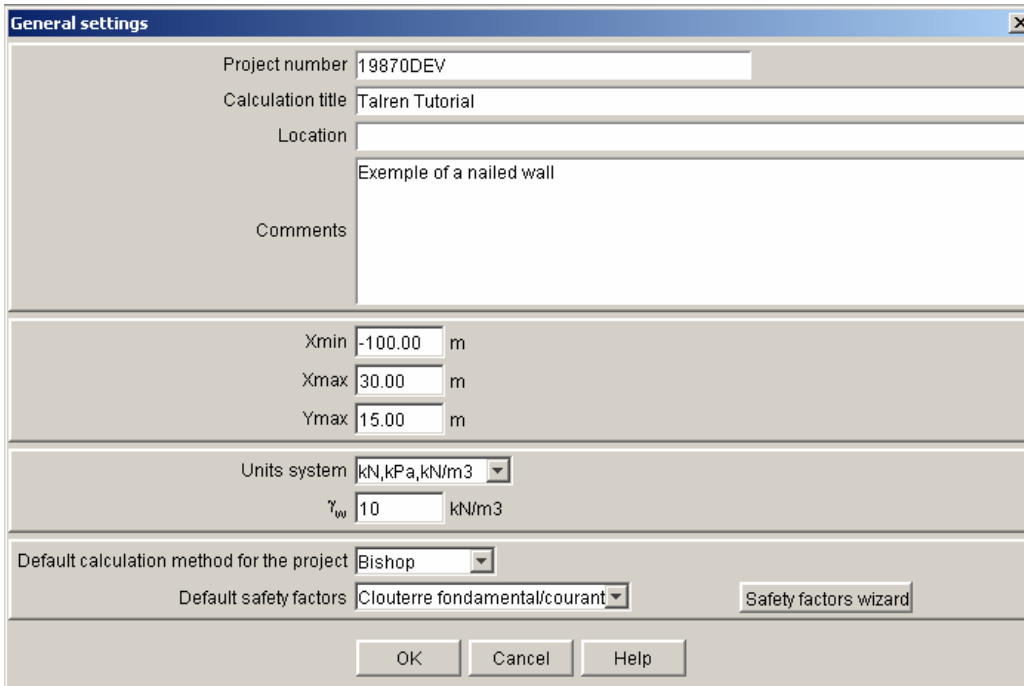
Generally, definition or modification of most geometrical data is possible with the mouse (buttons of the "Data" toolbar), but the equivalent operations are also available through dialogue boxes (accessible by the various items of the "Data" menu).

The user can therefore most often choose between various solutions to perform the same operation.

Elements properties should be input through dialogue boxes. When validating dialogue boxes, Talren 4 checks data and may produce error messages (corrections must be then made before validating the dialogue box) or warnings (it is possible to continue).

4.2. General settings

They are displayed in a dialogue box (Figure 30), accessible either from the Data/General settings menu item, or from the $\boxed{i}$ button.

Figure 30: general settings dialogue box

The data to input in this dialogue box are the following:

- *Project number:* compulsory alphanumeric field (maximum 20 characters);
- *Calculation title*: compulsory alphanumeric field (maximum 40 characters);
- *Location and comments:* not compulsory;
- X_{min}, X_{max} and Y_{max}: X limits for the model, and maximum project level (compulsory values);

  IMPORTANT: these values are used to automatically calculate the slope boundary. They should be defined as the real minimum and maximum x-coordinates of the model (section 4.3.1.6).

- *Units and water unit weight*: 3 unit systems are proposed, respectively linked to kN, MN or tons. The water unit weight is changed automatically (but only after validation of the dialogue box) depending on the units. It is also possible to change it manually.

  It is possible to change units for an existing project. In this case, 2 options are proposed: either convert all data already typed in (in the case of a "true" unit change), or not convert the data already typed in (in the case where the data were already typed in in a different unit than the one displayed, this option enables to make values and units display consistent with one another).

- *Calculation method and default safety factors*: the default calculation method is the Bishop method. The "Default safety factor" parameter is empty by default when creating a new project: it is not compulsory to chose a set of safety factors here.

  These two options allow to initialise default values that will then be automatically proposed as default values for all new situations. For example, in the case of a project with many stages, if the user knows from the beginning that he will have to make all calculations according to the perturbations method, with the same partial safety factors, the user can define these parameters as default values for his project (in the general settings dialogue box). They will then be automatically proposed for each situation (but can always be punctually modified for each situation if necessary). Also refer to section 4.7.2 for the definition of partial safety factors, and to section 5.2.5 for the situations properties.

Four calculation methods are available: Fellenius, Bishop, Perturbations, and yield design calculation. If the user chooses perturbations method, an additional parameter appears: the exponent for $\tan(\alpha)$, that can be equal either to 1 (default value) or 2 (refer to technical manual for additional information about this parameter).

4.3. Geometry

The geometry includes:

- The initial slope boundary (groundlevel);
- The soil layer limits;
- The geometrical limits necessary for the definition of each contruction stage: excavation or embankment levels, limits for a soil reinforcement, etc.

4.3.1. Drawing the geometry with the mouse

4.3.1.1. Drawing

The [pencil icon] button (drawing of geometrical lines) enables to activate the drawing mode (the corresponding option of the "Data" menu does not activate the drawing mode, but opens a dialogue box containing tables with all points and segments: refer to beginning of section 4).

2 different methods are then possible (and can be combined at any moment). The user can:

- Either 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 are created. To define points without connecting them with segments, simply click on the points and release the mouse button after each click.

  IMPORTANT: By default, Talren 4 adopts a scale adapted to X_{min} and X_{max}, and to view Y_{max}. It may happen that the user needs to draw geometrical elements lower than the visible zone on the screen. To do so, the best solution consists in drawing a point at the lowest model level, using the method described in the following paragraph (typing in of the coordinates with the keyboard). After validation of this low point, a vertical scrollbar will appear, and will allow access to any part of the model (it is of course also possible to modify the zoom level to see the whole model, refer to section 3.3.2.7).

- Or type 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).
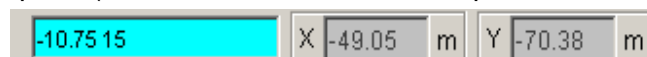
  The background of the cell will then become blue: this means that the next point typed in will be connected to the previous one with a segment. To "pull up the pen", press the "Esc" button of the keyboard. The cell background turns white again, meaning that the next point typed in will not be connected to the previous one with a segment. One can at any time change from the "mouse drawing mode" to the "coordinates drawing mode".

When coming back to the drawing mode after doing other operations, the first point that is drawn is not connected to the last point of the previous drawing session. It is independent. However, the second point of the new drawing session will be connected to the first point of the new drawing session and so on: a new polyline is created until it is interrupted by the right button of the mouse or by the "Esc" key of the keyboard.

If a point is drawn on an existing point, the software does not create the new point, but it connects the already existing point to the previous point with a segment (except if this point is the first of the drawing session) and to the next point (except if this point is the last).

In the same way, if two segments overlap each other, the software only saves the first segment created. However, if the extremities of the segments are not the same, all distinct points are created.

If a point is drawn on an existing segment, a new point is created, and connected to both extremities of the existing segment.

Finally, if a segment intersects an existing segment, a point is automatically generated at the intersection, and connected to the extremities of both segments that intersect.

Advices and complementary drawing tips:

- First of all, to facilitate the drawing with the mouse, it is recommended to activate the grid display and hooking. To do so, click on the "View" menu, and select the "Grid" option (section 3.3.2.2). The grid spacing can be set (choose 0.25 or 0.5 m for example), and both check-boxes should be activated. The mouse will then move on the screen by a 0.25 or 0.5 m increment (or any other chosen value). The smaller the spacing, the more precise the layout will be.

- As soon as the user "closes" a soil cluster, it turns grey: this means that Talren 4 identifies that the cluster was closed, and that it attributed default properties ("substratum") to this cluster (the user will later assign another soil layer (refer to section 4.6). These "substratum" properties are automatically created for each project, and if they are attributed to some soil clusters when calculating, Talren 4 automatically considers that failure surfaces shouldn't enter these substratum soil clusters.

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

- The mouse coordinates are always displayed in the status bar.

- The points and segments numbers can be displayed on the graphical view with the "View" menu: items "Point numbers" and "Segment numbers" (section 3.3.2.4).

- The mouse shape gives indications on the current drawing status, for example:

    o to add a new geometrical element,

    o to connect a segment to an existing point (start or end of a segment)

    o to add a point on an existing segment.

4.3.1.2. Selecting points or segments

The button activates the selection mode. The selection mode is also accessible by the pop up menu: right click in the graphical view.

All points and segments can be selected with the mouse (left click on an element).

4.3.1.3. Moving points or segments

After selection of a point or segment, it is possible to move it with the mouse by drag & drop (left click and move the mouse to another position).

When a point of the geometry is moved, the segments that are connected to this point are changed accordingly.

Depending on the move, additional points can be generated if necessary (movement of a point leading to the intersection of 2 segments for example).

Refer also to section 4.3.1.5.

4.3.1.4. Deleting points or segments

After selection of one or several elements, it is possible to delete them pressing the "Del" key, or clicking on the right mouse button and selecting "Delete" in the pop up menu.

4.3.1.5. Display and modification of the properties of a point or segment

It is possible to display the properties of an element: either by double-clicking on it, or, after having selected it (one at a time for this operation), by right-clicking and selecting "Edit" in the pop up menu. The relevant dialogue box is then displayed (coordinates for a geometrical point, extreme points for a segment). It is possible to change the properties of the element, and to validate them. The graphical display is updated if necessary.
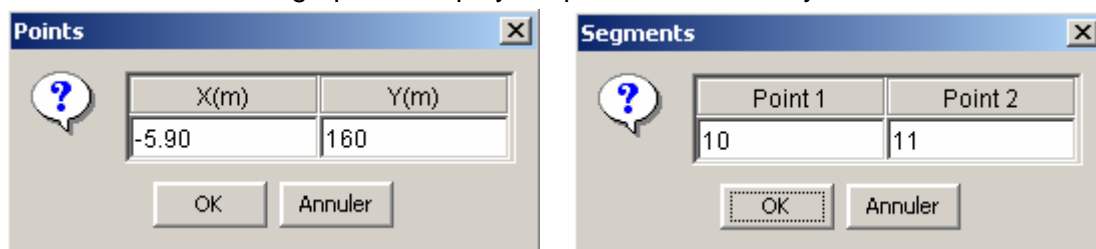
Figure 31: properties dialogue box for a point or segment

4.3.1.6. Slope boundary

The slope boundary is by default automatically identified, as soon as geometrical lines allow to identify a continuous polyline from X_{min} to X_{max}.

IMPORTANT: If no segment of the geometry reaches X_{min} or X_{max}, the boundary cannot be recognised, and the project cannot be calculated.

The slope boundary is calculated by default to be the groundlevel.

It is represented with a bold line on the drawing.

Refer also to section 4.3.2.

4.3.2. "Geometry" dialogue box

This dialogue box (Figure 32) is accessible through the "Project data" menu, option "Geometry".
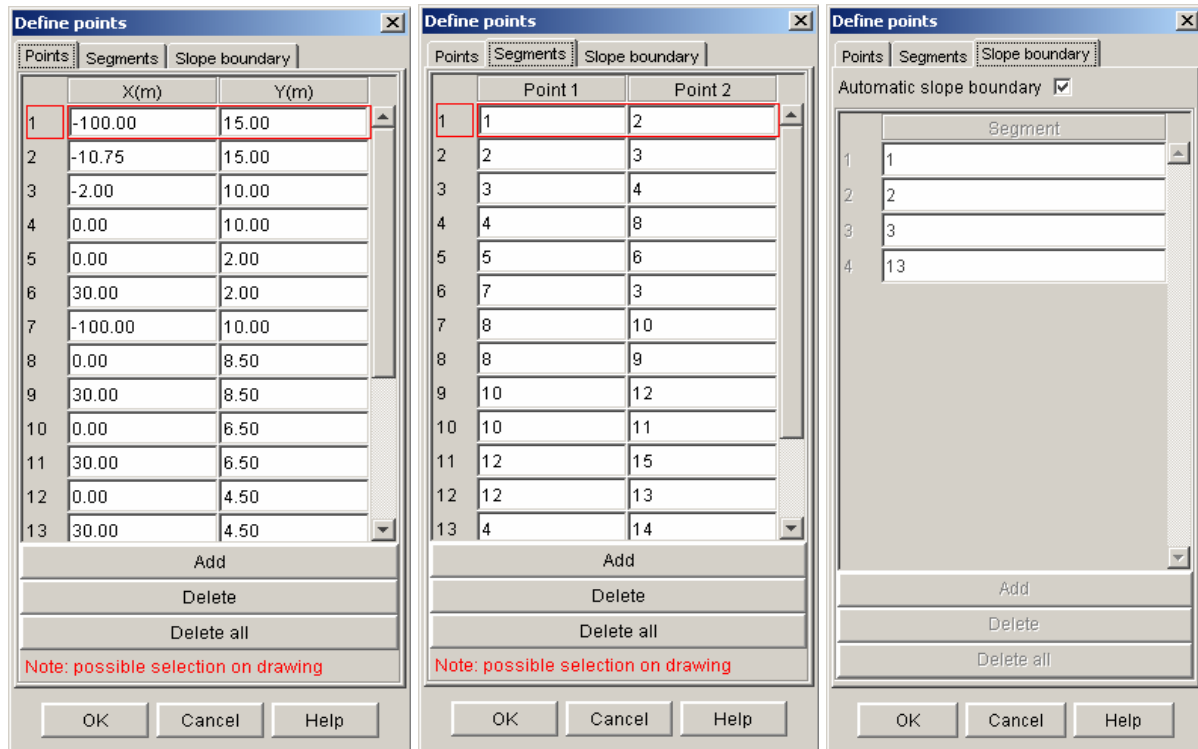
Figure 32: geometry dialogue box (3 tabs)

This dialogue box contains 3 tabs: "Points", "Segments" and "Slope boundary". It enables to view and edit the geometrical data.

However, it is not compulsory to go through this dialogue box, especially if the user chose to define the geometry with the mouse (section 4.3.1).

The data displayed in this dialogue box are the following:

- "Points" tab:
    - *Numbers of the points* (those are automatically attributed and cannot be edited).
    - *X and Y coordinates* (in m) of all geometry points.
- "Segments" tab:
    - *Number of the segments* (they are automatically attributed and cannot be edited).
    - *Extremities (point 1 and point 2) of each segment of the geometry*: each extremity is defined by the number of a point (as defined in the first tab).
- "Slope boundary" tab:
    - *"Automatic slope boundary"*: this checkbox is activated by default for a new file, and deactivated by default when loading former .tal files. If activated, the slope boundary will be automatically calculated. In the other case, it is necessary to define manually in the table the list of segments defining the slope boundary.

        IMPORTANT: it is compulsory to activate the "automatic slope boundary" checkbox if the project includes several construction stages, with activation/deactivation of soil clusters (and therefore possibly changes of the slope boundary from one stage to the other): if many stages are defined when the boundary is not "automatic", the boundary will not be changed from one stage to the other, and the slope boundary corresponding to the initial geometry will be taken into account in the calculations for all stages (which can lead to errors).
    - *Numbers of the segments of the slope boundary* (those are automatically attributed and cannot be edited).

o *List of the segments constituting the slope boundary*: this list cannot be edited if the "Automatic slope boundary" checkbox is activated.

For all 3 tabs, "Add", "Delete" and "Delete all" buttons enable either to add a new point or segment, or to delete the selected table line (point, segment or segment of the slope boundary), or to delete all elements defined in the active tab. Be careful, if [Delete all] is clicked in the Point tab for example, all points will be deleted, but all segments and slope boundary segments will be deleted too.

On the "Points" and "Segments" tab, the mention "possible selection on drawing" means that if you click on the graphical view while the "Points" tab is visible, for example, a new point corresponding to the mouse position will be added in the points list (the behaviour is the same for segments).

IMPORTANT:

- 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 (section 3.3.1.2), 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" (section 3.3.1.2).

- The order of the points of a segment (left and right) is not important.

- For vertical lines: Talren 4 allows for the definition of true vertical segments. The calculation follows the direction from the left to the right to determine whether vertical lines are upwards or downwards. For upwards vertical lines, the program will automatically define (the user doesn't have to care about that) a shift in X with 2 cm between the bottom and top points of the segment (for calculation only).

4.4. Loads

IMPORTANT:

- A distributed load should not be defined over several segments (Figure 33). Each distributed load must be included in one single segment. A load should be divided into several loads (one per segment) if necessary.

- Distributed loads must be defined on the slope boundary (neither above, nor below).

- It is not necessary to define geometrical points at the extremities of distributed loads.
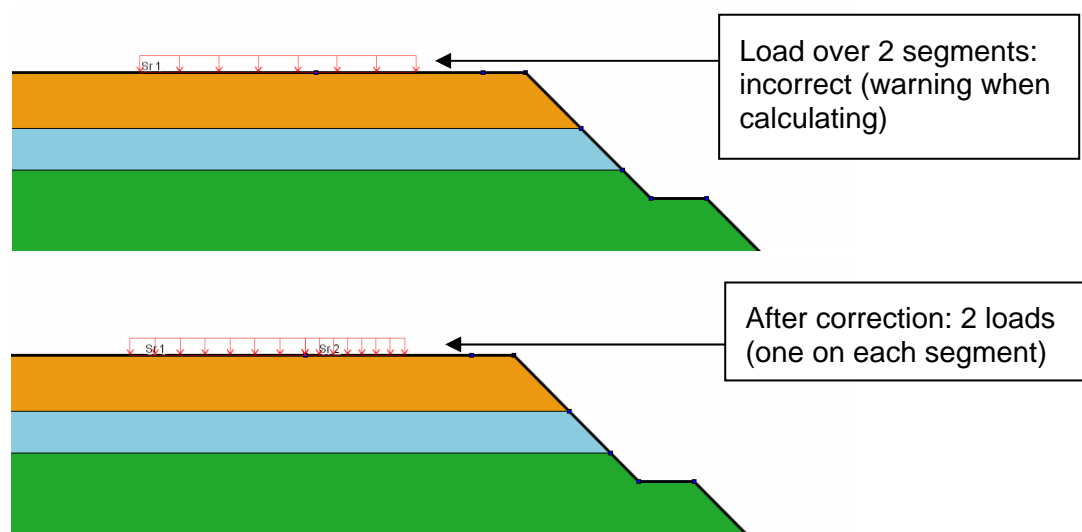
Figure 33: case of loads covering several segments

4.4.1. Defining loads with the mouse

4.4.1.1. Drawing distributed loads with the mouse

You should first click on the [⊞] button of the toolbar, then click with the left mouse button on two successive points of the drawing (to define the base segment for the load), or type in the coordinates of the two points with the keyboard (in the edition cell that appears in the status bar, refer to section 4.3.1.1). The load is then created between the two points with a default vertical inclination. It is displayed in red (if several distributed loads are defined, they are scaled taking into account their intensities).

It is necessary to complete the geometrical definition of the load with the definition of the load value. Refer to sections 4.4.1.7 or 4.4.2.1.
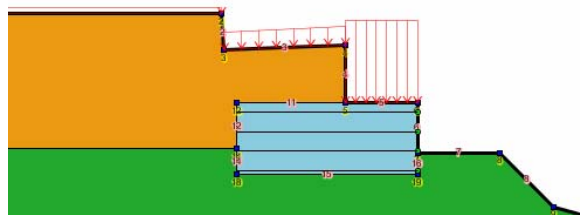
Figure 34: graphical display of a distributed load

4.4.1.2. Drawing linear loads and/or moments with the mouse

You should first click on the [↑] button of the toolbar, then click with the left mouse button at the position where the linear load and/or moment should be applied (or type in the coordinates of the point on the keyboard). It is then created with a default vertical inclination.

It is necessary to complete the geometrical definition of the linear load/moment with the definition of the load/moment values. Refer to sections 4.4.1.7 or 4.4.2.2.
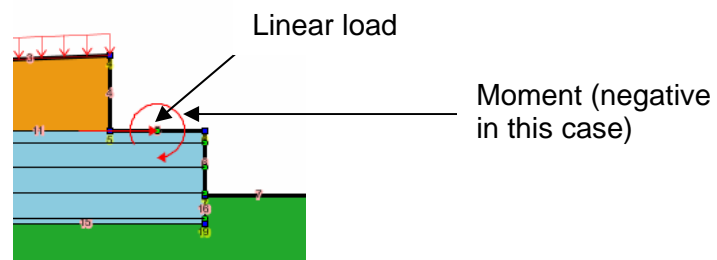
Figure 35: graphical display of a linear load and moment

4.4.1.3. Selecting loads

To activate the selection mode, click on the [↖] button of the general toolbar.

All loads can be selected with the mouse (with a left click on the point or segment where the load is applied).

4.4.1.4. Copy/paste loads

Refer to section 3.5.3.2.

4.4.1.5. Moving loads

After selecting a load, it is possible to move it with the mouse by drag & drop (left click and move the mouse to another position).

When the load is moved, the geometry segments or points are not moved at the same time.

4.4.1.6. Deleting loads

After selecting one or several loads, it is possible to delete them by pressing the "Del" key, or by clicking with the right mouse button and selecting "Delete" in the pop up menu.

4.4.1.7. Display and modification of the properties of a load

It is possible to display the properties of a load: either by double-clicking it, or, after selecting the element, by clicking on the right mouse button and selecting "Edit" in the pop up menu. The display of properties consists in the display of a dialogue box similar to the loads dialogue box, but with the properties of the selected load only. It is then possible to change the load properties, and to validate them. The graphical display is updated if necessary.
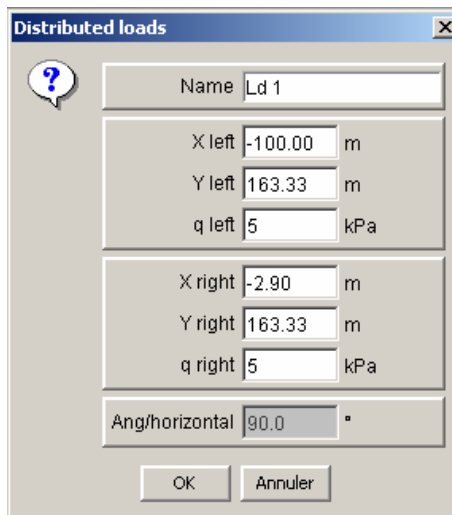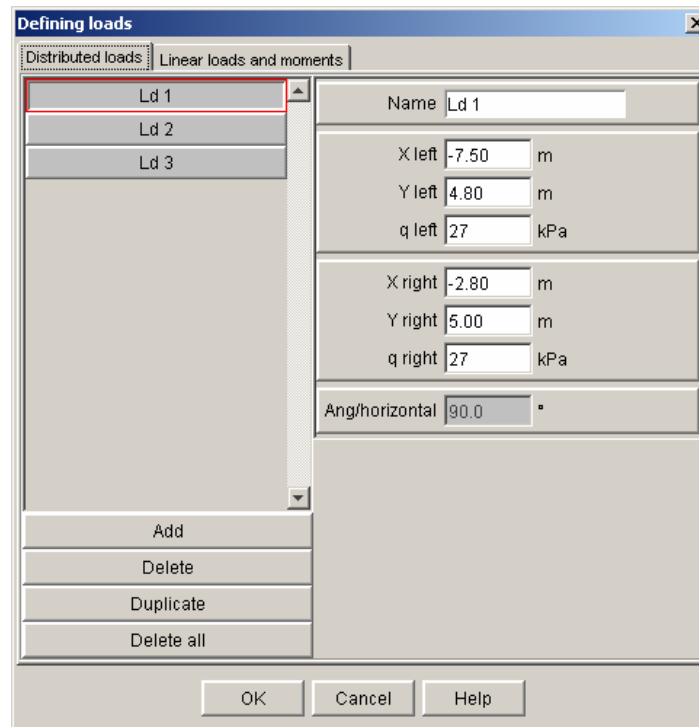
Figure 36: properties of a distributed load

4.4.2. "Loads" dialogue box

This dialogue box is accessible through the "Project data" menu, "Loads" option. It includes 2 tabs: "Distributed Loads", "Linear loads and moments". It allows to view and edit load properties. For each tab, the list on the left makes it possible to view and select project loads. In addition, for each tab, the buttons "Add", "Delete", "Duplicate" and "Delete all" allow to add a new load (distributed or linear load/moment), to remove the selected load, to duplicate the selected load (creation of a new load with the same properties), or to remove the whole list of loads in the active tab (either distributed loads or linear loads/moments).

4.4.2.1. Distributed Loads

Figure 37: loads dialogue box: Tab 1 (distributed loads)

The data displayed for each distributed load are as follows:

- *Name*: by default, Talren 4 names distributed loads "Sd i", but these names can be changed (10 characters maximum);
- X_{left} *and* Y_{left}: coordinates X and Y (m) of the left point of the load;
- Q_{left}: value of the load in this point (pressure);
- X_{right} *and* Y_{right}: coordinates X and Y (m) of the right point of the load;
- Q_{right}: value of the load in this point (pressure);
- *Ang/horizontal*: inclination of the load (compared to the horizontal line) in degrees: by default, the distributed loads are vertical and directed downwards (90°). It is possible change this angle only in the case of calculations using the yield design calculation method.

If the left and right points are not in the right order, they are automatically reversed during the validation of the dialogue box (a warning is displayed about that).

In the case of loads applied onto vertical lines (in the case of the yield design calculation method), the order of both end points is indifferent.

4.4.2.2. Linear loads / moments
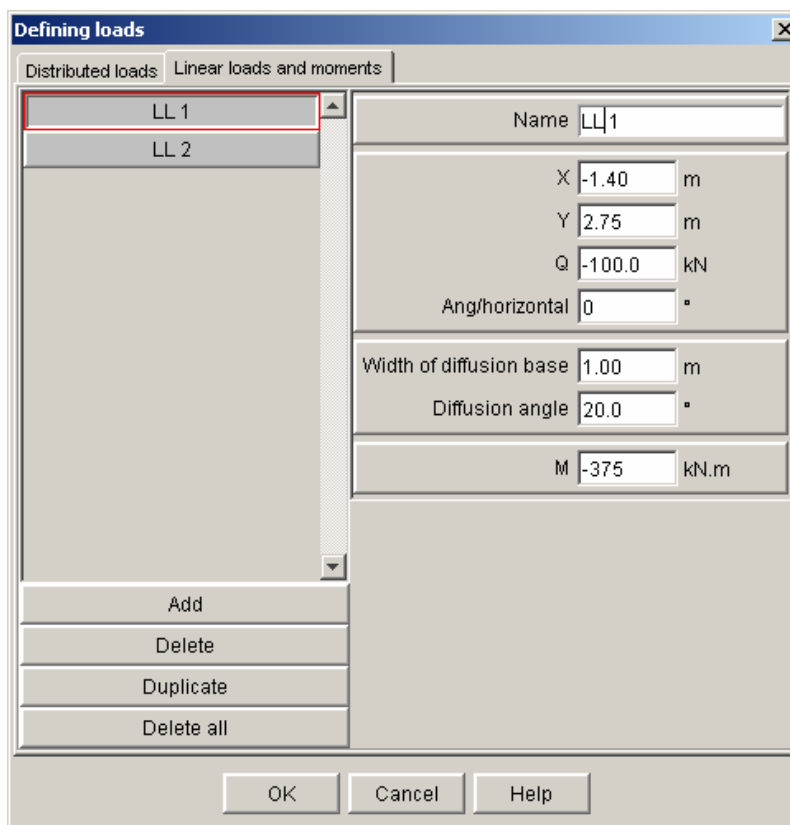
Figure 38: loads dialogue box: Tab 2 (linear loads/moments)

The data displayed for linear loads and/or moments are as follows:

- *Name*: by default, Talren 4 names these loads "Sl i", but these names can be changed (10 characters maximum);
- *X and Y*: coordinates (m) of the point of application of the load;
- *Q*: value of the load in this point (force/ml);
- *Ang/horizontal*: inclination of the load in degrees (compared to the horizontal line, positive in the trigonometrical direction): by default, linear loads are vertical and directed downwards (90°);
- *Width of the base of diffusion (m) and diffusion angle (°):* these 2 parameters possibly enable to take the load into account as a fictitious anchor (refer to technical manual).
- *M*: value of the additional moment (positive in the trigonometrical direction).

Each element can include either a load value, or a moment value, or both. An empty value for Q or M is equivalent to a zero value.

4.5. Reinforcements

4.5.1. Introduction to reinforcements

Type	Sollicitation mode	Interaction soil/reinforcement	Specific comments
Nails	Traction +shear	Interaction soil/nail	• Can work in calculated traction + imposed shear (can be traction only); this behaviour corresponds to 80% of the nailing projects. • Can work with no traction + calculated shear; this behaviour corresponds to slope reinforcements with piles or micro piles. • Can work in calculated traction + calculated shear. • Examples of specific normative documents: french standard XP P 94-240, Clouterre recommendations.
Anchors	Traction	Interaction soil/anchor	• Traction only. • Examples of specific normative document: TA 95 recommendations.
Strips (reinforced earth, Freyssisol (paraweb), VSol, geotextiles, geogrids, etc)	Traction	Interaction soil/strip	• Traction only. • Examples of specific normative documents: french standard XP P 94-220
Struts	Compression	No interaction	• Struts are considered as external compression forces. • They have no direct interaction (friction type) with the soil (but in the calculation, a strut acts on the forces normal to the failure surface).

Table 1: reinforcement types in Talren 4

Refer also to the technical manual (chapter C of this manual) for more information about reinforcements.

4.5.2. Definition of the reinforcements with the mouse

4.5.2.1. Drawing reinforcements with the mouse

You should first click on one of the buttons of the toolbar (⊞ for a nail, ⊞ for an anchor, ⊞ for a strip or ⊞ for a strut), then click with the left mouse button at the position of the reinforcement head (or type in its coordinates in the edition cell of the status bar). It is necessary to complete the definition of the reinforcement with its material and calculation characteristics. Refer also to chapters 4.5.2.6 or 4.5.3.
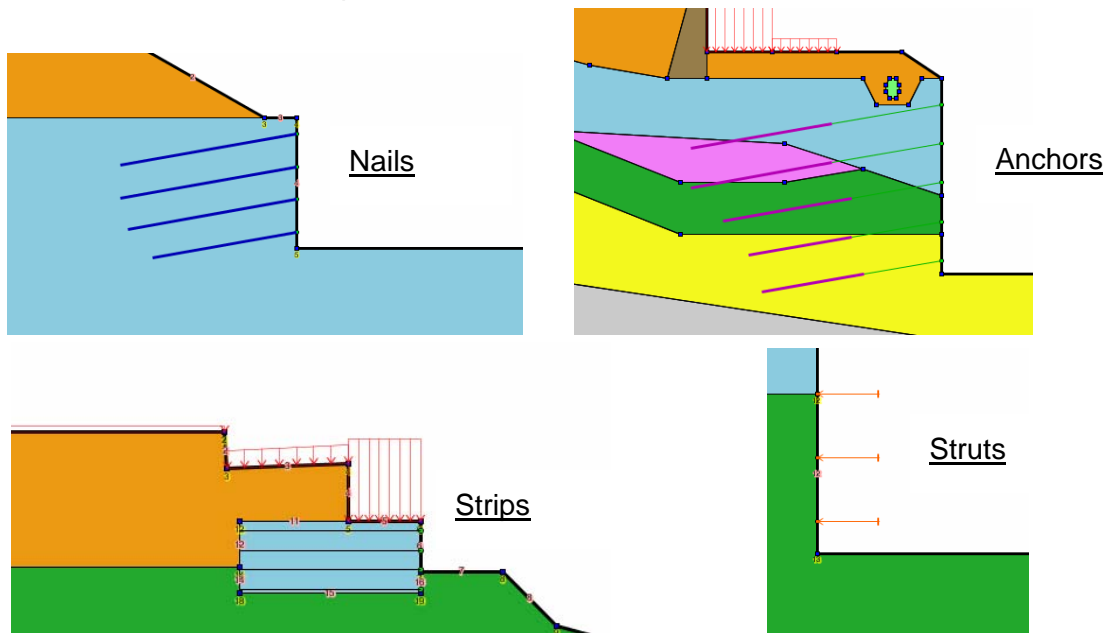
Figure 39: graphical display of reinforcements

4.5.2.2. Selecting reinforcements

To activate the selection mode, it is necessary to click on the button ⊞ of the general toolbar.
All reinforcements can be selected with the mouse (left click on the reinforcement head).

4.5.2.3. Copy/paste reinforcements

Refer to chapter 3.5.3.2.

4.5.2.4. Moving reinforcements

After selecting a reinforcement, it is possible to move it by drag & drop with the mouse (left click and move the mouse to another position).
When a reinforcement is moved, the points and segments of the geometry are not moved at the same time (geometrical independence).

4.5.2.5. Deleting reinforcements

After selecting one or more reinforcements, it is possible to delete them by pressing the "Del" key, or by right-clicking and choosing the "delete" option of the menu.

4.5.2.6. Display and modification of the properties of a reinforcement.

It is possible to display the properties of a reinforcement: either by double-clicking it, or, after selecting it, by right-clicking and selecting "Edit" in the pop up menu. The display of the

reinforcement properties consists in the display of a dialogue box similar to the reinforcements dialogue box, but specific to the selected reinforcement. It is then possible to view and edit the reinforcement properties, and to validate them. The graphical view is updated if necessary.
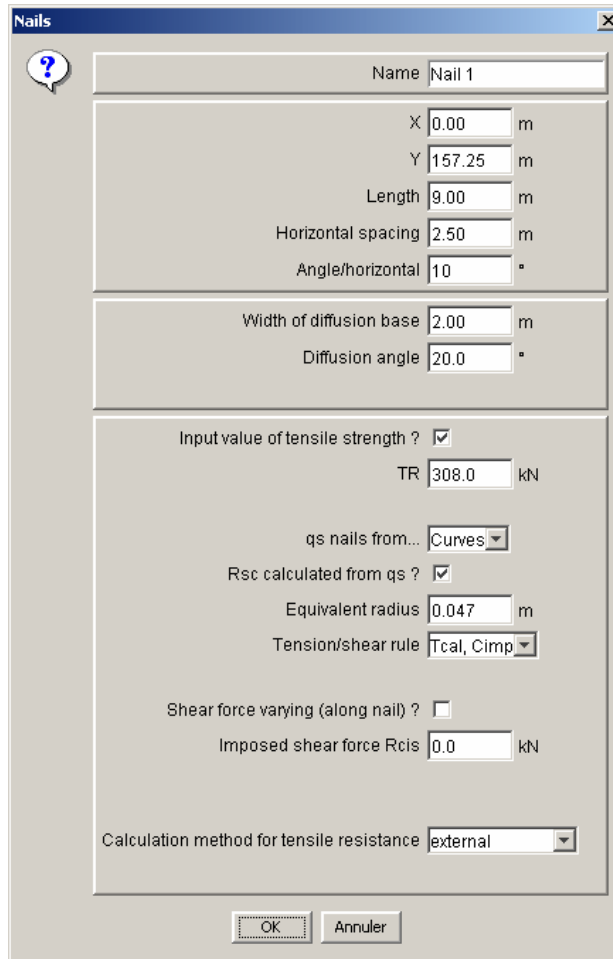
Figure 40: properties of a nail

4.5.3. "Reinforcements" dialogue box

This dialogue box is accessible through the menu "Data", option "Reinforcements". It includes 4 tabs, corresponding to the 4 reinforcement types which Talren 4 can take into account: nails, anchors, strips and struts. It allows for the edition of the reinforcements properties.

For each tab, the left list makes it possible to view the project reinforcements, and to edit their properties (select a reinforcement in the list to view its properties).

In addition, for each tab, the buttons "add", "remove", "duplicate" and "remove all" allow to add a new reinforcement, to remove the selected reinforcement, to duplicate the selected reinforcement (creation of a new reinforcement with the same properties), or to remove all the reinforcements defined in the active tab. The last button, "Display database", enables to display the reinforcements database.

4.5.3.1. Nails
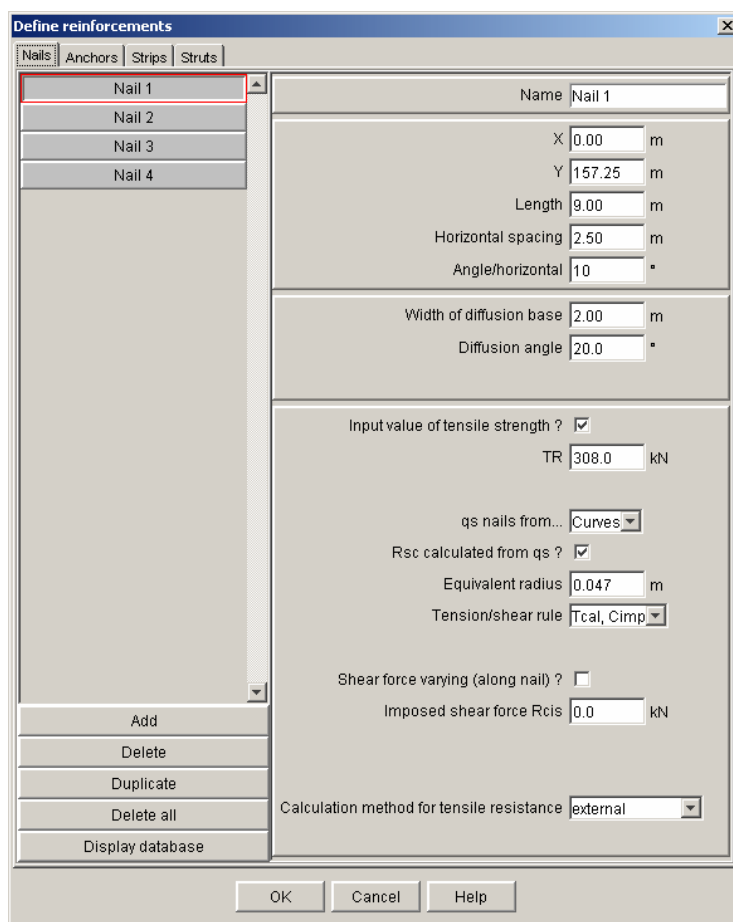
Figure 41: reinforcements dialogue box: tab 1 (nails)

Data displayed for each nail are as follows:

- *Name*: by default, Talren 4 names nails "Nail i", but these names can be changed (10 characters maximum)
- *X and Y* coordinates (m) for the nail head;
- *Length*: length of the nail (m);
- *Horizontal spacing* (m);
- *Angle/horizontal* (°): default value is zero. It is positive in the case illustrated on Figure 39 (trigonometrical direction);
- *Diffusion width (m) and diffusion angle (°):* in general, one can define for the diffusion width a value of 0,5 m for thin facings (reinforced concrete for example). For rigid walls (example: diaphragm walls), one can choose 1 or 2 m. The diffusion width can also be evaluated directly with $l_0 = {}^4\sqrt{\dfrac{4EI}{k_s B}} = {}^4\sqrt{\dfrac{4 \times facing\ stiffness}{soil\ reaction}}$.

  In the case of a support plate, the plate size can be used as diffusion width. These advices are valid for all types of reinforcements.

  On the other hand, the diffusion width and diffusion angle parameters are not used with the yield design calculation method (logarithmic spirals).

- *Input value of tensile strength (TR)*: this check box indicates whether the TR value (tensile strength of the nail)
  - o Is directly input as a data (force): Figure 41,

- o or must be calculated based on the steel bar diameter ($\varnothing_{bar}$ in m) and on the yield stress (σ_e, pressure unit) (Figure 42). This second choice is the default choice.
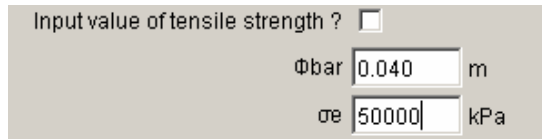
Figure 42: reinforcements dialogue box: tab 1 (nails)
when checkbox "Input value of tensile strength" is deactivated

- qs_{nails} *from*: 2 options are available: curves (default choice) and tests. The selected option will decide which value of the partial safety factor will be applied to the value of qs_{nails}. Refer also to chapter 4.7.2.3 for the description of partial safety factors, and chapter 4.6 for the application of specific safety factors;

  IMPORTANT: the option "qs_{nails} from" is automatically the same for all the nails of a project.

- *Rsc calculated from qs:* this checkbox indicates whether the Rsc value (pull-out resistance per meter length of nail)

  - o must be calculated based on the equivalent radius (drilling or grouting radius in m, used for the calculation of the grout perimeter) and qs_{nails} (defined for each soil layer, refer to chapter 4.6): Figure 41 (this first choice is the default option),

  - o or is directly input by the user (unit of force/m along nail): Figure 43.
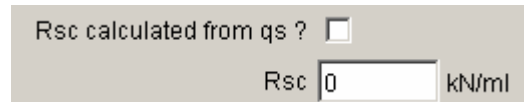
Figure 43: reinforcement dialogue box: tab 1(nails)
when checkbox "Rsc calculated from qs" is deactivated

- *Calculation mode (traction/shear rule)*: 3 options are available

  - o T_{cal}, C_{imp}: calculated traction, imposed shear;

  - o T_{nul}, C_{cal}: zero traction, calculated shear;

  - o T_{cal}, C_{cal}: calculated traction and shear.

  According to the selected option, the requested parameters are different.

  $\Rightarrow$ If reinforcement mode is (T_{cal}, C_{imp}) (Figure 41), it is necessary to provide the following data:

  - o *Shear force varying (along nail):* by default, shear force is imposed with only one value (checkbox deactivated). But it is possible to define several values of shear force along the nail (checkbox activated): usually, these values have been calculated first with a pile design software.

    - ▪ *Rcis (if checkbox "Shear force varying (along nail)" is deactivated)*: single value of the nail shear force (force unit);

    - ▪ *Variable shear wizard* (if checkbox "Shear force varying (along nail)" is activated, Figure 44). In this case, refer to the end of this chapter for the definition of the variable shear force;
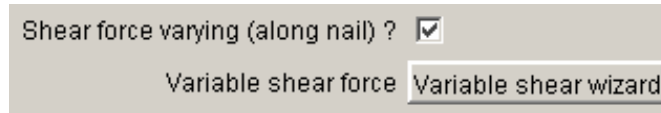
Figure 44: reinforcement dialogue box: tab 1 (nails)
when checkbox "Shear force varying…" is activated

- o *Traction calculation*: 2 options are available:
  - ▪ "external": pull-out resistance is calculated on the part of the reinforcement which is outside the failure surface;
  - ▪ "internal/external": pull-out resistance is calculated as the minimal traction between the external and internal parts of the reinforcement with respect to the failure surface;

⇒ If reinforcement mode is (T_{nul}, C_{cal}) (Figure 45), it is necessary to provide the following data:
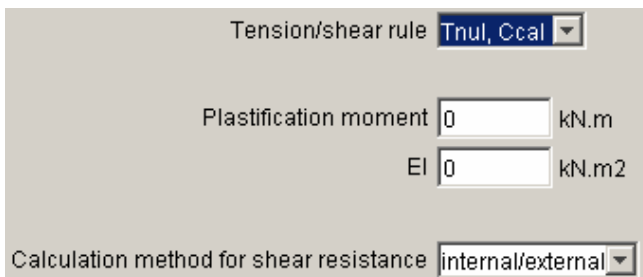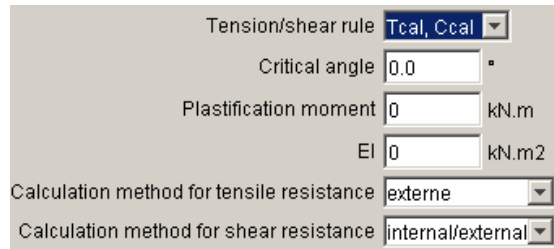
Figure 45: reinforcement dialogue box: tab 1 (nails)
with calculation mode (T_{nul}, C_{cal})

- o *Plastification moment*: plastification moment for the reinforcement (pure bending), calculated from the usual equation $M = \sigma_e . \dfrac{I}{v}$ (at the limit of the elastic domain, which is a simplification);
- o *EI* (unit consistent with kN.m2): reinforcement rigidity. It is possible to take into account the addition of the steel rigidity (bars, tube, etc) and of the grout or concrete rigidity*;*
- o *Shear calculation:* 2 options are available:
  - ▪ "external": shear force is calculated on the external part of the reinforcement with respect to the failure surface;
  - ▪ "internal/external": shear force is calculated as the minimal shear force between the external and internal parts of the reinforcement with respect to the failure surface;

Note: in the case (T_{nul}, C_{cal}), TR and Rsc are not requested (corresponding cells are hidden). But the equivalent radius should be defined.

⇒ If reinforcement mode is (T_{cal}, C_{cal}) (Figure 46), it is necessary to provide the following data:

Figure 46:reinforcement dialogue box: tab 1(nails)
with calculation mode (T_{cal}, C_{cal})

o *Critical angle* (for soil/nail interaction): it is advised to take for example 5°. This angle makes it possible to determine the couple traction/shear force that will be taken into account in the principle of maximum work (please refer to the technical manual to learn more about reinforcements);

o *Plastification moment*: plastification moment for the reinforcement (pure bending), calculated from the usual equation $M = \sigma_e . \dfrac{I}{v}$ (at the limit of the elastic domain, which is a simplification);

o *EI* (unit consistent with kN.m2): reinforcement rigidity. It is possible to take into account the addition of the steel rigidity (bars, tube, etc) and of the grout or concrete rigidity*;*

o Traction calculation: this choice can take 2 values:

  ▪ "external": pull-out resistance is calculated on the part of the reinforcement which is outside the failure surface;

  ▪ "internal/external": pull-out resistance is calculated as the minimal traction between the external and internal parts of the reinforcement with respect to the failure surface;

o *Shear calculation:* 2 options are available:

  ▪ "external": shear force is calculated on the external part of the reinforcement with respect to the failure surface;

  ▪ "internal/external": shear force is calculated as the minimal shear force between the external and internal parts of the reinforcement with respect to the failure surface;

Specific option: definition of varying shear force (along nail)

If calculation mode is (T_{cal}, C_{imp}), and the checkbox "Shear force varying (along nail)" is activated, the button" Variable shear wizard" is displayed (Figure 44). When selecting it, a new dialogue box is opened (Figure 47):
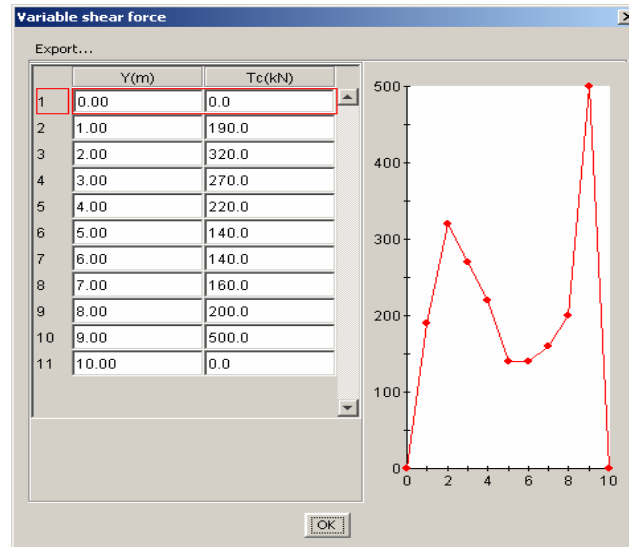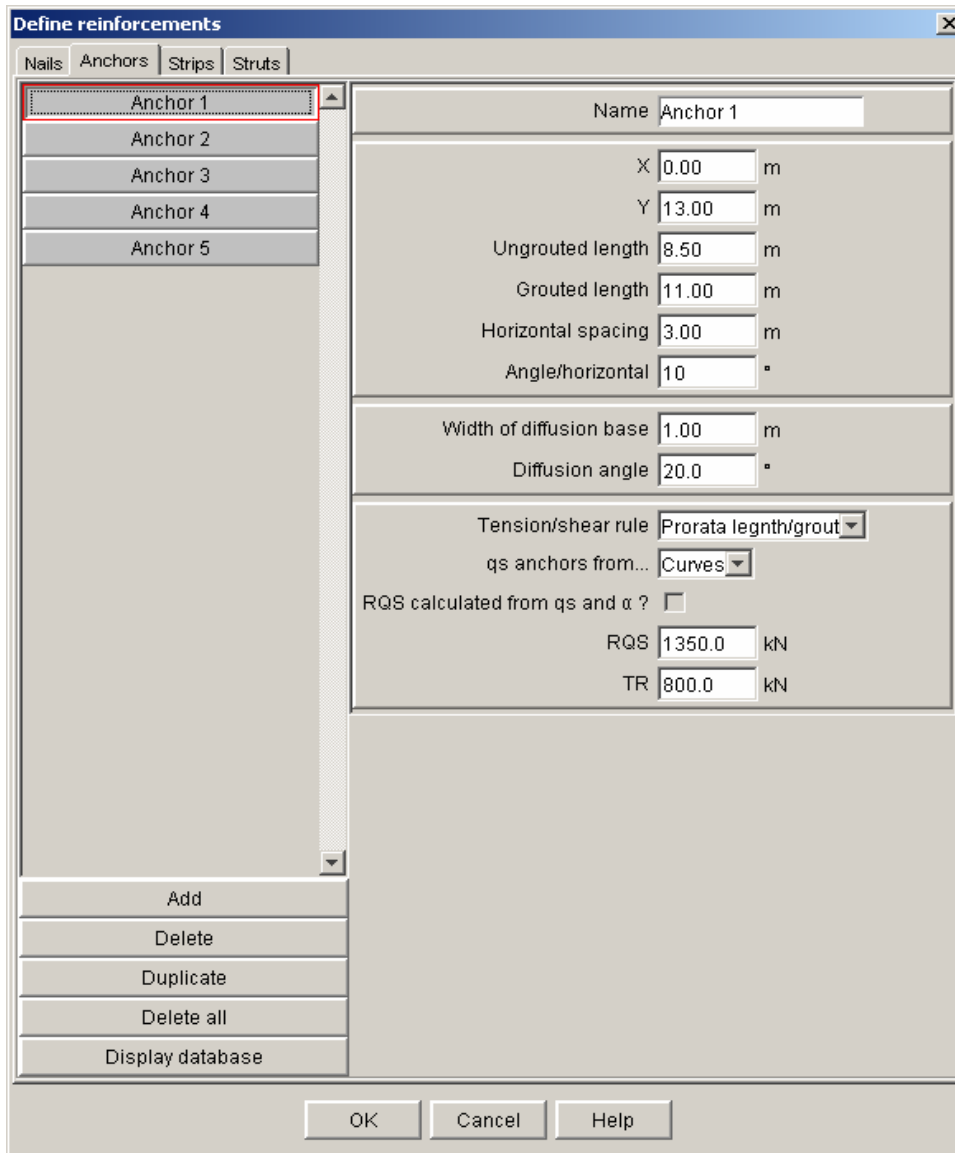
Figure 47: reinforcement dialogue box: tab 1 (nails),
definition of variable shear force

A table with 11 lines should then be filled in. The first column corresponds to the linear coordinate along the nail (starting at the reinforcement head), and the second to the value of the shear strength for each of these coordinates. The table lines are automatically ordered by increasing coordinate.

4.5.3.2. Anchors

Figure 48: reinforcements dialogue box: tab 2 (anchors)

The data displayed for each anchor are the following:

- *Name*: by default, Talren 4 names the reinforcements "Anchor i", but these names can be changed (minimum 10 characters).
- *X and Y*: coordinates (m) of the anchor head;
- *Ungrouted length* (m);
- *Grouted length* (m);
- *Horizontal spacing* (m);
- *Angle/horizontal* (°): the default value is zero. It is positive in the case illustrated on Figure 39 (trigonometrical direction);
- *Diffusion width* (m) *and diffusion angle* (°): please refer to the comments about nails (section 4.5.3.1);

- *Tension/shear rule:* 2 options are possible (illustrated on Figure 49):
  - o Proportional (prorata length/grout): the pull-out resistance is calculated using the ratio of the grouted length outside the failure surface compared to the total grouted length;
  - o Non proportional: the total pull-out resistance is taken into account if the failure surface intersects the anchor before the fictive anchoring point. On the contrary, the anchor is not taken into account at all if the failure surface intersects the anchor after the fictive anchoring point. The fictive anchoring point is at mid-grout length.

<u>Prorata length/grout (proportional)</u>

$$T_{cal} = \min\left(\frac{T_R}{\Gamma_{aanc}}, \frac{RQS \times l}{\Gamma_{qsanc} \times L} \right)$$

<u>Non proportional</u>

$T_{cal} = 0$

$$T_{cal} = \min\left(\frac{T_R}{\Gamma_{aanc}}, \frac{RQS}{\Gamma_{qsanc}} \right)$$

Fictive anchoring point
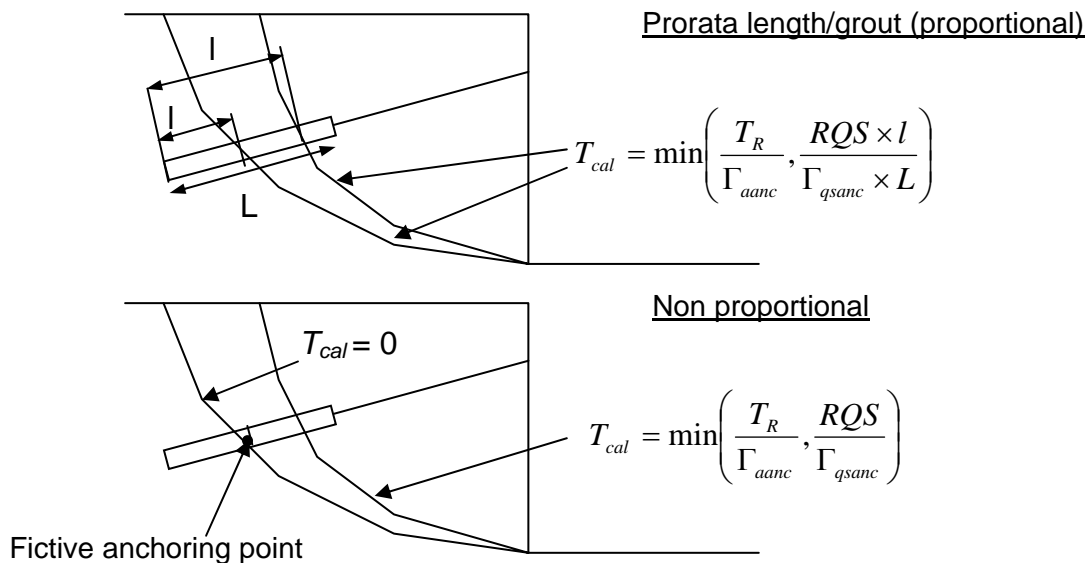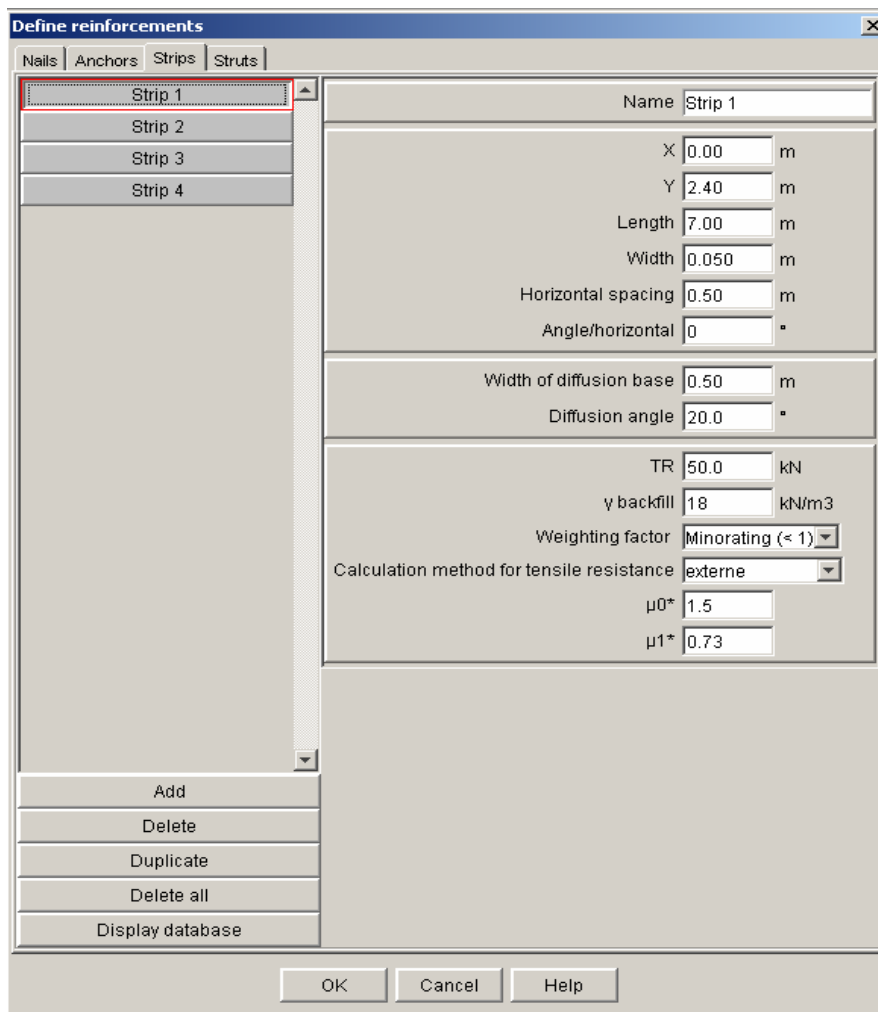
Figure 49: reinforcements dialogue box: tab 2 (anchors): calculation options

- *$qs_{anchors}$ from:* 2 possible options: curves (default choice) and tests. This choice decides the value of the safety factor applied to the anchor pull-out resistance. Also refer to section 4.7.2.3 for the description of the partial safety factors, and section 4.6 for the application of specific partial factors;

  IMPORTANT: the choice of "$qs_{anchors}$ from" is automatically the same for all the anchors of a project.

- *RQS calculated with qs and α:* this option is not available in this version of TALREN 4. The checkbox is always deactivated. One must therefore input the RQS value (force): total pull-out resistance of an anchor (equal to the unit skin friction (soil/anchor) by meter multiplied by the total grout length; be careful if the grout goes through several soil layers with different unit skin friction values; the α factor should also be taken into account, please refer TA 95 recommendations);

- *TR*: tensile strength of the anchor (force), generally calculated as the steel section multiplied by the steel yield stress.

4.5.3.3. Strips

Figure 50: reinforcements dialogue box: tab 3 (strips): strip 1

The data displayed for each strip are the following:

- *Name:* by default, Talren 4 names the strips "Strip i". These names can be changed (10 characters maximum);
- *X and Y:* (m) coordinates of the strip head;
- *Length* (m) *and width* (m): total length and width of each strip.
- *Horizontal spacing* (m);
- *Angle/horizontal* (°): the default value is zero (horizontal strips, as illustrated on Figure 39);
- *Diffusion width* (m) and *diffusion angle* (°): refer to comments given for nails (section 4.5.3.1).
- *TR*: tensile strength of a strip (force);
- $\gamma_{backfill}$: unit weight of the backfill used for the reinforcement;
- *Weighting factor :* it is a matter of defining whether the unit weight of the backfill will be considered as a favourable or unfavourable action (2 possible choices): this will change the partial safety factor taken into account for the backfill weight. By default, the weighting factor is minorating.

- *Calculation method for traction:* for strips, one can consider 2 failure types because of a lack of adherence: internal and external. Indeed, strip heads are not always anchored, and the failure can then occur on either side of the failure surface: Talren can (Figure 51):
  - ○ compare l_{int} and l_{ext} lengths and calculate the minimum $min(l_{int}, l_{ext})$: this corresponds to the "internal/external" choice,
  - ○ or consider only the external length (if the strip head is anchored, in the case of paraweb or reinforced earth for example): this corresponds to the "external" choice (default choice).
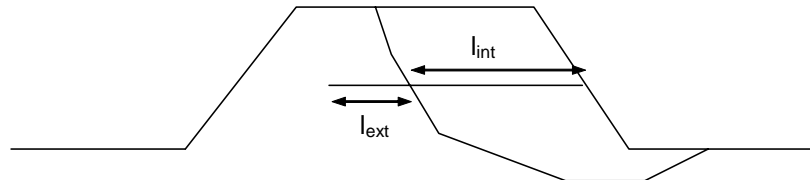
Figure 51: traction calculation

- μ_0^* and μ_1^*: unit soil/strip friction at groundlevel and at depth (> 6 m);

  The values of the μ_0^* and μ_1^* parameters are given in the reinforcement specific documentations. They are also defined in the standards and recommendations concerning this type of reinforcements (as for example the XP P 94-220-0 french standard, or the french document "Recommandations pour l'emploi des géotextiles dans le renforcement des ouvrages en terre").
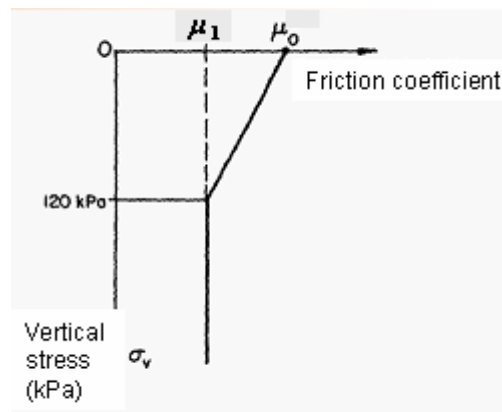
Figure 52: soil/strip interaction

IMPORTANT:

- For geotextiles forming continuous levels, TR should be defined as a force/ml, with a horizontal spacing of 1 m, a strip width of 1 m, and for example $L_B = 0.1$ m if the geotextile is simply placed without any anchoring system;
- The parameters ($\gamma_{backfill}$, weighting factor, μ_0^* and μ_1^*) have the same values for all strips. It is therefore possible to modify them for strip 1, but not for the other strips (which automatically take the same values for these parameters as strip 1): the corresponding edit cells are therefore disabled for the strips other than strip 1 (Figure 53).

Figure 53: reinforcements dialogue box: tab 3 (strips): strip 2 and more

4.5.3.4. Struts
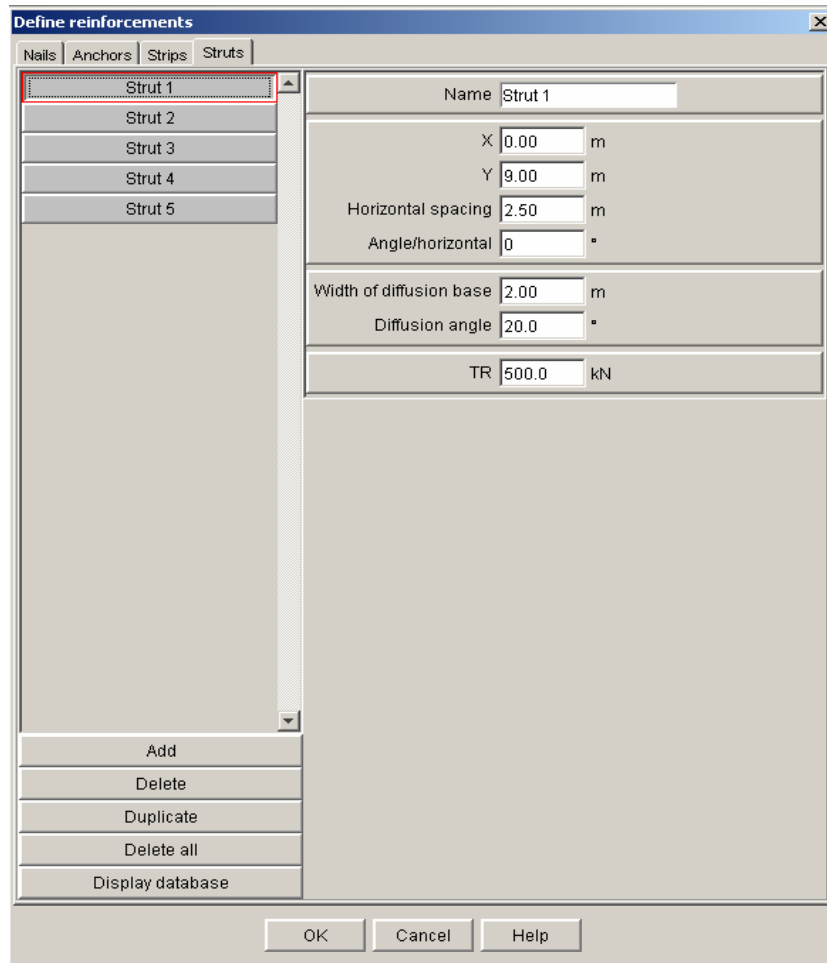
Figure 54: reinforcements dialogue box: tab 4 (struts)

The data displayed for each strut are the following:

- *Name* by default, Talren 4 names these reinforcements "Strut i". These names can be changed (10 characters maximum);
- *X and Y:* (m) coordinates of strut head;
- *Horizontal spacing* (m);
- *Angle/horizontal* (°): default value is zero (horizontal struts, as illustrated on Figure 39); it is positive in trigonometrical direction;
- *Diffusion width* (m) *and diffusion angle* (°): refer to comments given for nails (section 4.5.3.1);
- *TR*: ultimate intrinsic compression strength (force). It is usually calculated as the strut section multiplied by the yield stress.

4.6. Soil characteristics

The definition of soil properties is only possible through a dialogue box, accessible either with the "Project data" menu, "Soil characteristics" option, or by the [icon] button of the toolbar.

This dialogue box enables to view and edit soil properties.

The left list enables to view the project layers. When cliking on a soil layer in the list, its properties are displayed.

The "Add", "Delete", "Duplicate" and "Delete all" buttons enable to add a new soil layer, to delete the selected layer, to duplicate the selected layer (creation of a new layer with the same properties), or to delete all project layers. Finally, the last button, "Display database", enables to display the soils database.
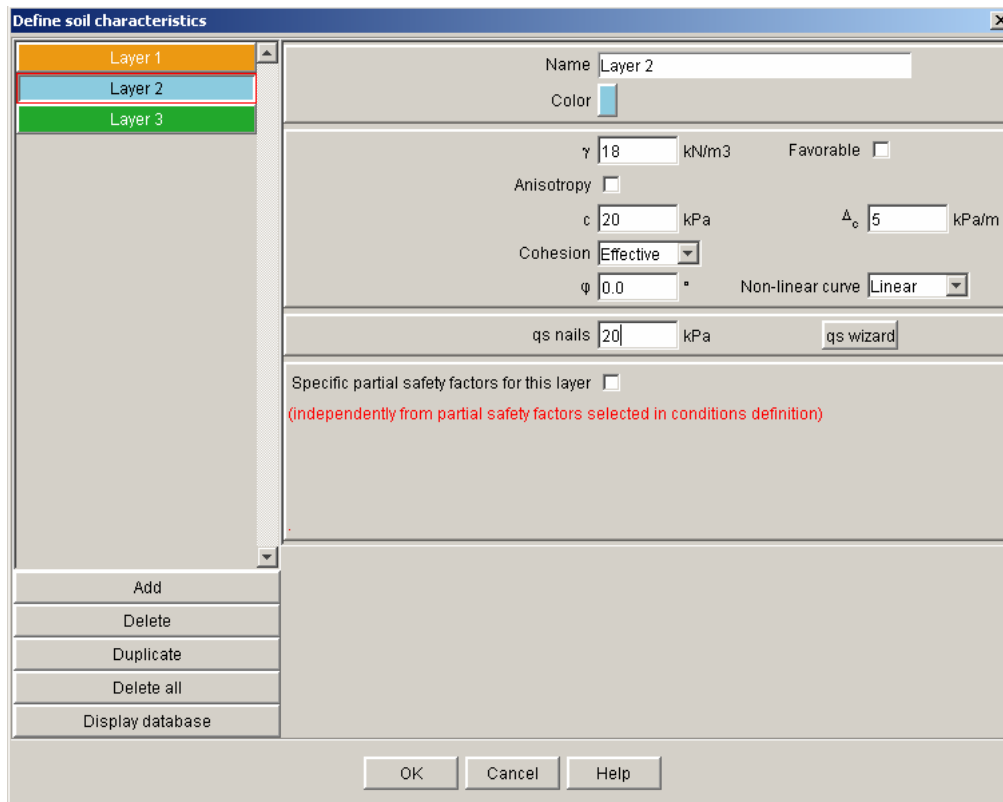
Figure 55: soil characteristics dialogue box

The data displayed for each soil layer are the following:

- *Name*: by default, Talren 4 names each layer "Layer i", but these names can be changed (20 characters maximum);

- *Colour or pattern:* it can be selected in the colour palette or in the patterns list, depending on the display option chosen for the soil layers painting (section 3.3.2.3). To access the colour palette, click on the coloured zone next to the "Colour" label.

- *Unit weight γ and favourable checkbox:* it is the unit weight of the soil layer. The "favourable" checkbox determines, depending on whether it is activated or not, the partial load factor that will be applied to the unit weight (also refer to section 4.7.2.3 for a description of the partial safety factors);

- *Anisotropy checkbox*: it is deactivated by default (no anisotropy). To take into account a cohesion anisotropy, this checkbox should be activated, and additional data should be defined (case 1 at the end of this section);

- *c, Δc and cohesion type:* respectively, cohesion at the top of the layer, cohesion variation with depth (per meter), and choice between effective or undrained cohesion. This last choice determines the partial safety factor that will be applied to the cohesion value (also refer to chapter 4.7.2.3 for a description of the partial safety factors).

    Cohesion variation with depth (Δc) is taken into account according to Figure 56: the layer thickness in each vertical slice (and not the depth compared to a horizontal level) is used for the calculation of the cohesion at a given depth.

    If Δc is zero, cohesion is constant for the whole soil layer.
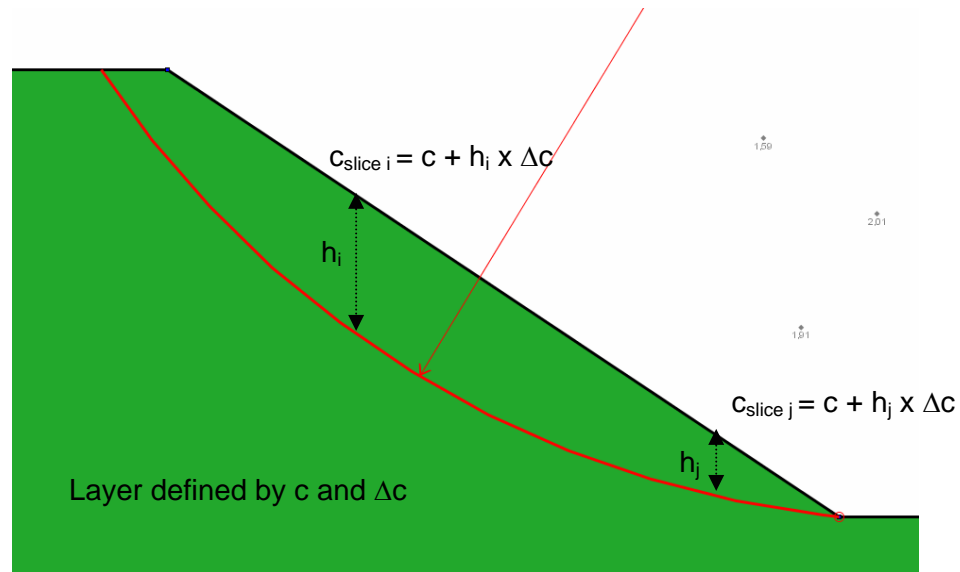
Figure 56: Δc (cohesion variation with depth)

- *"Non-linear curve" choice:* the default option is "Linear", which corresponds to the standard case. To define a non-linear shear strength curve, please refer to case 2 at the end of this section;

- φ *(friction angle)* in °;

- qs_{nails} *and qs wizard*: this line is visible only if at least one nail crosses the selected layer. qs_{nails} is the unit skin friction (soil/nails) for the layer. The qs wizard can help you to define the qs_{nails} value (also refer to section 4.7.1);

- *pl and K_sB* (pressure unit for the both parameters). These lines are only visible if at least one nail with the (T_{nul}, C_{cal}) or (T_{cal}, C_{cal}) calculation mode crosses the selected layer.

  K_sB can be calculated as a function of pressuremeter data E_m and α.

  o For $B \leq B_0 = 0,6$ m: $\dfrac{1}{K_s B} = \dfrac{1}{E_M} \dfrac{4(2,65)^{\alpha} + 3\alpha}{18}$

  o For $B \geq B_0 = 0,6$ m: $\dfrac{1}{K_s B} = \dfrac{2}{9 E_M} \dfrac{B_0}{B} \left(\dfrac{B}{B_0} 2,65 \right)^{\alpha} + \dfrac{\alpha}{6 E_M}$

Figure 57: soil characteristics dialogue box: pl and KsB

- *Specific partial safety factors*: these specific factors that can be activated with the corresponding checkbox and defined for each layer (Figure 58). They replace, for each layer where they are defined and activated, the corresponding values of the partial safety factors chosen in the situations.
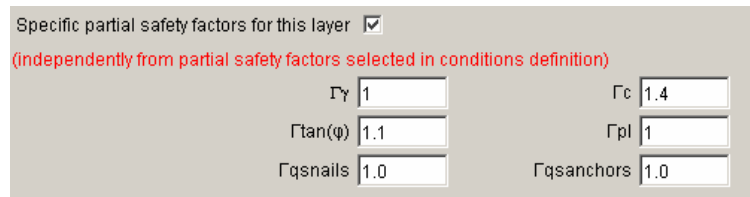
Figure 58: soil characteristic dialogue box: specific partial safety factors

This option should be used to apply different partial safety factors for various soil layers.

Particularly, this option is activated by default when loading Talren 97 files (.tal). Indeed, it was possible with Talren 97 to define different safety factors for each soil layer, and the activation of the option "specific partial safety factors" for each layer guarantees therefore that .tal files will be loaded with the correct values (section 7.1.1.5).

IMPORTANT: if specific partial safety factors are activated for a given layer:

- The specific load factor on the unit weight (Γ_γ) will be applied for the layer whether the weight action is indicated as favourable or unfavourable;

- In the same way, the specific partial safety factor on the cohesion (Γ_c) will be applied for the layer whether the cohesion is effective or undrained.

- Finally, the specific safety factors $\Gamma_{qsnails}$ and $\Gamma_{qsanchors}$ will be applied if relevant to the pull-out resistance of the nails (Rsc) or of the anchors (RQS), whether those are indicated as determined from curves or tests (sections 4.5.3.1 and 4.5.3.2).

When one or several layers have been created in the dialogue box and appear in the left list, they can be assigned to soil clusters by drag & drop: click on a soil layer in the list with the left mouse button; then without releasing the button, move the mouse onto a soil cluster; finally release the left mouse button: the selected layer is assigned to the soil cluster and appears in the same colour (or pattern).

It is also possible to assign soil layers to soil clusters after having closed the soil characteristics dialogue box: in the Selection mode, click with the right mouse button on a soil cluster, then choose "Edit" in the pop up menu. The window of Figure 59 is displayed, and it is possible to select which layer you want to assign to the soil cluster. Talren 4 proposes the list of available soil layers for the current project.
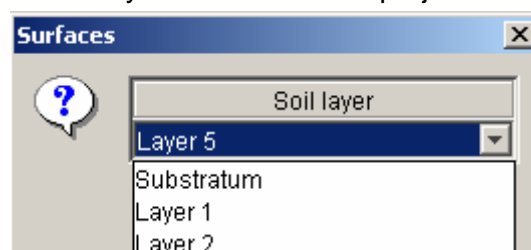
Figure 59: properties of a soil cluster: choice of a soil characteristics set
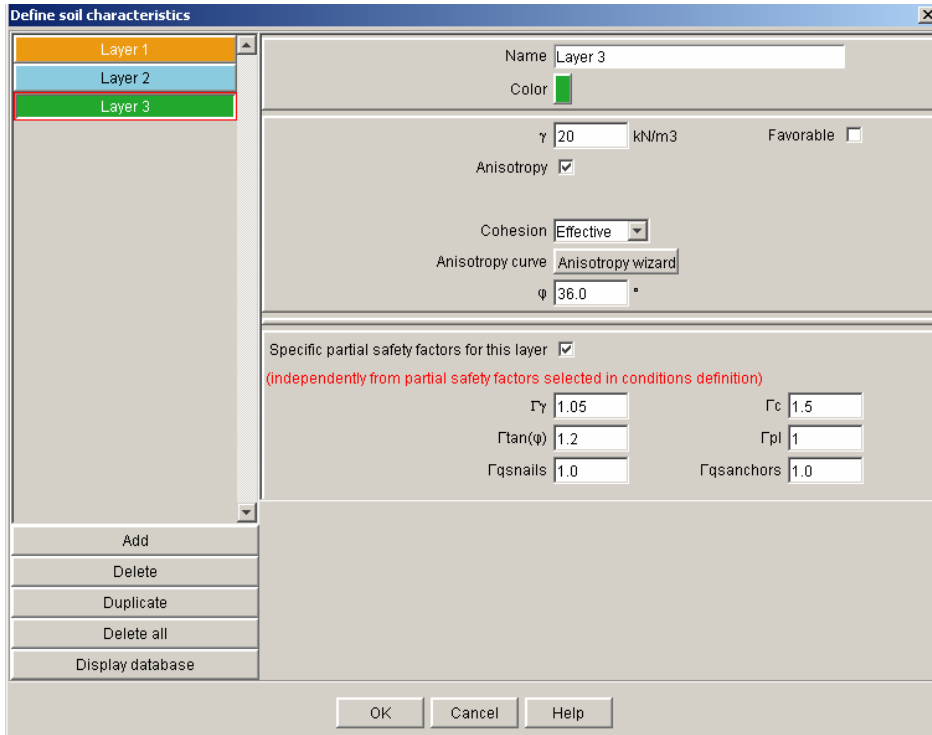
IMPORTANT

The "Substratum" set in the list is created automatically for each project, and if it is assigned to some soil clusters when running the calculation, Talren 4 will automatically consider that failure surfaces shouldn't enter these substratum soil clusters.

This "Substratum" set can be selected for one or several soil cluster(s), but its properties cannot be changed (anyway, failure surfaces cannot go through substratum clusters, and therefore there is no need for substratum properties). "Substratum" is the default set assigned to all new clusters (section 4.3.1.1)

Specific case 1: cohesion anisotropy

To define a cohesion anisotropy, the "Anisotropy" checkbox should be activated. The requested data are then different (Figure 60).

Figure 60: soil characteristics dialogue box: cohesion anisotropy

The cohesion type is still asked for, but a new button is displayed instead of the cohesion editing cell: "Anisotropy wizard".

Click on this button: a new dialogue box is then opened (Figure 61).

The 8 lines should be filled in. The first column corresponds to angles compared to the horizontal (°) (positive values in the trigonometrical direction), and the second to the cohesion value for each angle.

The table lines are automatically ordered by increasing angle values.
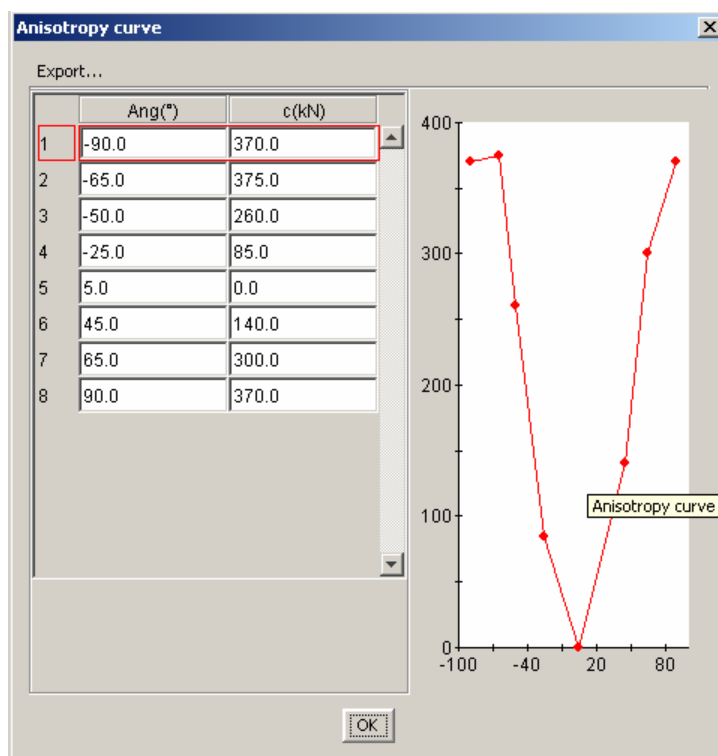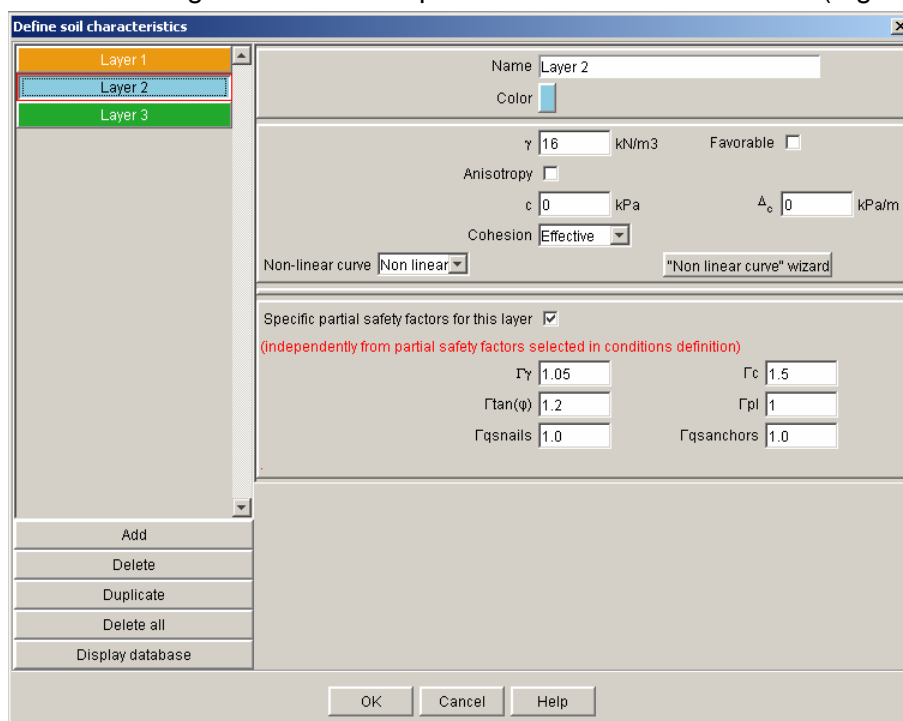
Figure 61: cohesion anisotropy

Specific case 2: non-linear shear strength curve

To define a non-linear shear strength curve, it is necessary to activate the "Non-linear" choice for the shear strength curve. The requested data are then modified (Figure 62).

Figure 62: soil characteristics dialogue box: choice of a non-linear shear strength curve

A new button appears to replace instead of the editing cell for the friction angle: "nonlinear curve wizard". Click on this button: a new specific dialogue box is opened (Figure 63).

Between 2 and 5 lines should be filled in the table (the "Add" button is visible as long as the table includes less than 5 lines). Each line corresponds to a (σ, τ) couple of the shear strength curve.

The lines in the table are automatically ordered by increasing σ values.
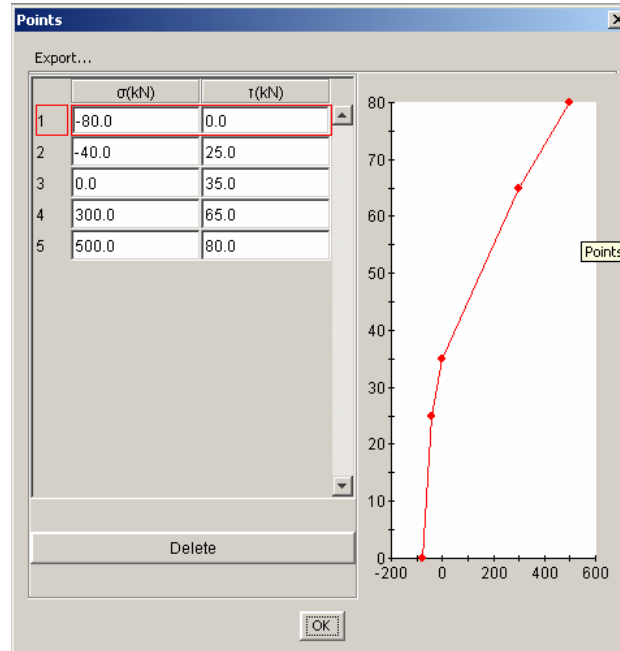
Figure 63: non-linear shear strength curve

4.7. WIZARDS AND DATABASES

These various wizards, as indicated in the following sub-sections, use or enable to save sets of predefined parameters.

4.7.1. Coefficient qs for nails

The qs_{nails} wizard allows to determine the values of the unit skin friction (qs_{nails}) for a given layer, depending on this layer properties. This qs_{nails} value enables to describe the soil/nail interaction behaviour (friction).

The qs_{nails} value should be defined for each layer crossed by one or more nails.

This wizard is accessible by a button from the soil characteristics dialogue box (next to the "qs nails" cell).
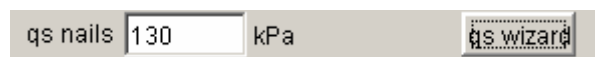
The user can not add new values to this wizard. The values provided in the wizard are issued from the Clouterre recommendations (as a function of the pressuremeter data pl). They are displayed as curves (Figure 64)
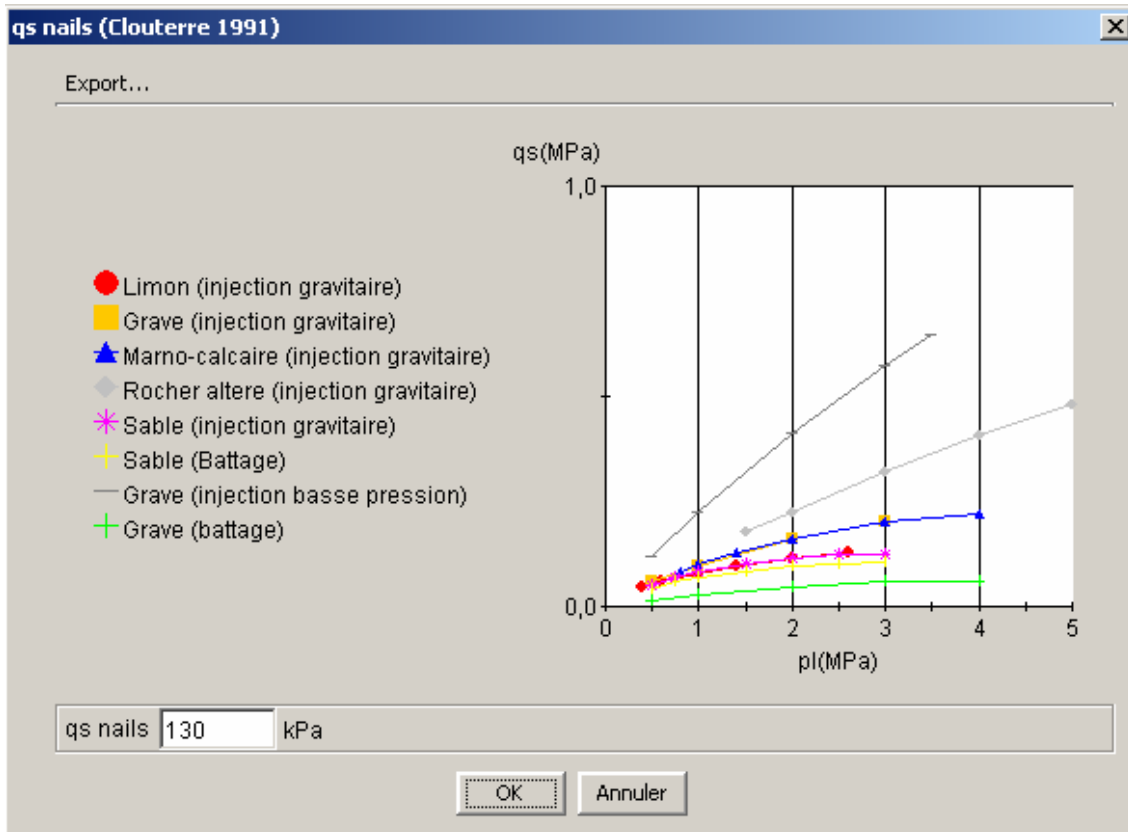
Figure 64: qs depending on pl
(extract from Clouterre recommendations 1991)

The user can click on a curve, and the software displays the corresponding value of qs .

- If the user validates the value (button "OK"), this value is converted into the current unit system, and is copied to the qs_{nails} cell of the current soil layer;
- If the user does not validate the value (button "Cancel"), the selected value is not taken into account in the current soil layer.

The use of the wizard is not compulsory: qs_{nails} values can also be input manually.

4.7.2. Database for partial safety factors

4.7.2.1. Principles

This wizard allows to define and save partial safety factors sets.

These partial factors are to be applied on some Talren data, and are imposed or proposed by some standards and recommendations.

This wizard is provided with several predefined sets (Figure 76), and the user can fill it with additional sets depending on his needs or on the standards evolutions. These predefined sets are those proposed in the French Clouterre recommendations, and in the french standards (XP P 94-240 and XP P 94-220-0). Traditional sets are also included.

Each standard or recommendations document usuallly provides several sets of partial safety factors corresponding to: fundamental or accidental load combinations, standard or sensitive sites, temporary or permanent situations. These distinctions are indicated in the title of each predefined set.

For more details about the predefined sets, please refer to chapter 6 of the technical manual (part C of this manual).

Name
Clouterre fundamental/standard
Clouterre fundamental/sensitive
Clouterre accidental/standard
Clouterre accidental/sensitive
XP P 94-240 fundamental 1-2a
XP P 94-240 fundamental 2b
XP P 94-240 accidental 1-2a
XP P 94-240 accidental 2b
XP P 94-220 fundamental/standard
XP P 94-220 fundamental/sensitive
XP P 94-220 accidental/standard
XP P 94-220 accidental/sensitive
Traditional/Provisional
Traditional/Permanent

Figure 65: predefined sets of partial safety factors

4.7.2.2. Handling of the sets of partial safety factors

This database is accessible from several moments in the project analysis. In a general way, in the Data mode, it is possible to select/view/edit/add sets of partial factors, whereas in the Stages/calculations mode, it is only possible to select and view them (no edition allowed):

- From the General settings dialogue box (section 4.2.), the user selects which safety factors set will be used as a default choice for calculations; among the list of sets in the local database (this local list is empty at first when creating a new Talren file).

Default calculation method for the project	Bishop ▼	
Default safety factors	Clouterre fondamental/courant ▼	Safety factors wizard

A button enables to display the wizard (in a separate window, Figure 66).

The list displayed on the right corresponds to the "general" database, that is shared between all Talren projects on a computer. The list on the left corresponds to the "local" (project) database, and includes the sets that are used for the current project. It is possible to transfer sets from a database to the other, in both 2 directions (provided no other set with the same name already exists in the destination list).

For example, on Figure 77, to transfer a set of safety factors from the global database to the local database, select the set of factors in the global list on the right (1) and click on the button "Transfer to current project" (2). The set that was last transferred is selected by default (framed in red in the left list. Fill in the missing values for the selected set if necessary (3). Then validate with "OK" (4).

In addition, the [Add], [Remove], and [Duplicate] buttons allow to add new sets of partial factors, to remove the selected set, to duplicate the selected set (creation of a new set with the same values), all these operations concerning the local database only. Finally, a last button, [Hide database], makes it possible to hide the right part of the screen (the global database).
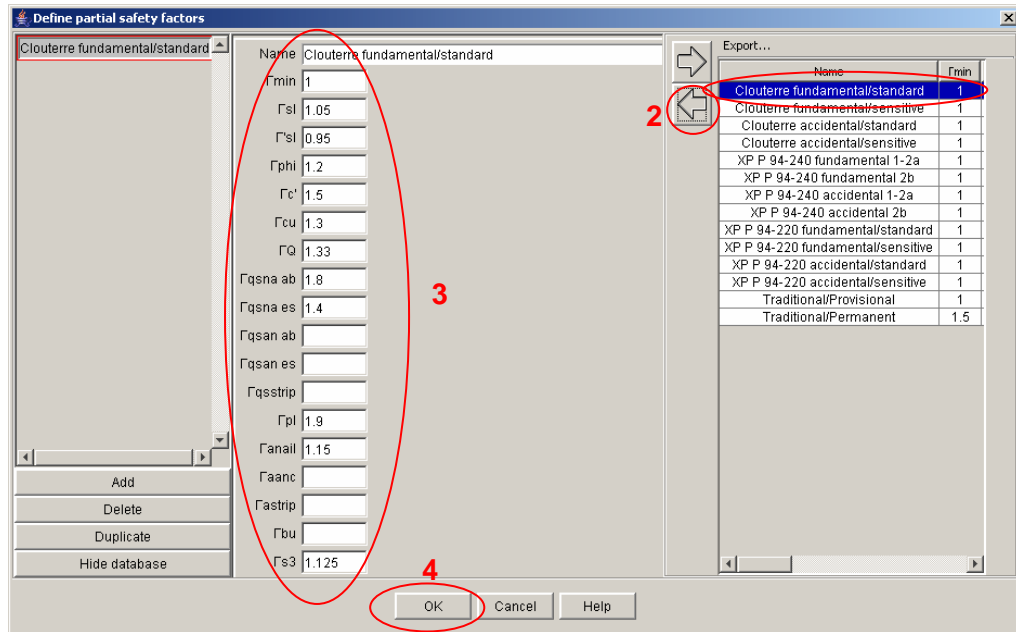
Figure 66: partial safety factors wizard

IMPORTANT: in the screen above, the partial safety factors relating to anchors, strips and struts are missing, because these values are not defined in the Clouterre recommendations. The other sets of partial safety factors are not complete either (the values defined in the various standards and recommendations alone are provided). To be able to validate the dialogue box, it is necessary to complete each set. Once completed, the sets can be transferred again to the global database (after changing their name), in order to share them with other projects.

Before returning to the project general settings, it is necessary to click in the left list on the set that that should be selected as the default set for the project, then close the dialogue box with "OK".

- With the option "Partial safety factors" from the menu "Options": this option directly activates the wizard, and allows the same operations as described previously (viewing partial safety factors, and if necessary adding new sets in Data mode).

- From the dialogue box for conditions properties (section 5.2.5.). It is possible to select the set of partial safety factors that will be used for the calculation of the current situation (if different from the project default safety factors) among the list in the local database. As above, a button makes it possible for the user to activate the wizard: he can then view the predefined sets, but can not add new sets (it is only possible in the "Project data" mode). Before returning to the current conditions properties, select a set in the left list: this set will be then selected for the properties of the current situation.

4.7.2.3. Detail of partial safety factors

Each set of partial factors includes the following values:

- Γ_{min}:   minimum safety factor to reach as a result to ensure stability (this value is only provided for information, it is not used in calculations);

- Γ_{s1}:   load factor for the soil unit weight, if this one is considered as unfavourable (section 4.6.);

- Γ'_{s1}:   load factor for the soil unit weight, if this one is considered as favourable (section 4.6.);

- Γ_{phi}:   partial safety factor for tan (ϕ);

- Γ_c: partial safety factor for effective cohesion (section 4.6.);

- Γ_{cu}:   partial safety factor for undrained cohesion (section 4.6.);

- Γ_Q: load factor applied for loads;

- $\Gamma_{qsna\ ab}$:   partial safety factor for qs_{nails} from curves (or directly for the value of the nails pull-out resistance when Rsc value is provided directly, section 4.5.3.1.);

- $\Gamma_{qsna\ es}$:   partial safety factor for qs_{nails} from tests (or directly for the value of the nails pull-out resistance when Rsc value is provided directly, section 4.5.3.1.);

- $\Gamma_{qsan\ ab}$:   partial safety factor for $qs_{anchors}$ from curves (or directly for the value of the anchors pull-out resistance when RQS value is provided directly, section 4.5.3.2.);

- $\Gamma_{qsan\ es}$:   partial safety factor for $qs_{anchors}$ from tests (or directly for the value of the anchors pull-out resistance when RQS value is provided directly, section 4.5.3.2.);

- $\Gamma_{qsstrip}$:   partial safety factor for the soil/strip interaction parameters μ_0^* and μ_1^*;

- Γ_{pl}:   partial safety factor for the pl values;

- Γ_{anail}:   partial safety factor for the nails tensile strength;

- $\Gamma_{aanc:}$ partial safety factor for the anchors tensile strength;

- Γ_{astrip}:   partial safety factor for the strips tensile strength;

- Γ_{bu}:   partial safety factor for the struts compression strength;

- Γ_{s3}:   partial safety factor for the calculation method.

Please refer to the technical manual (chapter C of this manual), section 6, for additional information about safety factors in Talren 4.

4.7.3. Soils database

This wizard allows to select sets of predefined soil properties and to import them in the current project.

This wizard is provided "empty". It can be completed progressively, in order to share soil properties between several Talren projects (but on the same computer).

This wizard is accessible:

- From the soil characteristics dialogue box (section 4.6.), when clicking on the button "Display database" (then replaced with the button "Hide database"): a new list appears on the right-hand side of the dialogue box (Figure 67). This right part is the "general" soils database.

    The usual list on the left can be considered as the local (project) soils database. It includes the layers defined for the current project.

    It is possible to transfer soil properties from a database to the other, in both directions, provided that no set with the same name already exists in the destination list.

    For example, to transfer a set of soil properties from the global database to the local database: select the set you want in the global list on the right (1), and click on the button "Transfer to the project" (2). The set of characteristics is then added to the left list, and can be viewed, edited, validated, etc (be careful: the layer colour is usually modified after a transfer from a database to the other).
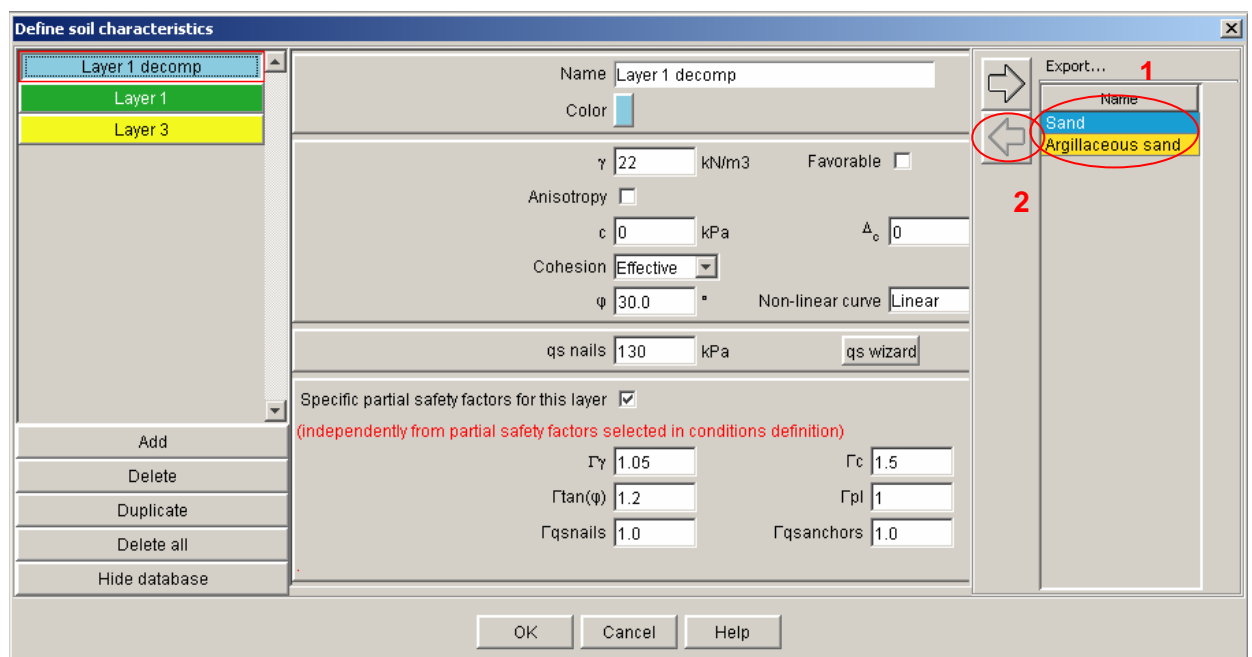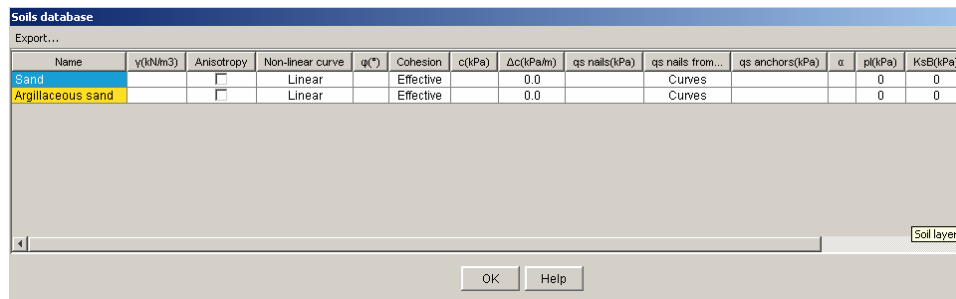
Figure 67: soil characteristics dialogue box: general soils database

- With the option "Soils database" from the "Options" menu: this option directly activates the wizard in a separate window (Figure 68), and makes it possible to view (only) the characteristics sets defined in the global database.

Figure 68: soils database

4.7.4. Reinforcements database

This wizard allows to select sets of predefined reinforcements and to import them in the current project.

This wizard is provided "empty". It can be completed progressively, in order to share reinforcements properties between several Talren projects (but on the same computer).

This wizard is accessible:

- From the reinforcements dialogue box, when clicking on the button "Display database" (then replaced with the button "Hide database"): a new list appears on the right side of the window (Figure 80). It is the general reinforcements database, that is shared between all projects on the same computer.
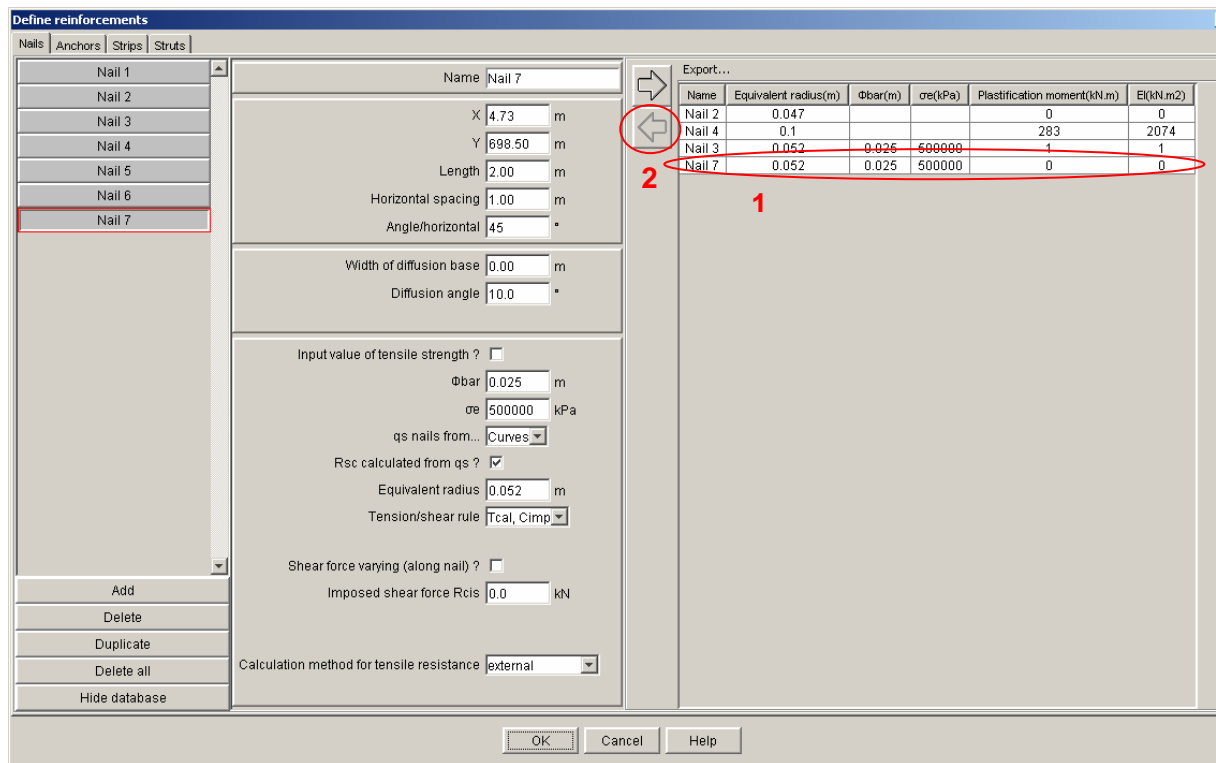
  The usual list on the left can be considered as the local (project) reinforcements database. It includes the reinforcements defined for the current project.

  It is possible to transfer reinforcements from a database to the other, in both directions, provided that no set with the same name already exists in the destination list.

  For example, to transfer a reinforcement from the global database to the local database: select the reinforcement you want in the global list on the right (1), and click on the button "Transfer to the project" (2). The reinforcement is then added to the left list, and can be viewed, edited, validated, etc.

  Note:
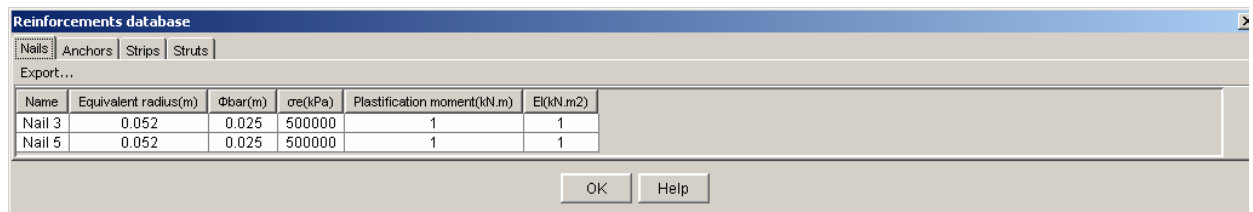
  o the material characteristics of each reinforcement alone are stored in the general database (the geometrical characteristics, for example, are not stored in the general database).

  o the reinforcements database is specific to each reinforcement type (there is a database for the nails, another for the anchors, etc), but it is activated in the same way for all the reinforcements.

Figure 69: reinforcements dialogue box: general reinforcements database

- with the "Reinforcements database" item of the "Options" menu: this option directly activates the wizard in a separate window (Figure 70), and enables to view (only) the reinforcements stored in the global database.

Figure 70: reinforcements database

5. "STAGES/CALCULATIONS" MODE

Also refer to section 2.1.2. for a general introduction about stages and conditions.
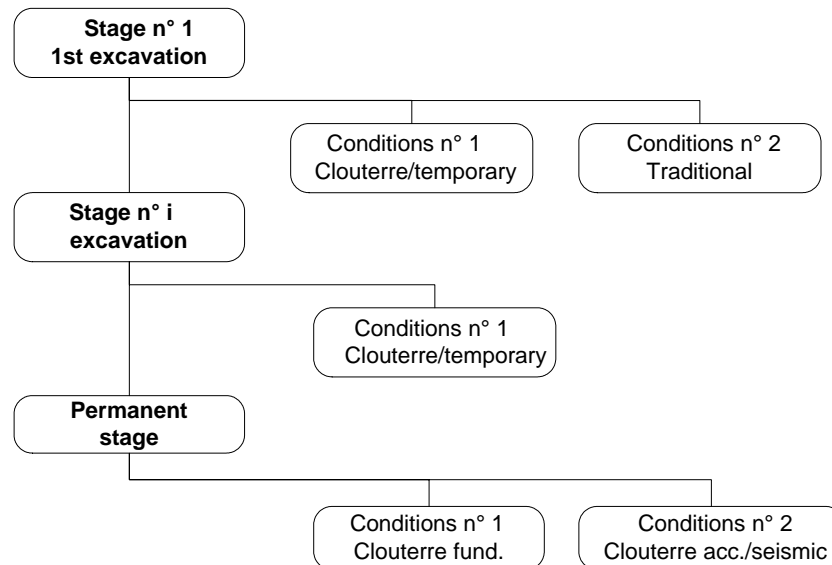
Figure 71: stages and conditions (situations)

Important

A calculation should be performed for each individual situation of each construction stage.

The calculation for each situation of each stage is independent from the calculations for the other situations and other stages. For example, the calculation result for situation m of stage n does not depend on the results of stage n-1 neither on the results of situation m-1 neither on any other result.

One should therefore run as many calculations for a project file as existing situations for all stages of the project file. On the other hand, at least one stage and one situation should be defined in order to run a calculation.

5.1. Stages

The construction stages enable to define within a single project file the various construction stages of a project for which stability should be checked. A construction stage corresponds to a cross-section of the project that should be analysed.

The changes from one stage to another correspond to a real physical evolution on the construction site.

The stages are numbered and the numbers define the succession of the stages.

Each stage is defined incrementally compared to the previous stage (or compared to the initial state of the project for the first stage). In other words, you only have to define for stage n what was modified compared to stage n-1: groundlevel change, activation of reinforcements or loadss, hydraulic conditions, etc.

Most possible actions for stages definition are accessible with the pop up menu (right-click on the stage label).

5.1.1. Principles

After completing the definition of the project data (data mode), you switch to the stages/calculation mode, and the first stage is automatically generated (one must define at

least one stage to be able to make a calculation). It is therefore not necessary to add or insert this first stage.

The user can then:

- Either keep to this single stage, which corresponds for example to the permanent stage of the project.
- Define (add/insert) many successive stages: the numbers of the phases are defined automatically by the software when creating each phase.

Table 2 provides an example of a succession of construction stages.

Stage n°	Settings associated to each stage	Summary of the elements taken into account for the display and calculation of this stage
	Project data	Initial groundlevel Nails n° 1, 2 et 3 Load n° 1 (no calculation allowed in data mode)
1	GL 1 (deactivation of soil clusters = excavation) Hydraulic conditions stage 1 Activation of nail level n° 1 Activation of load n° 1	GL 1 Hydraulic conditions 1 Nail n° 1 Load n° 1
2	GL 2 (deactivation of additional soil clusters = excavation) Hydraulic conditions stage 2 (*) Activation of the nail level n° 2	GL 2 Hydraulic conditions 2 Nails n° 1 et 2 Load n° 1
3	GL 3 (deactivation of additional soil clusters = excavation) Hydraulic conditions stage 3 (*) Activation of the nail level n° 3 Deactivation of load n° 1	GL 3 Hydraulic conditions 3 Nails n° 1, 2 and 3
4	GL 4 (deactivation of additional soil clusters = excavation) Hydraulic conditions stage 4 (*) Deactivation of nail n° 1	GL 4 Hydraulic conditions 4 Nails n° 2 and 3

(*) The hydraulic conditions for all stages can be identical or not.

Table 2: example of construction stages

5.1.2. Adding a new stage

The "Add a new stage" option of the "Stages and conditions" menu allows to create a new stage that is added as the last stage (no matter which stage is selected when adding the stage). This operation is also possible with a right click on an existing stage ("Add a stage" item in the pop up menu).

When adding a new stage (other than the first stage), it is generated to be identical to the stage immediately above in the list. The user can then define the changes compared to this preceding stage:

- Modification of the groundlevel (excavation, embankment, etc): activation or deactivation of soil clusters (section 5.1.4.1);
- Association of different soil properties (simulating soil substitution or soil improvement for example): the soil properties assigned with each soil cluster can be modified in a given stage. But all necessary soil properties must be available, i.e. must have been defined in the "Data" mode (section 5.1.4.2);
- Activation or deactivation of loads: distributed loads, linear loads/additional moments (section 5.1.4.3). Deactivation can of course only be selected for active loads;
- Activation or deactivation of reinforcements: nails, anchors, strips or struts (section 5.1.4.4). Deactivation can of course only be selected for active reinforcements;
- Change of hydraulic conditions (section 5.1.4.5);
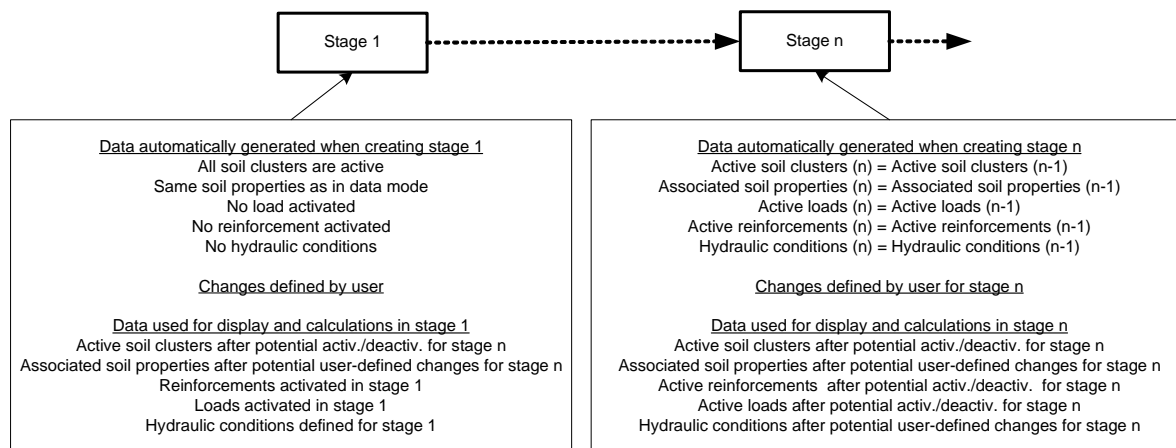
Figure 72: definition of successive stages

Figure 72 illustrates what happens when defining a stage:

- Hydraulic conditions are defined and stored independently for each stage, even if they are identical to those of the previous stage (it is the case by default);
- Association of soil properties with soil clusters is defined and stored independently for each stage, even if it is identical to the one of the previous stage (it is the case by default);
- Other elements (soil clusters, loads and reinforcements) are just activated or deactivated, but their properties are stored with general data.

It is possible to change the name of the new stage, either with the pop up menu (right click on the stage label, and "Rename" item from the pop up menu), or with a first click on the stage label to select it, and then a 2nd click to change this label. After changing the name, the number of the stage will automatically be added to the new name.

5.1.3. Inserting a new stage

The "Insert a new stage" item of the "Stages and conditions" menu enables to create a new stage and to insert it before the selected stage. This operation is also possible with a right click on the label of the stage before which the new stage should be inserted ("Insert a new stage" in the pop up menu).

The insertion of a stage at position n leads to an automatic shifting of all stages from the potition n, and of all data and operations defined for these stages: activation/deactivation of soil clusters, loads and of reinforcements, association of soil properties, hydraulic conditions.

The phase inserted at position n is generated to be identical to the stage before which it is inserted (the one which becomes number n+1).

The shift of the numbers of the stages following the inserted stage and the numbering of the new stage are made automatically.
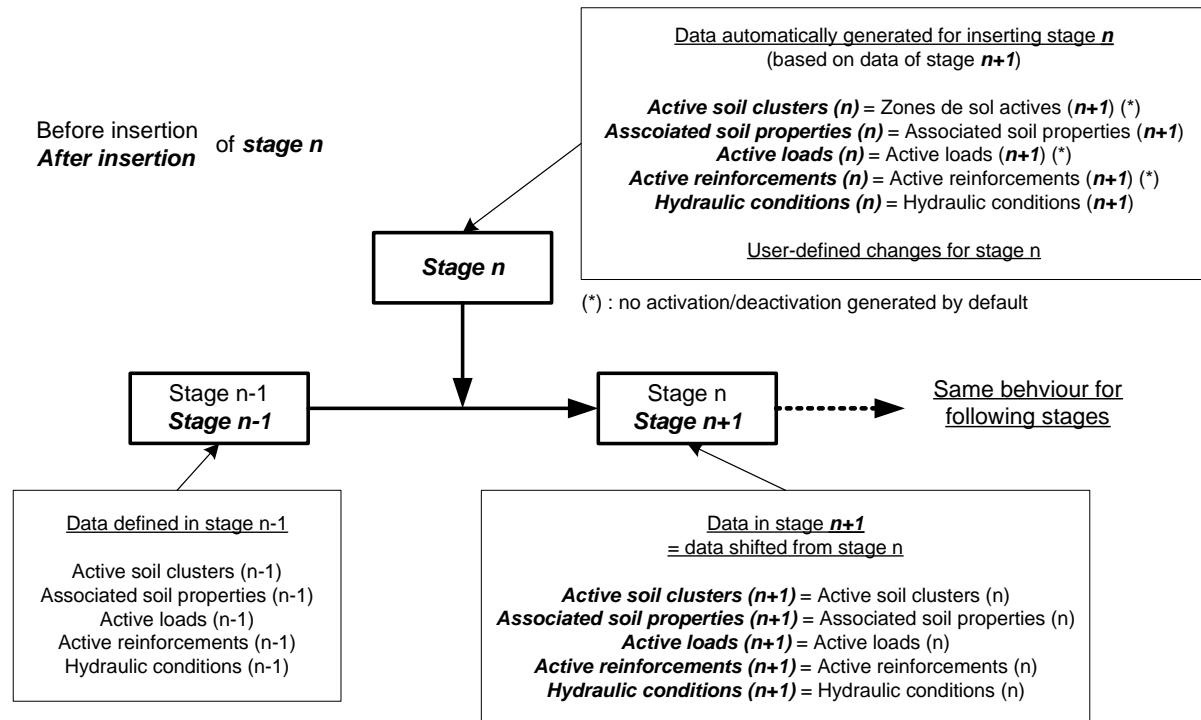
Figure 73: insertion of a stage at position n

The user can then define changes for the new stage n (section 5.1.4).

It is possible to change the name of the new stage, either with the pop up menu (right click on the stage label, and "Rename" item from the pop up menu), or with a first click on the stage label to select it, and then a 2nd click to change this label. After changing the name, the number of the stage will automatically be added to the new name.

5.1.4. Definition or modification of the stage properties

The stage properties and the way to define them are presented hereafter:

5.1.4.1. The slope geometry and boundary

It is possible, in each stage, to activate and/or deactivate soil clusters in the model. The geometrical limits of all soil clusters to be used have to be defined in the "Data" mode.

To activate/deactivate a soil cluster, the user can:

- o Either click on the soil cluster to activate/deactivate: this soil cluster will be alternatively activated then deactivated, etc.
- o Or click with the right mouse button in the soil zone to activate/deactivate, to display the pop up menu, and then choose the option "Activate/Deactivate".

The deactivated soil clusters are displayed with a white background.

5.1.4.2. Association of soil properties with soil clusters

It is possible, for each stage, to modify the soil characteristic sets attributed to soil clusters (to simulate a substitution or a soil improvement for example). All the characteristic sets to be used must have been previously defined in the "Data" mode.

To change the soil properties assigned to a cluster, the user can:

- Either open the table of soil characteristics, with the "View" menu (option "Table of soil characteristics"). It is then possible to drag & drop soil properties to soil clusters.
- Or click with the right mouse button in the soil cluster for which he wants to change the soil properties (choose "Edit" in the pop up menu): the list of available layers is displayed (Figure 74).
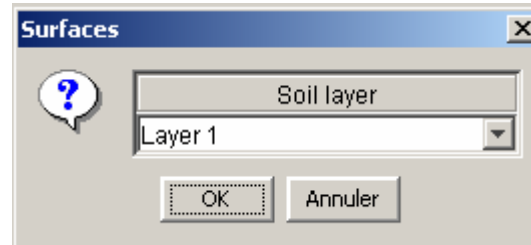
Figure 74: modification of the soil properties for a soil cluster, with the pop up menu

IMPORTANT

The "substratum" properties which appear in the list are created automatically for each project, and if assigned to some soil clusters when calculating, they automatically prevent failure surfaces from entering these substratum clusters.

These "substratum" properties can be selected, but can never be viewed or changed (anyway, failure surfaces cannot go through this layer, and there is therefore no need for substratum properties).

The "substratum" properties are the default properties assigned to new soil clusters (section 4.3.1.1)

5.1.4.3. Active loads

It is possible to activate loads defined in the "Data" mode, and/or to deactivate loads already activated in the previous stages.

To activate/deactivate a load (distributed or linear/moment), the user can:

- Either click on the load to activate/deactivate (on the baseline or on the application point of the load): this load will alternatively be activated then deactivated, etc.
- Or click with the right mouse button on the load to activate/deactivate (on the baseline or on the application point of the load), to display the pop up menu, then choose the menu option "Activate/Deactivate".

The deactivated loads appear in grey. The active loads are displayed in red.

5.1.4.4. Active reinforcements

It is possible to activate reinforcements defined in the "Data" mode, and/or to deactivate reinforcements already activated in the previous stages.

To activate/deactivate a reinforcement (nail, anchor, strip or strut), the user can:

- Either click on the reinforcement to activate/deactivate: this reinforcement will alternatively be activated then deactivated, etc.
- Or click with the right mouse button on the reinforcement to activate/deactivate, to display the pop up menu, then choose the menu option "Activate/Deactivate".

The deactivated reinforcements appear in grey.

5.1.4.5. Hydraulic conditions

Hydraulic conditions can be displayed with the "Stages and conditions" menu, option "Hydraulic conditions for the selected stage", with the same option in the pop up menu (right click on the stage label), or with the [icon] button of the contextual toolbar.

A dialogue box is opened (Figure 75), and several options are available.
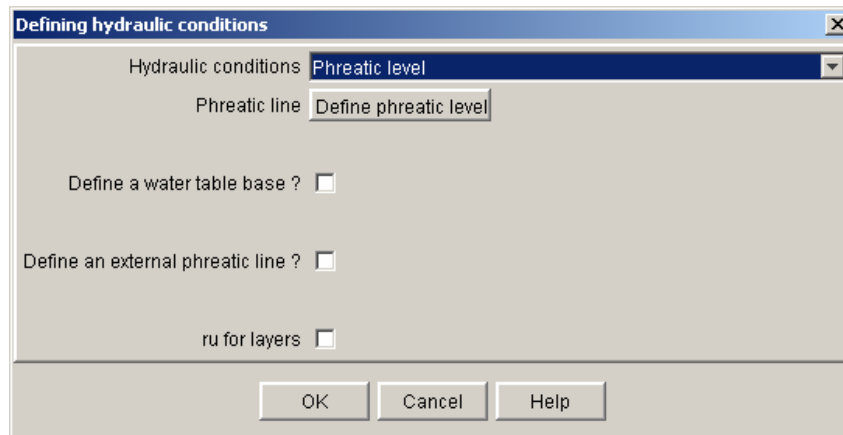
Figure 75: hydraulic conditions dialogue box

- <u>Phreatic line (i.e. groundwater level)</u>: select the "Phreatic level" option (Figure 75), and click on the "Define phreatic level" button. A new dialogue box opens (Figure 76), and enables to type in the points defining the groundwater level.
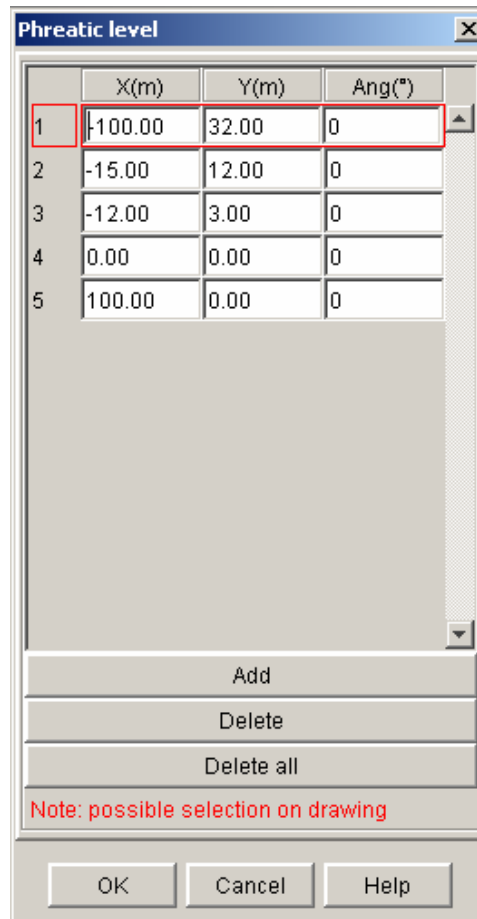
Figure 76: definition of the groundwater level

This dialogue box behaves in the same way as the "Geometry" dialogue box, "Points" tab (section 4.3.2): "Add", "Delete" and "Delete all" buttons enable to add a new point, to delete the selected point, or to delete all points defined for the groundwater level.

The "Possible selection on drawing" comment means that if you click on the graphical view, a new point corresponding to the mouse position will be added to the points list (the Ang column should be filled in for these new points).

The groundwater level is displayed with a thin blue line and classic symbols (blue triangles, Figure 77).

External water level

Water table base
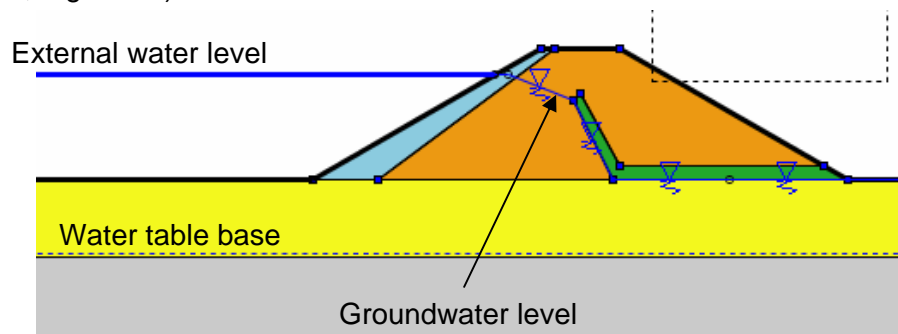
Groundwater level

Figure 77: graphical display for phreatic line

The data displayed in the groundwater dialogue box are the following:

o *Number of the points* (those are attributed automatically and cannot be edited);

o *X and Y coordinates* in m of all points describing the groundwater level;

o *Ang* (°): angle between the equipotential line (i.e. all the points have a constat head on a line) passing through the (X, Y) point and the vertical line (the angle is positive when contrary to trigonometrical direction).

IMPORTANT:

o The groundwater level should always be defined from X_{min} to X_{max}.

o The points of the groundwater level are automatically ordered by increasing X values when validating the dialogue box: if you define a zigzag for the groundwater level (by typing or drawing), it will automatically be "straightened" when validating the dialogue box. In the same way, if you add a point to an existing groundwater level, this point will first seem to be connected to the last point previously defined, but will be ordered after validation of the dialogue box (Figure 78).

Inserted point

Before validation

After validation
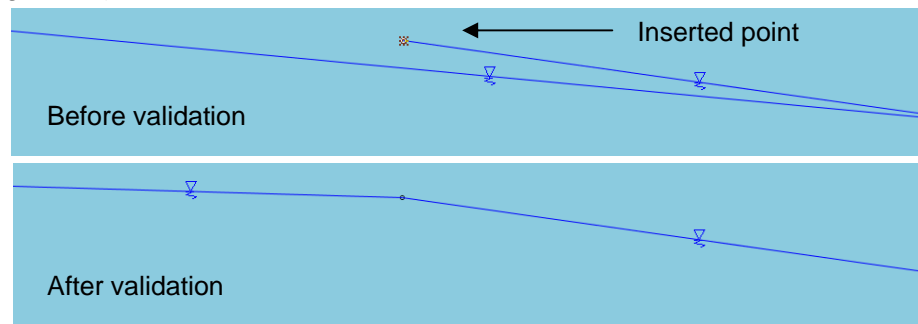
Figure 78: adding a point to an existing groundwater level

• It is possible to define a water table base, by activating the corresponding checkbox (Figure 79).
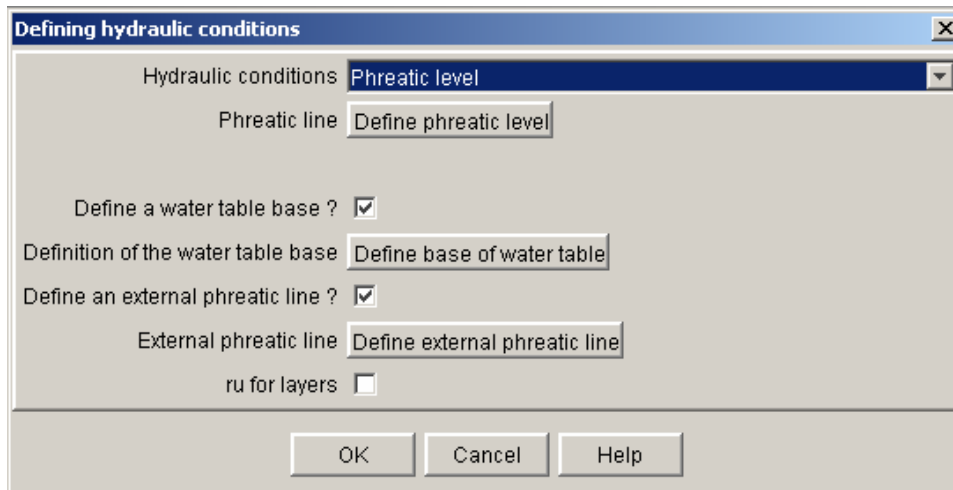
Figure 79: activation of a water table base and an external phreatic line

If water table base is defined, the groundwater is supposed to extend to infinity downwards.

If the checkbox is activated, the button "Define base of water table" is displayed. Click it. A new dialogue box opens (Figure 80), and enables to type in the points defining the water table base.

This window behaves in the same way as the dialogue box for the groundweter level definition (previous section).

Figure 80: definition of the water table base

The data requested in this dialogue box are the following:

o *Numbers of the points* (those are automatically attributed and cannot be edited);

o *X and Y coordinates* in m of all points describing the water table base;

The water table base is displayed with a thin blue dotted line (Figure 77).

IMPORTANT: external phreatic line

If the groundwater level is at least partially "out of the slope", it is compulsory to activate the option "Define an external phreatic line" (Figure 79).

If this option is activated, the button "Define the external phreatic line" is displayed. Click it. A new dialogue box opens (Figure 81), and enables to type in the points defining the external phreatic line.

This window includes a checkbox "Automatic calculation": this option is activated by default for new projects.

o If it is activated, the user cannot modify the table: the external groundwater level will be automatically calculated (the program will compare the groundwater level and the slope boundary).

o If it is not activated, the user should define manually the external groundwater level. The table behaves in the same way as the one for the groundwater level definition (previous sections).

Figure 81: definition of an external phreatic line

The data requested in this dialogue box are the following:

o *Numbers of the points* (those are automatically attributed and cannot be edited);

o *X and Y coordinates* in m of all points describing the external phreatic line;

The external phreatic line is displayed with a bold blue line (Figure 77).

- • Hydraulic data along polygonal failure surfaces: the option "Hydraulic data along with polygonal failure surface definition" should be selected (Figure 82). This option cannot and should not be used except in the case of polygonal failure surfaces (it cannot be combined with circular failure surfaces or logarithmic spirals).

  No other data is then requested in this dialogue box: additional data will be requested when defining the polygonal failure surfaces (section 5.2.5.4, and example of Figure 83).
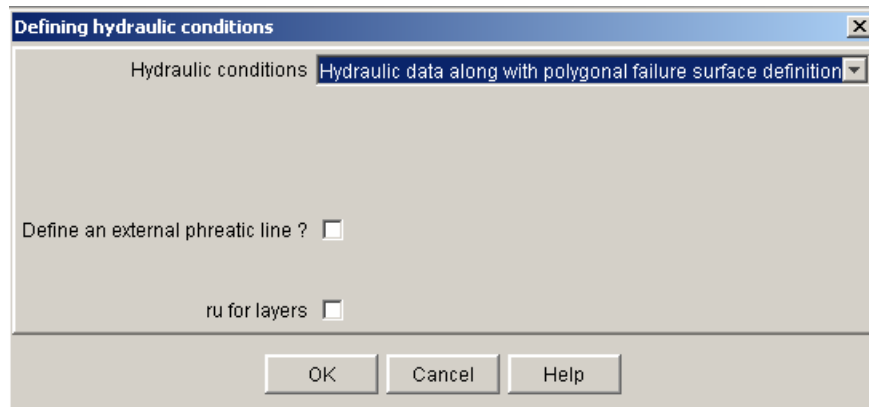
Figure 82: definition of hydraulic conditions:
data along polygonal failure surfaces
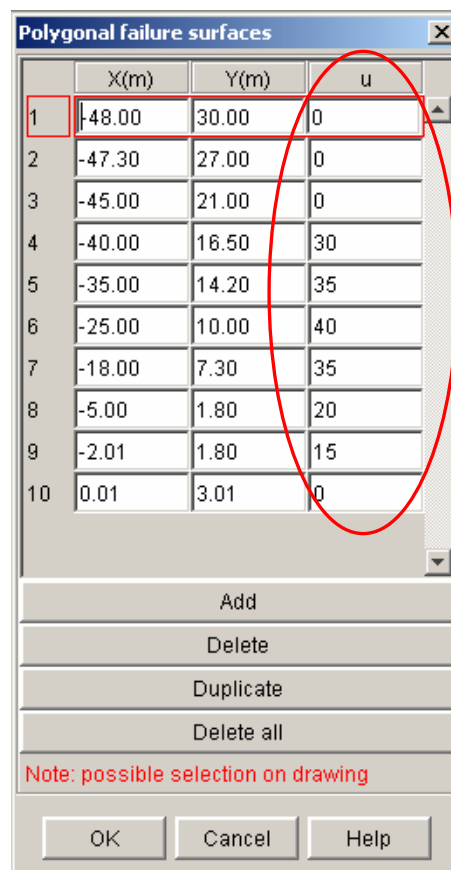
Figure 83: definition of hydraulic pressures along polygonal failure surfaces

No specific graphical display is associated with this type of hydraulic conditions.

IMPORTANT: external phreatic line

If the groundwater level is at least partially "out of the slope", it is compulsory to activate the option "Define an external phreatic line" (Figure 79).

If this option is activated, the button "Define the external phreatic line" is displayed. Click it. A new dialogue box opens (Figure 81), and enables to type in the points defining the external phreatic line.

Refer to the previous section for more information about this dialogue box. In addition, please note that the automatic calculation option for the external groundwater level cannot be used in the case of hydraulic data defined along the failure surface (the automatic calculation of the external groundwater level is correct only with the "phreatic level" option).

- Triangular mesh of pore pressures: select the relevant option in the top list (Figure 84).

The additional parameter "Subdivisions for calculation" is requested, its default value being equal to 5: it enables a "pre-classification" of the triangles in a number of zones (the parameter corresponds to the number of these zones), which accelerates the calculations in the case of hydraulic meshes of pore pressures (please refer to the technical manual for more information about this parameter)

The triangular mesh can be defined manually.

To do so, click on the button "Define nodes and triangles".
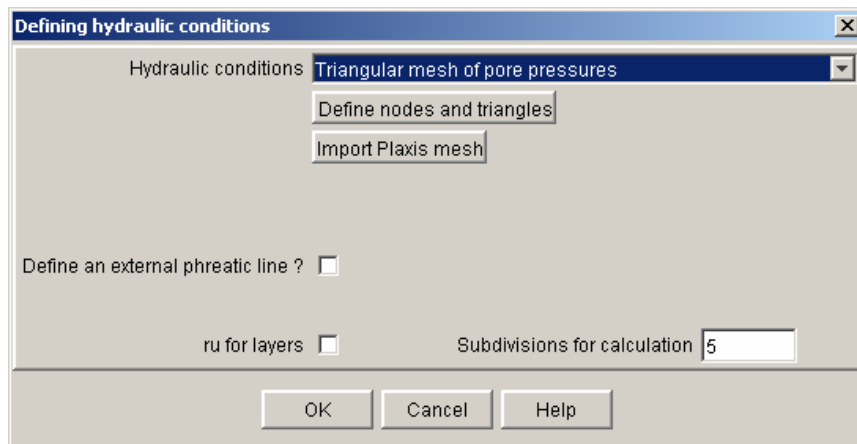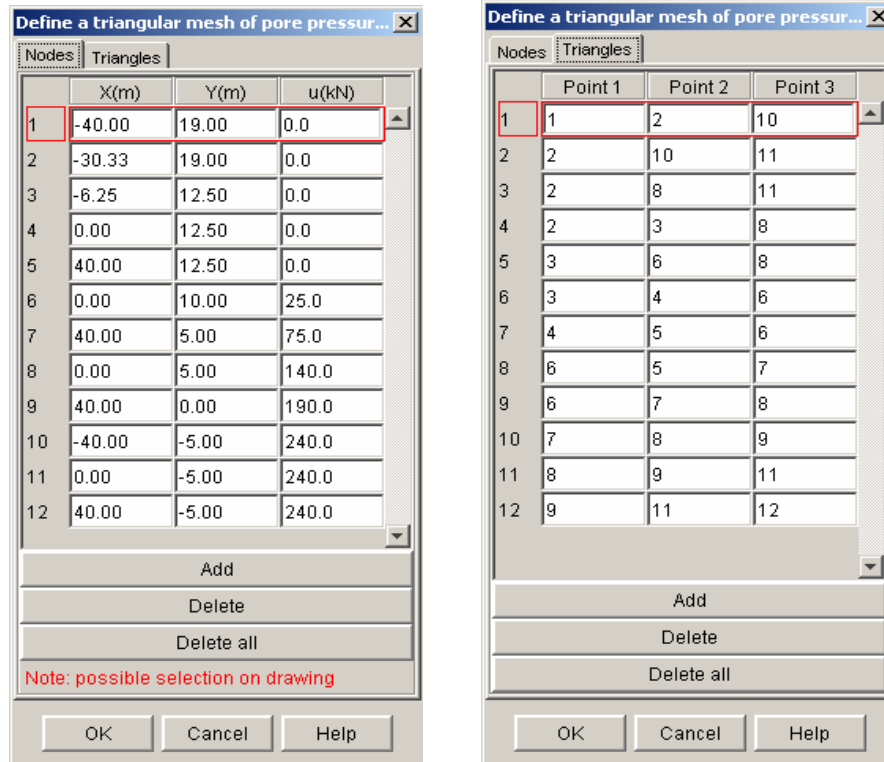
Figure 84: definition of a triangular pore pressures mesh

The dialogue box of Figure 85 then appears, with 2 tabs: "Nodes" and "Triangles".

Figure 85: definition of the pore pressures mesh (points and triangles)

This window is similar to the "Geometry" dialogue box (section 4.3.2): "Add", "Delete", and "Delete all" buttons enable for each tab to add a new point or triangle, to delete the selected point or triangle, or to delete all defined points and/or triangles. Be careful, you delete all points, then all triangles will be deleted too.

Note: it is possible to define these points and triangles on the graphical view.

The data requested in the dialogue box are the following:

o "Points" tab:
  ▪ *Numbers of the nodes* (those are allocated automatically and cannot be edited).
  ▪ *X and Y coordinates* in m for each node.
  ▪ *u:* value of pore pressure for each node (pressure).

o "Triangles" tab:
  ▪ *Numbers of the triangles* (those are allocated automatically and cannot be edited).
  ▪ *Points 1, 2 and 3* constituting each triangle: each point number refers to a point defined in the 1st tab.

An example of hydraulic mesh display is given on Figure 86 (blue points and triangles in thin blue dot line).
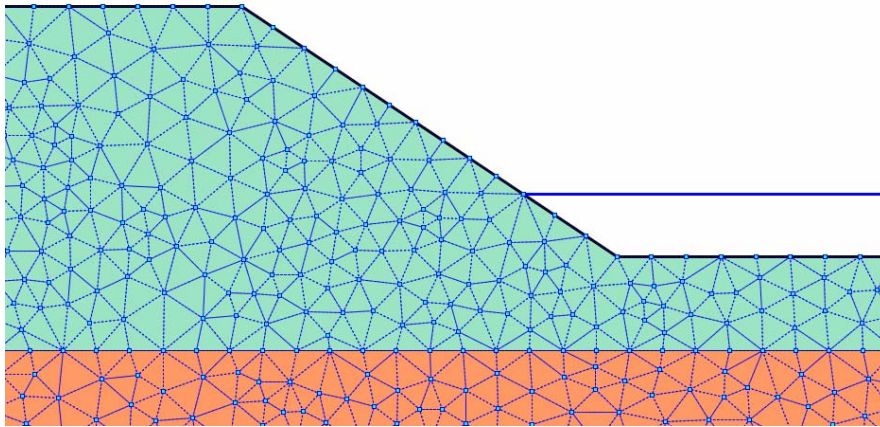
Figure 86: example of the graphical display for a hydraulic mesh

In the case of dams for example, it is sometimes necessary to first calculate the pore pressure mesh with a specific program (finite element type). The results of this hydraulic calculation can be imported into Talren 4 as a triangular mesh. Talren 4 enables to easily import pore pressure meshes resulting from a 2D finite element calculation made with Plaxis v8 software.

To use this option, click on the "Import Plaxis mesh" (Figure 84).

2 dialogue boxes appear successively and request:

- o The Plaxis file which includes the calculated pore pressures (Figure 87). Be careful: the Plaxis files generated for a given Plaxis project include a project.plx file, and a project.DTA directory, that itself contains a copy of the project.plx file. The project.plx file that should be selected is the one "outside" the project.DTA directory (and not the one inside this directory);

- o The calculation step from which the pore pressures should be imported (Figure 88). Talren 4 automatically proposes the list of Plaxis v8 calculation steps available for the selected plx file (this may take a few seconds): you should select one of these available calculation steps.
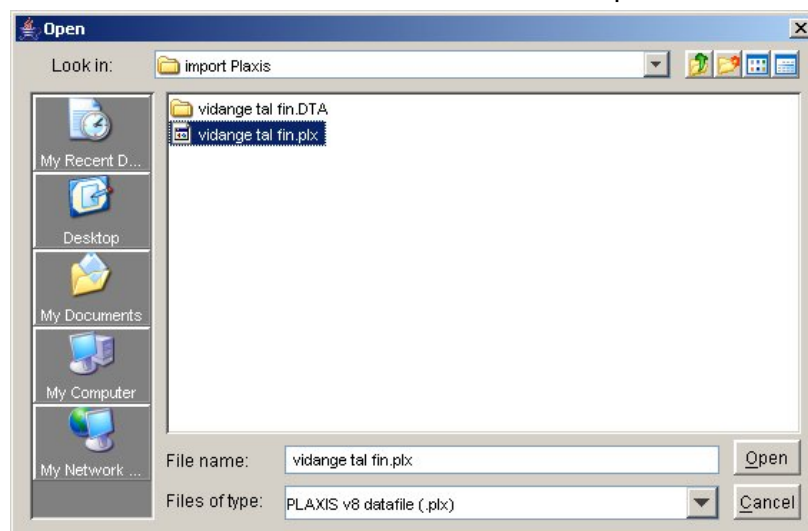
Figure 87: choice of the Plaxis file for pore pressures importation
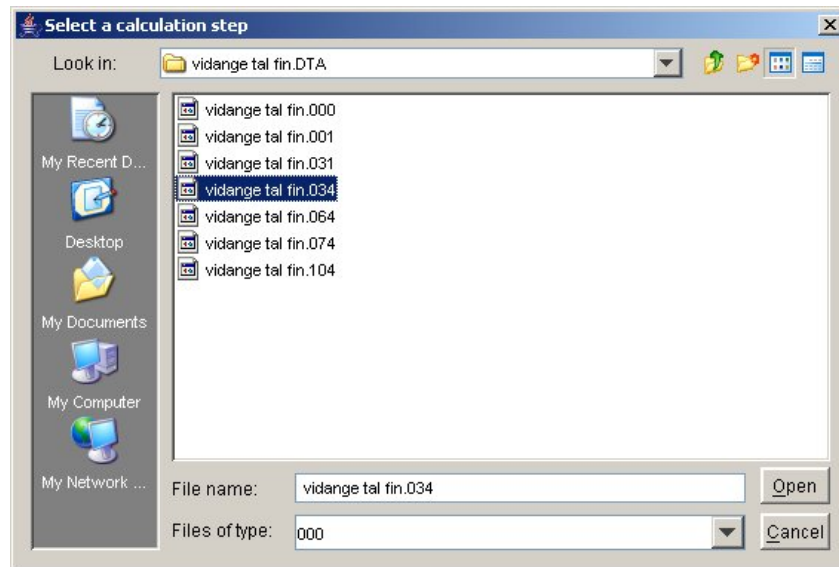
Figure 88: choice of the calculation step for pore pressures importation

After the validation of this 2nd dialogue box, the importation is performed, and the user can then either directly validate the imported mesh, or view the list of nodes and triangles, by clicking on the "Nodes and triangles" button.

IMPORTANT

- The u value that is imported from Plaxis v8 for each node is the value of the active pore pressure p_{active} (with change of sign, to make this data compatible with a Talren calculation) at the end of the selected calculation step. In Plaxis v8, the active pore pressure corresponds to the total pore pressure (permanent pore pressure + possible excess pore pressure).

- The imported triangles are the same as the triangles of the Plaxis mesh (same number). The imported nodes are the extreme nodes of the triangles (3 nodes per triangle).

- It is not possible to import pore pressures of Plaxis calculation step 0 (file .000), i.e. from the Plaxis initial conditions pore pressures. If you select this file, the importation will not work properly.

Note: there is no limit for the number of points and triangles that you can define or import. However, the more nodes and triangles you define or import, the longer the display and calculation times will be.

IMPORTANT: external phreatic line, example of Figure 86:

If the groundwater level is at least partially "out of the slope", it is compulsory to activate the option "Define an external phreatic line" (Figure 79).

If this option is activated, the button "Define the external phreatic line" is displayed. Click it. A new dialogue box opens (Figure 81), and enables to type in the points defining the external phreatic line.

Refer to the previous section for more information about this dialogue box. In addition, please note that the automatic calculation option for the external groundwater level cannot be used in the case of hydraulic data defined along the failure surface (the automatic calculation of the external groundwater level is correct only with the "phreatic level" option).

- <u>ru coefficients</u> for soil layers (refer to the technical manual for the meaning of this parameter): this option can be used alone (Figure 89, hydraulic conditions: "none"), or in addition to one of the types of hydraulic conditions described before.

  In this last case (combination of ru coefficients with another mode of definition of hydraulic conditions), the rule applied for the calculation is the following:

  o For each slice, the calculation checks which soil layer is present at the bottom of the slice axis.

  o If a value of ru different from zero has been defined for this layer, the calculation of the pore pressure u at the bottom of the slice is based on ru coefficient (refer to the technical manual).

  o But if the value of ru defined for this layer is equal to zero, then the calculation of the pore pressure at the bottom of the slice is based on the other mode of hydraulic conditions possibly defined (phreatic level, pressures along polygonal failure surfaces or triangular mesh).

It is therefore possible for example to take into account the ru coefficients inside a dike, and the groundwater level for the soil layers underneath the dike.

In all cases (ru used alone or in combination with one of the other types of hydraulic conditions), it is possible to define in addition an external groundwater level (refer to the other types of hydraulic conditions in the previous paragraphs for the definition of an external groundwater level).
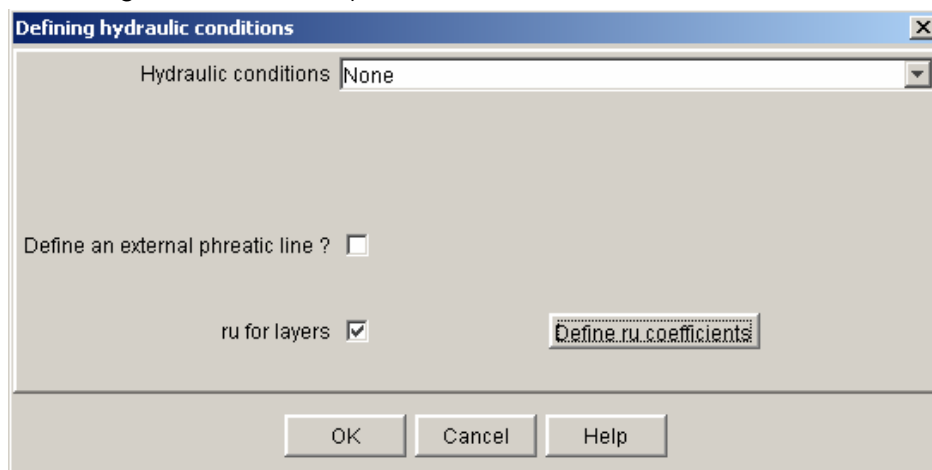
Figure 89: definition of ru coefficients for soil layers

Activate the ru checkbox. The button "Define ru coefficients" is displayed. Click it: the window of Figure 90 appears.

Figure 90: ru coefficients for soil layers

This dialogue box automatically displays the list of the project soil layers (1st column), that cannot be edited. The user should fill in the value of ru for each layer (default value of ru for each layer is zero).

A button at the bottom of the dialogue box enables if necessary to reset ru values for all layers to zero (for the current active stage only).

No specific graphical display is associated with ru coefficients.

5.1.4.6. Comments for a stage

It is possible to define comments specific to each stage (text zones), that will appear on the graphical view in the "Stages/Calculations" mode, depending on the active stage.

This option is accessible when a stage is selected, through the "Stages and conditions" menu, option "Comments for the selected stage" or with a right mouse click on the stage, and then the same option in the pop up menu. It enables to define or modify any number of comments for the selected stage.
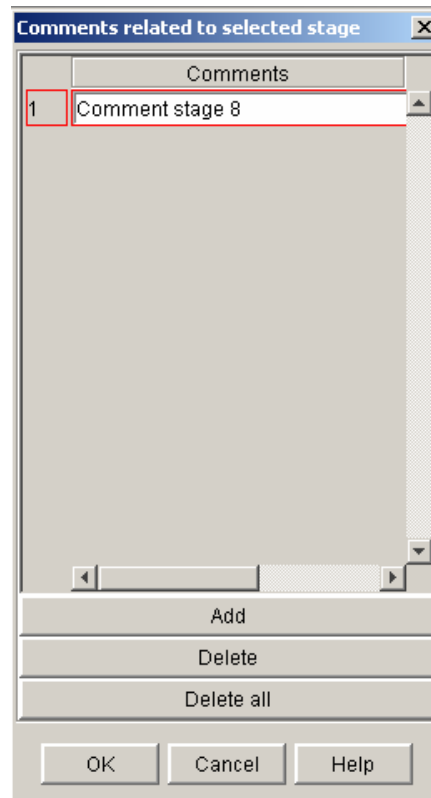
Figure 91: dialogue box for stage comments

To add a new stage comment, click on the "Add" button, then define the comment text on the new line in the table. To change an existing comment, click on the corresponding line and change the text.

The "Delete" and "Delete all" buttons enable to delete one or all comments (for the active stage).

After validating the dialogue box, the comments appear in blue font on the graphical view.

The user can move these comments with the mouse (click with the left mouse button on a stage comment, move the mouse and release the mouse button). It is also possible to delete a comment from the graphical view: select it with the mouse, and press the "Del" key of the keyboard.

The comments are of course saved with the project file.

Refer to sections 3.3.2.6 and 5.2.5.5.

5.1.5. Deleting a stage

The option "Delete the selected stage" of the "Stages and conditions" menu enables to delete the selected stage. The program will request a confirmation before really deleting the stage. This operation is also possible with a right mouse click on the label of the stage to be deleted (option "Delete selected stage" in the pop up menu).

Deleting a stage leads to deleting the actions and data attached to it: activation/deactivation of soil clusters, loads and reinforcements, attribution of soil properties, hydraulic conditions.

The soil clusters, loads or reinforcements are not deleted though, and will still be available in the project. Their activation/deactivation is postponed from stage n (deleted) to the next stage. This way, stage n+1 (renamed n) and the following stages are not affected by the removal of stage n.

Let's take the example of a nail activated in phase n (and therefore also active in phase n+1). This stage n is deleted. The nail should still be active in stage n+1 (renamed n), and so must now be activated in this stage n+1 (renamed n). An example is shown on Figure 92.

However, hydraulic conditions and association of soil properties with soil clusters previously defined in the deleted stage are really deleted, because they are stored independently for each stage (and therefore in particular for stage n+1 renamed n).

The shifting of the stages numbers following the removed stage is made automatically.

After deletion of stage n
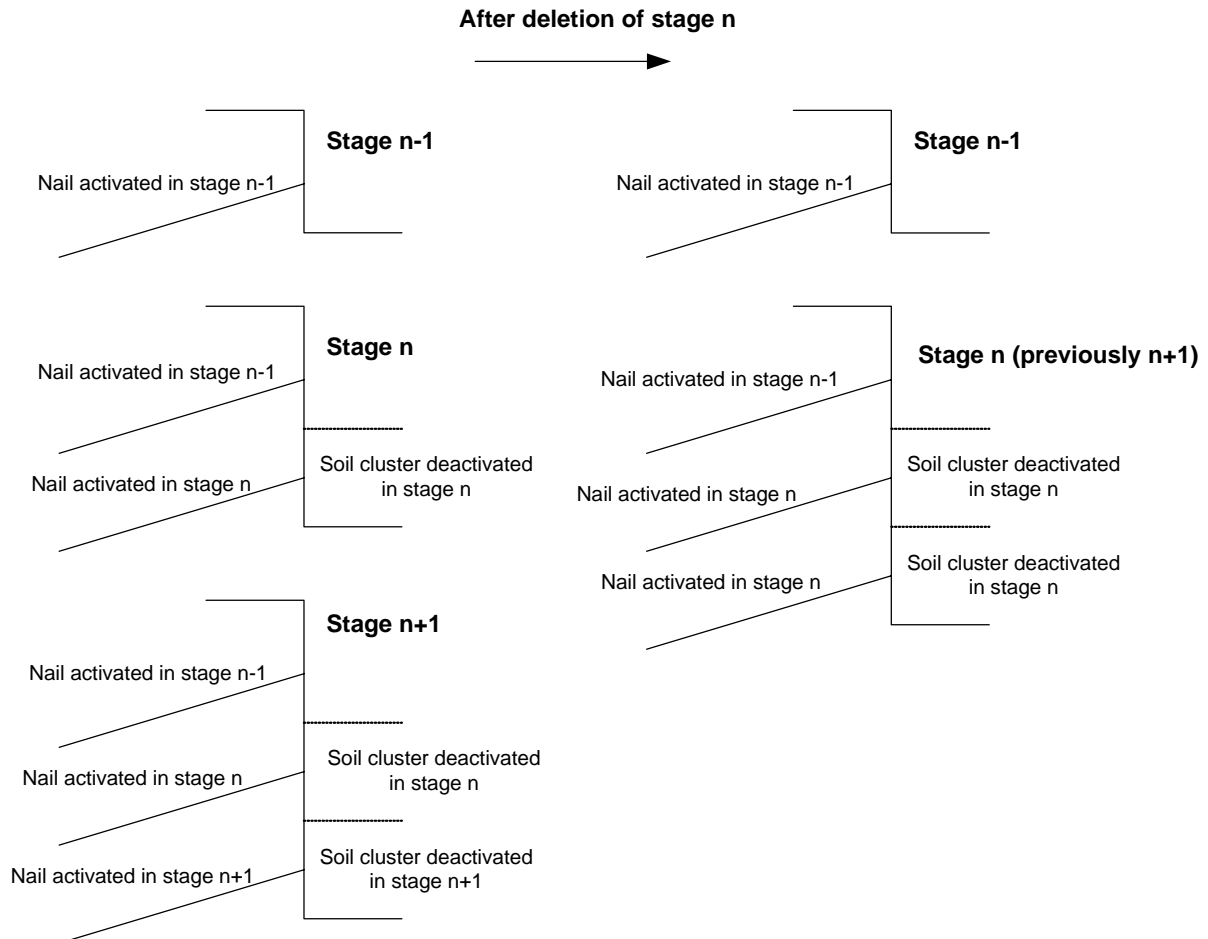
Figure 92: example of a stage removal

Before deleting stage n
After deleting stage n

Data removed when deleting stage **n**

Activations/deactivations of soil clusters (n)
Association of soil properties (n)
Activations/deactivations of loads (n)
Activations/deactivations of reinforcements (n)
Hydraulic conditions (n)

Stage n

Stage n-1
Stage n-1

Stage n+1
Stage n

Same thing for the
following stages

Data defined for stage n-1

Active soil clusters (n-1)
Associated soil properties (n-1)
Active loads (n-1)
Active reinforcements (n-1)
Hydraulic conditions (n-1)

Data for stage **n**
= data shifted from stage n+1

Active soil clusters (n) = Active soil clusters (n+1) (*)
Associated soil properties (n) = Associated soil properties (n+1)
Active reinforcements (n) = Active reinforcements (n+1) (*)
Active loads (n) = Active loads (n+1) (*)
Hydraulic conditions (n) = Hydraulic conditions (n+1)

(*) : including objects activated/deactivated on previous stage n
Activated objects (n) = Activated objects (n+1) + activated objects (n)
For example, on figure 92, the nail n is still there after deleting stage n,
although it was previously activated in stage n that is deleted.

Figure 93: deleting stage n

5.2. Situations (i.e. conditions)

Calculation situations (or conditions) enable to perform several different calculations for a same construction stage (i.e. a same cross-section): different types of failure surfaces, different partial safety factors, calculation with and without seismic accelerations for example, etc.

As opposed to stages, the situations are not defined incrementally: they are defined independently from one another, and are simply numbered to enable the user to refer to them.

Situations do not correspond to a true physical evolution of the construction site, but to different calculation assumptions (or conditions).

Most possible actions relative to situations are accessible through the corresponding pop up menu (right click on a situation label).
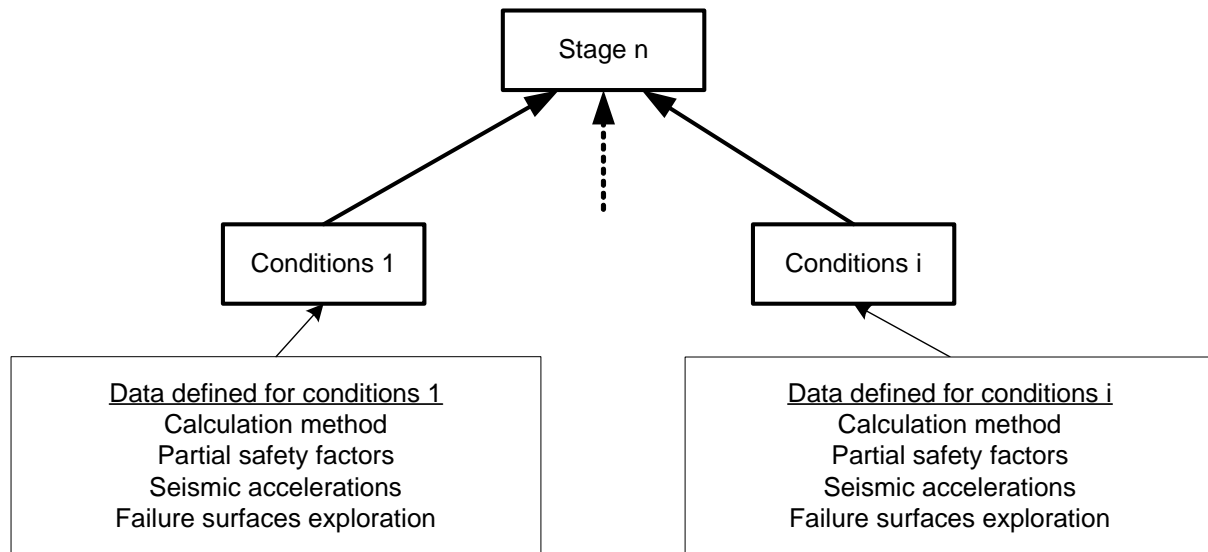
Figure 94: definition of several situations for a same stage

5.2.1. Principles

For each construction stage, a first set of conditions is generated automatically (at least one situation is necessary to perform calculations). The user then decides whether to create additional situations or not.

The situations are numbered automatically one after the other.

Once a situation is created (added), the user can define the following data for each of them:

- Calculation method;
- Partial safety factors;
- Seismic accelerations;
- Failure surfaces and associated parameters;

5.2.2. Adding a new set of conditions

The option "Add a new set of conditions" of the "Stages and conditions" menu enables to create a new situation which is added at the end of the situations list for the active stage (no matter which situation was selected when adding a new one). This operation is also possible with a right click on an existing situation, and the option "Add a new set of conditions" in the pop up menu.

Once the situation is created, it is possible to change its settings (section 5.2.5).

It is possible to the change the name of a situation, either with the pop up menu (right click on the situation label, option "Rename"), or with a first click on the situation label to select it, and a 2nd click to change this label. After changing the name, the number of the situation will be added automatically to the new name.

5.2.3. Duplicating a set of conditions

The option "Duplicate the selected set of conditions" of the "Stages and conditions" menu enables to create a new set of conditions in the same stage (added in the last position for the active stage), with the same settings as the duplicated situation. This operation is also possible with a right click on the situation to duplicate, and the choice "Duplicate the selected set of conditions" in the pop up menu.

Once the situation created, it is possible to change its settings (section 5.2.5).

5.2.4. Copy/paste a situation

The "Copy" option is accessible by a right click on the situation to be copied. After this operation, the "Paste" option in the same pop up menu is activated: it is possible to paste the copied situation either in the same stage, or in a different stage: in any case, the pasted situation is added in last position for the selected stage, and has the same settings as the copied situation.

Once the situation is pasted, it is possible to change its settings (section 5.2.5).

5.2.5. Definition or modification of the conditions settings

The conditions settings can be displayed with "Stages and conditions" menu and the "Define selected set of conditions" option, with the same option in the pop up menu (right click on the situation label), or with the [ ] button of the contextual toolbar. A dialogue box is opened (Figure 95). It is detailed in the following sub-sections.
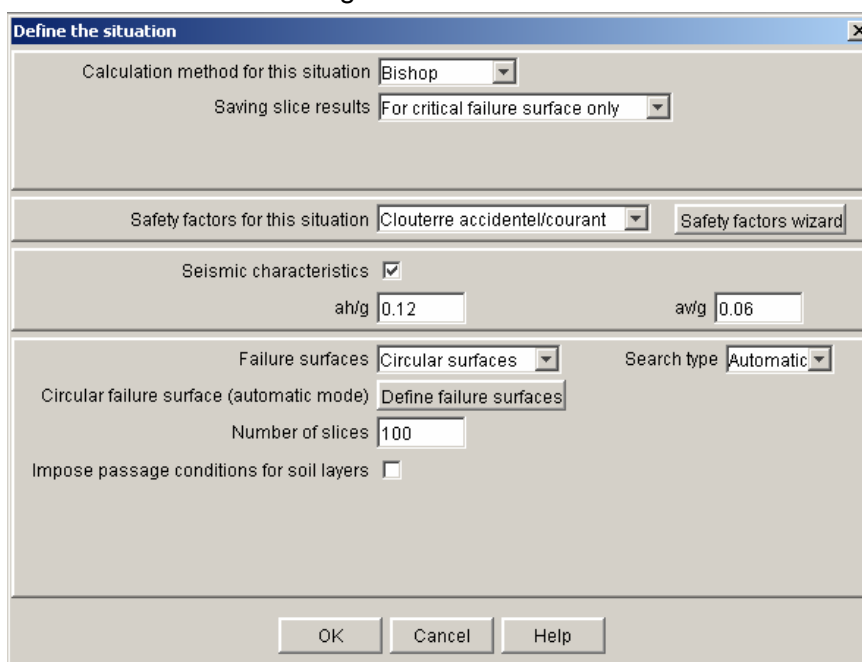
Figure 95: conditions dialogue box

5.2.5.1. Calculation options

Four calculation methods are available: Fellenius, Bishop, Perturbations and yield design calculation.

The default calculation method is the one defined in the project general settings (section 4.2), but it is possible to change it individually for each situation.

The saving option for detailed slice results determines the size of the results files: you can either save the detailed slice results for the critical failure surface only (recommended and default option), save the detailed slice results for all calculated failure surfaces (load and display times will increase), or on the contrary, save no detailed slice results at all. Also refer to section 6.4.3.

5.2.5.2. Partial safety factors

A set of partial safety factors should be selected from the list of available sets (i.e. the sets that are stored in the local project database, refer to section 4.7.2.2).

The partial safety factors wizard can be activated with the corresponding button to view the partial safety factors values (but edition is not allowed, because edition is possible only in the Data mode, refer to section 4.7.2.2).

The default set is the one selected in the project general settings, if any (section 4.2).

5.2.5.3. Seismic data

Seisms can be taken into account in TALREN 4 by the pseudo-static method (please refer to the technical manual for more information about the calculation method).

To use this option, the "Seismic characteristics" checkbox should be activated.

Additional data will then be requested (Figure 95):

- a_h/g (no unit): ratio horizontal acceleration / gravity;
- a_v/g (no unit): ratio vertical acceleration / gravity;

IMPORTANT:

- These accelerations can be positive (downwards or towards the right) or negative. It is generally advised to test the 4 possible signs combinations to check which combination is the most unfavourable.
- The horizontal acceleration is taken into account only for the soils and the "internal" phreatic level.
- The vertical acceleration is applied to the soils, the linear and distributed loads, and both the external and internal water levels.
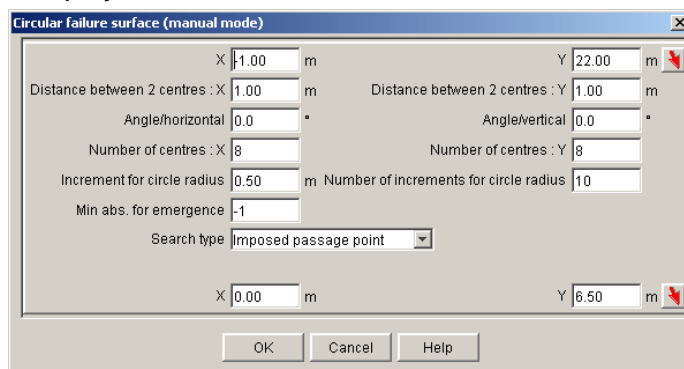
5.2.5.4. Failure surfaces

They can be circular (manual or automatic search), polygonal, or logarithmic spirals.

The requested data depend on the search type, and are described for each case hereunder.
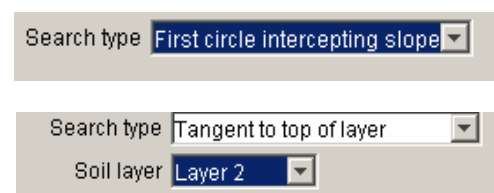
- Circular failure surfaces (manual mode): "Circular surfaces" choice and "Manual" search type (Figure 95).

  When clicking on the button "Define failure surfaces", the dialogue box of Figure 96 is displayed.

Figure 96: failure surfaces dialogue box: circular failure surfaces, manual search

The manual grid for the circles centres should be defined. For each centre, several circles may be calculated using different radius values.

The data requested in this dialogue box are the following:

- *X and Y:* (m) coordinate of a corner of the centres grid. They can be defined:
  o Either by defining X and Y values (m) in the corresponding cells;

  o Or by clicking on the [button icon] button, and then clicking the point on the graphical view.

- *X and Y increments*: distance (m) between 2 adjacent centres following the X and Y axis;
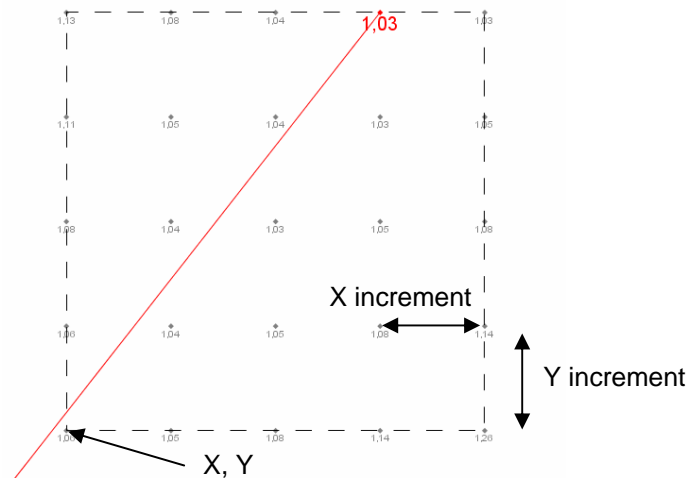
Figure 97: manual search for circular failure surfaces

- *Angle/horizontal and angle/vertical*: "Inclinations" of the grid axis compared to the horizontal and to vertical lines (trigonometrical direction);
- *Number of centres to explore in X and Y directions*;
- *Increment for circle radius* (m): value (positive or negative) of the radius increment;
- *Number of increments for circle radius:* maximum number of circles (with different radii) calculated for each centre;
- *Min Abs. for emergence XL* (m): value of the abscissa beyond which the failure surfaces are not taken into account, if these surfaces are completely on the left of the XL position. The XL value enables, in the case of complex slopes, to avoid some undesired circles. The position of the minimum abscissa for emergence is displayed on the graphical view (at the bottom) with the [symbol] symbol.
- *Search type*: 3 choices are available (Figure 96): imposed passage point, first circle intercepting slope, and tangent to top of layer. This choice is used to calculate the radius of the first circle associated with each centre. For the other circles associated with the same centre, the radius is determined by incrementation (positive or negative) of the radius (and the criterion is therefore no longer applied).

  Depending on this option, the requested data will vary:
  - *Imposed passage point:* in this case, this imposed passage point should be defined:
    - Either typing X and Y values in the edition cells;
    - Or by clicking on the [button] button, then on the position for the imposed passage point on the graphical view.
  - *First circle intercepting slope:* in this case, no additional parameter is requested.
  - *Tangent to top of layer:* in this case, the tangency layer should be selected among the list of the project layers.

- Circular failure surfaces (automatic search): "Circular surfaces" choice and "Automatic" search type (Figure 98).

Figure 98: failure surfaces dialogue box: circular failure surfaces, automatic search

When clicking on the "Define failure surfaces" button, the dialogue box of Figure 99 is displayed.
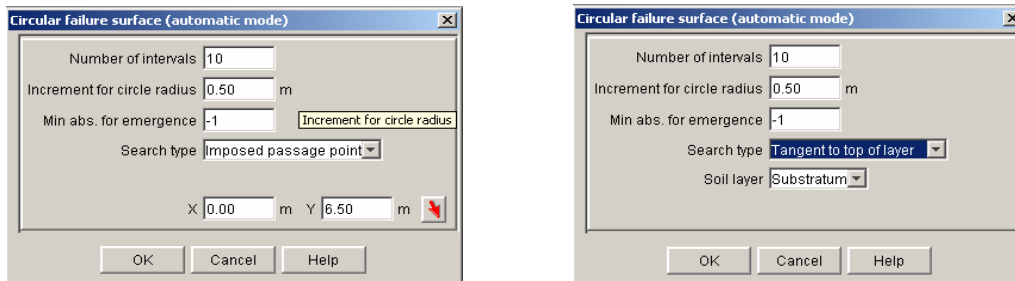
Figure 99: failure surfaces dialogue box: circular failure surfaces, automatic search

The data requested in this dialogue box are the following:

- *Number of intervals* (default value: 10): this setting determines the "accuracy" of the automatic search: number of explored directions, number of test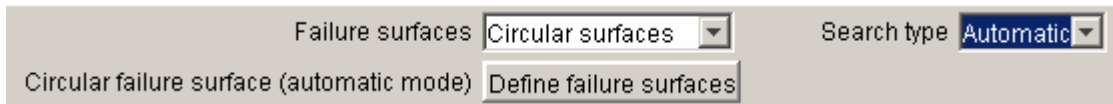ed distances for centres on each direction, and number of radius increments for each centre. Refer to section 2.1.3 and the technical manual for more information about the automatic search;

- *Increment for circle radius (m):* (positive) value of the radius increment. With the automatic search, Talren 4 automatically makes both a positive and negative incrementation (increasing and decreasing the circles radius);

- *Min abs. for emergence XL* (m): value of the abscissa beyond which the failure surfaces are not taken into account, if these surfaces are completely on the left of the XL position. The XL value enables, in the case of complex slopes, to avoid some undesired circles. The position of the minimum abscissa for emergence is displayed on the graphical view (at the bottom) with the [symbol] symbol.

- *Search type*: 2 choices are available (Figure 99): imposed passage point, and tangent to top of layer. This option is used to calculate the radius of the first circle associated with each centre. For the other circles associated with the same centre, the radius is determined by automatic incrementation (positive and negative) of the radius (and the criterion is therefore no longer applied).

  Depending on this option, the requested data will vary:

  o *Imposed passage point:* in this case, this imposed passage point should be defined:

    ▪ Either typing X and Y values in the edition cells;

    ▪ Or by clicking on the [button] button, then on the position for the imposed passage point on the graphical view.

  o *Tangent to top of layer:* in this case, the tangency layer should be selected among the list of the project layers.

- Polygonal failure surfaces:

  Select "Polygonal surfaces" (Figure 102), and click on the "Define failure surface" button. The dialogue box of Figure 101 is displayed.
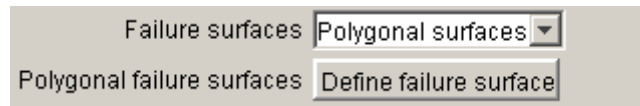
Figure 100: failure surfaces dialogue box: polygonal failure surfaces
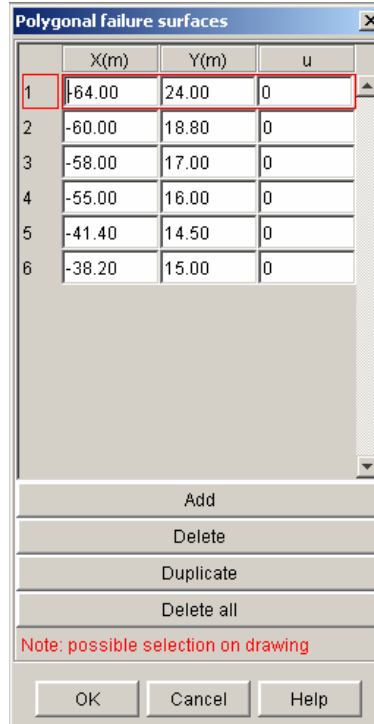
Figure 101: failure surfaces dialogue box: polygonal failure surfaces

This window behaves like the "Geometry" dialogue box, "Points" tab (section 4.3.2): "Add", "Delete" and "Delete all" buttons enable to add a new point, to delete the selected point or to delete all points defining the polygonal failure surface.

The "Possible selection on drawing" mention means that you can add points to the failure surface by clicking them on the graphical view (the u column should be filled after the points have been created).

The polygonal failure surface is represented by a thin black line, with black points (Figure 102).

Polygonal failure surface
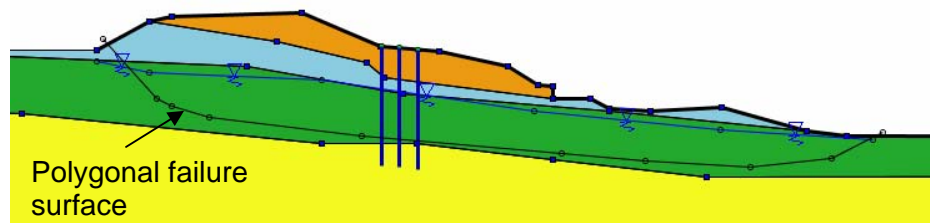
Figure 102: example of graphical display for polygonal failure surfaces

The data requested in the dialogue box are the following:

o *Numbers of the points* (those are automatically allocated and cannot be edited);

o *X and Y coordinates* in m of each point of the polygonal failure surface;

o *u* (pressure): pore pressure at (X,Y) point. This information is taken into account in the calculations only if the hydraulic conditions are set on "Hydraulic conditions along polygonal failure surfaces" for the active stage. If this type of hydraulic conditions is not active in the stage, the data in the u column will be ignored.

IMPORTANT:

o For each situation one single polygonal failure surface can be defined: to analyse several polygonal failure surfaces for a given construction stage, several situations should be defined.

o The polygonal failure surfaces should be defined so that the first and the last points are outside the slope boundary, and so that the 2nd and the next-to-last points are below the slope boundary (Figure 102).

o The points of a polygonal failure surface are automatically ordered by increasing X values when validating the dialogue box: if you define a zigzag failure surface (by typing or drawing), it will automatically be "straightened" when validating the dialogue box. In the same way, if you add a point to an existing failure surface, this point will first be connected to the last point previously defined, but will be ordered after validation of the dialogue box (Figure 103).
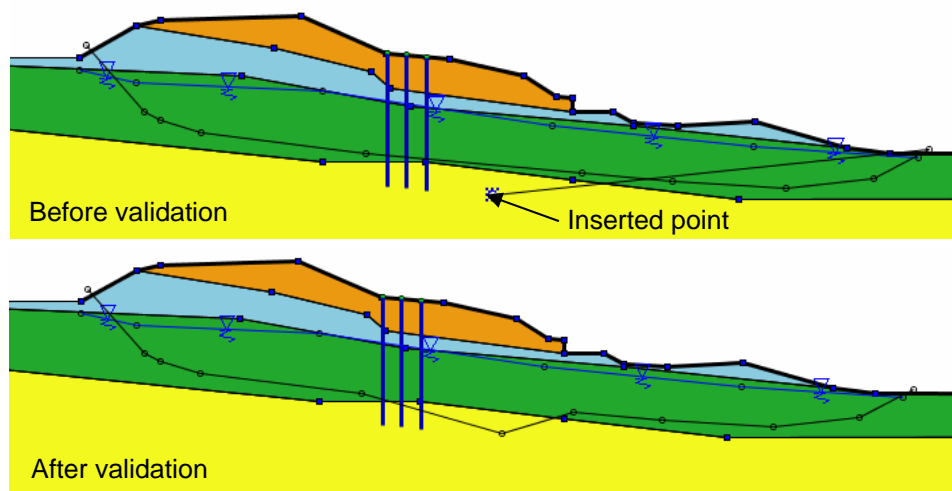
Figure 103: addition of a point to an existing polygonal failure surface

- Logarithmic spirals

  Chapter under construction

- Data requested for all types of failure surfaces (Figure 104)

  o *Number of slices:* the default value is set to 100 for new situations. There is no limit to the number of slices that you can define, but a large number of slices will slow down calculation and display .

  o *Impose passage conditions for soil layers:* this checkbox enables to define passage conditions (it is not compulsory). New data are then requested (Figure 104):

    ▪ *Condition n° 1*: choose the soil layer among the list of the project layers, then choose if the failure surface must pass out this layer, or on the contrary must pass in this layer;

    ▪ *Condition n° 2*: same type of data as for condition 1;

    ▪ *Combination mode*: it is possible to combine both conditions with "or" or "and".
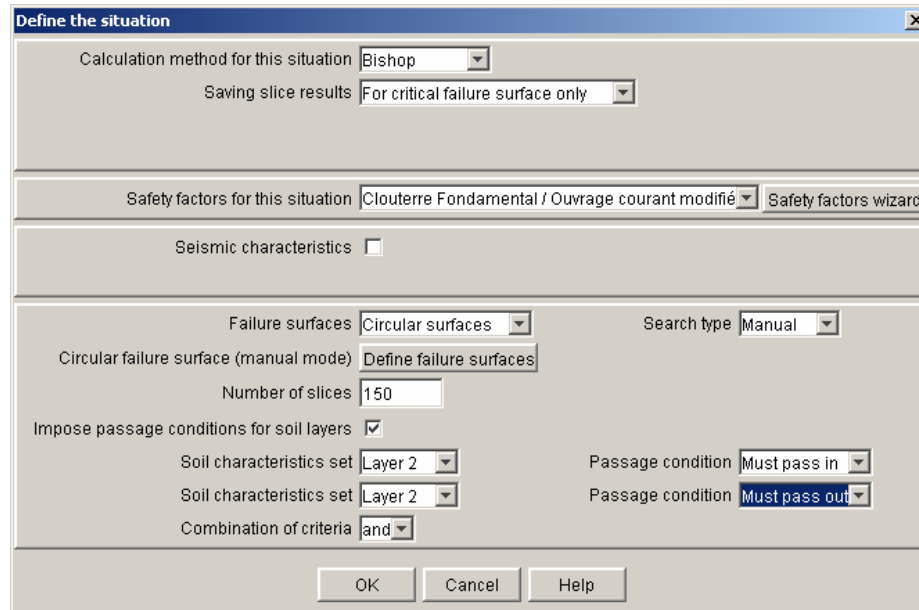
Figure 104: conditions dialogue box: impose passage conditions

IMPORTANT: if the "substratum" properties are assigned to some soil clusters, Talren 4 will automatically consider that the failure surfaces must pass out of these clusters (this condition is automatically defined and it is combined with the additional conditions defined by the user here).

5.2.5.5. Situation comments

It is possible to define comments specific to each situation (text zones) that will appear on the graphical view in "Stages/calculations" mode depending on the selected situation.

This option is accessible when a situation is selected, through the "Stages and conditions" menu, option "Comments for the selected conditions" or with a right mouse click on the situation, and then the same option in the pop up menu. It enables to define or modify any number of comments for the selected situation.
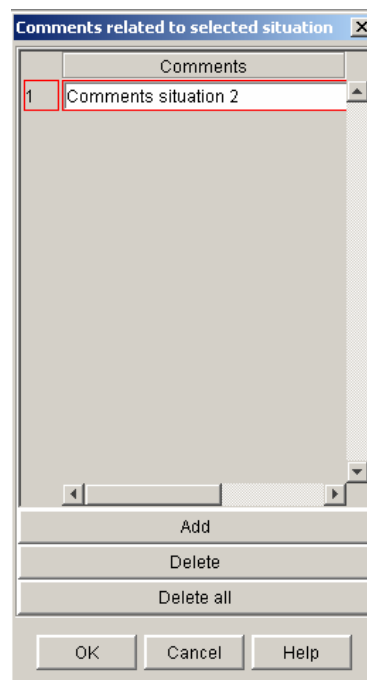
Figure 105: dialogue box for situation comments

To add a new situation comment, click on the "Add" button, then define the comment text on the new line in the table. To change an existing comment, click on the corresponding line and change the text.

The "Delete" and "Delete all" buttons enable to delete one or all comments (for the active situation).

After validating the dialogue box, the comments appear in green font on the graphical view.

The user can move these comments with the mouse (click with the left mouse button on a situation comment, move the mouse and release the mouse button). It is also possible to delete a comment from the graphical view: select it with the mouse, and press the "Del" key of the keyboard.

The comments are of course saved with the project file.

Refer to sections 3.3.2.6 and 5.1.4.6

5.2.6. Removing a situation

The option "Remove the selected set of conditions" of the "Stages and conditions" menu enables to delete the active situation. The software will request a confirmation before actually deleting the situation. This operation is also possible with a right click on the situation label, (option "Remove the selected set of conditions" in the pop up menu).

The removal of a situation leads to the removal of all data directly attached to it.
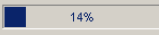
But situations are independent from each other, and deleting a situation has no influence on the other situations. The numbers of the situations following the one that was just deleted are automatically shifted.

5.3. Calculation and results display

5.3.1. Calculation

3 options are available to perform calculations:

- Calculate the selected set of conditions: "Calculation and results" menu or Σ button;

- Calculate all conditions of the selected stage: "Calculation and results" menu or Σ_Σ button;

- Calculation all: "Calculations and result" menu or $\dot{\Sigma}_\Sigma$ button.

During the calculations, a window (Figure 106) indicates the progression of the requested operations. A gauge in the status bar ▮ 14% displays the progress of the calculation itself for each situation.
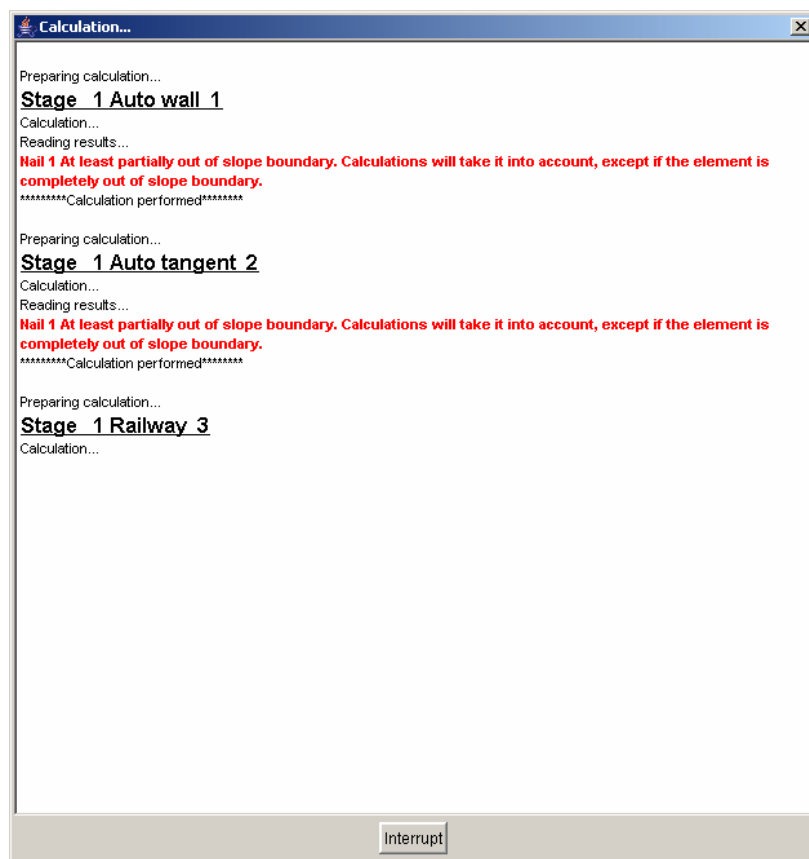
Figure 106: calculations progress

The button "Interrupt" enables to interrupt the calculation.

Talren 4 makes data checking before running the calculation, and if it detects errors or forbidden values, it displays red warnings or errors in the calculation window.

In case of warnings, the calculation is performed (it's up to the user to decide whether they are valid or not considering the warnings). But in case of errors, the calculation is not performed: data should be corrected as indicated before running the calculation again.

At the end of the calculations, the calculation window closes automatically, except if errors or warnings are displayed: it is then up to the user to close this window after reading the messages (the "Interrupt" button of Figure 106 is then replaced with the "Close" button).

When calculations have been performed, it is possible to display several types of results (graphs and tables). The various options are detailed in the following sub-sections.

Situations for which calculations have been performed (i.e. results are available) are displayed in bold characters in the stages explorer.

On the other hand, if the calculations have not yet been performed, or have failed, the output icons and menu options are disabled.

5.3.2. Default output display

After calculation, the default ouput types displayed on the graphical view are (Figure 107):

- The value of the minimum safety factor (red font), at the bottom of the graphical view;
- The values of the safety factor obtained for each failure surface:
  - o In the case of circular failure surfaces: the minimum safety factor obtained for each centre is displayed close to 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 this minimum next to each centre);
  - o In the case of polygonal failure surfaces: the safety factor calculated for the failure surface defined in the situation is displayed next to this failure surface;
  - o In the case of logarithmic spirals: the minimum safety factor obtained for each spiral is displayed next to its pole.
- The critical failure surface: painted in red on the drawing. The zoom level is optimised automatically after calculation to display the critical failure surface (centre, pole, etc).
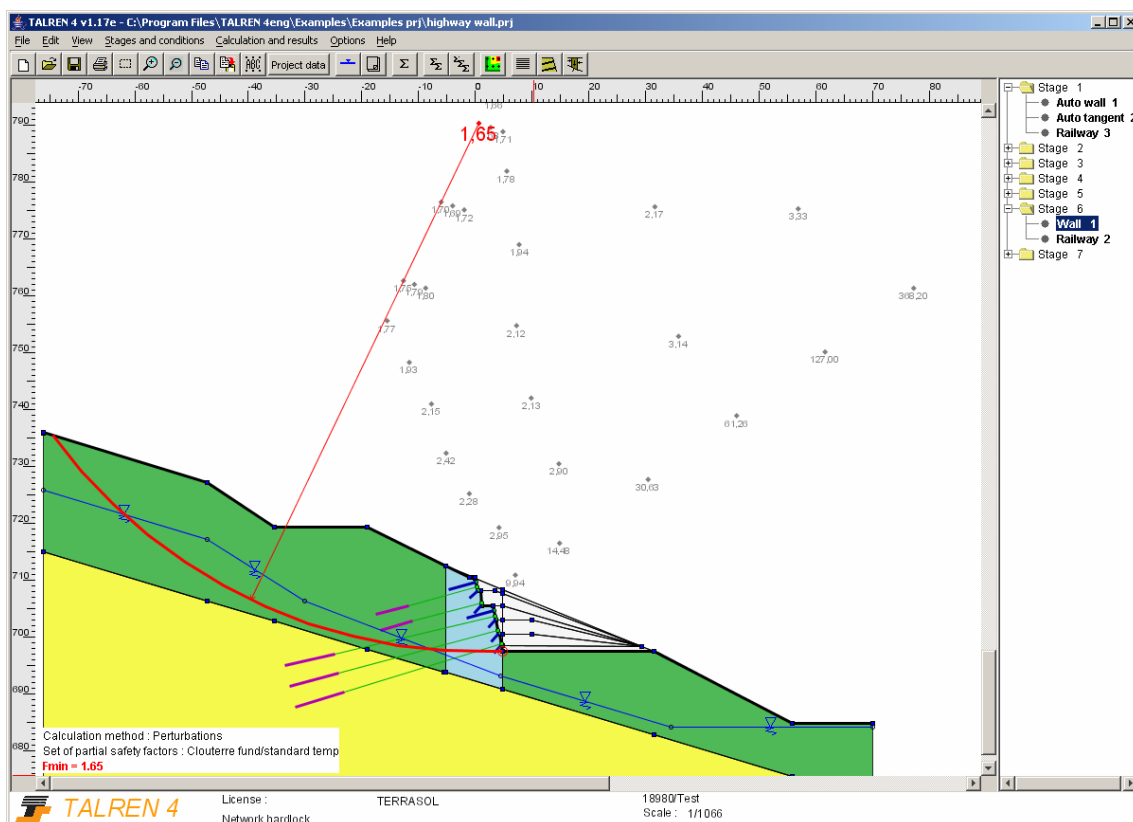
Figure 107: default output display after the calculation of a situation (example in the case of circular failure surfaces)

5.3.3. Settings for the graphical output display

The settings for the graphical ouput display can be edited with the "Calculation and results" menu, option "Graphical display settings for output", by the ⬛ button of the toolbar, or by the pop up menu on the graphical view (click on the right mouse button on the graphical view). The dialogue box illustrated on Figure 108 is displayed. It enables to request the display of (in addition to the default display, section 5.3.2):

- o   All calculated circles (Figure 109);
- o   All circles corresponding to a certain range of safety factor values (Figure 110);
- o   Isovalues (colour shadings, Figure 111): here also, a range of safety factor values can be defined;
- o   Indications on the design criterion for the reinforcements (Figure 111): 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, refer to section 5.3.5).
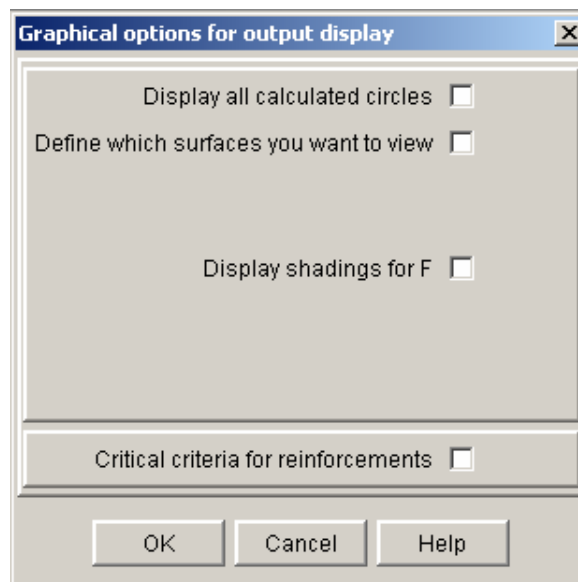
Figure 108: graphical output display settings
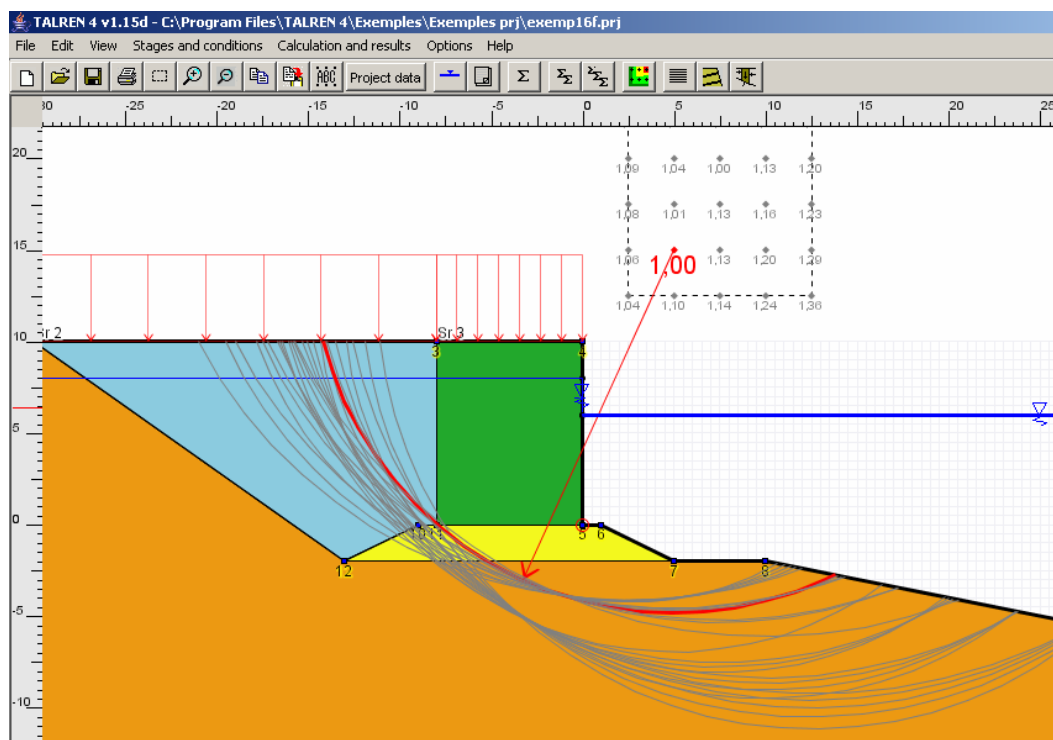
Figure 109: display of all calculated circles
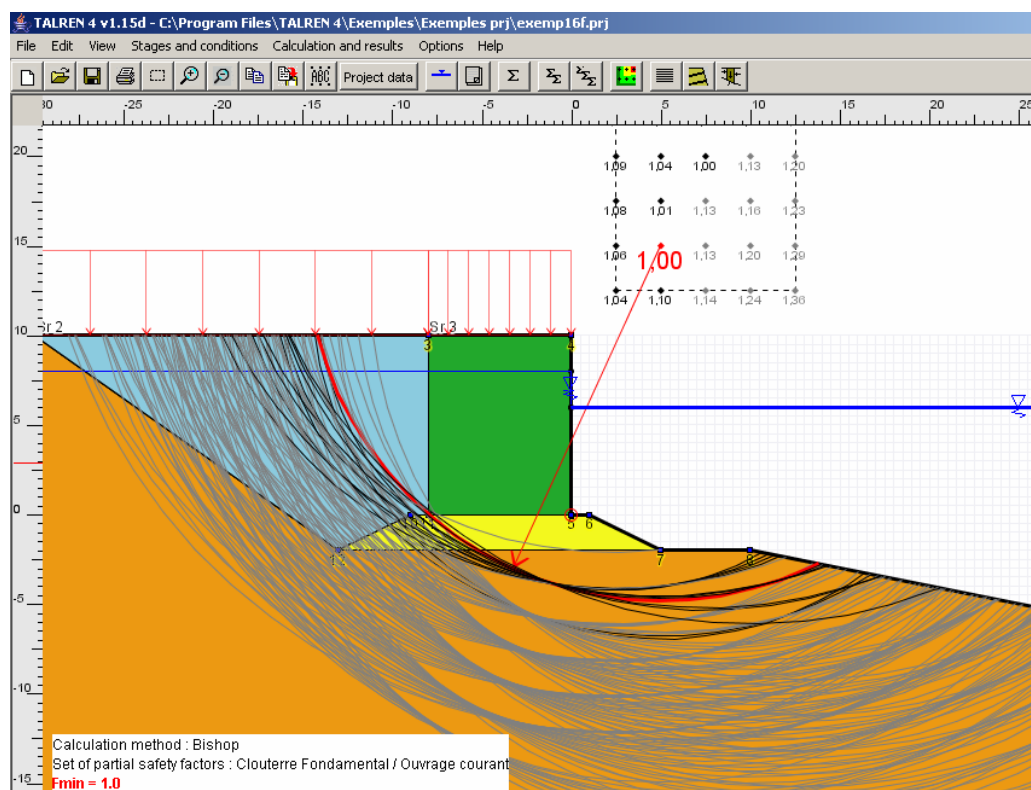
Figure 110: display of circles corresponding to a given range of safety factor values
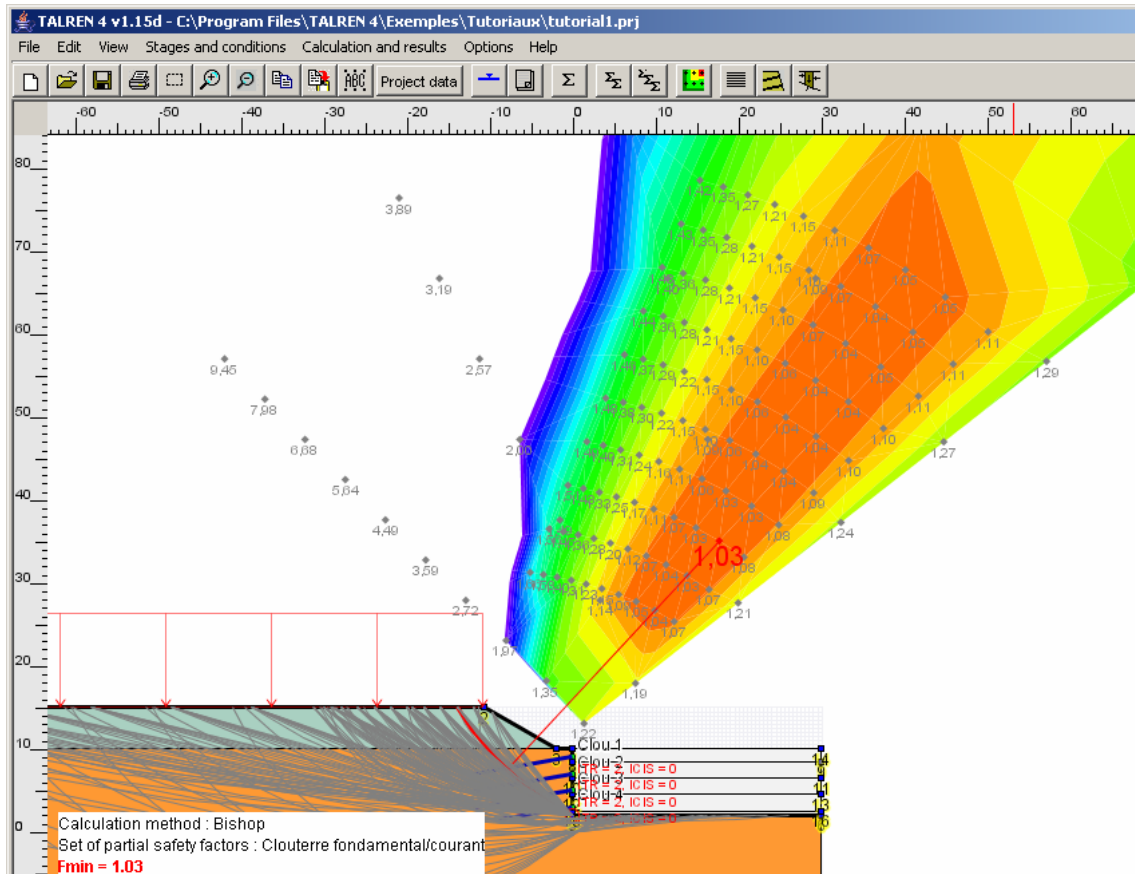
Figure 111: display of colour shadings and reinforcements design criteria

5.3.4. Detailed results for failure surfaces

The table of detailed results for failure surfaces can be displayed with the "Calculation and results" menu, option "Detailed output", or with the ▤ button of the toolbar. One of the tables illustrated in Figure 112, Figure 113 and Figure 114 is displayed.

By default:

- o The green highlighted line corresponding to the critical failure surface is selected when opening the dialogue box.
- o 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 (Figure 112) enables to request the display of all calculated circles (Figure 113).

The button "Legend for F values" enables to display the description of a legend for the safety factor values (Figure 115).

The columns displayed in the table depend on the type of failure surface (circular, polygonal, spiral), and on the calculation method (Fellenius, Bishop, Perturbations, yield design calculation).

The 3 values F-SOL, F-SURCH and F-TOTAL are displayed for the calculation methods Fellenius and Bishop. For the other calculation methods, F-TOTAL alone is displayed:

- • *F-SOL*: safety factor calculated without taking into account the loads nor the reinforcements;
- • *F-SURCH*: safety factor calculated with loads, but without reinforcements;

- *F-TOTAL*: global safety factor (taking into account loads and reinforcements).

In the case of the perturbations calculation method, 3 additional columns appear: λ and μ, and NSPEQ (number of solutions for the 3rd degree equation).

<u>Note</u>: When a line is selected in the table, the corresponding failure surface is displayed with a bold line on the graphical view.

X0(m)	Y0(m)	Radius(m)	MMOT(kN)	F-SOL	F-SURCH	F-TOTAL
-9.51	299.5	298.15	635212.51	4.689	4.537	4.537
3.48	396.5	395.01	833276.01	4.481	4.357	4.395
16.47	493.5	491.77	956615.95	4.278	4.178	4.323
29.47	590.5	589.23	1140333.7	4.162	4.078	4.230
42.47	687.5	686.81	1324051.3	4.079	4.005	4.162
55.46	784.5	784.46	1507765.4	4.015	3.949	4.111
68.45	881.5	882.16	1691479.1	3.968	3.909	4.082
12.22	25.33	26.33	25905.229	0.621	0.617	1.069
16.43	29.21	31.78	30484.667	0.597	0.594	1.070
20.63	33.1	37.31	35063.969	0.582	0.579	1.077
24.84	36.98	42.9	39643.472	0.572	0.569	1.082
29.04	40.86	48.51	44222.342	0.565	0.563	1.089
33.25	44.75	54.15	48802.248	0.560	0.558	1.095
37.45	48.63	59.8	53381.654	0.556	0.554	1.101
41.66	52.51	65.47	57960.711	0.553	0.551	1.106
45.86	56.39	71.15	62539.779	0.551	0.549	1.110
50.07	60.28	76.83	67119.671	0.549	0.547	1.114
9.87	26.68	26.58	30370.37	0.702	0.693	1.037
13.77	30.88	31.99	35805.993	0.672	0.665	1.031
17.66	**35.07**	**37.49**	**41242.506**	**0.654**	**0.648**	**1.031**
21.56	39.27	43.05	46678.258	0.642	0.636	1.033
25.45	43.46	48.65	52114.297	0.634	0.628	1.037
29.34	47.66	54.27	57550.366	0.627	0.621	1.038
33.24	51.85	59.91	62986.462	0.623	0.617	1.042
37.13	56.05	65.57	68422.249	0.618	0.613	1.045
41.03	60.24	71.24	73858.501	0.615	0.610	1.048
44.92	64.44	76.91	79293.745	0.612	0.608	1.050
7.64	27.76	26.86	34197.499	0.793	0.779	1.051
11.25	32.2	32.22	40342.515	0.757	0.745	1.040

☐ Display all results

OK   Help   Legend for F values

*Figure 112: detailed results for failure surfaces
(example for circular failure surfaces, Bishop calculation method)*

X0(m)	Y0(m)	Radius(m)	MMOT(kN)	F-SOL	F-SURCH	F-TOTAL
9.87	26.68	25.08	19210.475	0.722	0.716	1.297
9.87	26.68	24.58	15954.911	0.735	0.730	1.658
9.87	26.68	24.08	12922.011	0.754	0.752	2.555
13.77	30.88	31.99	35805.993	0.672	0.665	1.031
13.77	30.88	32.49	40917.842	1.591	1.560	2.044
13.77	30.88	32.99	45909.956	1.648	1.613	1.984
13.77	30.88	31.49	31038.243	0.678	0.671	1.190
13.77	30.88	30.99	26550.794	0.686	0.680	1.114
13.77	30.88	30.49	22337.417	0.697	0.692	1.337
13.77	30.88	29.99	18392.946	0.712	0.708	1.776
13.77	30.88	29.49	14711.4	0.735	0.733	3.005
17.66	**35.07**	**37.49**	**41242.506**	**0.654**	**0.648**	**1.031**
17.66	35.07	36.99	35650.872	0.661	0.655	1.199
17.66	35.07	36.49	30381.188	0.670	0.664	1.129
17.66	35.07	35.99	25427.99	0.683	0.678	1.378
17.66	35.07	35.49	20785.307	0.700	0.696	1.888
17.66	35.07	34.99	16447.431	0.725	0.724	3.525
21.56	39.27	43.05	46678.258	0.642	0.636	1.033
21.56	39.27	42.55	40254.654	0.650	0.644	1.211
21.56	39.27	42.05	34195.987	0.660	0.655	1.144
21.56	39.27	41.55	28495.907	0.674	0.669	1.416
21.56	39.27	41.05	23150.025	0.692	0.689	1.996
21.56	39.27	40.55	18151.611	0.719	0.719	4.097
25.45	43.46	48.65	52114.297	0.634	0.628	1.037
25.45	43.46	48.15	44852.957	0.642	0.637	1.221
25.45	43.46	47.65	38000.13	0.653	0.648	1.158
25.45	43.46	47.15	31549.855	0.667	0.663	1.451
25.45	43.46	46.65	25496.84	0.687	0.684	2.097
25.45	43.46	46.15	19834.887	0.716	0.716	4.737

☑ Display all results

OK   Help   Legend for F values

*Figure 113: detailed results for failure surfaces
(example for circular failure surfaces, Bishop calculation method)*

*Figure 114: detailed results for failure surfaces
(example for circular failure surfaces, perturbations calculation method)*

Figure 115: legend for F values

5.3.5. Forces in reinforcements

The table of detailed results for reinforcements can be displayed with the "Calculations and results" menu, option "Forces in reinforcements", or with the ⬛ button of the contextual toolbar. The dialogue box illustrated on Figure 116 is displayed: the left part displays the list of calculated failure surfaces, the right part displays, for the failure surface selected in the left list, the forces and results for all active reinforcements. In the example of the figure hereunder, the project contains nails and anchors, thus these 2 tabs are displayed.

By default, the green highlighted line corresponding to the critical failure surface is selected in the left list.

Note: When a line is selected in the left list, the corresponding failure surface is displayed with a bold line on the graphical view.

The columns contents depend on the reinforcements type (active tab). The button "Legend for reinforcements" on this screen enables to view the legend for the different columns of the table (Figure 117).

Figure 116: detailed results: forces in reinforcements

Figure 117: legend for the reinforcements results

5.3.6. Detailed results for slices

The table of detailed results for slices can be displayed with the "Calculation and results" menu, option "Slices results", or with the ⊞ button of the toolbar. The dialogue box illustrated on Figure 118 is displayed: the left part displays the list of calculated failure surfaces, the right part displays, for the failure surface selected in the left list, the detailed results for each slice (both as a table and as a graphic).

By default, the green highlighted line corresponding to the critical failure surface is selected when opening the dialogue box.

Note:

- If the option to save the detailed results for the critical failure surface only was selected in the situation properties, only the critical surface will appear in the left list (as is the case on the following figure).

- However, if the option to save the detailed results for all calculated failure surfaces was selected instead, all failure surfaces will appear in the left list. The table includes the critical circle 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 in the left table only the circle associated with this minimum.

  The checkbox "Display all results" below the table enables to request the display in the left list of all calculated circles (display can take a long time in this case).

When you select a line in the left list, the corresponding failure surface is displayed with a bold line on the graphical view.

The columns of the table on the right part of the dialogue box have the following meaning:

- *Slice N°*: incremental numbering;
- *DL* (m): slice width;
- *X* (m): abscissa of the slice axis;
- *Y* (m): level of the slice bottom for X position;
- *A* (rad): angle of the slice base compared to the horizontal line;
- *GH* (force unit): slice weight / ml;
- *YGS*: level of the slice gravity centre;
- *IS*: number of the active soil layer at point (X,Y);
- *u* (pressure unit): pore pressure at point (X,Y);
- *UNE* (pressure unit): hydrostatic pressure above the slope boundary in case of an external water level;
- *RDS* (pressure unit): supplement of normal stress due to reinforcements calculated in (X,Y);
- *SIG-TOTAL* (pressure unit): total normal stress calculated in (X,Y);
- *TAU* (pressure unit): shear stress along the failure surface calculated in (X,Y).
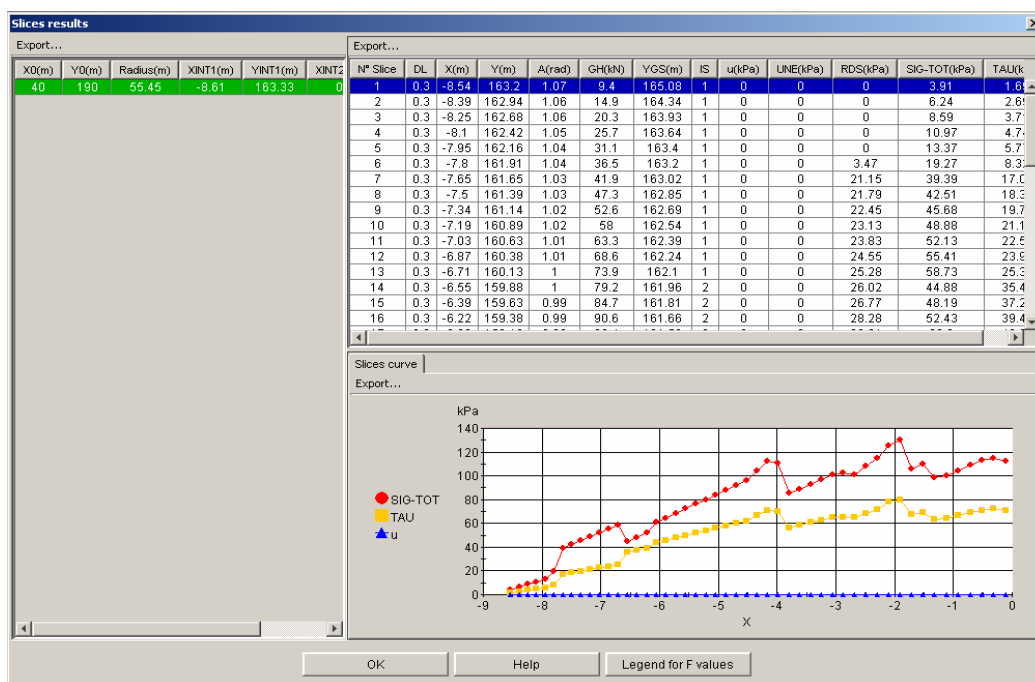
Figure 118: slices results
(case of detailed results saved for critical failure surface only)

6. PRINTING

It's possible to print:

- Only the active stage and situation;
- All situations of the active stage;
- All situations of all stages.

Several printing settings are available: they correspond to various detail levels. The printing settings are valid for all the situations of all selected stages if relevant.
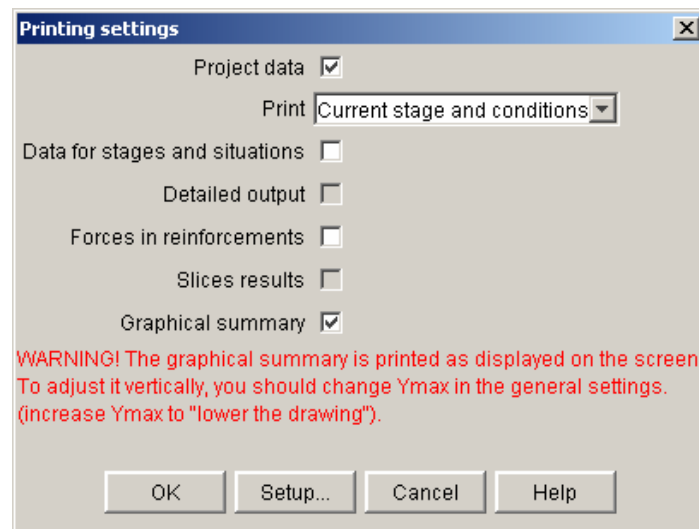
Figure 119: printing settings

The different types of printing are detailed hereunder.

6.1. Project data

This option is the only printing option proposed in the "Project data" mode.

In the "Stages/Calculation" mode, it is activated by default.

This printing summarises all data and assumptions defined by the user in the data mode.

- General settings;
- Soil characteristics;
- Geometry;
- Loads;
- Reinforcements.

6.2. Graphical summary

This option is available only in the "Stages/Calculation" mode.

It provides a page in landscape format with for each situation:

- The project layout, displayed in the same way as on the screen (same zoom level, etc); if the results are available for the printed situation(s), the critical failure surfaces are displayed too;
- The minimum safety factor obtained after calculation (if results are available);
- A bar at the bottom with the software version, title of the project, filename, stage and situation names, printing date and time, company name (from the hardlock).

6.3. Data for stages and situations

This option is only available in the "Stages/Calculation" mode.

The printed document summarises all data and assumptions defined for the printed stage and situation:

- Data for the stage:
  - o Hydraulic conditions;
  - o Active slope boundary;
  - o List of active loads;
  - o List of active reinforcements;
- Data for the situation:
  - o Calculation method;
  - o Partial safety factors;
  - o Seismic conditions;
  - o Description of the failure surfaces;
  - o Summarised results are also printed if available: minimum safety factor and description of the critical failure surface.

6.4. Detailed results

6.4.1. Detailed results for failure surfaces

This option is available only in the "Stages/Calculations" mode, if results are available.

The printed document includes the same table as in the corresponding dialogue box (section 5.3.4).

6.4.2. Results relative to reinforcements

This option is available only in the "Stages/Calculations" mode, if results are available.

The printed document includes the same table as in the corresponding dialogue box (section 5.3.5).

6.4.3. Results detailed by slices

This option is available only in the "Stages/Calculations" mode, if results are available.

The printed document includes the same table as in the corresponding dialogue box (section 5.3.6): the printed table is the one corresponding to the critical failure surface.

6.5. *Printer settings*

When first printing in each Talren 4 session, the following dialogue box appears automatically after validation of the dialogue box represented on Figure 119.

When printing again in the same Talren 4 session, this dialogue box does no longer appear automatically, but it can be displayed manually: click on the "Setup" button (Figure 119).
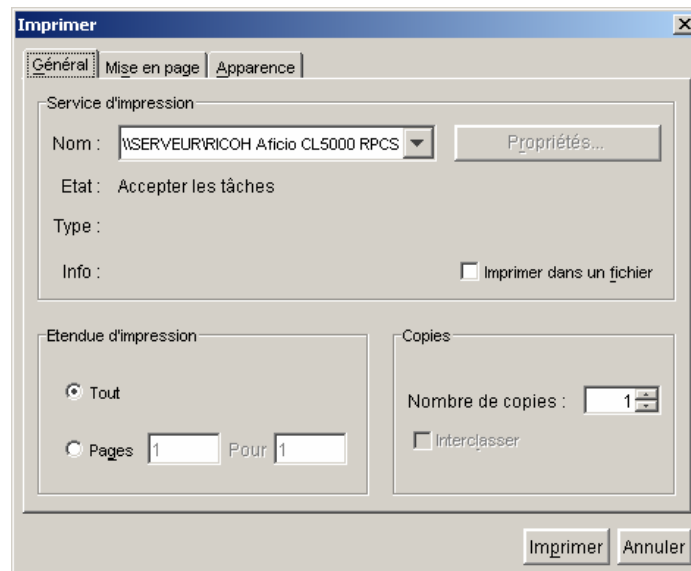
Figure 120: printer settings

This dialogue box enables to setup the printer settings, according to the Windows® environment.

7. FILES HANDLED BY TALREN 4

7.1. Files that can be opened/saved with TALREN 4

The file format associated with Talren 4 is .prj. With Talren 4, it is therefore possible to save and open .prj files.

It is also possible to open (load):

- .tal files saved with Talren 97 (previous version of the Talren software);
- .plx files saved with Plaxis v8.

Select the relevant file extension type in the "open" dialogue box (example Figure 121).
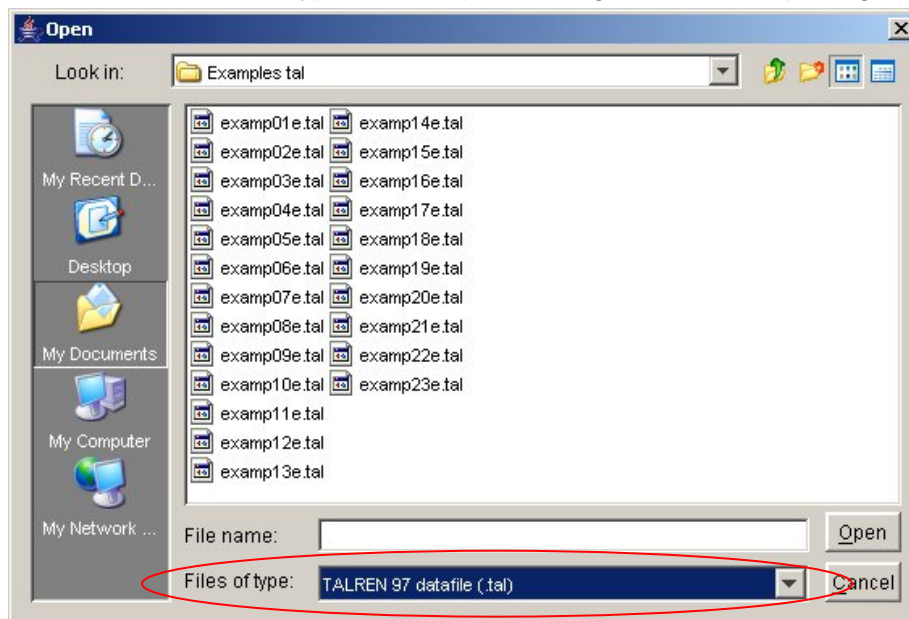
Figure 121: "open" dialogue box

Some specific aspects about loading .tal files and .plx files are underlined in the following sections.

However, TALREN 4 only allows to save one file type: it is the .prj format.

7.1.1. Loading .tal files saved with Talren 97

Talren 4 enables to open (load) data files created with previous versions of Talren 97, but not with the DOS versions (older than Talren 97).

Except for a couple of exceptions (for instance refer to sections 7.1.1.2 and 7.1.1.3), it is possible to load a Talren 97 file, to switch to the "Stages/Calculation" mode, to perform the calculation for the single situation: the result provided by Talren 4 will be identical to the one provided previously by Talren 97.

However, some new options in Talren 4 lead to default choices when loading .tal files saved with Talren 97.

It is therefore strongly recommended for Talren 97 users to read the following sub-sections, in order to know and use the possibilities of the Talren 4 software when reading Talren 97 files.

One can also refer to chapter D of the manual: tutorial 2 describes in a detailed manner how to work with .tal files.

IMPORTANT: The .tal files opened with Talren 4 (menu File/Open, selection of the file extension .tal) can only be saved in the .prj format.

7.1.1.1. General settings

The titles, comments and units are those defined in the .tal file.

The dimensions of the model (X_{min}, X_{max} and Y_{max}) are automatically initialised when reading the geometry of the .tal file. Default calculation method and partial factor sets are also initialised based on the .tal data.

For each .tal file, a special set of partial safety factors called "Coefficients TAL" is automatically created in the local project database: it is filled with the partial factors defined in the .tal file (example on Figure 122).

Also refer to sections 7.1.1.3, 7.1.1.4 and 7.1.1.5 for specific aspects about partial safety factors.
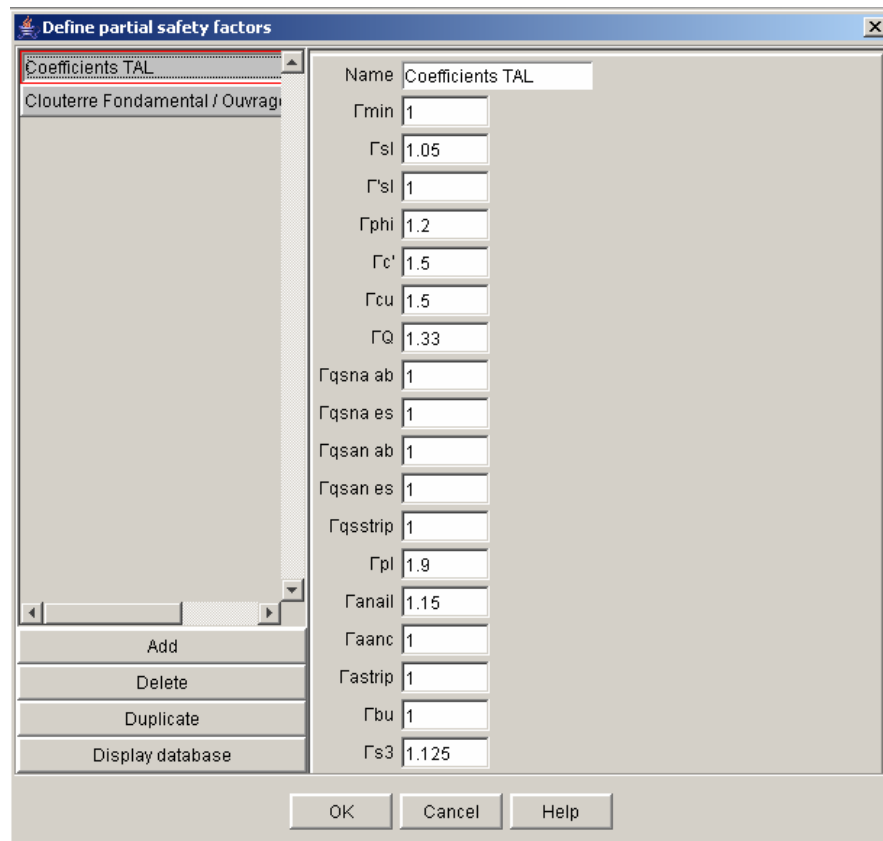
Figure 122: partial factor set "Coefficients TAL"

7.1.1.2. Geometry

When reading .tal files, the slope boundary is never set to "automatic" (because Talren 4 displays the user-defined Talren 97 slope boundary).

Except for specific cases (slope boundary on purpose different from groundlevel), it is recommended to activate the "automatic slope boundary" in the "Geometry" dialogue box (as indicated in the figure hereunder) when first opening .tal files.

IMPORTANT: It is even compulsory to activate the "automatic slope boundary" mode in the case where the user wishes to define additional construction stages for his project, with activation/deactivation of soil clusters (and therefore potentially change of the slope boundary): if several stages are defined and the boundary is not "automatic", the boundary will not automatically change with each stage, and the same initial slope boundary will be taken into account for all calculated phases and situations (which can lead to errors).
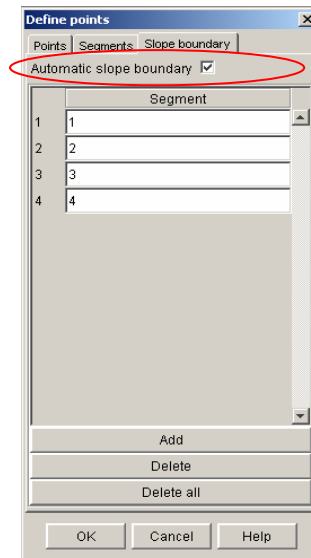
Figure 123: definition of the slope boundary

Generally, when reading a .tal file, no particular problem is raised about the geometry, except if the .tal file contains vertical limits without corresponding segments: take for example the simple case of 2 adjacent horizontal segments, one with soil layer 1 below, and the other with soil layer 2 below, without the vertical limit between both soil layers being defined with a geometrical segment (this was possible in Talren 97, Figure 124).

Reading this kind of file will raise a problem in Talren 4, which needs closed soil clusters to be able to associate them with different soil layers: in the previous example, the "missing" segment should be added manually, and then the soil layers should be assigned to both clusters again.

Example of a Talren 97 file with 2 layers, without a vertical separation line

After importation in Talren 4: Talren was not able to identify the soil clusters

After drawing a vertical line and associating both soil layers
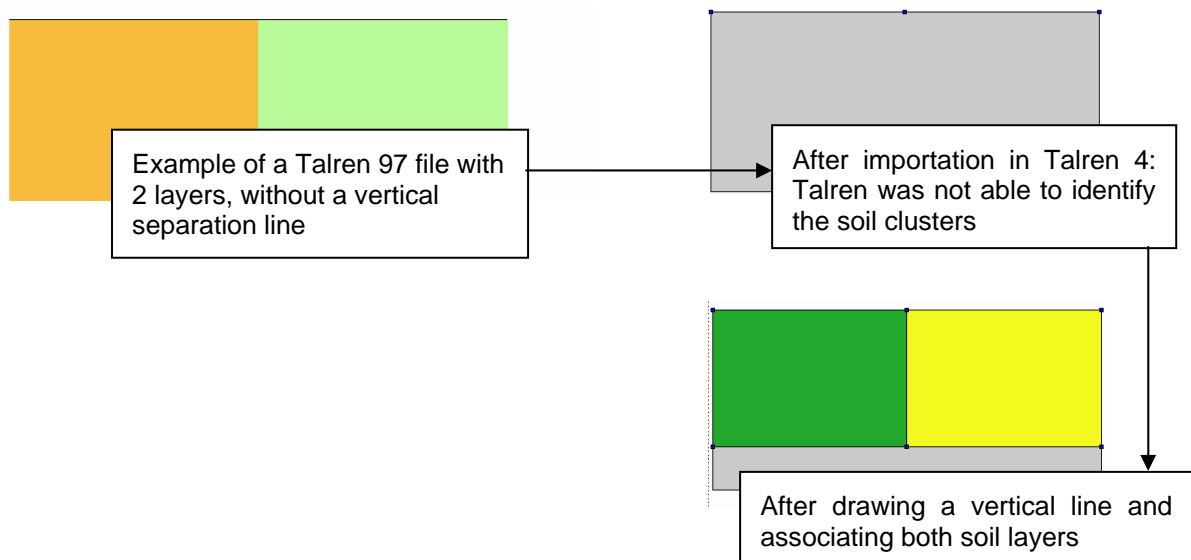
Figure 124: specific case of Talren 97 file reading

Note: the previous comments apply to vertical geometrical lines inside the model: the vertical lines delimiting the model in X_{min} and X_{max} are created automatically in Talren 4 when reading a Talren 97 file (it is not necessary to draw these lines).

7.1.1.3. Loads

In Talren 97, the partial load factors were specific to each load. In Talren 4, this partial load factor is common for all loads (one single value for each set of partial safety factors): it is the value defined in "Coefficients TAL" (section 7.1.1.1).

When reading a Talren 97 file including one or more loads, all defined with the same load factor, Talren 4 will define the load factor of the "Coefficients TAL" set to be equal to this same value, and the Talren 4 file will be equivalent to the Talren 97 file.

However, when reading a Talren 97 file including several loads with different load factors, it is necessary to adjust the load values, to make the Talren 4 equivalent to the Talren 97 file although the same load factor will be used for all loads.

7.1.1.4. Reinforcements

There is no special comment about reinforcements reading, except for the nails and anchors. For the nail data, Talren 4 needs to know whether the unitary skin friction (qs_{nails}) is issued from tests and curves. This datum did not exist in Talren 97. It enables to choose the relevant partial safety factor (in the Clouterre recommendations for example, the partial safety factor for qs_{nails} is not the same whether qs is known from tests or curves).

When reading a .tal file:

- This datum is set to "Curves" (default value);
- In the specific "Coefficients TAL" set generated for the project (section 7.1.1.1), the values of the safety factors for $qs_{nails\ curves}$ and $qs_{nails\ tests}$ are set to the same value.

In this way, the Talren 4 calculations will give the same results as the Talren 97 calculations.

IMPORTANT: if you later change the set of safety factors (and choose for example a Clouterre set), you should make sure to select for each nail whether the qs value is issued from curves or tests, so that when calculating, the relevant safety factor is applied (also refer to section 4.7.2.3 for the description of the partial safety factors).

The same comments apply to anchors, about $qs_{anchors}$.

7.1.1.5. Soil properties

The important point to underline here is the use of "specific partial safety factors" when reading .tal files. Indeed, Talren 97 enabled to define partial safety factors, but not to save them as sets associated with a given standard. Talren 4, when reading .tal files, cannot therefore associate a partial safety factor set (Clouterre for example) with the individual values defined in the .tal file (that's why a specific "coefficients TAL" set is created when reading each tal file, refer to section 7.1.1.1).

Moreover, it was possible in the .tal files to define different partial safety factors for each soil layer. So when reading tal files, specific partial safety factors are activated and defined for each layer: these factors replace for each layer, the global set selected in the situations, and ensure therefore that .tal files will provide the same results with Talren 4 as with Talren 97.
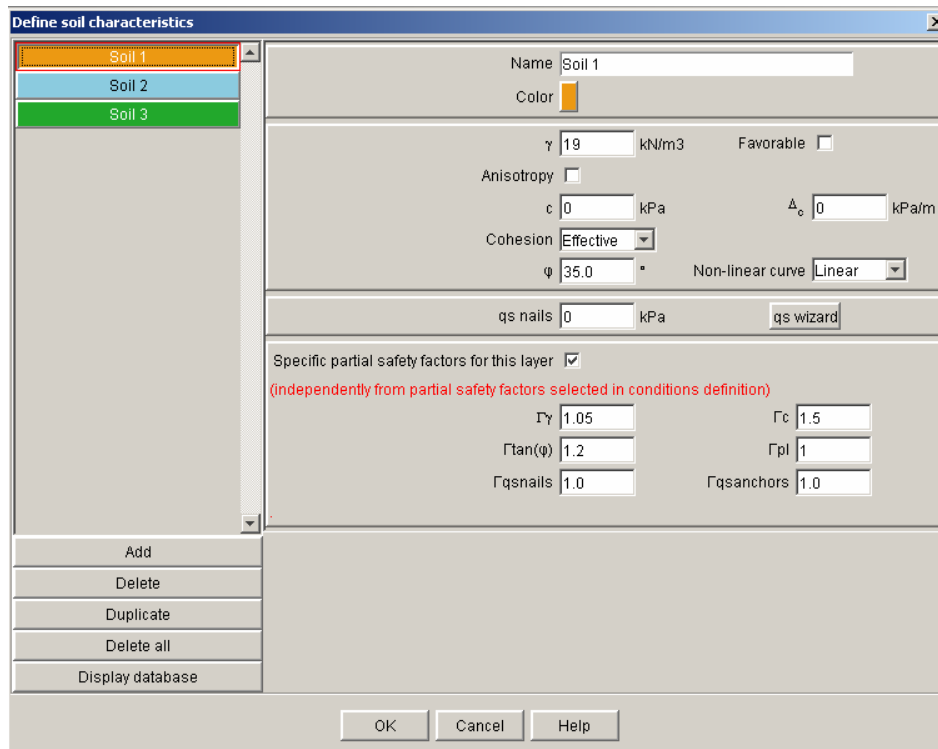
Figure 125: soil characteristics and specific partial safety factors

However, if the .tal file includes the same partial safety factors for all layers, it is possible to deactivate the use of specific partial safety factors for each layer, and to use for all situations the factor set created automatically "Coefficients TAL" (or even another predefined set if relevant or necessary).

When reading a .tal file:

- The cohesion type is set to "Effective" for all layers;

- The action of the soil weight is considered to be unfavourable for all layers;

- In the specific factors for each layer, the partial factors for the cohesion and the soil weight are identical to those that were defined for each layer in Talren 97;

- In the specific "Coefficients TAL" set generated for the project, the values of the safety factors for c' and c_u are the same.

This way, Talren 4 will provide the same results as Talren 97.

IMPORTANT: in the case where you change the safety factors set (and choose for example a Clouterre set), you should make sure to select the right cohesion type for each layer (effective or undrained): this choice will decide which safety factor will be applied. In the same way, you should check if the soil weight is an unfavourable action (default choice) or not. Also refer to section 4.7.2.3 for the description of the partial safety factors.

7.1.1.6. Definition of the construction stages

When reading a .tal file, Talren 4 automatically creates a stage and a situation for this stage, which corresponds to the calculation performed in the .tal file, except in the case of polygonal failure surfaces: in this case Talren 4 automatically creates a stage and as many situations as polygonal failure surfaces defined in the .tal file (there can only be one single polygonal failure surface for each situation in Talren 4).

7.1.1.7. Hydraulic conditions (properties of stage 1)

Phreatic level

A single point was sufficient in Talren 97 to define a horizontal groundwater level.

However, in Talren 4, 2 extremities should be defined for the groundwater level: if in the Talren 97 file, the groundwater level is defined with a single point, Talren 4 automatically adds a 2nd point at the same level when reading the file.

This comment also applies if necessary to the water table base.

External phreatic line

As for the slope boundary (section 7.1.1.2), there is in Talren 4 an "automatic" mode for the external water level, that is deactivated by default when reading a .tal file (because Talren reads the previously user-defined data), but can be activated (it is even recommended, except if the user wishes on purpose to impose an external phreatic level that is different from the one calculated by default).

Hydraulic pore pressure mesh

No special comment.

ru coefficients

In Talren 97, the ru coefficients were defined in the soil characteristics.

In Talren 4, the hydraulic conditions can be changed for each stage, and the ru coefficients are defined (and therefore can be edited) in the hydraulic conditions for each stage (with a table enabling to define ru for each soil layer).

7.1.1.8. Situations: partial safety factors (properties of stage 1 / situation 1 or i)

The "Coefficients TAL" set is selected by default for the situation (section 7.1.1.1).

7.1.1.9. Situations: failure surfaces (properties of stage 1 / situation 1 or i)

There is no special comment about failure surfaces settings.

In the case of polygonal failure surfaces, one single failure surface can be defined for each situation: Talren 4 therefore creates as many situations as polygonal failure surfaces when reading a Talren 97 file.

7.1.2. Reading .plx files saved with Plaxis v8

Talren 4 enables to read Plaxis files created with Plaxis v8 (but not previous files created with previous versions of Plaxis, or with other Plaxis products). These files have the .plx extension (be careful: the plaxis v7 files also have the .plx extension, but can not be loaded with Talren 4).

IMPORTANT: the Plaxis files for a given project include a project.plx file, and a project.DTA directory, which itself includes a copy of the project.plx file. To open the project with Talren 4, you should select the project.plx file that is "outside" the project.DTA directory (and not the one inside this directory).

The data imported when opening a Plaxis data file are the following:

- Project limits (X_{min}, X_{max} and Y_{max}), initialised from the dimensions of the Plaxis model;
- The project geometry (all points and segments, association of soil clusters with soil properties), as defined in the Input module of Plaxis (section 7.1.2.1);
- Some soil properties (section 7.1.2.2);
- Loads, as defined in the Input module of Plaxis (section 7.1.2.3);
- Groundwater conditions from the Plaxis initial conditions.

The staged construction defined in Plaxis is not imported (section 7.1.2.4).

The imported data should be completed (for instance the file number, the project title and the situation properties are missing) before being able to perform a calculation.

After the importation of a Plaxis data file, Talren 4 displays the following warning:
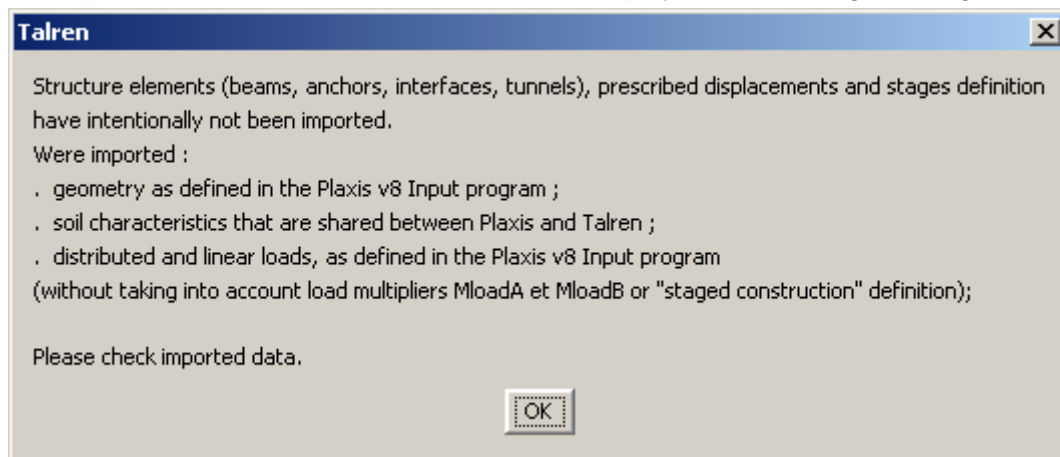

Figure 126: warning displayed when reading a Plaxis v8 file

It is strongly recommended that Plaxis v8 users read the following sections, in order to know and use the possibilities of Talren 4 when reading Plaxis v8 files.

Please also refer to chapter D of the manual: tutorial 3 describes in a detailed manner an example of .plx file reading.

IMPORTANT: The .plx files opened with Talren 4 (menu File/Open, selecting the .plx file extension) can only be saved with the .prj format.

7.1.2.1. Geometry

When reading a Plaxis file, Talren 4 automatically creates vertical segments on the left and right boundariess of the model: indeed, these segments have been defined in Plaxis, and are therefore read in Talren 4 like the others.

These segments are not necessary, but they are no problem either. The user can therefore keep or delete them at his convenience.

On the other hand, the slope boundary is not a Plaxis datum: it is therefore calculated in automatic mode when reading a Plaxis file in Talren 4.

7.1.2.2. Soil properties

The soil characteristics compatible between Talren 4 and Plaxis v8 (in other words read by Talren 4 when opening a Plaxis v8 file) are the following:

- Unit weight: it is the unsaturated unit weight (γ_{unsat}) that is used in Talren 4.
- Cohesion: c_{ref} value. Be careful, the possible linear cohesion variations with depth that can be defined with Plaxis v8 are not read by Talren 4.
- Friction angle (phi).

These 3 data are read for each layer defined in Plaxis v8 (independently from the behaviour laws chosen in Plaxis v8, and other soil parameters possibly defined with Plaxis v8).

7.1.2.3. Loads

No special comment.

7.1.2.4. Construction stages

A single stage is automatically created in Talren (as it is always the case). But no data are imported from Plaxis concerning the stage (neither hydraulic conditions, nor activation/deactivation of soil clusters, loads, etc).

A single situation is also automatically created for this first stage (as it is always the case). No data are imported from Plaxis for the situation.

A Plaxis v8 file behaves therefore as a new file as far as the "Stages/calculation" mode is concerned.

7.2. Other files handled by TALREN 4

7.2.1. Files provided during installation

In addition to the system files (DLL, etc) and the example files, files provided during the program installation (in the Talren 4 installation directory) are the following:

- talren4.jar: it is the Talren 4 "exe" file.
- run.bat: this file should be executed to run Talren 4 in the best conditions (this enables a better optimisation of your computer memory than running directly talren4.jar).
- Properties, fra and ang files: these files include Talren 4 settings (language, hardlock type, etc.) as well as all text resources.
- bd files: these are the global Talren databases (partial safety factor sets, soils and reinforcements databases).

7.2.2. Result files

Several result files are generated by Talren 4 for each calculation (situation).

- –n.prr or -c.prr file: result file in XML format, that contains all the available results for a situation if they exist. This file is created in the same directory as the project file. It has the same name as the .prj file, with the stage and situation number added at the end. For example, the result file for situation 2 of stage 1 of the project.prj file will be named project12-c.prr. The –c notation indicates that the file actually contains up-to-date results, whereas the –n notation indicates that it is not the case.

When sending a Talren 4 project to another user, the files to be sent are:

- o The prj file, which contains all the data;
- o The –c.prr file (optional result file).

7.2.3. <u>Temporary files</u>

Talren 4 handles additional file types linked to the calculation (data and results). You don't have to know these files. They are however listed here.

- .ta2 file: text file necessary to the calculation module, generated by the user interface when the user asks for a calculation. This file is created for each calculated situation. The .ta2 filename is composed of the .prj file name, followed by the number of the stage and the number of the situation. For example when calculating situation 3 of stage 2 of the example.prj file, the ta2 file is named example23.ta2. This file is created in the same directory as the project file.

- talren.ava file: file generated by the calculation module, and allowing the interface to know the calculations progress (display of the gauge during the calculation). This file is created in the same directory as the project file. It is unique in each directory: It is therefore overwritten each time a new calculation is asked for in the same directory.

- talren.fic file: text file used as a link between the user interface and the calculation module. This file is generated in the installation directory of Talren 4 at the beginning of each calculation.

- .re1 file: formatted text file generated by the calculation module and listing calculation data. The filename is the same as ta2 filename. The .re1 file is created in the same directory as the project file.

- .res file: same comments as for the .re1 file. The .res file contains the detail of the calculation results for failure surfaces. These results are displayed in the "Results for failure surfaces" dialogue box.

- .ren file: same comments as for the .re1 file. The .ren file contains the detail of reinforcements results. These results are displayed in the "Forces in reinforcements" dialogue box.

- .tra file: same comments as for the .re1 file. The .tra file contains the details of the slices results (this file can therefore be very big: often several Mb). These results are displayed in the "Slices results" dialogue box.

- .trb file: same comments as for the .re1 file. The .trb file is specific to the yield design calculation method.

- -n.rrp or –c.rrp file: same comments as for the .re1 file. It is a temporary result file (these files are deleted when the project is saved and/or closed).

- .err file: this file contains relevant error messages generated by the calculation module (it is almost always empty). These error messages, if any, are displayed in the calculation window: the user therefore automatically views these messages. The name of the .err file is the same as for the ta2 file. The .err file is created in the same directory as the project file.