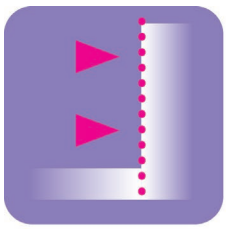


K-Réa v4

B. USER'S MANUAL

TABLE OF FIGURES

TABLE OF TABLES

B.1. Introduction

B.1.1. Presentation of K-Réa

K-Réa is used to study the behaviour of planar or circular retaining walls subjected to a series of construction phases, using the reaction coefficients calculation method.

K-Réa analyses two types of projects:

- "**Single wall"** projects: for a single retaining wall;

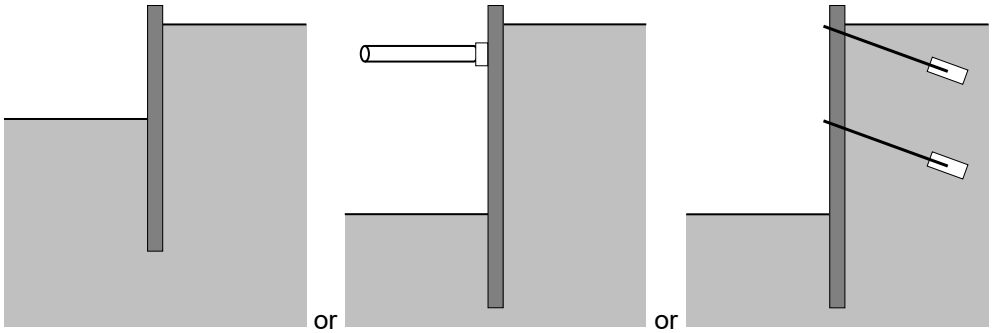


Figure B1 : Examples of "single wall" projects

- "**Double wall"** projects: for two walls joined by a set of anchors.
  Note: in this manual the term *double wall* refers to both double walls and back walls.

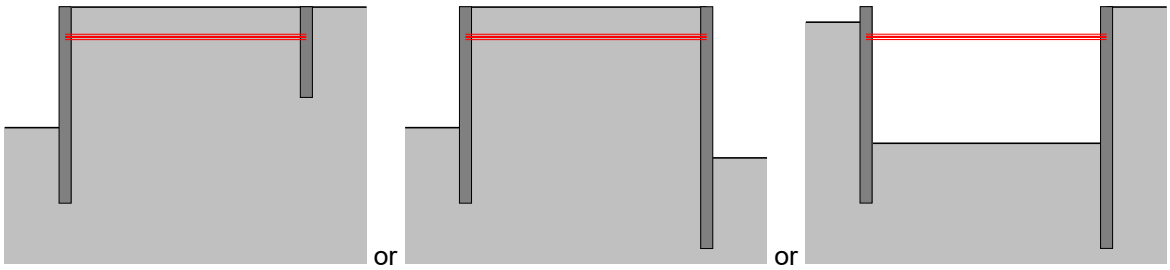


Figure B2 : Examples of "double wall" projects

K-Réa runs in a Windows® environment, which makes the interface intuitive and particularly simple to use for data input. In addition, the graphical display of the calculation phasing allows "real time" monitoring of the input of validated actions.

The main steps in the definition of a project are as follows:

1. Define general project data: project type (single wall or double wall), units system (metric or imperial), hydraulic options, consideration of 2nd order moments and activation of ULS checks;
2. Define characteristics of soil layers (numerous wizards are available);
3. Define wall properties (several wizards are available);
4. Define calculation phasing: creation of phases and choice of actions in each phase (excavations, installation of anchors, etc.);
5. Start calculations;
6. Display results.

Part A of the manual is devoted to installation of the K-Réa software.

This part B presents all the interface elements and summarises the operations to be performed for defining and exploiting a K-Réa project.

Part C of the manual is devoted to the technical manual defining the theoretical framework of the calculation methods used in K-Réa.

Part D gives application examples, tutorials and details K-Réa interface operations.

B.1.2. Conventions

B.1.2.1. Units

Two units systems are available in K-Réa: the metric system and the imperial system. The metric system can itself be broken down into 3 subsystems based on kN, MN or t. The units system is chosen for each project in the **Data, Title and Options** Menu.

The K-Réa calculations are performed for a unit length of wall, so virtually all the data and results are relative to this convention. The /ml (per linear metre) or /lft (per linear foot) unit is explicitly recalled for all the parameters defined per unit length (data and results). If this suffix (/ml or /lft) is absent, this means that the parameter in question is not expressed in terms of unit length.

The following table gives the correspondences for each type of physical quantity between the "metric (kN)" and "imperial" units systems. The (/ml) or (/lft) suffix is also added to the physical quantities expressed per unit of length or surface.

Physical quantity	Unit in the metric system	Unit in the imperial system
Distance or elevation	m	Ft
Displacement	mm	In
Force	kN/ml	kip/lft
Pressure	kN/m/ml	kip/ft/lft
Unit weight	kN/m^3	Kcf
Product of inertia	kNm^2/ml	$kip.ft^2/lft$
Linear stiffness	kN/m/ml	kip/ft/lft
Surface stiffness	$kN/m^2/ml$	ksf/lft
Moment	kN.m/ml	kip.ft/lft

Table B 1: Correspondence between metric and imperial units systems

In this manual, the units corresponding to each type of data or result will be given in "metric (kN)" system and "imperial" system. However the "metric (MN)" and "metric (t)" systems are also available in the software.

B.1.2.2. Double wall projects

For double wall projects, K-Réa uses the notation **Wall 1** to designate the **left-hand** wall and the **Wall 2** to designate the **right-hand** wall.
For wall 1 it is advisable to define that which will have the longest construction phasing. This is not an obligation, but a simple recommendation for optimising the operations during definition of the phasing.

For a double wall project, each side of each wall is identified by convention using the following notation:

- "**Left"** or "**Right/E.2R"** for wall 1 (left-hand wall);
- "**Left/E.2R"** or "**Right**" for wall 2 (right-hand wall).

B.1.2.3. Sign conventions

These are described in part C of the manual.

B.1.2.4. Conventions concerning phasing actions

All the phasing actions are graphically represented in the K-Réa interface. What is taken into account in the calculation therefore corresponds to what is represented on the screen.

All K-Réa projects consist of an initial phase and a set of standard phases. As indicated by its name, the initial phase is used to define the status of the soil prior to any interaction with the wall; in other words, this is a stress and resistance initialisation phase. The actions applicable during this particular phase are listed below:

- Hydraulic action;
- Excavation and Backfill actions;
- The reduced earth pressure action, which is in fact only applicable in the initial phase;
- The Caquot Overload and Boussinesq Overload actions.

During the standard phases, a large number of actions is proposed. These are placed in action groups, each dealing with a particular aspect of the problem being studied.

These actions are summarised in the following table:

Group	Actions
Hydraulic	Hydraulic action (1)
Earthworks	Excavation (1) Fill (1) Installation of sheeting (soldier pile) (2)
Soil properties	Redefinition of soil layers(3) Imposed pressure diagrams (4)
Wall properties	Modification of wall stiffness(5) Modification of wall structure (5)
Anchors	Tie (6) Strut(6) Circular waling (6) Clamping(7) Surface support (6) Linear anchor (8) Slab anchor (8)
Loads – Forces - Moments	Caquot overload (6) Boussinesq overload (6) Graux overload (6) Line force (9) Linear moment (9) Horizontal load (9)

Action (X) is applicable:
(1) once per phase / wall / side. In a given phase, the "Excavation" and "Fill" actions cannot coexist on the same side of the wall;
(2) only on the wall sections concerned by the "reduced active earth pressure" action defined in the initial phase;
(3) once per phase / wall / side / soil layer;
(4) per phase / wall / side as many times as necessary;
(5) per phase / wall as many times as necessary;
(6) per phase / wall / side as many times as necessary. This action can also be modified and/or deactivated during phasing;
(7) per phase / wall as many times as necessary. This action can also be modified and/or deactivated during the course of phasing;
(8) per phase as many times as necessary. It is only accessible in double wall projects. This action can also be modified and/or deactivated during the course of phasing;
(9) per phase / wall as many times as necessary. This action can also be modified and/or deactivated during the course of phasing.

All the actions are presented in detail in chapter B.5.

B.1.3. K-Réa v4 data file extension

The extension of the K-Réa v4 data files is **.K4P.** Only this file is necessary when you wish to exchange your calculation data with another K-Réa v4 user.

K-Réa v4 can be used to read and import K-Réa v3.1 projects. The extension of the K-Réa v3.1 data files is **.K3P**.

K-Réa v4 can also be used to read and import K-Réa v3.0 projects. The extension of the K-Réa v3.0 data files is **.KRP**.

B.2. General presentation of interface

B.2.1. Start window

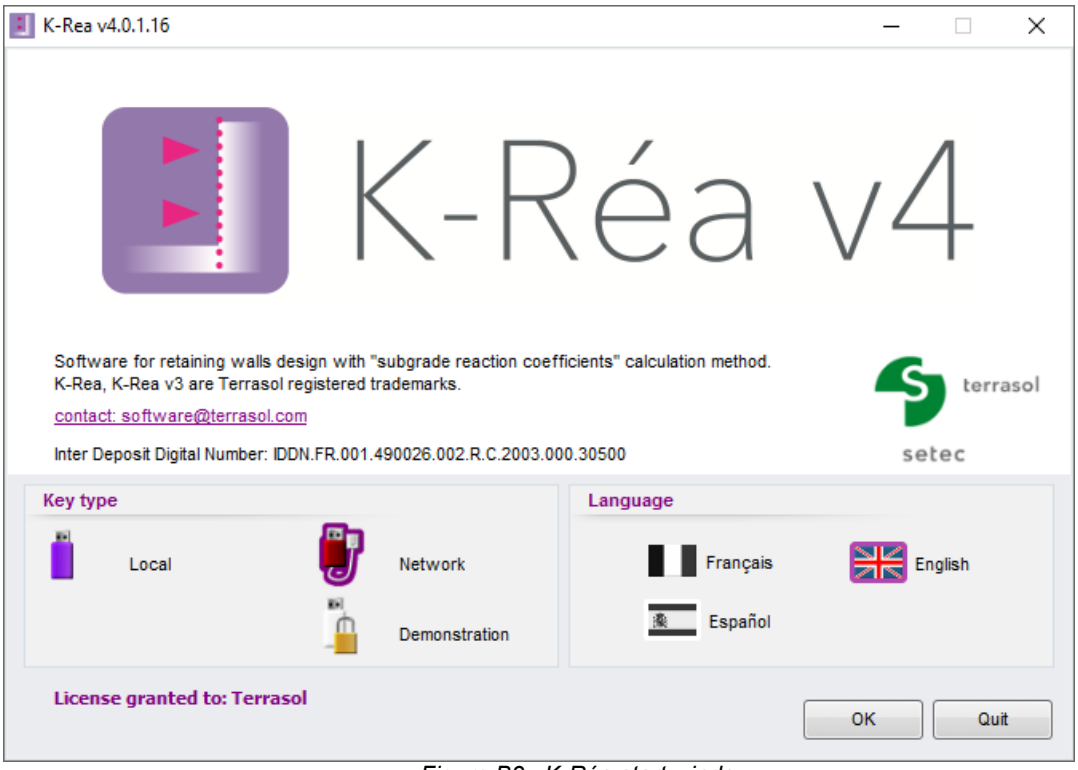


Figure B3 : K-Réa start window

The start window is used to:

- Choose the language to be used by the K-Réa interface;
- Select the type of licence you have. If you do not have a licence, you will only be able to access the demonstration mode;
- Start the K-Réa software;
- Send an email to the Terrasol Software Departement;
- Access the Terrasol website (click the Terrasol logo).

The installed version of K-Réa is also indicated.

B.2.2. Main window

The main window of K-Réa is used to access all the functions available for defining a project. The secondary windows correspond to the data input windows, wizards or results.

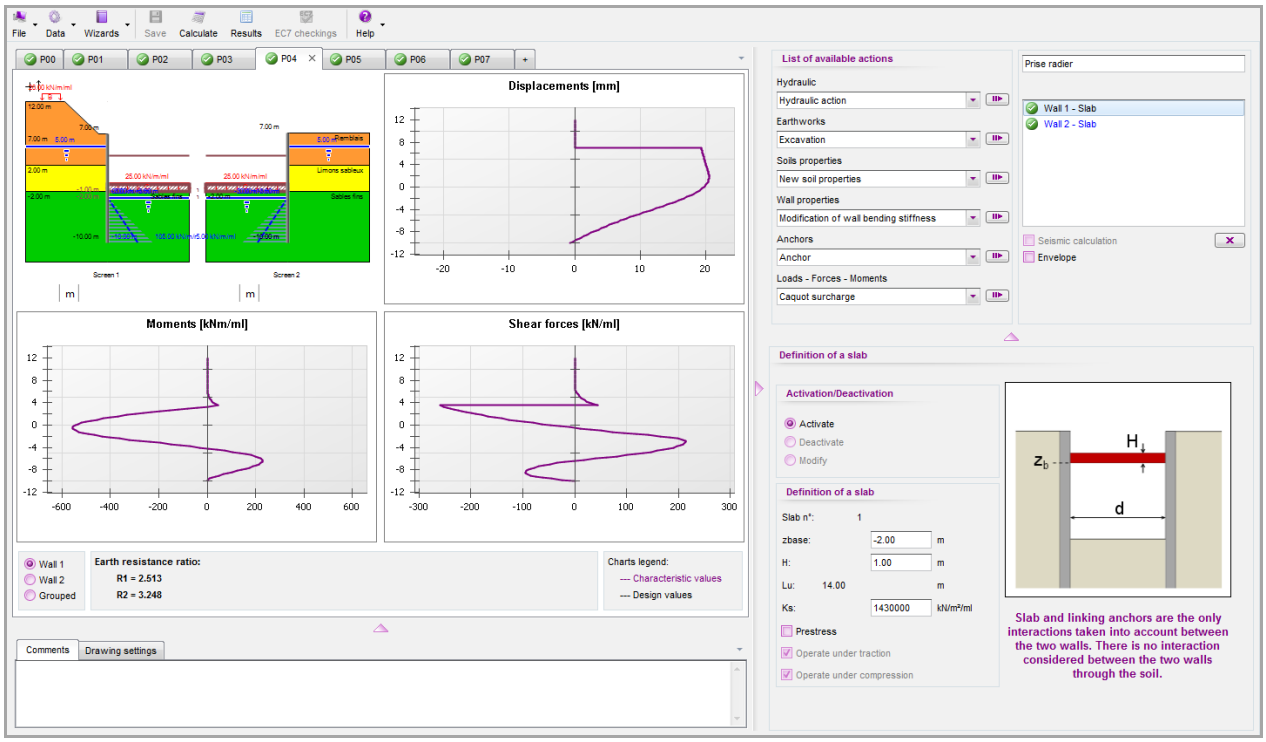


Figure B4 : Main window

The following are displayed in the main window:

- The title bar specifying the project name and file path;
- The menu bar, described in detail in chapter B.2.3;
- The buttons bar and menu lines, presented in chapter B.2.4;
- The project cross-section, presented in the form of a tab per phase;
- The phasing management frame, detailed in chapter B.4;
- A "Comments" tab, which will be printed in the graphic summary of the phase (1 comment per phase);
- A "Drawing settings" tab, used to display the characteristics of the actions. This tab concerns the entire project;
- The unit and elevation (or depth), corresponding to the position of the mouse when it is on the drawing of the project cross-section, shown at the bottom left of the graphic.

B.2.3. Menus

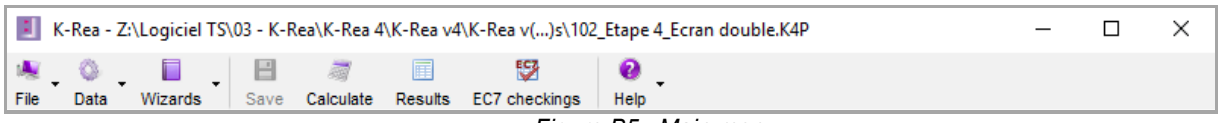


Figure B5 : Main menu

The menus are accessible simply by clicking their title and can call up sub-menus. They are used to manage all the functions linked to the Windows® environment and those specific to K-Réa.

B.2.3.1. File menu

This menu allows access to various file and print options.

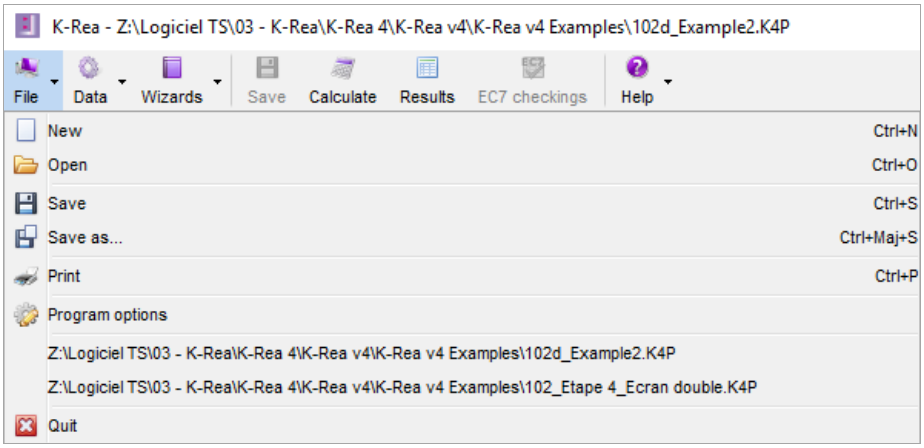


Figure B6 : File menu

The sub-menus are described below:

- **New** : define a new project by accessing the first K-Réa dialogue box (Title and Options).
- **Open** : access the operating system tree to select an existing project to be opened. The project file extension in K-Réa v4 is ".K4P". It is also possible to import projects created with older versions 3 and 3.1 of K-Réa, for which the project files carry extensions ".KRP" and ".K3P" respectively, by changing the extension filter (drop-down list at bottom-right of window).
- **Save** : save the data input in the file corresponding to the current project.
- **Save as** : save the data input in a file other than that activeoine. The name given to the new file must comply with Windows® writing format.
- **Print** : access the print dialogue box. This function is only accessible if the project has already been calculated.
- **Program options** : display the default settings: the folder in which the project files are saved and the unit system:

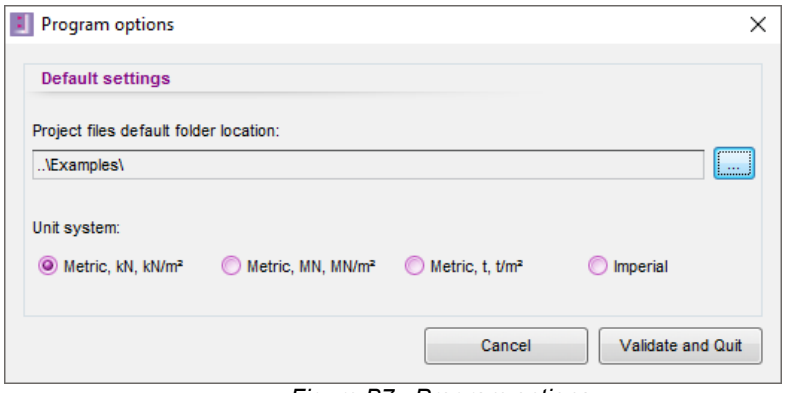


Figure B7 : Program options

- **History**: this display groups shortcuts for the last five projects opened. Direct access is only possible provided they have not been moved or deleted.
- **Quit** : quit current project.

B.2.3.2. Data menu

This menu gives access to the dialogue boxes defining the project data: the project type (single wall or double wall), the calculation options, the characteristics of the soils and of the wall(s). The content of these boxes and their use are described in detail in chapter B.3.

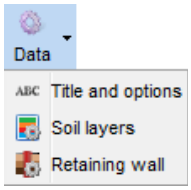


Figure B8 : Data Menu

- **Title and Options** : choice of the project type, input of a title and definition of the calculation options (units, calculation pitch, ULS checks, 2nd order moments, advanced options, etc.).
- **Soil layers definition** : definition of the characteristics of the soil layers (intrinsic parameters, characteristics of the soil-wall interaction). In the case of double wall projects, a "soil model" is allocated to each wall so as to be able to differentiate between the layers specific to wall 1 and those specific to wall 2.
- **Retaining wall definition** : definition of the characteristics of the wall(s) (dimensions, properties).

B.2.3.3. Wizards Menu

This menu is used to access the wizards dialogue boxes to determine the active/passive earth pressure coefficients and the reaction coefficient, as well as to define the load cases. The content of these boxes and how they are used are described in detail in paragraphs §B.3.2 and §B.3.5.

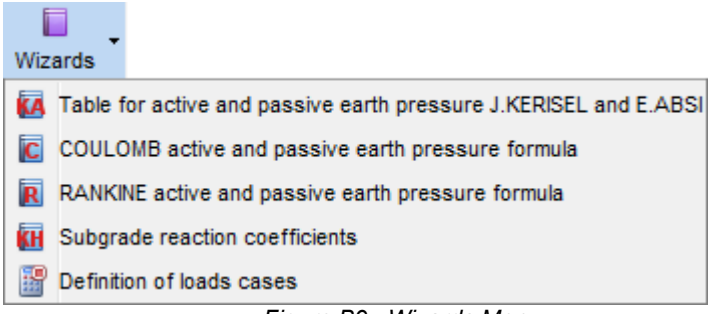


Figure B9 : Wizards Menu

- **Table for active and passive earth pressure J. KERISEL and E. ABSI** : determine the active and passive earth pressure coefficients as a function of the internal friction angle and the characteristic values needed for reading of the J. Kérisel and E. Absi tables (see § B.3.2.2 and part C of the manual);
- **COULOMB active and passive earth pressure formula** : calculate the active and passive earth pressure coefficients using the sliding wedge method (see § B.3.2.2 and part C of the manual);
- **RANKINE active and passive earth pressure formula** : calculate the active and passive earth pressure coefficients using the Rankine formula (see § B.3.2.2 and part C of the manual);
- **Subgrade reaction coefficients** : evaluate the reaction coefficient using the Balay or Schmitt formulas or by reading the Chadeisson curves (see § B.3.2.4 and part C of the manual);
- **Definition of load cases** : create load families used to examine the influence of several load combinations in a single project (see § B.3.5 and part C of the manual).

B.2.3.4. Calculate/Results/EC7 checks Menus

The 3 menus enable to start the calculation and explore the results.

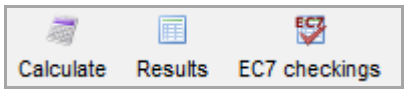


Figure B10 : Calculation/Results Menu

This menu is used to manage the calculations and results.

- **Calculate** : start calculation of the whole project;
- **Results** : open a window containing the summary of results and the envelope curves, preceded by a summary of the data and followed by results per phase in the form of graphics and tables;
- **EC7 checks** : open the EC7 checks if they were activated in "Title and Options". Three types of checks are available: passive earth pressure safety check, vertical equilibrium and stability of the anchoring block (Kranz).

B.2.3.5. Help Menu

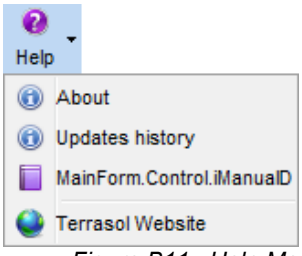


Figure B11 : Help Menu

This menu is used to access the following options:

- **About** : gives information about the software (the version number in particular) and access to information about the system;
- **Updates history** : opens the file logging the various software updates;
- **Terrasol website** : link to the Terrasol website where the manuals are available in the "Downloads" section.

B.2.3.6. Phasing management

Phasing is managed using the context menu on the tabs of each calculation phase (right click.

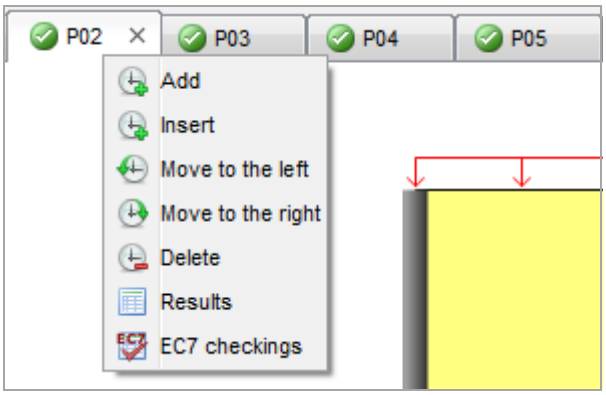


Figure B12 : Phasing management Menu

- **Add:** creates a new calculation phase after the last phase created. It is also possible to add a phase by clicking the tab to the right of the last phase ;
- **Insert**: inserts a new calculation phase before the phase selected;
- **Move to the right**: modifies the calculation phases sequence by moving the selected phase forward by a single position in relation to the others;
- **Move to the left:** modifies the calculation phases sequence by moving the selected phase backward by a single position in relation to the others;
- **Delete**: deletes the selected calculation phase after confirmation;
- **Results**: opens the results window for the selected phase, accessible if the project has already been calculated;
- **EC7 cheks**: opens the checks window for the selected phase;

These modifications are applied in the context of phasing management presented in chapter B.4.2.

B.2.3.7. Keyboard short-cuts

Some of the menu options previously described can be accessed directly. The following list summarises all the keyboard shortcuts available in K-Réa (some of them are also explained in the menus):

- **Ctrl+N**: creates a new project;
- **Ctrl+O**: opens a file to be selected from the file explorer;
- **Ctrl+S**: saves the current file;
- **Ctrl+Shift+S**: saves the current file under a new name;
- **Ctrl+P**: opens the print wizard dialogue box;
- **Ctrl+Q**: starts calculations;
- **Ctrl+A**: interrupts calculations;
- **Ctrl+R**: opens the results window.

B.2.3.8. Context menus

These additional menus are not permanently displayed in the K-Réa main window. Right-clicking certain zones calls them up.

Context menu for graphics and tables

Each results graphic or table has a specific context menu. This context menu is accessible by right-clicking the graphic or table in question.

A right click inside the frame of a graph calls up an export wizard for exporting the graphic in image format:

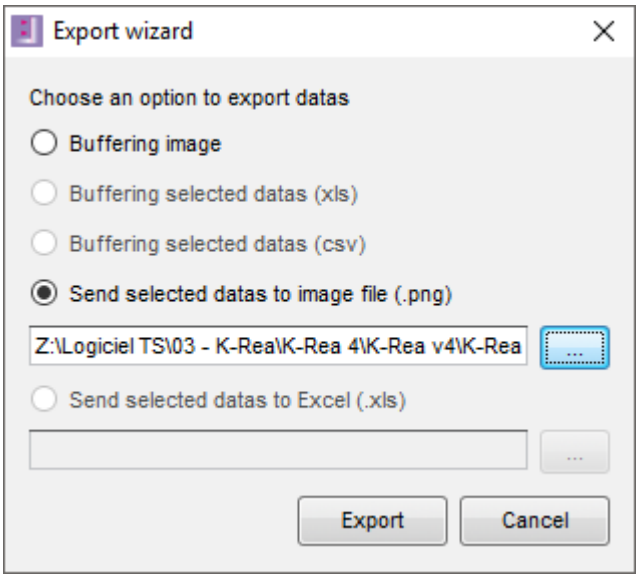


Figure B13 : Help Menu

- **Copy to clipboard as an image:** copies the graphic in image format to the Windows® clipboard so that it can be pasted into a document (Microsoft Excel®, Microsoft Word®, etc.);
- **Save as image (.png)**: creates an image file with extension ".png" in your environment.

Note: the other export formats are shaded because not applicable to graphics. They are functional when exporting tables.

Right clicking in a table (see figure below) calls up a table export wizard:

- **Copy to clipboard (xls)** ⇨ copies the selected table in Excel format to the Windows® clipboard so that it can be pasted into a document (Microsoft Excel®, Microsoft Word®, etc.);
- **Copy to clipboard (csv)** ⇨ copies the selected table in CSV format (values separated by decimal points) to the Windows® clipboard so that it can be pasted into a document (Microsoft Excel®, Microsoft Word®, etc.);
- **Save as Excel file (.xls)** ⇨ creates an Excel file with extension ".xls", containing the selected data.

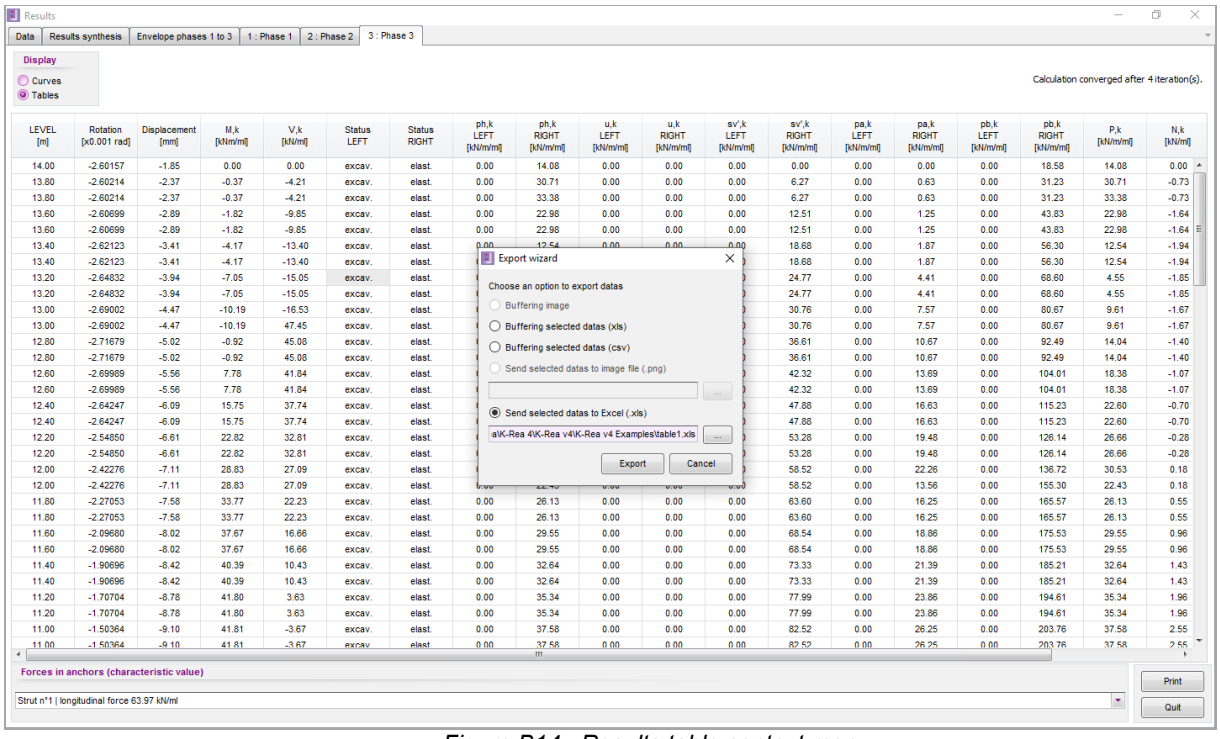


Figure B14 : Results table context menu

Summary tables context menu

The same menu as described above appears when right clicking on the table in the data tab, where it is possible to export the definition of soil layers. The copy to clipboard option is also accessible for the wall definition table.

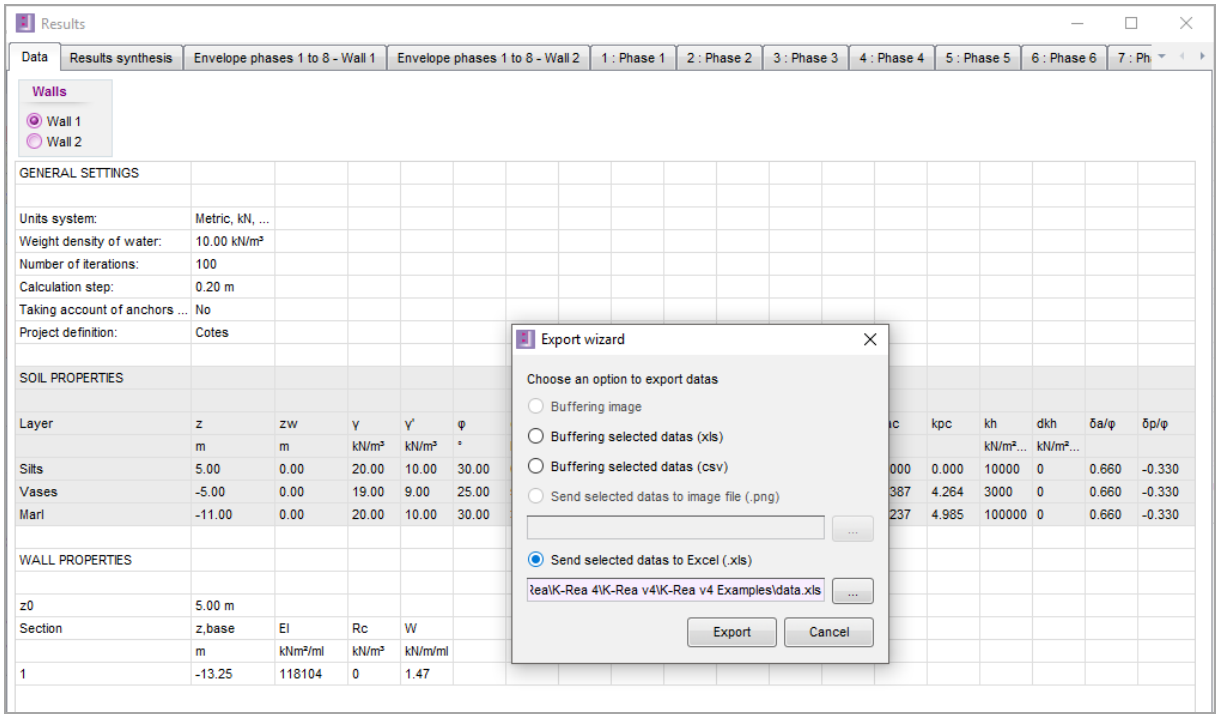


Figure B15 : Soil layers definition table context menu

B.2.4. Buttons bar and menu items

Each menu item has an icon illustrating the corresponding action. They are listed below:

Buttons corresponding to the items of the **File** menu  :

 Initialises creation of a new project;

 Opens an existing project from file explorer;

 Saves changes to current project;

 Saves current project under a new name and/or path;

 Opens print dialogue box (only accessible if the project has already been calculated);

 Opens the "Program options" dialogue box (default choice of project files folder and unit system);

 Closes current project.

Buttons corresponding to items of **Data** menu  :

 Opens the "Title and Options" dialogue box;

 Opens the "Soil layers" definition dialogue box;

 Opens the "Retaining wall" definition dialogue box.

Buttons corresponding to items of **Wizards** menu  :

 KERISEL and ABSI active and passive earth pressure coefficients wizard;

 COULOMB active and passive earth pressure coefficients wizard;

 RANKINE active and passive earth pressure coefficients wizard;

 Subgrade reaction coefficients wizard;

 Load cases wizard.

Buttons corresponding to items of Help menu  :

 Opens the "About" K-Réa window;

 Opens the updates history;

 Opens the K-Réa manuals;

 Opens the Terrasol website.

Buttons

 **Save**: saves the current project;

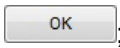
 **Calculate**: starts the calculations for the current project;

 **Results**: displays the project results (only accessible if the project has already been calculated);

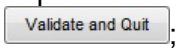
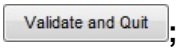
 **EC7 checks**: displays the results of the Eurocode 7 checks (only accessible if the project has already been calculated), if they were requested upstream in the "Title and Options" window.

B.2.5. Overall procedure for defining a new project

This paragraph summarises the procedure to be followed for defining data, running the calculation and viewing the results of a K-Réa project:

1. Start K-Réa and select the type of licence to be used: Local, Network or Demonstration. Demonstration mode gives access to all K-Réa functions but does not enable a calculation to be started;
2. Click  ;
3. Choose **New Project**;
4. In the **Title and Options** dialogue box, define the type of project (**Single wall** or **Double wall**) and fill out the general project options. If necessary, activate consideration of 2nd order moments, ULS checks and define the corresponding parameters. Then click  ;
5. Save the project.

If a Single wall project was selected:

6. In the **Soil layers** definition dialogue box, define the parameters of the various soil layers using the available wizards. These data can be imported from the general soils database or conversely saved in the database, then click:  ;
7. In the **Retaining wall** definition dialogue box, define the characteristics of the wall using the wizard, then click:  **;**
8. Apply the actions of the initial phase via the **Choice of actions** frame. Once the action has been correctly defined, its graphical representation appears in the phasing management tab. At the same time, the action is given a green tick in place of the red cross, the behaviour of the associated phase tab is the same if all the actions have been correctly defined;
9. Click tab  in the phasing management frame to add a phase. Apply the required actions to the new phase via the **List of available actions** frame**;**
10. Repeat the previous step until the final phase is performed;
11. Start calculations by clicking the **Calculate**  button on the buttons bar;
12. Finally, click the **Results**  button on the buttons bar to view the results. They contain a synthesis, envelopes and results detailed per phase. If the ULS checks were activated in "Title and Options", the results of these checks can also be accessed via button 

If a Double wall project was selected:

The principle is the same as for a Single wall project. By convention;

- Wall 1 corresponds to the left-hand wall, its actions are shown in black in the actions definition frame;
- Wall 2 corresponds to the right-hand wall, its actions are shown in blue in the actions definition frame.

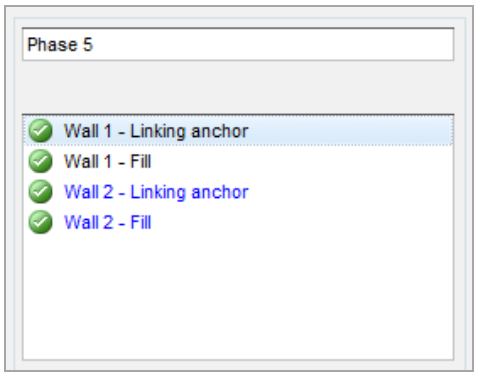


Figure B16 : Double wall project: List of actions

6. In the **Soil layers** definition dialogue box, fill out the soil model parameters for wall 1, using the available wizards. These data can be imported from the general soils database or conversely saved in the database;
7. Choose tab "Wall 2", define the soil model for wall 2 repeating the same approach as followed for wall 1, or click Import model to import the soil model of wall 1, then click Validate and Quit;
8. In the **Retaining wall** definition dialogue box, define the characteristics of wall 1 using the wizard if necessary;
9. Choose tab "Wall 2", do the same as for definition of the characteristics of wall 1 or click Import model to import the properties of wall 1 to those of wall 2, then click Validate and Quit;
10. Phasing management follows the same procedure as for a single wall (add a phase by clicking tab +). The only difference lies in the definition of actions, each of which also needs to be assigned to one of the two walls;
11. Start calculations by clicking the **Calculate** button on the buttons bar;
12. Finally, click the **Results** button on the buttons bar to consult the results. On each of the tabs in the results window, it is possible to switch between the wall 1 and wall 2 results.

This summarised procedure is described in detail in the rest of this document and through the tutorials provided in part D of the manual.

B.2.6. Operation of the updater

By default, the K-Réa shortcut activated at initial installation of the software starts **K-Réa (TerrasolUpdater)**.

The Updater allows for automatic updating of the K-Réa software. The TerrasolUpdater utility automatically logs into an updates server to check whether a new version of the K-Réa software is available. If it is, it prompts the user to update their software and they may then either accept or refuse.

- If they accept, the Updater downloads and installs the update then automatically starts the K-Réa software;
- If they refuse, the Updater does not install any update and automatically starts the K-Réa software.

Finally, if no new update is available, the Updater automatically starts the K-Réa software.

If a user is several updates late when starting the Updater, they are all proposed in turn. If the user refuses a given update "n", the following ones (n+1, etc.) are not proposed, until update "n" has been accepted.

When no internet connection is available, or the various security gateways prevent access to the updates server, the Updater does not appear and starts the K-Réa software directly.

B.3. Project data

Data input, except for phasing, is performed through 3 dialogue boxes: **Title and Options**, **Soil layers** definition and **Retaining wall** definition, all accessible from the Data menu. An additional dialogue box, in the Wizards menu, can be used for **Load cases definition**.

Chapters B.3.1 to B.3.3 describe the working of the Data menu dialogue boxes using the case of a single wall project.

Chapter B.3.4 gives the input particularities of double wall projects.

Chapter B.3.5 is devoted to the definition of load combinations.

B.3.1. Title and Options

B.3.1.1. General options

The project general options are to be defined in the **Title and Options** dialogue box accessible from the **Data** menu.

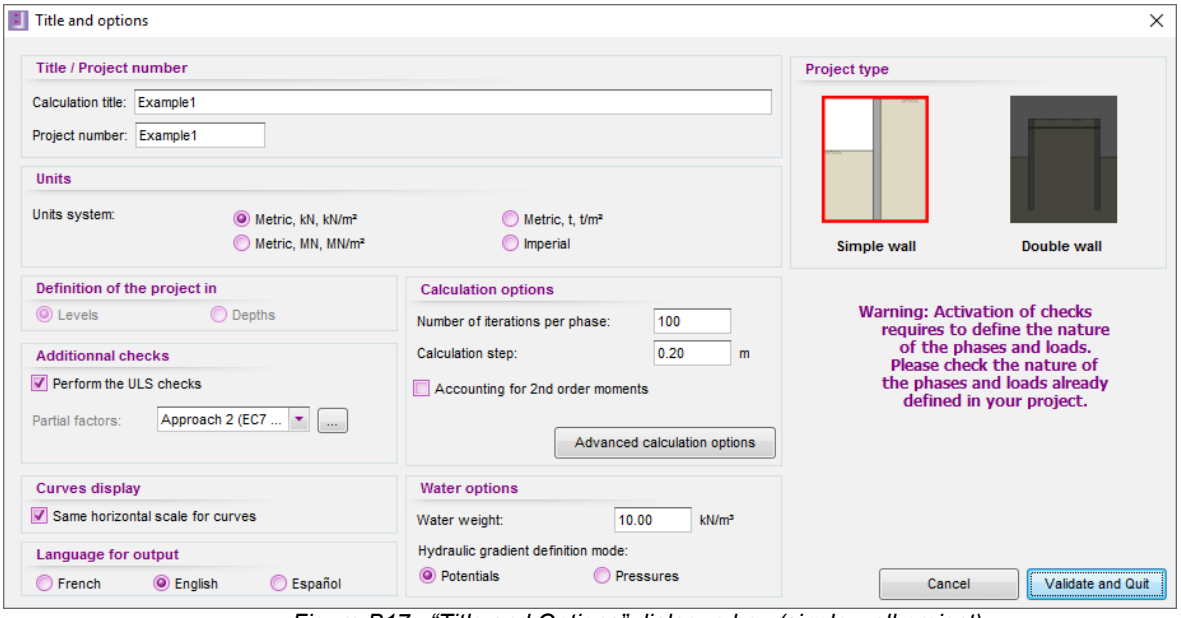


Figure B17 : "Title and Options" dialogue box (single wall project)

This dialogue box is divided into frames presented below:

- **Title / Project number:** this zone is used to input a Title and a project reference (filling out not mandatory);
- **Units:** it is possible to work in the metric system (kN, MN or t) or the imperial system. The units chosen are valid for the input data and results. If the project input data already exist, K-Réa converts these values to the new system of units;
- **Definition of the project in:** this option is used to orient the vertical axis upwards "Levels" or downwards "Depths". It is valid for all project levels and can no longer be modified once the project has been created. The "Levels" option is selected by default;

- **Additional checks**: this frame is used to activate the ULS checks, in particular concerning the passive earth pressure safety checks, vertical equilibrium and stability of the anchoring block (Kranz). The set of partial factors chosen by default corresponds to the 2/2* approach of standard NF P94-282, although the sets resulting from the other Eurocode 7 approaches are also available. Modification of an existing set or creation of a personalised set is feasible by clicking the button to the right of the drop-down list [...] (see chapter B.3.1.2);

  These checks are accessible for both single wall and double wall projects.

- **Calculation options**: in this frame it is possible to take account of 2nd order effects (see Part C of the manual) by simply ticking the corresponding box ("Accounting for 2nd order moments", which is unticked by default). It is also possible to modify the maximum number of iterations per phase, by default equal to 100, as well as the calculation step, corresponding to the wall breakdown step, by default given a value of 0.20 m (or 0.66 ft in the imperial system).

  In this same frame, a button gives access to the advanced calculation options, the content of which is illustrated in the following figure.

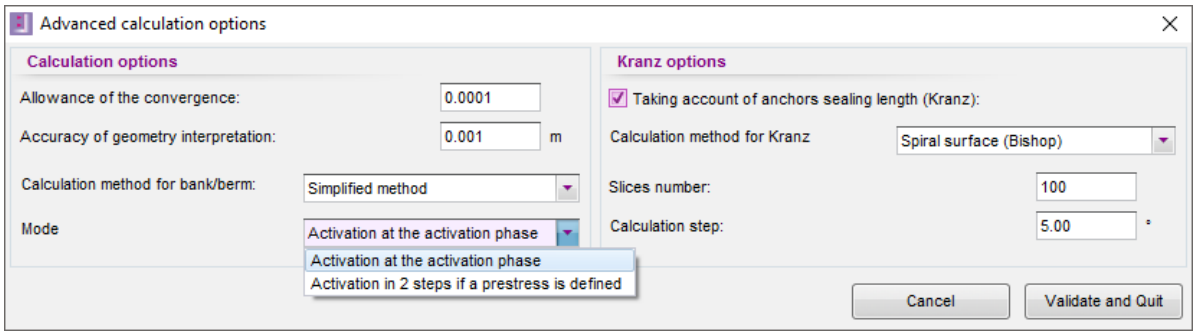


Figure B18 : "Advanced calculation options" dialogue box

The "Calculation options" frame is used to specify the following:

- The relative tolerance controlling convergence, by default set at 10^{-4};
- The accuracy of geometry interpretation (in m), by default equal to 10^{-3} m, or one mm;
- The choice of calculation method for banks and berms:
  - Equivalent loads: the effect of a bank or berm is simulated by superposing overloads equivalent to their weight. This approach is not recommended, in accordance with standard NF P94-282;
  - Simplified method: the effect of a bank or berm is simulated in accordance with the approaches proposed in appendix D to standard NF P94-282;
  - Slope calculation (line surfaces): the effect of a bank or berm with respect to the limit active/passive earth pressure diagrams is estimated using the kinematic yield design method considering plane rupture kinematics;
  - Slope calculation (arc surfaces): the effect of a bank or berm with regard to limit active/passive earth pressure diagrams is estimated using the yield design kinematic method considering rotational failure kinematics with arcs of logarithmic spiral.
- Anchors work mode (except for embedded anchors): it is possible to choose between operation as of the activation phase or operation only as of the next phase (then only pre-stressing is taken into account in the activation phase). This

choice applies to all the anchors which will be defined in the project (see part C of the Manual for the theoretical differences between to these two options);

- Reference method for recalculation of k_a/k_p: to be chosen from the three methods available: Kérisel (corresponding to the Kérisel and Absi tables), Coulomb and Rankine.

  Comments: Depending on the parameter applied to the tangent of the friction angle ($\gamma_{\varphi'}$, see B.3.1.2), the friction angle calculation value may differ from its characteristic value defined by the user. In this case, the values of k_a/k_p will be automatically recalculated by the calculation engine. The above option can be used to set the recalculation method for these parameters.

The "Kranz Options" frame defines the parameters for checking the stability of the anchoring block (Kranz):

- Taking account of anchors embedded length in the equivalent anchor force calculation for a wall anchored by several tie-rods. This option is activated by default.
- Calculation method for Kranz for examining the limit equilibrium of the anchoring block: slices method (Bishop) with a choice between Planar Surface or Spiral Surface, or yield design method with multi-spiral failure surface.

The theoretical aspects of these options are detailed in part C of the manual.

- **Curves display:** by ticking the "Same horizontal scale for curves" box, a common scale is assigned to all the graphics presenting the same type of result. By default, this box is not ticked.
- **Language for output:** these buttons are used to select the language used to print the project. The default language corresponds to that chosen by the user on opening K-Réa. If a different language is chosen during definition of the project, only project printing will be affected and not the interface.
- **Water options:** this frame is used to declare the water weight and the hydraulic gradient definition mode (Potentials or Pressures). By default the water weight is taken as equal to 10 kN/m^3 (0.0624 kcf in imperial system) and the hydraulic gradient is defined in terms of hydraulic potentials.
- **Project type:** this frame is reserved for the choice of the type of project (Single wall or Double wall) by selecting the corresponding icon. If a double wall project is selected, K-Réa asks for input of the distance between the two walls, in the **Double wall options** frame. As this chapter is devoted to the single wall case, refer to chapter B.3.4 for the particularities of double wall projects.

B.3.1.2. Definition of partial factors dialogue box

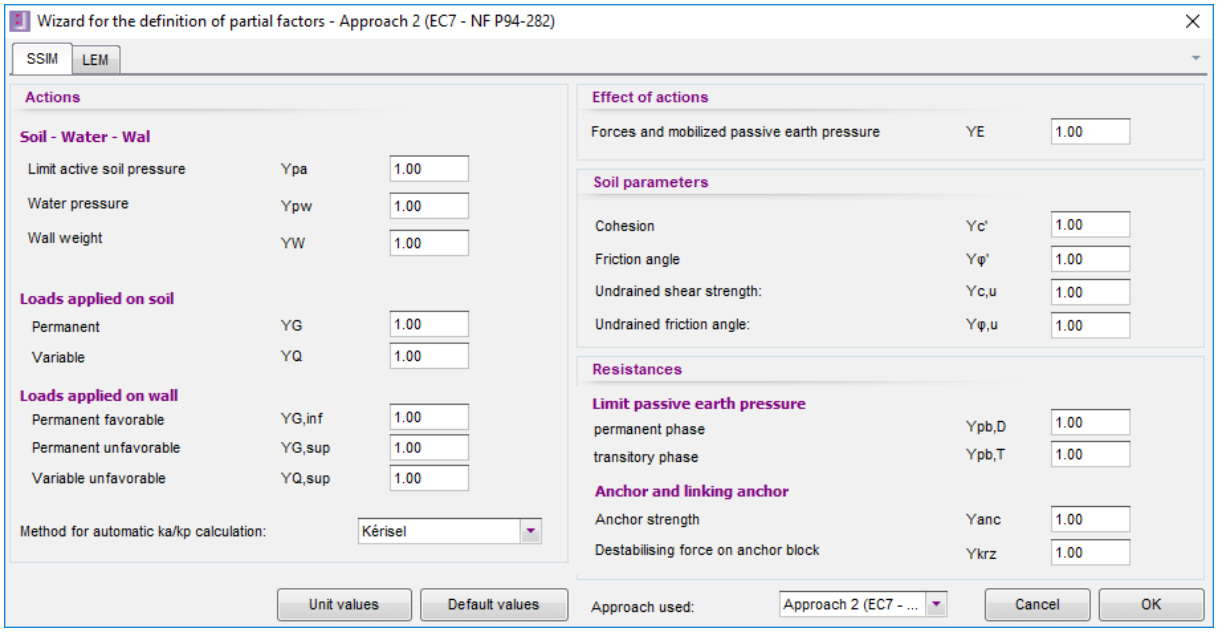


Figure B19 : Partial factors definition dialogue box

This window is used to view and modify the partial safety/weighting factors used in the ULS calculations and checks. Two sets of parameters are available, each grouped under a tab SSIM LEM, one corresponding to the SSIM calculation and the other to the LEM. Some parameters are common to the two sets of parameters and can be modified in the SSIM tab and only viewed in the LEM tab.

The default values proposed are those resulting from one of the approaches (1, 2 or 3) of Eurocode 7 and its French application standard NF P94-282. It is however possible to modify them to adapt to application of other regulatory references.

The Default values button is used to reset the factors with the values corresponding to the approach chosen from the "Partial factors" drop-down list, which is also summarised in the title of the wizard window.

The Unit values button is used to assign a value of 1.0 to all factors. In this way, it is possible to run checks without applying any weighting.

The definitions of the various factors are presented below:

<u>Actions and Effects of actions</u>:

- **Soil – Water – Wall:** factors applied to the limit active soil pressure (γ_{pa}), water pressure (γ_{pw}) and weight of the wall (γ_W);
- **Loads applied on soil:** factors applied to overloads acting on the soil as a function of their nature (permanent γ_G / variable γ_Q);
- **Loads applied on wall:** factors applied to overloads applied directly to the wall as a function of its nature (permanent/variable) and their character (favourable/unfavourable). Several possible combinations: permanent favourable ($\gamma_{G,inf}$), permanent unfavourable ($\gamma_{G,sup}$) and variable unfavourable ($\gamma_{Q,sup}$);
- **Effect of actions** (γ_E): factor applied to the forces, loads and mobilised passive earth pressure.

Resistance parameters:

- $\gamma_{c'}$: applied to soil layers cohesion for drained behaviour;
- $\gamma_{\varphi'}$: applied to the tangent of the friction angle of the soil layers for drained behaviour;
- γ_{cu} : applied to soil layers cohesion for undrained behaviour;
- $\gamma_{\varphi u}$: applied to the tangent of the friction angle of the soil layers for undrained behaviour;

Resistances:

- $\gamma_{pb,D}$ and $\gamma_{pb,T}$: applied to the soil limit passive earth pressure as a function of the nature of the phase (permanent or transitory respectively);
- γ_{anc} : applied to the resistance of anchors;
- γ_{krz} : applied to the destabilising anchor force when checking the stability of the anchoring block (Kranz).

The γ_{anc} and γ_{krz} factors apply exclusively for an SSIM type calculation.

The following table summarises the set of default values proposed for the SSIM model, for each calculation approach:

Category	Partial factor	Symbol	Approach 1.1	1.2	2/2*([1])	3
Soil-Water-Wall	Limit active soil pressure	γ_{Pa}	1.35	1.00	1.00	1.00
	Water pressure	γ_{Pw}	1.35	1.00	1.00	1.00
	Weight of wall	γ_{w}	1.35	1.00	1.00	1.00
Load applied on soil	Permanent	γ_{G}	1.00	1.00	1.00	1.00
	Variable	γ_{Q}	1.11	1.30	1.11	1.30
Load applied on wall	Permanent favourable	$\gamma_{G,inf}$	1.00	1.00	1.00	1.00
	Permanent unfavourable	$\gamma_{G,sup}$	1.35	1.00	1.00	1.35
	Variable unfavourable	$\gamma_{Q,inf}$	1.50	1.30	1.11	1.50
Effects of actions		γ_{E}	1.00	1.00	1.35	1.00
Soil parameters	Cohesion (drained behaviour)	$\gamma_{c'}$	1.00	1.25	1.00	1.25
	Friction angle (drained behaviour)	$\gamma_{\varphi'}$	1.00	1.25	1.00	1.25
	Cohesion (undrained behaviour)	γ_{cu}	1.00	1.40	1.00	1.40
	Friction angle (undrained behaviour)	$\gamma_{\varphi u}$	1.00	1.40	1.00	1.40
Resistances	Limit passive soil pressure – Permanent phase	$\gamma_{pb,D}$	1.00	1.00	1.40	1.00
	Limit passive soil pressure – Transitory phase	$\gamma_{pb,T}$	1.00	1.00	1.10	1.00
	Resistance of anchors	γ_{anc}	1.10	1.10	1.00	1.00
	Destabilising anchor force (Kranz)	γ_{krz}	1.00	1.00	1.10	1.00

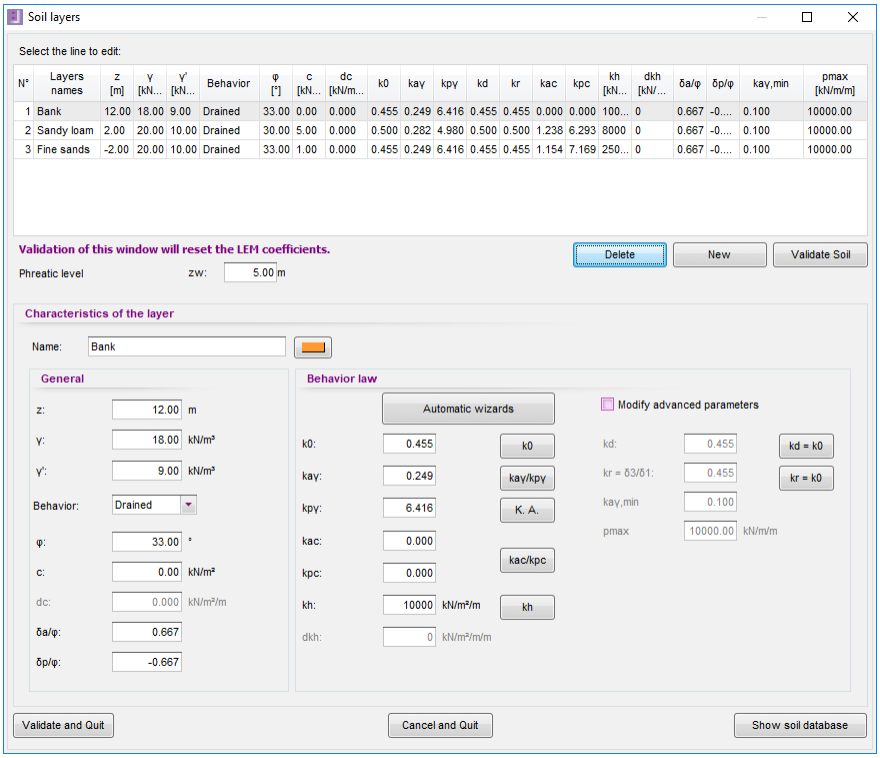
Table B 2: Default values of partial factor sets proposed in Eurocode 7

([1]) Model 2* is only valid for the SSIM method (see part C of the manual for more details and in particular for the values proposed for the LEM model).

B.3.2. Definition of soil layers

B.3.2.1. Soil characteristics definition dialogue box

This dialogue box is accessible by clicking the **Data** menu then **Soil layers**. For each wall, it is used to define the intrinsic parameters and the characteristics defining the soil-wall interaction (behaviour law).



Data summary and selection table

Input frame

Figure B20 : Soil layers characteristics dialogue box

The dialogue box is divided into two parts. At the top, a summary table showing all the data and enabling a layer to be selected. At the bottom, an input frame for defining or modifying the characteristics of the layer selected in the summary table, and for accessing the Soils Database by clicking Show soil database.

The soils database (DB) is used to keep a record of the soil layer and its characteristics. They can thus be reused to define a new soil layer. To save a soil layer, after validating the soil in the summary table, click Show soil database, then the purple arrow pointing to the right. Conversely, if one wishes to use a layer already in the memory, click Show soil database, then select the layer required by clicking it and then clicking the purple arrow pointing to the left. A layer can be removed from the memory by selecting it and then clicking the trash can icon under the list.

Once input of a layer is complete, click the summary table in the upper part to validate it or click the Validate Soil button and then New to begin input of a new layer or Validate and Quit to end input of all layers. The soil layers are automatically reorganised as a function of their upper level (z). A soil layer can be deleted by clicking Delete.

In the input zone, it is first of all necessary to enter z_w: initial level of the top of the water table (m, ft). This level is common to all the soil layers which will be defined by the concerned wall.

In the other boxes one must then define the characteristics of the soil layers, beginning with the upper layer:

- Name of soil layer;
- Colour assigned to the soil layer: a simple click on the colour associated with the soil layer displays the colour selection dialogue box (see chapter B.3.2.5).

General Frame:

- **z** : level of head of layer (m, ft);
- γ : wet unit weight (characterising the soil situated above the groundwater table) (kN/m^3, kcf);
- γ' : buoyant unit weight (characterising the soil situated below the groundwater table) (kN/m^3, kcf);
- **Behaviour**: drained or undrained (the choice only impacts the safety factor on cohesion and internal friction angle)
- φ : internal friction angle (°);
- **c** : cohesion (kN/m², ksf);
- **dc** : variation in cohesion per unit depth in the layer – the reference being the initial head of the layer (kN/m²/m, ksf/ft);
- δ_a/φ : obliquity of limit active earth pressure stresses (ratio between angle of active pressures and angle of internal friction);
- δ_p/φ : obliquity of limit passive earth pressure stresses (ratio between angle of passive pressures and angle of internal friction).

Behaviour law:

- k_0 : horizontal active earth pressure coefficient at rest;
- $k_{a\gamma}$: horizontal active earth pressure coefficient;
- $k_{p\gamma}$: horizontal passive earth pressure coefficient;
- k_{ac} : horizontal active earth pressure coefficient applied to cohesion;
- k_{pc} : horizontal passive earth pressure coefficient applied to cohesion;
- k_h : horizontal soil reaction coefficient (kN/m^2/ml, ksf/lft);
- dk_h : variation in horizontal soil reaction coefficient per unit depth in the layer – the reference being the initial head of layer (kN/m^2/m/ml, ksf/ft/lft);
- k_d : unloading ratio of soil;
- k_r : reloading ratio of soil;
- $k_{a\gamma,min}$: minimal active earth pressure coefficient (horizontal);
- p_{max} : limit pressure not to be exceeded (kN/m/ml, kip/lft).

A tick box is used to activate the advanced parameters (**dc, dk_h, k_d**, **k_r**, **$k_{a\gamma,min}$** and **p_{max}**) so that they can be modified. When this box is not ticked, these parameters take the following default values:

- **dc = 0** (kN/m²/m, ksf/ft);
- **dk_h = 0** (kN/m^2/m/ml, ksf/ft/lft);
- $k_d = k_0$;
- $k_r = k_0$;
- $k_{a\gamma,min} = 0.10$;
- **p_{max} = 10^4** (kN/m/ml, kip/lft).

Two additional buttons are accessible under the soil characteristics frame. The Cancel and Quit button cancels the modifications made and closes the dialogue box, while the Validate and Quit button saves the latest inputs before closing the window.

Caution: closing dialogue boxes by clicking the cross at the top-right of the window will close them without saving any modifications made (equivalent to clicking the Cancel and Quit button).

Wizards are available for determining the various coefficients and are accessible via the buttons in the dialogue box.

Clicking the Automatic wizards button enables the values of the k_0, $k_{a\gamma}$, $k_{p\gamma}$, k_d, k_r, k_{ac} and k_{pc} coefficients to be calculated in one go, according to the following choices:

- $\mathbf{k_0}$: Jaky formula for a normally consolidated horizontal soil (OCR overconsolidation ratio = 1 and β "TN angle" = 0°);
- $\mathbf{k_{a\gamma}}$: Kerisel and Absi wizard – Active earth pressure, weighted soil, no overload for a vertical wall and a horizontal soil (λ "obliquity of wall" = 0° and β "Ground level angle" = 0°);
- $\mathbf{k_{p\gamma}}$: Kerisel and Absi wizard – Passive earth pressure, weighted cohesionless soil, no overload for a vertical wall and horizontal soil (λ "obliquity of wall" = 0° and β "Ground level angle" = 0°);
- $\mathbf{k_d} = \mathbf{k_r} = \mathbf{k_0}$;
- If cohesion is nil, then: $\mathbf{k_{ac}} = \mathbf{k_{pc}} = 0$;
- If cohesion is not nil: $\mathbf{k_{ac}}$ and $\mathbf{k_{pc}}$ are obtained using the corresponding wizard (no value to be input, this wizard already knows everything)

The button located next to each coefficient can be used to access the corresponding wizard for a manual definition of the parameters of the behaviour law.

All the wizards are described in the following chapters.
These wizards are accessible via:

- Initial definition of soil layers;
- Definition of a bank or berm (fill) (see chapter B.5.2.2);
- Redefinition of soil layers (see chapter B.5.3.1);
- Definition of LEM coefficients (see chapter B.5.7.1);
- or, independently of the current project, via the **Wizards** menu.

CAUTION: WIZARDS ARE PROVIDED TO HELP THE USER, WHO HOWEVER REMAINS RESPONSIBLE FOR THEIR USE.

B.3.2.2. Wizards for determining active and passive earth pressures

Wizard $k_{a\gamma}/k_{p\gamma}$ calculates the values of the active and passive earth pressures using one of the three methods proposed:

- reading the Kerisel & Absi tables
- the Coulomb sliding wedge method
- the Rankine formula

The theoretical bases used by these wizards are detailed in part C of the manual. This chapter simply describes how they are used.

a) KERISEL and ABSI active and passive earth pressure tables

This wizard takes the form of a dialogue box as illustrated below. It comprises a free consultation part (right-hand side) and a settings part to determine coefficients by interpolation (the actual wizard – left-hand side).

The values of the friction angle and those of the obliquities input in the soil layers characteristics definition box are automatically recovered.

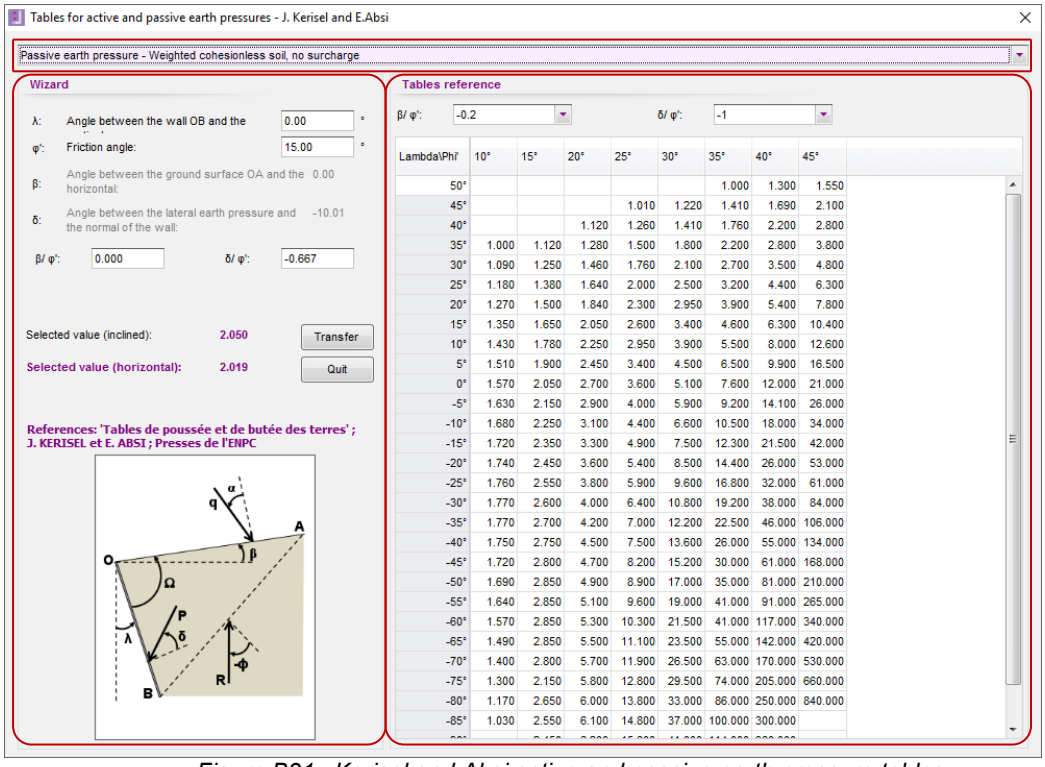
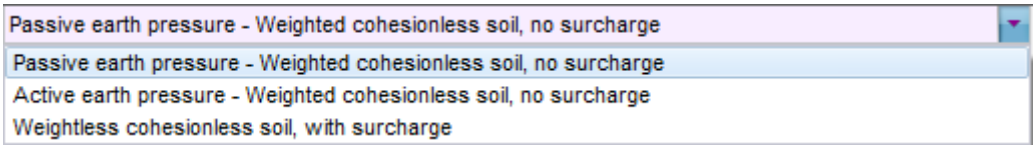


Figure B21 : Kerisel and Absi active and passive earth pressure tables

The type of table to be used should be selected from the drop-down list at the top of the screen:



The contents of the "Tables reference" (right) is updated according to the type of tables selected.

In the "Wizard" frame, enter the data needed to determine the (active or passive earth pressure) coefficient in the input frame at the top-left of the window:

- λ : inclination of wall (i.e. angle between wall and vertical) - 0° by default (°);
- φ : soil friction angle (°);
- β/φ : angle of soil free surface normalised by φ;
- δ/φ : obliquity of stresses to the normal of the wall normalised by φ;
- α : obliquity of overload on free surface (°);
- Ω : angle between free surface and wall (= $\pi/2 + \beta - \lambda$) (°).

The chosen value corresponding to the input data is displayed at the bottom of the interactive frame, plus the horizontal value which is that used in the calculations.

This value can then be transferred to the layer data currently being input, by using the Transfer button. To close the wizard, click Quit.

Note 1: The transfer button only authorises values calculated with a zero wall inclination ($\lambda=0$) in order to comply with the reaction coefficients calculation method. The values calculated with a inclination other than zero ($\lambda\neq0$) may be consulted but are not directly usable with the calculation method, which is why they cannot be transferred to the project.

Note 2: K-Réa performs linear interpolation for values which are not given in the tables.

CAUTION: WIZARDS ARE PROVIDED TO HELP THE USER, WHO HOWEVER REMAINS RESPONSIBLE FOR THEIR USE.

b) COULOMB active and passive earth pressure formula

This wizard takes the form of a dialogue box as illustrated below:

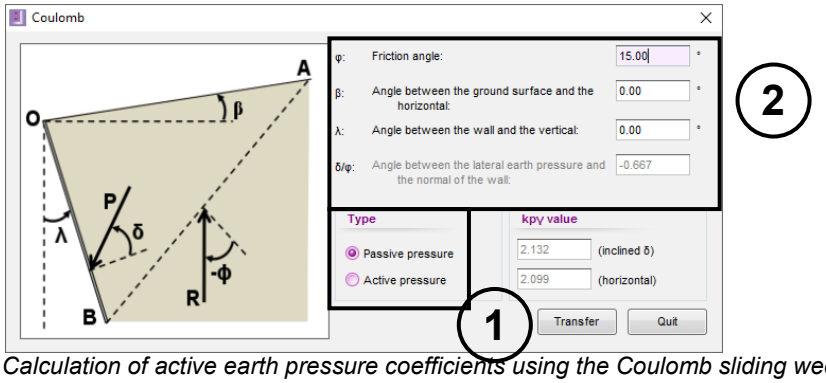


Figure B22 : Calculation of active earth pressure coefficients using the Coulomb sliding wedge method

The values of the friction angle and of the angles input into the soil layers characteristics definition box are retrieved automatically.

1) Select the type of calculation:
   - Active pressure;
   - Passive pressure.

2) Input the data needed for the calculation:
   - **φ** : friction angle (°);
   - β : angle between the ground surface and the horizontal (°);
   - λ : angle between the wall and the vertical – 0° by default (°);
   - **δ/φ** : angle between the lateral earth pressure and the normal of the wall.

The calculated values (inclined and horizontal) are displayed at the bottom-right of the window. They may be transferred to the data of the layer currently being input by clicking the Transfer button. To close the wizard, click Quit.

Note 1: The transfer button only authorises values calculated with a zero wall inclination ($\lambda=0$) in order to comply with the reaction coefficients calculation method. The values calculated with an inclination other than zero ($\lambda\neq0$) may be consulted but are not directly usable with the calculation method, which is why they cannot be transferred to the project.

CAUTION: WIZARDS ARE PROVIDED TO HELP THE USER, WHO HOWEVER REMAINS RESPONSIBLE FOR THEIR USE.

c) RANKINE active and passive earth pressure formula

This wizard takes the form of a dialogue box as illustrated below:

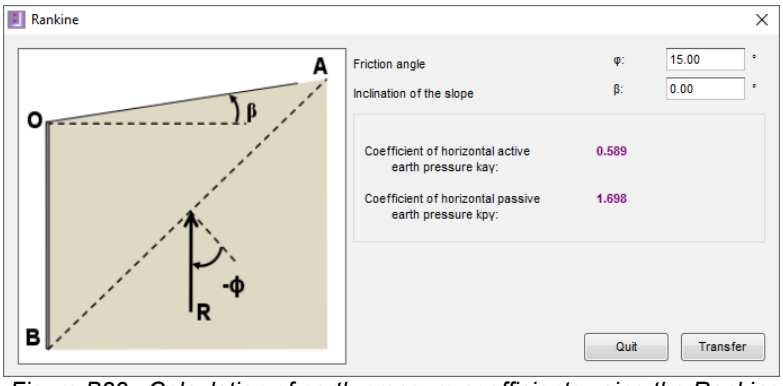


Figure B23 : Calculation of earth pressure coefficients using the Rankine method

Input of the friction angle φ and inclination of the slope β values is sufficient to calculate the active and passive earth pressure coefficients using the Rankine formula. Their horizontal projections are displayed in the lower part of the window.

Note: the wizard automatically retrieves the friction angle input in the soil layers characteristics definition box.

These values can then be transferred to the data for the layer currently being input by clicking the Transfer button. To close the wizard, click Quit.

CAUTION: WIZARDS ARE PROVIDED TO HELP THE USER, WHO HOWEVER REMAINS RESPONSIBLE FOR THEIR USE.

B.3.2.3. Wizard for determining coefficients k_{ac} and k_{pc}

This wizard proposes a calculation method to determine the active and passive earth pressure coefficients applied to the cohesion term.

The formulas used are given in part C of the manual. Only the use of the wizard is described below.

This wizard takes the form of a dialogue box illustrated below:

Figure B24 : Calculation of coefficients k_{ac}/k_{pc}

Input the data necessary for this calculation:

- **δa/φ** : ratio between obliquity of active earth pressure stresses and friction angle;
- **δp/φ** : ratio between obliquity of passive earth pressure stresses and friction angle;
- **φ** : friction angle (°).

Note: the wizard automatically retrieves the values input in the soil layers characteristics definition box.

The values of k_{ac} and k_{pc} are displayed in the bottom-left of the window.

These values can then be transferred to the data of the layer currently being input by clicking the Transfer button. To close the wizard, click Quit.

CAUTION: WIZARDS ARE PROVIDED TO HELP THE USER, WHO HOWEVER REMAINS RESPONSIBLE FOR THEIR USE.

B.3.2.4. Wizards for determining the reaction coefficient

This wizard proposes three methods for determining the reaction coefficient: application of the Balay formula, application of the Schmitt formula and reading of the Chadeisson curves.

The theoretical bases used for these 3 methods are detailed in part C of the manual. Only the actual use of the wizard is described below.

The wizard takes the form of a single window, the upper part of which contains the choice of calculation method and a recapitulation of the values determined, the central part of which contains the parameters to be input for the calculation and, finally, the lower part of which contains the proposed value of k_h and the comments associated with the methods used.

Once the input parameters have been entered, the value of kh obtained can be transferred using the Transfer button.

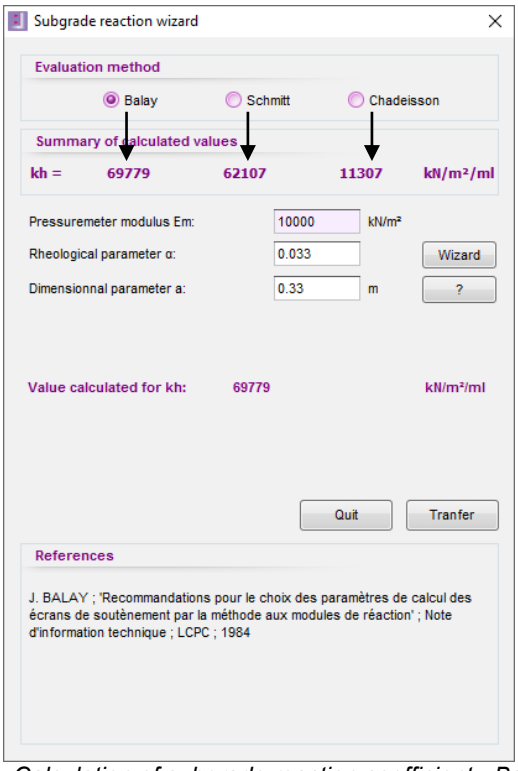


Figure B25 : Calculation of subgrade reaction coefficient - Balay method selected

a) BALAY formula

The following parameters must be input:

- E_m: pressuremeter strain modulus (kN/m², KsF) of the soil layer
- α: soil layer rheological coefficient

  Note: a wizard is provided for determining this coefficient (can be consulted simply by clicking the adjacent Wizard >> button). It is extracted from the Fascicule 62 (LCPC-SETRA).

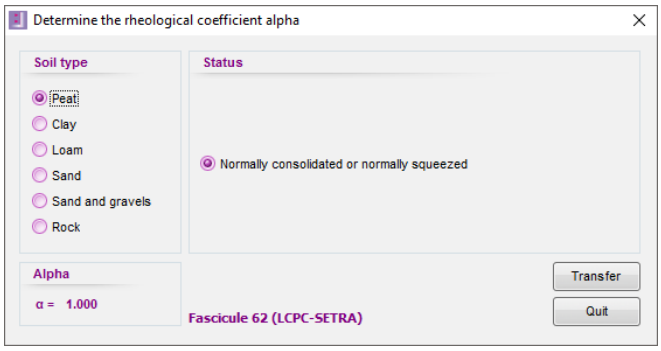


Figure B26 : Determination of rheological parameter α

- **a**: parameter to be defined as a function of wall embedment depth, the supported height and the relative position of the layer concerned with respect to the bottom of the excavation. The dimension of parameter "a" is a length. A help diagram illustrates the choice of this parameter (see Figure B 27).

  Note: in certain special cases (same layer encountered on either side of bottom of excavation), the choice of parameter "a" requires differentiation between 2 layers with the same characteristics except for the value of k_h. This latter must be calculated for each side of the wall, allocating the appropriate value to parameter "a".

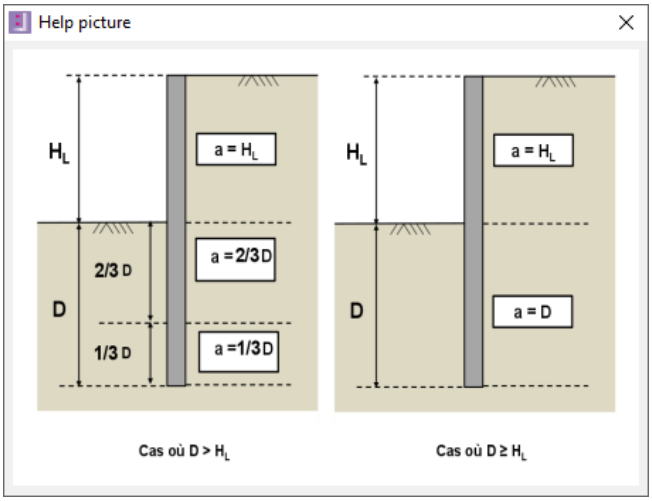


Figure B27 : Help diagram for defining dimensional parameter a

b) SCHMITT formula

Input parameter:

- **E_m**: pressuremeter strain modulus of the layer (kN/m², ksf);
- α: rheological coefficient of the layer. A wizard is available for determining this parameter by clicking Wizard >>;
- **Section**: if the EI product of the wall has already been input in the **Retaining Wall definition** window, the "Section" parameter gives direct access to the EI values for the various wall sections. If not, this parameter may be left blank;
- **EI**: average product of inertia of the wall (kN.m²/ml, kip.ft²/lft).

The interest of this approach is that it takes account of the variation in the reaction coefficient with the stiffness of the wall: the stiffer the wall, the smaller the reaction coefficient, which is an accurate reflection of reality.

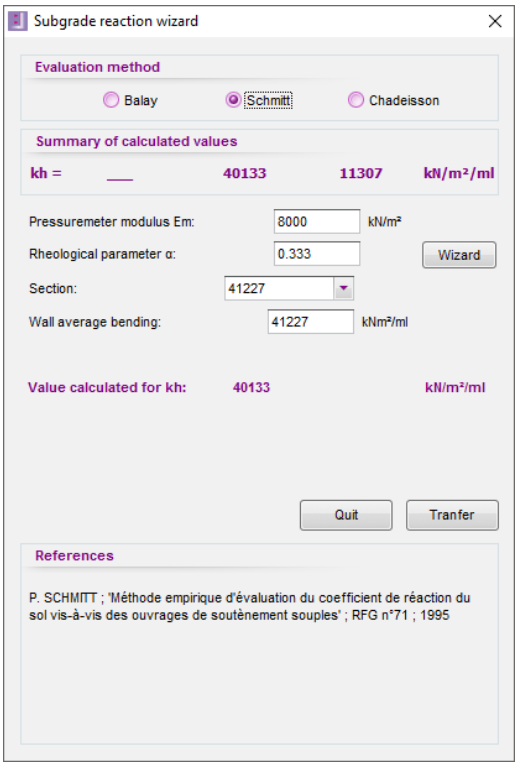


Figure B28 : Calculation of reaction coefficient - Schmitt method selected

c) CHADEISSON curves

The curves are read automatically after inputting the cohesion value and that of the friction angle. It is possible to check the value of k_h proposed by direct reading of the curves.

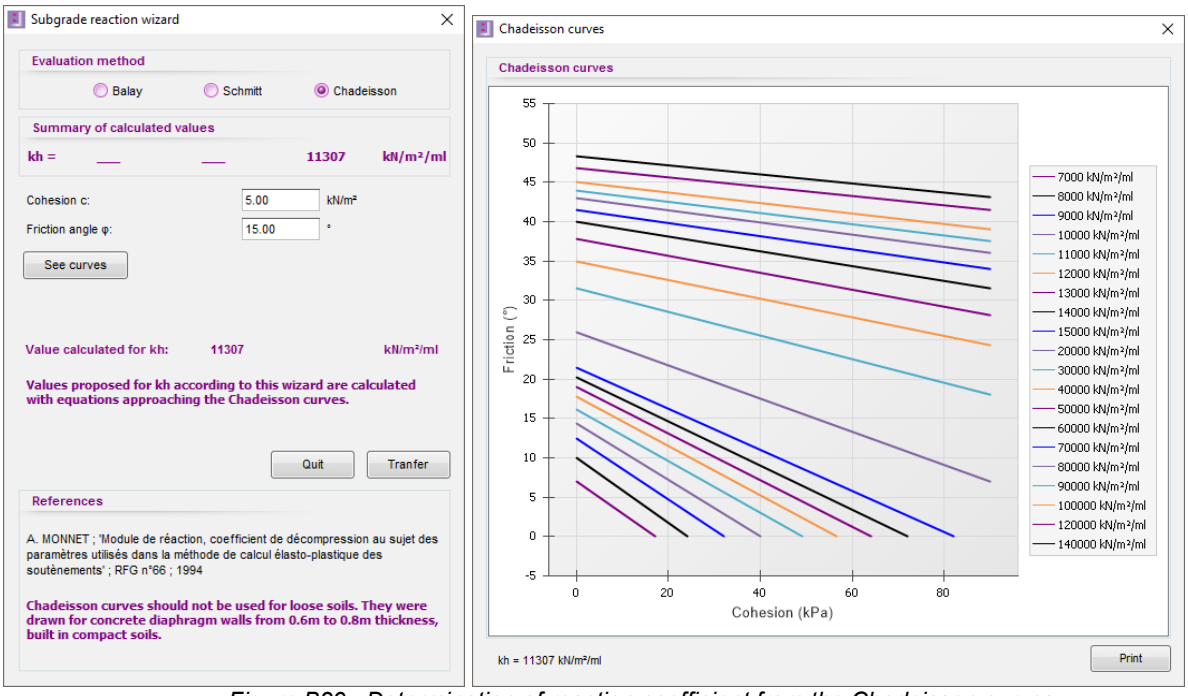


Figure B29 : Determination of reaction coefficient from the Chadeisson curves

The original curves were only approximately recreated. There may thus be a slight difference between what can be seen on the screen and what the user can do on paper.

B.3.2.5. Colour of layers

This wizard is accessible by opening the **Data** menu, then **Soil layers definition,** as well as in the **Fill** action. This action triggers opening of the soil layers definition dialogue box. Then click the layer colour to be modified in order to open the "Colours" wizard.

This enables a colour to be chosen other than that selected by default at creation of the soil layers. If the modifications are validated, the soil layers will appear with the new colours in the project cross-section.

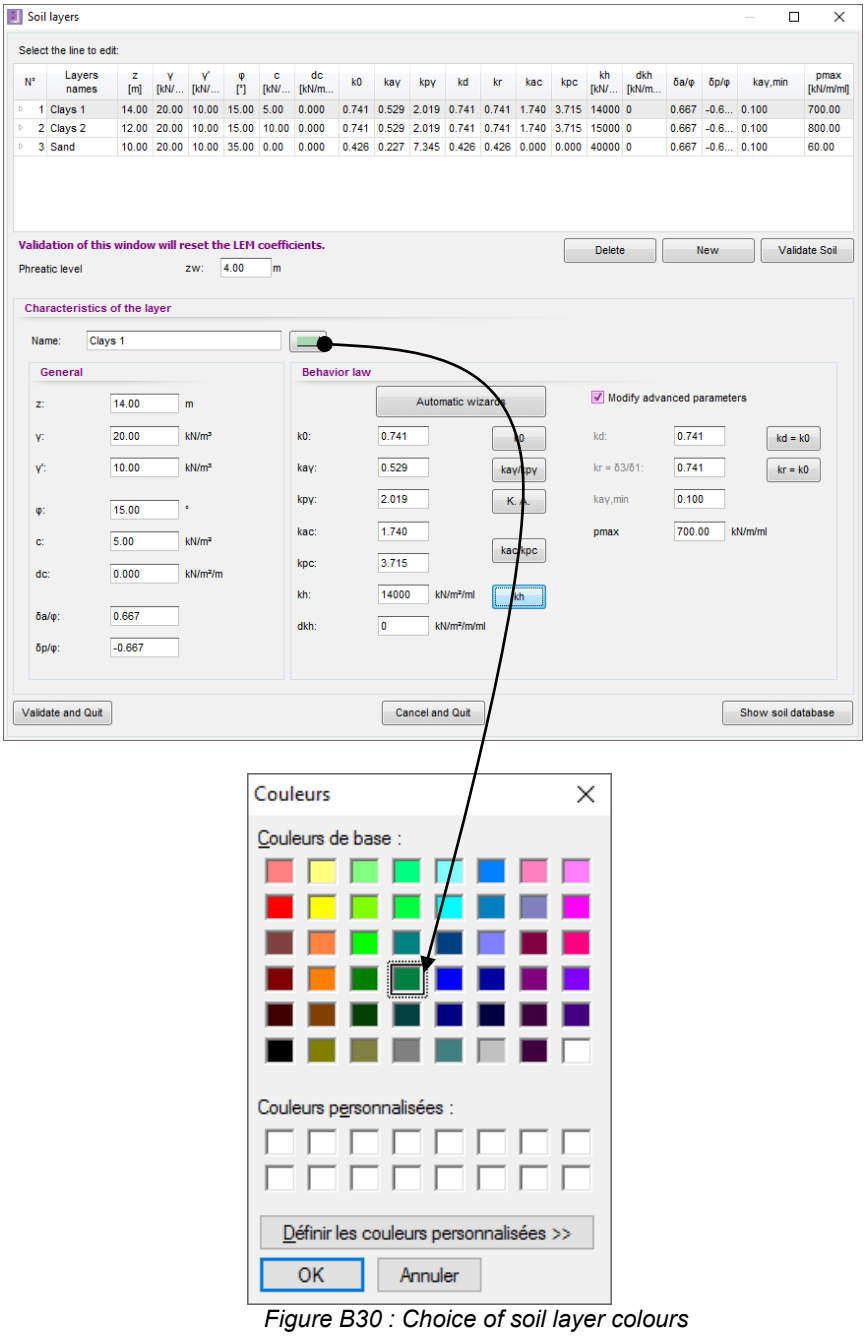


Figure B30 : Choice of soil layer colours

B.3.3. Retaining wall definition

B.3.3.1. Retaining wall definition window

This dialogue box is accessible by clicking the **Data** menu then **Retaining Wall** definition. It is used to define the wall characteristics needed for the calculation.

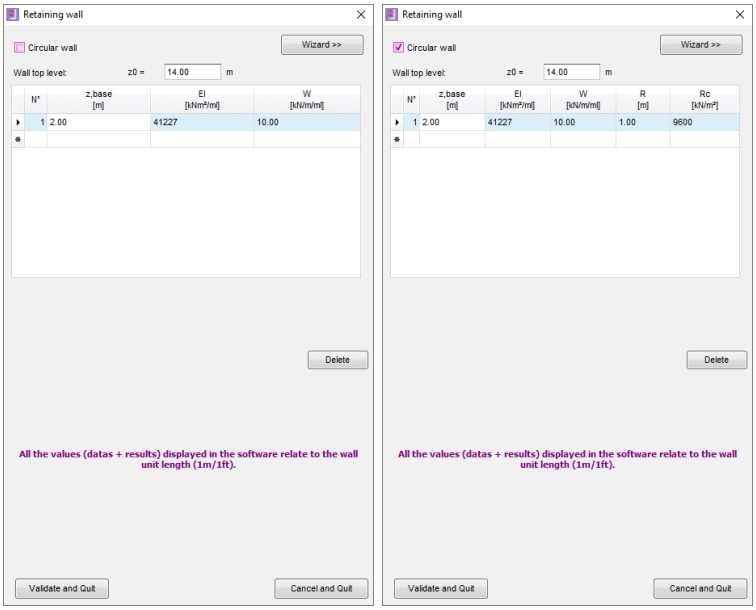


Figure B31 : Retaining Wall definition:
Plane wall (left) and circular wall (right)

The check box at the top of the dialogue box can be used to activate definition of a **circular wall** if the wall has circular stiffness (see right-hand figure).

The wall is defined by:

- z_0 : level (elevation or depth) of top of wall (m, ft);
- z_{base} : level (elevation or depth) of base of wall or lower level of sections making up the wall (m, ft);
- **EI** : product of inertia of wall (kN.m²/ml, kip.ft²/lft). This parameter can be determined for each section with the wizard proposed (Wizard >> button described in next chapter);
- **W** : surface weight of wall (kN/m²/ml, ksf/lft). The "Sheet pile wall" wizard on the right automatically transfers this value at the same time as the product of inertia;
- If the wall has cylindrical stiffness, the user must enter the following parameters after ticking the box (circular wall):
  - R_c : cylindrical stiffness of wall (kN/m³, kip/ft³).
  - **R** : average radius of wall (m, ft).

THE WIZARDS PROVIDE THE USER WITH HELP BUT THEIR USE REMAINS THE RESPONSIBILITY OF THE USER.

To close the wall definition window, click either Validate and Quit to save the wall model or Cancel and Quit to abandon the last modifications made.

B.3.3.2. Wizards for determining wall stiffness

K-Réa proposes several wizards for three types of wall families: continuous wall, composite wall and sheet pile wall. They can be accessed from the wall definition window (**Data** menu, then **Retaining wall definition**).

For each family, specific data are requested for calculation of the EI product and the R_c value if the "Circular wall" option is activated.

The theoretical bases used by these various wizards are detailed in part C of the manual. Only the actual use of the wizards is described below.

a) Continuous wall

This wizard is used to calculate the EI product (and the cylindrical rigidity R_c as applicable) of a continuous wall.

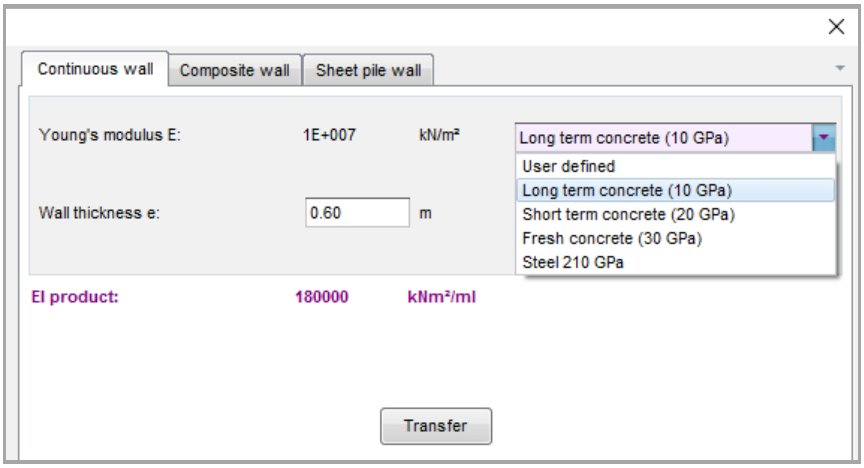


Figure B32 : Determining the EI product for continuous plane walls

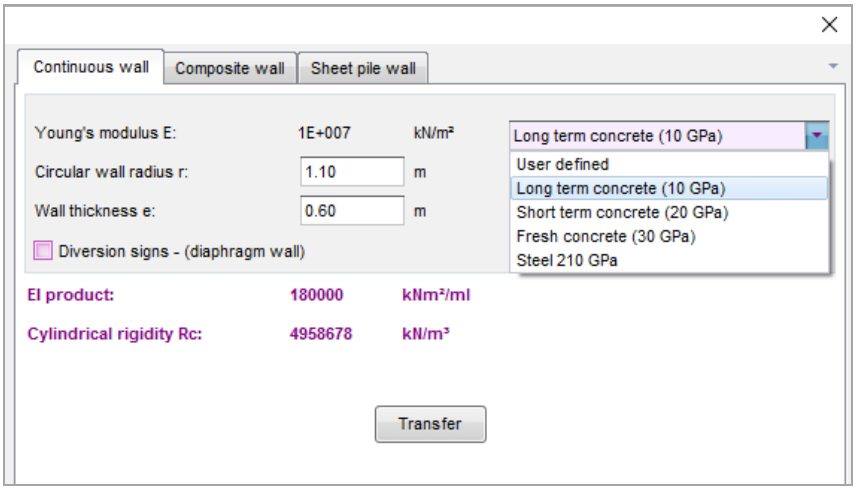


Figure B33 : Determining EI and Rc for continuous circular walls

The following data are required:

- **E**: Young's modulus of the wall material (kN/m², ksf). The value can be input manually (personalised choice) or loaded from the drop-down list (typical values for concrete or steel);
- **e**: wall thickness (m, ft);
- **R**: average wall radius if previously defined as a circular wall (m, ft).

The value obtained for EI and for R and Rc as applicable, can then be sent to the input zone on the left, using the Transfer button.

b) Composite wall

This wizard is used to calculate the EI product per unit length (m or ft) of elements (piles, sections, rectangular piles, etc.) either spaced or abutting and making up a plane wall. For each of the 3 types proposed, the user must characterise:

- the main elements (concrete only, steel only, or combination);
- the facing.

K-Réa then gives the EI product per equivalent unit length of continuous wall that the user may transfer to the retaining wall definition table.

Case of solid circular piles

To calculate the EI product per unit length of a composite wall for which the circular piles are solid, click the "Composite wall" tab then select "Solid circular piles" as shown in the following figure:

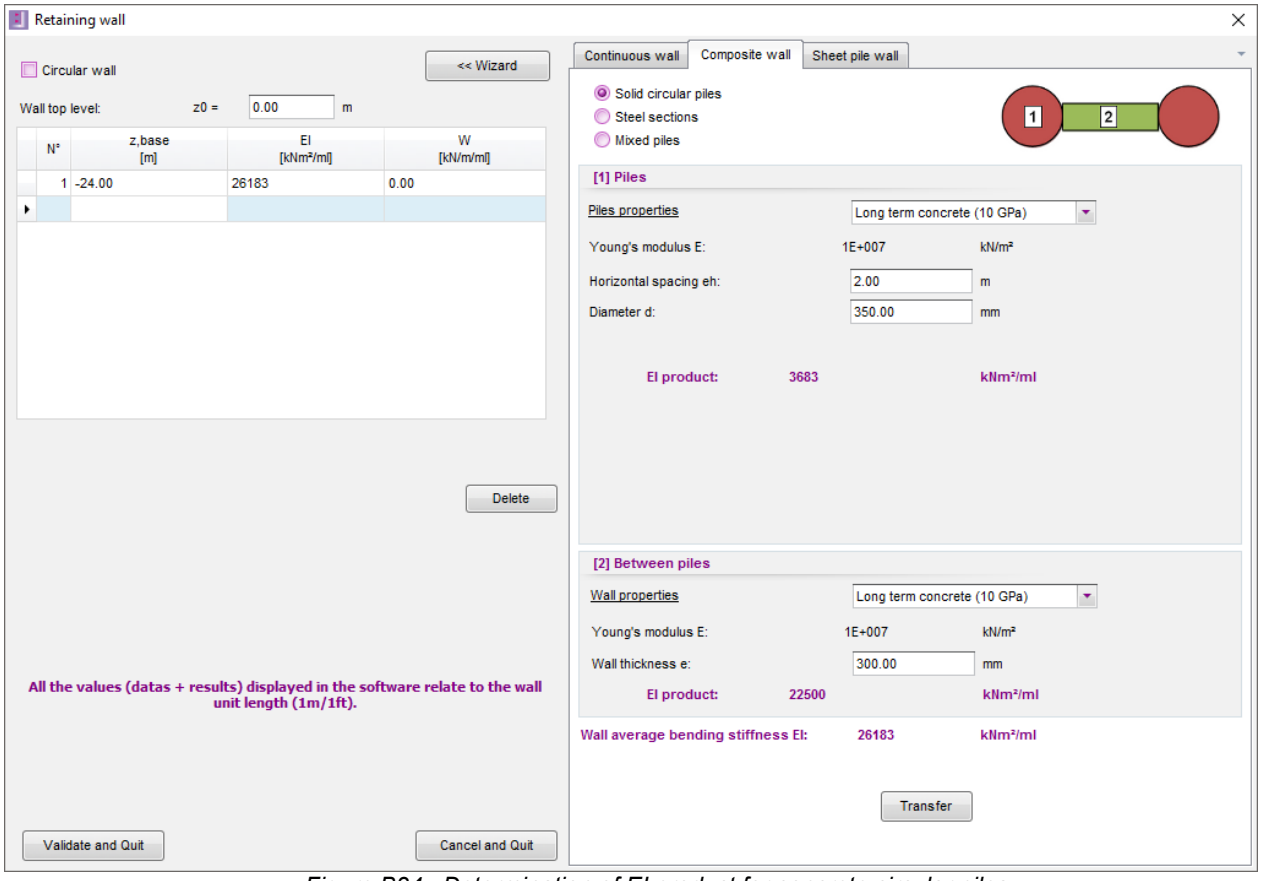


Figure B34 : Determination of EI product for concrete circular piles

The following data must be input to define the piles:

- **E** : Young's modulus of the pile. The value can be input manually or chosen from the adjacent drop-down list (kN/m², ksf);
- $\mathbf{e_h}$: horizontal spacing between piles center (m, ft);
- **d** : pile diameter (mm, in).

The definition of the facing is based on the following 2 parameters:

- **E** : Young's modulus of the facing (kN/m², ksf), to be input manually or chosen from the adjacent drop-down list;
- **e** : facing thickness (m, in).

K-Réa displays the equivalent EI product per unit length of composite wall at the bottom of the screen.

Case of steel sections

To calculate the EI product per unit length of a composite wall in which the piles are made of steel sections, click the "Composite wall" tab, then select "Steel sections" as shown in the following screenshot:

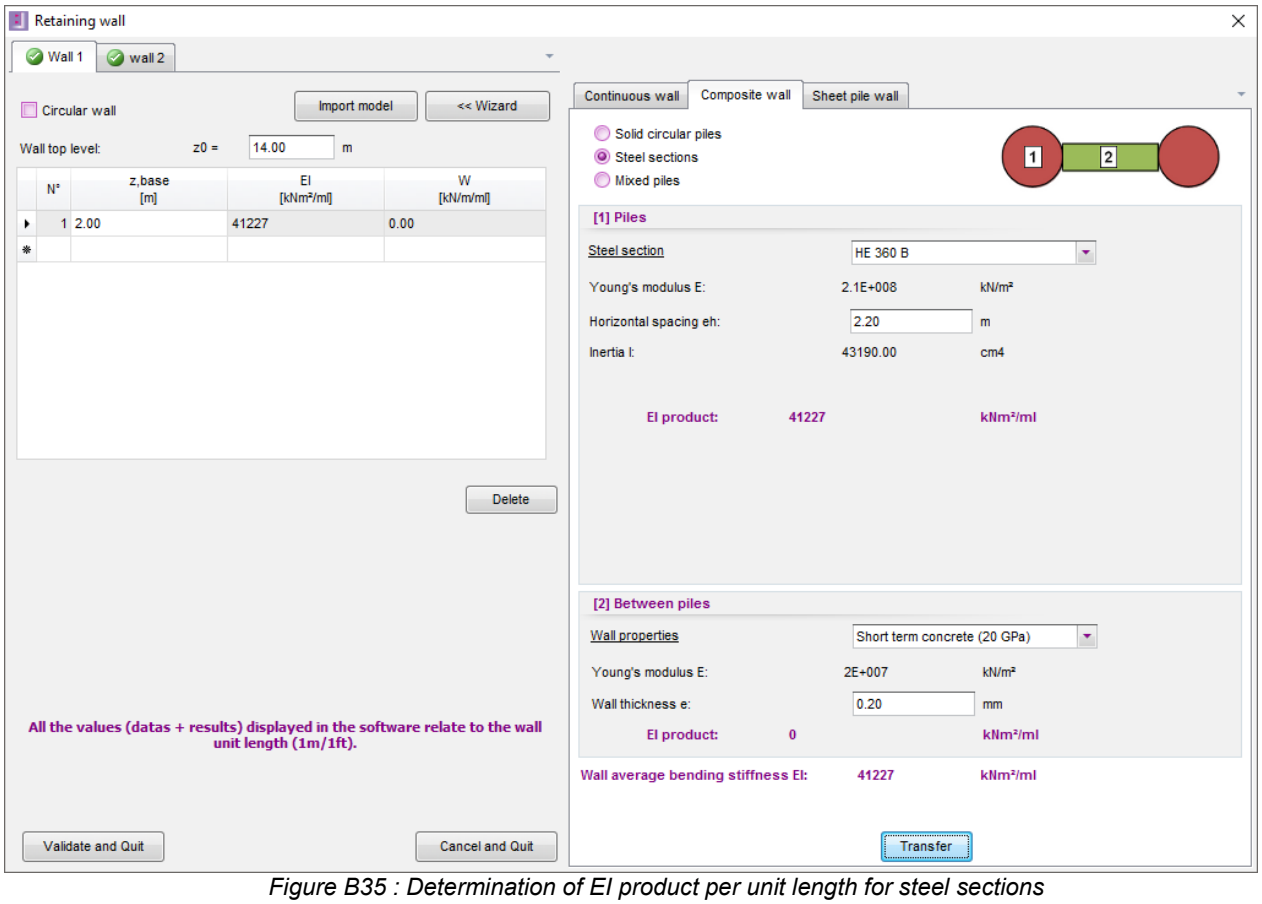


Figure B35 : Determination of EI product per unit length for steel sections

Then choose the type of section with the "Steel section" drop-down menu comprising the following categories:

- Special section: used to input an inertia value manually;
- Circular and hollow section: used to calculate the inertia for circular and hollow sections as a function of their outside diameter and thickness;

- IPE, IPE A, IPE O (*): European I beams;
- IPN (*): normal European beams;
- HE AA, HE A, HE B, HE M (*): European wide flange beams;
- HL (*): European very wide flange beams;
- HD (*): wide flange beams;
- HP (*): pile beams;
- TUBEUROP diameter/thickness: hot-finished hollow round sections extracted from the TUBEUROP range;
- TUBES STAD diameter/thickness: hollow round sections extracted from the TUBES STAD range.

(*) Data taken from the ArcelorMittal range.
Note: the list of supplier ranges does not claim to be exhaustive.
The following must be input in all cases:

- **e_h** : horizontal spacing between piles centers (m, ft).

If the user chooses "Circular hollow section" (case of tubes), they must also input:

- **d** : section diameter (mm, in);
- **e** : section thickness (mm, ft).

If however a "Special section" is chosen, the following must be defined:

- **I** : inertia of section (cm^4/m, In^4/ft).

The data then to be input to define the facing between the piles are as follows:

- **E** : Young's modulus of facing (kN/m^2, ksf);
- **e** : thickness of facing (m, ft).

At the bottom of the screen, K-Réa displays the equivalent EI product per unit length of composite wall.

Case of mixed piles

To calculate the EI product per unit length of a composite wall with mixed piles (steel section + concrete), click the "Composite wall" tab, then "Mixed piles", as shown in the following screenshot:

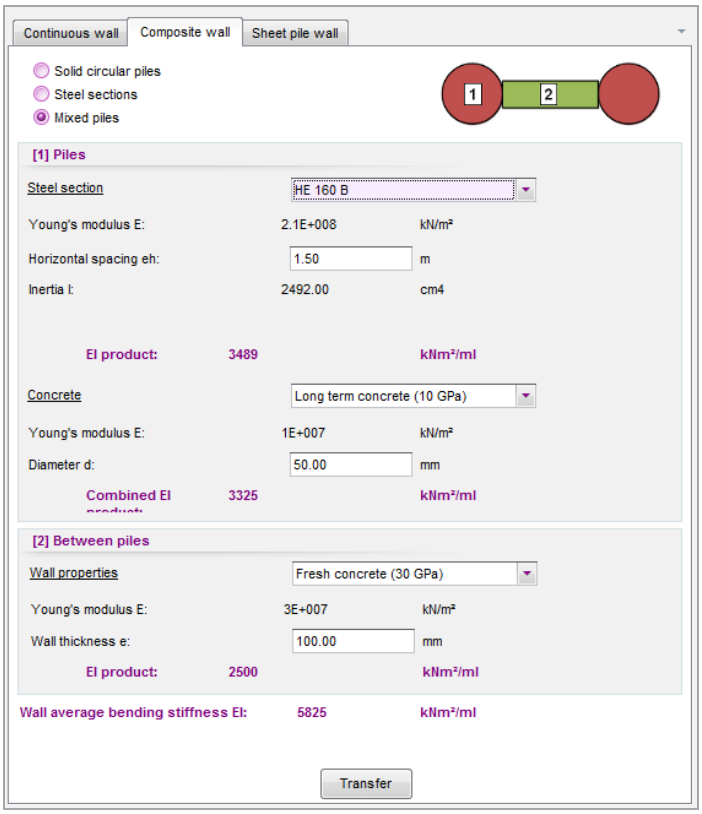


Figure B36 : Determination of EI product per unit length for mixed piles

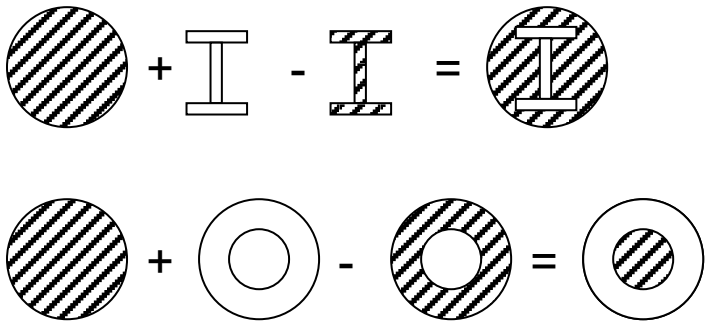
The following must then be input:

- for the pile: the characteristics of both the steel section and the concrete (see previous paragraphs);
- for the facing: its characteristics, in the same way as the rest of the composite walls.

The EI product of inertia of the mixed pile is calculated as follows:

$$(EI)_{\text{mixed section}} = E_{\text{concrete}} \times I_{\text{solid section}} + (E_{\text{steel}} - E_{\text{concrete}}) \times I_{\text{pile cross-section}}$$

This formula can be applied to several geometries:



c) Sheet pile wall

This wizard gives access to the catalogue of ArcelorMittal sheet piles.

Select the required type of wall then, depending on your choice, adjust the sheet pile characteristics:

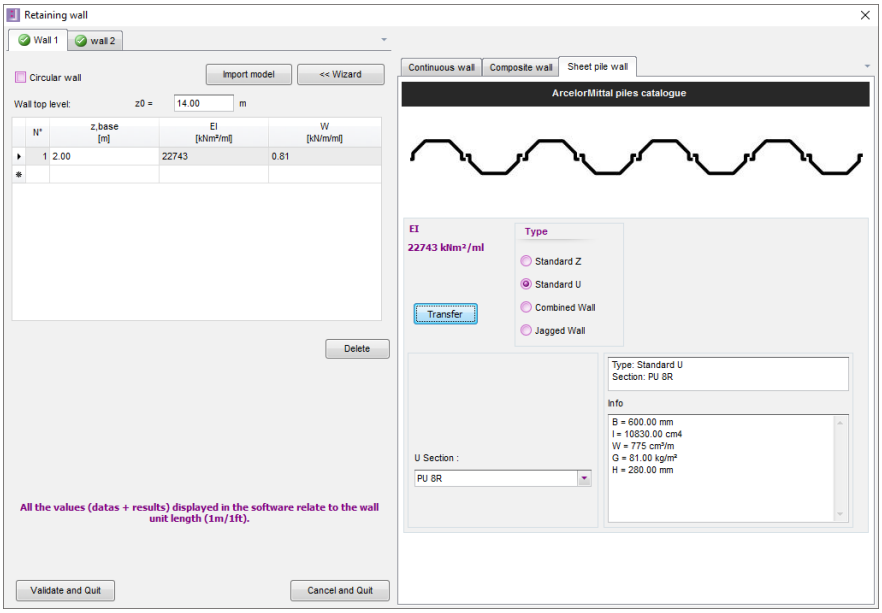


Figure B37 : Catalogue of ArcelorMittal sheet piles

Click the Transfer button to import the value of the product of inertia per unit length and that of the surface weight, into your project.

Depending on the type of sheet pile wall selected:

- Standard Z:

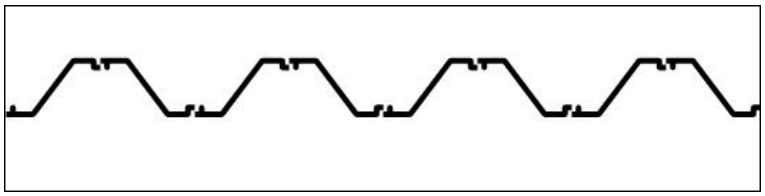


Figure B38 : Schematic of a " Standard Z" type sheet pile wall

Select the sheet pile section from the "Z Section" drop-down menu, then click "Transfer".

- Standard U:

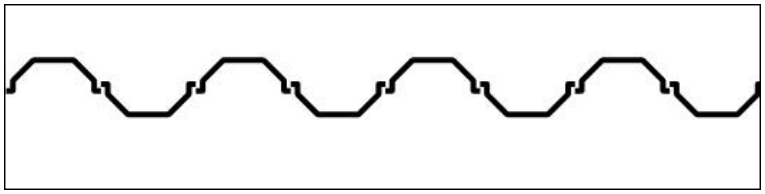


Figure B39 : Schematic of a " Standard U" type sheet pile wall

Select the sheet pile section from the "Z Section" drop-down menu, then click "Transfer".

- Combined wall:

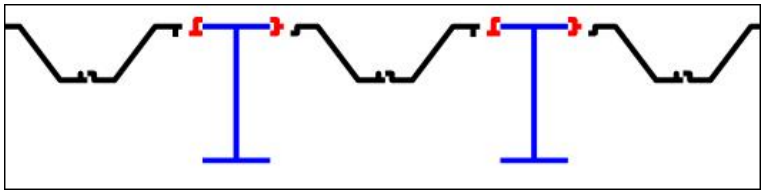


Figure B40 : Schematic of a combined wall

Select the type of combined wall and then its characteristics, first of all by choosing from the options displayed to the right of the type of wall, then from the drop-down menus lower down. Click the Transfer button.

d) <u>Jagged wall:</u>

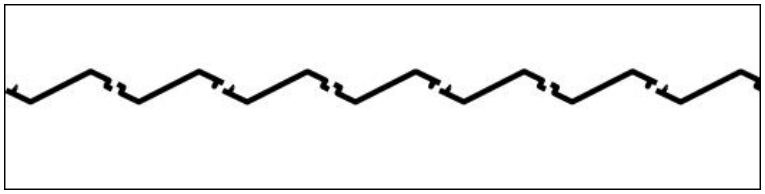


Figure B41 : Schematic of a jagged wall

Select the type of jagged wall and then its characteristics from among the proposed options. Click the Transfer button.

The frame at the bottom-right of the catalogue shows the characteristics of the sheet pile wall selected, that is:

- **B** : width of sheet pile wall (mm or in);
- **I** : inertia of sheet pile wall (cm^4/m or in^4/ft);
- **W** : modulus of sheet pile wall (cm^3/m or in^3/ft);
- **G** : weight of sheet pile wall (kg/m^2 or lb/ft^2);
- **H** : height of sheet pile wall (mm or in).

The actual catalogue is only available in the metric system, but in the case of a project using imperial units, all the values displayed are converted and appear on the screen in the imperial system (in addition to the values from the catalogue in the metric system). The EI product is also converted into $kip.ft^2/lft$.

B.3.4. Input of data for double wall projects

In the case of double wall projects, the data to be input are the same as those described previously, but they must be input for each of the 2 walls. The additional parameters and particularities encountered when defining a double wall project are presented below.

B.3.4.1. Title and options (case of double wall)

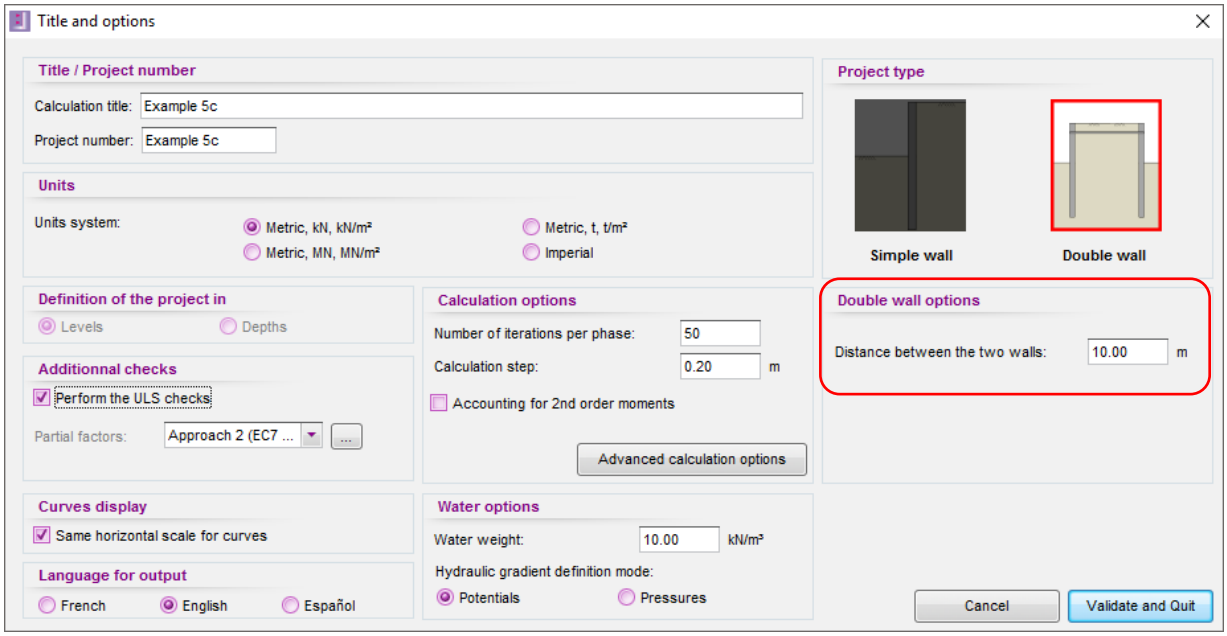


Figure B42 : "Title and Options" dialogue box (double wall project)

When choosing a double wall project type, an additional parameter needs to be filled out:

- Distance between the two walls (m, ft).

  It is important to note that this parameter is only used when defining the anchoring block, when ULS checks are requested. Its value has no influence on the other results.

Data input then refers to Wall 1 (left) and Wall 2 (right). See chapter B.1.2.2 for the conventions concerning these 2 walls and the choice of wall 1 and wall 2.

B.3.4.2. Definition of soil layers for a double wall

In the case of a double-wall project, the "Soil layers" definition dialogue box comprises two tabs, "Wall 1" and "Wall 2".

Input is exactly the same as for a single wall project (see § B.3.2).

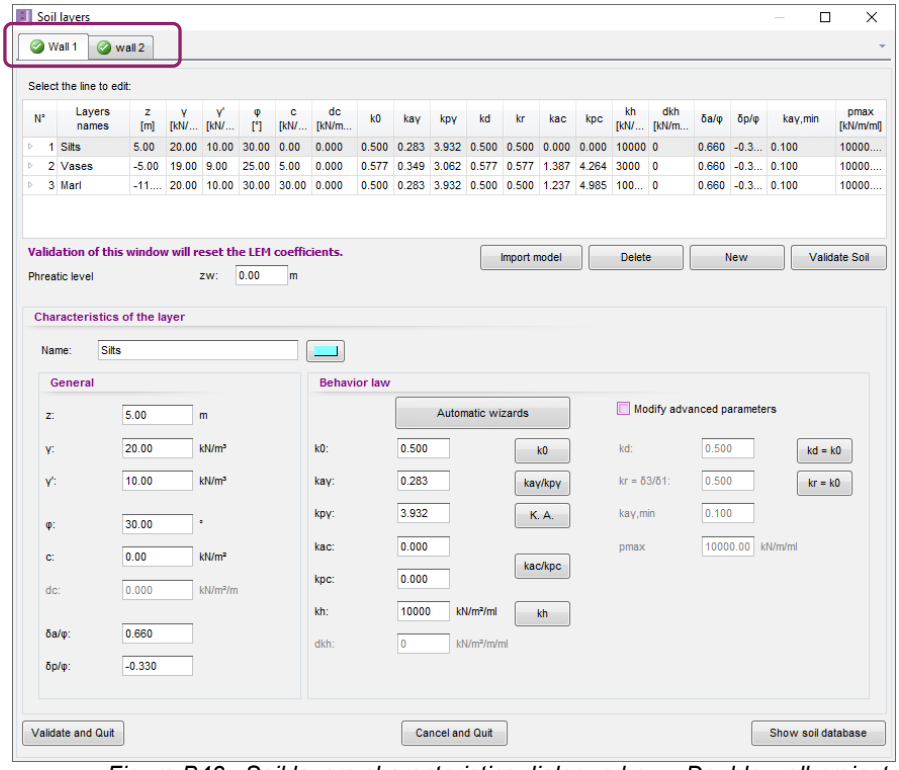


Figure B43 : Soil layers characteristics dialogue box – Double wall project

After entering the data for wall 1, it is possible to transfer the soil model from wall 1 to wall 2, by clicking the Import model button from the "Wall 2" tab.

B.3.4.3. Definition of characteristics of the two walls

The principle is exactly the same as that previously described for the definition of a single wall. The user must first of all input the data for wall 1 and then those for wall 2. As with definition of the soils, it is possible to export the data from wall 1 to wall 2.

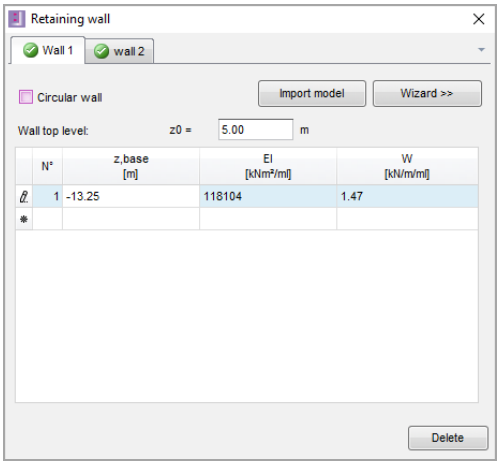


Figure B44 : Retaining wall definition window – Double wall project

B.3.5. Definition of load cases

In K-Réa it is possible to create several load combinations in a single project. Unlike the first 3 windows described in chapters B.3.1 to B.3.3, the definition of the load cases is optional.

The definition of the load cases involves three steps:

1. Activate the calculation of load combinations: open the **Definition of load cases** window accessible from the Wizards menu and tick the box at the top of the window;
2. Define the load families for the project, giving them a name and specifying the weighting factor to be allocated to each combination. This is to be done in the upper part of the window;
3. Define the calculation combinations to be considered (combination 1, 2, etc.) in each calculation phase. This is to be done in the lower part of the window.

When defining the phasing, each overload (on the soil or wall) will have to be assigned to a load family.

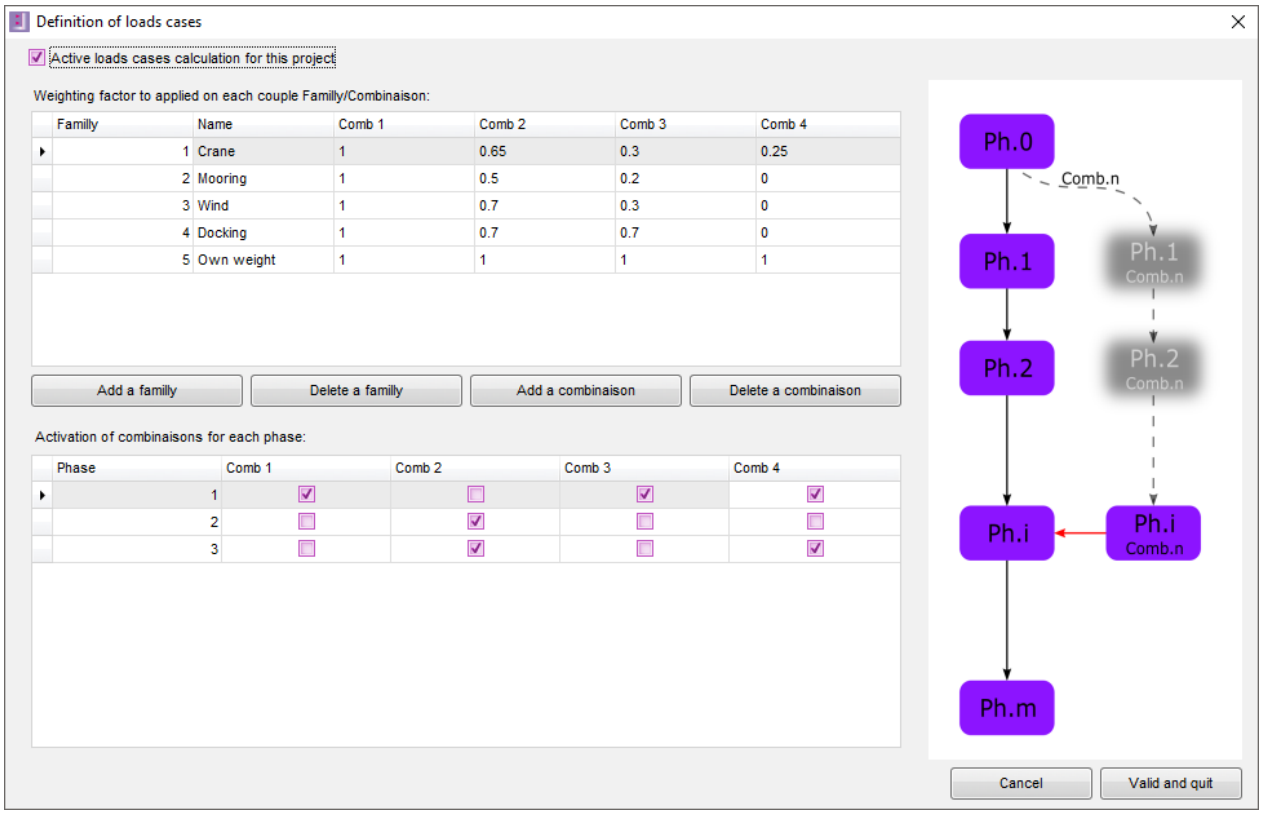


Figure B45 : Window for defining families and load combinations

The results of these load cases will appear in addition to those of the basic calculation. They can be accessed by choosing "Combinations" in the **Results** window and in the **EC7 Checks** window. This list of choices is accessible in all phases in which at least one combination was requested. If several combinations were considered in a given phase, a drop-down list can be used to navigate between them.

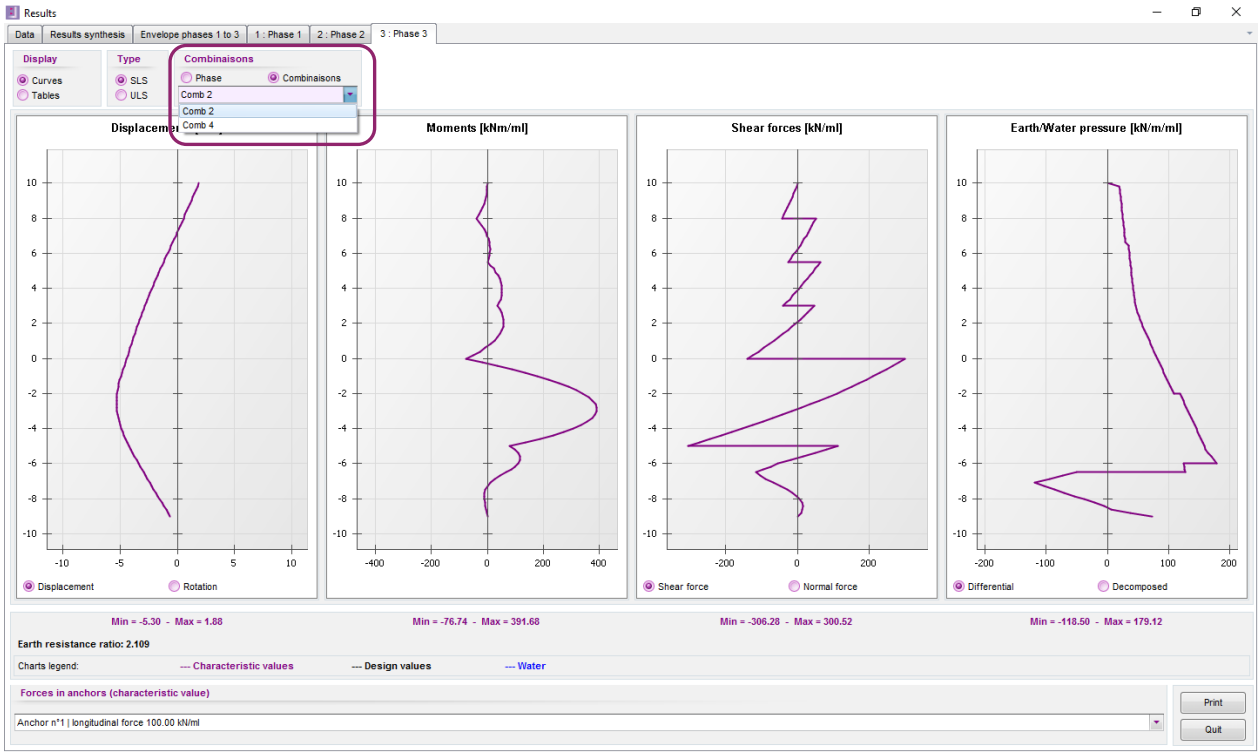


Figure B46 : Results window giving access to the results per load combination

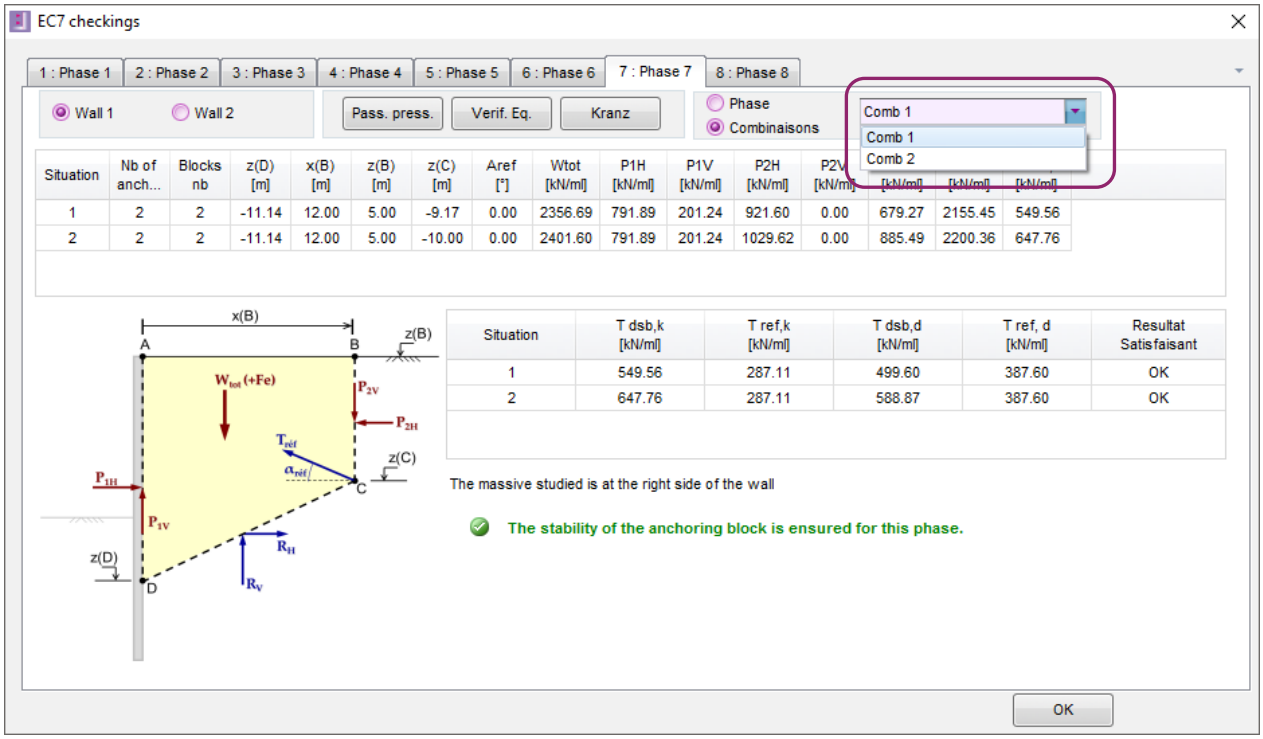


Figure B47 : EC7 Checks window giving access to the results per load combination

B.4. Definition of phasing

After inputting the soil and wall data, the calculation phasing must be defined, representing the project construction and service steps.

The choice of the kinematics of phasing may have a significant influence on the results, in particular owing to non-linearities as a result of soil plastification and changes in the stiffness of the wall and its anchors during the phasing.

Generally speaking, the phasing should be defined as close as possible to reality, breaking it down as much as possible and avoiding defining actions with opposite effects (fill followed by excavation for example) in a given phase, for a given side and wall.

This chapter gives a general description of the operations used to define the calculation phasing and the procedure for creating phasing for simple and double wall projects respectively. The actions that can be defined in a given phase are described in detail in chapter B.5.

B.4.1. Presentation

Phasing is managed via 3 zones:

- the phases management zone (create, delete, browse, etc.);
- the zone for choosing the actions to be applied in each phase;
- the zone for defining the parameters of each action.

The working of these various zones is explained in the following sub-chapters.

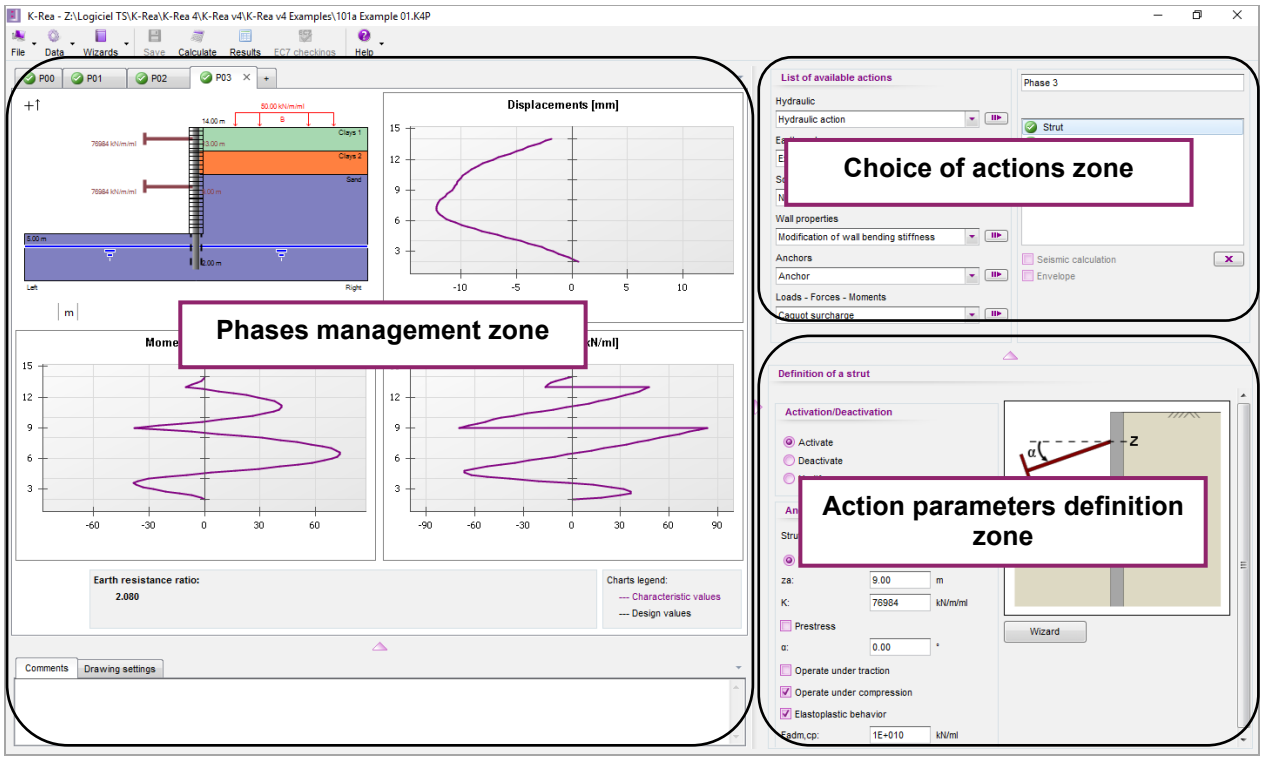


Figure B48 : Main window – Phasing management

B.4.2. Phases management frame

The following figure shows the phases management zone, situated on the left-hand side of the main window. It is used to create and control the calculation phases.

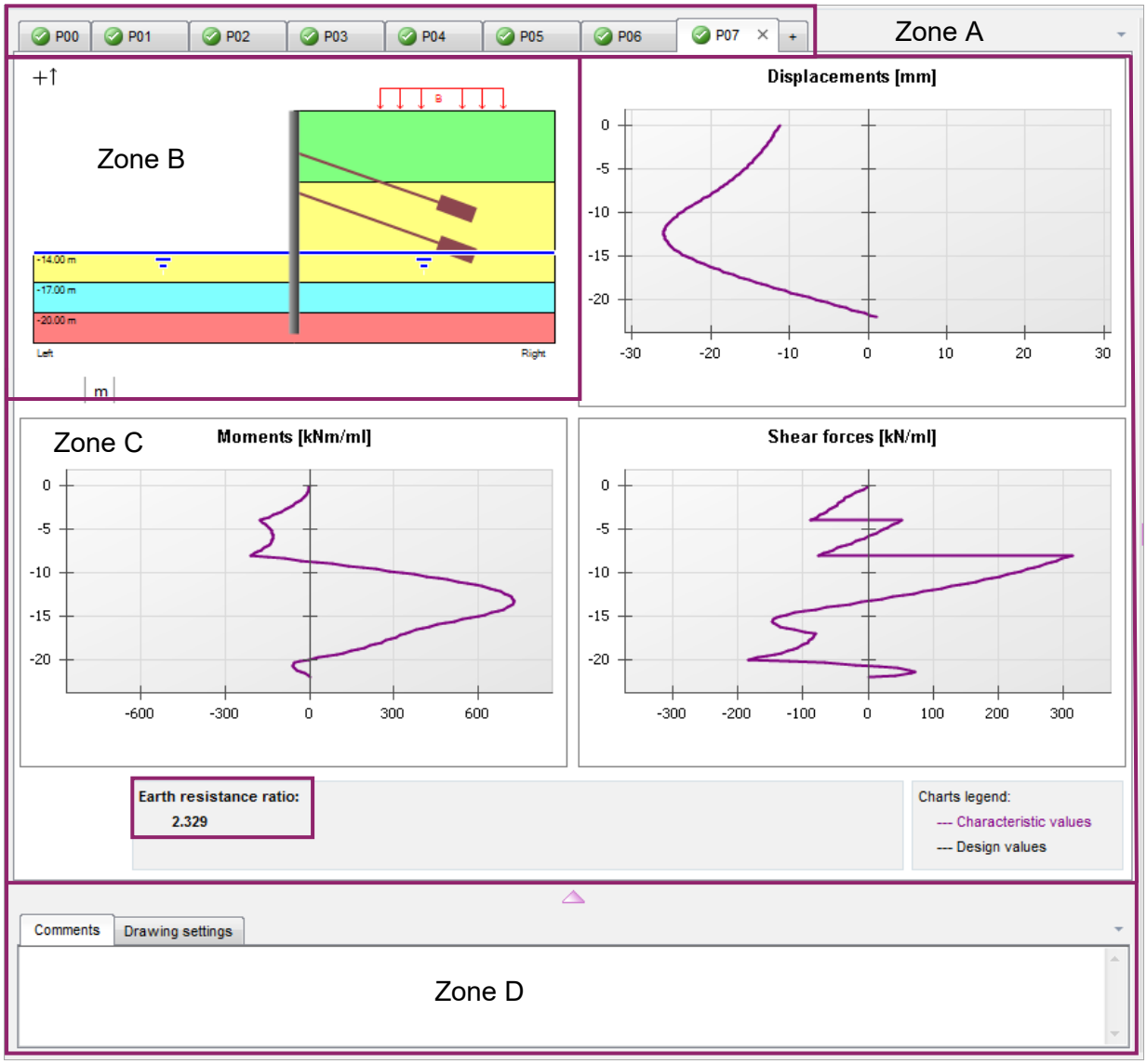


Figure B49 : Phasing management zones

Zone A presents the tabs corresponding to the calculation phases. It is possible to browse between the various phases of the project simply by clicking the tab of the phase one wishes to view.

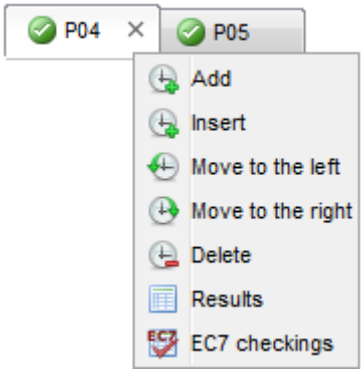


Figure B50 : Phasing management menu

- **Add**: adds a phase after the last phase;
- **Insert**: adds a phase to the left of the selected phase;
- **Move to the left**: shifts the selected phase to the left;
- **Move to the right**: shifts the selected phase to the right;
- **Delete**: deletes the selected phase, with a confirmation message;
- **Results**: opens the results window for the selected phase;
- **EC7 checks**: only accessible in projects for which the ULS checks are activated in the "Title and Options" menu. If this is the case, this button opens the EC7 checks for the selected phase.

Zone B is used to display the project cross-section corresponding to the current phase.

Zone C is used to display the results curves after starting the calculations. These are displacement curves, shear forces and bending moments diagrams concerning the wall for the current phase. The **earth resistance ratio(s)** on the excavation side are also displayed[1].

Zone D is reserved for drawings settings and comments. It is immediately below the phasing management frame and is used for:

- entering comments concerning the current phase. These are then recalled subsequently for information when printing the results.

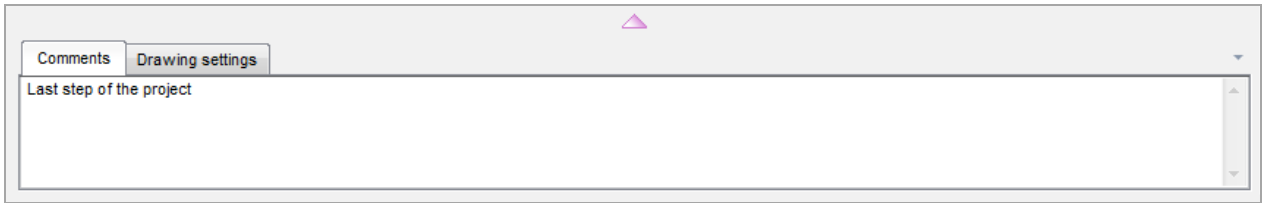


Figure B51 : "Comments" tab

- configuring the display, the project cross-section, the elevations of the soil layers and actions (groundwater, anchors, overloads, etc.), as well as for showing the names of the soil layers or the characteristics of the actions (stiffness of anchors, overload values, etc.).

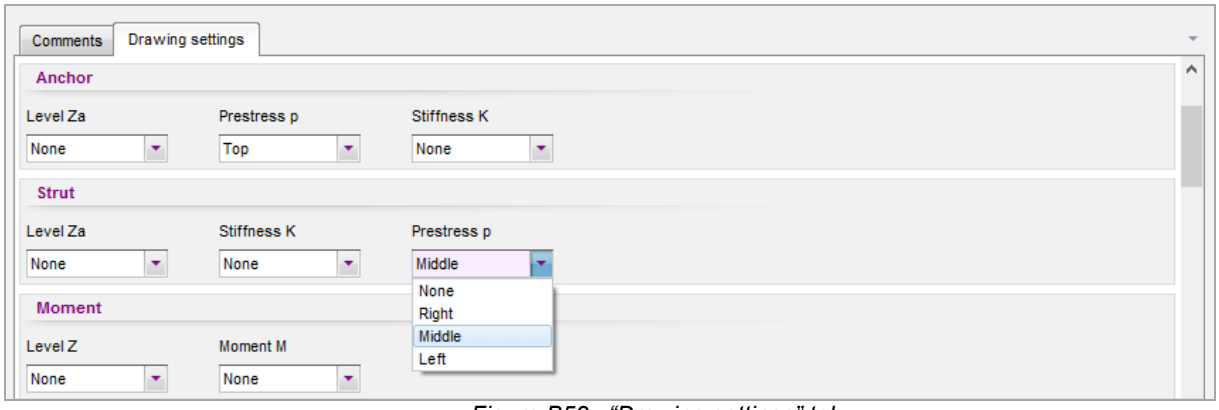


Figure B52 : "Drawing settings" tab

[1] The earth resistance ratio does not appear in zone C if the ULS checks are activated. This parameter is however still available in the "Results synthesis" (see § B.6.2.3).

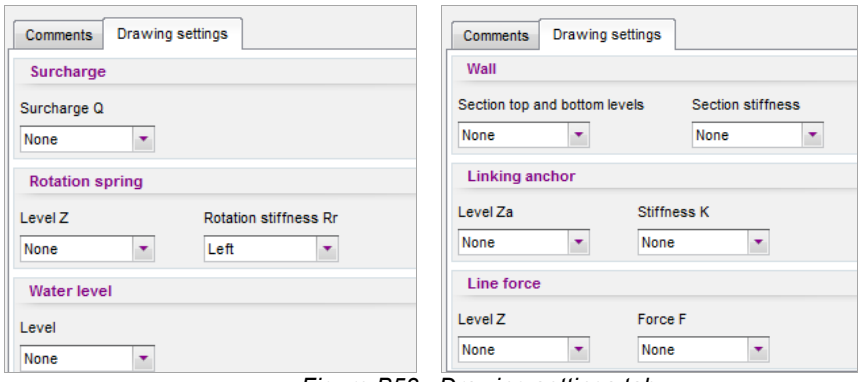


Figure B53 : Drawing settings tab
(examples of display settings for certain actions)

Note 1: the graphics can be enlarged to fill the entire phasing management frame by double-clicking on them. The mouse can be used to hover over the curves to display the value of the curve at the chosen point, in the form of a tooltip. In the case of a calculation with ULS check, two curves appear for the bending moments and the shear forces. The curve in pink/purple indicates the characteristic values (without weighting) and the black curve displays the design values (with weighting).
A right-click on these same graphics gives access to the context menu (see chapter B.2.3.8).
A double-click returns the graphic to its original size.

Note 2: in **zone B**, the graphical representation of the soil layers, walls and actions follows a vertical scale but not a horizontal one. The coordinates system used is not orthonormal and the distance between the two walls is not at the same scale as the vertical coordinates. The length and inclination of the tie-rods and the spacing between the two walls (for a double wall project) coming from the graphical representation should not therefore be considered.

Note 3: the space given to the project cross-section, the graphical results, the comments and the settings in the main window may vary:

- clicking the ▷ arrow enables all the figures to occupy the entire window;
- clicking/dragging the ▷ arrow enables the user to split the window horizontally to about two-thirds;
- clicking the △ arrow:
  - displays the "Comments" or "Drawing settings" tab over the full height of the window, or
  - displays the definition of the action over the full height of the window
- click/dragging the △ arrow enables the user to split the window.

B.4.3. Actions selection frame

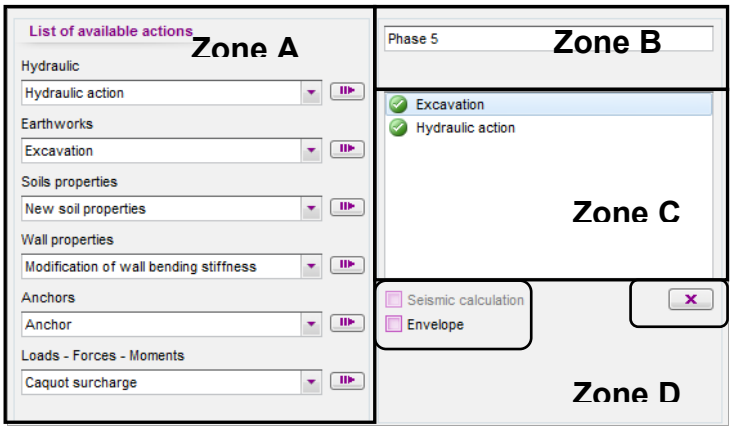


Figure B54 : Actions selection frame for a calculation phase

The choice of actions frame is displayed at the top-right of the main window. It can be used to apply the required actions in the selected calculation phase.

The title of the current phase is given in **zone B**. This title is recalled in the "Results" window to make it easier to read and interpret. It is possible to rename the selected phase by modifying the text input in this zone.

The actions in K-Réa on the whole define the wall support and loading conditions and their evolution during the construction phases. They can also be used to characterise the wall-soil and wall-anchors interactions.

These actions are classified by groups:

- **Hydraulic:** this group contains the hydraulic action used to define the groundwater levels and any gradients;
- **Earthworks:** contains the operations performed on the soil (excavation or fill) and offers the possibility of simulating bank or berm type geometries. It also includes an "install sheeting" action, the activation of which is condition-based (see § B.5.2.3);
- **Soil properties:** this group contains 2 actions. One is used to modify the inherent properties of the soil layers and the other to impose limit or at rest pressure diagrams;
- **Wall properties:** this group contains 2 actions. The first allows modification of the inherent properties of the wall, limited to its initial height. The second acts more on the structure of the wall and offers the possibility of defining wall extension upwards or downwards;
- **Anchors:** comprises 5 anchor types (tie, strut, clamping, circular waling and surface support) for single walls and 2 additional anchor types for double wall projects (linking anchor and slab anchor);
- **Loads, forces, moments**: comprises 3 types of overloads applicable to the soil (Caquot, Boussinesq and Graux), as well as 3 types of loads applicable directly to the wall (line force, linear moment and horizontal load).

Zone A contains drop-down menus corresponding to each group, for choosing the actions to be applied in the current phase.

The complete table of available actions and the detailed description of the parameters of each action are given in chapter B.5.

To apply an action, click inside the frame of the drop-down menu corresponding to the action group to be carried out.

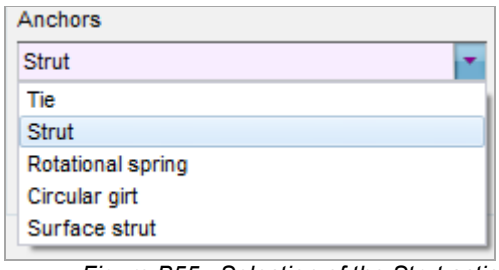


Figure B55 : Selection of the Strut action

Then click the transfer button  to include it in the list of actions to be performed (**zone C**) during the current phase.

It then appears in the definition frame for the action selected under the choice of actions frame. The parameters needed to define the action are to be input in this frame.

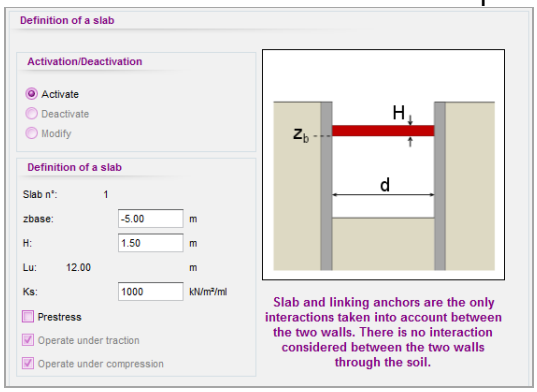


Figure B56 : Definition of parameters of an action

All the actions in a phase will appear in the form of a list classified in order of declaration in **zone C** situated under the name of the current phase.

Note 1: the  button is used to delete the selected action from the list of actions for the current phase.

Note 2: the actions are marked:

- with a green tick when correctly defined;
- with a red cross when incomplete or not correctly defined.

A tooltip in the action definition frame and in the list of actions gives a certain amount of information about the reason for the input being considered invalid:

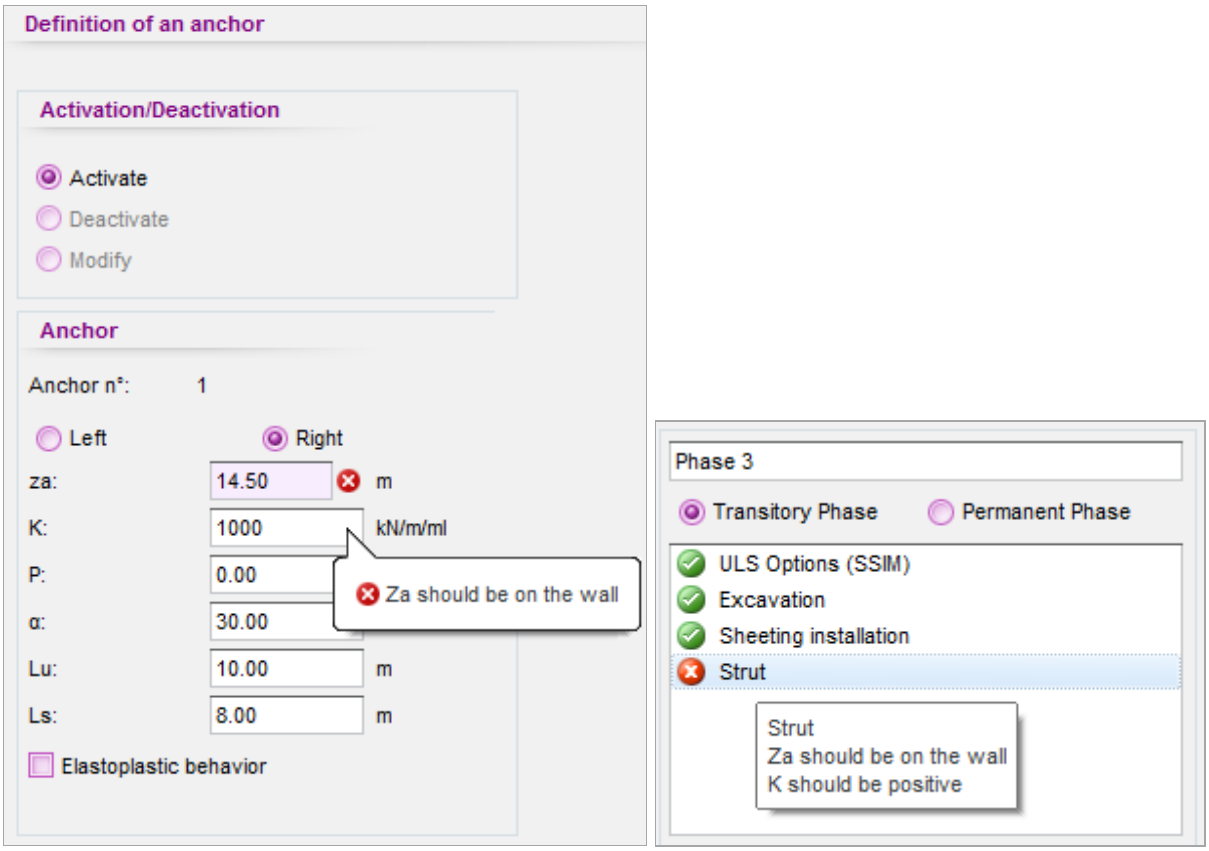


Figure B57 : Information about the reason for input being considered invalid

The mechanical properties of the actions are described in chapter B.5.

In the case of a project with ULS checks, two additional selections are needed to characterise the nature of the phase and the calculation model:

- **Transitory phase / Permanent phase**: this choice determines selection of the partial factor for the mobilisable passive earth pressure considered in the ULS passive earth pressure safety check (see § B.3.1.2);
- **Cantilever wall (LEM calculation):** this choice is only accessible for phases with no "active" support (element of "Anchors" group). It is activated by default in this case but can be deactivated by the user whenever they wish. If the box is ticked, the wall will be considered to be cantilever for the phase selected and the ULS checks will be made on the basis of a LEM calculation instead of a SSIM calculation (see part C of the manual). If the box is ticked, K-Réa will automatically create a **LEM coefficients** action (see § B.5.7.1).

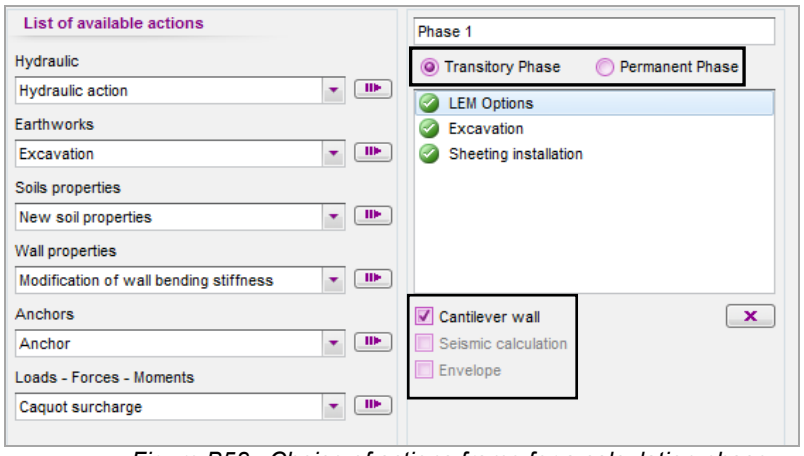


Figure B58 : Choice of actions frame for a calculation phase (case of a project with ULS checks)

In any case (projects with or without ULS checks), the **"Seismic calculation"** and "**Envelope"** check boxes are available.

The **"Seismic calculation"** check box is used to reserve the phase in question for a calculation with earthquake. The particularity of such a phase is that:

- it is a dead end in that it does not modify the main phasing consisting of "non-seismic" phases;
- no action in the "choice of actions" frame can be added by the user. Conversely, the "Seismic calculation" option is deactivated and inaccessible in the phases containing at least one non-automatic action.

This action is defined in detail in § B.5.8.

The "**Envelope"** check box is used to assemble the results curves from several successive phases. For example:

- If no "Envelope" box is ticked in a project, an envelope curves family will appear and collate the extreme displacement, shear force and bending moment values calculated on the basis of all the phases calculated.
- If a project contains 5 phases in addition to the initial phase and the "Envelope" box was ticked only in phase 3, two envelope curves (extreme displacement, shear force and bending moment values) will be represented for phases 1 to 3 and 4 to 5 respectively.

Note: The envelope box cannot be selected in the initial phase, nor in phase 1 (the results of phase 1 constitute their own envelope) nor in the last phase.
The **Delete** button deletes the selected action from the list of actions for the current phase. Caution, to simplify the operation, the action is deleted without requesting confirmation.

B.4.4. Actions definition frame

The actions definition frame appears at the bottom-right of the K-Réa interface (just below the choice of actions frame). It is used to define the selected action.

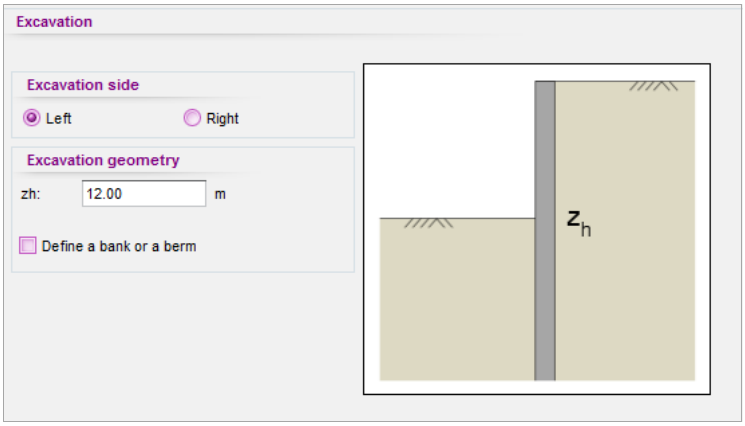


Figure B59 : Actions definition frame (example of an excavation)

Each action has its own actions definition frame. It generally consists of a left-hand part for inputting parameters defining the action and a right-hand part showing a schematic illustrating these parameters.

Figure B 59 shows the definition frame for an excavation. The list of choices appears in pink/purple and the boxes to be filled out appear in white. This example illustrates excavation on the left-hand side to level -5.00 m.

All the definition frames are explained in chapter B.5 in the paragraphs dedicated to the corresponding actions.

B.4.5. Validation / Calculation / Results

An action is validated if its mandatory parameters have been duly filled out. It is then given a green tick in the list of actions.

The Calculate and Results buttons are accessible from the buttons bar.

- starts the calculation;
- opens the results window (deactivated if the project has not yet been calculated).
- opens the EC7 checks results. This button is only active if the ULS checks were requested in the project options and if the project has been calculated. These checks are also accessible from the results window, for each phase (see chapter B.6.4).

B.4.6. Definition of phasing for a "Single wall" type project

B.4.6.1. Projects without ULS checks

By default, K-Réa always creates a first calculation phase "P00" entitled "initial phase", as illustrated in the following figure:

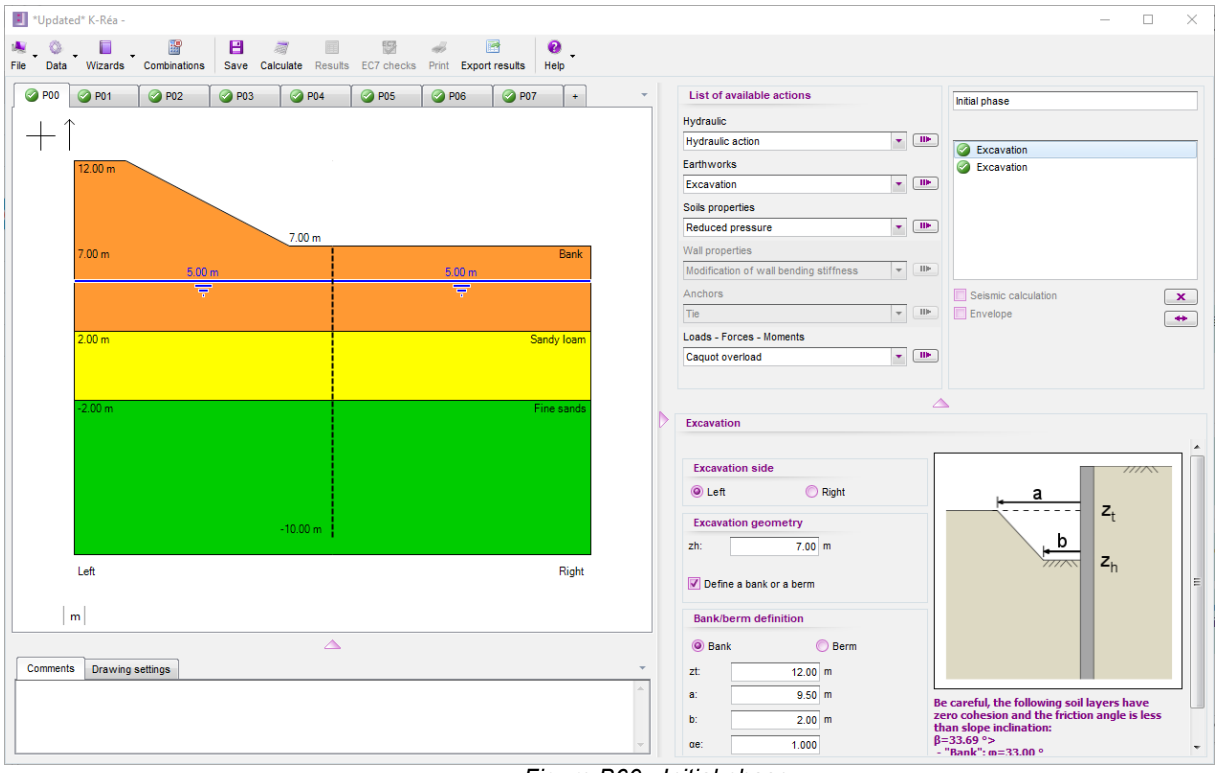


Figure B60 : Initial phase

The actions available in the initial phase are as follows:

- "Hydraulic";
- "Excavation" and "Fill";
- "Reduced pressure";
- "Caquot overload" and "Boussinesq overload".

The working of the actions frame was described in chapter B.4.3 and details of the actions are given in §B.5. If actions are applied in the initial phase, they will be represented on the project cross-section after validation.

Then, to create phase 1, click the + tab to the right of that of the initial phase, or use the context menu of the initial phase tab (right-click then "Add"). This action will create a new tab "P01" which by default will be named "Phase 1". The actions required for this phase 1 must be defined (as a function of your project). After each action creation in phase 1, the graphical representation of the project will be updated accordingly.

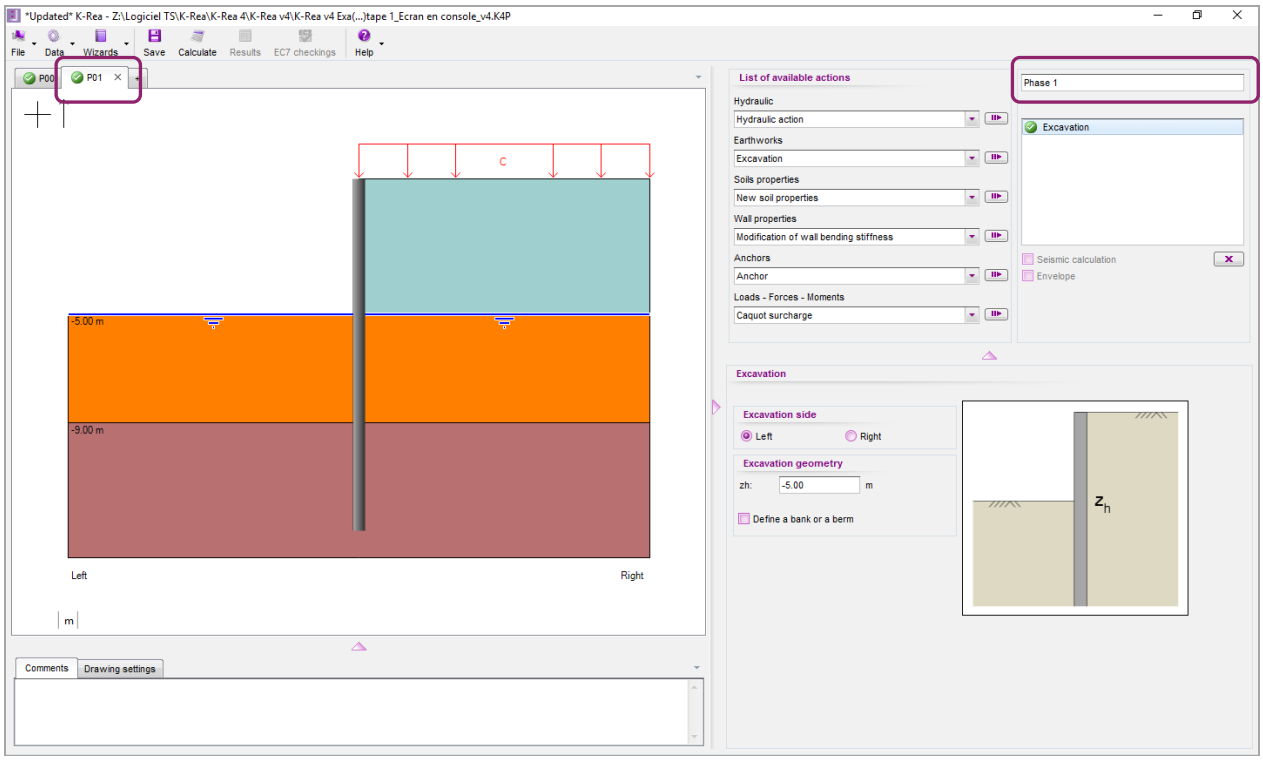


Figure B61 : Creation and display of phase 1 of a project

Then simply do the same for the following phases, until the phasing for the project concerned has been completed. Each time a new phase is added, the corresponding tab will appear after the existing tabs and will carry a name such as "PXX" where XX corresponds to the phase order in the project. By default, it will also be given a modifiable title such as "Phase XX".

The tabs in the phases management frame allow rapid browsing between the project calculation phases simply by clicking (during definition of phasing but also after calculation to view the results).

B.4.6.2. Projects with ULS checks

In the case of a project with ULS checks, K-Réa requires the input of additional data for definition of phases and actions.

Each phase must therefore be defined either as a **Transitory phase** or a **Permanent phase**.

For each phase, one must also state whether the wall is to be considered cantilever (LEM calculation) or anchored (SSIM calculation). The **Cantilever wall** option is automatically deactivated and inaccessible when an anchor is active in the phase in question, except for a tie without pre-stressing, which is not active in its installation phase.

In the phases for which the wall is considered to be cantilever, K-Réa automatically creates an **LEM options** action. This latter can be used to check over-excavations, the LEM calculation method used, the passive earth pressure elevation and, if necessary, the counter passive earth pressure parameters, its inclination angle in particular, for the current phase (see chapter B.5.7.1).

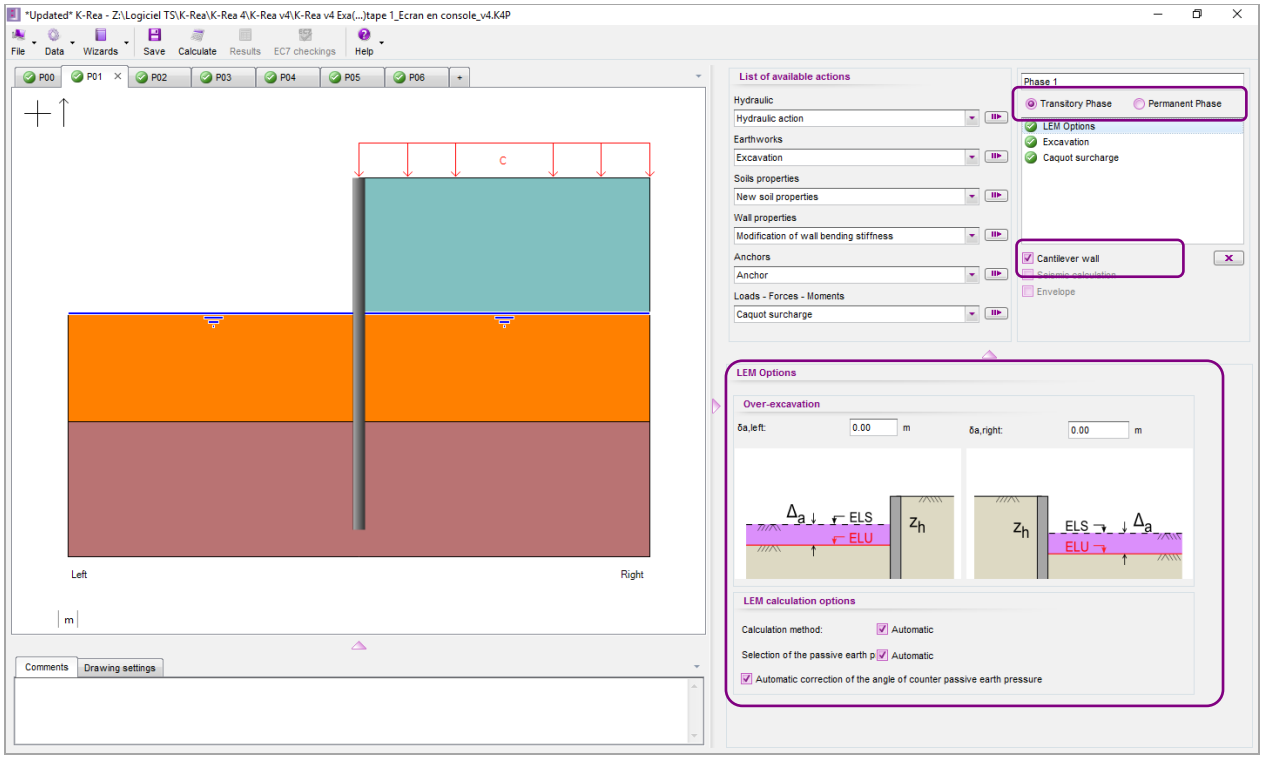


Figure B62 : Example of additional data for projects with ULS checks if the wall is considered to be cantilever

In the phases for which the wall is anchored, K-Réa automatically creates an **ULS options (SSIM)** action. This is used to check any over-excavations (see chapter B.5.7.1 and chapter B.5.7.2).

B.4.7. Definition of phasing for a "Double wall" type project

The phasing creation/management principle for a double wall project is the same as for a single wall project. The actions to be applied to each of the 2 walls will have to be defined.

Two buttons therefore appear at the top of the definition frame for each action, for allocation to wall 1 or wall 2. This concerns all the actions compatible with a single wall, the only exceptions being linking and slab anchors.

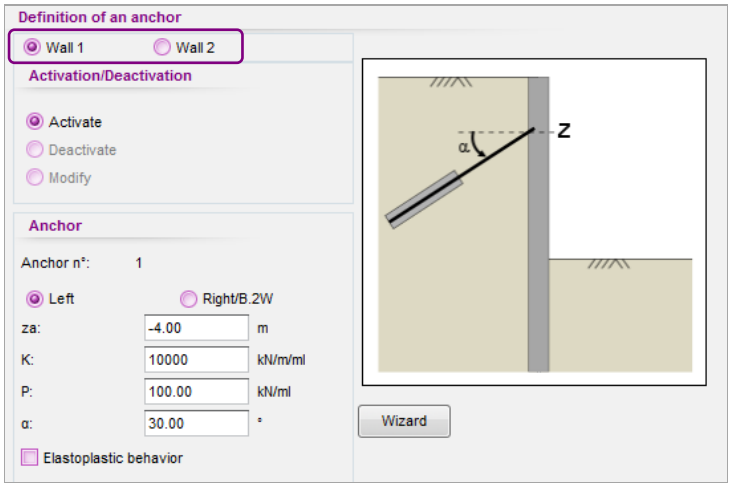


Figure B63 : Double-wall project: choice of Wall 1 / Wall 2

Moreover, the actions for the phase in progress carry the prefix of the wall to which they are linked:

- Wall 1: text in black;
- Wall 2: text in blue.

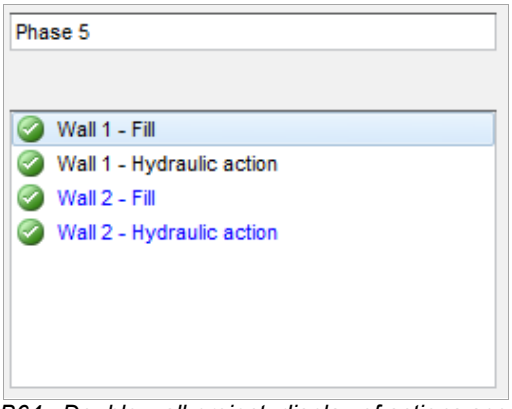


Figure B64 : Double wall project: display of actions specific to each wall

Finally, the two walls can be connected by one or more anchors, with no limit on their number. These latter may be either linking anchors (defined as ties or struts) or slabs. The interactions between the two walls are through these anchors only. K-Réa considers no interaction between the 2 walls via the soil medium separating them (see part C of the manual).

To find out more about the anchors, refer to chapters B.5.5.6 and B.5.5.7.

B.5. Description of actions defined during phasing

K-Réa proposes a total of 25 actions, 3 of which are automatic, for simulating the construction phasing of a given project. The following table presents these actions by group:

Hydraulic	Hydraulic action (1)
Earthworks	Excavation (1) Fill (1) Sheeting installation (solider pile)
Soil properties	Reduced pressure (2) Redefinition of soil layers Imposed pressure diagram
Wall properties	Modification of wall bending stiffness Wall modification
Anchors	Tie Strut Rotational spring Circular waling Surface strut Linking anchor (3) Slab (3)
Loads, forces, moments	Caquot overload (1) Boussinesq overload (1) Graux overload Apply a line force Apply a moment Horizontal load on wall
Actions created automatically	LEM options (4) ULS options (SSIM) (4) Earthquake (Seismic calculation) (4)

(1) This action exists in the initial and other phases;
(2) This action exists only in the initial phase;
(3) This action is exclusively available for double wall type projects;
(4) This action is automatically created by K-Réa according to the status of the options detailed in § B.4.3. The user must always check the pre-defined parameters, or even modify and supplement them if such an action is present.
The remainder of the actions only exists in the standard phases, or in all the phases except for the initial phase.

Table B 3: Actions available for definition of phasing

An action is applied by means of the actions selection frame described in chapter B.4.3.

This chapter describes the actions in detail. Each sub-paragraph will comprise the principle of an action, possibly illustrated by a schematic or screenshot, followed by the designation of the corresponding input data.

B.5.1. Hydraulic action

This command is used to define the hydraulic conditions and if necessary correct the water pressures to take account either of a hydraulic gradient associated with the flow regime involved with groundwater drawdown, or the presence of impermeable passages or perched aquifers in the various layers.

It is important to note that definition of a hydraulic action has an impact both on the calculation of the horizontal water pressure on the wall and on that of the effective vertical stress in the soil (for more information, consult part C of the manual).

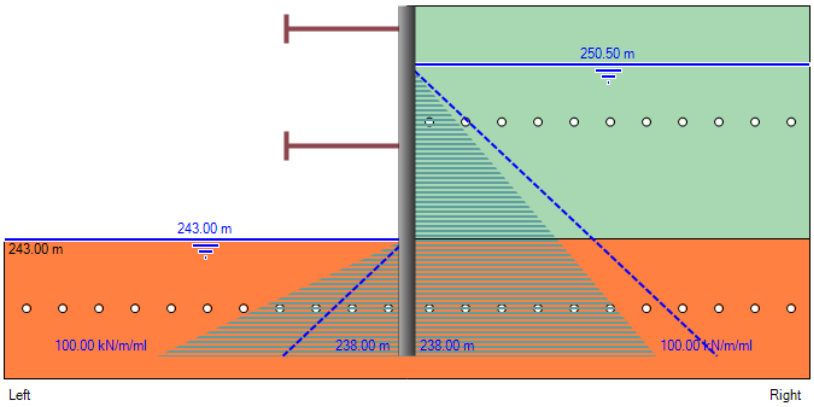


Figure B65 : Example of definition of a hydraulic gradient

Parameters to be filled out:

- Side concerned by the hydraulic action:
  - **"Left** or **Right"** for a single wall project;
  - "**Left** or **Right/E.2R"** (wall 1) or "**Left/E.2R** or **Right"** (wall 2) for a double wall project;
- z_w: level (elevation or depth) of the hydrostatic water table.

It is also possible to impose a pressure or potential diagram, for example to define a hydraulic gradient. To do this, the corresponding box must be ticked and the following filled out point by point:

- **Level** (elevation or depth) of the diagram point;
- **Hydraulic Potential** or **Pressure** (according to the option chosen in the "Title and options" window) to be considered for this point.

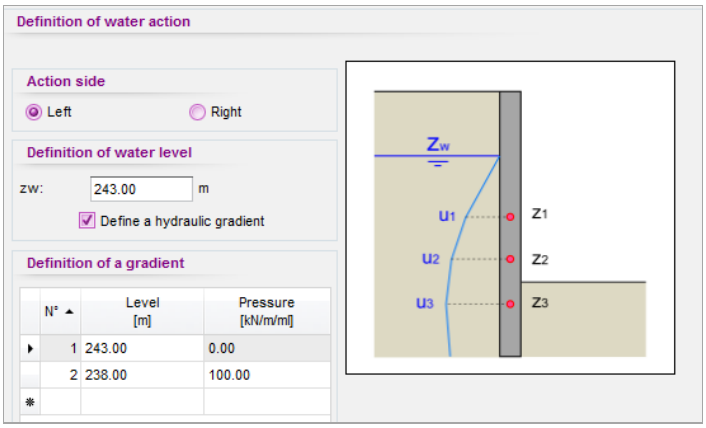


Figure B66 : Water action definition frame

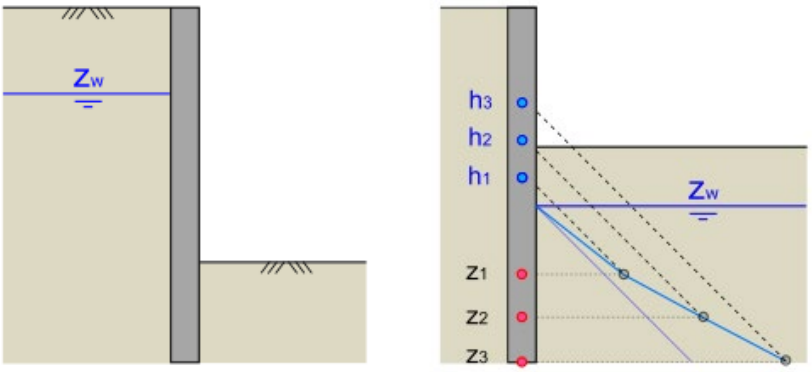


Figure B67 : Definition of hydraulic conditions: hydrostatic regime (left) and hydraulic gradient (right)

B.5.2. "Earthworks" actions

B.5.2.1. Excavation

This action is used to define an excavation on one side of the wall. The excavation level is by default horizontal. An additional option allows definition of an excavation in the form of a bank or berm.

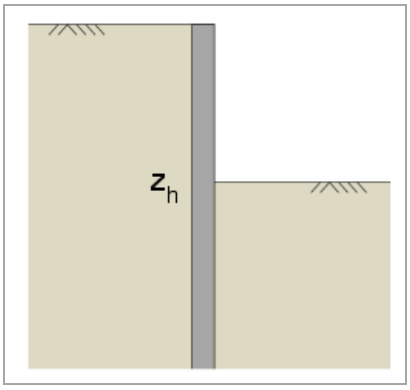


Figure B68 : Definition of an excavation

Parameters to be input:

- Side concerned by the excavation:
  - **"Left** or **Right"** for a single wall project;
  - "**Left** or **Right/E.2R"** (wall 1) or "**Left/E.2R** or **Right"** (wall 2) for a double wall project;
- z_h: excavation level (elevation or depth) (m or ft).

It is also possible to define an excavation in the form of a bank or berm. The additional parameters to be input are:

- **"Bank** or **Berm**" according to the geometry to be defined;
- z_t: level of head of bank or berm (m or ft);
- **a**: distance between top of bank or berm and wall > 0 (m or ft);
- **b**: distance between base of bank or berm and wall > 0 (m or ft);
- α_e: multiplication factor (initially set at 1.00) used to correct the Boussinesq overloads (see § B.5.6.2 and part C of the manual).

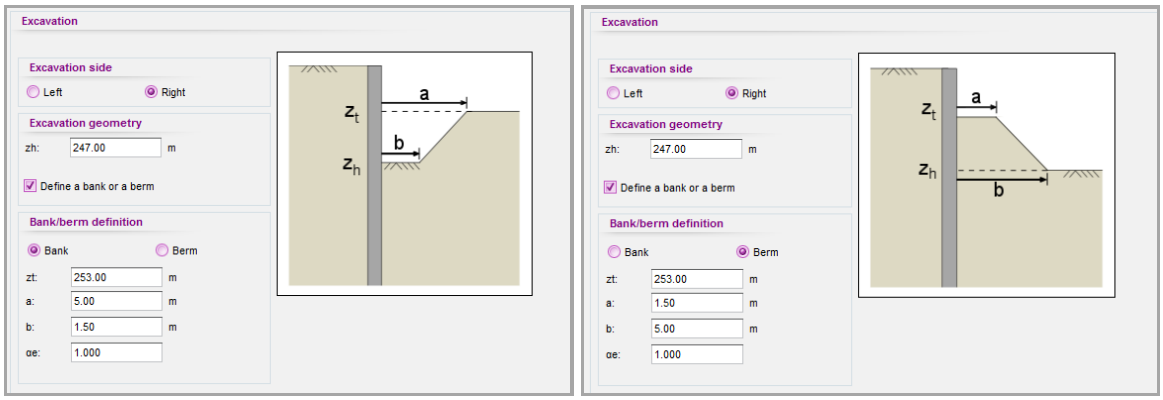


Figure B69 : Definition of an excavation in the form of a bank (left) or berm (right)

A few recommendations concerning the "excavation" action:

This action cancels the Caquot, Boussinesq or Graux type overloads on the soil on the side concerned by the excavation. To maintain them, they have to be redefined.

K-Réa does not verify the stability of the bank or berm. This type of verification is the responsibility of the user and must be done beforehand using an appropriate model (for example with Talren v5).

A horizontal excavation action systematically cancels out the geometric effects linked to the existence of a bank or berm, regardless of the side of the earthworks.

The level z_h of a new excavation must be below that of the previous level. In the case of a bank or berm, the z_t level must also meet this condition.

The lower level of a fill (z_b) will by default be considered to be equal to the level in contact with the wall previously. If a bank or berm already existed, the level z_b will be equal to its upper level, that is $z_b = z_{t0}$.

	Former geometry	Subsequent action	
		Excavation	Fill
Project defined in levels	Bank between $[z_{h0}, z_{t0}]$	$z_t \leq z_{t0}$ $z_h \leq z_{h0}$	$z_t \geq z_{h0}$ $z_h \geq z_{h0}$
	Berm between $[z_{h0}, z_{t0}]$	$z_t \leq z_{t0}$ $z_h \leq z_{h0}$	$z_t \geq z_{t0}$ $z_h \geq z_{t0}$
Project defined in depths	Bank between $[z_{h0}, z_{t0}]$	$z_t \geq z_{t0}$ $z_h \geq z_{h0}$	$z_t \leq z_{h0}$ $z_h \leq z_{h0}$
	Berm between $[z_{h0}, z_{t0}]$	$z_t \geq z_{t0}$ $z_h \geq z_{h0}$	$z_t \leq z_{t0}$ $z_h \leq z_{t0}$

Table B 4: Criteria for defining excavation and fill levels after a bank or berm ground level geometry

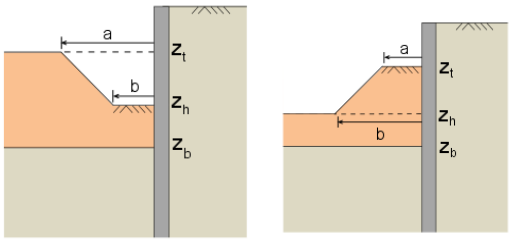


Figure B70 : Help diagrams for a fill action after a bank left) or berm (right) type geometry

B.5.2.2. Fill

This function is used for a fill with a base resting on the existing soil. The upper surface of the fill is by default horizontal, but a bank or berm type geometry can be defined using the appropriate option

The parameters to be input are:

- **Name of layer** constituting the fill;
- Colour of fill on the cross-section (choice from a colour pallet);
- Side concerned by the fill: **"Left** or **Right"** for a single wall; "**Left** or **Right/E.2R"** (wall 1) or "**Left/E.2R** or **Right"** (wall 2) for a double wall project;
- z_t: level of top of fill (m or ft);
- z_b**:** level of base of fill (m or ft) automatically pre-defined by the interface.

If the definition of a bank or berm is activated, the following additional parameters must be defined:

- **"Bank** or **Berm"** type of geometry to be defined;
- z_h: level of base of bank or berm (m or ft);
- **a:** distance between top of bank or berm and wall > 0 (m or ft);
- **b:** distance between base of bank or berm and wall > 0 (m or ft);
- α_e: multiplication factor (initially set at 1.00) used to correct the Boussinesq overloads (see § B.5.6.2 and part C of the manual).

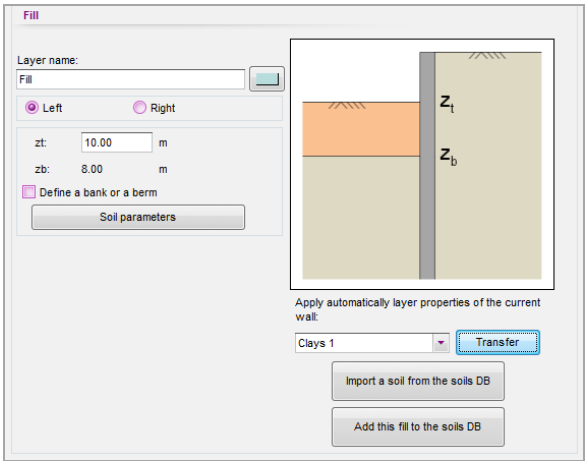


Figure B71 : Fill definition zone

It is possible:

- to input or modify the parameters of a fill by clicking the Soil parameters button;
- to import existing soil characteristics into the current project, by selecting from the drop-down list under the help diagram and clicking the Transfer button;
- to import existing soil characteristics into the database by clicking the Import a soil from the soils DB button;
- to save a fill in the soils database by clicking the Add this fill to the soils DB button.

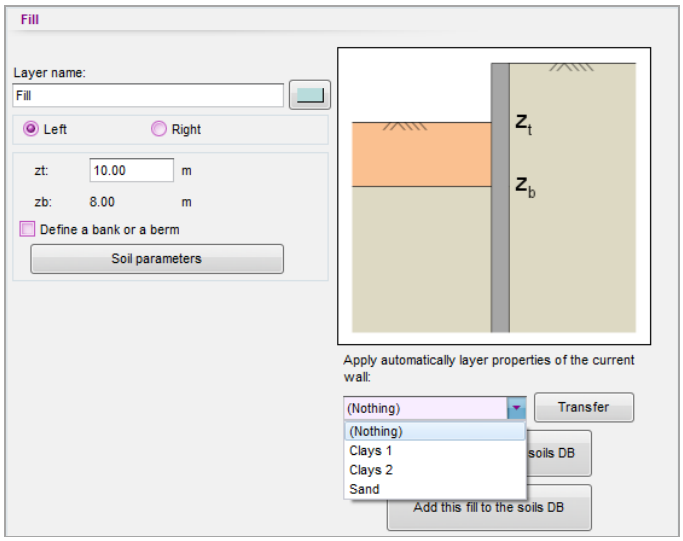


Figure B72 : Previously defined soil layer properties can be imported

To define the parameters of the soil making up the fill, click Soil parameters to open the input form.

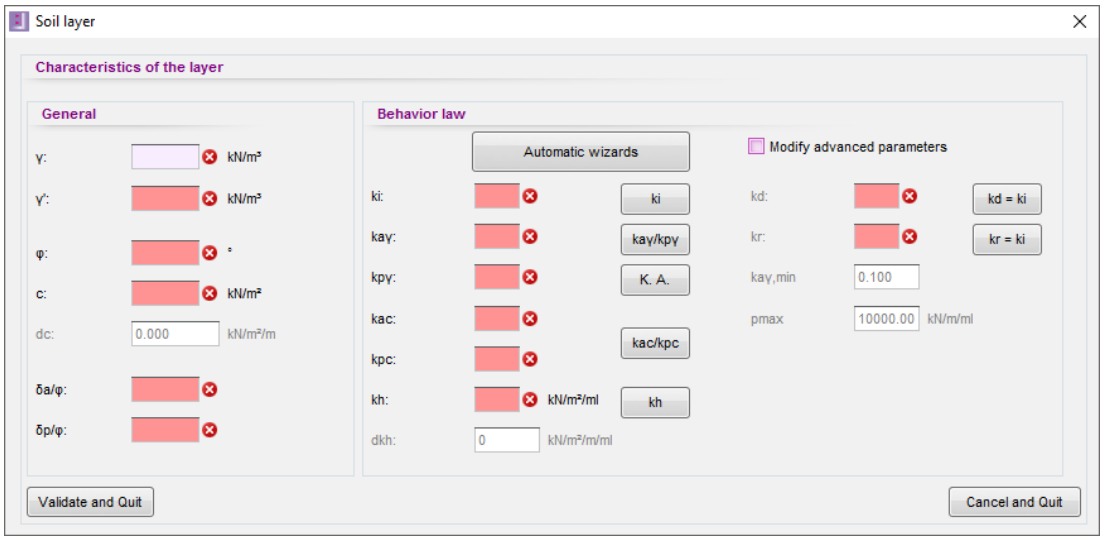


Figure B73 : Fill soil definition window

The parameters to be input are as follows:

- γ and γ': wet and buoyant unit weights (kN/m^3 or kcf);
- **φ:** internal friction angle (°);
- **c** and **dc**: cohesion and its variation until depth (kN/m^2 and kN/m^2/m or ksf and ksf/ft);
- δ_a/φ and δ_p/φ: angle of limit active and passive earth pressures;
- k_i: initial active earth pressure coefficient;
- $k_{a\gamma}$, $k_{p\gamma}$, k_{ac} and k_{pc}: active and passive earth pressure coefficients;
- k_h and dk_h: horizontal reaction coefficient and its variation with depth (kN/m^2/ml and kN/m^2/m/ml or ksf/lft and ksf/ft/lft);
- k_d and k_r: unloading and reloading ratios;
- $k_{a\gamma,min}$ and p_{max}: allowable minimum active pressure coefficient and maximum pressure.

Tick the "**Modify advanced** parameters" box to modify **dc**, **dk$_h$**, **k$_d$**, **k$_r$**, **k$_{a\gamma,min}$** and **p$_{max}$**.

The wizards present in this window correspond to the same wizards available in the soil layers definition window.

A few recommendations concerning the "Fill" action:

K-Réa does not verify the stability of the bank or berm. This type of verification is the responsibility of the user and must be done beforehand using an appropriate model (for example with Talren v5).

Part C of the manual details how a fill is taken into account, in particular regarding the installation phase.

The k_i coefficient is commonly taken within the range [k_a, k_0]: the use of k_0 generally corresponds to the case of compacted fill, while the use of k_a is to be preferred for fills put into place using gravity alone.

It is possible to create several successive fills with different characteristics on the same side of the wall (in successive phases).

This action cancels the Caquot, Boussinesq or Graux type overloads on the soil on the side concerned by the fill. To maintain them, they have to be redefined.

The lower level of a fill (z_b) will by default be considered to be equal to the level in contact with the wall previously. If a bank or berm already existed, the level z_b will be equal to its upper level, that is $z_b = z_{t0}$.

Table B3 summarises the application of these rules.

B.5.2.3. Sheeting installation (soldier pile)

This action simulates the installation of shielding for a composite wall (e.g.: soldier pile).

It is only available during phasing if a "**Reduced pressure"** action was defined during the initial phase.

The parameter to be filled out to define installation of sheeting is:

- z: lower level of sheeting installation zone (m, ft).

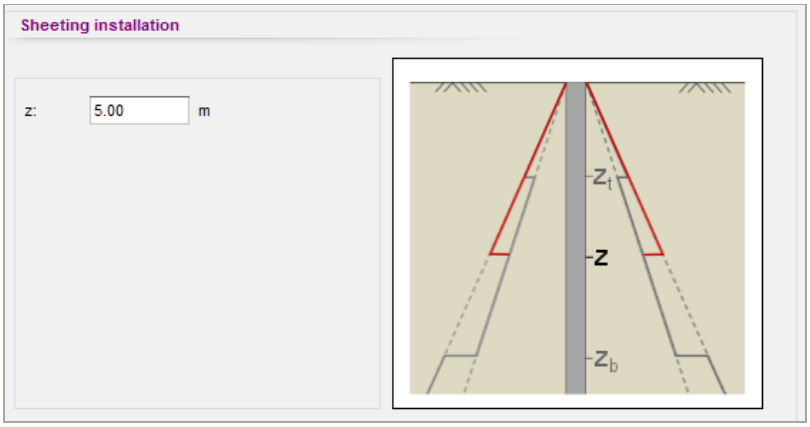


Figure B74 : Sheeting installation definition frame

The active earth pressure which was reduced by the "Reduced pressure" action in the initial phase is restored to 100% of its value above level **z,** in the same way as for passive earth pressure, water pressures and reaction coefficient.

Sheeting installation always starts at the top of the wall and is completed by descending through the successive installation phases.

B.5.3. Soil characteristics

B.5.3.1. Redefinition of soil layers

This action is used to modify the characteristics of a soil layer during phasing. The parameters of this layer can be modified on the left, right or both sides of the wall.

The following data are to be input:

- **"Wall 1"** or "**Wall 2"** only for a double wall project;
- "Left", "Right" or "Left and right";
- **Name of soil layer** to be modified: can be selected with the mouse from the drop-down list;
- New parameters: input window accessible via the Soil parameters button.

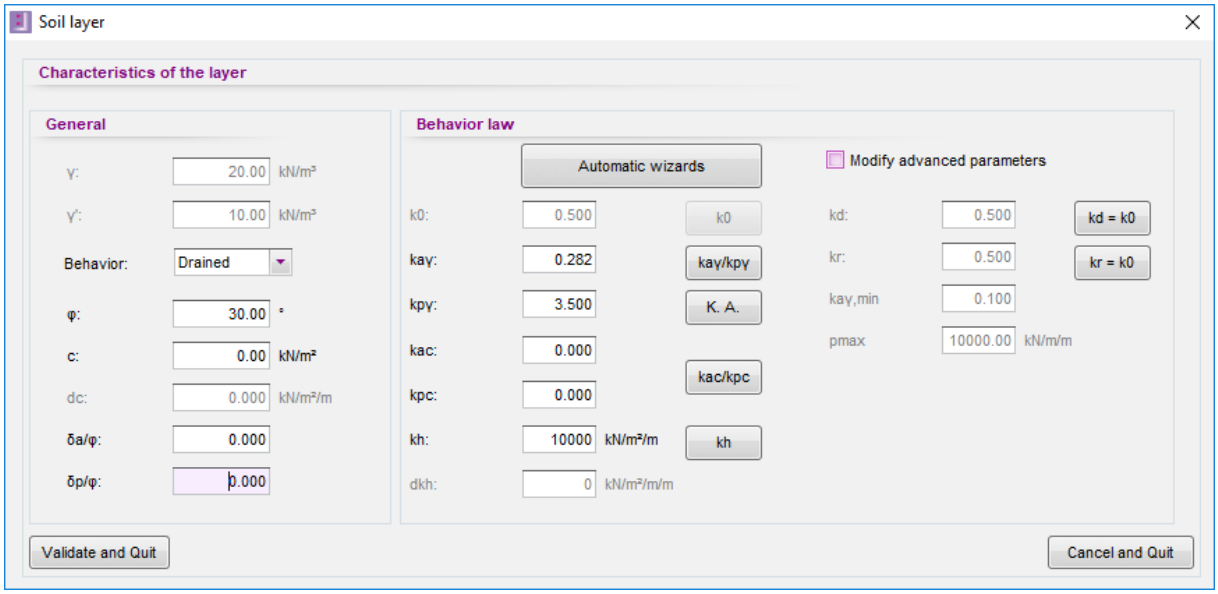


Figure B75 : Window for redefining a soil layer during phasing

The modifiable parameters are:

- **Behaviour**: drained or undrained (the choice only impacts the safety factor on cohesion and internal friction angle)
- **φ:** internal friction angle (°);
- **c** and **dc**: cohesion and its variation with depth (kN/m^2 and kN/m^2/m or ksf and ksf/ft);
- **δ_a/φ** and **δ_p/φ**: angle of limit active and passive earth pressure stresses;
- **$k_{a\gamma}$**, **$k_{p\gamma}$**, **k_{ac}** and **k_{pc}**: active and passive earth pressure coefficients;
- **k_h** and **dk_h**: horizontal reaction coefficient and its variation with depth (kN/m^2/ml and kN/m^2/m/ml or ksf/lft and ksf/ft/lft);
- **k_d** and **k_r**: unloading and reloading ratios;
- **$k_{a\gamma,min}$** and **p_{max}**: allowable minimum active earth pressure coefficient and maximum pressure.

Tick the "**Modify advanced parameters" box to modify dc**, **dk_h**, **k_d**, **k_r**, **$k_{a\gamma,min}$** and **p_{max}**. The non-modifiable parameters γ, γ' and **k_0**, are shown for information.

This action can be used to modify one or more soil layer parameters independently of each other. The wizards in the soil layers definition window are also available in this window.

Modification of the parameters applies to the layer chosen in the current phase. To modify the parameters of another layer, one must add another "**Soil layers redefinition"** action to the list of actions of the same phase, choose the other layer to be modified and input the new parameters. This procedure is to be repeated as many times as necessary.

In the new parameters definition frame for a soil layer, the Properties phase-1 button is used to automatically input the parameters by recovering all of those defined in the previous phase. Similarly, the Properties phase n 0 button can be used to import the initial parameters of the soil layer in question. The values copied can then be modified by consulting the soil parameters input wizard window: Soil parameters.

B.5.3.2. Imposed pressures diagram

This action can be used to impose the pressure diagrams (limit or at rest) on one side of the wall. It applies if the configuration of the terrain does not enable the active earth pressure coefficients (limit or at rest) applied by default in K-Réa to be used.

The limit active or passive earth pressures diagram can be fitted using the yield design kinematic calculation method implemented in Talren. The method enables all types of situations to be considered, in particular highly complex geometries (any natural terrain profile with soil layers that are not necessarily horizontal and regular). It is also possible to take account of reinforcements if one wishes to consider their role in the stability of the wall. The yield design calculation method also enables the presence of seismic conditions to be taken into account if the user so wishes.

To impose a pressure diagram, fill out the following parameters:

- **"Left"** or "**Right"** for a single wall project;
- "Wall 1" or "Wall 2";
  "Left" or "**Right/E.2R"** (wall 1) or "**Left/E.2R"**or "**Right"** (wall 2) for a double wall project;
- For each type of pressure, two management modes are proposed:
  - **Automatic**: in this case the calculation engine automatically generates the limit and at rest soil pressure diagrams as if the "Imposed pressures diagram" action did not exist;
  - **Imposed**: in this case, the calculation engine uses the imposed diagram defined point by point by the user and, as necessary, supplements it with that generated automatically. The following diagrams can be modified:
    - Limit active earth pressure $\mathbf{p_a}$;
    - Pressure at rest $\mathbf{p_0}$;
    - Limit passive earth pressure $\mathbf{p_b}$.

When at least one of the diagrams is in "Imposed" mode, a table appears at the bottom of the action frame and is used to define a diagram point by point:

- **z**: level of diagram point (m, ft);
- $\mathbf{p_a}$: imposed active earth pressure limit value (kN/m/ml, kip/lft)
- $\mathbf{p_0}$: imposed pressure value at rest (kN/m/ml, kip/lft);
- $\mathbf{p_b}$: imposed passive earth pressure limit value (kN/m/ml, kip/lft).

A graphical interpretation is only proposed for the imposed diagrams. The colour code is as follows:

- Black: pressure at rest $\mathbf{p_0}$;
- Blue: limit passive earth pressure $\mathbf{p_b}$;
- Red: limit active earth pressure $\mathbf{p_a}$.

The limit pressure values to be input correspond to the horizontal projection of the possibly angled active/passive earth pressure diagram. As previously mentioned, this diagram can be fitted using the yield design kinematic calculation method implemented in Talren.

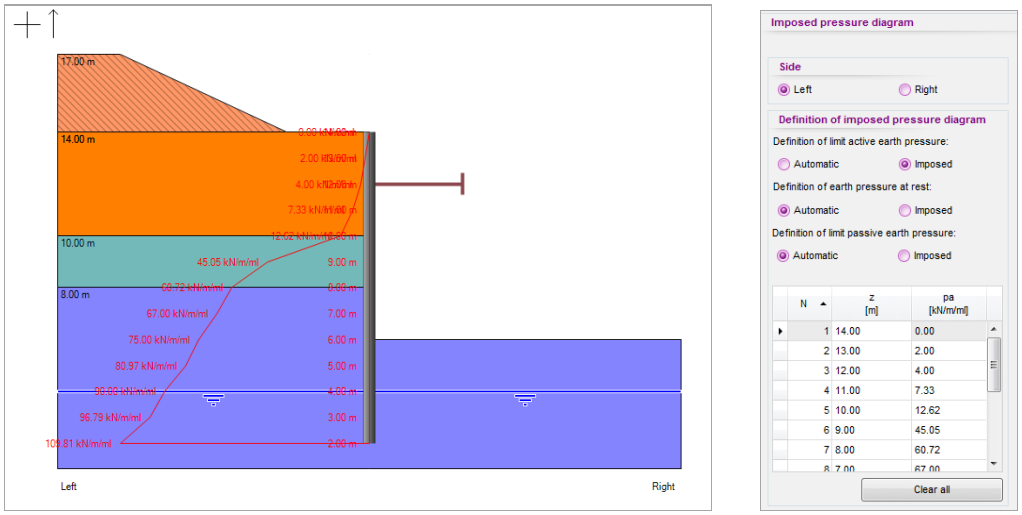


Figure B76 : Definition of an imposed pressure diagram

B.5.3.3. Reduced pressure

The "Reduced pressure" action is only accessible in the initial phase.

This command is used for a composite wall (e.g. soldier pile, wall with shorter middle panel, etc.). It simulates the fact that the soil and water pressures only apply to a proportion of the width of the wall **R** (between 0 and 1) over a height bounded by an upper level $\mathbf{z_t}$ and lower level $\mathbf{z_b}$. The passive earth pressure applied to the same width can be multiplied by an ultimate passive earth pressure ratio C. The scope of the reaction coefficient is reduced accordingly.

The parameters needed for defining this action are:

- $\mathbf{z_t}$: upper level of the reduced pressure application zone (m or ft);
- $\mathbf{z_b}$: lower level of the reduced pressure application zone (m or ft);
- **R**: reduction coefficient applied to the soil pressures (active/passive) and to the reaction coefficient;
- **C**: additional ultimate passive earth pressure ratio applied to the passive earth pressure only.

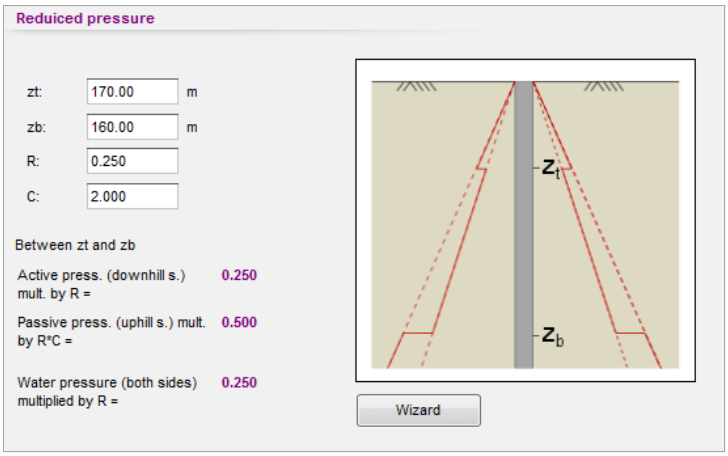


Figure B77 : Reduced pressure action definition frame

In practice, this action affects the soil and water pressures at zone [z_t , z_b] in the following way:

- The limit active earth pressure is multiplied by R
- The limit passive earth pressure is multiplied by R x C
- The water pressure is multiplied by R
- The reaction coefficient is multiplied by R

Additional information and an application example are given in parts C and D of the manual respectively.

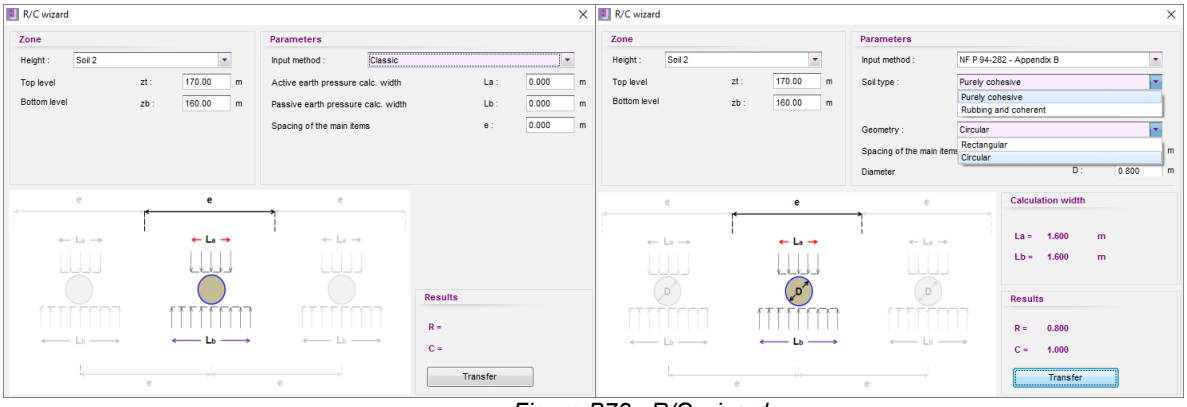


Figure B78 : R/C wizard

B.5.4. Characteristics of the wall

B.5.4.1. Modification of wall stiffness

This action is used to modify the mechanical characteristics of a wall section bounded by its upper and lower levels.

The parameters to be input are as follows:

- **"Wall 1"** or "**Wall 2"** for a double wall project;
- **z1:** upper level of the section concerned by the change (m, ft);
- **z2:** lower level of the section concerned by the change (m, ft);
- **EI**: product of inertia (kNm²/ml, kip.ft²/lft);
- **R_c**: cylindrical stiffness (kN/m²/ml, ksf/lft) if the wall was defined as circular in the project data definition;
- **R:** wall radius (m, ft) if the wall was defined as circular in the project data definition;
- **W**: surface weight of wall (kN/m/ml, ksf/lft).

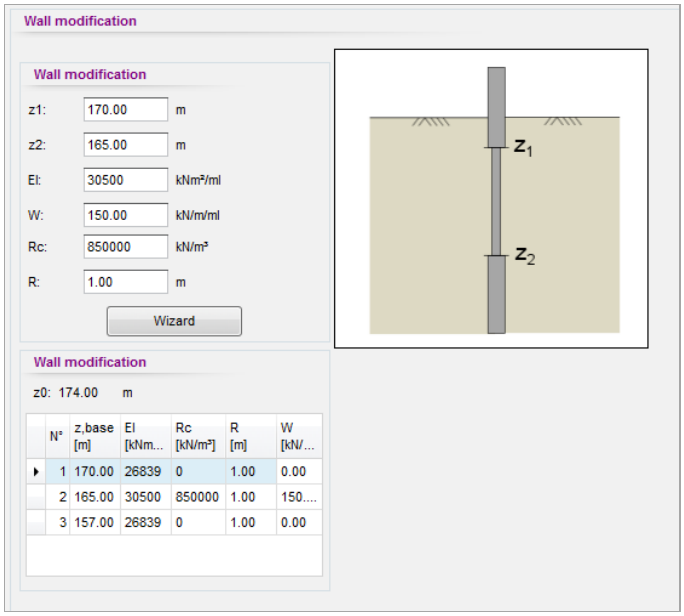


Figure B79 : Wall modification frame

B.5.4.2. Modification of wall structure

This action is used to extend the top of the wall upwards or the base of the wall downwards.

The parameters to be input are:

- **"Wall 1"** or "**Wall 2"** for a double wall project;
- **z0**: new level of top of wall (m, ft);
- **EI**: product of inertia (kNm^2/ml, kip.Ft²/lft);
- **Rc**: cylindrical stiffness (kN/m^3, KcF) if the wall was defined as circular in the project data definition;
- **R:** wall radius (m, ft) if the wall was defined as circular in the project data definition;
- **W**: surface weight of wall (kN/m/ml, ksf/lft).

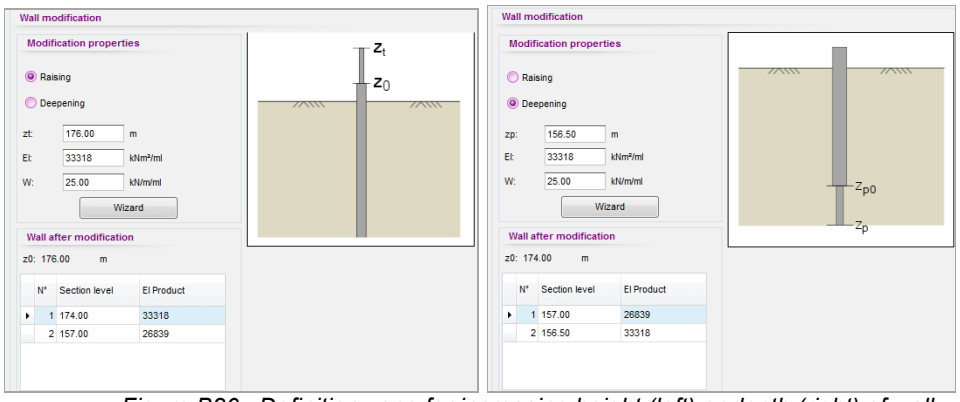


Figure B80 : Definition zone for increasing height (left) or depth (right) of wall

B.5.5. Anchors and supports

The anchors, except for active anchors, work in one of the two modes defined in "Advanced calculation options" (in the "Title and Options" window). These modes are considered as follows:

- Activation as of the installation phase: in this case, the anchor stiffness is **always** taken into account as of the installation phase;
- Activation in two stages if pre-stressing is active: the stiffness is only taken into account as of the phase following installation, **provided that the anchor is initially pre-stressed.** If this is not the case, the anchor works as in the first mode.

B.5.5.1. Tie

This action is used to activate, modify or deactivate a layer of bedded ties. For simplification purposes, we will refer to "layer of ties" as "tie" in the documentation and in the interface. By convention, the force in a tie is positive in traction.

When installing a tie (installation phase), only the pre-stressing is taken into account in the calculation. The stiffness of the tie is activated as of the following phase.

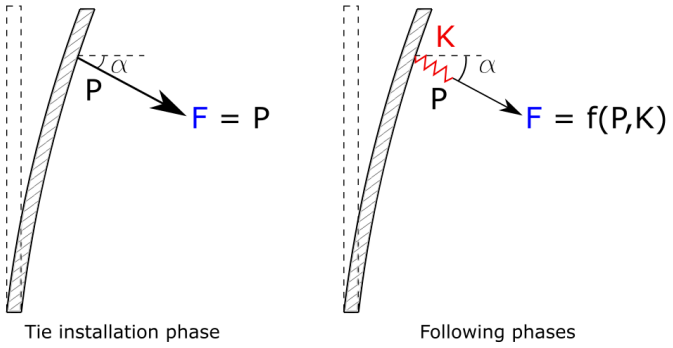


Figure B81 : Behaviour of a tie with pre-stressing

The parameters needed to define a tie are:

- Choice of wall: "**Wall 1"** or "**Wall 2"** for a double wall project;
- Choice of **"Activate":** chosen by default when defining a new tie;
- Choice of side: **"Left"** or "**Right"** for a single wall project, "**Left"** or "**Right/E.2R"** (wall 1) or "**Left/E.2R"** or "**Right"** (wall 2) for a double wall project;
- z_a: level of application (m or ft);
- **K**: axial stiffness (kN/m/ml or kip/lft);
- **P**: pre-stressing counted in positive traction (kN/ml or kip/lft);
- α: angle of tie with respect to the horizontal, counted as positive if the angle is downwards (°);
- L_u: useful length of tie (in m or ft) usually taken as equal to the distance, along the axis of the tie, between the anchor head and the point corresponding to the middle of the bedding (also see the Kranz check in part C of the manual for validation of the value of L_u to be considered and in part D for examples of its definition). This parameter is only required if the ULS checks are activated;
- **L_s:** length of the grouted embedment (in m or ft). It is to be filled out if the "consideration of the grouted embedment length" option was chosen in the "Kranz Options" grouped in the "Advanced calculation options" (see Part D of the manual for examples of definition of L_s);

If the "Elastoplastic behaviour" box is ticked, a "plastification" plateau can be defined for the traction force:

- **$F_{adm,tr}$**: allowable traction force (kN or kip)

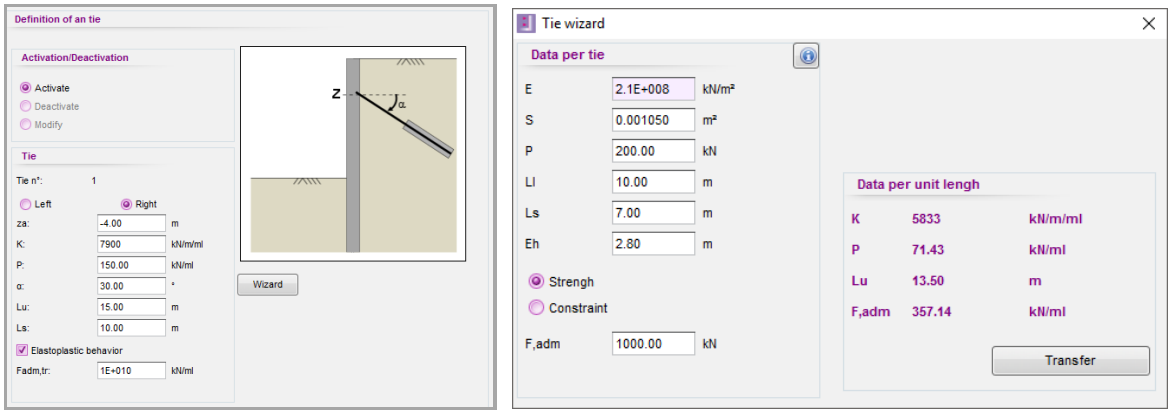


Figure B82 : Tie definition and corresponding wizard

In K-Réa, the characteristics of a layer of ties are given per unit length of wall.

The Wizard >> button gives access to the **Tie Wizard** (right-hand part of the above figure). It is used to calculate the stiffness of the tie and the pre-stressing force per unit length, on the basis of the formulas given in the technical manual (Part C of the manual).

The data to be input into the tie wizard are as follows:

- eh: spacing of ties (distance between axis) (m);
- E: Young's modulus (kN/m²);
- S: calculation cross-section (m²);
- P: pre-stressing (kN);
- LI: free length (m);
- Ls: grouted length.

After entering the input data, the values of K, P and Lu (for the ULS calculations only) are calculated per unit length. Click the Transfer button to transfer the values to the project.

A help diagram is available by clicking the button:

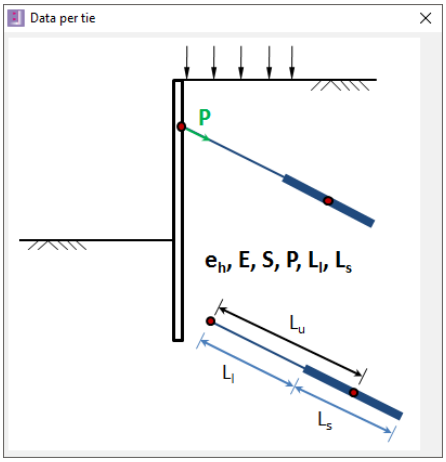


Figure B83 : Tie wizard help - schematic

This "Data per tie" figure clarifies the meaning of the wizard input parameters.

To **modify a tie** that was previously defined, select "**Modify**". The modifiable values are as follows:

- **K**: axial stiffness per unit length (kN/m/ml or kip/lft);
- **P**: pre-stressing (kN/ml or kip/lft);
- $\mathbf{F_{adm,tr}}$: allowable traction force (kN or kip).

The following figure illustrates the appearance of a drop-down list containing the ties active in the phase in question. Each tie is identified by:

- its identification number;
- its application level;
- its stiffness;
- its pre-stressing;
- its inclination angle.

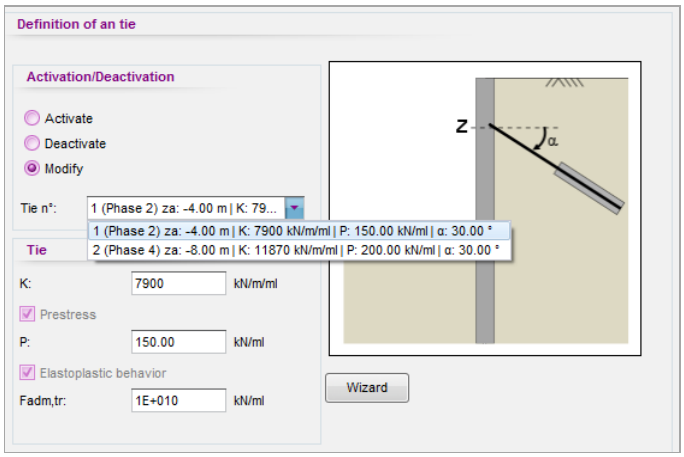


Figure B84 : Modification of a tie

To **deactivate a tie** previously defined, select **"Deactivate".** The following figure shows the drop-down list available containing all the ties that the user can deactivate. Each tie is identifiable by the following:

- its declaration number;
- its application level;
- its stiffness;
- its pre-stressing;
- its inclination angle.

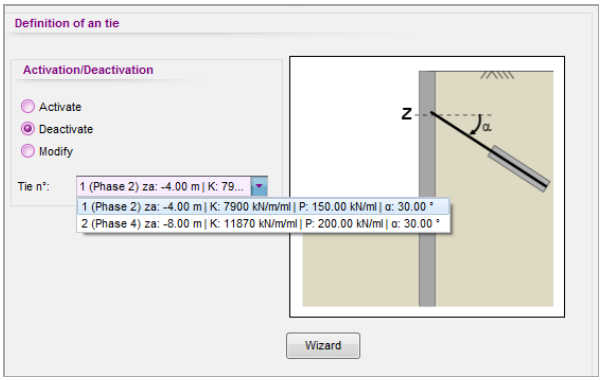


Figure B85 : Deactivation of a tie

B.5.5.2. Strut

This action is used to activate, modify or deactivate a layer of struts. For simplification purposes, we will refer to "layer of struts" as "strut" in the documentation and in the interface. By convention, the force in a strut is positive in compression.

The parameters to be input to define a strut are:

- **"Wall 1"** or "**Wall 2"** for a double wall project;
- **Activate:** chosen by default when defining a new strut;
- **"Left"** or "**Right"** for a single wall project, "**Left"** or "**Right/E.2R"** (wall 1) or "**Left/E.2R"** or "**Right"** (wall 2) for a double wall project;
- z_a: application level (m or ft);
- **K**: axial stiffness (kN/m/ml or kip/lft);
- **P**: pre-stressing counted positively in compression (kN/ml or kip/lft);

- α: inclination angle of strut with respect to the horizontal considered to be positive if the angle is downwards;
- **Operate under traction:** if ticked, the strut can operate under traction;
- **Operate under compression:** if ticked, the strut can operate under compression;

If the "**elastoplastic behaviour**" box is ticked, it is possible to define:

- $\mathbf{F_{adm,tr}}$: allowable traction force (kN/ml or kip/lft), only accessible if the "operate under traction" option is ticked;
- $\mathbf{F_{adm,cp}}$: allowable compression force (kN/ml or kip/lft), only accessible if the "operate under compression" option is ticked.

It should be recalled that the calculation is made for a unit length of wall.

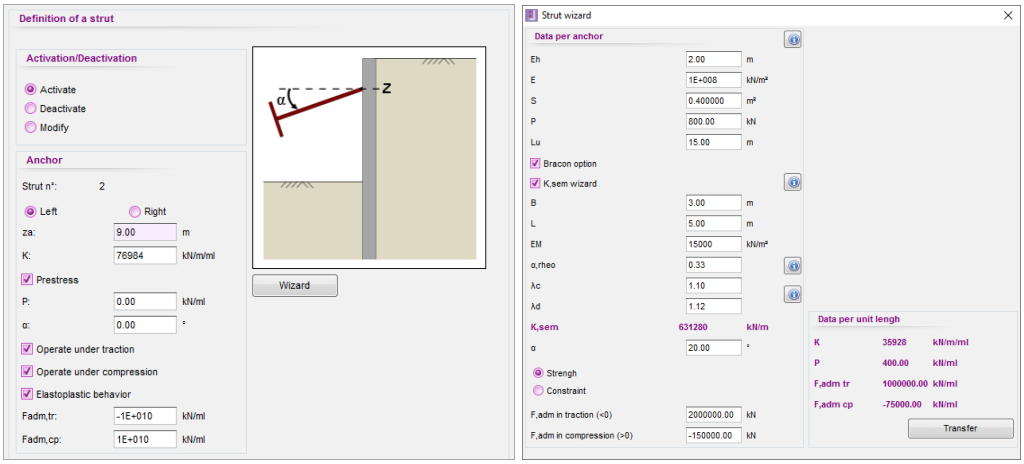


Figure B86 : Definition of a strut and corresponding wizard

The Wizard >> button gives access to the **Strut wizard** (see above figure). This is used to calculate the stiffness of the strut and its pre-stressing, if any, per unit length, using the formulas given in the technical manual (Part C of the manual). It also includes a tool for evaluating the equivalent stiffness of a diagonal strut.

The input data to be entered into the strut wizard are as follows:

- eh: spacing of struts (m);
- E: Young's modulus (kN/m²);
- S: cross-section (m²);
- P: pre-stressing (kN);
- L_u: useful length (m);
- F_{adm} (>0): allowable ultimate compression force;
- $\sigma_{,adm}$ (>0): allowable ultimate compression stress.

Ticking the "diagonal strut" option gives access to the display of the following parameters:

- $K_{,sem}$: vertical stiffness of support base (kN/m);
- α: strut inclination angle with respect to the horizontal (°);

Ticking the "Wizard K,sem" option gives access to the following parameter:

- B: width of footing B (m);
- L: length of footing L (m), (L ≥ B);
- E_m: pressuremeter modulus (kN/m²);
- $\alpha_{,rheo}$: rheological factor;
- λ_c and λ_δ: coefficients corresponding to the volume and deviatoric part of settlement.

Once the values have been input, K-Réa displays the values selected for K, P and $F_{,adm\ cp}$ or $\sigma_{,adm\ cp}$, in the right-hand part of the window.

Four help diagrams can be obtained by clicking the buttons:

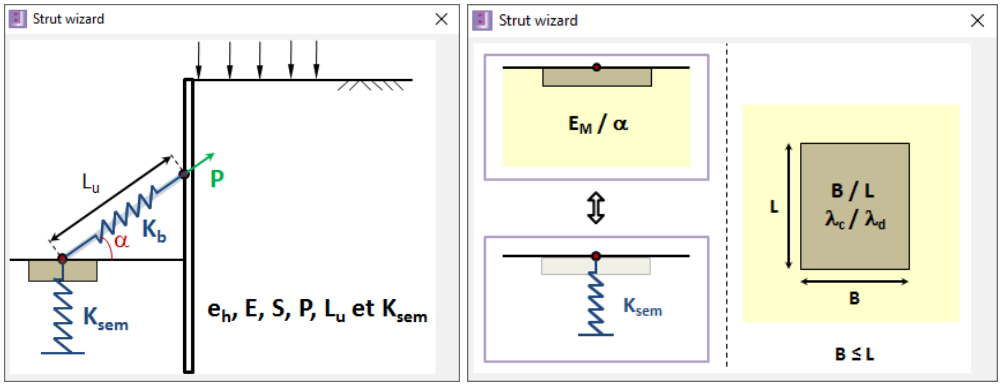


Figure B87 : Strut wizard help - Schematics

Strut wizard

Rheological factor

TYPE OF SOIL	Peat	Clay		Silt		Sand		Sand and gravel		Type	Rock
	α	E_M/p_l	α	E_M/p_l	α	E_M/p_l	α	E_M/p_l	α		α
Overconsolidated or very tight	1	> 16	1	> 14	2/3	> 12	1/2	> 10	1/3	Very little fractured	2/3
Normally consolidated		9 à 16	2/3	8 à 14	1/2	7 à 12	1/3	6 à 10	1/4	Normal	1/2
Under consolidated altered and disturbed or loose		7 à 9	1/2	5 à 8	1/2	5 à 7	1/3	--		Very fractured – Very altered	1/3 - 2/3

Figure B88 : Strut wizard help – Rheological factor

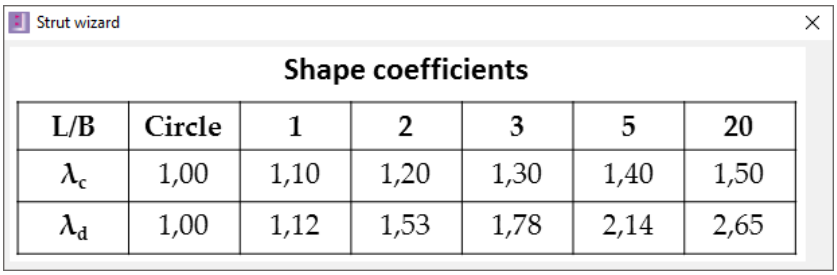


Figure B89 : Strut wizard help – Form factors

To **modify a strut** previously defined, select "**Modify**". The modifiable values are:

- **K**: stiffness per unit length (kN/m/ml or kip/lft);
- **P:** pre-stressing (kN/ml or kip/lf);
- $\mathbf{F_{adm,tr}}$: allowable traction force (kN/ml or kip/lft), visible if the "Operate under traction" option was ticked when the strut was defined;
- $\mathbf{F_{adm,cp}}$: allowable compression force (kN/ml or kip/lft), visible if the "Operate under compression" option was ticked when the strut was defined.

The following figure shows the appearance of a drop-down list with all the existing and modifiable struts, each identified by its characteristics:

- its declaration number;
- its application level;
- its stiffness;
- its pre-stressing;
- its inclination angle.

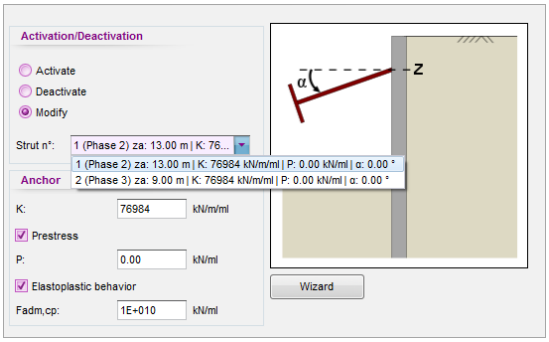


Figure B90 : Modification of a strut

To **deactivate a strut** previously defined, select **"Deactivate**". The strut deactivation frame illustrated in the following figure then appears. From the drop-down list select the strut to be modified, identified by the following:

- its declaration number;
- its application level;
- its stiffness;
- its pre-stressing;
- its inclination angle.

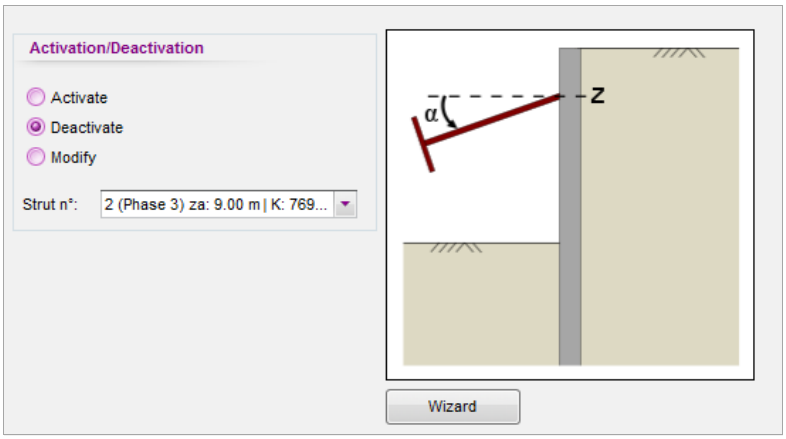


Figure B91 : Deactivation of a strut

B.5.5.3. Rotational spring

Rotation spring corresponds to an anchor which opposes the rotation of the wall. This command is used to activate, modify or deactivate this type of anchor. By convention, the moment induced in a rotational spring is positive clockwise.

The parameters to be defined are as follows:

- **"Wall 1"** or "**Wall 2"** for a double wall project;
- **Activate:** choice by default to define a new rotational spring;
- z_a: application level (m or ft);
- R_r: rotational stiffness (kNm/rad/ml or kip.ft/rad/lft);
- M_{init}**:** initial positive clockwise moment (the reaction induced on the wall is positive in the counter-clockwise direction) (kNm/ml or kip.ft/lft).

If the "**elastoplastic behaviour**" box is ticked, it is possible to define:

- $M_{adm,ho}$**:** allowable clockwise moment (kNm/ml or kip.ft/lft), visible only if the "Clockwise work" option is ticked;
- $M_{adm,ah}$**:** allowable counter-clockwise moment (kNm/ml or kip.ft/lft), only visible if the "Counter clockwise work" option is ticked.

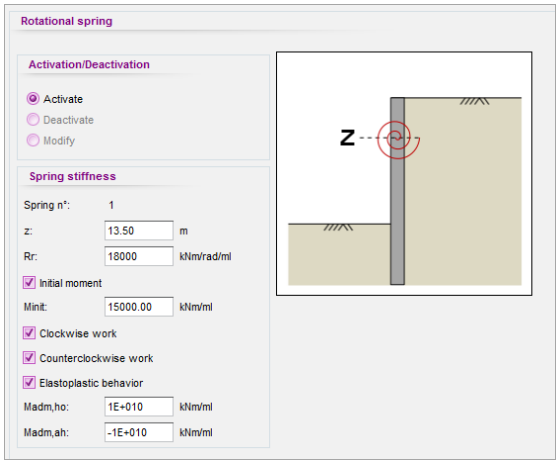


Figure B92 : Definition of a rotational spring

To **modify a rotational spring** previously defined, select "**Modify**". The modifiable values are:

- **R_r**: rotational stiffness (kNm/rad/ml or kip.ft/rad/lft);
- **M_{init}:** initial moment (kNm/ml or kip.ft/lft);
- **$M_{adm,ho}$:** allowable clockwise moment (kNm/ml or kip.ft/lft), only accessible if the "Clockwise work" option was ticked when defining the rotational spring;
- **$M_{adm,ah}$:** allowable counter-clockwise moment (kNm/ml or kip.ft/lft), only accessible if the "Counter-clockwise work" option was ticked when defining the rotational spring.

The following figure shows the drop-down list consisting of the rotational springs previously defined and which are modifiable. Each of them is identified by the following:

- its declaration number;
- its application phase;
- its stiffness;
- its initial moment.

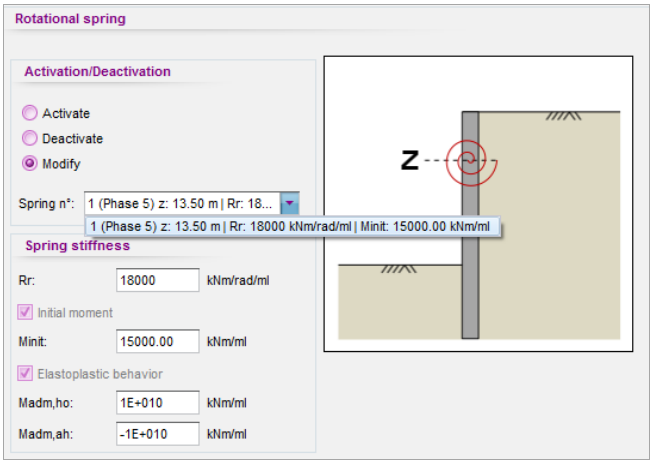


Figure B93 : Modification of a rotational spring

To **deactivate a rotational spring** previously defined, select **"Deactivate"**. The deactivation frame illustrated in the following figure then appears. From the drop-down list select the rotational spring to be deactivated, identified by the following:

- its declaration number;
- its application level;
- its stiffness;
- its initial moment.

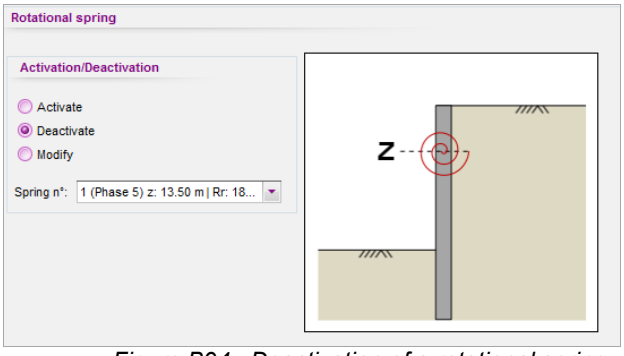


Figure B94 : Deactivation of a rotational spring

B.5.5.4. Circular waling

This action is used to activate, modify or deactivate a circular waling. By convention, the force in a waling is positive in compression.

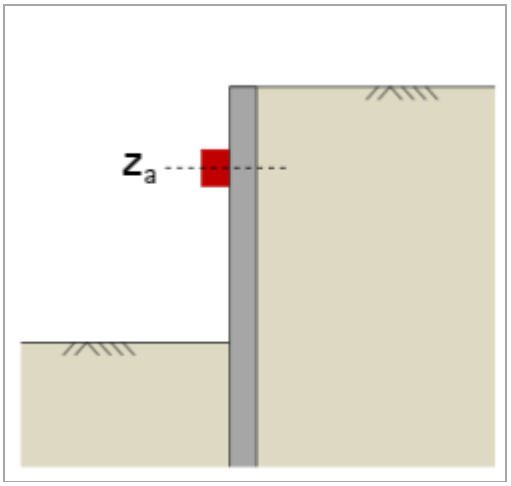


Figure B95 : Definition of a circular waling

The parameters to be input to define a circular waling are :

- **"Wall 1"** or "**Wall 2"** for a double wall project;
- **Activate:** chosen by default to define a new waling;
- Waling installation side:
  - **"Left"** or "**Right"** for a single wall project
  - **"Left"** or "**Right/E.2R"** (wall 1) or "**Left/E.2R"** or "**Right"** (wall 2) for a double wall project;
- z_a: application level (m or t);
- **R**: radius (m or ft);
- **ES**: orthoradial stiffness (kN or kip) used to calculate the cylindrical stiffness R_c of the waling (kN/m^2 or ksf). If the waling consists of a steel section, the ES value is simply equal to the product of the cross-section of the steel section S by the Young's modulus of the steel;
- **P**: orthoradial pre-stressing positive in compression (kN or kip).

If the **"elastoplastic behaviour"** box is ticked, it is possible to define:

- **$F_{adm,tr}$:** allowable traction force (kN or kip), only accessible if the "operate under traction" option is ticked.
- **$F_{adm,cp}$:** allowable compression force (kN or kip), only accessible if the "operate under compression" option is ticked.

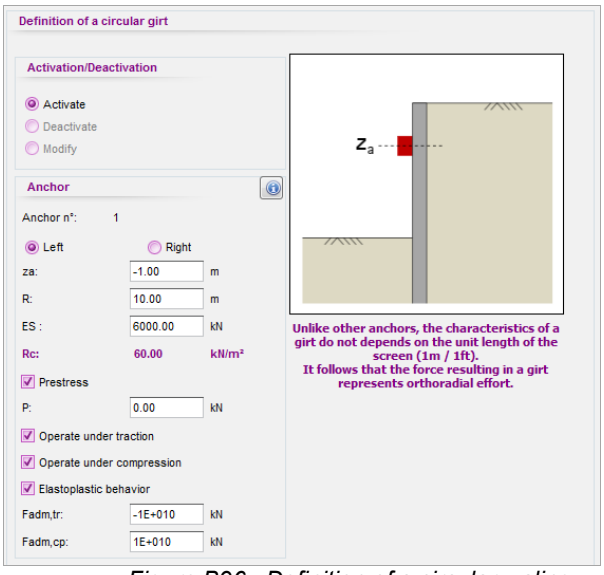


Figure B96 : Definition of a circular waling

Unlike the other anchors and supports, the characteristics of a waling are unrelated to the unit length of the wall. The orthoradial force in a circular waling is expressed in kN or in kip.

A help diagram is available by clicking the button:

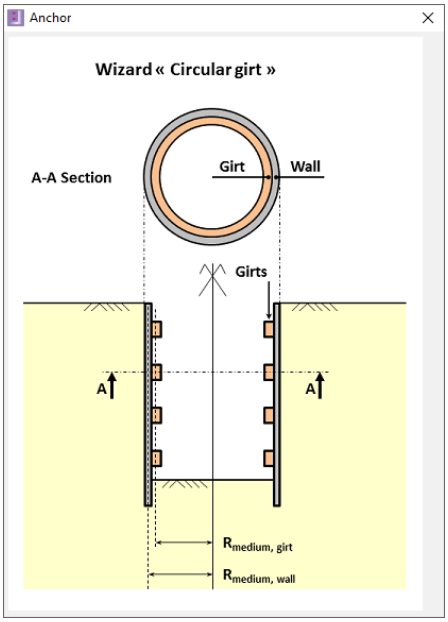


Figure B97 : Help for definition of a waling

To **modify a waling** previously defined, select "**Modify"**. The modifiable data are:

- **Rc**: cylindrical stiffness (kN/m² or ksf);
- **P**: pre-stressing (kN or kip);
- **$F_{adm,tr}$:** allowable traction force (kN or kip), only visible if the "operate under traction" option was ticked when the waling was defined;
- **$F_{adm,cp}$:** allowable compression force (kN or kip), only visible if the "operate under compression" option was ticked when the waling was defined.

The following figure shows the list of existing walings that can be modified, each waling being identified by:

- its declaration number;
- its application level;
- its radius;
- its orthoradial stiffness;
- its pre-stressing.

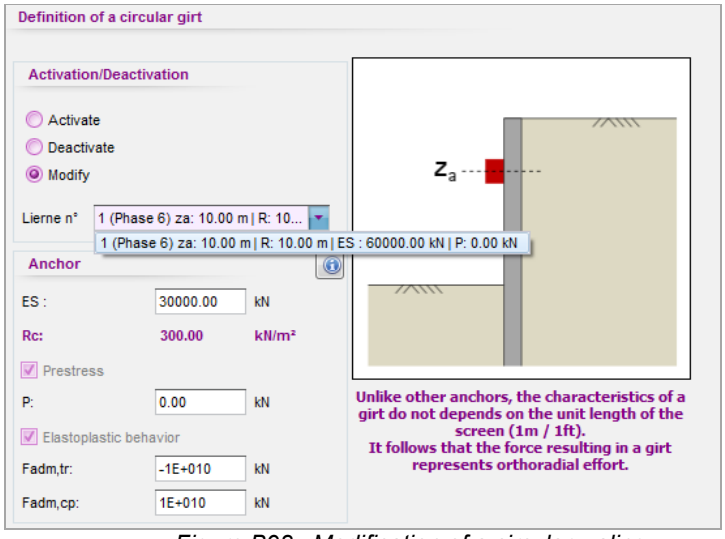


Figure B98 : Modification of a circular waling

To **deactivate a waling** previously defined, select "**Deactivate"**. The deactivation frame illustrated in the following figure then appears. From the drop-down list, select the waling to be deactivated, identified by the following:

- its declaration number;
- its application level;
- its radius;
- its circular stiffness;
- its pre-stressing.

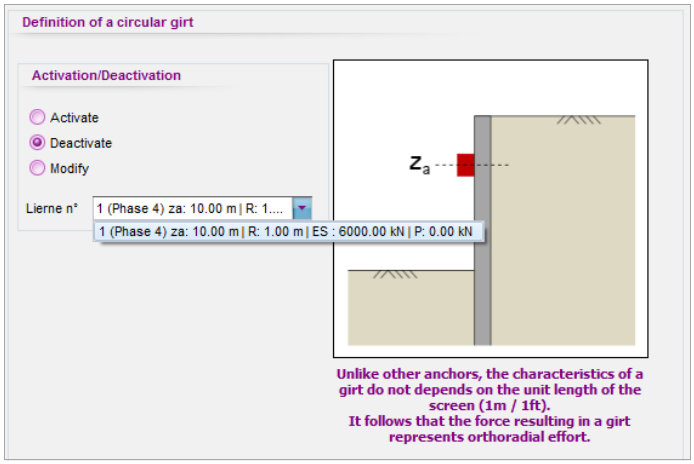


Figure B99 : Deactivation of a circular waling

B.5.5.5. Surface strut

This action can be used to activate, modify or deactivate a surface strut. By convention, the force in such a strut is positive in compression.

The parameters necessary for defining a surface strut are:

- **"Wall 1"** or "**Wall 2"** for a double wall project;
- **Activate:** chosen by default to define a new surface strut;
- Surface strut installation side:
  - **"Left"** or "**Right"** for a single wall project
  - **"Left"** or "**Right/E.2R"** (wall 1) or "**Left/E.2R"** or "**Right"** (wall 2) for a double wall project;
- z_{sup}: upper level of the surface strut (m or ft);
- z_{inf}: lower level of the surface strut (m or ft);
- k_s: surface stiffness (kN/m²/ml or ksf/lft);
- p_s: surface pre-stressing (kN/m/ml or kip/lft);
- **Operate under traction**: this box is ticked by default. The surface strut can thus work in traction;
- **Operate under compression**: this box is ticked by default. The surface strut can thus work in compression.

It should be remembered that the calculation is performed for a wall unit length.

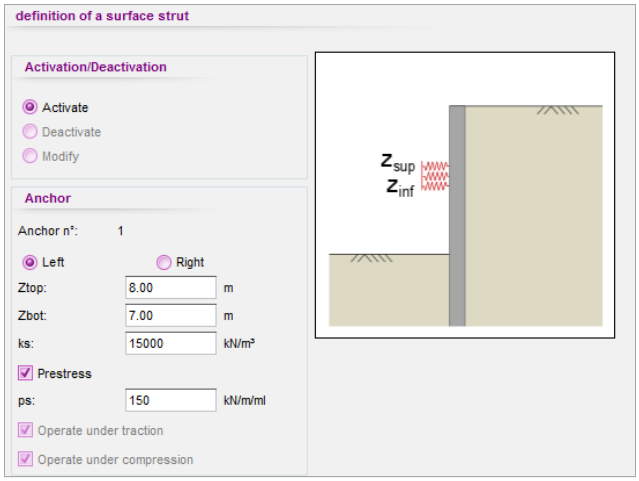


Figure B100 : Definition of a surface strut

To **modify a surface strut** previously defined, select "**Modify"**. The modifiable values are:

- **ks**: surface stiffness (kN/m^3 or ksf);
- **ps:** surface pre-stressing (kN/m/ml or kip/lft).

The following figure shows the drop-down list containing the surface anchors previously defined and which are modifiable, each of which is identified by:

- its declaration number;
- its upper and lower levels;
- its surface stiffness;
- its surface pre-stressing.

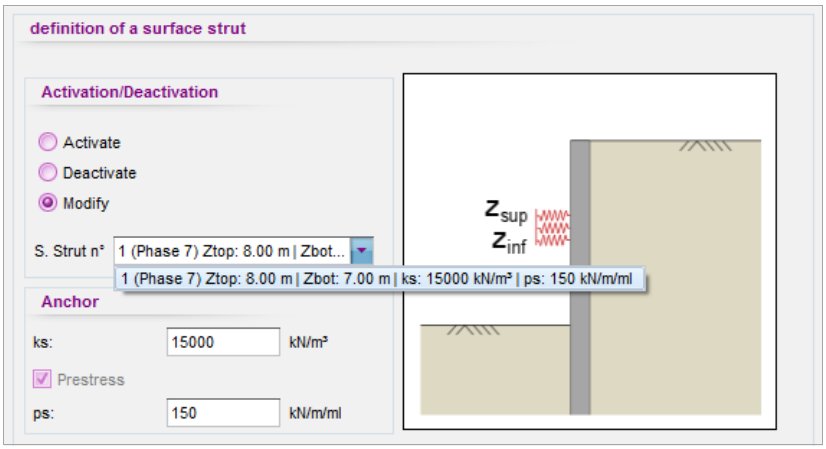


Figure B101 : Modification of a surface strut

To **deactivate a surface strut** previously defined, select "**Deactivate"**. The following figure shows the drop-down list consisting of surface struts previously defined and which are modifiable, each of which is identified by:

- its declaration number;
- its upper and lower levels;
- its surface stiffness;
- its surface pre-stressing.

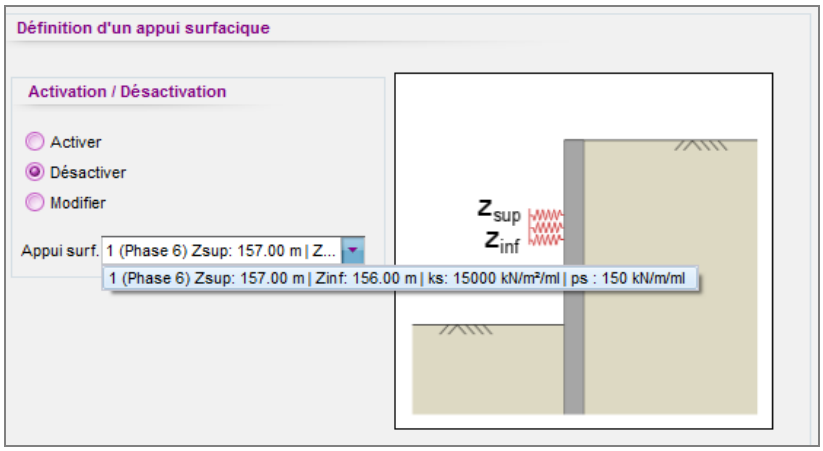


Figure B102 : Deactivation of a surface strut

B.5.5.6. Linking anchor

This action is only available for a double wall project and can be used to activate, modify or deactivate a linking anchor joining the two walls.

The parameters to be input to define a linking anchor are:

- **Activate**: chosen by default to define a new strut;
- **Tie** or **Strut:** choice of type of anchor. By convention, the force in a tie is positive in traction, that in a strut is positive in compression;
- z_{aa}: anchor level on wall 1 (m or ft);
- z_{ab}**:** anchor level on wall 2 (m or ft);
- **d**: distance between the two walls. This value is recovered directly from the "Title and Options" tab, where it was filled out as a project parameter. It is recalled for information (m or ft);

- α: this value is not to be defined by the user and is automatically calculated as a function of the distance d and anchor levels z_{a1} and z_{a2};
- **K**: axial stiffness per unit length (kN/m/ml or kip/lft);
- **P**: pre-stressing per unit length (kN/ml or kip/lft).

If the **"elastoplastic behaviour"** box is ticked, the following can be defined:

- **$F_{adm,tr}$:** allowable traction force (kN/ml or kip/lft), only accessible if the "Operate under traction" option is ticked;

It should be recalled that the calculation is made for a unit length of wall.

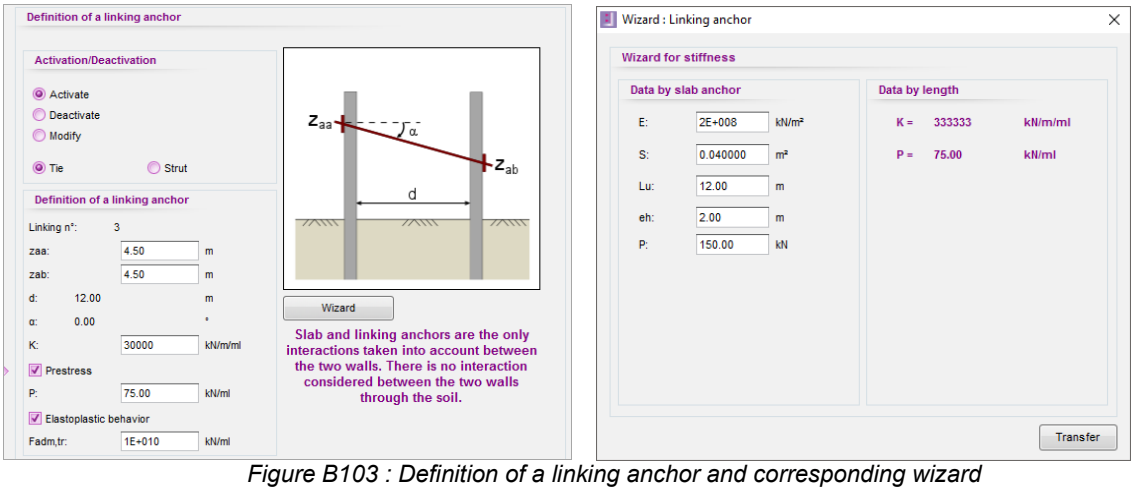


Figure B103 : Definition of a linking anchor and corresponding wizard

The Wizard >> button gives access to the **Linking anchor wizard**. This latter can be used to calculate the stiffness of the connecting anchor and its pre-stressing, if any, per unit length, using the formulas indicated in the technical manual (see Part C of the manual).

To **modify a linking anchor** previously defined, select **"Modify"**. The modifiable values are:

- **K**: stiffness (kN/m/ml or kip/lft);
- **P**: pre-stressing (kN/ml or kip/lft);
- **$F_{adm,tr}$:** allowable traction force (kN/ml or kip/lft), accessible if the "Operate under traction" option was ticked when the linking anchor was defined.

The following figure shows the drop-down list consisting of the linking anchors previously defined and which are modifiable. Each linking anchor is identified by the following:

- its declaration number and its installation phase;
- its anchor levels (z_{a1} and z_{a2});
- its stiffness.

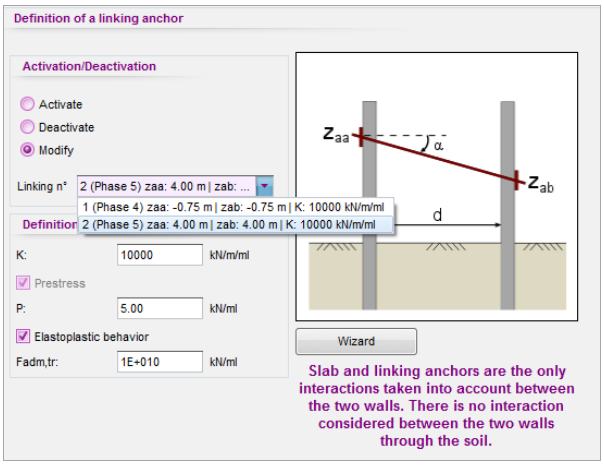


Figure B104 : Modification of a linking anchor

To **deactivate a linking anchor** previously defined, select "**Deactivate"**. The following figure shows the drop-down list containing the linking anchors previously defined. Each linking anchor is identified by the following:

- its declaration number;
- its anchor levels (z_{a1} and z_{a2});
- its stiffness.

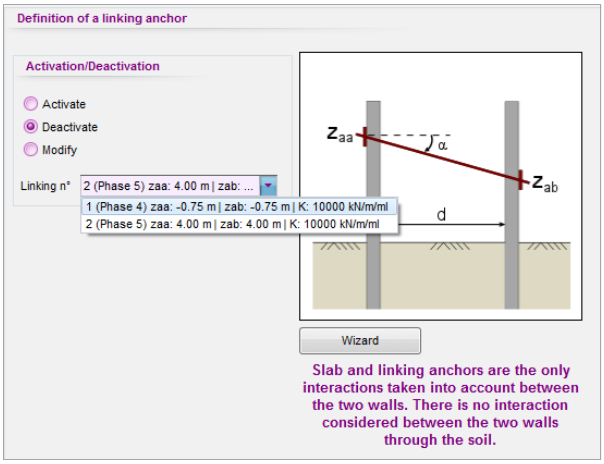


Figure B105 : Deactivation of a linking anchor

B.5.5.7. Slab anchor

This action is only available for a double wall project and is used to activate, modify or deactivate a slab anchor between the two walls.

The parameters to be input to define a slab anchor are as follows:

- **Activate:** chosen by default to define a new strut;
- $\mathbf{z_{base}}$: level of base of slab anchor (m or ft);
- **H:** height (thickness) of slab anchor (m or ft);
- **d**: distance between the two walls. This value is recovered directly from the "Title and Options" tab, where it was input as a project parameter. It is recalled for information (m or ft);
- $\mathbf{k_s}$: surface stiffness (kN/m²/ml or ksf/lft);
- $\mathbf{p_s}$: surface pre-stressing (kN/m/ml or kip/ft/lft);

- **Operate under traction**: this box is ticked and modification is blocked. The slab anchor can thus work in traction;
- **Operate under compression**: this box is ticked and modification is blocked. The slab anchor can thus work in compression.

It should be recalled that the calculation is performed for a unit length of wall.

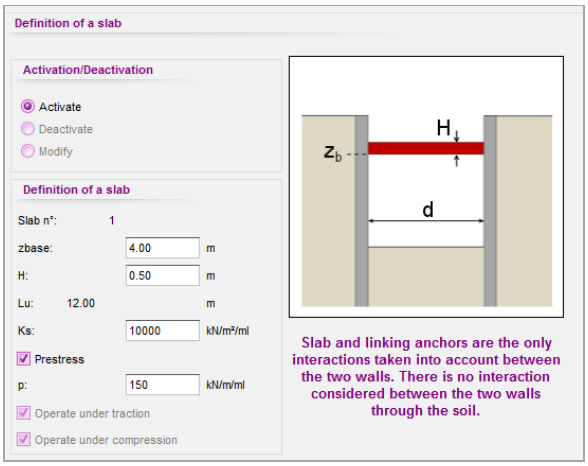


Figure B106 : Definition of a slab anchor

To **modify a slab anchor** previously defined, select "**Modify"**. The modifiable values are:

- **k_s**: surface stiffness (kN/m²/ml or ksf/lft);
- **p_s:** surface pre-stressing (kN/m/ml or kip/lft).

The following figure shows the drop-down list of slab anchors previously defined and which are modifiable. Each slab anchor is identified by the following:

- its declaration number and installation phase;
- the levels of the base of its ends;
- its surface pre-stressing.
- its surface stiffness.

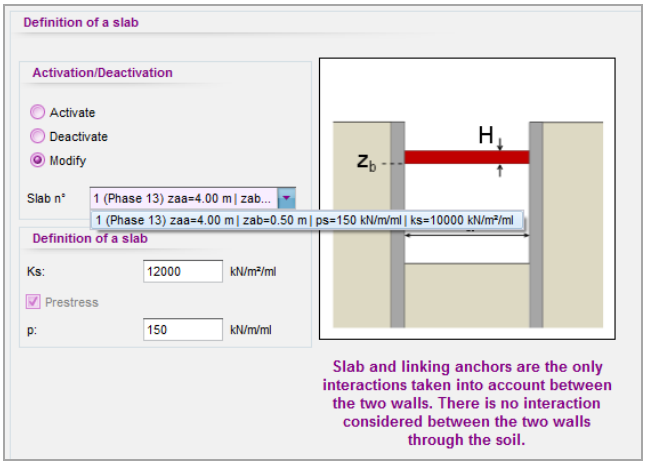


Figure B107 : Modification of a slab anchor

To deactivate a **slab anchor** previously defined, select "**Deactivate"**. The following figure shows the drop-down list of slab anchors previously defined. Each slab anchor is identified by the following:

- its declaration number and installation phase;
- the levels of the base of its ends;
- its surface pre-stressing;
- its surface stiffness.

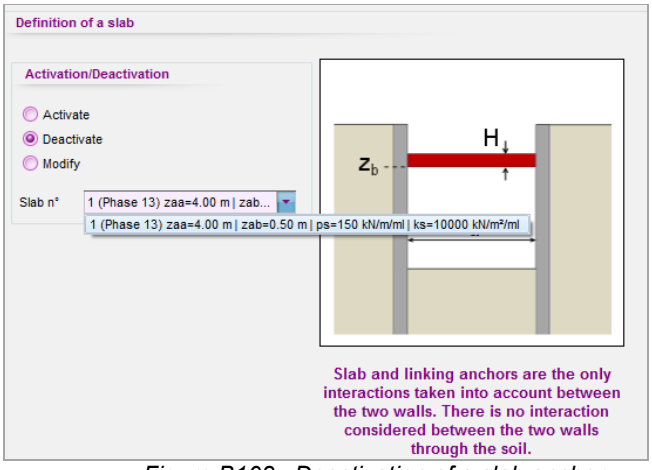


Figure B108 : Deactivation of a slab anchor

B.5.6. Soil and wall loading

B.5.6.1. Caquot overload

This action is used to apply a semi-infinite vertical overload to the soil situated on one side of the wall (details of how this type of overload is taken into account in the calculations are given in part C of the manual).

The parameters to be input to define a Caquot overload are:

- **"Wall 1"** or "**Wall 2"** for a double wall project;
- **Activate:** chosen by default to define a new overload;
- Overload application side:
  - **"Left"** or "**Right"** for a single wall project
  - **"Left"** or "**Right/E.2R"** (wall 1) or "**Left/E.2R"** or "**Right"** (wall 2) for a double wall project;
- **z**: overload application level (m or ft);
- **q**: overload amplitude (kN/m/ml or kip/lft).
- **Action nature**: if ULS checks were requested, the nature of the overload must also be defined, by specifying whether it is **permanent** or **variable**. This choice determines the value of the partial factor to be applied to the overload (see § B.3.1.2).
- **Family**: if the load cases calculation was activated (see §B.3.5), the overload must be allocated to a loads family using the drop-down list containing all the previously defined families.

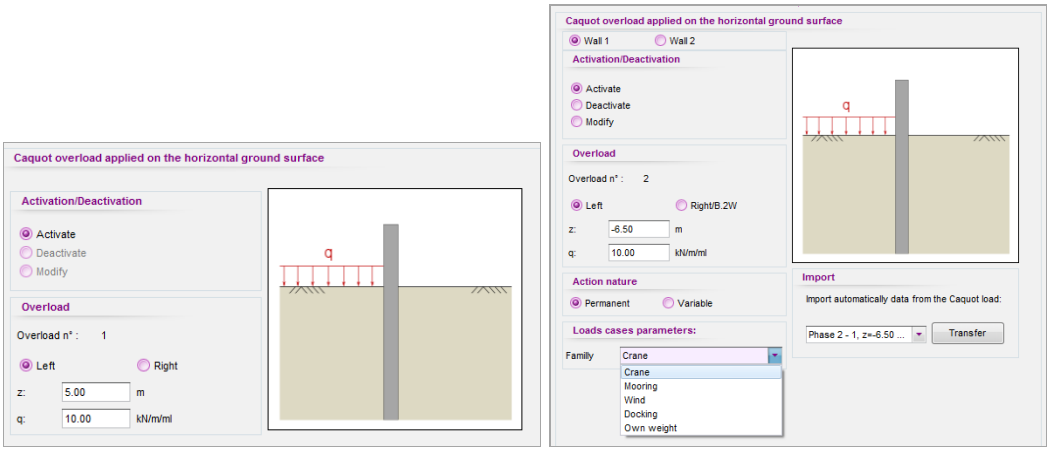


Figure B109 : Definition of Caquot overload

It is also possible to automatically import the properties of a previously defined Caquot overload into the project: select the overload to be imported from the drop-down list and then click the Transfer button.

The "Excavation" and "Fill" actions cancel out the Caquot overloads previously defined on the same side of the wall.

To **modify a Caquot overload** previously defined, select "**Modify**". The modifiable values are:

- **q**: overload amplitude (kN/m/ml or kip/lft);
- **Action nature**: if ULS checks are requested.

The following figure shows the drop-down list containing the previously defined Caquot overloads which are still present. Each Caquot overload is identified by the following:

- its definition phase and its declaration number;
- its application level;
- its amplitude.

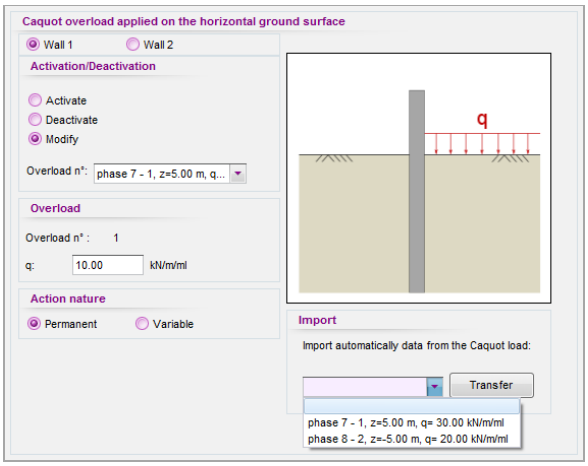


Figure B110 : Modification of a Caquot overload

To deactivate a previously defined **Caquot overload**, select "**Deactivate"**. The following figure shows the drop-down list containing the previously defined Caquot overloads which are still present. Each Caquot overload is identified by the following:

- its definition phase and its declaration number;
- its application level;
- its amplitude.

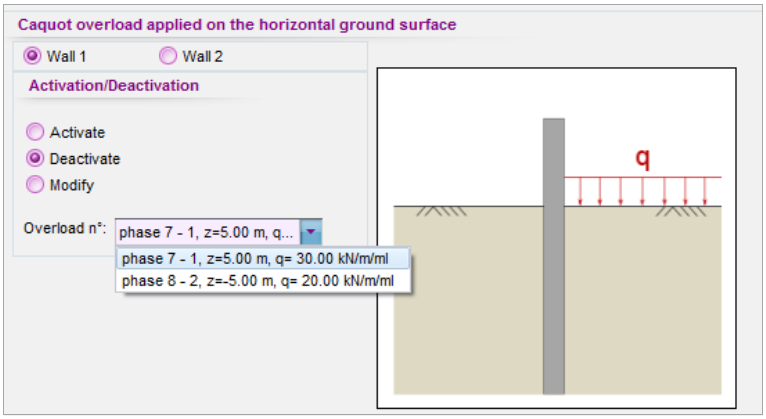


Figure B111 : Deactivation of a Caquot overload

B.5.6.2. Boussinesq overload

This action is used to apply a localised vertical Boussinesq type overload to a limited width in the soil on one side of the wall (part C of the manual details how Boussinesq overloads are taken into account in the calculations).

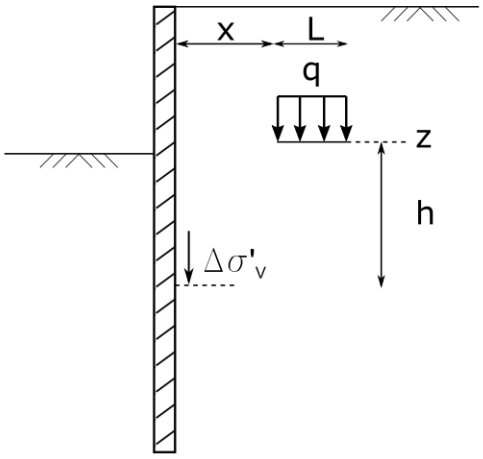


Figure B112 : Schematic of a Boussinesq overload

The parameters to be input to define a Boussinesq overload are:

- **"Wall 1"** or "**Wall 2"** for a double wall project;
- **Activate:** chosen by default to define a new overload;
- Side of application of the Boussinesq overload:
  - **"Left"** or "**Right"** for a single wall project,
  - **"Left"** or "**Right/E.2R"** (wall 1) or "**Left/E.2R"** or "**Right"** (wall 2) for a double wall project;
- **z**: level of application of the overload (m or ft);
- **x**: distance from the wall > 0 (m or ft);
- **L**: width of application > 0 (m or ft);
- **q**: overload amplitude (kN/m/ml or kip/lft);
- α_e: multiplication coefficient used to correct the Boussinesq overload (taking account of the wall effect) The αe = (x+2)/(x+1) button gives this coefficient the value derived from the following formula:

  $\alpha_e = (x + 2) / (x + 1)$
- **Action nature**: if ULS checks were requested, the nature of the overload must also be defined by specifying whether it is **permanent** or **variable**. This choice determines the value of the partial factor to be applied to the overload (see § B.3.1.2).
- **Family**: if the load cases calculation has been activated (see §B.3.5), the overload must be allocated to a loads family using the drop-down list containing all the previously defined families.

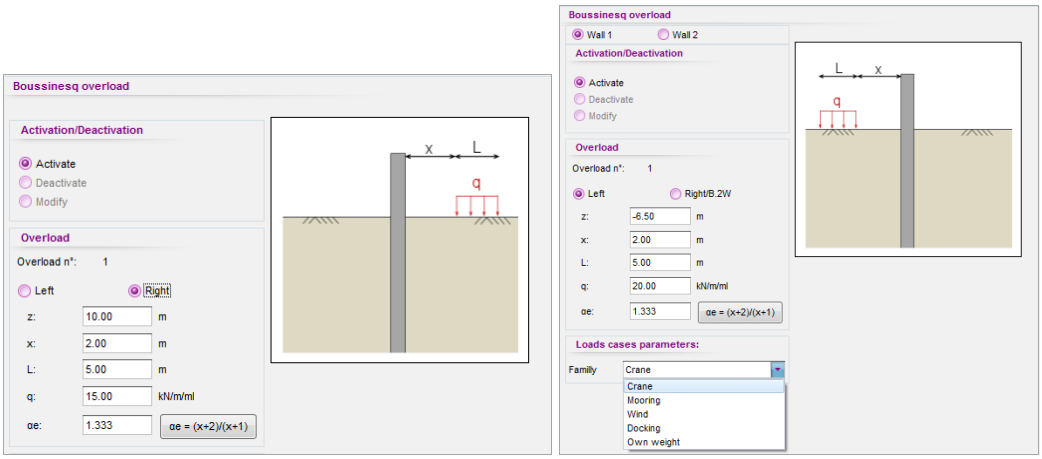


Figure B113 : Definition of Boussinesq overload

The "Excavation" and "Fill" actions cancel the previously defined Boussinesq overloads on the same side of the wall.

To **modify a Boussinesq overload** previously defined, select "**Modify"**. The modifiable values are:

- **q**: overload amplitude (kN/m^2 or KsF);
- **Action nature**: if ULS checks were requested.

The following figure shows the drop-down list containing the previously defined Boussinesq overloads which are still present. Each Boussinesq overload is identified by the following:

- its declaration number and its definition phase;
- its application level;
- its amplitude.

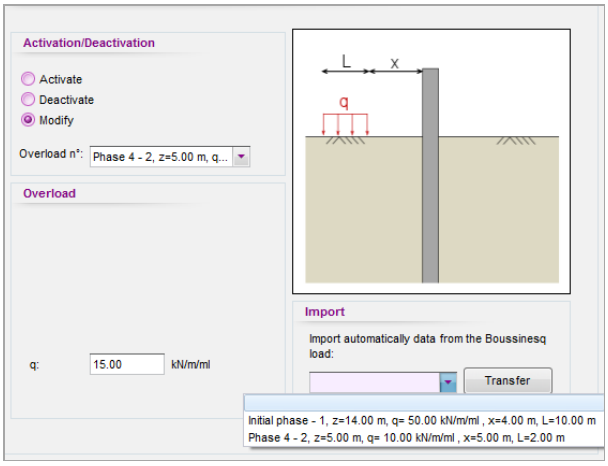


Figure B114 : Modification of a Boussinesq overload

To deactivate a previously defined **Boussinesq overload,** select "**Deactivate"**. The following figure shows the drop-down list containing the previously defined Boussinesq overloads which are still present. Each Boussinesq overload is identified by the following:

- its declaration number and its definition phase;
- its application level;
- its amplitude.

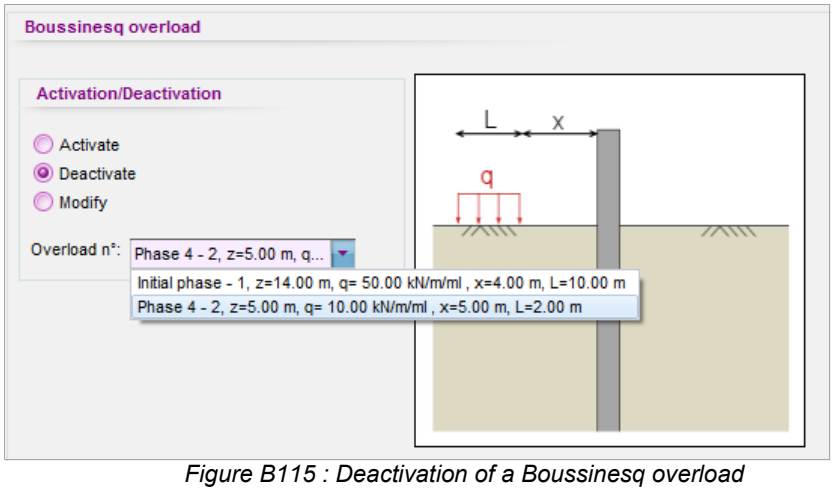


Figure B115 : Deactivation of a Boussinesq overload

B.5.6.3. Graux overload

This action is used to apply a Graux type localised vertical overload over a limited width in the soil situated on one side of the wall. This overload is "uniformly" diffused within the diffusion cone in accordance with the following figure (part C of the manual details how this type of Graux overload is taken into account in the calculations).

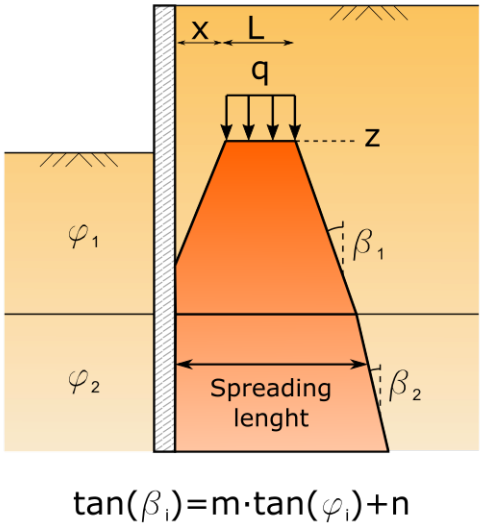


Figure B116 : Graux overload schematic

The parameters to be input to define this overload are as follows:

- **"Wall 1"** or "**Wall 2"** for a double wall project;
- **Activate:** chosen by default to define a new overload;
- Side of application of the Graux overload:
  - **"Left"** or "**Right"** for a single wall project;
  - **"Left"** or "**Right/E.2R"** (wall 1) or "**Left/E.2R"** or "**Right"** (wall 2) for a double wall project;
- **z**: level of application of the overload (m or ft);
- **x**: distance to the wall > 0 (m or ft);
- **L**: application width > 0 (m or ft);
- **m**: proportionality term between the tangent of the friction angle and the tangent of the diffusion angle (-);
- **n**: constant term in the diffusion angle tangent formula (-);

  Note: the particular case in which m = 1 and n = 0 guarantees diffusion at a phi angle value in each layer. In addition, choosing m = 0 simulates constant angle diffusion independent of the friction angle of the soils encountered.

- **q**: overload amplitude (kN/m^2 or ksf);
- **Action nature**: if ULS checks were requested, the nature of the overload must also be defined, specifying whether it is **permanent** or **variable**. This choice determines the value of the partial factor to be applied to the overload (see § B.3.1.2).

- **Family**: if the load cases calculation was activated (see §B.3.5), the overload must be allocated to a loads family using the drop-down list containing all the previously defined families.

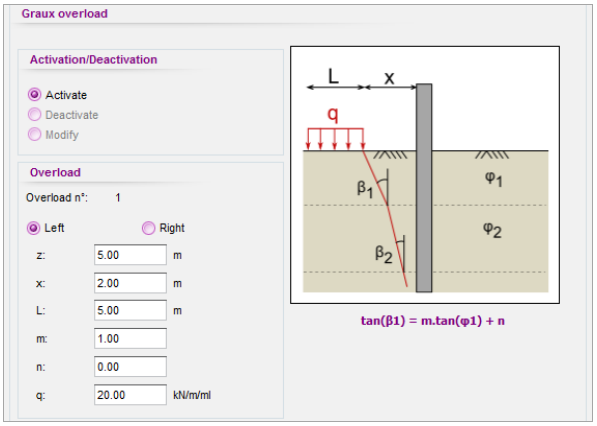


Figure B117 : Definition of a Graux overload

The "Excavation" and "Fill" actions cancel the previously defined Graux overloads on the same side of the wall.

To **modify a Graux overload** previously defined, select "**Modify**". The modifiable values are:

- **q**: overload amplitude (kN/m/ml or kip/lft);
- **Action nature**: if ULs checks are requested.

The following figure shows the drop-down list containing the previously defined Graux overloads still present. Each Graux overload is identified by the following:

- its definition phase and its declaration number;
- its application level;
- its position with respect to the wall and its amplitude.

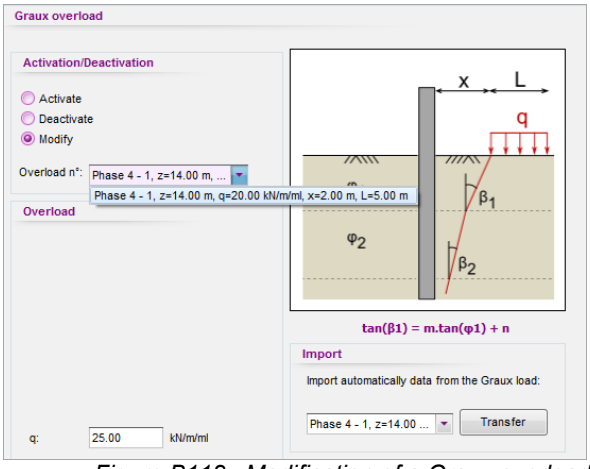


Figure B118 : Modification of a Graux overload

To deactivate a previously defined **Graux overload**, select "**Deactivate"**. The following figure shows the drop-down list containing the previously defined Graux overloads still present. Each Graux overload is identified by the following:

- its definition phase and its declaration number;
- its application level;
- its position with respect to the wall and its amplitude.

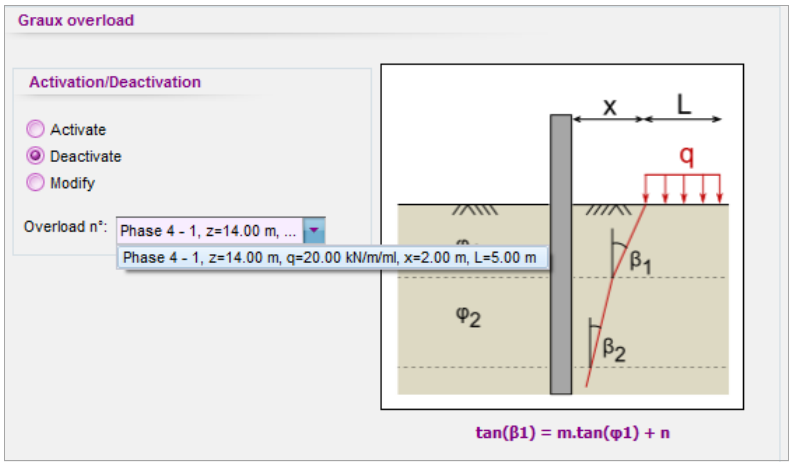


Figure B119 : Deactivation of a Graux overload

B.5.6.4. Line force

This action is used to manage (activate, modify or deactivate) the line forces applied directly to the wall.

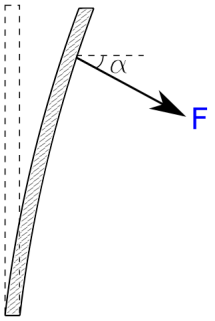


Figure B120 : Line force schematic

The parameters needed for definition of a line force are:

- **"Wall 1"** or "**Wall 2"** for a double wall project;
- **Activate:** chosen by default to define a new force;
- **z**: level of application (m or ft);
- **F**: amplitude of the force (kN/ml or kip/lft);
- α: angle of the force with respect to the horizontal, counted positively clockwise (°);
- **Action nature**: if ULS checks were requested, the nature of the force must also be defined, specifying whether it is **permanent** or **variable** on the one hand, and **favourable** or **unfavourable** on the other. These choices will affect the value of the partial factor to be applied to the load value (see § B.3.1.2 of the manual).
- **Family**: if the load cases calculation was activated (see §B.3.5), the overload must be allocated to a loads family using the drop-down list containing all the previously defined families.

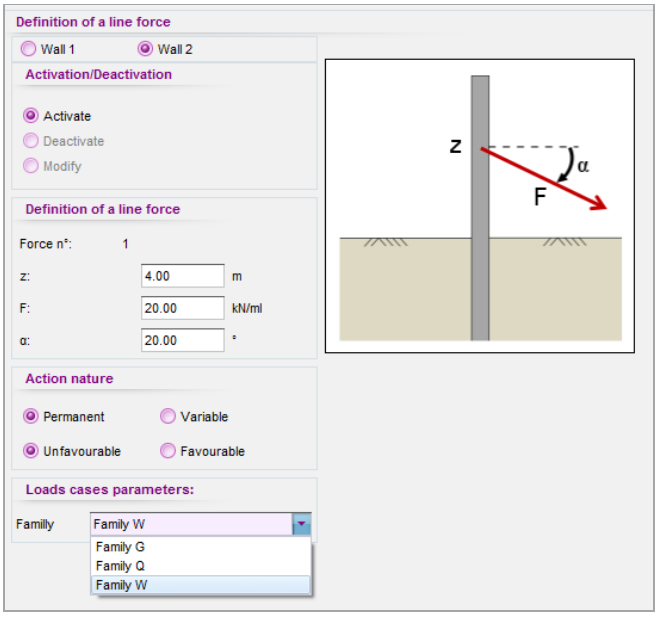


Figure B121 : Definition of a line force

To **modify a line force** previously defined, select **"Modify"**. The following figure shows the drop-down list containing the previously defined line forces which are still present. Each line force is identified by the following:

- its declaration number;
- its application level;
- its amplitude;
- its angle.

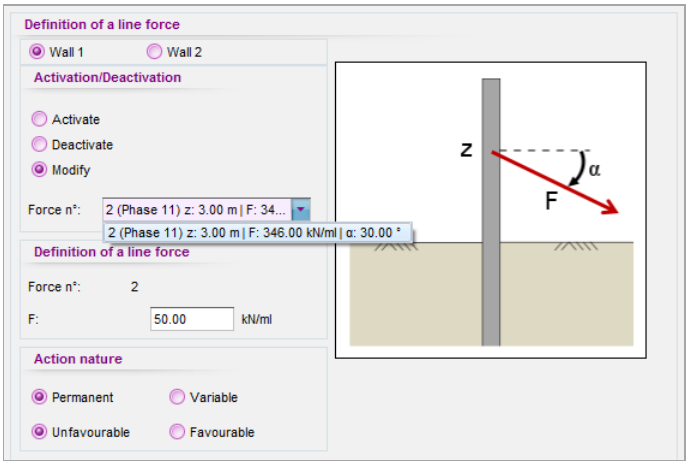


Figure B122 : Modification of a line force

The modifiable values are:

- **F**: force amplitude (kN/ml or kip/lft);
- **Action nature**: if ULs checks are requested.

To deactivate a previously defined **line force**, select "**Deactivate"**. The following figure shows the drop-down list containing the previously defined line forces which are still present. Each line force is identified by the following:

- its declaration number and its definition phase;
- its application level;
- its amplitude;
- its angle.

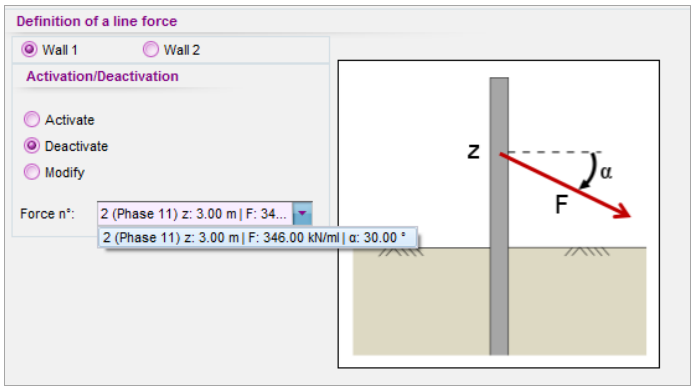


Figure B123 : Deactivation of a line force

B.5.6.5. Moment

This action is used to manage (activate, modify or deactivate) the linear moments applied directly to the wall.

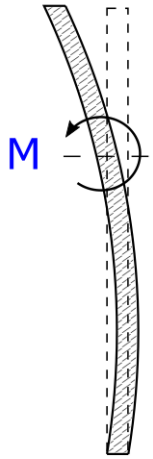


Figure B124 : Moment schematic

The parameters to be input to define a moment are:

- **"Wall 1"** or "**Wall 2"** for a double wall project;
- **Activate:** chosen by default to define a new moment;
- **z:** level of application (m or ft);
- **M:** amplitude of moment (kNm/ml or kip/lft);
- **Action nature:** if ULS checks were requested, the nature of the load must also be defined, specifying whether it is **permanent** or **variable** on the one hand and **favourable** or **unfavourable** on the other. These choices will affect the value of the partial factor to be applied to the load value (see § B.3.1.2 of the manual).

- **Family**: if the load cases calculation was activated (see §B.3.5), the overload must be allocated to a loads family using the drop-down list containing all the previously defined families.

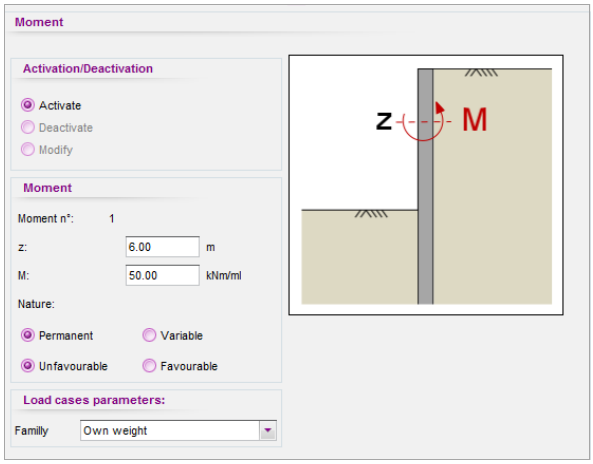


Figure B125 : Definition of a moment

To **modify a moment** previously defined, select "**Modify"**. The following figure shows the drop-down list containing the previously defined moments which are still present. Each moment is identified by the following:

- its declaration number and the definition phase;
- its application level;
- its amplitude;
- its angle.

The modifiable values are:

- **M:** moment amplitude (kNm/ml or kip.ft/lft);
- **Action nature:** if ULS checks were requested, the nature of the load must also be defined, specifying whether it is **permanent** or **variable** on the one hand and **favourable** or **unfavourable** on the other. These choices will affect the value of the partial factor to be applied to the load value (see § B.3.1.2 of the manual).

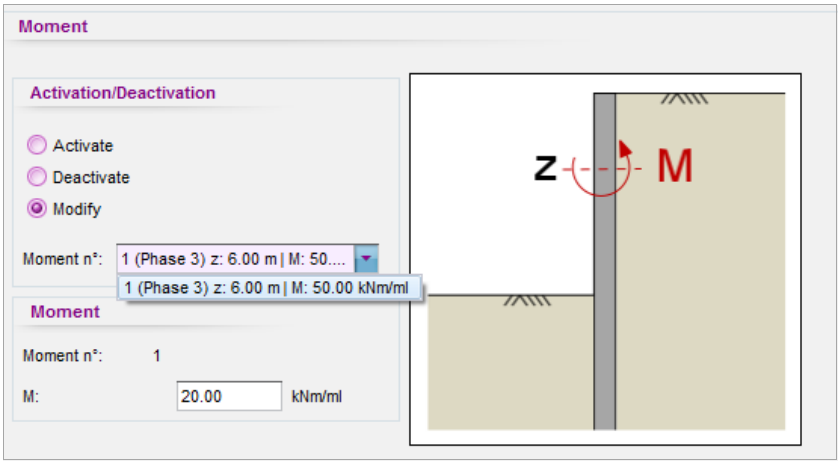


Figure B126 : Modification of a moment

To **Deactivate a moment** previously defined, select "**Deactivate"**. The following figure shows the drop-down list containing the previously defined moments which are still present. Each moment is identified by the following:

- its declaration number and its definition phase;
- its application level;
- its amplitude.

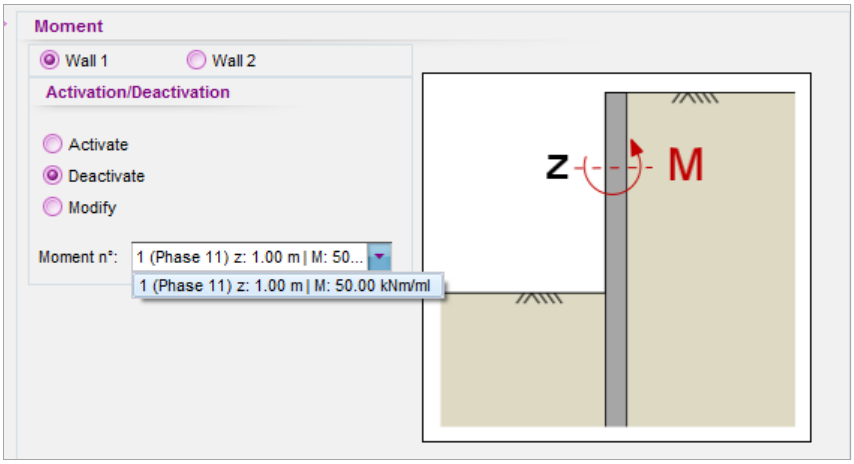


Figure B127 : Deactivation of a moment

B.5.6.6. Horizontal load

This command is used to manage (activate, modify or deactivate) the horizontal loads applied directly to the wall.

The parameters to be input to define this type of load are:

- **"Wall 1"** or "**Wall 2"** for a double wall project;
- **Activate:** chosen by default to define a new load;
- z_t: upper level of load (m or ft);
- z_b: lower level of load (m or ft);
- α: angle of the load with respect to the horizontal, counted positively clockwise (°).
- q_{ht}: load amplitude at level z_t (kN/m/ml or kip/lft);
- q_{hb}: load amplitude at level z_b (kN/m/ml or kip/lft).
- **Action nature:** if ULS checks were requested, the nature of the load must also be defined, specifying whether it is **permanent** or **variable** on the one hand and **favourable** or **unfavourable** on the other. These choices will affect the value of the partial factor to be applied to the load value (see § B.3.1.2 of the manual).
- **Family**: if the load cases calculation was activated (see §B.3.5), the overload must be allocated to a loads family using the drop-down list comprising all the previously defined families.

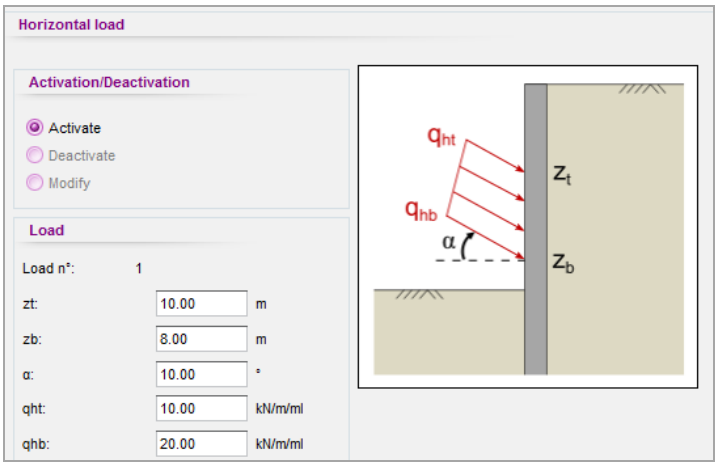


Figure B128 : Definition of a horizontal load

The overload value between q_{ht} and q_{hb} is obtained by linear interpolation between levels z_t and z_b.

To **modify a horizontal load** previously defined, select "**Modify"**. The modifiable values are:

- q_{ht}: load amplitude at level z_t (kN/m/ml or kip/lft);
- q_{hb}: load amplitude at level z_b (kN/m/ml or kip/lft).
- **Action nature:** if ULS checks were requested, the nature of the load must also be defined, specifying whether it is **permanent** or **variable** on the one hand and **favourable** or **unfavourable** on the other. These choices will affect the value of the partial factor to be applied to the load value (see § B.3.1.2 of the manual).

The following figure shows the drop-down list containing the previously defined horizontal loads which are still present. Each horizontal load is identified by the following:

- its declaration number and its definition phase;
- its application levels (top and bottom);
- its amplitude at levels z_t and z_b;
- its angle.

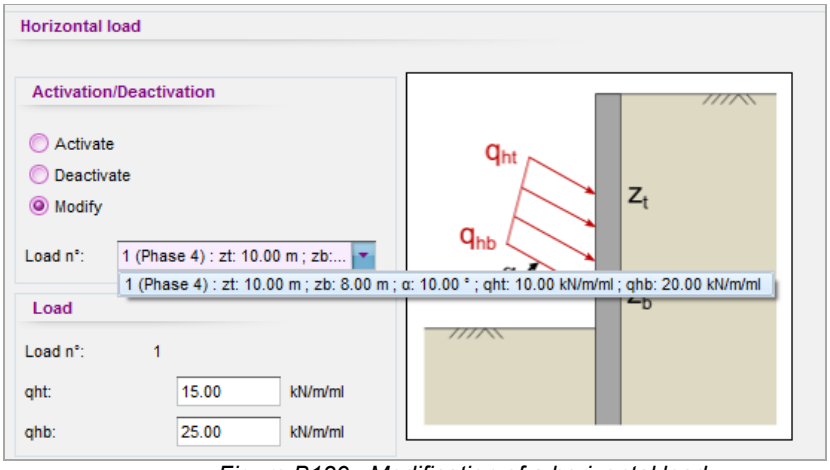


Figure B129 : Modification of a horizontal load

To deactivate a previously defined **horizontal load**, select "**Deactivate"**. The following figure shows the drop-down list containing the previously defined horizontal loads which are still present. Each horizontal load is identified by the following:

- its declaration number and its definition phase;
- its application levels (top and bottom);
- its angle;
- its amplitude at levels z_t and z_b.

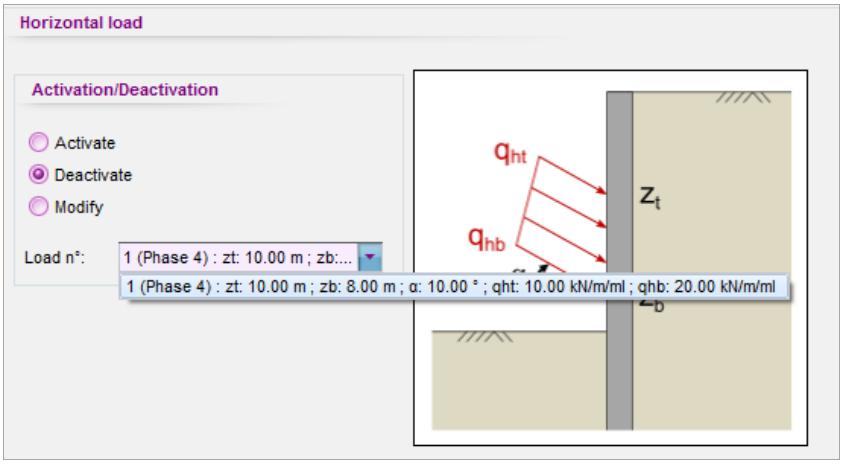


Figure B130 : Deactivation of a horizontal load

B.5.7. Automatic actions

B.5.7.1. LEM Options (Limit Equilibrium Method)

This action is used to check the available options in the ULS checks frame for the phases during which the wall is defined as cantilever (LEM calculation), in other words, the phases for which the "**Cantilever wall"** box is ticked.

This command comprises 2 sections:

- **Over-excavation:** definition of the over-excavations to be taken into account in the ULS checks;
- **LEM calculation options:** configuration of the options used in the LEM calculation.

The parameters of this automatic action are as follows:

- **Δa_{left}:** over-excavation value on the left side of the wall to be taken into account in the ULS checks (LEM);
- **Δa_{right}**: over-excavation value on the right side of the wall to be taken into account in the ULS checks (LEM).

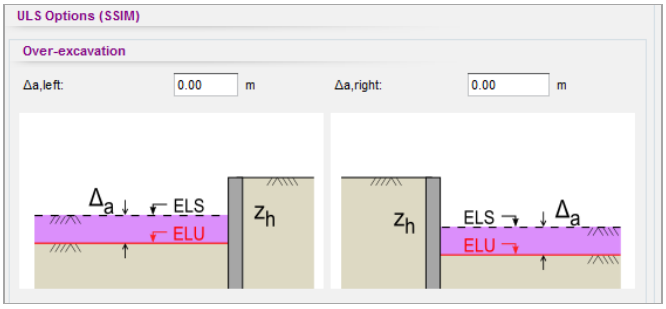


Figure B131 : Modification of over-excavation parameters

- **Calculation method:** calculation method used for the LEM checks. The "Automatic" choice is selected by default. In this case, method D is used. If you uncheck this option, you can choose to apply method F or method D for this phase (see Part C of the manual for details about these two calculation methods). When method D is selected, an additional option can be used to define the design embedment depths.

  Three choices are then available for the base of the embedment depth considered in the LEM calculation:

  - **Wall base:** the base of the embedment depth considered corresponds to the base of the wall (default option);
  - **Level z_c – 0.2 x f_0**: the embedment depth base considered corresponds to the point on the wall z_{base} = z_c - 0.2 x f_0 and is automatically evaluated by the calculation engine once z_c and f_0 have been calculated;
  - **Personalized**: the base of the embedment depth considered is set by the user.

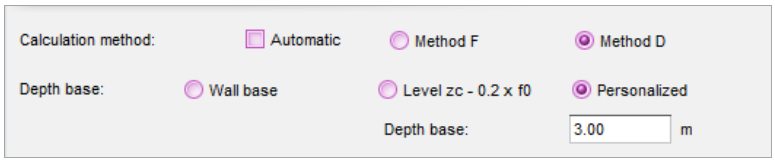


Figure B132 : Modification of calculation method parameters

- **Selection of passive earth pressure side**: passive earth pressure side considered during the checks performed for this phase. The "Automatic" box is selected for this option by default. In this case, the side with the lowest passive earth pressures ratio in the SSIM calculation is chosen. If you deselect this option, you can force the choice of passive earth pressure side considered (see Part C of the manual).

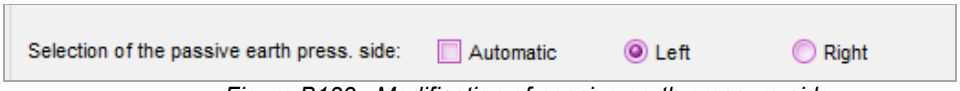


Figure B133 : Modification of passive earth pressure side

- **Automatic correction of counter passive earth pressure angles:** allows automatic correction of the counter passive earth pressure angles when evaluating the checks for this phase (see Part C of the manual + Part D/Tutorial 3).

  When the user deactivates this option, a table containing the soil layers present in the current phase appears. This table enables the user to define personalized counter passive earth pressure parameters.

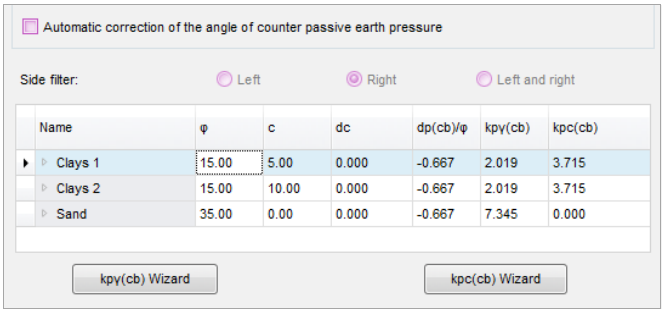


Figure B134 : Modification of characteristics of counter passive earth pressure for a LEM calculation

By default, for each soil layer, the counter passive earth pressure parameters ($\delta_{p(cb)}/\varphi$, $k_{p\gamma(cb)}$ and $k_{pc(cb)}$) are considered to be equal to those of the passive earth pressure (values of δ_p/φ, $k_{p\gamma}$ and k_{pc}).

In order to be able to simultaneously check the vertical and horizontal forces with compatible pressures, the user may need to modify the angle of the counter passive earth pressure $\delta_{p(cb)}/\varphi$ (see Example 3 in part D of the manual).

These parameters are used for ULS (LEM) verification calculations as described in part C of the manual.

B.5.7.2. ELU Options (SSIM)

This command is used to check the options available for ULS checks for the phases during which the wall is considered to be anchored (SSIM calculation), in other words, the phases for which the "**Cantilever wall"** box is not ticked.

This command comprises 2 sections:

- **Over-excavation:** definition of over-excavations to be taken into account in the ULS checks;
- **Kranz Options:** configuration of options used for the Kranz calculation.

The parameters of this command are as follows:

- **Δa_{left}**: over-excavation value on the left side of the wall to be taken into account during the ULS (SSIM) checks;
- **Δa_{right}**: over-excavation value on the right side of the wall to be taken into account during the ULS (SSIM) checks.

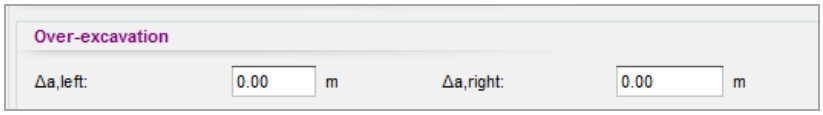


Figure B135 : Modification of over-excavation parameters

- **Position z_D of the null shear force point**: used to choose how the z_D point used during the Kranz calculation is set (see Part C of the manual). The "Automatic" option is selected by default, in which case the calculation engine considers the shear force point to be the lowest point between the base of the wall and the bottom of the excavation. If this option is deselected, it is possible to impose the level considered for the z_D point.

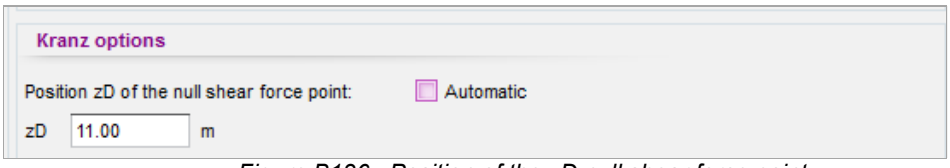


Figure B136 : Position of the zD null shear force point

B.5.8. Earthquake (seismic calculation)

This action is used to define the characteristics of a seismic calculation with respect to a phase for which the **"Seismic calculation"** box was ticked.

It should be noted that this option is only available in the phases for which the user has not yet inserted actions. Conversely, activation of this option deactivates the possibility of creating new actions in the current phase. The phase is then considered to be an "orphan". This means that the initial state reference for a phase without earthquake defined after a phase with earthquake will be the last phase without earthquake previously defined and not

the seismic calculation phase. The following diagram explains the phasing when phases with earthquake are present.

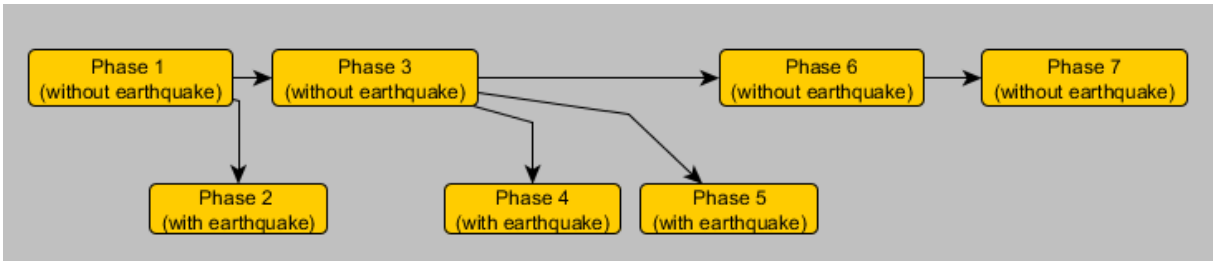


Figure B137 : Calculation phasing with processing of phases with earthquake

This command comprises 2 sections:

- **Acceleration parameters:** characteristics of the earthquake;
  - **k_h:** horizontal seismic coefficient;
  - **k_v:** vertical seismic coefficient;
  - **XP:** passive earth pressure limitation factor (≤1).
- **Soil parameters:** soil behaviour during seismic calculation. For each soil layer:
  - γ: wet unit weight (kN/m^3, kcf);
  - γ_d: dry unit weight (kN/m^3, kcf);
  - **Water behaviour left** and **right**: used to choose an open behaviour (highly permeable soil) or closed behaviour (low permeability soil) for the soil layer selected on each side of the wall.

The calculation of seismic effects is carried out using the "pseudo-static" method. The K-Réa calculation engine uses the above parameters for the following operations:

- Correction of limit active/passive earth pressure diagrams on each side of the wall taking account of seismic action;
- Correction of hydraulic profiles on each side of the wall at the levels where the groundwater is free and/or those where the soil was declared as an "open" medium.
- Application of a force of inertia associated with the mass of the wall.

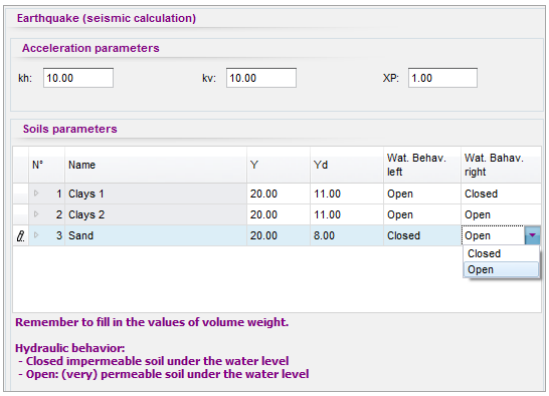


Figure B138 : Seismic calculation

The mathematical description of the calculation method used is detailed in part C of the manual.

B.6. Calculations and results

The calculations carried out by K-Réa are performed for a unit length of wall, so most of the data and results are relative to this unit length. The unit /ml (per linear metre) or /lft (per linear foot) is explicitly recalled in the results provided.

B.6.1. General presentation

B.6.1.1. Calculation

Click the button on the buttons bar to start the calculations for all the calculation phases and ULS checks, if requested.

The calculations can be made at any time (in initial phase, during phasing, or in the final phase) provided that the soil, wall and action data have been correctly filled out.

B.6.1.2. Calculations flowchart

The available results depend on the type of calculation performed. The following calculation flowchart explains the results obtained for the various types of available calculation. Part C of the manual gives detailed explanations about the various types of calculation mentioned in this flowchart.

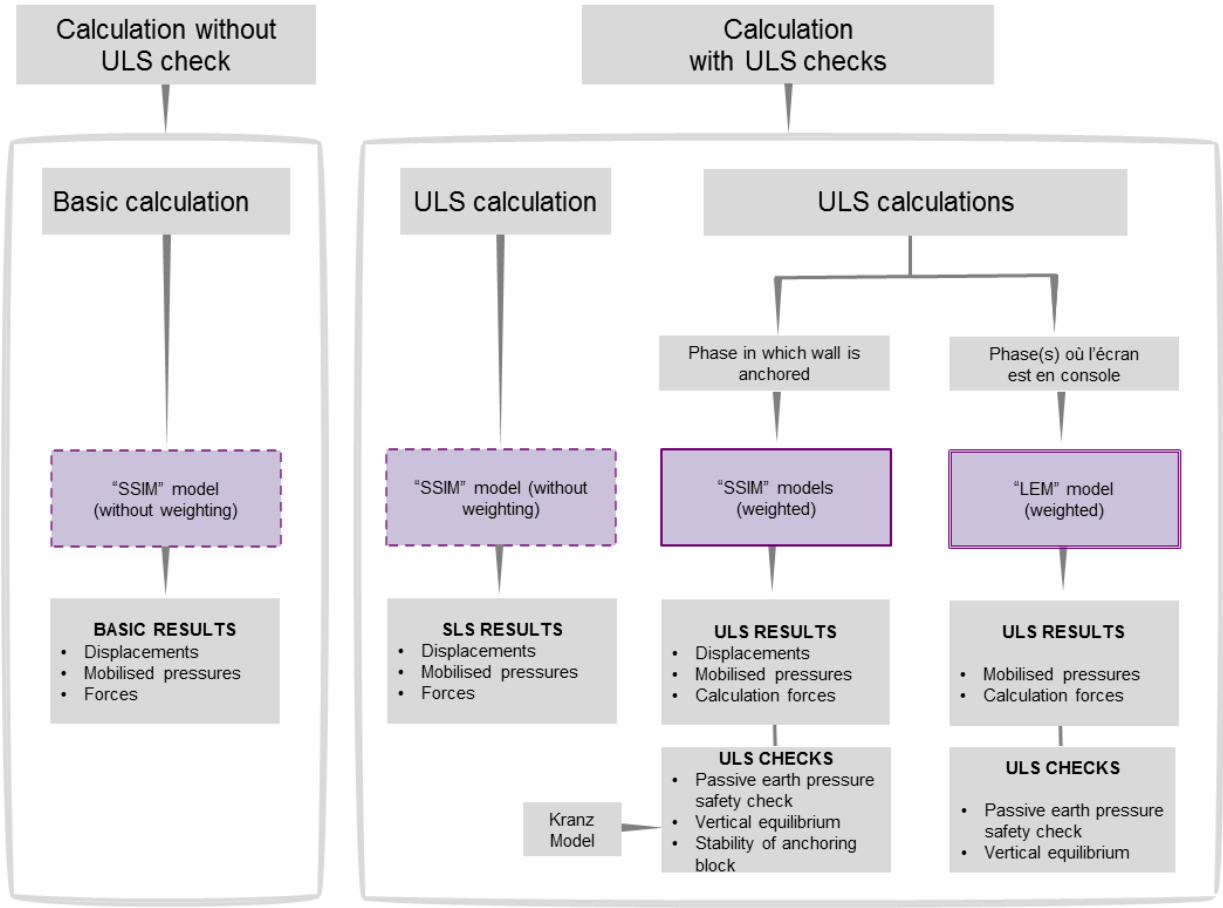


Figure B139 : Calculation flowchart and results obtained for each type of calculation

B.6.1.3. Results for a calculation without ULS checks

All the phases are processed using the "basic" SSIM calculation without weighting on the soil characteristics and overloads.

The results obtained include: wall displacements, bending moments, shear forces, mobilised pressures and support reactions.

Chapter B.6.2 gives a detailed presentation of the results of the "basic" SSIM calculation.

B.6.1.4. Calculation with ULS checks

For each phase, two calculations are performed:

- **A "SLS" calculation:** calculation based on an SSIM model carried out without weighting on the characteristics of soils and overloads. The results of this calculation are strictly identical to those of a calculation "without ULS checks": displacements, bending moments, shear forces, mobilised pressures and support reactions.

  See chapter B.6.2.5 for a detailed presentation of the SLS results as displayed in K-Réa (the presentation of these results is very similar to that of the basic SSIM calculation results.

- **A "ULS" calculation:** calculation based on a weighted SSIM model supplemented by a LEM calculation for those phases in which the wall is considered to be cantilever.

  The available results are as follows:

  - Wall displacements (for those phases in which the wall is anchored);
  - Bending moments and shear forces calculation values;
  - Mobilised pressures calculation values;
  - Forces in supports calculation values.

  See chapter B.6.3 for a detailed presentation of the ULS results (SSIM) as displayed in K-Réa.

  The results of the following ULS checks are also available:

  - Verification of passive earth pressure safety check;
  - Verification of vertical equilibrium;
  - Verification of stability of anchoring block (Kranz) for phases in which wall is anchored.

  See chapter B.6.4.3 for a detailed presentation of the verification results as displayed in K-Réa.

B.6.2. Calculation without ULS checks

B.6.2.1. Results available in the K-Réa main window

After the end of the calculations, some of the results are displayed in graphic form in the main window in the context of management of the current phase: displacements, bending moments and shear forces.

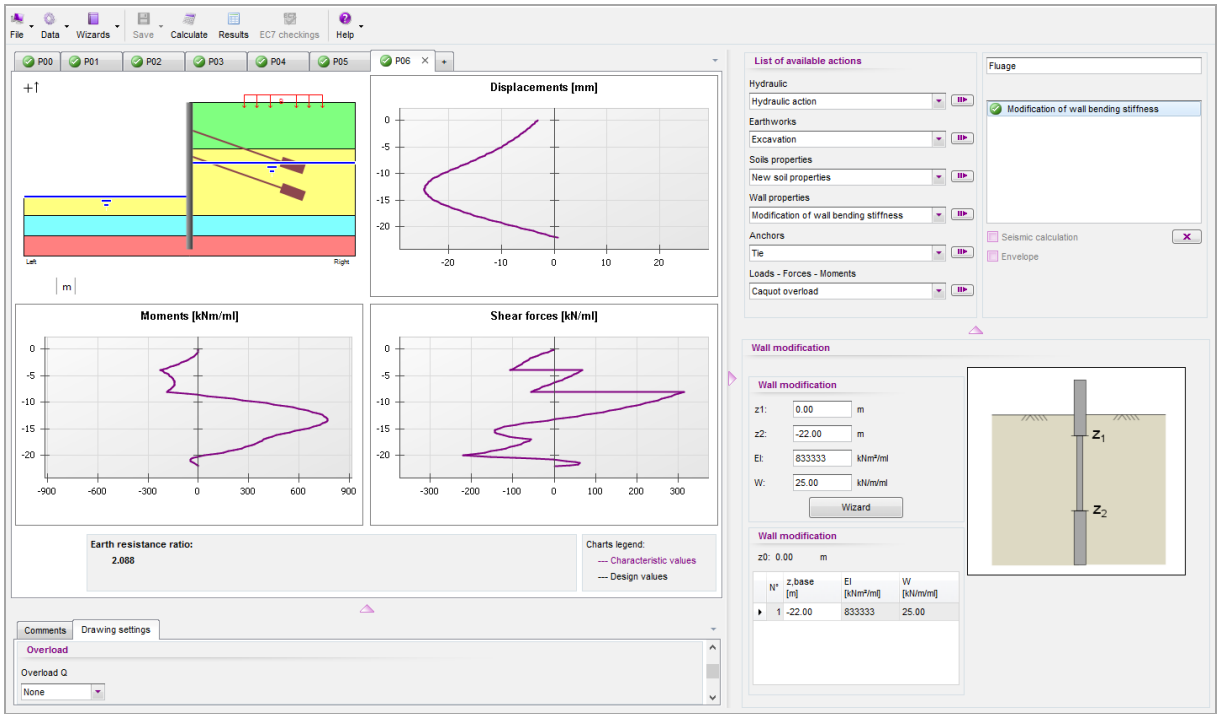
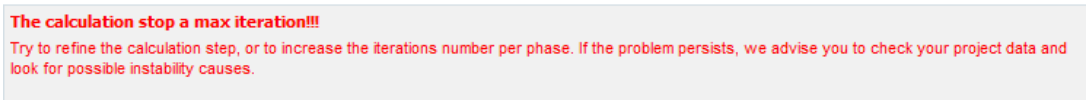


Figure B140 : Display of SSIM results (without ULS checks) in the main window

The (mobilisable passive earth pressure/mobilised passive earth pressure) ratio is also displayed in the main window.

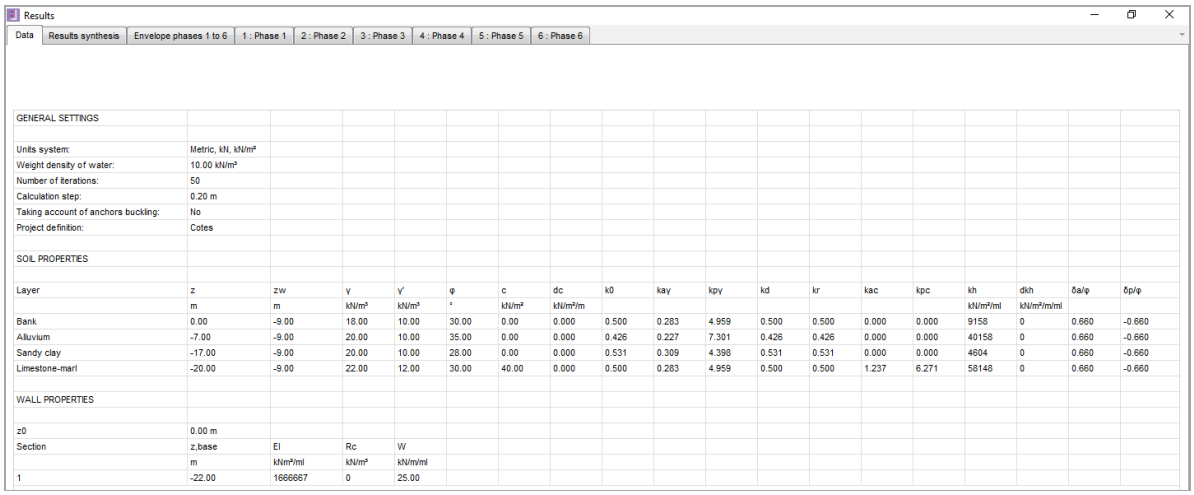
A warning message appears in the phasing management frame if the calculation reaches the maximum number of iterations per phase. For reasons of consistency, this message also appears for all the phases following a phase in which the calculation did not converge.



It is of course possible to access more detailed results by opening the results window: this is a specific window which follows the same operating principle as the phasing management frame, using tabs. For this, click the "**Results"** button on the buttons bar.

B.6.2.2. Results window / "Data" tab

When the results window appears, it opens by default on the first tab, which is a recapitulation of the general project data:



GENERAL SETTINGS

Units system:	Metric, kN, kN/m²
Weight density of water:	10.00 kN/m³
Number of iterations:	50
Calculation step:	0.20 m
Taking account of anchors buckling:	No
Project definition:	Cotes

SOIL PROPERTIES

Layer	z	zw	γ	γ'	φ	c	dc	k0	kay	kpy	kd	kr	kac	kpc	kh	dkh	δa/φ	δp/φ
	m	m	kN/m³	kN/m³	°	kN/m²	kN/m²/m								kN/m³/ml	kN/m³/m/ml		
Bank	0.00	-9.00	18.00	10.00	30.00	0.00	0.000	0.500	0.283	4.959	0.500	0.500	0.000	0.000	9158	0	0.660	-0.660
Alluvium	-7.00	-9.00	20.00	10.00	35.00	0.00	0.000	0.426	0.227	7.301	0.426	0.426	0.000	0.000	40158	0	0.660	-0.660
Sandy clay	-17.00	-9.00	20.00	10.00	28.00	0.00	0.000	0.531	0.309	4.398	0.531	0.531	0.000	0.000	4604	0	0.660	-0.660
Limestone-marl	-20.00	-9.00	22.00	12.00	30.00	40.00	0.000	0.500	0.283	4.959	0.500	0.500	1.237	6.271	58148	0	0.660	-0.660

WALL PROPERTIES

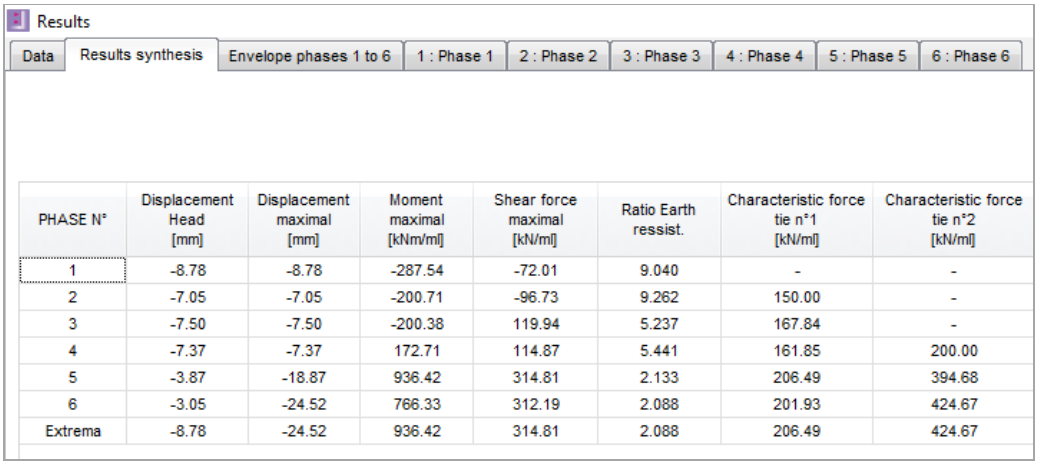
z0	0.00 m			
Section	z,base	EI	Rc	W
	m	kNm²/ml	kN/m²	kN/m/ml
1	-22.00	1666667	0	25.00

Figure B141 : Results window: "Data" tab

For a single wall project, this tab groups together all the soil and wall characteristics data along with the chosen calculation options and the actions performed in the initial phase (e.g. reduced pressure).

B.6.2.3. Results window / "Results synthesis" tab

The "Results synthesis" tab, which follows that dedicated to data, gives a summary table of the extreme values obtained for the main types of results, for each calculation phase and overall for the phasing as a whole (last line of table).



PHASE N°	Displacement Head [mm]	Displacement maximal [mm]	Moment maximal [kNm/ml]	Shear force maximal [kN/ml]	Ratio Earth ressist.	Characteristic force tie n°1 [kN/ml]	Characteristic force tie n°2 [kN/ml]
1	-8.78	-8.78	-287.54	-72.01	9.040	-	-
2	-7.05	-7.05	-200.71	-96.73	9.262	150.00	-
3	-7.50	-7.50	-200.38	119.94	5.237	167.84	-
4	-7.37	-7.37	172.71	114.87	5.441	161.85	200.00
5	-3.87	-18.87	936.42	314.81	2.133	206.49	394.68
6	-3.05	-24.52	766.33	312.19	2.088	201.93	424.67
Extrema	-8.78	-24.52	936.42	314.81	2.088	206.49	424.67

Figure B142 : Display of results synthesis table (single wall, without ULS checks)

The types of results for which these extreme values are given are as follows:

- **Displacement at head** of wall (in mm or in);
- **Maximum displacement** obtained along the wall (in mm or in);
- **Maximum bending moment** obtained along the wall (in kNm/ml or kip.ft/lft);
- **Maximum shear force** obtained along the wall (in kN/ml or kip/lft);
- **Maximum arch pressure** obtained along the wall, only for walls defined as being circular (in kN/ml);
- **Maximum normal force** obtained along the wall (in kN/ml);
- **Earth pressures ratio:** (mobilisable passive earth pressure / mobilised passive earth pressure) ratio. It should be noted that unlike the other columns, the extreme value presented in the last row for the passive earth pressures ratio is the minimum value encountered on all the phases (and not the maximum value, as is the case for the other columns);
- **Characteristic force ties:** forces taken up by the anchors for each phase (variable unit).

B.6.2.4. Results window/ "Envelope" tab(s)

The tab(s) which follow "Results synthesis" in the results window is (are) devoted to the envelope displacements, bending moments and shear forces.

- If no "intermediate" envelope was requested at definition of phasing, a single "Envelope" tab is available. It corresponds to the envelopes calculated for the entire phasing defined.
- If intermediate envelopes were requested at definition of phasing, several envelope tabs are created and correspond to the breakdown imposed by the user with the ticked boxes.

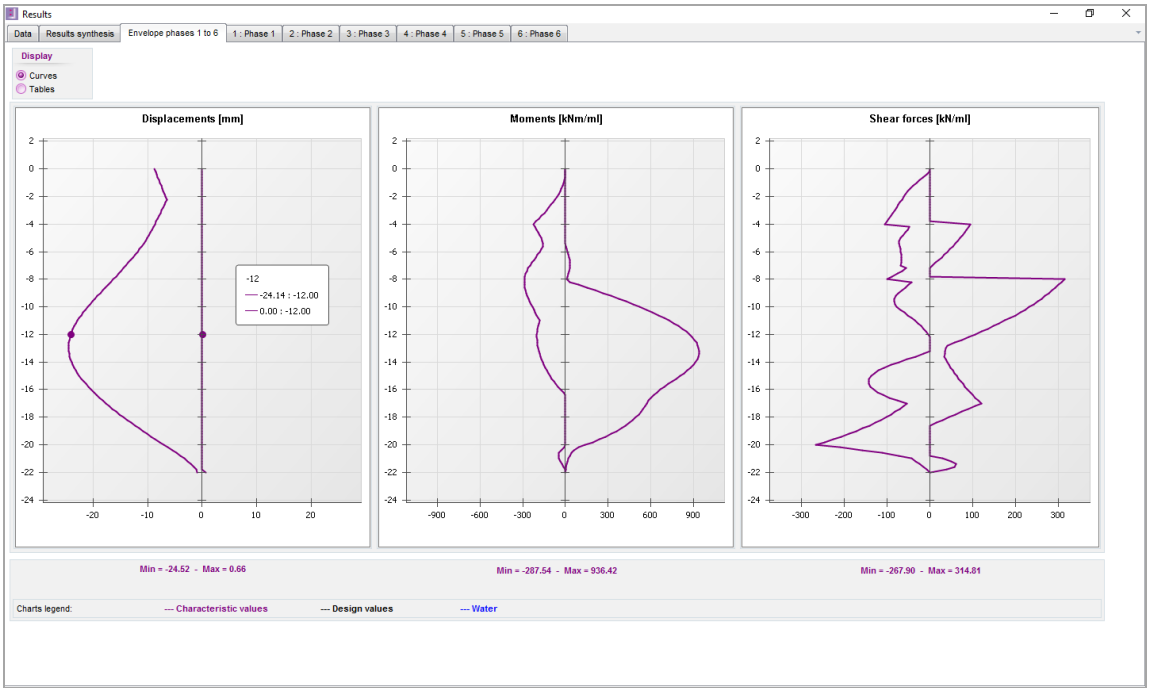


Figure B143 : Display of envelopes for phases 1 to 5

In each "Envelope" tab, it is possible to switch to "Curves" or "Table" display mode from the selection list.

Two buttons appear in each tab in the results window. The Print button is used to open the print dialogue box and the Quit button is used to close the results window.

B.6.2.5. Results per phase: graphical representation

The following tabs correspond to the phases defined in the project. They give the curves ("Curves" option checked by default): displacements, bending moments, shear forces, rotations, arch pressures, normal force and soil and water pressures.

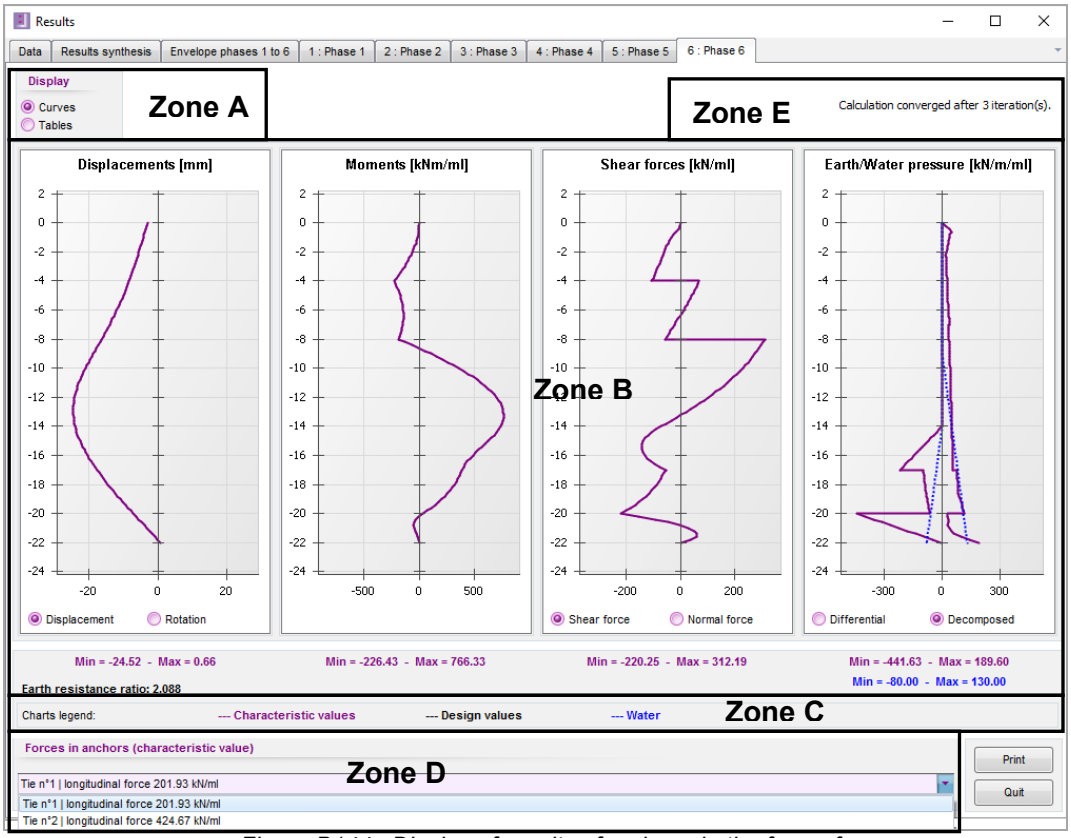
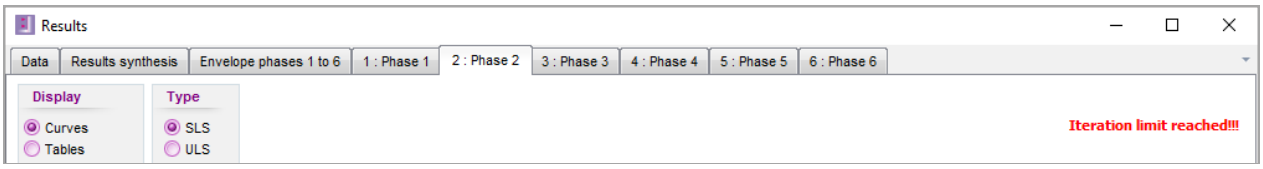


Figure B144 : Display of results of a phase in the form of curves (single wall, without ULS checks)

Each "phase" tab comprises **five zones**:

- **Zone A:** choice of type of display (curves or tables).
- **Zone B:** devoted to the display of the actual results:
- **Zone C**: reserved for display of the (mobilisable passive earth pressure / mobilised passive earth pressure) ratio.
- **Zone D**: contains a drop-down list giving the forces in the anchors active in the phase concerned.
- **Zone E**: devoted to information or warning messages, such as that regarding calculation convergence. If this was not reached after the maximum number of iterations set in the data (see chapter B.3.1), the calculation stops (to avoid an infinite loop) and a message appears to alert the user.



In zone B, the following curves are displayed:

- curve of wall displacements / rotations;
- curve of wall bending moment / arch pressure;
- curve of calculated wall shear force / axial force;
- curve of **earth and water pressures**: in the case of a "decomposed" display, the solid purple line curves correspond to the earth pressures and the dotted blue lines to the water pressures. The curves corresponding to the negative values are those of the pressures which apply to the left-hand side of the wall. Conversely, those which correspond to the positive values are the pressure values which apply to the right-hand side of the wall. It is also possible to request display of the differential pressure calculated by adding the earth pressures on either side of the wall and the water pressures.

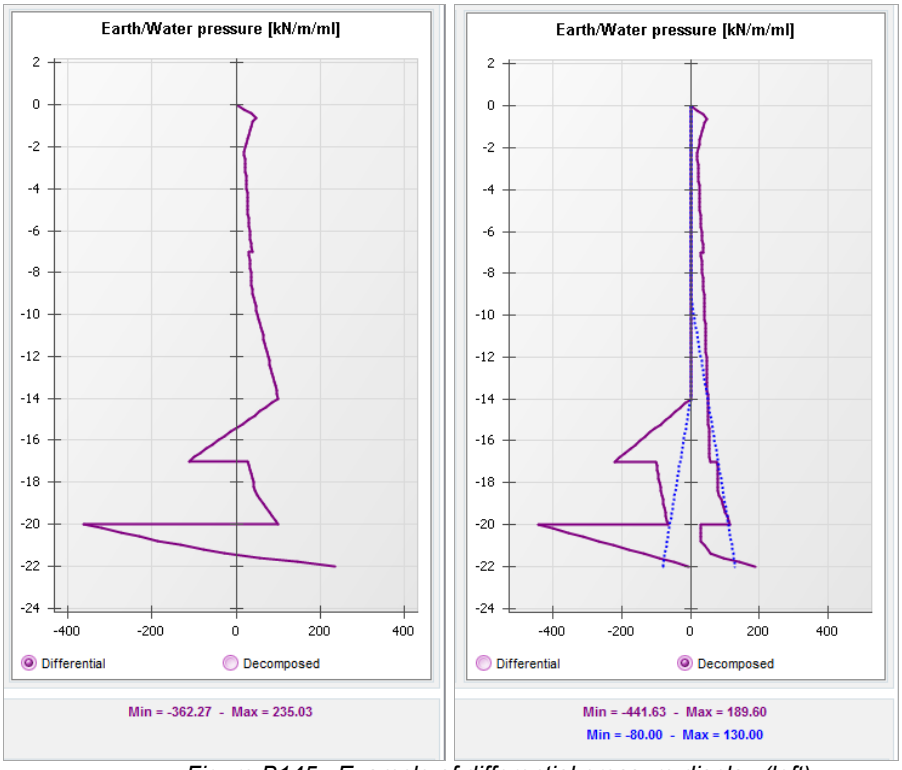
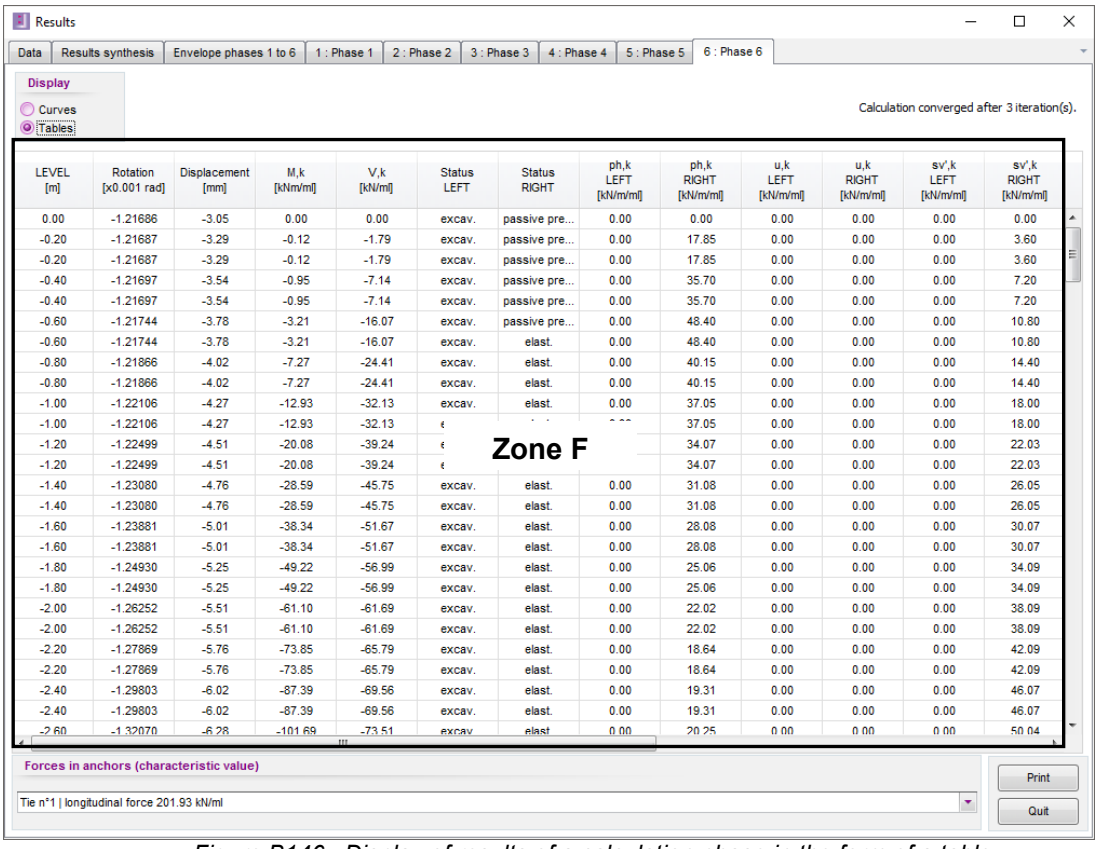


Figure B145 : Example of differential pressure display (left) and decomposed pressures display (right)

The minimum and maximum values of each of the curves appear under each of the curves.

B.6.2.6. Results per phase: tables of values

It is possible to switch to a display in the form of results tables, by selecting the "Tables" option.



LEVEL [m]	Rotation [x0.001 rad]	Displacement [mm]	M,k [kNm/ml]	V,k [kN/ml]	Status LEFT	Status RIGHT	ph,k LEFT [kN/m/ml]	ph,k RIGHT [kN/m/ml]	u,k LEFT [kN/m/ml]	u,k RIGHT [kN/m/ml]	sv',k LEFT [kN/m/ml]	sv',k RIGHT [kN/m/ml]
0.00	-1.21686	-3.05	0.00	0.00	excav.	passive pre...	0.00	0.00	0.00	0.00	0.00	0.00
-0.20	-1.21687	-3.29	-0.12	-1.79	excav.	passive pre...	0.00	17.85	0.00	0.00	0.00	3.60
-0.20	-1.21687	-3.29	-0.12	-1.79	excav.	passive pre...	0.00	17.85	0.00	0.00	0.00	3.60
-0.40	-1.21697	-3.54	-0.95	-7.14	excav.	passive pre...	0.00	35.70	0.00	0.00	0.00	7.20
-0.40	-1.21697	-3.54	-0.95	-7.14	excav.	passive pre...	0.00	35.70	0.00	0.00	0.00	7.20
-0.60	-1.21744	-3.78	-3.21	-16.07	excav.	passive pre...	0.00	48.40	0.00	0.00	0.00	10.80
-0.60	-1.21744	-3.78	-3.21	-16.07	excav.	elast.	0.00	48.40	0.00	0.00	0.00	10.80
-0.80	-1.21866	-4.02	-7.27	-24.41	excav.	elast.	0.00	40.15	0.00	0.00	0.00	14.40
-0.80	-1.21866	-4.02	-7.27	-24.41	excav.	elast.	0.00	40.15	0.00	0.00	0.00	14.40
-1.00	-1.22106	-4.27	-12.93	-32.13	excav.	elast.	0.00	37.05	0.00	0.00	0.00	18.00
-1.00	-1.22106	-4.27	-12.93	-32.13	excav.	elast.	0.00	37.05	0.00	0.00	0.00	18.00
-1.20	-1.22499	-4.51	-20.08	-39.24	excav.	elast.	0.00	34.07	0.00	0.00	0.00	22.03
-1.20	-1.22499	-4.51	-20.08	-39.24	excav.	elast.	0.00	34.07	0.00	0.00	0.00	22.03
-1.40	-1.23080	-4.76	-28.59	-45.75	excav.	elast.	0.00	31.08	0.00	0.00	0.00	26.05
-1.40	-1.23080	-4.76	-28.59	-45.75	excav.	elast.	0.00	31.08	0.00	0.00	0.00	26.05
-1.60	-1.23881	-5.01	-38.34	-51.67	excav.	elast.	0.00	28.08	0.00	0.00	0.00	30.07
-1.60	-1.23881	-5.01	-38.34	-51.67	excav.	elast.	0.00	28.08	0.00	0.00	0.00	30.07
-1.80	-1.24930	-5.25	-49.22	-56.99	excav.	elast.	0.00	25.06	0.00	0.00	0.00	34.09
-1.80	-1.24930	-5.25	-49.22	-56.99	excav.	elast.	0.00	25.06	0.00	0.00	0.00	34.09
-2.00	-1.26252	-5.51	-61.10	-61.69	excav.	elast.	0.00	22.02	0.00	0.00	0.00	38.09
-2.00	-1.26252	-5.51	-61.10	-61.69	excav.	elast.	0.00	22.02	0.00	0.00	0.00	38.09
-2.20	-1.27869	-5.76	-73.85	-65.79	excav.	elast.	0.00	18.64	0.00	0.00	0.00	42.09
-2.20	-1.27869	-5.76	-73.85	-65.79	excav.	elast.	0.00	18.64	0.00	0.00	0.00	42.09
-2.40	-1.29803	-6.02	-87.39	-69.56	excav.	elast.	0.00	19.31	0.00	0.00	0.00	46.07
-2.40	-1.29803	-6.02	-87.39	-69.56	excav.	elast.	0.00	19.31	0.00	0.00	0.00	46.07
-2.60	-1.32070	-6.28	-101.69	-73.51	excav.	elast.	0.00	20.25	0.00	0.00	0.00	50.04

Figure B146 : Display of results of a calculation phase in the form of a table

Each column recalls the physical value in the title, the application side (left or right) and the usual units defined for the project.

The columns are described below:

- **Depth or level** (m, ft): elevation/depth of calculation points for wall elements. The calculation step input in the "Title and options" window is a maximum spacing value between two successive points. K-Réa adjusts this spacing when necessary as a function of the soil interfaces and anchors (1 point for anchors and 2 points for soil layer interfaces).
- **Rotation** (rad): rotation (or distortion) of wall at calculation point.
- **Displacement** (mm, in): lateral displacement of wall at calculation point.
- $\mathbf{M_k}$ (kNm/ml, kip.ft/lft): characteristic value of bending moment in wall.
- $\mathbf{V_k}$ (kN/ml, kip/lft): characteristic value of shear force in wall.
- **Status**: gives the status of the soil at the nodes, with the following notation:
  - Excavation: the soil is excavated in front of the left or right hand side of the wall;
  - Unsticking: the soil and wall are no longer in contact (negative pressure replaced by minimum pressure, more details in part C of the manual);
  - Active earth pressure: the soil in contact with the wall is in active earth pressure state;
  - Elastic: the soil in contact with the wall is in elastic phase;
  - Passive earth pressure: the soil in contact with the wall is in passive earth pressure state.
- $\mathbf{p_{h,k}}$ (kN/m/ml, kip/lft): characteristic value of the mobilised effective horizontal pressure;

- u_k (kN/m/ml, kip/lft): characteristic value of the water pressure calculated as a function of the unit volume of water at the node considered;
- $\sigma'_{v,k}$ (kN/m/ml, kip/lft): characteristic value of the effective vertical stress at the point considered;
- $p_{a,k}$ (kN/m/ml, kip/lft): characteristic value of the effective active earth pressure (mobilisable active earth pressure);
- $p_{b,k}$ (kN/m/ml, kip/lft): characteristic value of the effective passive earth pressure (mobilisable passive earth pressure);
- p_k (kN/m/ml, kip/lft): characteristic value of the differential pressure, calculated as follows:

$$p_k = p_{h,k}^{right} - p_{h,k}^{left} + u_{,k}^{right} - u_{,k}^{left}$$

- $F_{v,k}$ (kN/m, kip): characteristic value of the of the arch pressure;
- $N_{,k}$ (kN/ml, kip/lft): characteristic value of the normal force;
- $p_{0,k}$ (kN/m/ml, kip/lft): initial pressure value.

B.6.3. Calculation with ULS checks (main results)

In the case of single walls with ULS checks, K-Réa is used to display the SLS results on the one hand and the ULS results on the other.

For each ULS result, the "k" index shows that this is a characteristic value while the "d" index shows that this is a "design" value.

B.6.3.1. Main window

In the main window, only the ULS calculation results are displayed. The display of the results differs according to whether or not the wall is anchored in the phase considered.

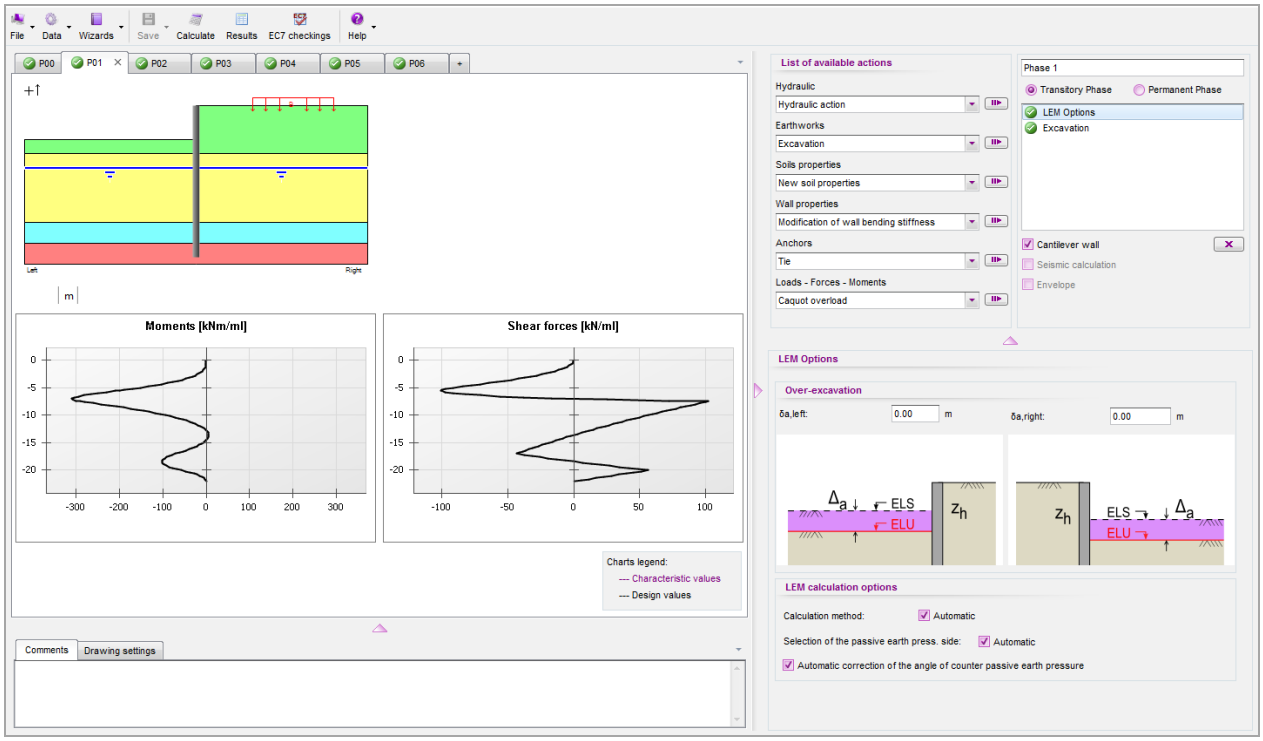


Figure B147 : ULS results of a phase in which the wall is cantilever (LEM calculation) – The displacements are not displayed

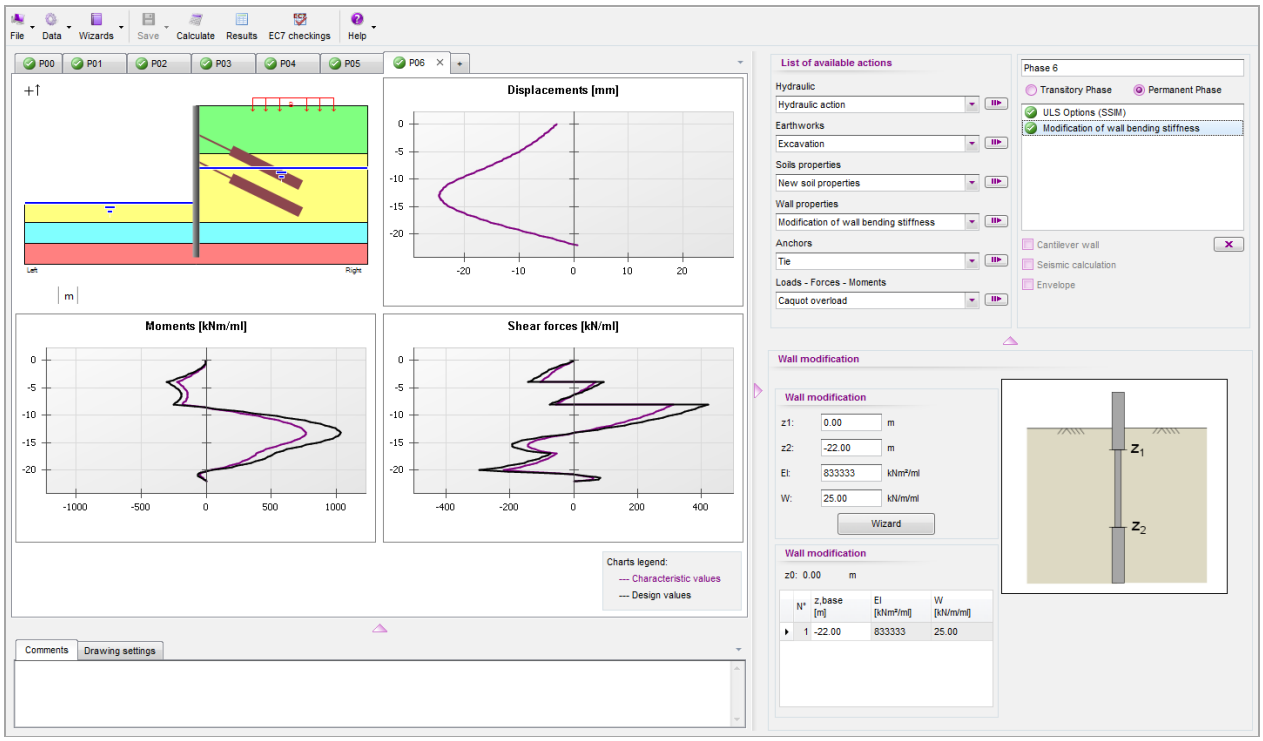


Figure B148 : ULS results of a phase in which the wall is anchored (SSIM calculation)

In the upper part of the window, K-Réa enables the user to switch between the SLS results and ULS results at any moment (whether for the "phases", "results synthesis" or "envelopes" tabs).

In addition, when the display requested is that of the ULS results, 3 additional buttons appear and give access to the results of the ULS checks (see Part C of the manual).

B.6.3.2. SLS results per phase

The results of a SLS calculation are the same as those of a basic SSIM calculation. The contents of chapters B.6.2.2 to B.6.2.5 therefore remain valid.

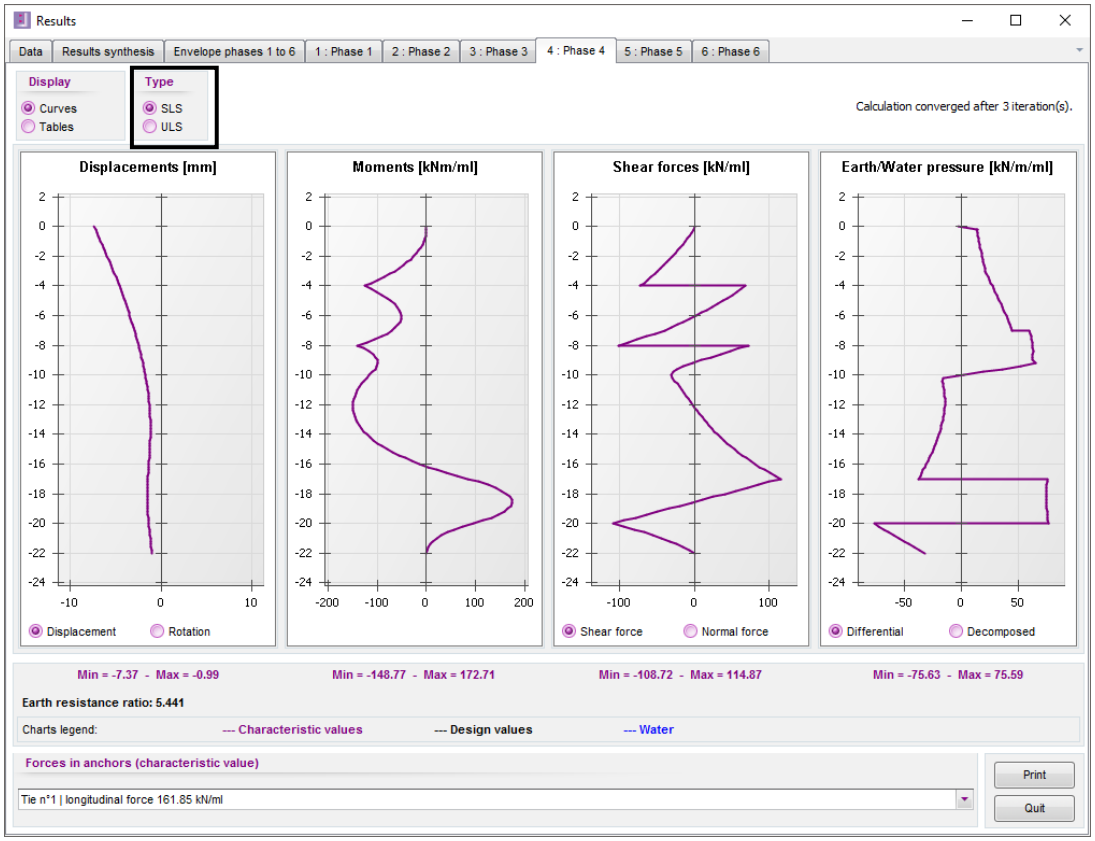


Figure B149 : Display of SLS results in the results window

B.6.3.3. ULS results per phase: LEM calculation (cantilever wall)

In this case, the LEM calculation performed gives the following results on the curves and in the tables (see Figure B 150 and Figure B 151):

- only the design values (d index) of the bending moments and shear forces are available,
- the results in terms of displacements are not displayed (neither on the curves nor in the tables), because this is a limit equilibrium calculation.

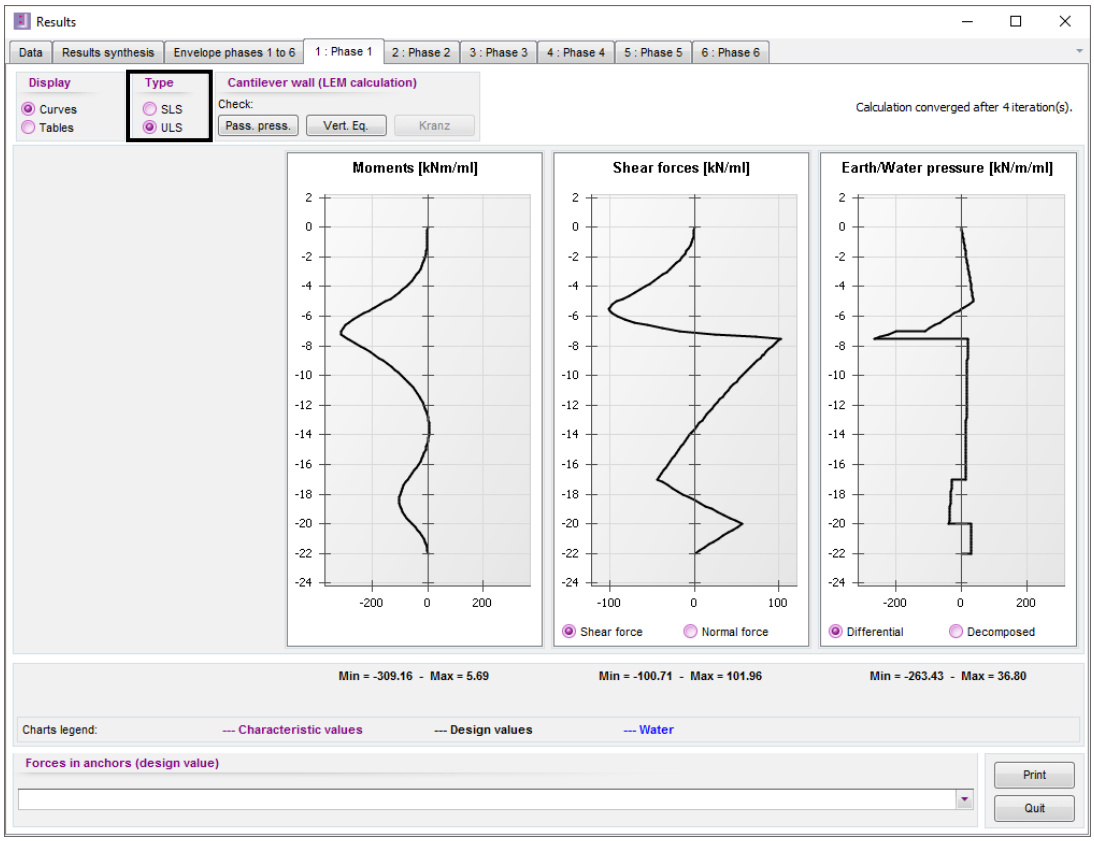


Figure B150 : Results window – ULS results (LEM) - Curves

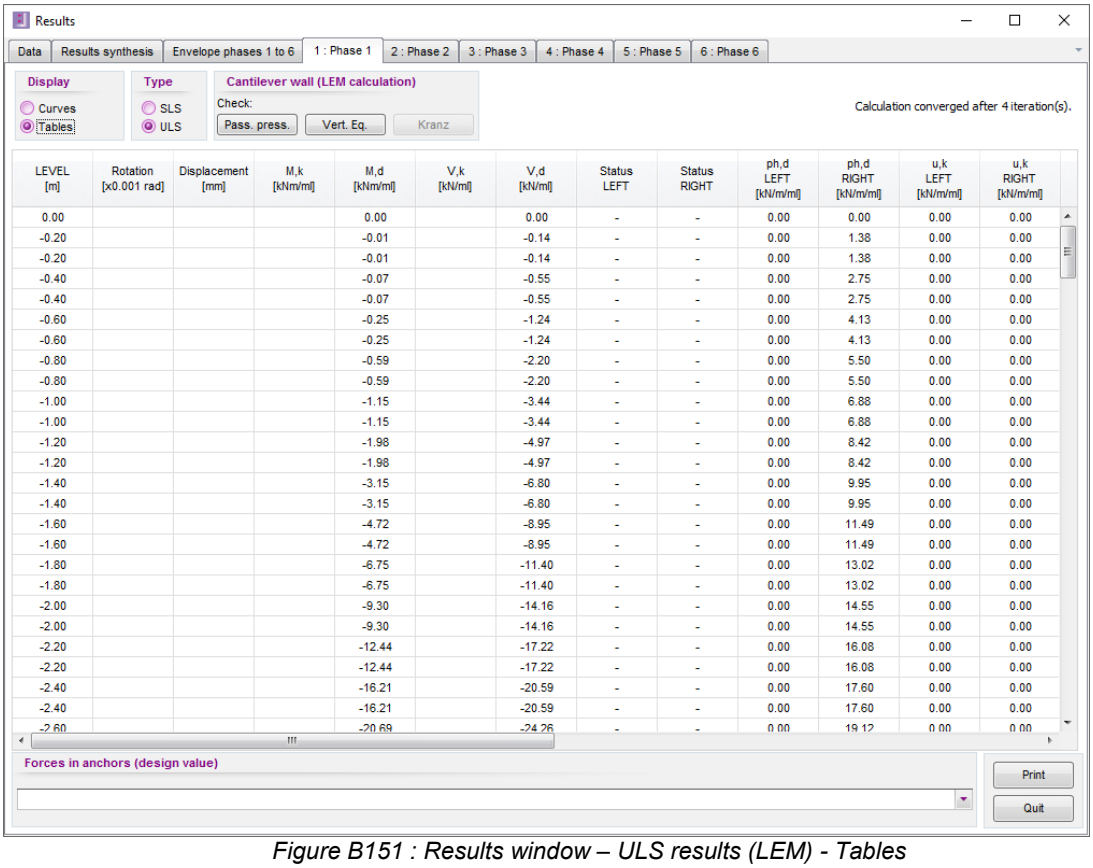
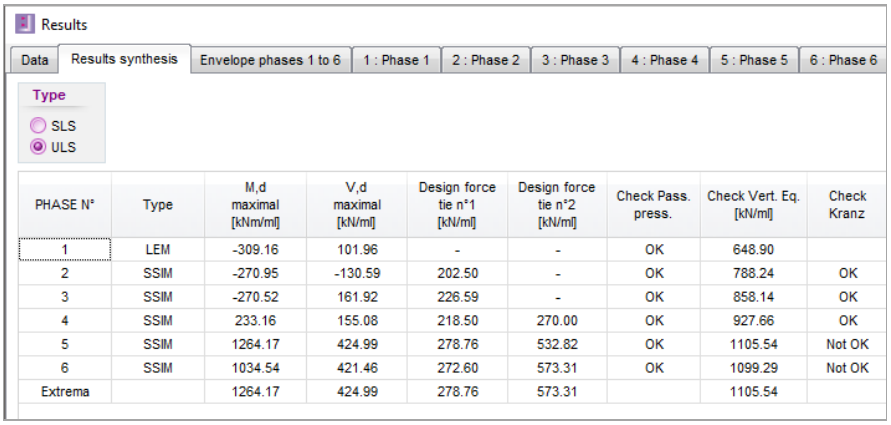


Figure B151 : Results window – ULS results (LEM) - Tables

In addition, extra columns appear in the **Results synthesis** tab (see Figure B 152):

- **Type:** indicates the type of calculation performed (LEM or SSIM);
- **Check pass. press.:** indicates the passive earth pressure safety check results;
- **Check Vert Eq.:** indicates the vertical resultant calculated when checking vertical equilibrium (positive value if the vertical forces resultant is directed downwards).


Results

Data | Results synthesis | Envelope phases 1 to 6 | 1 : Phase 1 | 2 : Phase 2 | 3 : Phase 3 | 4 : Phase 4 | 5 : Phase 5 | 6 : Phase 6

Type
SLS
ULS

PHASE N°	Type	M,d maximal [kNm/ml]	V,d maximal [kN/ml]	Design force tie n°1 [kN/ml]	Design force tie n°2 [kN/ml]	Check Pass. press.	Check Vert. Eq. [kN/ml]	Check Kranz
1	LEM	-309.16	101.96	-	-	OK	648.90	
2	SSIM	-270.95	-130.59	202.50	-	OK	788.24	OK
3	SSIM	-270.52	161.92	226.59	-	OK	858.14	OK
4	SSIM	233.16	155.08	218.50	270.00	OK	927.66	OK
5	SSIM	1264.17	424.99	278.76	532.82	OK	1105.54	Not OK
6	SSIM	1034.54	421.46	272.60	573.31	OK	1099.29	Not OK
Extrema		1264.17	424.99	278.76	573.31		1105.54	

Figure B152 : Results window – ULS results synthesis for cantilever phases only

For the rest, the content of chapters B.6.2.2 to B.6.2.5 remains valid.

B.6.3.4. ULS results per phase: SSIM calculation (anchored wall)

In this case, the SSIM calculation gives the following results:

- the curves and tables presenting the results in characteristic values (k index) and in design values (d index) (Figure B 153 and Figure B 154);
- a key is given under the curves to distinguish between the results in characteristic values (purple curves) and the results in design values (black curves).

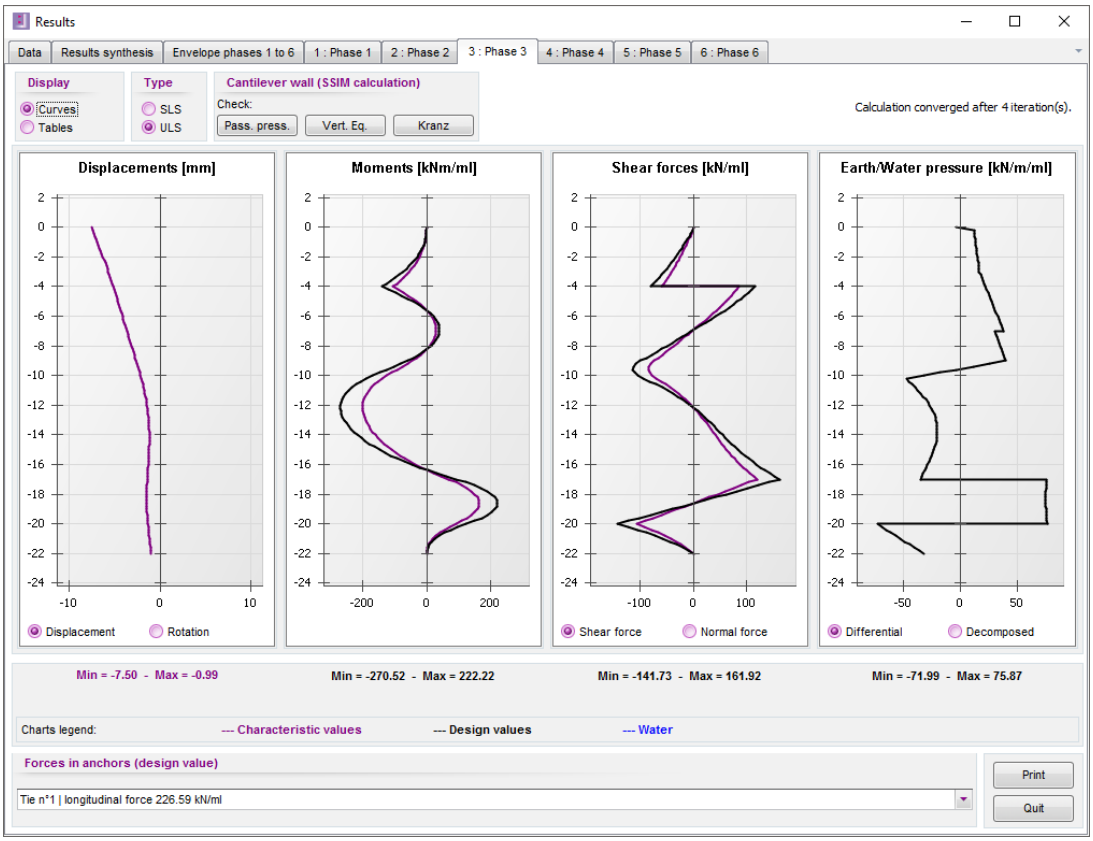
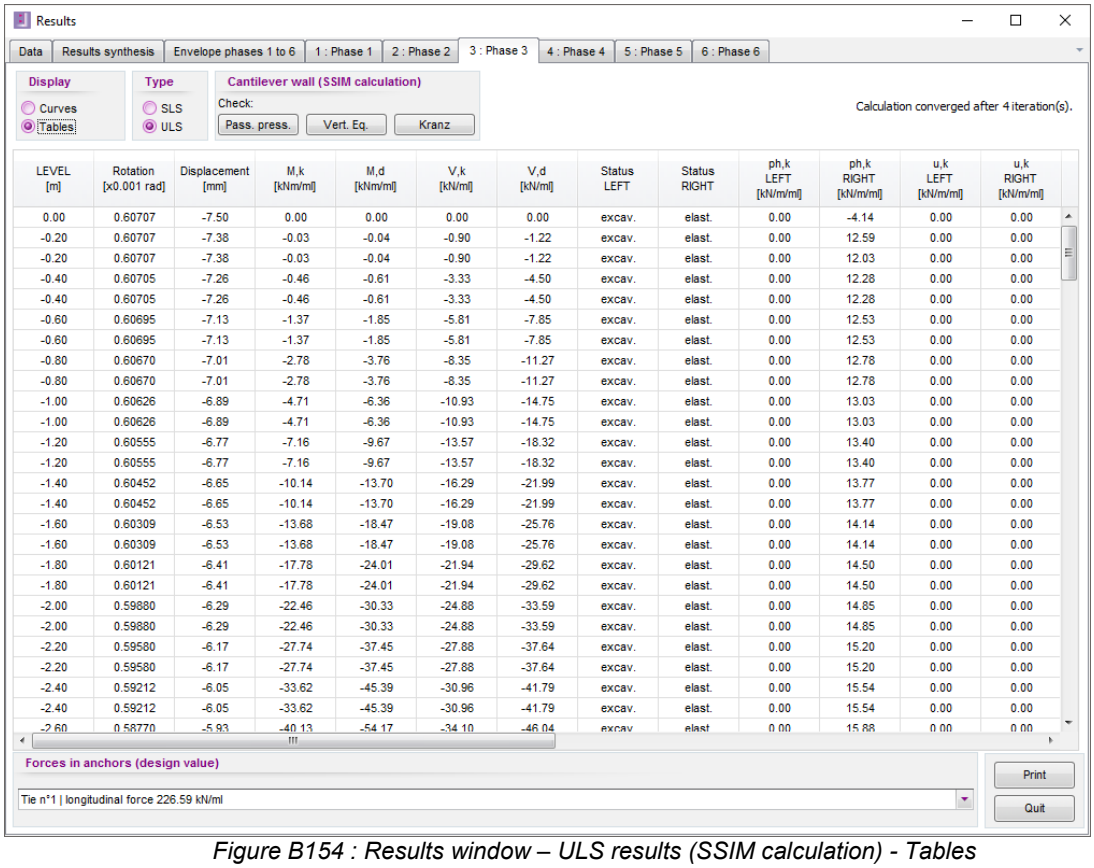


Figure B153 : Results window – ULS results (SSIM calculation) - Curves

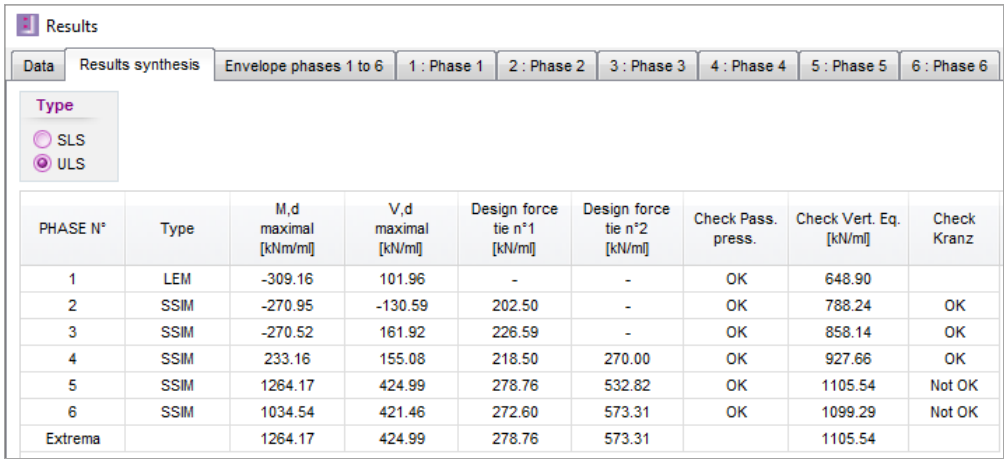


LEVEL [m]	Rotation [x0.001 rad]	Displacement [mm]	M,k [kNm/ml]	M,d [kNm/ml]	V,k [kN/ml]	V,d [kN/ml]	Status LEFT	Status RIGHT	ph,k LEFT [kN/m/ml]	ph,k RIGHT [kN/m/ml]	u,k LEFT [kN/m/ml]	u,k RIGHT [kN/m/ml]
0.00	0.60707	-7.50	0.00	0.00	0.00	0.00	excav.	elast.	0.00	-4.14	0.00	0.00
-0.20	0.60707	-7.38	-0.03	-0.04	-0.90	-1.22	excav.	elast.	0.00	12.59	0.00	0.00
-0.20	0.60707	-7.38	-0.03	-0.04	-0.90	-1.22	excav.	elast.	0.00	12.03	0.00	0.00
-0.40	0.60705	-7.26	-0.46	-0.61	-3.33	-4.50	excav.	elast.	0.00	12.28	0.00	0.00
-0.40	0.60705	-7.26	-0.46	-0.61	-3.33	-4.50	excav.	elast.	0.00	12.28	0.00	0.00
-0.60	0.60695	-7.13	-1.37	-1.85	-5.81	-7.85	excav.	elast.	0.00	12.53	0.00	0.00
-0.60	0.60695	-7.13	-1.37	-1.85	-5.81	-7.85	excav.	elast.	0.00	12.53	0.00	0.00
-0.80	0.60670	-7.01	-2.78	-3.76	-8.35	-11.27	excav.	elast.	0.00	12.78	0.00	0.00
-0.80	0.60670	-7.01	-2.78	-3.76	-8.35	-11.27	excav.	elast.	0.00	12.78	0.00	0.00
-1.00	0.60626	-6.89	-4.71	-6.36	-10.93	-14.75	excav.	elast.	0.00	13.03	0.00	0.00
-1.00	0.60626	-6.89	-4.71	-6.36	-10.93	-14.75	excav.	elast.	0.00	13.03	0.00	0.00
-1.20	0.60555	-6.77	-7.16	-9.67	-13.57	-18.32	excav.	elast.	0.00	13.40	0.00	0.00
-1.20	0.60555	-6.77	-7.16	-9.67	-13.57	-18.32	excav.	elast.	0.00	13.40	0.00	0.00
-1.40	0.60452	-6.65	-10.14	-13.70	-16.29	-21.99	excav.	elast.	0.00	13.77	0.00	0.00
-1.40	0.60452	-6.65	-10.14	-13.70	-16.29	-21.99	excav.	elast.	0.00	13.77	0.00	0.00
-1.60	0.60309	-6.53	-13.68	-18.47	-19.08	-25.76	excav.	elast.	0.00	14.14	0.00	0.00
-1.60	0.60309	-6.53	-13.68	-18.47	-19.08	-25.76	excav.	elast.	0.00	14.14	0.00	0.00
-1.80	0.60121	-6.41	-17.78	-24.01	-21.94	-29.62	excav.	elast.	0.00	14.50	0.00	0.00
-1.80	0.60121	-6.41	-17.78	-24.01	-21.94	-29.62	excav.	elast.	0.00	14.50	0.00	0.00
-2.00	0.59880	-6.29	-22.46	-30.33	-24.88	-33.59	excav.	elast.	0.00	14.85	0.00	0.00
-2.00	0.59880	-6.29	-22.46	-30.33	-24.88	-33.59	excav.	elast.	0.00	14.85	0.00	0.00
-2.20	0.59580	-6.17	-27.74	-37.45	-27.88	-37.64	excav.	elast.	0.00	15.20	0.00	0.00
-2.20	0.59580	-6.17	-27.74	-37.45	-27.88	-37.64	excav.	elast.	0.00	15.20	0.00	0.00
-2.40	0.59212	-6.05	-33.62	-45.39	-30.96	-41.79	excav.	elast.	0.00	15.54	0.00	0.00
-2.40	0.59212	-6.05	-33.62	-45.39	-30.96	-41.79	excav.	elast.	0.00	15.54	0.00	0.00
-2.60	0.58770	-5.93	-40.13	-54.17	-34.10	-46.04	excav.	elast.	0.00	15.88	0.00	0.00

Figure B154 : Results window – ULS results (SSIM calculation) - Tables

Additional columns also appear in the **Results synthesis** tab (see B.6.2.3):

- **Type:** indicates the type of calculation performed (LEM or SSIM);
- **Check pass. press.:** indicates the result of the passive earth pressure safety check;
- **Check Vert. Eq.:** indicates the resultant calculated during the vertical check (positive value if the vertical forces resultant is directed downwards);
- **Check Kranz:** indicates the result of the Kranz check.



PHASE N°	Type	M,d maximal [kNm/ml]	V,d maximal [kN/ml]	Design force tie n°1 [kN/ml]	Design force tie n°2 [kN/ml]	Check Pass. press.	Check Vert. Eq. [kN/ml]	Check Kranz
1	LEM	-309.16	101.96	-	-	OK	648.90	
2	SSIM	-270.95	-130.59	202.50	-	OK	788.24	OK
3	SSIM	-270.52	161.92	226.59	-	OK	858.14	OK
4	SSIM	233.16	155.08	218.50	270.00	OK	927.66	OK
5	SSIM	1264.17	424.99	278.76	532.82	OK	1105.54	Not OK
6	SSIM	1034.54	421.46	272.60	573.31	OK	1099.29	Not OK
Extrema		1264.17	424.99	278.76	573.31		1105.54	

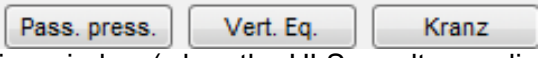
Figure B155 : Results window – ULS results summary for a project with cantilever and anchored phases

For the rest, the contents of chapters B.6.2.2 to B.6.2.5 remain valid.

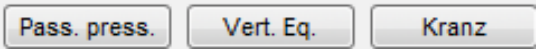
B.6.4. ULS checks

When the ULS checks have been activated for a project, K-Réa provides the results of three types of ULS checks performed for each phase defined in the project.

These results are displayed in a specific window, which can be accessed in two ways:

- By clicking directly on the "EC7 checks button  .
- By clicking one of the 

buttons from the detailed results presentation window (when the ULS results are displayed).

The specific ELU checks results window (Figure B 156) then opens and by default is positioned on the phase currently selected before the results display request: either the phase displayed in the main window in the case of access by the  button, or the phase displayed in the results window in the case of access from one of the 

buttons.

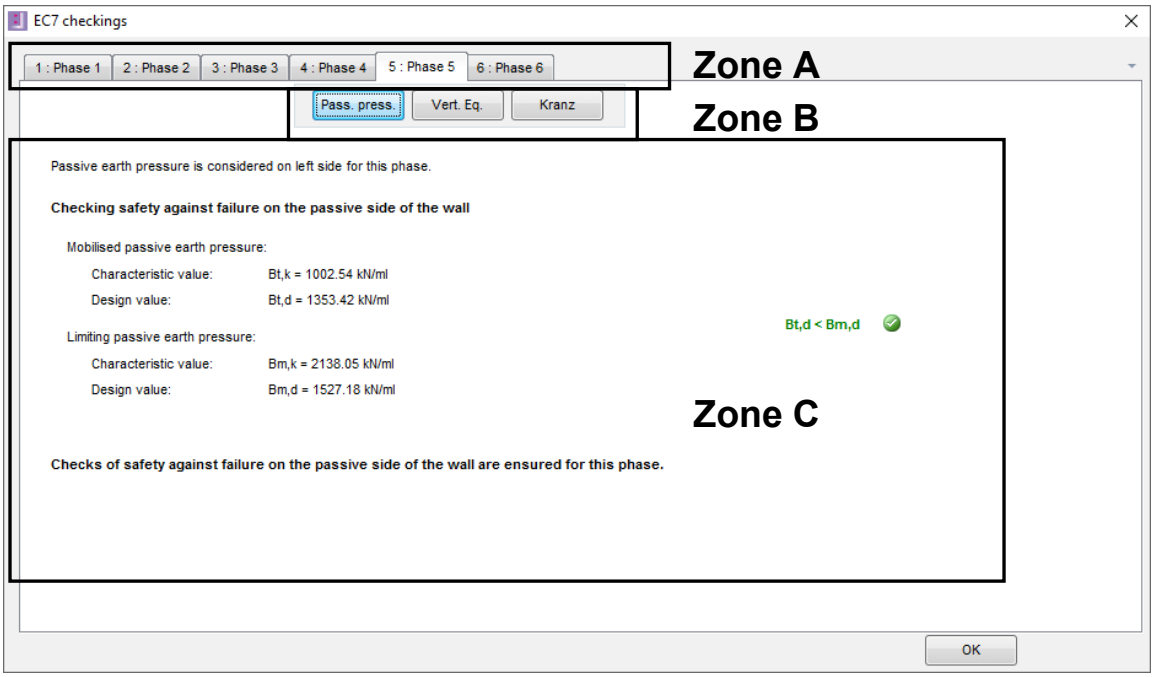


Figure B156 : ULS checks results display window

The check results display window contains 3 zones:

- **Zone A:** corresponds to the tabs used to select the phase for which the checks are displayed.
- **Zone B:** corresponds to the tabs used to select the ULS check for which one wishes to display the results:
  - The passive earth pressure safety check;
  - The vertical equilibrium check;
  - The check on the stability of the anchoring block via the simplified **Kranz method** (only available if at least one anchor is active in the selected phase).
- **Zone C:** requested results display zone.

The following sub-chapters specify the various results displayed for each type of check. These results and their notations refer to part C of the manual for details of the calculation methods applied (in accordance with French standard NF P94-282).

B.6.4.1. Passive earth pressure safety check

B.6.4.1.1. Case of an anchored wall (SSIM calculation)

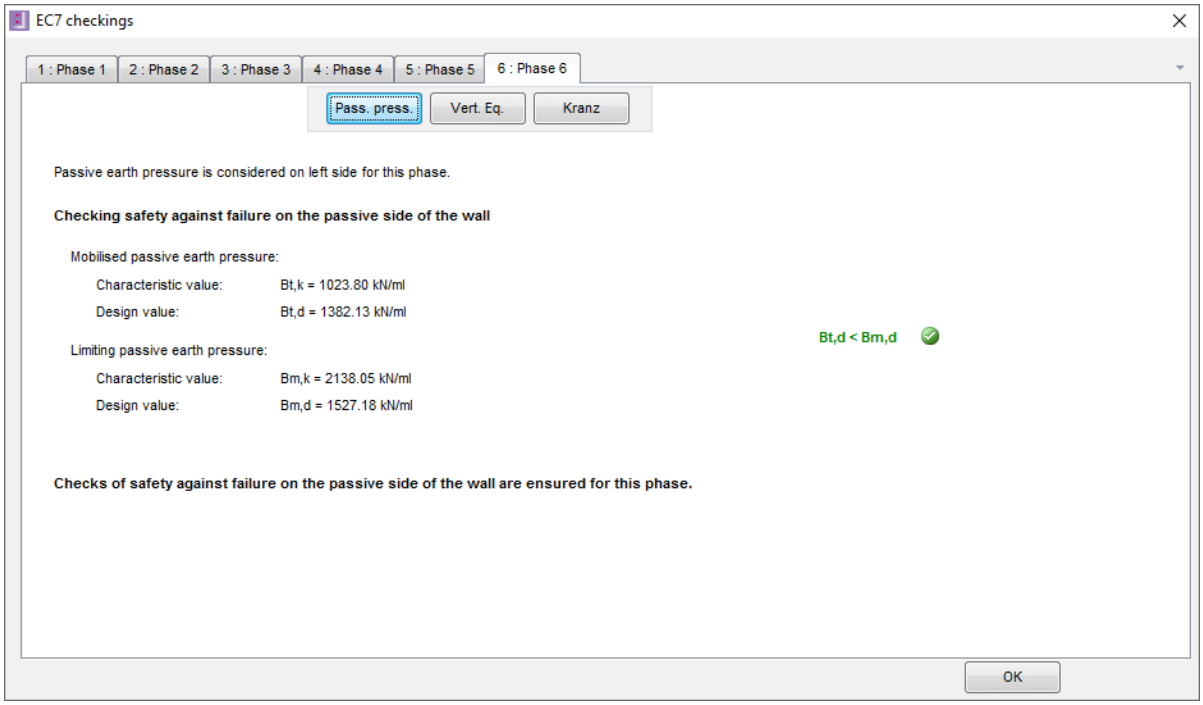


Figure B157 : ULS check – Results of passive earth pressure safety check – Anchored phase (SSIM calculation)

In the case of an anchored wall, the passive earth pressure safety check is based on the evaluation of the following parameters:

- **$B_{t,k}$:** characteristic value of mobilised passive earth pressure resultant (in kN or kp);
- **$B_{t,d}$:** calculation value of mobilised passive earth pressure resultant (in kN or kip);
- **$B_{m,k}$:** characteristic value of mobilisable passive earth pressure resultant (in kN or kip);
- **$B_{m,d}$:** calculation value of mobilisable passive earth pressure resultant (in kN or kip).

K-Réa compares the value of $B_{t,d}$ with that of $B_{m,d}$ and an indicator shows the result of the check for the selected phase: green circle if the mobilised passive earth pressure $B_{t,d}$ is less than the mobilisable passive earth pressure $B_{m,d}$, or red one otherwise.

For more details on this check, refer to part C of the manual.

B.6.4.1.2. Case of a cantilever wall (LEM calculation)

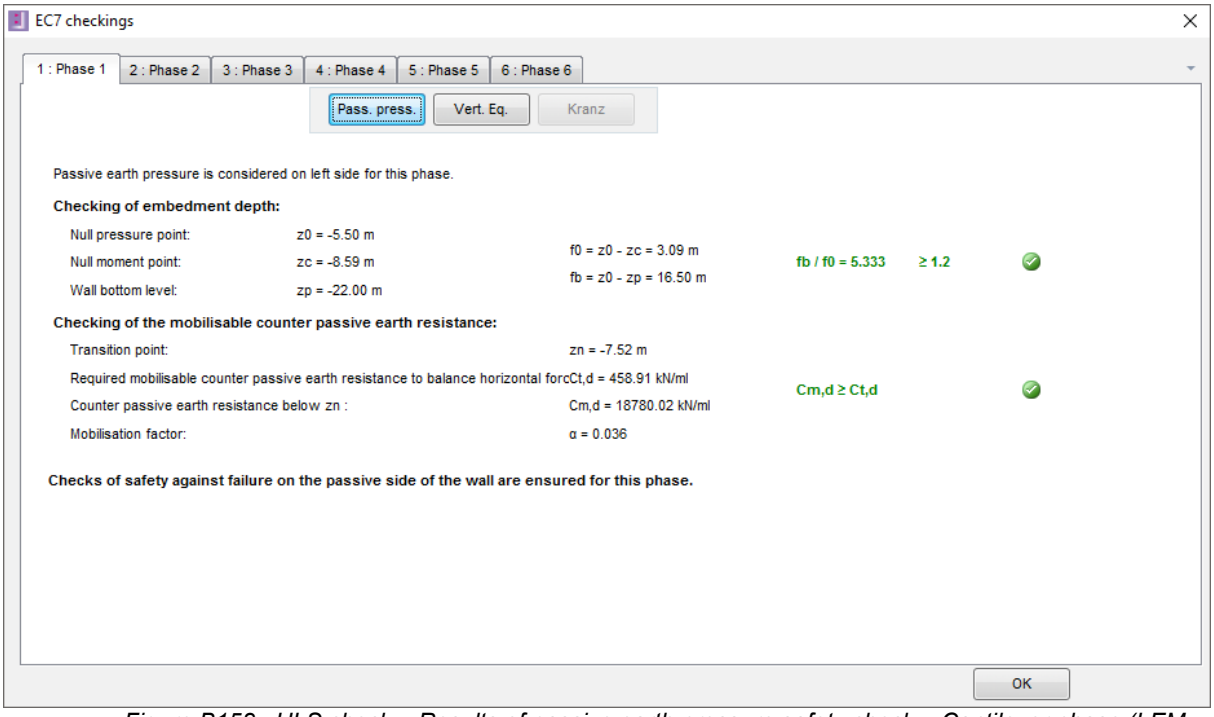


Figure B158 : ULS check – Results of passive earth pressure safety check – Cantilever phase (LEM calculation)

In the case of an anchored wall, the passive earth pressure safety check is based on a LEM type calculation (limit equilibrium model) which involves the following intermediate parameters:

- **z_0:** elevation/depth of null differential pressure point (in m or ft);
- **z_c:** elevation/depth of null moment point (in m or ft);
- **z_p:** elevation/depth of base of wall (in m or ft);
- **f_0:** "available" wall embedment depth under z_0 (in m or ft);
- **f_b:** minimum embedment depth, under z_0, needed to obtain equilibrium of moments (in m or ft);
- **f_0/f_b:** ratio between the two previously calculated embedment depths (no units);
- Embedment depth height verification indicator: this indicator is green if the verification is positive (available embedment depth greater than minimum embedment depth with a safety coefficient greater than 1.2); it is red otherwise.
- **z_n** (only if method D was chosen; otherwise point z_n is implicitly the same as point z_c)**:** elevation/depth of transition point (in m or ft);
- **$C_{t,d}$:** value of calculation of resultant of counter passive earth pressure necessary for equilibrium of the horizontal forces (in kN/ml or kip/lft);
- **$C_{m,d}$:** value of calculation of resultant of counter passive earth pressure mobilisable under the transition point (in kN/ml or kip/lft);
- **α:** mobilisation factor (function of ratio $C_{t,d}$ / $C_{m,d}$, see Part C);
- **Counter passive earth pressure verification indicator:** this indicator is green if the verification is positive (value of α is 1 or less); it is red otherwise.

For more details on this verification, please refer to part C of the manual.

B.6.4.2. Verification of vertical equilibrium of wall

B.6.4.2.1. Case of anchored wall (SSIM calculation)

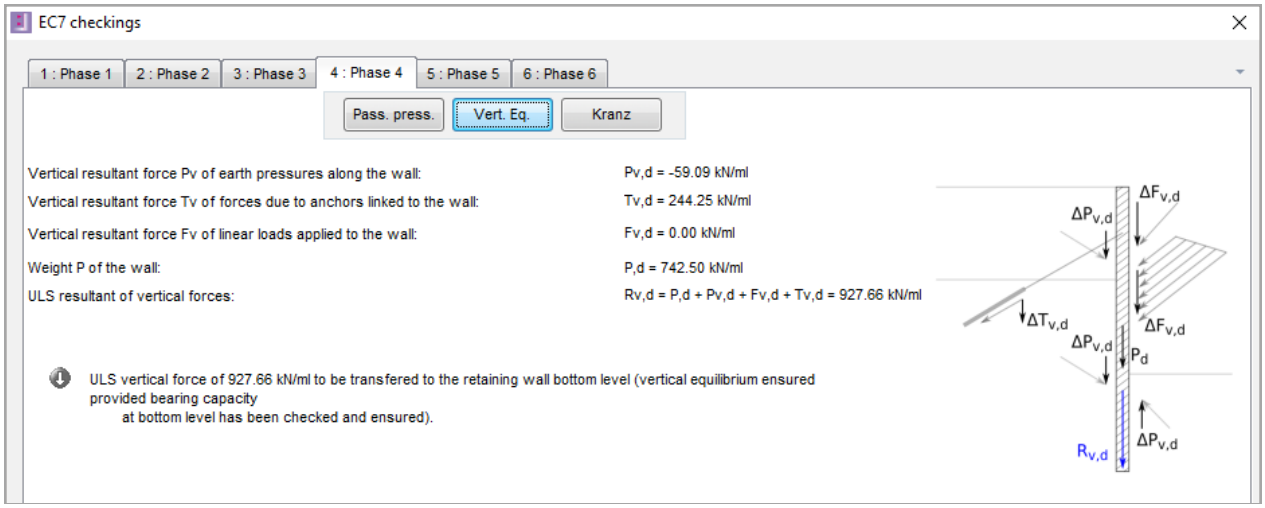


Figure B159 : ULS check– Vertical equilibrium results – Anchored wall (SSIM calculation)

In the case of an anchored wall, the vertical equilibrium check involves the following parameters:

- **$P_{v,d}$:** calculation value of vertical resultant of earth pressures over the height of the wall (in kN/ml or kip/lft);
- **$T_{v,d}$:** calculation value of vertical resultant of forces due to anchors connected to the wall (in kN/ml or kip/lft);
- **$F_{v,d}$:** calculation value of vertical resultant of linear overloads applied over the height of the wall (in kN/ml or kip/lft);
- **P_d:** calculation value of own weight of wall (in kN/ml or kip/lft);
- **$R_{v,d}$:** calculation value of resultant of vertical forces at ULS (in kN/ml or kip/lft). A symbol at the bottom of the window indicates if this resultant is directed upwards or downwards.

The verification of vertical equilibrium of the wall is considered to be satisfactory when the resultant of the vertical forces is positive, it is then by convention directed "downwards".

To the left of the conclusion, a grey icon with a down arrow tells the user that the resultant of the vertical forces is positive and directed "downwards".

If not, a red icon with an upwards arrow informs the user that the resultant of the vertical forces is negative and directed "upwards". The concluding sentence in this case will also be written in red.

The user must check that the wall base bearing is guaranteed taking account of the value obtained for $R_{v,d}$.

For more details concerning this verification, please refer to part C of the manual.

B.6.4.2.2. Case of a cantilever wall (LEM calculation)

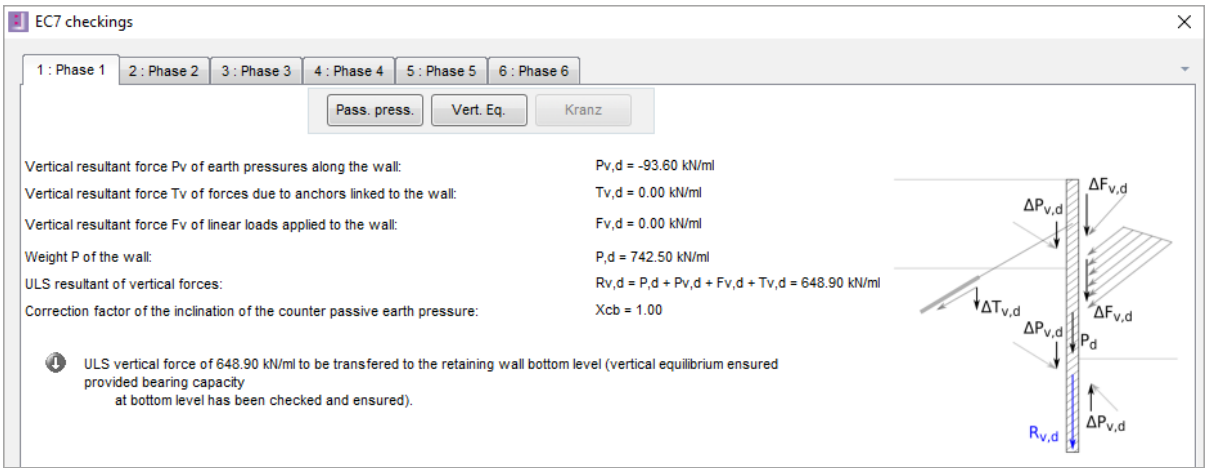


Figure B160 : ULS check– Vertical equilibrium results – Cantilever wall (LEM calculation)

For the phases in which the wall is considered to be cantilever, the vertical equilibrium check involves the following parameters:

- **$P_{v,d}$:** calculation value of vertical resultant of earth pressures over the height of the wall (in kN/ml or kip/lft);
- **$T_{v,d}$:** calculation value of vertical resultant of forces due to anchors connected to the wall (in kN/ml or kip/lft);
- **$F_{v,d}$:** calculation value of vertical resultant of linear overloads applied to the height of the wall (in kN/ml or kip/lft);
- **P_d:** calculation value of own weight of wall (in kN/ml or kip/lft);
- **$R_{v,d}$:** calculation value of resultant of vertical forces (in kN/ml or kip/lft). A symbol at the bottom of the window indicates whether this resultant is directed upwards or downwards.
- **X_{cb}:** counter passive earth pressures angle correction factor. This factor, determined automatically by K-Réa, acts on the angle of the passive earth pressure initially defined to correct the angle of the counter passive earth pressure, such as to obtain a vertical resultant ($R_{v,d}$) that is directed downwards:

$$(\delta/\varphi)_{\text{counter-passive earth pressure}} = X_{cb} \times (\delta/\varphi)_{\text{passive earth pressure}}$$

The verification of vertical equilibrium is considered to be satisfactory when the resultant of the vertical forces is positive, it then being directed "downwards" by convention. To the left of the conclusion, a grey icon with a down arrow informs the user that the resultant of the vertical forces is positive and directed "downwards". Otherwise, the icon is red with an up arrow informing the user that the resultant of the vertical forces is negative and directed "upwards". The concluding sentence in this case will also be written in red.

The user shall ensure that the wall base bearing is guaranteed taking account of the value obtained for $R_{v,d}$. For more details concerning this check, please refer to part C of the manual.

B.6.4.3. Verification of stability of the anchoring block (Kranz)

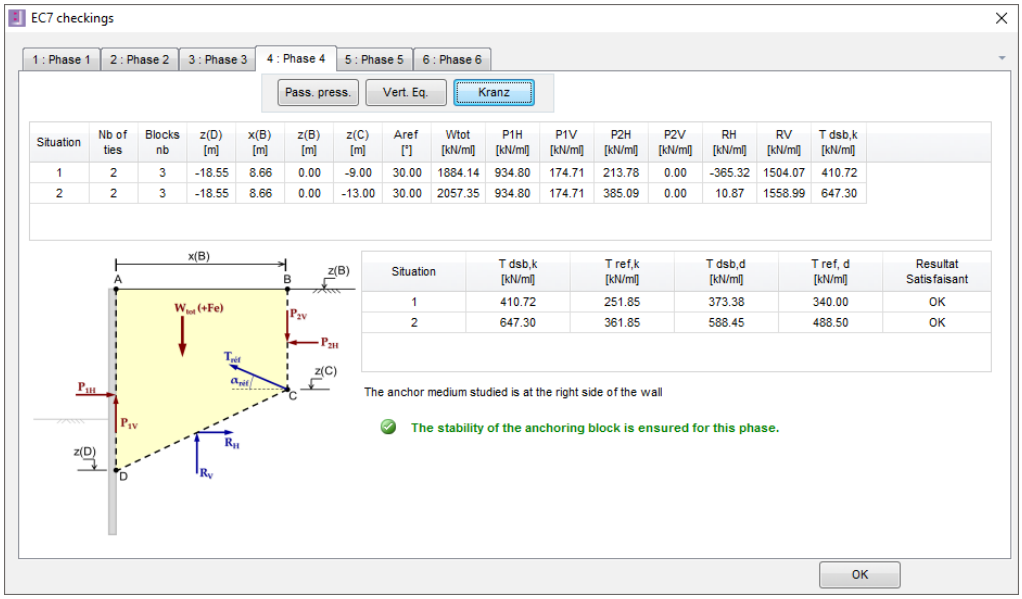


Figure B161 : ULS checks – Kranz check

The check on the anchoring block stability (Kranz) is only available for the phases in which at least one tie has been defined.

The first table gives the intermediate calculation results:

- **Situation:** situation number (the situation number corresponds to the number of active anchors in the phase concerned);
- **Spiral angle:** angle at pole of spiral (°);
- **Nb of ties:** number of ties considered in each situation;
- **Blocks nb:** number of blocks defined during discretisation of the anchoring block (as a function of the number of layers intersected along the base of the anchoring block);
- **z(D):** elevation/depth of null shear force point (in m or ft);
- **x(B):** distance between vertical projection of point C and the head of the wall (in m or ft);
- **z(B):** elevation/depth of soil (in m or ft);
- **z(C):** elevation/depth of effective anchor point of tie (in m or ft), corresponding to the useful length L_u defined for the anchor;
- **A_{ref}:** angle of anchor with respect to the horizontal (in °);
- **W_{tot}:** total weight of block for the situation considered (in kN/ml or kip/lft);
- **P_{1H}:** horizontal component of wall reaction on the anchoring block (in kN/ml or kip/lft);
- **P_{1V}:** vertical component of wall reaction on the anchoring block (in kN/ml or kip/lft);
- **P_{2H}:** horizontal component of the active earth pressure force exerted upstream of the anchoring block (in kN/ml or kip/lft);
- **P_{2V}:** vertical component of the active earth pressure force exerted upstream of the anchoring block (in kN/ml or kip/lft);
- **R_H:** horizontal component of the soil reaction under the anchoring block (in kN/ml or kip/lft);
- **R_V:** vertical component of the soil reaction under the anchoring block (in kN/ml or kip/lft);
- **$T_{dsb,k}$:** characteristic value of the destabilising anchor force (in kN/ml or kip/lft).

The second table gives the results of the check:

- **$T_{dsb,k}$:** characteristic value of the destabilising anchor force (in kN/ml or kip/lft), this value is identical to that of the last column in the previous table;
- **$T_{ref,k}$:** characteristic value of reference anchor force resulting from the ULS SSIM calculation (in kN/ml or kip/lft);
- **$T_{dsb,d}$:** calculation value of destabilising anchor force (in kN/ml or kip/lft);
- **$T_{ref,d}$:** calculation value of reference anchor force (in kN/ml or kip/lft).
- **"OK"** (or "**non OK"**): for each situation, the last column shows whether the check is satisfactory, that is if $T_{ref,d}$ is less than $T_{dsb,d}$.

In the lower part of the window, an indicator specifies whether the Kranz check is satisfactory for all the situations studied (the check is only satisfactory overall if it is satisfactory for each calculation situation): if this is the case, the indicator is a green circle.

For more details concerning this check, please refer to part C of the manual.

B.6.5. Double wall project

B.6.5.1. Main results

For each wall, the results presented are the same as for a single wall project without ULS checks.

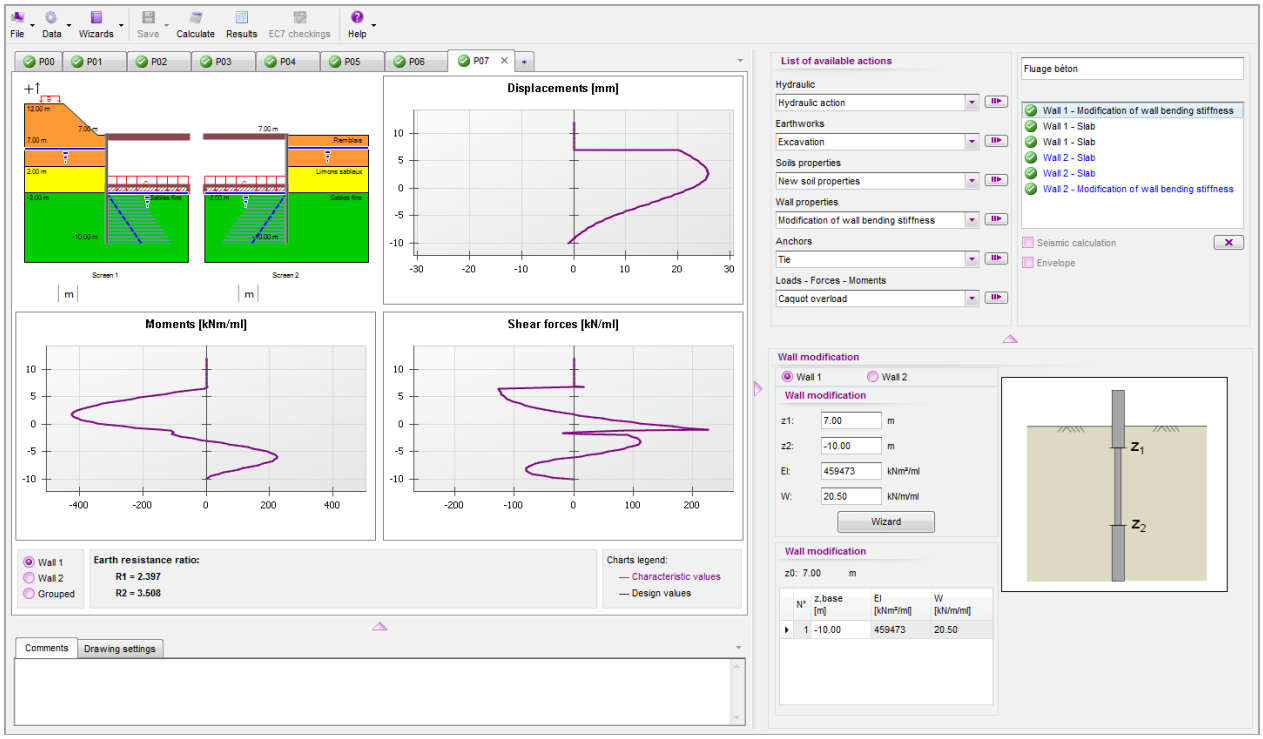


Figure B162 : Main window – Results of a double wall project

It is also possible to choose simultaneous (superposed) display of the results curves of both walls, by clicking the "**Grouped**" choice (in the list of choices under the moments curve). In this case, the curves on the active screen appear in thick purple, while those of the other wall appear in thin purple. The passive earth pressures ratio obtained is also displayed for each wall (see Figure B 163).

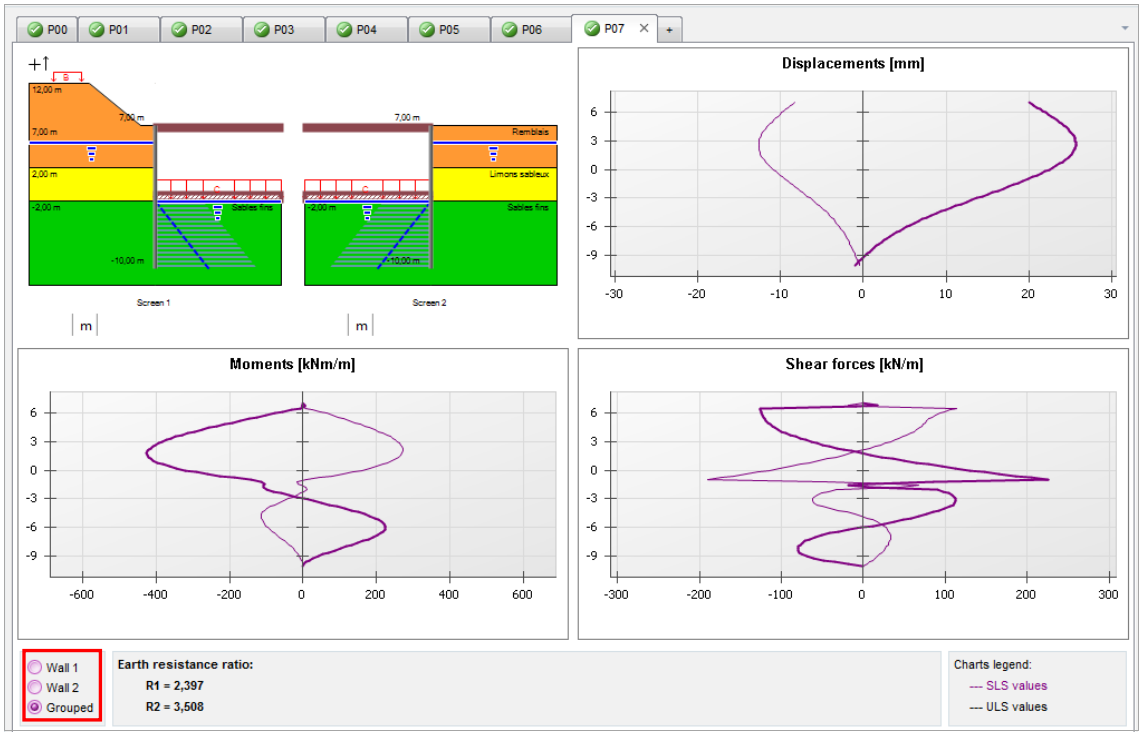


Figure B163 : Main window – Results of a double wall project

As with the single wall projects, the detailed results are accessible via the "Results" button.

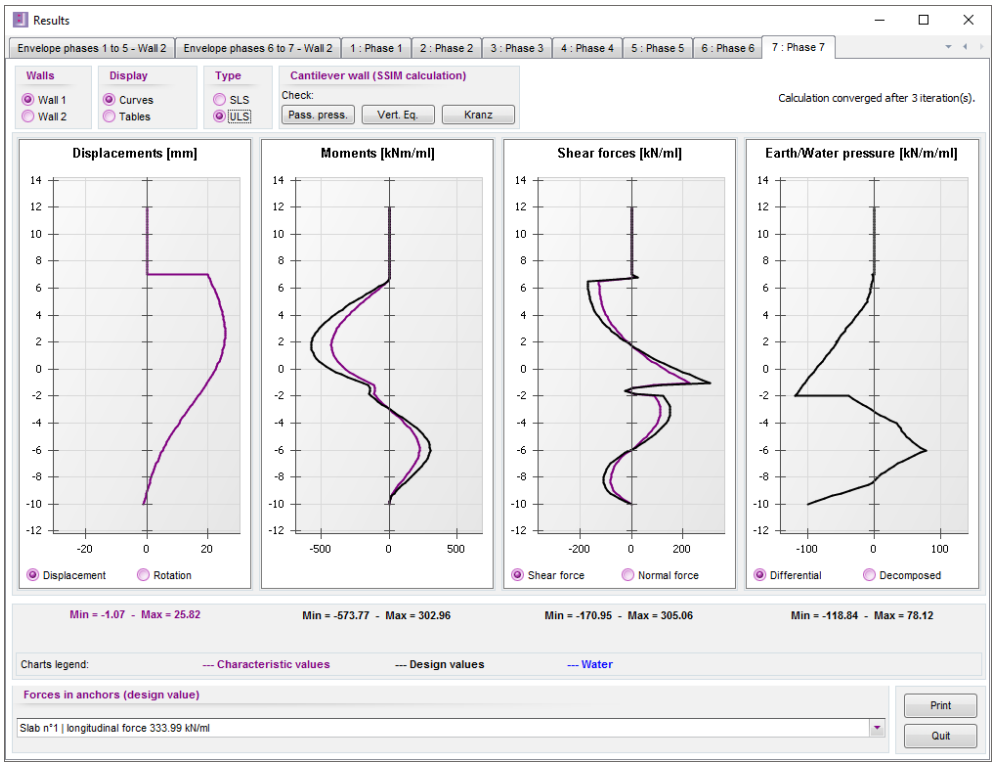


Figure B164 : Results window – Case of a double wall project

The results given for each wall are as specified in chapter B.6.2 (results of a single wall calculation without ULS checks).

At any moment it is possible to go from the results of one wall to those of the other via the selection buttons in the results window (Data, Phases, Results Synthesis and Envelopes).

B.6.5.2. ULS checks

K-Réa is used to carry out ULS checks for double wall type projects if the "ULS checks" box was activated in the "Title and Options" window.

The passive earth pressure safety checks for each wall are performed and their presentation is comparable to that of a single wall.

The use of the Kranz check is broadened by allowing validation of the distance between walls, while verifying the stability of everything between the two walls on the basis of the forces behind each wall. For more details, please refer to part C of the manual.

B.6.6. Export des résultats sous format de fichiers texte

K-Réa can export the results presented in the tables and graphs as a text file (.txt). This feature allows the coupling of K-Réa with other post-calculation tools. The generated files are exported in the directory selected by the user.

The set of generated text files are summarized in the following table:

File	Description
01-KR_ELS_Wall	SLS Results
02-KR_ELS_Reactions	SLS Results: forces in the supports
03-KR_ELU_MISS_Wall	ULS Results – MISS
04-KR_ELU_Reactions	ULS Results: forces in the supports
05-KR_ELU_MEL_F_Wall	ULS Result – MEL F
06-KR_ELU_MEL_D_Wall	ULS Results – MEL D
07-KR_ELU_MISS_Synthesis.txt	Main results of the passive earth pressure and vertical balance checks at ULS
08-KR-KRANZ-Results.txt	Kranz verification results for plane surfaces
09-KR-Phase_And_Combinations.txt	Relationship between the successive calculation phases calculated by the calculation kernel, the combination to which they belong and the index of the calculation phase on the interface

B.6.6.1. Reading the file 01-KR_ELS_Wall

This file contains the values of displacements, internal forces and various pressures of the soil at every level of the wall (2 values per element) resulting from SLS calculation.

Header of the file (only once at the beginning of the file)

Pour k=1, nEcr loop on the walls
- **nLignes(k)** number of lines in the result block of each wall

Rest of file:

For k=1, nEcr loop on the walls
For i=1, nEl(k) loop on the elements of each wall
For j=1,2 two nodes of each element (top and bottom)

1) iPhase index of the phase
2) k wall index
3) I element index
4) j =1 top node / =2 bottom node
5) Z node level
6) ry rotation
7) wx displacement
8) M bending moment
9) V shear effort
10) EtatG soil status on the left (-2=exc. / -1= detach. / 0=active / 1=elastic / 2=passive)
11) EtatD soil status on the right (-2=exc. / -1= detach. / 0=active / 1=elastic / 2=passive)
12) phG horizontal pressure mobilized on the left side
13) phD horizontal pressure mobilized on the right side
14) pwG hydrostatic water pressure on the left side
15) pwD hydrostatic water pressure on the right side
16) sigG effective vertical stress on the left side
17) sigD effective vertical stress on the right side
18) paG active soil pressure on the left side
19) paD active soil pressure on the right side
20) pbG passive soil pressure on the left side
21) pbD passive soil pressure on the right side
22) Nvte arch effort
23) poG initial pressure on the left side
24) poD initial pressure on the right side
25) pvG vertical component of the soil reaction on the left side
26) pvD vertical component of the soil reaction on the right side
27) N effort axial
28) fiG friction angle on the left side
29) fiD friction angle on the right side
30) coG cohesion on the left side
31) coD cohesion on the right side
32) dcG cohesion increment on the left side
33) dcD cohesion increment on the right side
34) kaG horizontal active pressure coefficient on the left side
35) kaD horizontal active pressure coefficient on the right side
36) kpG horizontal passive pressure coefficient on the left side
37) kpD horizontal passive pressure coefficient on the left side
38) kacG active pressure coefficient on the cohesion on the left side
39) kacD active pressure coefficient on the cohesion on the right side

40) kpcG passive pressure coefficient on the cohesion on the left side
41) kpcD passive pressure coefficient on the cohesion on the right side
42) daG obliquity of the active stress on the left side
43) daD obliquity of the active stress on the right side
44) dpG obliquity of the passive stress on the left side
45) dpD obliquity of the passive stress on the right side

B.6.6.2. Reading the file 02-KR_ELS_Reactions

This file contains the reactions in the supports from an SLS calculation.

For k=1, 7 *loop the types of supports*
For i=1, nAnc *loop on the number of anchors per type*
iPhase k i Fanc

1) iPhase index of the phase
2) k anchor type index
   - 1: anchor
   - 2: strut
   - 3: circular waling
   - 4: linking anchor
   - 5: rotational stiffness
   - 6: surface support
   - 7: slab
3) i anchor number (is initialized for each type)
4) Fanc axial effort
5) Manc embedding torque

B.6.6.3. Reading the file 03-KR_ELU_MISS_Wall

This file contains the values of displacements, internal forces and various pressures of the soil at every level of the wall (2 values per element) resulting from a ULS calculation.

Header of the file (only once at the beginning of the file):

For k=1, nEcr loop on the walls
- **nLignes(k)** number of lines in the result block of each wall

Suite du fichier:

For k=1, nEcr loop on the walls
For i=1, nEl(k) loop on the elements of each wall
For j=1,2 deux nœuds de chaque élément (sup et inf)

1) iPhase index of the phase
2) k wall index
3) i element index
4) j =1 top node / =2 bottom node
5) Z node level
6) Md bending moment (design value)
7) Vd shear effort (design value)
8) ph,k(G/D) characteristic value of the mobilized horizontal pressure (2 columns)
9) u,k(G/D) characteristic value of the water pressure (2 columns)
10) pa,k(G/D) characteristic value of the effective pressure in active side (2 columns)

11) pb,k(G/D) characteristic value of the effective pressure in passive side (2 columns)
12) pd_eff characteristic value of the effective differential pressure
13) σ'v(G/D) characteristic value of the effective vertical stress (2 columns)
14) Nvte design value of the arch force
15) pvG vertical pressure on the left side
16) pvD vertical pressure on the right side
17) Nd design value of axial force
18) fiG friction angle on the left side
19) fiD friction angle on the right side
20) coG cohesion on the left side
21) coD cohesion on the right side
22) dcG cohesion increment on the left side
23) dcD cohesion increment on the right side
24) kaG horizontal active pressure coefficient on the left side
25) kaD horizontal active pressure coefficient on the right side
26) kpG horizontal passive pressure coefficient on the left side
27) kpD horizontal passive pressure coefficient on the right side
28) kacG horizontal active pressure coefficient on the cohesion on the left side
29) kacD horizontal active pressure coefficient on the cohesion on the right side
30) kpcG horizontal passive pressure coefficient on the cohesion on the left side
31) kpcD horizontal passive pressure coefficient on the cohesion on the right side
32) daG obliquity of the active stress on the left side
33) daD obliquity of the active stress on the right side
34) dpG obliquity of the passive stress on the left side
35) dpD obliquity of the passive stress on the right side

Attention: pvG and pvD are characteristic values (if approach 2)

B.6.6.4. Reading the file 04-KR_ELU_Reactions

This file contains the reactions of the linear supports (tie rods / struts / links / rotational stiffness) resulting from an ULS calculation.

For k=1, 7 loop on the types of supports
For i=1, nAnc loop on the number of anchors per type
iPhase k i Fanc,d

1) iPhase index of the phase
2) k anchor type index
   - 1: anchor
   - 2: strut
   - 3: circular waling
   - 4: linking anchor
   - 5: rotational stiffness
   - 6: surface support
   - 7: slab
3) i anchor number (is initialized for each type)
4) Fanc axial effort (design value)
5) Manc embedding torque (design value)

B.6.6.5. Reading the file 05-KR_ELU_MEL_F_Wall

This file contains the values of the vertical and horizontal forces and pressures of the soil at every level of the wall (2 values per element) resulting from a calculation with the ULS of type MEL F.

Header of the file (only once at the beginning of the file):

For k=1, nEcr loop on the walls
- **nLignes(k)** number of lines in the result block of each wall

For k=1, nEcr loop on the walls
For i=1, nEl(k)+4 loop on the elements of each wall
For j=1,2 two nodes of each element (top and bottom)

iPhase k i j Z Md Vd ph,k(G/D) u,k(G/D) pa,k(G/D) pb,k(G/D) pd_eff σ'v(G/D) Nvte pvG pvD Nd
fiG fiD coG coD dcG dcD kaG kaD kpG kpD
kacG kacD kpcG kpcD daG daD dpG dpD

Attention: pvG, pvD and Nd are design values

B.6.6.6. Reading the file 06-KR_ELU_MEL_D_Wall

This file contains the values of the vertical and horizontal forces and pressures of the soil at every level of the wall (2 values per element) resulting from a calculation with the ULS of type MEL D.

Header of the file (only once at the beginning of the file):

For k=1, nEcr loop on the walls
- **nLignes(k)** number of lines in the result block of each wall

For k=1, nEcr loop on the walls
For i=1, nEl(k)+4 loop on the elements of each wall
For j=1,2 two nodes of each element (top and bottom)

iPhase k i j Z Md Vd ph,k(G/D) u,k(G/D) pa,k(G/D) pb,k(G/D) pd_eff σ'v(G/D) Nvte pvG pvD Nd
fiG fiD coG coD dcG dcD kaG kaD kpG kpD
kacG kacD kpcG kpcD daG daD dpG dpD

Attention: pvG, pvD and Nd are design values

B.6.6.7. Reading the file 07-KR_ELU_MISS_Synthesis.txt

This file contains the main results of the passive pressure and vertical balance checks at ULS.

For each phase, the following block is repeated as many times as there are walls:

iPhase	iEcr	BM_G,d	BM_D,d	BL_G,d	BL_D,d	Rapp_G	Rapp_D
iPhase	iEcr	Pvd	Fvd_Anc	Fvd_Ecr	Fvd_pds	Rvd	

Parameter description:

- iPhase — phase index
- iEcr — wall index
- BM_G — mobilized passive earth soil pressure on the left side
- BM_D — mobilized passive earth soil pressure on the right side
- BL_G — mobilizable passive earth soil pressure on the left side
- BL_D — mobilizable passive earth soil pressure on the right side
- Rapp_G — ratio of mobilizable and mobilized passive earth soil pressures on the left side
- Rapp_D — ratio of mobilizable and mobilized passive earth soil pressures on the right side
- Pvd — resultant of the vertical components of the soil pressures
- Fvd_Anc — resultant of the vertical components of the anchor reactions
- Fvd_Ecr — resultant of the vertical components of the loads applied on the wall
- Fvd_pds — weight of the wall
- Rvd — resultant force of vertical balance

B.6.6.8. Reading the file 08-KR-KRANZ-Results.txt

This file contains the results of the Kranz checks for plane surfaces.

For k = 1, nEcr (loop on the walls)

iPhase k iCote nSit

For i = 1, nSit(k) → number of situations on the retained side

iPhase	**k**	**iCote**	**i**	**Nta**	**Nb**	**theta**	**Zd**	**Xb**	**Zb**
Zc	**αref**	**Wtot**	**P1H**	**P1V**	**P2H**	**P2V**	**RH**	**RV**	
Tdsb,k	**Tref,k**	**Tdsb,d**	**Tref,d**						

- iPhase — phase index
- k — wall index
- iCote — index of the retained side for the wall k (1: left side / 2: right side)
- nSit — number of situations
- I — situation number
- Nta — number of tie rods taken into account
- Nb — number of subdivisions into sub-blocks
- Theta — angle of the spiral
- Zd — zero shear level
- Xb — abscissa of the point « B »
- Zb — level of the point « B »
- Zc — level of the "equivalent" anchor point for the examined massif
- αref — inclination of the reference anchor force
- Wtot — total weight of the massif taking into account the external loads
- P1H — horizontal component of the downstream soil pressure

- P1V vertical component of the downstream soil pressure
- P2H horizontal component of the upstream soil pressure
- P2V vertical component of the upstream soil pressure
- RH horizontal component of the reaction under the massif
- RV vertical component of the reaction under the massif
- Tdsb,k destabilizing anchor force (characteristic value)
- Tref,k reference anchor force (characteristic value)
- Tdsb,d destabilizing anchor force (design value)
- Tref,d reference anchor force (design value)

B.6.6.9. Reading the file 09-KR-Phase_And_Combinations.txt

This file contains the relation between the successive calculation phases calculated by the calculation kernel, the combination to which they belong and the index of the calculation phase on the user interface.

For each phase calculated by the kernel, a line appears in this file:

iPhaseMoteur iCombinaison iPhaseUtilisateur

- iPhaseMoteur index of the calculation phase generated by the kernel
- iCombinaison index of the calculation combination to which the phase belongs
- iPhaseUtilisateur index of the calculation phase within the staged construction defined by the user on the interface

B.7. Printing

This is accessible by the shortcut buttons present in the various Print windows or by the **File Menu**, then **Print**.

B.7.1. Single wall type projects without ULS checks

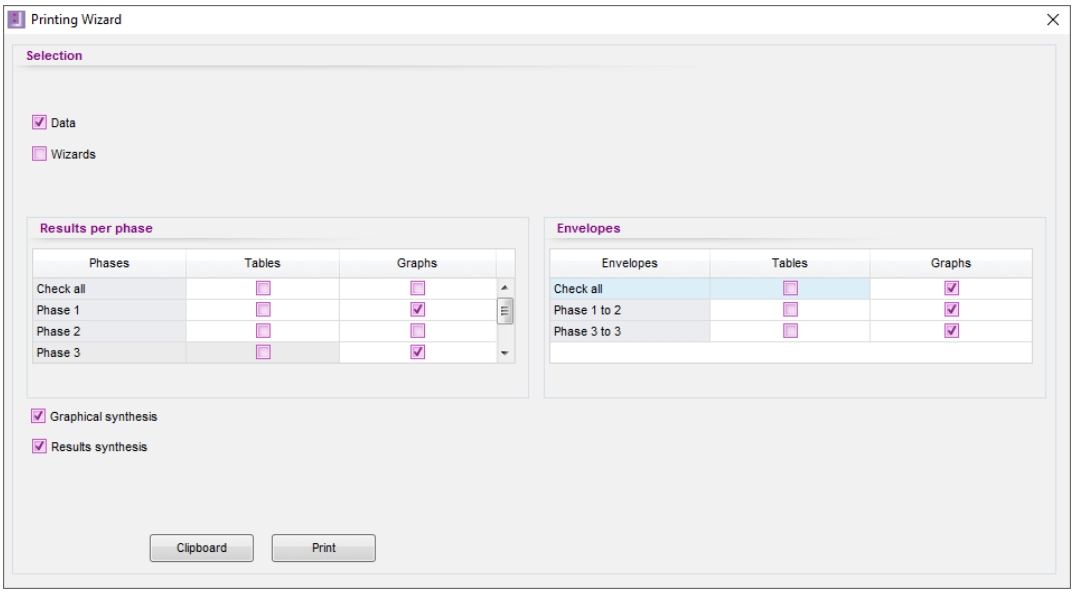


Figure B165 : Printing wizard for a single wall project without ULS checks

By a simple click, the print dialogue box is used to select the items to be printed:

- **Data:** prints the first tab in the results window containing the reminder of the soil characteristics, the retaining wall and the selected options.
- **Wizards**: prints the wizards input data and the results obtained.
- **Graphical synthesis**: prints the synthesis of the project phasing. This synthesis includes the project cross-section for each phase and the parameters of the various actions defined.
- **Results per phase**: used to print the results of all the calculation phases (**Check all**) or only those selected. it is possible to print the results tables (left-hand column) and/or the Graphs (right-hand column).
- **Envelops**: prints the calculated envelopes in table and/or graphic format.
- **Results synthesis**: prints the synthesis table of the results obtained.

These "printouts" can be:

- Either copied to the clipboard for subsequent inclusion in another document;
- Or sent to a printer.

The choice is made by clicking the appropriate button at the bottom of the printing wizard window, once the elements to be printed have been selected.

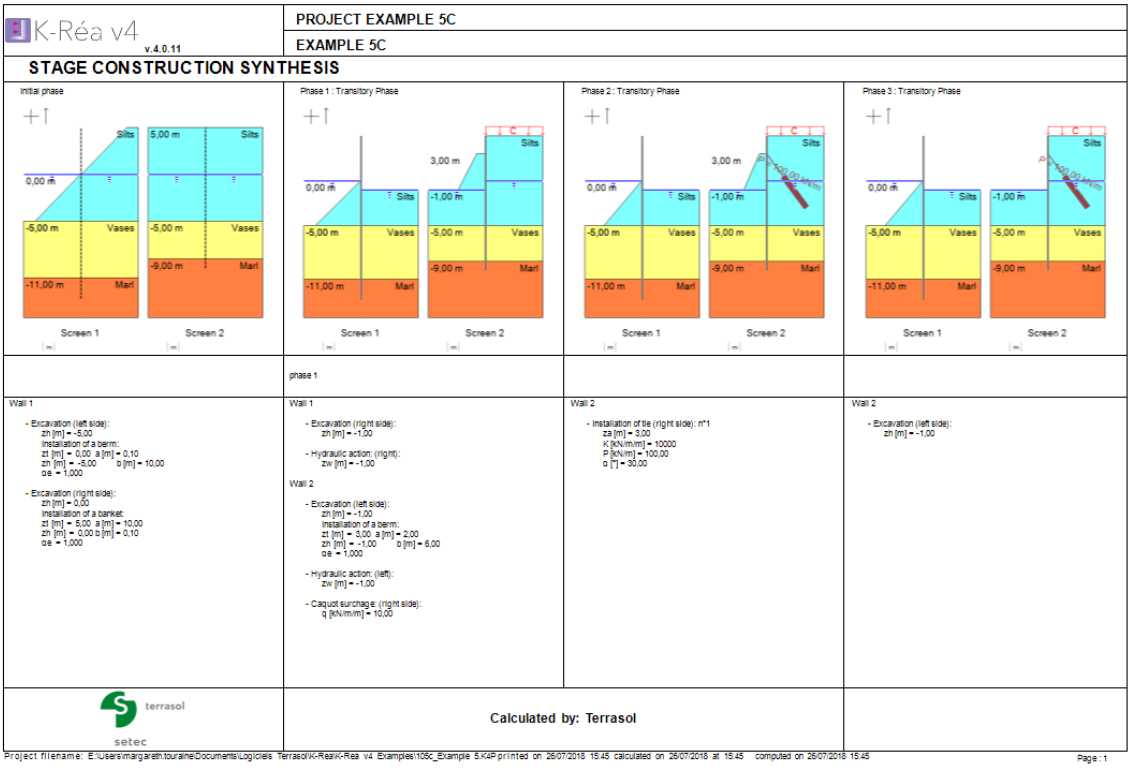


Figure B166 : Example of printing of graphical synthesis of phasing

B.7.2. Single wall type projects with ULS checks

When printing the calculation results for a single wall project with ULS checks, the type of results to be printed must be chosen:

- **SLS results**: these are results produced by the "standard" SSIM calculation without weighting.
- **ULS results**: these are results produced by the calculation with weighting (LEM or SSIM depending on the phases) performed in the context of the ULS checks.

The choice between these two types of results is by means of the selection made at the top of the printing wizard window.

SLS results

If the "SLS" option is selected in the printing wizard (see following figure), the available printing options are the same as for a single wall calculation without ULS checks (see chapter B.7.1).

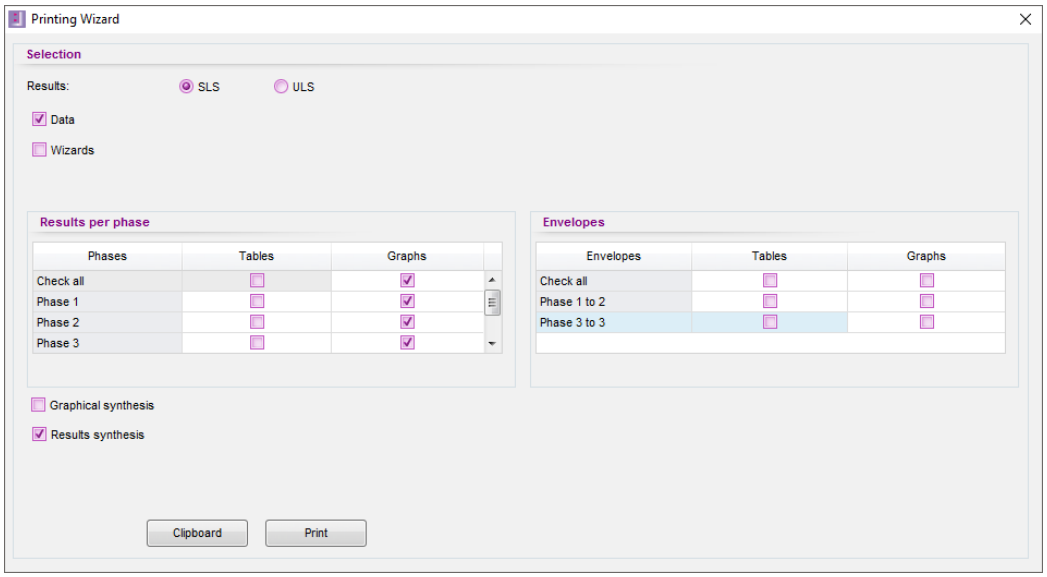


Figure B167 : Printing wizard for a single wall project with ULS checks, with selection of SLS results for printing

Note: as these are the SLS results, only the characteristic values (K index) of the unweighted calculation results will be printed.

ULS results

If the "ULS" option was selected in the printing wizard (see figure below), the available printing options correspond to the results available for a calculation with ULS checks.

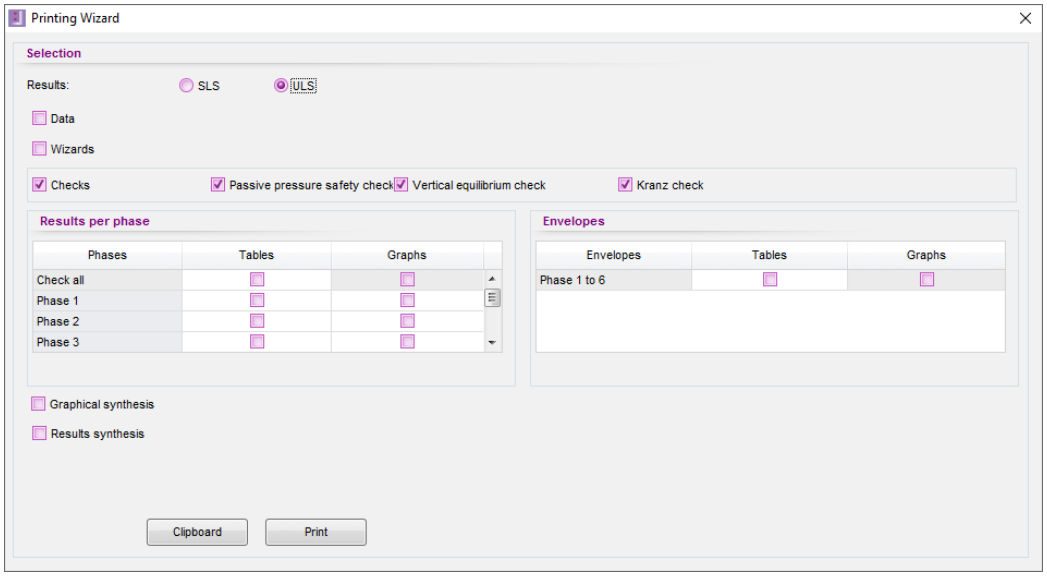


Figure B168 : Printing wizard for a single wall project with ULS checks, with selection of ULS results for printing

The printing options are as follows:

- **Data:** prints the first tab in the results window containing the reminder of the soil characteristics, the wall and the selected options (same as previously).
- **Wizards**: prints the wizards input data and the results obtained.
- **Graphical synthesis**: prints the synthesis of the phases (same as previously).
- **Results per phase**: used to print the results of all the calculation phases (**Check all**) or only those selected. It is possible to print the results tables (left-hand column) and/or the graphs (right-hand column).
- **Checks**: prints the results of the ULS checks performed for each phase (check on passive earth pressure safety, check on vertical equilibrium and check on stability of anchoring block in the case of a project including one or more ties).
- **Detailed Kranz check**: prints the details of the calculations performed in the context of the Kranz check (in addition to the Kranz synthesis results already printed if the **Checks** box above is ticked).
- **Results synthesis**: prints the synthesis of results obtained for displacements, bending moments, shear forces and forces in the anchors.

Note: as these results are ULS, the characteristic values (k index) and design values (d index) of the results will be printed for the phases in which the walls are anchored (SSIM calculation). For those phases in which the wall is assumed to be cantilever (LEM calculation), only the design values (d index) of the results will be printed (see Part C of the manual).

B.7.3. Double wall type projects

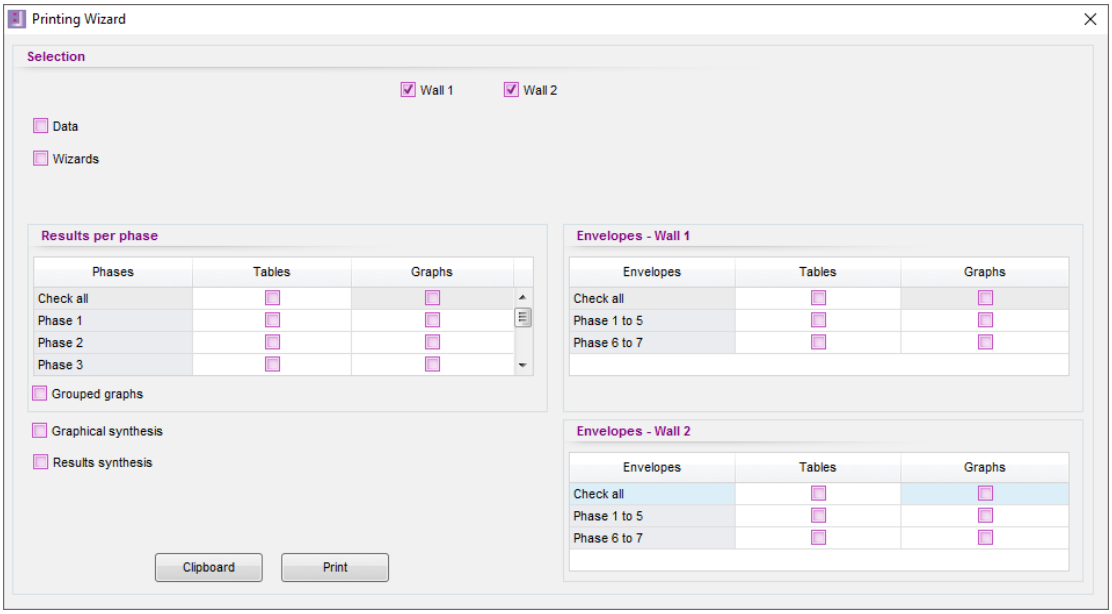


Figure B169 : Printing wizard for a double wall project

The printing options available for double wall projects are on the whole the same as for single walls without ULS checks, but they are doubled, to enable the user to choose whether to print the results for wall 1 and/or for wall 2. By default, the wizard proposes printing the results for both walls.

In addition, for the results per phases and the envelopes in graph format, it is possible to superpose the results of the 2 walls on the same graphs, by ticking the "**Grouped graphs"** box.

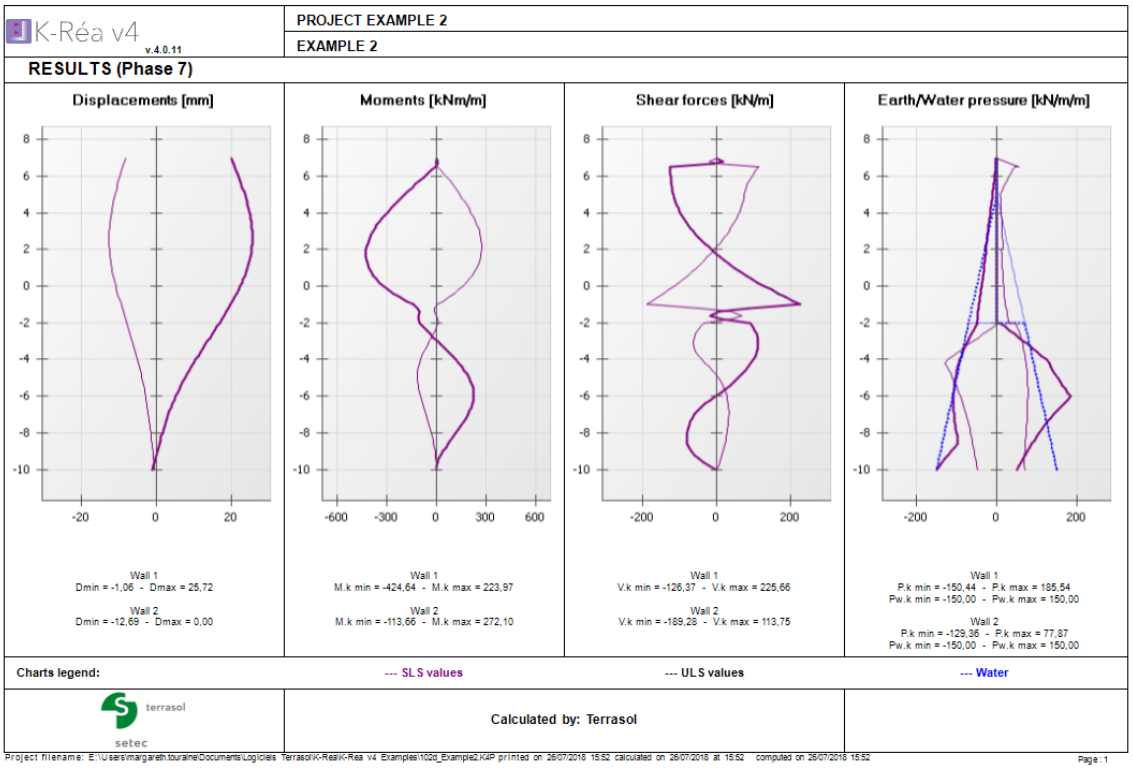


Figure B170 : Extract of printout of a phasing graphical synthesis for a double wall project