Using the Help

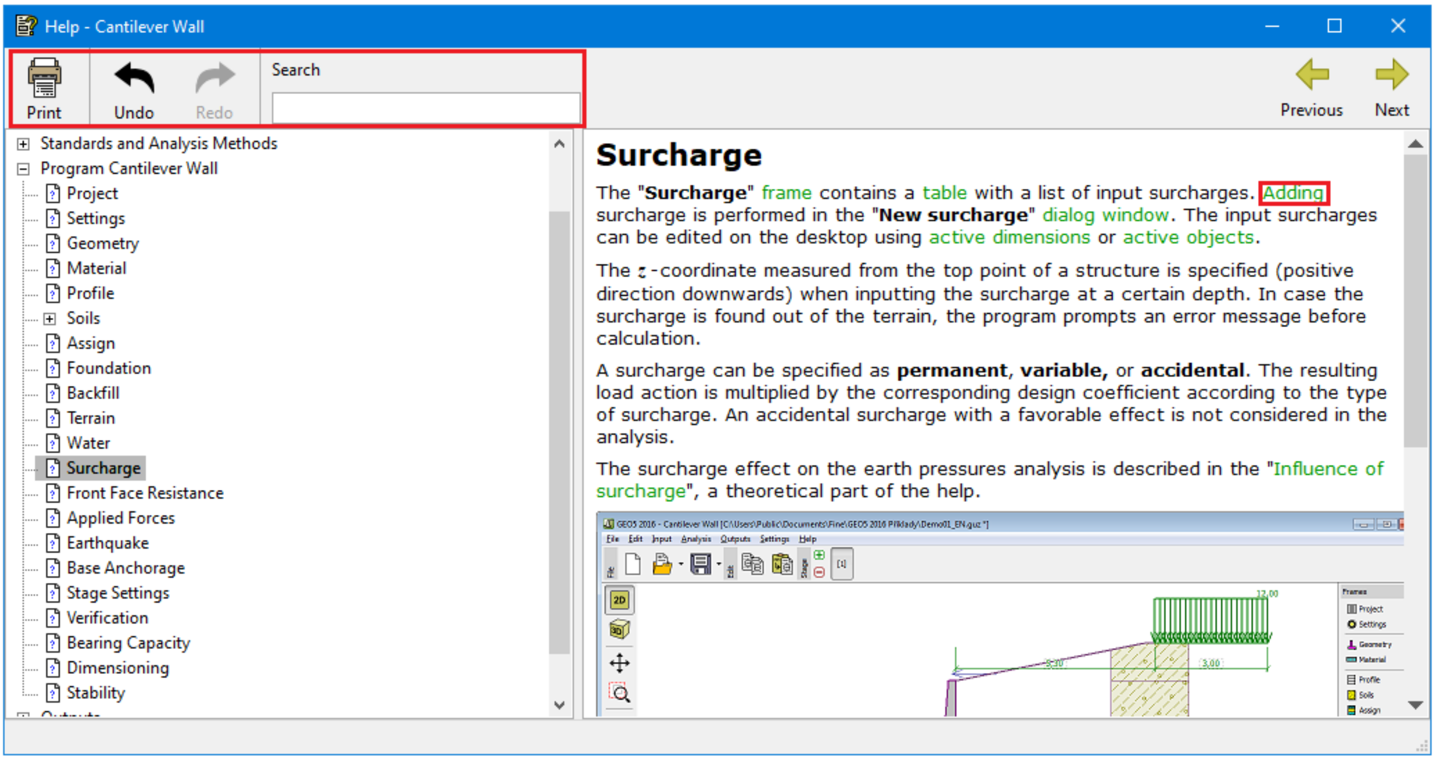
The contextual help for all GEO5 programs is displayed in a standard dialog window. Help can either be launched by directing through the program menu (items "**Help**", "**Content**"), or by pressing the "**F1**" button anywhere in the program.

Some dialog windows (e.g. "**Add new soils**") allow opening a corresponding chapter of the Help by pressing the help button "?".

The dialog window contains:

- A toolbar with basic control buttons. The "**Print**" button opens the print dialog window. The ("**Back**/**Forward**") buttons allow listing through pages, which have been recently opened. The **"Previous/Next"** buttons allow listing through pages in the "**Tree**" up and down.
- Input field "Search".
- "**Tree**", which contains the list of Help items - individual items in the tree are **opened**/**closed** by clicking the symbols "⊞"/"⊟" in front of the name.
- The window for displaying the help - the window header contains the name of the currently shown page.

The text of each help contains further cross-references to other items. The text of these references is colored in green.

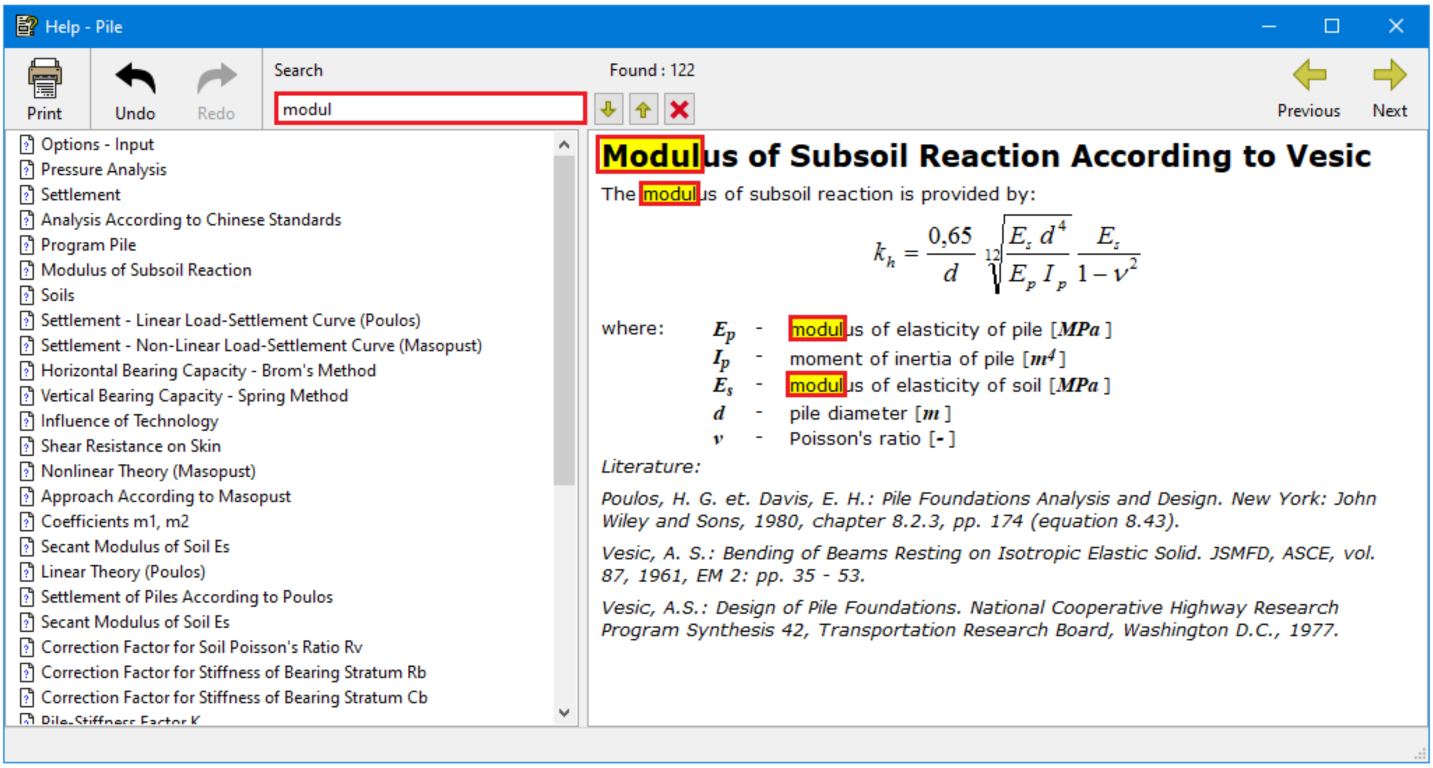


"Help - GEO5" dialog window

Using the Search Function

The "**Search**" function allows you to find an arbitrary text in the help items:

- Write the search texts into the "**Search**" field.
- The searching is launched automatically by text input.
- The algorithm searches the entered text in each expression also in which it is only partially contained.
- The searched text is highlighted in yellow.
- The list of found pages containing the searched text is displayed in the left part of the dialog window.
- The buttons "↑ / ↓" allow listing through found pages in the "**Tree**" up and down.
- The searching is terminated, and the original help tree is shown by using the "✖", "**Undo**" buttons or by clicking on any context link.



Dialog window "Help" - "Search" tab

User Interface

GEO5 programs are standard 32 or 64 bit Windows applications and respect the standard properties of the Windows interface.

The **user environment** is described on the following pages:

- Application Window
- Control Menu
- Tools and Control Bars
- 3D Visualization
- Drawing Settings
- Annotations
- Frames
- Tables
- Dialog Windows

Mouse functions are described on the following pages:

- Active Dimensions and Objects
- Mouse Functions
- Mouse Context Menu

All GEO5 programs support two **sets of units** (metric/imperial).

Clipboard functions are described on the following pages:

- Copy to Clipboard
- Geoclipboard

Programs allow setting other individual settings for **Print parameters**, **Copy to clipboard** and **Input parameters** (**Undo/Redo** function, **snap to grid**, horizontal and vertical **rulers**) in the "Options" dialog window.

32/64-bit Version

All **GEO5** programs (from the 2022 edition) **can be installed in 32, or 64-bit versions**.

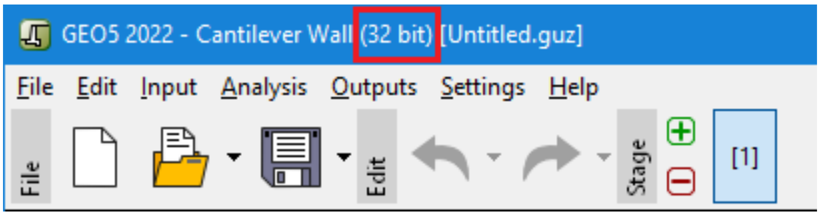
The **version is selected during the installation** - it is possible to select a 32 or 64-bit version in the list in the installer. It is possible to install and use both versions on one PC.

The more powerful **64-bit version is recommended** especially for the complex task in the Stratigraphy and FEM programs, where it allows us **faster calculation or model generation**.

The 32-bit version can be launched on 32-bit Windows only. On 64-bit Windows, both versions can be launched.

Both versions are completely compatible, the tasks created in the 32-bit version can be opened and edited in the 64-bit version (and vice versa).

The **version type is shown** in the window header and also when launching each application on the start screen.

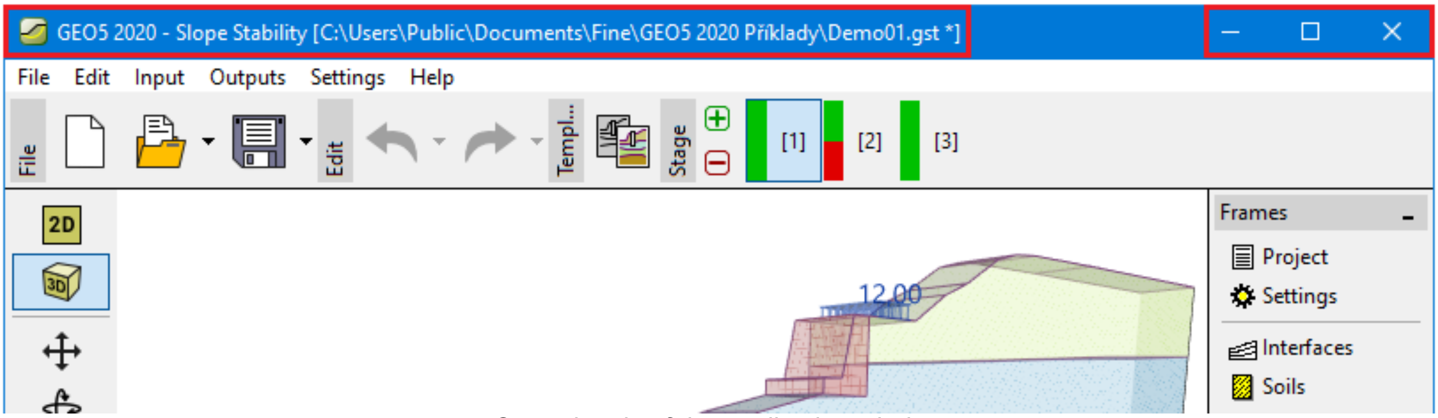


Version type in the window header

Version type on the start screen

Application Window

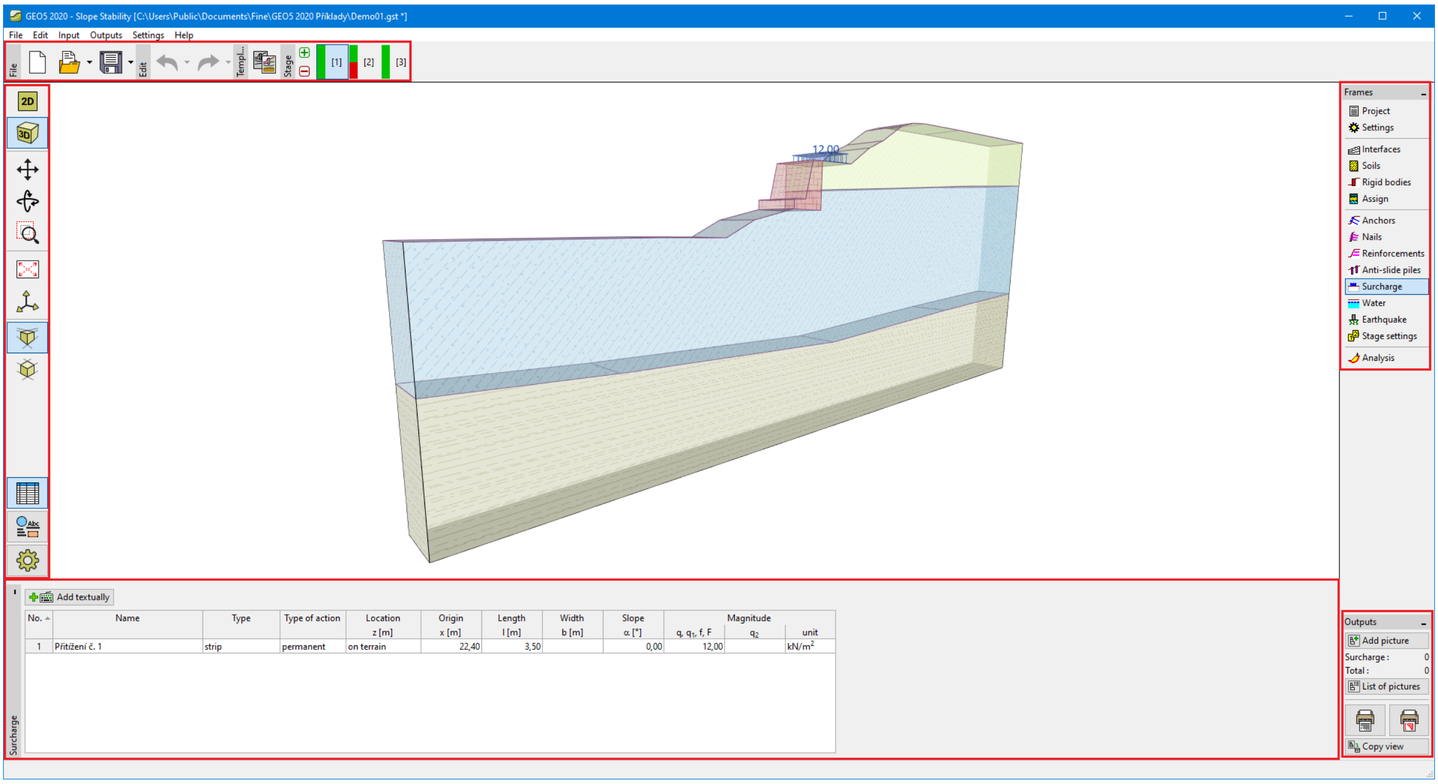
The application is launched in a standard dialog window containing all the managing tools typical for a Windows environment (minimizing, maximizing, and closing the program window). The window header displays information about the currently executed task (file name and location) and used version (32 or 64-bit) - see the picture:



Control tools of the application window

The program window consists of a control menu, tool and control bars, and desktop, which visualizes the executed task.

The bottom part of the desktop displays frames that allow the user to introduce various input parameters into the task. Location of the elements on the desktop is evident from the following figure:



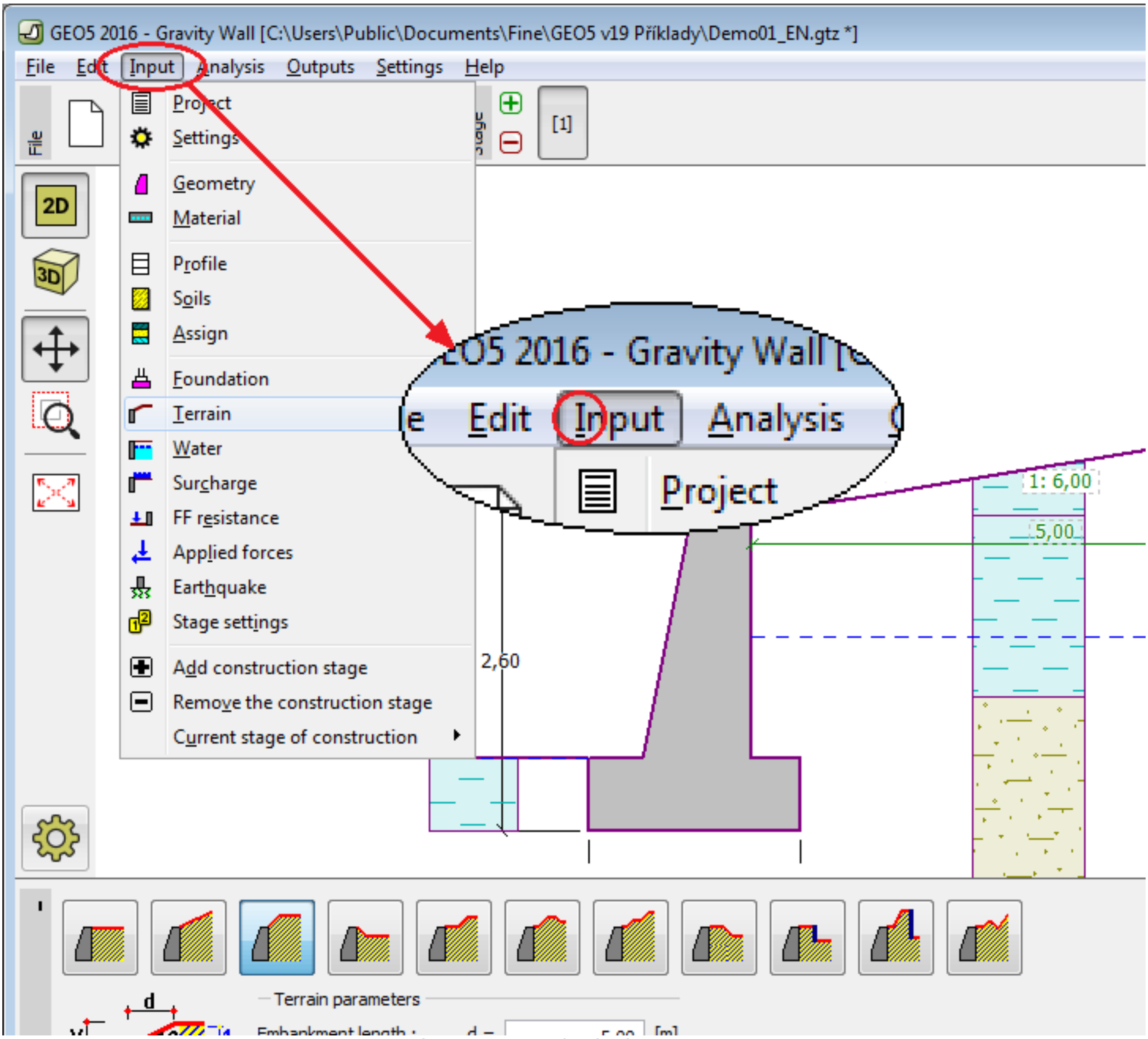
Program tools arrangement

Control Menu

An item in the menu can be selected by clicking the **left mouse button** over it, or alternatively by using the keyboard by pressing **ALT + underlined letter** in the selected item menu.

As typical for the Windows environment, some options in the menu can be replaced by buttons on individual toolbars, or with keyboard shortcuts (when existing it is displayed next to the item in the menu - for example, **Save - CTRL + S**).

Some options in the program can only be set by using the menu - e.g., "Options".

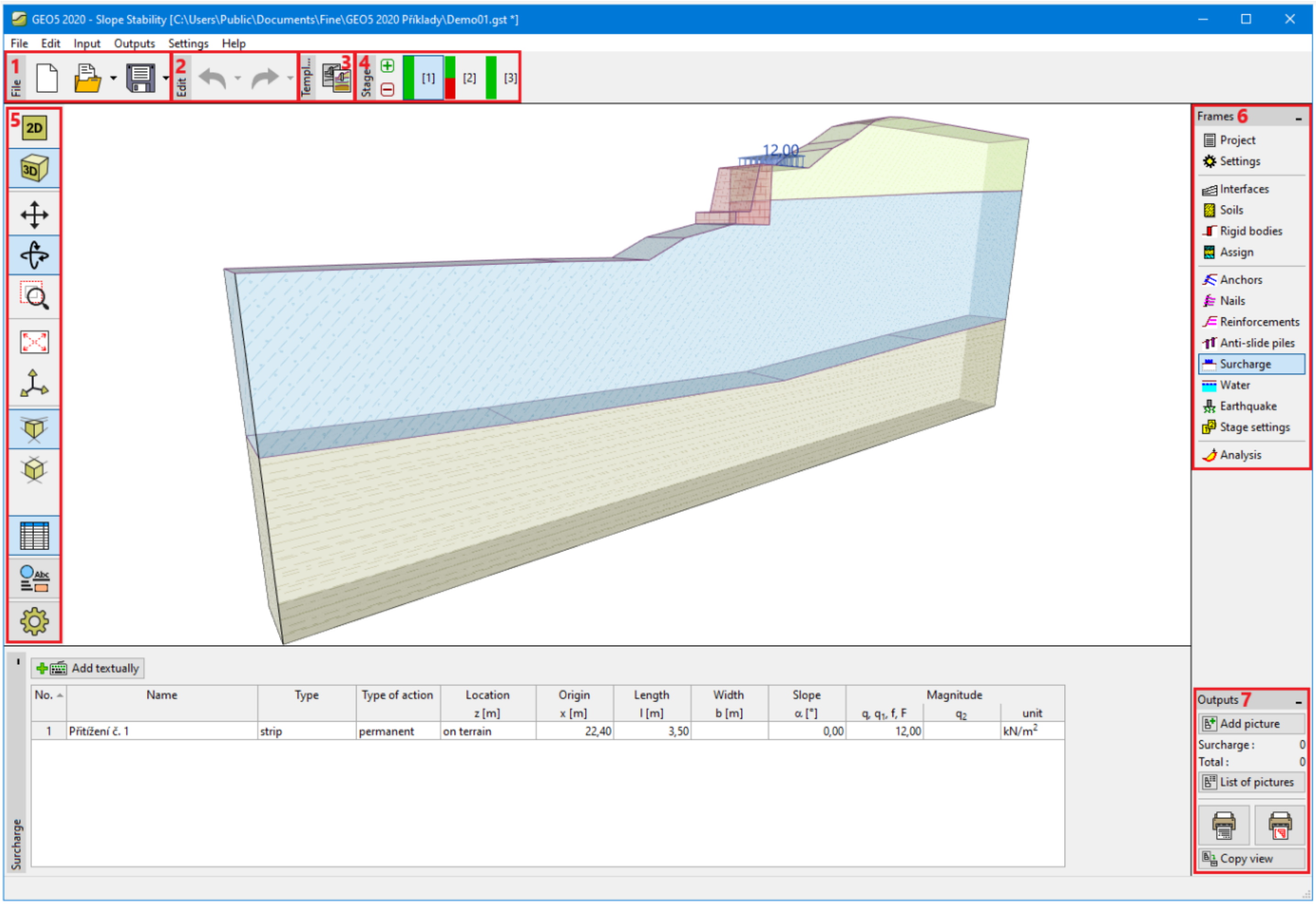


Control menu of a GEO5 program

Tool and Control Bars

Each program contains the following tool and control bars:

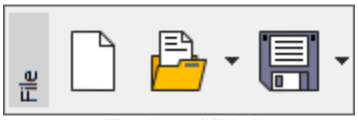
- (1) File
- (2) Edit
- (3) Template
- (4) Stage
- (5) Visualization
- (6) Modes
- (7) Outputs



Tool and Control Bar Location on the Desktop

File

Buttons on the toolbar are for working with files. The toolbar contains the following buttons:



Toolbar "File"

Several buttons are divided into two parts and the button can control more functions (the right part with the arrow).

Using the button for more functions

Individual buttons functions are the following:

New file

- Opens a new file - if there is an existing task opened in the same window, the program prompts the user to save unsaved data.

Open a file

- Opens an existing file - if there is an existing task opened in the same window, the program prompts the user to save unsaved data.

Open recent files

- Opens a list of recently edited files.

Save data into a file

- Saves data of a currently opened task - if no name is assigned to the task, the program opens the "**Save as**" dialog window.

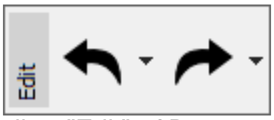
Save as

- Opens the "**Save as**" dialog window - currently running task can be saved under a different name or to a different location.

Edit

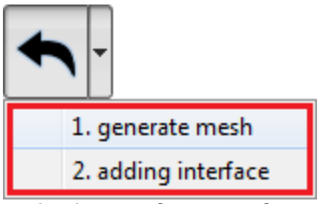
Buttons on the toolbar are used for controlling data in a running task. The toolbar has a different appearance in 1D and 2D programs. The toolbar contains the following buttons:

Toolbar "Edit" - 1D programs



Toolbar "Edit" - 2D programs

Several buttons are divided into two parts, and the button can control more functions (the right part with the arrow).

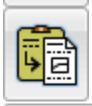


Using the button for more functions

Individual buttons functions are the following:

Copy data to the clipboard

- Copies the data from the current task to clipboard



Paste

- Opens a dialog window and pastes selected data from a different GEO5 program - for example, from the "**Earth Pressures**" program to the "**Gravity Wall**" program.

Undo

- Returns the last performed step (the function is available only in programs with the 2D environment and must be allowed in "Options").

Undo (more steps)

- Opens a list of steps that can be undone.

Redo

- Restores one returned step (the function is available only in programs with the 2D environment and must be allowed in "Options").

Redo (more steps)

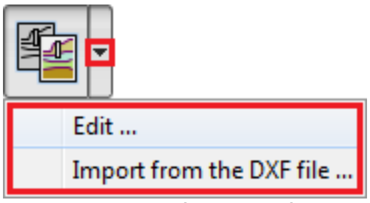
- Opens a list of steps that can be redone.

Template

Buttons on the toolbar are used to work with DXF templates. The toolbar contains the following buttons:

Toolbar "Template"

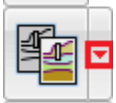
Several buttons are divided into two parts, and the button can control more functions (the right part with the arrow). The button gets this function only after loading the template.

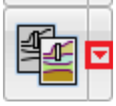


Using the button for more functions

Individual buttons functions are the following:

 **Template**
- Imports template from a DXF file. If the template is loaded, it turns **on/off** its visualization on the desktop.

 **Edit**
- Opens a dialog window for template layer editing.

 **Import from the DXF file**
- Opens a dialog window for the import of a new template from a DXF file.

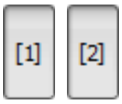
Construction Stages

This toolbar manages the construction stages. The following picture shows the location of individual buttons:

Toolbar "Stages"

 **Adds construction stage**
- adds a new construction stage at the end of the list

 **Removes the construction stage**
- removes the last construction stage from the list

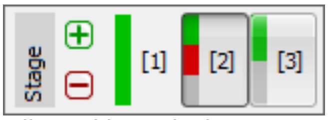


Construction stage 1,2 ...
- switches between individual stages of construction - the selection is performed by using the left mouse button

This bar allows us to define the stages of the construction. Construction stages serve to model gradual building of the construction (**essential for programs** "**Sheeting check**", "**Settlement**", "**FEM**"). This function can also be used for parametric studies, and in each construction stage, different soil assignments or different design coefficients can be assumed. It is rather advantageous to model earthquake effects on a structure in a separate stage of construction as it is then possible to assume different factors of safety or different design coefficients.

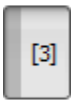
Some types of input (soil assignment, anchors, supports…) there always exists a relationship between construction stages (Heredity).

Some programs show construction stage analysis status using a color stripe.



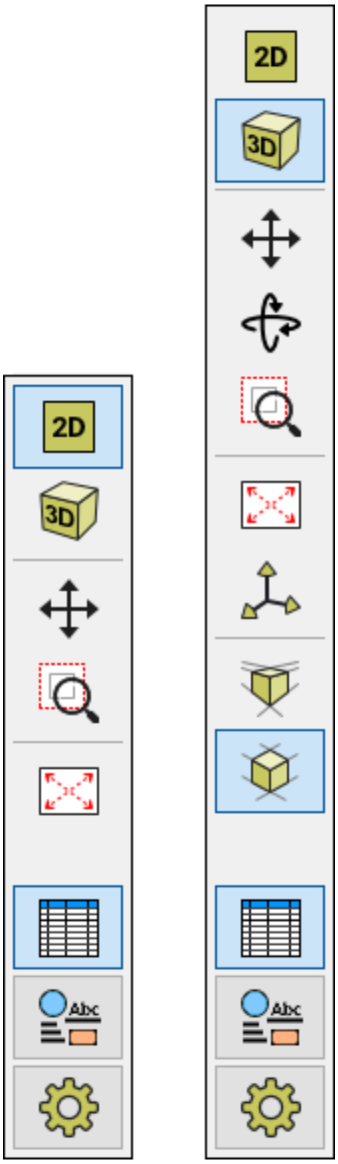
"Stages" toolbar with analysis status color stripes

The colors have the following meaning:

-  green - there is an analysis in the construction stage which IS SATISFACTORY
-  red - there is an analysis in the construction stage which IS NOT SATISFACTORY
-  grey - there is an analysis in the construction stage which has not been performed yet

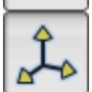
Visualization

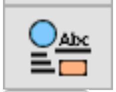
Buttons on the toolbar allow the user to change the setting of visualization on the desktop. The toolbar has a different appearance in 2D and 3D mode. The toolbar contains the following buttons:



Toolbar "Visualization" - toolbar appearance in 2D and 3D mode

Functions of individual buttons are the following:

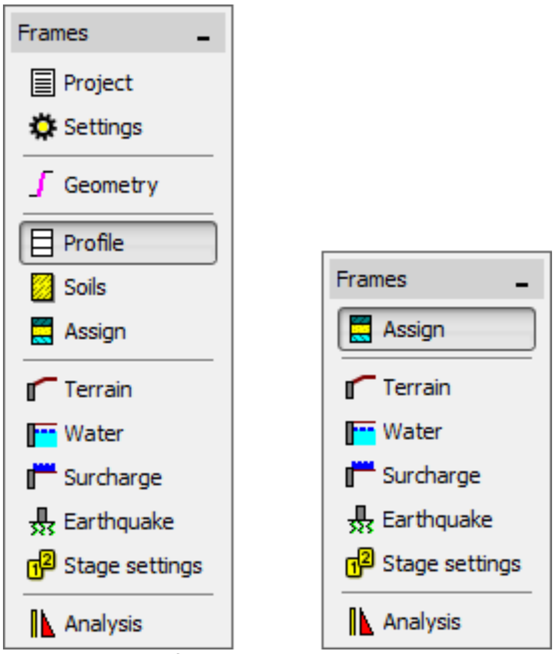
	**2D view**	• Applies the 2D visualization mode.
	**3D view**	• Applies the 3D visualization mode.
	**Move the displayed area**	• Moves the current view in any direction - to proceed move mouse in the desired location while keeping the left mouse button pressed.
	**Rotates the scene**	• Rotates the displayed drawing in any direction (3D view) - to move the drawing slide the mouse while pressing the left mouse button.
	**Shows the marked area**	• Shows and scales up the marked area - the area is selected by using the left mouse button.
	**Modify the scale**	• Scales the view so that all objects are visible on the desktop (by pressing the left mouse button).
	**Pre-defined 3D view**	• Sets the predefined 3D view of drawing (3D view).
	**2D view**	• Displays the view towards the Z -axis

	**Perspective view**	• Sets the perspective view of drawing (3D view).
	**Axonometric view**	• Sets the axonometric view of drawing (3D view).
	**Modes**	• The button opens the frame "Modes" for **data input and calculations**
	**Annotations**	• The button opens the frame "Annotations"
	**Drawing Settings**	• The button opens the frame "Drawing Settings".

Modes

The vertical toolbars allow the user to select the desired mode of inputting data (Project, Geometry, Profile, etc.), including analysis type and verification. The selection of the mode from this bar displays in the bottom part of the desktop the corresponding frame for data input.

The toolbar only contains those frames, where the input of data makes sense. This means that if a task has more construction stages, the toolbar is complete in the first stage; however, some items are missing in further construction stages where given data cannot be changed.



Control bar "Frames" for switching between input data modes

Outputs

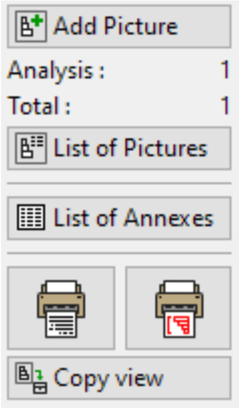
A standalone toolbar serves to manage pictures and output documents.

The "**Add picture**" button opens the "New picture" dialog window. The next line in the bar provides the number of stored pictures in the given mode of data input. The "**Total**" line shows the total number of stored pictures for this file. The "**Picture list**" button opens the list of pictures.

"**List of Annexes**" button opens the list of annexes.

The two other buttons open the "Print and export document" and "Print and export desktop view" dialog windows.

The "**Copy picture**" button saves the current view from the desktop to the clipboard.

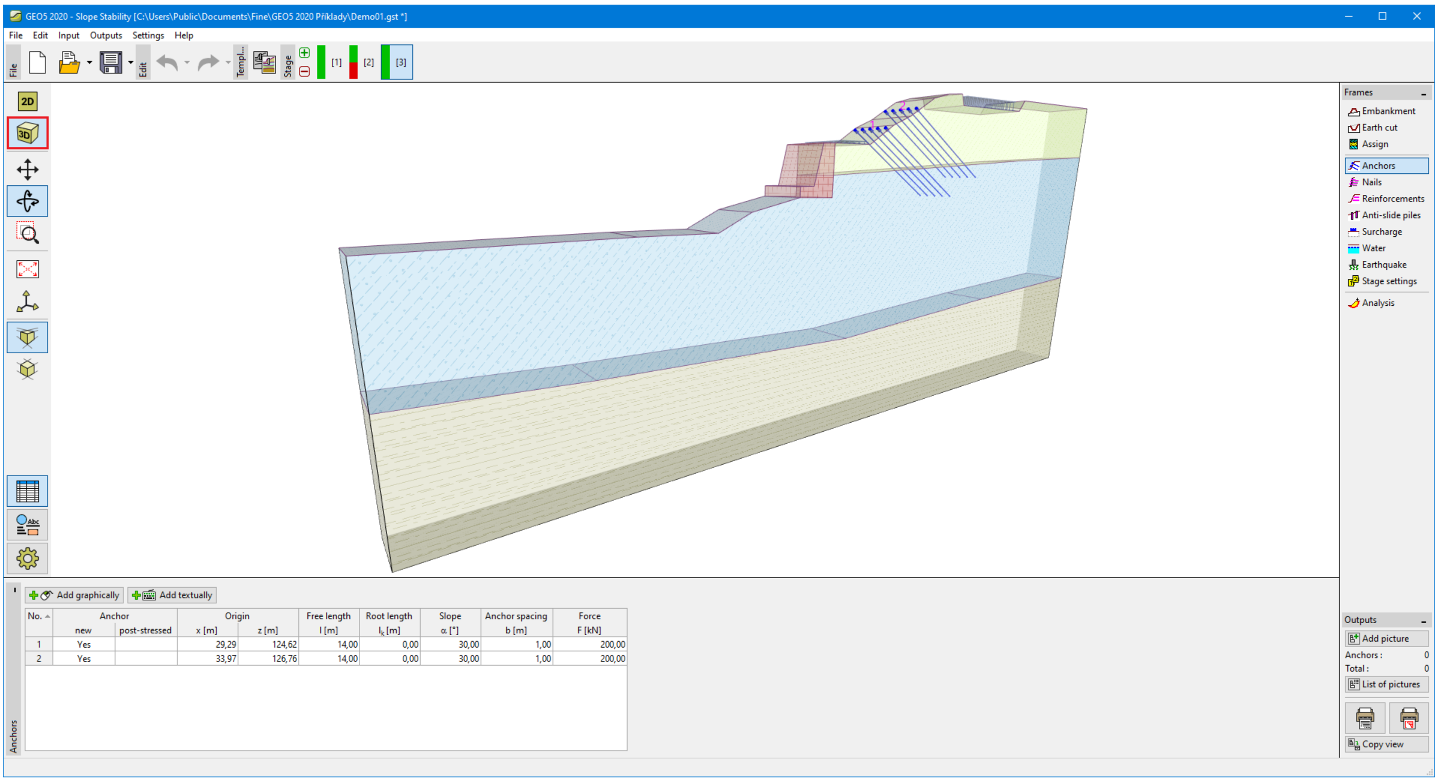


Control bar "Outputs"

3D Visualization

Programs (1D and 2D) enable 3D visualization on the desktop. 3D visualization is informative and serves for better orientation in the structure (e.g. dislocation of objects) and for results presentations.

3D visualization is set on the "Visualization" toolbar, which also contains tools for working with the visualization.



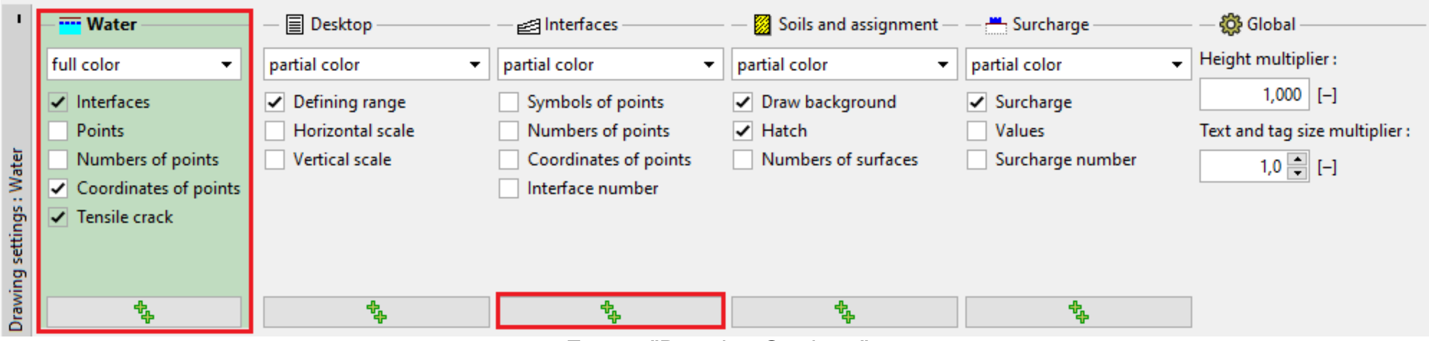
3D visualization

Drawing Settings

The "**Drawing Settings**" frame is used to set the parameters of the visualization (what is showed on the desktop). In any frame, it is possible to switch to the "**Drawing Settings**" mode, just by clicking the button on the "Visualization" toolbar.

Button of drawing settings

Individual columns in the window correspond with the individual frames. On the left, there is always the settings for the current frame (frame "**Water**" on the figure). In other columns, visualization of other objects is defined. It is possible to set visualization only of the currently viewed objects.



Frame "Drawing Settings"

The "**Use everywhere**" button in the bottom part of each column sets the defined parameters of visualization to **all** frames.

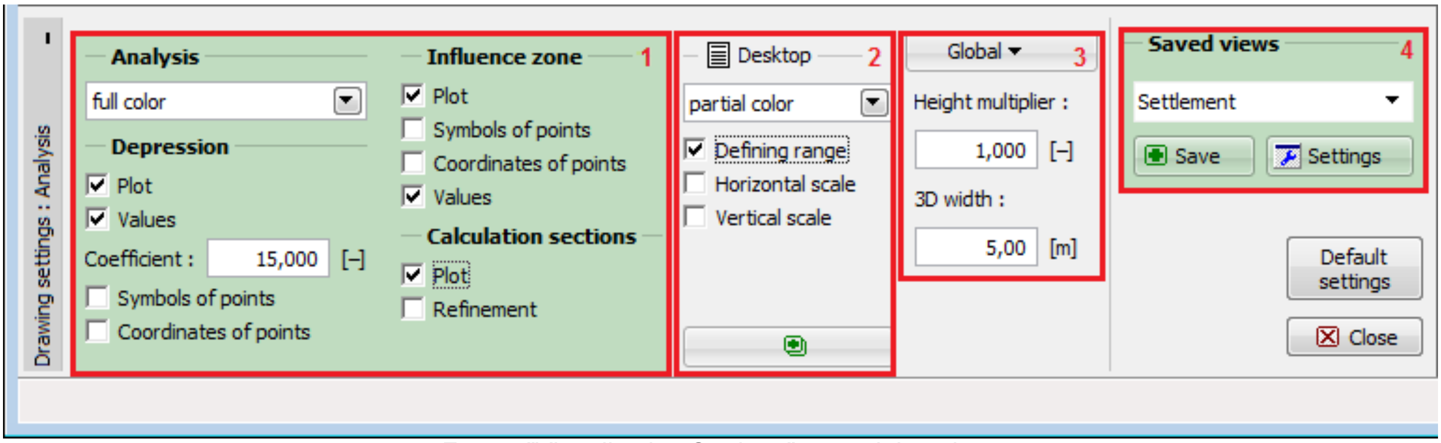
The "**Default settings**" button sets all the parameters back to default.

The settings in this frame serves mainly for defining parameters of the visualization on the desktop - the settings of parameters of drawing for outputs (print) are defined in the "Add Picture" mode.

The frame can also contain columns with special settings, which are only displayed in several cases in some programs:

- Visualization settings of analysis results (**1**).
- Desktop (2D programs) - defines several special settings (defining range, scale) and are global settings for all frames (**2**).
- Global - Height multiplier (2D programs) - enables change of scale in the vertical direction (z) (**3**).
- Global - Stretch Drawing (1D programs) - enables change of scale in the horizontal direction (x) (**3**).
- Global - 3D width - defines the width of the visualized structure in 3D visualization mode (**3**).
- Store views (programs Settlement, FEM) (**4**).

Settings of background colors, styles of lines, and fills are defined for all programs in the "Drawing Styles Administrator".

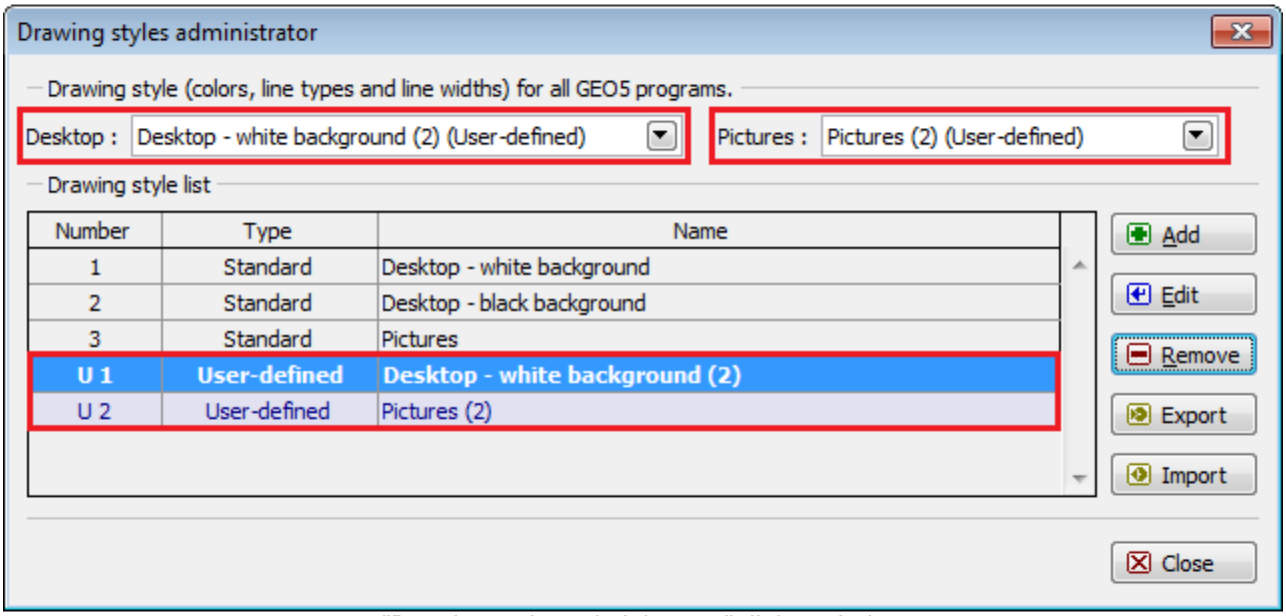


Frame "Visualization Settings" - special settings

Drawing Styles Administrator

The **Drawing styles administrator** enables globally to set the picture styles (colors of displayed objects, colors of lines, line styles, filling styles, selections colors). The dialog window is available from the menu (items "**Settings**", "**Drawing style**"). The user can define the style of visualization on the desktop or in the outputs for all GEO5 programs.

The style of the structure visualization is defined in the "**Desktop**" and "**Pictures**" combo lists.



"Drawing styles administrator" dialog window

The program contains three predefined drawing styles - two for desktop visualization (black and white background) and one for pictures. There is a possibility to input user-defined drawing styles, too.

How to create a user-defined Drawing style:

Select one of the styles in the "**Drawing style list**".

Press the "**Add**" button - then the "**Add drawing style**" dialog window will open.

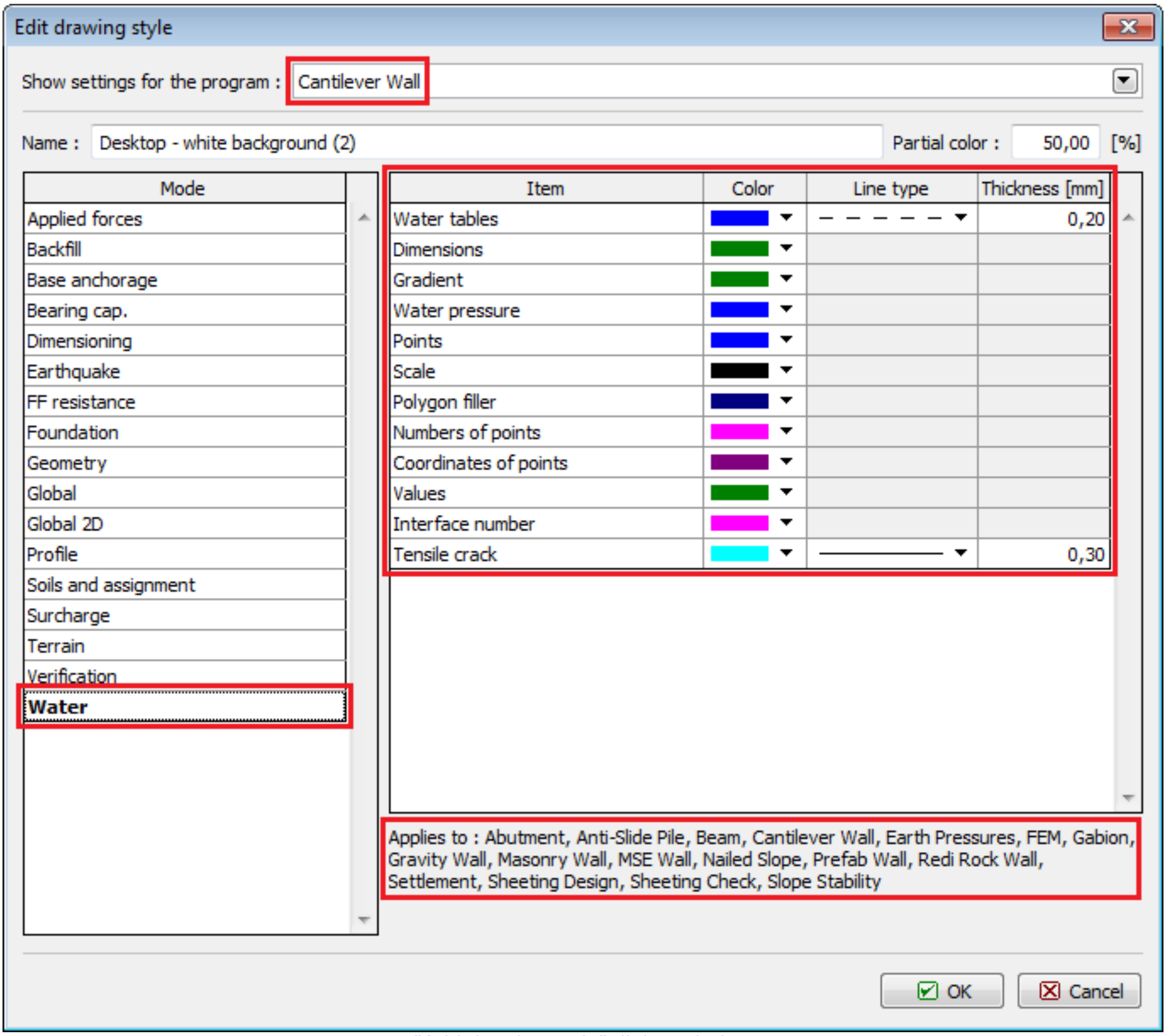
In the "**Show settings for the program**" combobox it is possible to filter modes according to individual programs or show modes for all GEO5 programs.

In the box "**Name**" input the name of a new settings.

In the "**Mode**" list (left part of the window) **select the mode** (e.g. Applied forces, Backfill, Base anchorage...), in which you want to change the style of drawing. In the right part of the window input, new drawing styles of objects in selected mode (e.g. Water tables, Dimensions, Gradient...).

The list of affected GEO5 programs is shown in the bottom part of the window.

Example: If we change drawing style (color, line type, thickness) of "Tensile crack" (mode Water) in the Cantilever wall program, the new settings will **be valid for all programs** in the list in the bottom part of dialog window ("**Abutment**", "**Anti-slide pile**", "**Beam**"....). New settings will be stored by pressing the "**OK**" button. These new settings will be added to the "**Drawing style list**" and will be available in the "**Desktop**" and "**Pictures**" combo lists. After adding a new style, it is **necessary to select it** in the corresponding combo list for visualization on the desktop or print.



"Add drawing style" dialog window

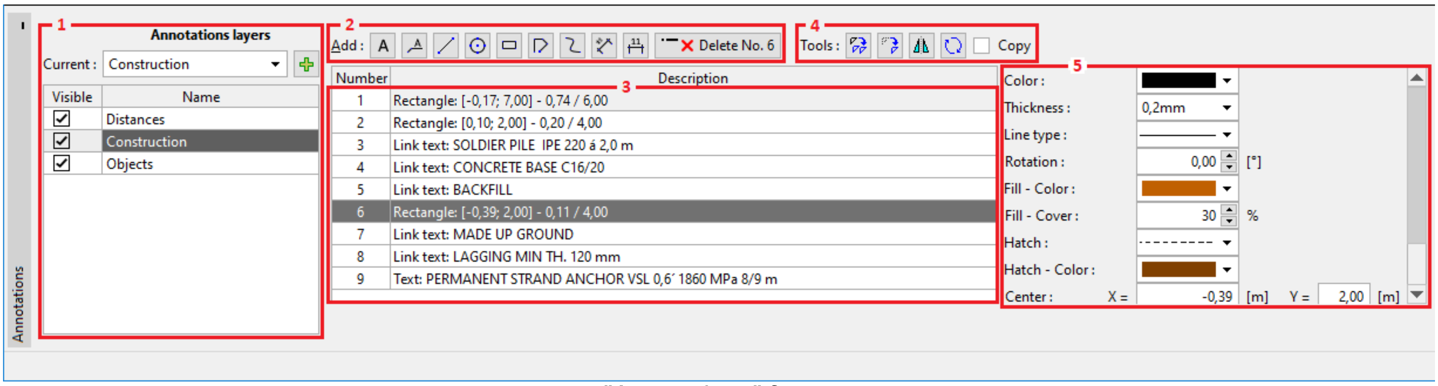
Annotations

In all GEO5 programs, it is possible to change drawing settings on the desktop. Some information can sometimes be missing (additional descriptions, distances, other structures...). In this case, it is possible to use the "**Annotations**" frame:

- add user objects (drawings) to the desktop display
- edit defined objects and use special tools for their editing
- define objects in layers

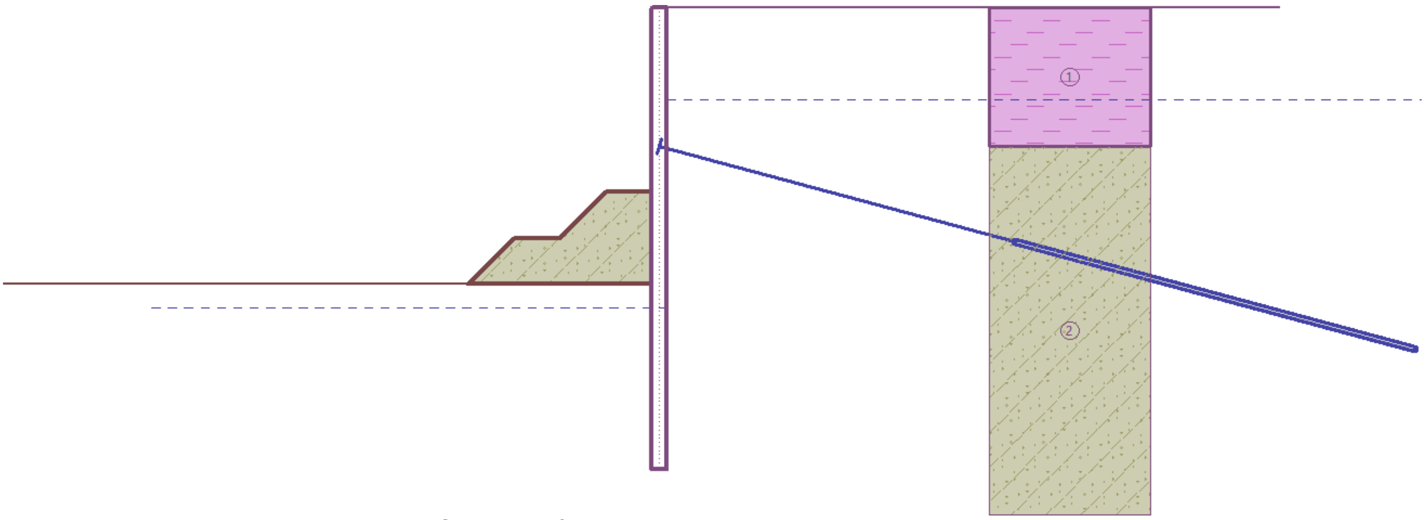
The "**Annotations**" mode is launched by the button on the "Visualization" toolbar. In this mode, the frame contains:

- Table of layers (**1**)
- Buttons for objects input (**2**)
- Table of entered objects (**3**)
- "Tools" button (**4**)
- Editable properties of objects (**5**)

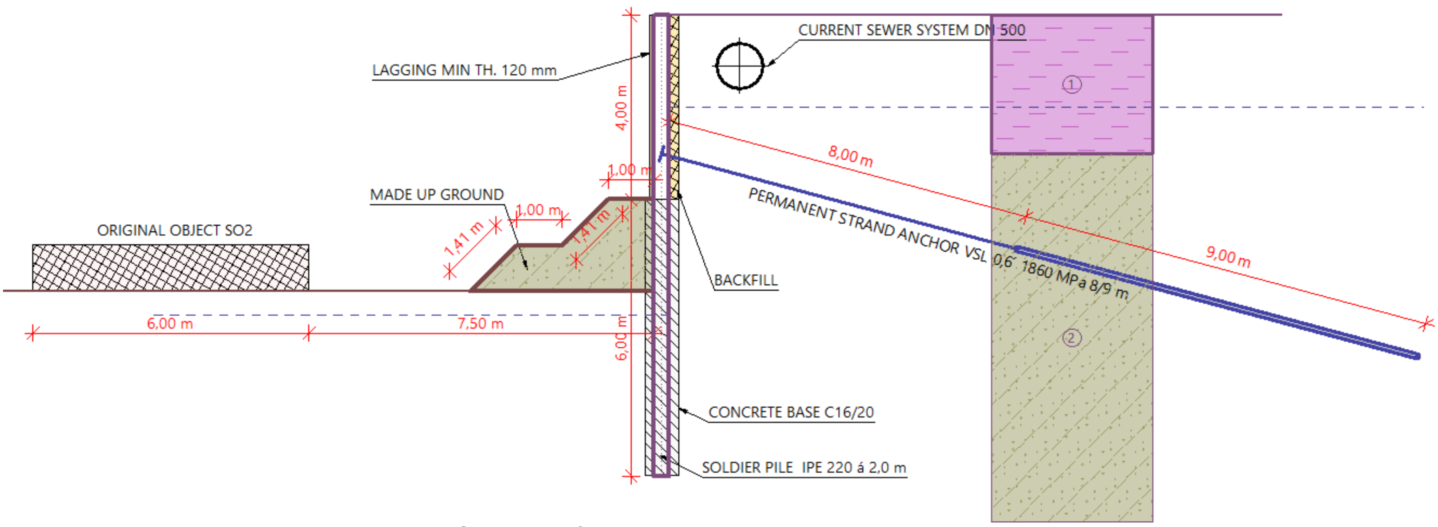


"Annotations" frame

In the following pictures, there is a simple example of "**Annotations**" using.



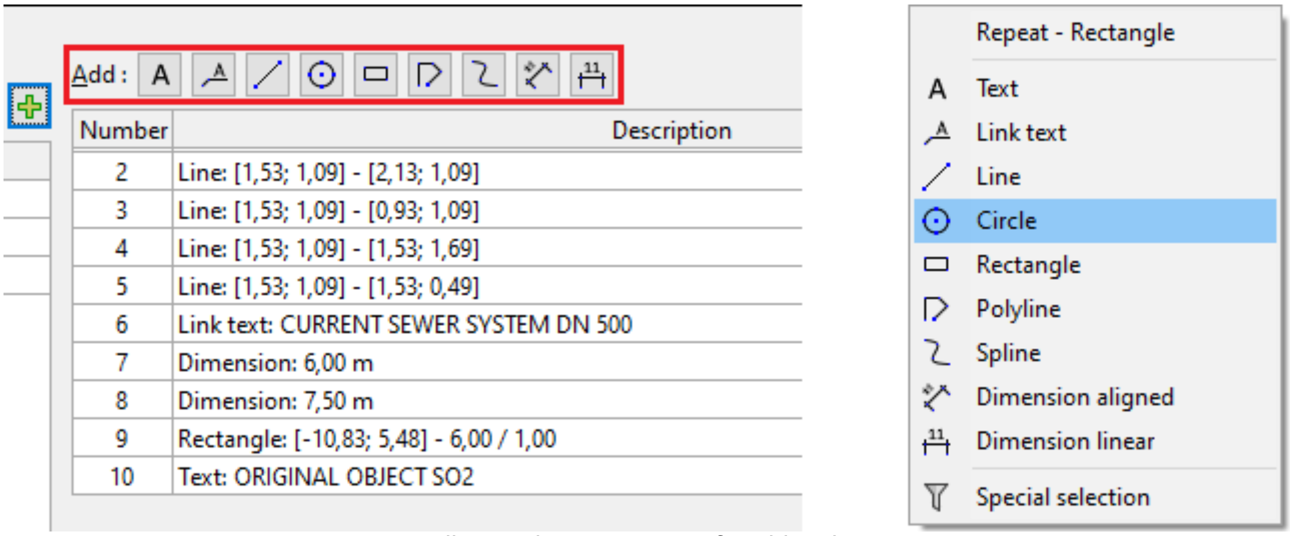
Scheme of shoring structure without annotations



Scheme of shoring structure with annotations

Objects

Objects can be added graphically by a mouse. The type of an object is selected using buttons in the "Annotations" frame or by context menu (right mouse button).



toolbar and context menu for object input

The programs allow inputing following objects:

Object	Description of object input
Text	• place the text by the left mouse button
Link text	• place the link text by the left mouse button
Line	• define the start and the end point by the left mouse button
Circle	• define the center point and the radius by the left mouse button
Rectangle	• define one corner and the length/height of a rectangle by the left mouse button
Polyline	• define all points of a polyline by the left mouse button
Spline	• define all points of spline by the left mouse button
Dimension aligned	• define the start and end point of dimension by the left mouse button, define the distance between the object and its dimension dragging the mouse
Dimension linear	• define the start and end point of dimension by the left mouse button, define the distance between the object and its dimension by dragging the mouse in the horizontal or vertical direction

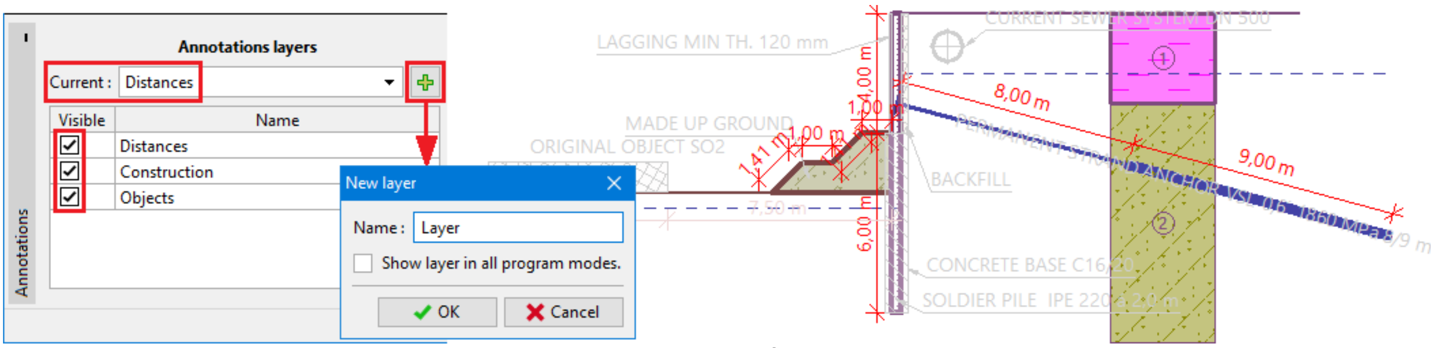
Objects are entered in layers. The selected objects can be edited graphically or textually. Using "Tools", objects can be copied, shifted, or mirrored.

Layers

This tool allows creating an arbitrary number of layers. Individual objects are saved into these layers.

Layer controls in the "**Annotations**" frame are:

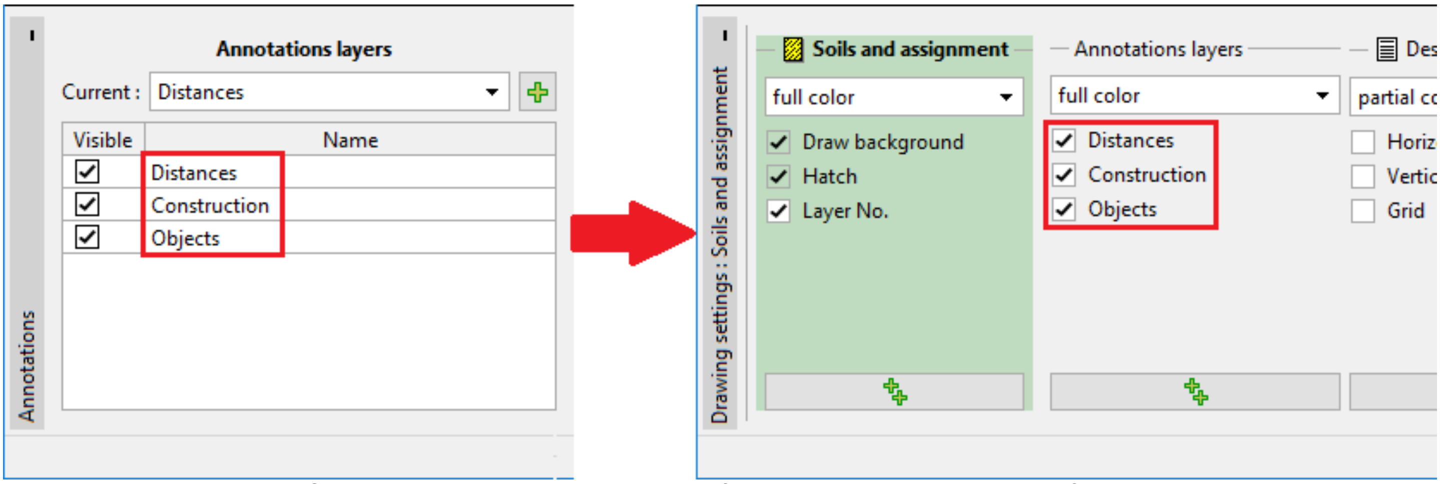
- button opens the "New layer" dialog window to enter the name of the layer. "**Show layer in all program modes**" check button switches this layer visible in all program modes.
- "Current" combo list - select the layer, which is being edited (the objects are displayed in full color)
- "Visible" column - select visible/invisible layers.



"Annotations" frame

The defined layers from the "**Annotations**" frame are displayed in the "**Annotations layers**" column in "Drawing settings".

These layers can be turned on/off in all program modes.

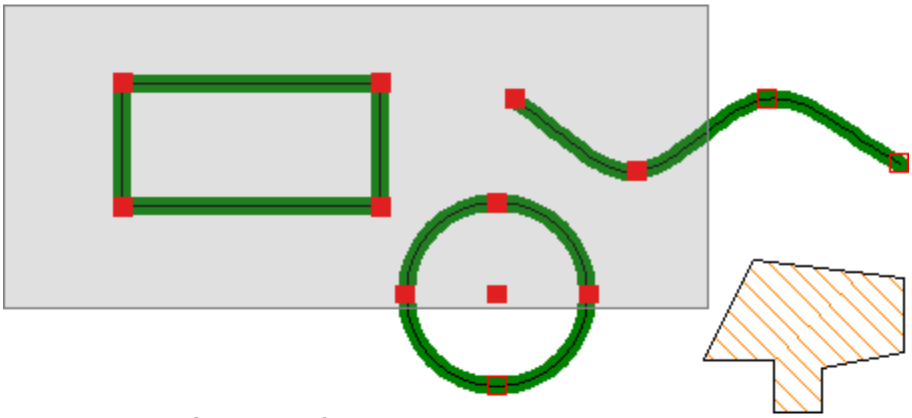


Connection between "Annotations" frame and "Drawing settings" frame

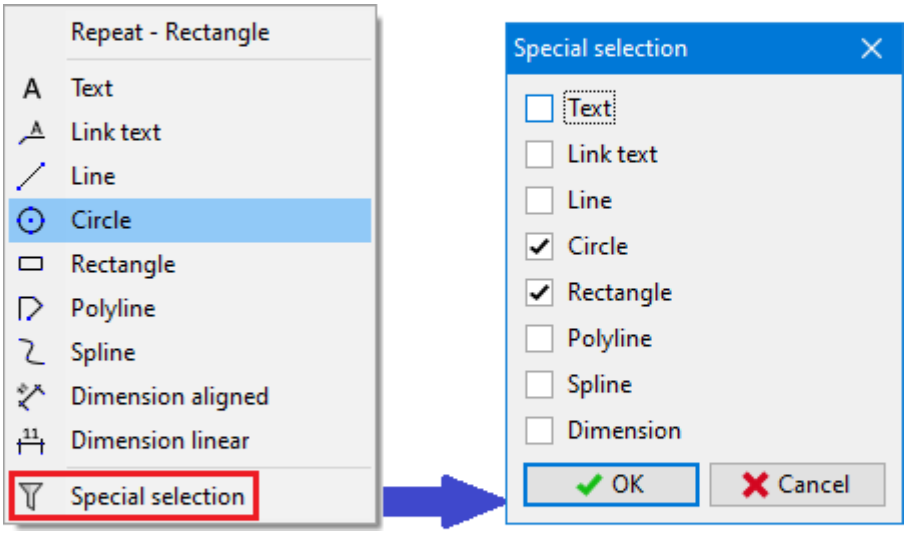
Graphic editing

Defined objects can be selected and edited.

- **one object** - can be selected by the left mouse button
- **multiple objects** - selected by "**SHIFT**" key + left mouse button
- **multiple objects** - selected by a rectangle (dragging the left mouse button)


Selection of multiple objects using a rectangle

- **special selection** - in the dialog window from the mouse context menu - "**Special selection**" item



Multiple selection using a dialog window

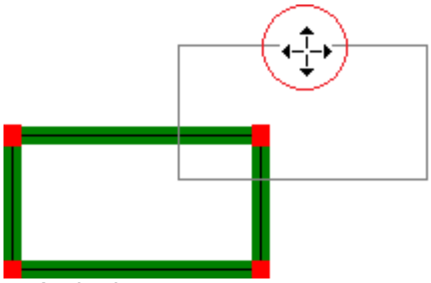
Selected objects can be:

- edited - in the table as "text editing"

- deleted - "**Delete**" button or "**DELETE**" key
- shifted, copied using tools

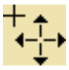
Shift of objects

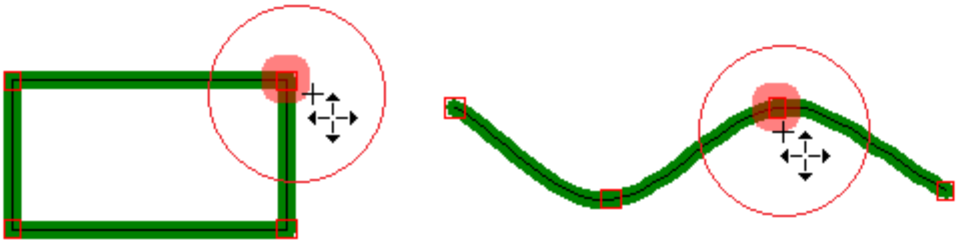
- we select one or more objects
- the cursor is changed to 
- the shift of an object in any direction



Shift of an object by the mouse

The next option is "**Geometry editing**"

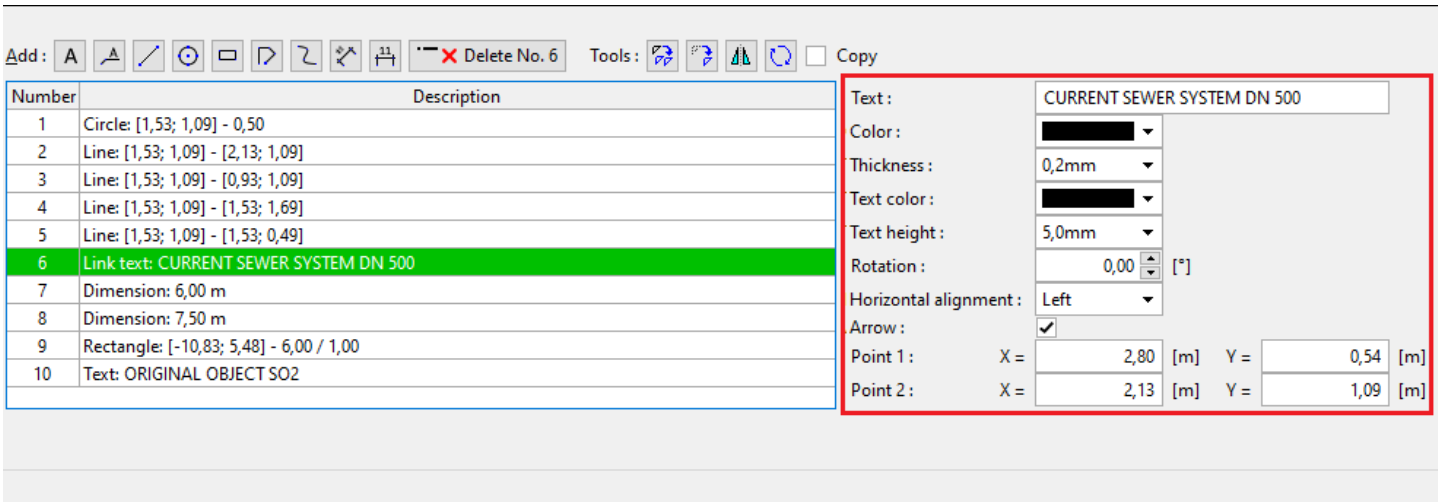
- we select an object
- the cursor is changed to 
- we snap one of the object nodes, and we change the geometry dragging the mouse



Change of geometry - the node shift

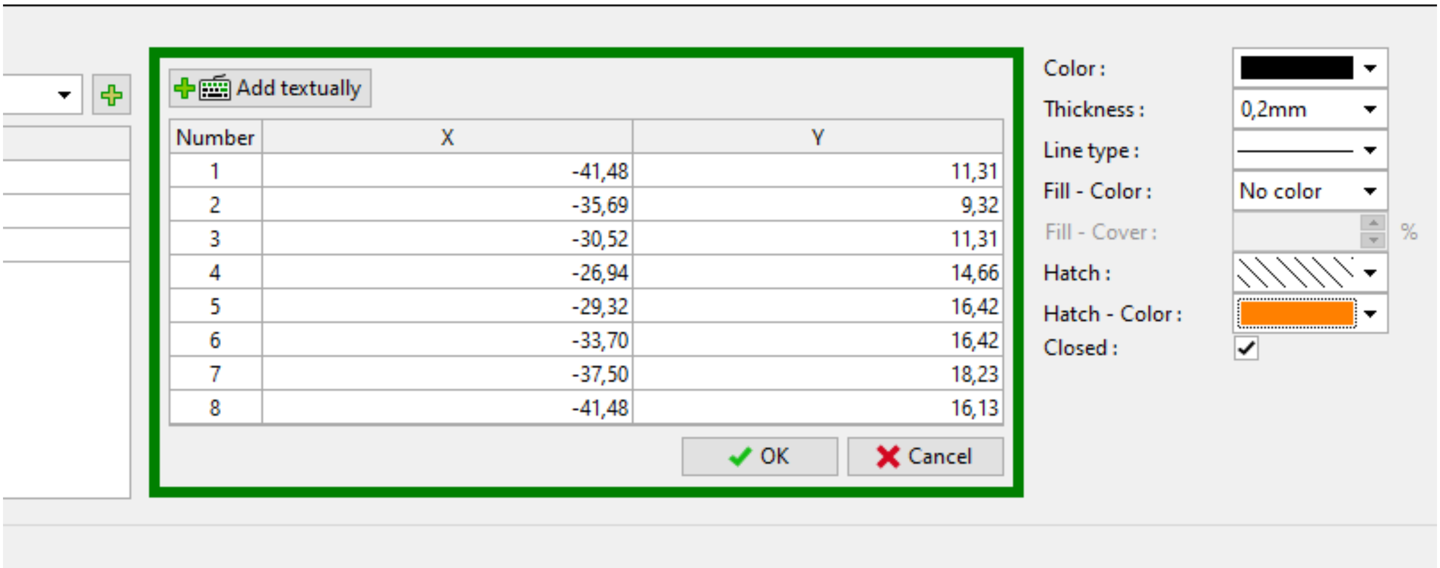
Text editing

Defined objects can be edited in the right part of the frame next to the table of objects. The properties of an object selected in the table are displayed and can be edited.



Text editing of object

"**Spline**" or "**Polyline**" object types allow showing a table with coordinates. (After double click on the row).

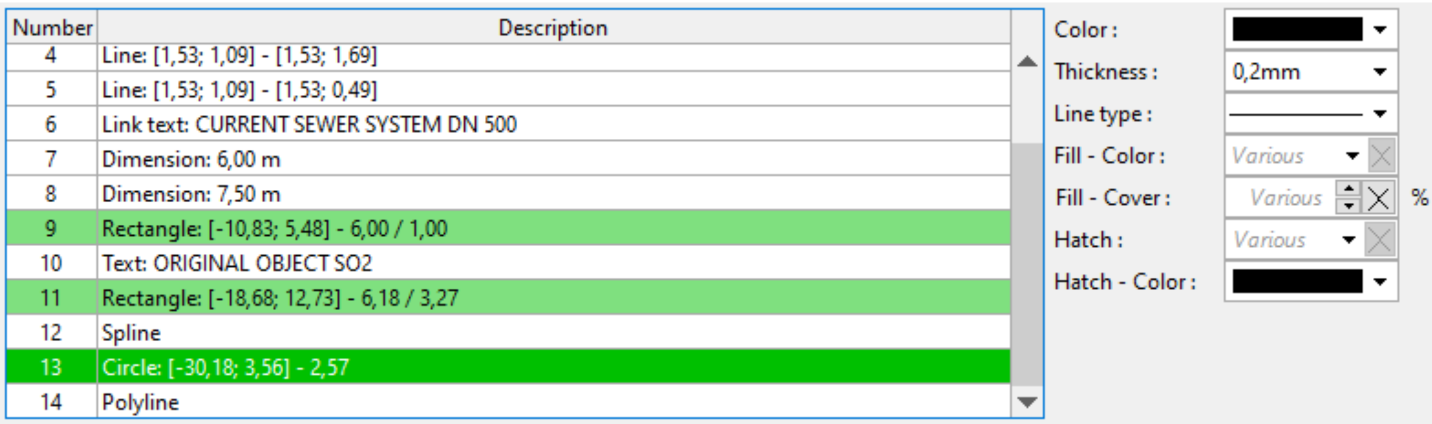


Number	X	Y
1	-41,48	11,31
2	-35,69	9,32
3	-30,52	11,31
4	-26,94	14,66
5	-29,32	16,42
6	-33,70	16,42
7	-37,50	18,23
8	-41,48	16,13

*Text editing (**Splines** and **polylines**)*

Multiple editing

- selecting objects by "**CTRL**" key + the left mouse button
- the **common editable properties of all selected objects** are available next to the table



Number	Description
4	Line: [1,53; 1,09] - [1,53; 1,69]
5	Line: [1,53; 1,09] - [1,53; 0,49]
6	Link text: CURRENT SEWER SYSTEM DN 500
7	Dimension: 6,00 m
8	Dimension: 7,50 m
9	Rectangle: [-10,83; 5,48] - 6,00 / 1,00
10	Text: ORIGINAL OBJECT SO2
11	Rectangle: [-18,68; 12,73] - 6,18 / 3,27
12	Spline
13	Circle: [-30,18; 3,56] - 2,57
14	Polyline

Multiple editing

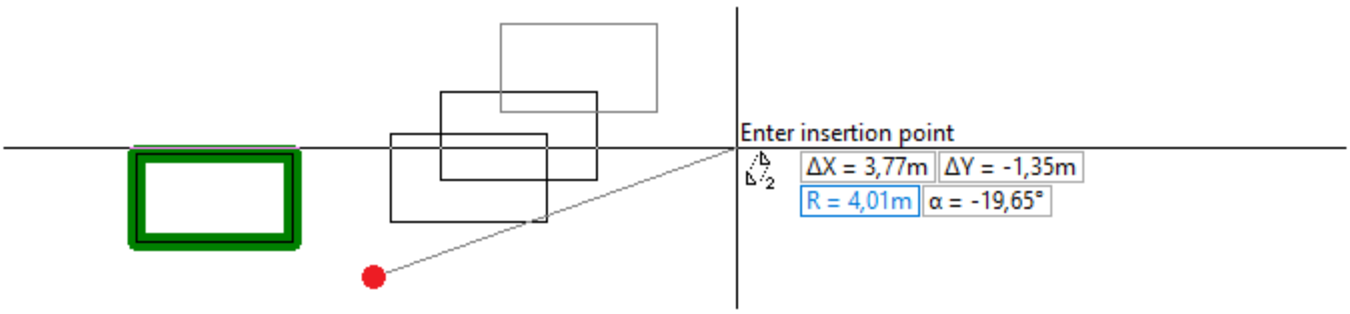
The objects can also be edited on the desktop using the mouse - "Graphic editing".

Tools

The tools allow fast and easy edit of objects. There are frequent editing steps carried out: (the original object is in bold)

Multiple copying

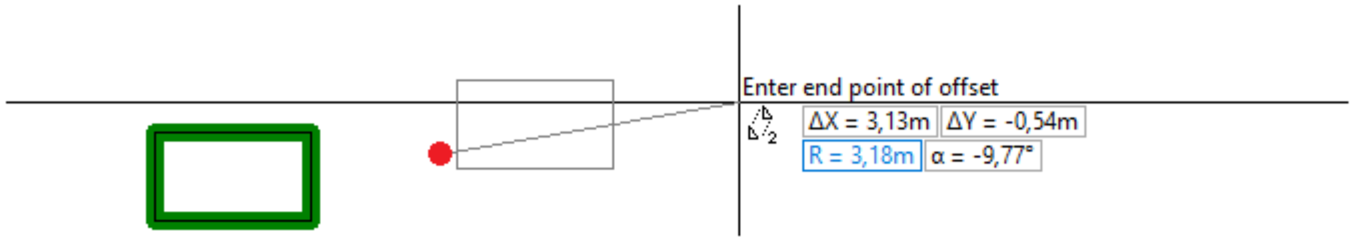
- enter the reference point (colored in red in the picture)
- enter the insertion point (repeatedly according to the needed number of copies of the object)
- end the entering (the right button of the mouse or ESC)



Multiple copying of objects

Moving

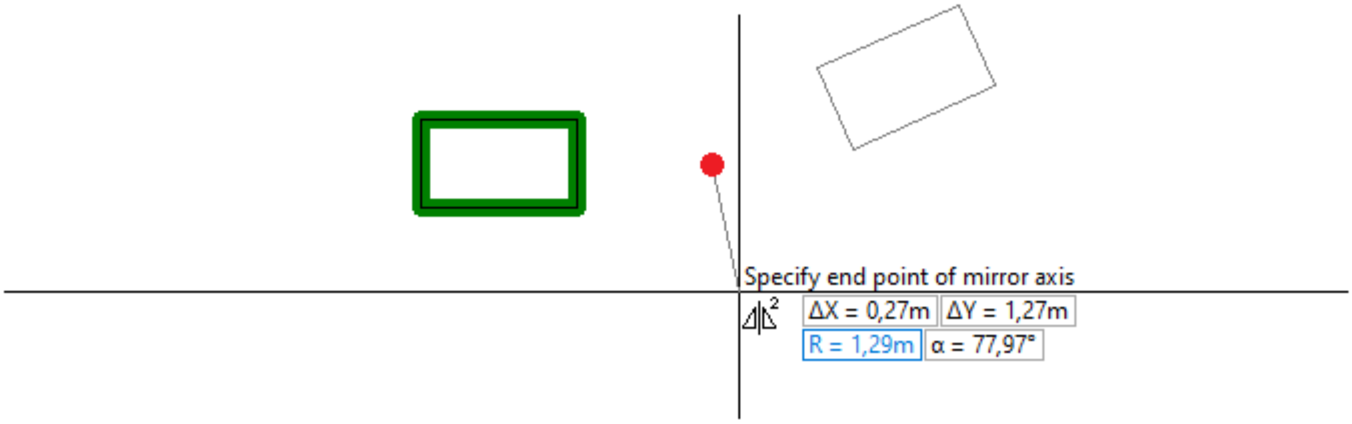
- enter the starting point of the displacement (colored in red in the picture)
- enter the end point of the displacement
- **Advice:** An object can be moved by using the left button of the mouse



Moving the object

Mirroring

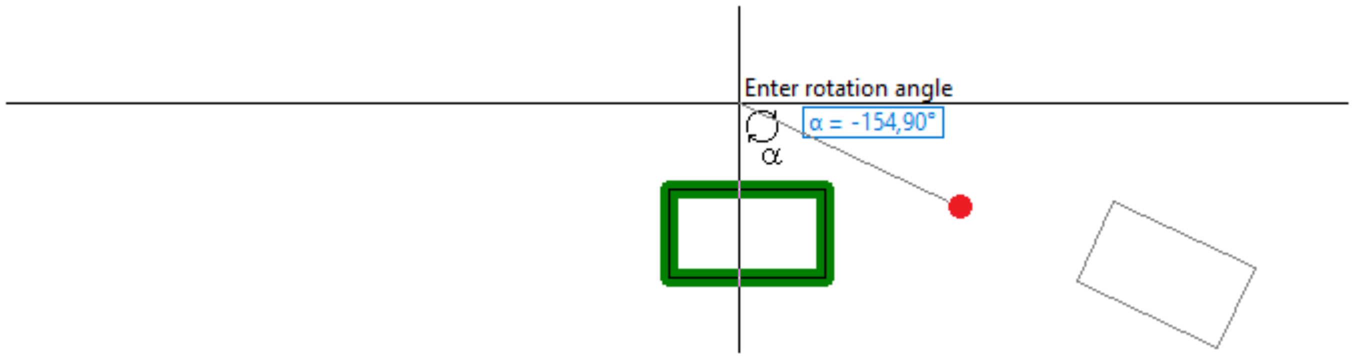
- enter the starting point of the mirror axis (colored in red in the picture)
- enter the end point of the mirror axis



Object Mirroring

Rotating

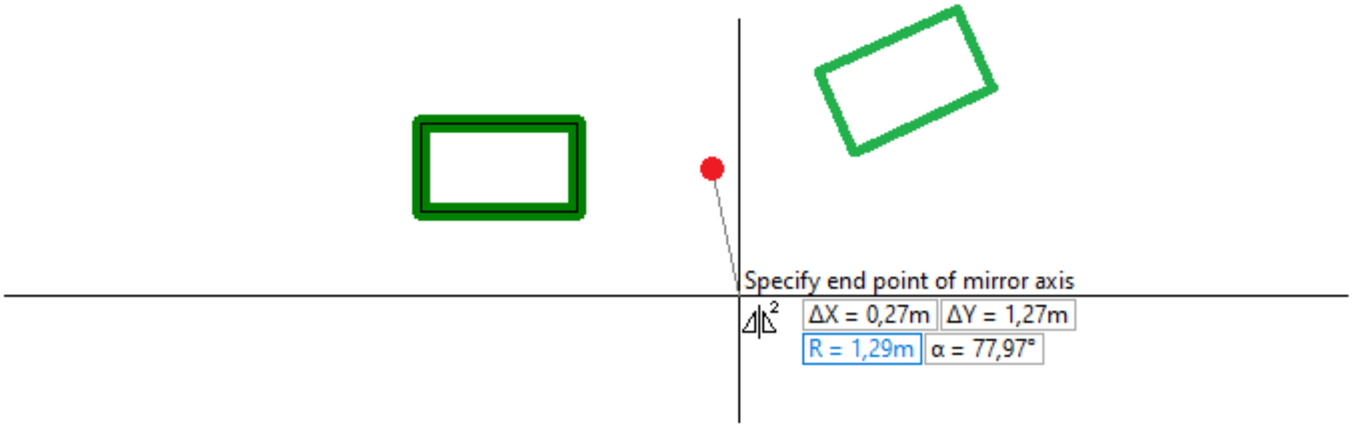
- enter the middle of the rotation (colored in red in the picture)
- enter the rotation angle



Object rotation

☑ Copy Copy of the object

- if the option is marked, the original object remains in the same place, and a copy is created (at moving, mirroring and rotating of the object)



Object Mirroring with a copy

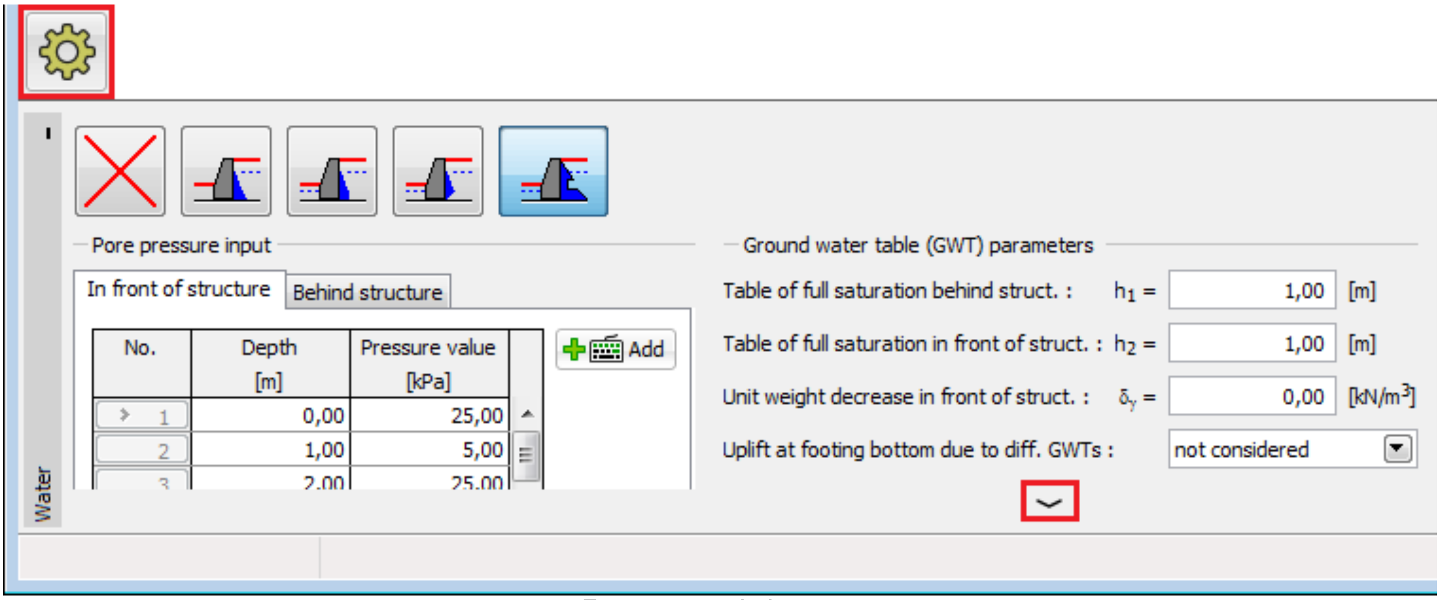
Frames

The Frame is a permanently opened window in the bottom part of the application window. Frames are changed depending on the selected input data mode of a given task selected from the "Frames" control bar and using the button on the "Drawing Settings" control bar. The Frame may contain the following items: table, combo list, fields for data input (h_1 , h_2 ….), and command buttons.

When selecting data by using the keyboard, use the "**Tab**" function key together with cursor arrows for moving within the selected element (for example, combo list). The selection of the checkboxes is made by the space button. When selecting by using the mouse, we use the left mouse button.

The buttons that open dialog windows ("**Add**", "**Add in dialog**", and "**Add graphically**") can only be controlled by the left mouse button.

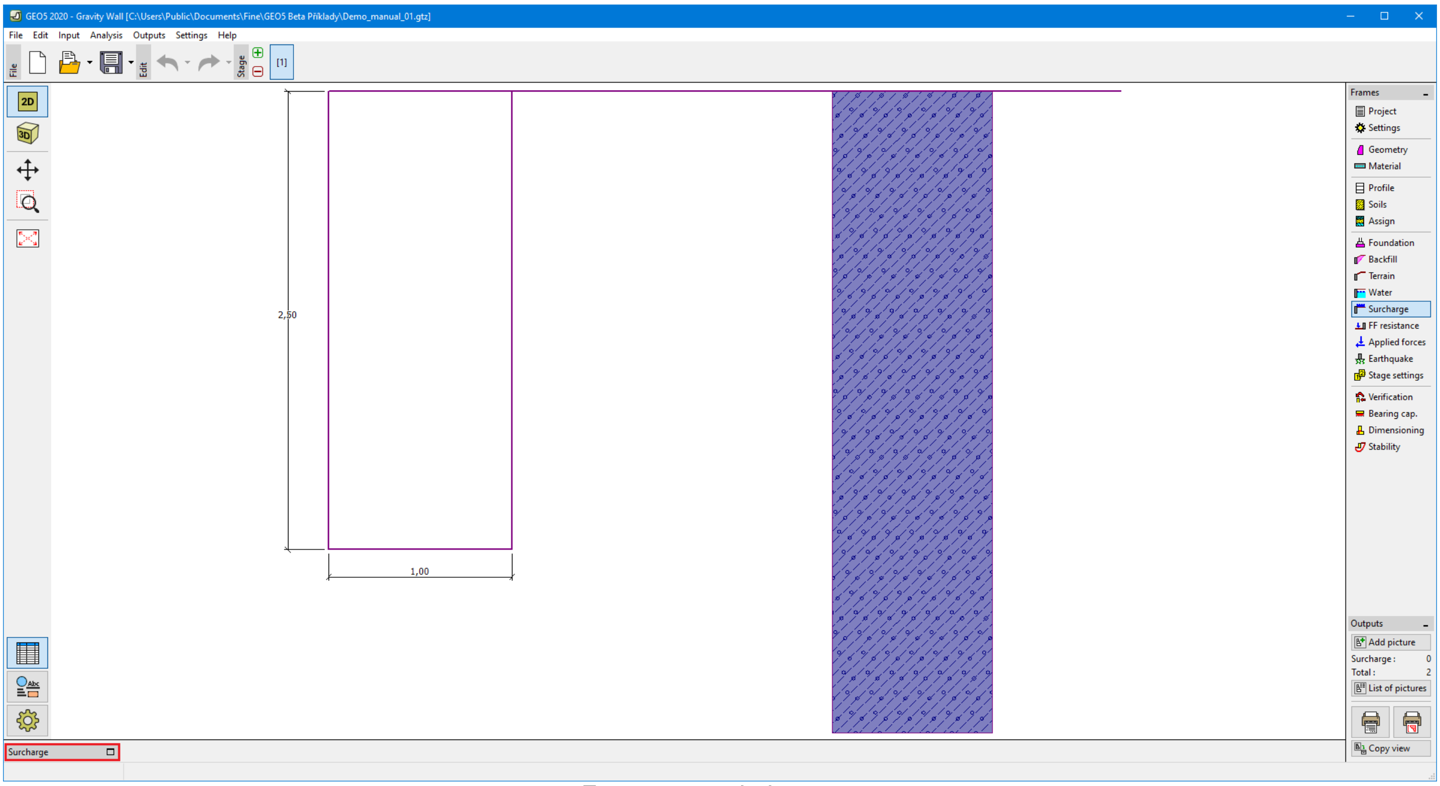
If the frame window is not wide enough (or high enough) so that all elements could be visualized, it is possible to move the frame in the vertical or horizontal direction using the buttons.



Frame control elements

The Frame can be minimized by using the button in the upper left corner. In this case, the frame space is taken by the drawing space. The height of the frame can also be changed by clicking the left mouse button on the upper frame edge and dragging upwards or downwards. It is recommended for the use of active dimensions and active objects.

To maximize the frame back, press the button in the left bottom corner of the desktop showing the frame name.



Frame control elements

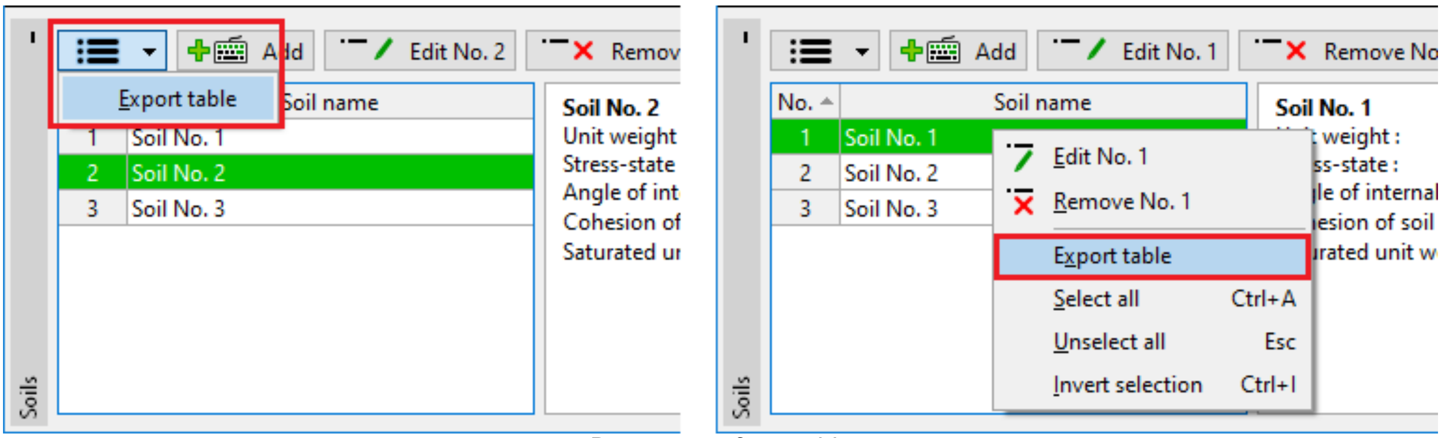
Tables

A table is a list of input data (for example, a list of surcharges, soils, profile interface etc.). The table header contains a list of items (surcharge, name, width, size…).

Data in some tables can be **sorted by columns** (by clicking on the right part of the column head - with an arrow). This way, we can sort the data in the table **alphabetically** or by **value**.

Special table options

Special table features are available using the "**Options**" (hamburger) button . This button is visible in all GEO5 programs tables. Usually, the **export of data** (in XLSX, XLS, ODS, CSV, and HTML formats) is available here, but some tables have additional features. These **special features can be also shown** using the right mouse button on the table header.

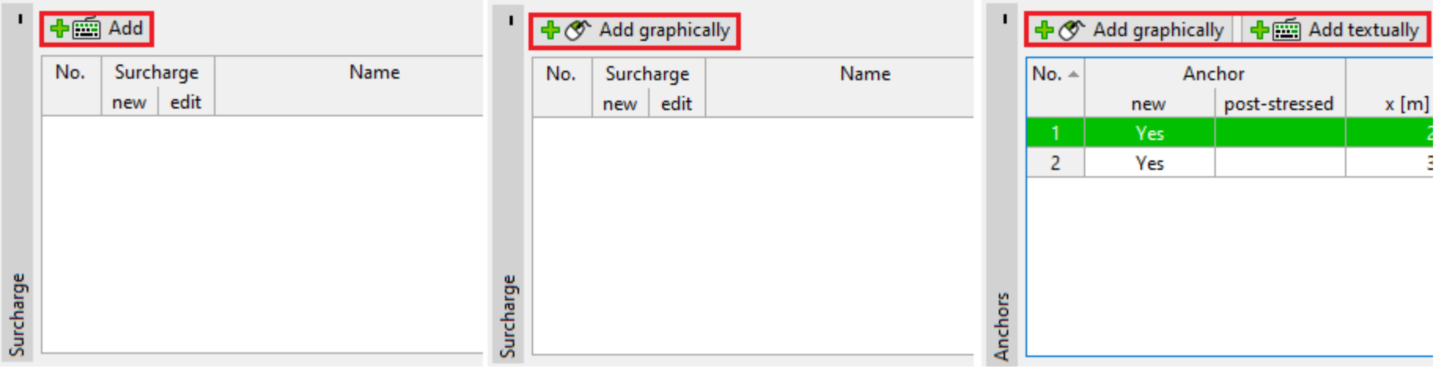


Data export from table

Adding new items to the table

The form of the table depends on the selected frame. Graphical bonds between the items in the table and on the desktop are important. Some data (objects) can only be input textually, others also graphically by using the mouse. If the program enables both textual and graphical input, both buttons are shown in the table.

If the table still **does not contain any items**, or **no item is selected in the table**, only the "**Add**", "**Add textually**" or "**Add graphically**" buttons are visible above the table. Using these buttons, new items can be added into the table.

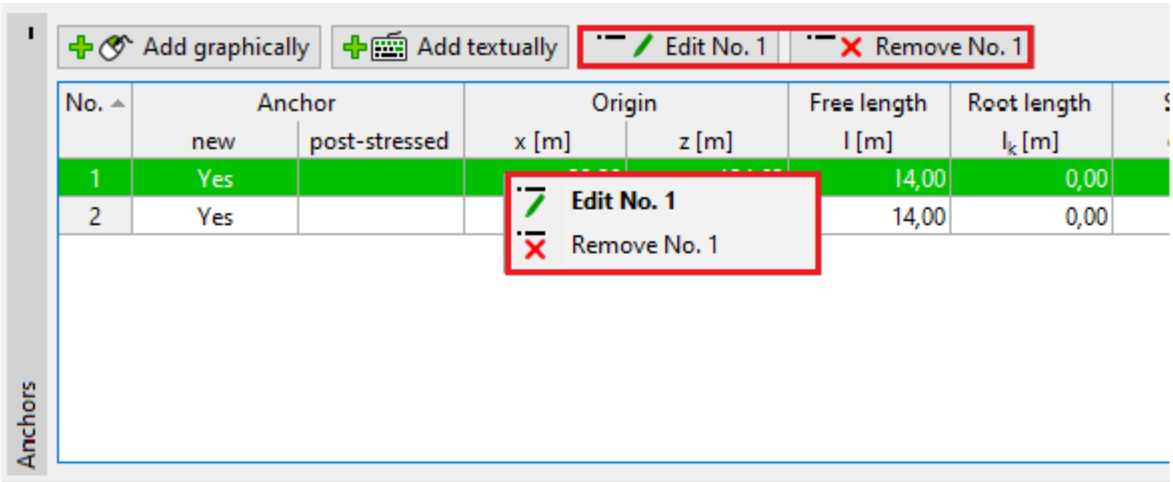


Adding a new item into the table

Editing table items

If a row is selected (highlighted green), the "**Edit**" and "**Remove**" buttons are activated above the table. The **number of items** is given in the brackets. Using these buttons, individual rows can be edited.

The selection of items is made by pressing the left mouse button. Clicking the right mouse button opens the **context menu**.

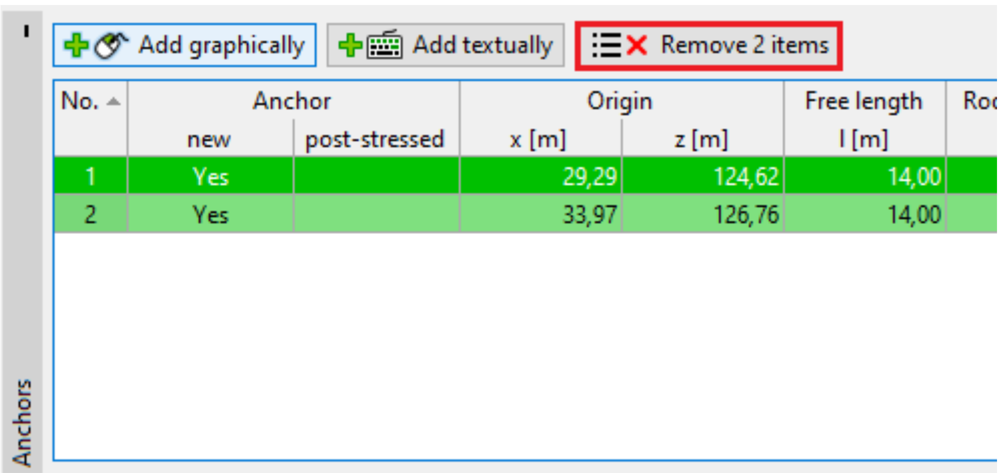


Editing table items

If the operation **enables** it (e.g. **removal of an item** from a table), it is possible to edit more items (rows) at a time. There are two selected rows on the figure below. From the picture, we can see that the only option, in this case, is the removal of both items from the table, and therefore, the "**Edit**" button is not available.

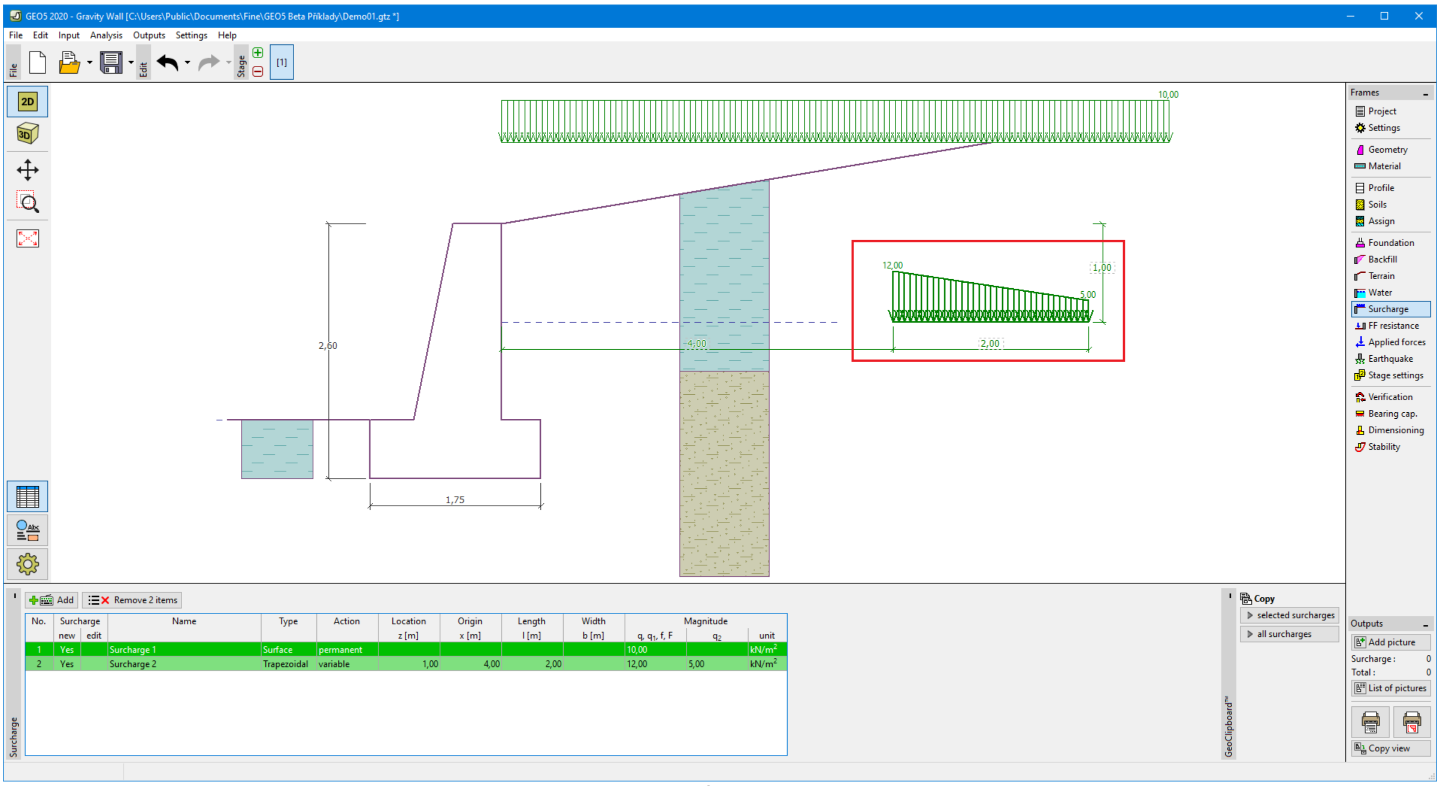
The selection of more items (rows) in a table is made by holding CTRL and clicking the left mouse button. By holding **SHIFT** and clicking the left mouse button, all rows above or below the selected rows are selected.

By clicking outside of the selected table rows, the selection is canceled.



Editing more table items

The selection of rows in the table corresponds with the visualization of the objects on the desktop (and reverse). If a row or multiple rows are selected in the table, the relevant objects on the desktop are highlighted with the same color. If the mouse marker is over one of the objects, the objects are shown **bold**. After clicking on "**Remove**", the objects and table rows are shown **red**.



Visualization of selected objects

Marking objects by using these colors is implicitly set. This setting, however, can be modified in the "Drawing styles administrator" dialog window.

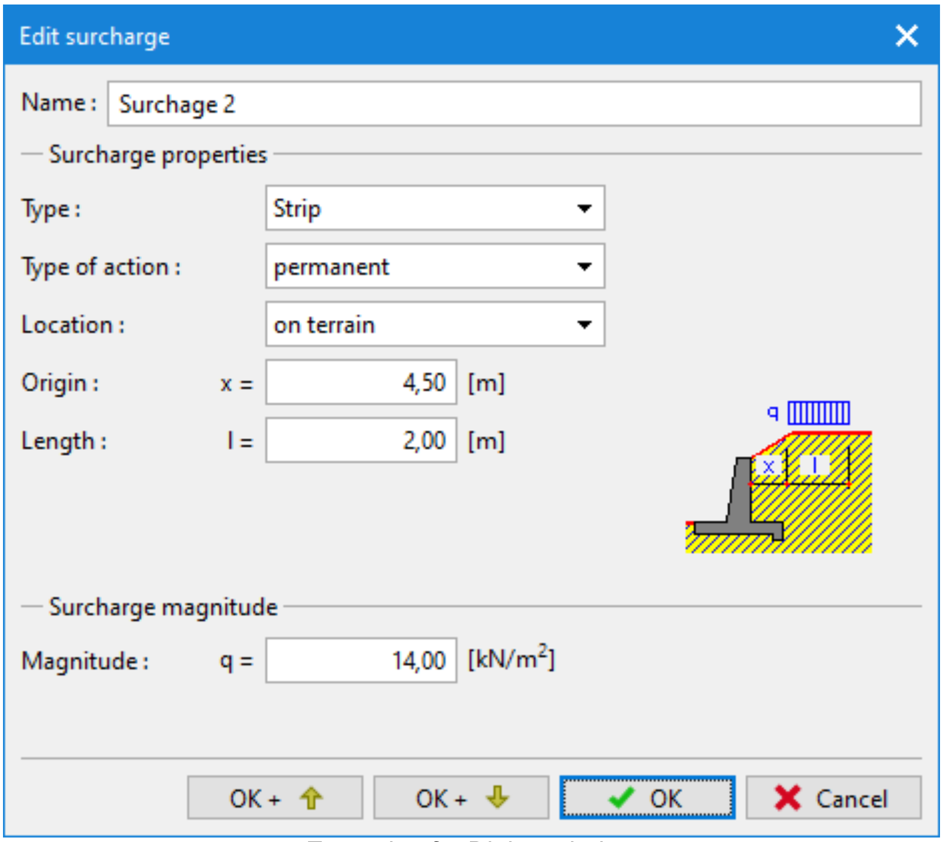
Dialog Windows

A dialog window is one of the elements that allows inputing data into the program. In all GEO5 programs, dialog windows apply to conventional windows management typical for the Windows environment. A left mouse button is used when selecting objects in the window or alternatively the "**Tab**" function key when using the keyboard. When moving inside an object (for example input field) use the arrow buttons and the "**ENTER**" key.

A dialog window can contain the following items: table, combo list, fields for inputting data (number, text) and command buttons. The "**OK**" command button confirms the selection, while the "**Cancel**" button leaves the input mode.

Providing the window contains a certain non-typical control element (or this element has some other than typical effect) its function is described in the corresponding data input regime.

As an example consider the following picture showing the "**Edit surcharge**" dialog window that contains the "**OK+**" and "**OK+**" buttons. These buttons allow the user to move within the list of input surcharges and at the same time to confirm changes made in the window.

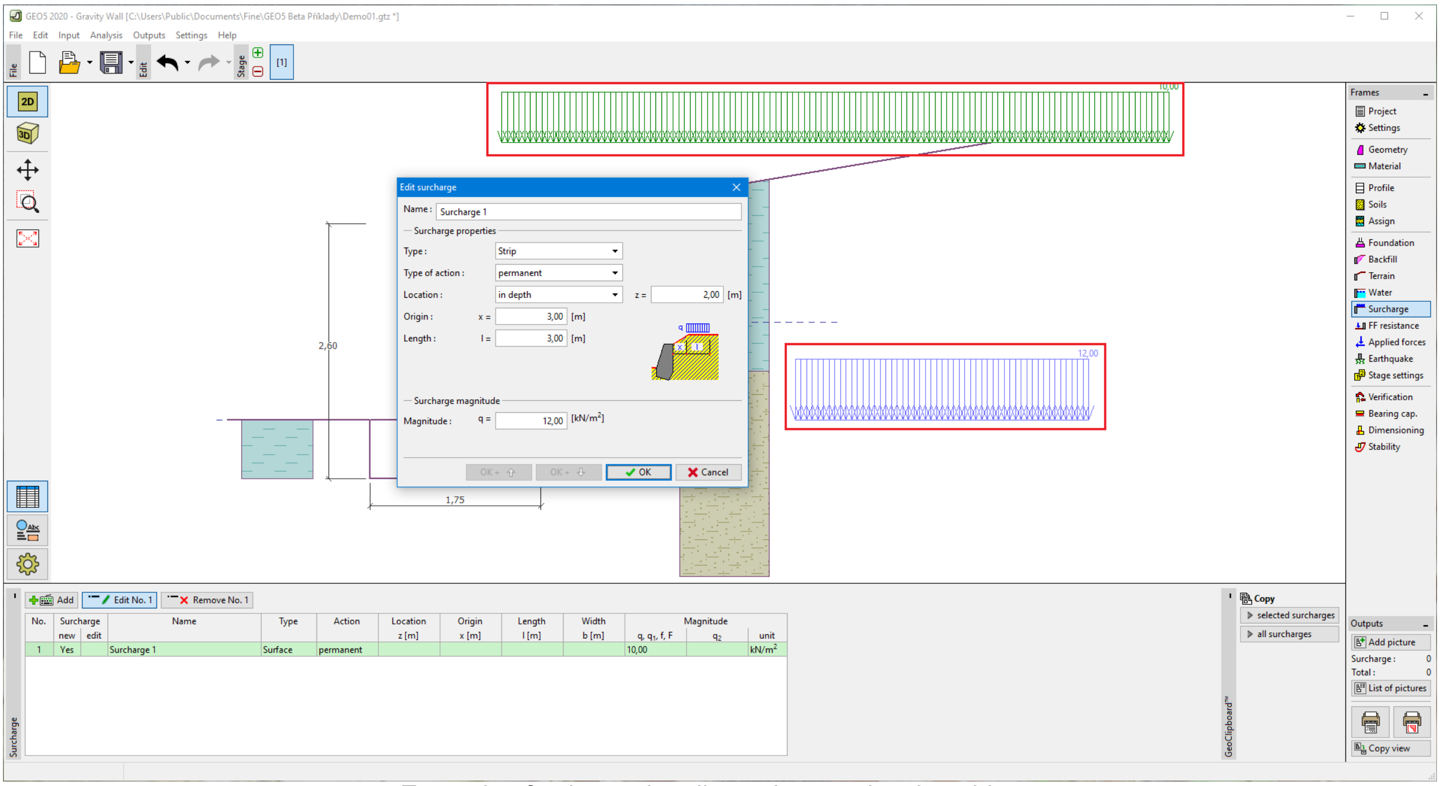


Example of a Dialog window

Active Dimensions and Objects

The system of active dimensions and objects allows faster editing of input data.

- **Active dimension** is a dimension that can be edited directly on the desktop. The value of the active dimension is labeled by a frame (dashed line). Clicking the left mouse button on the value then changes the frame view (is plotted by a solid line), the cursor starts to blink, and the dimension can be edited. The "**Enter**" button closes the editing mode. The change is immediately displayed on the desktop.
- **Active object** functions in a similar way. Clicking the object (double click) then activates the editing mode. In this case, however, the values are not edited directly on the desktop, but rather in the dialog window originally used to create the object. The picture shows an example of an active object (surcharge), when clicking on the desktop opens the "**Edit surcharge**" dialog window. With active objects, it is also possible to use the context menu option.

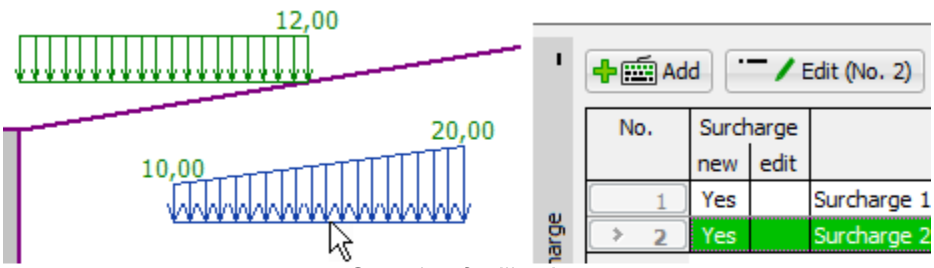


Example of using active dimensions and active objects

Mouse Functions

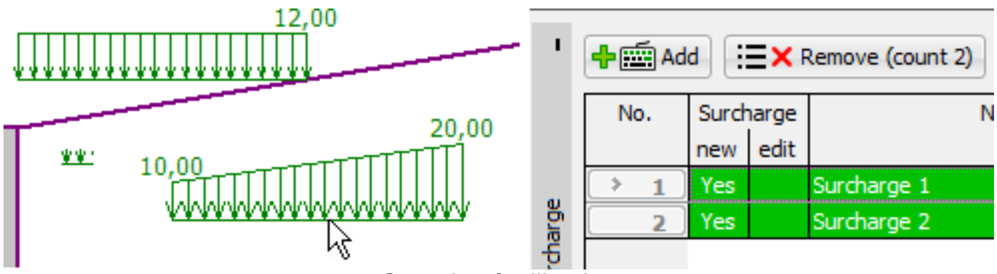
As well as other Windows applications, GEO5 programs use the mouse for controls.

Utilization	Mouse Icon	Mouse Button	Function Description
Selection		left	Active object or table row is selected by clicking the left mouse button.



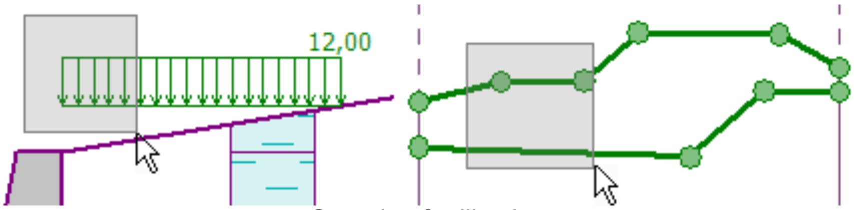
Sample of utilization

Utilization	Mouse Icon	Mouse Button	Function Description
Multiselection		CTRL + left, SHIFT + left	An arbitrary number of active objects or table rows is added to selection (or removed from) by holding **CTRL** and clicking the left mouse button. All rows of a table below or above the selected row are selected by holding **SHIFT** and clicking the left mouse button.



Sample of utilization

Utilization	Mouse Icon	Mouse Button	Function Description
Graphical Selection		left	By holding the left mouse button and dragging the mouse, a highlighted area is created. Active objects in the area (even partially selected) are added to the selection.

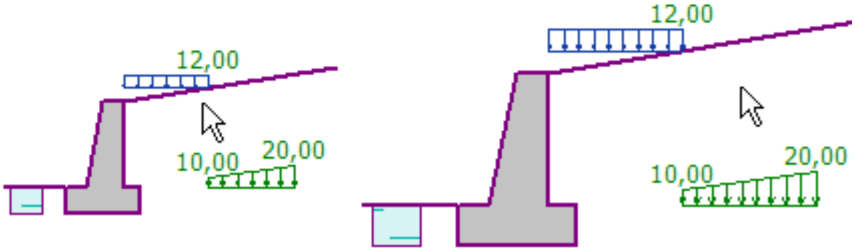


Sample of utilization

Scale Change 

Middle (mouse wheel)

Rotating the mouse wheel zooms in or out, keeping the center of the frame.

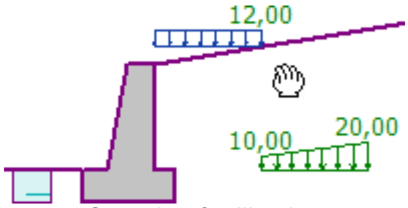


Sample of utilization

Scale Change 

Middle (mouse wheel)

Double-clicking the mouse wheel scales the visualization to the maximum zoom so that all objects are still visible on the desktop.

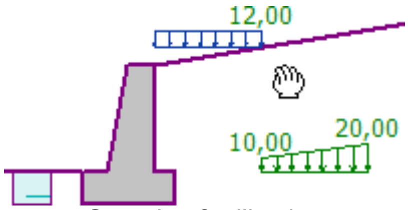


Sample of utilization

Shift 

Middle (mouse wheel)

By pressing and holding the middle mouse button (wheel), and moving the mouse, we can shift the view in any direction.

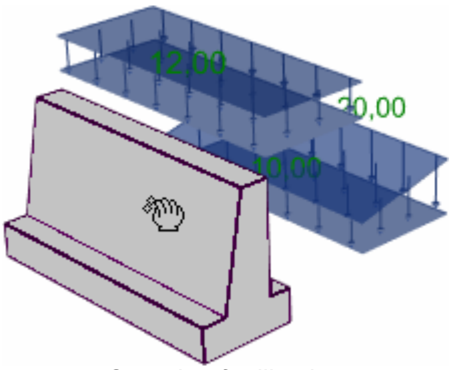


Sample of utilization

Rotation 

Middle (mouse wheel)

Holding the **CTRL** button and pressing the middle mouse button (wheel) will rotate the 3D visualization scene.

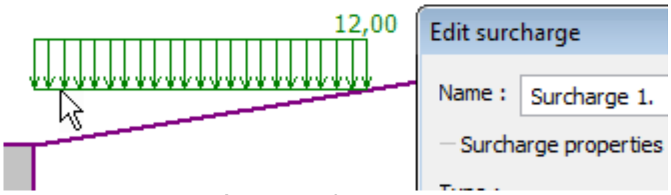


Sample of utilization

Open a Dialog Window 

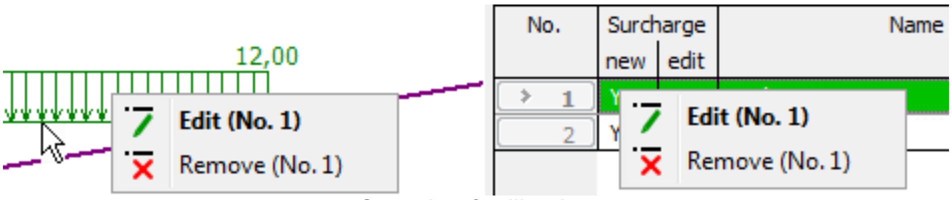
left

Double-clicking the left mouse button on an active object or table row opens the relevant dialog window.



Sample of utilization

Context Menu		right	Clicking the right mouse button on an active object or table row opens the context menu.



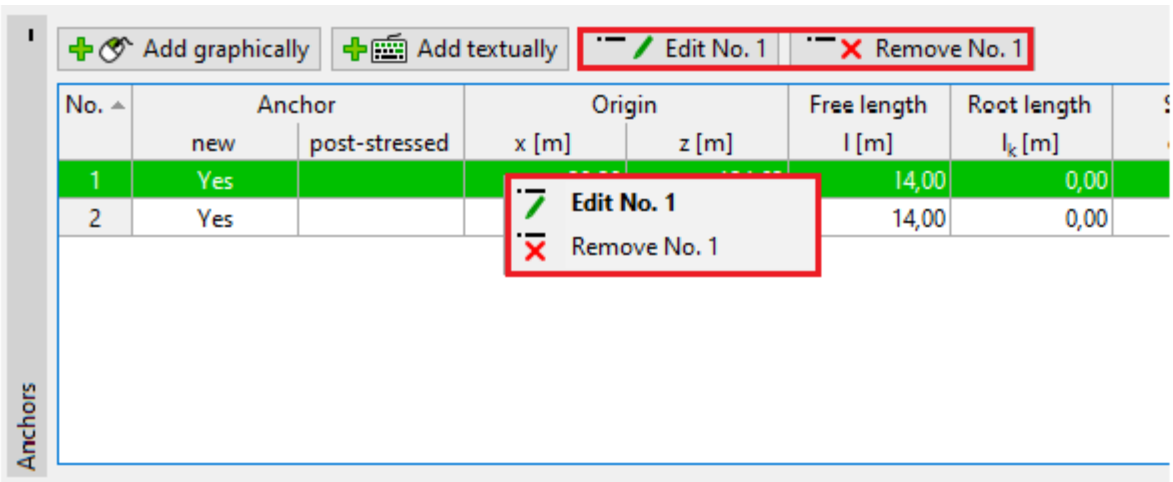
Sample of utilization

Mouse Context Menu

Programs have an implemented context menu of the right mouse button. Context Menu opens by clicking the right mouse button on an object or table row.

Context menu when editing tables

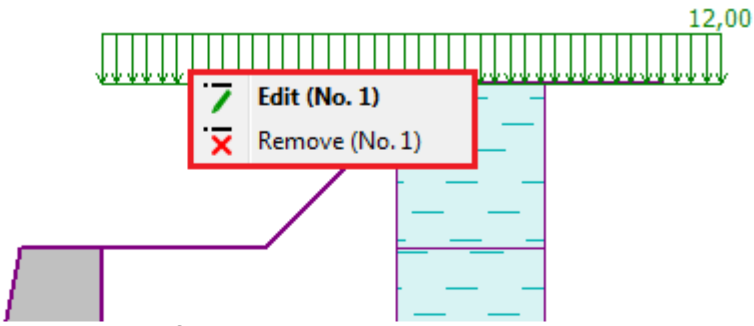
If a row in a table is selected by the right mouse button, the context menu will appear. The number in the brackets next to individual items shows the number of the edited objects. The required item in the context menu can be selected by either left or right mouse button.



Context menu when editing tables

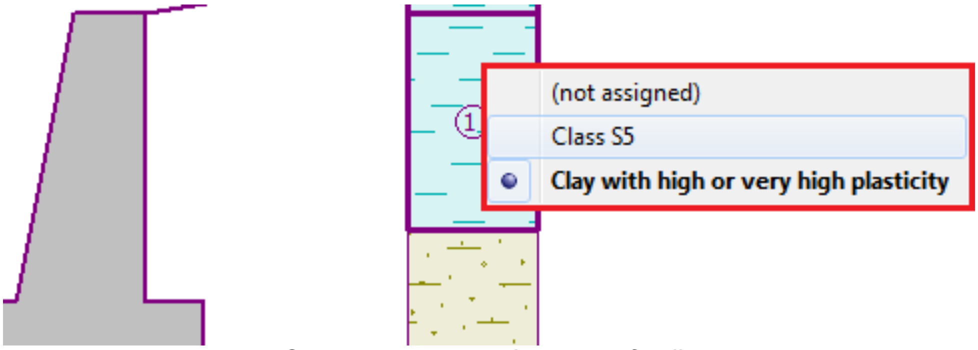
Context menu when editing objects

If an object is selected by the right mouse button, the context menu will appear. The number in the brackets next to individual items shows the number of the edited object (corresponds with the number of the table item). The required item in the context menu can be selected by either left or right mouse button.



Context menu when editing objects

Context menu can also be used for the assignment of soils.

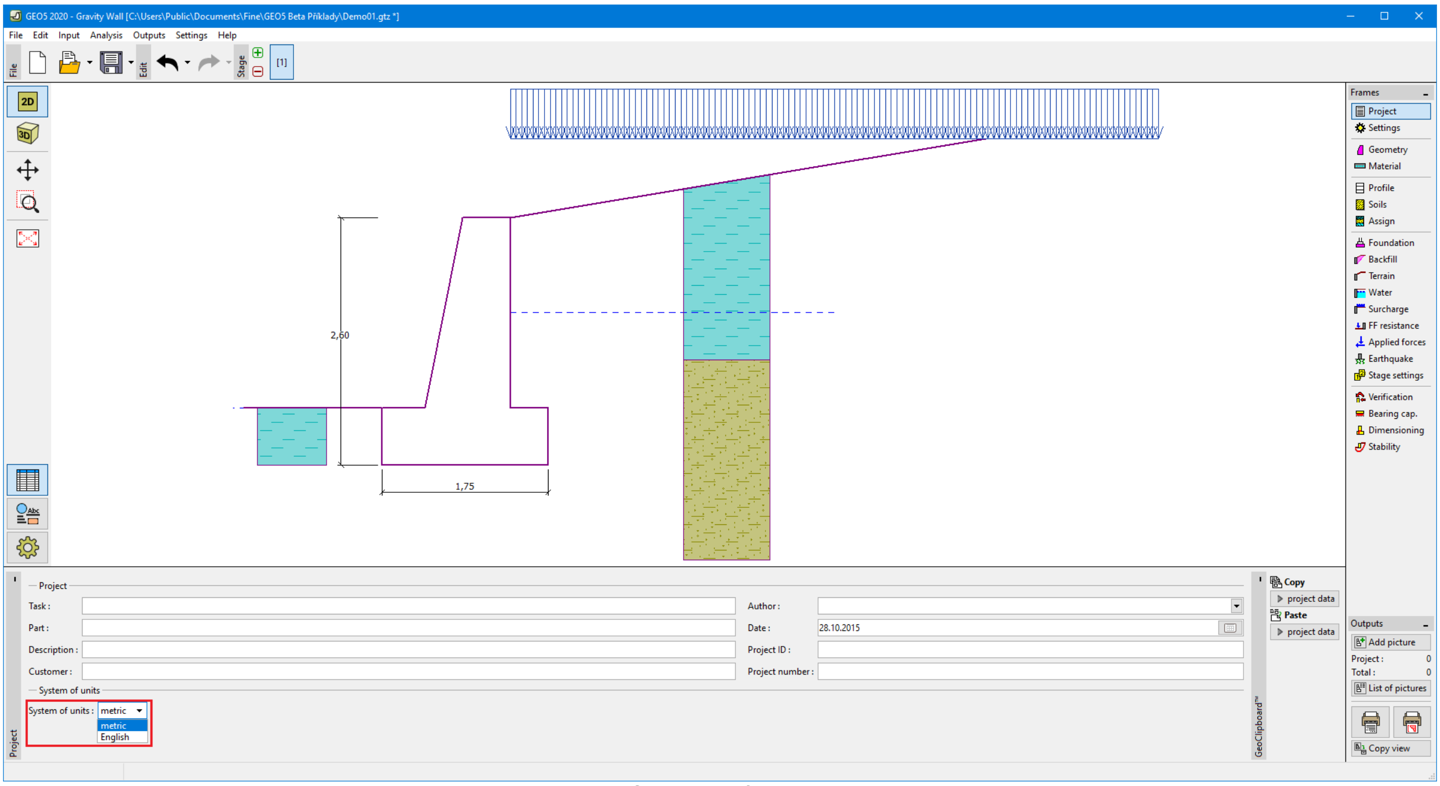


Context menu - assignment of soils

Alternatively, it is possible to use active dimensions and objects.

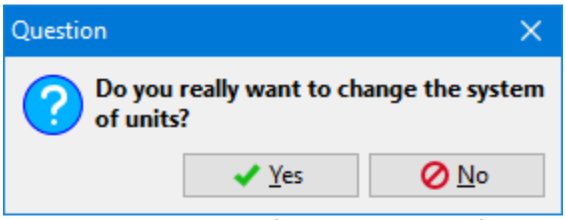
Units - Metric/Imperial

The program allows selecting either **metric or imperial** units in the "**Project**" frame.



Change of units

Use the combo list to select the desired type of units. A prompt message appears requesting to confirm the selection.



Dialog window to confirm the change of units

Copy to Clipboard

The program allows using the Windows clipboard in two different ways:

- It is possible to copy the current desktop view. The picture can then be inserted into an arbitrary editor (MS Word, Paintbrush, Adobe Photoshop, etc.). Copying the pictures to clipboard can either be done by using the control menu (items "**Outputs**", "**Copy picture**") or by the button on the "Outputs" control bar. The settings of parameters are defined in the "Options" dialog window, "Copy to clipboard" tab.

- It is possible to copy the program input data (soil parameters, profile and interfaces, surcharges, water impact, terrain, etc.). The copied data can then be pasted into the same or another GEO5 program as a new project. Copying to the clipboard can be either done using the control menu (items "**Edit**", "**Copy data**", "**Paste data into new project**") or using the button on the "Edit" toolbar.

It is also possible to use a special GeoClipboard™ of the GEO5 software, which enables data transfer between input modes or construction stages of one or more programs.

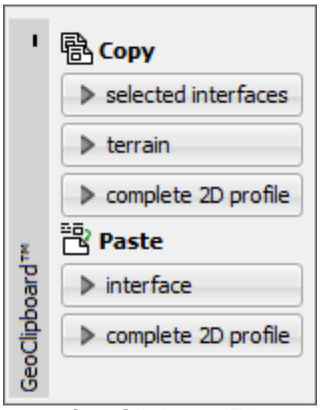
GeoClipboard™

GeoClipboard™ is a special clipboard used in GEO5 software. It allows copying and paste data between modes and stages of one or more programs.

Basic characteristics of the **GeoClipboard™** are:

- GeoClipboard™ can simultaneously contain different data, e.g. copying soils after interfaces doesn't remove interface data
- data is saved after program exit and computer restart till they are replaced by other data from the same data category
- every computer user has its own GeoClipboard™
- during pasting data form GeoClipboard™ the preview of changes is always shown, and the user can change pasting parameters

GeoClipboard™ controls are always placed into the relevant frame. It looks like this:



GeoClipboard™

GeoClipboard™ is implemented for the following data:

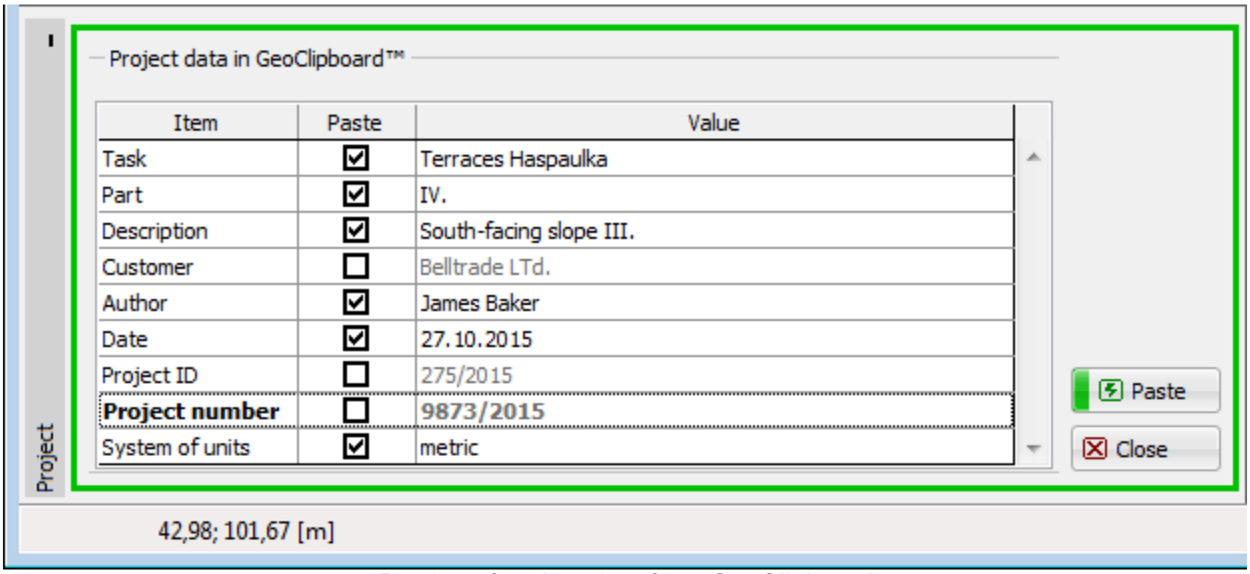
- project data
- 2D interfaces, including copying the analyzed GWT and topology of stages in FEM
- soils and rigid bodies
- 2D assignment
- field tests

Copying and Pasting Project Data

GeoClipboard allows copying and pasting project data.

When copying, all entered non-empty data is copied.

Pasting of project data is done in the following window:



Pasting of project data from GeoClipboard

In this window, it can be specified which project data is pasted ("**Paste**" column). Pasting is done, and data are changed when the "**Paste**" button is pressed.

Copying and Pasting 2D Interfaces

GeoClipboard allows us to copy and paste 2D interfaces between the following modes:

- Interface
- Embankment and Earth cut
- Water
- Incompressible Subsoil

It also allows us to copy data from the FEM program:

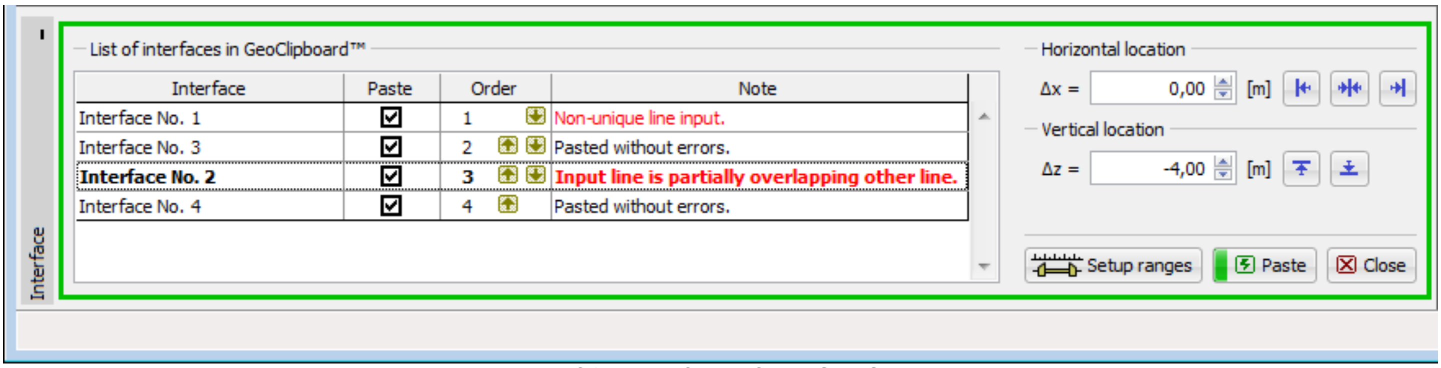
- in "**Analysis**" regime it allows us to copy analyzed GWT, especially after water flow analysis
- in "**Activity**" regime it allows us to copy interfaces concerning active and inactive areas

It is possible to copy the following items:

- the current interface
- selected interfaces
- the terrain of the current stage
- the complete 2D profile

Pasting of interfaces

Pasting interfaces from GeoClipboard is the same process as if the user enters interfaces step by step. The pasting parameters can be entered in the following window:



Pasting of 2D interfaces from GeoClipboard

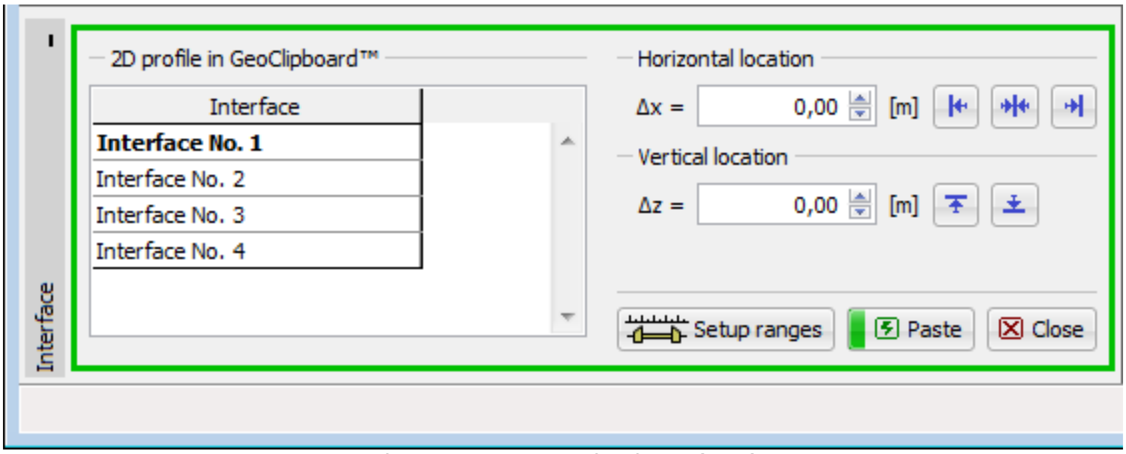
In this window, it can be specified, which interfaces are pasted ("**Paste**" column) and in which order of pasting (change by mouse click on ⇧ ⇩ or by **Ctrl**+**Shift**+**up or down arrow** keys) and locate it to the desired place. In the same mode, there is a possibility to paste only the standalone interface, thereafter the "**Paste**" column acts like a radio button, and the "**Order**" column is hidden.

The program always shows a preview of changes caused by pasting, and the result of pasting is described in the "**Note**" column. Data are changed when the "**Paste**" button is pressed.

There is a possibility to repeatedly paste one interface with a changed location to enter the skew-layered profile.

Pasting of complete 2D profile

This mode allows the pasting of a complete 2D profile from GeoClipboard to the current 2D profile. The interfaces to be pasted cannot be specified, only locating to the desired place can be done. The program shows the preview of the resulting 2D profile. The profile is pasted and the data is changed when the "**Paste**" button is pressed.



Pasting of complete 2D profile from GeoClipboard

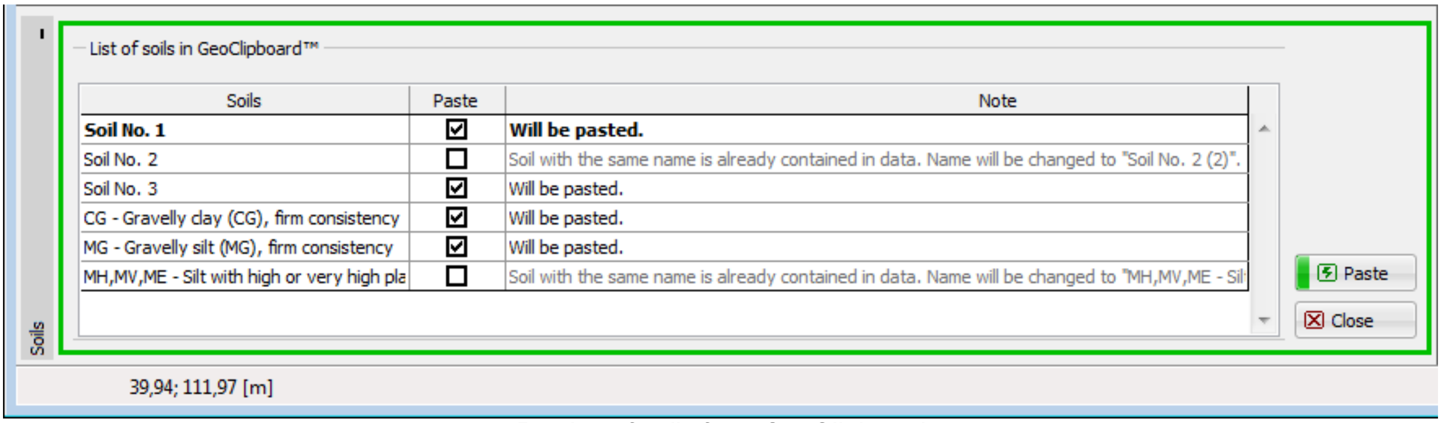
Copying and Pasting Soils and Rigid Bodies

GeoClipboard allows us to copy and paste soils and rigid bodies.

It is possible to copy the following items:

- current soil (rigid bodies)
- selected soils (rigid bodies)
- all soils (rigid bodies)

Pasting of soils (rigid bodies) is done in the following window:



Pasting of soils from GeoClipboard

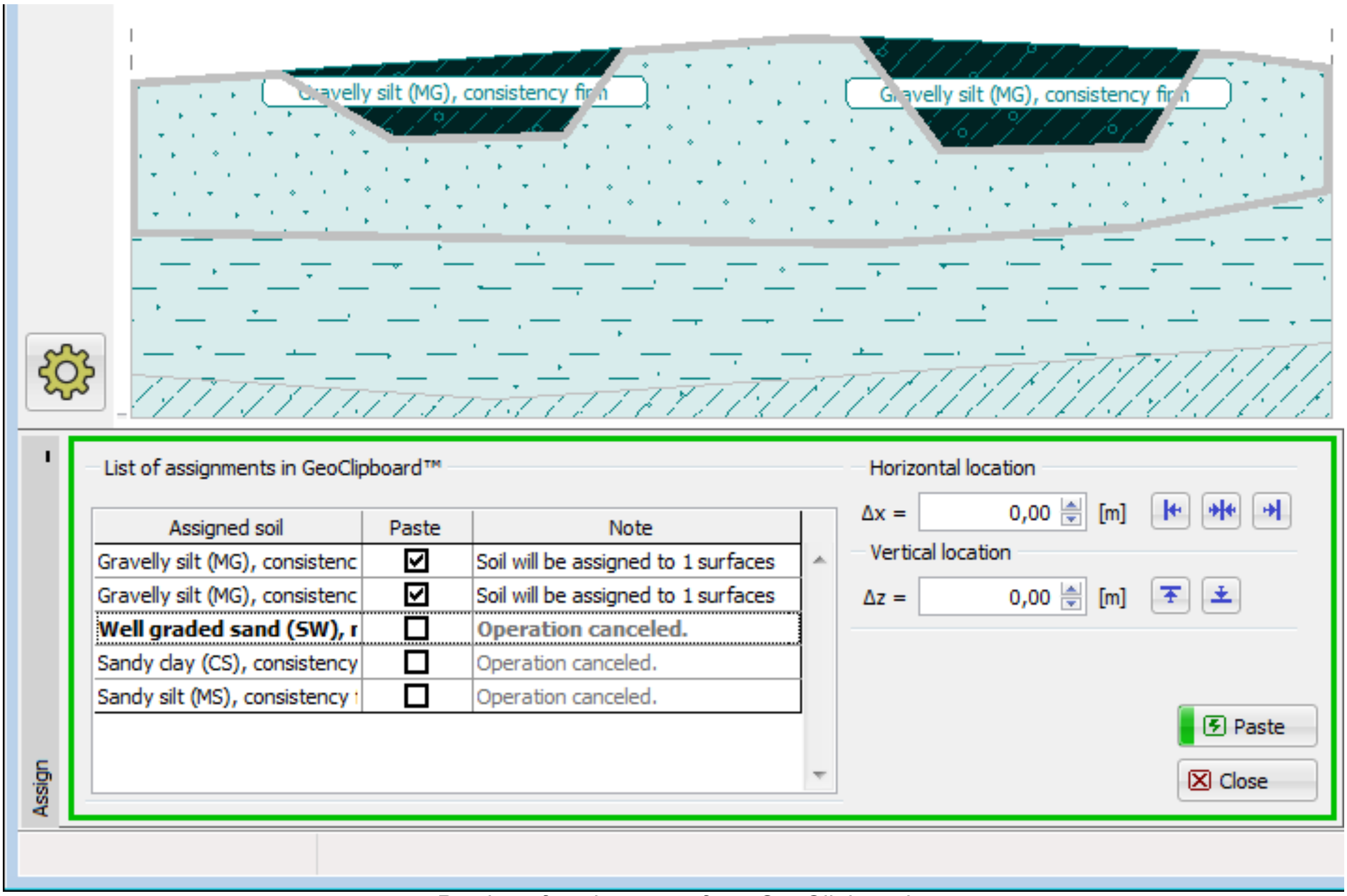
In this window, it can be specified which soils (rigid bodies) are pasted ("**Paste**" column). If the name of the pasted soil (rigid body) is identical to the name of the soil (rigid body) already contained in the data, the program changes the name to be unique. Pasting is done and the data is changed when the "**Paste**" button is pressed.

Copying and Pasting 2D Assignment

GeoClipboard allows us to copy and paste assignments.

While copying the surfaces as polygons with assigned soils are copied to the GeoClipboard.

Pasting of assignments is done in the following window:



Pasting of assignments from GeoClipboard

In this window, it can be specified which 2D assignments of soils are pasted ("**Paste**" column). The location of the pasted surfaces can be changed. The resulting state is available as a preview, with marked surfaces and information about newly pasted soil assignments. The assigned surface modifies assignments in data of all surfaces included in it.

If any assigned soils are not included in the data, they are added to the list of soils.

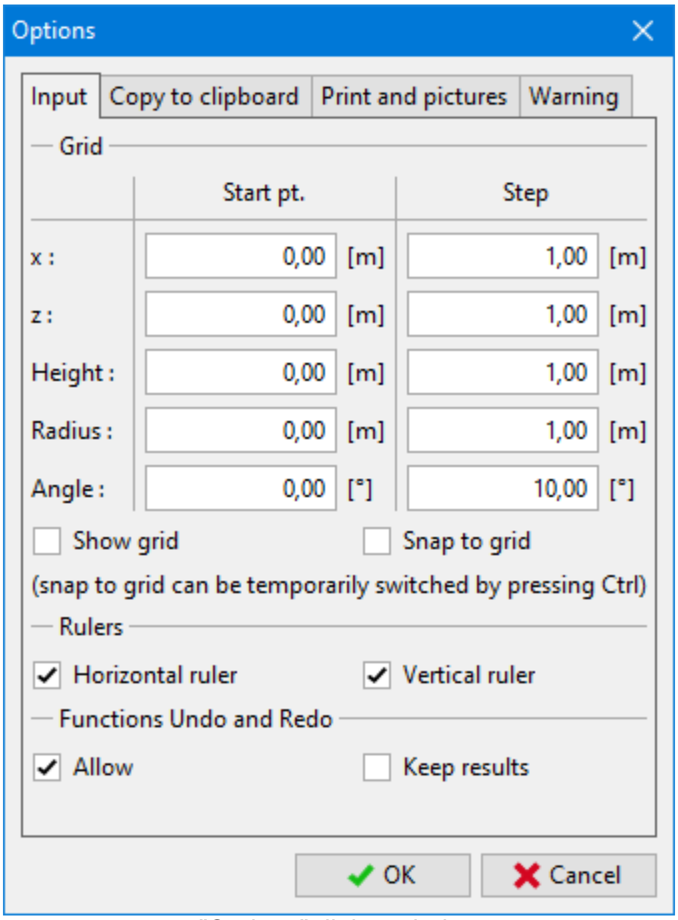
Pasting is done, and data are changed when the "**Paste**" button is pressed.

Options

"**Options**" dialog window allows us to set some of the program's special functions (Input, Copy to clipboard, print view, etc.).

The **"Options"** dialog window is opened from the control menu (items "**Settings**", "**Options**").

The window contains individual tabs (number and content may vary depending on the program), which allows specifying corresponding settings.



"Options" dialog window

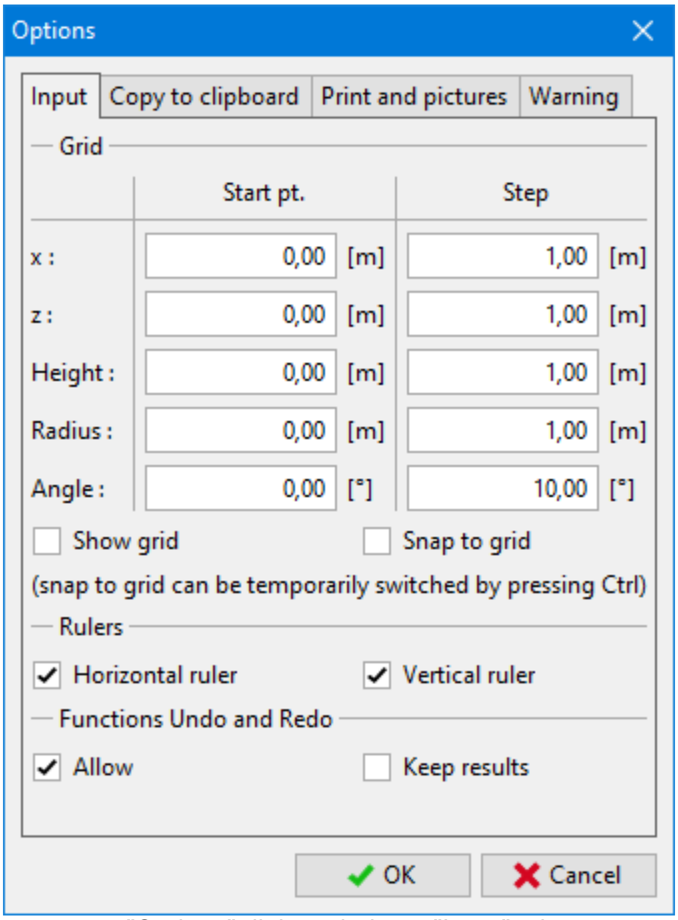
Options - Input

The "**Input**" tab allows setting the "**Grid**" parameters and parameters of "**Undo**" and "**Redo**" functions.

This tab is implemented **only in 2D programs** (FEM, Slope stability, Settlement, Beam, etc.).

Grid	• sets the grid origin and step in the x and z directions
Show grid	• shows/hides grid on the desktop
Snap to grid	• turns on/off the snap to grid option using the mouse (when shifting the mouse, the cursor jumps over the defined grid - a point off the grid can be specified by holding the "**CTRL**" key)
Horizontal rule	• shows/hides horizontal rule with a scale of distances on the desktop
Vertical rule	• shows/hides vertical rule with a scale of distances on the desktop
"Undo and Redo" functions	• turns on/off the possibility of using these functions in the program (on toolbar these buttons are "foggy")

In some modules, it is possible to specify if the results are stored with undo data by checking "**Keep results**". Storing results with undo can be time-consuming. If the results are not stored, it is necessary to make the calculation again after pressing "**Undo**".



"Options" dialog window - "Input" tab

Options - Copy to Clipboard

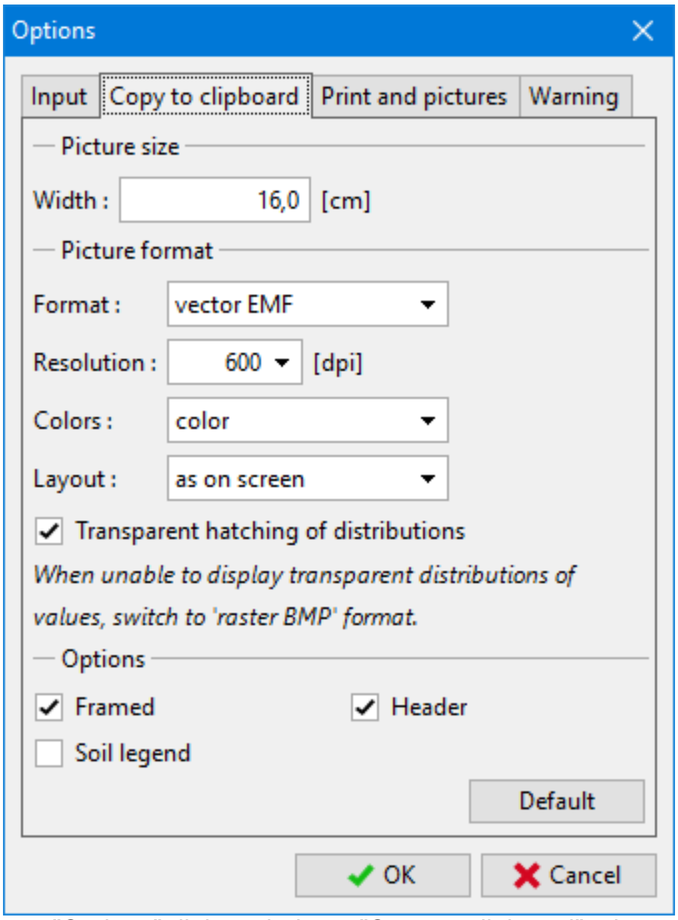
The "**Copy to clipboard**" tab allows setting the controlling parameters:

Picture size
- This setting defines the picture size. Enter the picture width, and the height is calculated automatically according to the picture contents.

Picture format
- This setting defines the picture format (*.EMF, *.WMF, *.BMP), its resolution, color, and layout. The recommended setting is displayed in the figure (format: *.EMF, resolution: 600 dpi, color).

Options
- This setting defines the picture frame and header. If both options are checked, the picture contains both the frame and the header.
- "Soils legend" option adds legend of used soils into the picture.

The "**Default**" button in the window sets original implicit values.



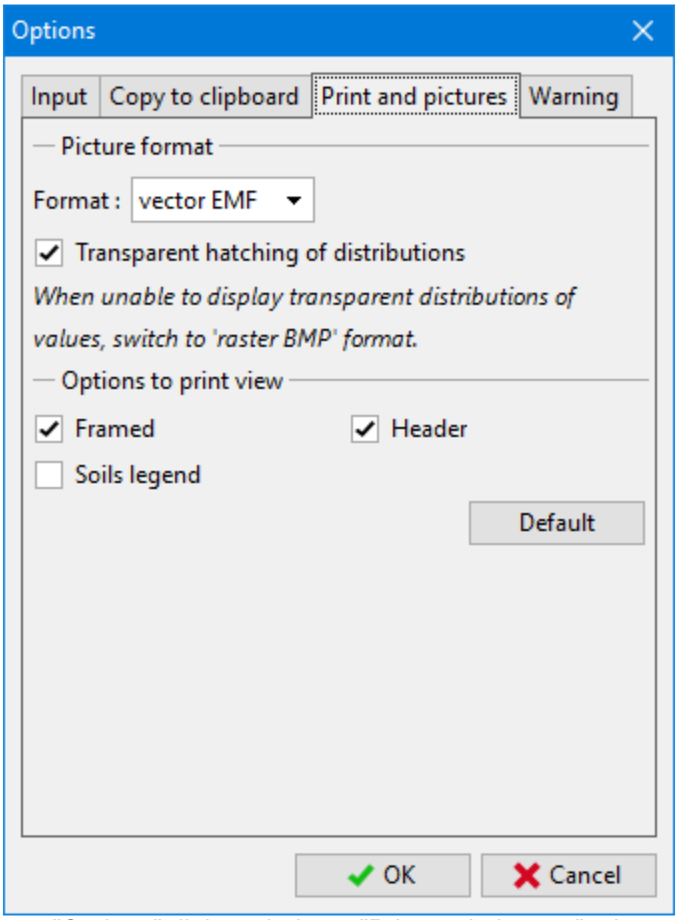
"Options" dialog window - "Copy to clipboard" tab

Options - Print and Pictures

The dialog window opens in the menu ("**Settings**" and "**Options**" items). The "**Print and pictures**" tab allows setting the picture parameters assumed for "Print and export desktop view" and "Print and export document" dialog windows.

Picture format	This setting defines the picture format (*.EMF, *.WMF, *.BMP).
Options to print view	This setting defines the picture frame and header. If both options are checked, the picture contains both the frame and the header. "**Soils legend**" option adds legend of used soils into the picture. This is valid only for the "**Print and export picture**".

The "**Default**" button in the window sets original implicit values.



"Options" dialog window - "Print and pictures" tab

Common Input

This chapter contains explanations common to more GEO5 programs:

- Input and Edit of Soils
- Interface in 2D Environment
- Input of Objects and Data
- Assigning Soils
- Design Coefficient
- Running Several Analyses / Verifications

A very important function in all GEO5 programs is the possibility to define Construction Stages.

Some GEO5 programs allow running other GEO5 programs with automatic data transfer.

Basic functions for work with **graphical outputs** in FEM, Settlement programs are described on the following pages:

- Saved Views
- Setting a Color Range

The following chapters describe **Data Import**:

- DXF Import and Export
- Table Data Import
- Import LandXML

Clipboard functions are described in the following chapters:

- Copy to Clipboard
- Geoclipboard

Input and Edit of Soils

In the "**Add new soil**" dialog window, we input the name of a soil and parameters that should be obtained from laboratory measurements or geological surveys.

All input fields in the window are mandatory. The only exception is the value of $\gamma\ sat$ (unit weight of saturated soil) in the section "**Uplift**". If this field remains empty, the program automatically adds the value of γ (unit weight of soil).

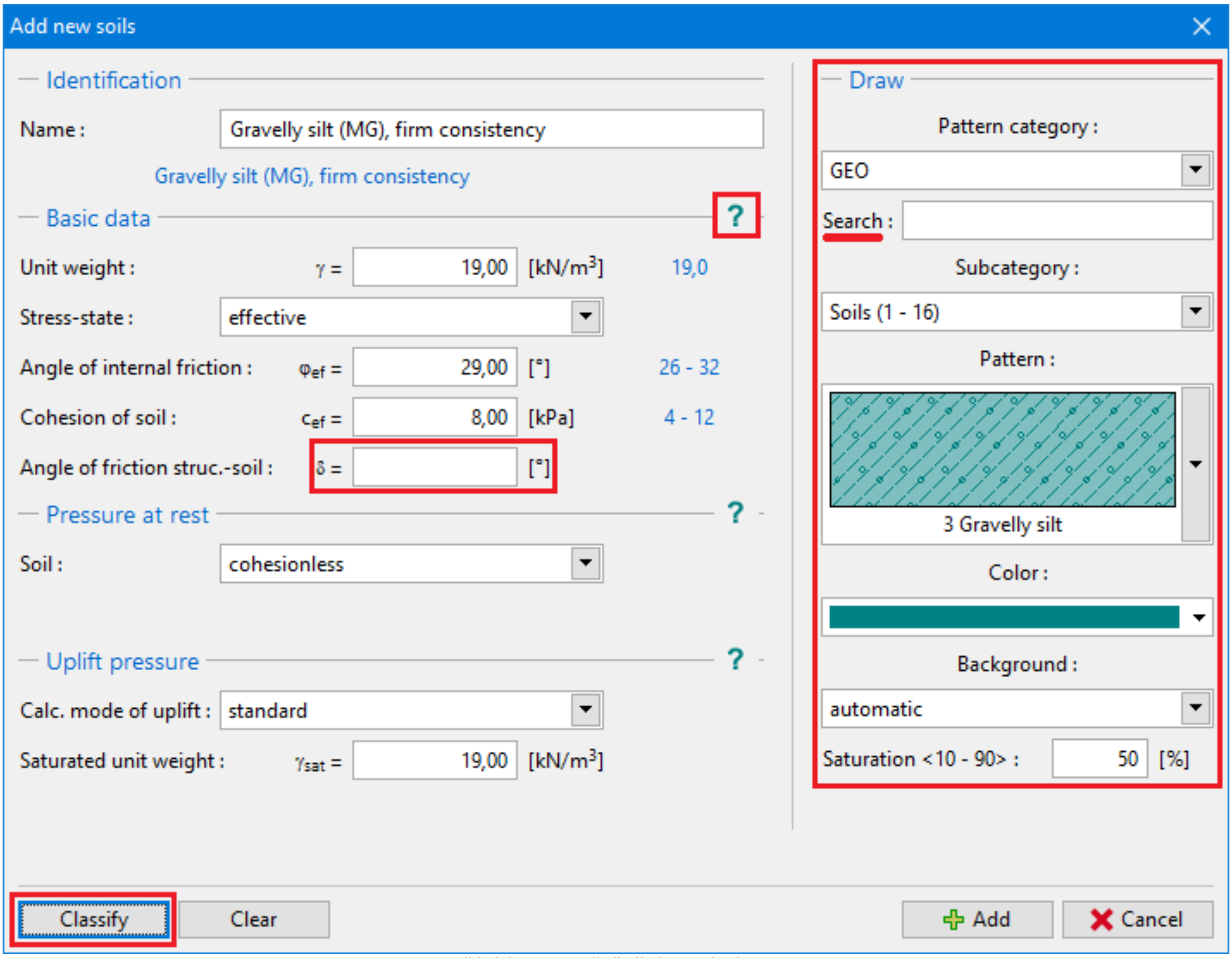
The Hint button "?" provides information about the theory of analyses linked to individual input values.

The Color and pattern category of soil can be set in the combo lists in the right part of the dialog window.

It is possible to search for a specific pattern in each category using a number of patterns (e.g. GEO category - Clay pattern - number 4).

If no geological survey or laboratory experiments are available, the soil can be specified with the help of the soil database containing approximate values of basic characteristics. The "**Classify**" button opens the "Classification of soils" dialog window with values offered to insert into the window. The "**Delete**" button allows removing information about classified soil from the catalog. Soil parameters that do not appear in the catalog ("**Friction angle structure-soil**" in the picture) must be assigned manually in any case. The characteristics of rocks is not listed in the implemented database, these parameters must also be defined manually. Approximate parameters of rocks are presented in the theoretical part of the Help (for calculation of rock stability or for analysis of bearing capacity of foundation on bedrock).

The specified soil is added onto the list of soils by pressing the "**Add**" button.



"Add new soils" dialog window

Classification of Soils

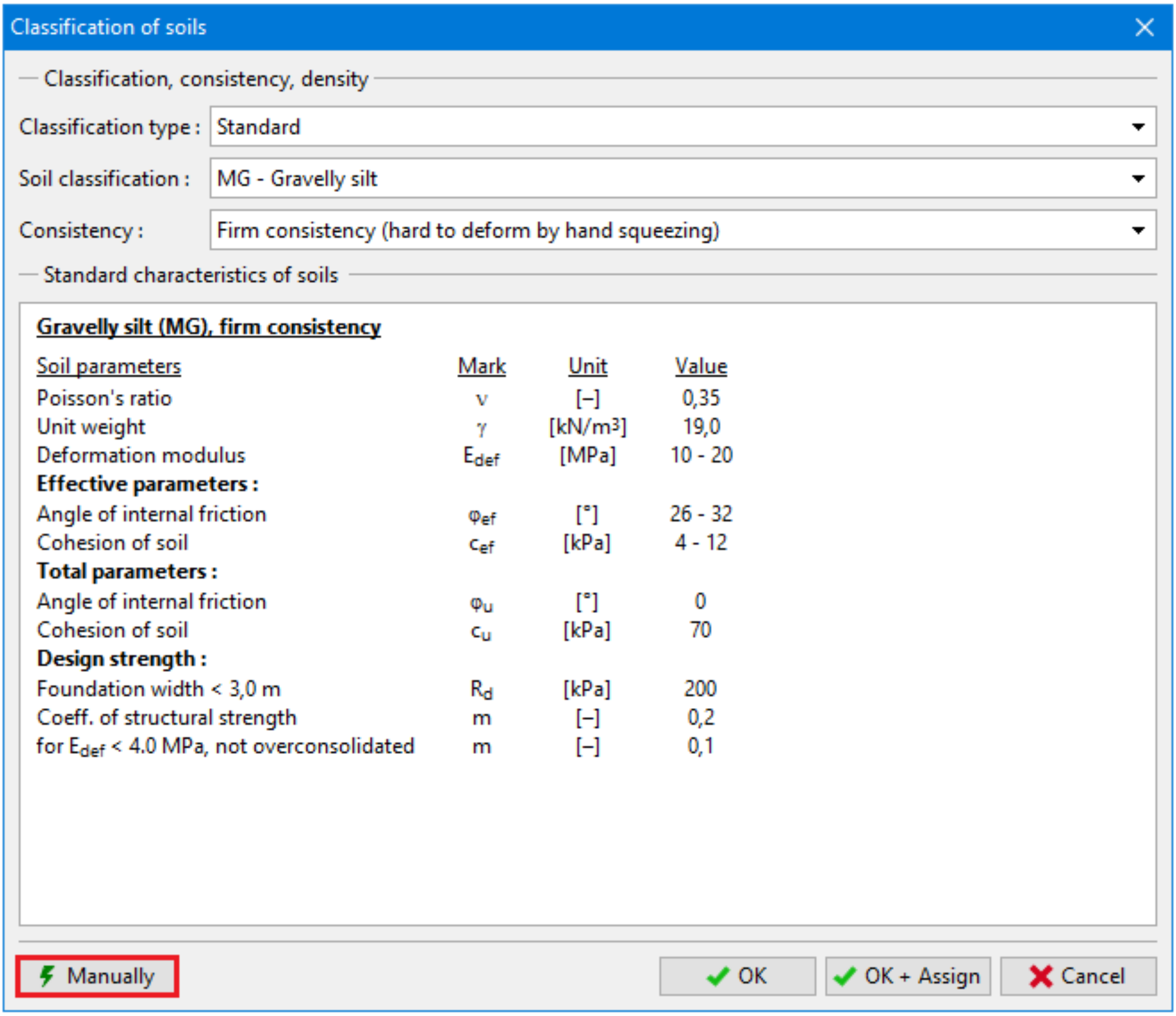
Approximate values for a specific soil can be obtained from the implemented catalogs of soils. One of the three **classification types** can be selected:

- **Standard**
- **China**

- **EAU 2012** - German standard

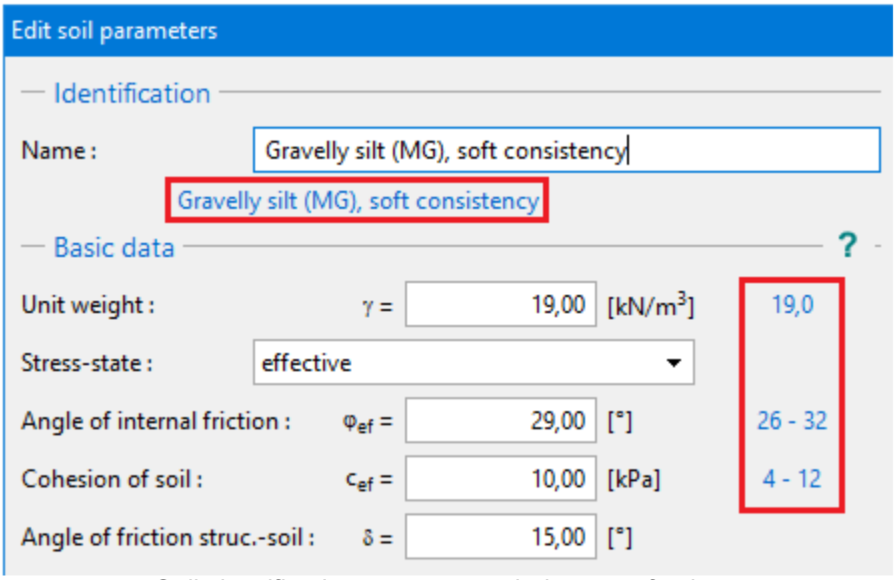
Next, the soil and relevant properties are selected (e.g. consistency, density,...). The soil properties obtained from the catalog appear in the window.

The "**Manually**" button (available only for **Standard** classification type) opens the "Manual classification of soils" dialog window that allows classifying the soil if its parameters are known, e.g. from laboratory measurements (grading, moisture, compactness….).



"Classification of soils" dialog window

Pressing the "**OK**" button shows recommended values next to corresponding input fields (see the picture) in the "**Add new soils**" dialog window. Pressing the "**OK+Assign**" button then assigns the average values of soil parameters into individual input fields. The "**Cancel**" button leaves the window with no action.



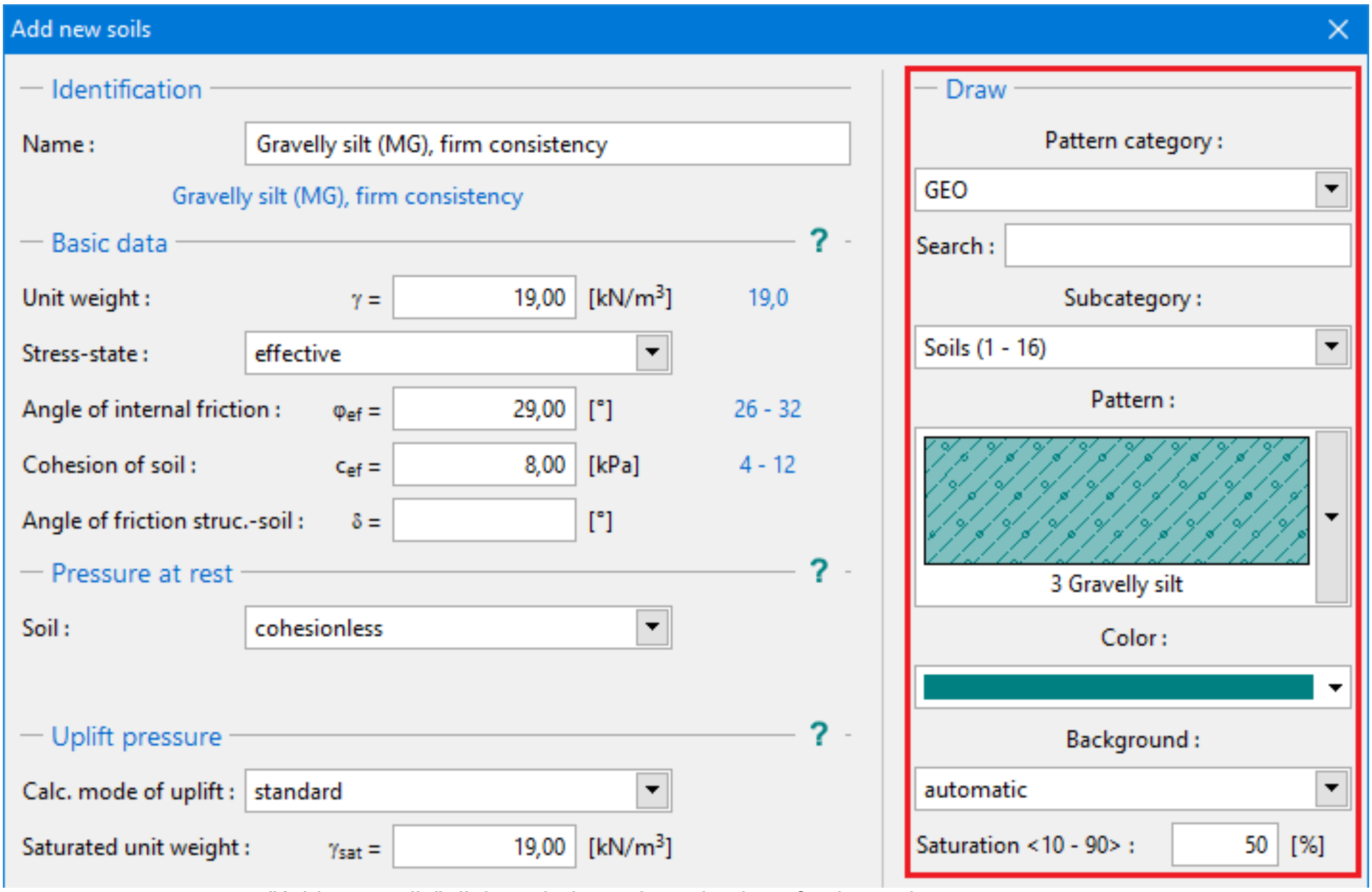
Soil classification - recommended range of values

Soil and Rock Symbols

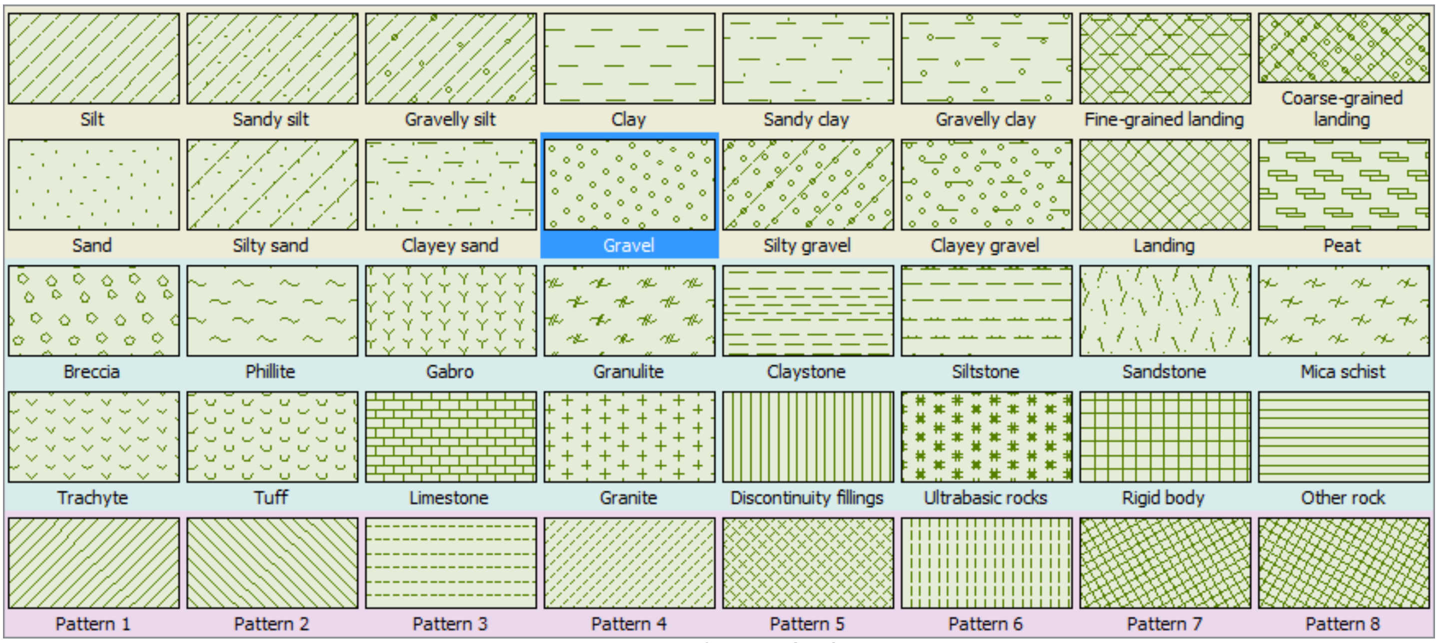
It is possible to select a pattern category from a combo list for each input soil (patterns of GEO5 software, full color, gINT patterns, or patterns according to the YS 5204 Chinese standard 1) and the color of the pattern. These are displayed in the input profile.

The color you choose from the combo list is used to plot soils or rocks on the desktop and to pictures, which are either stored on the "Picture list" or printed using "Print and export desktop view" (to visualize the same (full) colors in pictures, the option "**full color**" in the "Drawing Settings" must be set).

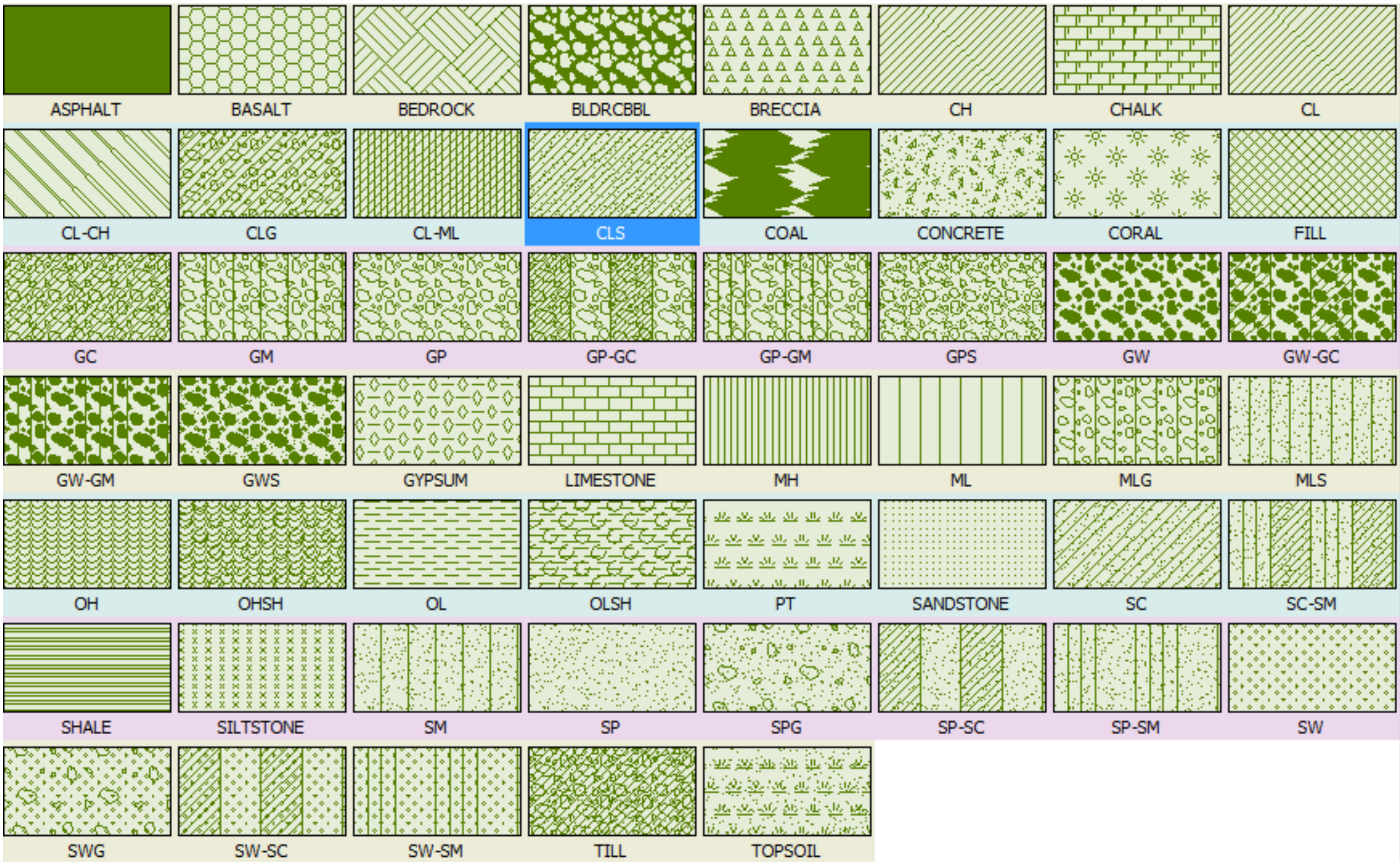
The pattern color should be chosen concerning the desktop background or printout paper so that it is sufficiently visible.



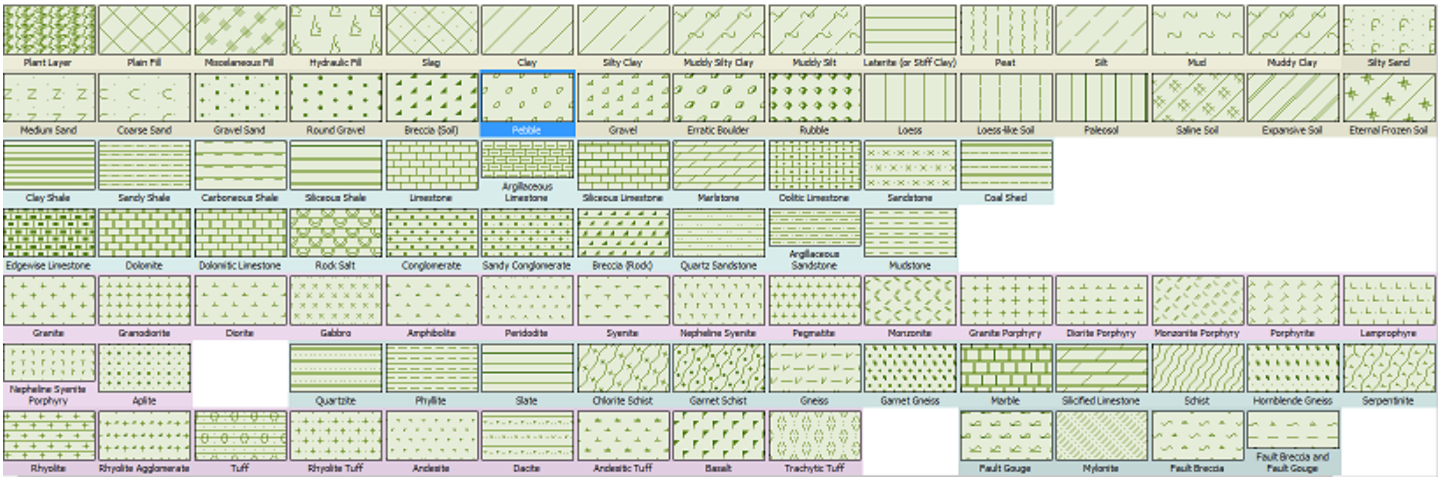
"Add new soils" dialog window - the selection of color and pattern category



Patterns of soils- GEO5



Patterns of soils - gINT



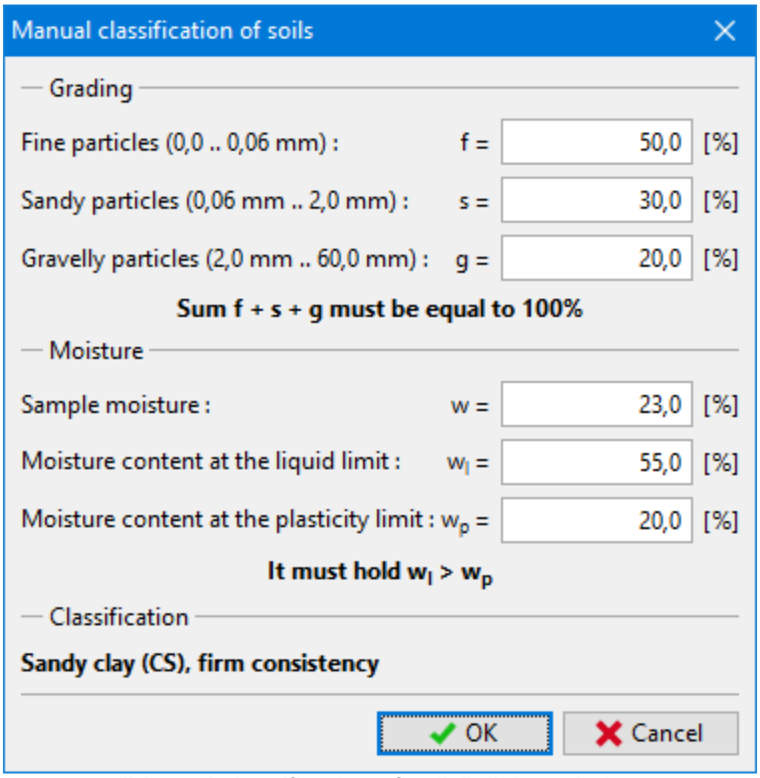
Patterns of soils - YS 5204 - 2000

Literature:

1 - YS 5204-2000 - Specification for mapping symbol of geotechnical investigation report

Manual Classification of Soils

This dialog window allows specifying the soil parameters, which are then added into the catalog of soils. The "**OK**" button returns back to the "**Classification of soils**" dialog window with the setting and classified soil.



"Manual classification of soils" dialog window

Interfaces in 2D Environment

The left part of the frame contains a table with a list of interfaces. Above the table, there are two basic buttons required for interface input:

Setup Ranges Opens the "World coordinates" dialog window that allows setting the world dimensions (left and right edge).

Add interface Turns on the mode for inputting a new interface - individual interfaces can be added in an arbitrary order. Each interface is automatically stored on the list of interfaces when leaving the input mode.

Interfaces are ordered in the table downwards, and it is possible to edit and delete them.

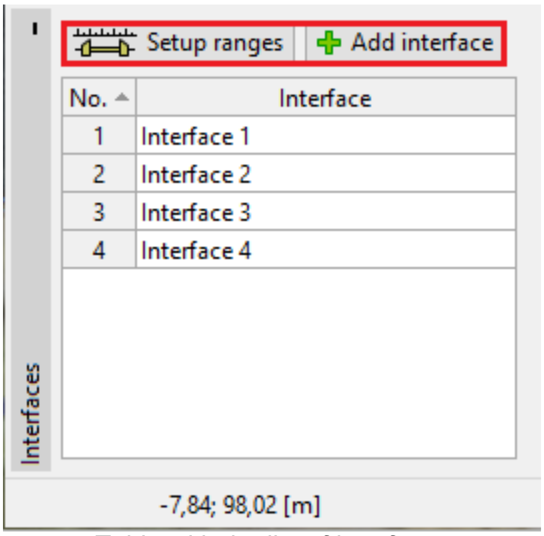
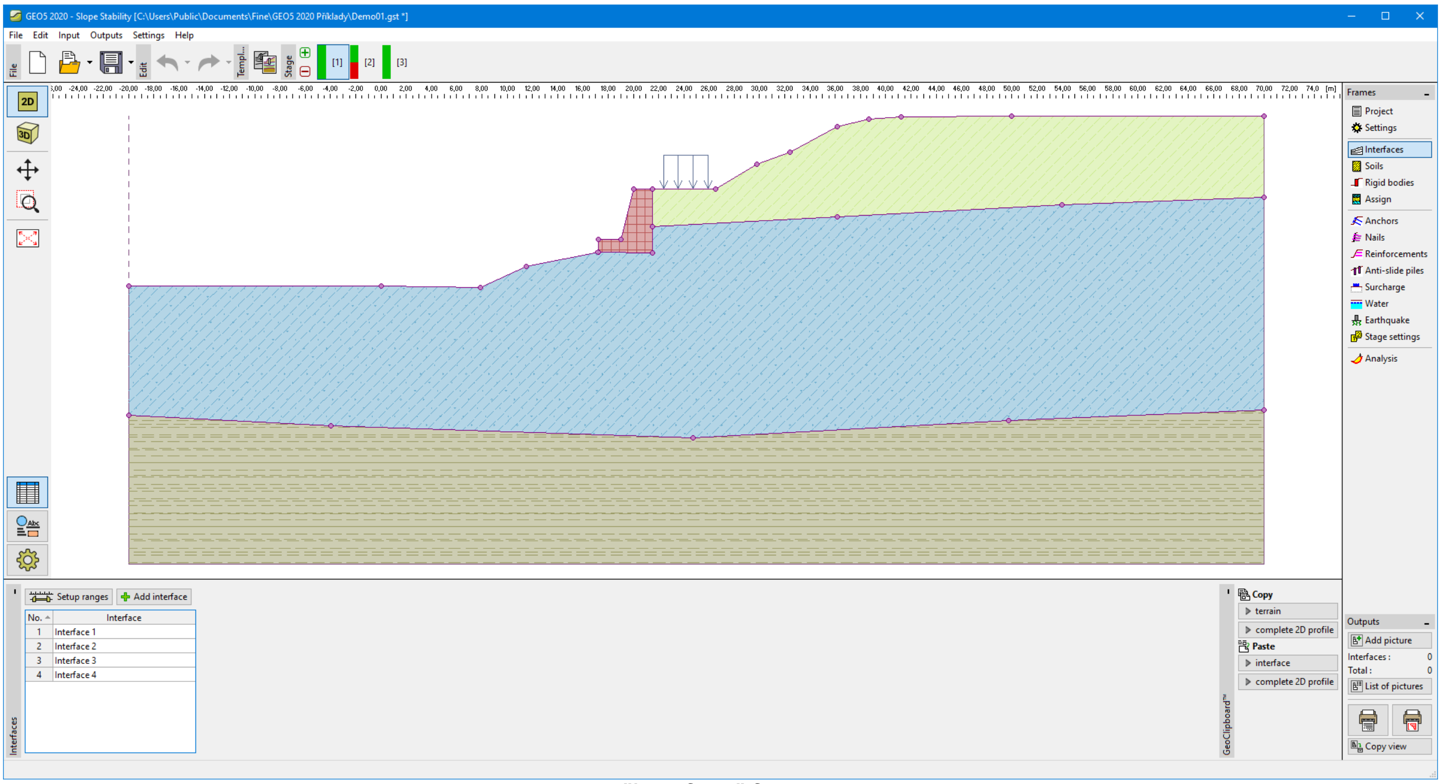


Table with the list of interfaces

Every change made to a given interface can be reverted using the "UNDO/REDO" buttons on the toolbar.

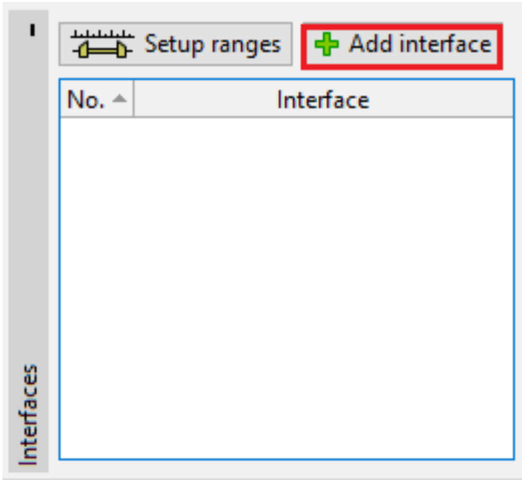
It is possible to copy and paste interfaces using GeoClipboard.



"Interface" frame

Adding Interface

The "**Add interface**" button starts the mode of inputting points of a new interface. The mouse mask is changed to an axes cross, and the visualization of the frame changes.



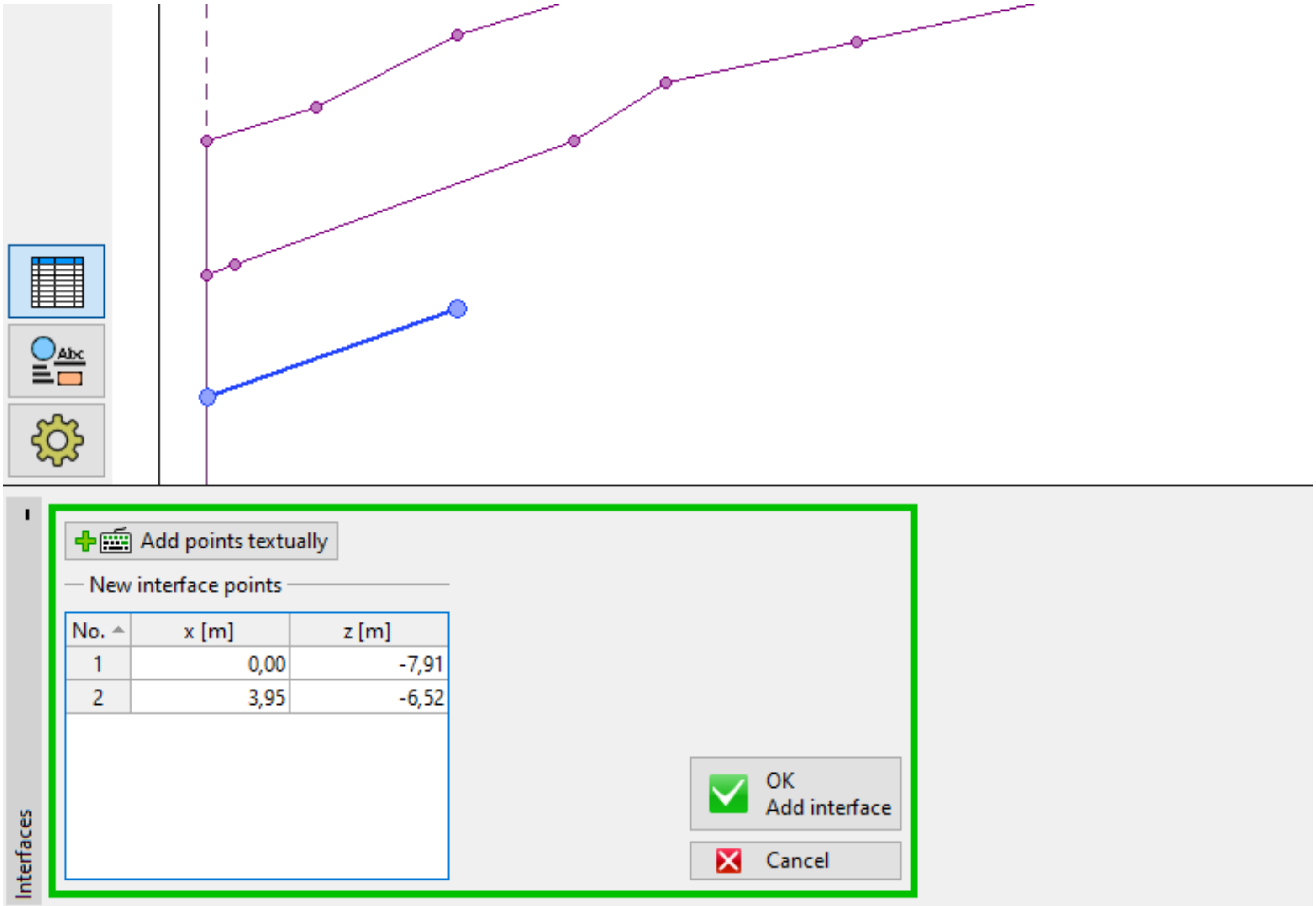
"Interface" frame

By pressing the left mouse button on the desktop, it is possible to input points of the interface. The coordinates (x , z) of every input point are added into the "**New interface points**" table. Alternatively, it is possible to add points in a dialog window by pressing the "**Add point in dialog**" button. An input point is always rounded to four significant figures (two decimal places) - input by mouse and keyboard is then completely equivalent.

During the input, it is possible to edit and delete individual points.

Input is terminated by pressing the "**OK Add interface**" button (adds the input interface onto the interface list), or by pressing the "**Cancel**" button (input interface is discarded).

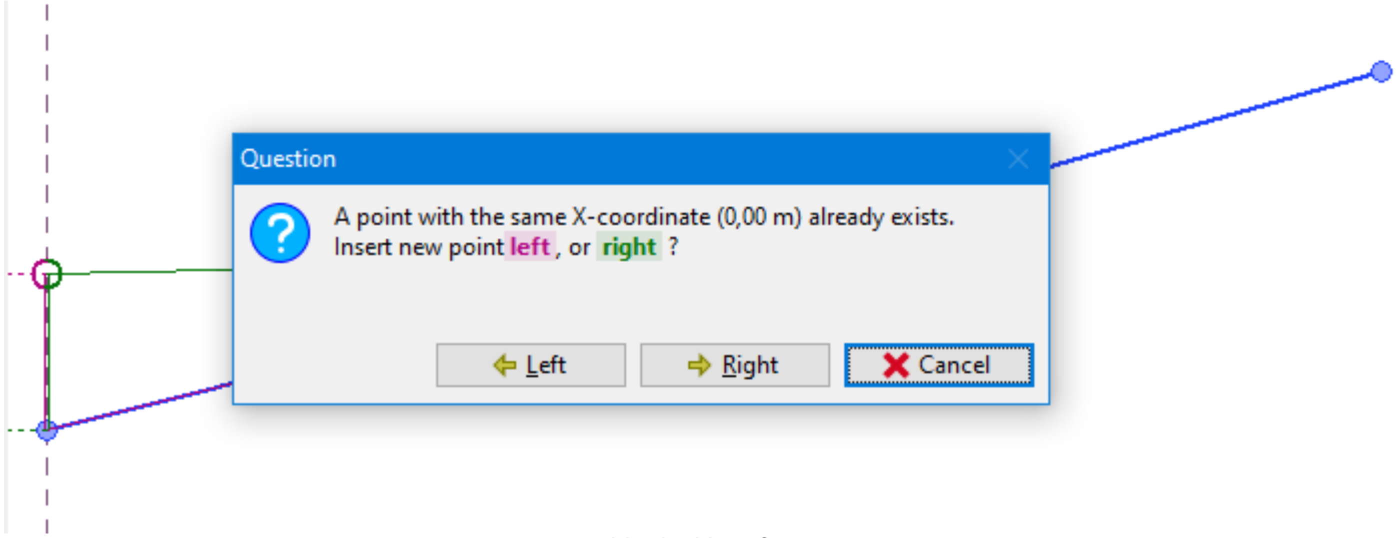
When inputting, it is possible to use the function of the grid.



Mode of interface points input

When inputting points, it is possible to use templates obtained from DXF import.

The program also allows introducing vertical interfaces - in such a case, the program requests to insert the point to the **left** or **right**. The buttons for confirming the action are colored - the same color is also used to visualize both input variants on the desktop.



Vertical interfaces

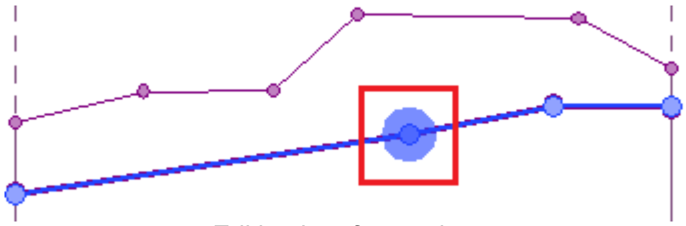
The program also contains an automatic corrector of input interface that determines the interface end points and then adds the interface onto the list of interfaces.

Editing Interface Points

When inputting or editing an interface, it is possible to edit or remove individual points, graphically and textually in a dialog window.

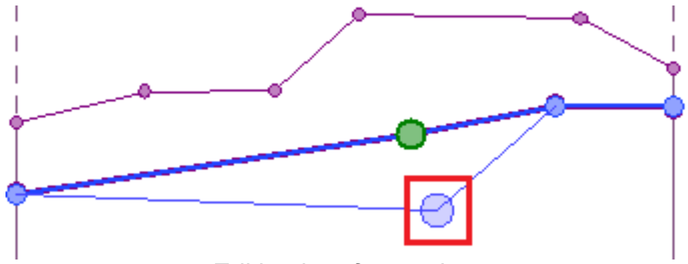
Editing interface point

Move the mouse cursor on the point, which is to be edited. The area around the point is highlighted.



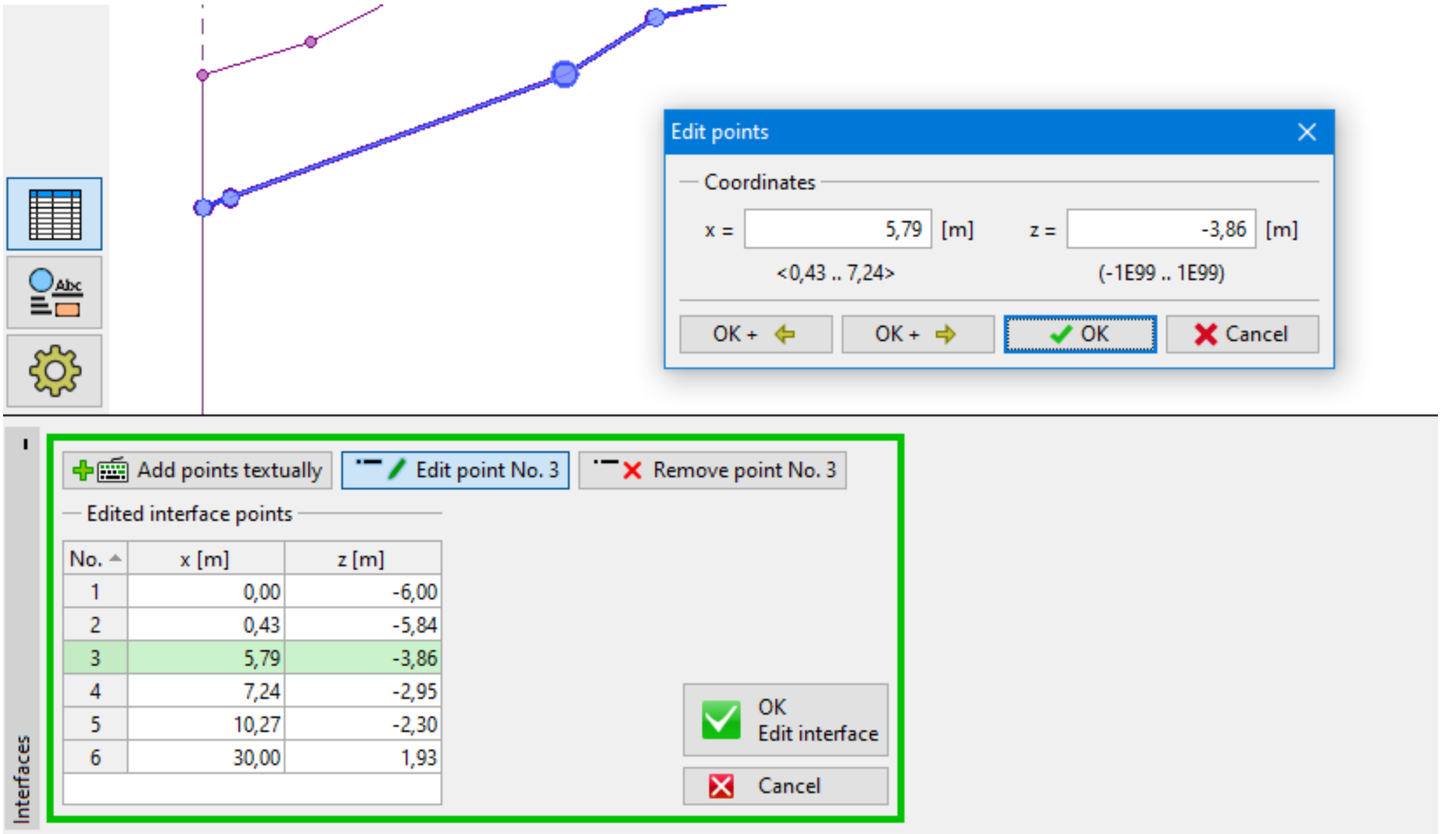
Editing interface points

The point is moved by holding the left mouse button and dragging. By releasing the left mouse button, the point is changed.



Editing interface points

Alternatively, it is possible to press "**Edit point**" in the table and change the coordinates in the dialog window. Editing is terminated by clicking the button "**OK**".

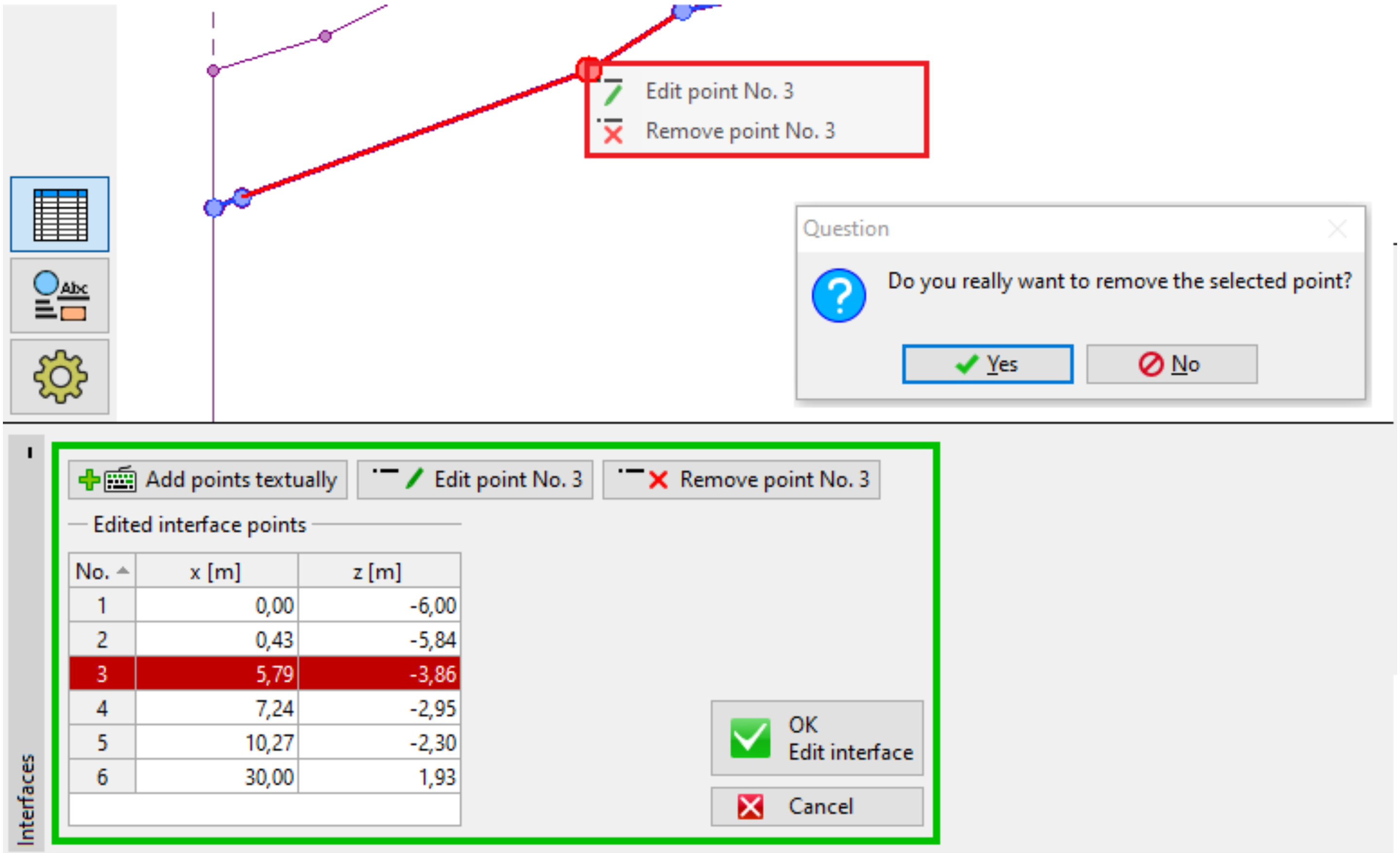


Editing interface points

Editing interface points - result of coordinates change

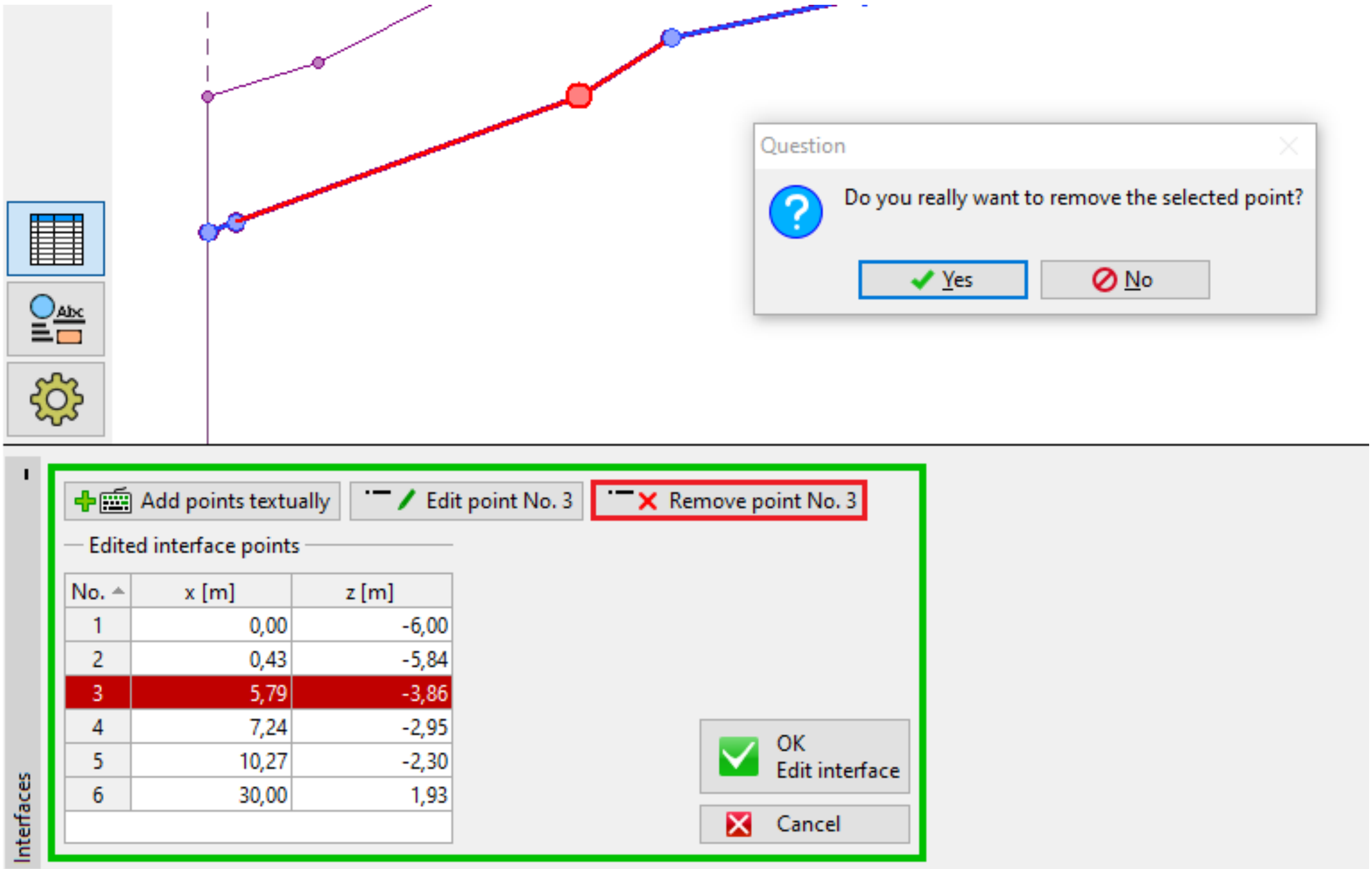
Removing interface point

Move the mouse cursor on the point, which is to be deleted. The area around the point is highlighted. After clicking the right mouse button, the context menu appears. Select the "**Remove point**" item. The program highlights the point and the lines which are affected by deleting the point in red. After confirmation by the user, the program deletes the point and adjusts the interface.



Removal of interface point - graphically

Alternatively, it is possible to select the point in the table and click on "**Remove point**". The further process is identical to the one described earlier.

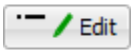


Removal of interface point - textually

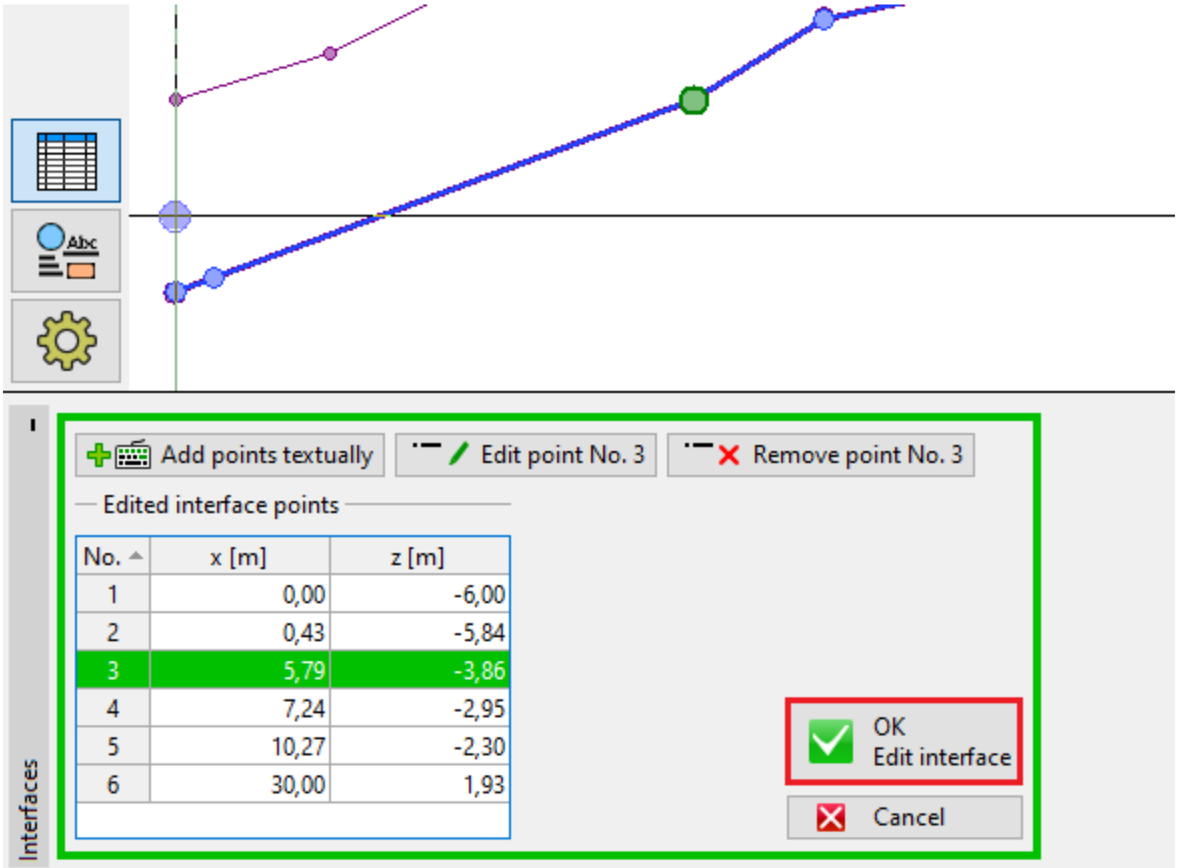
Editing and Removing Interface

Editing Interface

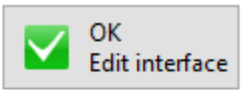
An interface is selected in the table or on the desktop by pressing the left mouse button.

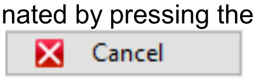
By double-clicking the mouse button on the interface, or by pressing the 

button, the interface editing mode is turned on. The interface being edited is highlighted in blue, and it is possible to change its shape.



Editing Interface

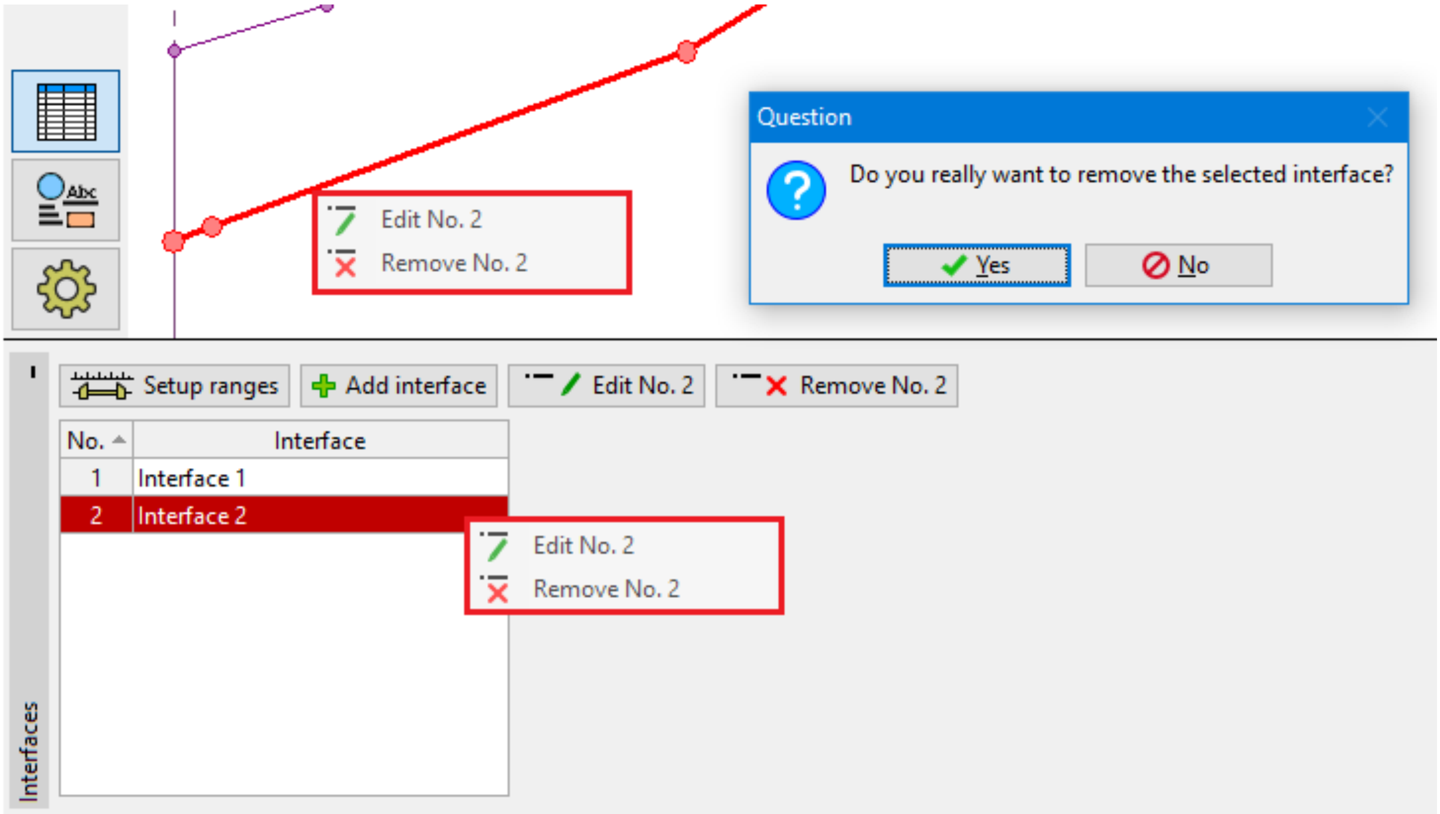
Input is terminated by pressing the 

button (adds the input interface onto the interface list), or by pressing the 

button (changes are discarded).

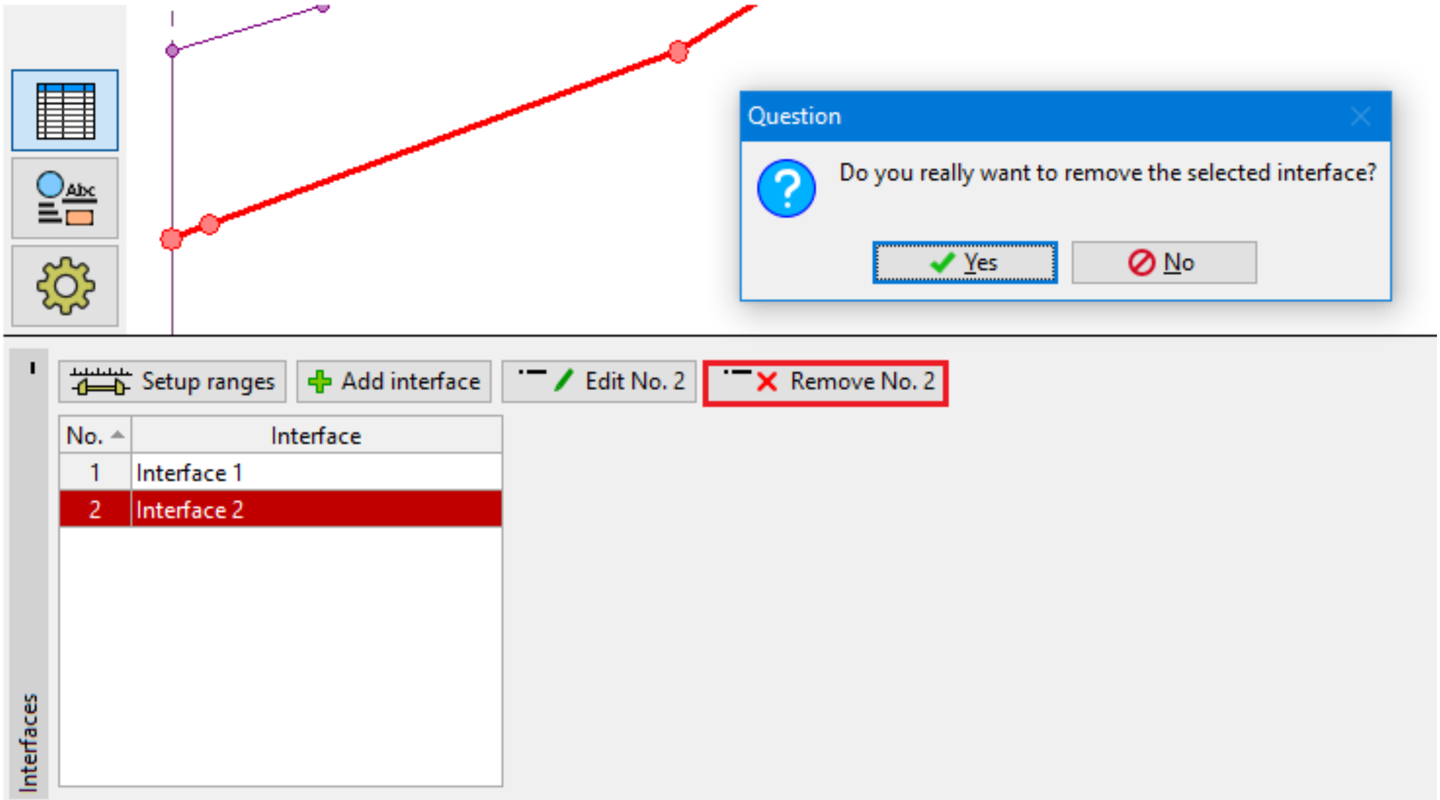
Deleting Interface

From the table list, or on the desktop, select an interface. Pressing the right mouse button on the interface (table row) opens the context menu. After pressing the "**Remove**" button, the program highlights the interface and the table row in red. After confirmation by the user, the interface is removed.



Removing Interface

Alternatively, it is possible to select an interface in a table and press "**Remove**". The further process is the same as described earlier.



Removing Interface

After editing interface, an automatic interface corrector (the same as after adding interface) sets off, which checks the interface shape, and corrects the end points if required.

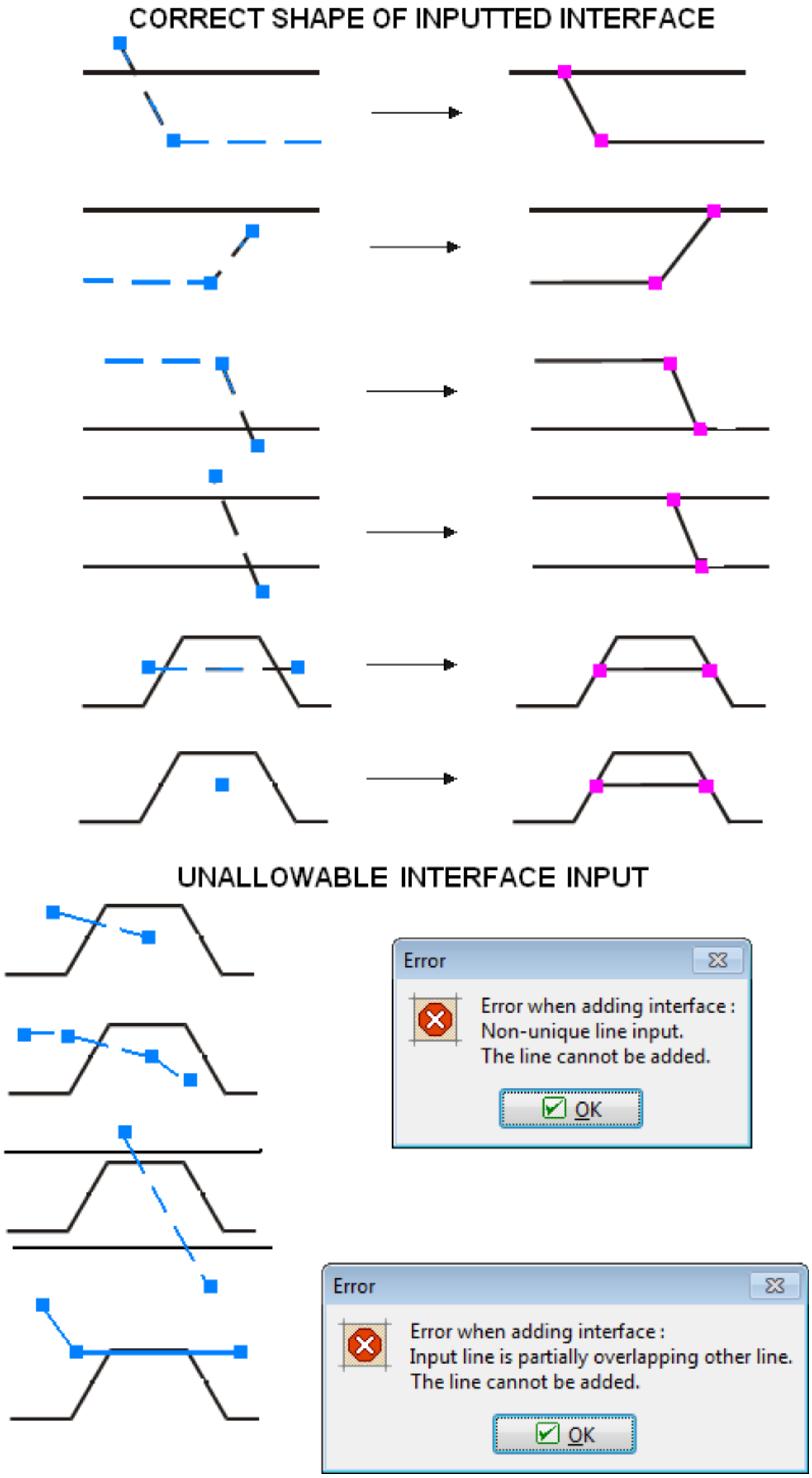
Input Interface Corrector

When the inputting or editing process is completed, the program automatically modifies the input interface in order to comply with the software requirements, i.e. the end points touch the world edges or other interfaces. The automatic corrector can be further used to simplify the input process - for example, if only one point is used to specify an interface, the program automatically creates a horizontal interface containing the defined point.

If two interfaces collide, the corrector creates new end points of the current interface. These points then also become the points of the interface being touched. All lines of individual interfaces thus start and end in a point.

In case of an incorrect input (see the picture below), the interface cannot be stored. In this case, the interface must be modified, or the inputting process must be stopped using the "**Cancel**" button.

Here are some examples of interface corrector functions (correct and incorrect input):

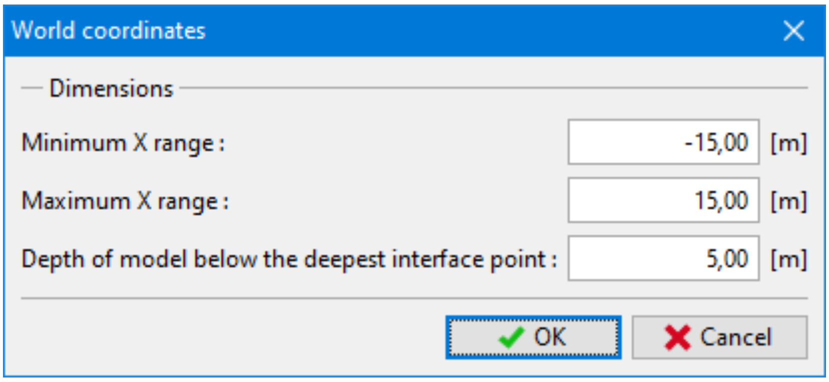


Correct and incorrect interface shapes

World Coordinates

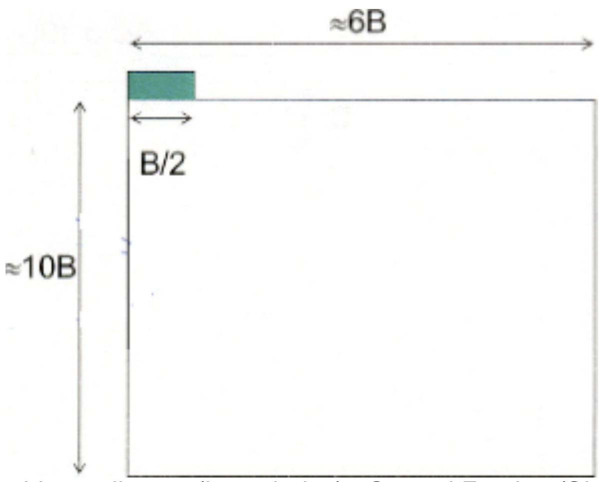
This dialog window is used to specify the world coordinates (dimensions) for a given task - left and right edges. The depth from the lowest point of the interface is for most of the programs an auxiliary input - it does not have any effect on the analysis itself. In the FEM program, the determination of correct world dimensions is very important and can effect the analysis results substantially.

The world coordinates can be changed at any time - when increasing dimensions, all input interfaces are automatically prolonged; when reducing dimensions, all points falling off the new world coordinates are automatically removed.

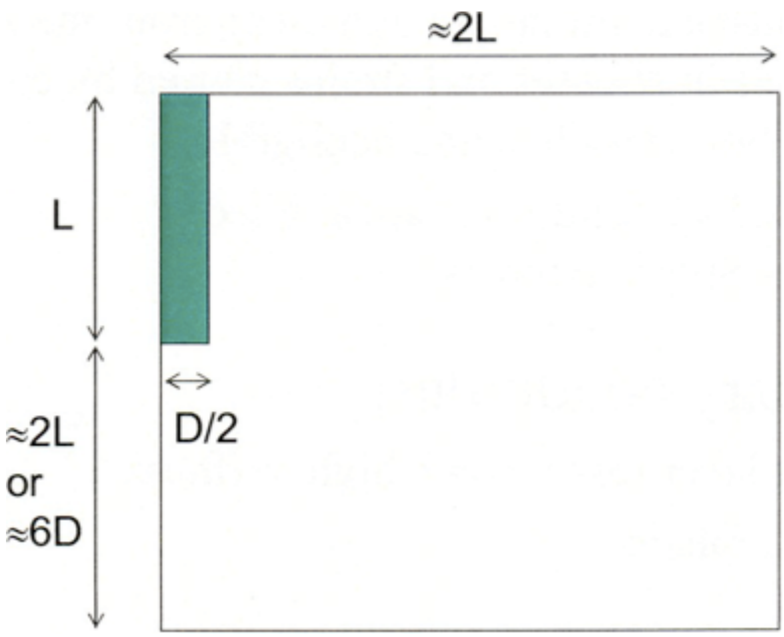


"World coordinates" dialog window

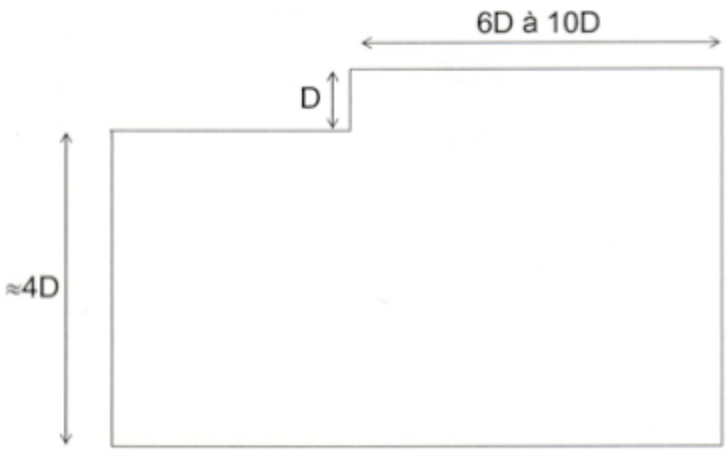
Recommended world coordinates for the FEM program are evident from the pictures.



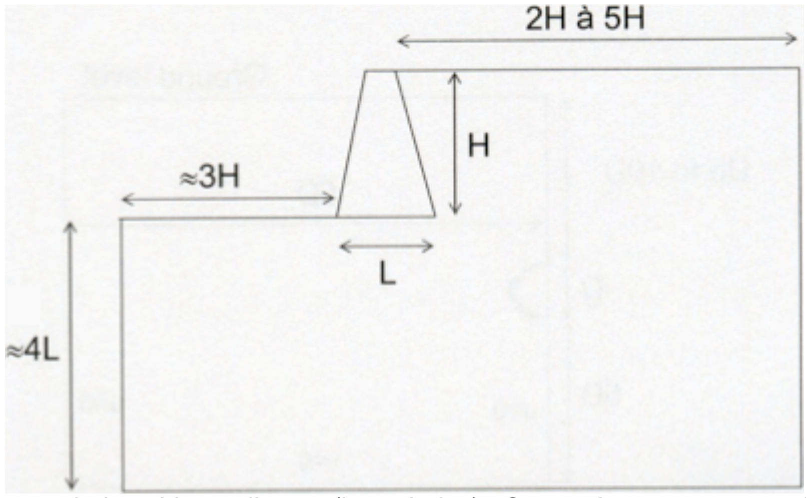
Recommended world coordinates (boundaries) - Spread Footing (Shallow foundations)



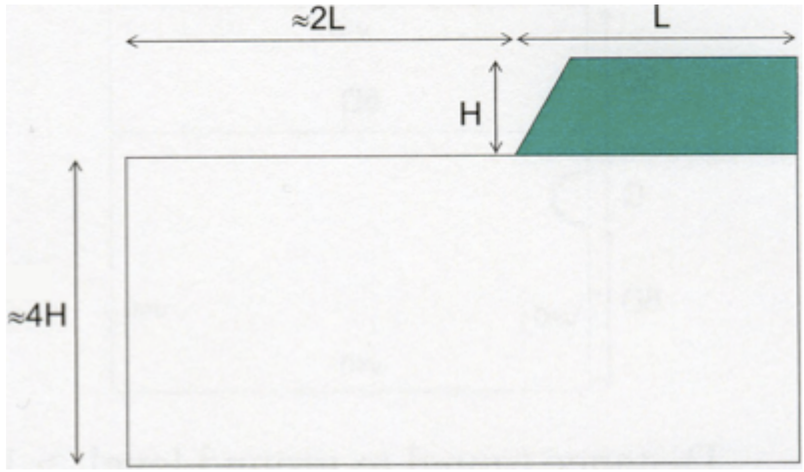
Recommended world coordinates (boundaries) - Deep and pile foundations



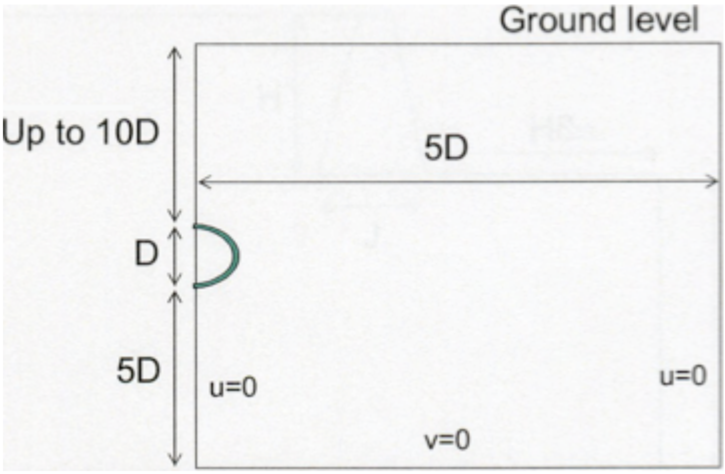
Recommended world coordinates (boundaries) - Excavations



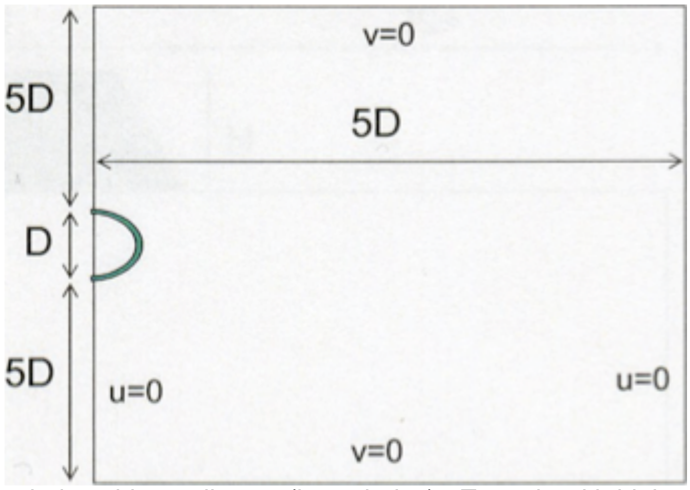
Recommended world coordinates (boundaries) - Supporting structures and Walls



Recommended world coordinates (boundaries) - Embankments and Slopes



Recommended world coordinates (boundaries) - Shallow tunnels saved



Recommended world coordinates (boundaries) - Tunnels with high overburden

Input of Objects and Data

Input data can be entered to the GEO5 programs in several ways. Data is input in frames and dialog windows. Access to individual frames (input modes) is provided by the "Frames" toolbar. Apart from direct input in the frames, it is also possible to add data (coordinates of points, profile depths...) and objects (surcharge, anchors, props, georeinforcements etc.) using the "**Add**", "**Add in dialog**" and "**Add graphically**" buttons.

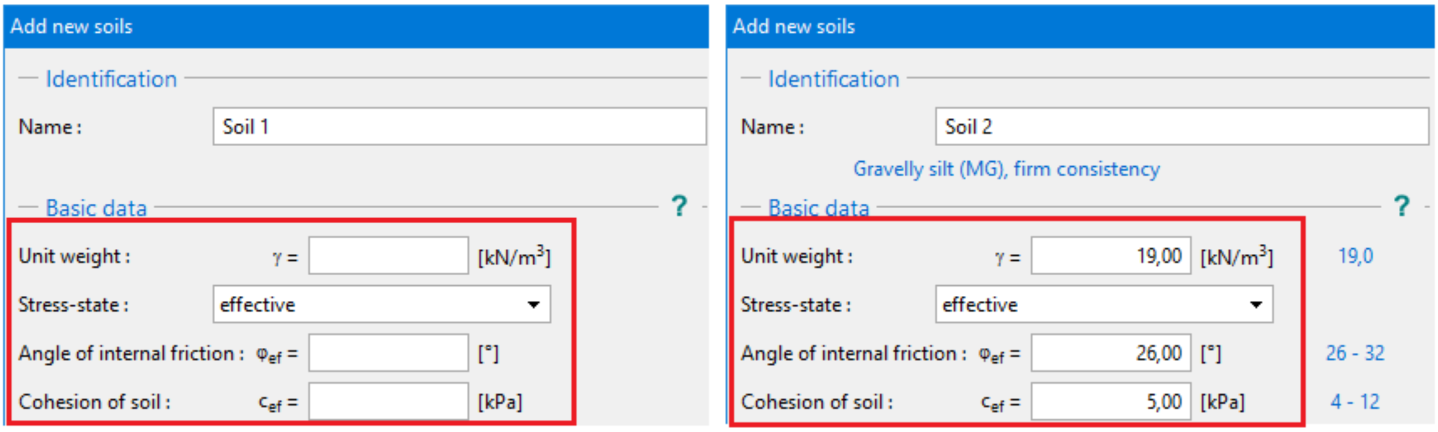
Add

The "**Add**" button opens a dialog window where required data is input (for example, parameters of soils, surcharge, forces, etc.). After confirmation, the data are saved, and the item is added to the table.

Visualization of the buttons

If the frame is used the first time (the table with the list of items is empty), or the table contains items, but none of those are selected, the programs opens an empty dialog window, and all the data needs to be input. If there is a selected item in the table and then the dialog window is opened, it will use the data from the selected item, which can be further modified and saved as a new item in the table list.

The figure below shows an easy example. The frame was used the first time, all fields in the dialog window are empty (left picture). The parameters are input, and added into the table. The table contains the selected item "**Soil 1**". Then, the "**Add**" button is used again, and the dialog window contains pre-defined parameters from "**Soil 1**". After the required revision of the parameters, it is possible to save the new data as another item of the table list.

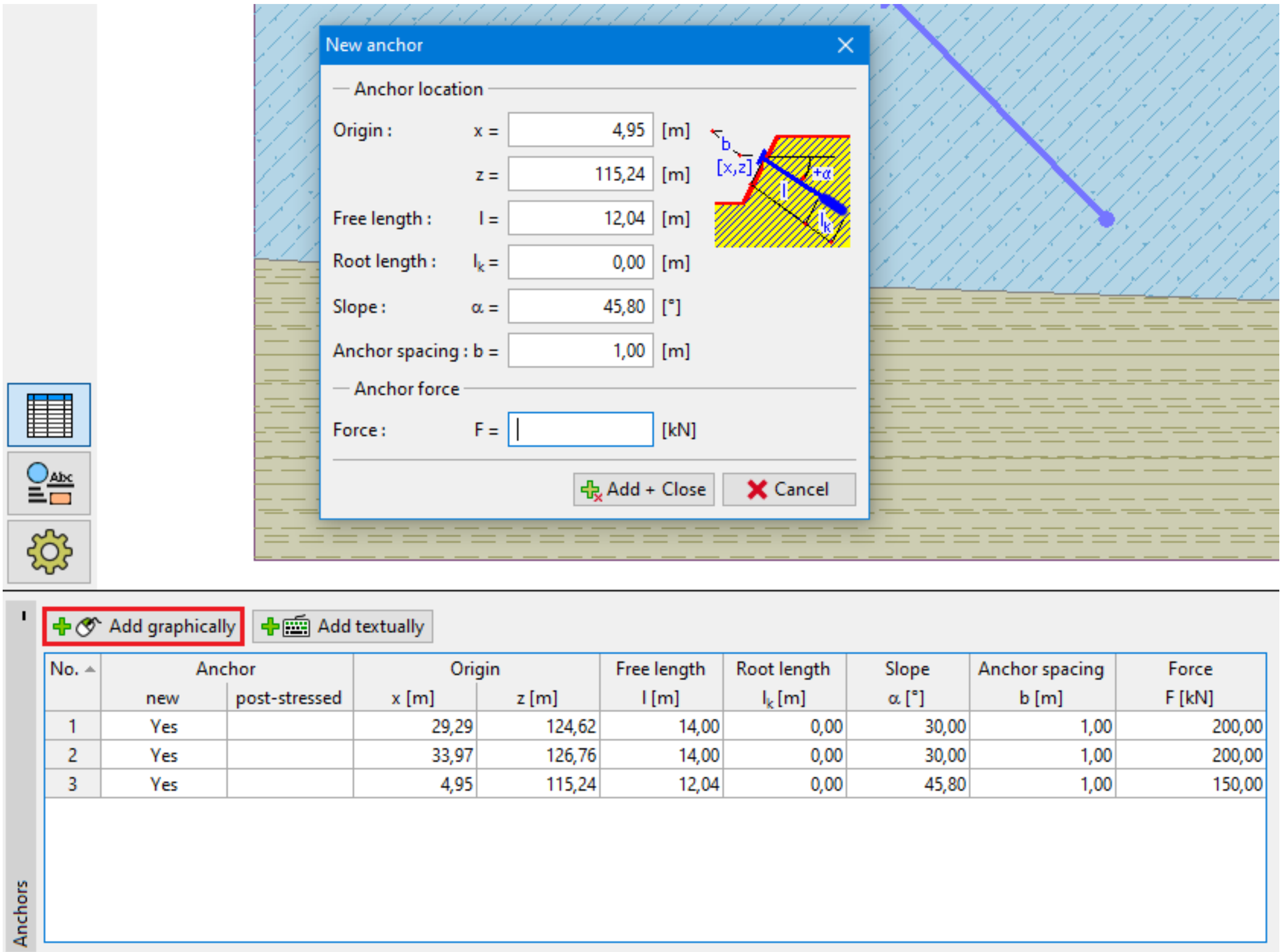


Input modes - using existing data

Add Graphically

Add Graphically

The "**Add graphically**" button turns on the mode of graphical input. The mouse mask changes to an axes cross, and using the left mouse button, it is possible to add the required object. For example, when adding anchors, by clicking on the desktop, the initial and end point of the anchor are input. After inputting the second point, the program opens a dialog window. From now on, the process is the same as described in the chapter "**Add**".



Graphical input mode

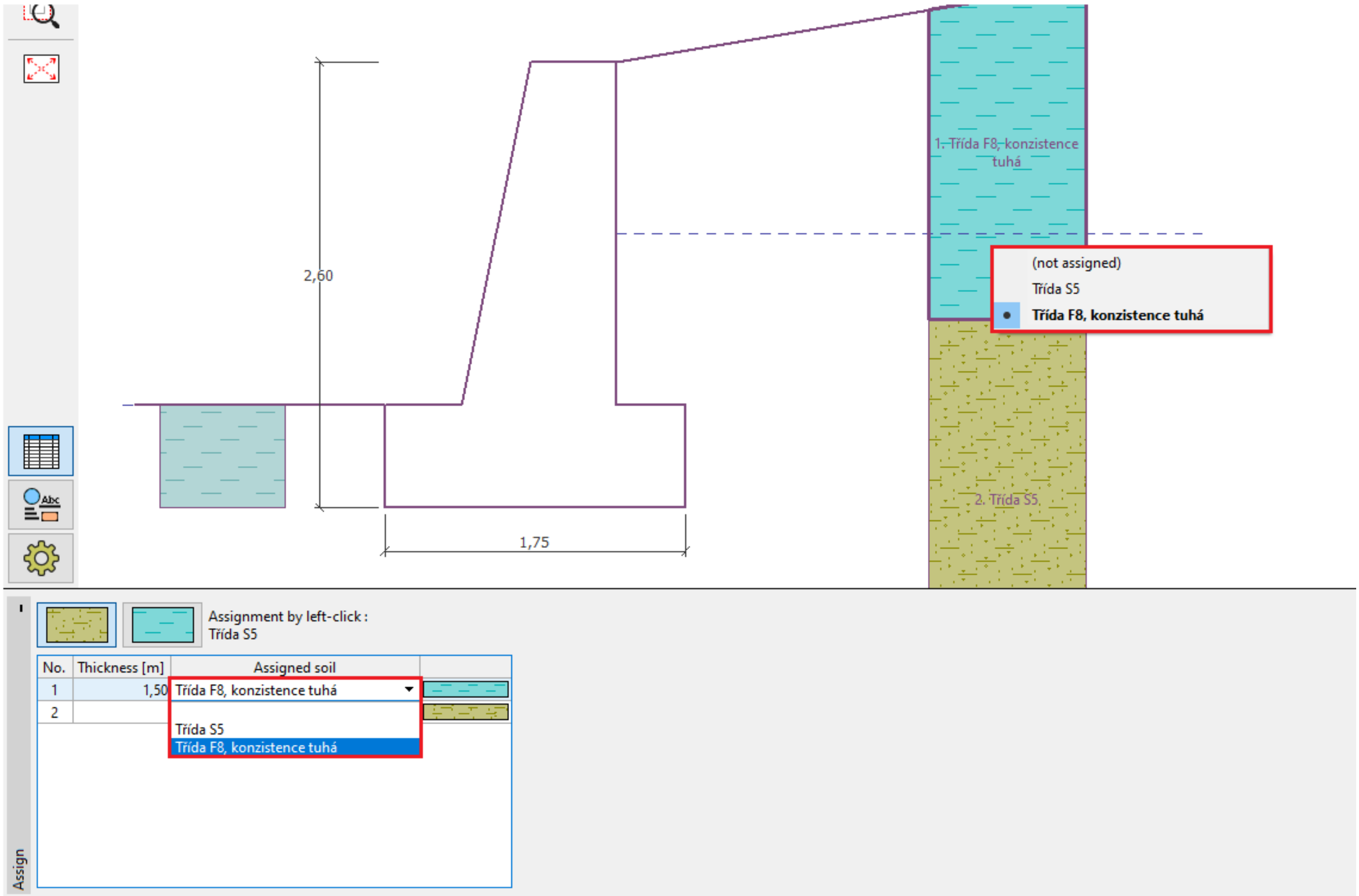
Assigning Soils

Three options are available to assign soils into individual profile layers.

Clicking the left mouse button on the toolbar button above the table selects the desired soil (positioning the mouse cursor in the bar above the soil button displays a bubble hint with the soil name). The soil is inserted by moving the mouse cursor (the cursor mask changes into a "**hand**") first into a specific layer and then by pressing the left mouse button.

The second option requires to open a combo list of a specific interface and then select the desired soil to be assigned. All changes in the soil assignment are automatically displayed on the desktop.

Last option is to use the mouse context menu.



"Assign" Frame

Design Coefficients

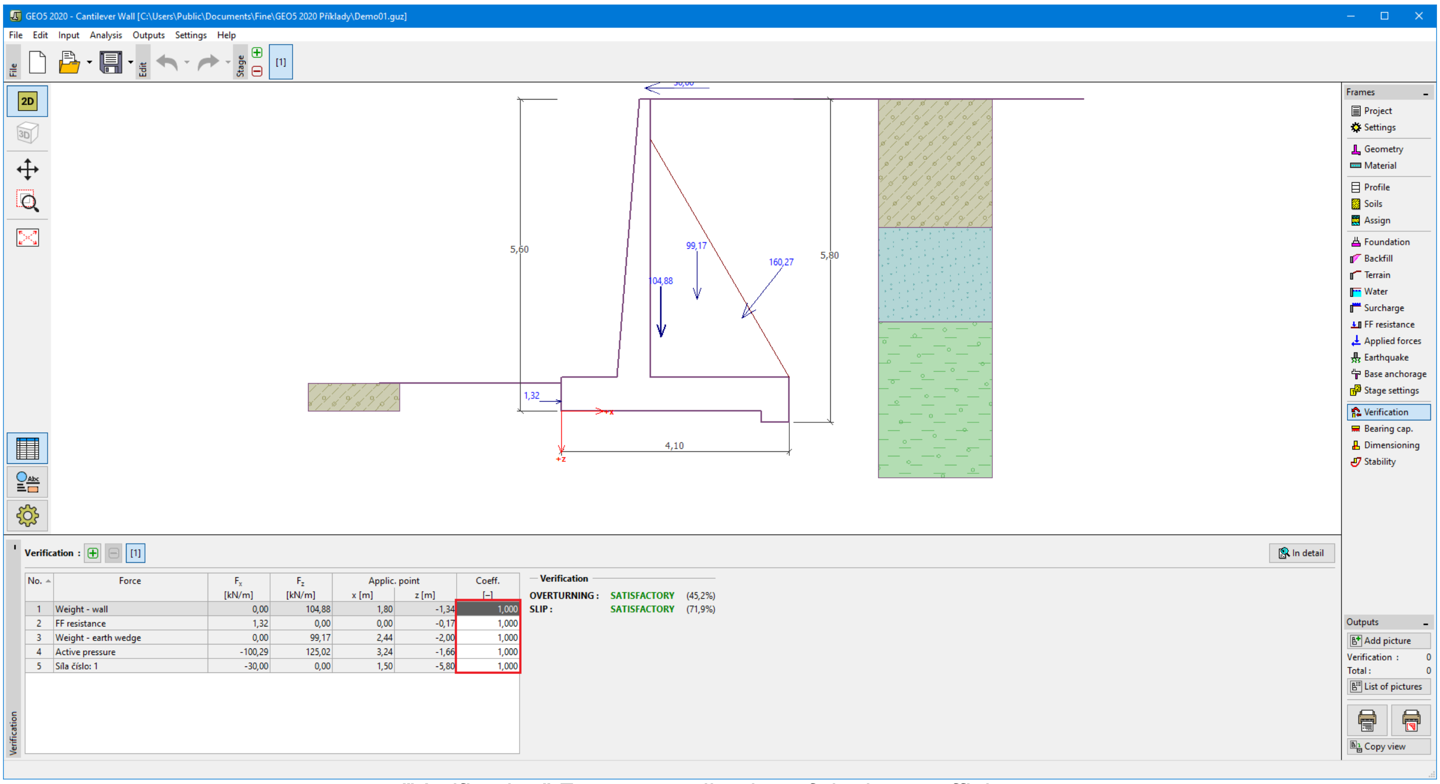
The "**Analysis**" or "**Verification**" (for verification methodology - Classical way) frames that display the list of computed forces allow specifying design coefficients. A design coefficient multiplies the corresponding force. When inputting the coefficient, the results are automatically recomputed, and the desktop shows modified forces.

Design coefficients are advantageous, for example, for:

- Structure testing when a structure response to an increase of force specified directly in the analysis window can be visualized.
- Excluding several forces from verification or their reduction.
- Specifying design combinations - e.g. different coefficients can be assigned in the sense of EC to main load variables and side variable loads.

The following combinations can be specified, for example, when performing the wall verification:

	Analysis 1	**Analysis 2**	**Analysis 3**
• **Wall**	1.0	1.0	1.0
• **Active pressure**	1.0	1.0	1.0
• **Surcharge 1**	1.0	0.5	0.5
• **Surcharge 2**	0.5	1.0	0.5
• **Surcharge 3**	0.5	0.5	1.0



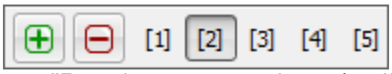
"Verification" Frame - application of design coefficients

Running Several Analyses / Verifications

Most frames that display the analysis results allow defining more than one analysis to be run. Several analyses within one construction stage are carried out, for example for:

- Dimensioning structure in more locations
- Analyses of various slip surfaces
- Verification with various design coefficients

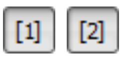
The toolbar in the top part of the frame allows managing individual analyses.



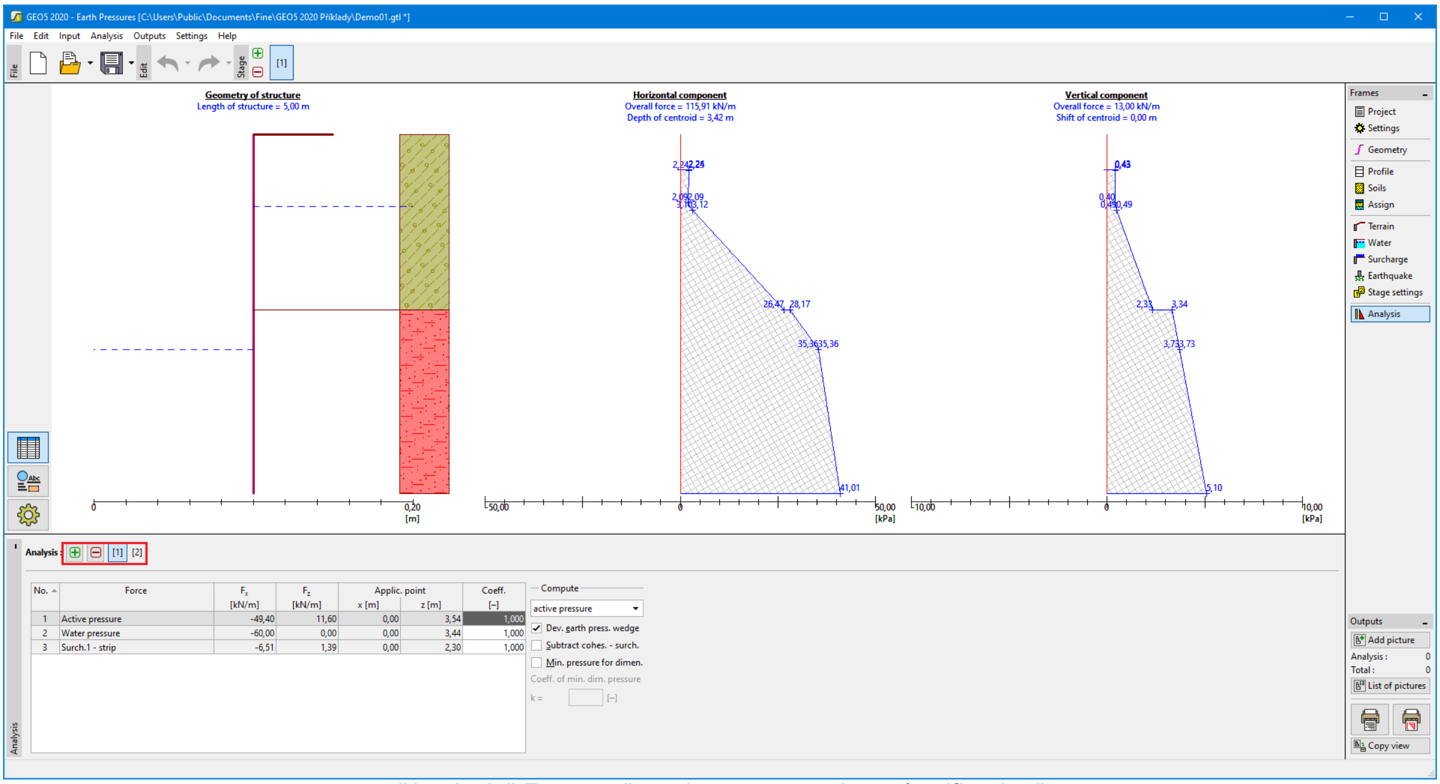
"Analysis" Frame - "Running more analyses/verification" toolbar

 **Add** • adds additional analysis onto the toolbar

 **Remove** • removes the **currently selected** analysis



Analysis 1,2,3 ... • switches between individual analyses



"Analysis" Frame - "running more analyses/verification"

Program Connection

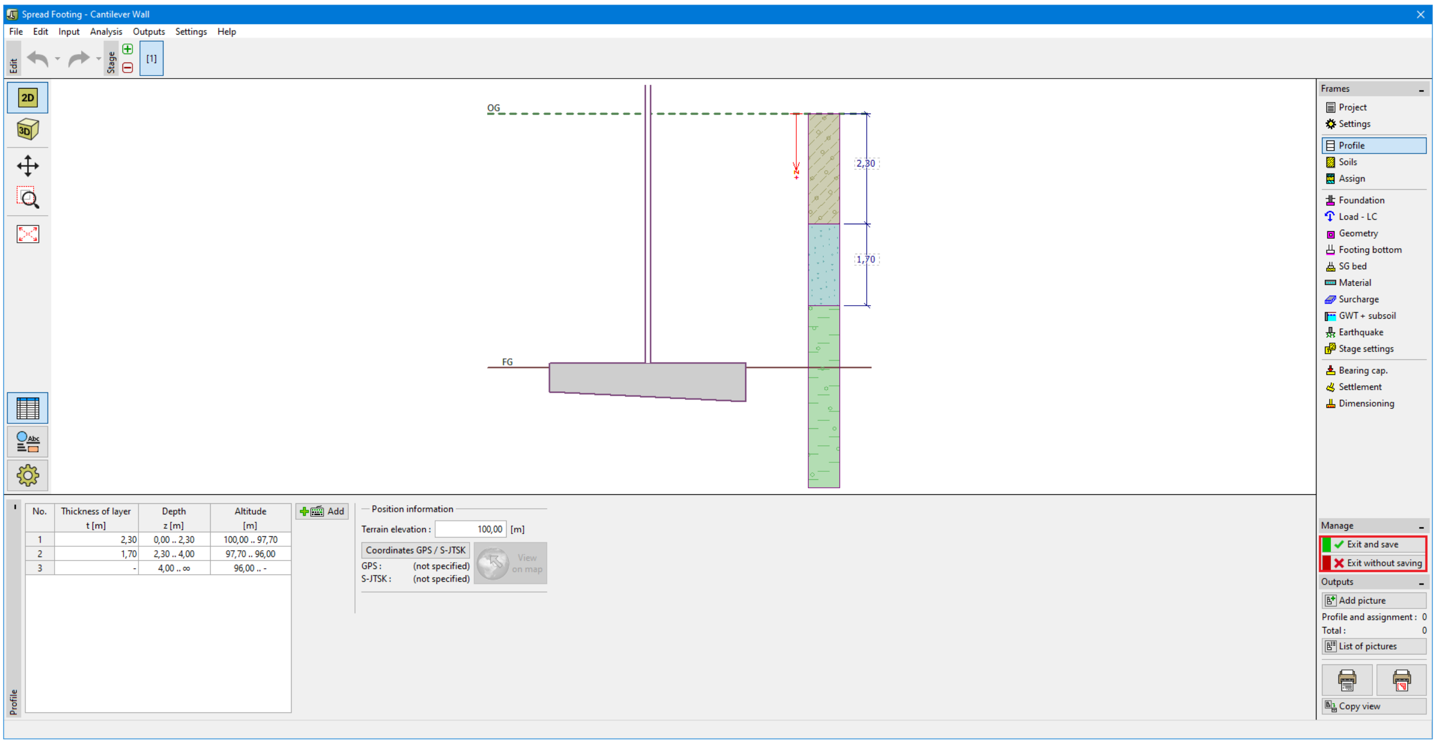
In some cases, it is possible to launch another program from a currently running program. For example, the "**Cantilever wall**" program allows running the "**Slope stability**" program in order to verify the external stability of a structure, or the "**Spread footing**" program in order to verify the bearing capacity of a footing of a structure.

The new program loads the data of the structure and then it behaves as a standalone program - closing the program, however, is different. Pressing the "**Exit and save**" button (on the right below the toolbars) closes the program and the analysis data is passed to the original running program. This is not the case if closing the program by pressing the "**Exit without saving**" button.

When running it for the first time, the program creates data of a structure and passes on the structure dimensions, geology, loads, surcharges and other data. The program then asks you to **input some additional data**, e.g. the analysis method, analysis setting, slip surfaces, stages of construction, etc.

When running it again (always necessary if some changes were made in the original program) the program regenerates the data to be passed on, but **keeps the data already input to this program**. For example, when connecting the original program with the "**Spread footing**" program the new program keeps the additionally input sand-gravel cushion together with input soil - the footing dimensions, foundation geometry, and geological profile are, however, regenerated.

Some actions are not allowed in the new program - e.g. to change the basic setting of the project, unit, etc. The generated task, however, can be saved into new data using the "**Save as**" button and work with it as with any other independent task.



"Spread footing" launched from the "Cantilever wall" program

Saved Views

The programs offer a number of ways of displaying results. A specific option can be selected from the "Drawing Settings : Analysis" frame. Quite often, it is necessary to go through complex and tedious settings of views - for example, if we are interested in the distribution of internal forces developed in beams using FEM, it is necessary to turn off the color range, draw only undeformed structure, select a variable to be displayed, select a suitable magnification, etc.

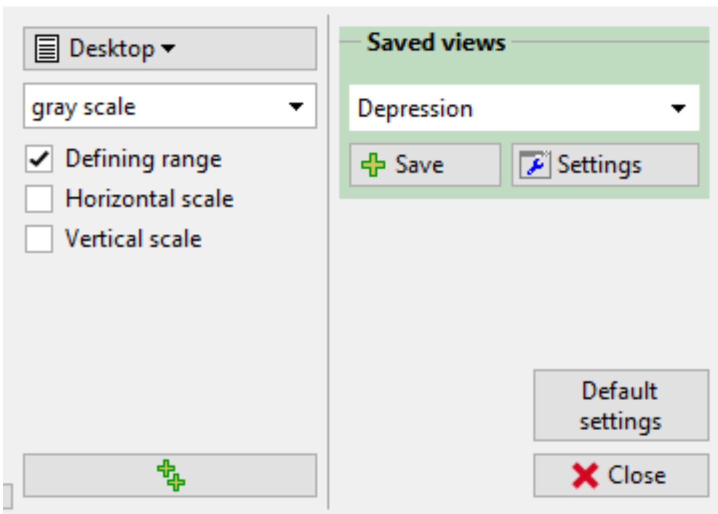
To simplify the way of managing individual views the programs allow us to **store the current view** using the "**Saved views**" bar, and also to **go from one view to another** in a relatively simple way.

The stored view keeps:

- All settings from the "Drawing Settings : Analysis" frame.
- Drawn variables
- Color range
- Picture zoom

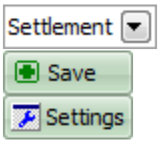
The view is stored for all stages of construction - if it is not possible in a certain construction stage to perform such a setting (e.g. in the first construction stage the settlement and depression are not defined) the programs display the closest possible setting and the defined view is switched to **<none>**.

Control elements are shown in the "**Analysis**" frame in the "Drawing Settings" frame.



"Saved views" - control elements

The following control units are available to **manage views**:



Select view	•	Combo list allows selecting an already specified and stored view.
Store current view	•	Opens the "**New view**" dialog window to store a new view.
Open view manager	•	Opens the "Picture manager" window with a list of views.

Drawing Settings : Analysis

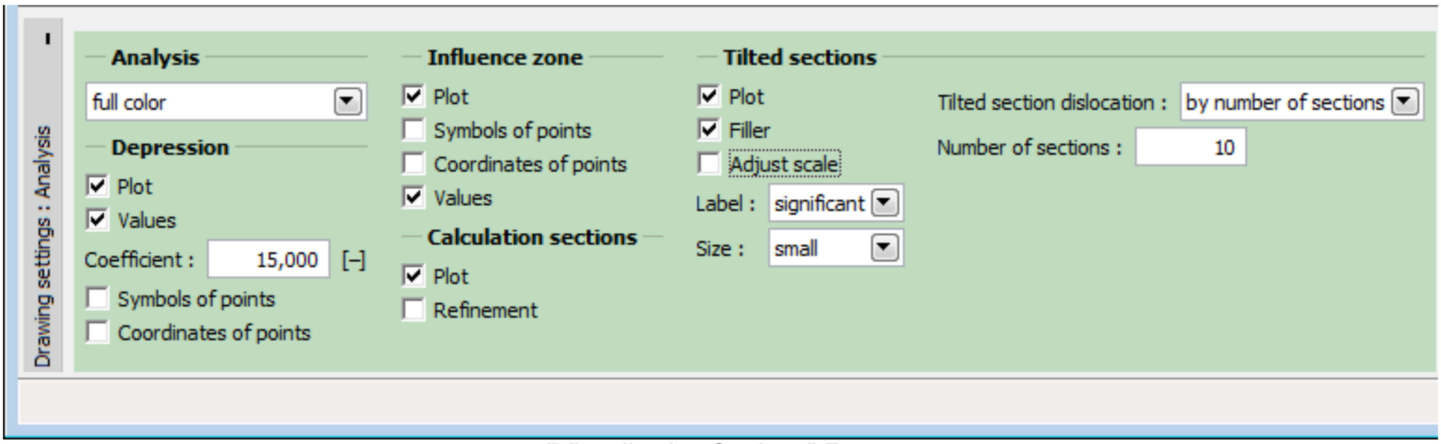
The "**Drawing Settings : Analysis**" frame provides tools for a clear way of displaying the results both on the screen and in the printed document:

- Parameters to draw **depression line** and **influence zone**
- Setting surface views and color scale drawing
- Setting and drawing **tilted sections**

The programs based on the **finite element method** further allows setting:

- Parameters to draw the finite element **mesh**
- Parameters to draw construction - **deformed/undeformed** (note that undeformed option must be selected when displaying beam internal forces)
- **Distribution of internal forces** on interfaces and beam elements

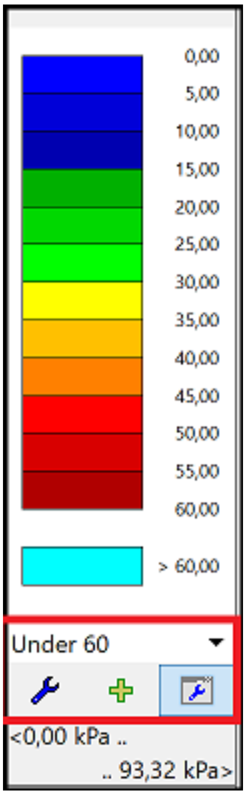
All information specified in this frame (including the setting of the current magnification) can be stored using the "Saved Views" bar.



"Visualization Settings" Frame

Setting a Color Scale

A color scale is an important tool providing a clear way of visualization of results.



Color scale with control buttons

The following control buttons are available to **manage scales**:

	**Select scale**	• a combo list allows us to select from the defined scales
	**Edit**	• opens the "Scale color definition" dialog window to edit current scale
	**Save**	• names and saves current scale into the "**Color scales manager**"
	**Scale manager**	• opens the "**Color scales manager**" dialog window

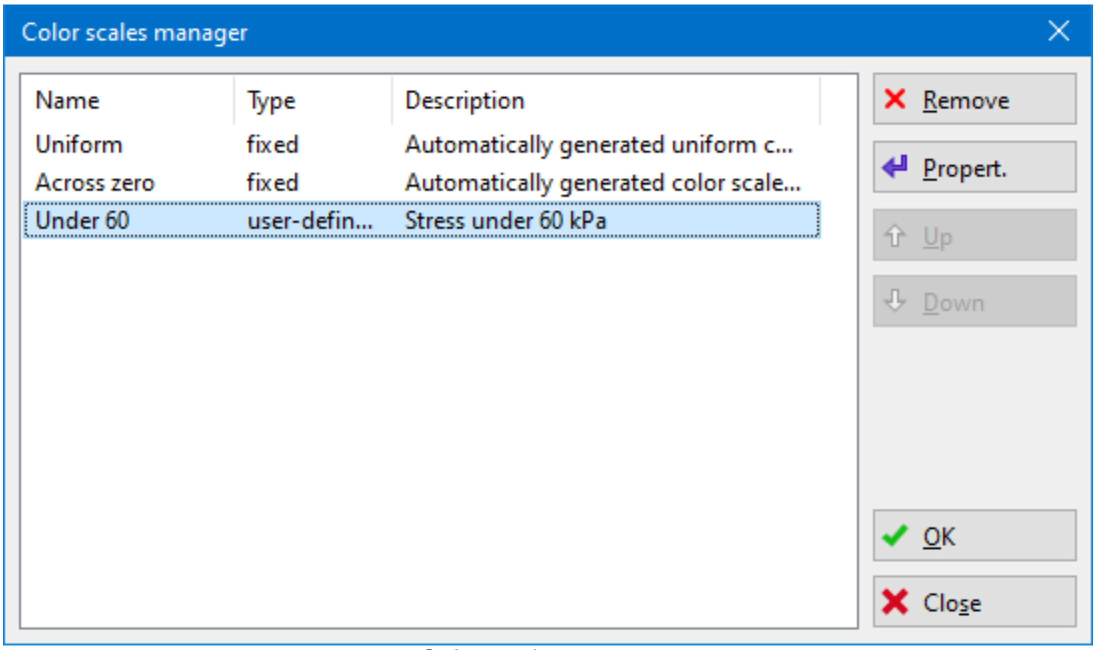
The program offers two predefined types of color scales - "**Uniform**" and "**Across zero**". Both scales have a moving minimum and maximum value and predefined colors. The scale is **automatically regenerated** whenever a variable or a stage of construction is changed.

Current scale range and unit are displayed below the control buttons

The "**Uniform**" scale means that colors are uniformly spread from the minimum to the maximum value.

The "**Across zero**" scale colors the positive values using warm colors (yellow, red), and negative values with cold colors (green, blue).

The program allows to define user-defined scales and save them into the "**Scales manager**" for easier use.



Color scales manager

Color Scale

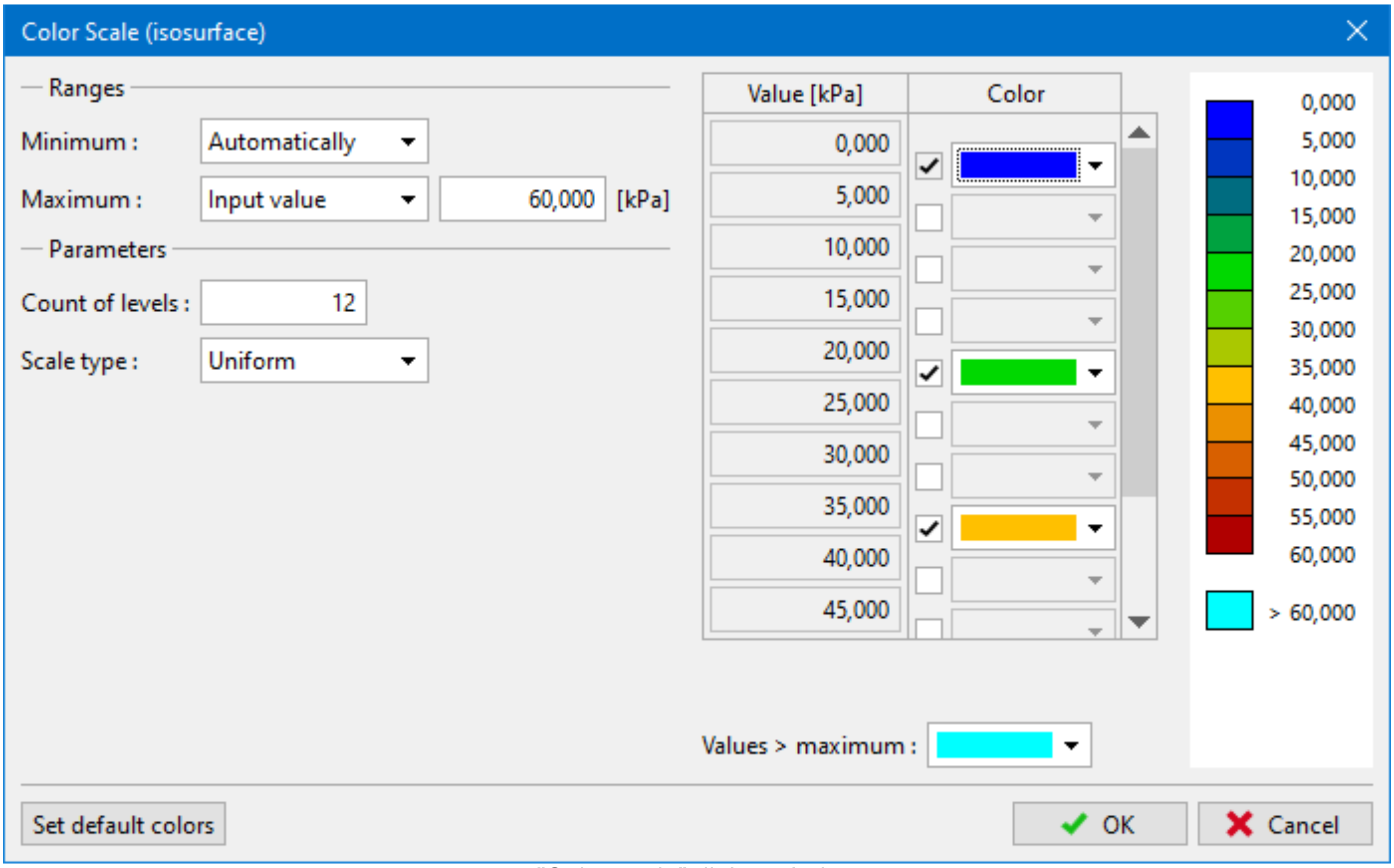
The "**Color scale**" dialog window lets you create a **user-defined color scale**.

In the "**Ranges**" section, the minimum and maximum of scale can be determined automatically, or it can be defined by **input value**. If the scale is limited by input value, The values outside the range (below the minimum or above the maximum value) are drawn with colors specified in the right bottom part of the window. The input minimum and maximum values **are linked to the same unit** - e.g. when specifying the range of 0 - 200 kPa , this range is kept the same for all variables specified in kPa - when changing the currently displayed variable to the settlement variable, the current range is switched to that corresponding to the settlement unit.

The "**Uniform**" scale type means that colors are smoothly spread from the minimum to the maximum value of the scale. The "**Divided by value**" scale type draws the colors the space above the selected value with warm colors (yellow, red) and the space below the selected value with cold colors (green, blue)."Color scale" dialog window

In the window, it is also possible to edit:

- count of levels (minimum 4, maximum 100)
- scale colors
- scale type



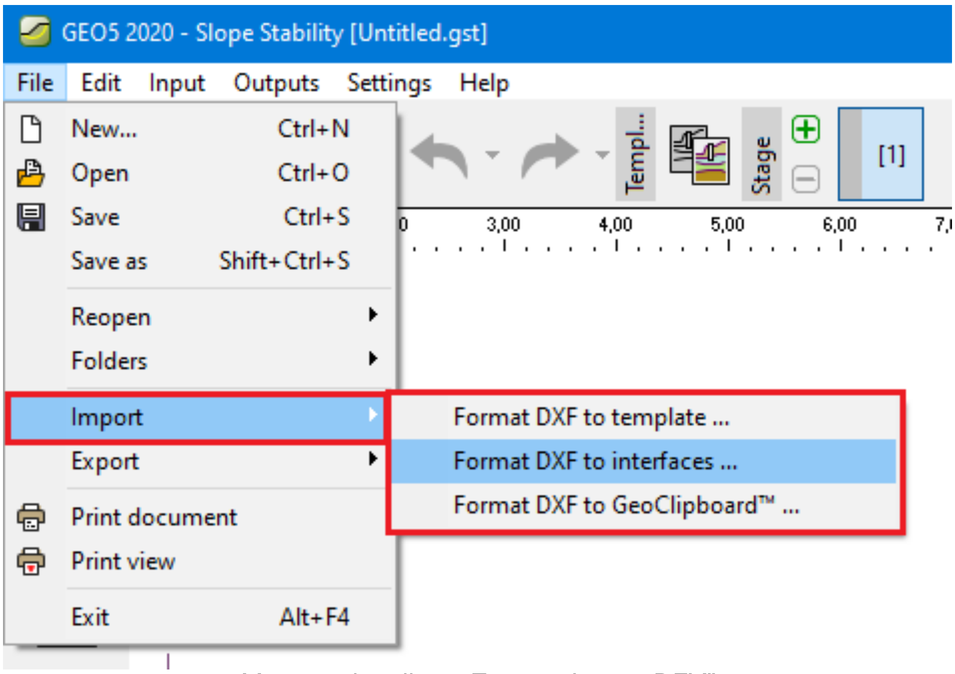
"Color scale" dialog window

DXF Import and Export

2D programs ("**Slope stability**", "**Settlement**", "**FEM**", "**Beam**") allow importing and exporting data in DXF format.

1D programs ("**Cantilever wall**", "**Gabion**", "**Spread footing**", "**Pile**"…) only allow exporting data in the *.DXF format.

The program main menu (the "**File**" item) contains "**Import**" - "**Format DXF to template**", "**Format DXF to interface**", "**Format DXF to GeoClipboard**" and "**Export**" - "**Format DXF**" items.



Menu and toolbar - Export - Import DFX"

Data import proceeds in several steps:

- Loading data into template
- Loading data into interfaces

- Inputting data using a template
- Modifying template during data input

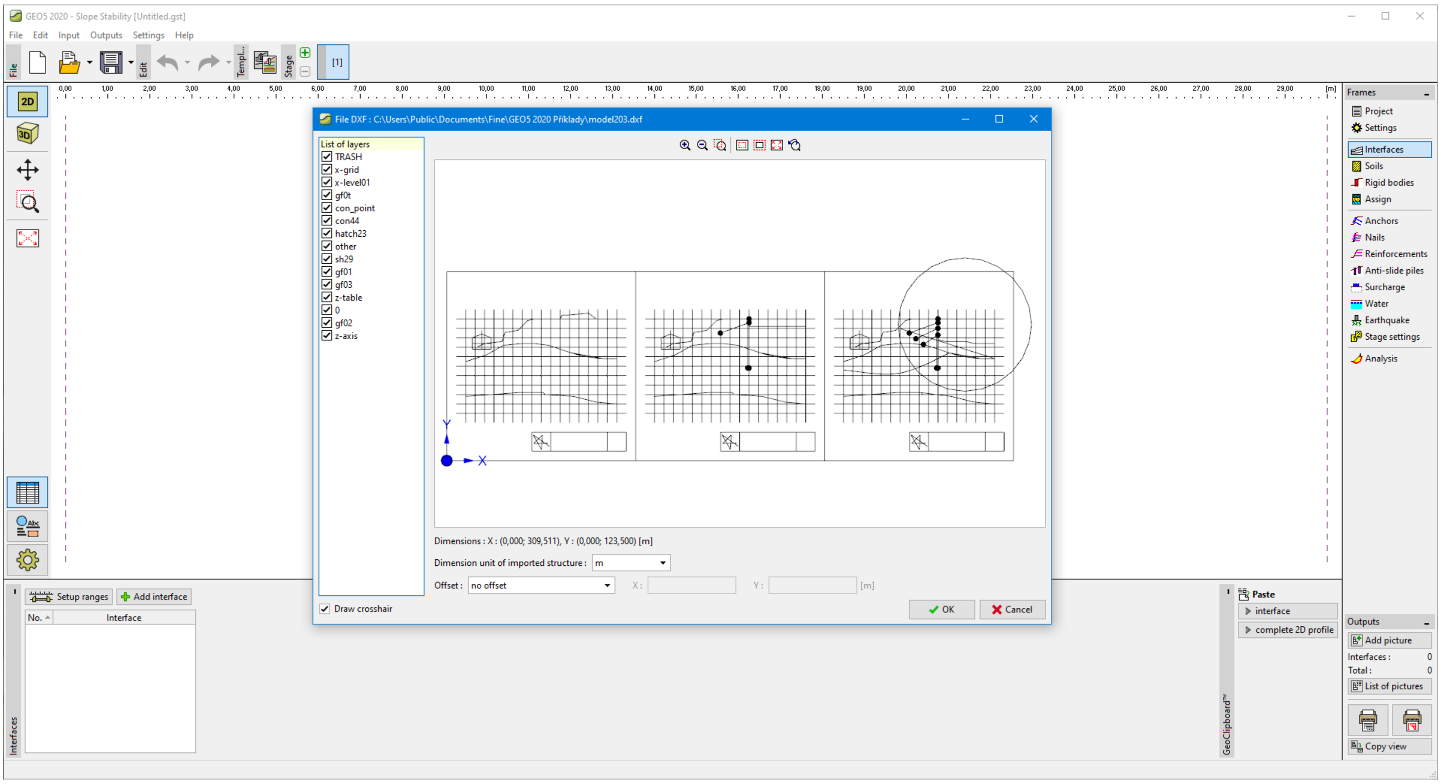
The data input to the program can be exported in DXF format anytime.

Loading Data into Template

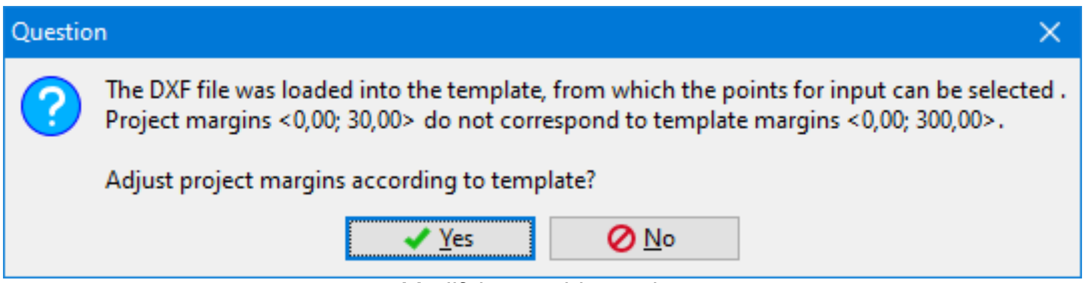
From the program menu ("**File**") select the "**Import**", "**Format DXF into template**" items and then select the file to be imported. The loaded data is displayed in the DXF Import dialog window and then it is loaded into the template. The layer selection can be modified anytime.

When importing data, it is possible to adjust the world margins based on the imported data - this is particularly useful when defining a new task.

The imported data is not transferred directly into the program. It is loaded into a template, through which it is transferred into program data later on. When the data is loaded, the template with data is displayed on the desktop and the buttons on a horizontal toolbar, which is used to manage the template, are made available.



Loading data into a template



Modifying world margins

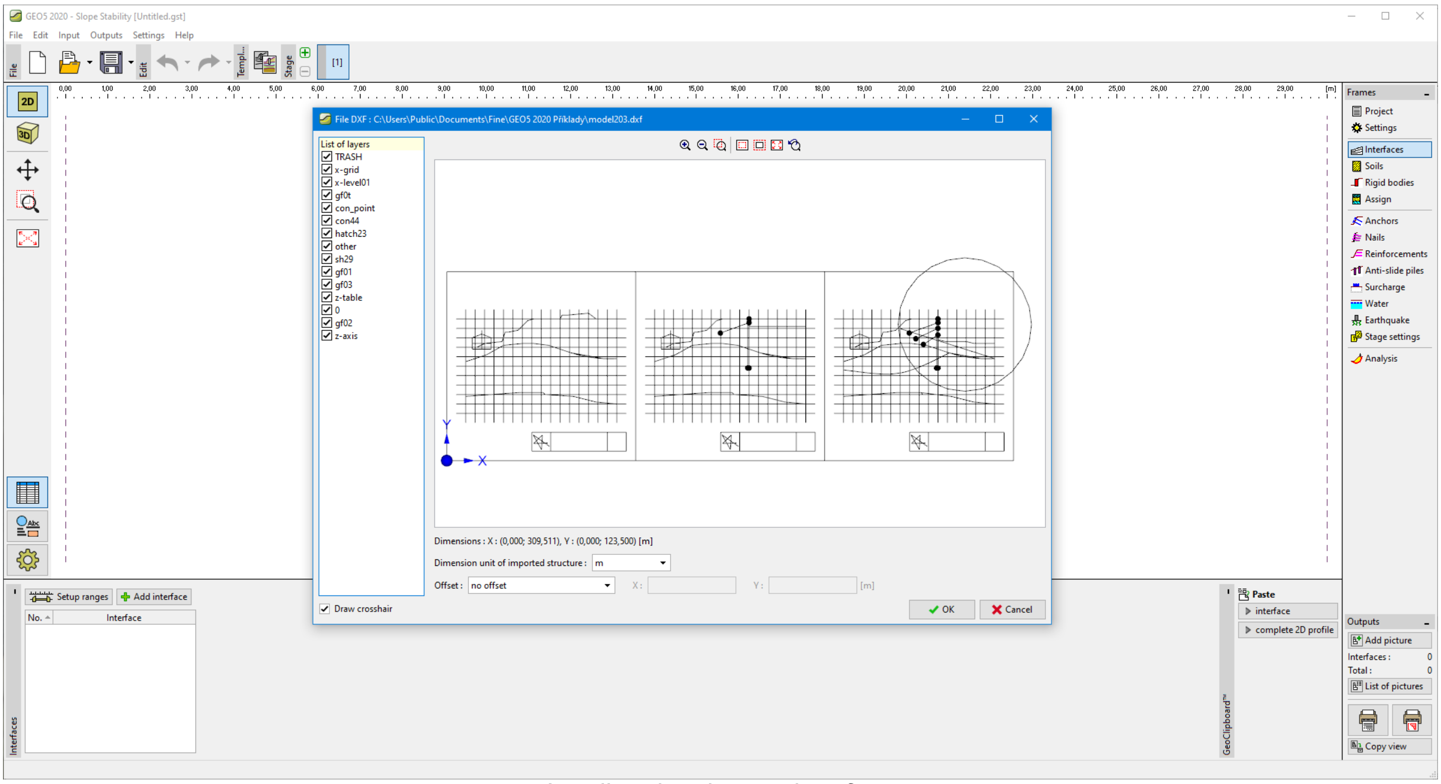
Loading Data into Interfaces

Select the "**Import**", "**Format DXF into interface**" items from the program menu ("**File**"), and then select the file to be imported.

The data is loaded and displayed in the DXF Import dialog window which allows selecting individual layers and specify other parameters. The program automatically adjusts the world margins for the loaded data.

After pressing the "**OK**" button, a new file is created, and DXF data is loaded into interfaces.

If not all selected layers of the structure are successfully loaded, the program allows us to use the loaded DXF file as a template.



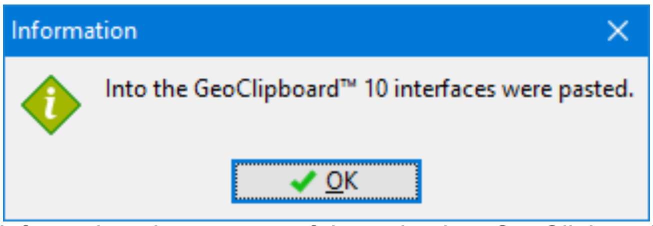
Loading data into an interface

Loading Data into GeoClipboard

Select the "**Import**", "**Format DXF into GeoClipboard**" items from the program menu ("**File**"), and then select the file to be imported.

The data is loaded and displayed in the DXF Import dialog window, which allows selecting individual layers and to specify other parameters.

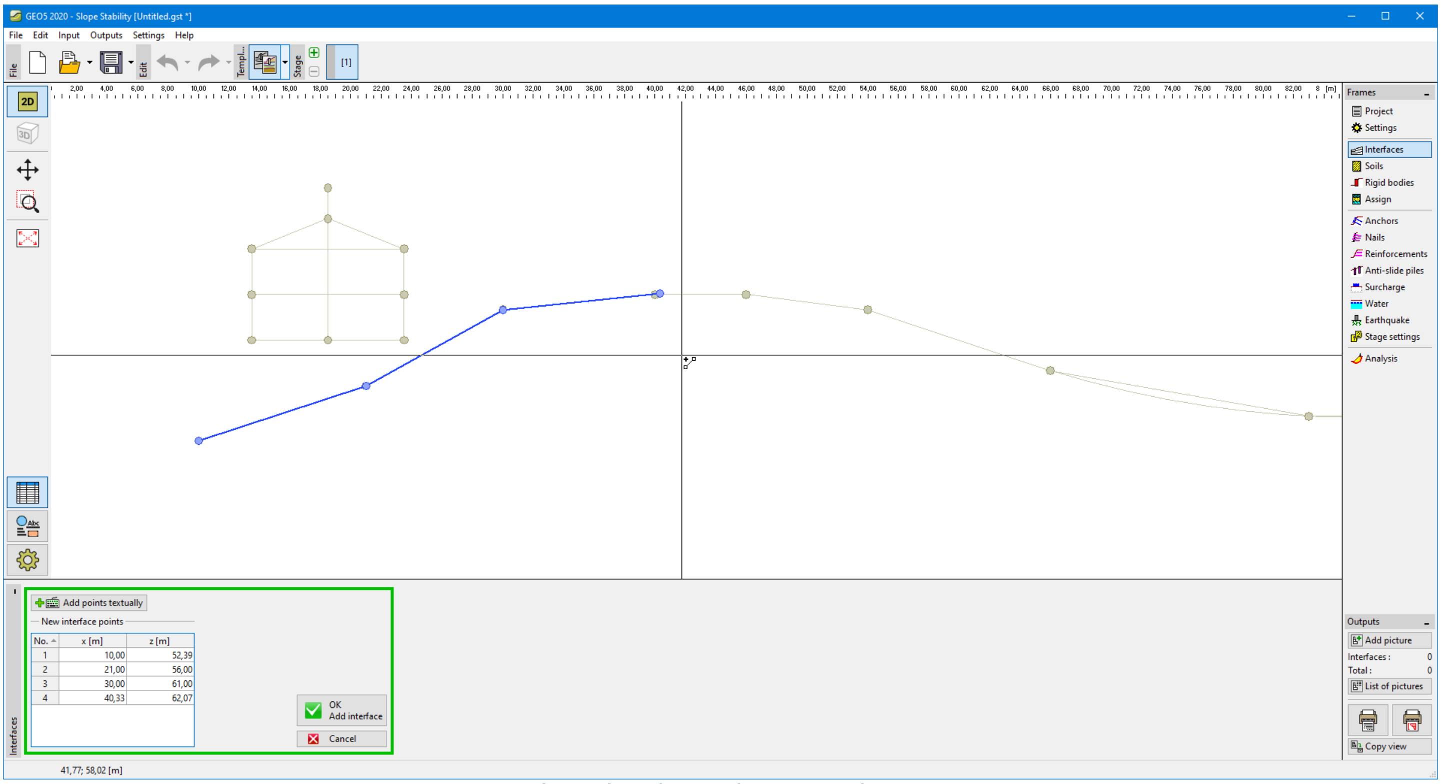
After pressing the OK button, the selected layers of the DXF file are converted into interfaces and copied into GeoClipboard. Data from GeoClipboard can be afterward pasted into various places in the program.



Information about successful copying into GeoClipboard

Data Input using a Template

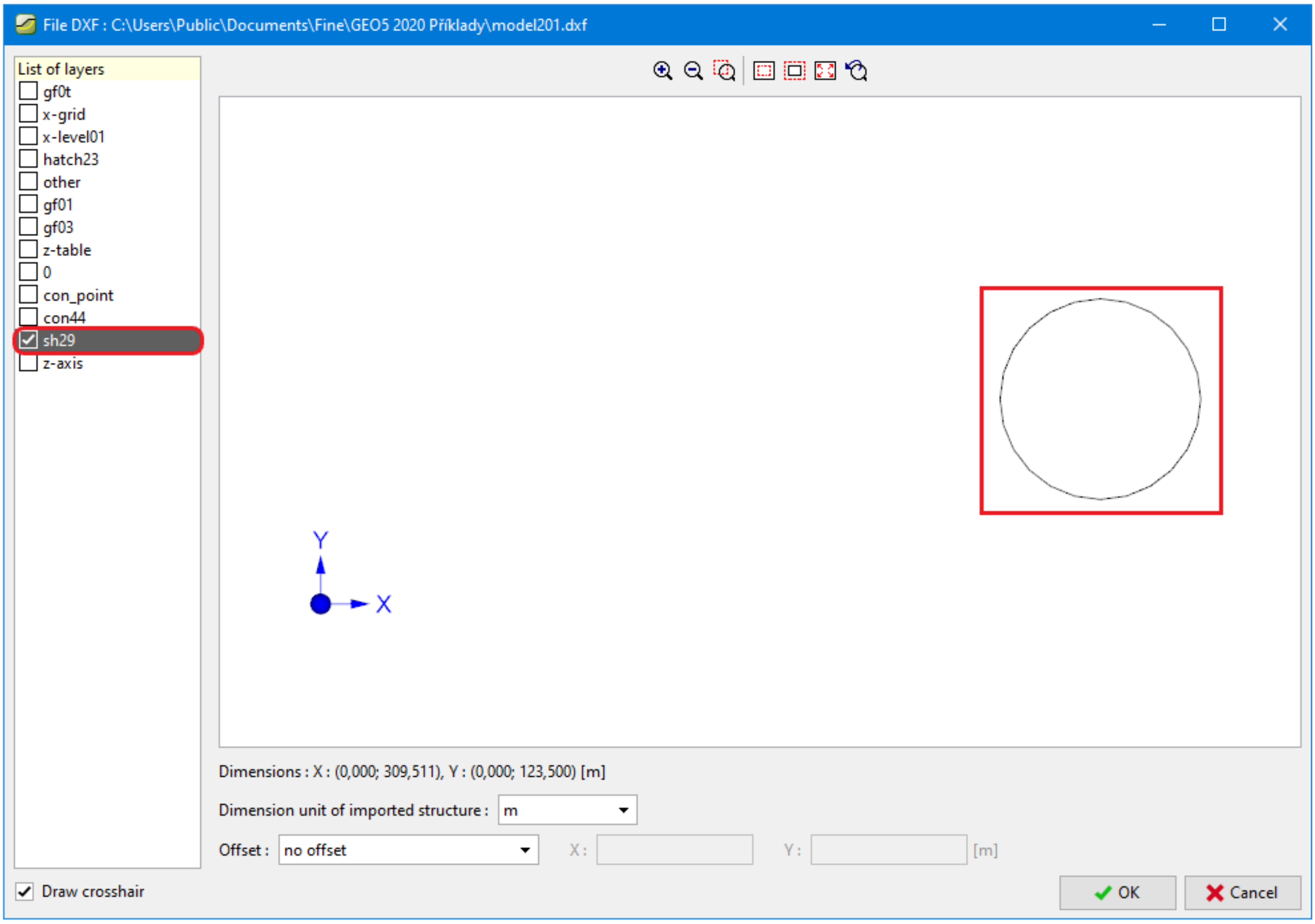
Inputting data using a template is more or less the same process as a standard input of data into the program. The possibility of adding a point from a template to the data being input is the main difference. During the input, the mouse cursor appears as an axial cross - when approaching the template, it turns into a small cross, and long axes disappear. When a point is now input (using the left mouse button), the point from the template is inserted (the input point now has the same coordinates as the point in the template). To accelerate the input of individual lines, it appears useful to employ the zooming tools. After interfaces are put in, the procedure can be applied to inputting other entities. During data input, it is possible to modify the template anytime.



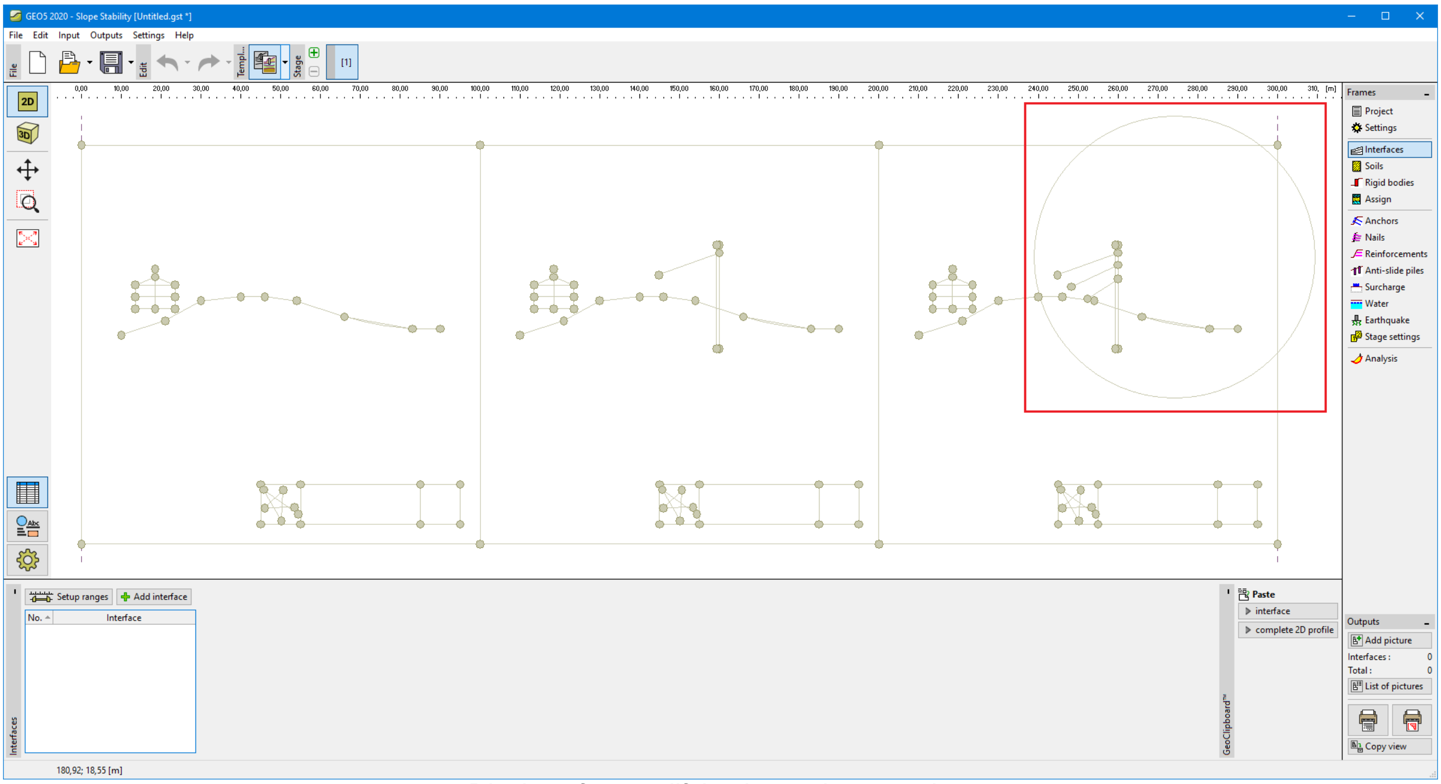
Inputting data using a template

Modification of Template During Data Input

When inputting data, the template can be modified anytime by pressing the "**Modify**" button on the "**Template DXF**" toolbar. This opens a dialog window with individual layers of the template. For example, when inputting anchors, it is possible to turn off all layers except anchors - inputting anchors then becomes simple and clear.



Turning on/off layers in a template



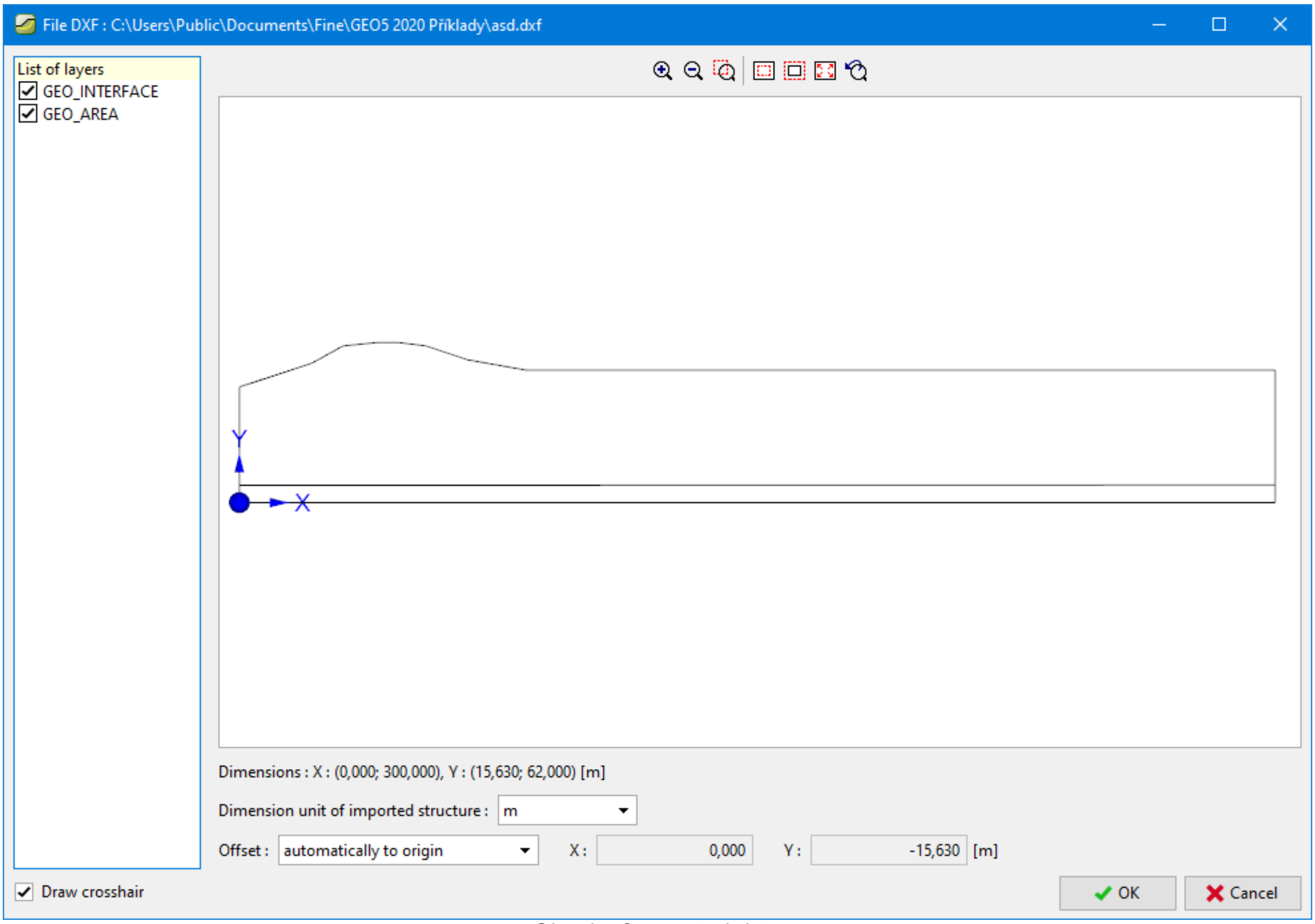
Display after modifying layers in a template

DXF Export

From the program menu ("**File**" item) select the "**Export**" item, "**Format DXF**". Next, select the file name intended for the export. Using a dialog window, the program then provides information on the performed data export.

Information regarding the performed data export

The exported data can be verified by importing them back into the program.



Check of exported data

DXF Import

In the DXF Import dialog window, the parameters of the DXF import are specified.

In the upper part, there is a preview of the imported data.

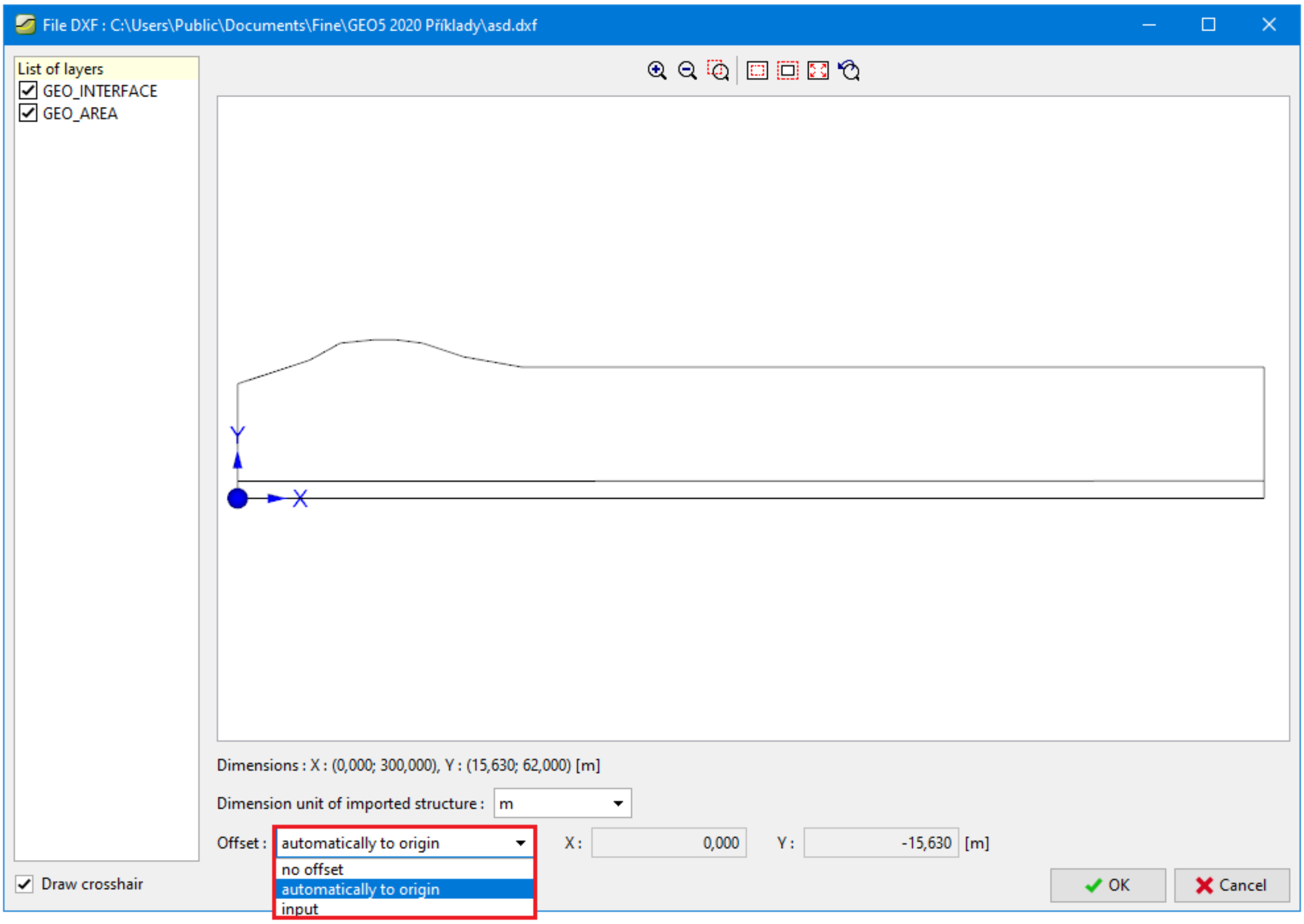
In the left part, the layers to be read can be selected.

In the lower part, the unit used when creating the DXF file can be specified. The program attempts to estimate it, but it is **always necessary to check** whether the unit is specified correctly.

The structure can also be moved. The program offers the following options:

- **do not move** - this option reads the data the way they were entered
- **automatically from zero** - this option moves the left bottom corner of the structure to the coordinate system origin
- **input** - this option allows defining the shift manually

If the program does not allow inputing arcs and circles, the way of splitting it into line segments can be specified.



Dialog window for DXF Import and its parameters

Table Data Import

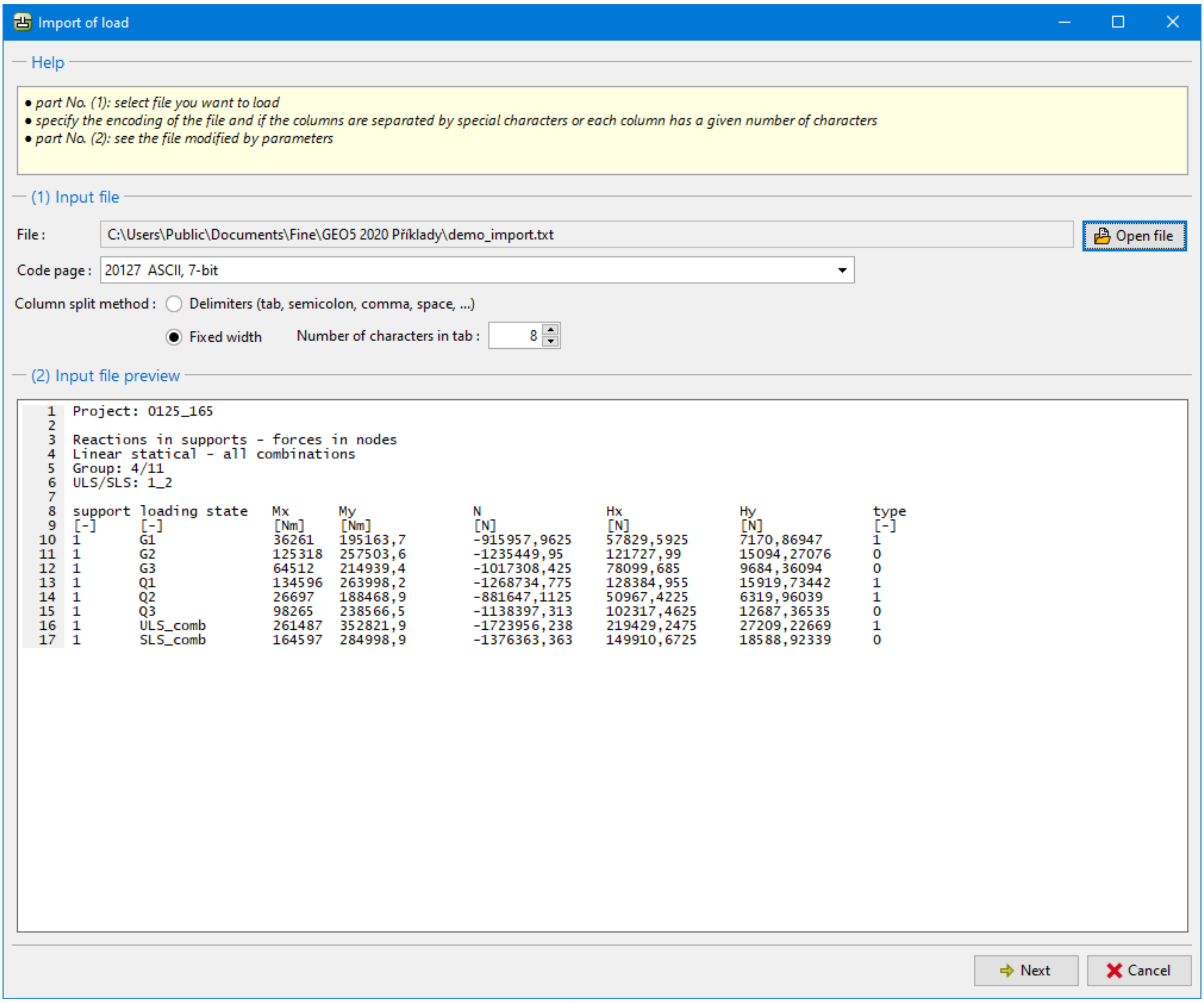
Using this tool, it is possible to import table data (i.e. data organized into columns) into tables in the program, e.g. load, coordinates, etc. The tool can read the following formats:

- a text file separated by **delimiters**, e.g. commas, semicolumns, tabs, CSV file extension
- a general text file with a **fixed width of columns**, data is organized into columns using spaces or tabs, most frequent TXT file extension
- Microsoft Office Excel **tables** (XML Office Open) - the XLSX file extension
- OpenOffice system **tables** (OpenDocument) - the ODS file extension

The import is organized gradually into three steps, which moreover varies according to the loaded format. At the top of the window, there is a brief help displayed. Analyzing the format and the file content, the program attempts to suggest the best parameters of the transfer. If the user changes some parameters, the program tries to remember these changes and use them appropriately for other files.

1st step: select a file, determine a file type and view its contents

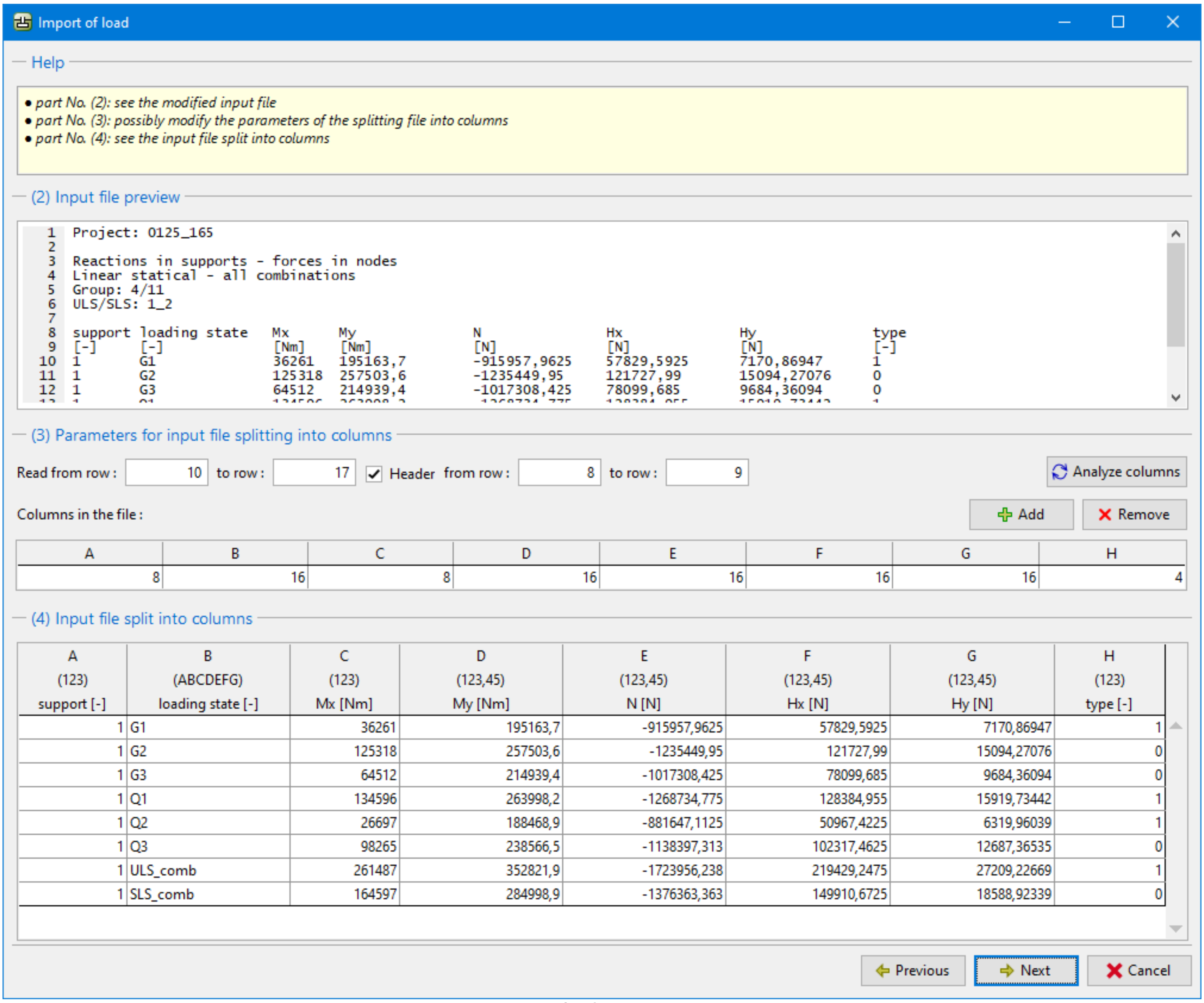
In this step, the visible parts are (1) Input file and (2) Input file preview.



1st step

2nd step: splitting the input file into columns

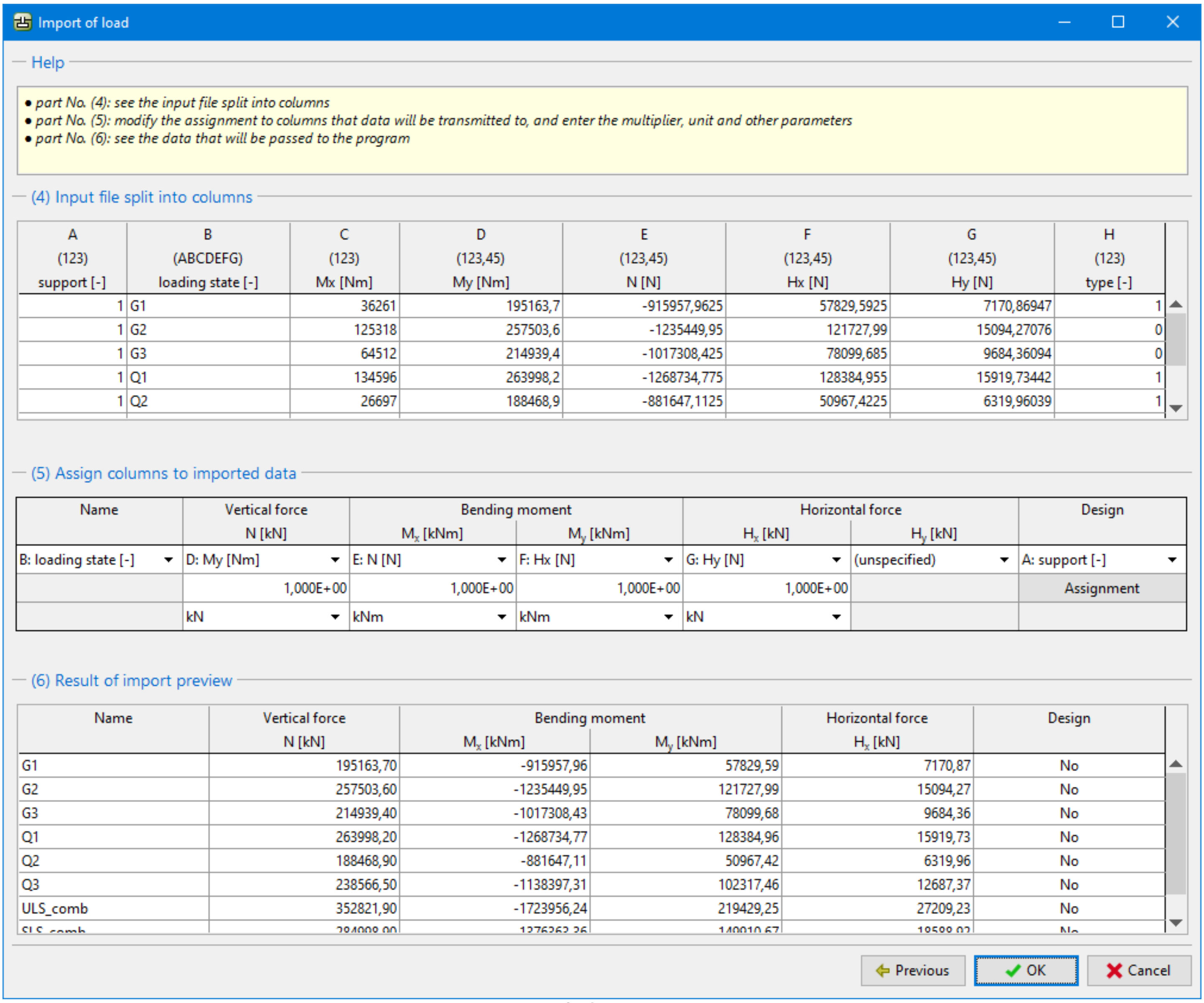
In this step, the visible parts are (2) Input file preview, (3) Parameters for input file splitting into columns, and (4) Input file split into columns.



2nd step

3rd step: Assigning columns to data

In this step, the visible parts are (4) Input file split into columns, (5) Assign columns to imported data, and (6) Result of import preview.



3rd step

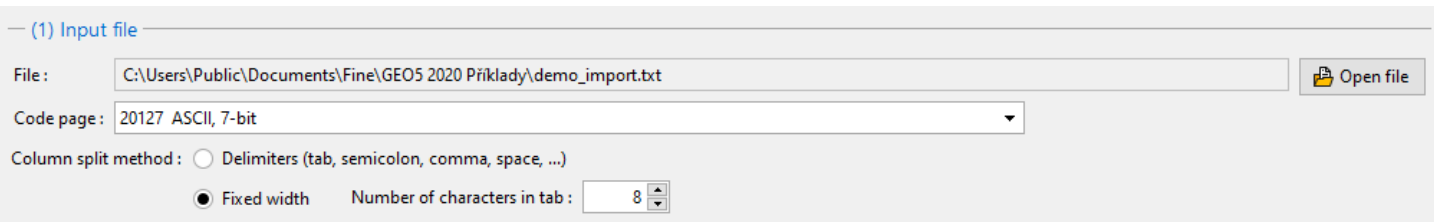
After pressing the "**OK**" button, the data is transferred into the program.

(1) Input File

In this section, the input file and its basic parameters are specified. The file is opened in the standard way by pressing the "**Open file**" button. The program analyzes the input file and fills in the data in this section.

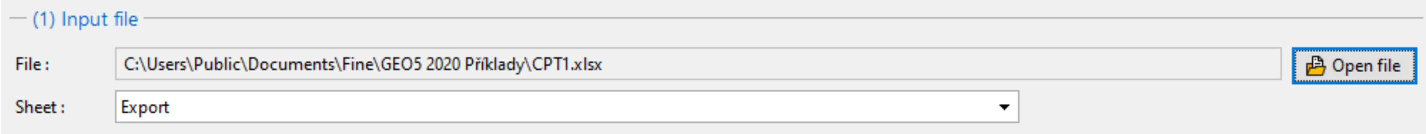
If the **text file** is being imported, the following parameters are determined:

- Encoding - encoding (language) in which the file is written can be changed
- The way of separating columns - it is specified whether the file is separated by special characters (which are then entered here) or whether the columns have a fixed width
- The number of characters in a tab - the program replaces the tabs with spaces for further processing, this parameter can affect the way how



Text file

If the **spreadsheet file** is imported (e.g. Excel), here it is possible to determine which sheet is imported.

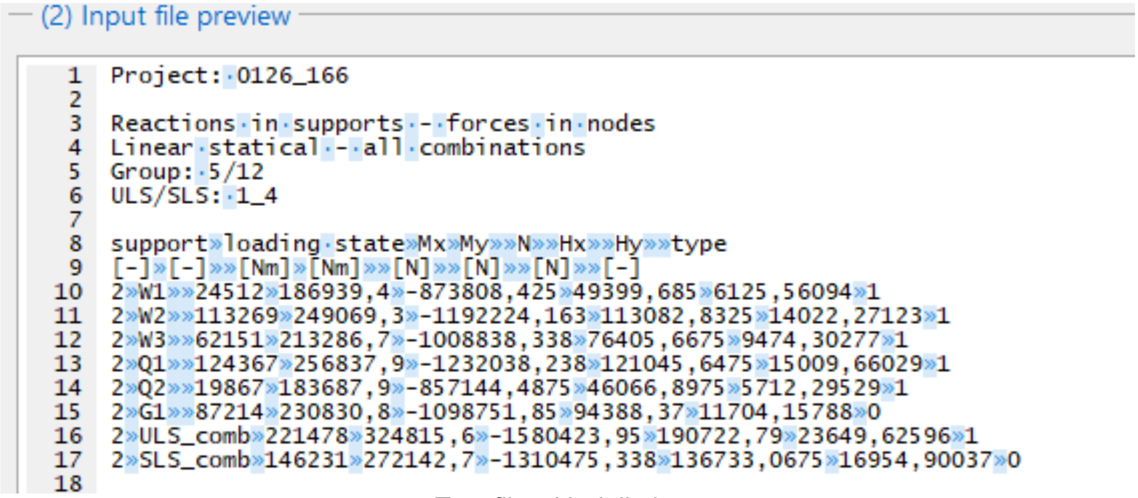


Spreadsheet file

In both cases, it is possible to check the result in (2) Input file preview. If everything is OK, go on by clicking "**Next**".

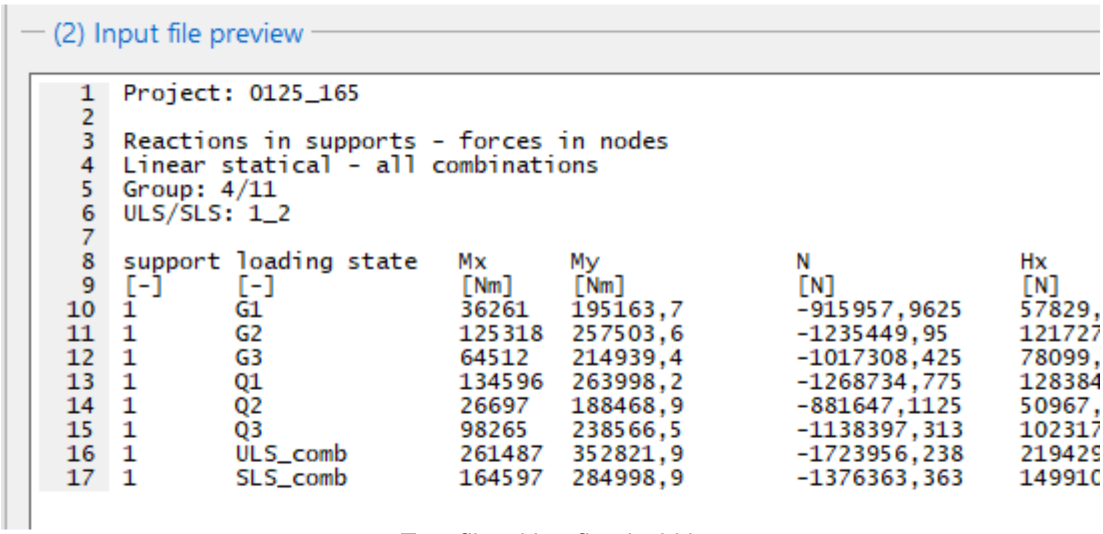
(2) Input File Preview

If the **text file** with **delimiters** is being imported, commonly used delimiters are highlighted in the preview.



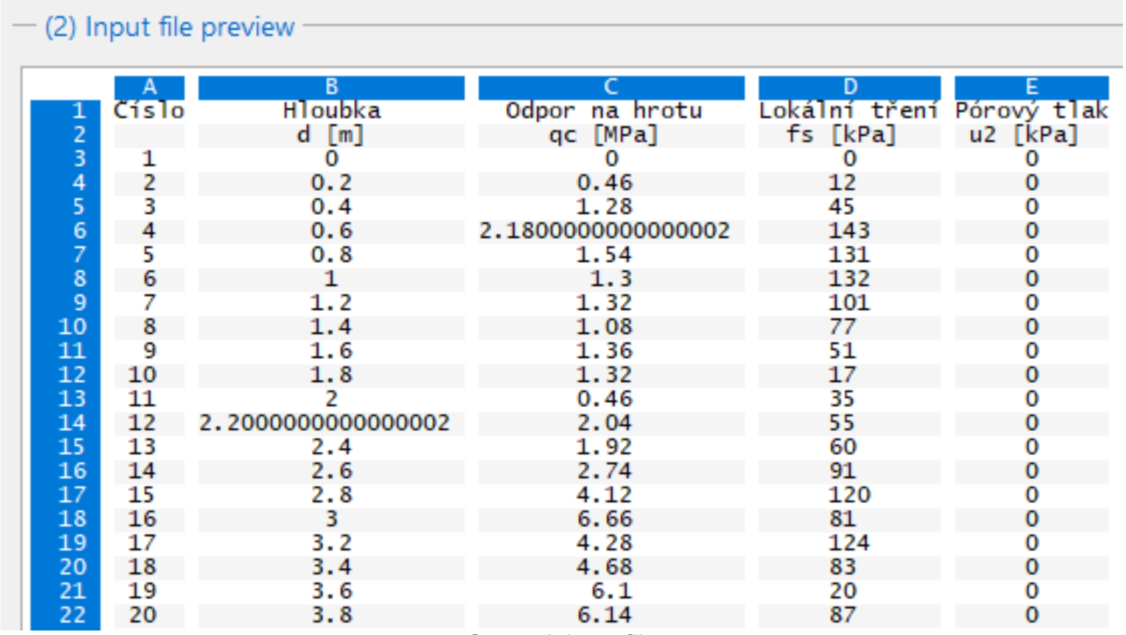
Text file with delimiters

If the **text file** with a **fixed width** is being imported, the preview looks like this.



Text file with a fixed width

If the **spreadsheet file** is being imported, the preview contains cell addresses.



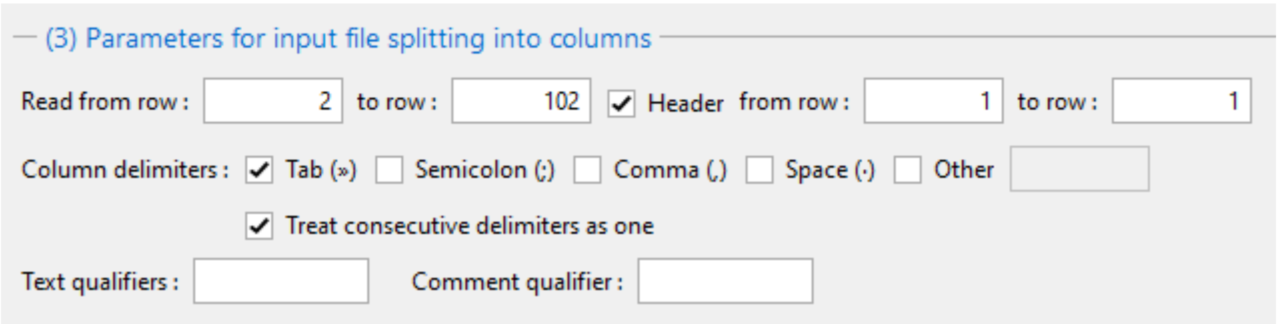
	A	B	C	D	E
1	Číslo	Hloubka	Odpor na hrotu	Lokální tření	Pórový tlak
2		d [m]	qc [MPa]	fs [kPa]	u2 [kPa]
3	1	0	0	0	0
4	2	0.2	0.46	12	0
5	3	0.4	1.28	45	0
6	4	0.6	2.1800000000000002	143	0
7	5	0.8	1.54	131	0
8	6	1	1.3	132	0
9	7	1.2	1.32	101	0
10	8	1.4	1.08	77	0
11	9	1.6	1.36	51	0
12	10	1.8	1.32	17	0
13	11	2	0.46	35	0
14	12	2.2000000000000002	2.04	55	0
15	13	2.4	1.92	60	0
16	14	2.6	2.74	91	0
17	15	2.8	4.12	120	0
18	16	3	6.66	81	0
19	17	3.2	4.28	124	0
20	18	3.4	4.68	83	0
21	19	3.6	6.1	20	0
22	20	3.8	6.14	87	0

Spreadsheet file

(3) Parameters for Input File Splitting into Columns

The program analyzes the input file and fills out this part with the obtained parameters.

If the **text file** with **delimiters** is being imported, the following parameters are specified:

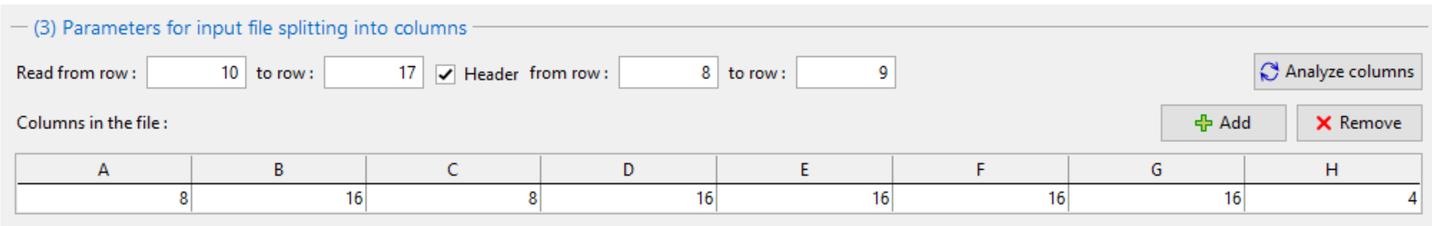


Text file with delimiters

- the first and the last row to be loaded are determined if any row contains a header and possibly the first and the last row of the header
- determine the column separators by checking the switches of each type, or check "**Other**" and add another separator into the input line
- the "**Treat consecutive delimiters as one**" switch determines how the program will handle delimiters following straight one after another (even various types of delimiters)
- text qualifiers specify whether the text columns are marked left and right with a character
- comment qualifier specifies a character from which the contents of the file is ignored to the end of the row

The "**Analyse delimiters**" button reanalyses the parameters after changing the row range.

If the **text file** with a **fixed width** is being imported, a column count and a width of each column is specified.



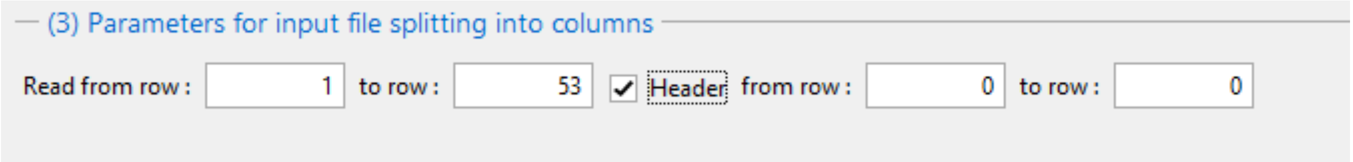
Text file with a fixed width

- the first and the last row to be loaded are determined if any row contains a header and possibly the first and the last row of the header
- the "**Add**" button inserts a column at the end of the list, the "**Remove**" button removes the last column

- in the "**Columns in the file**" table, a width of each column is specified

The "**Analyse columns**" button reanalyses the parameters after changing the row range.

If the **spreadsheet file** is being imported, only the first and the last row to be loaded are determined if a row contains a header and possibly the first and the last row of the header



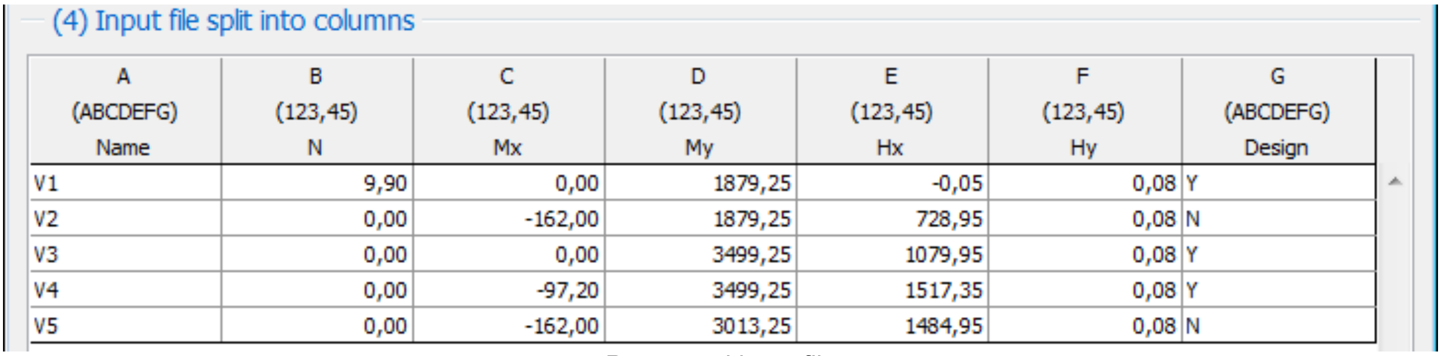
Spreadsheet file

In all cases, it is possible to check the result in part (4) Input file split into columns. If everything is OK, go on by clicking "**Next**".

(4) Input File Split into Columns

In this part, the input file split by parameters is displayed. The first header row contains the letters starting from A, the second row specifies the type of data in the column, the third header row possibly contains the retrieved header from the imported file. There can occur the following data types:

- (ABCDEFG) is a general text
- (123,45) is a number with the decimal point
- (123) is a number without the decimal point


(4) Input file split into columns

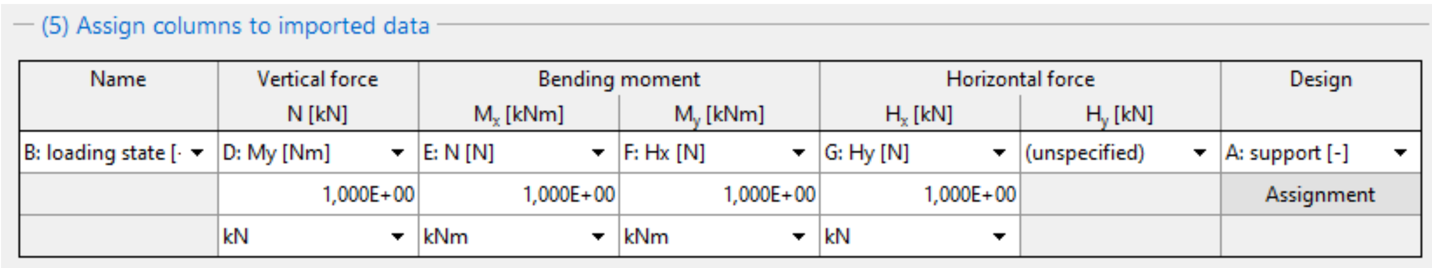
A (ABCDEFG) Name	B (123,45) N	C (123,45) Mx	D (123,45) My	E (123,45) Hx	F (123,45) Hy	G (ABCDEFG) Design
V1	9,90	0,00	1879,25	-0,05	0,08	Y
V2	0,00	-162,00	1879,25	728,95	0,08	N
V3	0,00	0,00	3499,25	1079,95	0,08	Y
V4	0,00	-97,20	3499,25	1517,35	0,08	Y
V5	0,00	-162,00	3013,25	1484,95	0,08	N

Processed input file

(5) Assign Columns to Imported Data

The program prepares the initial assignment of columns. It is then possible to change the assignment manually. The system remembers the user changes and uses it, preferably in the same cases.

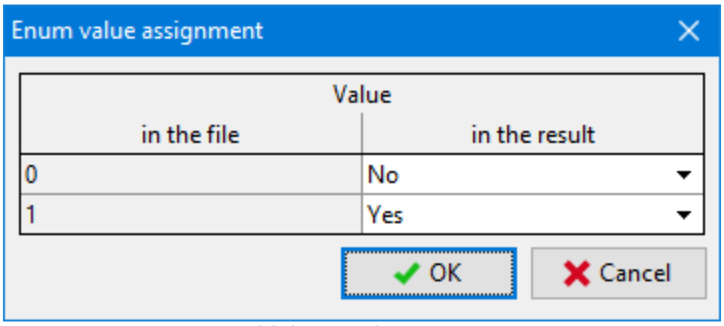
Changes in the assignment of columns are carried out in the following table:


(5) Assign columns to imported data

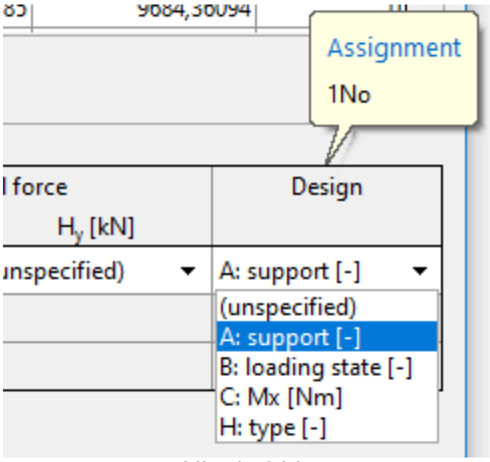
Name	Vertical force N [kN]	Bending moment M_x [kNm]	 M_y [kNm]	Horizontal force H_x [kN]	 H_y [kN]	Design
B: loading state [·	D: My [Nm]	E: N [N]	F: Hx [N]	G: Hy [N]	(unspecified)	A: support [-]
	1,000E+00	1,000E+00	1,000E+00	1,000E+00		Assignment
	kN	kNm	kNm	kN		

Column assignment

- The table header contains columns that are required by the program mode into which the import is being done.
- In the first table row, the input file column is assigned to be imported into the appropriate column of data. The columns can be used repeatedly, but you can only assign a compatible data type, i.e. you cannot use for example a text column for a number.
- In the second table row, a multiplier for numbers can be entered. Pressing the "**Assign**" button in the window enables us to specify how the values of the **Yes/No** type and similar ones are treated. The current assignments and other information are displayed in the hint-bubble if the appropriate column is focused on.



Value assignment



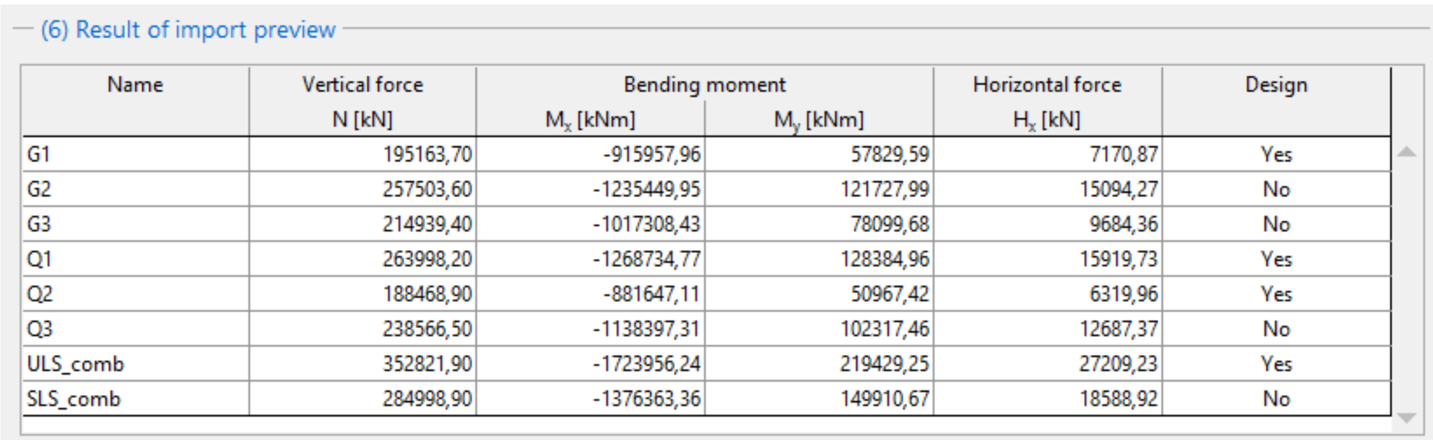
Hint bubble

In all cases, it is possible to check the result in part (6) Result of import preview. If any problems arise, an error message is displayed in the window.

If everything is OK, you can complete the import by pressing the "**OK**" button.

(6) Result of Import Preview

This section shows the data that is going to be imported into the program.



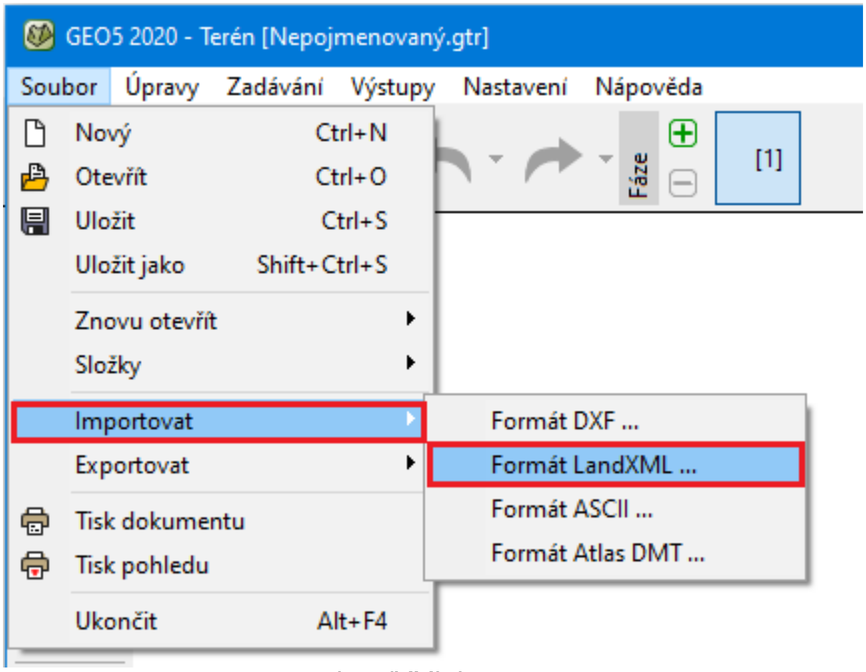
(6) Result of import preview

Name	Vertical force N [kN]	Bending moment M_x [kNm]	M_y [kNm]	Horizontal force H_x [kN]	Design
G1	195163,70	-915957,96	57829,59	7170,87	Yes
G2	257503,60	-1235449,95	121727,99	15094,27	No
G3	214939,40	-1017308,43	78099,68	9684,36	No
Q1	263998,20	-1268734,77	128384,96	15919,73	Yes
Q2	188468,90	-881647,11	50967,42	6319,96	Yes
Q3	238566,50	-1138397,31	102317,46	12687,37	No
ULS_comb	352821,90	-1723956,24	219429,25	27209,23	Yes
SLS_comb	284998,90	-1376363,36	149910,67	18588,92	No

Result of the import

Import LandXML

The "**Terrain**" and "**Stratigraphy**" programs allow you to import data in the LandXML format. Select the "**Import**", "**Format LandXML**" items in the program menu ("**File**"), and then select the file to be imported in a standard way.



LandXML Import

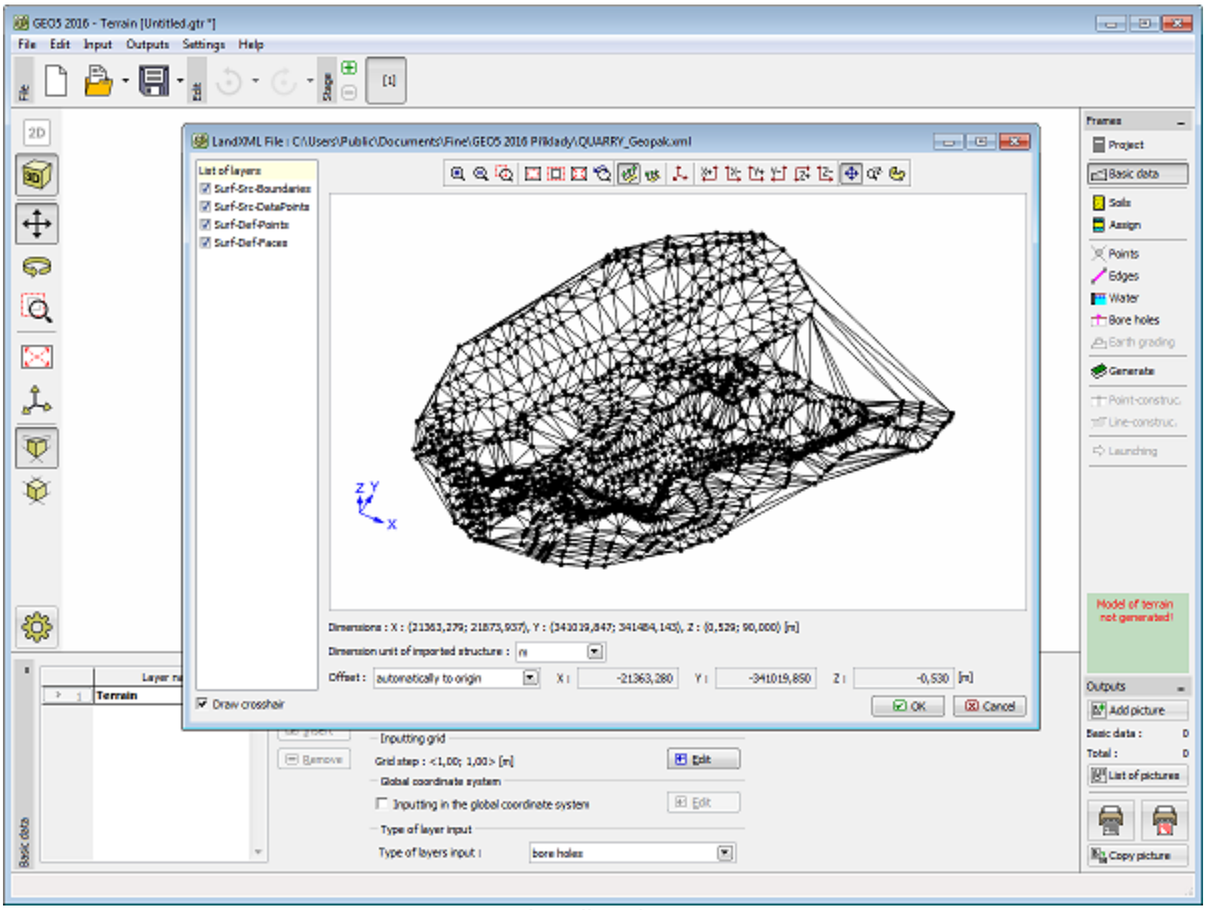
The data is loaded and displayed in the dialog window, which allows selecting individual layers to be loaded as points and interfaces. The same dialog window also allows us to modify the unit used when creating the LandXML file. The structure can also be moved. The program offers the following options:

- **do not move** - this option reads data in the same way as they were input
- **automatically from zero** - this option moves the left bottom corner of the structure to the coordinate system origin
- **input** - this option allows us to define the shift manually

After the import having been finished, the program automatically adjusts the world margins based on the loaded data.

Supported LandXML elements: Units, Alignments, CgPoints, Parcels, PlanFeatures, Roadways, Surfaces, Survey.

Not supported LandXML elements: GradeModel, Spiral curves except clothoid.

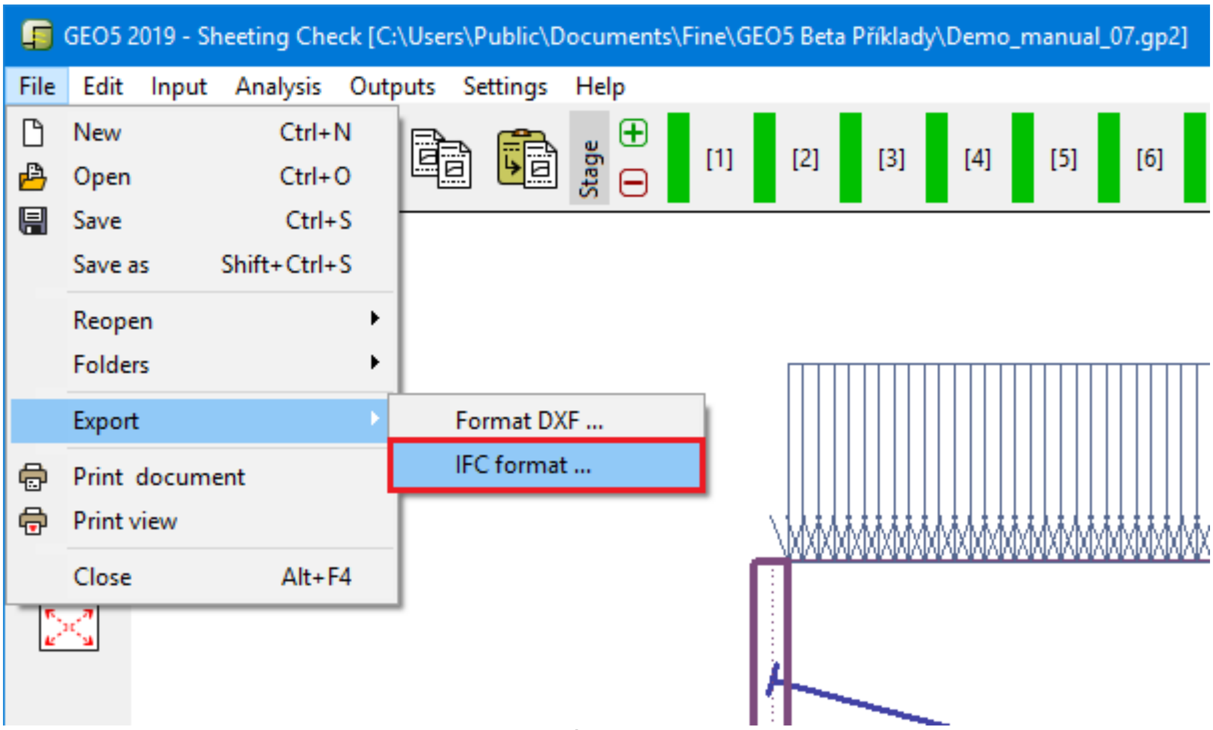


Loading data

Export IFC (BIM)

GEO5 programs allow exporting data into the **IFC format**. This format is used as an **exchange platform between the programs in the building information modeling area (also known as BIM)**.

The option of the export is available from the menu "**File**"/"**Export**"/"**IFC format**".

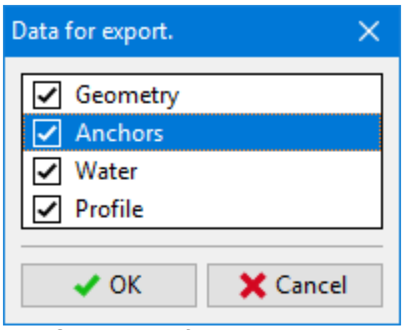


IFC Export

The programs allow exporting **graphical data** corresponding with the constructions in the **3D visualization**. Another part of the IFC export is non–graphical data which can be imported into third-party programs. As an example of an export we may set:

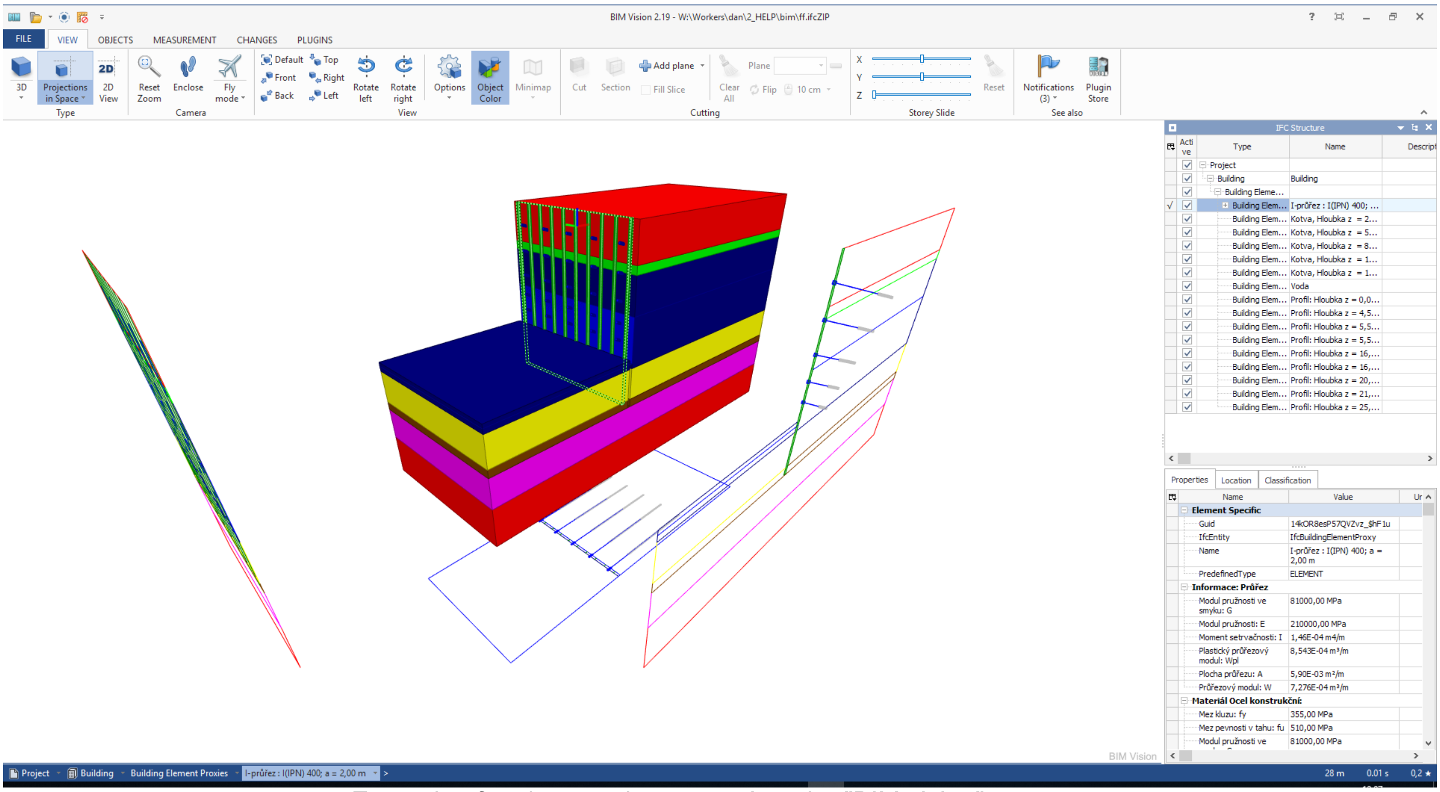
- material properties of the structure (concrete, steel, masonry)
- soil parameters (2D interfaces, 1D profile, front face resistance...)
- cross-sectional properties of shoring structures (piles, sheet piles, rectangular walls...)
- parameters of anchor elements (anchors, props, nails)
- properties next to the structure (surcharge, water, front face resistance, applied forces...)
- basic parameters of field tests and soil profiles

Before the export, it is possible to select the required data.



Selection of required data

The final structure of visualized data depends entirely on the program which we use. For clarity and functionality, we can recommend, for example, *BIM Vision, Tekla BIMsight, Xbim Xplorer*.



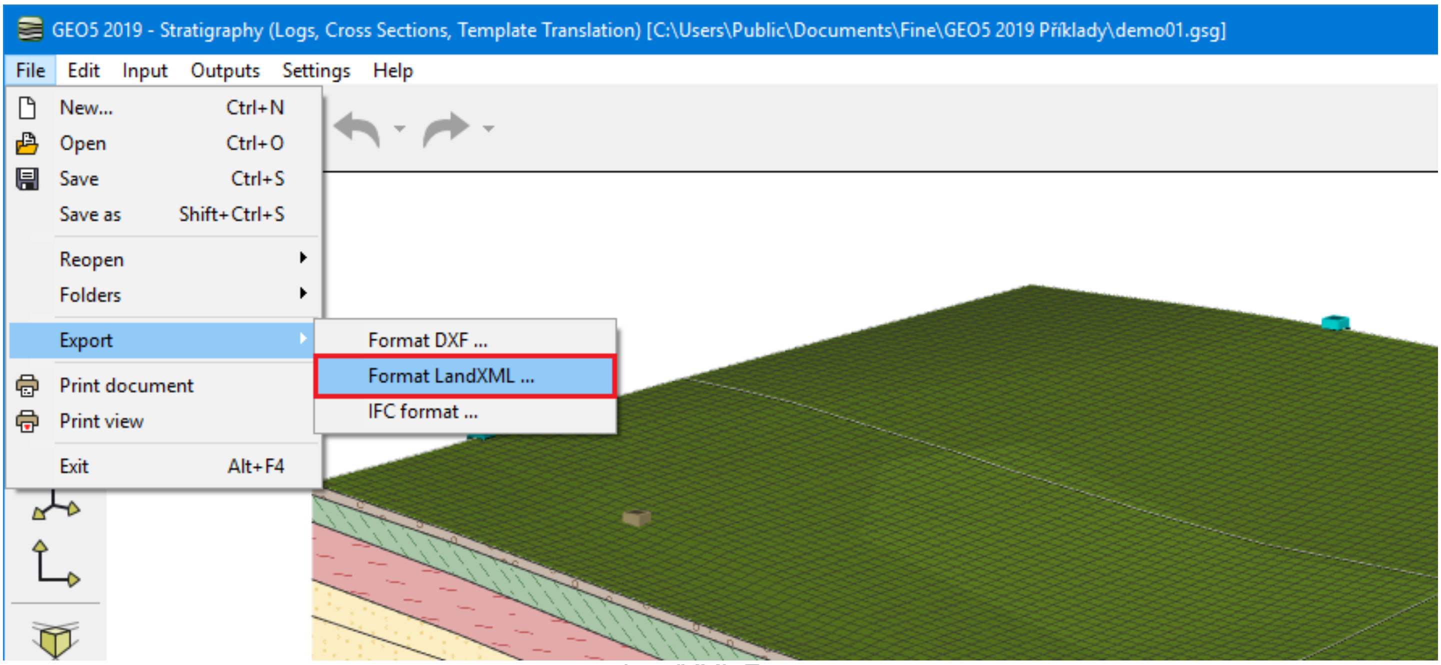
Example of an imported structure into the "BIM vision" program

The export of IFC data is going to be clarified and amended within the next development according to real needs.

Export LandXML

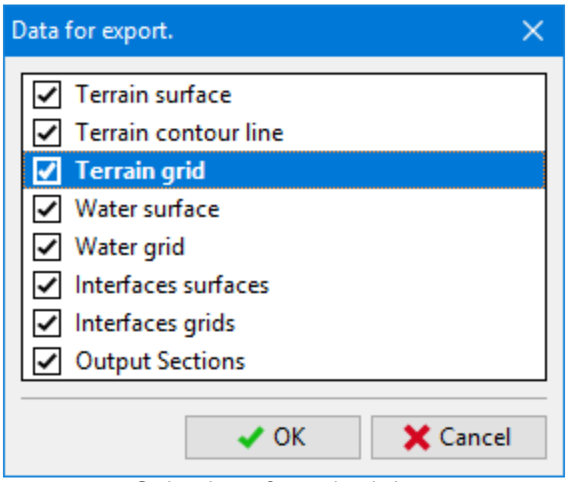
The **Stratigraphy** program allows exporting data into the **LandXML format**. This format is used as an **exchange platform between the programs in the area of civil engineering, geological surveys, and GIS**.

The option of the export is available from the menu "**File**" / "**Export**" / "**LandXML Format**".



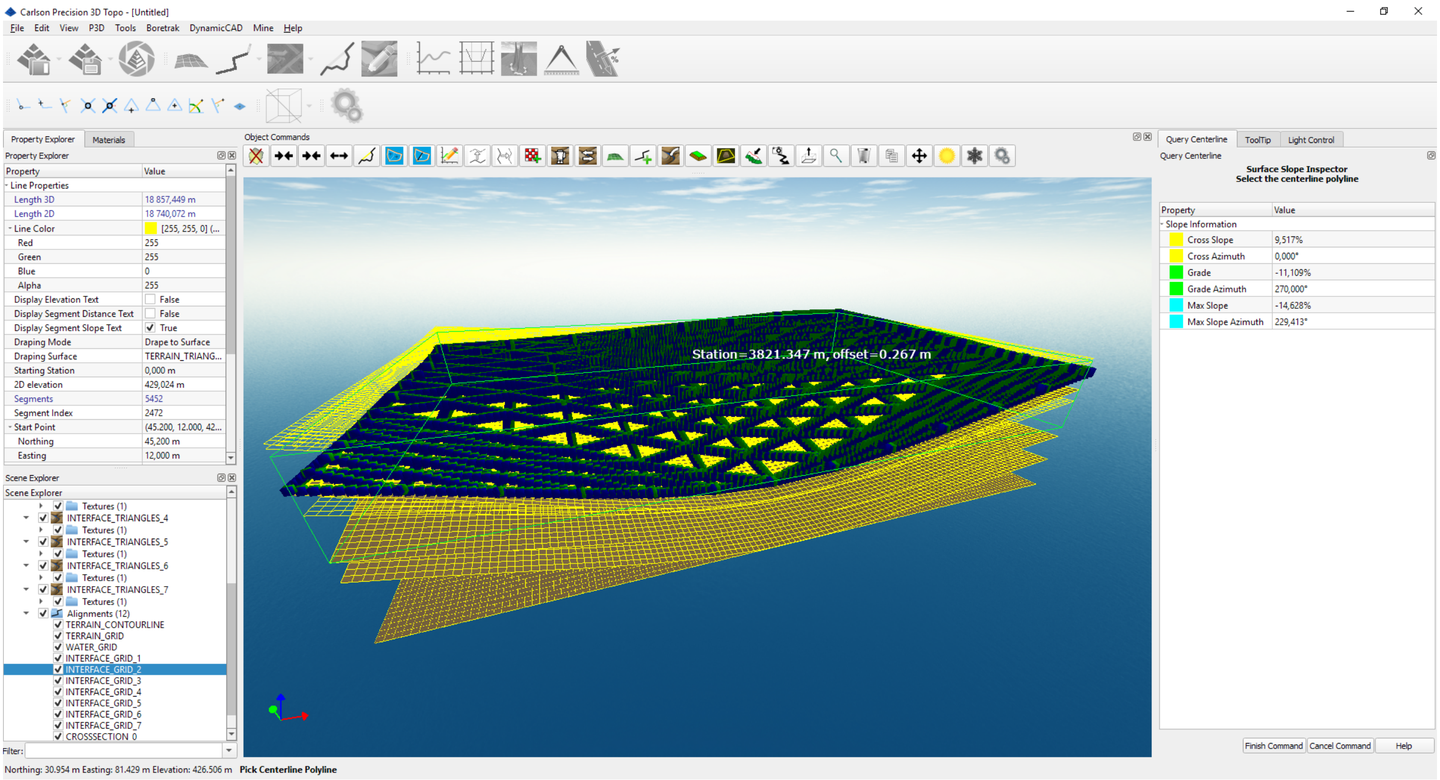
LandXML Export

The program allows us to export **graphical data** corresponding with the constructions in the **3D visualization**. Before the export, it is possible to select the required data for the final LandXML file.



Selection of required data

The final structure of visualized data may depend on the program that we use. For clarity and functionality, we can recommend e.g. "***Carlson Precision 3D***" program.



Example of an imported structure into the "Carlson Precision 3D" program

The export of LandXML data is going to be clarified and amended within the next development according to real needs.

Heredity - Construction Stage

Construction stages (Toolbar Construction Stages) allow creating the construction step by step and check all construction stages (this is necessary in the **Sheeting Check, Settlement, FEM** programs).

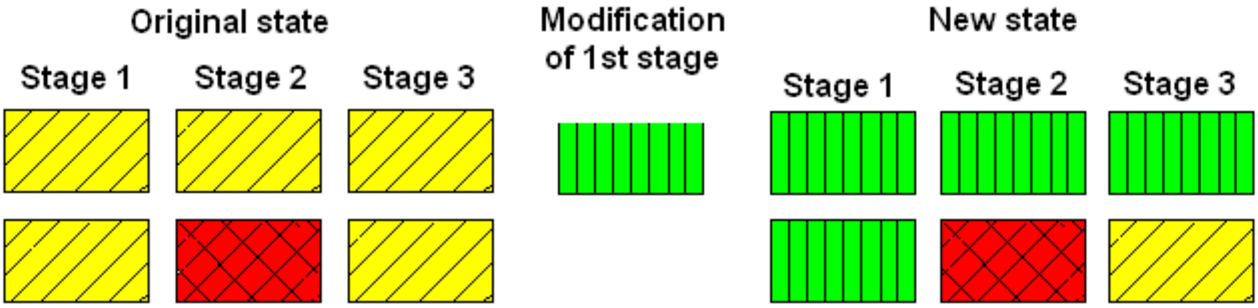
When creating a new stage of the construction, or editing the previous stage, the properties of the construction are **automatically passed over** between the construction stages (heredity of properties).

There are two types:

Defined heredity - (anchors, supports, surcharges…) - these objects always remember the stage in which they were created. This is also the stage where these objects can be edited. In all subsequent stages, these objects can be either removed, or it is possible to change some of their properties (post-stressing anchor, change of surcharge magnitude, translating support…).

Automatic heredity - (assigning of soils, terrain profile, influence of water, analysis setting…) - for such types of inputs, the properties from the previous stage are carried over to a new one when being created. When the properties in the current construction stage are changed, the program proceeds as follows:

- If the property in the next stage remains the same as in the previous one, it also receives the tag new - this change also applies to all subsequent stages.
- If the property in the next stage differs from the one in the previous stage (this means that this property has already been changed in the next stage), then this change is not carried over to subsequent stages.

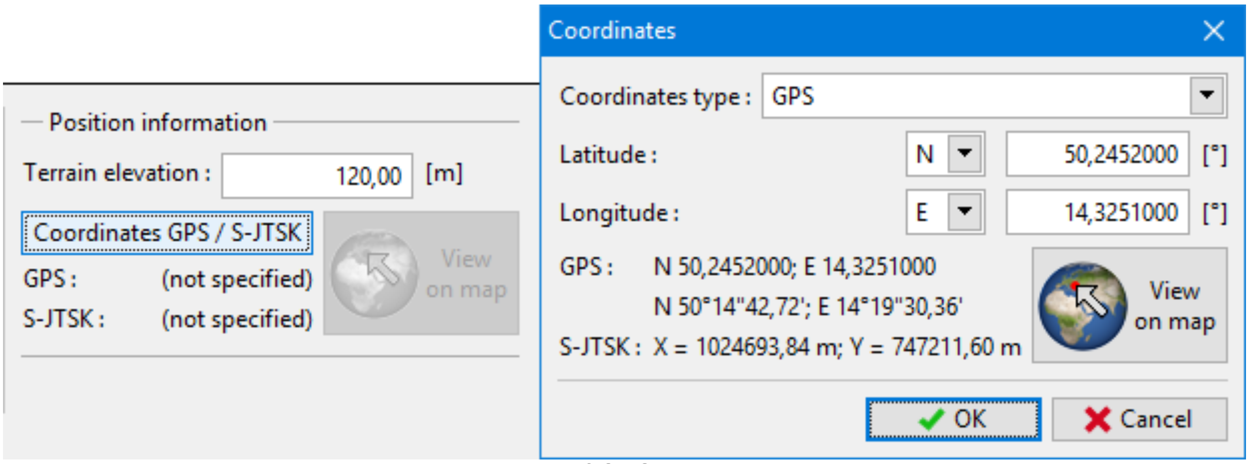


Changes within stages of a construction - automatic heredity

Site view on Google Maps

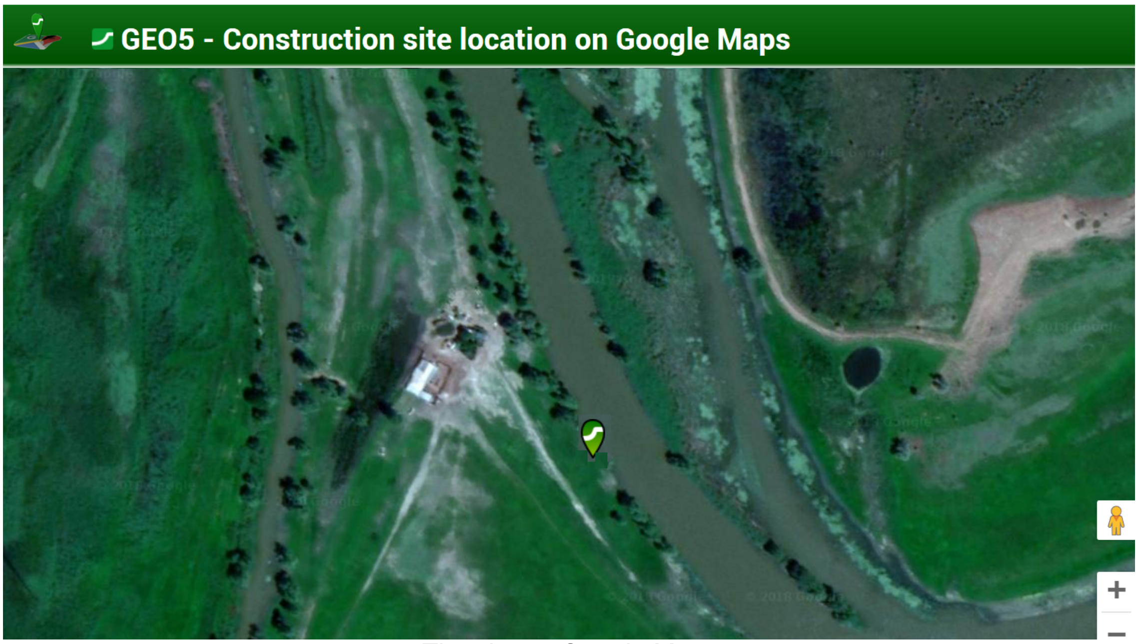
GEO5 programs allow showing the location of the construction on Google Maps. In most GEO5 programs, it is possible to define GPS coordinates in the frame "**Profile**".

In the Czech version of programs, local S-JTSK coordinates can be introduced, too.



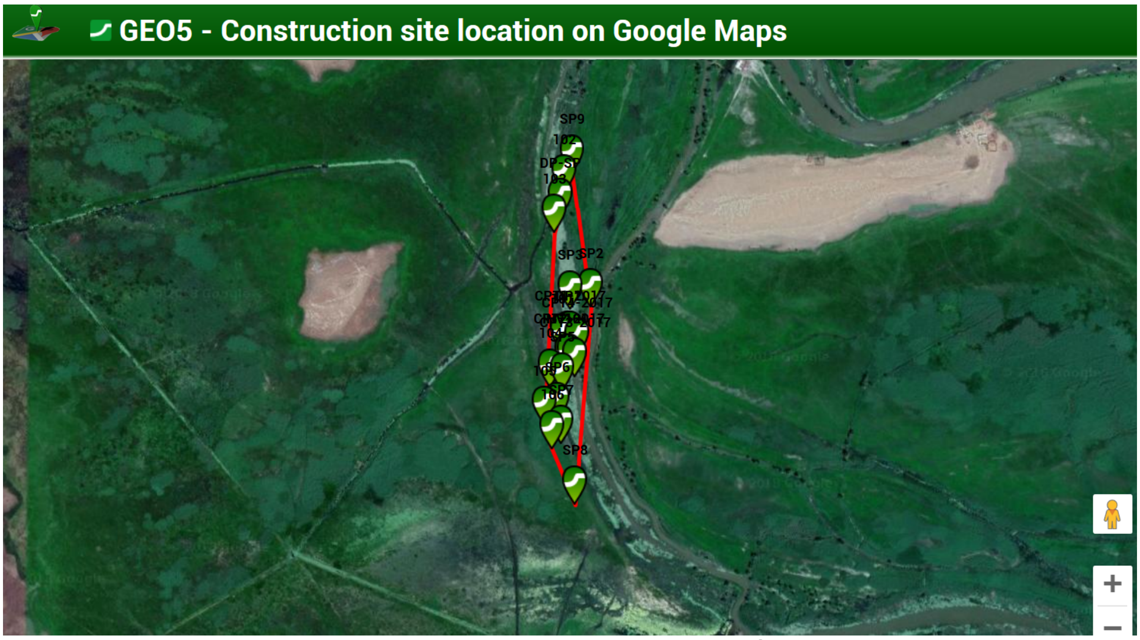
The input of GPS coordinates

By pressing the **"View on map"** button, you can show the location on the Google Maps.



The site on Google Maps

In the program "**Stratigraphy**" it is possible to show the shape of the construction site and input boreholes and field tests in one of **more than 6000 global coordinate systems** from the **MapTiler** database (https://epsg.io/). The coordinate system is selected in the "Settings" frame.



The construction site with provided tests on Google Maps

Standards and Analysis Methods

GEO5 allows the setting of standards and analysis methods centrally for all GEO5 programs.

In all GEO5 programs, these parameters can be specified in the "**Settings**" **frame**, which enables the user to:

- select analysis parameters in the Settings list
- store and manage settings in the Settings Administrator
- create and manage new user settings

The program allows performing the structure verification according to five methodologies:

- Verification according to factor of safety
- Verification according to limit states
- Verification according to EN 1997
- Verification according to LRFD
- Verification according to the Chinese standards

The programs allow us to define design situations (for different construction stages), which may differ by the partial factors.

Administrator

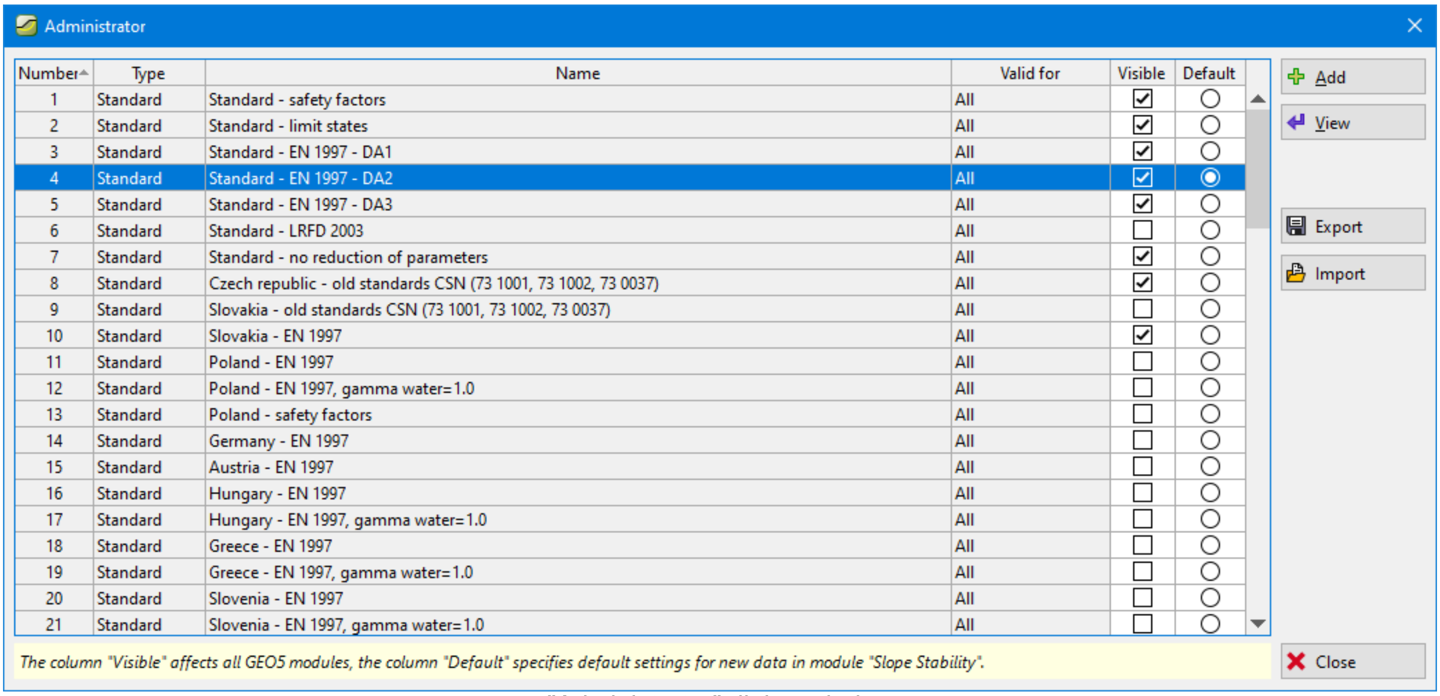
The Administrator is the main tool for managing individual "Settings" or "Template sets". In particular, it enables the user to:

- **determine the visibility of Settings** (**Template set**) in the "Settings list (List of tests templates)" (it is determined by checking the box in the "**Visible**" column)
- **specify the default Settings** (**Template set**) for new data files of the current program (the "**Default**" column)
- **view basic Settings** (**Template set**)which are currently selected - available by pressing the "**View**" button
- **add user Settings** (**Template set**) - pressing the "**Add**" button opens "**New settings**" dialog window with a copy of the currently selected **Settings** / **Template set**
- **edit input user Settings** (**Template set**) - by pressing the "**Edit**" button
- **delete user Settings** (**Template set**) - by pressing the "**Remove**" button

In addition, the **Settings Administrator** enables exporting and importing**Settings** (**Template set**) stored on the disk.

Visibility and default settings (template set) are switched by using the mouse or pressing the Space key (visibility) or

Shift+Space (default).



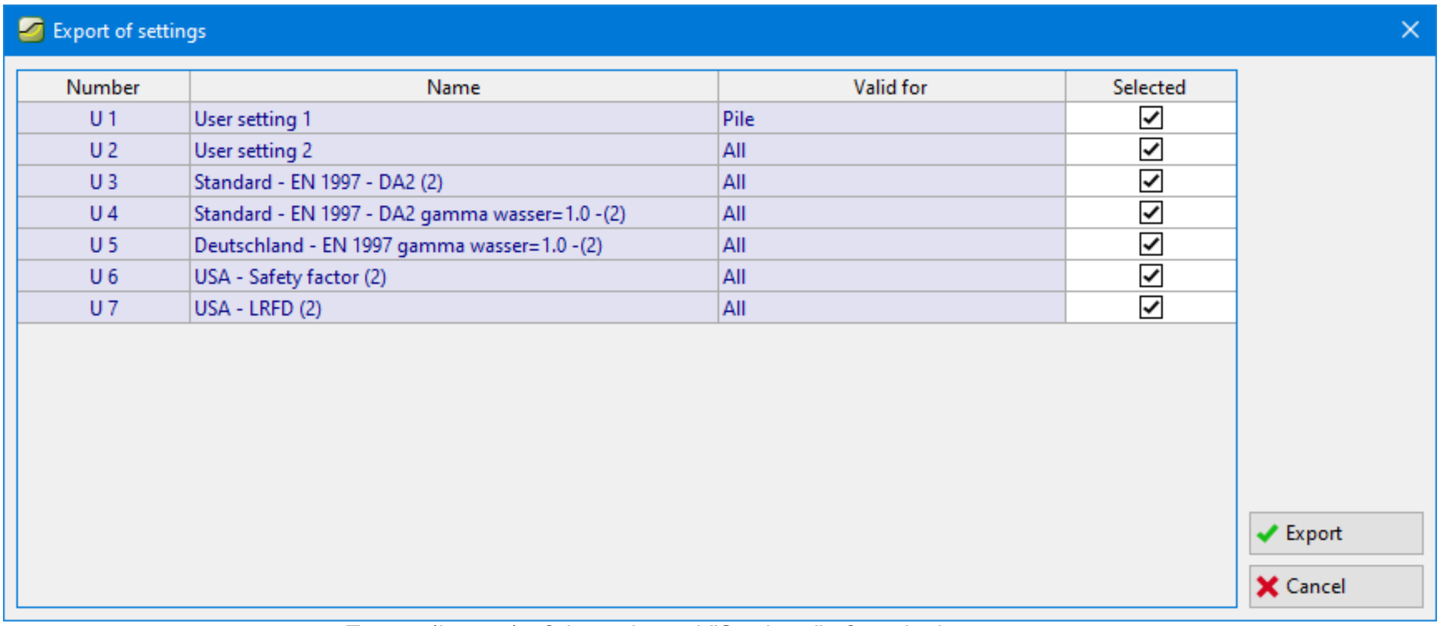
Number	Type	Name	Valid for	Visible	Default
1	Standard	Standard - safety factors	All	☑	○
2	Standard	Standard - limit states	All	☑	○
3	Standard	Standard - EN 1997 - DA1	All	☑	○
4	Standard	Standard - EN 1997 - DA2	All	☑	◉
5	Standard	Standard - EN 1997 - DA3	All	☑	○
6	Standard	Standard - LRFD 2003	All	☐	○
7	Standard	Standard - no reduction of parameters	All	☑	○
8	Standard	Czech republic - old standards CSN (73 1001, 73 1002, 73 0037)	All	☑	○
9	Standard	Slovakia - old standards CSN (73 1001, 73 1002, 73 0037)	All	☐	○
10	Standard	Slovakia - EN 1997	All	☑	○
11	Standard	Poland - EN 1997	All	☐	○
12	Standard	Poland - EN 1997, gamma water=1.0	All	☐	○
13	Standard	Poland - safety factors	All	☐	○
14	Standard	Germany - EN 1997	All	☐	○
15	Standard	Austria - EN 1997	All	☐	○
16	Standard	Hungary - EN 1997	All	☐	○
17	Standard	Hungary - EN 1997, gamma water=1.0	All	☐	○
18	Standard	Greece - EN 1997	All	☐	○
19	Standard	Greece - EN 1997, gamma water=1.0	All	☐	○
20	Standard	Slovenia - EN 1997	All	☐	○
21	Standard	Slovenia - EN 1997, gamma water=1.0	All	☐	○

"Administrator" dialog window

Import and Export of Settings

The selected user settings (template) in the "Administrator (Administrator of tests templates)" can be saved into a ("**Export**") file in formats ***.gxc** ("**Settings**") or ***.gxt** ("**Templates**") and subsequently loaded ("**Import**") on a different computer that has GEO5 programs installed. It enables sharing analysis settings (or templates) between several users, for example, in companies that share more licenses distributed among several subdivisions.

These formats may be particularly useful when solving various problems with our hotline.



Number	Name	Valid for	Selected
U 1	User setting 1	Pile	☑
U 2	User setting 2	All	☑
U 3	Standard - EN 1997 - DA2 (2)	All	☑
U 4	Standard - EN 1997 - DA2 gamma wasser=1.0 -(2)	All	☑
U 5	Deutschland - EN 1997 gamma wasser=1.0 -(2)	All	☑
U 6	USA - Safety factor (2)	All	☑
U 7	USA - LRFD (2)	All	☑

Export (Import) of the selected "Settings" of analysis parameters

Settings List

The "**Settings list (List of template sets)**" dialog window allows choosing the current "Settings", which will then drive both the calculation and verification analysis of the given task or "Template set" for geological documentation.

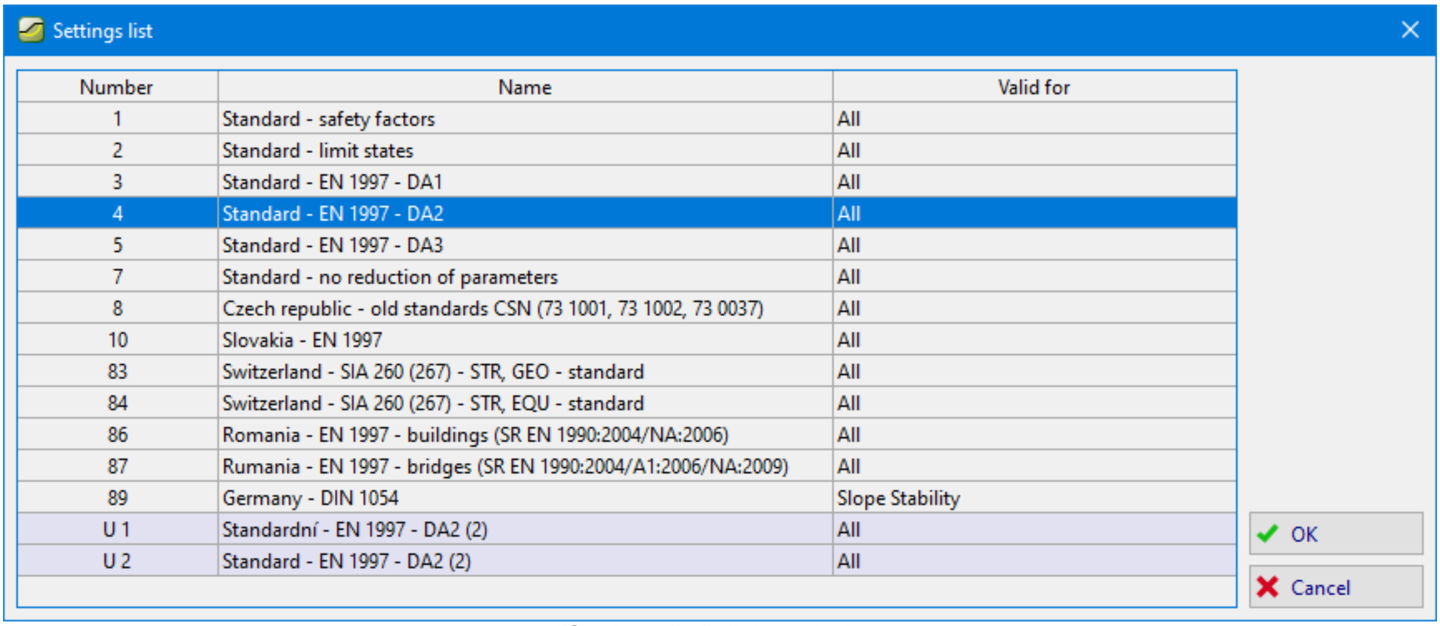
This list contains two types of settings (template sets):

- **basic**, which accompanies the software distribution and cannot be edited or deleted
- **user**, which is defined by the user

The list applies to all GEO5 programs, only some of the Settings can be restricted to a specific program.

For lucidity, only Settings (Template sets), which are checked in the "Settings Administrator" as visible, are displayed. When running the program the first time, the Settings (Template set) visibility is determined according to the country of destination. Subsequently, the program remembers the changes made by the user.

To work efficiently with the GEO5 programs, it is sufficient for most countries to create one or several specific "Settings" or "Template sets". Then, for the **solution of individual tasks,** the user just selects the **particular Setting** or **Template set**. The analysis methods, values of coefficients, and the verification methodology than do not need to be specified, which results in a lucid and simpler work with the given program.



Number	Name	Valid for
1	Standard - safety factors	All
2	Standard - limit states	All
3	Standard - EN 1997 - DA1	All
4	Standard - EN 1997 - DA2	All
5	Standard - EN 1997 - DA3	All
7	Standard - no reduction of parameters	All
8	Czech republic - old standards CSN (73 1001, 73 1002, 73 0037)	All
10	Slovakia - EN 1997	All
83	Switzerland - SIA 260 (267) - STR, GEO - standard	All
84	Switzerland - SIA 260 (267) - STR, EQU - standard	All
86	Romania - EN 1997 - buildings (SR EN 1990:2004/NA:2006)	All
87	Rumania - EN 1997 - bridges (SR EN 1990:2004/A1:2006/NA:2009)	All
89	Germany - DIN 1054	Slope Stability
U 1	Standardní - EN 1997 - DA2 (2)	All
U 2	Standard - EN 1997 - DA2 (2)	All

"Settings list" dialog window

Analysis Settings

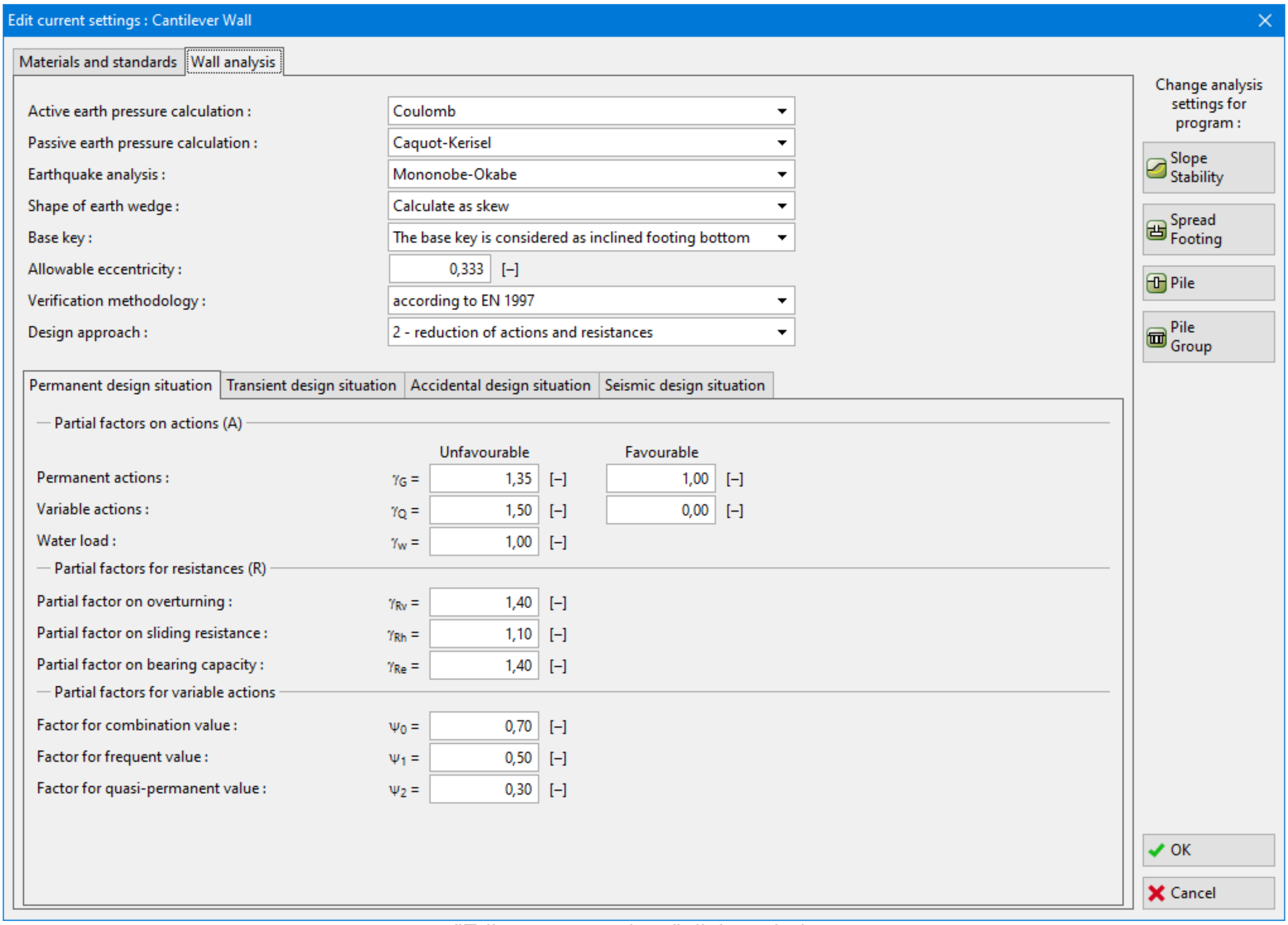
An analysis setting is a set of data, which is key for performing various calculations in the program. These include, in particular:

- methods and theories of the analyses
- verification methodology; the way of proving the safety of the structure (factor of safety, limit states, EN 1997, LRFD, Chinese standards)
- actual values of reduction coefficients and factors of safety for individual design situations

An analysis setting is typically the same for a large number of tasks - because of this, the program enables creating a "Settings list". individual Settings can be edited, exported and imported in the "Settings Administrator".

A setting can be **valid** for **all GEO5 programs** or for **one selected program** only.

- Materials and Standards
- Wall Analysis
- Pressure Analysis
- Stability Analysis
- Settlement
- Spread Footing
- Anchors
- Pile
- Pile CPT
- Micropiles
- Pile Group

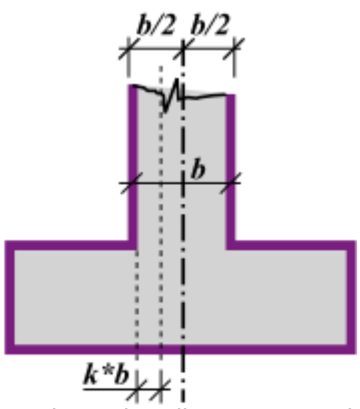


"Edit current settings" dialog window

Materials and Standards

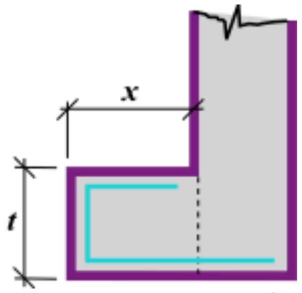
This tab allows inputing materials and analysis standards for:

- Concrete structures
- Concrete structures of bridge abutments
- EC 2 coefficients are specified for the analysis of concrete structures according to EN 1992-1-1. Both default and user-defined values can be adopted.
- **Parameter of cross-section determination** - this coefficient determines the location of the critical section for the calculation of bending moment within the wall foundation. The default value for concrete and steel-reinforced concrete columns is $\boldsymbol{k} = 0$, i.e. the critical section is found at an exposed column face. For masonry structures, it is recommended to choose the value of this coefficient equal to 0,25.



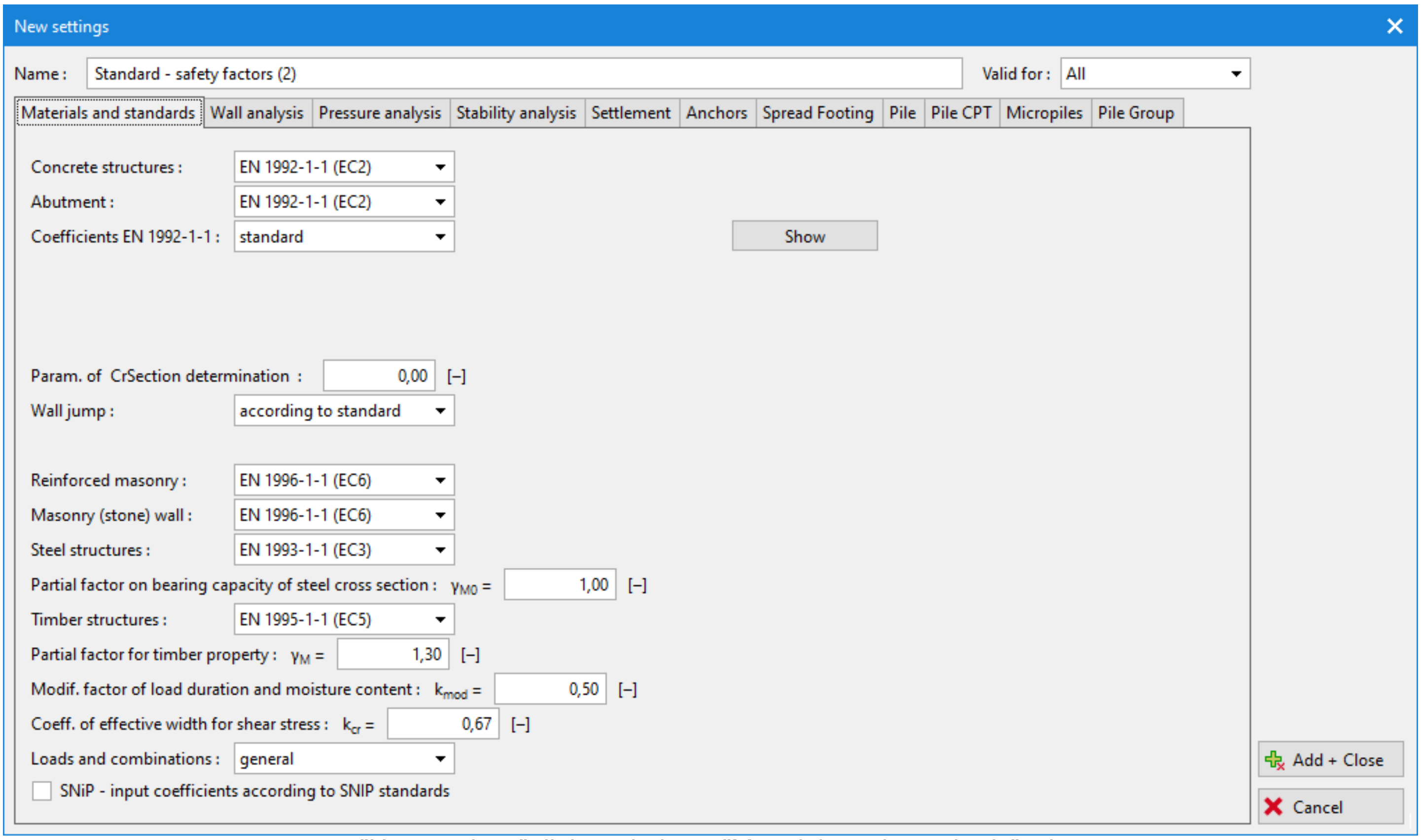
Critical section for the maximum bending moment within the wall foundation

- **Parameter of wall jump** k - this parameter determines whether the load-bearing reinforcement of the wall front jump is verified or not. For jump length $x \leq k*t$ the reinforcement verification is not performed. When choosing the "**according to standard**" option the short wall jump parameter is automatically back-calculated according to the selected standard for dimensioning of RC structures. When choosing the "**input parameter**" option the value of the parameter k is input by the user. The parameter of a wall short jump is considered for the analysis of walls and for the analysis of spread footings.



Scheme for determining the parameter of a short jump of wall

- Masonry structures (defined in the "**Masonry Wall**" program only)
- Steel structures (defined in "**Slab**", "**Sheeting Check**" and "**Anti-Slide Pile**" programs)
- Loads and combinations (defined in "**Slab**" and "**Beam**" programs only)
- SNiP - enables inputting design coefficients in the sense of Russian standards SNiP.



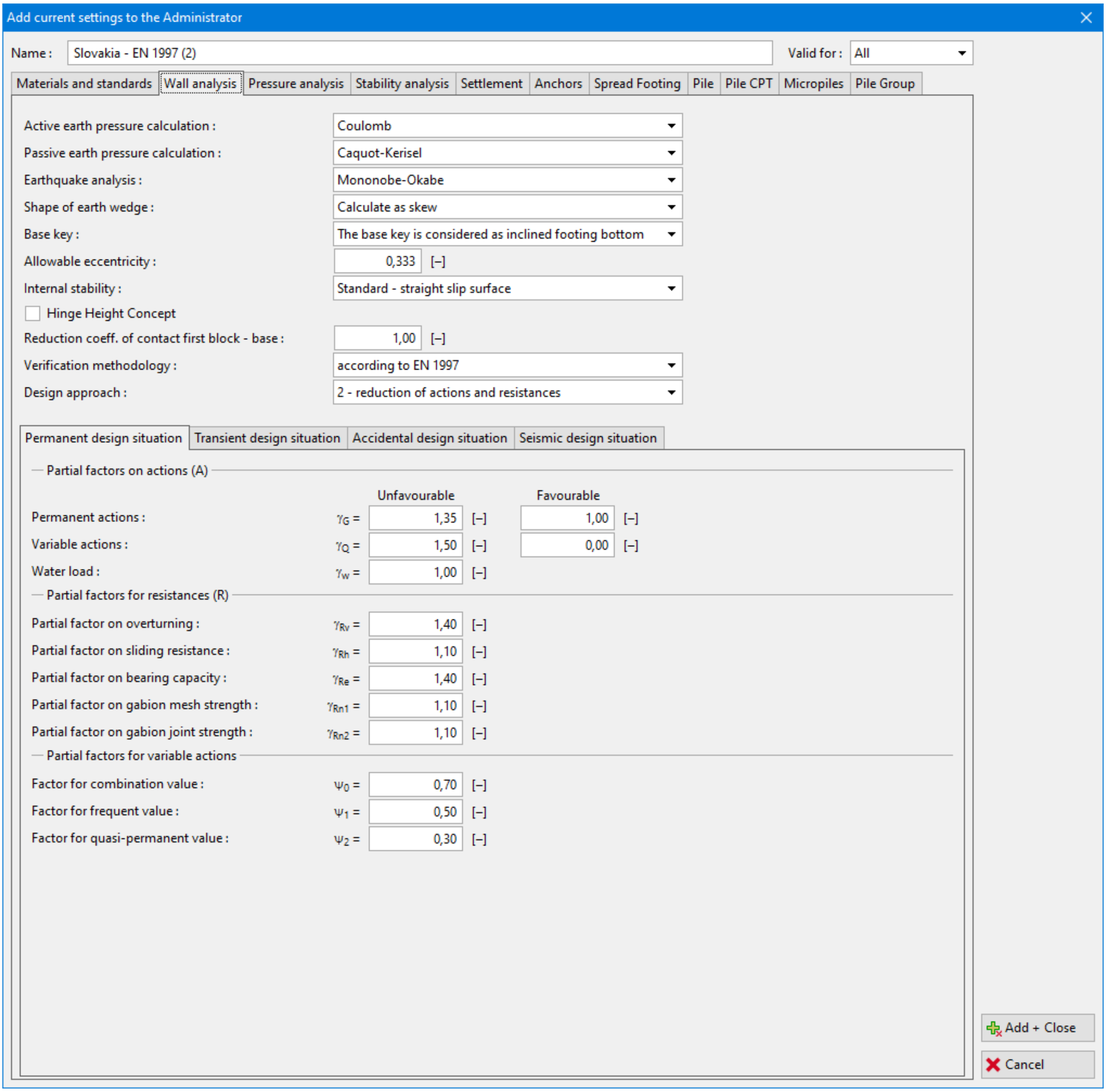
"New settings" dialog window - "Materials and standards" tab

Wall Analysis

This tab allows inputing parameters for the wall analysis:

- Active earth pressure calculation (Caquot, Coulomb (CSN 730037), Müller-Breslau (DIN 4085), Mazindrani (Rankine), Absi, SP 22.13330.2016).
- Passive earth pressure calculation (Caquot-Kerisel (CSN 730037), Coulomb, Müller-Breslau, Sokolovski (DIN 4085), Mazindrani (Rankine), Absi, SP 22.13330.2016).
- Earthquake analysis (Mononobe-Okabe, Arango, JTJ 004-89, JTS 146-2012, SL 203-97).
- Shape of earth wedge (Calculate as skew, Consider always vertical).
- Base key (The base key is considered as front face resistance; the base key is considered as inclined footing bottom).

- **Allowable eccentricity** - for assessment of contact stress at footing bottom is the value assumed of maximum allowable eccentricity in the range of 0.1 to 0.4.
- Internal stability - this way of calculation is adopted in "**MSE Wall**". The slip surface has a different shape (straight, broken) according to the selected standard of calculation.
- Hinge Height Concept - represents the way of analysis of precast walls according to AASHTO, in which the favorably acting gravity force of a part of the structure is reduced. It is used only in the "**Redi Rock Wall**" program.
- In the case of verification methodology according to the limit states and factor of safety, it is possible to reduce the parameters of the foundation - soil interface. The reduction coefficient of structure - soil interface μ represents the amount of wall resistance against slipping resp. against translation when in contact with the soil.
- When running the verification analysis according to the theory of limit states, the program enables the reduction of the tangent of the angle of internal friction φ employing the coefficient $\gamma_{m\varphi}$.
- Verification methodology (factor of safety, limit states, analysis according to EN 1997, analysis according to LRFD, analysis according to the Chinese standards).
- Design situations are specified for all verification methodologies.

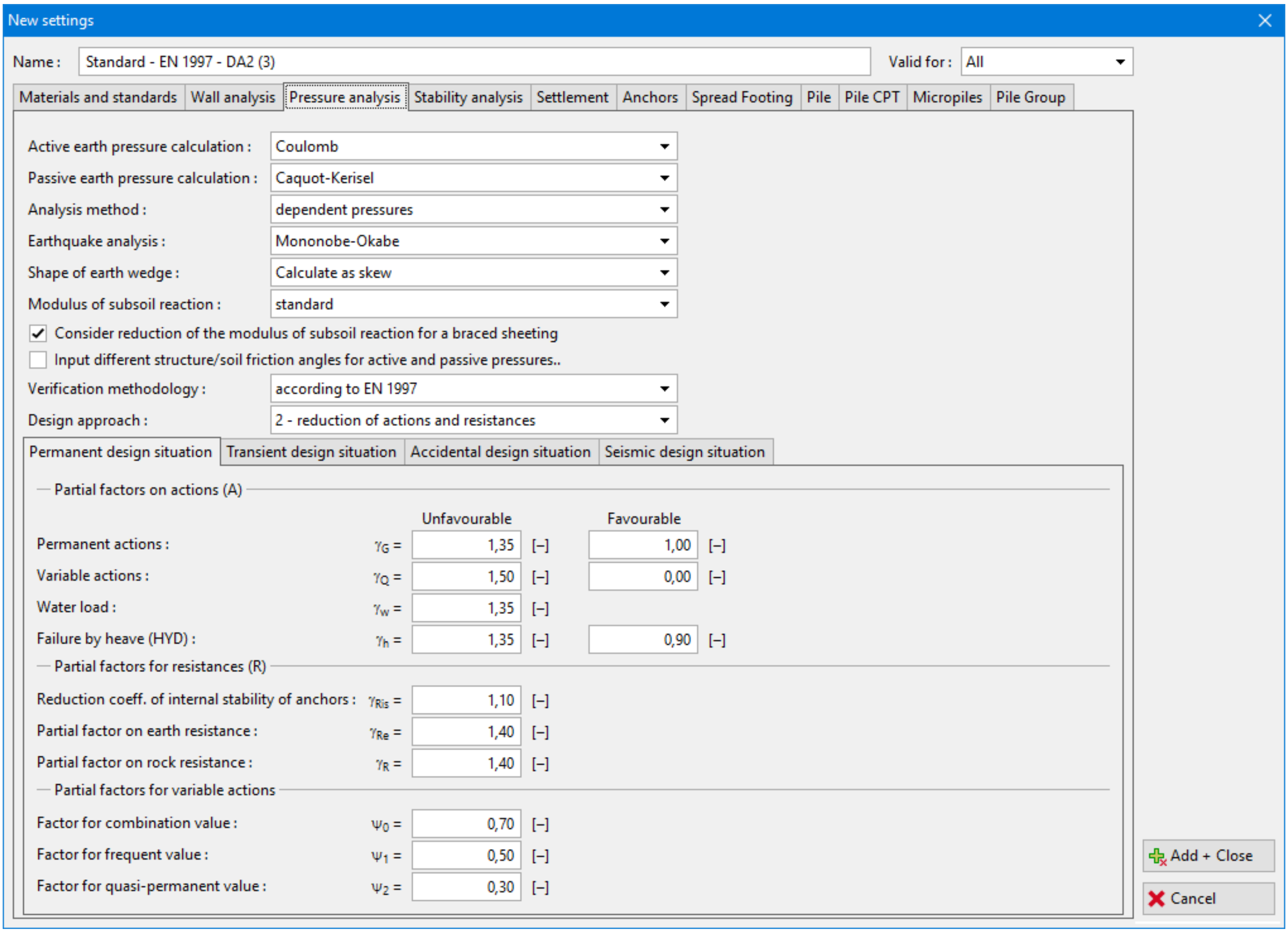


"New settings" dialog window - "Wall analysis" tab

Pressure Analysis

This tab allows inputing parameters for the analysis of excavations and earth pressures:

- Active earth pressure calculation (Caquot, Coulomb (CSN 730037), Müller-Breslau (DIN 4085), Mazindrani (Rankine), Absi, SP 22.13330.2016).
- Passive earth pressure calculation (Caquot-Kerisel (CSN 730037), Coulomb, Müller-Breslau, Sokolovski (DIN 4085), Mazindrani (Rankine), Absi, SP 22.13330.2016).
- Method of calculation (dependent pressures, JGJ 120-2012).
- Earthquake analysis (Mononobe-Okabe, Arango, JTJ 004-89, JTS 146-2012, SL 203-97).
- Modulus of subsoil reaction (standard, input, pressuremeter PMT, dilatometric DMT, CUR 166, Chinese standards). Standard settings contain recommended international methods for calculation of modulus of subsoil reaction (for **Sheeting Check** program in the frame "ModulusK_h") - other methods described herein are used only in specific countries.
- Shape of earth wedge (Calculate as skew, Consider always vertical).
- Consider reduction of the modulus of subsoil reaction for a braced sheeting - this option is used only in "**Sheeting Check**" and "**Anti-Slide Pile**" programs, when the program, during the analysis, reduces the values of the modulus of subsoil reaction automatically.
- Verification methodology (factor of safety, limit states, analysis according to EN 1997, analysis according to LRFD, analysis according to the Chinese standards).
- Design situations are specified for all verification methodologies.



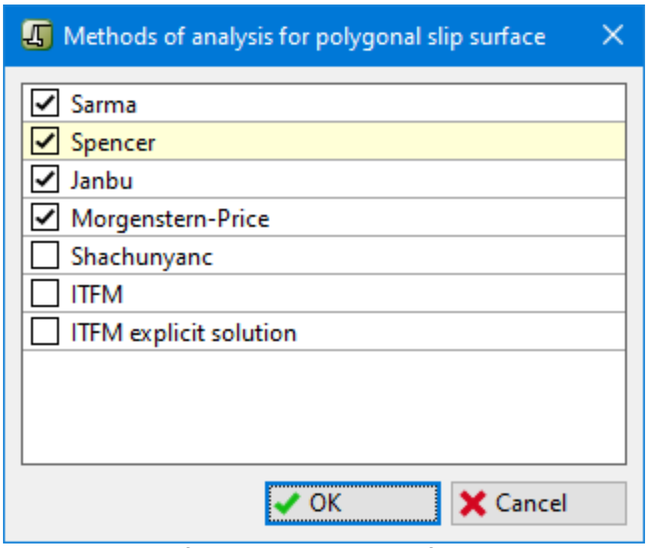
"New settings" dialog window - "Earth pressures" tab

Stability Analysis

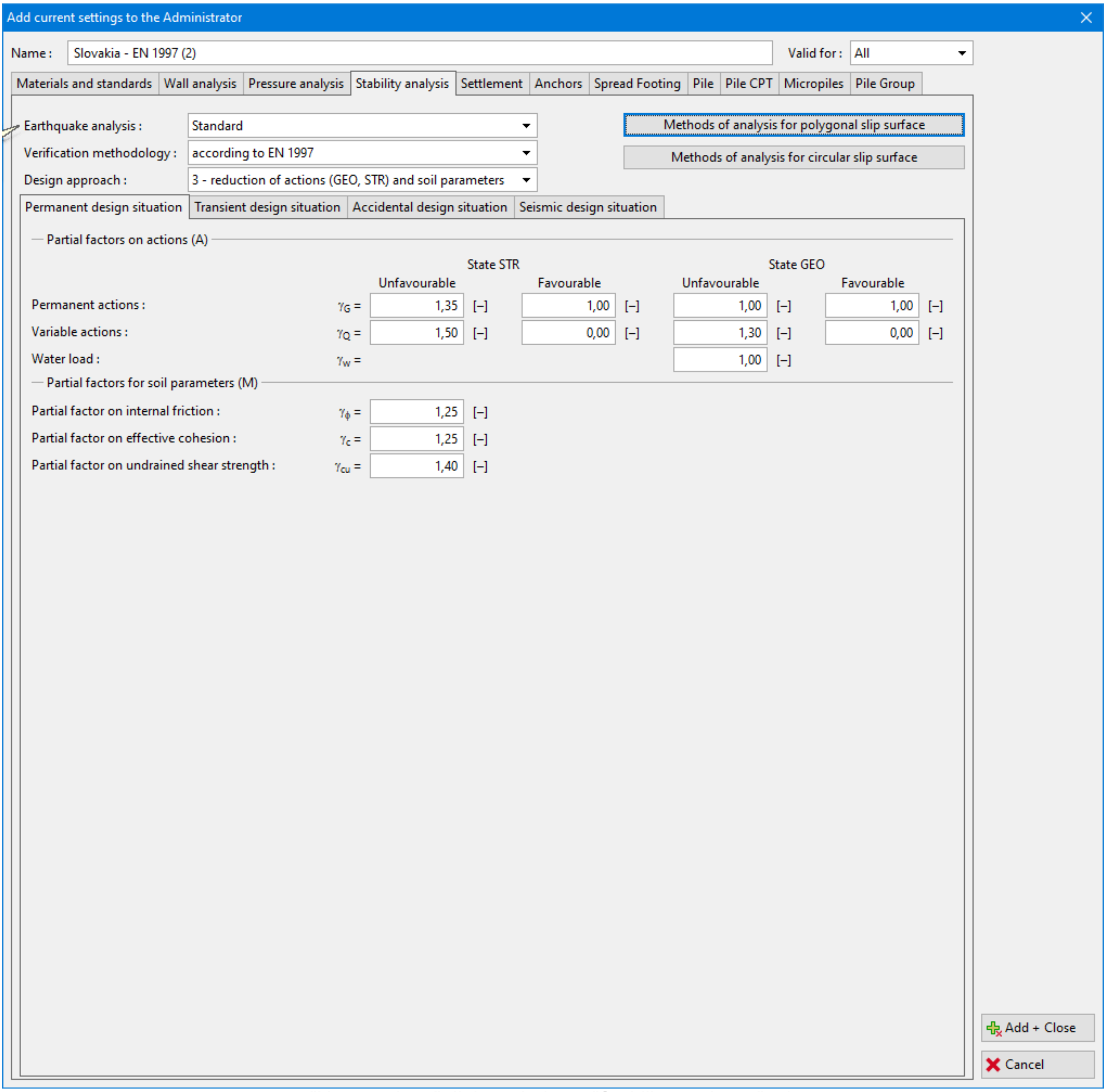
The tab allows inputing parameters for stability analysis:

- Earthquake analysis (Standard, GB 50111-2006, NB 35047-2015, GB 50330-2013, JTG B02-2013)
- Verification methodology (factor of safety, limit states, analysis according to EN 1997, analysis according to LRFD, analysis according to Chinese standards, analysis according to DIN 1054).
- Design situations are specified for all verification methodologies

- The "**Methods of analysis for polygonal slip surface**" and "**Methods of analysis for circular slip surface**" buttons open a dialog window that allows us to select the analysis method. The program allows us to calculate the results of all analysis methods for the selected slip surface (polygonal, circular). However, some of them are very exotic and known only in the countries of their origin. Thus the methods, the user is not interested in, can be turned off.



Dialog window - "Analysis methods for polygonal slip surface" - selecting the method of analysis



"New settings" dialog window - "Stability analyses" tab

Settlement

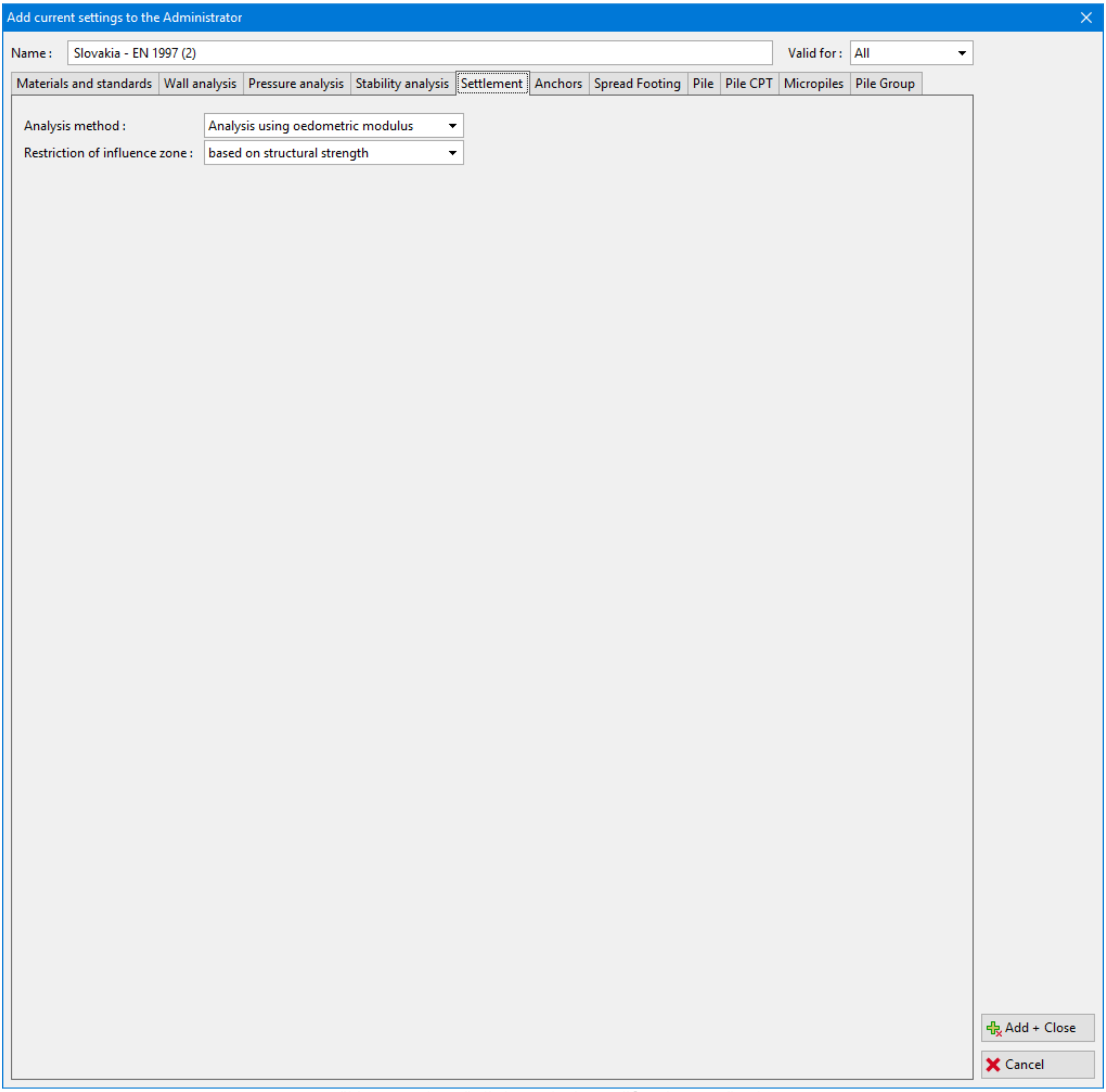
This tab allows inputing parameters for the settlement analysis:

Analysis methods:

- CSN 73 1001 (Analysis using oedometric modulus)
- Analysis using compression coefficient
- Analysis using compression index
- NEN (Buismann, Ladd)
- Soft soil model
- Janbu's theory
- Analysis using constrained modulus

Restriction of the influence zone:

- based on structural strength
- by a percentage of σ_{or} (The coefficient to bound the influence zone is input in [$\%$])



"New settings" dialog window - "Settlement" tab

Spread Footing

This tab allows inputing parameters for the analysis of bearing capacity of foundation:

Analysis for drained conditions:

- standard approach
- CSN 73 1001
- PN-81B-03020
- IS:6403-1981
- EC 7-1 (EN 1997-1:2003)
- NCMA
- GB 50007-2002
- SNiP 2.02.01-83
- DS/EN 1997-1 DK NA:2013
- Meyerhof
- Vesic

- DIN 4017
- CTE DB SE-C
- B1/VM4
- SP 22.13330-2016

Analysis for undrained conditions:

- standard approach
- CSN 73 1001
- IS:6403-1981
- EC 7-1 (EN 1997-1:2003)
- DS/EN 1997-1 DK NA:2013
- CTE DB SE-C
- B1/VM4

Analysis of spread footing on rock subsoil:

- standard approach
- CSN 73 1001
- EC 7-1 (EN 1997-1:2003)

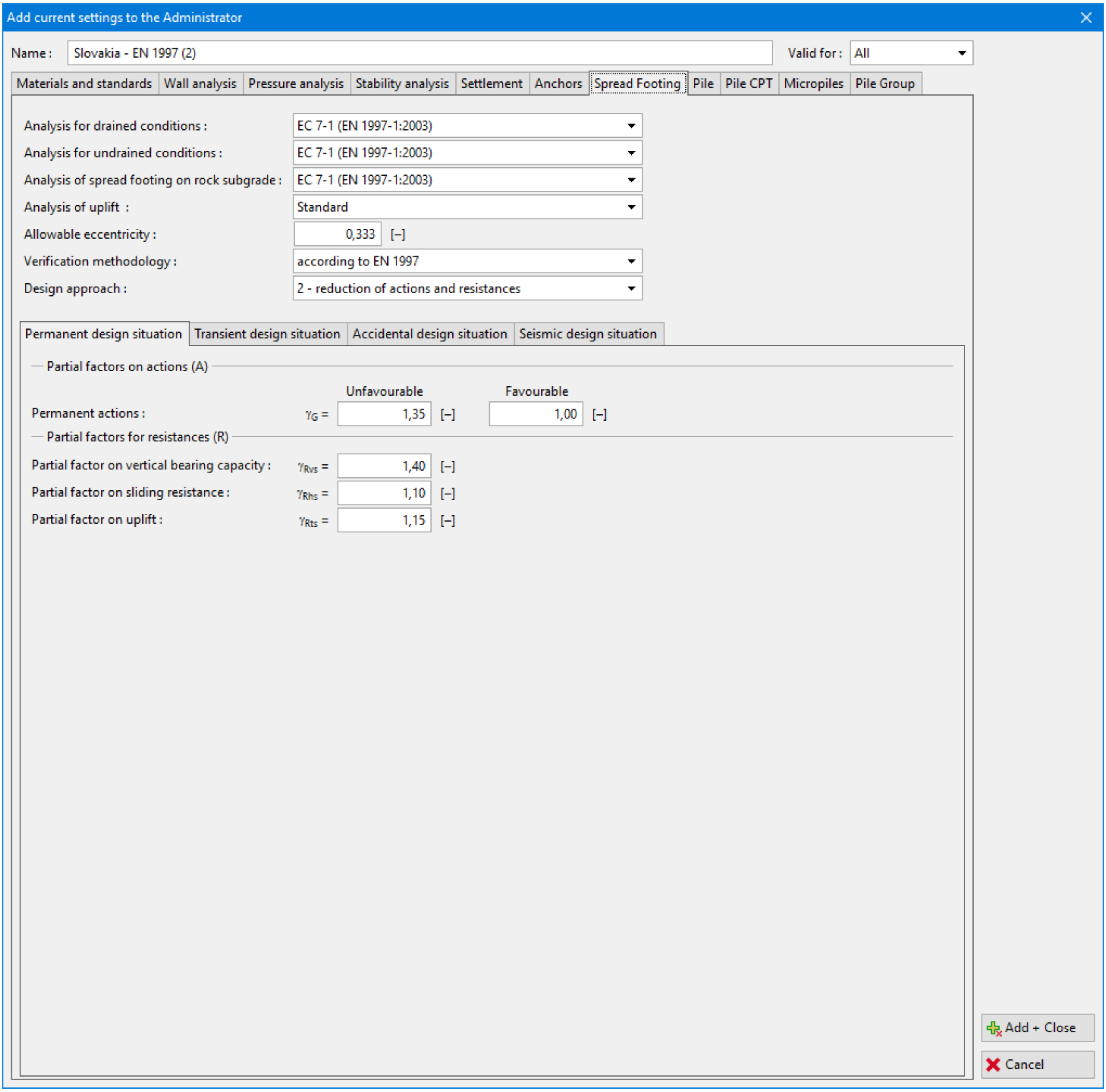
Analysis of uplift

- standard approach
- cone method
- DL/T 5219-2005
- EN 50341

Allowable eccentricity - The value of maximum allowable eccentricity in the range of 0.1 to 0.4 is input for verification of the eccentricity of foundation.

Verification methodology (factor of safety, limit states, analysis according to EN 1997, analysis according to LRFD, analysis according to Chinese standards).

Design situations are specified for all verification methodologies.



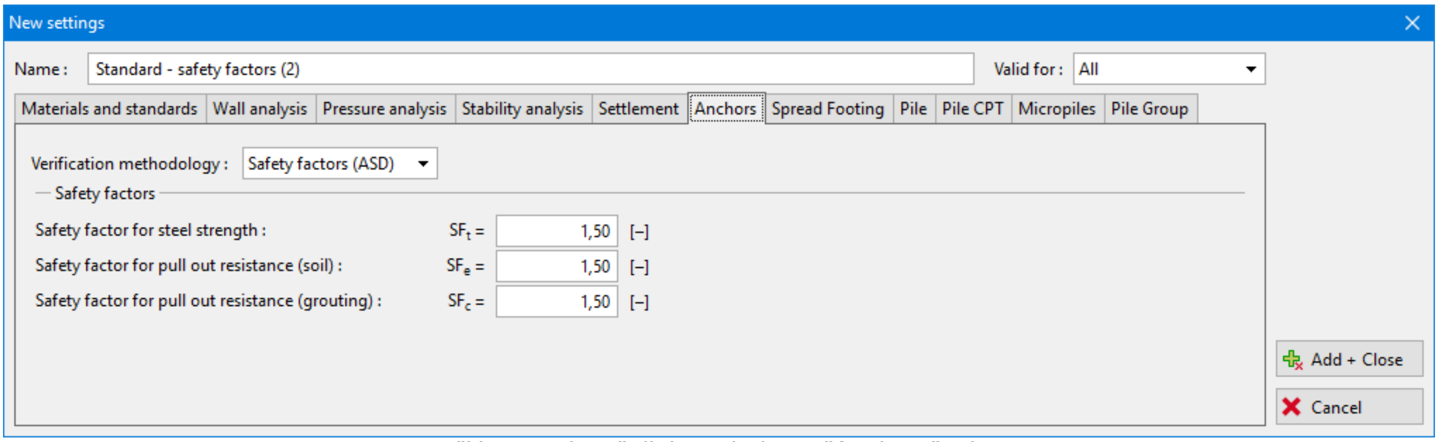
"New settings" dialog window - "Spread Footing" tab

Anchors

This tab allows inputing parameters for the analysis of **anchors bearing capacity**:

Verification methodology (factor of safety, limit states).

Input safety factors or coefficients of reduction reduce the computed bearing capacities of the anchor.



"New settings" dialog window - "Anchors" tab

Pile

This tab allows inputing parameters for the analysis of pile:

Analysis for drained conditions:

- CSN 73 1002
- Effective stress
- NAVFAC DM 7.2
- CTE-DB SE-C

Analysis for undrained conditions:

- Tomlinson
- NAVFAC DM 7.2
- CTE-DB SE-C

Load-settlement curve:

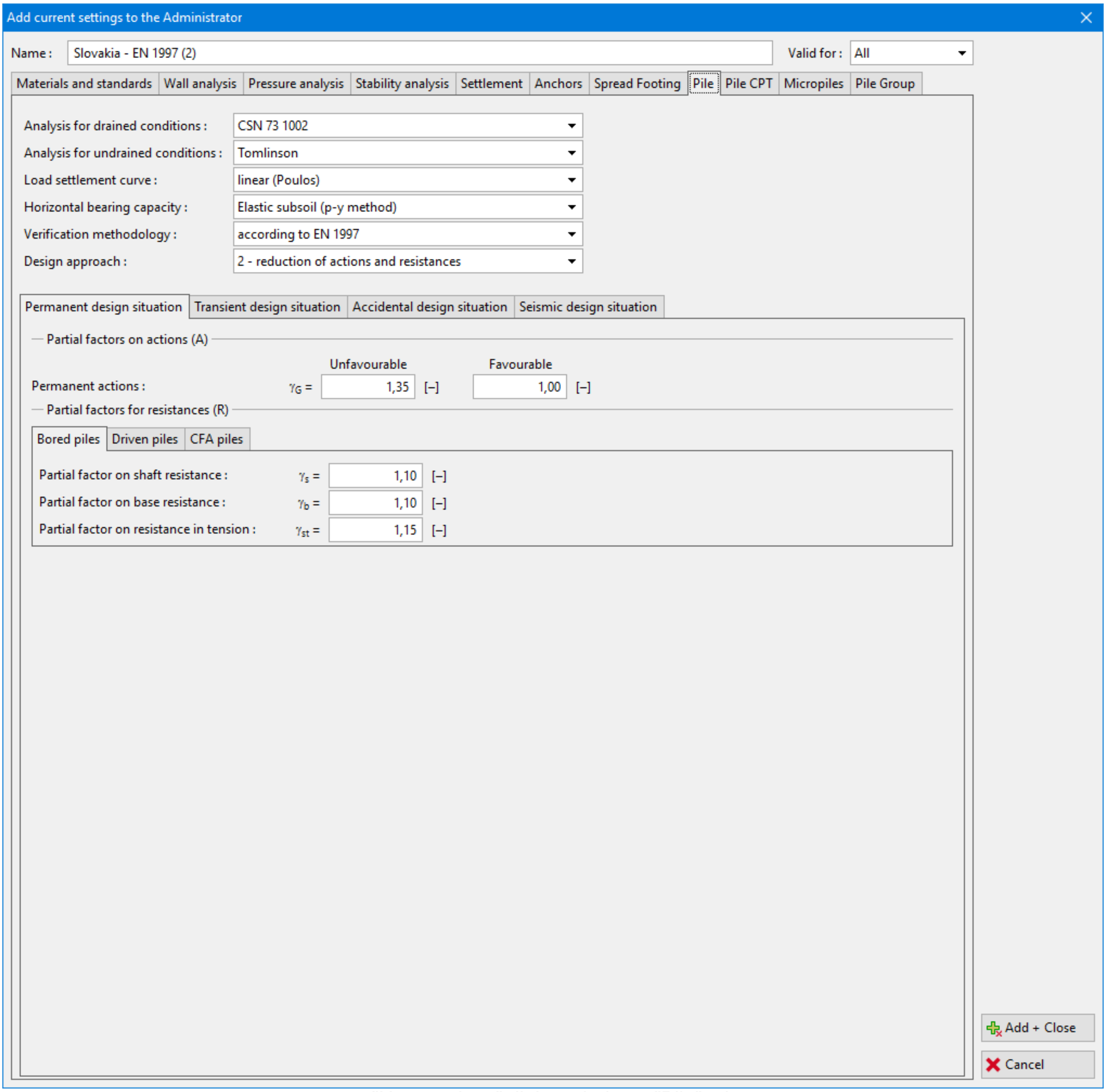
- nonlinear (Masopust)
- linear (Poulos)

Horizontal bearing capacity:

- Elastic subsoil (p-y method)
- Broms method

Verification methodology (factor of safety, limit states, analysis according to EN 1997, EA-Pfähle).

Design situations are specified for all verification methodologies.



"New settings" dialog window - "Pile" tab

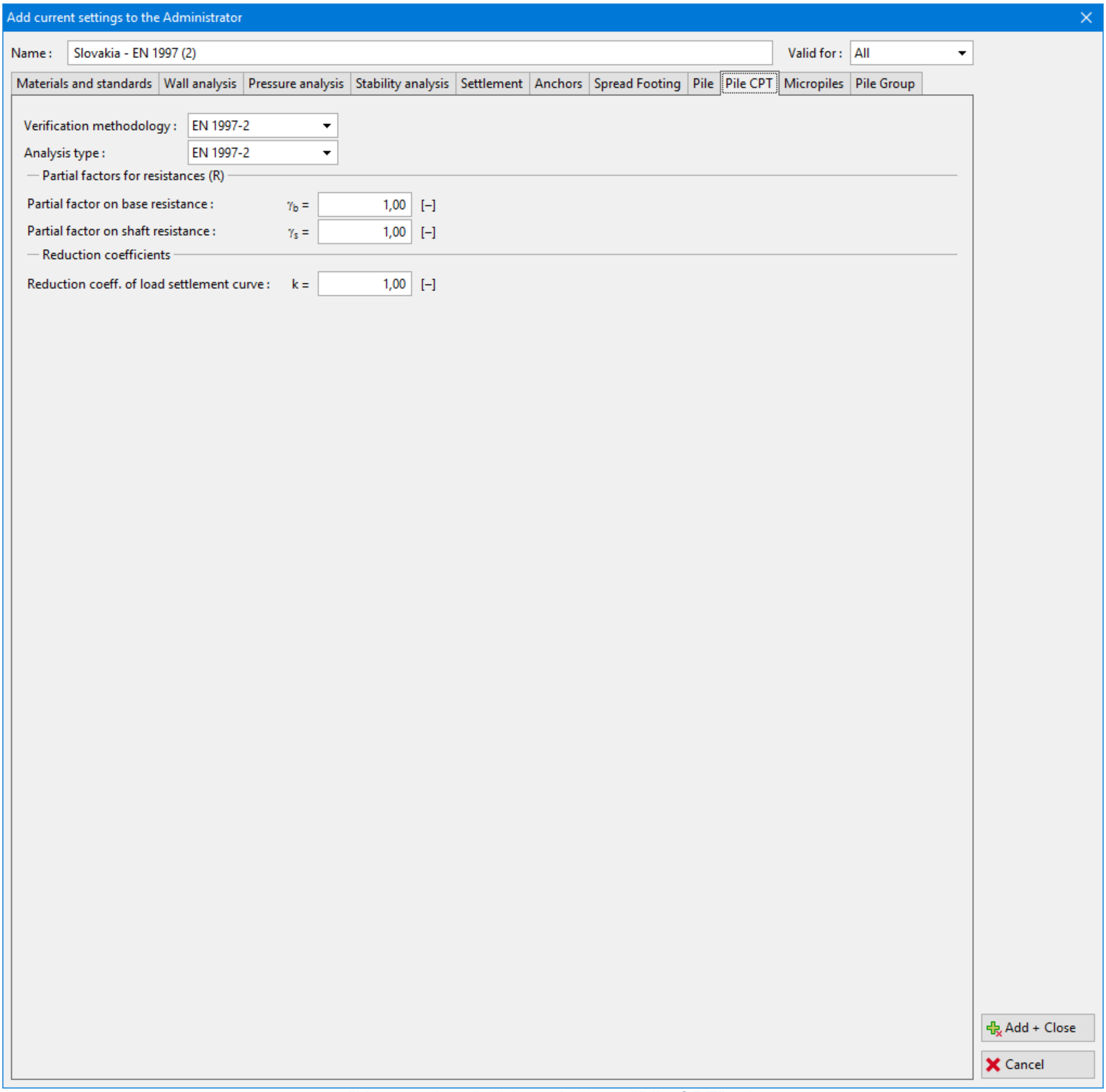
Pile CPT

This tab allows inputing parameters for the analysis of pile CPT:

Verification methodology (factor of safety, limit states, NEN 6743, EN 1997-2).

Analysis type:

- EN 1997-2
- NEN 6743
- LCPC (Bustamante)
- Schmertmann
- NBN EN1997-1 ANB



"New settings" dialog window - "Pile CPT" tab

Micropiles

This tab allows inputing parameters for the analysis of micropiles:

Calculation of stem bearing capacity:

- geometric method (Euler)
- Salas theory
- Véas-Souche theory

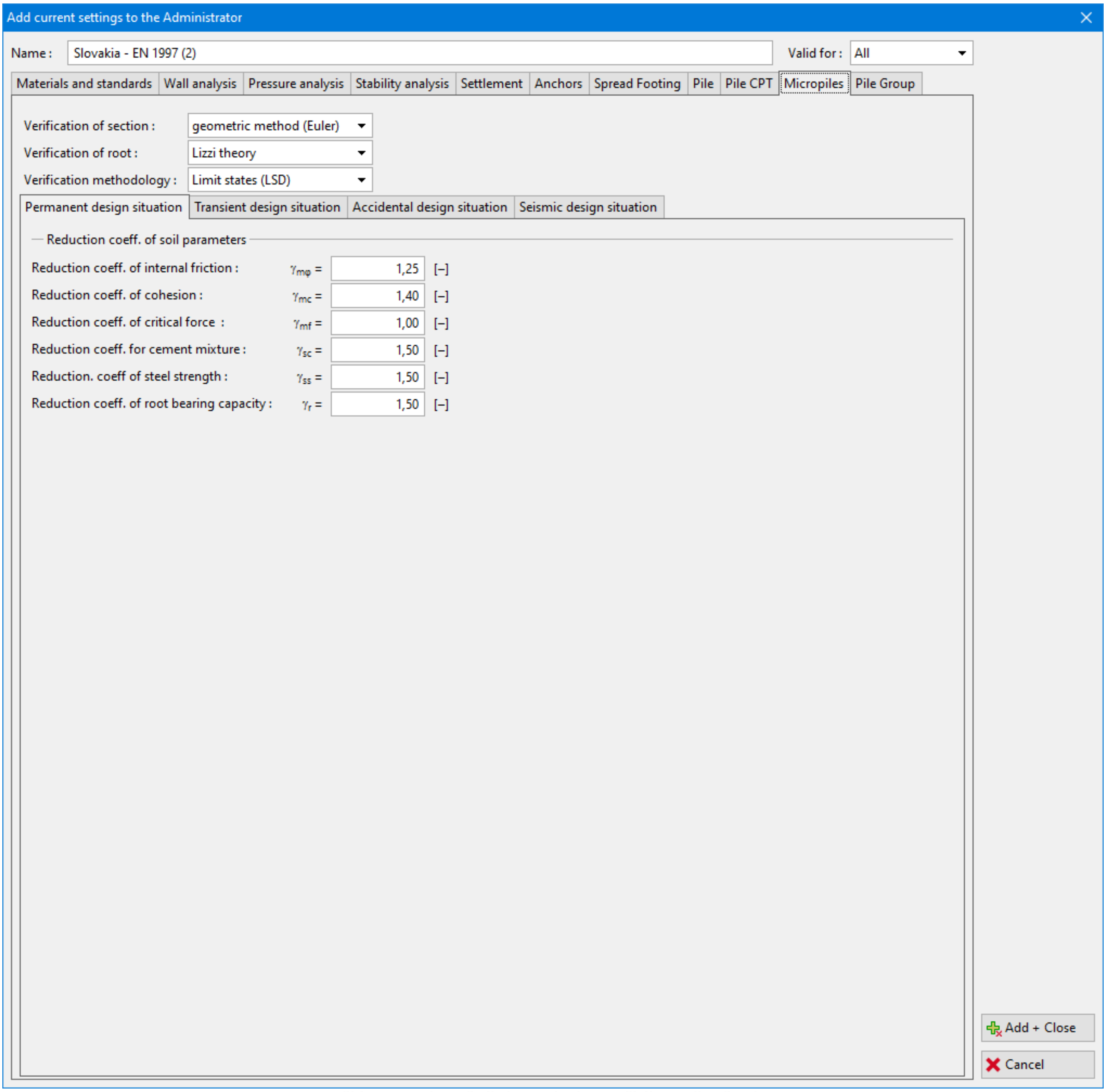
Calculation of root bearing capacity:

- Lizzi theory
- Littlejohn theory
- Bowles theory
- Zweck theory
- Véas theory
- root in the rock

- Bustamante (SPT, Pressuremeter PMT)

Verification methodology (factor of safety, limit states).

Design situations are specified for all verification methodologies.

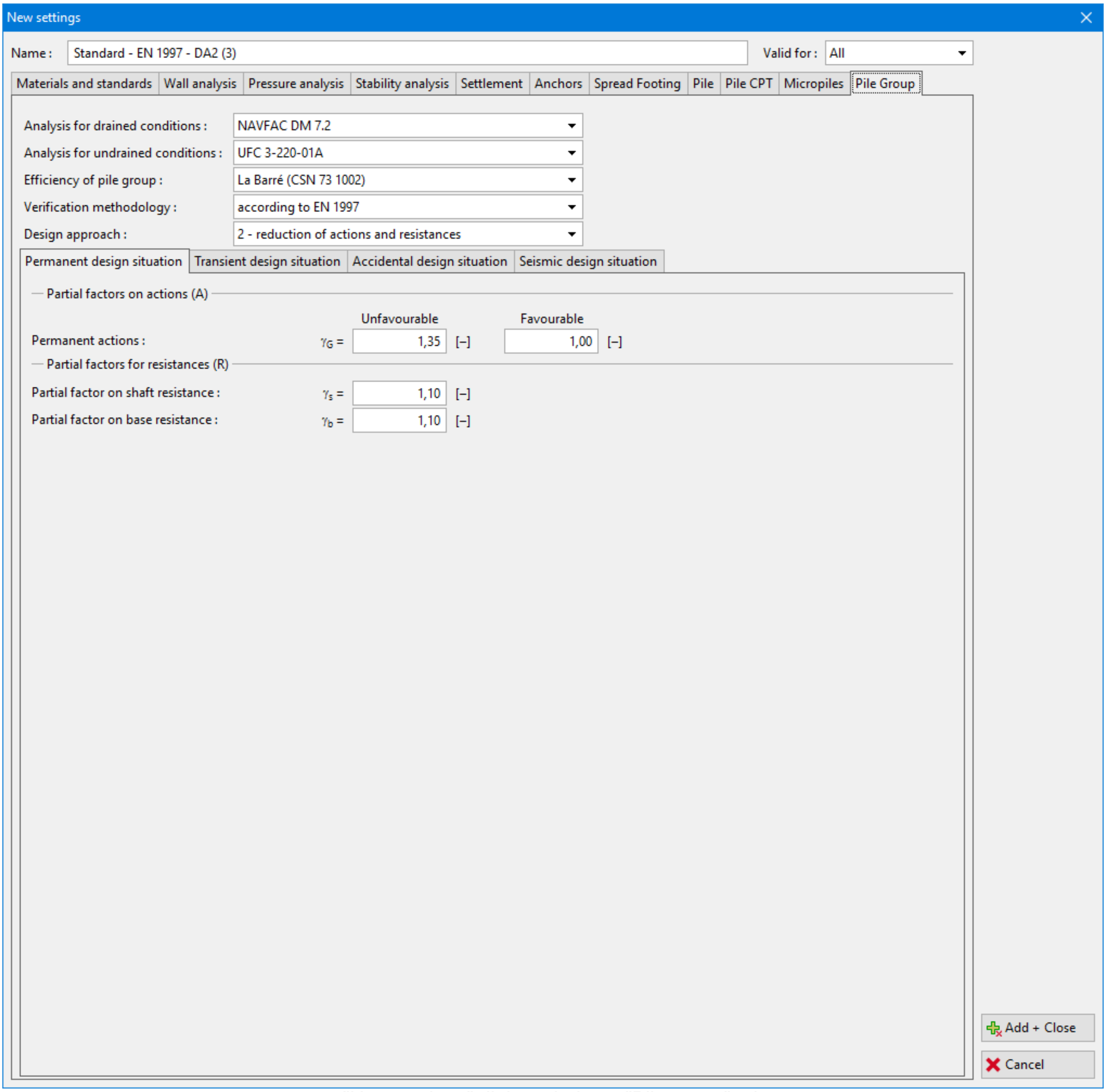


"New settings" dialog window - "Micropiles" tab

Pile Group

This tab allows inputting parameters for the analysis of pile groups:

- Analysis for drained conditions: CSN 73 1002, Effective stress, NAVFAC DM 7.2
- Analysis for undrained conditions: (UFC 3-220-01A, Masopust)
- Efficiency of a pile group: UFC 3-220-01A, La Barré (CSN 73 1002), Seiler-Keeney, input efficiency
- Verification methodology (factor of safety, limit states, analysis according to EN 1997).
- Design situations are specified for all verification methodologies.



"New settings" dialog window - "Pile group" tab

Adding New Settings

The program contains a relatively large number of **basic Settings** applicable to individual countries and theoretical approaches. Despite that, it is quite probable that most users will require to modify it and create their **user-defined Settings**.

A setting can be valid for all GEO5 programs or for the current program only (this can be specified in the right upper corner of the dialog window).

Pressing the "**Add to administrator**" button opens a dialog window, which displays the current setting of the program:

- if the "**Input for the current task**" is the current setting, the window is opened in the regime pertinent to the current program
- if the current setting is selected from the **Settings list**, a copy of this Setting with the same validity is opened

After editing and specifying the name of the **new Setting**, this Setting is saved into the "Settings Administrator" by pressing the "**Add**" button so it can be subsequently selected from the "Settings list".

It is reasonable to create user-defined Settings, for example:

1) based on countries and standards

- settings for Boguto
- settings for Borito
- settings for Borito, bridge structures

2) based on the investor

- settings for highways
- settings for railways
- settings for buildings

3) based on analysis methods

- analysis based on Mazindrani
- analysis based on Coulomb

4) individually

- my way
- Peter's way

The goal is to create a "Settings list" so that the user does not need to care for the way of inputting various types of analysis and coefficients. The created settings can be "Exported" and made available to other users or within the company. Providing such Settings have a broader validity, the FINE company will implement them into the pre-defined Settings available to all users of the GEO5 software.

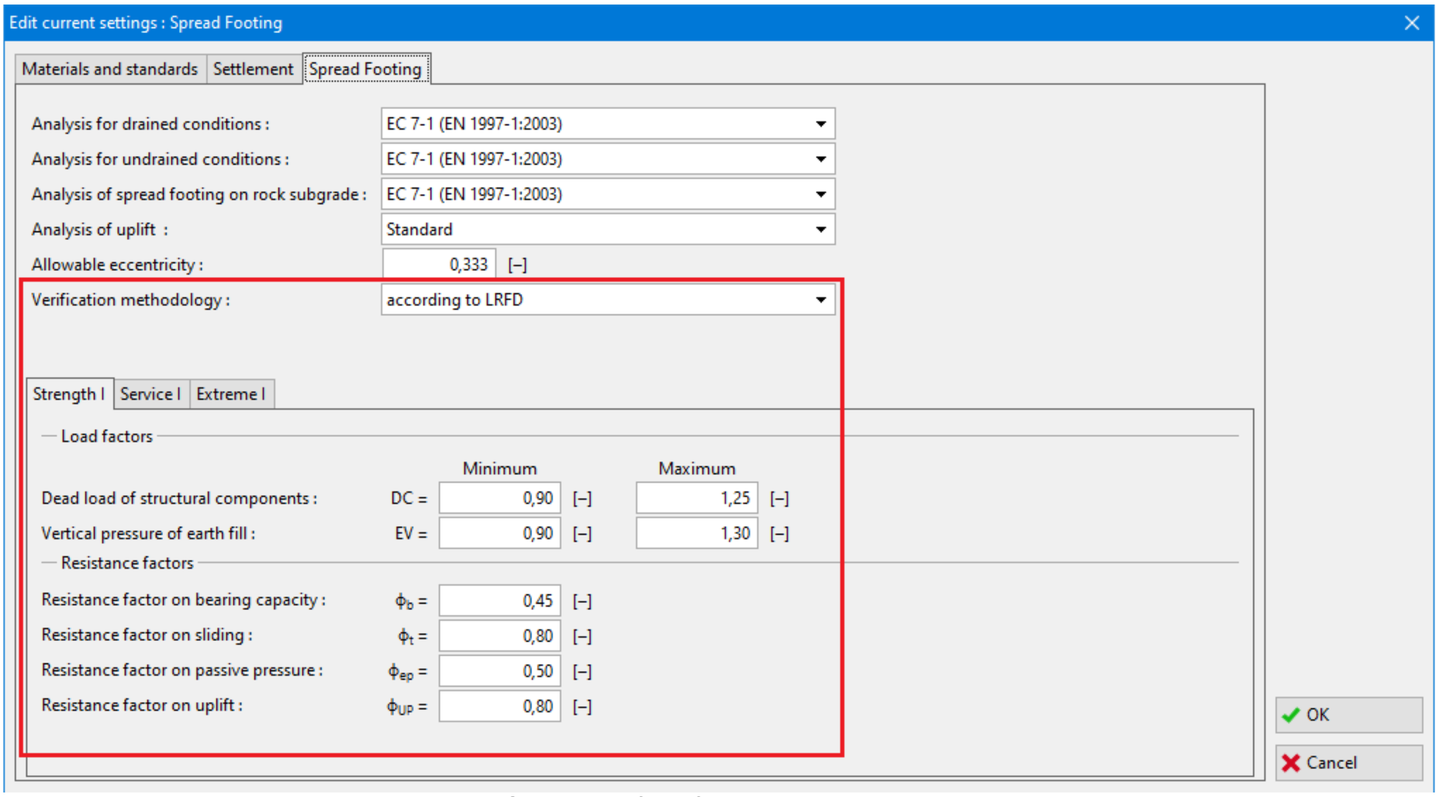
Verification Methodology

The program allows performing the structure verification according to these methodologies:

- Verification according to factor of safety
- Verification according to limit states
- Verification according to EN 1997
- Verification according to LRFD
- Verification according to the Chinese standards
- Verification according to BS8006 (the MSE Wall program)
- Verification according to DIN1054 (the Slope Stability program)
- Verification according to EA-Pfähle (the Pile program)

Specific calculations (e.g. pressure analysis, determination of bearing capacity of foundation soil) are the same for all verification methodologies - they differ only in the way of introducing the design coefficients, combinations, and the procedure for verifying the structure safety.

Verification methodology can be selected in the "**Settings**" dialog window.



Selection of verification methodology

Analysis According to the Safety Factor (ASD)

The verification methodology of structure safety based on the "**Safety factor**" is historically the oldest and most widely used approach. The principal advantage is its simplicity and lucidity.

In general, safety is proved using the safety factor:

$$FS = \frac{X_{pas}}{X_{act}} > FS_{req}$$

where: FS - Computed safety factor

X_{pas} - A variable resisting the failure (resisting force, strength, capacity)

X_{act} - A variable causing the failure (sliding force, stress)

FS_{req} - A Required factor of safety

When performing the analysis using the "**Safety factor**", neither the load nor the soil parameters are reduced by any of the design coefficients.

A more detailed description for individual programs and types of structures can be found in the following chapters (Walls and retaining structures, Slope Stability, Spread Footing, Pile, Rock Stability, Micropile, Pile CPT, Pile Group).

Analysis According to the Theory of Limit States (LSD)

The verification methodology based on the "**Limit states**" theory proves the safety by comparing a resisting variable (resisting force, strength, bearing capacity) and a variable causing failure (sliding force, stress).

$$X_{pas} > X_{act}$$

where: X_{pas} - A variable resisting the failure (resisting force, strength, capacity)

X_{act} - A variable causing the failure (sliding force, stress)

X_{act} is in general determined from the design parameters of soils and load:

- soil parameters are reduced by corresponding coefficients
- load (its action) is increased by corresponding coefficients

X_{pas} is determined based on the following assumptions:

- soil parameters are reduced by corresponding coefficients

- the calculated structure resistance is reduced by a corresponding coefficient

The verification based on "**Limit states**" is a more modern approach than the "Safety factor". However, it is less lucid.

Modern standards used for verification of structure safety (EN 1997, LRFD) are based on the concept of limit states. In addition, they introduce various values for the coefficients of partial factors for favorably and unfavorably acting loads.

A more detailed description for individual programs and types of structures can be found in the following chapters (Walls and retaining structures, Slope Stability, Spread Footing, Pile, Rock Stability, Micropile, Pile CPT, Pile Group).

Verification According to EN 1997

Designing a structure according to EN 1997-1 essentially follows the theory of limit states.

Partial factors, adjusting the characteristic values of load, material, and resistance, are introduced into the analysis depending on the selected "Design approach".

Partial factors are identical for all analyses in a given program. However, a "Design situation" can be selected for individual stages.

The programs can be grouped into several categories based on the selected approach:

- Analysis of walls, supporting structures (Walls, Abutment, Nailed Slope)
- Analysis of sheeting structures (Sheeting Design, Sheeting Check, Earth Pressures)
- Foundation analysis (Spread Footing, Pile)
- Slope stability analysis

GEO5 programs support following **National Annexes**:

Finland, France, Poland, Germany, Slovakia, Austria, Singapore, Denmark, Belgium, the Netherland, the United Kingdom, Greece, Hungary, Bulgaria, Slovenia, Italy, Portugal, Norway, Romania.

Desired National Annex can be selected in "Settings List".

Partial Factors

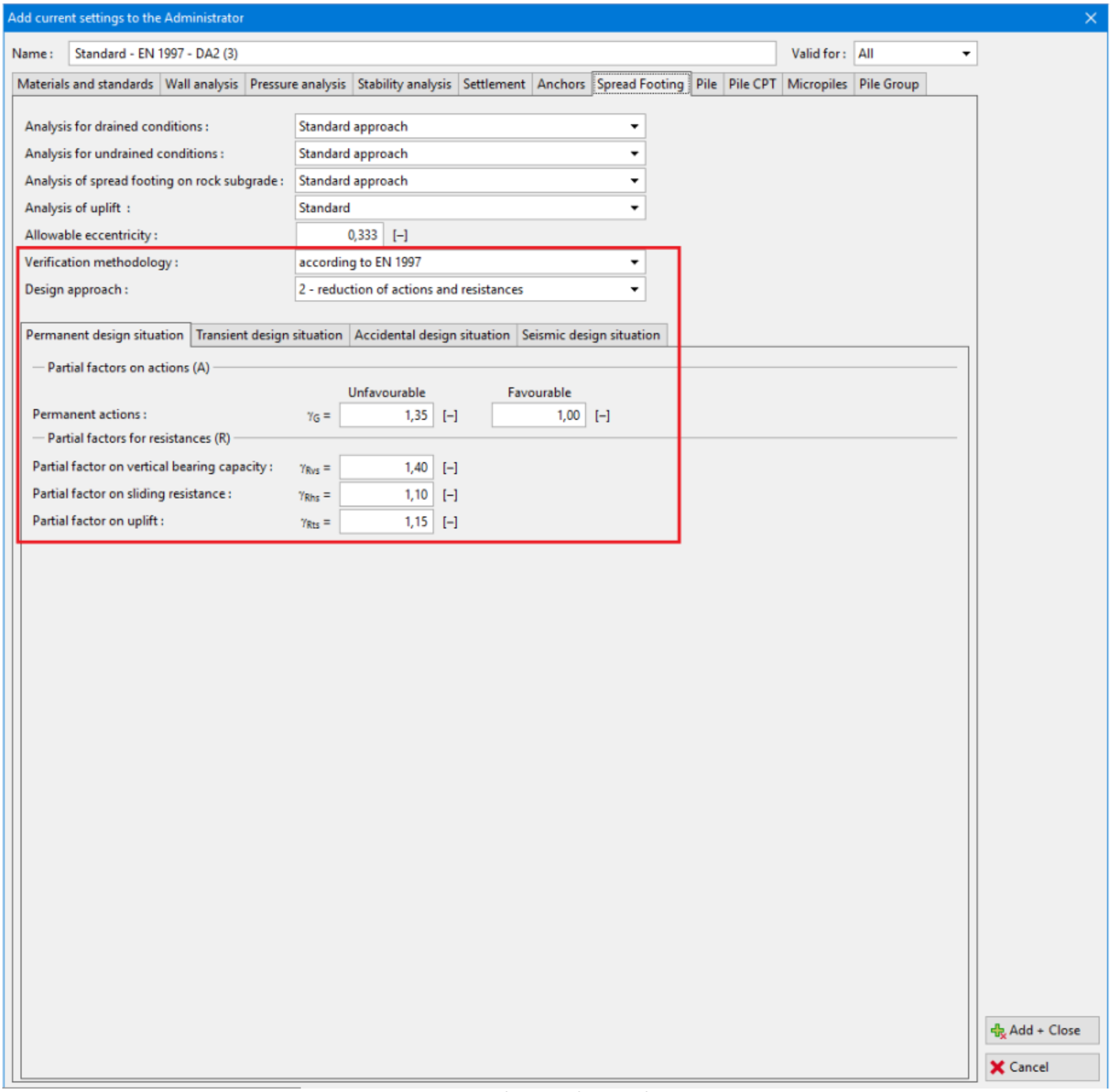
The "**Settings**" dialog window allows inputing the partial factors for the analysis based on EN 1997.

The "**Design approach**" combo list allows us to select one of the three "Design approaches". Depending on the selected design approach, the dialog window displays the **partial factors on actions**, **material or resistance** and coefficients of combination for variable load actions.

The section for inputting partial factors on actions also enables to input partial factors reducing the action of water.

The "Settings administrator" and the "Settings list" contain a large number of pre-defined settings for individual **countries EU - settings EN 1997** according to selected national annexes (NA). In most countries, only one Design approach is then specified depending on NAD and used program (a type of geotechnical task) - a larger number of pre-defined settings are available only for some countries.

The program enables to input each set of parameters four times - for individual design situations. The program then adopts the coefficients based on the design situation set in the "**Stage settings**" frame.



"New settings" dialog window - input of partial factors for the analysis based on EN 1997

Design Approaches

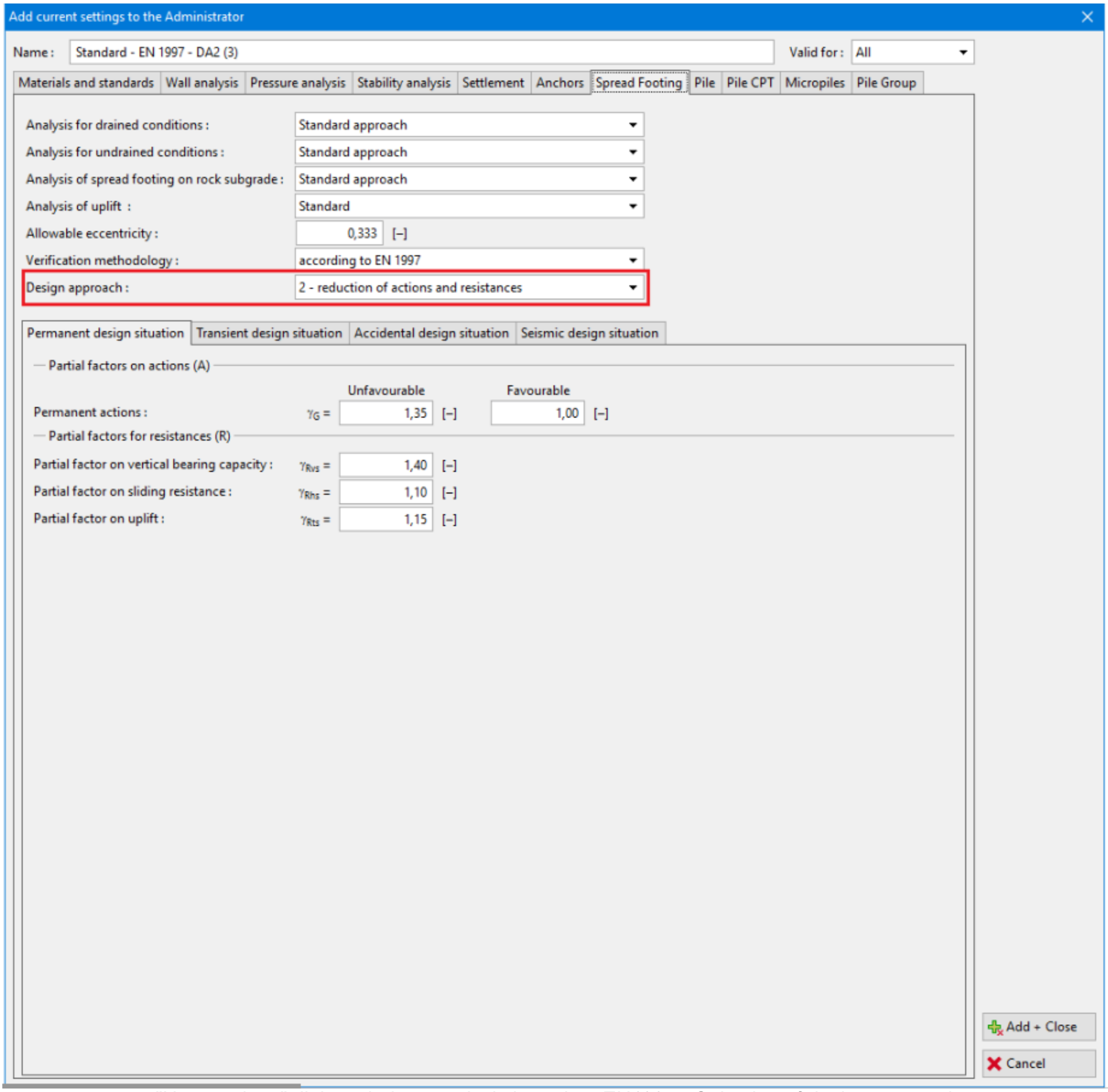
EN 1997-1 introduces three **design approaches** into the analysis; they differ by the application of partial factors.

According to EN 1997-1, the partial factors are generally applied to load actions, their impact on properties of foundation soil M, resistance R, or both. The values of partial factors not only differ by the assumed **design approach**, but also by the type of the analyzed geotechnical task (support structures, piles, etc.). The values of partial factors are in general specified by the Eurocode in **Annexes A**; **NA** specifies the national choice of values of partial factors. The program automatically displays the required coefficients depending on the selected design approach or on the selection of other parameters in the setting.

Regarding the fact that individual **Design approaches** introduce the partial factors into the analysis in a different way (e.g., partial factors on actions on a structure and the resulting structure resistance or actions and soil parameters) it is logical that the results attributed to these design approaches may also considerably differ. If the **National Annex** does not recommend a **Design approach** for a given geotechnical task, it is up to the designer to select it (and therefore also to evaluate whether the results correspond to the analyzed situation).

- Design approach 1 - Verification is performed for two sets of coefficients (**Combination 1 and Combination 2**) used in two separate analyses. Coefficients are applied to **load actions and to material parameters**.

- Design approach 2 - Applies partial factors to **load actions and material resistance** (bearing capacity).
- Design approach 3 - Applies partial factors to **load actions** and, at the same time, to the **material** (material parameters of soil).



"New settings" dialog window, analysis based on EN1997 - Selection of design approach

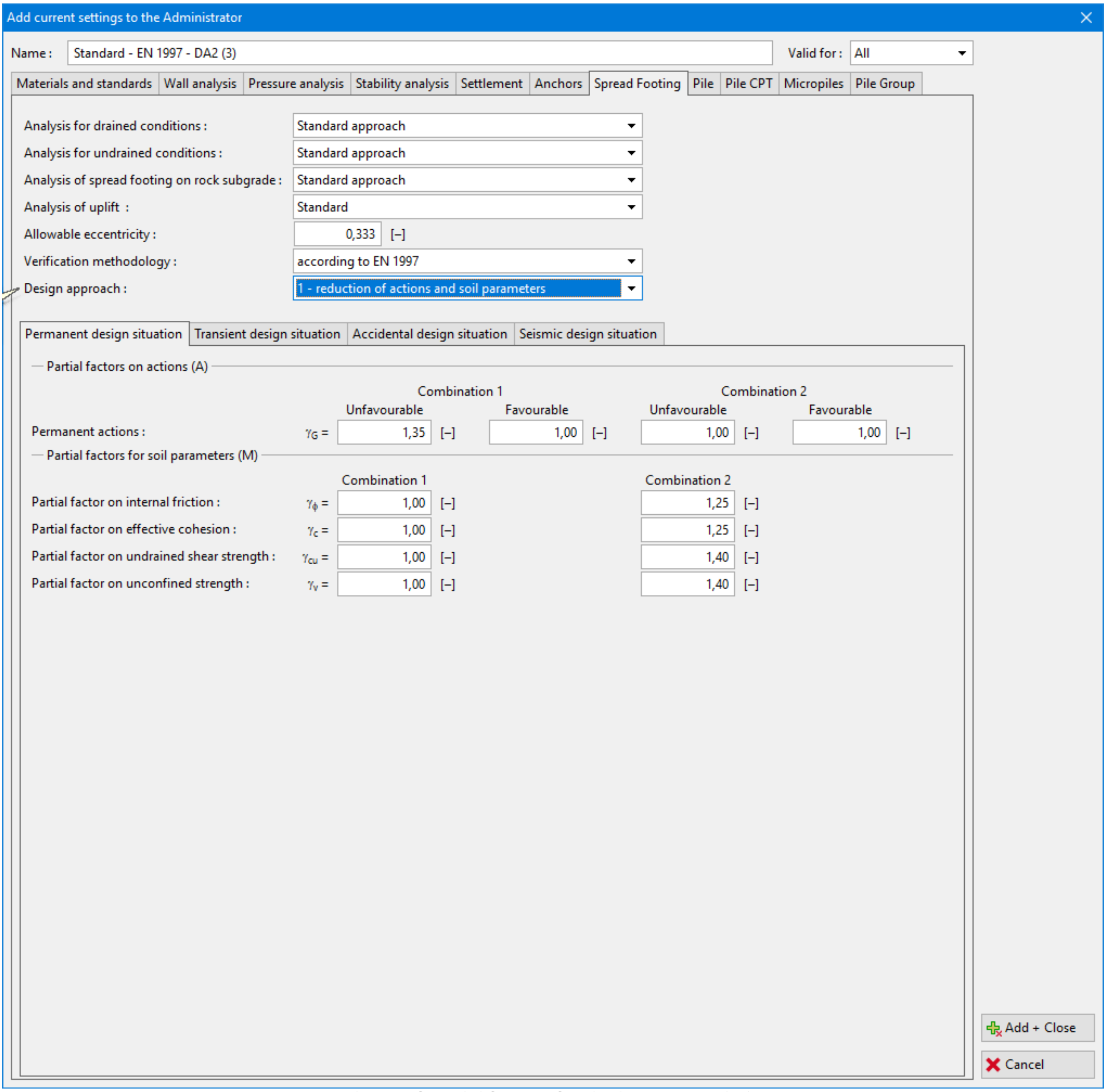
Design Approach 1

The verification analysis is performed for two sets of coefficients (**Combination 1** and **Combination 2**) used in two separate analyses. For **combination 1**, the partial factors are applied to **load actions only**, the remaining coefficients are set equal to 1.0. For **combination 2**, the partial factors are applied to **material parameters (material parameters of soil), and to variable load actions**, the remain coefficients are set equal to 1.0.

In programs analyzing walls and performing stability analyses, the analysis is carried out for **both combinations automatically,** and the results are presented for the **most unfavorable situation**. In the output protocol, there are calculations of both combinations described in detail.

This approach is not applicable to the "Sheeting Check" program. The combination, for which the analysis should be carried out, must be selected in the "Pressure Analysis" dialog windows.

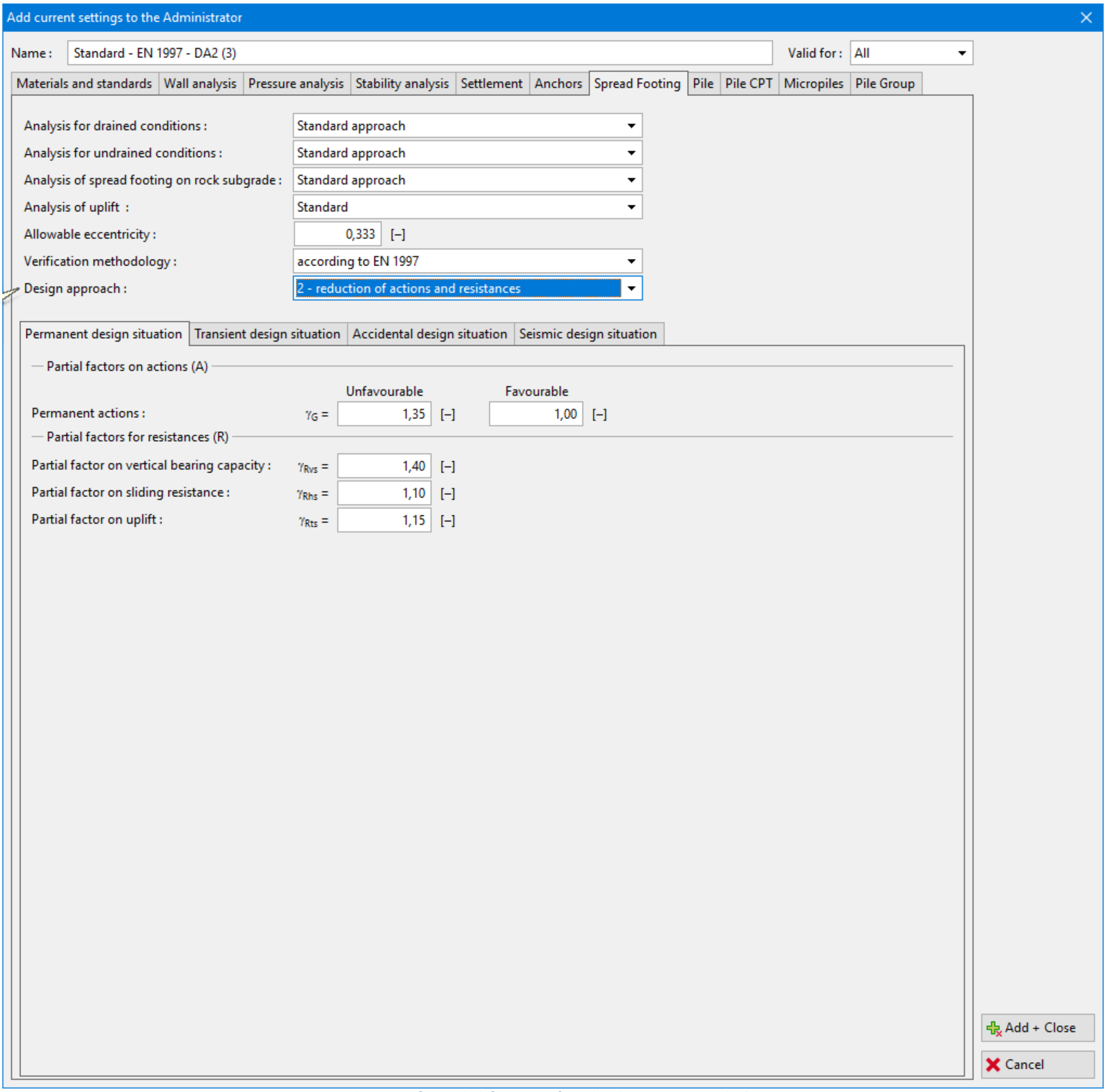
In "Spread Footing" and "Pile" programs, it is necessary to specify service load even for the bearing capacity analysis. The **design load** is considered with combination 1, the **service load** than with combination 2.



Input of partial factors for design approach 1

Design Approach 2

Design approach 2 applies the partial factors to **load actions** and to **material resistance** (bearing capacity).

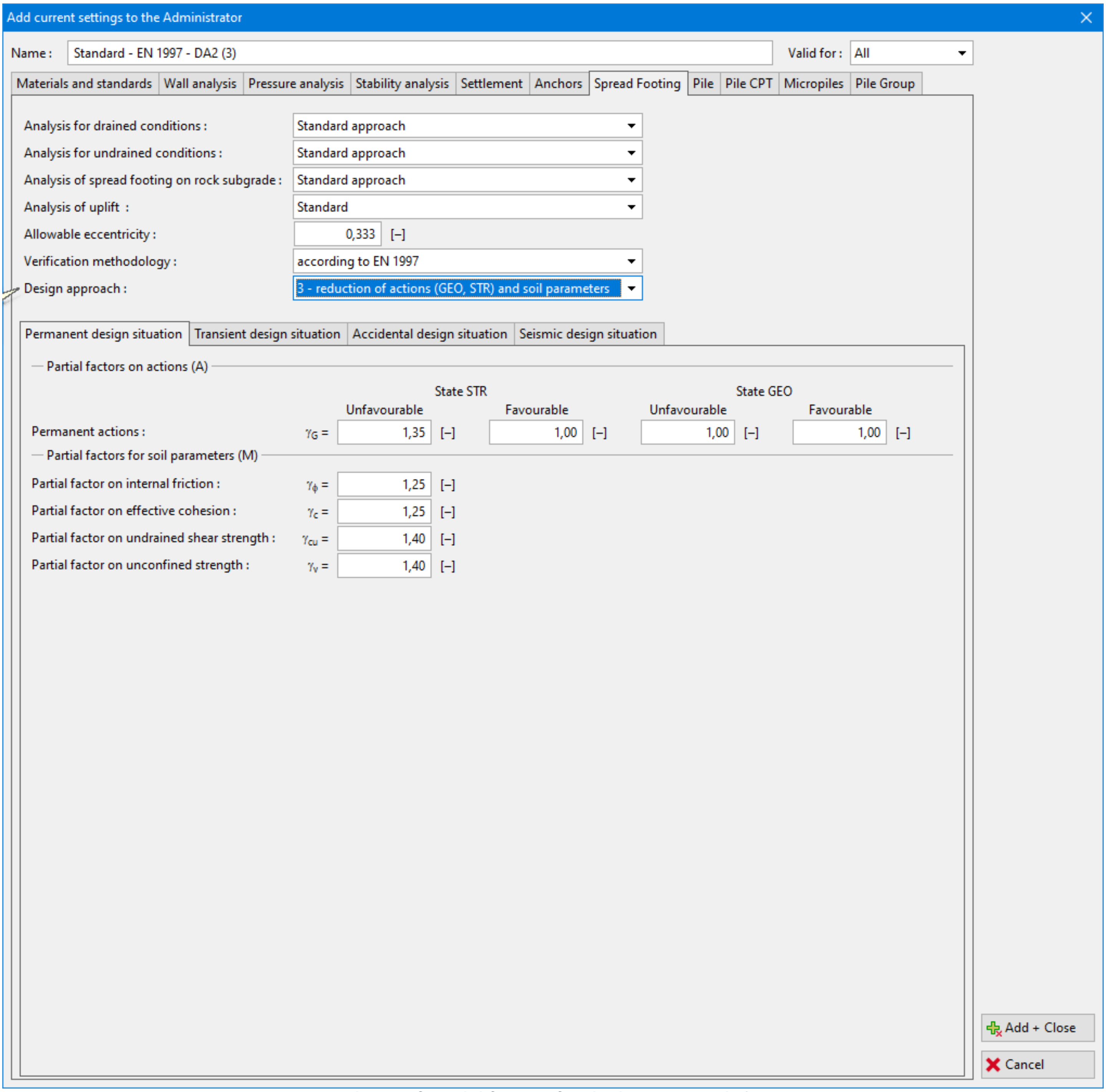


Input of partial factors for design approach 2

Design Approach 3

Design approach 3 applies the partial factors to **load actions** and at the same type to the **material** (material parameters of soil).

Contrary to other design approaches, it distinguishes **geotechnical loads - GEO State** (load actions caused by soils - e.g. earth pressures, pressures due to a surcharge, water action) and **loads applied to structures - STR State** (the program considers the structure self-weight, applied forces acting on the structure, anchors, geo-reinforcements, mesh overhangs). A different set of coefficients, specified in the "Partial factors" dialog window, is used for each type of load. Partial factors applied to geotechnical loads are mostly smaller than those applied to structure loads.



Input of partial factors for design approach 3

National Annex (NA)

The **National Annex (NA)** offers details on the method of application of the Eurocode at a national level (in individual EU countries) and it was usually issued together with ENV of the given country.

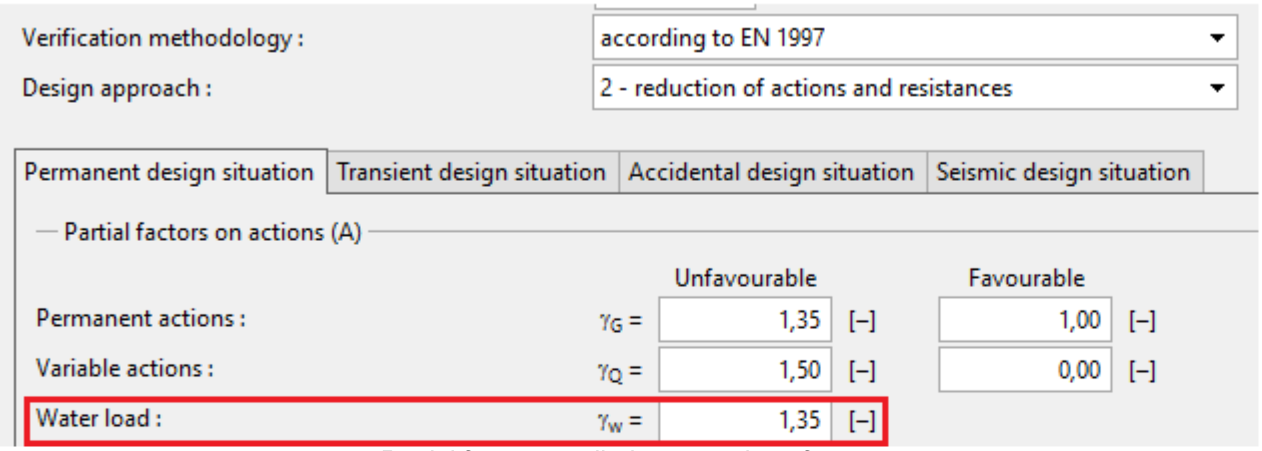
The National Annex, therefore, determines the choice of partial factors at a national level and the application of design approaches for individual geotechnical tasks. Owing to the fact that the content of NA remains open in some member countries, national annexes are not implemented into programs for all member countries.

Individual national annexes can be selected from the pre-defined settings available in the settings administrator and settings list.

New settings can be created by the user from the existing ones and in that way to define their **own national annexes**.

Partial Factors on Water

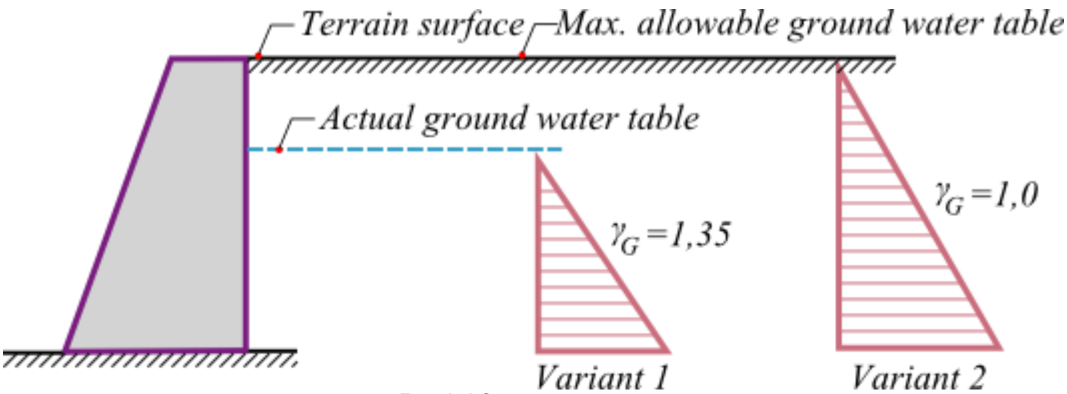
Partial factor on water action adjusts the force magnitude due to water action; the magnitude of pore pressure, respectively.



Partial factors applied to an action of water

The partial factor on water action can be input because EN 1997 offers several ways how to account for the influence of water. The two basic approaches are:

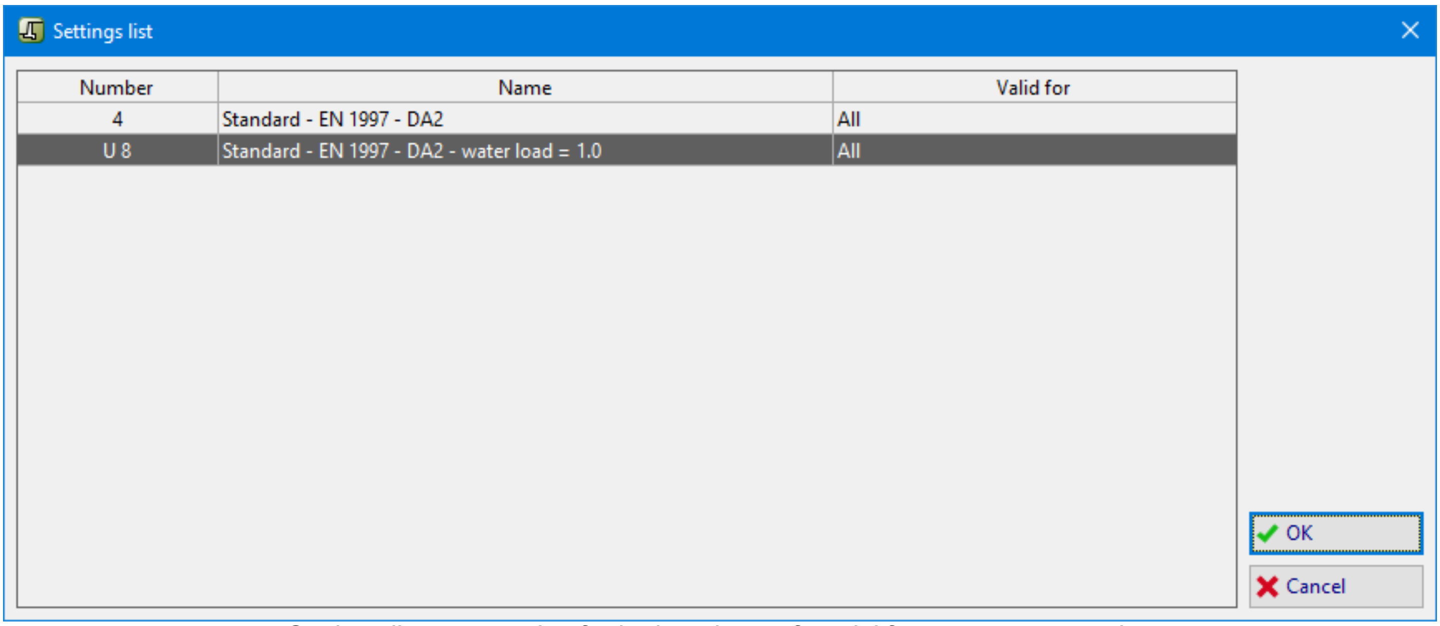
- **Variant 1** - the coefficient of water action is set to 1.3 or 1.35, respectively (some NA). In this case, the actual groundwater table is considered and its influence is multiplied by the input partial factor.
- **Variant 2** - the coefficient of water action is set to 1.0, or in other words, the action of water is not considered in the analysis. In this case, the maximum allowable groundwater table must be considered.



Partial factors on water

The selection of a particular option for the verification remains upon the user.

Providing the user adopts both options, we recommend introducing two settings in the "Settings administrator", which differ by the magnitude of coefficient γ_w .



Settings list - pre-setting for both variants of partial factors on water action

Analysis of Walls (Support Structures)

Analysis based on EN 1997 introduces several partial factors according to the selected Design approach (DA).

Designing a structure according to EN 1997-1 essentially follows the analysis of limit states.

Load reduction (DA1, DA2, DA3):

All design approaches consider partial factors reducing load. These are used to multiply all forces entering the analysis. For actual verification of individual modes of failure, the program determines whether the **force or pressure acts favorably or unfavorably**. Depending on that, these actions are then multiplied by the corresponding partial factor. Information regarding the applied partial factors is stored in the analysis protocol.

Forces acting on construction

Name	F_{hor} [lbf/ft]	App.Pt. Z [ft]	F_{vert} [lbf/ft]	App.Pt. X [ft]	Coeff. overtur.	Coeff. sliding	Coeff. stress
Weight - wall	0.0	-6.10	10005.0	6.01	1.000	1.000	1.350
FF resistance	-124.6	-0.67	0.0	0.00	1.000	1.000	1.000
Weight - earth wedge	0.0	-7.00	8243.4	8.64	1.000	1.000	1.350
Active pressure	9031.7	-6.49	12610.5	11.10	1.000	1.350	1.350
Force No. 1	2000.0	-23.00	0.0	5.10	1.350	1.350	1.350

Analysis protocol

The frame analysis allows defining "**Secondary variable actions**" - corresponding partial factors are then multiplied by the combination coefficients of the load.

When analyzing supporting structures, the water load and corresponding partial factor for water is very important.

Reduction of material (DA1, DA3):

Soil parameters are automatically reduced by corresponding partial factors.

Reduction of resistance (DA2):

Corresponding magnitudes of resistant forces, moments, and bearing capacities are reduced.

When performing analysis according to the Design approach 1, all verifications are carried out twice for both combinations of load. For a given limit state, the highest stressed design is displayed on the desktop.

Analysis of Sheeting Structures

Analysis based on EN 1997 introduces several partial factors according to the selected Design approach (DA).

Designing a structure according to EN 1997-1 essentially follows the analysis of limit states.

Load reduction (DA1, DA2, DA3):

In programs that consider the overall earth pressure in the analysis (**Earth Pressures, Sheeting Design, Sheeting Check**) the partial factors are used to multiply individual components of pressure acting on a structure.

The basic assumption of the analysis is that the **active earth pressure acts unfavorably** whereas the **passive earth pressure** is considered as **favorable**. Individual pressure diagrams are therefore multiplied by the corresponding partial factor.

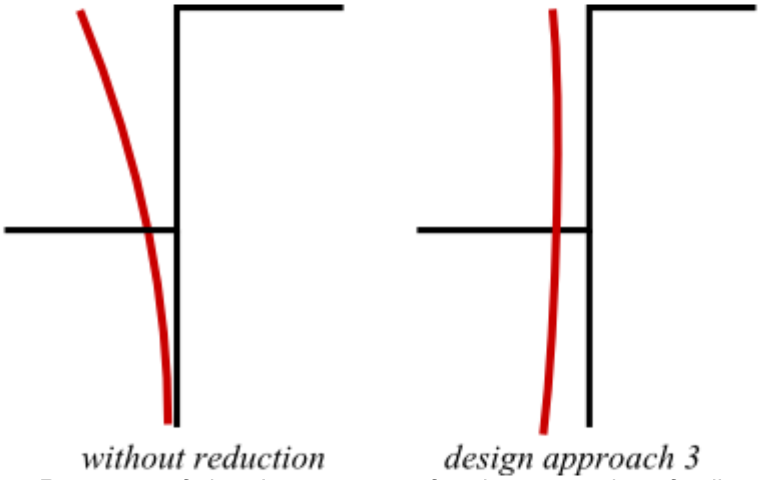
Reduction of material (DA1, DA3):

Parameters of soils are automatically reduced by the corresponding partial factors.

Reduction of resistance (DA2): is considered. Partial factors on resistance reduce the passive earth pressure in front of the structure.

In **simple words**, DA1 - Combination 2, DA2, and DA3 **increase the magnitude of active pressure** and **reduce the magnitude of passive pressure**, while DA1 - combination 1 only **increases the magnitude of active pressure**.

This approach may, therefore, **change in some cases the structure behavior** and deliver **misleading results**. Caution must, therefore, be exercised when reducing input parameters.



Response of sheeting structure after the excavation of soil

Analysis of Foundations (Spread Footing, Piles)

Analysis based on EN 1997 introduces several partial factors according to selected Design approach (DPA).

Designing a structure according to EN 1997-1 essentially follows the analysis of limit states.

Load reduction (DA1, DA2, DA3):

Load of the foundation is taken as a result of the analysis of the upper structure.

- load cases are determined according to rules provided by EN 1990:2002
- combinations of load cases are calculated according to EN 1991

The results of calculated combinations then serve as an input to "**Spread footing**" and "**Pile**" programs.

Either **design** (bearing capacity analysis, dimensioning of foundation) or **service** (analysis of settlement) **load** is considered. In Design approach 1, the analysis is performed for both the input design load (combination 1) and input service load (combination 2).

Only the **structure self-weight** or the **weight of soil above footing** is multiplied by the partial factors in the program. The specified design load must be determined in accordance with the **EN 1990** and **EN 1991** standards - individual components of the load must be **multiplied** by the corresponding partial factors - **the program does not change the input load any further**.

Reduction of material (DA1, DA3):

Parameters of soils are automatically reduced by the corresponding partial factors.

Reduction of load (DA2), for piles (DA1, DA2, DA3):

The partial factors in the "**Pile**" program are dependent on the type of pile (**bored, driven, CFA**). The window allows us to define all partial factors. The analysis then adopts partial factors depending on the type of pile selected in the "**Geometry**" frame. Verification of the **tensile pile** always considers the pile self-weight. For the **compressive pile,** the pile self-weight can be neglected depending on the settings in the "**Load**" frame. The actual verification analysis is performed according to the theory of limit states.

Vertical and horizontal bearing capacity of the foundation is reduced in the "**Spread Footing**" program.

Slope Stability Analysis

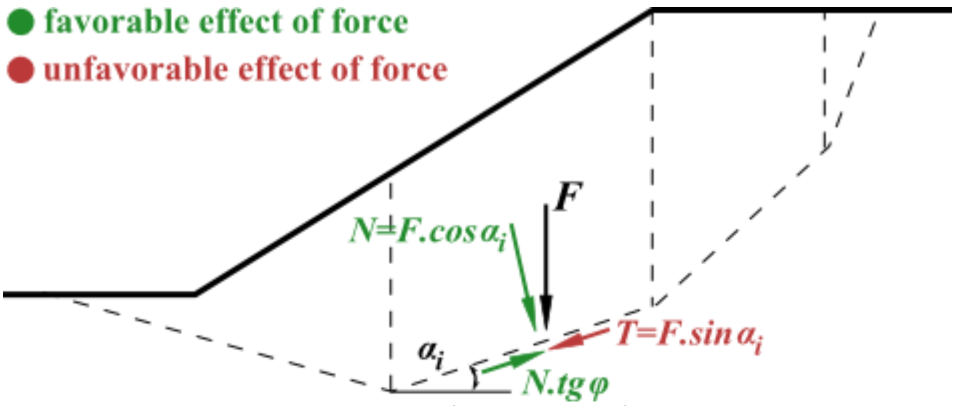
Analysis based on EN 1997 introduces several partial factors according to the selected Design approach (DA).

Designing a structure according to EN 1997-1 essentially follows the analysis of limit states.

Load reduction (DA1, DA2, DA3):

Loads acting on a strip are reduced in the analysis by partial factors. Depending on the inclination of the slip surface, the program evaluates whether the **gravity force** acting on a given block is favorable or not. If the favorable action of the force is greater than the unfavorable one, the program adopts the favorable coefficient. Based on that, the weight of block W is multiplied by the partial factor for the permanent load.

The water load is reduced by the partial factor, which multiplies the resulting pore pressure and forces due to unconfined water above the terrain.



Determining whether the force action is favorable or not

For input surcharges, the program first evaluates whether these act favorably or not, and then multiplies the **overall load** by the corresponding partial factor.

Reduction of material (DA1, DA3):

Parameters of soils are automatically reduced by the corresponding partial factors.

Reduction of resistance (DA2):

Resistance on a slip surface is reduced.

WARNING !!! **When calculating slope stability based on total parameters, DA2 and DA1 (comb. 1) give unrealistic results** caused by a different reduction of the self-weight of massive (favorable and unfavorable). If adopting these approaches **we recommend adjusting the partial factors manually** (i.e. to increase the reduction partial factor on the resistance on the slip surface and to decrease reduction partial factors on load actions).

Load Combinations

Actions of loads that act simultaneously are introduced into the analysis with the help of load combinations defined in **EN 1990 Basis of Structural Design**. Most of the loads are considered permanent. Surcharges and input forces can be specified as a variable load. The program automatically determines the values of individual partial factors depending on whether the given load acts in favor or unfavorably.

By default, the variable loads are considered as **major**. Nevertheless, the "**Verification**" and "**Dimensioning**" frames allow specifying the variable loads as **minor** - such load is then multiplied by the corresponding coefficient reducing its magnitude. Providing that all loads are considered in the basic combination as minor, the program prompts a warning and the verification is not accepted.

Four types of combinations can be specified in the "**Stage settings**" frame:

Permanent and transient design situation:

$$\sum_{j\geq 1} \gamma_{G,j} \cdot G_{k,j} + \gamma_{Q,1} \cdot Q_{k,1} + \sum_{i>1} \gamma_{G,i} \cdot \psi_{0,i} \cdot Q_{k,i}$$

where:

- $G_{k,j}$ - characteristic value of j^{th} permanent load
- $\gamma_{G,j}$ - partial factor of j^{th} permanent load
- $Q_{k,i}$ - characteristic value of a secondary i^{th} variable load
- $Q_{k,1}$ - characteristic value of a primary variable load
- $\gamma_{Q,i}$ - partial factor of i^{th} variable load
- ψ_0 - factor for the combination value of a variable load

Accidental and seismic design situation:

$$\sum_{j\geq 1} G_{k,j} + A_d + \psi_{1,i} \cdot Q_{k,1} + \sum_{i>1} \psi_{2,i} \cdot Q_{k,i}$$

where:

- $G_{k,j}$ - characteristic value of j^{th} permanent load
- $Q_{k,1}$ - characteristic value of primary variable load
- $\psi_{1,i}$ - factor for frequent value of variable load
- $\psi_{2,i}$ - factor for quasi-permanent value of variable load
- A_d - design value of an extreme load

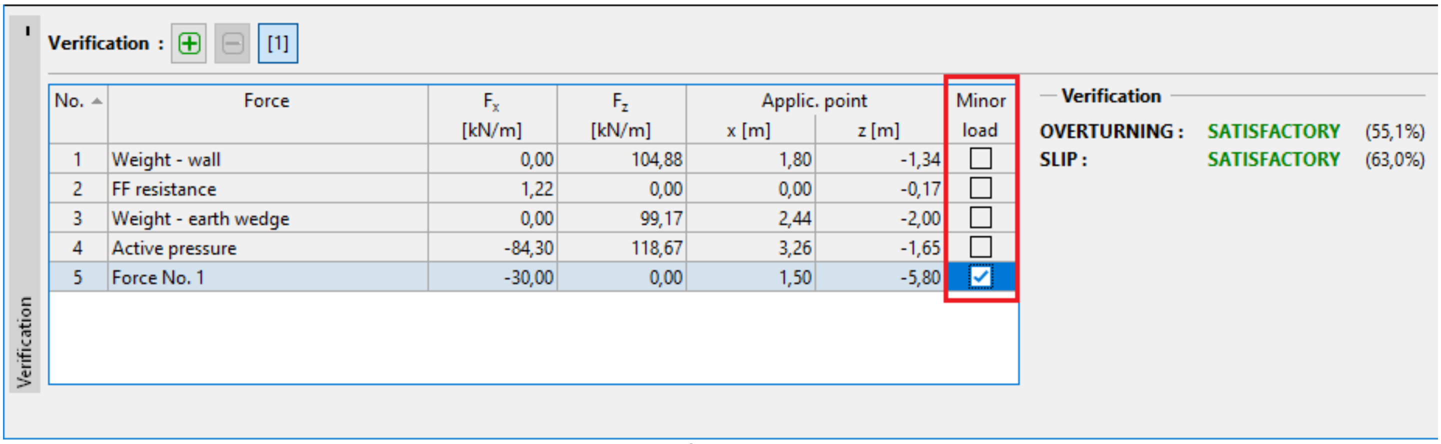
Seismic design situation:

$$\sum_{j\geq 1} G_{k,j} + A_{Ed} + \sum_{i>1} \psi_{2,i} \cdot Q_{k,i}$$

where:

- $G_{k,j}$ - characteristic value of j^{th} permanent load
- $Q_{k,i}$ - characteristic value of secondary j^{th} variable load
- $\psi_{2,i}$ - factor for quasi-permanent value of variable load
- A_{Ed} - design value of a seismic load

Load partial factors and **combination coefficients** are introduced in the "Partial factors" dialog window.



Input of minor loads

Analysis According to LRFD

When performing analysis according to LRFD (**Load Resistance Factor Design**) we follow the theory of limit states.

The analysis according to LRFD introduces two types of design coefficients:

- coefficients modifying the load magnitude (**Load factors**)
- coefficients reducing the soil resistance (**Resistance factors**)

LRFD is implemented in the program to perform:

- Analysis of retaining walls (support structures)
- Analysis of spread foundations
- Slope stability analysis

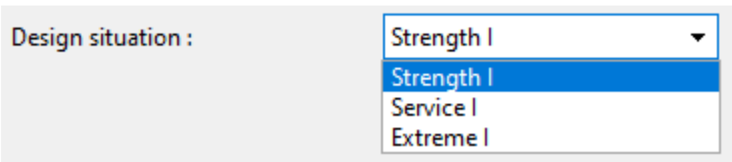
LRFD introduces new types of **design situations** (Strength I, Service I, Extreme I).

LRFD - Design Situations

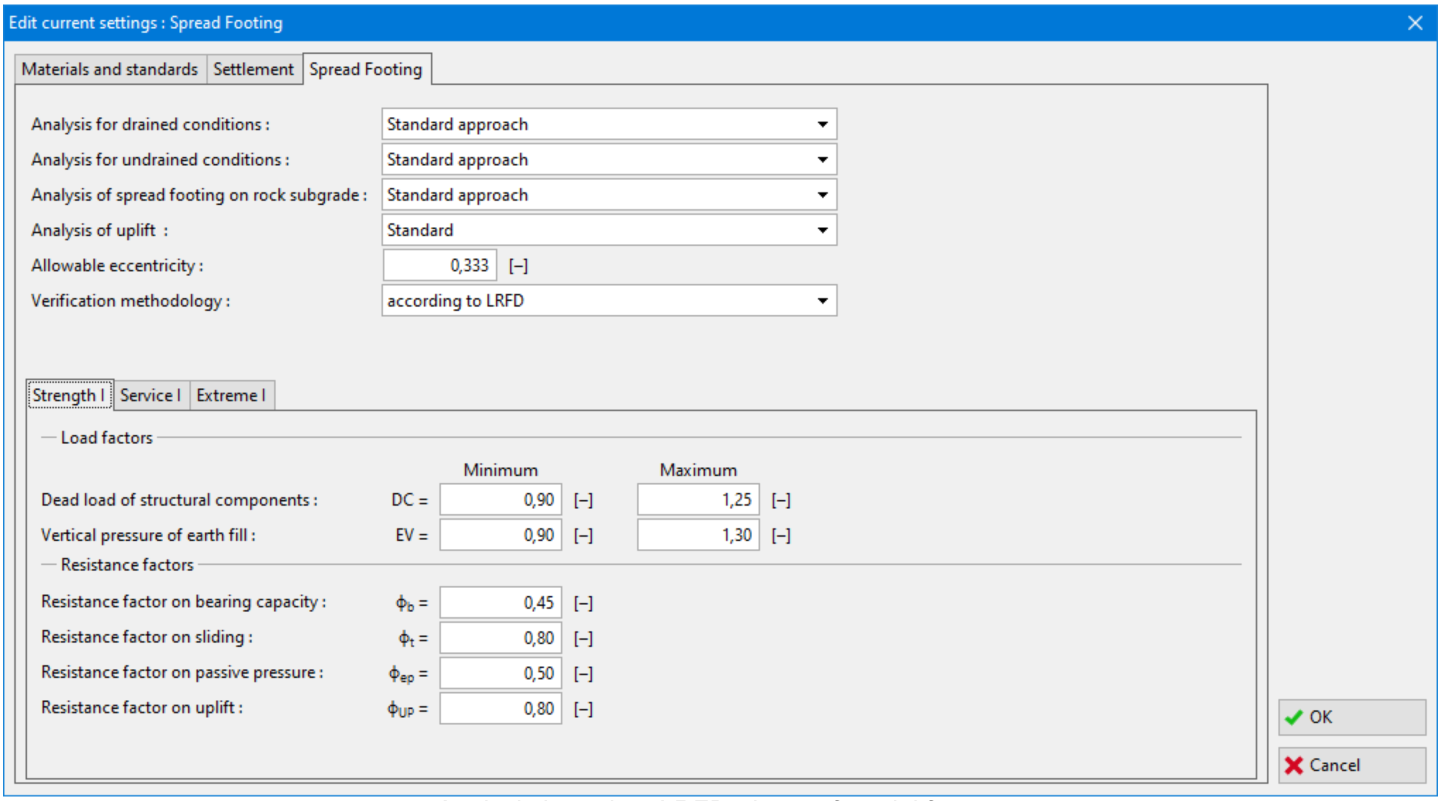
LRFD introduces the following design situations for the analysis of support structures (retaining walls), foundations, and slope stability:

- **Strength I**: : the basic design situation that reduces the structure resistance and the magnitude of the load.
- **Service I**: this design situation uses the values of partial factors (**load**, **resistance reduction**) equal to 1.0 for most cases.
- **Extreme I**: this design situation uses the values of partial factors (**resistance reduction**) equal to 1.0 for most cases.

The type of design situation is selected in the "**Stage settings**" frame. The values of partial factors (**load**, **resistance reduction**) can be modified in the "**Settings**" frame.



LRFD - Selection of design situation



Analysis based on LRFD - input of partial factors

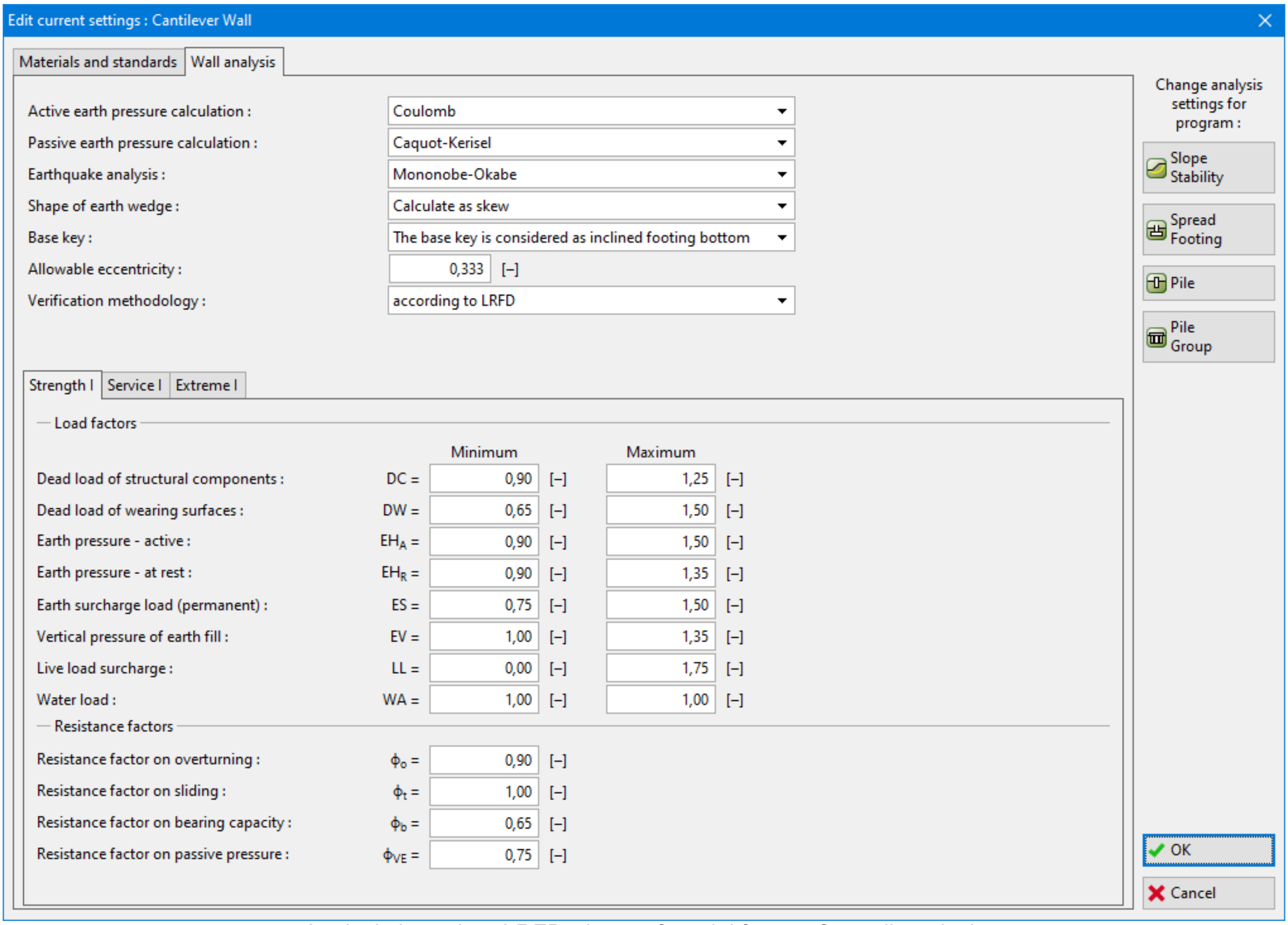
LRFD - Analysis of Retaining Walls (Support Structures)

Analysis according to LRFD introduces two types of design coefficients - coefficients modifying the load magnitude (**Load factors**) and coefficients reducing the soil resistance (**Resistance factors**).

These coefficients enter the analysis according to the selected design situation.

When evaluating individual cases of a failure, the program determines whether the **force or pressure acts favorably or unfavorably**. It is then multiplied by the corresponding partial factor.

When performing the final verification, the overall resistance of the structure against failure is multiplied by the corresponding resistance factor.



Analysis based on LRFD - input of partial factors for wall analysis

For support structures (walls), the information about the applied design factors is provided in the analysis protocol.

Forces acting on construction

Name	F_{hor} [lbf/ft]	App.Pt. Z [ft]	F_{vert} [lbf/ft]	App.Pt. X [ft]	Coeff. overtur.	Coeff. sliding	Coeff. stress
Weight - wall	0.0	-6.10	10005.0	6.01	0.900	0.900	1.250
FF resistance	-124.6	-0.67	0.0	0.00	1.000	1.000	1.000
Weight - earth wedge	0.0	-7.00	8243.4	8.64	1.000	1.000	1.350
Active pressure	9031.7	-6.49	12610.5	11.10	1.000	1.500	1.500
Surcharge n.1	67.8	-8.95	96.4	10.05	0.000	0.000	1.500
Force No. 1	2000.0	-23.00	0.0	5.10	1.350	1.350	1.350

Analysis protocol

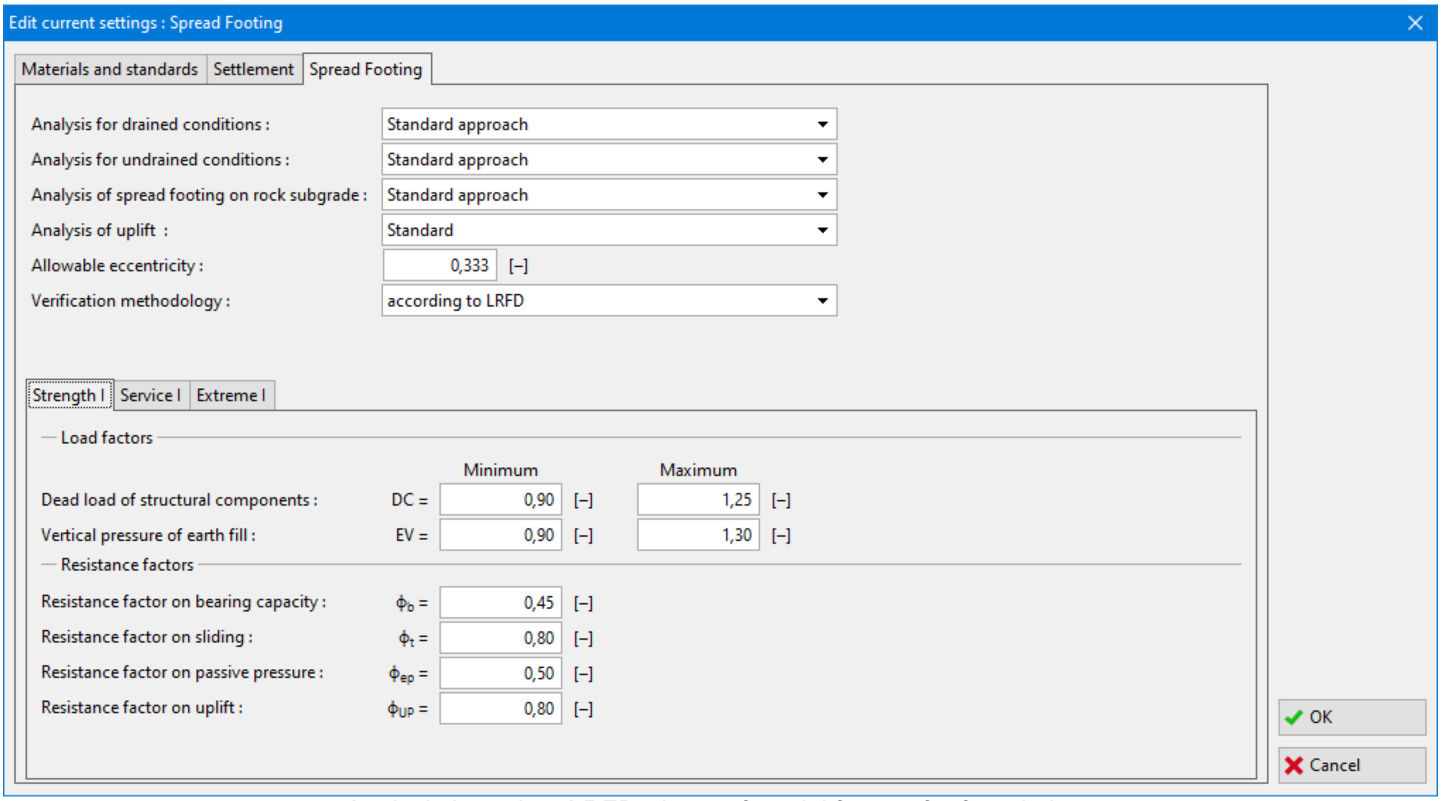
LRFD - Analysis of Spread Foundations

Analysis according to LRFD introduces two types of design coefficients - coefficients modifying the load magnitude (**Load factors**) and coefficients reducing the soil resistance (**Resistance factors**).

These coefficients enter the analysis according to the selected design situation.

Partial factors for load are used to multiply the **self-weight of spread foundation** and the **weight of soil above foundation (overburden)** only. Individual components of the load must be **multiplied** by corresponding partial factors - the input load is not adjusted by the program in any way.

When performing the final verification, the overall resistance of the structure against failure is multiplied by the corresponding resistance factor.



Analysis based on LRFD - input of partial factors for foundations

LRFD - Slope Stability Analysis

Analysis according to LRFD introduces two types of design coefficients - coefficients modifying the load magnitude (**Load factors**) and coefficients reducing the soil resistance (**Resistance factors**).

These coefficients enter the analysis according to the selected design situation.

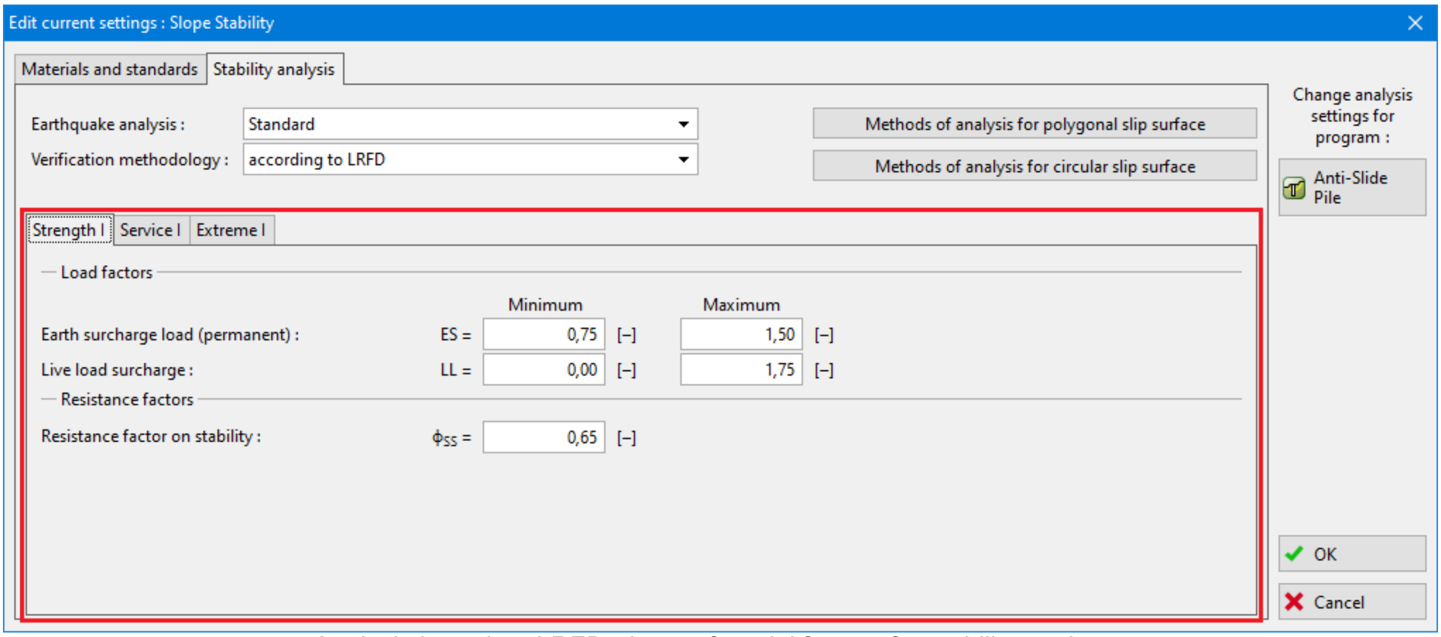
The input loads are verified, whether they act favorably or unfavorably. The **surcharge magnitude** is then multiplied by the corresponding partial factor for the load (**ES** or **LL**, respectively).

Resistance reduction (**Resistance factors**):

The overall resistance on the slip surface is reduced by the ϕ_{SS} partial factor. When evaluating the safety, the following condition on the slip surface must be satisfied:

$$\phi_{SS} F_{pas} \geq F_{act}$$

where:	ϕ_{SS}	-	resistance factor on stability
	F_{pas}	-	resisting (passive) forces acting on the slip surface
	F_{act}	-	active forces acting on the slip surface



Analysis based on LRFD - input of partial factors for stability analyses

Analysis According to Chinese Standards

GEO5 programs allow performing various analyses based on the methodologies provided by the Chinese standards.

Geotechnical analyses are verified using the safety factor. Neither calculated forces nor soil parameters are reduced by any coefficient.

Dimensioning of steel-reinforced concrete and masonry structures follows the GB 50153-2008 or JTS D30-2004 standards. In this case, each force entering a combination is multiplied by the corresponding coefficient.

Another coefficient influencing the dimensioning is the **Coefficient of structure importance** (GB 50153-2008, 8.2.2-1) to be specified in the "Settings" frame when performing the structure verification according to GB 50010-2010.

Earthquake analysis and seismic combination analysis according to GB 50010-2010 further exploit **Seismic coefficients of strength** (GB 50011-2010), which increase the calculated bearing capacity of a cross-section. These coefficients are specified in the "**Settings**" frame in the "Materials and standards" tab.

Analysis of sheeting structures follows the JGJ 120-2012 standard (Technical specification for retaining and protection of building foundation excavations). This is e.g. a determination of modulus of subsoil reaction or ditch bottom verification.

Analysis According to BS8006

Designing a structure according to BS 8006-1 essentially follows the theory of limit states.

Partial factors, adjusting the characteristic values of load, material, and resistance, are introduced into the analysis depending on the selected **structure type**.

The structure type is selected in the "Stage Settings":

- Wall
- Slope

Wall Partial Factors

The verification analysis is performed for two sets of coefficients:

- **Combination A** - generates maximum foundation bearing pressure
- **Combination B** - generates the worst case for sliding along the base

The following partial factors are applied:

Partial factors of soil properties		
Partial factor on internal friction	f_{ms}	1.00
Partial factor on effective cohesion	f_{ms}	1.60
Partial factor on undrained shear strength	f_{ms}	1.00

Resistance factors		

Resistance factor on sliding	f_s	1.20
Resistance factor on bearing capacity	f_{ms}	1.35
Resistance factor for pullout of reinforcement	f_p	1.30
Partial factors of slip on reinforcement	f_s	1.30

Partial factors		**Combination A**	**Combination B**
Dead load of structural components	f_{fs}	1.50	1.00
Earth pressure - active	f_{fs}	1.50	1.50
Permanent load - on the block	f_s	1.50	1.00
Permanent load - behind the block	f_s	1.50	1.50
Variable load - on the block	f_q	1.50	0.00
Variable load - behind the block	f_q	1.50	1.50
Loads acting favourable	f	1.00	1.00

Slope Partial Factors

The calculation applies the following partial factors:

Partial factors of soil properties		
Partial factor on internal friction :	f_{ms}	1.00
Partial factor on effective cohesion :	f_{ms}	1.60
Partial factor on undrained shear strength :	f_{ms}	1.00

Resistance factors		
Resistance factor on sliding :	f_s	1.20
Resistance factor for pullout of reinforcement :	f_p	1.30
Partial factors of slip on reinforcement :	f_s	1.30

Partial factors		
Dead load of structural components :	f_{fs}	1.50
Earth surcharge load (permanent) :	f_f	1.20
Live load surcharge :	f_q	1.30
Loads acting favourable :	f	1.00

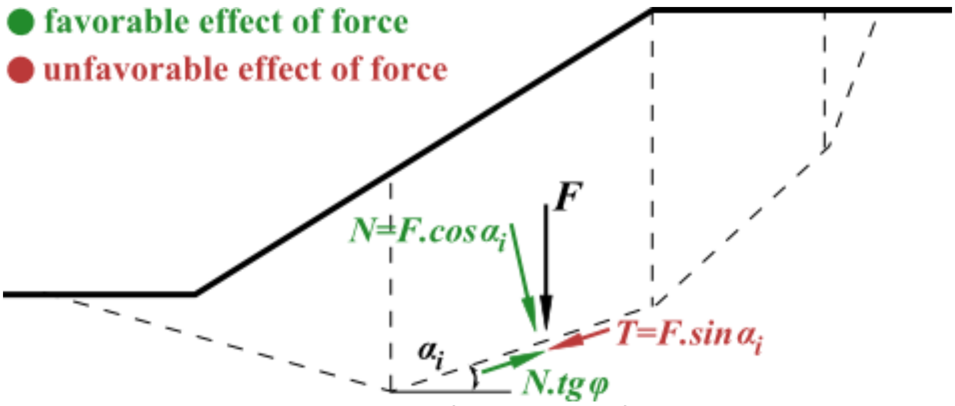
Analysis according to DIN1054

Designing a structure according to DIN1054 essentially follows the analysis of limit states.

Load reduction:

Loads acting on a strip are reduced in the analysis by partial factors. Depending on the inclination of the slip surface, the program evaluates whether the **gravity force** acting on a given block is favorable or not. If the favorable action of the force is greater than the unfavorable one, the program adopts the favorable coefficient. Based on that, the weight of block W is multiplied by the partial factor for the permanent load.

The water load is reduced by the partial factor, which multiplies the resulting pore pressure and forces due to unconfined water above the terrain.



Determining whether the force action is favorable or not

For input surcharges, the program first evaluates whether these act favorably or not, and then multiplies the **overall load** by the corresponding partial factor.

Reduction of material:

Parameters of soils are automatically reduced by the corresponding partial factors.

Design Situations

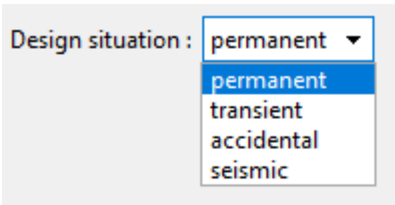
The programs allow defining four design situations, which may differ by the analysis coefficients. These are:

- **Permanent design situation** - the most common situation and type of verification, adopted when proving the safe design of a structure for the assumed lifetime.
- **Transient design situation** - can be used for temporary structures (i.e. construction stages). As a rule, lower safety is required in comparison to the permanent design situation.
- **Accidental design situation** - adopted for extraordinary loads (e.g. blast, vehicle impact, flood, fire, etc.). The values of partial factors are typically equal to one.
- **Seismic design situation** - applied to the analysis of earthquakes. It might seem similar to the accidental design situation, but for an earthquake higher safety is sometimes required. In some countries, the required safety is even the same as for the permanent design situation.

LRFD introduces new types of design situations (**Strength I**, **Service I**, **Extreme I**).

Safety coefficients and partial factors are specified in the analysis settings.

The corresponding design situation for a given construction stage is selected in the "**Stage settings**" frame.



Selection of design situation

Individual Programs

This chapter contains a basic description of individual ways of inputting data into the program:

- Earth Pressure
- Cantilever Wall
- Gravity Wall
- Prefab Wall
- Masonry Wall
- Gabion
- Abutment
- Nailed Slope
- Redi-Rock Wall
- Sheeting Design
- Sheeting Check
- Anti-Slide Pile
- Shaft

- Slope Stability
- Rock Stability
- MSE Wall
- Spread Footing
- Spread Footing CPT
- Pile
- Pile CPT
- Pile Group
- Micropile
- Slab
- Beam
- Settlement
- Ground Loss
- Stratigraphy (and modules Logs, Cross Sections, Earthworks)
- FEM (and modules Consolidation, Water Flow, Tunnel, Earthquake)

Program Earth Pressure

This program computes basic earth pressures (active pressure, passive pressure, and pressure at rest) acting upon an arbitrarily shaped structure.

The help in the "Earth Pressure" program includes the following topics:

- The input of data into individual frames:

Project	Settings	Geometry	Profile	Soils	Assign
Terrain	Water	Surcharge	Earthquake	Stage Settings	Analysis

- Standards and analysis methods

- Theory for analysis in the "**Earth Pressure**" program:

Stress in Soil Body	Earth Pressures

- Outputs

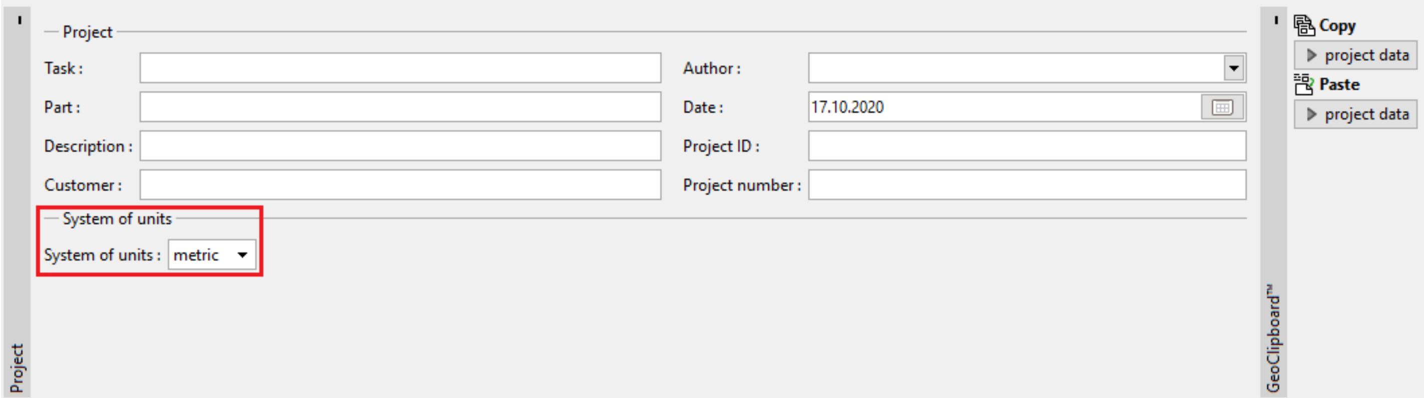
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The **"Project"** frame is used to input basic project data. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



"Project" frame

Settings

The "**Settings**" frame allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

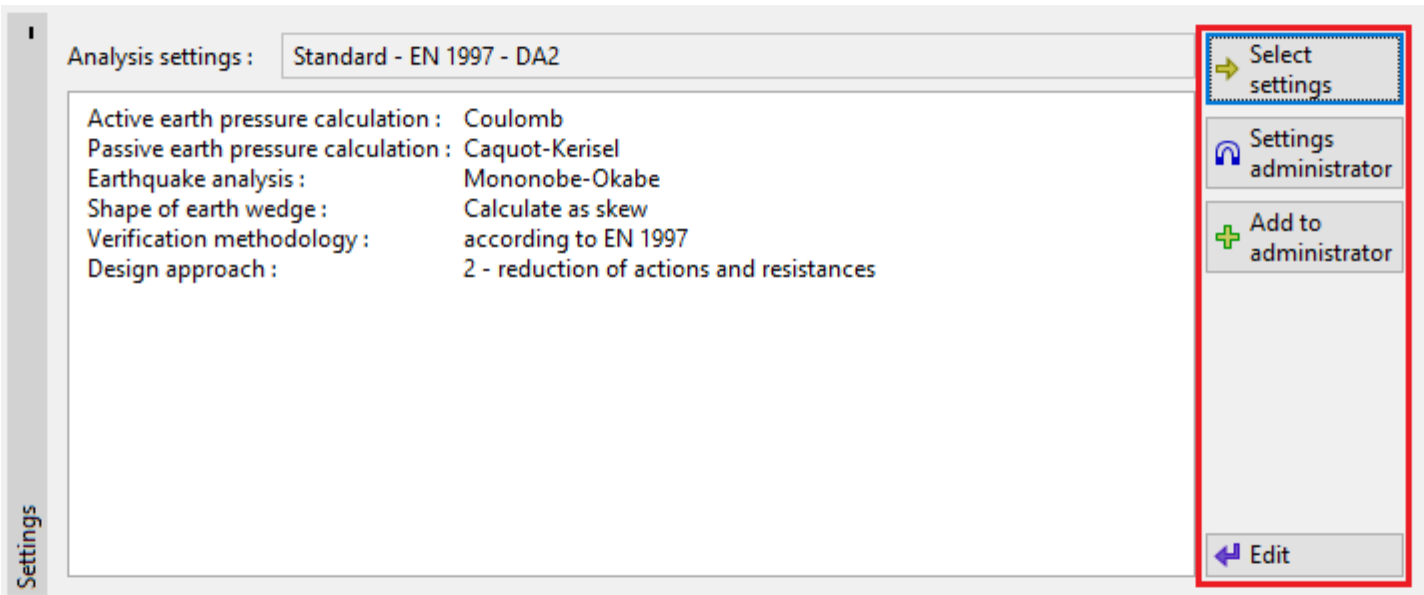
The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows us to view and modify an individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. The title changes to "**Input for the current task**" by modifying any of the parameters. Individual analyses are then performed with this **local setting**. If this setting is suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

Settings of analysis parameters are performed in the "Materials and standards" and the "Pressure Analysis" tabs.



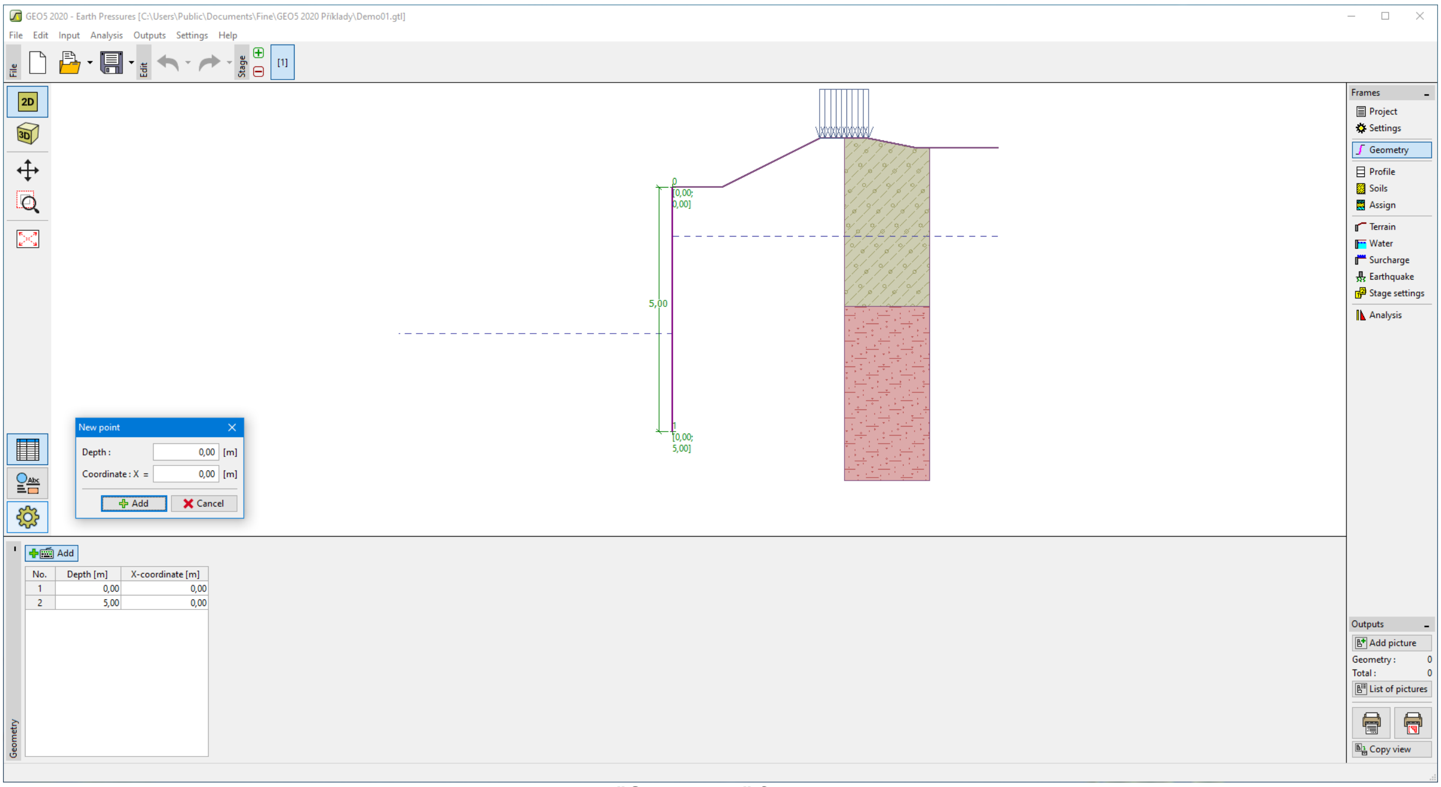
"Settings" frame

Geometry

The "**Geometry**" frame contains a table listing the points of a structure. Adding (editing) points is performed in the "**New point**" dialog window.

The existing geometry points can also be edited on the desktop using active objects - double-click on a selected point opens a dialog window to edit the point.

The program makes it possible to export the geometry of a structure in the *.DXF format.



"Geometry" frame

Profile

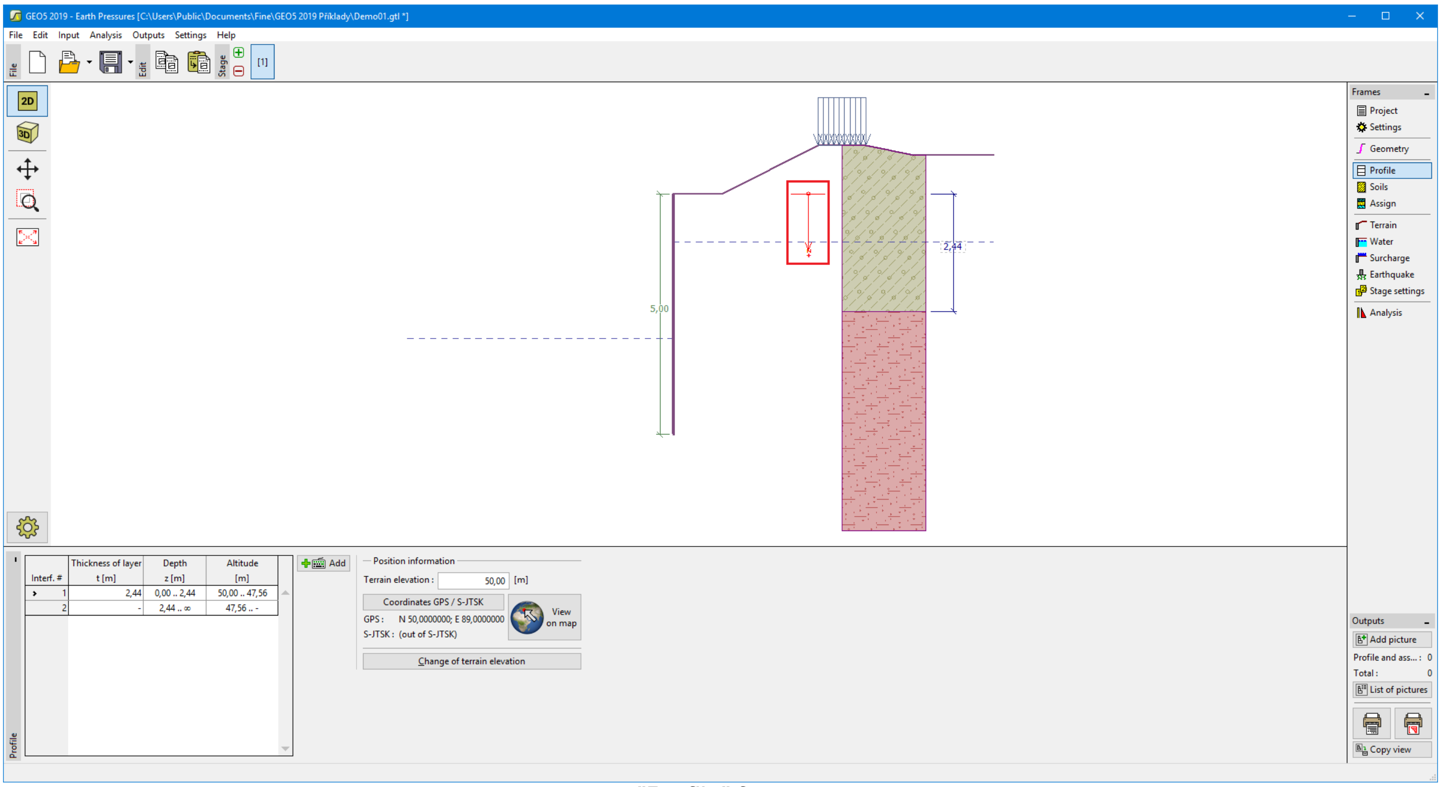
The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.



"Profile" frame

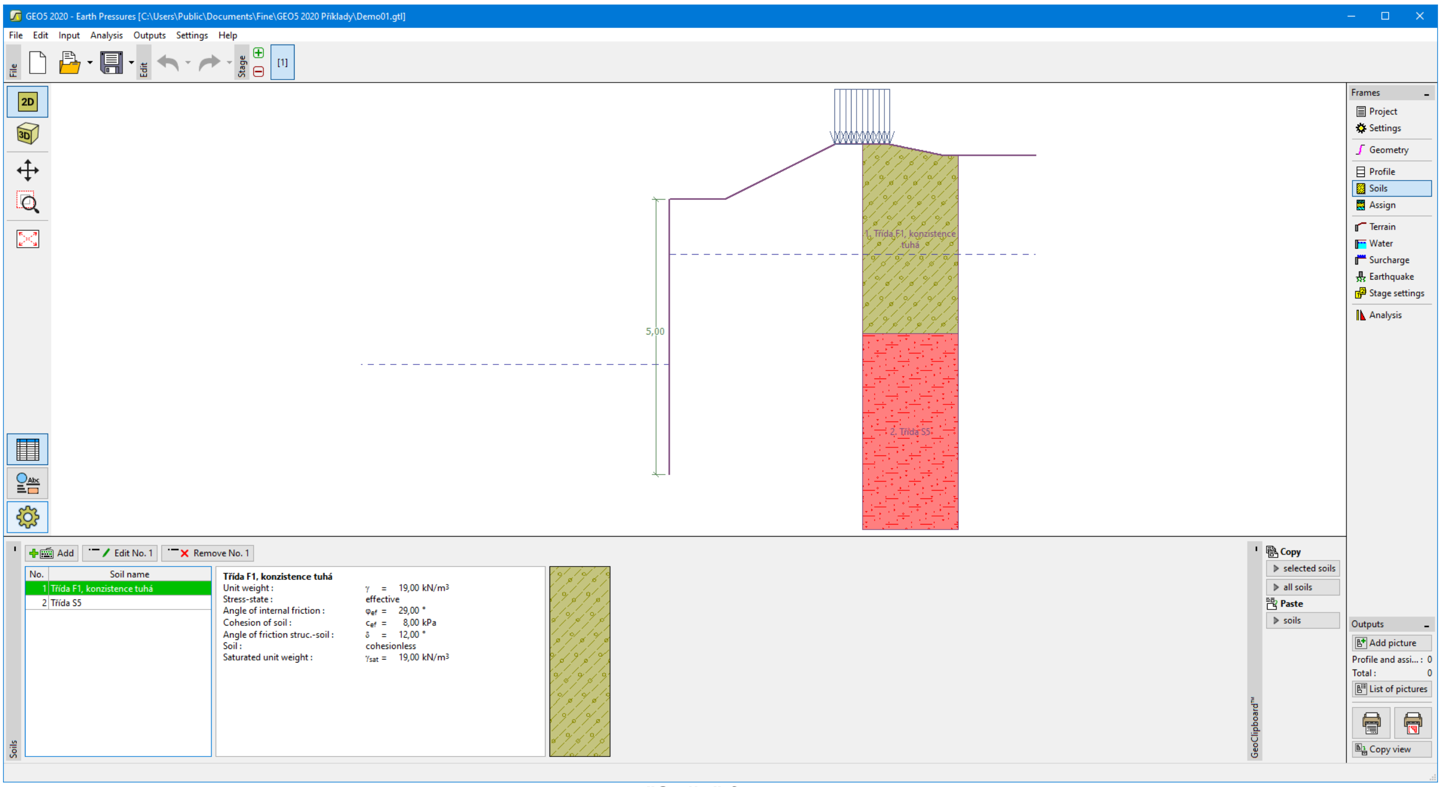
Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding a soil is performed in the "Add new soils" dialog window.

The soil characteristics entered in the program are further specified in the following chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



"Soils" frame

Basic Data

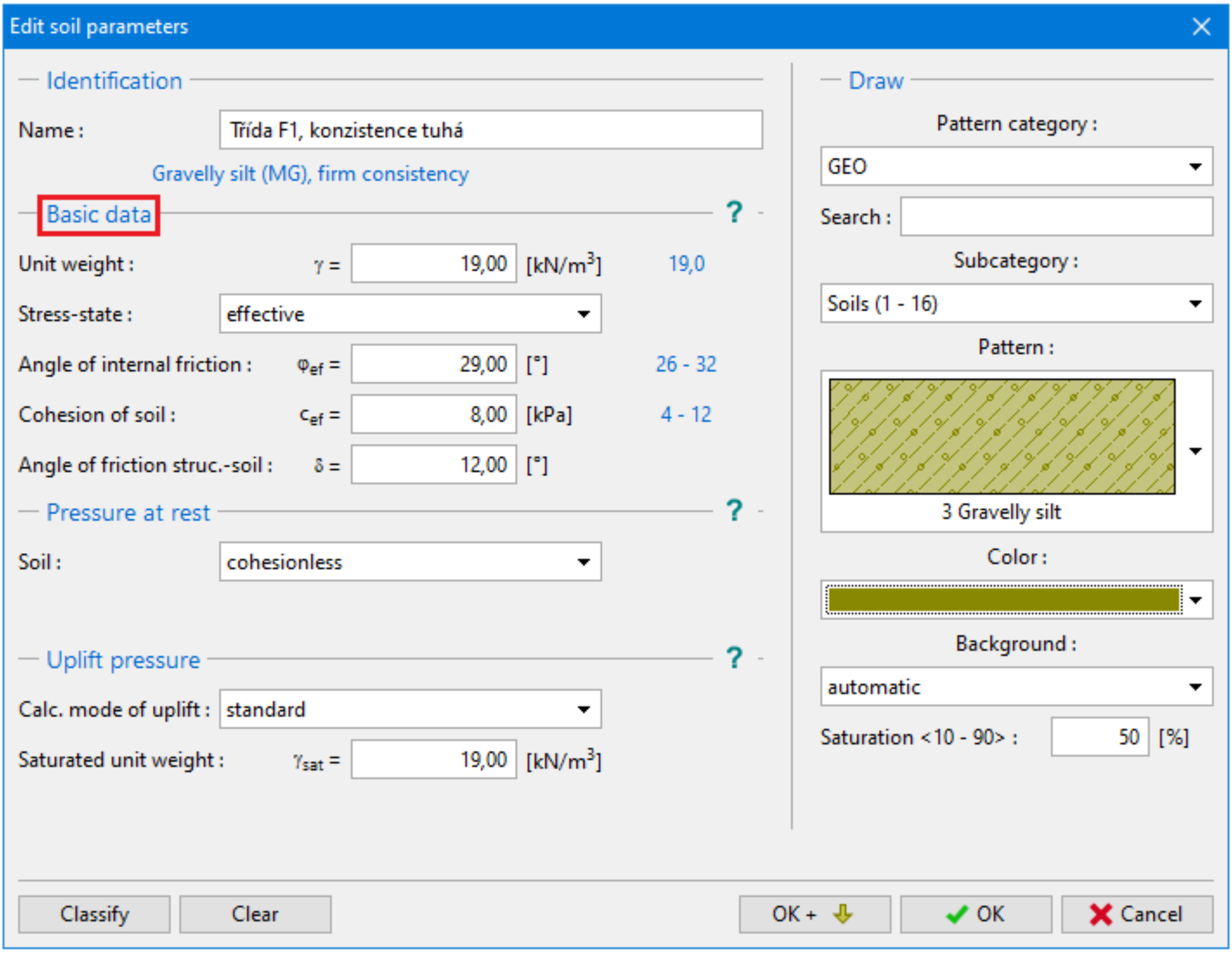
This part of the window allows us to introduce basic soil parameters - **unit weight, angle of internal friction, and cohesion**. The particular values are obtained from a geotechnical survey or from laboratory experiments. If this data is not available, it is possible to exploit the built-in soils database, which contains values of selected soils characteristics. The rocks characteristics are not listed in the built-in database, these parameters must be defined manually.

Parameters of the internal friction and cohesion angle are specified as **effective or total** depending on the settings in the "**Stress analysis**" combo list. Whether to use effective or total parameters, it depends primarily on the type of soil and load, structure duration, and water conditions.

For effective stress, specify the angle of internal friction between the soil and the structure, which depends on the structure material and type of soil. Possible values of this parameter are listed in the table of recommended values.

For total stress, specify the adhesion of soil to the structure face a .

The associated theory is described in detail in the chapter "Earth pressures".

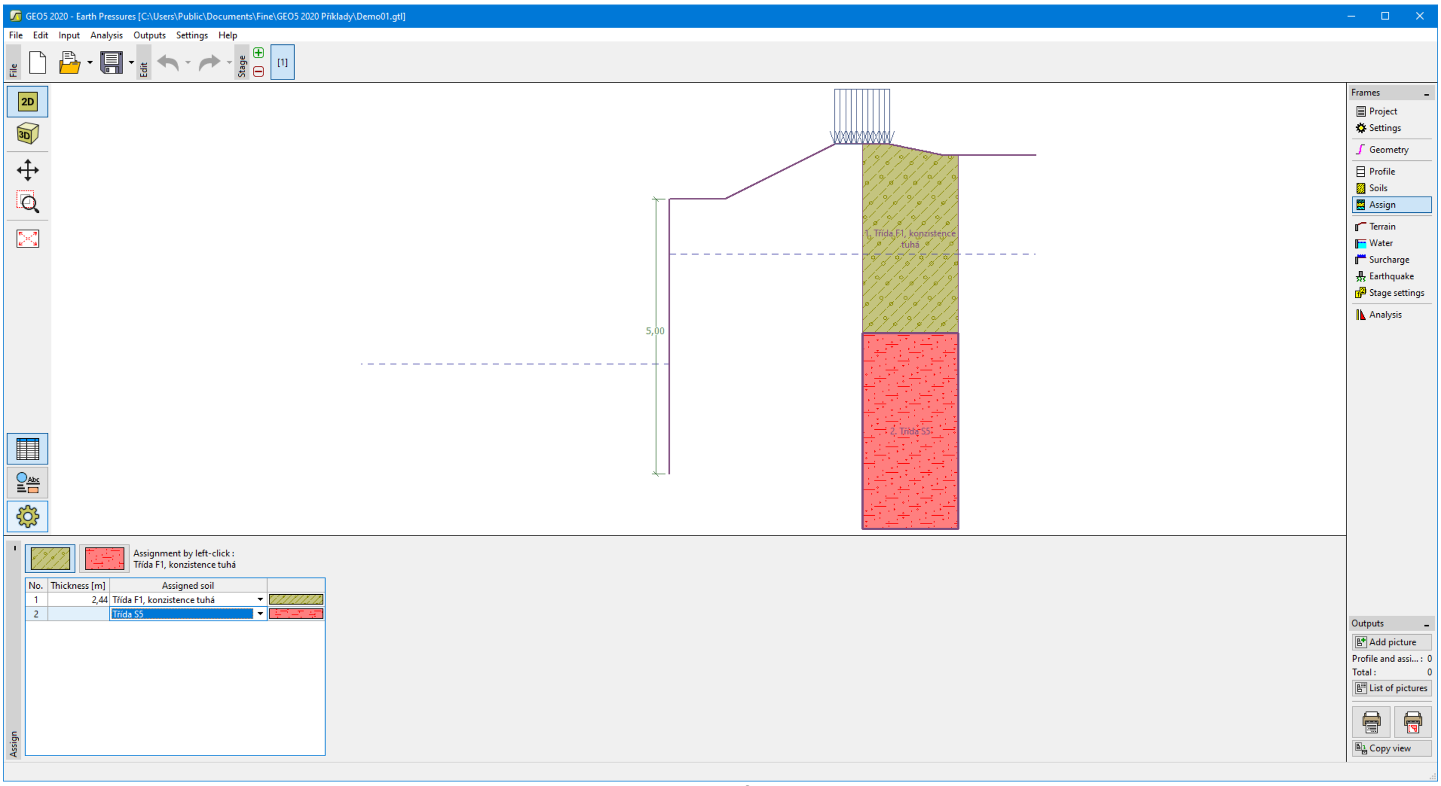


"Add new soils" dialog window

Assign

The "**Assign**" frame contains a list of profile layers and assigned soils. The list of soils is graphically represented by using buttons in the bar above the table or is accessible from a combo list for each profile layer.

The procedure to assign a soil to a layer is described in detail herein.



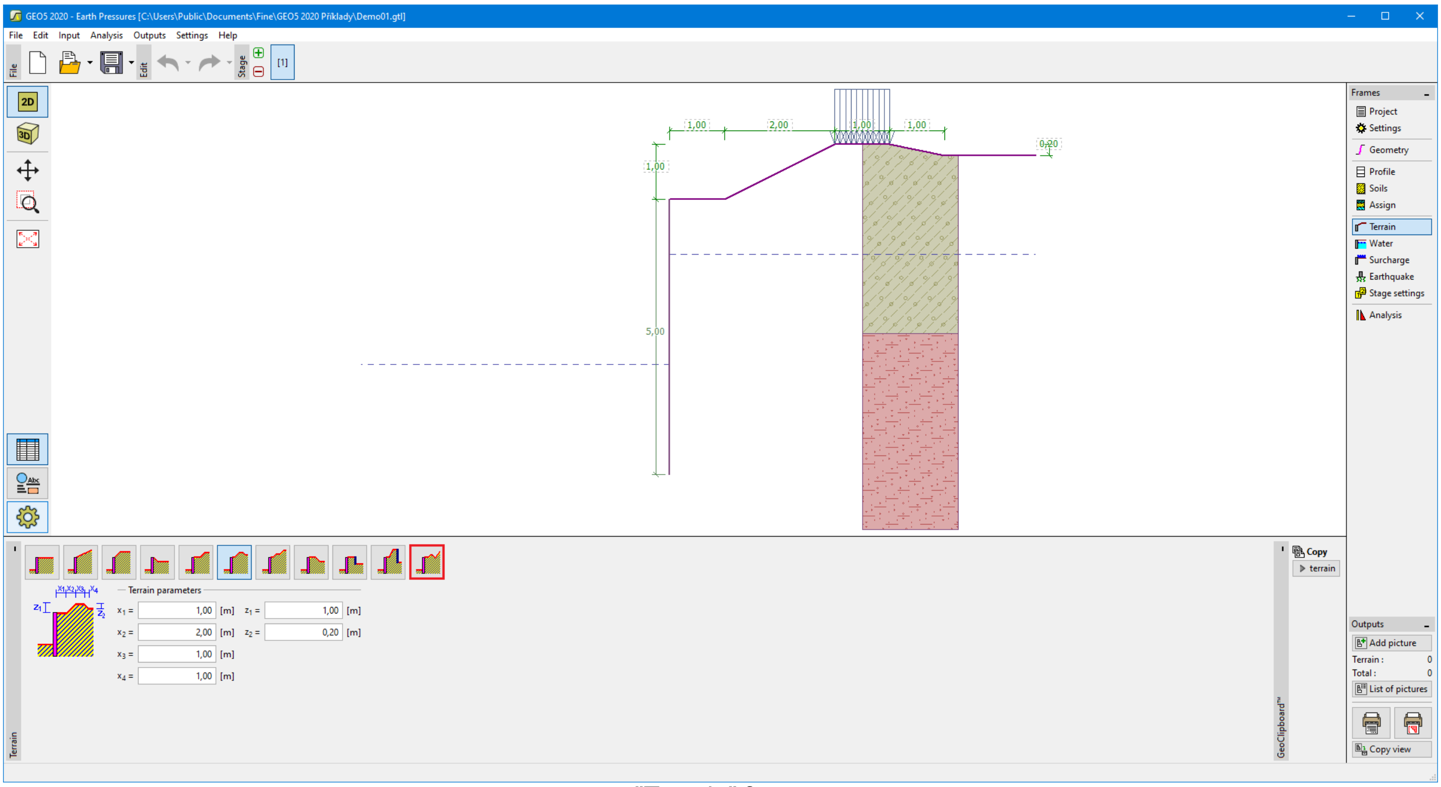
"Assign" frame

Terrain

The "**Terrain**" frame allows, by pressing the button, to specify the terrain shape. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop by using active dimensions.

The last option in the menu is a general shape of a terrain. In this case, the frame contains a table with a list of terrain points. The first point with coordinates [0; 0] coincides with the top point of a structure.

The analysis of earth pressures in case of an inclined terrain is described in "Distribution of earth pressures for broken terrain", a theoretical part of the help.



"Terrain" frame

Water

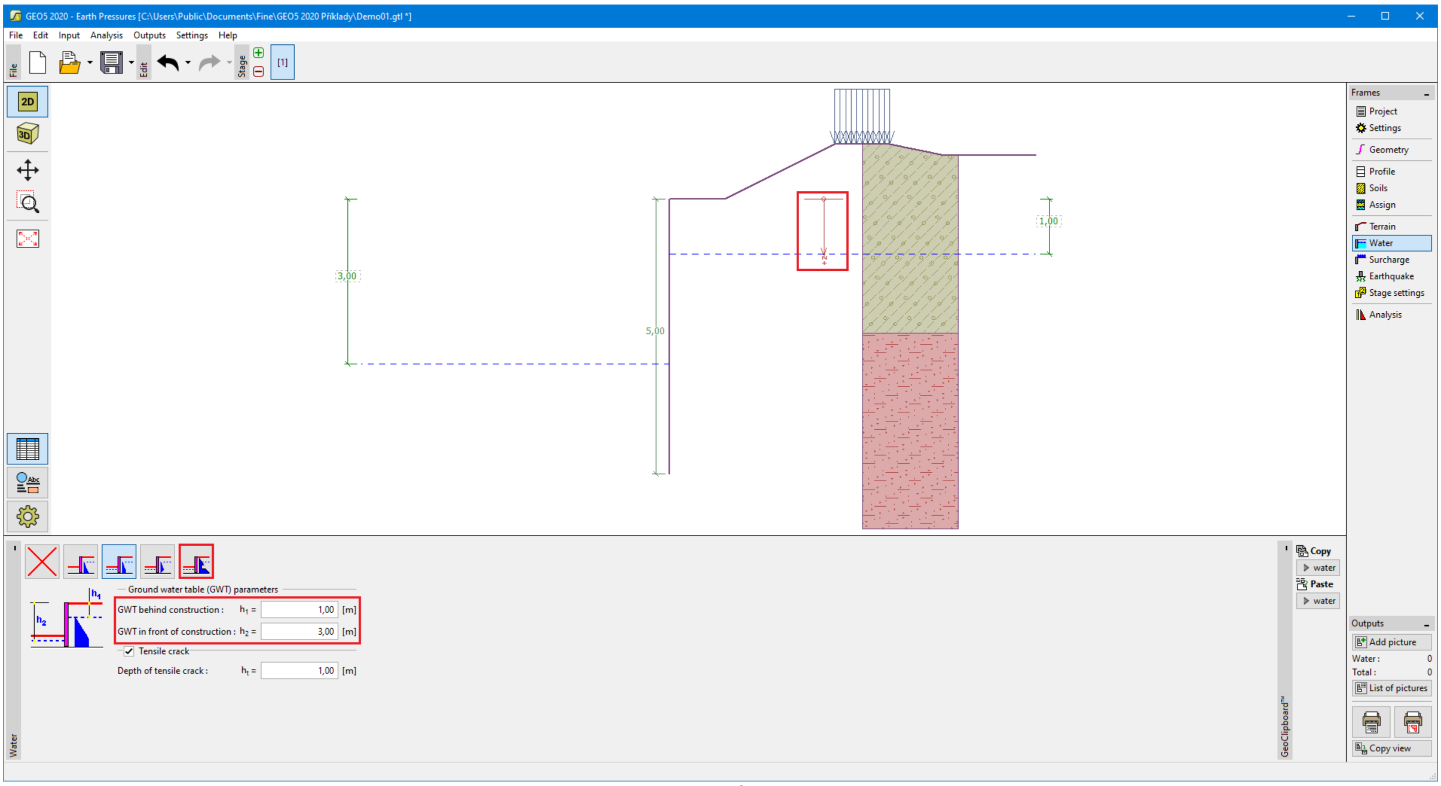
The "**Water**" frame allows, by pressing the button, to select the type of water. The selected type with a graphic hint of input values is displayed in the left part of the frame. Water parameters (h_1 , h_2 ...) can be edited either in the frame by inserting values into input fields, or on the desktop by using active dimensions.

The last option is a manual input of pore pressure both in front and behind the structure. Two tabs, "**In front of structure**" and "**Behind structure**", appear with tables. The table contains the values of pore pressure in front of the structure or behind it.

The groundwater table can also be specified **above the structure** or earth profile, respectively - in such a case, the depth of water is input with a negative value.

The earth pressures analysis with the influence of water is described in the "Influence of water", a theoretical part of the help.

The program further allows us to specify a depth of tensile cracks filled with water.



"Water" frame

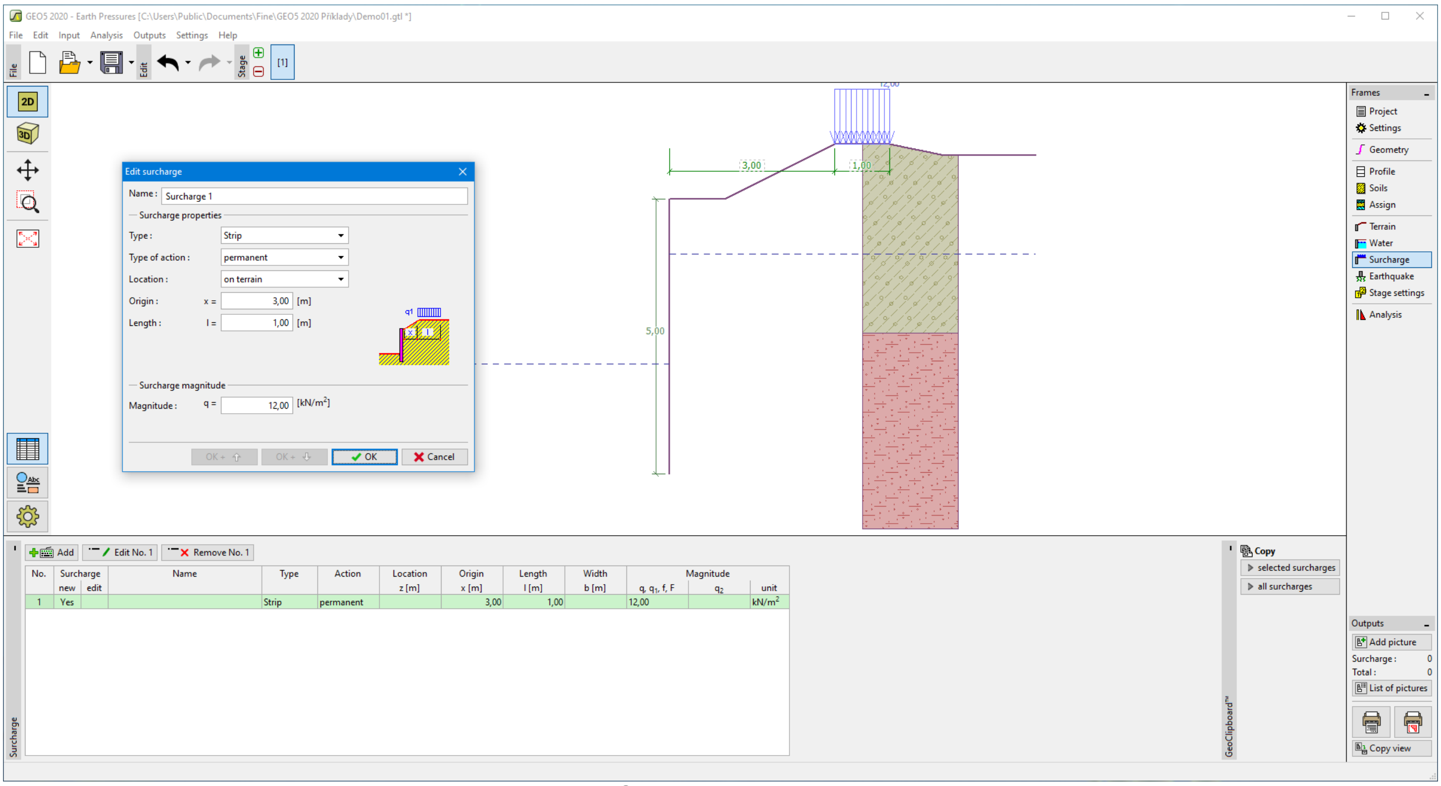
Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop using active dimensions or active objects.

The z -coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. When the surcharge is entered off the terrain, the program prompts an error message before calculation.

A surcharge can be specified as **permanent, variable, or accidental**. The resulting load action is multiplied by the corresponding design coefficient according to the type of surcharge. An accidental surcharge with a favorable effect is not considered in the analysis.

The surcharge effect on earth pressure analysis is described in the "Influence of surcharge", a theoretical part of the help.



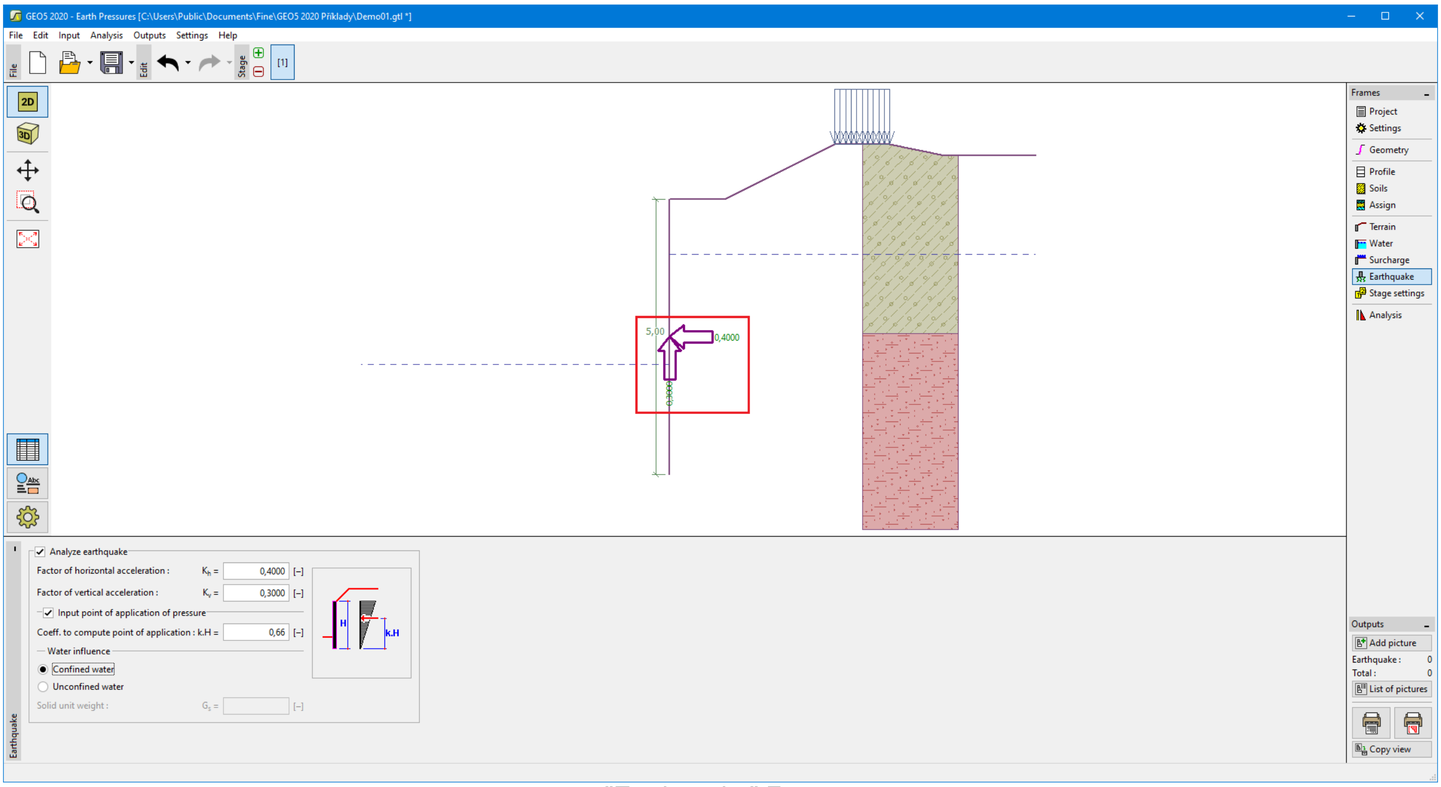
"Surcharge" Frame

Earthquake

The "**Earthquake**" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements, the coefficients k_h and k_v can be calculated according to EN 1998-5.

The analysis of earth pressures with earthquake influence is described in the "Influence of earthquake", a theoretical part of the help.



"Earthquake" Frame

Stage Settings

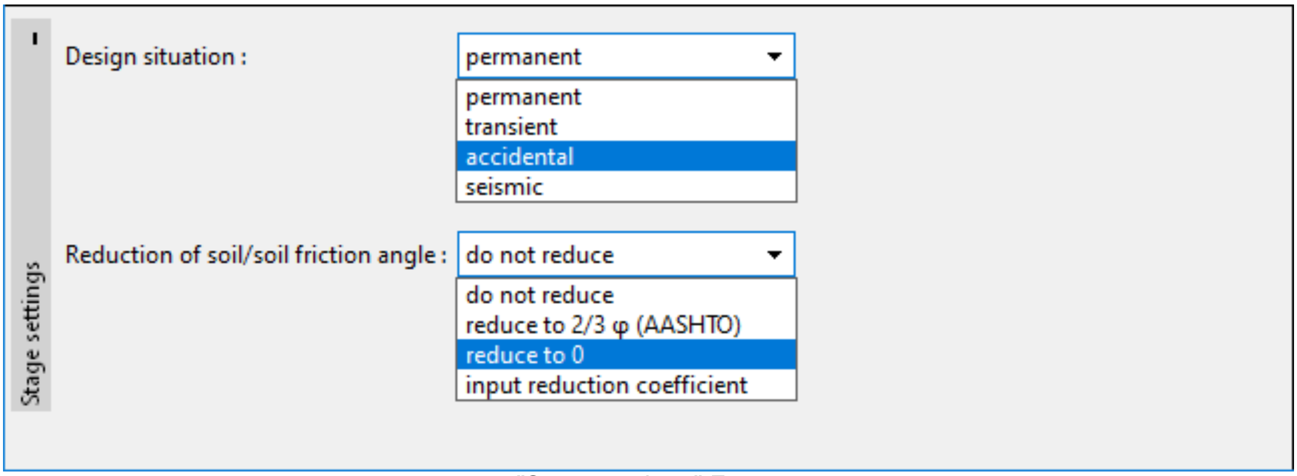
The "**Stage settings**" frame serves to input settings valid for a given construction stage.

The selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology.

The reduction of soil/soil friction angle can be considered in one of following ways:

- **do not reduce**
- **reduce to 2/3φ (AASHTO)**
- **reduce to 0**
- **input reduction coefficient**



"Stage settings" Frame

Analysis

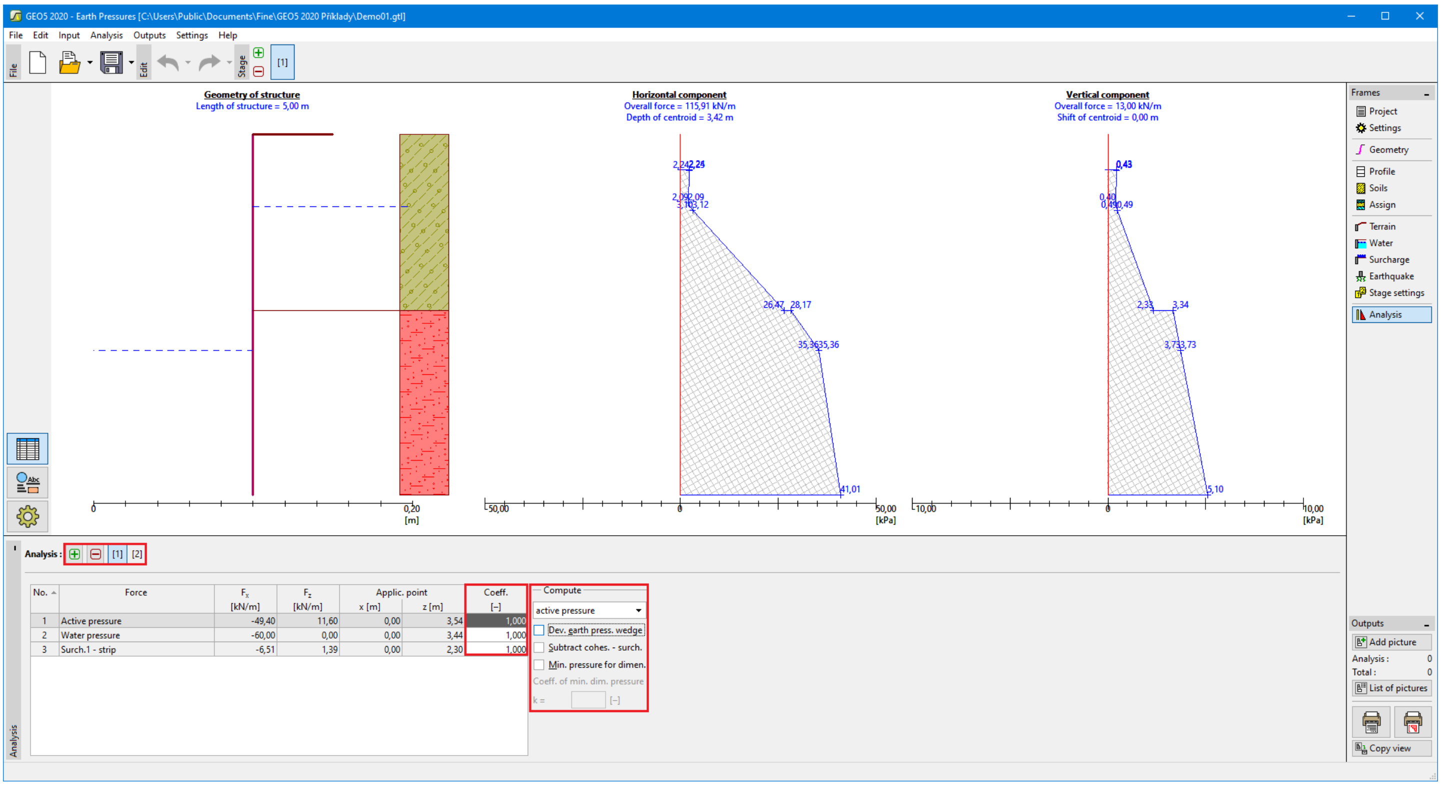
The "**Analysis**" frame shows the analysis results. Several computations can be carried out for a specified task.

The frame appearance is adjusted based on the selected verification methodology:

- Verification according to the factor of safety or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows for specifying whether the load acting on a structure is considered as the secondary one. This is explained in more detail in the "Load combinations" section. Providing the analysis is carried out according to "Design approach 1", it is necessary to enter the combination number in the right part of the window.
- Analysis according to LRFD - in this case, the last column is not displayed at all.

The "**Analysis**" frame displays the analysis results. The frame serves to select the type of computed earth pressure (active pressure, pressure at rest, passive pressure, increased active pressure). Two options, "Create soil wedge" and "Minimum dimensioning pressure", are available when computing the active earth pressure.

The analysis results are displayed on the desktop and are updated immediately for an arbitrary change in input data or setting. Visualization of results can be adjusted in the "Drawing Settings" frame.



"Analysis" Frame

Program Cantilever Wall

This program is used to verify the cantilever wall design. It offers a number of wall shapes and analyzes reinforced concrete cross-sections.

The help in the "Cantilever Wall" program includes the following topics:

- The input of data into individual frames:

Project	Settings	Geometry	Material	Profile	Soils	Assign
Foundation	Backfill	Terrain	Water	Surcharge	Front Face Resistance	Applied Forces
Earthquake	Base Anchorage	Stage Settings	Verification	Bearing Capacity	Dimensioning	Stability

- Standards and analysis methods

- Theory for analysis in the "**Cantilever Wall**" program:

Stress in Soil Body	Earth Pressures	Analysis of Walls	Analysis of Foundation Bearing Capacity	Dimensioning of Concrete Structures

- Outputs

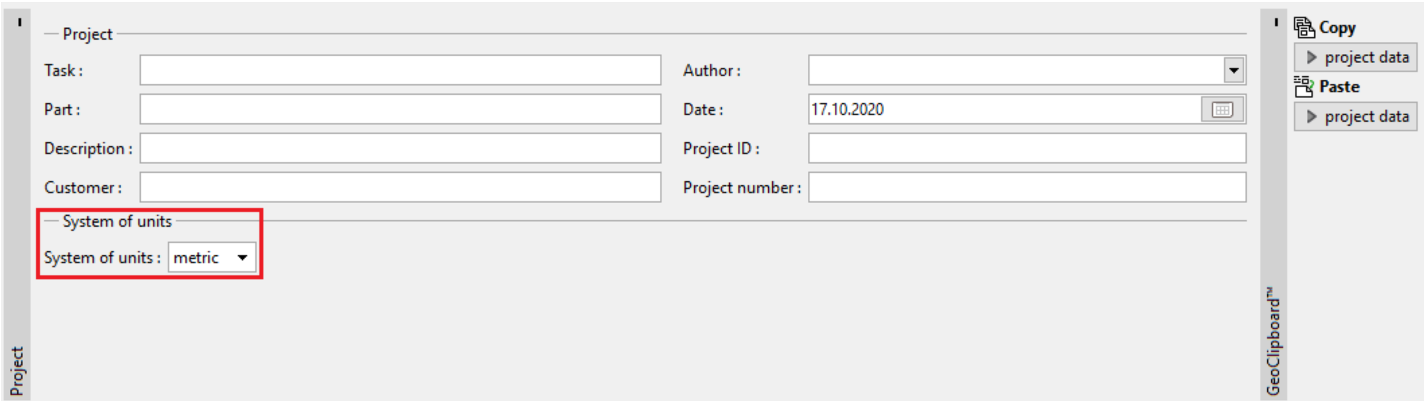
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The **"Project"** frame is used to input basic project data. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



"Project" frame

Settings

The "**Settings**" frame allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows us to view and modify an individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

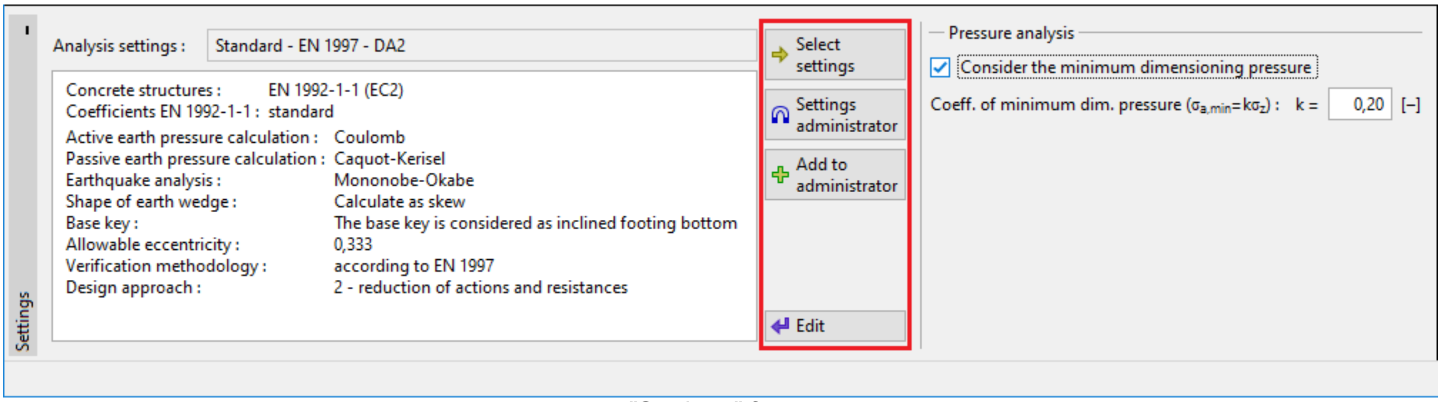
The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. The title changes to "**Input for the current task**" by modifying any of the parameters. Individual analyses are then performed with this **local setting**. If this setting is suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

The program allows us to specify a value of the minimum dimensioning pressure (by selecting the option "**Consider the minimum dimensioning pressure**").

Settings of analysis parameters are performed in the "Materials and standards" and "Wall analysis" tabs.



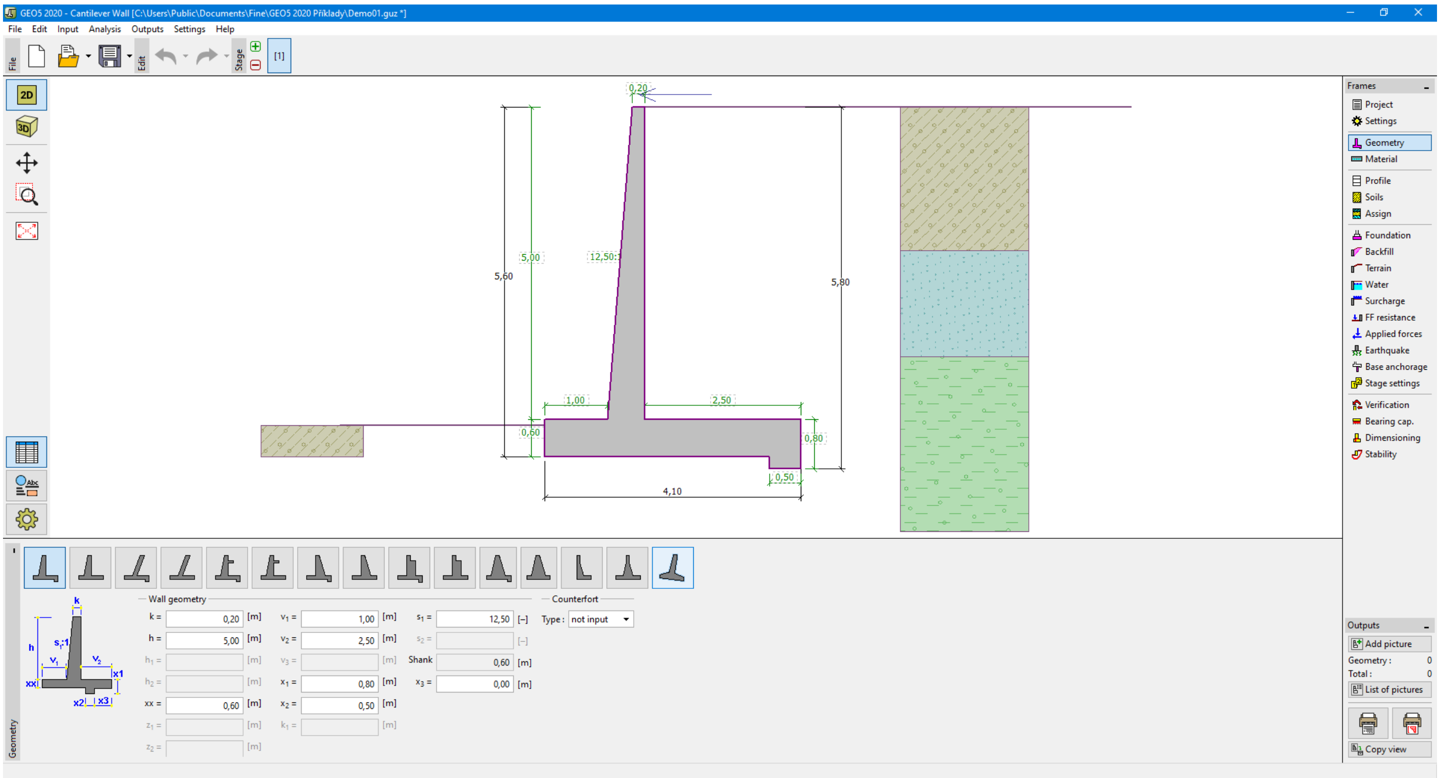
"Settings" frame

Geometry

The "**Geometry**" frame allows us to select a wall shape. The buttress or counterfort wall can be defined for some wall shapes. The selected shape with a graphic hint appears in the left part of the frame. The shape of a wall can be edited either in the frame by inserting values into input fields, or on the desktop by using active dimensions.

In case the structure is composed of inclined segments, it is required to enter the ratio of sides of an inclined 1:x segment. **The straight structure** is specified by entering the value zero.

The program makes it possible to export the geometry of a structure in the *.DXF format.



"Geometry" frame

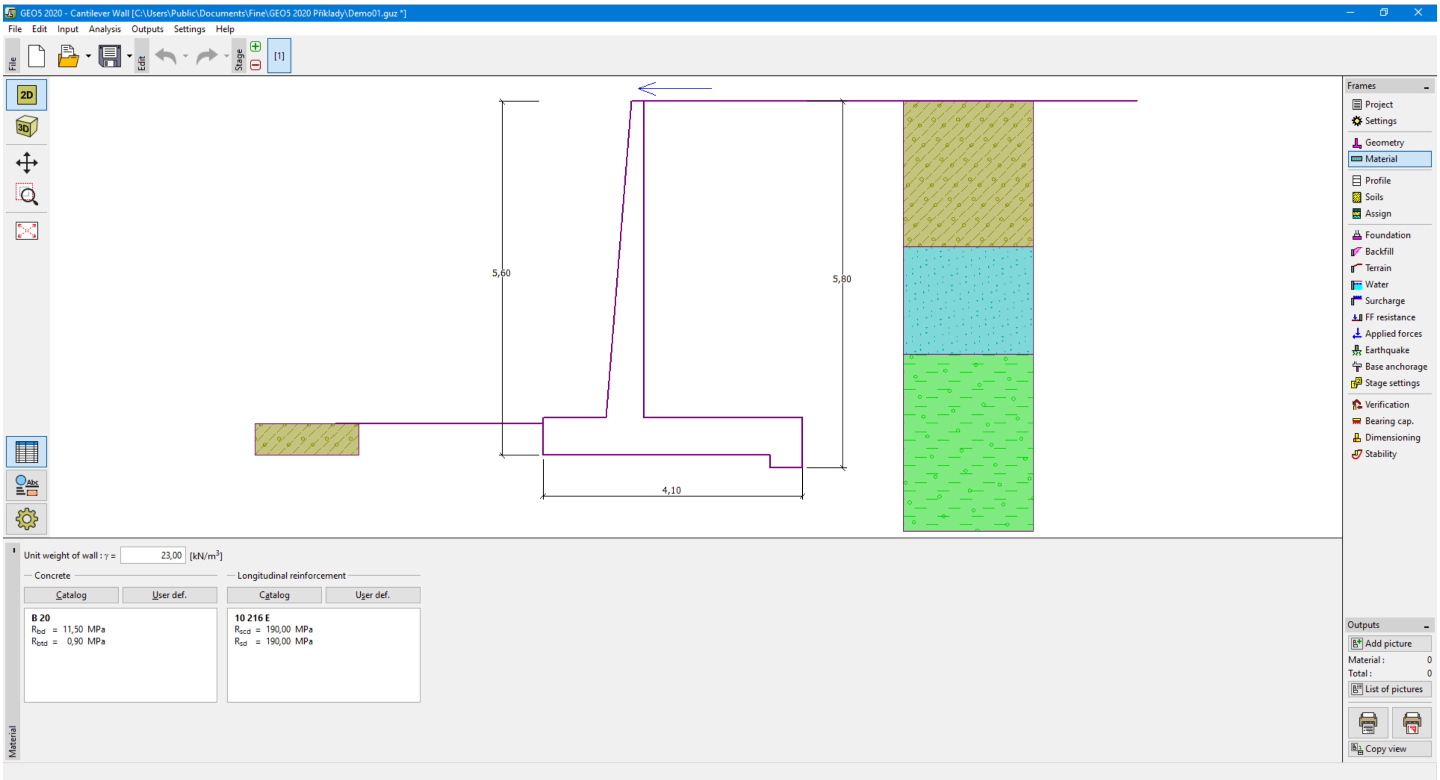
Material

The **"Material"** frame allows us to enter material parameters. The input field in the upper part of the frame serves to specify the **structure unit weight**.

Two options can be used for setting the parameters of the material:

- The "**Catalog**" button opens the "**Catalog of materials**" dialog window, where the required material can be selected.
- The "**User-defined**" button opens the "**Editor of material**", which allows us to input the specification of material parameters manually by the user.

The content of catalogs depends on the selection of relevant standard for the dimensioning of concrete structures in the "Materials and standards" tab.



"Material" frame

Profile

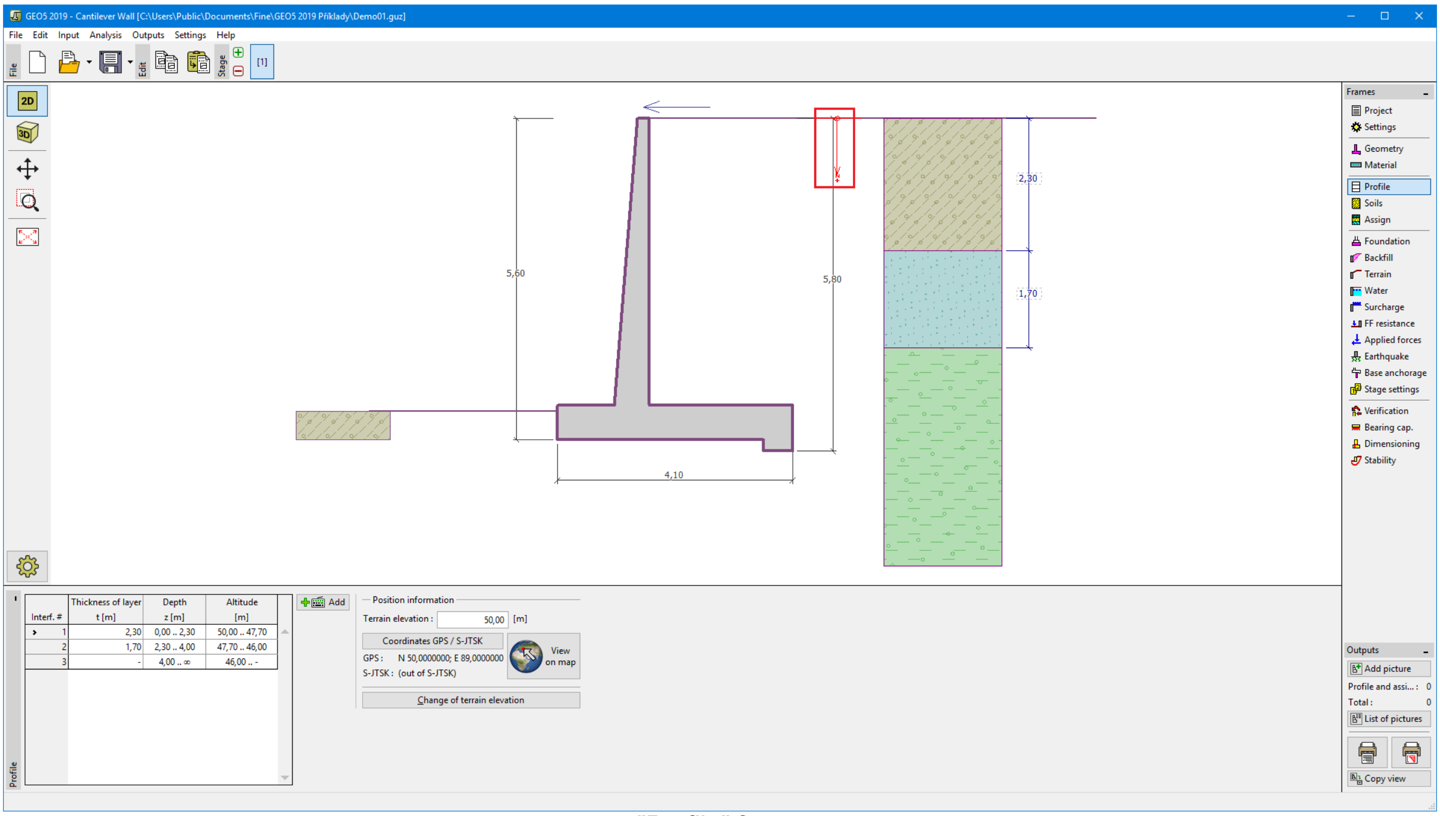
The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.



"Profile" frame

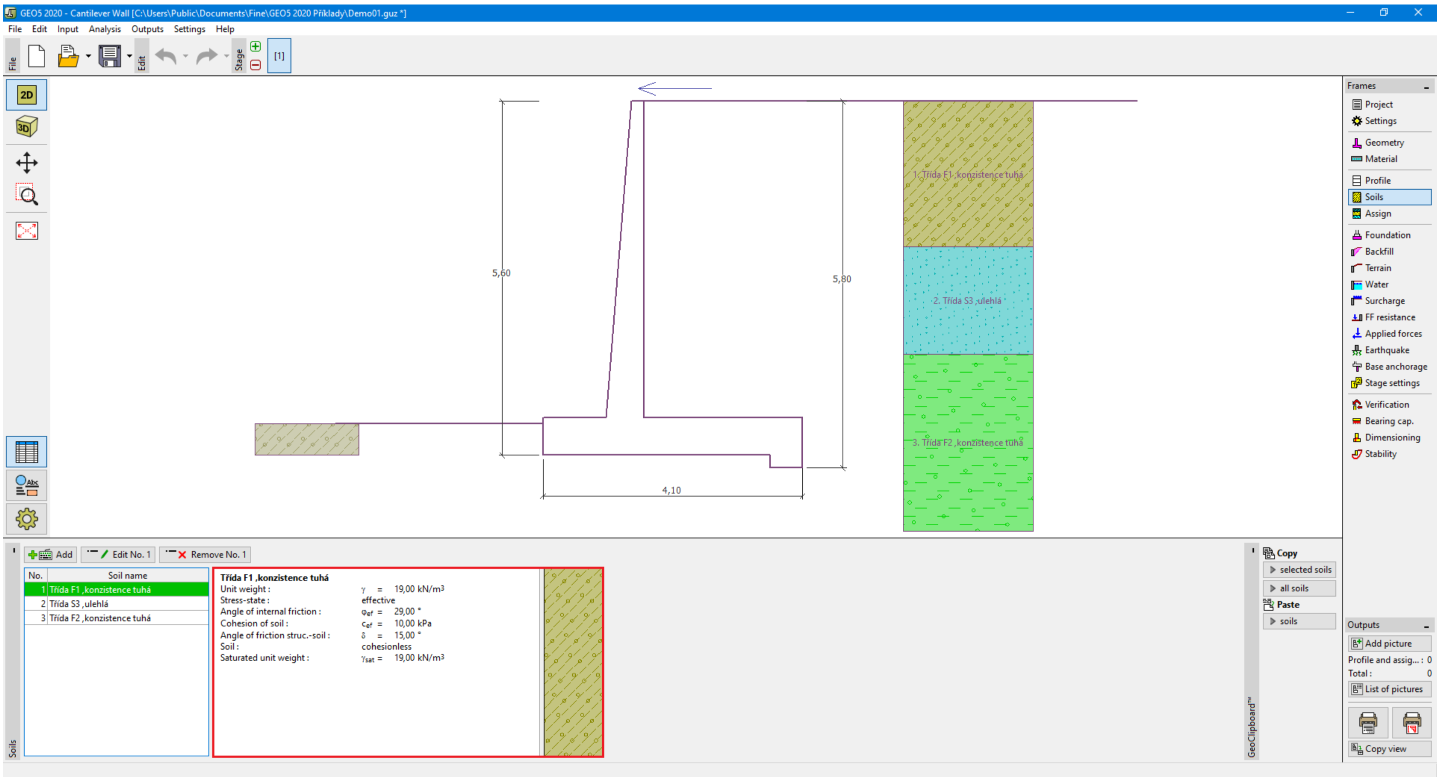
Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about a currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding a soil is performed in the "Add new soils" dialog window.

The soil characteristics entered in the program are further specified in the following chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

The data of input soils can be copied within all GEO5 programs using "GeoClipboard".



"Soils" Frame

Basic Data

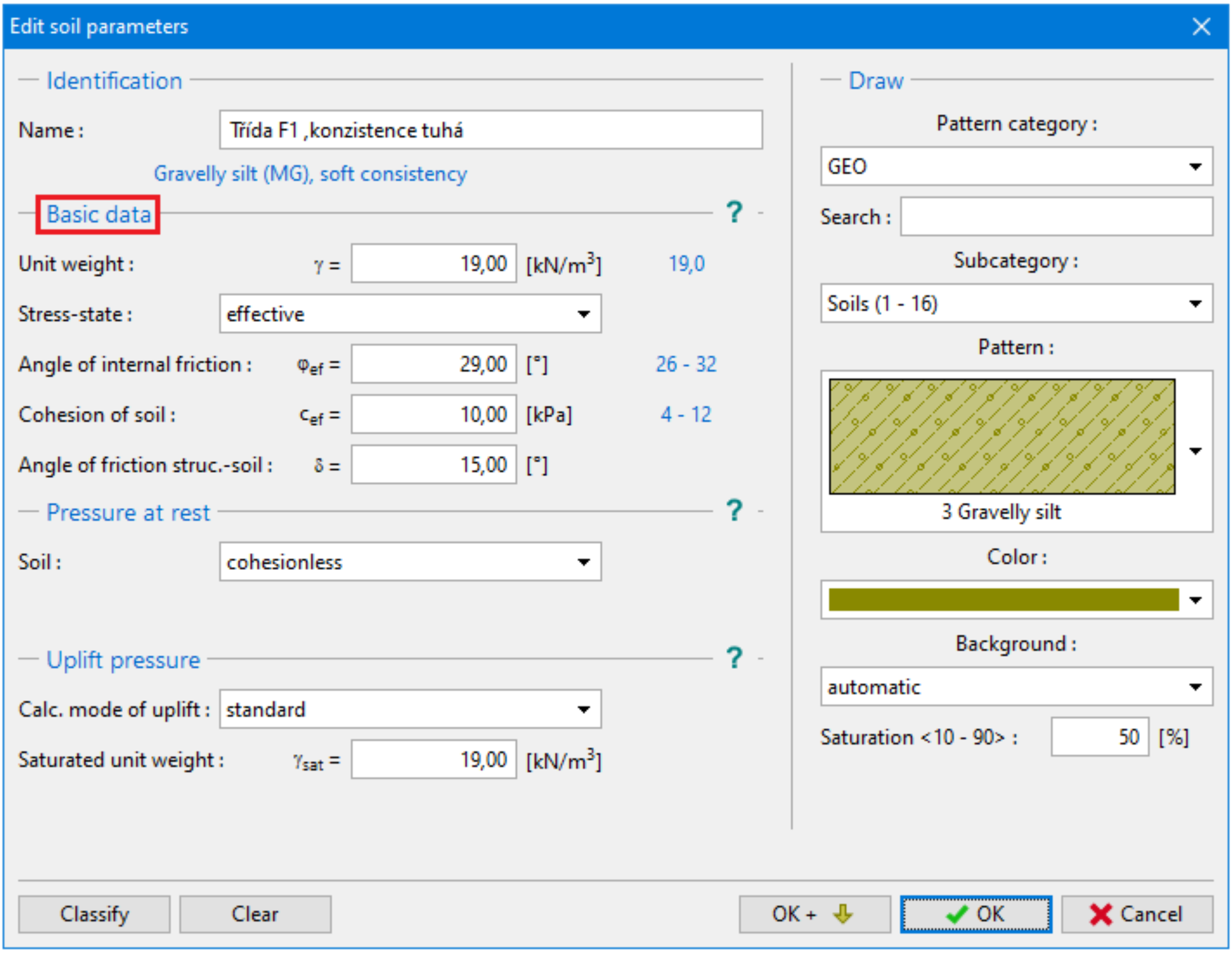
This part of the window allows us to introduce basic soil parameters - **unit weight, angle of internal friction, and cohesion**. The particular values are obtained from a geotechnical survey or from laboratory experiments. If this data is not available, it is possible to exploit the built-in database of soils, which contains values of selected soils characteristics. The rocks characteristics are not listed in the built-in database, these parameters must be defined manually.

Effective or total parameters of an internal friction angle and cohesion are specified depending on the setting in the "**Stress analysis**" combo list. Whether to use effective or total parameters, it depends primarily on the type of soil, type of load, structure duration, and water conditions.

For effective stress, specify the angle of internal friction between the soil and structure, which depends on the structure material and the type of soil. Possible values of this parameter are listed in the table of recommended values.

For total stress, specify the adhesion of soil to the structure face a .

The associated theory is described in detail in the "Earth pressures" chapter.

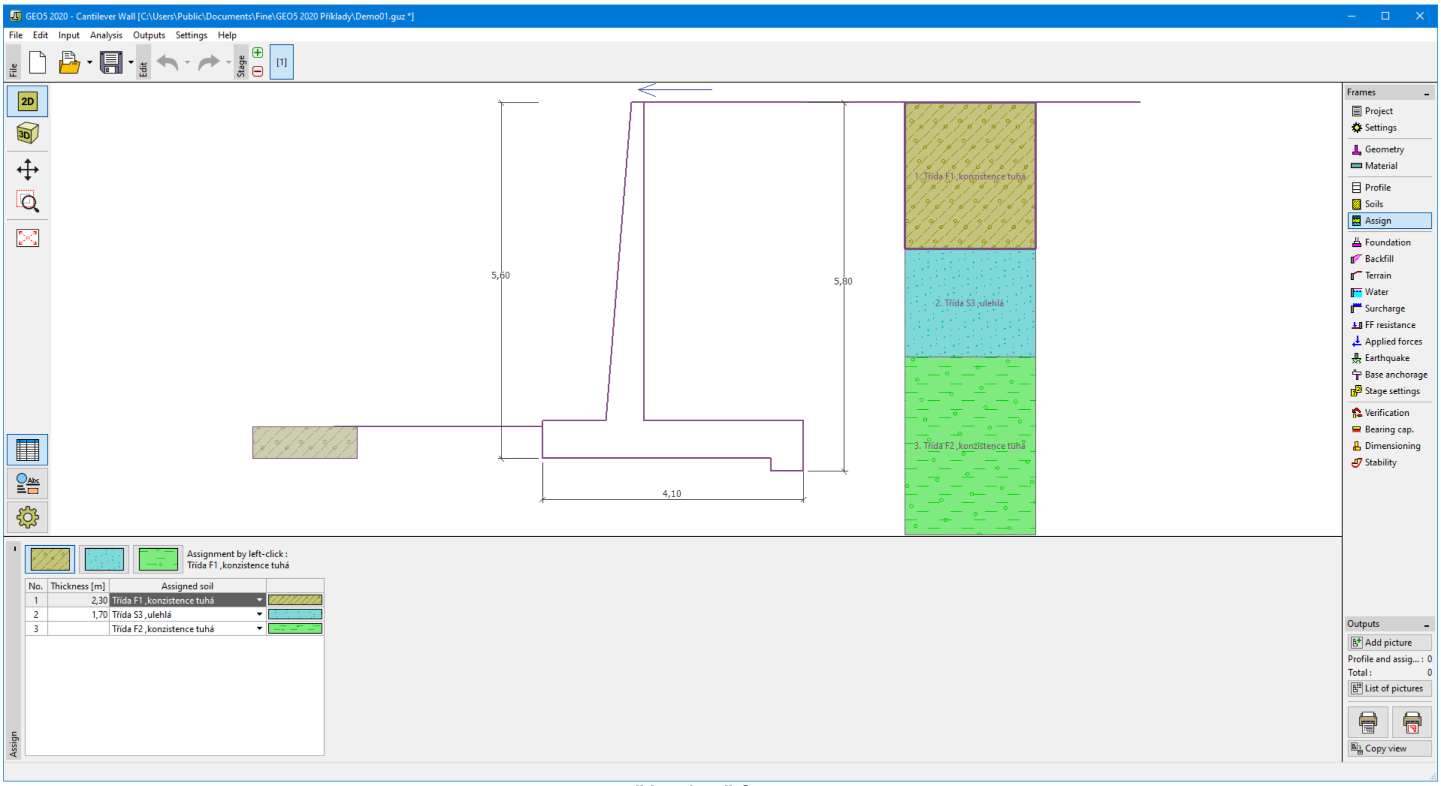


"Add new soils" - "Basic data" dialog window

Assign

The "**Assign**" frame contains a list of profile layers and assigned soils. The list of soils is graphically represented by using buttons in the bar above the table or is accessible from a combo list for each profile layer.

The procedure to assign a soil to a layer is described in detail herein.



"Assign" frame

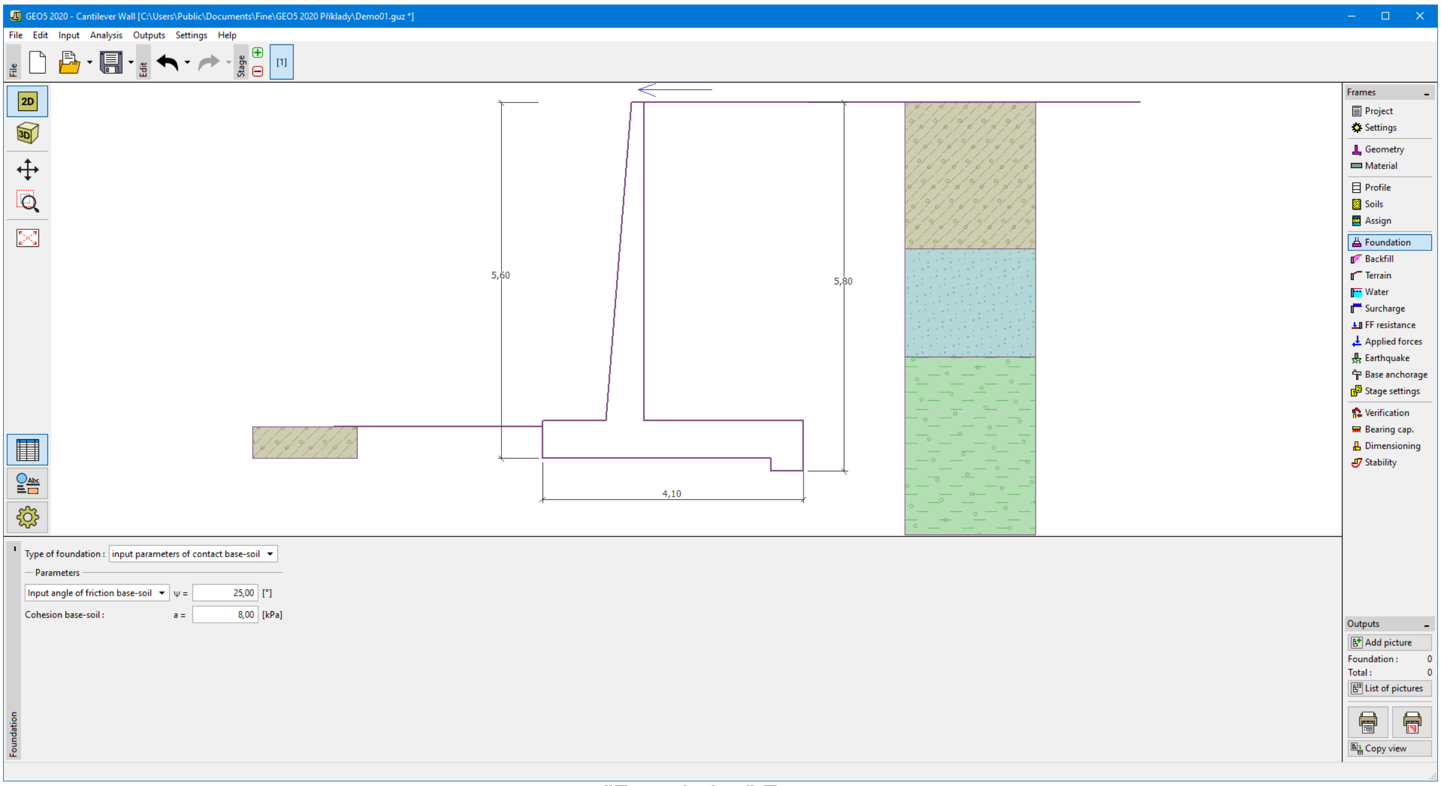
Foundation

The "**Foundation**" frame serves to specify the wall foundation type. The following wall foundation types are available:

- **soil from geological profile** - the wall is founded on the soil assigned from the geological profile specified in the "Profile" frame.
- **input parameters of contact base-soil** - parameters of the contact between the footing bottom and the structure are specified. The "**input angle of friction base-soil**" option requires inputting a friction angle ψ [°] between the foundation and the soil. The "**input friction coefficient**" option requires specifying the friction coefficient μ [-]. Both options require inputting the cohesion a [kPa] between the foundation (base) and the soil.
- **strip foundation** - strip foundation material is represented either by **soil** (input in "Soils" frame), or by concrete - requires inputting the **unit weight of foundation material** γ and **parameters of contact base-soil** (friction coefficient f, cohesion c, additional resistance F).
- **pile foundation** - the wall can be founded on one row of piles or two rows of piles, respectively.

The **Strip foundation** and the **pile foundation** can be adopted for the wall foundation only if the wall type with **straight footing bottom without jump** is selected in the "Geometry" frame. The wall foundation geometry (**strip foundation**, **pile foundation**) can be modified either in the frame by entering specific values into the inputting fields or on the desktop by using active dimensions.

The input data introduced in this frame influence the actual wall analysis (check for slip) and the bearing capacity of foundation soil.

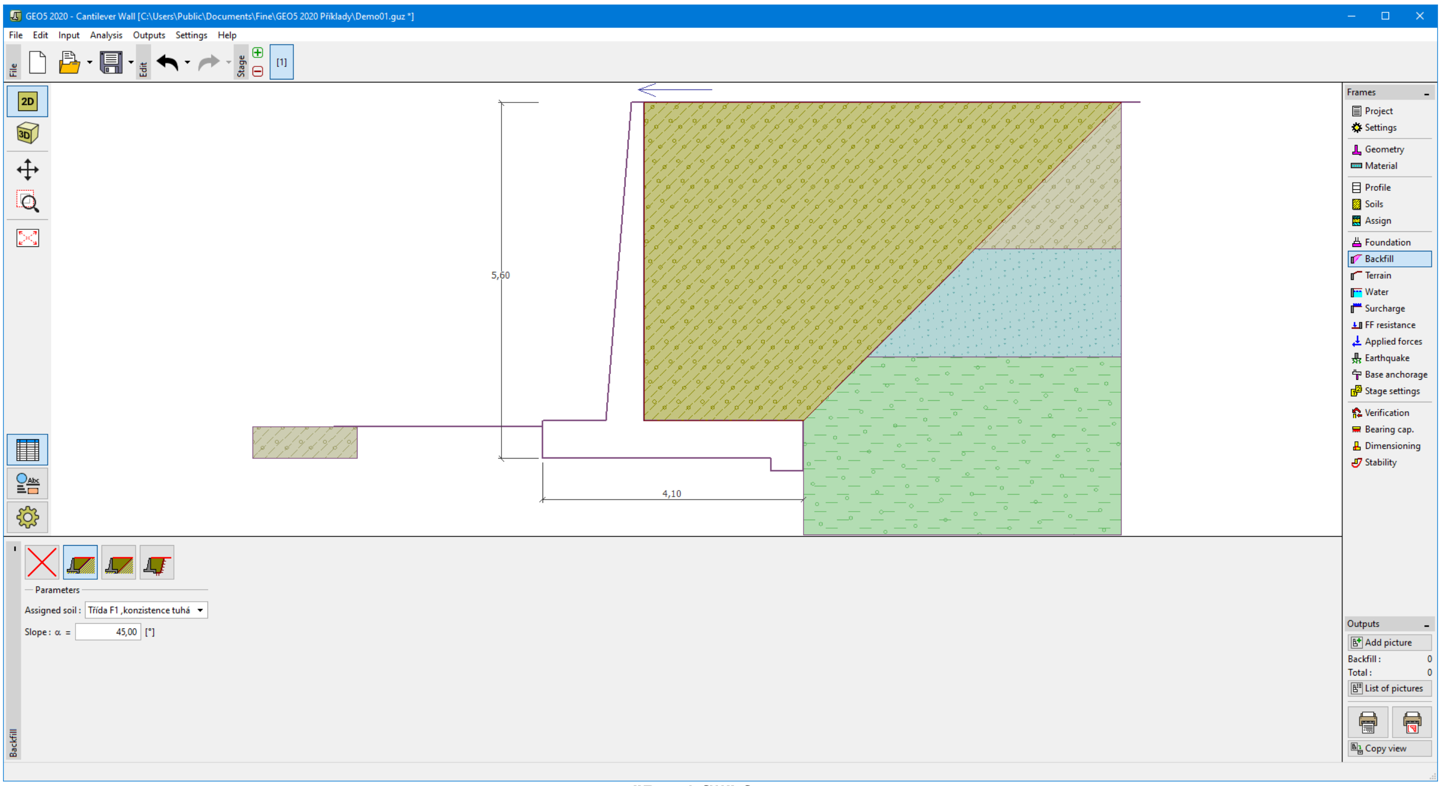


"Foundation" Frame

Backfill

The "**Backfill**" frame allows the selection of backfills behind the structure.

The analysis of earth pressures with the influence of the backfill is described in the theoretical part of the help: "Influence of Backfill".



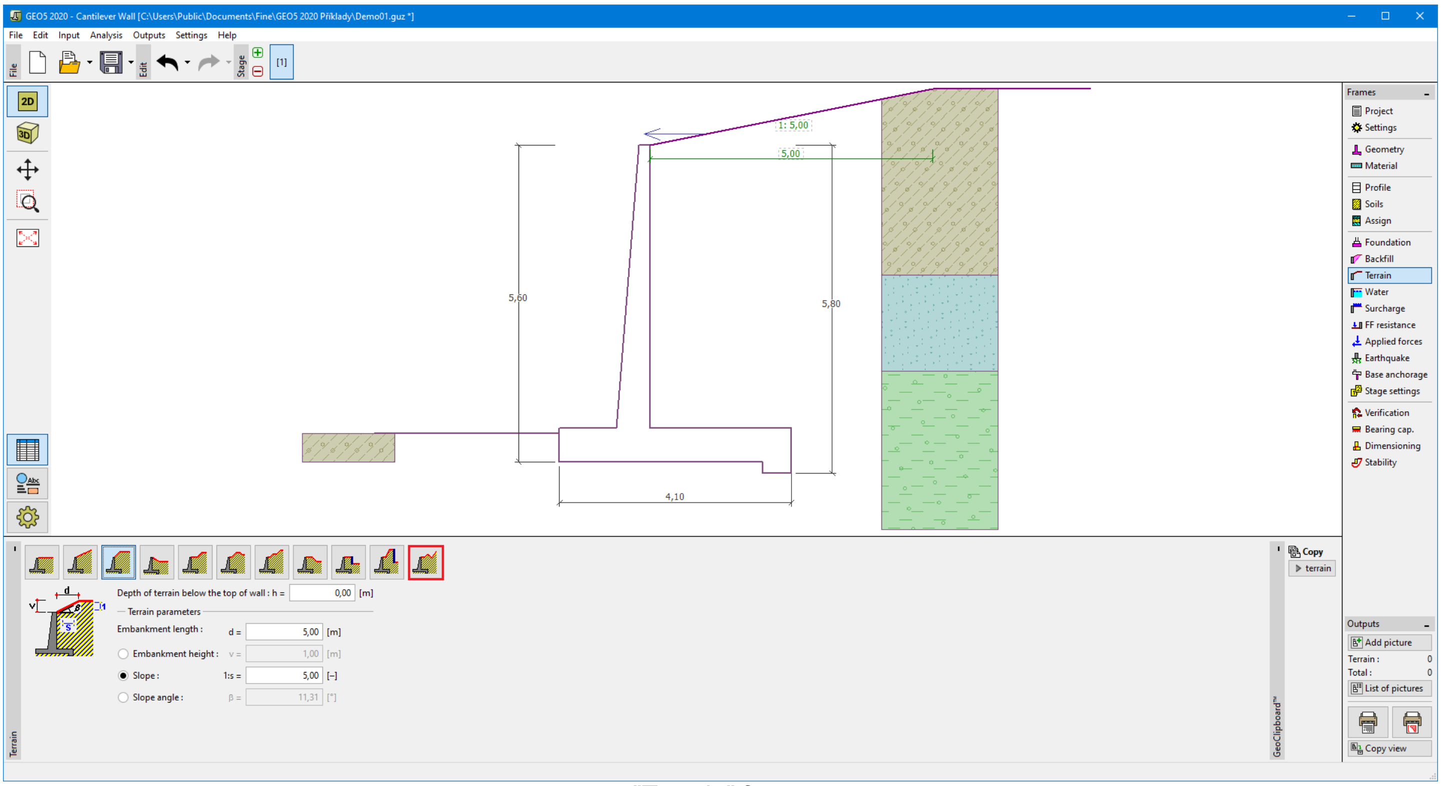
"Backfill" frame

Terrain

The "**Terrain**" frame allows, by pressing the button, to specify the terrain shape. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop by using active dimensions.

The last option in the menu is a general shape of a terrain. In this case, the frame contains a table with a list of terrain points. The first point with coordinates [0; 0] coincides with the top point of a structure.

The analysis of earth pressures in case of inclined terrain is described in "Distribution of earth pressures for broken terrain", a theoretical part of the help.



"Terrain" frame

Water

The "**Water**" frame allows, by pressing the button, to select a water type. The selected type with a graphic hint of input values is displayed in the left part of the frame. Water parameters (h_1 , h_2 ...) can be edited either in the frame by inserting values into input fields, or on the desktop by using active dimensions.

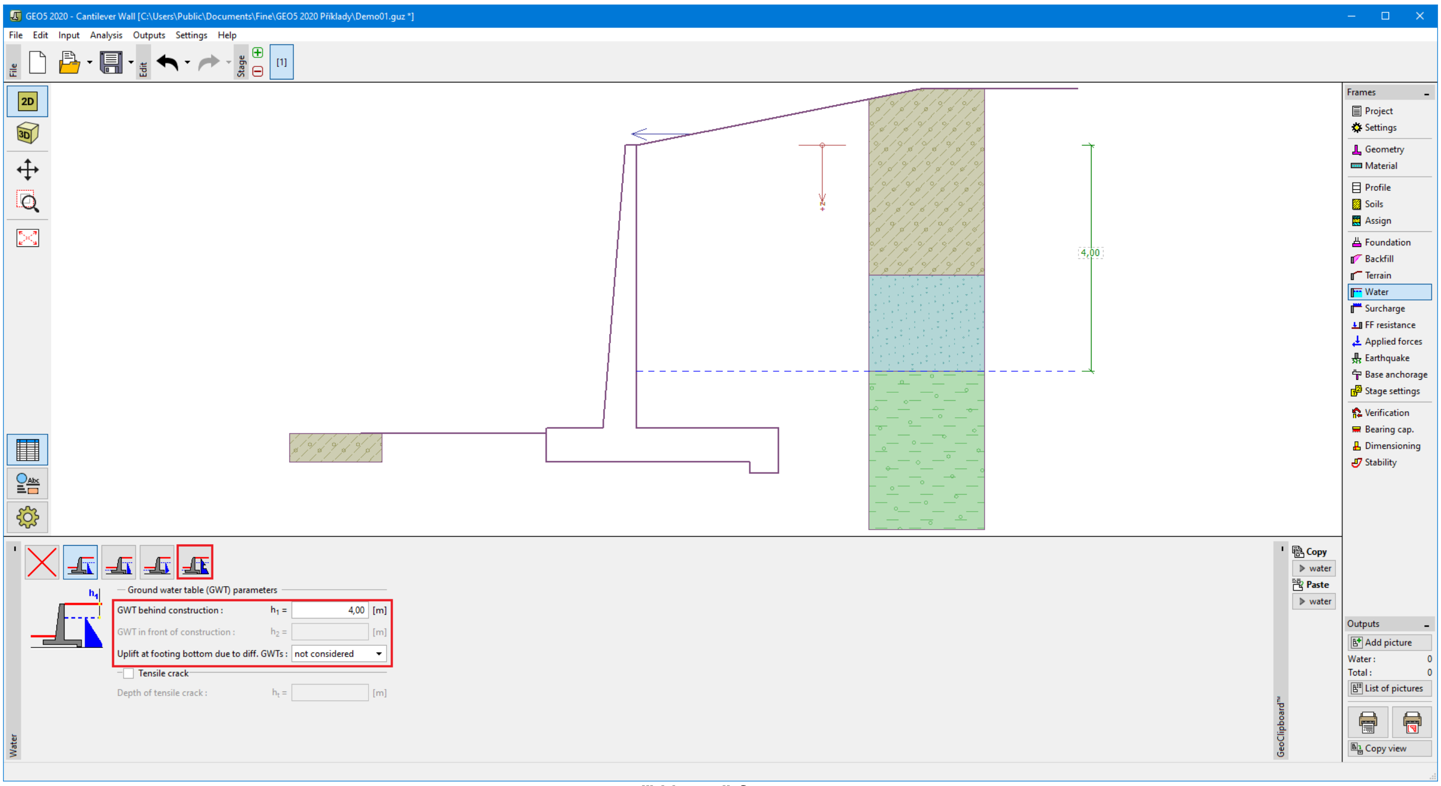
The combo list serves to specify whether the influence of water uplift pressure due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be linear, parabolic, or it may not be considered at all. When verifying the wall, the uplift pressure in the base of footing due to the different water tables is introduced as a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tabs, "**In front of structure**" and "**Behind structure**", appear with tables. The table is filled with values of pore pressure in front of the structure or behind it at a depth of "z " (z -axis).

The groundwater table can also be specified **above the structure** or earth profile, respectively - in such a case, the depth of water is input with a negative value.

The earth pressures analysis with the influence of water is described in the "Influence of water", a theoretical part of the help.

The program further allows us to specify a depth of tensile cracks filled with water.



"Water" frame

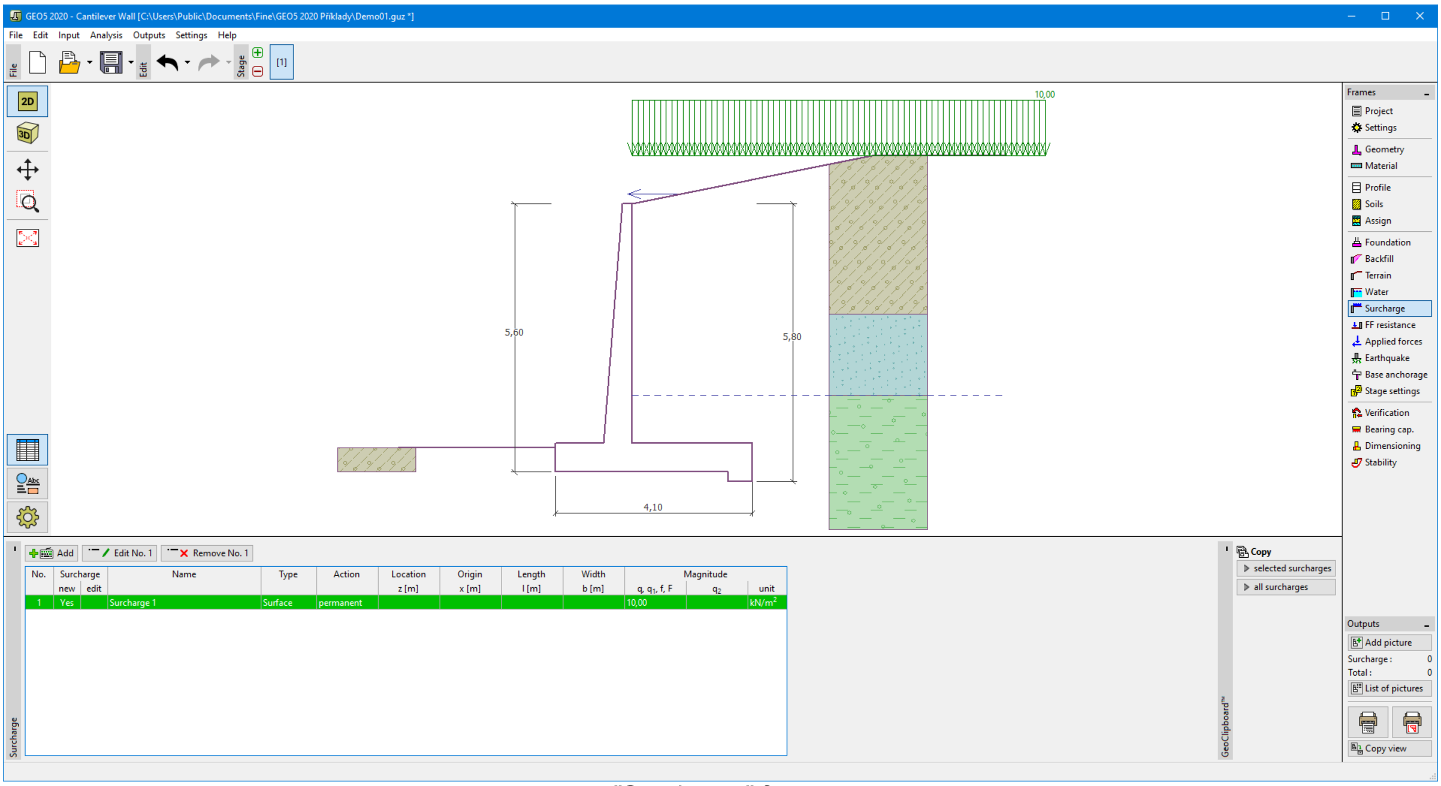
Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop using active dimensions or active objects.

The z -coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case the surcharge is found out of the terrain, the program prompts an error message before calculation.

A surcharge can be specified as **permanent**, **variable,** or **accidental**. The resulting load action is multiplied by the corresponding design coefficient according to the type of surcharge. An accidental surcharge with a favorable effect is not considered in the analysis.

The surcharge effect on the earth pressures analysis is described in the "Influence of surcharge", a theoretical part of the help.



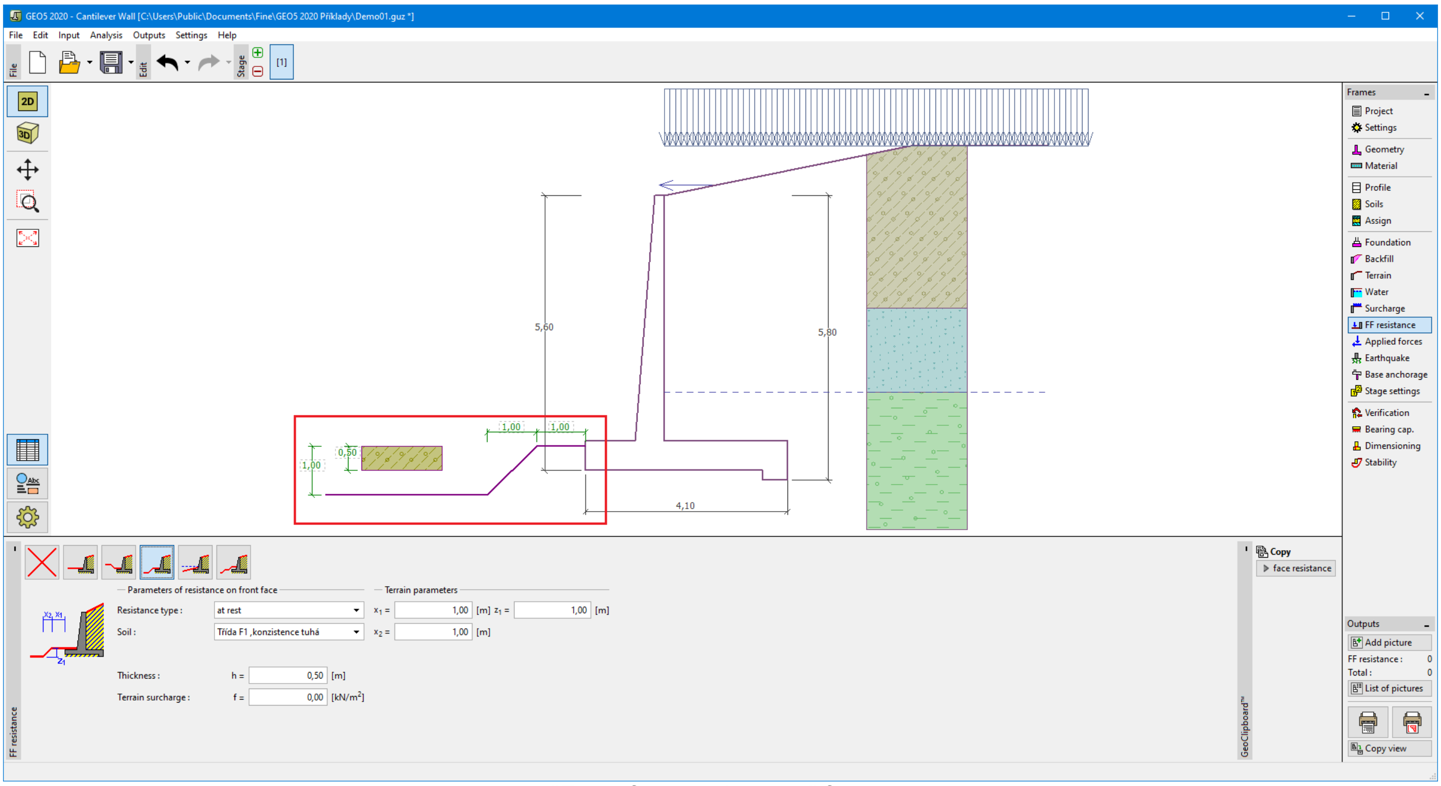
"Surcharge" frame

Front Face Resistance

The "**Front face resistance**" frame allows us to specify the terrain shape in front of the structure and parameters of front face resistance. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop using active dimensions.

Combo lists in the frame allow selecting the type of resistance and a soil (from the soils introduced in the "Soils" frame). The terrain surcharge magnitude and soil thickness (above the wall lowest points) can also be specified.

The resistance on a structure front face can be specified as the pressure at rest, passive pressure, or reduced passive pressure. The resulting force due to the reduced passive pressure is calculated as a resultant force caused by pressure at rest and passive pressure multiplied by corresponding coefficients.



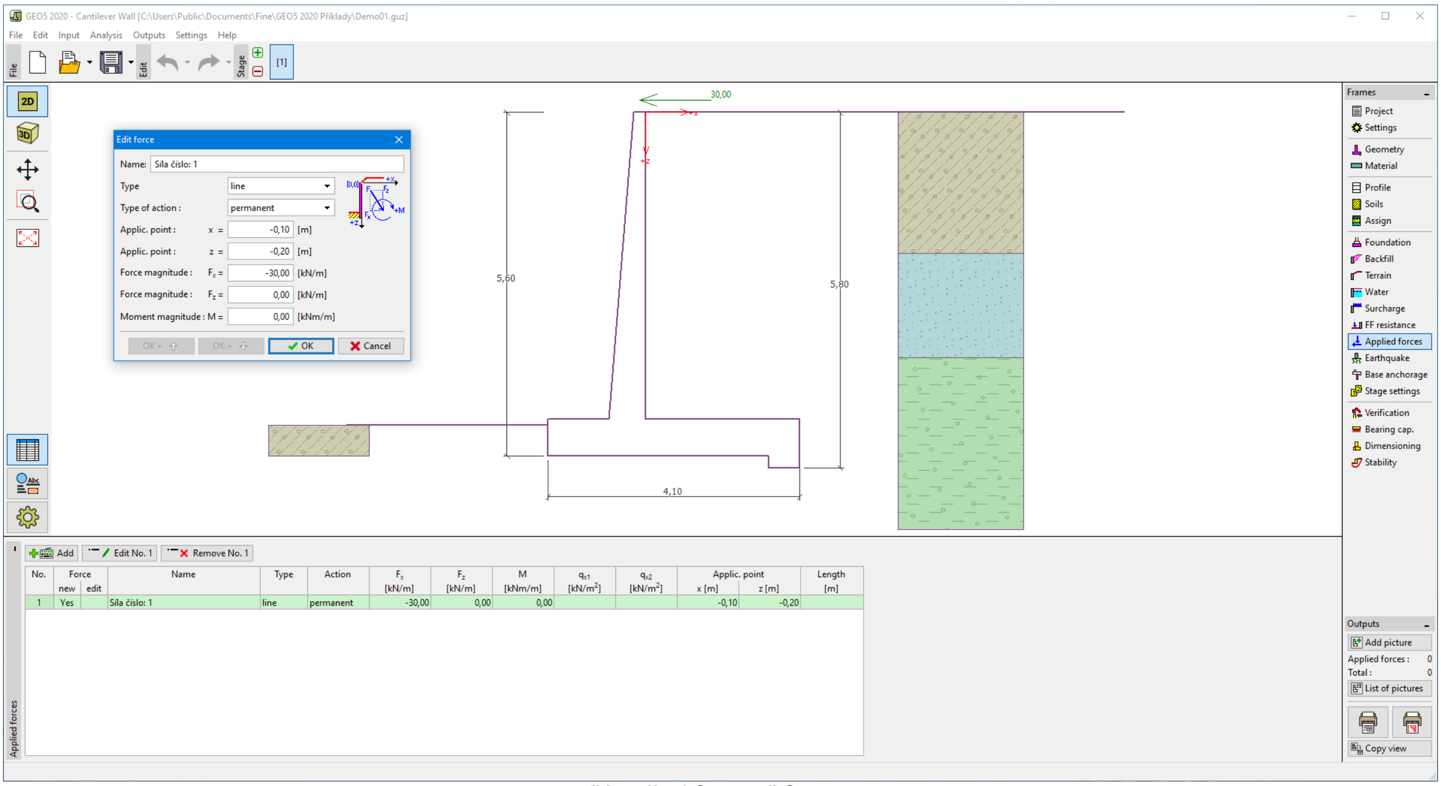
"Front face resistance" frame

Applied Forces

The "**Applied forces**" frame contains a table with a list of forces acting on a structure. Adding forces is performed in the "**New force**" dialog window. The input forces can also be edited on the desktop by using active objects.

Applied forces represent an additional load on the wall structure, sheeting, or an MSE wall. By using them, we can model e.g. an anchoring crash barrier, a crash vehicle, load from billboards and hoardings etc. The program doesn`t adjust applied forces in the calculation in any way except multiplying them with corresponding coefficients according to the selected load type (EN1997, LRFD).

It is necessary to define external load acting on the ground surface as a surcharge.



"Applied forces" frame

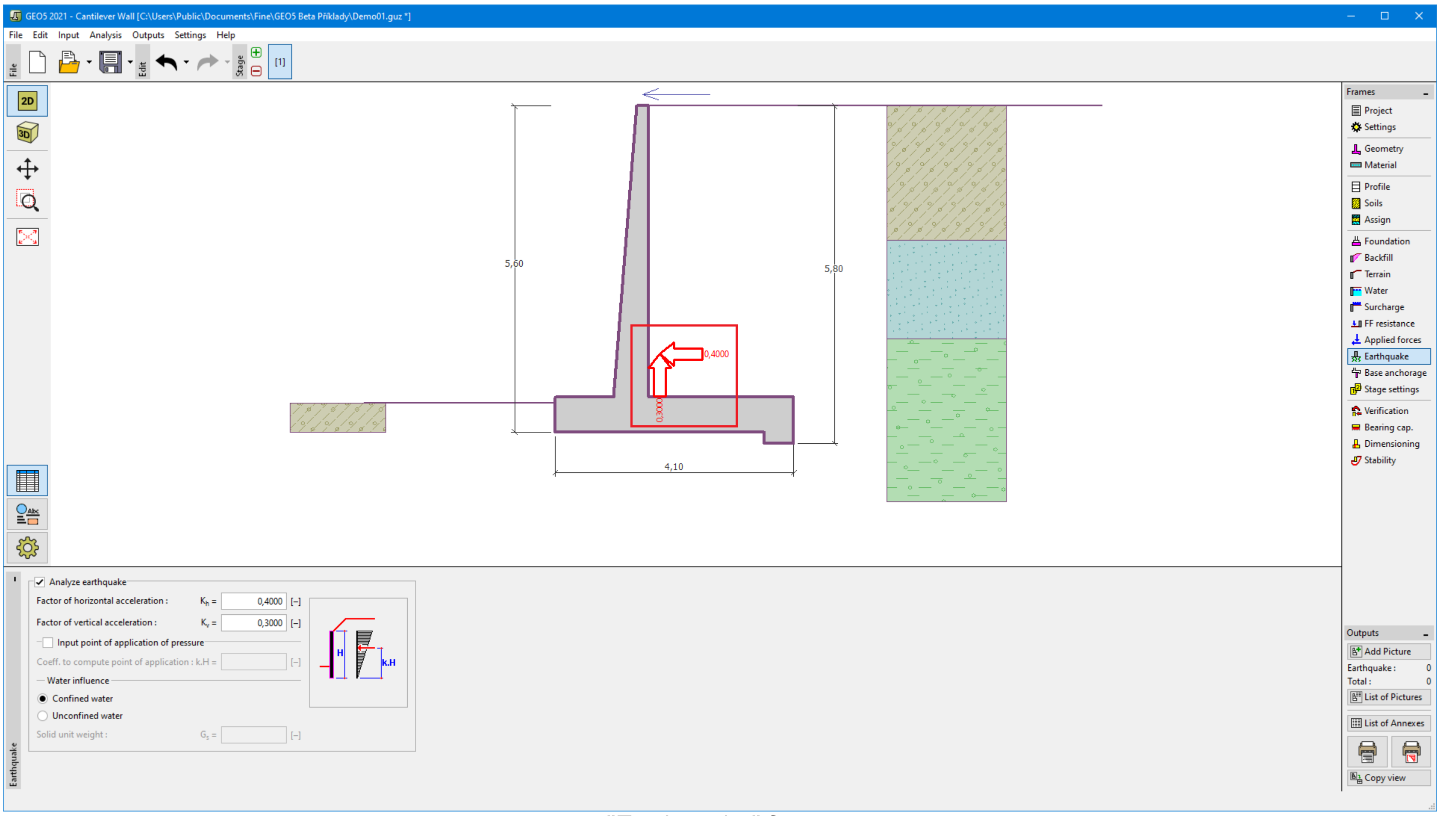
Earthquake

The "**Earthquake**" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements, the coefficients k_h and k_v can be calculated according to EN 1998-5.

The earth pressures analysis with earthquake influence is described in the "Influence earthquake", a theoretical part of the help.

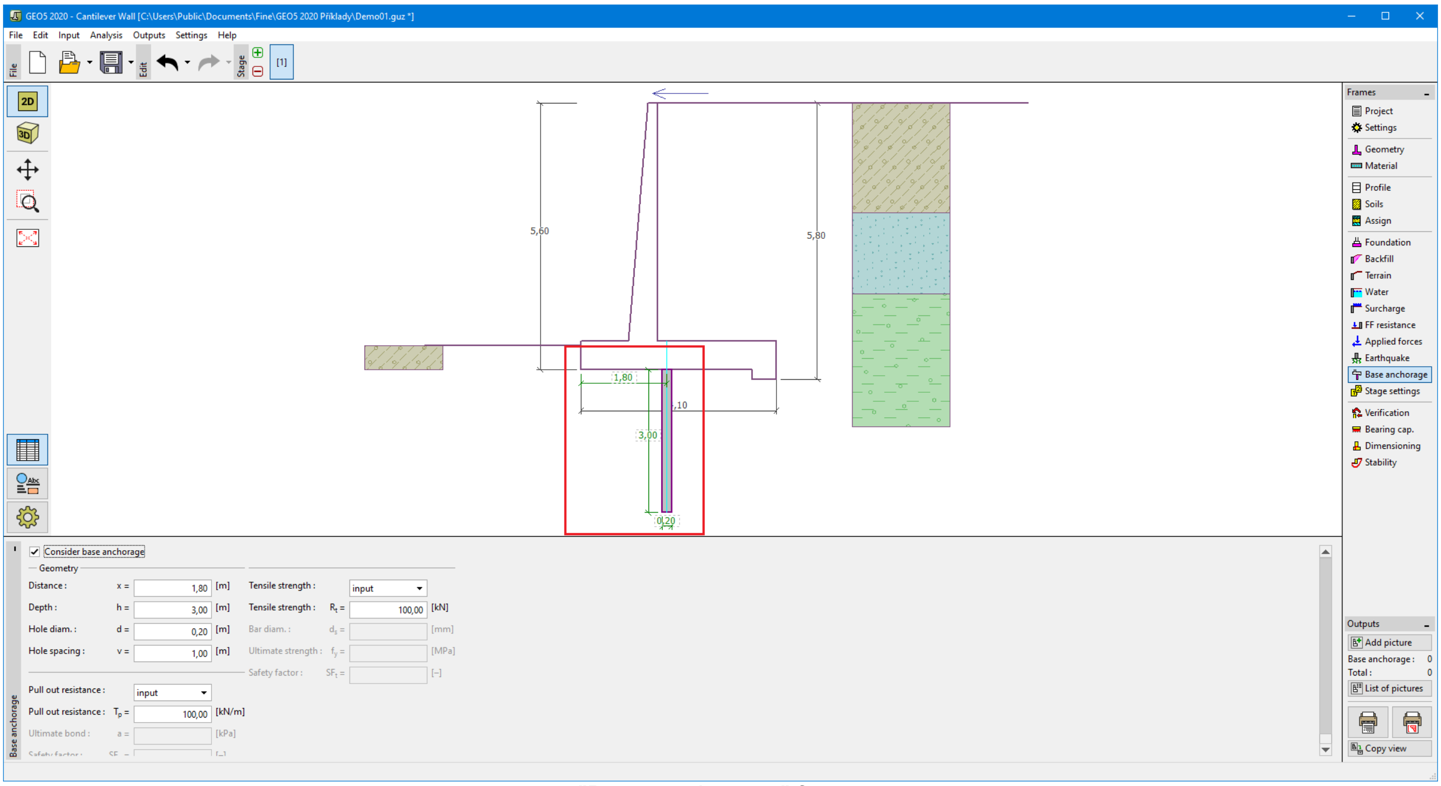
For the LRFD Verification methodology, it is possible to define coefficients for seismic combinations according to the AASHTO.



"Earthquake" frame

Base Anchorage

The "**Base anchorage**" frame serves to input parameters (anchorage geometry, bearing capacity against pulling-out and pulling-apart) specifying an anchorage of the wall foundation. The geometry of the footing anchorage can be edited either in the frame by inserting values into the inputting boxes or on the desktop by using active dimensions. The bearing capacity values can be either input or computed by the program from the input parameters.



"Base anchorage" frame

Stage Settings

The "**Stage settings**" frame serves to input settings valid for a given construction stage.

The selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

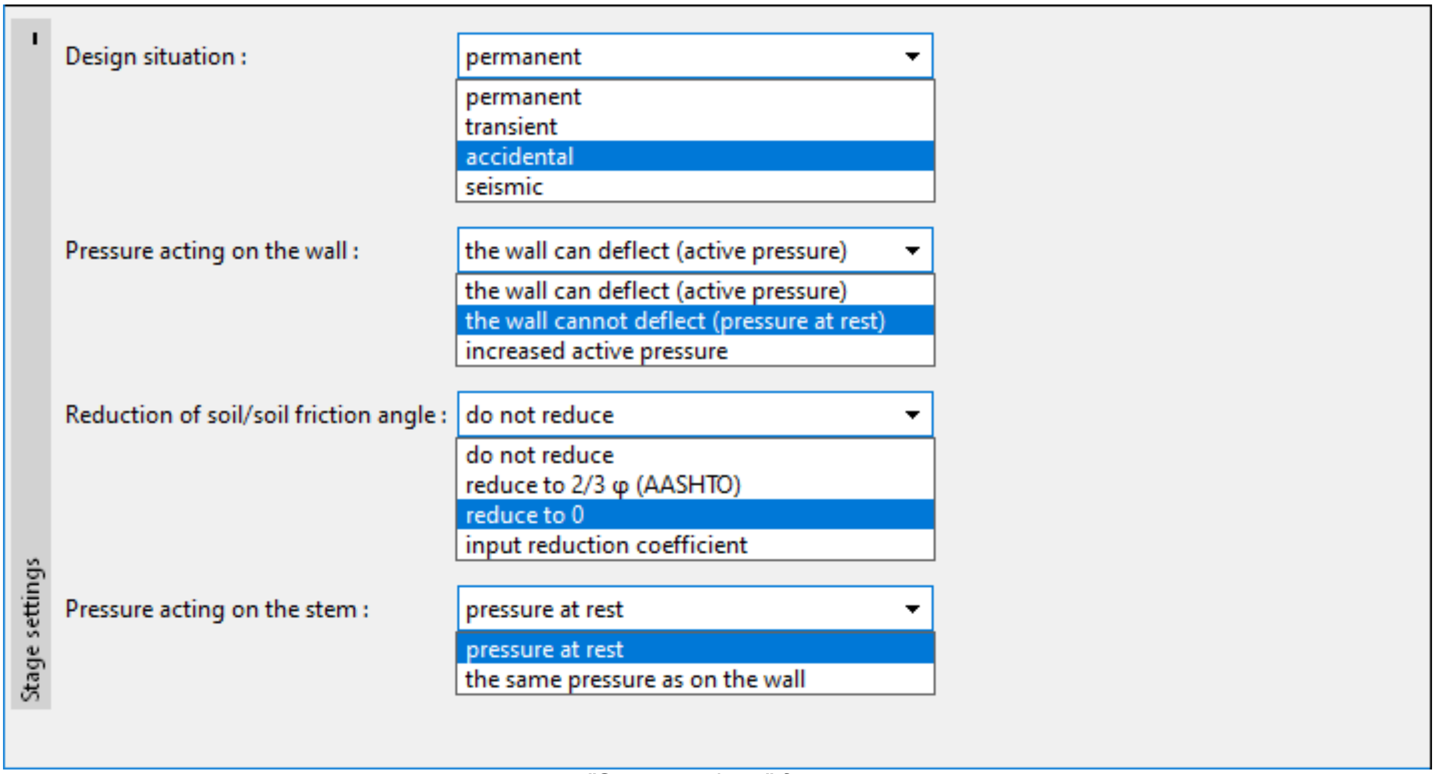
The frame view depends on the selected verification methodology.

Next, the frame serves to specify the pressure type acting on a wall based on the allowable wall deformation. Providing the wall is free to move, the active pressure is assumed, otherwise, the pressure at rest is used. The third option enables to load the wall by increased active pressure.

The reduction of soil/soil friction angle can be considered in one of following ways:

- **do not reduce**
- **reduce to 2/3φ (AASHTO)**
- **reduce to 0**
- **input reduction coefficient**

The wall stem is loaded either by pressure at rest or by the same pressure as that acting on the entire wall.



"Stage settings" frame

Verification

The "**Verification**" frame shows the analysis results. Several computations can be carried out for a single task.

The frame appearance is adjusted based on the selected verification methodology.

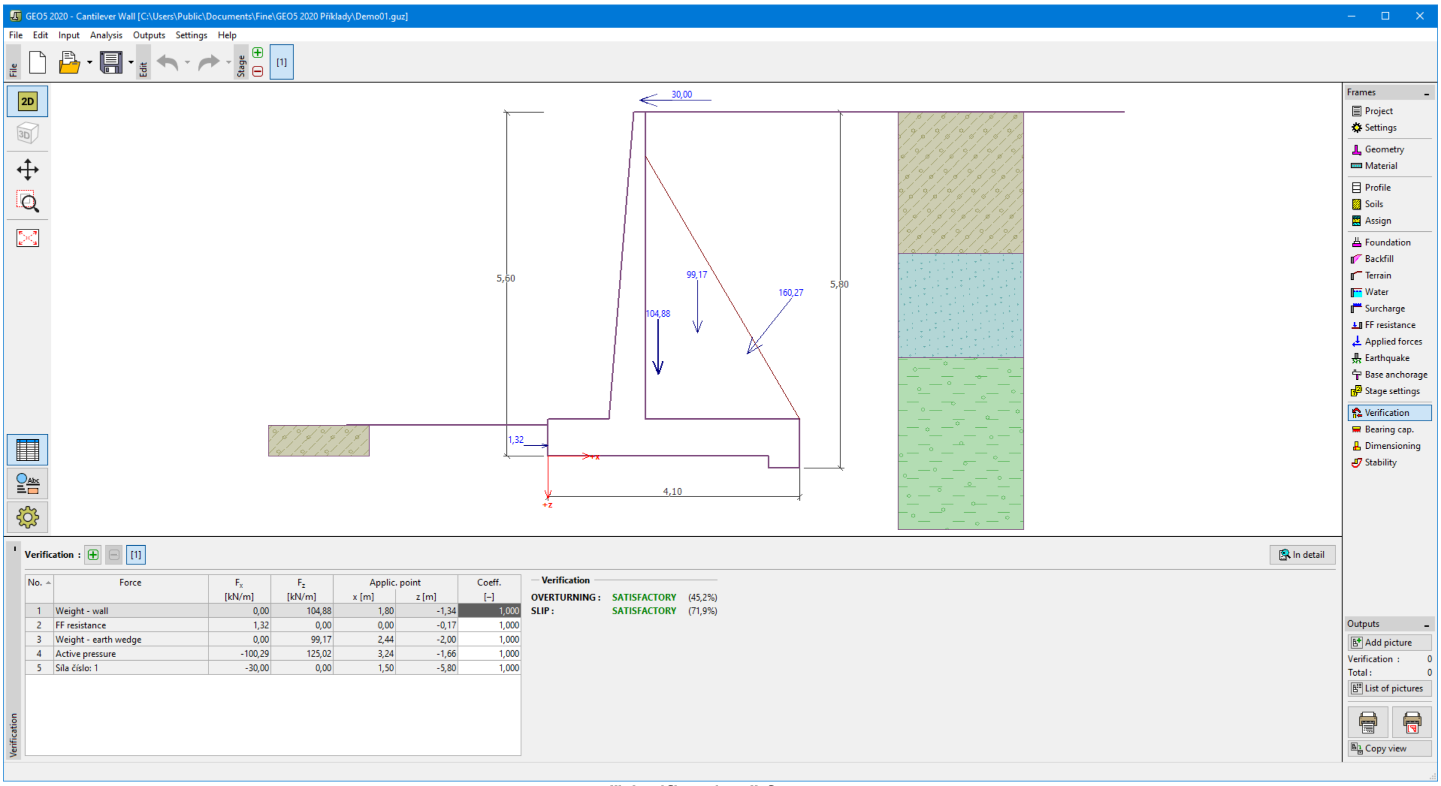
- Verification according to the factor of safety or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows us to specify whether the load acting on a structure is considered to be a secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in this case, the last column is not displayed at all.

The wall is loaded either by active pressure or pressure at rest depending on input in the "Stage settings" frame.

The procedure for wall verification is described in the theoretical part of the help.

The computed forces are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of wall verification against **overturning** and **translation**. The "**In detail**" button opens the dialog window, which contains a detailed listing of the verification analysis results.

Visualization of results can be adjusted in the "Drawing Settings" frame.



"Verification" frame

Bearing Capacity

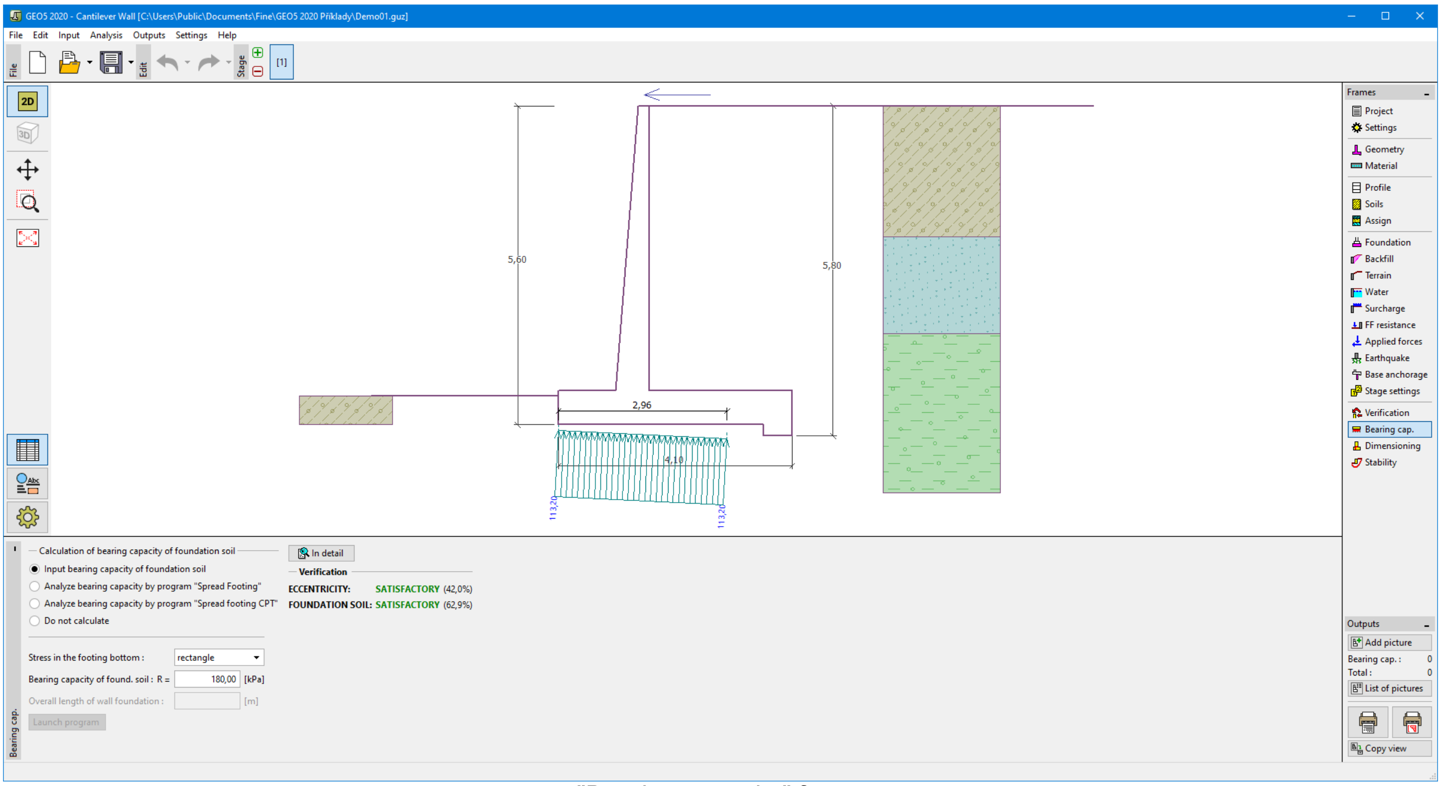
The "**Bearing capacity**" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the "Verification" frame. The "**Spread footing**", "**Spread footing CPT**", "**Pile**" and "**Pile group**" programs consider all verifications as load cases. In the "**Pile CPT**" program, just normal load is used.

The frame contains the following analysis options:

- **Insert bearing capacity of foundation soil** — The foundation soil bearing capacity is input. The **eccentricity** and bearing capacity analysis results are displayed in the right part of the frame. The "**In detail**" button opens a dialog window that displays a detailed listing of the results.
- **Analyze bearing capacity by "Spread footing" program** — Pressing the "**Run program Spread footing**" button opens the program "**Spread footing**" which allows us to calculate the soil bearing capacity or settlement and rotation of the footing. Pressing the "**OK**" button leaves the analysis mode - the results and all plots are transferred into the "**Cantilever wall**" program. The "**Spread footing**" program must be installed for the button to be active. The overall length of the wall foundation is input.
- **Analyze bearing capacity by "Spread footing CPT" program** — The procedure is identical as if calculating soil bearing capacity by the **"Spread footing"** program.
- **Analyze bearing capacity by "Pile" program** — The procedure is identical as if calculating soil bearing capacity by the **"Spread footing"** program. The **"Run program Pile**" is available if the wall has a pile foundation (frame "Foundation"). Pile spacing s is input.
- **Analyze bearing capacity by "Pile CPT" program** — The procedure is identical as if calculating soil bearing capacity by the "**Pile**" program.
- **Analyze bearing capacity by "Pile group" program** — The procedure is identical as if calculating soil bearing capacity by the "**Spread footing**" program. The "**Run program Pile group**" is available if the wall has a pile foundation with more than one pile ("Foundation" frame). Pile spacing s , a total number of pile rows n and loading length l are input.
- **Do not calculate (pile footing)** — The foundation soil bearing capacity is not calculated.

The program allows us to specify a shape of stress in the footing bottom.

Visualization of results can be adjusted in the "Drawing Settings" frame.



"Bearing capacity" frame

Dimensioning

The "**Dimensioning**" frame serves to design and verify the reinforcement of the wall cross-section - the cross-section subjected to dimensioning is selected from the list at the left part of the screen.

- **Wall stem verification - front reinforcement**
- **Wall stem verification - back reinforcement**
- **Wall jump verification**
- **Wall heel verification**
- **Stress relief plate check**

For the Counterfort wall and Buttress wall, the following cross-sections are verified:

- **Wall stem verification - front vertical and horizontal reinforcement**
- **Wall stem verification - back vertical and horizontal reinforcement**
- **Wall jump verification - upper and lower reinforcement**
- **Wall heel verification - upper and lower reinforcement**

- **Verification of rib (buttress and counterfort wall)**

The frame appearance is adjusted based on the selected verification methodology.

- Verification according to the factor of safety or the theory of limit states - the last column in the table allows us to input the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows us to specify whether the load acting on a structure is considered to be a secondary one. This is explained in more detail in the "Load combinations" section.
- Analysis according to LRFD - in this case, the last column is not displayed.

Calculation of forces and their action on the analyzed cross-section is described here.

The wall stem and construction joint are always loaded by the pressure at rest. When verifying the front wall jump, the wall is loaded either by the active pressure or the pressure at rest depending on input specified in the "Stage settings" frame.

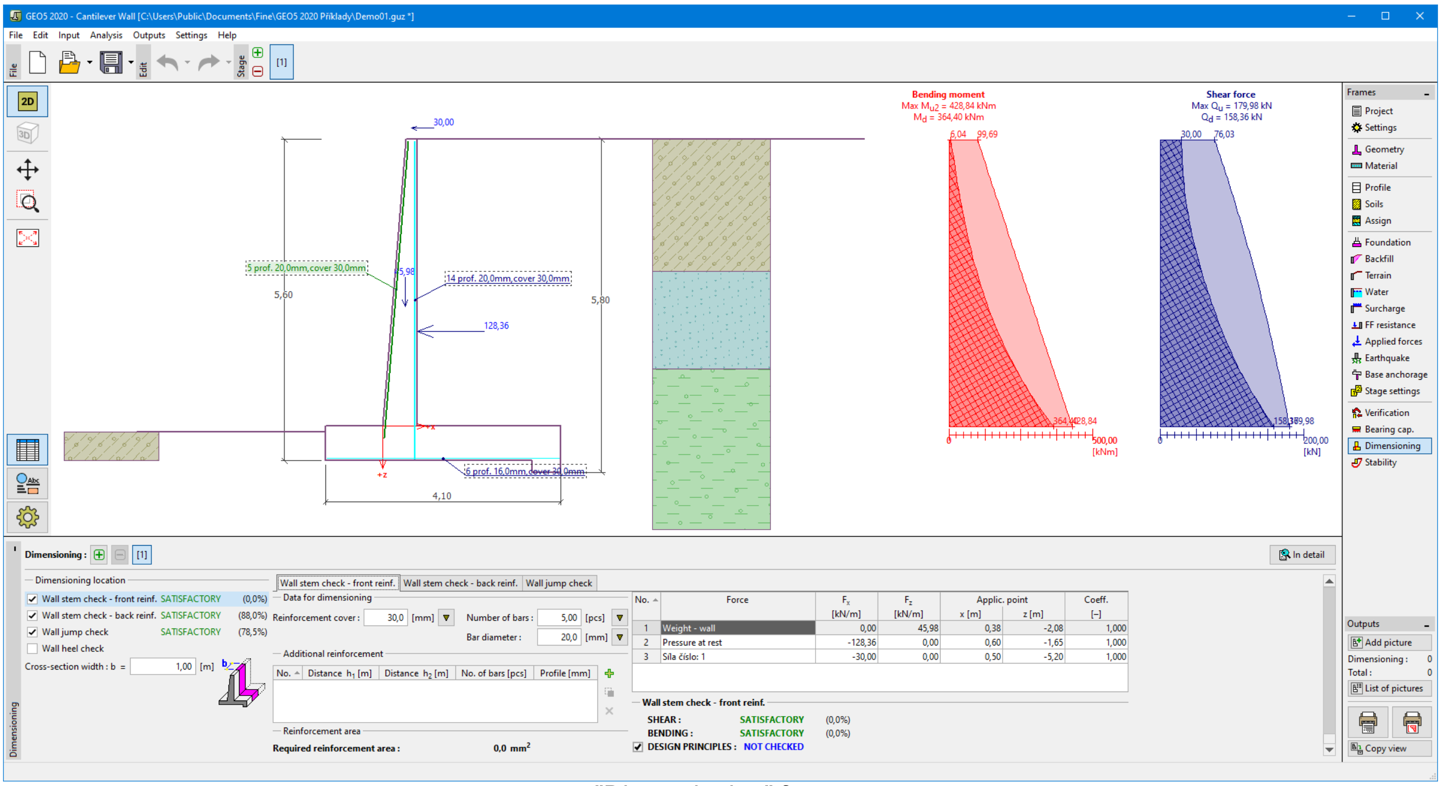
The way to calculate the internal forces in individual cross-sections is described in the theoretical part of this documentation. In addition, force from earth pressure at rest is taken into account when considering earthquake analysis.

Dimensioning of the reinforced concrete is performed according to the standard set in the "Materials and standards" tab.

In the case of wall dimensioning according to EN 1992, the crack width can be verified.

Several computations for different reinforcement variants can be carried out. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "**In detail**" button opens the dialog window that contains a detailed listing of the dimensioning results.

Visualization of results can be adjusted in the "Drawing Settings" frame.

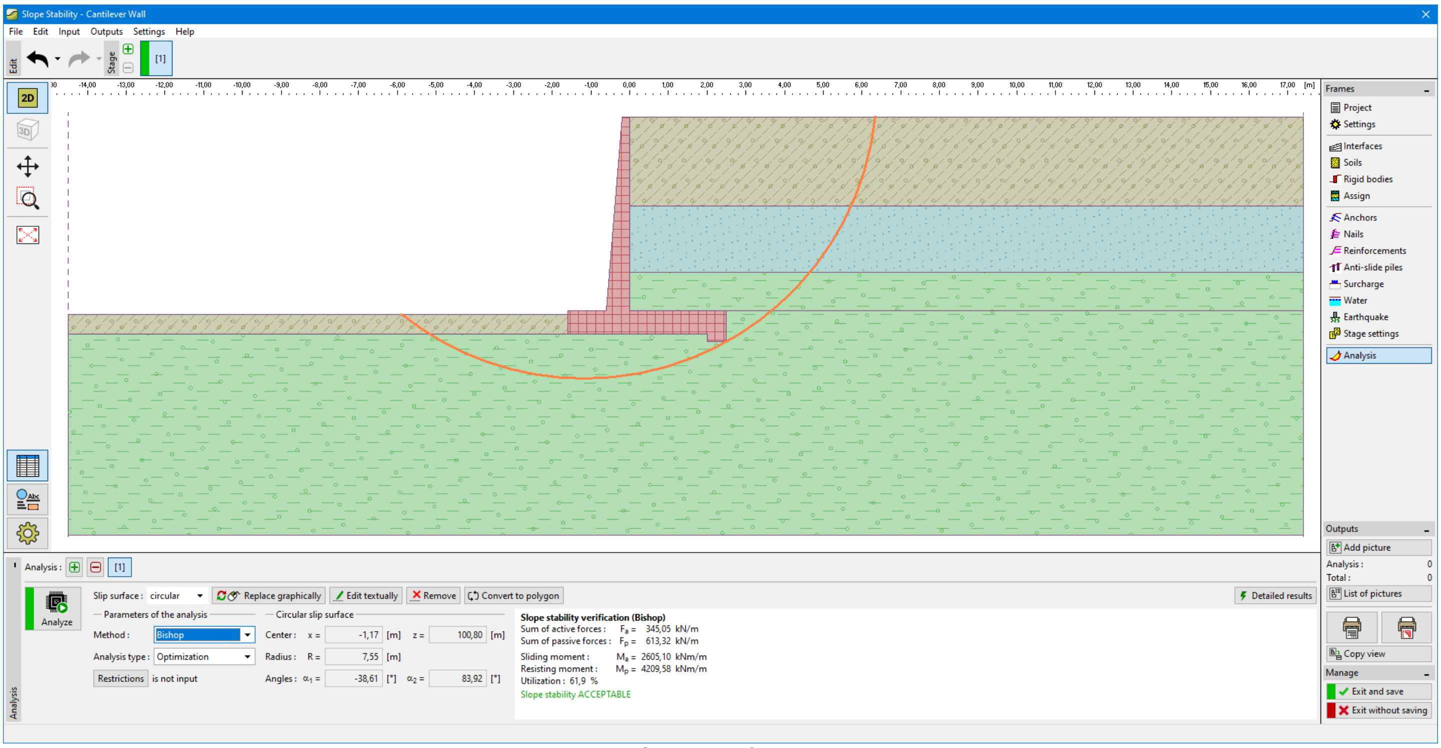


"Dimensioning" frame

Stability

Pressing the "**Stability**" button launches the "**Slope stability**" program. This program allows us to check the overall stability of the analyzed structure. The button is available only if the "**Slope stability**" program is installed.

After completing all analyses, press the "**Exit and save**" button to leave the program - all data are carried over to the analysis protocol of the "**Cantilever wall**" program.



"Stability" frame

Program Gravity Wall

This program is used for the analysis of gravity walls. It offers a range of wall shapes and verifies mass concrete cross-sections.

The help in the program "Gravity Wall" includes the following topics:

- The input of data into individual frames:

Project	Settings	Geometry	Material	Profile	Soils	Assign
Foundation	Backfill	Terrain	Water	Surcharge	Front Face Resistance	Applied Forces
Anchors	Earthquake	Stage Settings	Verification	Bearing Capacity	Dimensioning	Stability

- Standards and analysis methods

- Theory for analysis in the program "**Gravity wall**":

Stress in Soil Body	Earth Pressures	Analysis of Walls	Analysis of Foundation Bearing Capacity	Dimensioning of Concrete Structures	Masonry Cross Sections Verification

- Outputs

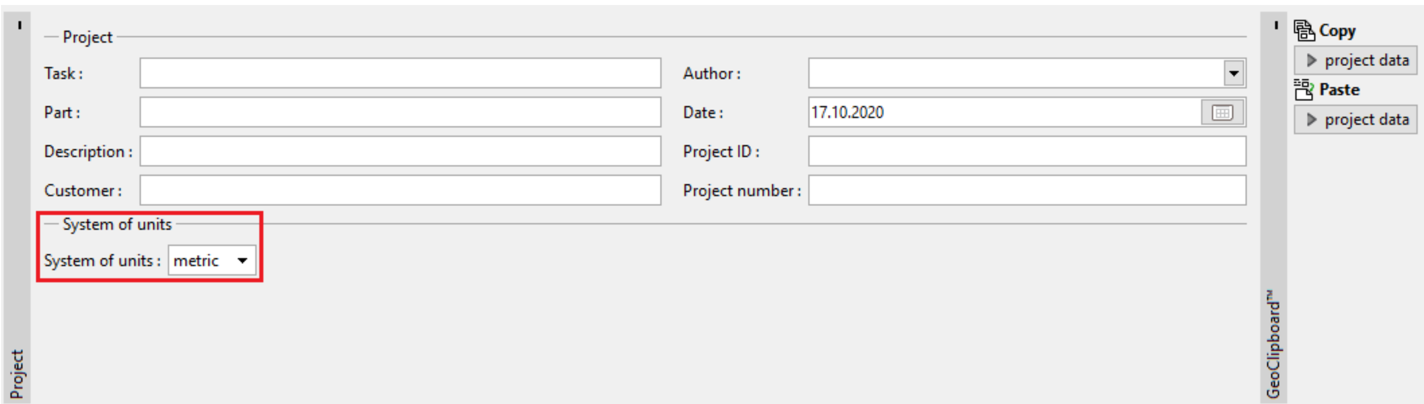
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The **"Project"** frame is used to input basic project data. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows the user to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The "**Settings**" frame allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows us to view and modify an individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

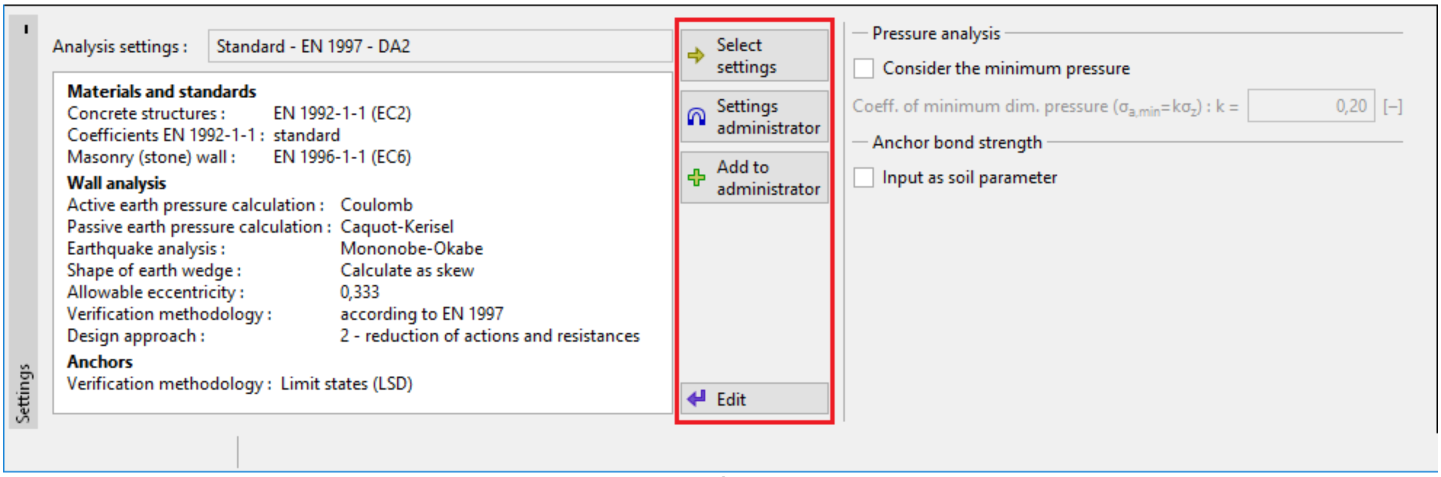
The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. The title changes to "**Input for the current task**" by modifying any of the parameters. Individual analyses are then performed with this **local setting**. If this setting is suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

The program allows us to specify a value of the minimum dimensioning pressure (by selecting the option "**Consider the minimum dimensioning pressure**").

When calculating the pull-out resistance of anchor, bond strength can be inserted as a soil parameter.

Settings of analysis parameters are performed in the "Materials and standards" and "Wall analysis" tabs.



Frame "Settings"

Geometry

The "**Geometry**" frame allows us to select a desired wall shape. The selected shape with a graphic hint appears in the left part of the frame. The shape of the wall can be edited either in the frame by inserting values into the input fields, or on the desktop using active dimensions.

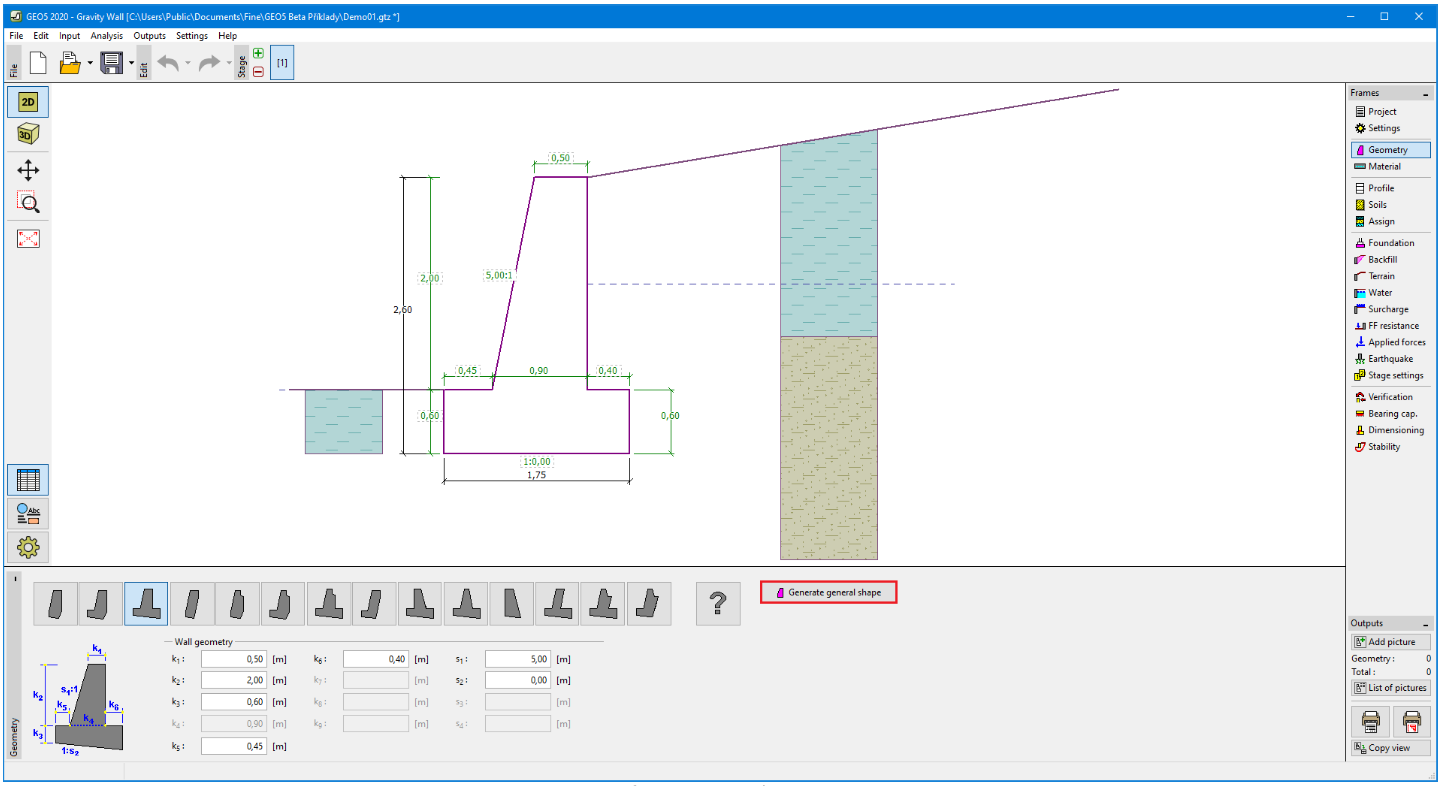
In case the structure is composed of inclined segments, it is required to enter the ratio of sides of an inclined segment $1{:}x$. A **straight structure** is specified by entering the value zero.

Defined wall shapes

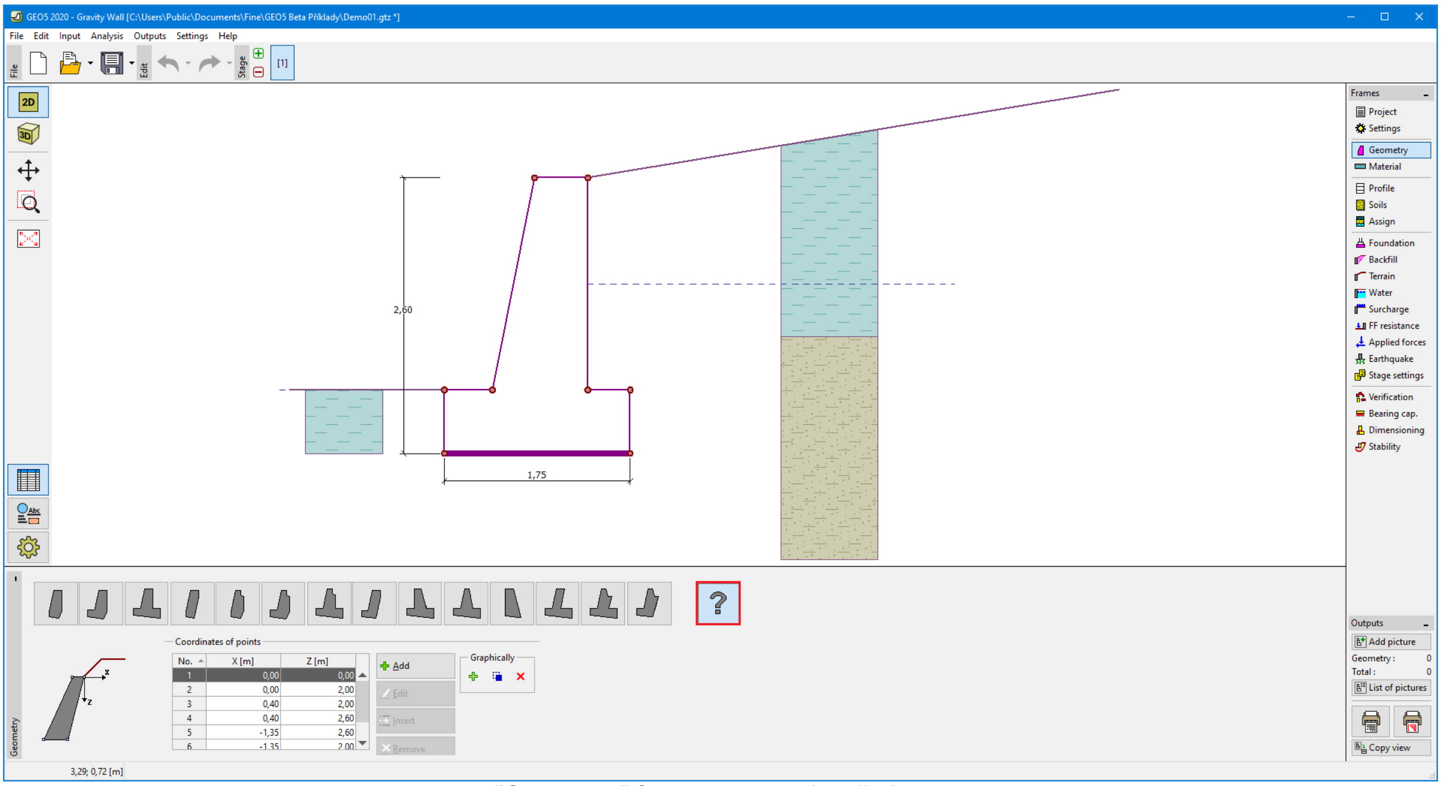
If the defined wall shapes are not satisfactory for the wall geometry input, the program allows us to enter a general shape of gravity wall. The general wall shape is entered by coordinates of points, but it is also possible (by pressing the "**Generate general shape**" button) to generate coordinates of the structure from an already input predefined wall.

General wall shape input

The program allows us to export the geometry of the structure in *.DXF format.



"Geometry" frame



"Geometry" frame - general wall shape

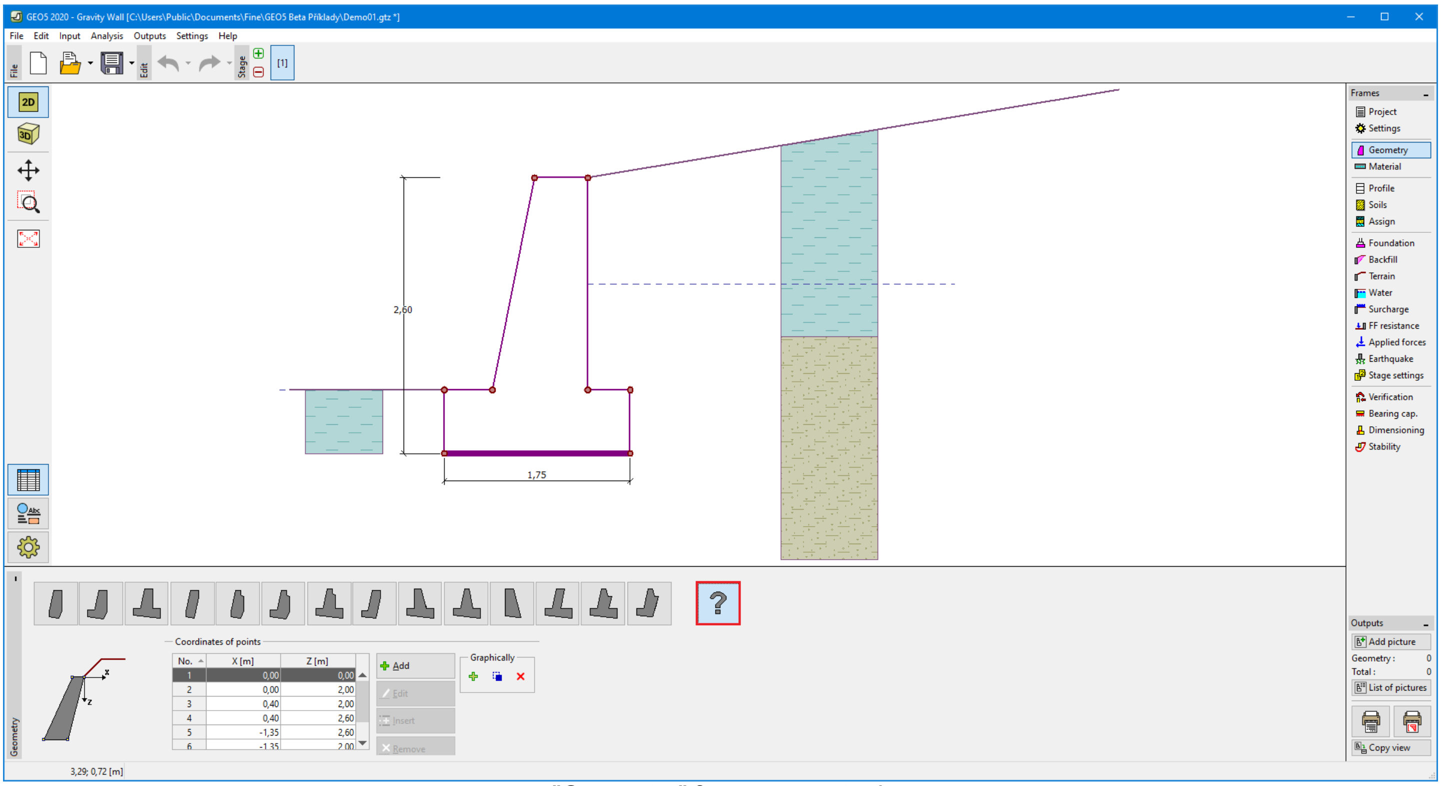
General Wall Shape

Input of general wall shape in a new task

The program allows us to input general wall shape in two ways:

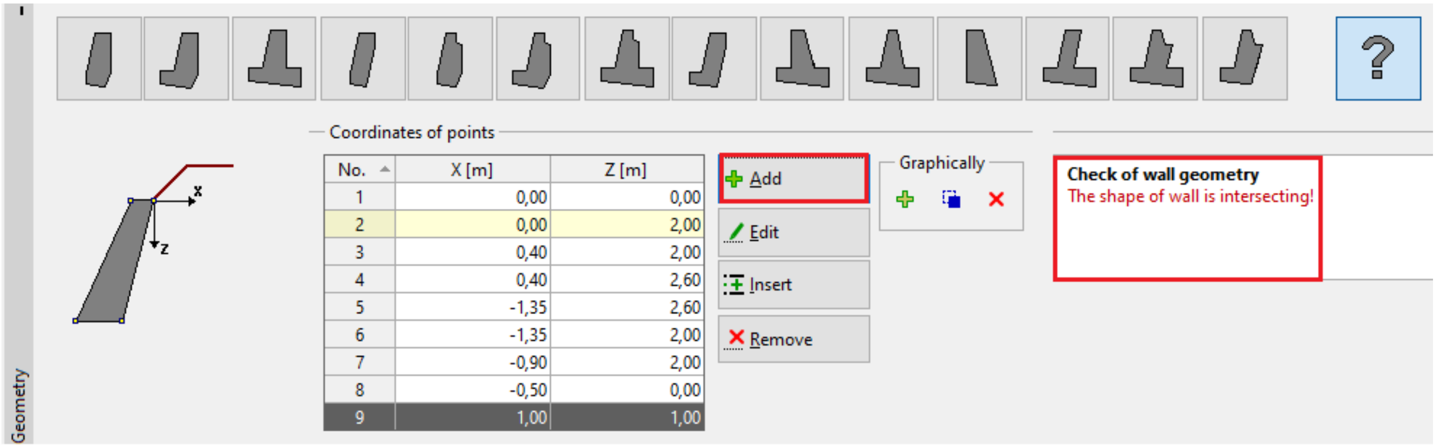
1. Input of general wall shape using points

Pressing the icon with a question mark on the **"Wall geometry scheme"** toolbar will create a rectangular shape and the first structure point [0,0]. A minimum input number of points is 3 (in case a smaller number is input, the program will show an error message).



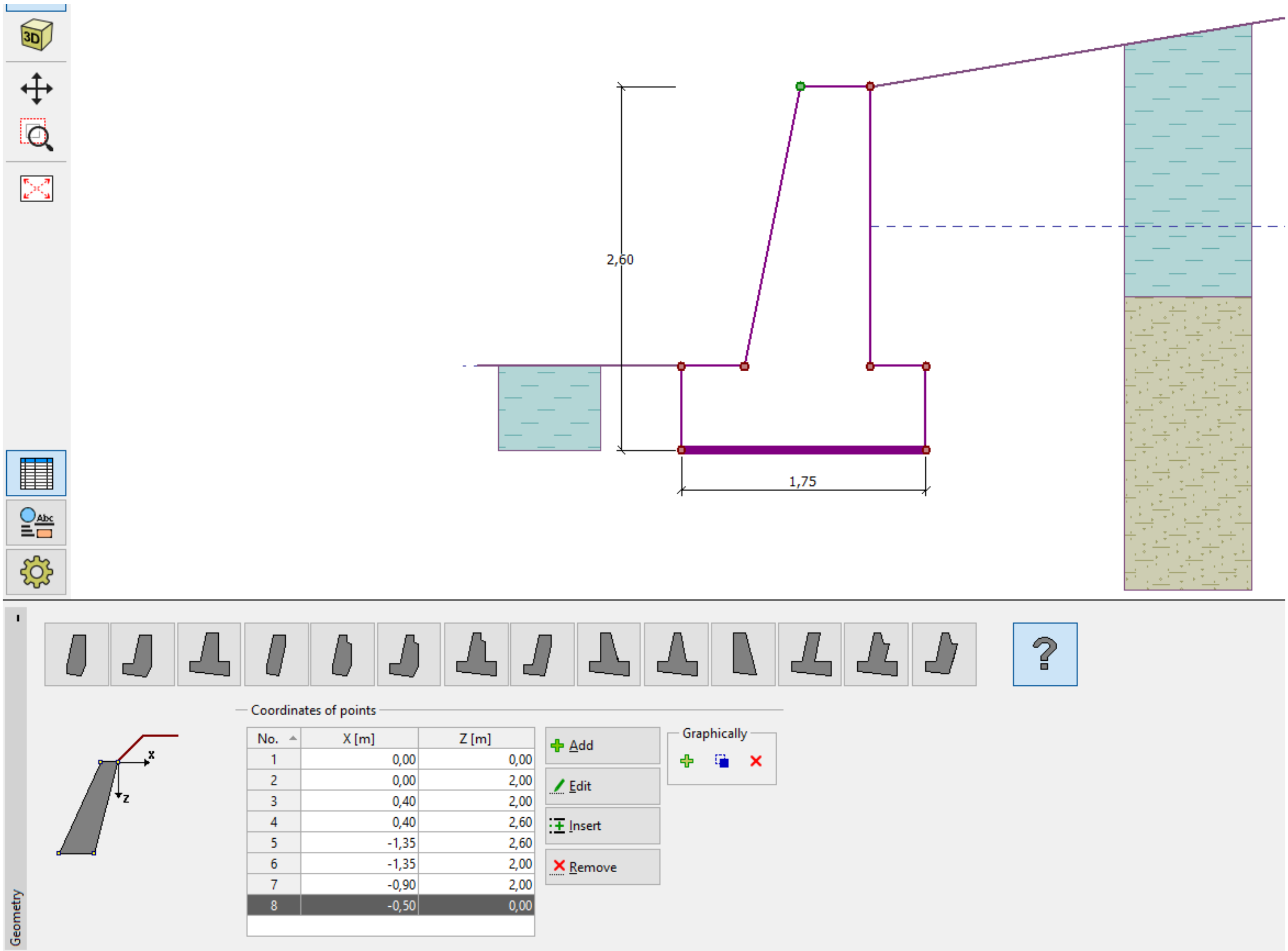
"Geometry" frame - new task

By using the **"Add"** button, which opens the **"New point"** dialog window, more structure points are input (it is possible to input points by clicking on the desktop).



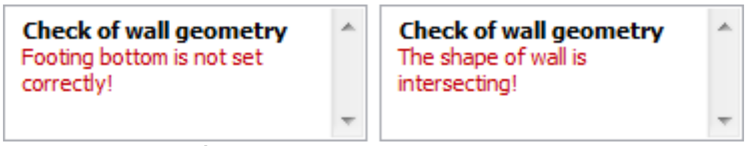
"Geometry" frame - general wall shape input using points

Input points are being added to the table, and it is then possible to edit them, insert more in between and delete them using the desired buttons **"Edit"**, **"Insert"** and **"Remove"** or by clicking the points on the desktop in the corresponding mode. Points can be moved right on the desktop by mouse after clicking on the special icon .



"Geometry" frame- edit points

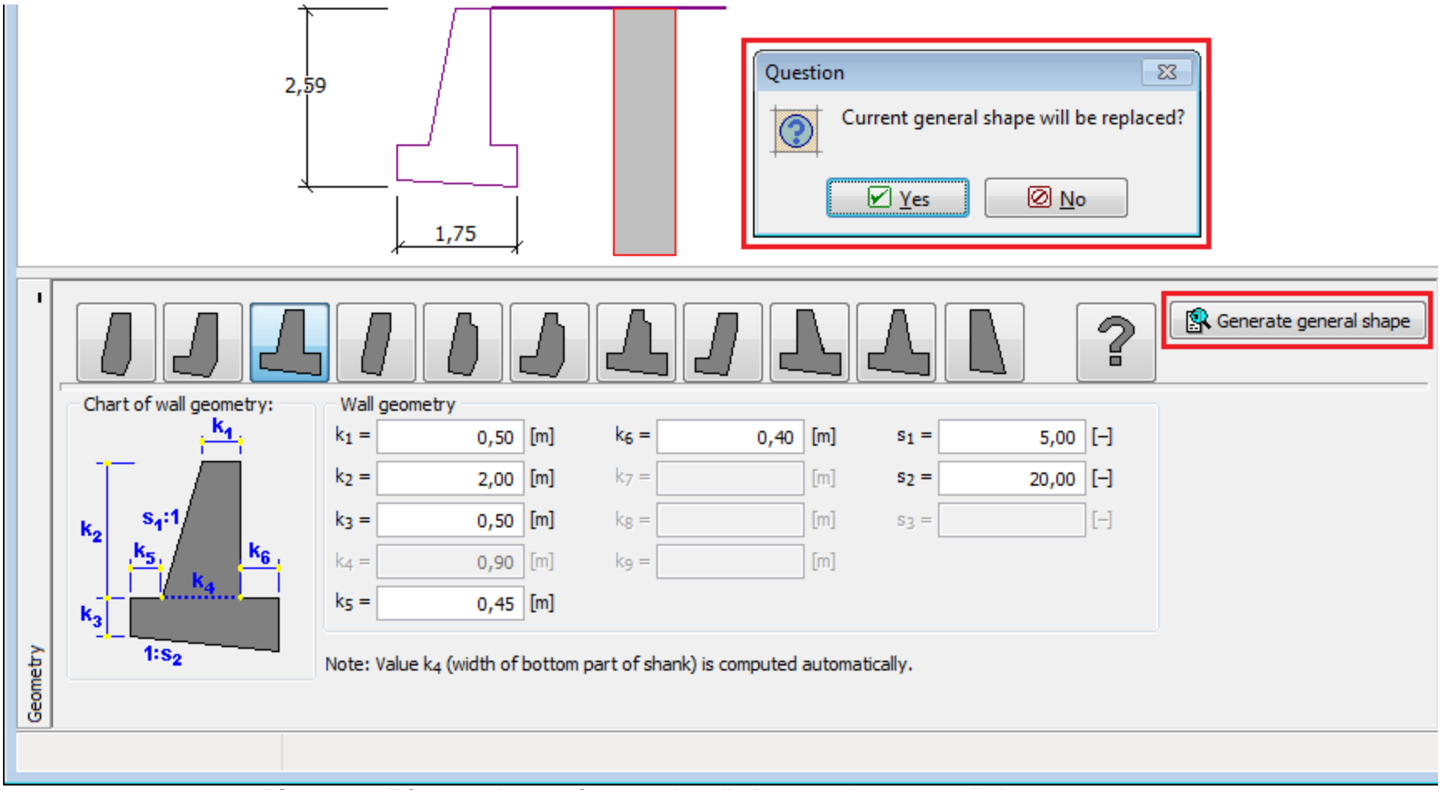
The maximum inclination of the footing bottom is considered 45° . In case of the wrong inclination of the footing bottom input, or if the wall shape is intersecting, the program checks the general geometry of the wall and warns the user of an error. In that case, the wall geometry has to be changed.



Frame "Geometry" - point input error message

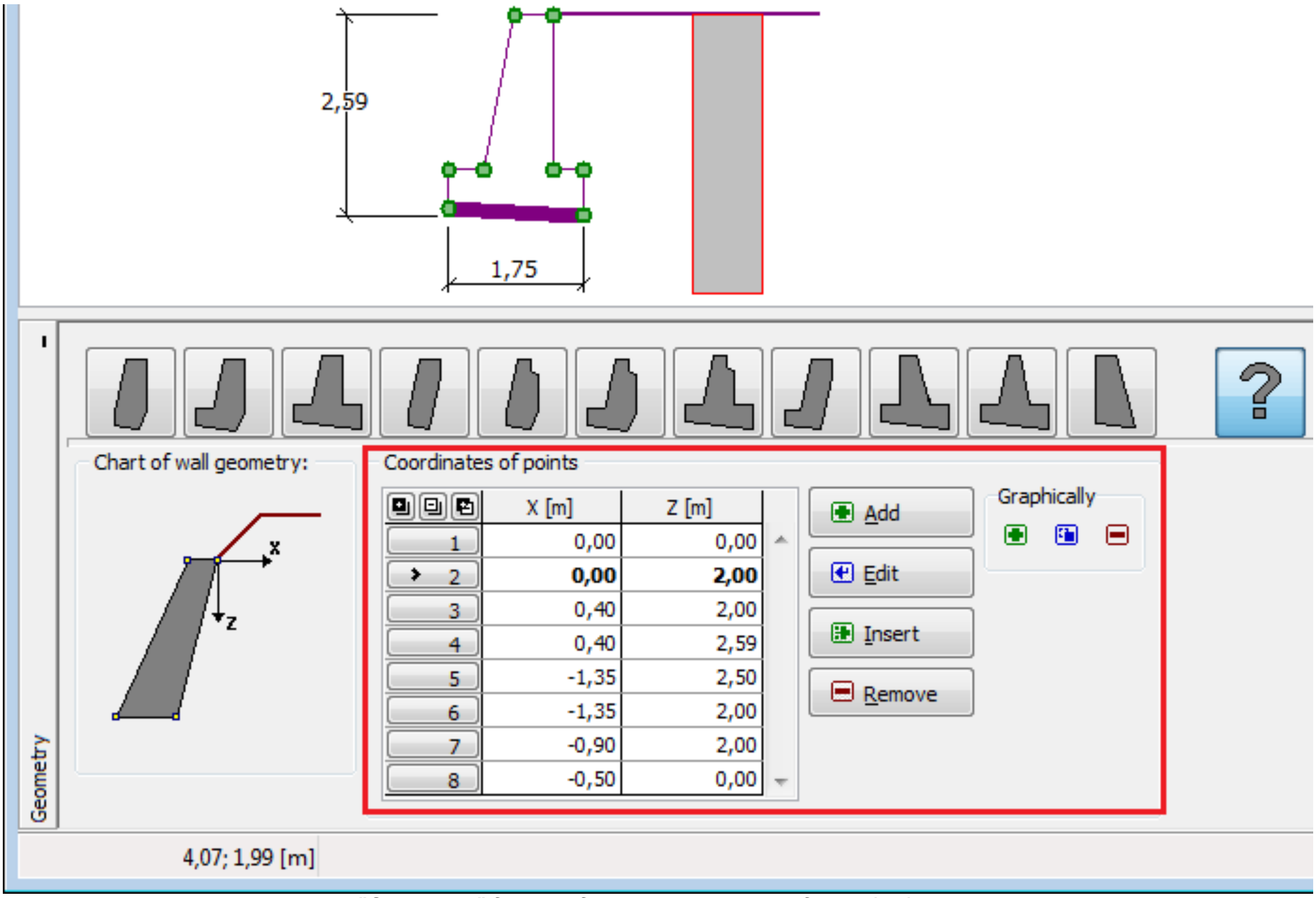
2. Input of general wall shape using general shape generator

The structure defined by the scheme of construction and its dimensions can be taken to the general wall shape input by pressing the "**Generate general shape**" button. It is possible to work with the newly generated points and edit the generated wall shape.



"Geometry" frame - input of general wall shape using general shape generator

Frame appearance is then changed as in the first case of general wall shape input. It is possible to work with the picture of the structure as already described.



"Geometry" frame - frame appereance after point input

Material

The **"Material"** frame allows us to enter material parameters. The input field in the upper part of the frame serves to specify the **wall unit weight**.

If the **wall is made from concrete**, the program offers two options:

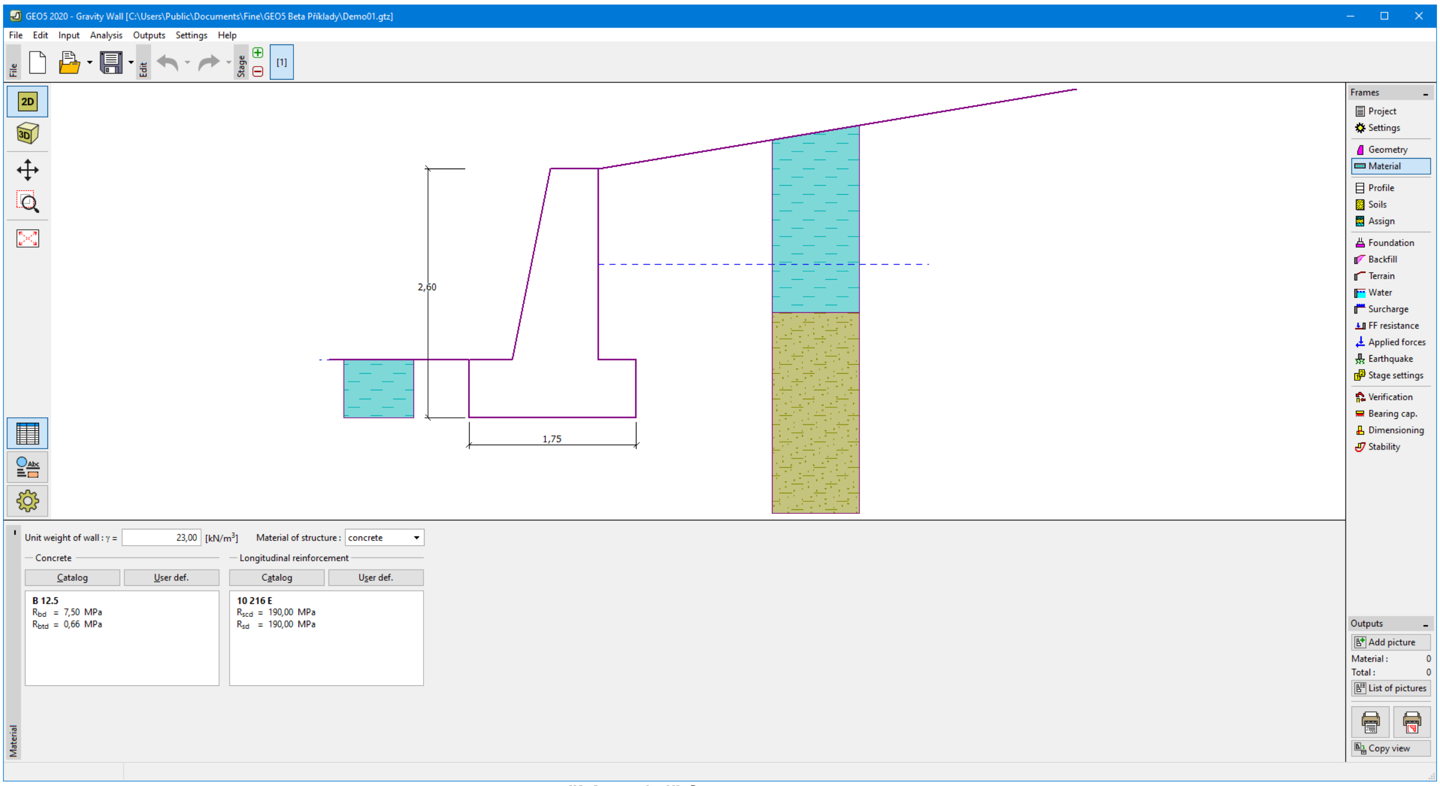
- The "**Catalog**" button opens the "**Catalog of materials**" dialog window, where the required material can be selected.
- The "**User-defined**" button opens the "**Editor of material**", which allows us to input the specification of material parameters manually by the user.

The content of catalogs depends on selecting the relevant standard for dimensioning of concrete and steel structures in the "Materials and standards" tab.

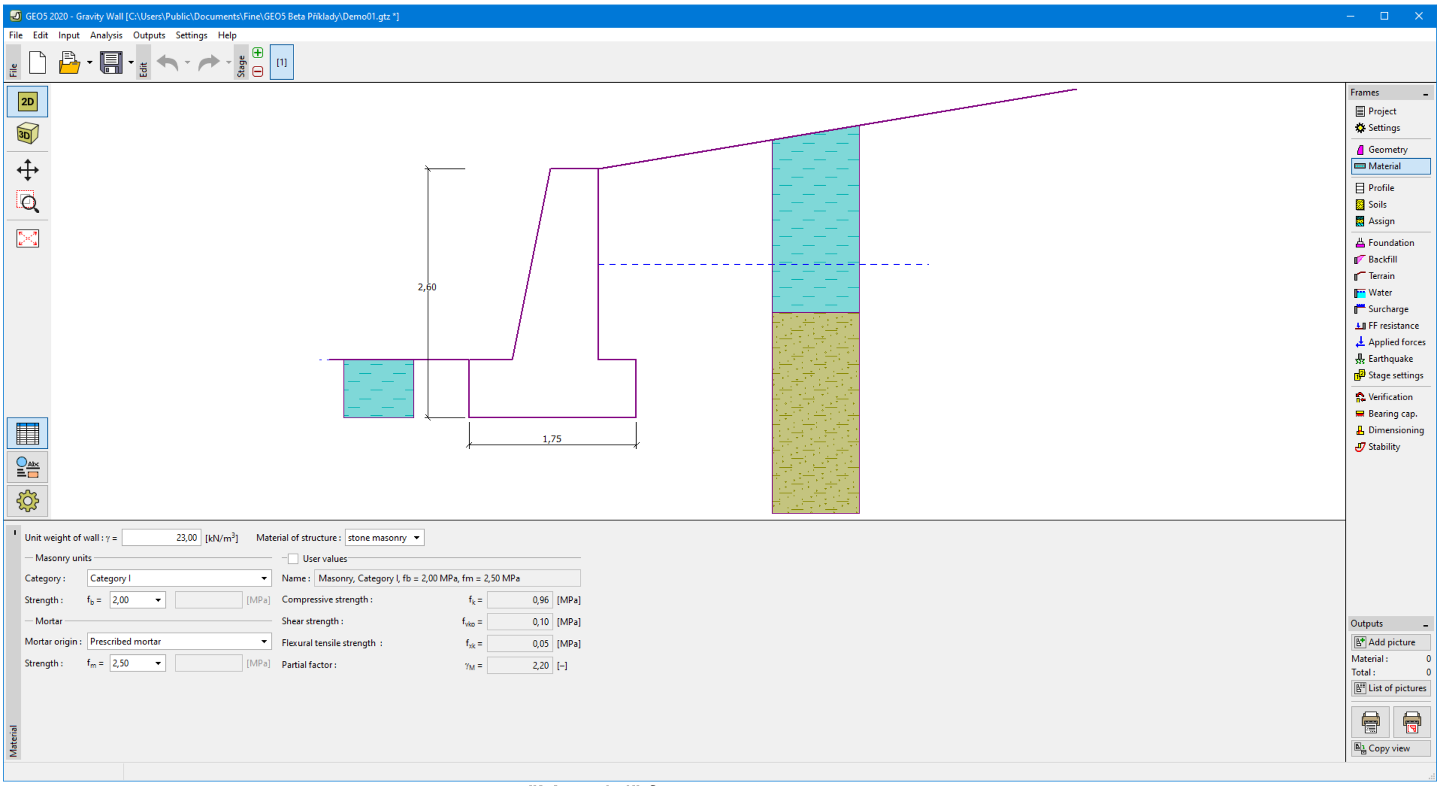
If the wall is made from stone masonry, material characteristics of masonry units according to the selected standard are specified in the "Materials and standards" tab:

- EN 1996-1-1 - the strength of masonry units f_b , mortar origin and strength of mortar f_m
- GB 50003-2011 - a type of masonry units, strength grade of stone, strength grade of mortar

If the **"User values"** button is checked, it is possible for the users to input user-defined material characteristics.



"Material" frame- concrete



"Material" frame - stone masonry

Profile

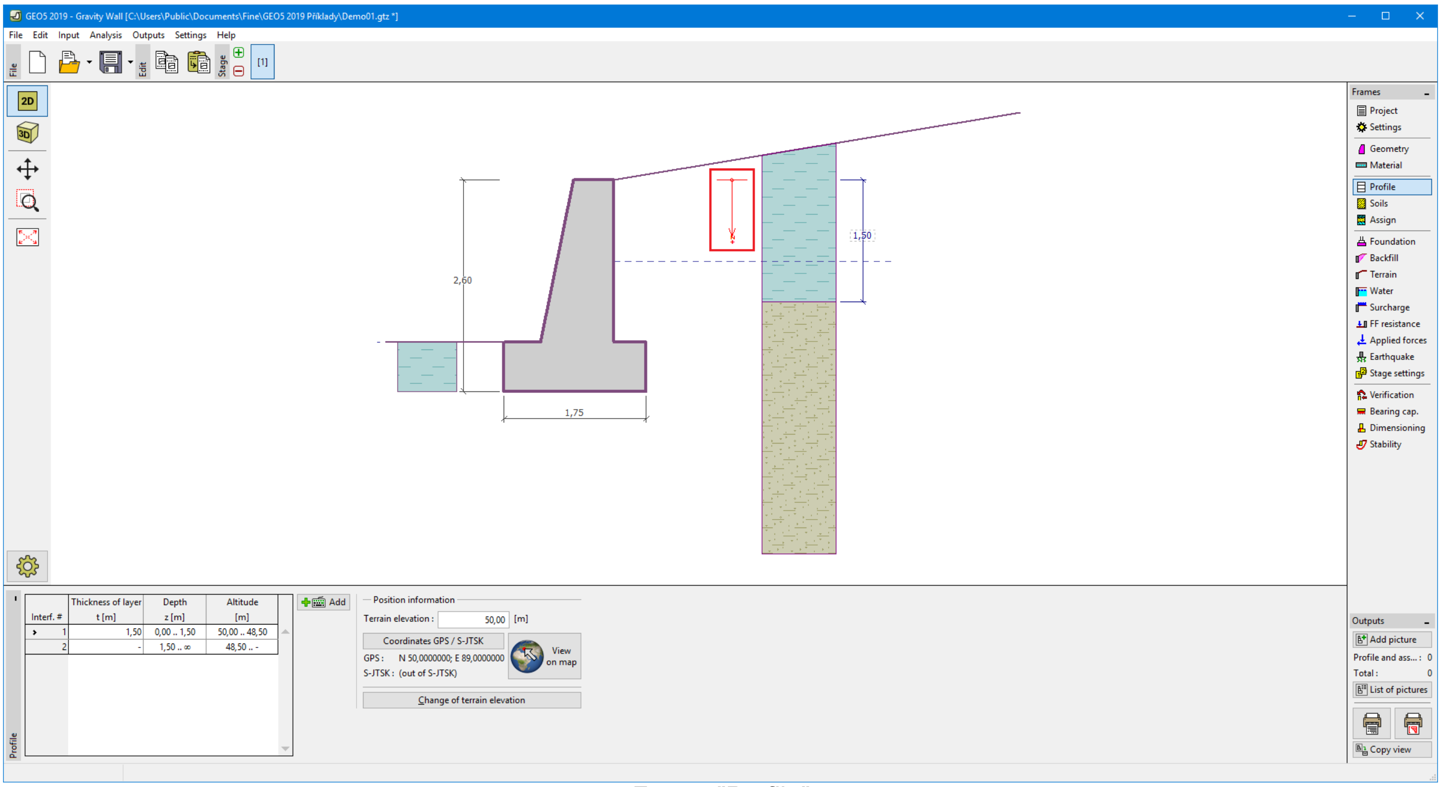
The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.



Frame "Profile"

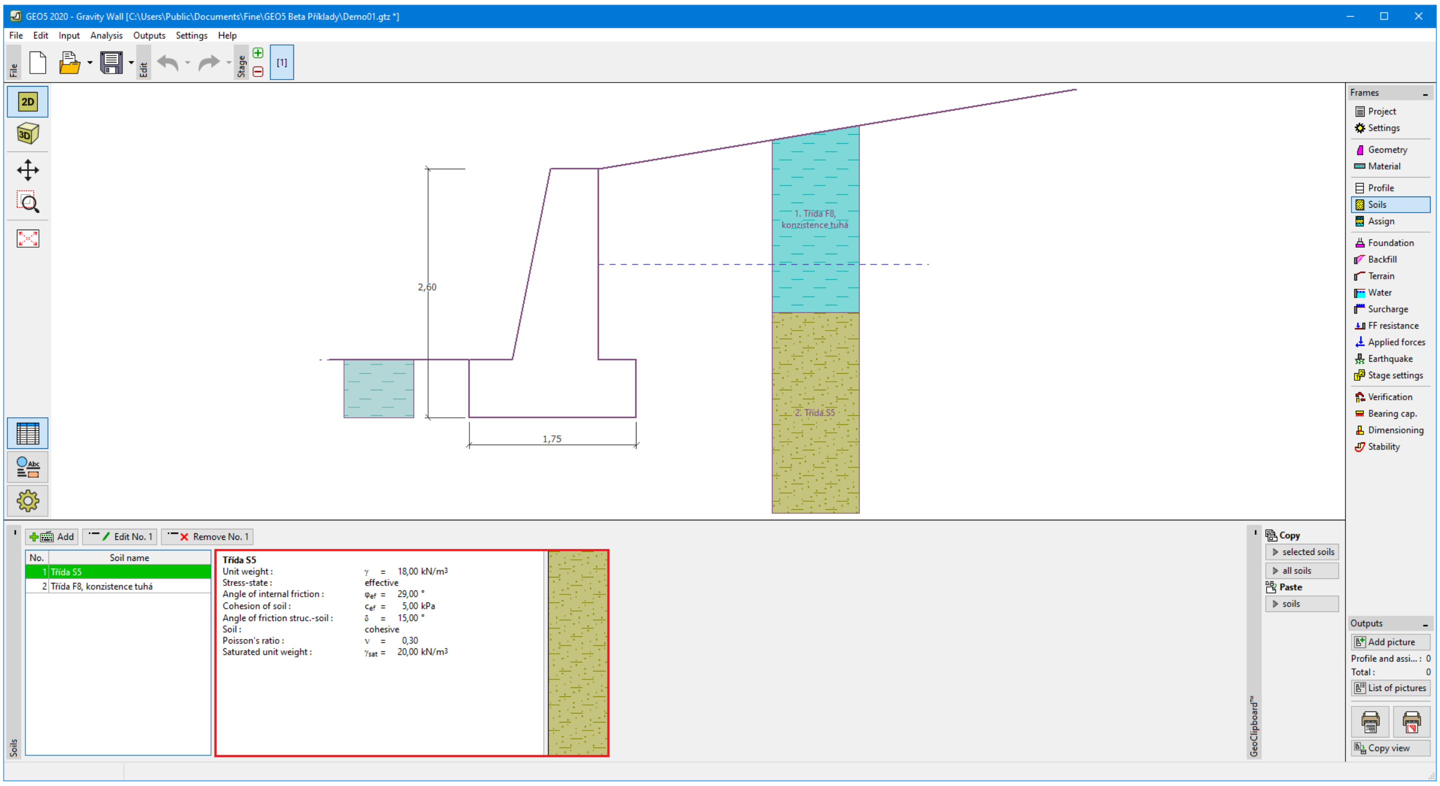
Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about a currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding a soil is performed in the "Add new soils" dialog window.

The soil characteristics entered in the program are further specified in the following chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

The data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

Basic Data

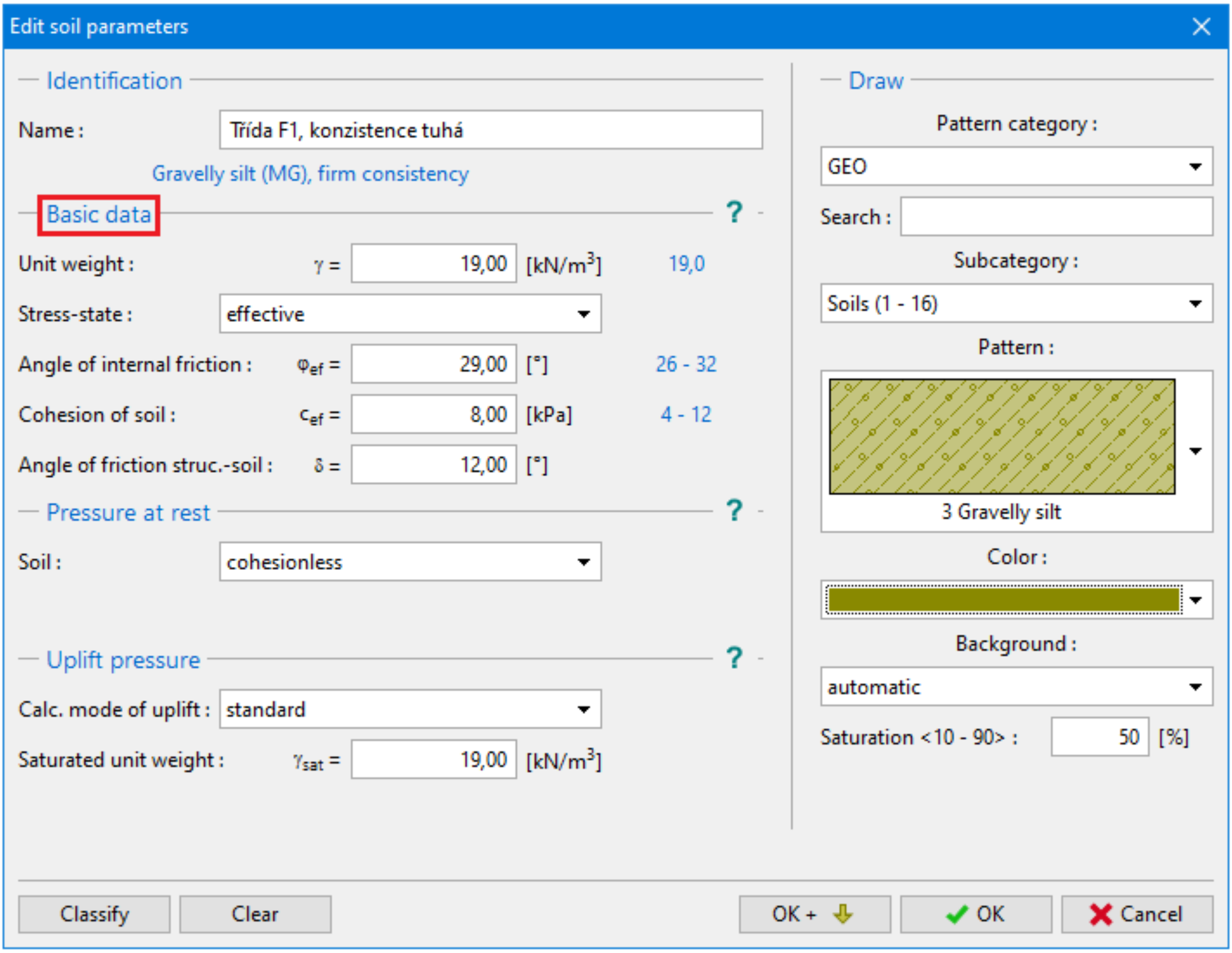
This part of the window allows us to introduce basic soil parameters - **unit weight, angle of internal friction, and cohesion**. The particular values are obtained from a geotechnical survey or from laboratory experiments. If this data is not available, it is possible to exploit the built-in database of soils, which contains values of selected soils characteristics. The rocks characteristics are not listed in the built-in database, these parameters must be defined manually.

Effective or total parameters of an internal friction angle and cohesion are specified depending on the setting in the "**Stress analysis**" combo list. Whether to use effective or total parameters, it depends primarily on the type of soil, type of load, structure duration, and water conditions.

For effective stress, specify the angle of internal friction between the soil and structure, which depends on the structure material and the type of soil. Possible values of this parameter are listed in the table of recommended values.

For total stress, specify the adhesion of soil to the structure face a .

The associated theory is described in detail in the "Earth pressures" chapter.

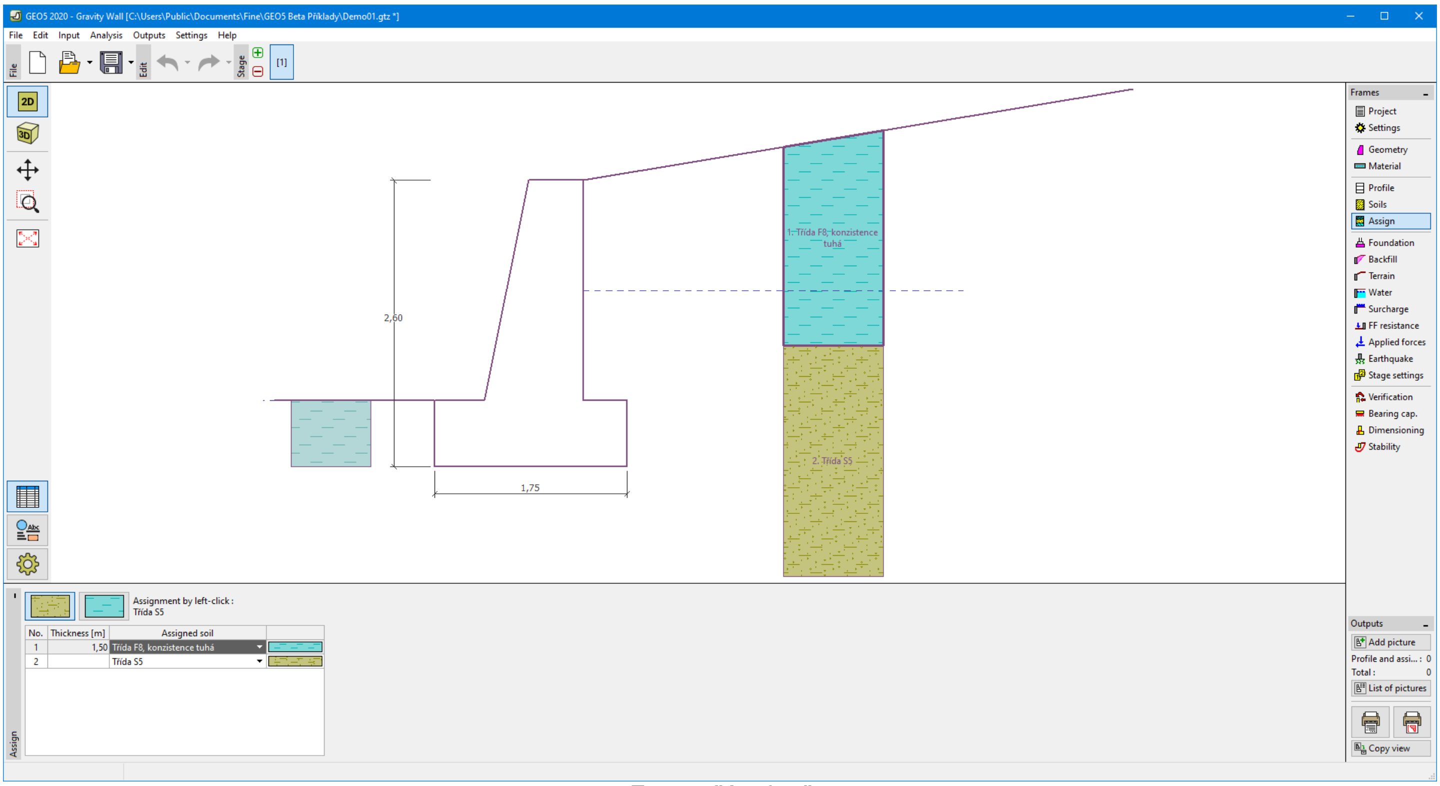


Dialog window "Add new soils" - "Basic data"

Assign

The "**Assign**" frame contains a list of profile layers and assigned soils. The list of soils is graphically represented by using buttons in the bar above the table or is accessible from a combo list for each profile layer.

The procedure to assign a soil to a layer is described in detail herein.



Frame "Assign"

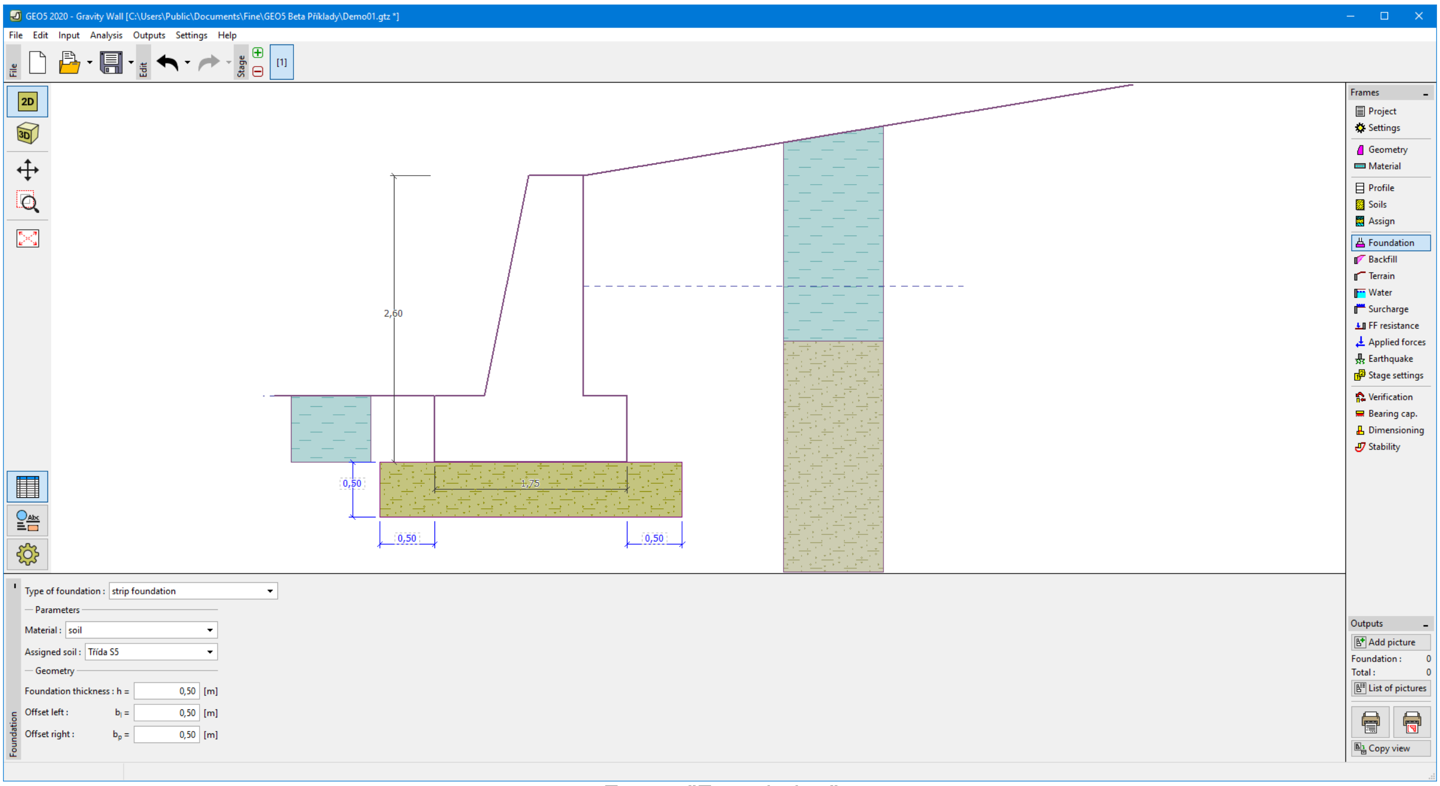
Foundation

The "**Foundation**" frame serves to specify the wall foundation type. The following wall foundation types are available:

- **soil from geological profile** - the wall is founded on the soil assigned from the geological profile specified in the "Profile" frame.
- **input parameters of contact base-soil** - parameters of the contact between the footing bottom and the structure are specified. The "**input angle of friction base-soil**" option requires inputting a friction angle ψ [°] between the foundation and the soil. The "**input friction coefficient**" option requires specifying the friction coefficient μ [-]. Both options require inputting the cohesion a $[kPa]$ between the foundation (base) and the soil.
- **strip foundation** - strip foundation material is represented either by **soil** (input in "Soils" frame), or by concrete - requires inputting the **unit weight of foundation material** γ and **parameters of contact base-soil** (friction coefficient f, cohesion c, additional resistance F).
- **pile foundation** - the wall can be founded on one row of piles or two rows of piles, respectively.

The **Strip foundation** and the **pile foundation** can be adopted for the wall foundation only if the wall type with **straight footing bottom without jump** is selected in the "Geometry" frame. The wall foundation geometry (**strip foundation**, **pile foundation**) can be modified either in the frame by entering specific values into the inputting fields or on the desktop by using active dimensions.

The input data introduced in this frame influence the actual wall analysis (check for slip) and the bearing capacity of foundation soil.

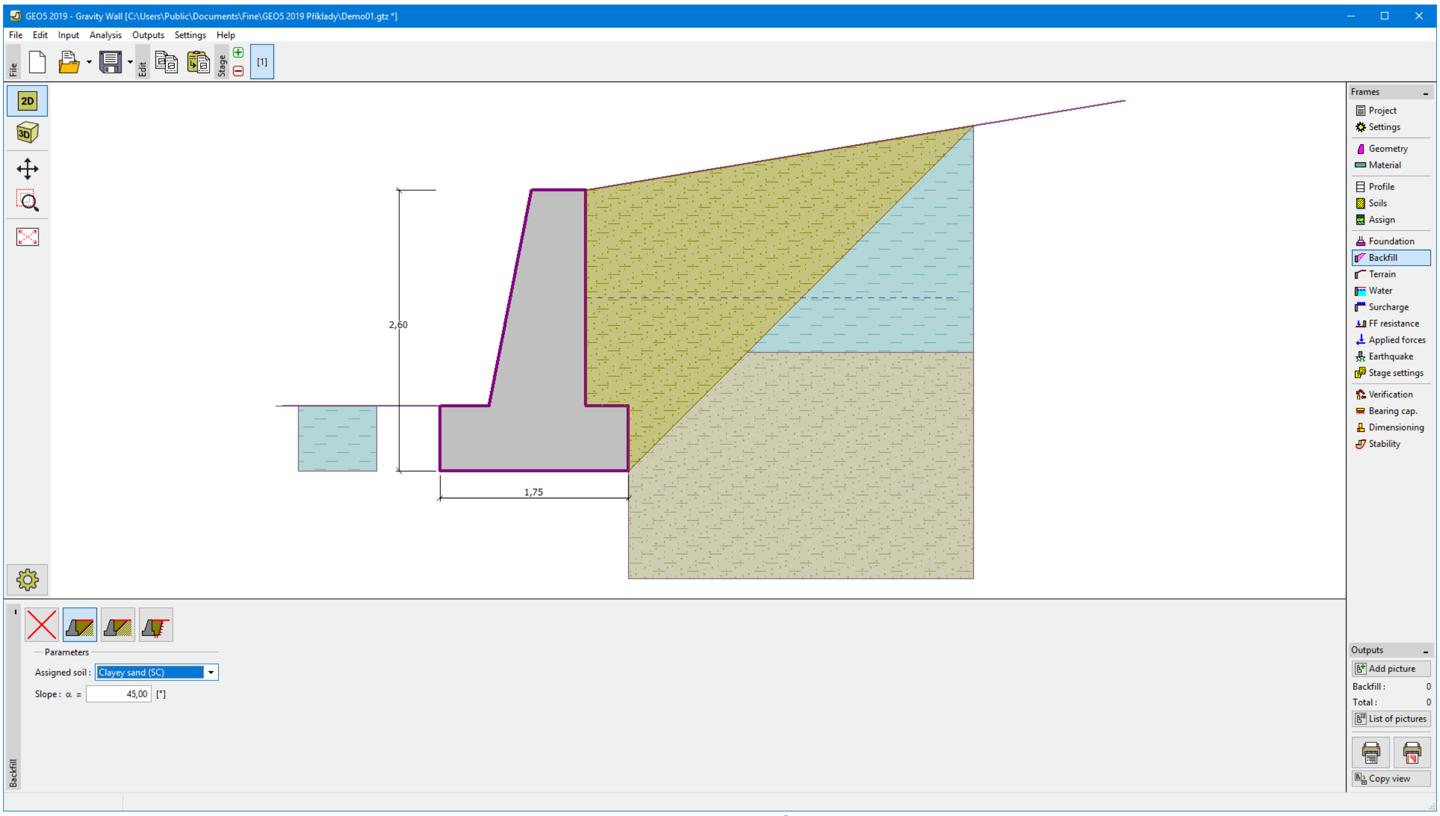


Frame "Foundation"

Backfill

The "**Backfill**" frame allows the selection of backfills behind the structure.

The analysis of earth pressures with the influence of the backfill is described in the theoretical part of the help: "Influence of Backfill".



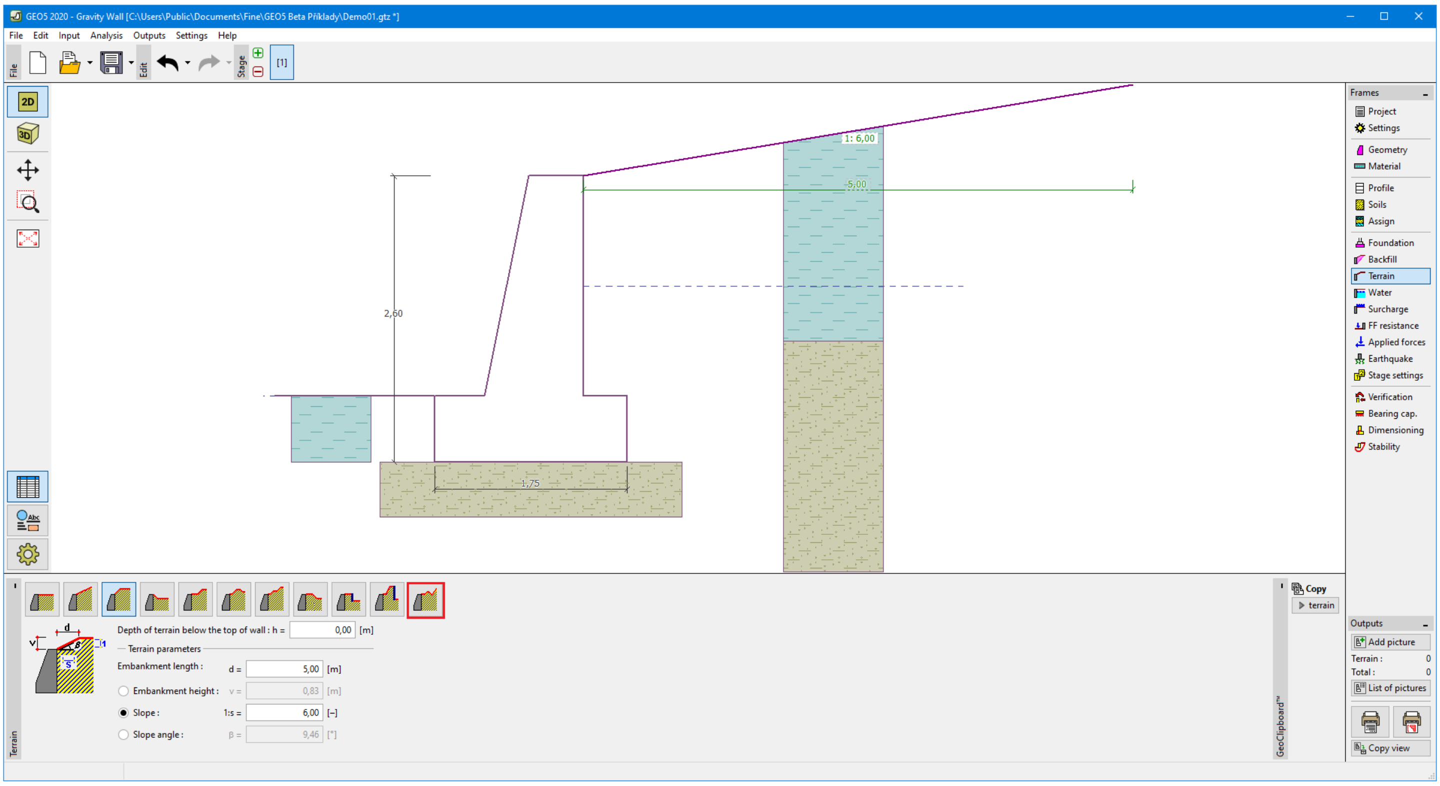
Frame "Backfill"

Terrain

The "**Terrain**" frame allows, by pressing the button, to specify the terrain shape. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop by using active dimensions.

The last option in the menu is a general shape of a terrain. In this case, the frame contains a table with a list of terrain points. The first point with coordinates [0; 0] coincides with the top point of a structure.

The analysis of earth pressures in case of inclined terrain is described in "Distribution of earth pressures for broken terrain", a theoretical part of the help.



Frame "Terrain"

Water

The "**Water**" frame allows, by pressing the button, to select a water type. The selected type with a graphic hint of input values is displayed in the left part of the frame. Water parameters (h_1 , h_2 ...) can be edited either in the frame by inserting values into input fields, or on the desktop by using active dimensions.

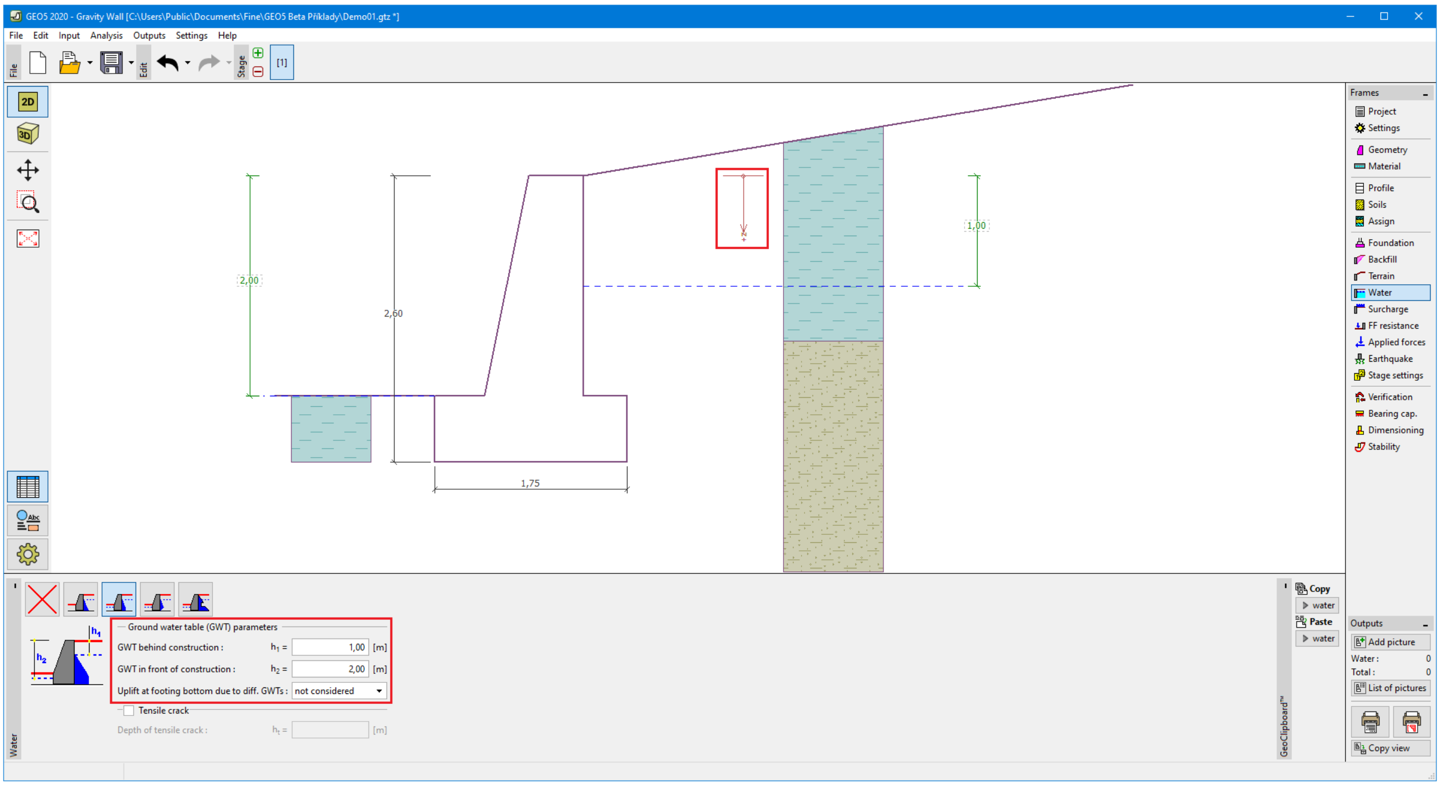
The combo list serves to specify whether the influence of water uplift pressure due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be linear, parabolic, or it may not be considered at all. When verifying the wall, the uplift pressure in the base of footing joint due to different water tables is introduced in terms of a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tabs, "**In front of structure**" and "**Behind structure**", appear with tables. The table contains the values of pore pressure in front of the structure or behind it.

The groundwater table can also be specified **above the structure** or earth profile, respectively - in such a case, the depth of water is input with a negative value.

The earth pressures analysis with the influence of water is described in the "Influence of water", a theoretical part of the help.

The program further allows us to specify a depth of tensile cracks filled with water.



Frame "Water"

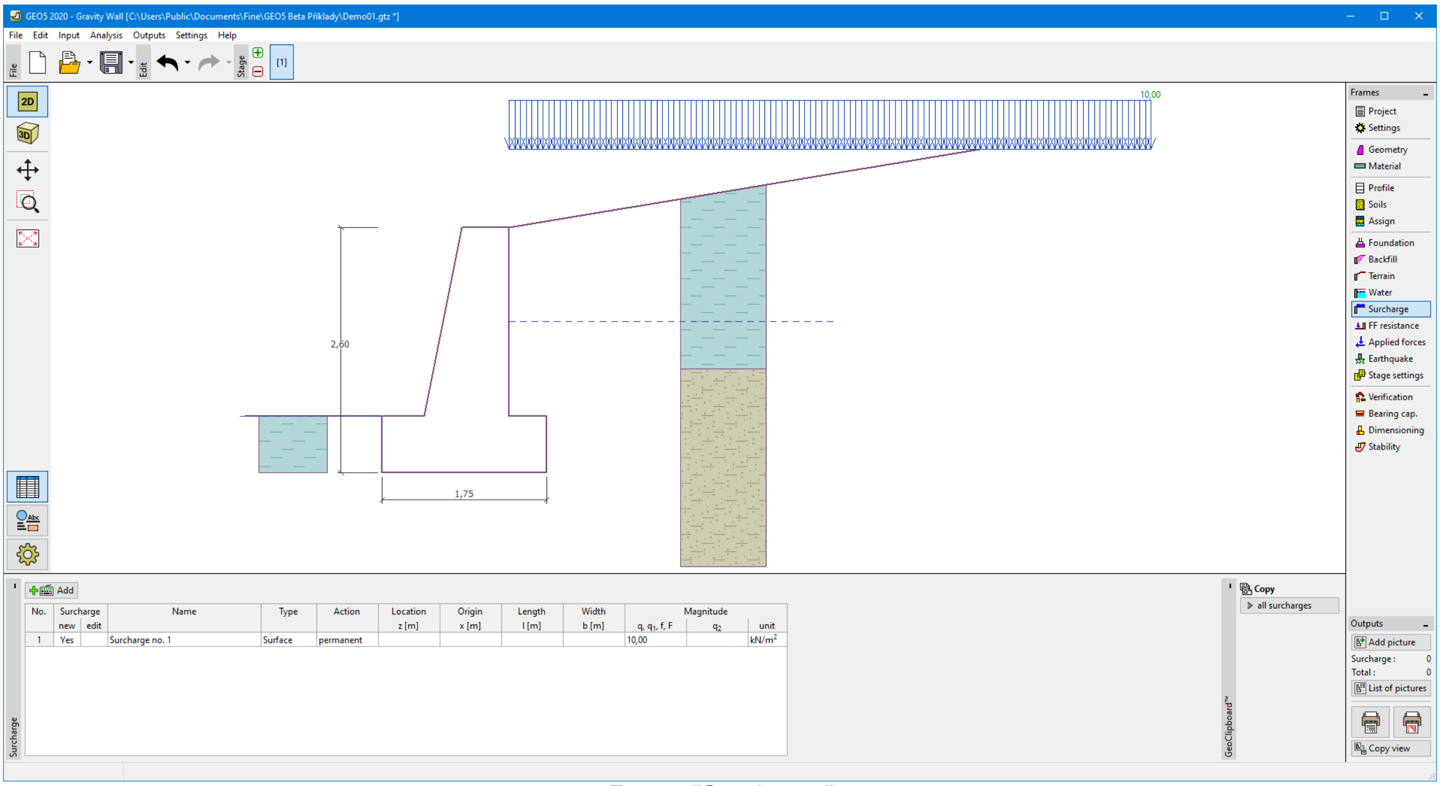
Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop using active dimensions or active objects.

The z -coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case the surcharge is found out of the terrain, the program prompts an error message before calculation.

A surcharge can be specified as **permanent**, **variable,** or **accidental**. The resulting load action is multiplied by the corresponding design coefficient according to the type of surcharge. An accidental surcharge with a favorable effect is not considered in the analysis.

The surcharge effect on the earth pressures analysis is described in the "Influence of surcharge", a theoretical part of the help.



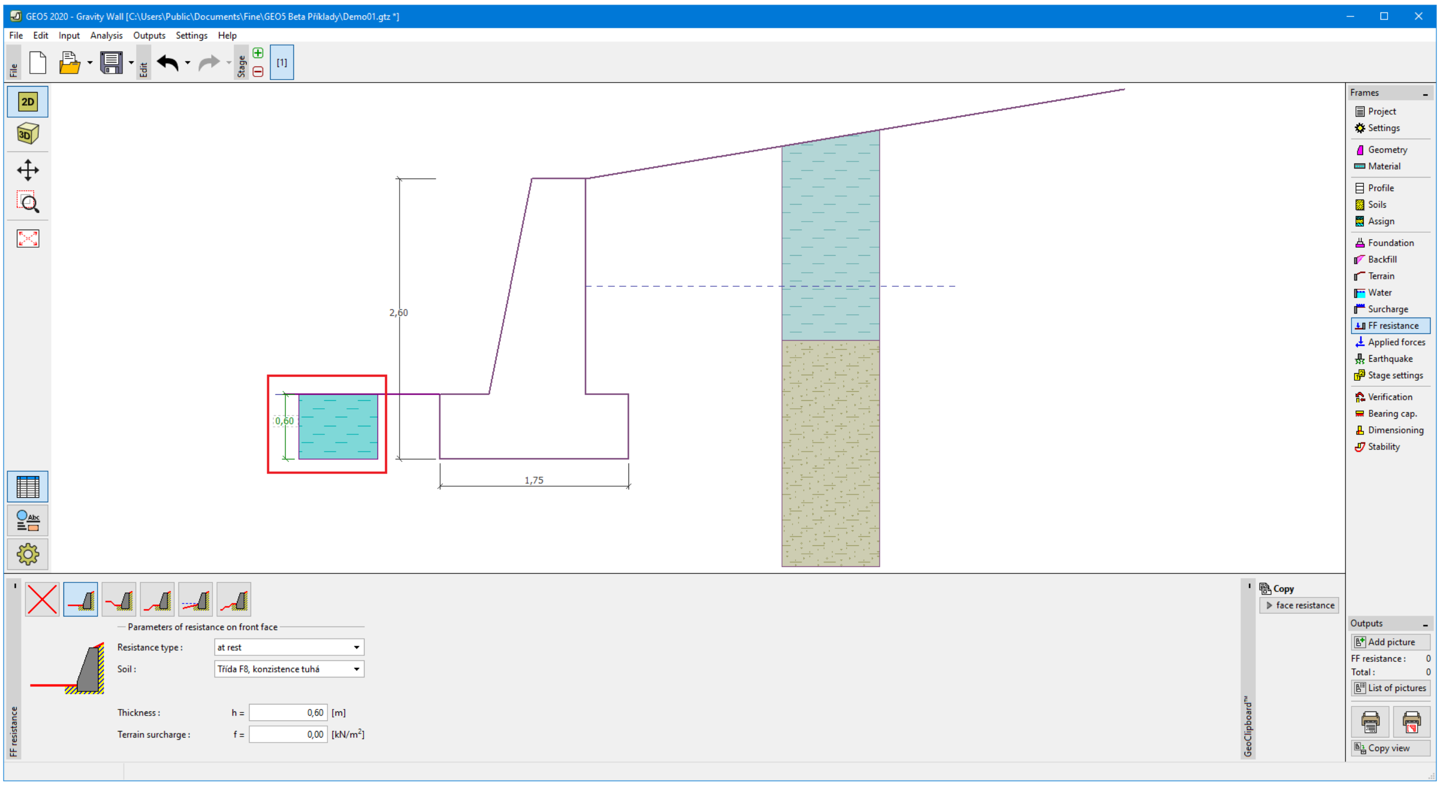
Frame "Surcharge"

Front Face Resistance

The "**Front face resistance**" frame allows us to specify the terrain shape in front of the structure and parameters of front face resistance. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop using active dimensions.

Combo lists in the frame allow selecting the type of resistance and a soil (from the soils introduced in the "Soils" frame). The terrain surcharge magnitude and soil thickness (above the wall lowest points) can also be specified.

The resistance on a structure front face can be specified as the pressure at rest, passive pressure, or reduced passive pressure. The resulting force due to the reduced passive pressure is calculated as a resultant force caused by pressure at rest and passive pressure multiplied by corresponding coefficients.



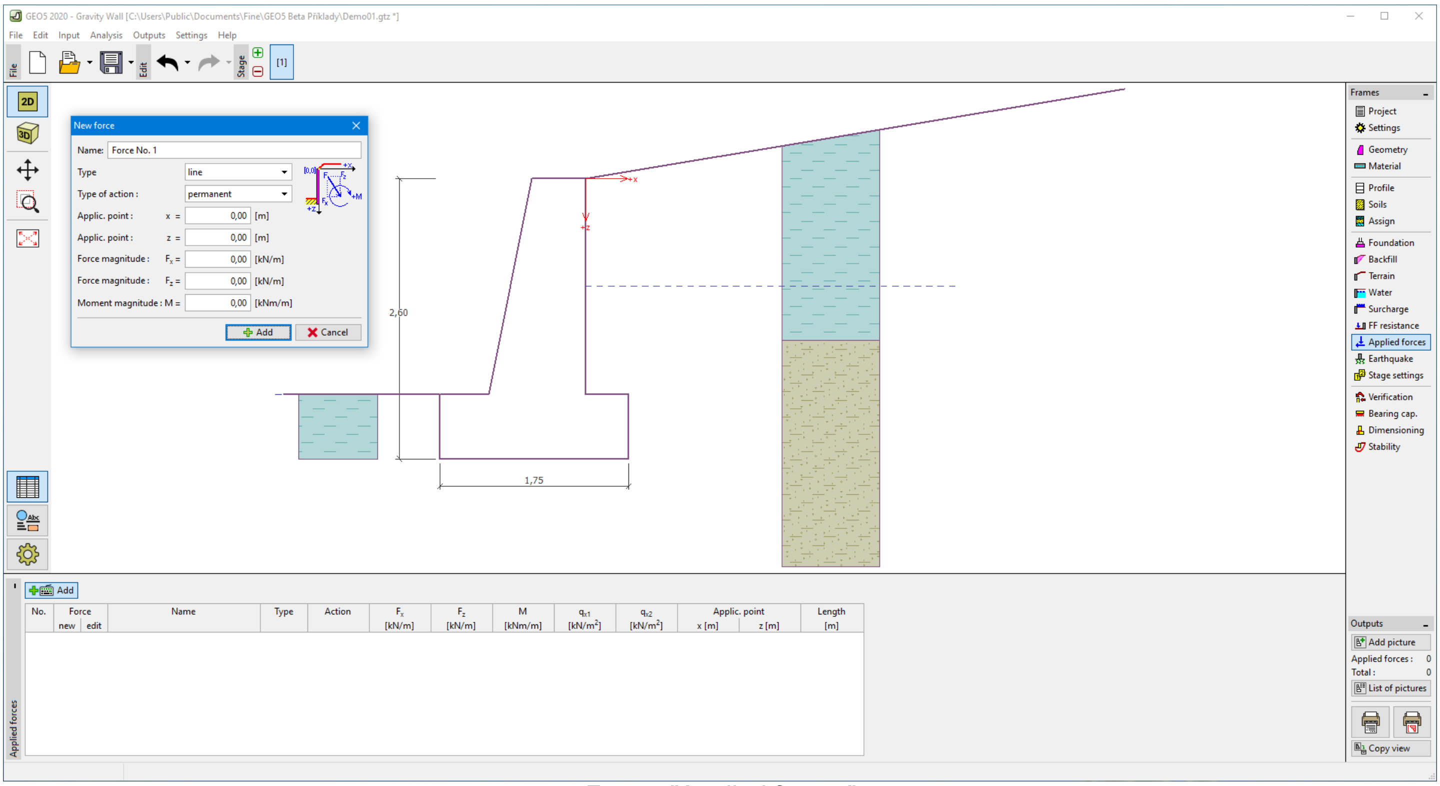
Frame "Front face resistance"

Applied Forces

The "**Applied forces**" frame contains a table with a list of forces acting on a structure. Adding forces is performed in the "**New force**" dialog window. The input forces can also be edited on the desktop by using active objects.

Applied forces represent an additional load on the wall structure, sheeting, or an MSE wall. By using them, we can model e.g. an anchoring crash barrier, a crash vehicle, load from billboards and hoardings etc. The program doesn`t adjust applied forces in the calculation in any way except multiplying them with corresponding coefficients according to the selected load type (EN1997, LRFD).

It is necessary to define external load acting on the ground surface as a surcharge.



Frame "Applied forces"

Anchors

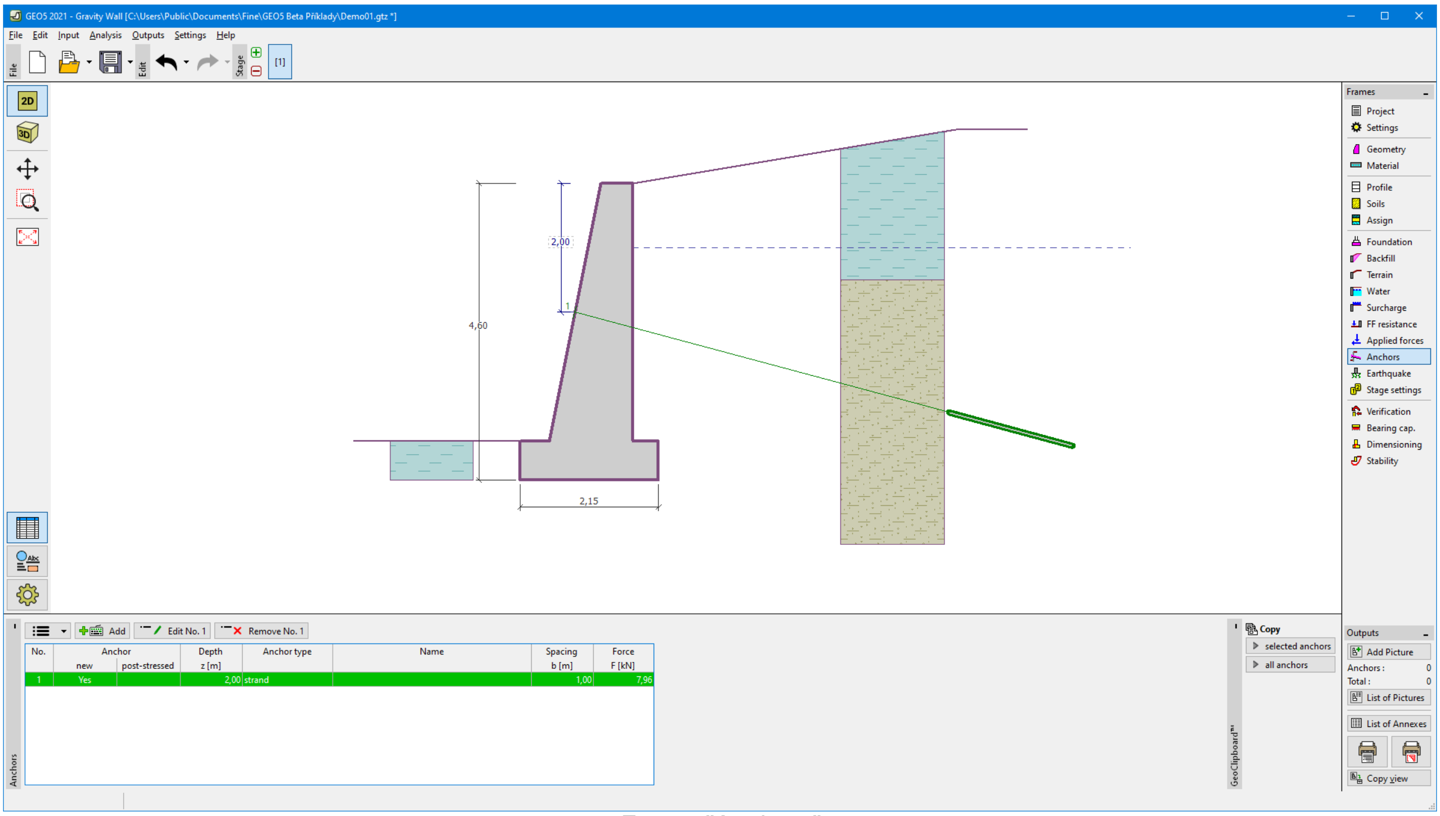
The "**Anchors**" frame contains a table with a list of input anchors. Adding (editing) anchors is performed in the "**New anchor**" dialog window. The input anchors can be edited on the desktop with the help of active objects.

Anchors can be entered as pre-stressed (not specified, prestressed bars, strand) and non-prestressed (helical, non-prestressed bars, deadman).

In the case of **not specified** anchor type, the anchor force is input directly. In other cases, the anchor force is calculated from the anchor strength, pull-out resistance (soil), and pull-out resistance (grouting).

Computed **bearing capacities** of anchors **are reduced** by corresponding safety factor or reduction coefficient, which are defined in the "Settings" frame, "Anchors" tab.

In **subsequent stages** the anchor cannot be changed.



Frame "Anchors"

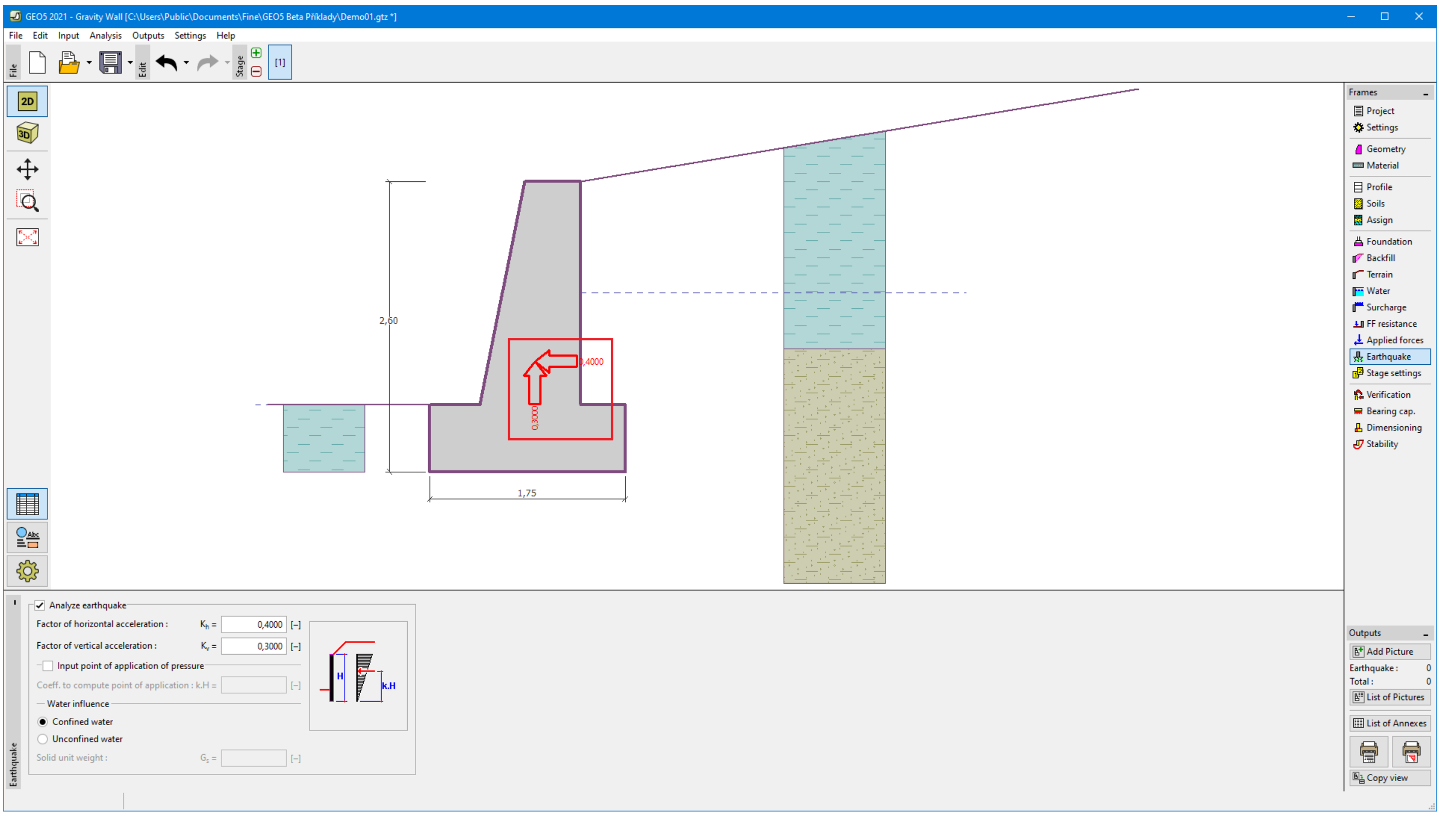
Earthquake

The "**Earthquake**" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements, the coefficients k_h and k_v can be calculated according to EN 1998-5.

The earth pressures analysis with earthquake influence is described in the "Influence earthquake", a theoretical part of the help.

For the LRFD Verification methodology, it is possible to define coefficients for seismic combinations according to the AASHTO.



Frame "Earthquake"

Stage Settings

The "**Stage settings**" frame serves to input settings valid for a given construction stage.

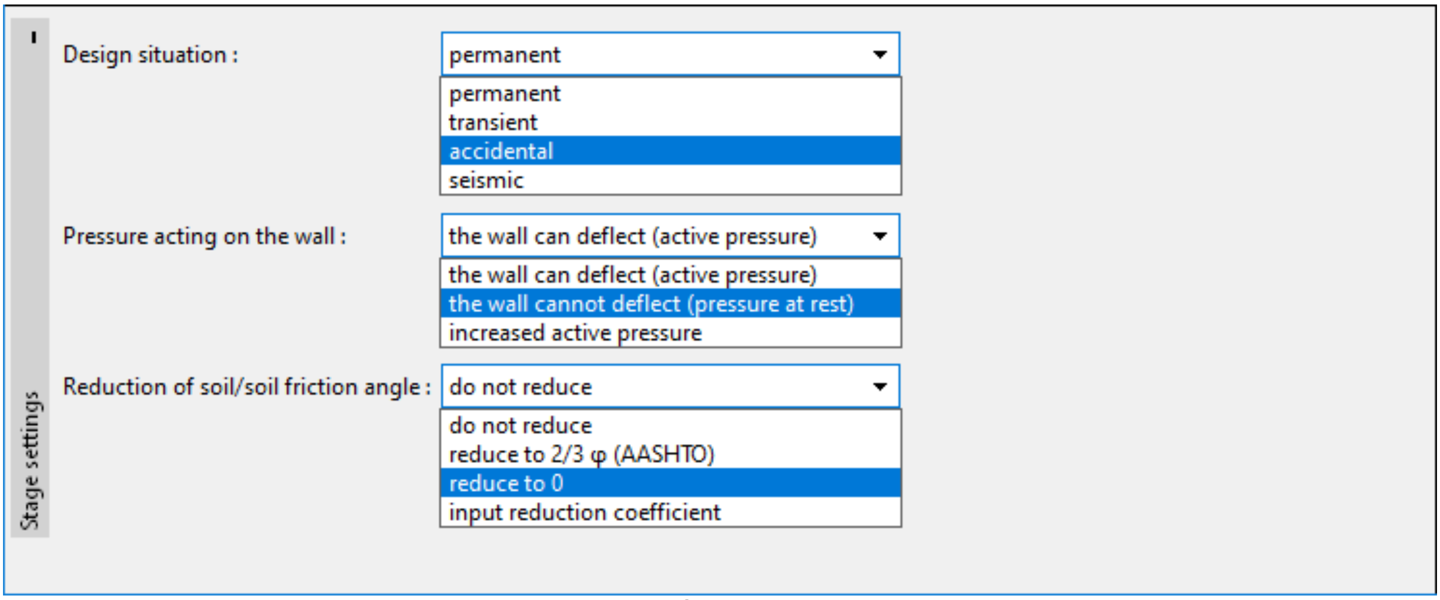
The selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology.

Next, the frame serves to specify the pressure type acting on a wall based on the allowable wall deformation. Providing the wall is free to move, the active pressure is assumed, otherwise, the pressure at rest is used. The third option enables to load the wall by increased active pressure.

The reduction of soil/soil friction angle can be considered in one of following ways:

- **do not reduce**
- **reduce to 2/3φ (AASHTO)**
- **reduce to 0**
- **input reduction coefficient**



Frame "Stage settings"

Verification

The "**Verification**" frame shows the analysis results. Several computations can be carried out for a single task.

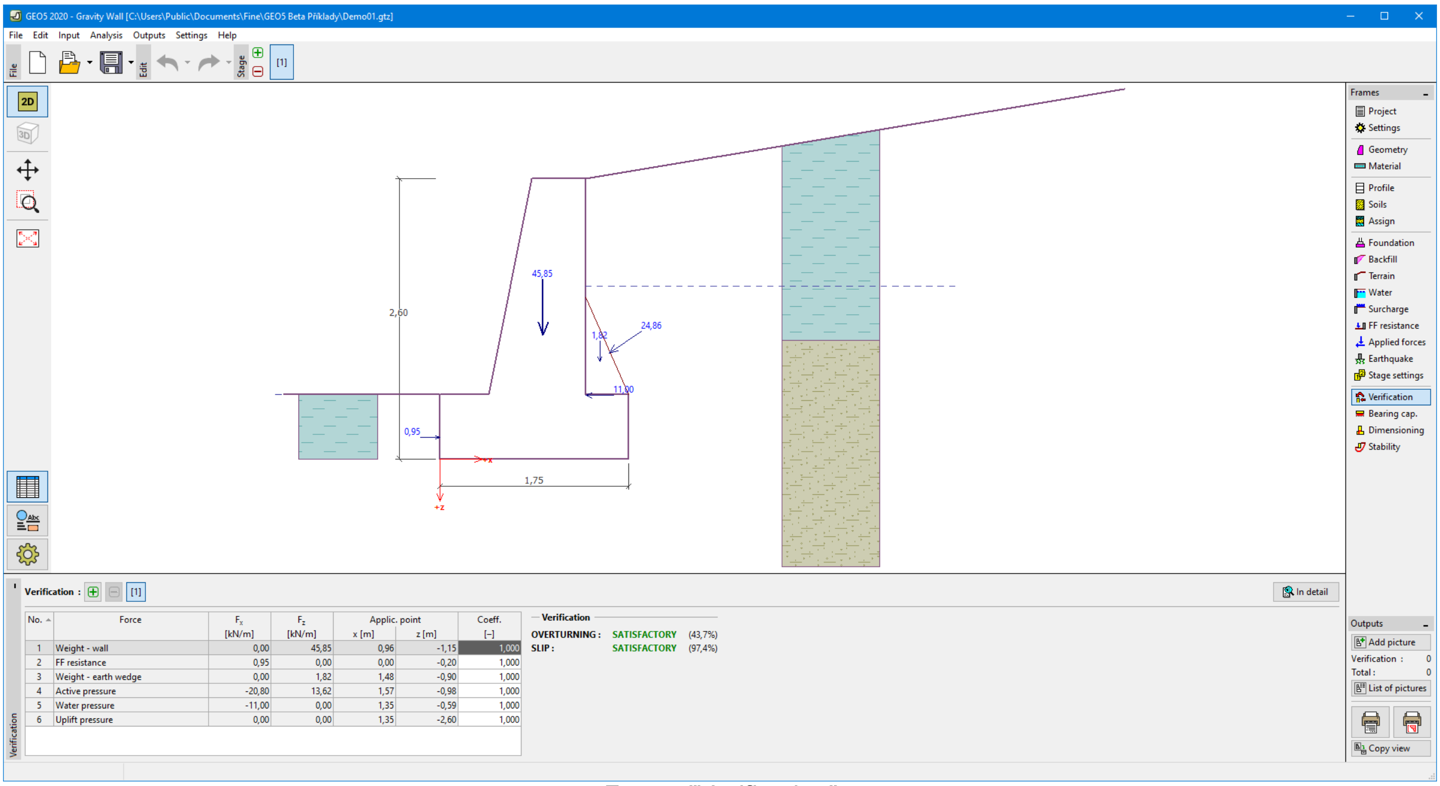
The frame appearance is adjusted based on the selected verification methodology.

- Verification according to the factor of safety or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows us to specify whether the load acting on a structure is considered to be a secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in this case, the last column is not displayed at all.

The procedure for wall verification is described in the theoretical part of the help.

The computed forces are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of wall verification against **overturning** and **translation**. The "**In detail**" button opens the dialog window, which contains a detailed listing of the verification analysis results.

Visualization of results can be adjusted in the "Drawing Settings" frame.



Frame "Verification"

Bearing Capacity

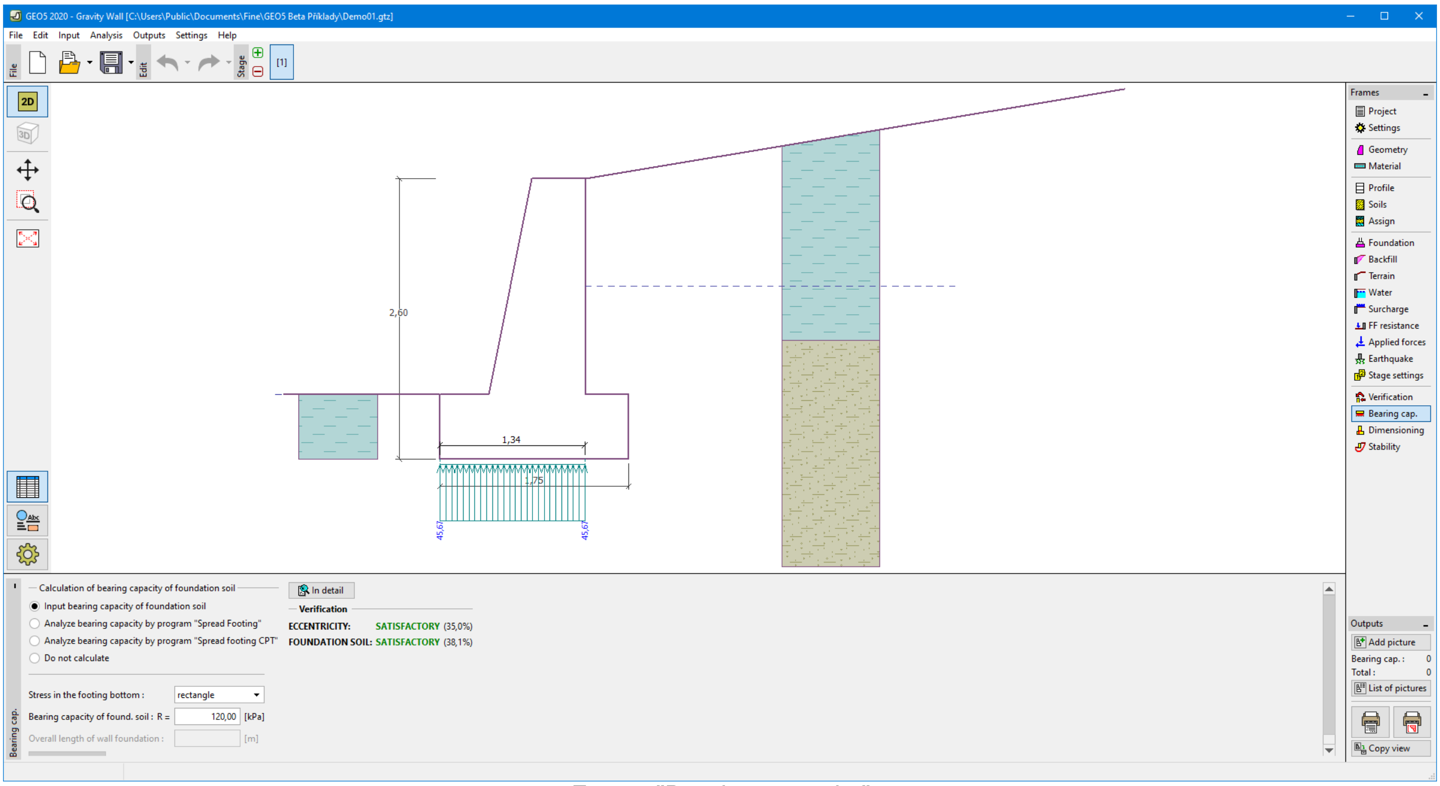
The "**Bearing capacity**" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the "Verification" frame. The "**Spread footing**", "**Spread footing CPT**", "**Pile**" and "**Pile group**" programs consider all verifications as load cases. In the "**Pile CPT**" program, just normal load is used.

The frame contains the following analysis options:

- **Insert bearing capacity of foundation soil** — The foundation soil bearing capacity is input. The **eccentricity** and bearing capacity analysis results are displayed in the right part of the frame. The "**In detail**" button opens a dialog window that displays a detailed listing of the results.
- **Analyze bearing capacity by "Spread footing" program** — Pressing the "**Run program Spread footing**" button opens the program "**Spread footing**" which allows us to calculate the soil bearing capacity or settlement and rotation of the footing. Pressing the "**OK**" button leaves the analysis mode - the results and all plots are transferred into the "**Gravity Wall**" program. The "**Spread footing**" program must be installed for the button to be active. The overall length of the wall foundation is input.
- **Analyze bearing capacity by "Spread footing CPT" program** — The procedure is identical as if calculating soil bearing capacity by the **"Spread footing"** program.
- **Analyze bearing capacity by "Pile" program** — The procedure is identical as if calculating soil bearing capacity by the **"Spread footing"** program. The **"Run program Pile"** is available if the wall has a pile foundation (frame "Foundation"). Pile spacing s is input.
- **Analyze bearing capacity by "Pile CPT" program** — The procedure is identical as if calculating soil bearing capacity by the "**Pile**" program.
- **Analyze bearing capacity by "Pile group" program** — The procedure is identical as if calculating soil bearing capacity by the "**Spread footing**" program. The "**Run program Pile group**" is available if the wall has a pile foundation with more than one pile ("Foundation" frame). Pile spacing s , a total number of pile rows n and loading length l are input.
- **Do not calculate (pile footing)** — The foundation soil bearing capacity is not calculated.

The program allows us to specify a shape of stress in the footing bottom.

Visualization of results can be adjusted in the "Drawing Settings" frame.



Frame "Bearing capacity"

Dimensioning

The "**Dimensioning**" frame serves to design and verify the reinforcement of the wall cross-section - the cross-section subjected to dimensioning is selected in the combo list.

- **Wall stem verification**
- **Construction joint verification** - **depth** of a construction joint from the construction top edge is specified
- **Wall jump verification**

The frame appearance is adjusted based on the selected verification methodology.

- Verification according to the factor of safety or the theory of limit states - the last column in the table allows us to input the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows us to specify whether the load acting on a structure is considered to be a secondary one. This is explained in more detail in the "Load combinations" section.
- Analysis according to LRFD - in this case, the last column is not displayed.

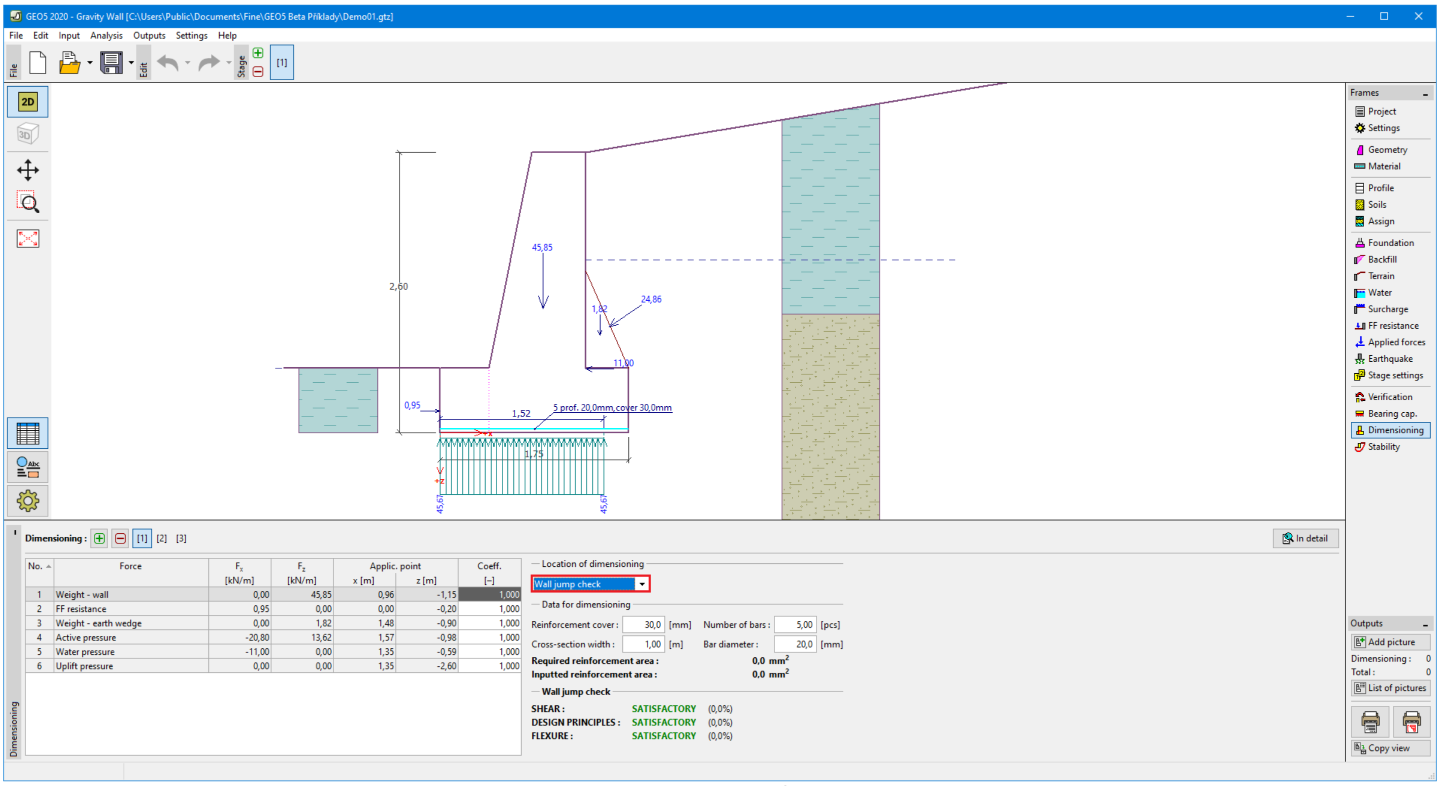
Calculation of forces and their action on the analyzed cross-section is described here. The wall stem and construction joint are always loaded by the pressure at rest.

The way to calculate the internal forces in individual cross-sections is described in the theoretical part of this documentation.

Dimensioning of the reinforced concrete is performed according to the standard set in the "Materials and standards" tab.

Several computations for different reinforcement variants can be carried out. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "**In detail**" button opens the dialog window that contains a detailed listing of the dimensioning results.

Visualization of results can be adjusted in the "Drawing Settings" frame.

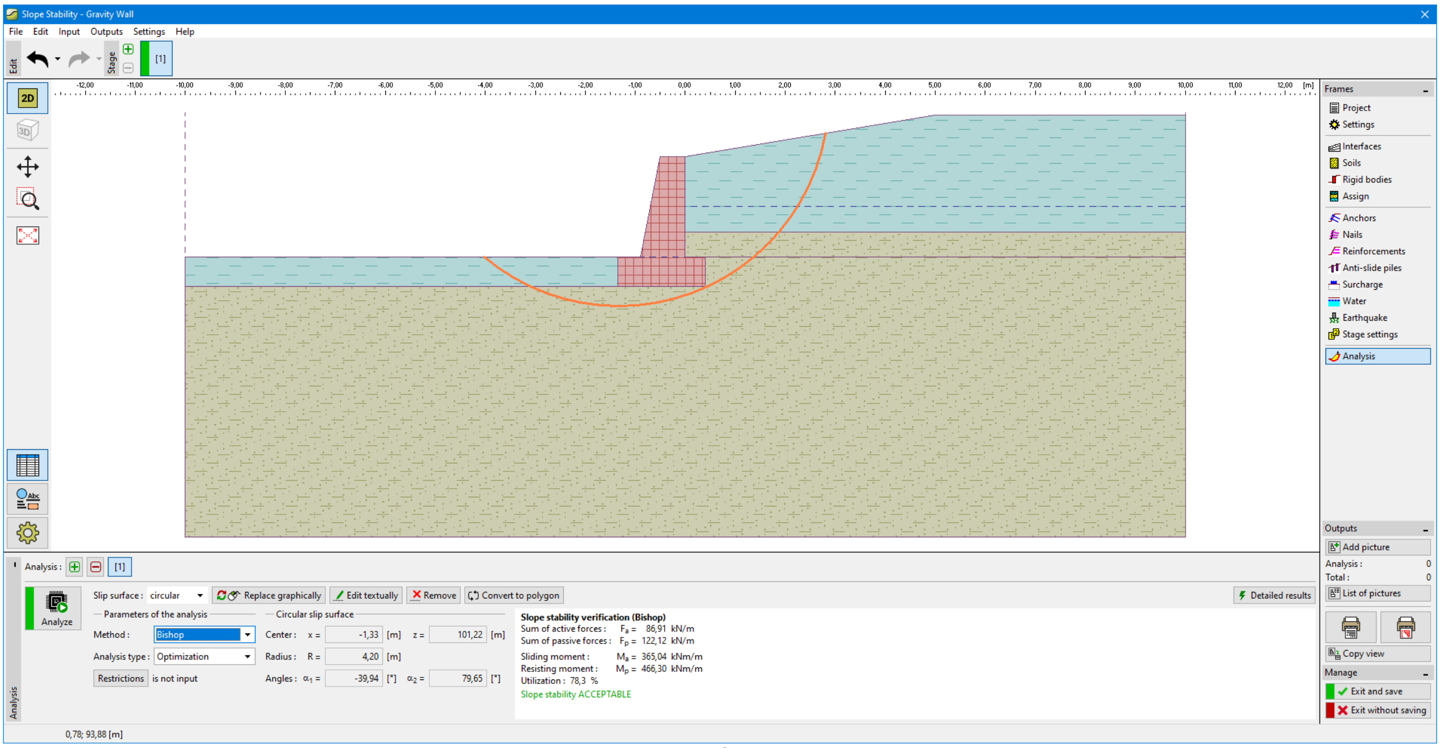


"Dimensioning" frame

Stability

Pressing the "**Stability**" button launches the "**Slope stability**" program. This program allows us to check the overall stability of the analyzed structure. The button is available only if the "**Slope stability**" program is installed.

After completing all analyses, press the "**Exit and save**" button to leave the program - all data are carried over to the analysis protocol of the "**Gravity wall**" program.



Frame "Stability"

Program Prefab Wall

This program is used to verify retaining walls made of prefabricated blocks.

The help in the program "Prefab Wall" includes the following topics:

- The input of data into individual frames:

Project	Settings	Geometry	Profile	Soils	Assign	Foundation
Backfill	Terrain	Water	Surcharge	Front Face Resistance	Applied Forces	Earthquake
Stage Settings	Verification	Bearing Capacity	Dimensioning	Slip on Georeinforcement	Stability	

- Standards and analysis methods

- Theory for analysis in the program "**Prefab Wall**":

Stress in Soil Body	Earth Pressures	Analysis of Walls	Analysis of Foundation Bearing Capacity

- Outputs

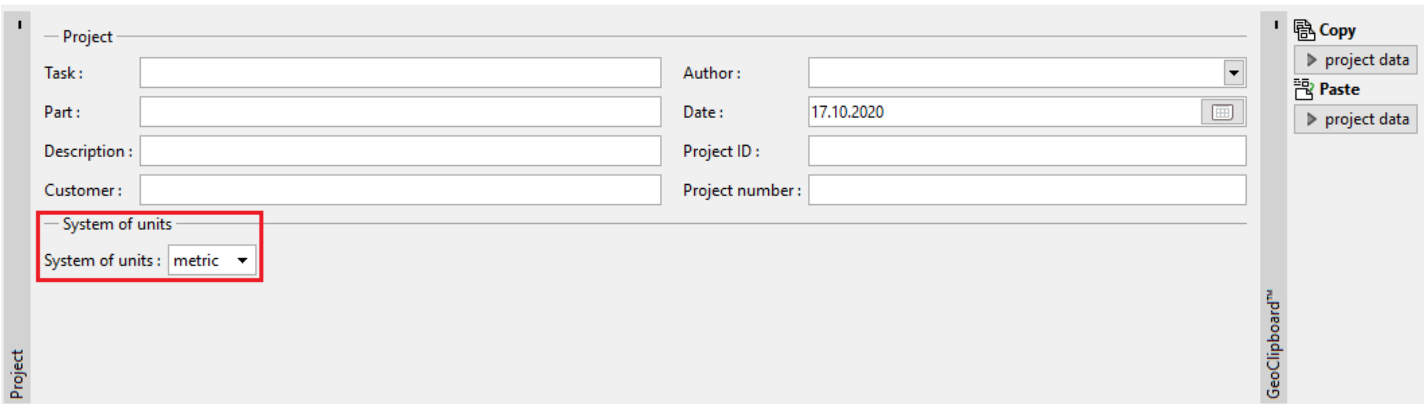
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The frame **"Project"** is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The frame "**Settings**" allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

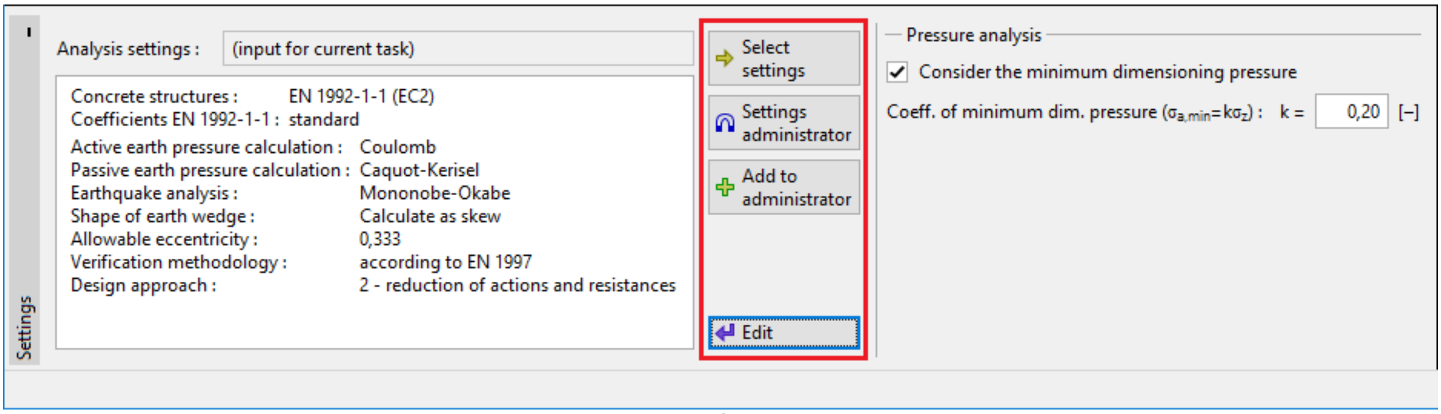
The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local**

setting. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

The program allows us to specify a value of the minimum dimensioning pressure (by checking the option "**Consider the minimum dimensioning pressure**").

Settings of analysis parameters are performed in the "Materials and standards" and "Wall analysis" tabs.



Frame "Settings"

Geometry

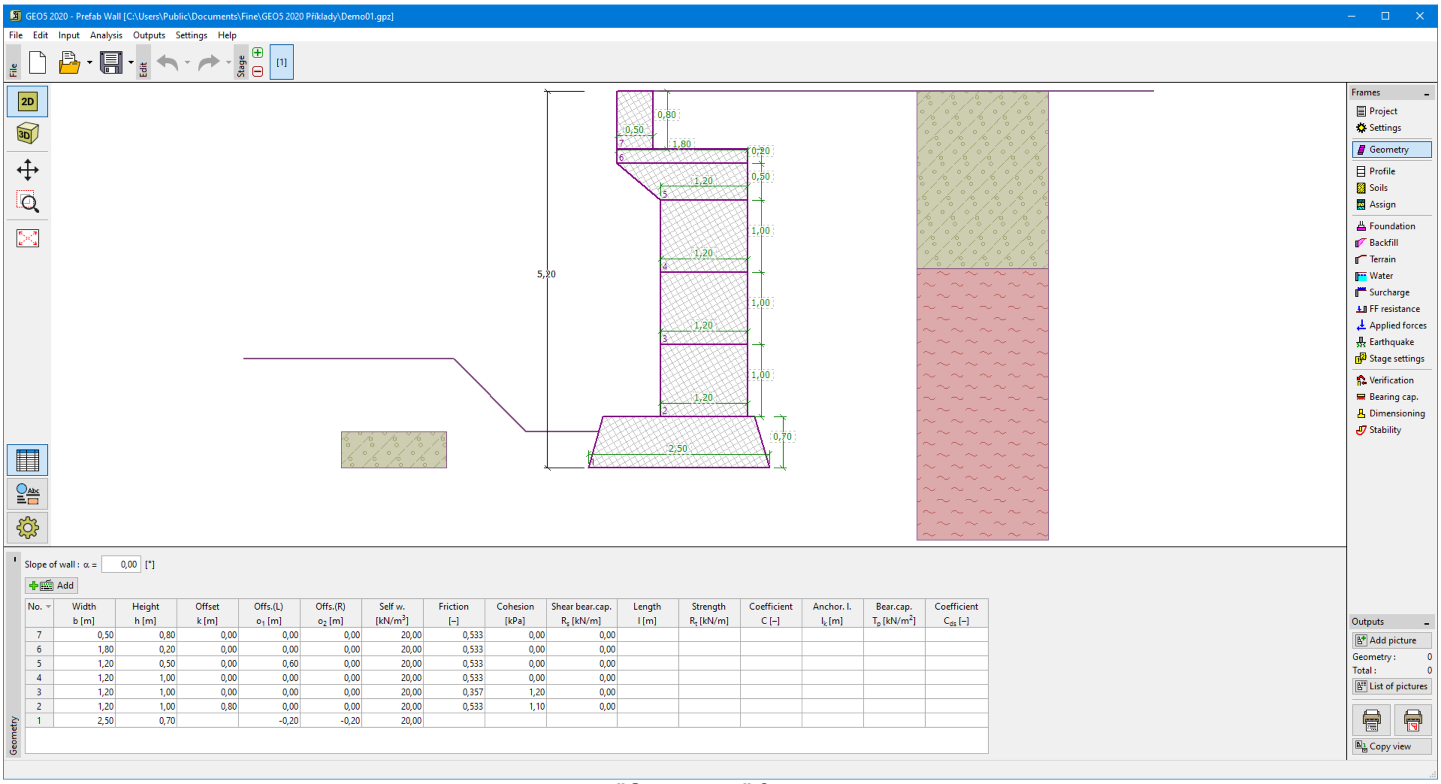
The "**Geometry**" frame contains a table with a list of input structural precast units (blocks) of a wall (the lowest block is labeled as No. 1). Adding blocks is performed in the "**New block**" dialog window.

This dialog window serves to define **the geometry of a block**, parameters of reinforcement (length, anchorage length, tensile strength and pull out resistance) and **material characteristics** (self-weight, shear resistance between two blocks, cohesion). The **merge blocks** option is used for more detailed block geometry modelling.

The program allows us to add (insert) another block in between two already existing blocks of a structure. Inserting a new block is performed in the "**Insert block**" dialog window that complies with the "**New block**" dialog window.

The input blocks can be further edited on the desktop with the help of active dimensions or active objects - double-clicking on a structure opens a dialog window with a given block. **When using the regime of active objects the visualization of detailed dimensions must be turned in the "Drawing Settings" frame.**

The program makes it possible to export the geometry of a structure in the *.DXF format.



"Geometry" frame

Profile

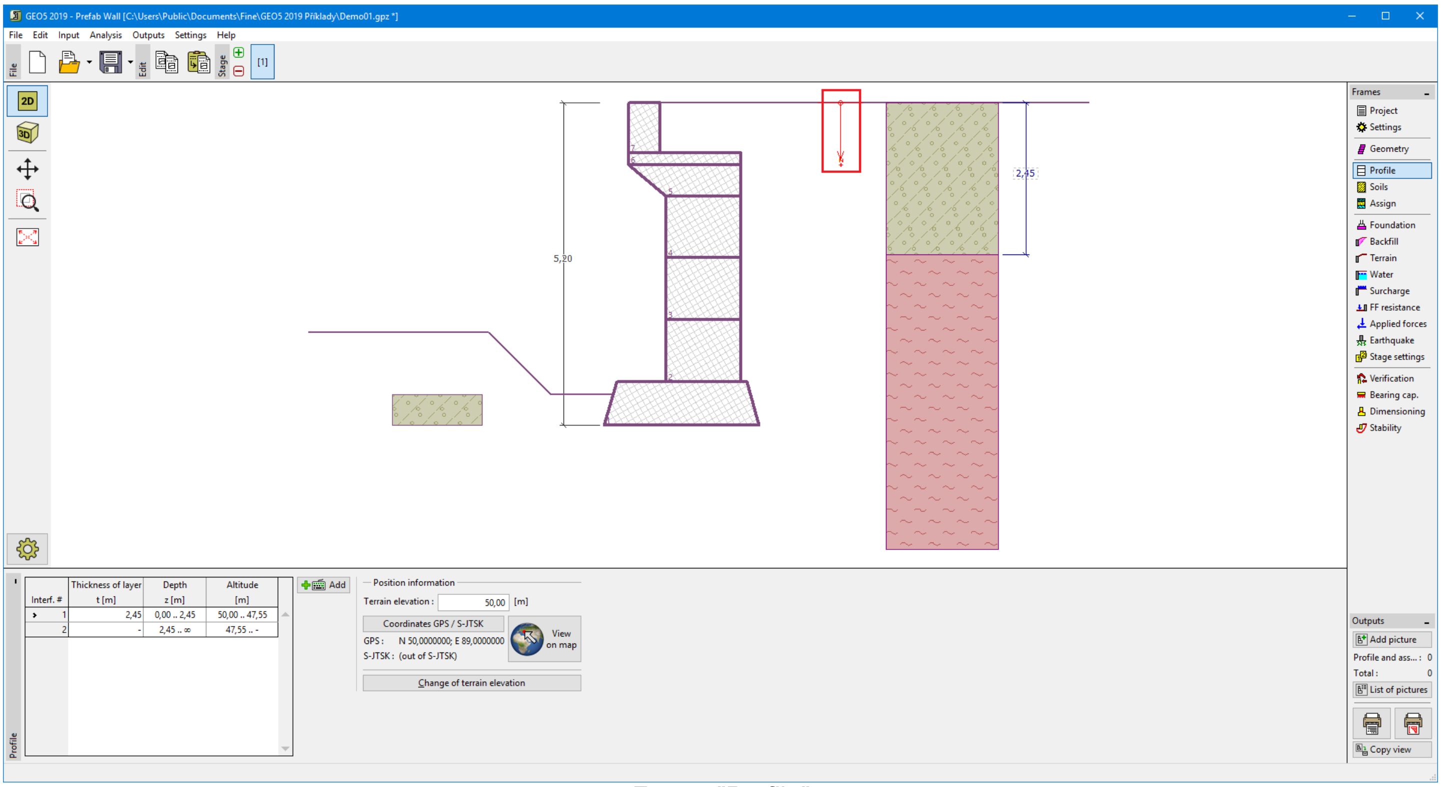
The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.



Frame "Profile"

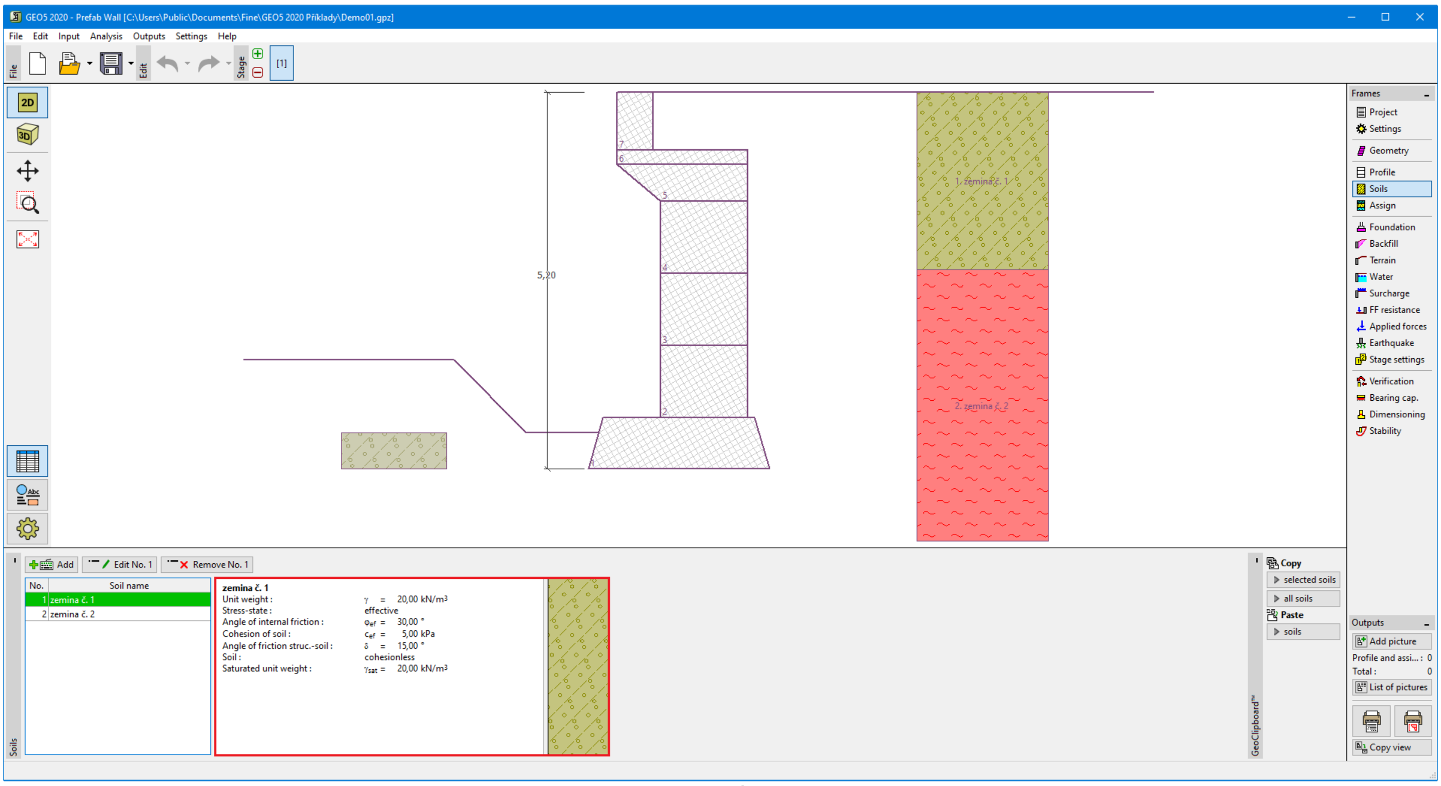
Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

Basic Data

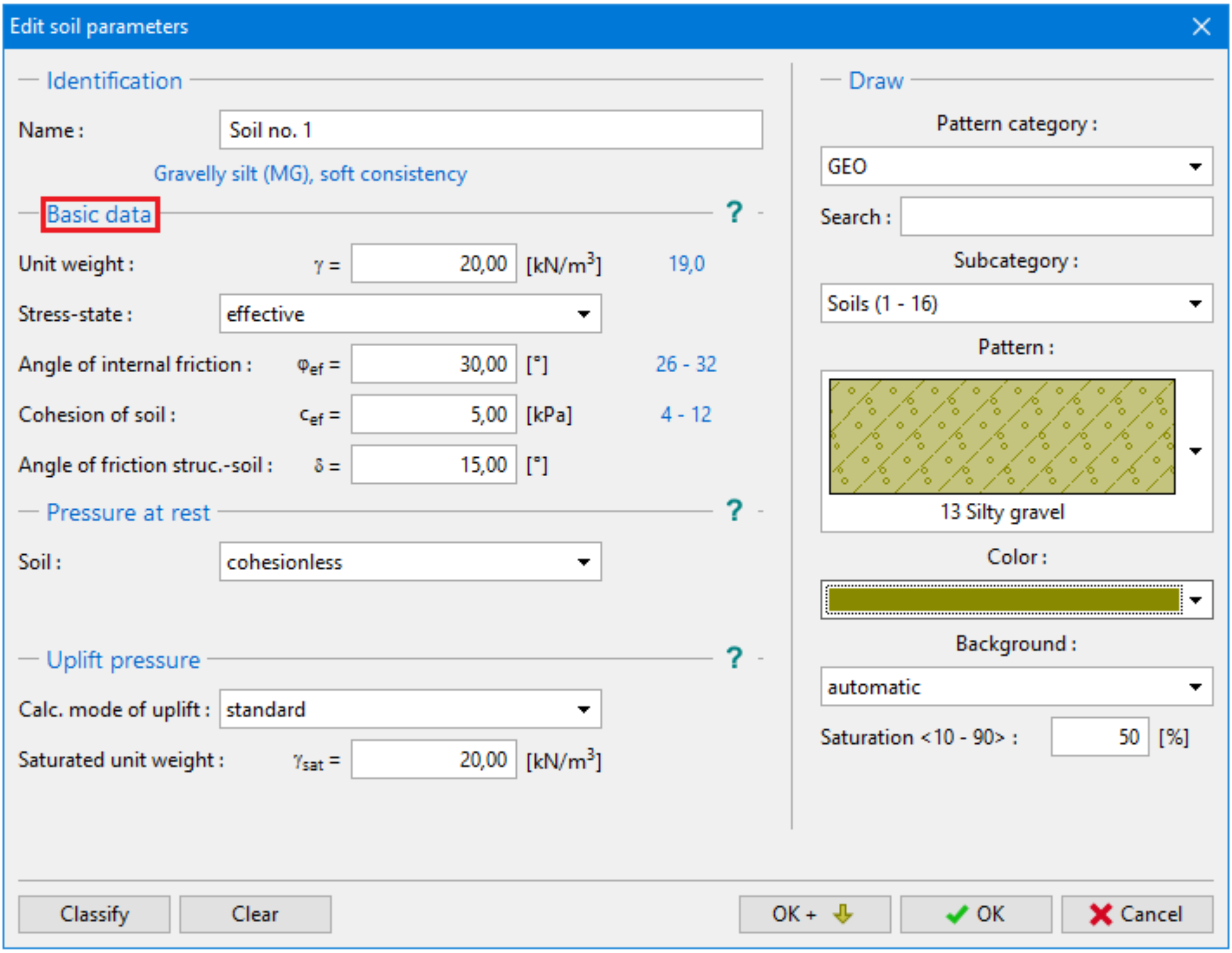
This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction, and cohesion**. The particular values are obtained from a geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit the built-in database of soils, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually.

Either **effective or total** parameters of the angle of internal friction and cohesion are specified depending on the setting in the "**Stress analysis**" combo list. Whether to use effective or total parameters depends primarily on the type of soil, type of load, structure duration, and water conditions.

For effective stress further needs to specify the angle of internal friction between the soil and structure, which depends on the structure material and the type of soil. Possible values of this parameter are listed in the table of recommended values.

For total stress further needs to specify the adhesion of soil to the structure face a .

The associated theory is described in detail in chapter "Earth pressures".

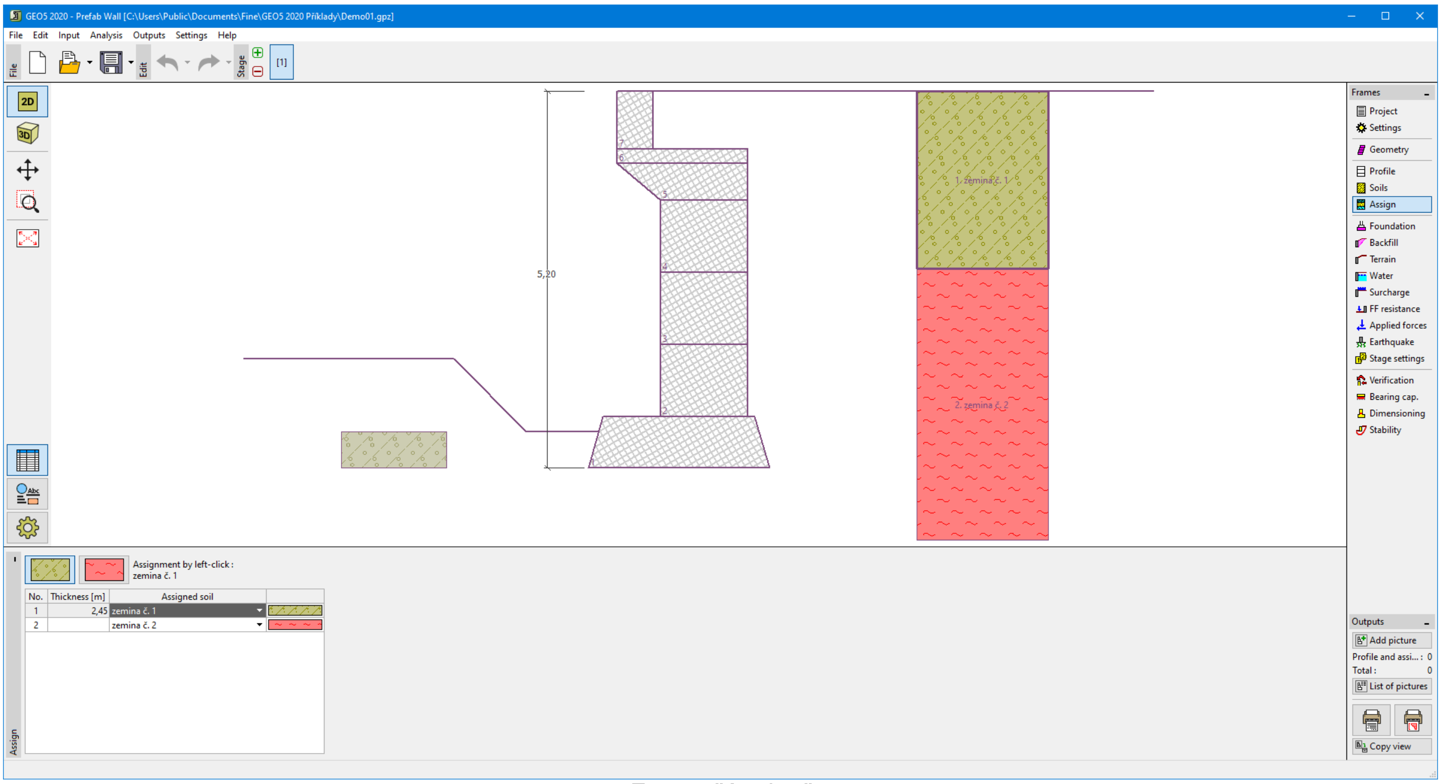


Dialog window "Add new soils" - "Basic data"

Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail herein.



Frame "Assign"

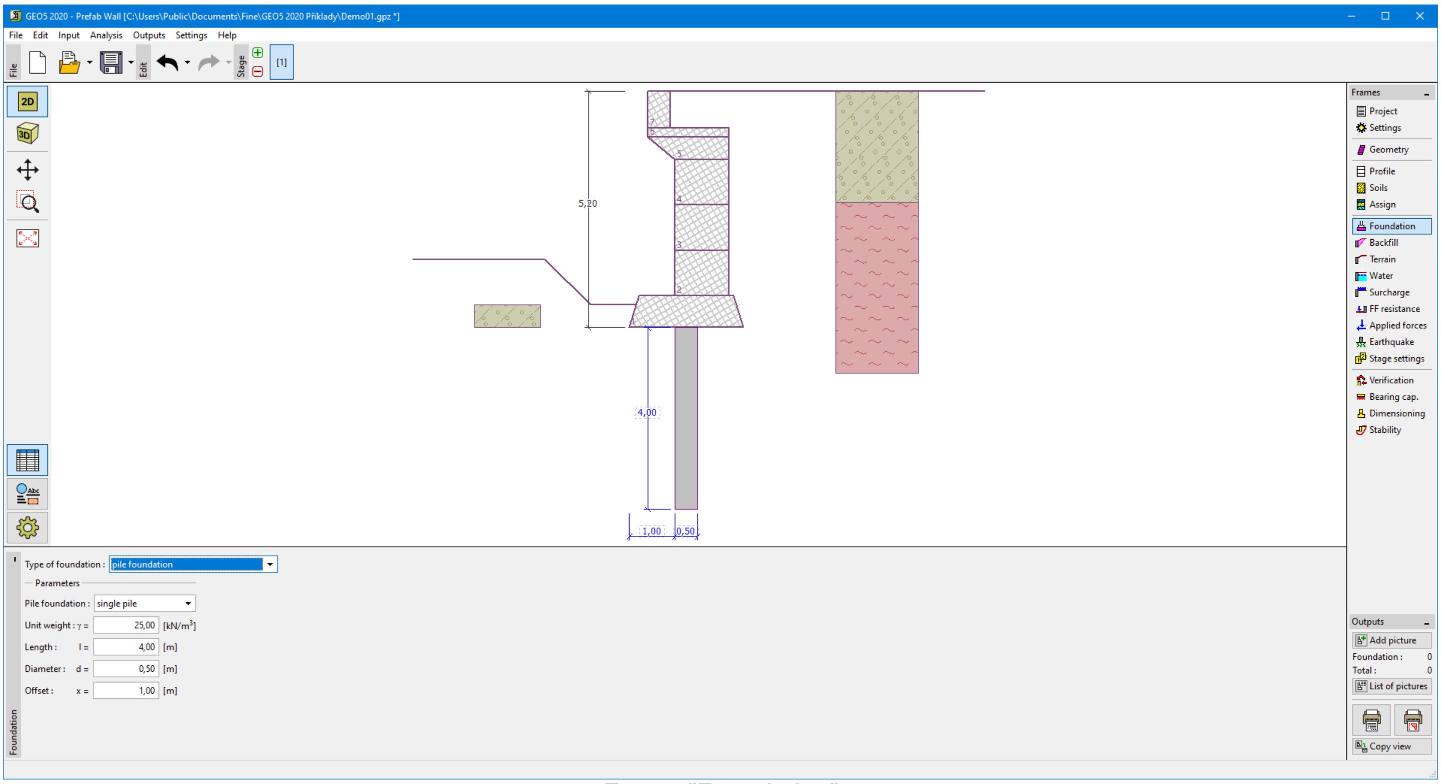
Foundation

The "**Foundation**" frame serves to specify the type of wall foundation. The following types of wall foundation are available:

- **soil from geological profile** - the wall is founded on the soil assigned from the geological profile specified in the "Profile" frame.
- **input parameters of contact base-soil** - parameters of the contact between footing bottom and structure are specified. Option "**input angle of friction base-soil**" requires inputting the friction angle ψ [$°$] between foundation and soil. Option "**input friction coefficient**" requires specifying the friction coefficient μ [**-**]. Both options require inputting the cohesion a [kPa] between foundation (base) and soil.
- **strip foundation** - strip foundation material is represented either by **soil** (input in "Soils" frame), or concrete - requires inputting the **unit weight of foundation material** γ and **parameters of contact base-soil** (friction coefficient f, cohesion c, additional resistance F).
- **pile foundation** - the wall can be founded on one row of piles or two rows of piles, respectively.

Strip foundation and **pile foundation** can be adopted for the wall foundation only if the type wall with **straight footing bottom without jump** is selected in the "Geometry" frame. The geometry of the wall foundation (**strip foundation**, **pile foundation**) can be modified either in the frame by entering specific values into the inputting fields or on the desktop with the help of active dimensions.

The input data introduced in this frame influence the actual wall analysis (check for slip) and further the bearing capacity of foundation soil.

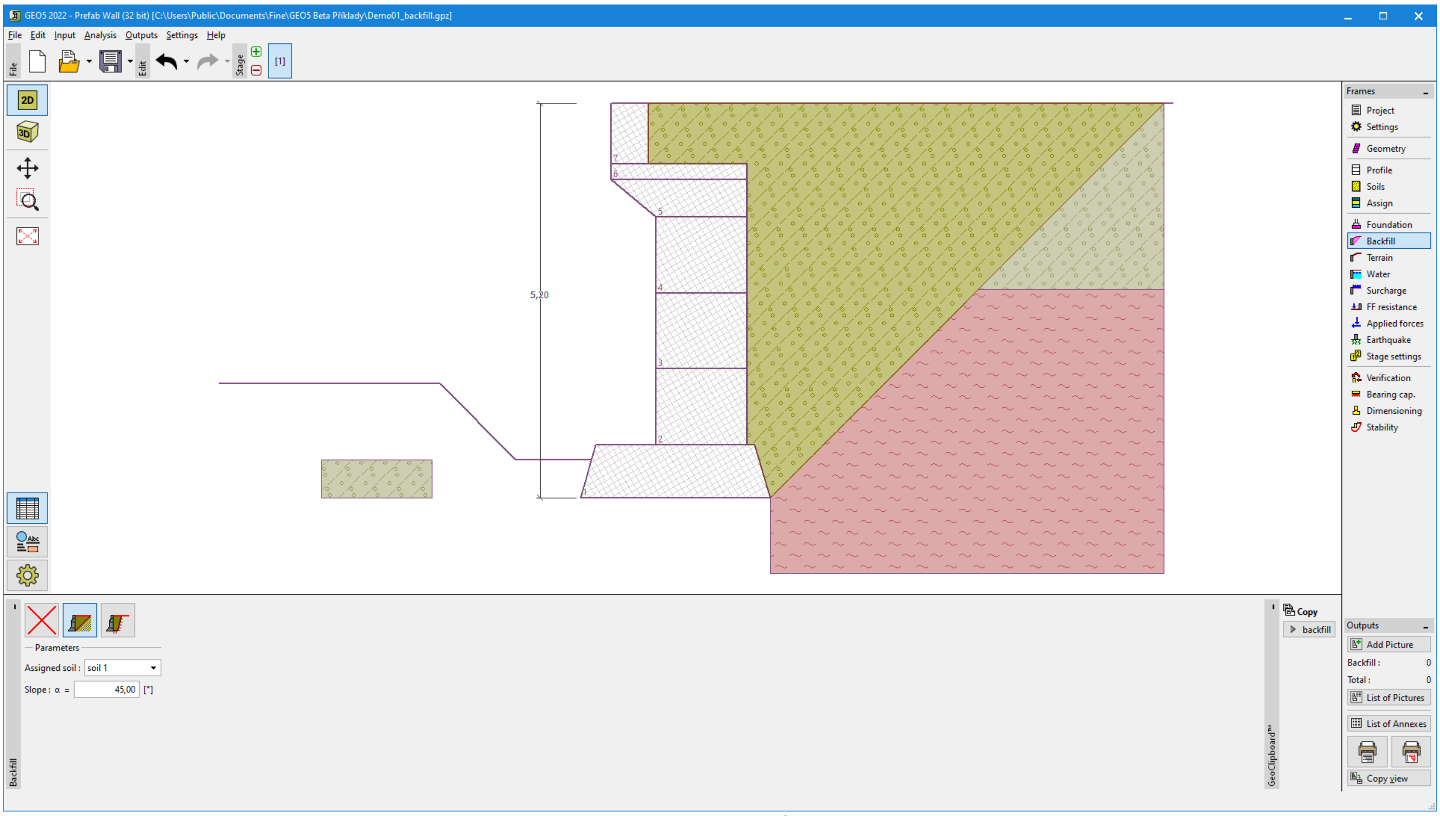


Frame "Foundation"

Backfill

The frame "**Backfill**" allows the selection of backfills behind the structure.

Analysis of earth pressures with the influence of backfill is described in the theoretical part of the help: "Influence of Backfill".



Frame "Backfill"

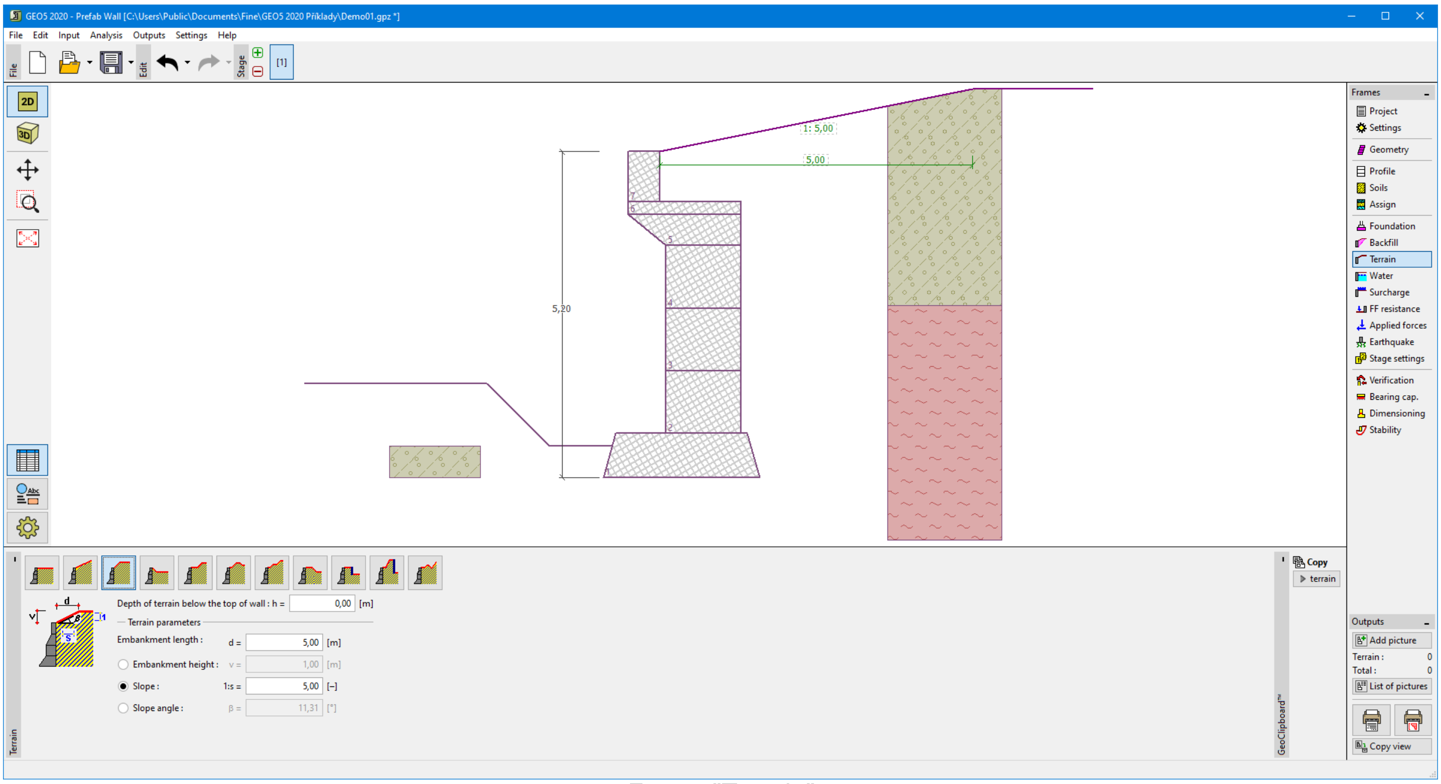
Terrain

The "**Terrain**" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by

inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case, the frame contains a table with a list of terrain points. The first point with coordinates [0; 0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "Distribution of earth pressures for broken terrain".



Frame "Terrain"

Water

The "**Water**" frame allows, by pressing the button, for selecting the type of water. The selected type with a graphic hint of input values is displayed in the left part of the frame. Water parameters (h_1 , h_2 ...) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

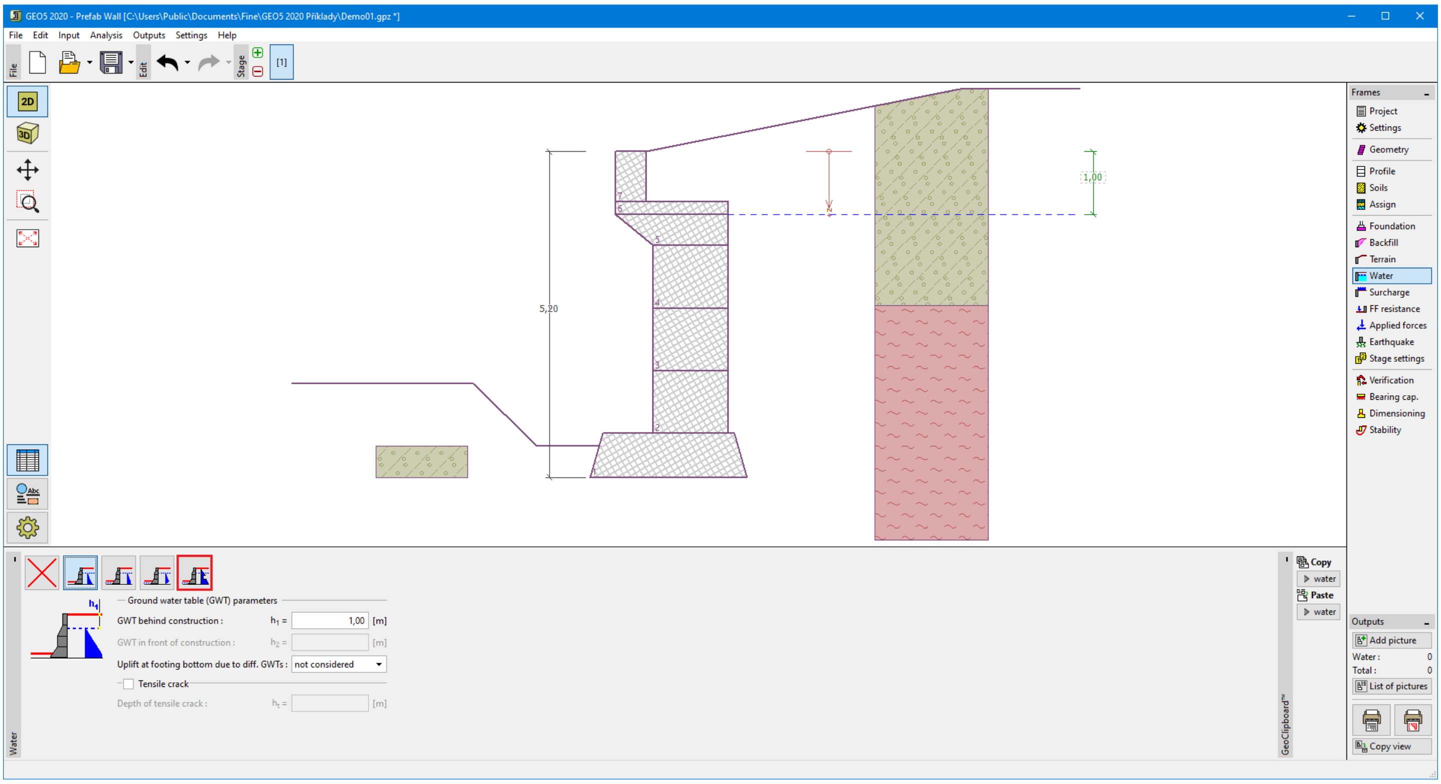
The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be linear, parabolic, or it may not be considered at all. When verifying the wall, the uplift pressure in foundation joint due to different water tables is introduced in terms of a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tabs "**In front of structure**" and "**Behind structure**" appear with tables. The table contains the values of pore pressure in front of the structure or behind it.

The groundwater table can also be specified **above the structure** or earth profile, respectively - in such a case, the depth of water is input with a negative value.

Analysis of earth pressures with the influence of water is described in the theoretical part of the help, chapter "Influence of water".

The program further allows for specifying a depth of tensile cracks filled with water.



Frame "Water"

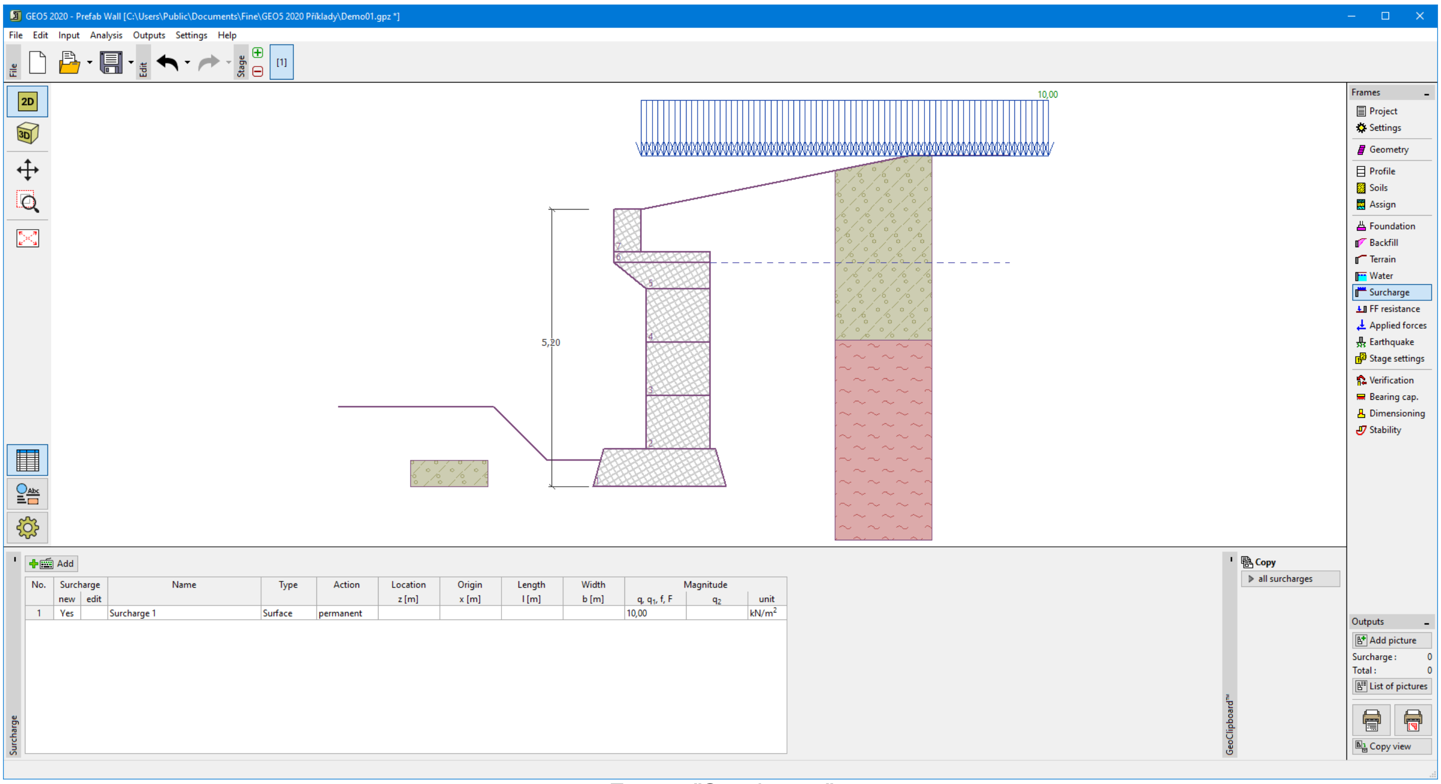
Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding (editing) surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The z -coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case when the surcharge is found out of the terrain the program prompts an error message before calculation.

Either **permanent**, **variable,** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with a favorable effect is not considered in the analysis.

Analysis of earth pressures due to surcharges is described in the theoretical part of the help, chapter "Influence of surcharge".



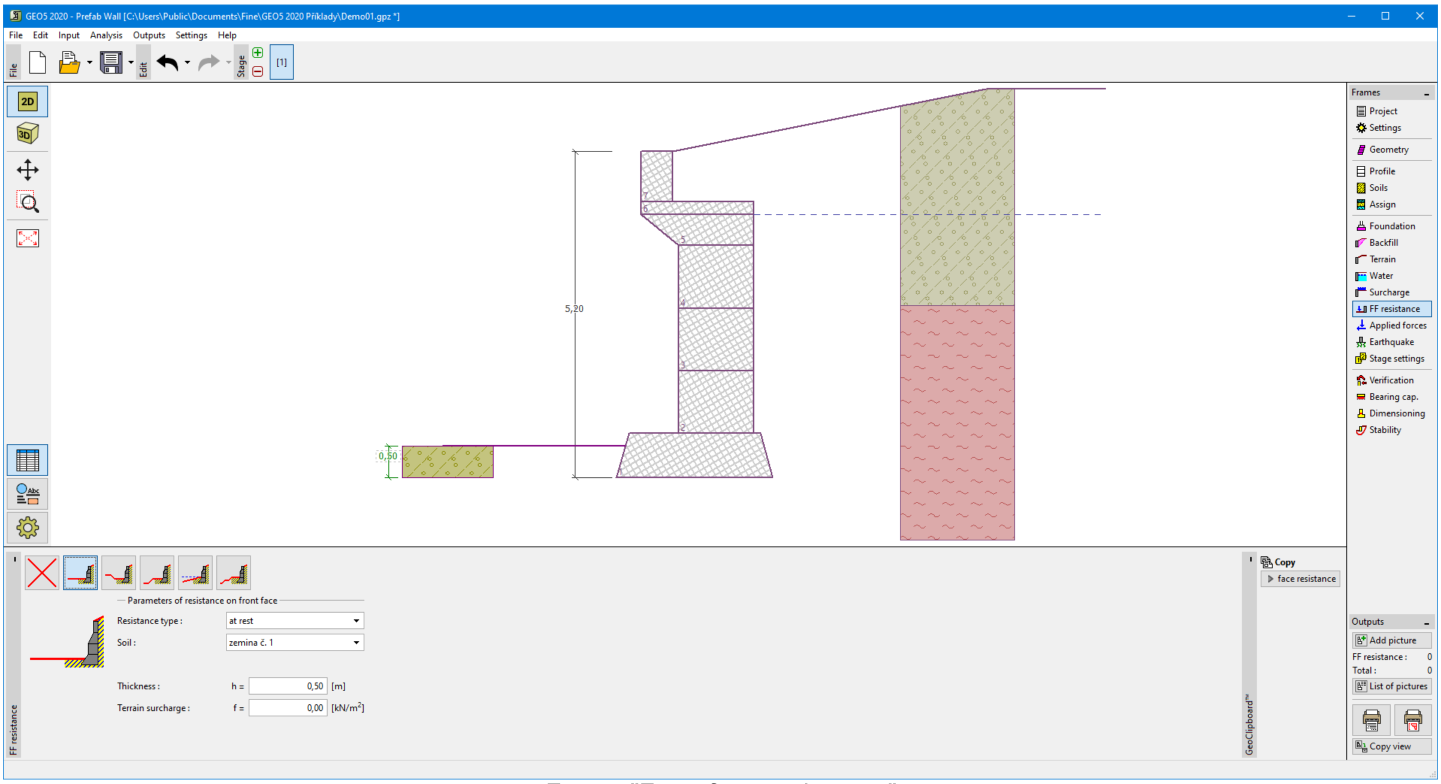
Frame "Surcharge"

Front Face Resistance

The "**Front face resistance**" frame allows us to specify the terrain shape in front of the structure and parameters of front face resistance. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop using active dimensions.

Combo lists in the frame allow selecting the type of resistance and a soil (from the soils introduced in the "Soils" frame). The terrain surcharge magnitude and soil thickness (above the wall lowest points) can also be specified.

The resistance on a structure front face can be specified as the pressure at rest, passive pressure, or reduced passive pressure. The resulting force due to the reduced passive pressure is calculated as a resultant force caused by pressure at rest and passive pressure multiplied by corresponding coefficients.



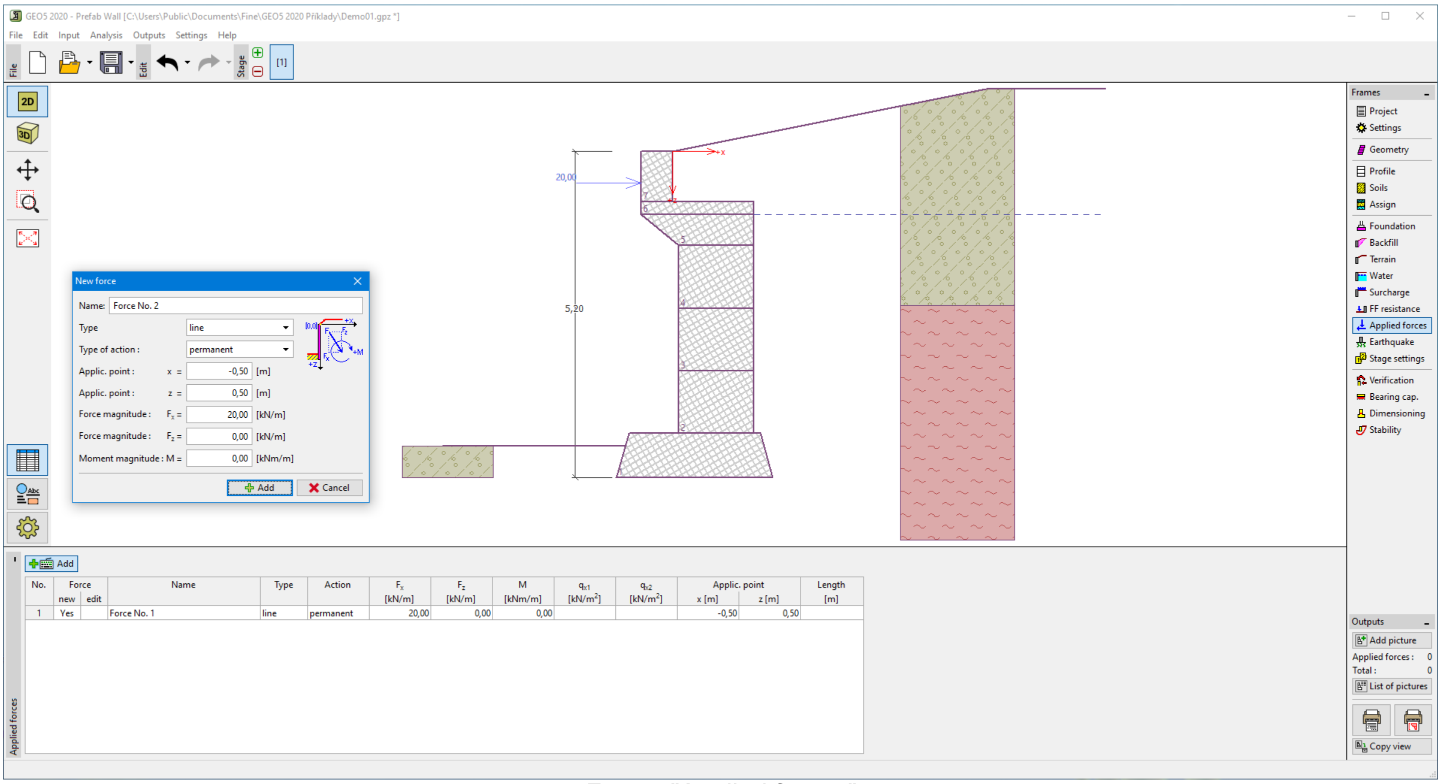
Frame "Front face resistance"

Applied Forces

The "**Applied forces**" frame contains a table with a list of forces acting on a structure. Adding forces is performed in the "**New force**" dialog window. The input forces can also be edited on the desktop with the help of active objects.

Applied forces represent an additional load on the structure of the wall, sheeting or MSE wall. We can model such as an anchoring crash barrier, crash vehicle, load from billboards and hoardings etc. The program does not modify the applied forces in the calculation except multiplying them with the corresponding coefficients according to the selected type of load (EN1997, LRFD).

External load acting on the terrain is necessary to define as a surcharge.



Frame "Applied forces"

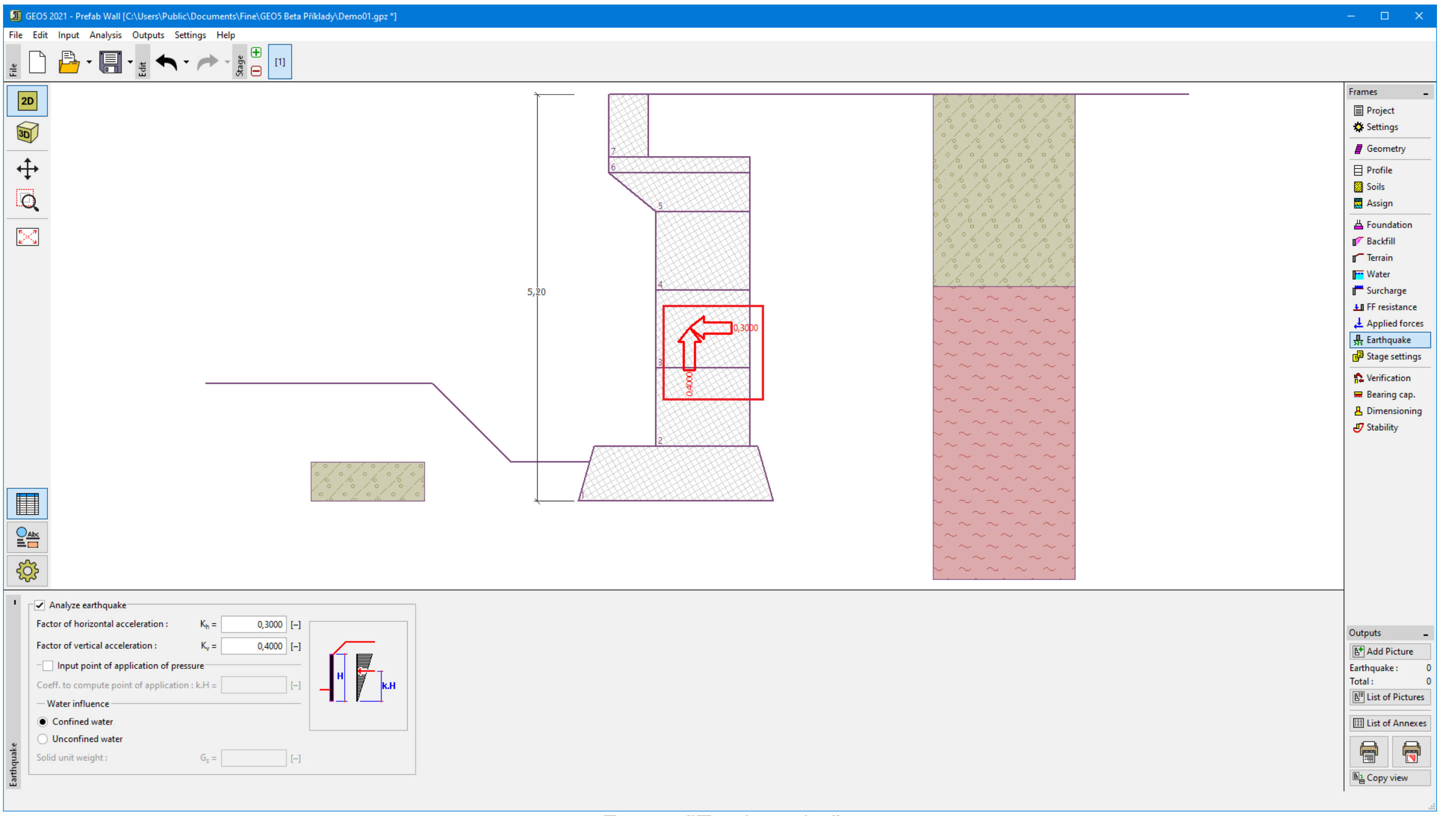
Earthquake

The "**Earthquake**" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements, the coefficients k_h and k_v can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for an earthquake is described in the theoretical part of the help, chapter "Influence earthquake".

For the LRFD Verification methodology, it is possible to define coefficients for seismic combinations according to the AASHTO.



Frame "Earthquake"

Stage Settings

The frame "**Stage settings**" serves to input settings valid for a given construction stage.

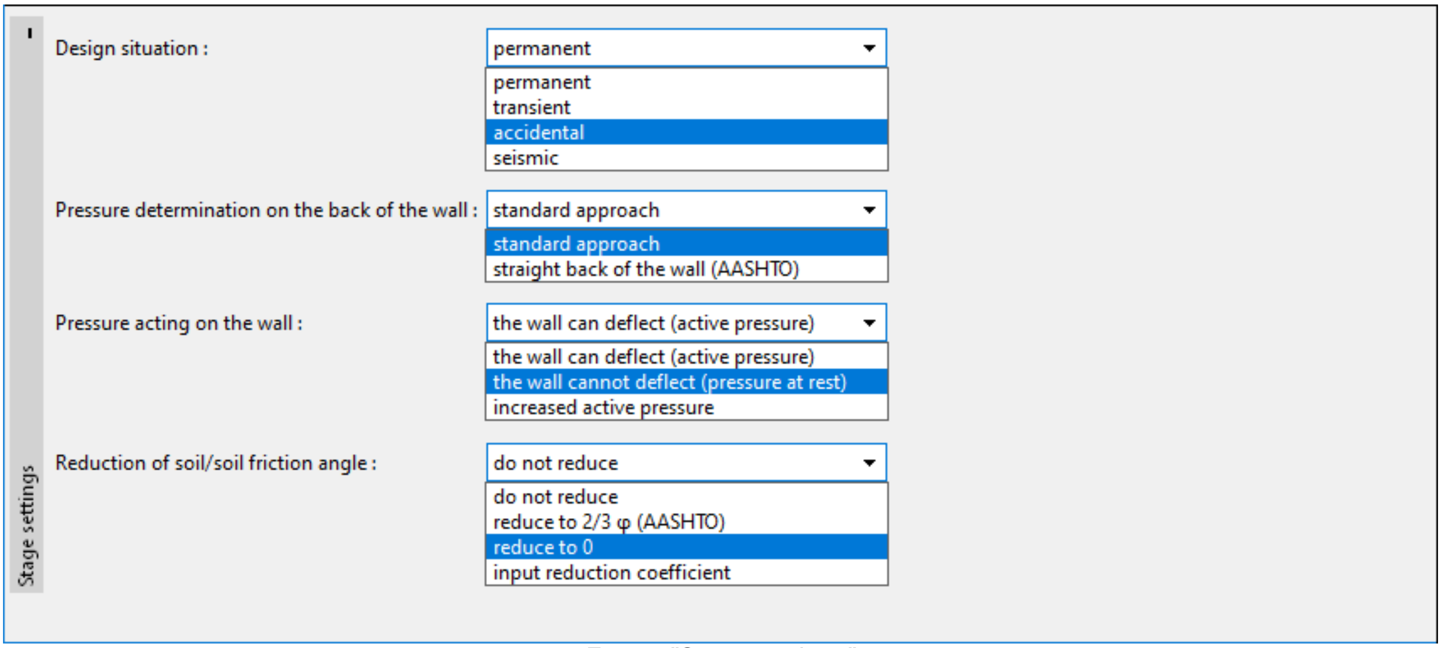
The selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology.

It is also possible to consider the simplified straight back of the wall for pressure calculation.

The reduction of soil/soil friction angle can be considered in one of following ways:

- **do not reduce**
- **reduce to 2/3φ (AASHTO)**
- **reduce to 0**
- **input reduction coefficient**



Frame "Stage settings"

Verification

The frame "**Verification**" shows the analysis results. Several computations can be carried out for a single task.

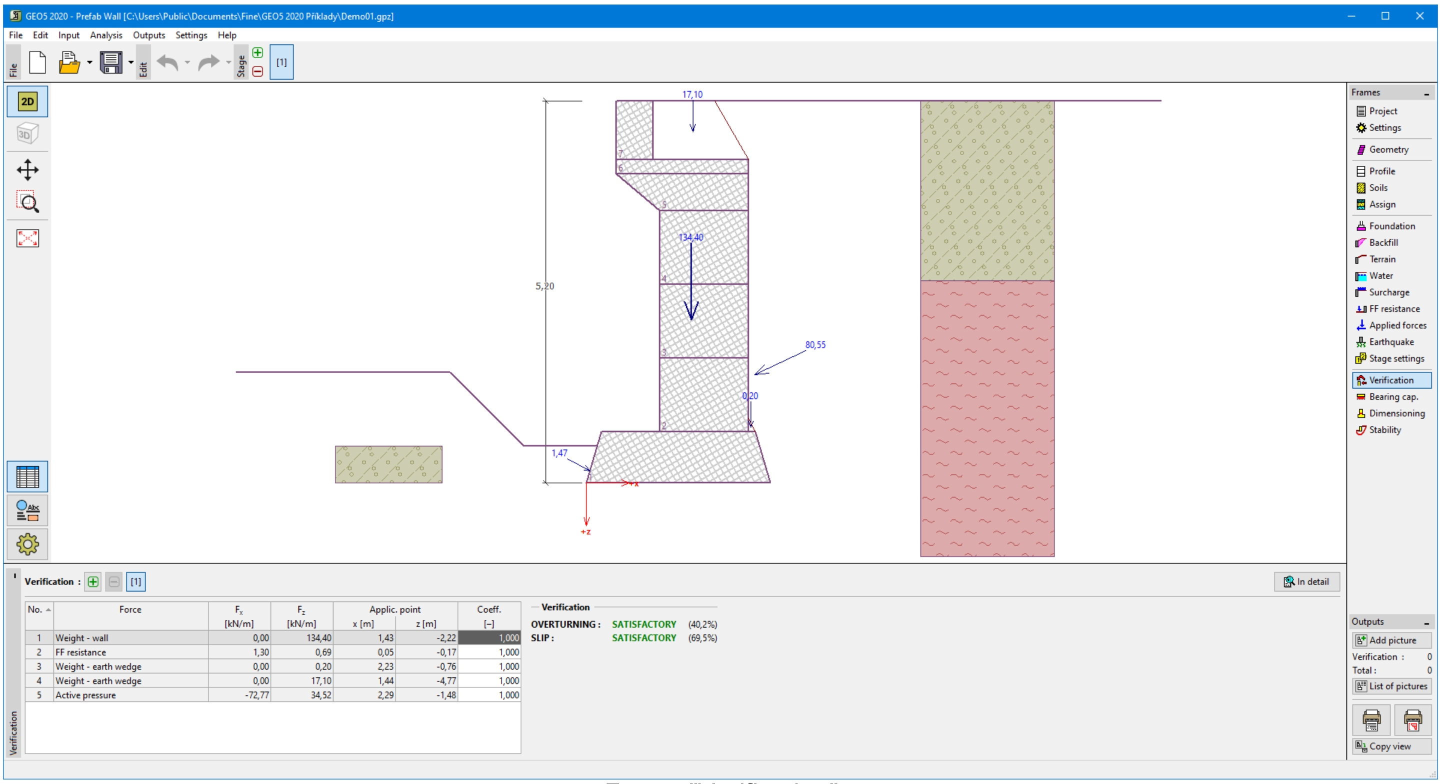
The frame appearance is adjusted based on the selected verification methodology.

- Verification according to the factor of safety or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows for specifying whether the load acting on a structure is considered as the secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in this case, the last column is not displayed.

The procedure for wall verification is described in the theoretical part of the help.

The computed forces are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against **overturning and translation**. The "**In detail**" button opens the dialog window, which contains a detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Verification"

Bearing Capacity

The "**Bearing capacity**" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the frame "Verification". The programs "**Spread footing**", "**Spread footing CPT**", "**Pile**" and "**Pile group**" then consider all verifications as load cases. In the program "**Pile CPT**", just normal load is used.

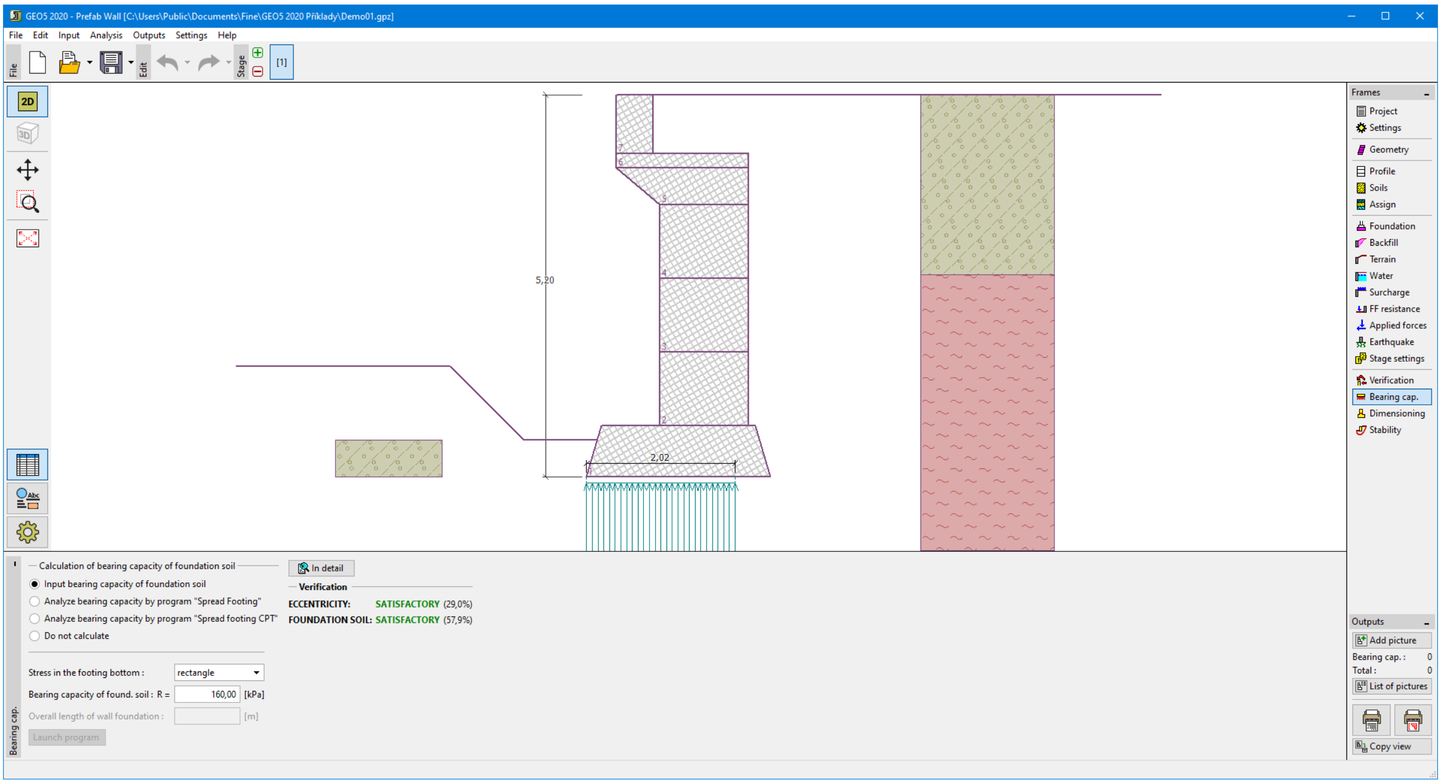
The frame contains the following analysis options:

- **Insert bearing capacity of foundation soil** – The foundation soil bearing capacity is input. The **eccentricity** and bearing capacity analysis results are displayed in the right part of the frame. The "**In detail**" button opens a dialog window that displays a detailed listing of the results.
- **Analyze bearing capacity by program "Spread footing"** – Pressing the "**Run program Spread footing**" button opens the program "**Spread footing**" which allows us to calculate the soil bearing capacity or settlement and rotation of the footing. Pressing the "**OK**" button leaves the analysis mode - the results and all plots are transferred into the program "**Prefab wall**". The "**Spread footing**" program must be installed for the button to be active. The overall length of the wall foundation is input.
- **Analyze bearing capacity by program "Spread footing CPT"** – The procedure is identical as if calculating soil bearing capacity by the **"Spread footing**" program.

- **Analyze bearing capacity by program "Pile"** – The procedure is identical as if calculating soil bearing capacity by the "**Spread footing**" program. The "**Run program Pile**" is available if the wall has a pile foundation (frame "Foundation"). Pile spacing s is input.
- **Analyze bearing capacity by program "Pile CPT"** – The procedure is identical as if calculating soil bearing capacity by the "**Pile**" program.
- **Analyze bearing capacity by program "Pile group"** – The procedure is identical as if calculating soil bearing capacity by the "**Spread footing**" program. The "**Run program Pile group**" is available if the wall has pile foundation with more then one pile (frame "Foundation"). Pile spacing s , the overall number of pile rows n and loading length l are input.
- **Do not calculate (pile footing)** – The foundation soil bearing capacity is not calculated.

The program allows us to specify a shape of stress in the footing bottom.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Bearing capacity"

Dimensioning

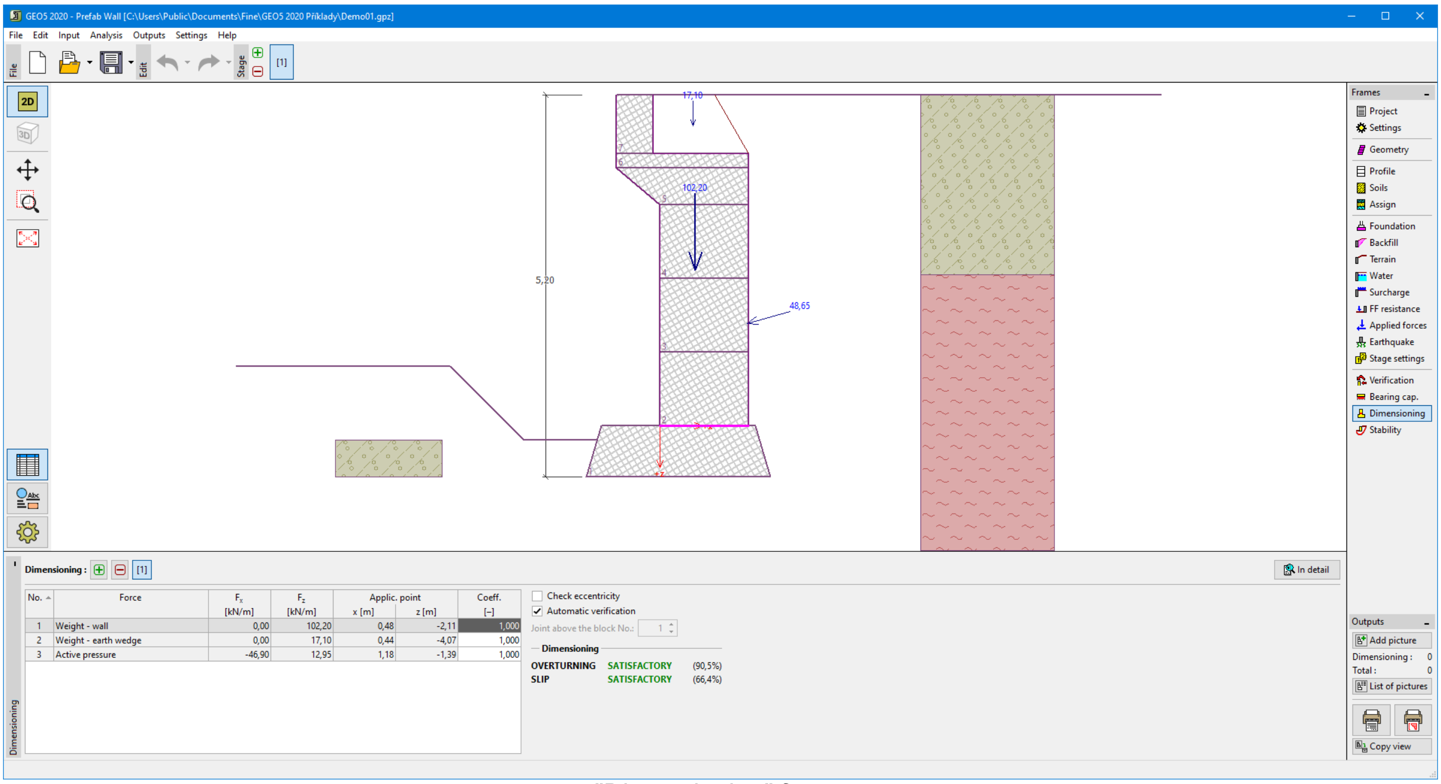
The "**Dimensioning**" frame allows us to verify joints between individual blocks of a wall. The "**Joint above block No.**" field serves to select the desired joint subjected to verification analysis. The verification against **overturning** and **translation** is performed in the same way as for the entire wall - friction between blocks and cohesion of a block material is input in the "Geometry" frame.

The frame appearance is adjusted based on the selected verification methodology.

- Verification according to the factor of safety or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows for specifying whether the load acting on a structure is considered as the secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in this case, the last column is not displayed.

Several computations for various cross-sections can be carried out. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "**In detail**" button opens the dialog window that contains a detailed listing of the dimensioning results.

Visualization of results can be adjusted in the frame "Drawing Settings".



"Dimensioning" frame

Slip on Georeinforcement

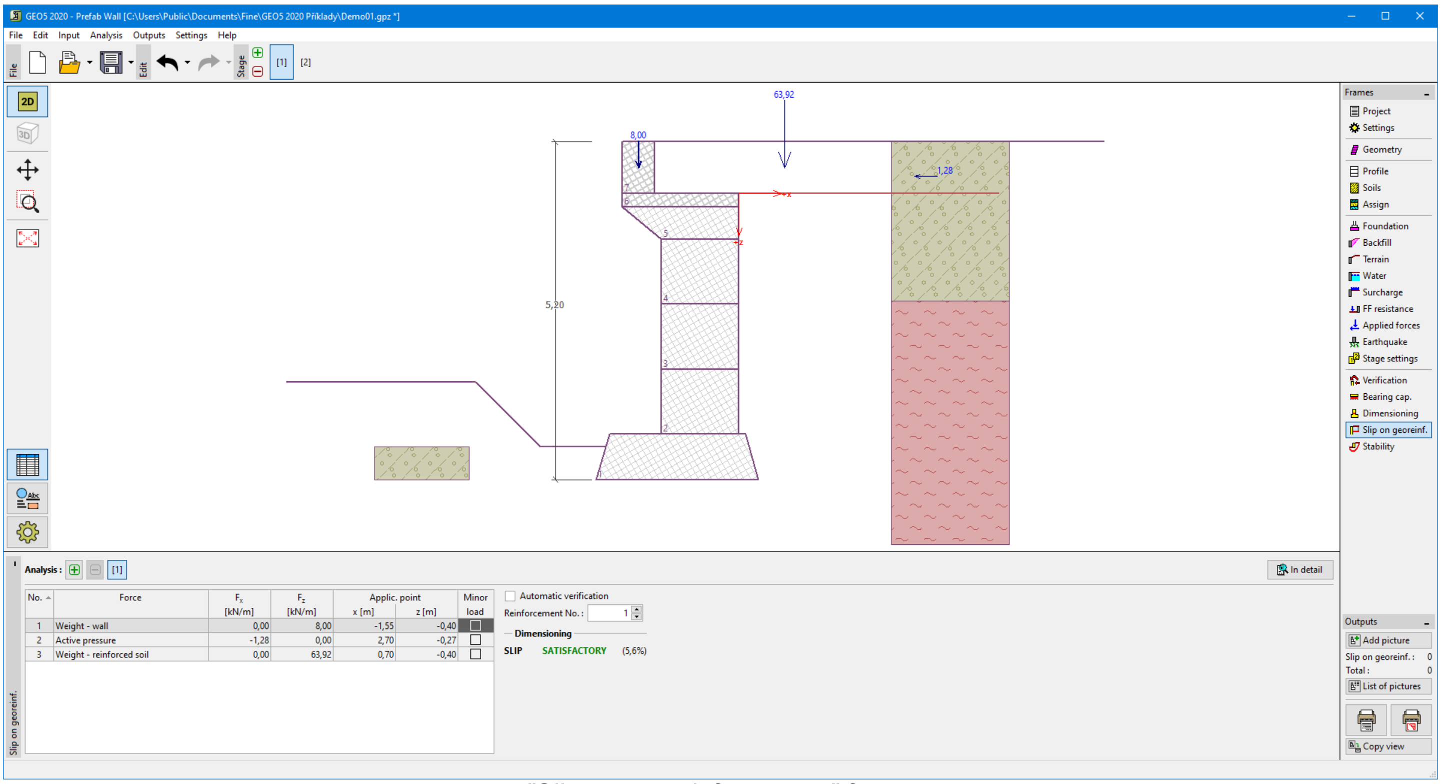
The frame serves to verify the limit state for slip along reinforcement - the frame is therefore accessible only in stages, where the reinforcements are defined.

The window requires inputting the reinforcement number - the forces entering verification analysis together with the shape of the sliding block are then displayed. The calculated forces are stored in the table.

Several calculations for various reinforcements can be carried out. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "**In detail**" button opens the dialog window that contains a detailed listing of the dimensioning results.

The verification procedure depends on settings in the "Wall analysis" tab - either based on **factors of safety** or according to the **theory of limit states**. The solution procedure is described herein.

Visualization of results can be adjusted in the frame "Drawing Settings".

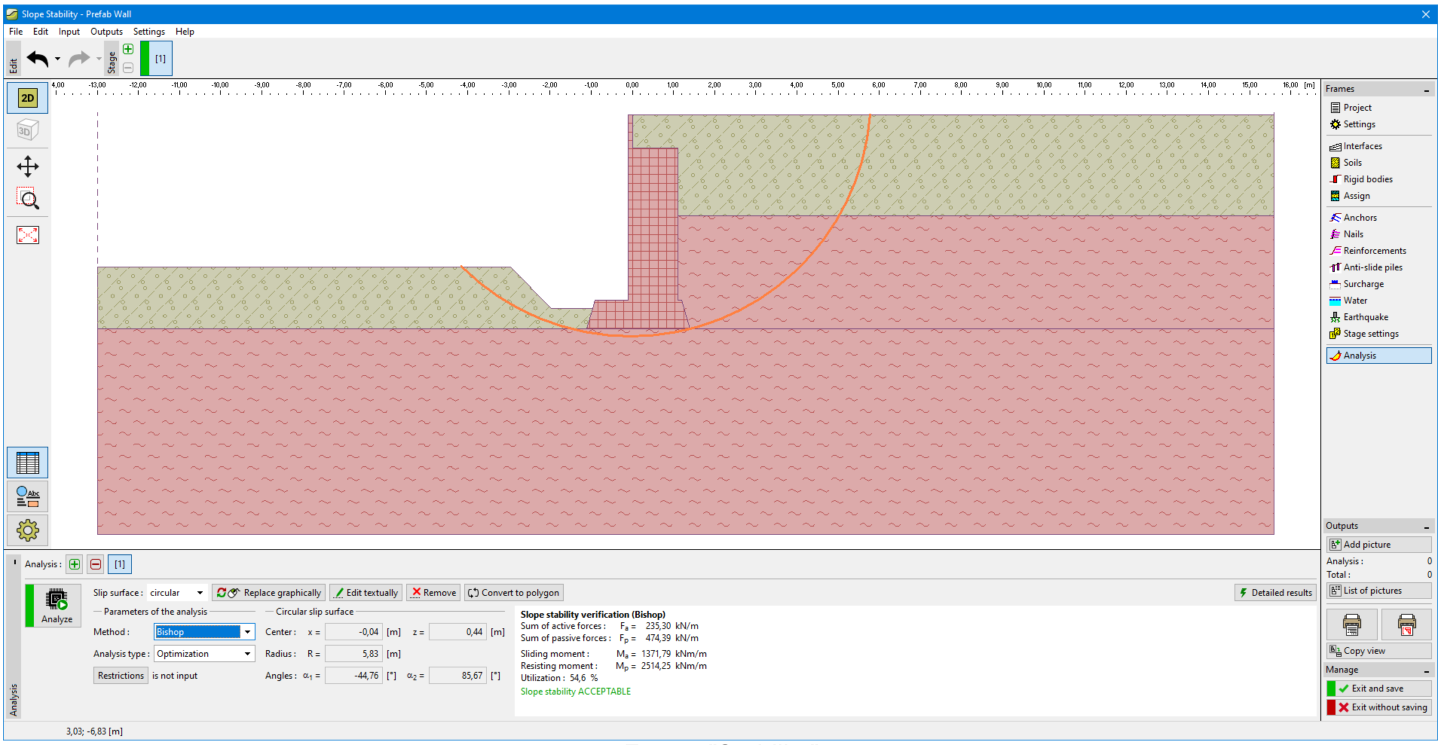


"Slip on georeinforcement" frame

Stability

Pressing the "**Stability**" button launches the "**Slope stability**" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program "**Slope stability**" is installed.

After completing all analyses press the "**OK**" button to leave the program - all data are then carried over to the analysis protocol of the "**Prefab wall**" program.



Frame "Stability"

Program Masonry Wall

This program is used for the design and analysis of reinforced masonry block wall according to various standards.

The help in the program "Masonry Wall" includes the following topics:

- The input of data into individual frames:

Project	Settings	Geometry	Types of Blocks	Material	Profile	Soils
Assign	Foundation	Backfill	Terrain	Water	Surcharge	Front Face Resistance
Applied Forces	Earthquake	Base	Stage Settings	Verification	Bearing Capacity	Dimensioning
Stability		Anchorage				

- Standards and analysis methods

- Theory for analysis in the program "**Masonry Wall**":

Stress in Soil Body	Earth Pressures	Analysis of Walls	Analysis of Foundation Bearing Capacity	Dimensioning of Concrete Structures	Masonry Cross Sections Verification

- Outputs

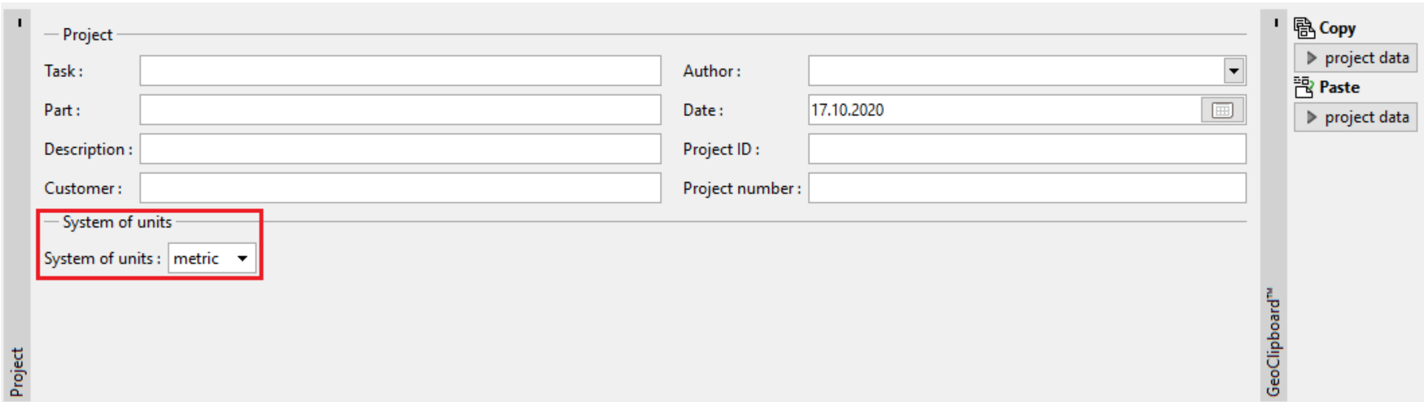
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The frame **"Project"** is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The frame "**Settings**" allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the

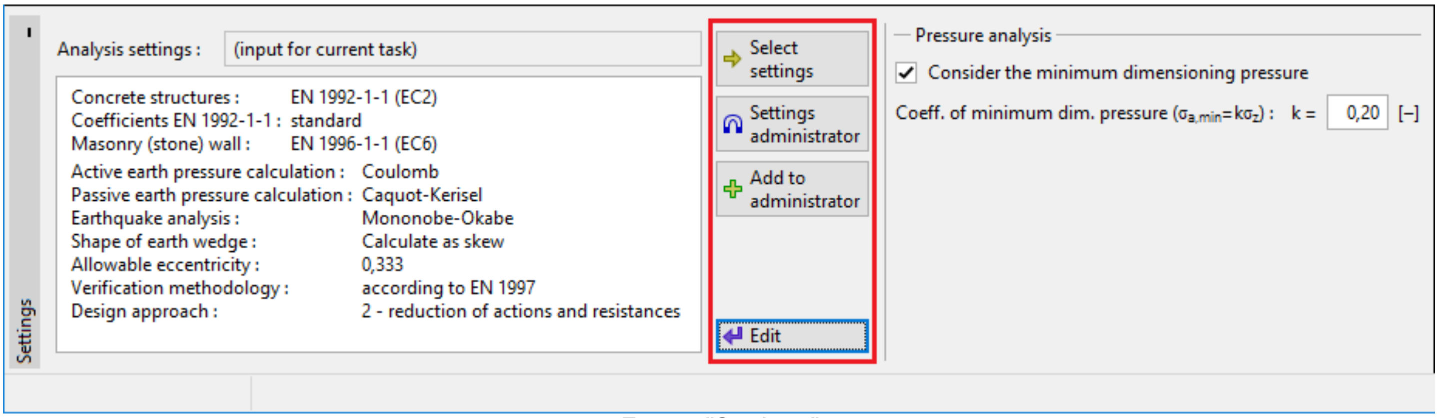
Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

The program allows us to specify a value of the minimum dimensioning pressure (by checking the option "**Consider the minimum dimensioning pressure**").

Settings of analysis parameters are performed in the "Materials and standards" and "Wall analysis" tabs.



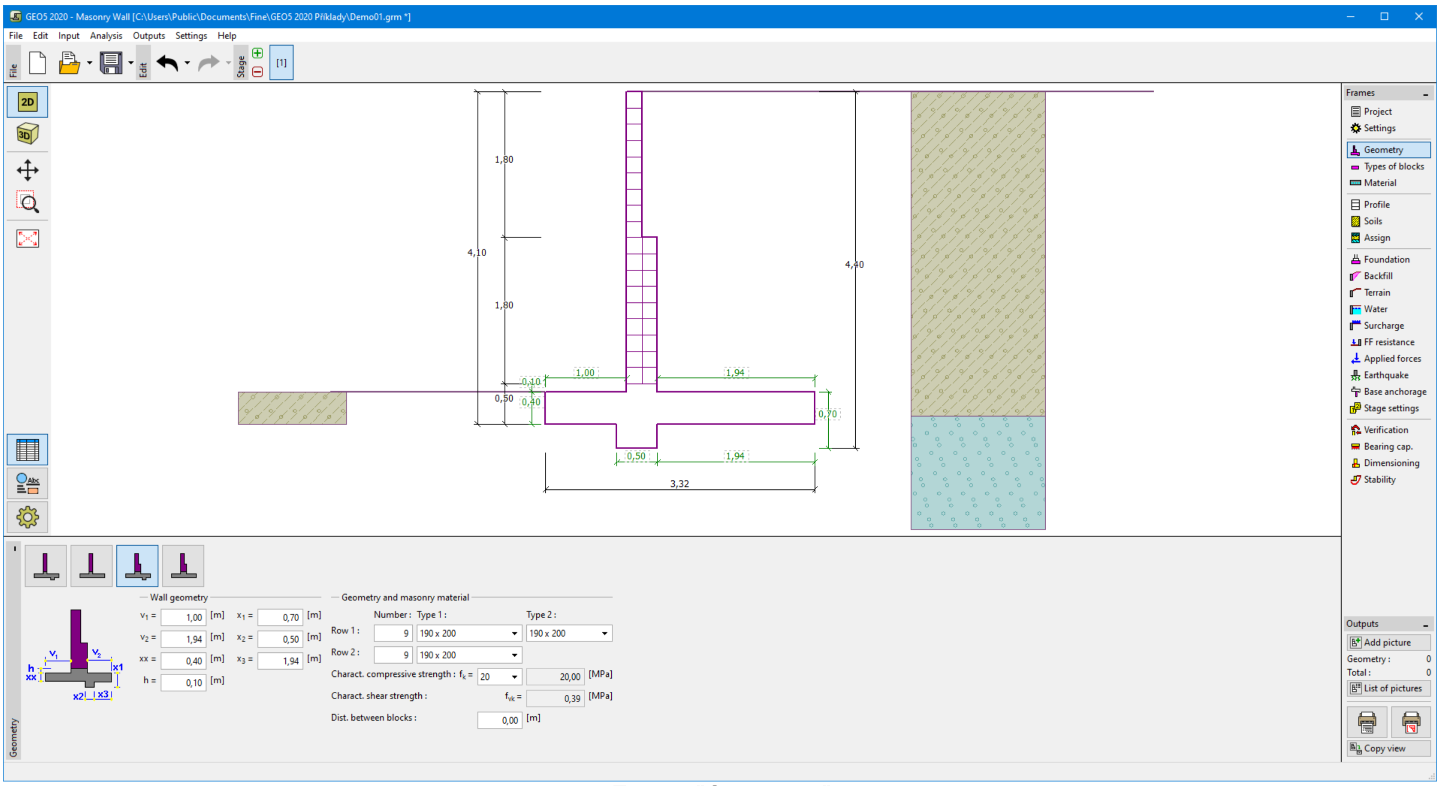
Frame "Settings"

Geometry

The "**Geometry**" frame allows us to select the wall shape by pressing the button. The selected shape with a graphic hint appears in the left part of the frame. The shape of a wall can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

Based on the selected shape of a wall, you specify, in the "**Geometry and masonry material**" frame, the number and dimensions of masonry blocks in individual columns, or if applicable, also the thickness of vertical joint between blocks. In addition, it is necessary to input the compressive strength of masonry, which serves as the basic input parameter for the bearing capacity verification of reinforced masonry (according to EN 1996-1-1 or AS 3700).

The program makes it possible to export the geometry of a structure in the *.DXF format.

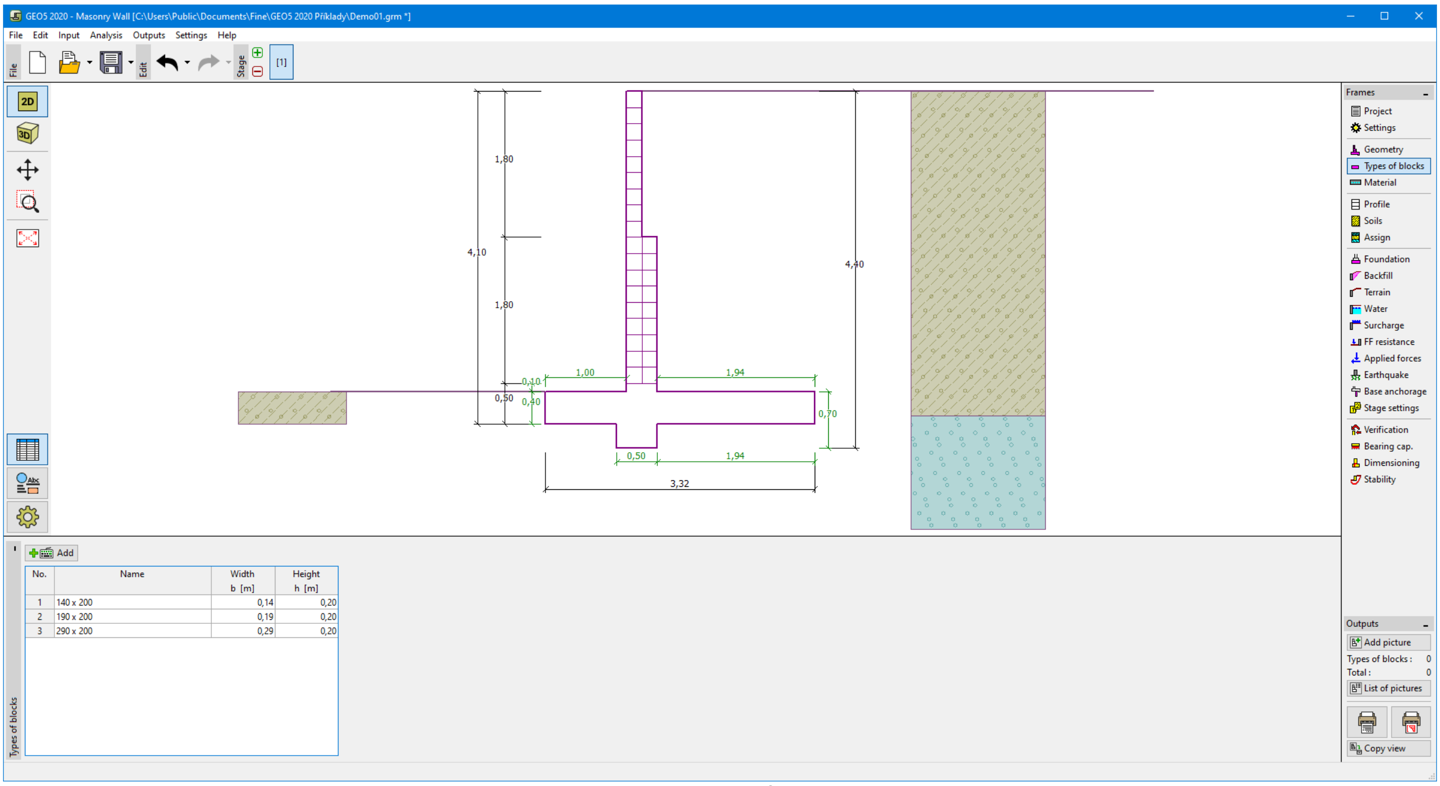


Frame "Geometry"

Types of Blocks

The "**Types of blocks**" frame contains a table with a list of input blocks. Adding (editing) blocks is performed in the "**New type of block (Edit type of block)**" dialog window.

This dialog window serves to define **the geometry of a block** (width and height).



Frame "Types of blocks"

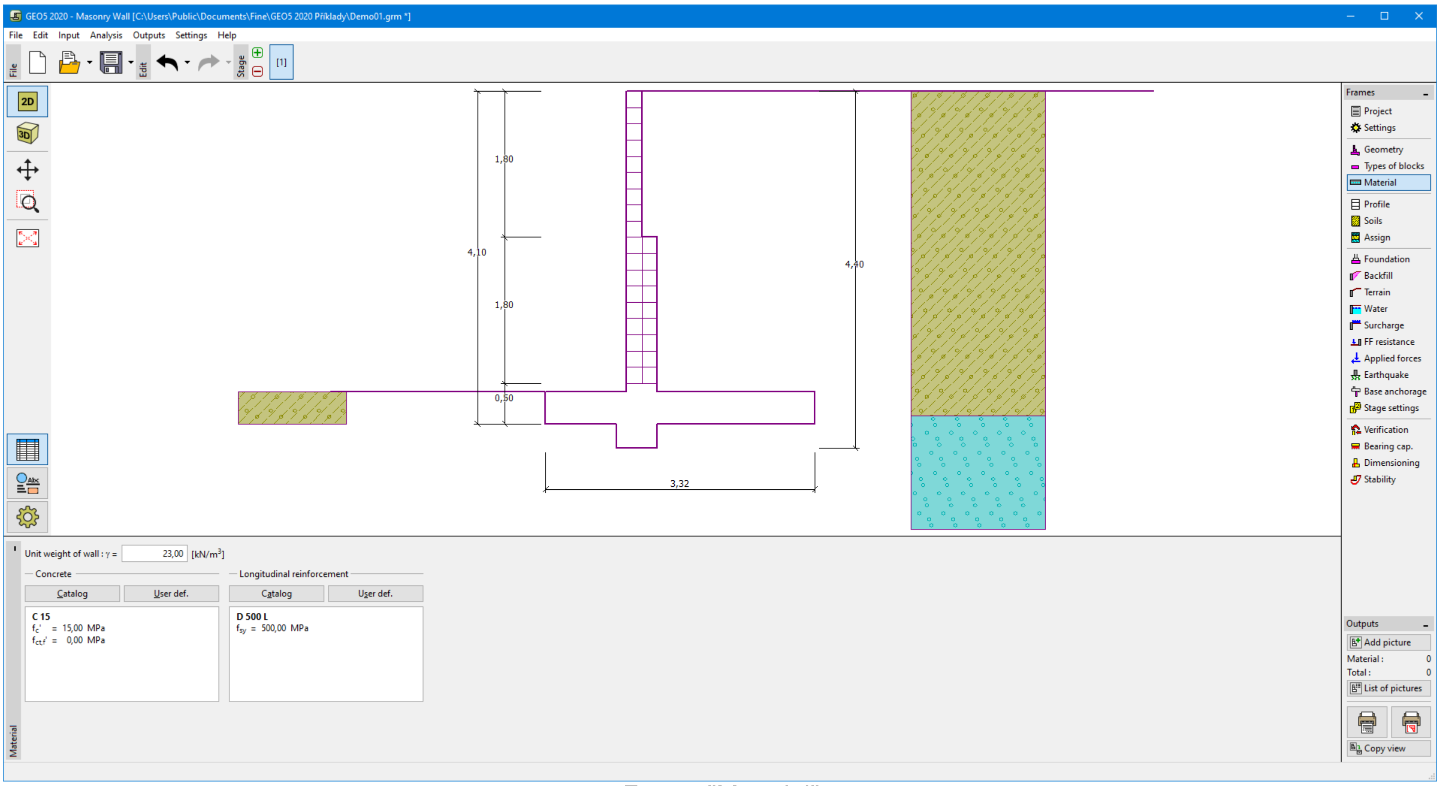
Material

The frame **"Material"** allows us to enter material parameters. The input field in the upper part of the frame serves to specify the **wall unit weight**.

Two options can be used for setting the parameters of the material:

- The "**Catalog**" button opens the "**Catalog of materials**" dialog window, where the required material can be selected.
- The "**User-defined**" button opens the "**Editor of material**", which allows us to input the specification of material parameters manually by the user.

The catalogs content depends on the selection of standard for the design of concrete structures set in the "Materials and standards" tab.



Frame "Material"

Profile

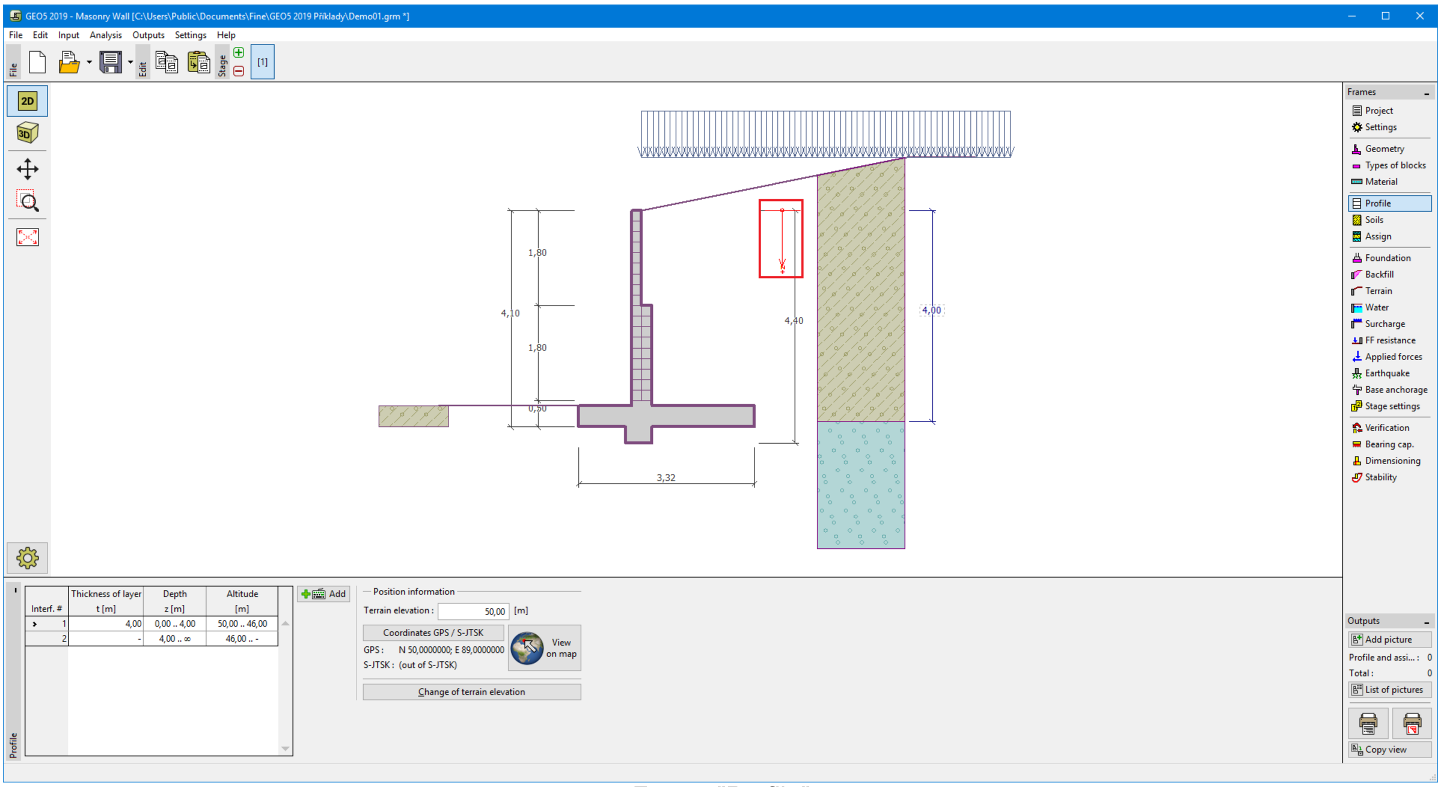
The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.



Frame "Profile"

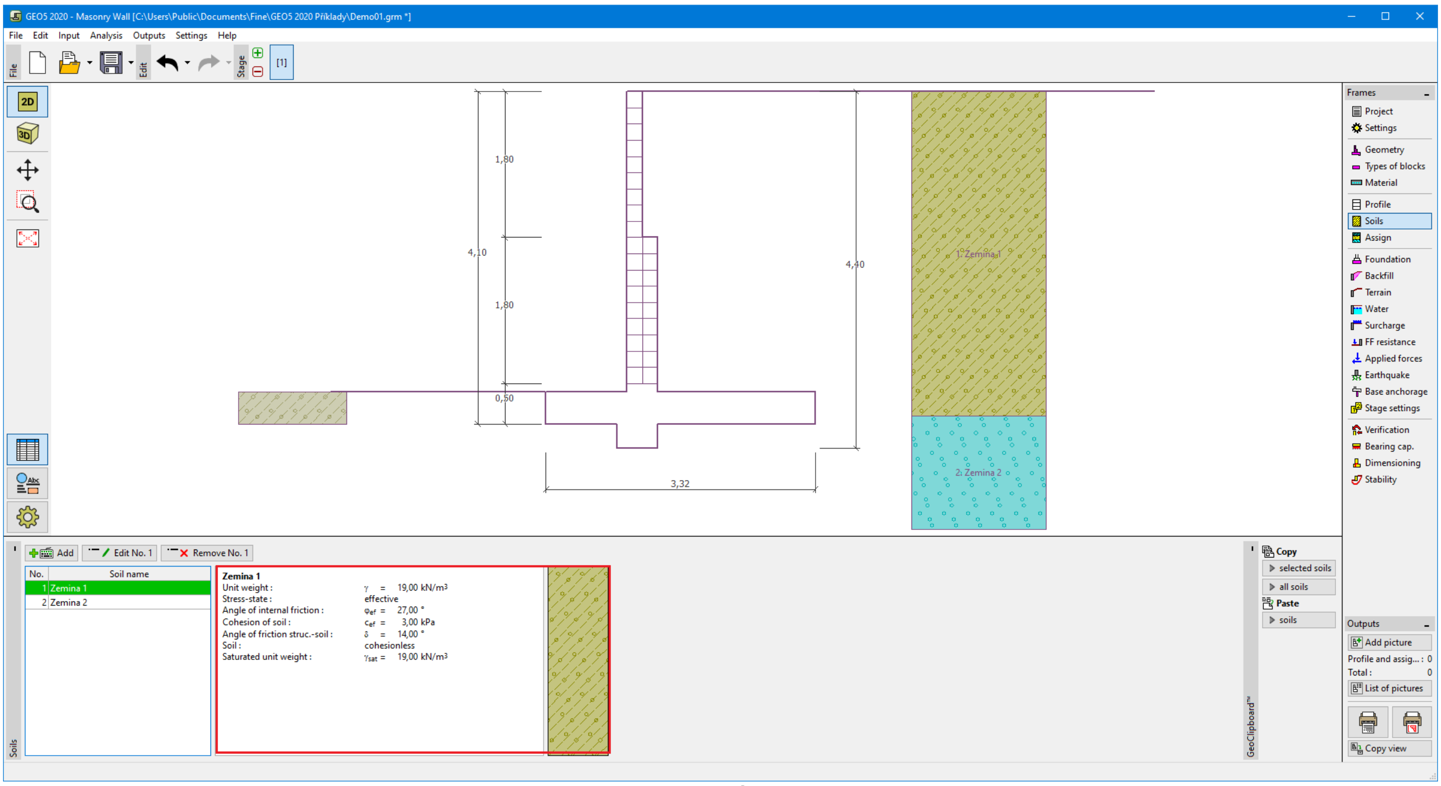
Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

Basic Data

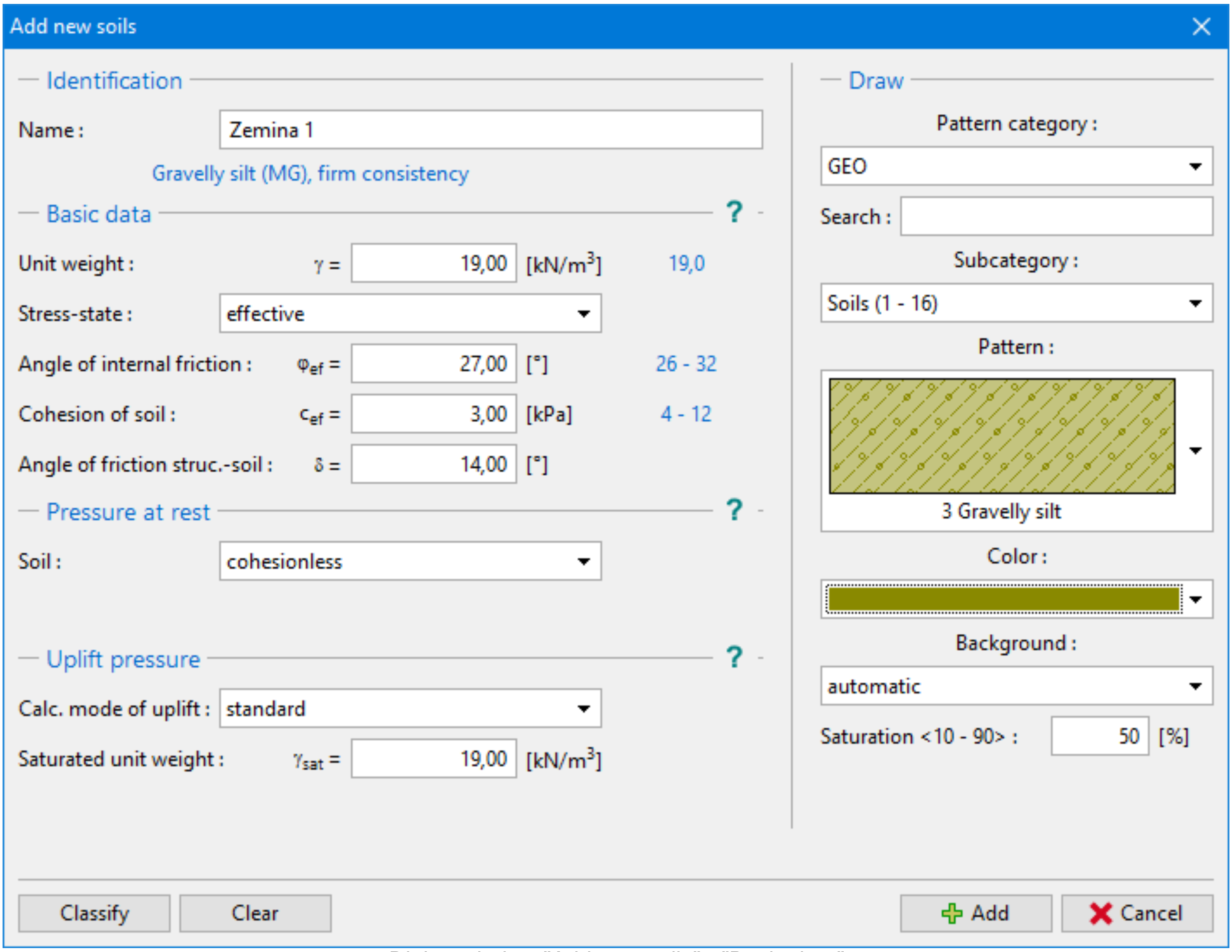
This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction, and cohesion**. The particular values are obtained from a geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit the built-in database of soils, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually.

Either **effective or total** parameters of the angle of internal friction and cohesion are specified depending on the setting in the "**Stress analysis**" combo list. Whether to use effective or total parameters depends primarily on the type of soil, type of load, structure duration, and water conditions.

For effective stress further needs to specify the angle of internal friction between the soil and structure, which depends on the structure material and the type of soil. Possible values of this parameter are listed in the table of recommended values.

For total stress further needs to specify the adhesion of soil to the structure face a .

The associated theory is described in detail in chapter "Earth pressures".

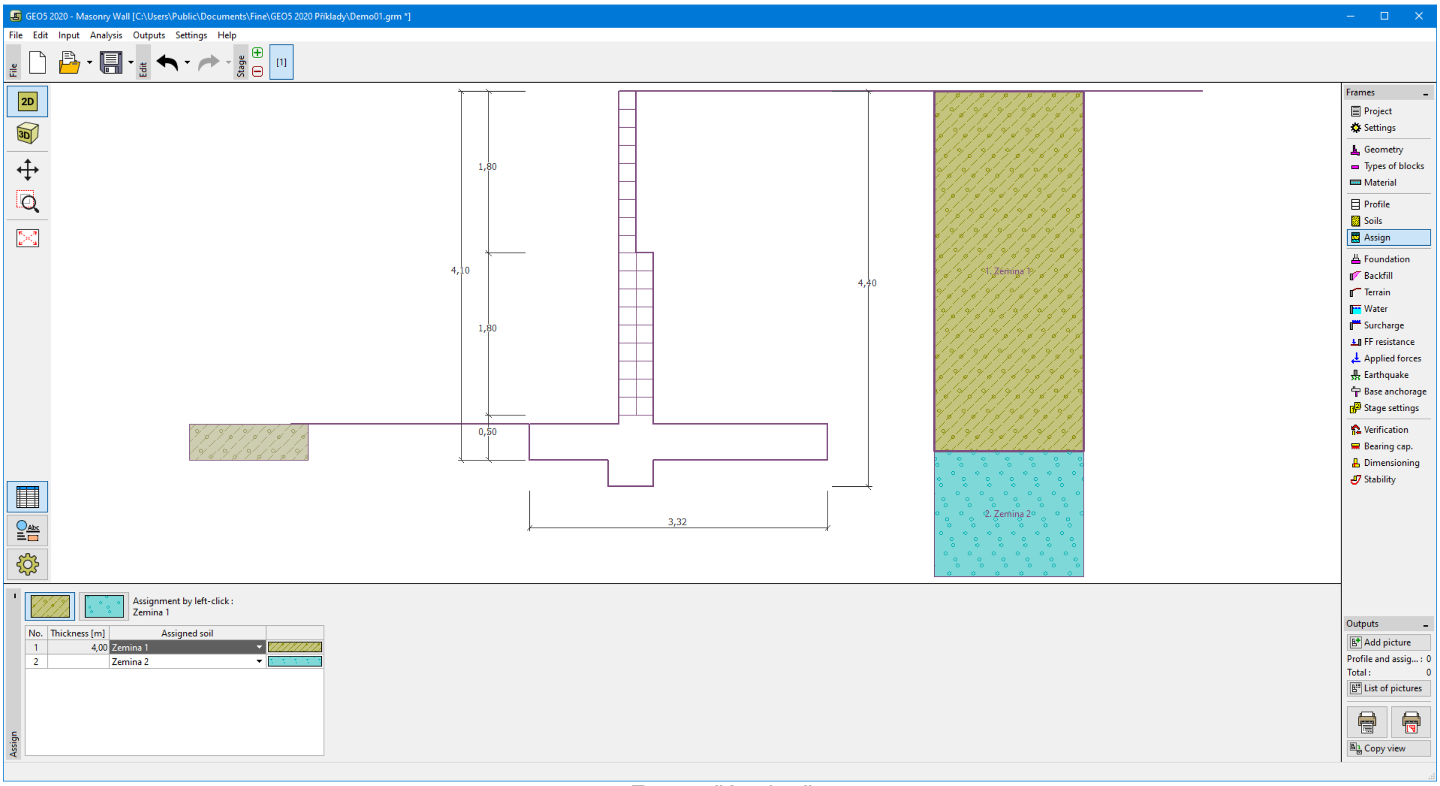


Dialog window "Add new soils" - "Basic data"

Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail herein.



Frame "Assign"

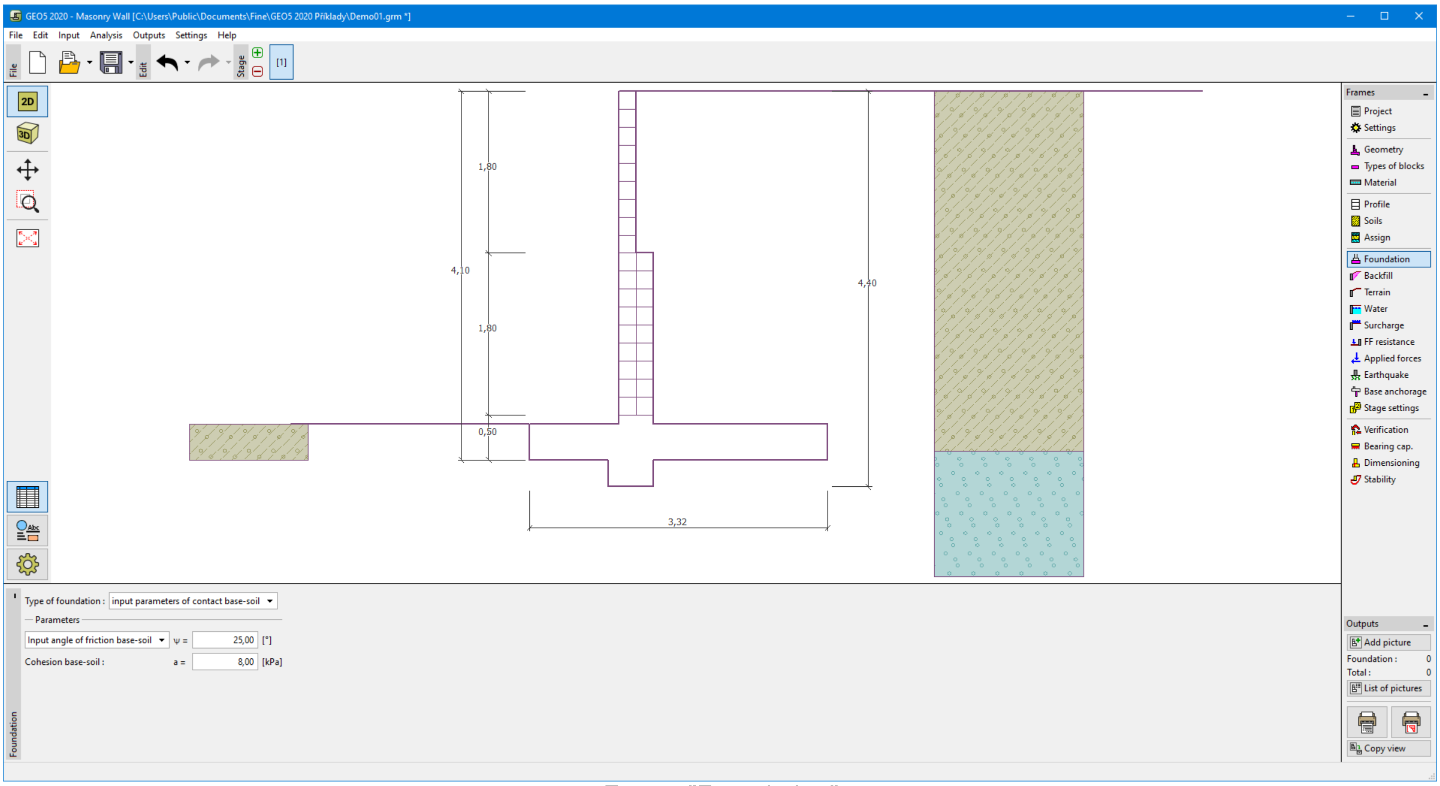
Foundation

The "**Foundation**" frame serves to specify the type of wall foundation. The following types of wall foundation are available:

- **soil from geological profile** - the wall is founded on the soil assigned from the geological profile specified in the "Profile" frame.
- **input parameters of contact base-soil** - parameters of the contact between footing bottom and structure are specified. Option "**input angle of friction base-soil**" requires inputting the friction angle ψ [°] between foundation and soil. Option "**input friction coefficient**" requires specifying the friction coefficient μ [-]. Both options require inputting the cohesion a [kPa] between foundation (base) and soil.
- **strip foundation** - strip foundation material is represented either by **soil** (input in "Soils" frame), or concrete - requires inputting the **unit weight of foundation material** γ and **parameters of contact base-soil** (friction coefficient f, cohesion c, additional resistance F).
- **pile foundation** - the wall can be founded on one row of piles or two rows of piles, respectively.

Strip foundation and **pile foundation** can be adopted for the wall foundation only if the type wall with **straight footing bottom without jump** is selected in the "Geometry" frame. The geometry of the wall foundation (**strip foundation**, **pile foundation**) can be modified either in the frame by entering specific values into the inputting fields or on the desktop with the help of active dimensions.

The input data introduced in this frame influence the actual wall analysis (check for slip) and further the bearing capacity of foundation soil.

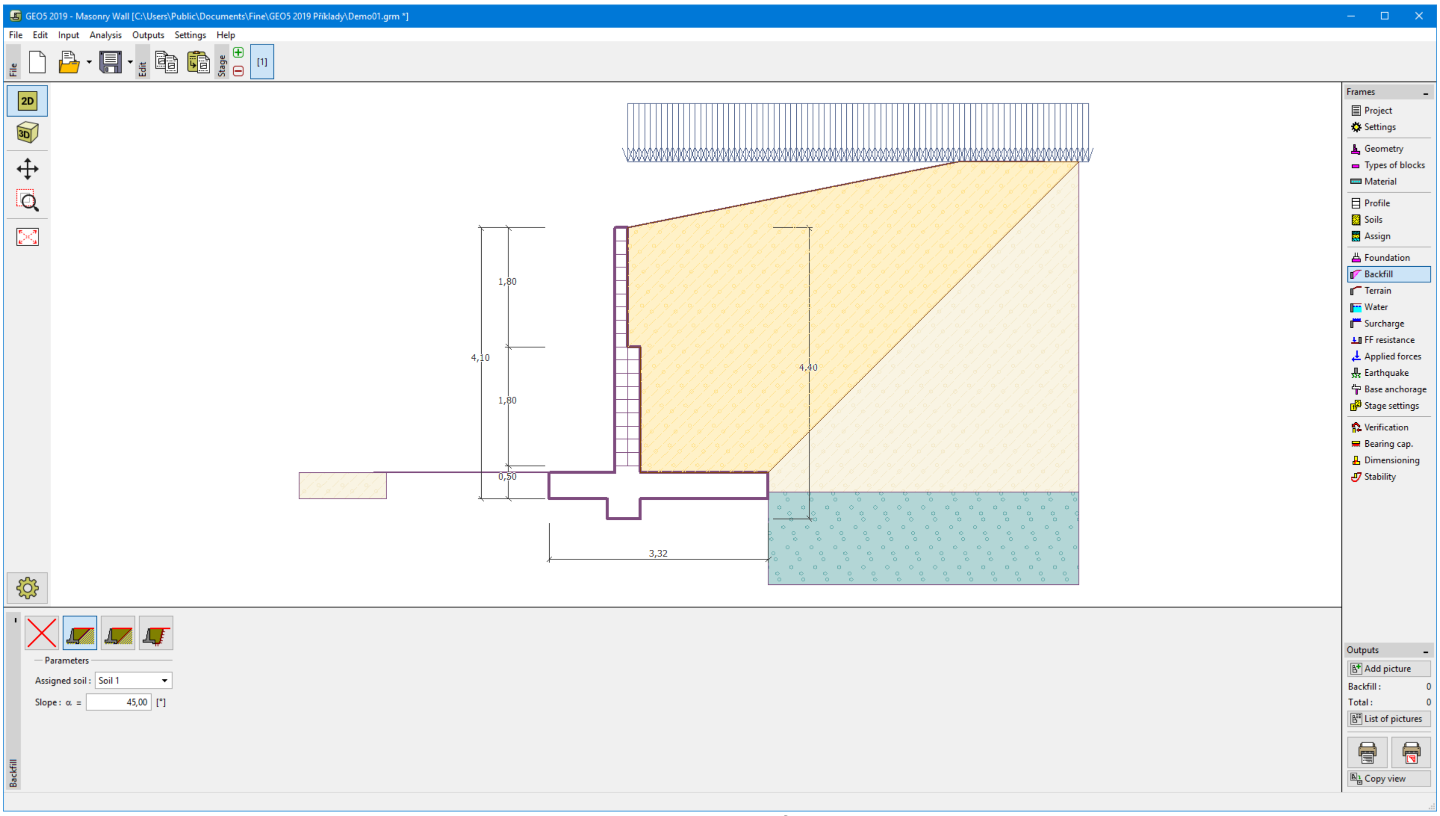


Frame "Foundation"

Backfill

The frame "**Backfill**" allows the selection of backfills behind the structure.

Analysis of earth pressures with the influence of backfill is described in the theoretical part of the help: "Influence of Backfill".



Frame "Backfill"

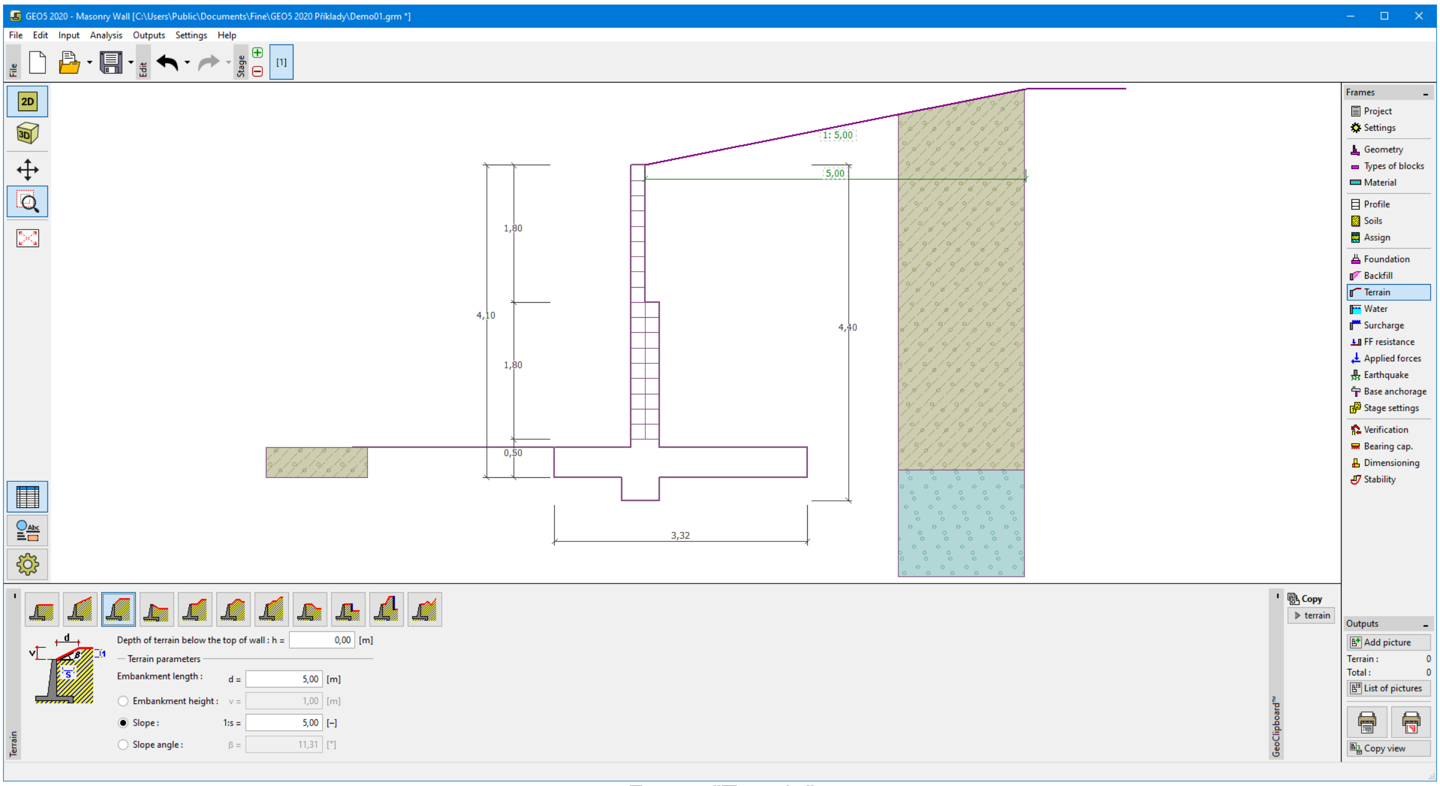
Terrain

The "**Terrain**" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by

inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case, the frame contains a table with a list of terrain points. The first point with coordinates [0, 0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "Distribution of earth pressures for broken terrain".



Frame "Terrain"

Water

The "**Water**" frame allows, by pressing the button, for selecting the type of water. The selected type with a graphic hint of input values is displayed in the left part of the frame. Water parameters (h_1 , h_2 ...) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

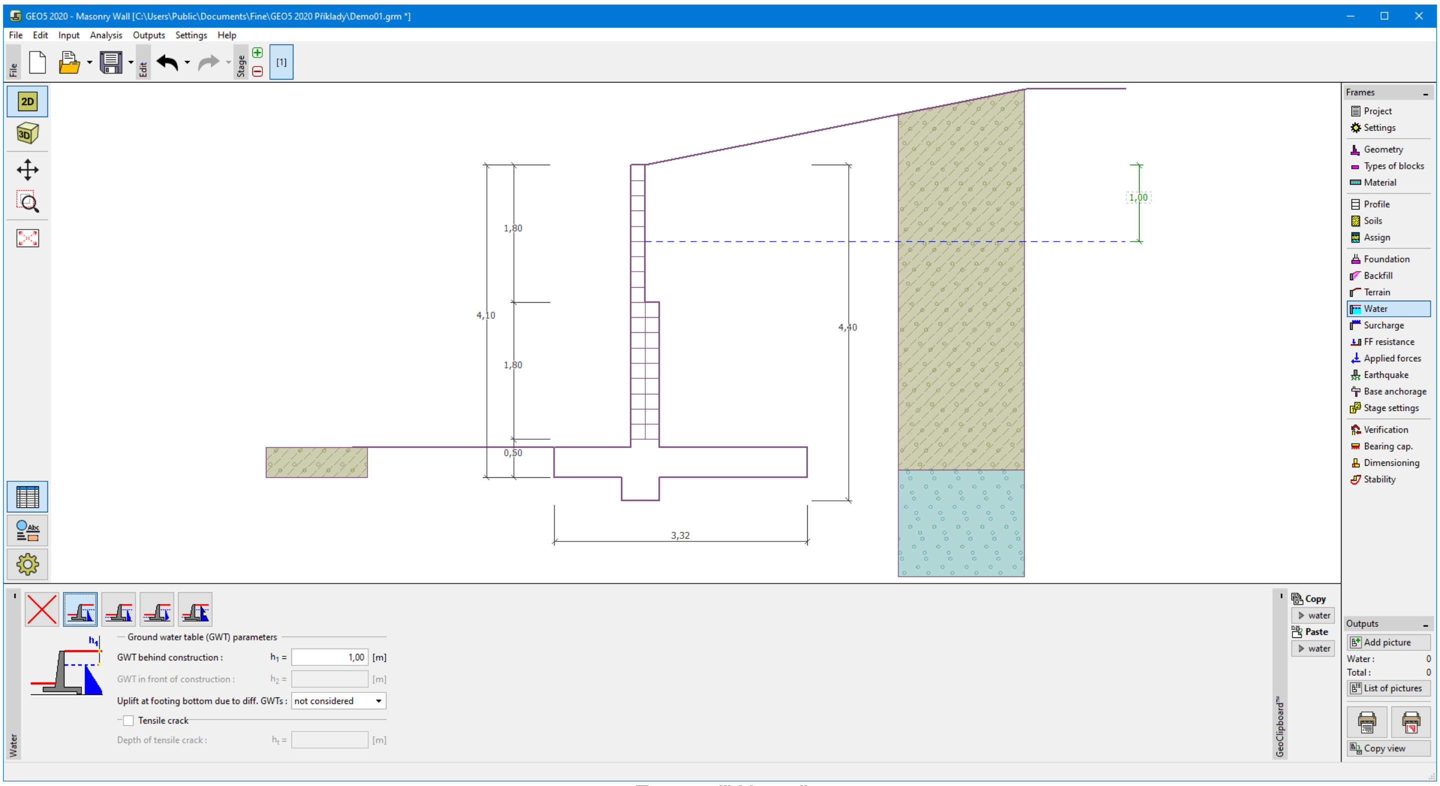
The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be linear, parabolic, or it may not be considered at all. When verifying the wall, the uplift pressure in the base of footing due to the different water tables is introduced as a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tabs "**In front of structure**" and "**Behind structure**" appear with tables. The table contains the values of pore pressure in front of the structure or behind it.

The groundwater table can also be specified **above the structure** or earth profile, respectively - in such a case, the depth of water is input with a negative value.

Analysis of earth pressures with the influence of water is described in the theoretical part of the help, chapter "Influence of water".

The program further allows for specifying a depth of tensile cracks filled with water.



Frame "Water"

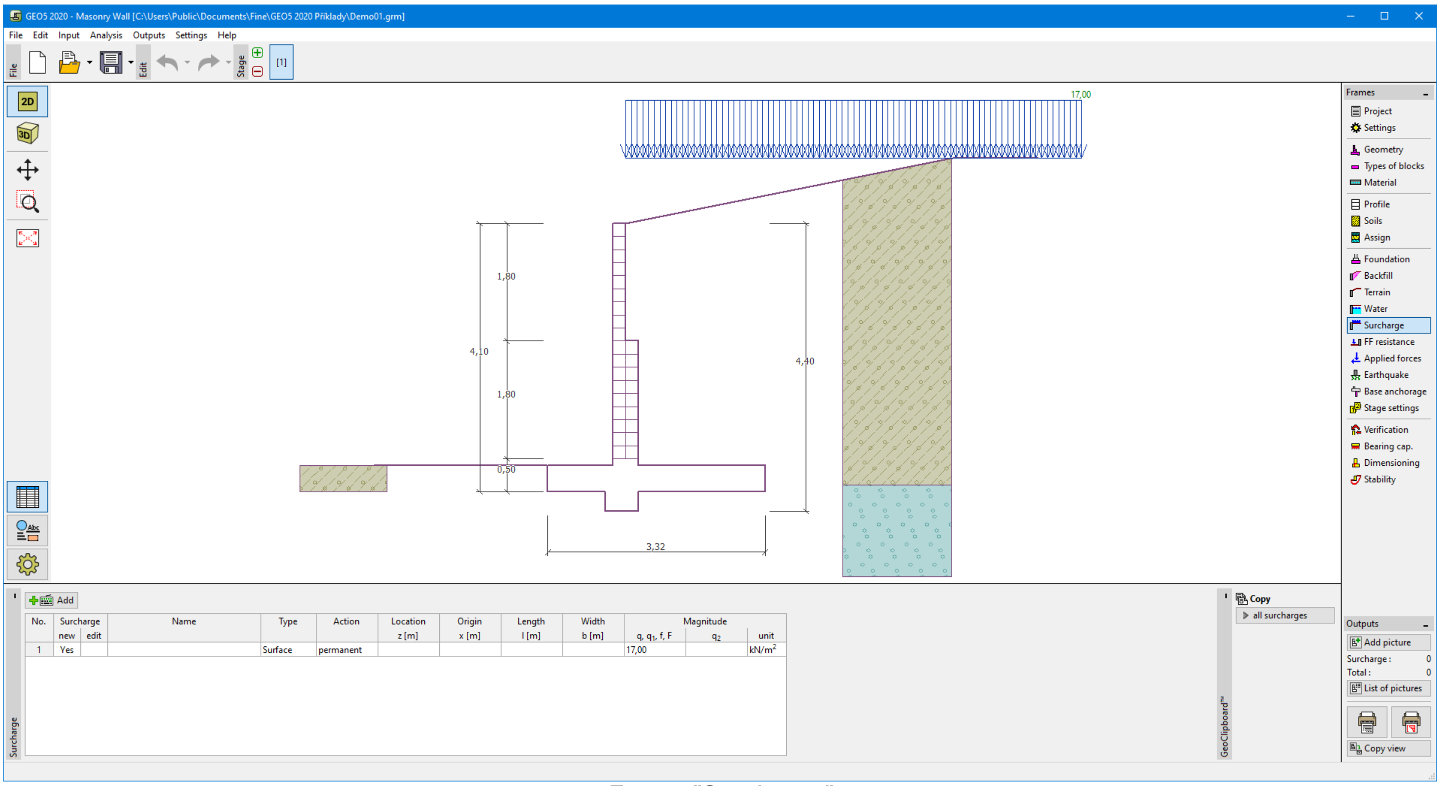
Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The z -coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case when the surcharge is found out of the terrain the program prompts an error message before calculation.

Either **permanent**, **variable,** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with a favorable effect is not considered in the analysis.

Analysis of earth pressures due to surcharges is described in the theoretical part of the help, chapter "Influence of surcharge".



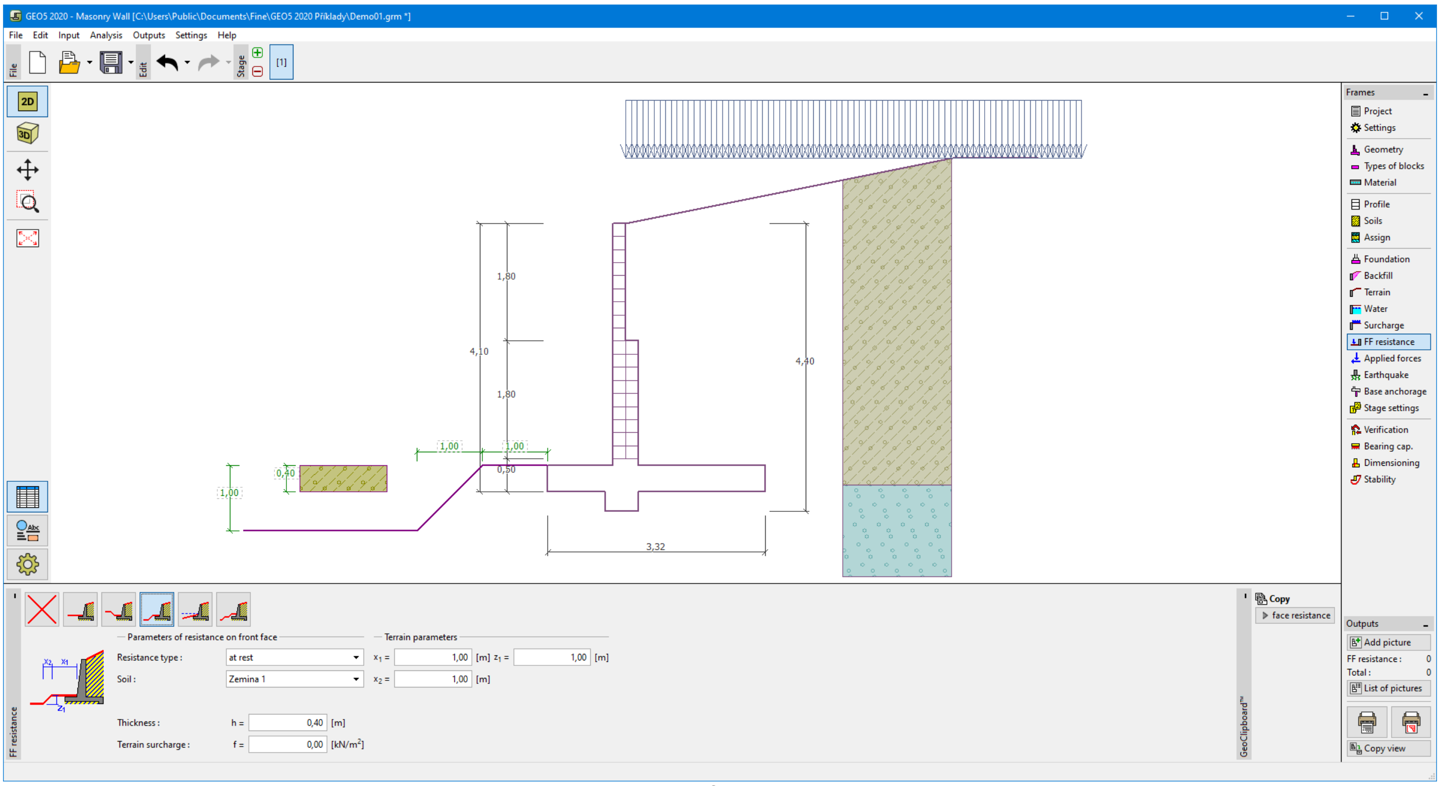
Frame "Surcharge"

Front Face Resistance

The "**Front face resistance**" frame allows us to specify the terrain shape in front of the structure and parameters of front face resistance. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop using active dimensions.

Combo lists in the frame allow selecting the type of resistance and a soil (from the soils introduced in the "Soils" frame). The terrain surcharge magnitude and soil thickness (above the wall lowest points) can also be specified.

The resistance on a structure front face can be specified as the pressure at rest, passive pressure, or reduced passive pressure. The resulting force due to the reduced passive pressure is calculated as a resultant force caused by pressure at rest and passive pressure multiplied by corresponding coefficients.



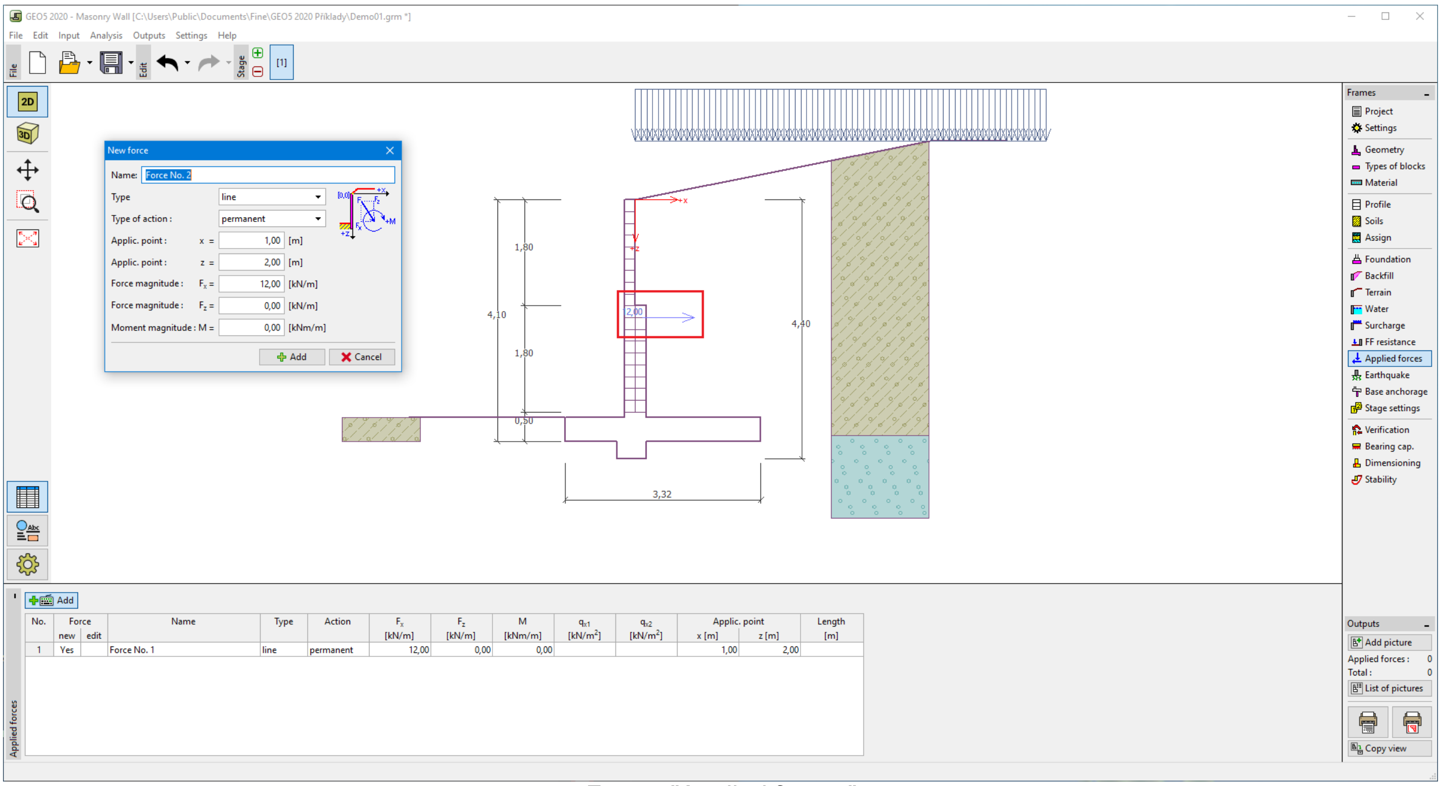
Frame "Front face resistance"

Applied Forces

The "**Applied forces**" frame contains a table with a list of forces acting on a structure. Adding forces is performed in the "**New force**" dialog window. The input forces can also be edited on the desktop with the help of active objects.

Applied forces represent an additional load on the structure of the wall, sheeting or MSE wall. We can model such as an anchoring crash barrier, crash vehicle, load from billboards and hoardings etc. The program does not modify the applied forces in the calculation except multiplying them with the corresponding coefficients according to the selected type of load (EN1997, LRFD).

External load acting on the terrain is necessary to define as a surcharge.



Frame "Applied forces"

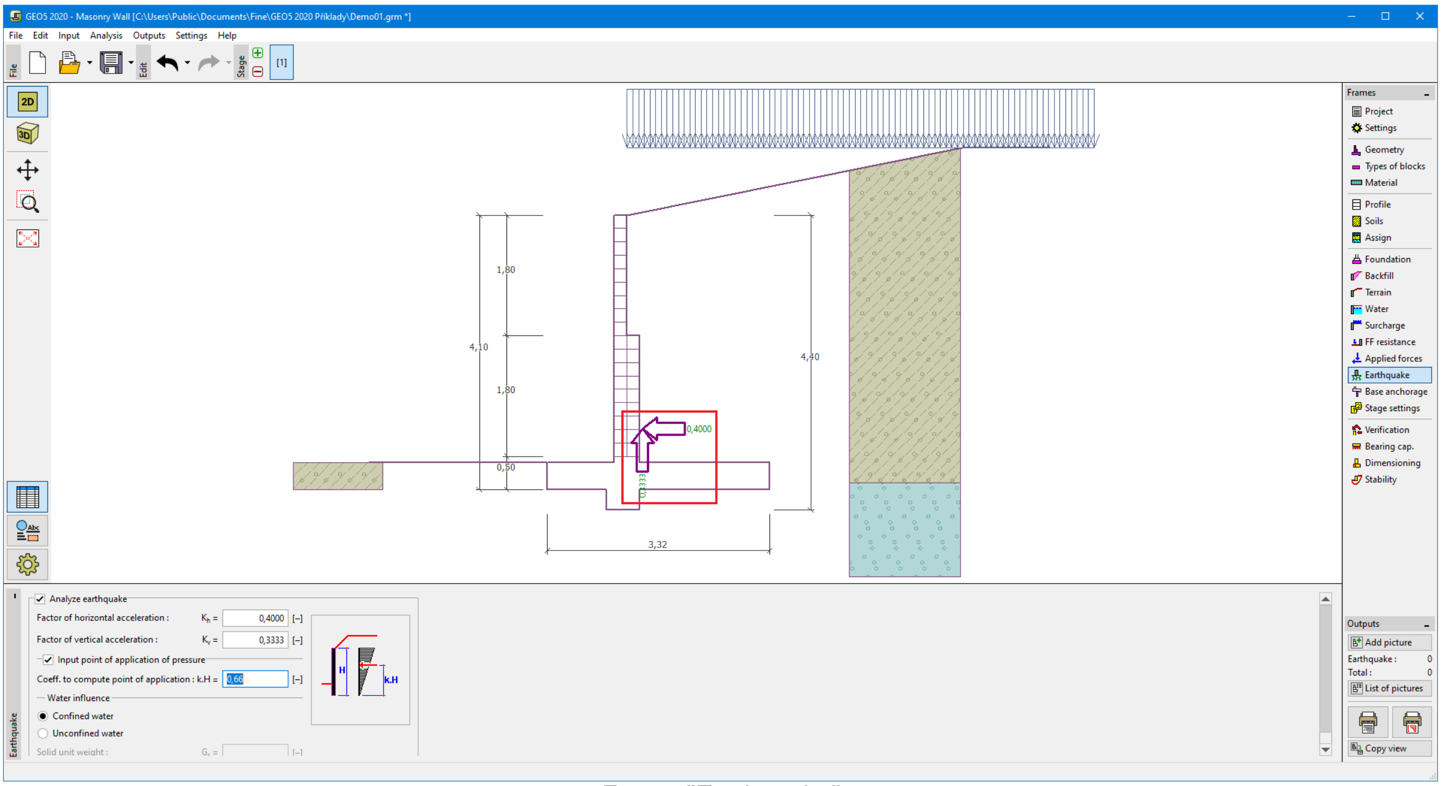
Earthquake

The "**Earthquake**" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements, the coefficients k_h and k_v can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for an earthquake is described in the theoretical part of the help, chapter "Influence earthquake".

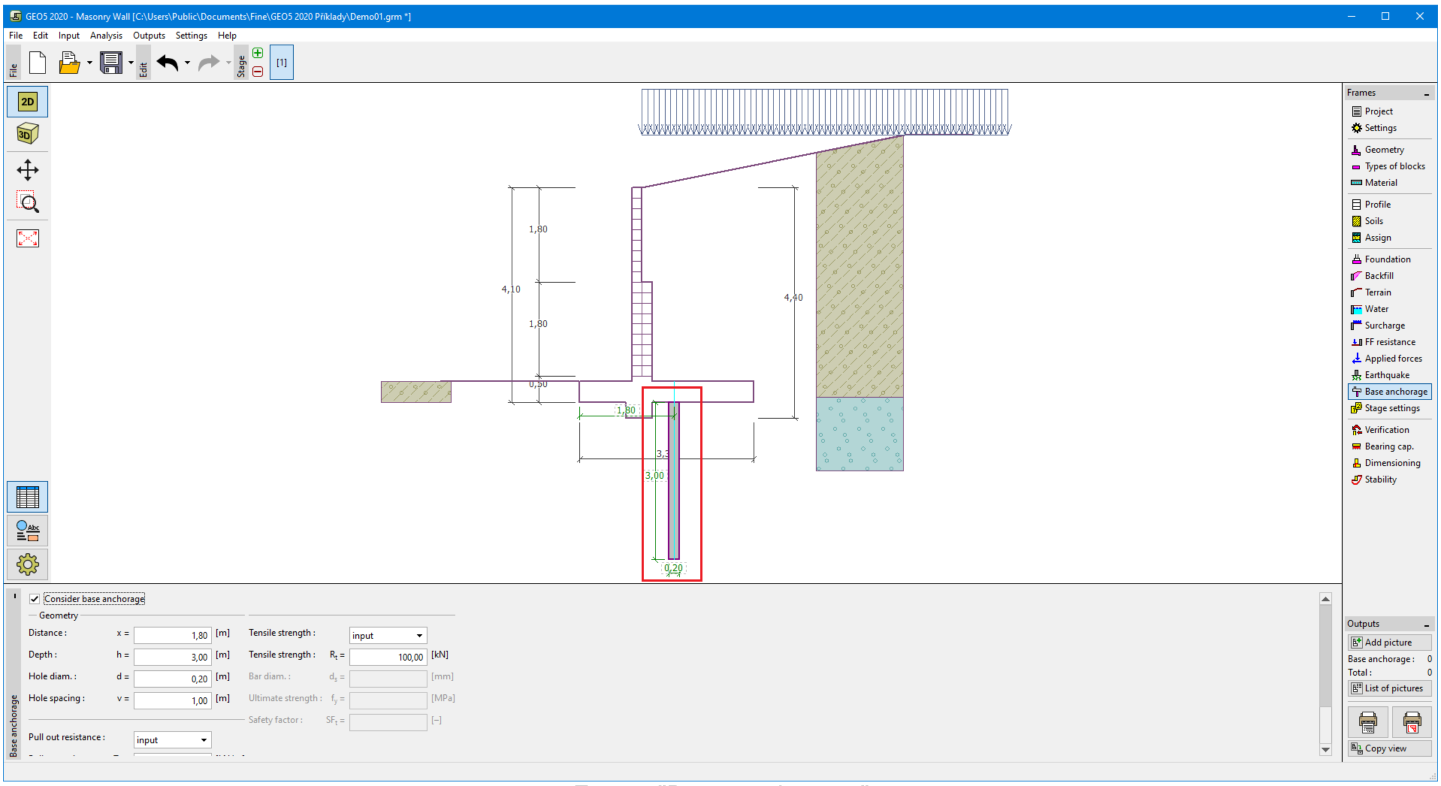
For the LRFD Verification methodology, it is possible to define coefficients for seismic combinations according to the AASHTO.



Frame "Earthquake"

Base Anchorage

The frame "**Base anchorage**" serves to input parameters (anchorage geometry, bearing capacity against pulling-out and pulling-apart) specifying an anchorage of the wall foundation. The geometry of footing anchorage can be edited either in the frame by inserting values in the inputting boxes or on the desktop with the help of active dimensions. The bearing capacity values can be either input or computed by the program from the input parameters.



Frame "Base anchorage"

Stage Settings

The frame "**Stage settings**" serves to input settings valid for a given construction stage.

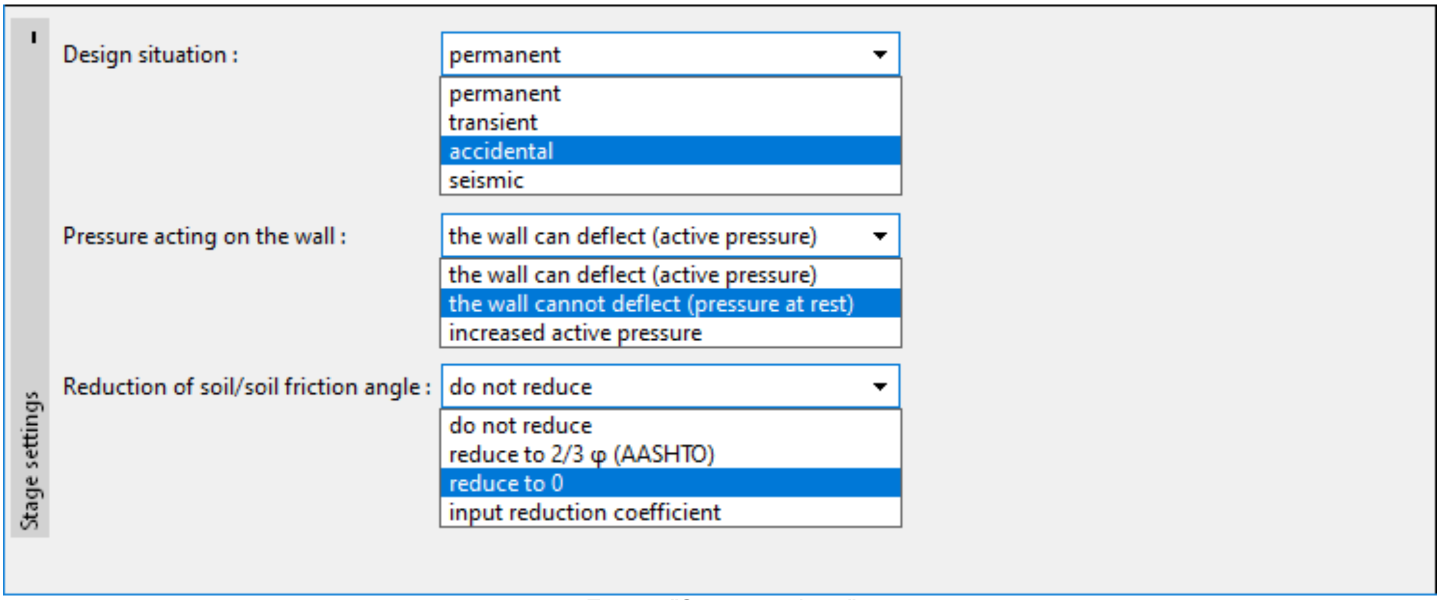
The selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology.

Next, the frame serves to specify the type of pressure acting on a wall based on the allowable wall deformation. Providing the wall is free to move, an active pressure is assumed, otherwise, a pressure at rest is used. The third option enables to load the wall by increased active pressure.

The reduction of soil/soil friction angle can be considered in one of following ways:

- **do not reduce**
- **reduce to 2/3φ (AASHTO)**
- **reduce to 0**
- **input reduction coefficient**



Frame "Stage settings"

Verification

The frame "**Verification**" shows the analysis results. Several computations can be carried out for a single task.

The frame appearance is adjusted based on the selected verification methodology.

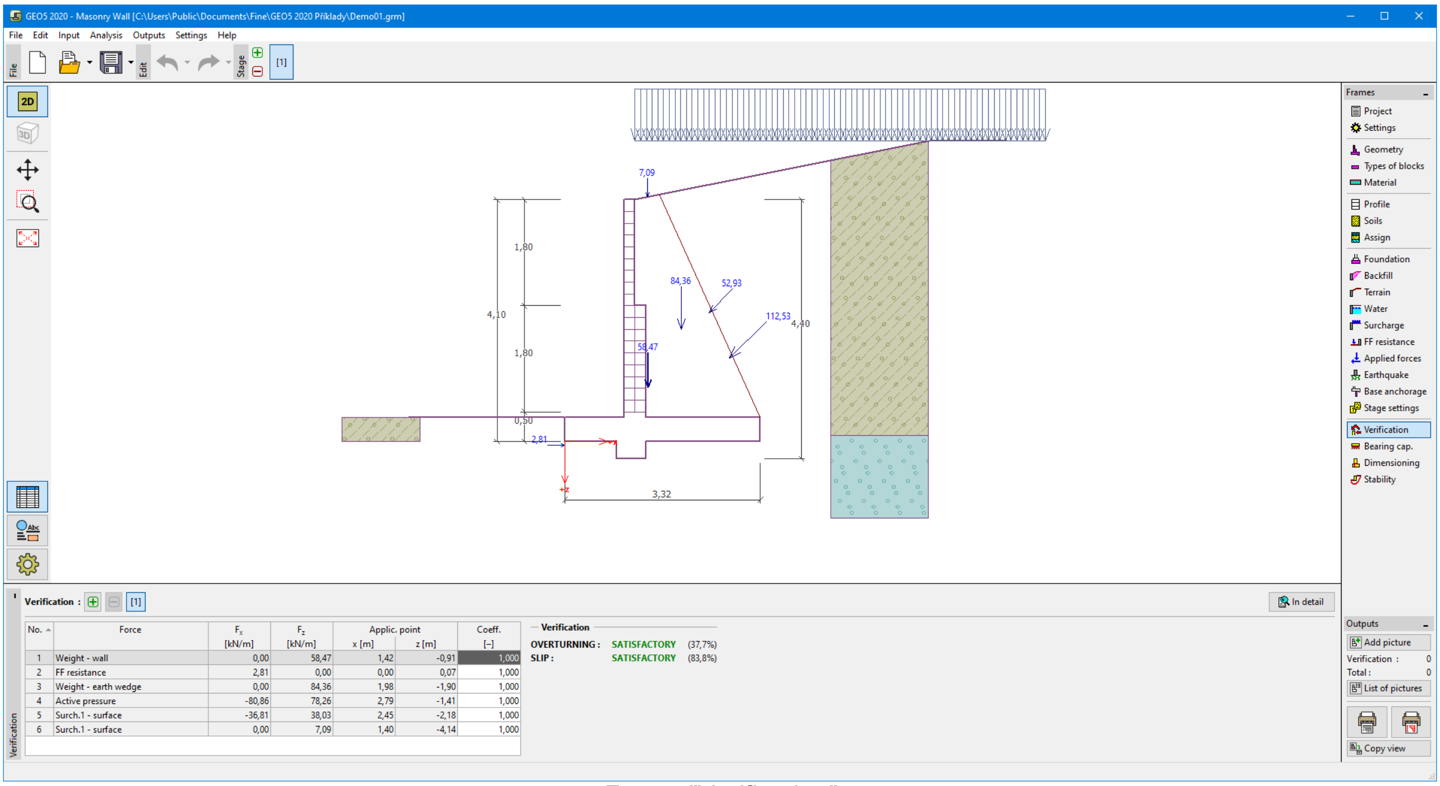
- Verification according to the factor of safety or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows for specifying whether the load acting on a structure is considered as the secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in this case, the last column is not displayed.

The wall is loaded either by active pressure or pressure at rest depending on input in the frame "Stage settings".

The procedure for wall verification is described in the theoretical part of the help.

The computed forces are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against **overturning and translation**. The "**In detail**" button opens the dialog window, which contains a detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Verification"

Bearing Capacity

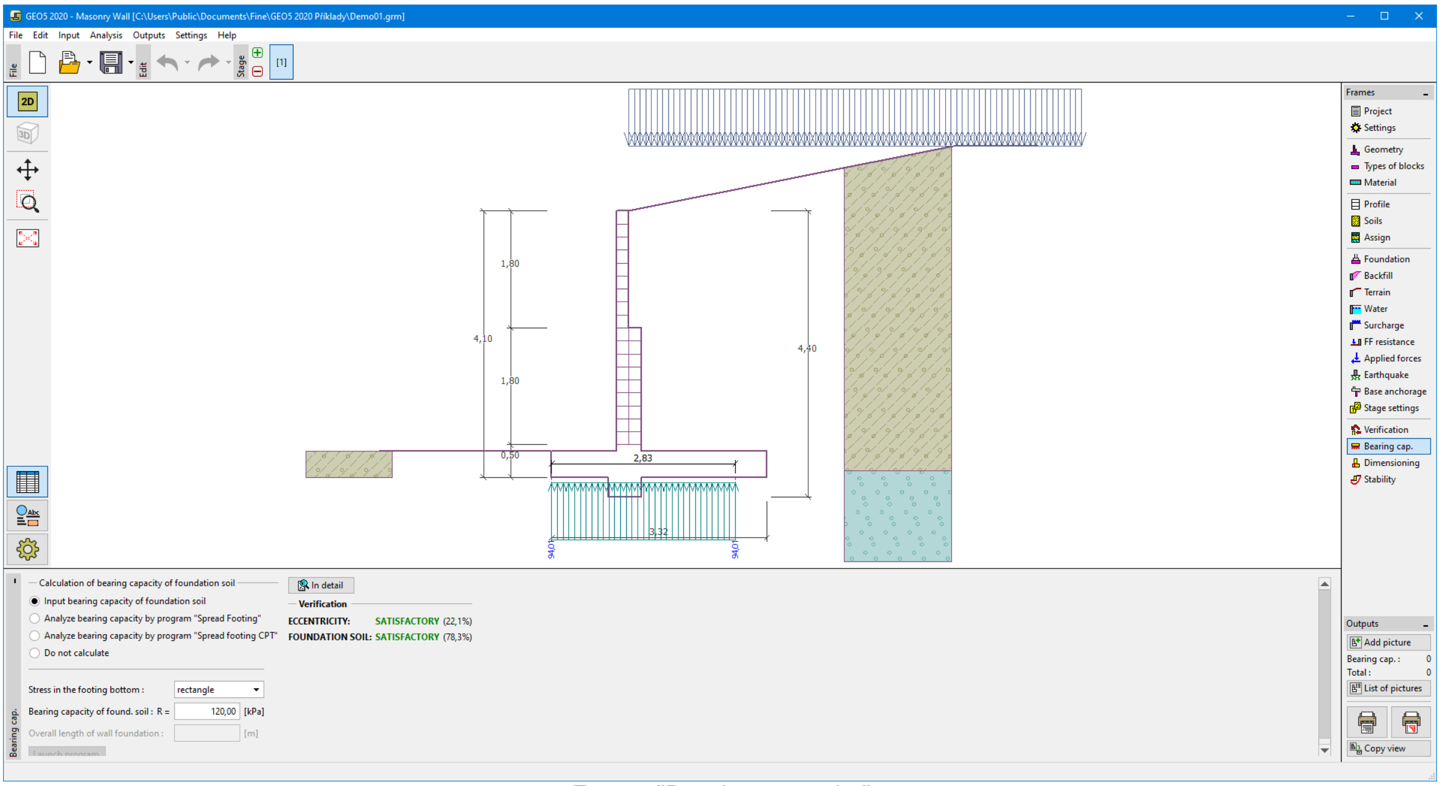
The "**Bearing capacity**" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the frame "Verification". The programs "**Spread footing**", "**Spread footing CPT**", "**Pile**" and "**Pile group**" then consider all verifications as load cases. In the program "**Pile CPT**", just normal load is used.

The frame contains the following analysis options:

- **Insert bearing capacity of foundation soil** — The foundation soil bearing capacity is input. The **eccentricity** and bearing capacity analysis results are displayed in the right part of the frame. The "**In detail**" button opens a dialog window that displays a detailed listing of the results.
- **Analyze bearing capacity by program "Spread footing"** — Pressing the "**Run program Spread footing**" button opens the program "**Spread footing**" which allows us to calculate the soil bearing capacity or settlement and rotation of the footing. Pressing the "**OK**" button leaves the analysis mode - the results and all plots are transferred into the program "**Masonry wall**". The "**Spread footing**" program must be installed for the button to be active. The overall length of the wall foundation is input.
- **Analyze bearing capacity by program "Spread footing CPT"** — The procedure is identical as if calculating soil bearing capacity by the **"Spread footing"** program.
- **Analyze bearing capacity by program "Pile"** — The procedure is identical as if calculating soil bearing capacity by the "**Spread footing**" program. The "**Run program Pile**" is available if the wall has a pile foundation (frame "Foundation"). Pile spacing s is input.
- **Analyze bearing capacity by program "Pile CPT"** — The procedure is identical as if calculating soil bearing capacity by the "**Pile**" program.
- **Analyze bearing capacity by program "Pile group"** — The procedure is identical as if calculating soil bearing capacity by the "**Spread footing**" program. The **"Run program Pile group**" is available if the wall has pile foundation with more then one pile (frame "Foundation"). Pile spacing s , the overall number of pile rows n and loading length l are input.
- **Do not calculate (pile footing)** — The foundation soil bearing capacity is not calculated.

The program allows us to specify a shape of stress in the footing bottom.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Bearing capacity"

Dimensioning

The "**Dimensioning**" frame serves to design and verify the reinforcement of the wall cross-section - the cross-section subjected to dimensioning is selected from the list at the right part of the frame.

- **Construction joint verification** - the number of a joint between masonry blocks is input
- **Wall jump verification**
- **Verification of heel of wall**

The frame appearance is adjusted based on the selected verification methodology.

- Verification according to the factor of safety or the theory of limit states - the last column in the table allows us to input the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows us to specify whether the load acting on a structure is considered to be a secondary one. This is explained in more detail in the "Load combinations" section.
- Analysis according to LRFD - in this case, the last column is not displayed.

Calculation of forces and their action on the analyzed cross-section is described here.

The wall is loaded either by the active earth pressure or by the pressure at rest depending on the setting in the "Stage settings" frame.

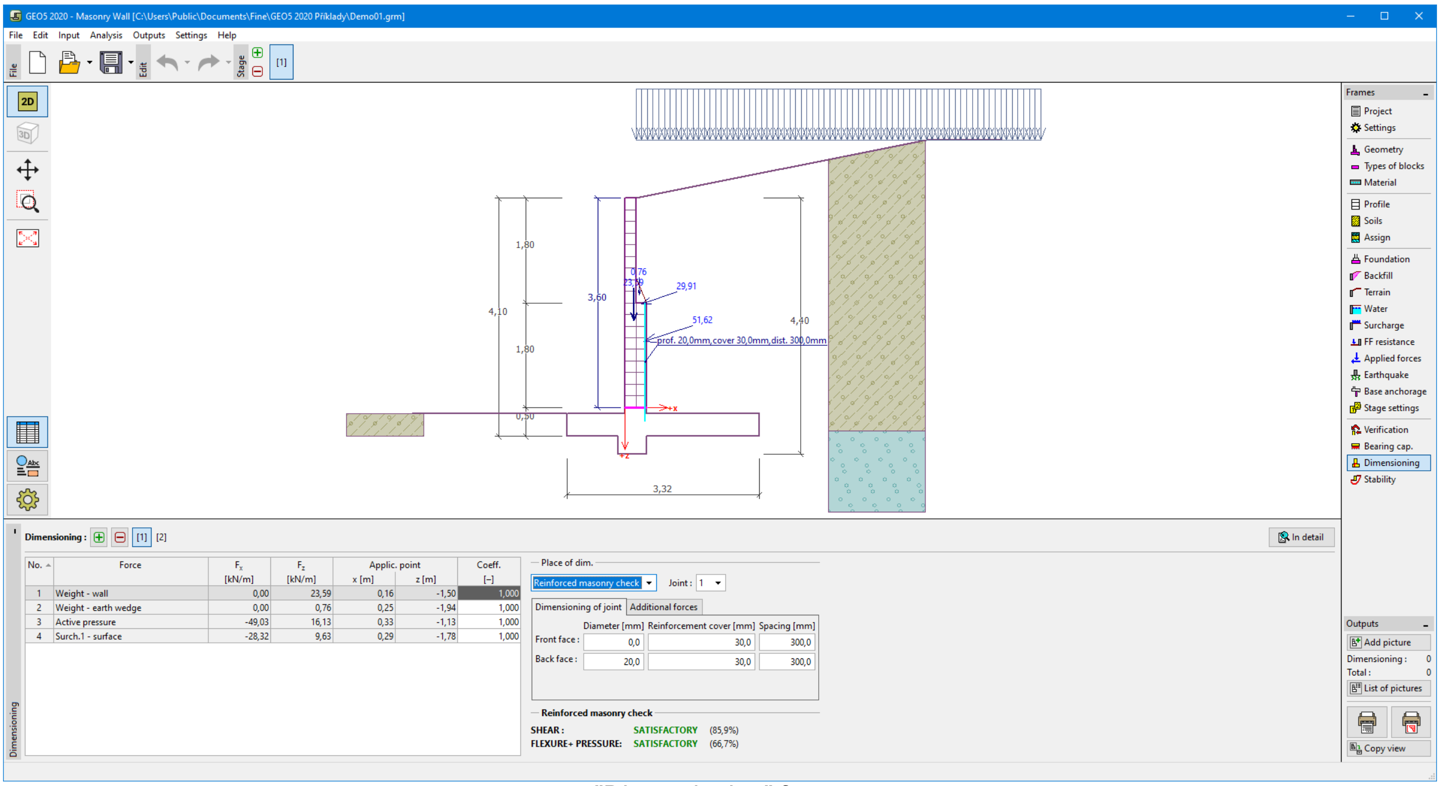
The way to calculate the internal forces in cross-sections is described in the theoretical part of this documentation. In addition, force from earth pressure at rest is taken into account when considering earthquake analysis.

Joints between masonry blocks are verified according to AS 3700 or EN 1996-1-1 depending on the setting in the "Materials and standards" tab. The program verifies the bearing capacity for bending, shear and combination of compression and bending. Reinforcement can be specified on both front and back sides of a structure. An additional load applied to a cross-section (bending moment, compressive normal force and shear force) can also be specified. These additional forces are added to the computed ones.

Dimensioning of the reinforced concrete is performed according to the standard set in the "Materials and standards" tab.

Several computations for different reinforcement variants can be carried out. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "**In detail**" button opens the dialog window that contains a detailed listing of the dimensioning results.

Visualization of results can be adjusted in the "Drawing Settings" frame.

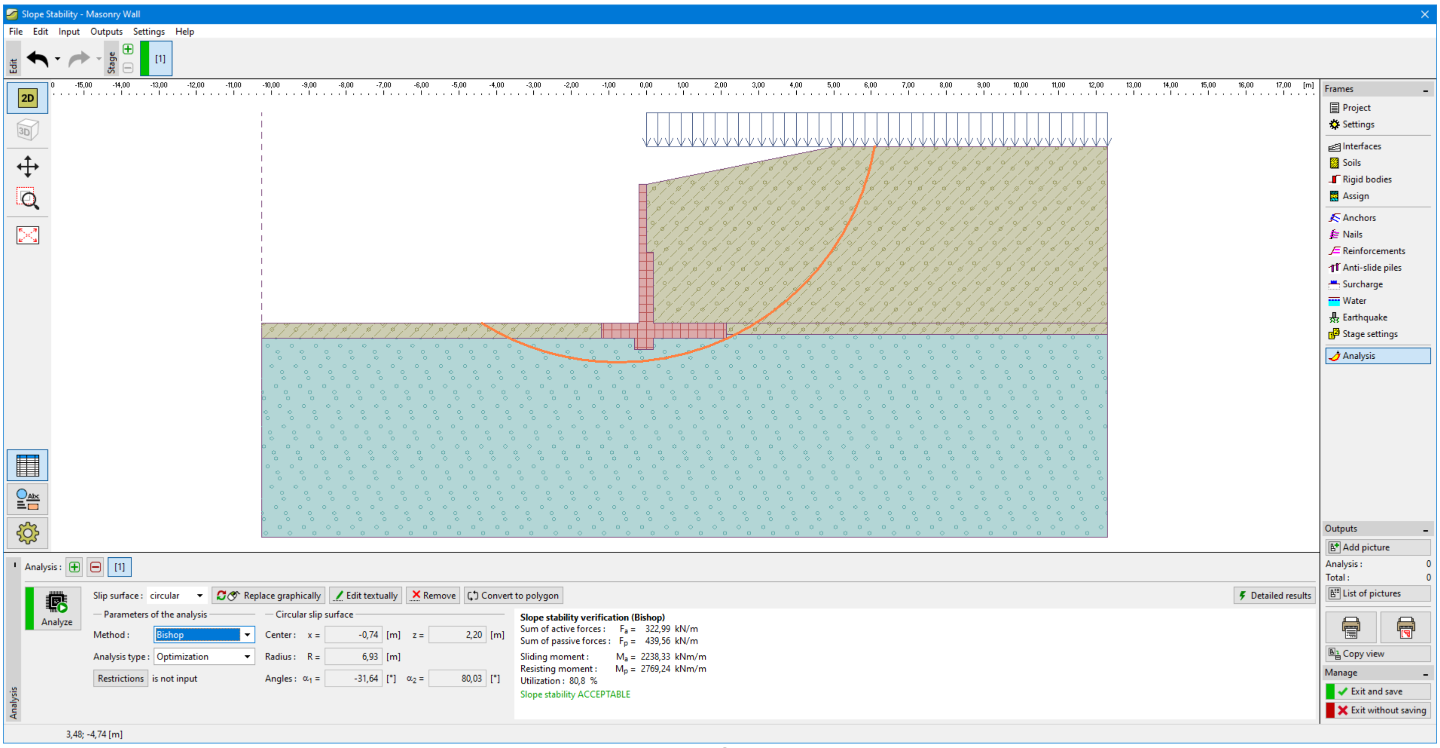


"Dimensioning" frame

Stability

Pressing the "**Stability**" button launches the "**Slope stability**" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program "**Slope stability**" is installed.

After completing all analyses press the "**OK**" button to leave the program - all data are then carried over to the analysis protocol of the "**Masonry wall**" program.



Frame "Stability"

Program Gabion

This program is used for the analysis of gabions. It allows analysis of any structure shapes including overhangs requiring anchoring.

The help in the program "Gabion" includes the following topics:

- The input of data into individual frames:

Project	Settings	Material	Geometry	Profile	Soils	Assign
Foundation	Backfill	Terrain	Water	Surcharge	Front Face	Applied Forces
					Resistance	
Earthquake	Stage Settings	Verification	Bearing Capacity	Dimensioning	Stability	

- Standards and analysis methods

- Theory for analysis in the program "**Gabion**":

Stress in Soil Body	Earth Pressures	Analysis of Walls	Analysis of Foundation Bearing Capacity

- Outputs

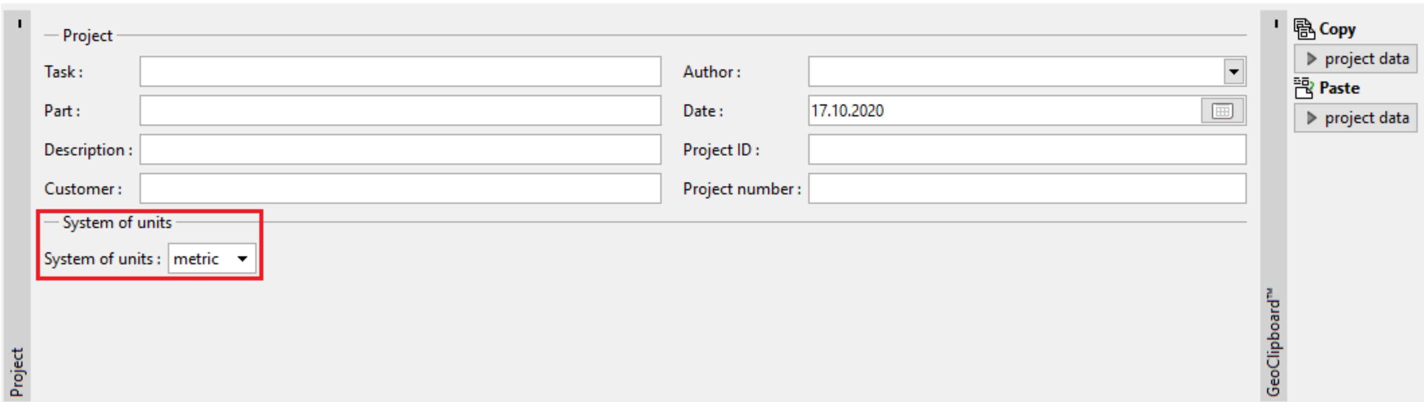
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The frame **"Project"** is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The frame "**Settings**" allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the

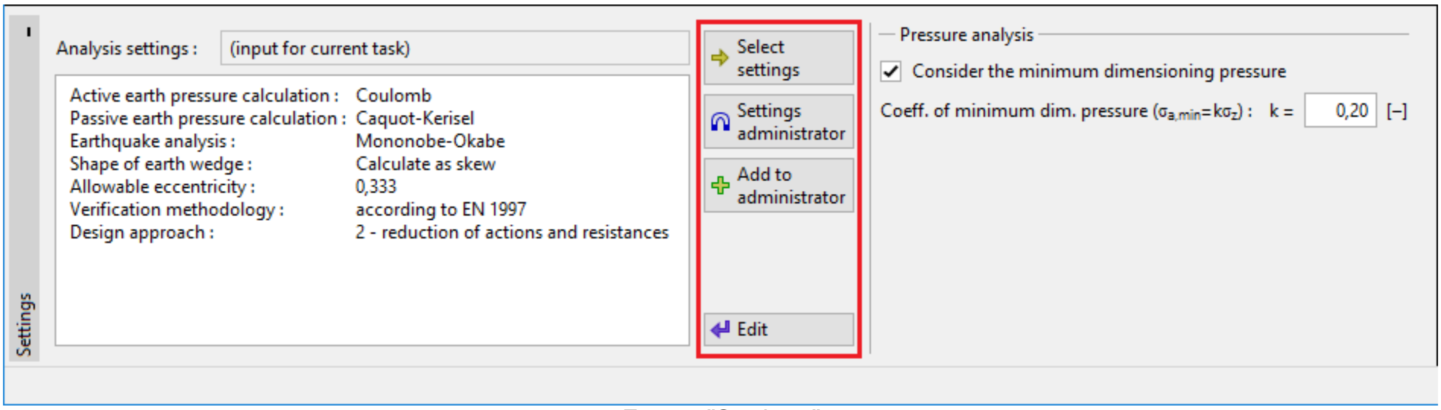
Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

The program allows us to specify a value of the minimum dimensioning pressure (by checking the option "**Consider the minimum dimensioning pressure**").

Settings of analysis parameters are performed in the "Materials and standards" and "Wall analysis" tabs.



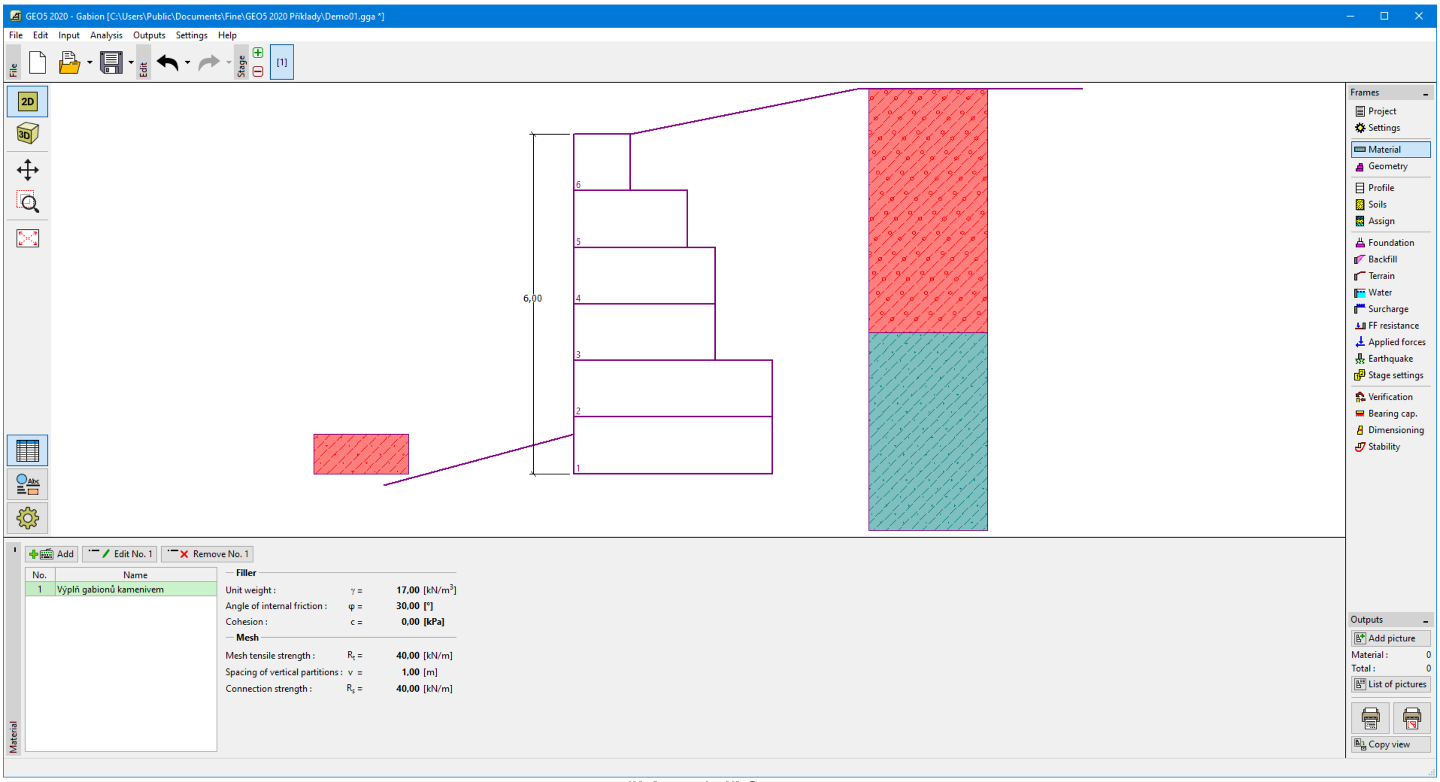
Frame "Settings"

Material

The "**Material**" frame contains a table with a list of input filling (aggregates) and material parameters of applied gabion wire netting. Adding material and netting is performed in the "**New material**" dialog window.

The material parameters of filling and netting of the currently selected gabion block are displayed in the right part of the frame.

An approximate value of the angle of internal friction of the material of gabion filler is for a well graded gravel in the range of 35 - 40, for a quarry masonry, it can be larger.



"Material" frame

Geometry

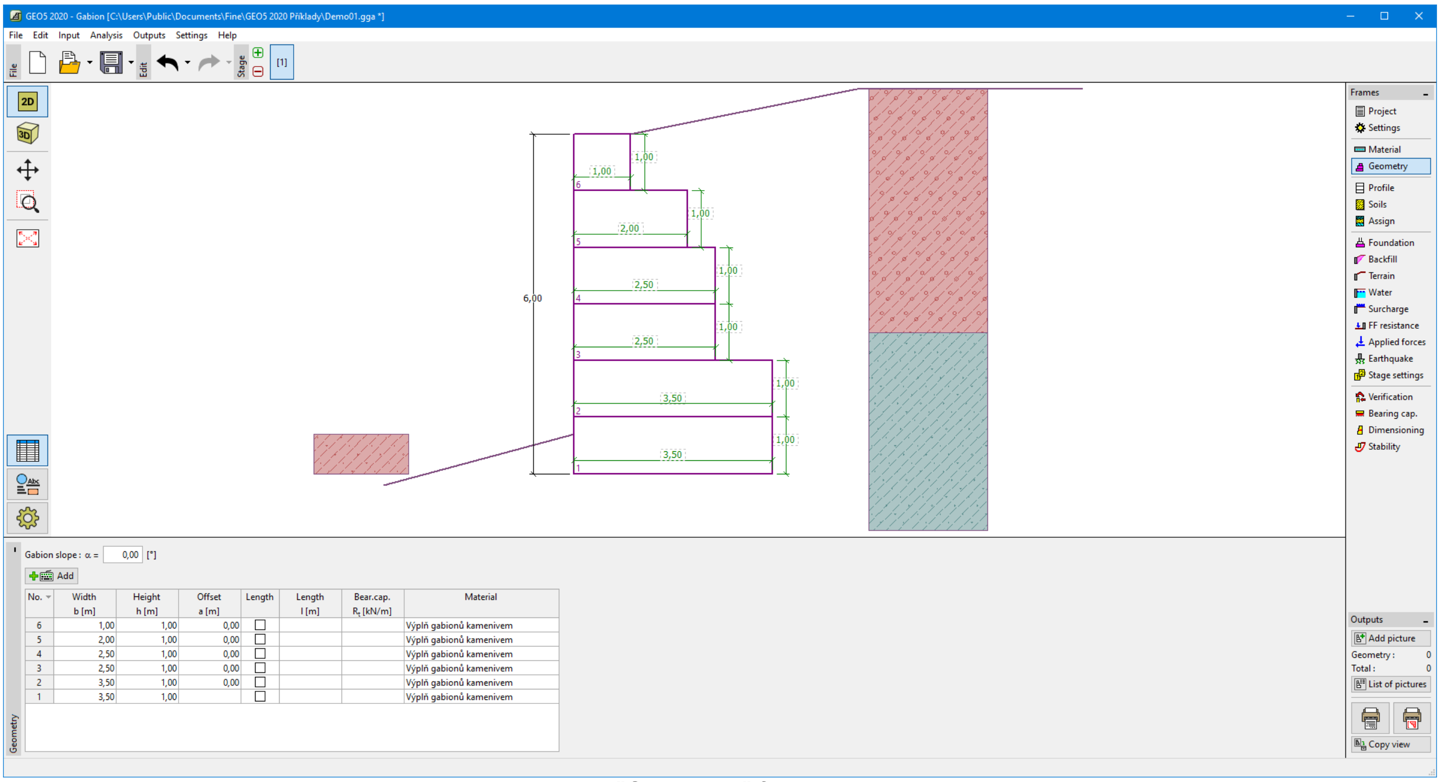
The "**Geometry**" frame contains a table with a list of input blocks of a wall (the lowest block is labeled as No. 1). Adding blocks is performed in the "**New block**" dialog window.

This dialog window serves to define **the geometry of a block**, and parameters of mesh overhang (overhang length, overhang anchorage, bearing capacity against pull out).

The program allows us to add (insert) another block in between two already existing blocks of a structure. Inserting a new block is performed in the "**Insert block**" dialog window that complies with the "**New block**" dialog window. The inserted block is ordered to proceed the currently selected block of a structure.

The input blocks can be can be further edited on the desktop with the help of active dimensions or active objects - double-clicking on a structure opens a dialog window with a given block. **When using the regime of active objects the visualization of detailed dimensions must be turned off in the** "Drawing Settings" **frame**.

The program makes it possible to export the geometry of a structure in the *.DXF format.



"Geometry" frame

Profile

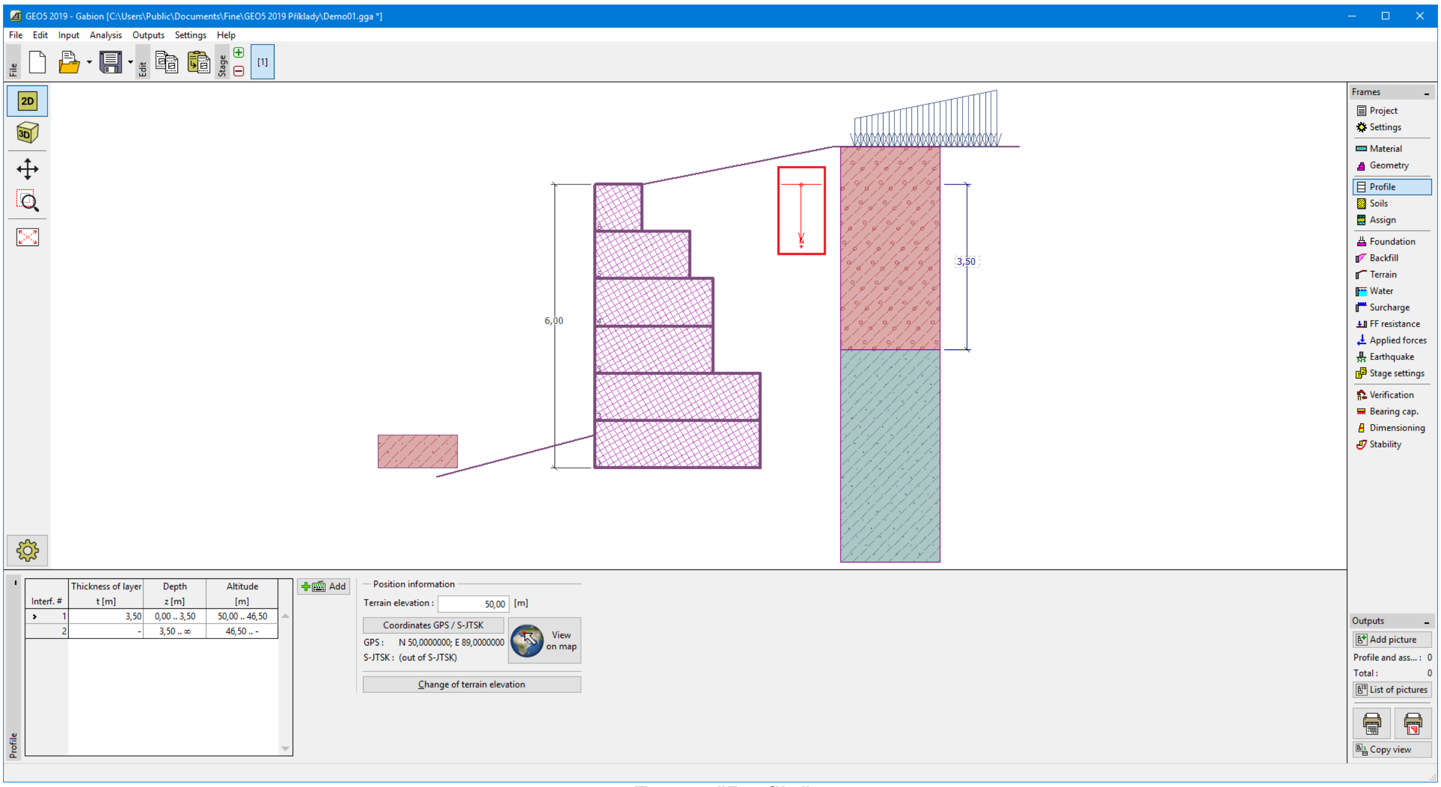
The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.



Frame "Profile"

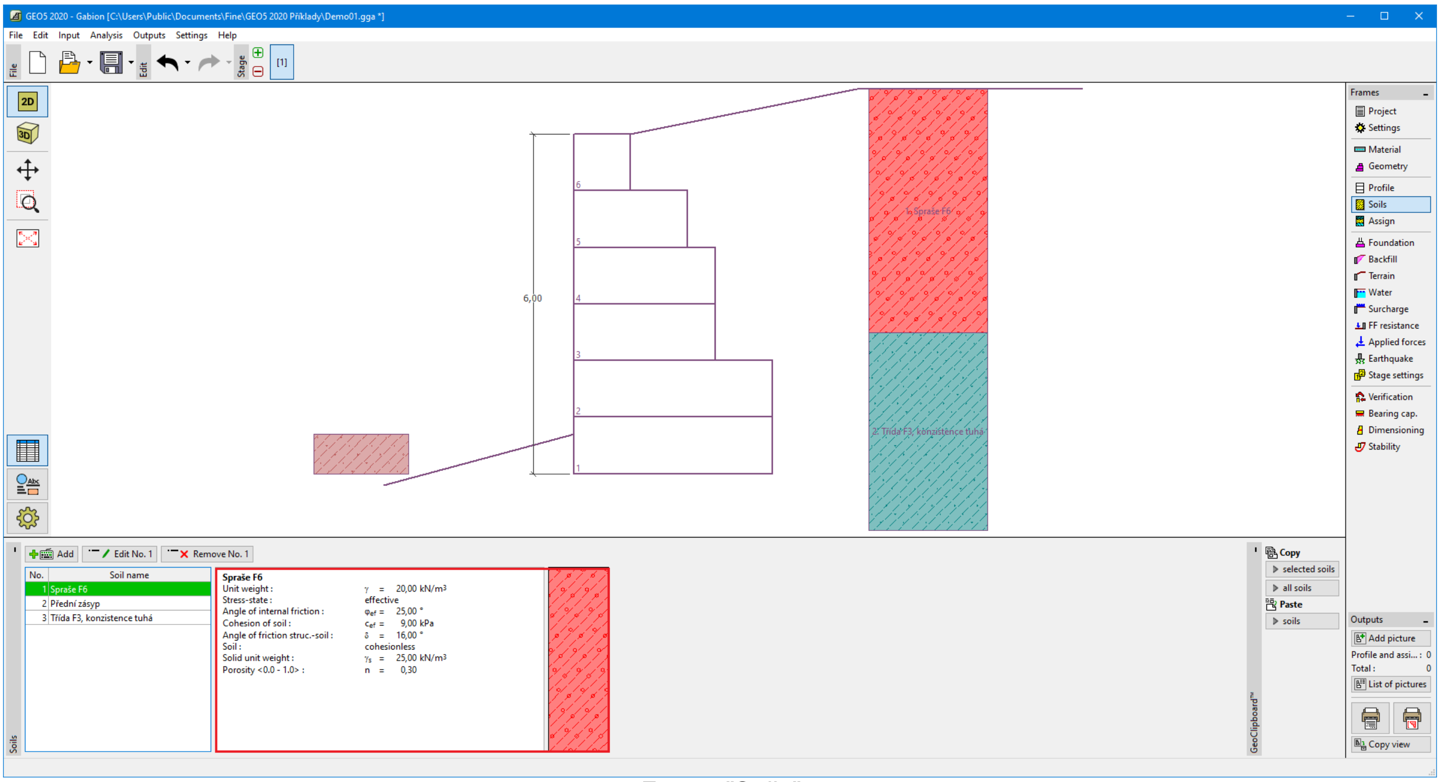
Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

Basic Data

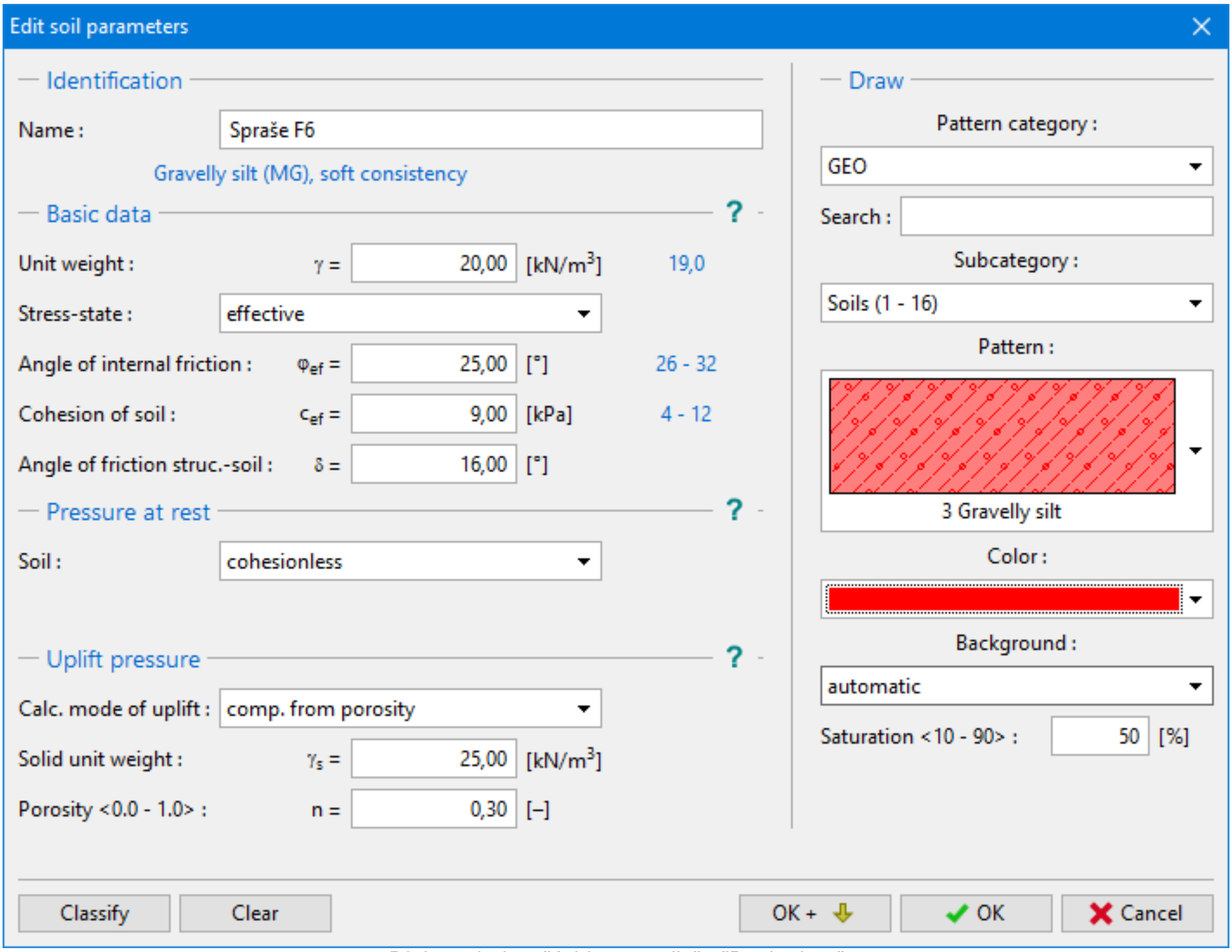
This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction, and cohesion**. The particular values are obtained from a geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit the built-in database of soils, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually.

Either **effective or total** parameters of the angle of internal friction and cohesion are specified depending on the setting in the "**Stress analysis**" combo list. Whether to use effective or total parameters depends primarily on the type of soil, type of load, structure duration, and water conditions.

For effective stress further needs to specify the angle of internal friction between the soil and structure, which depends on the structure material and the type of soil. Possible values of this parameter are listed in the table of recommended values.

For total stress further needs to specify the adhesion of soil to the structure face a .

The associated theory is described in detail in chapter "Earth pressures".

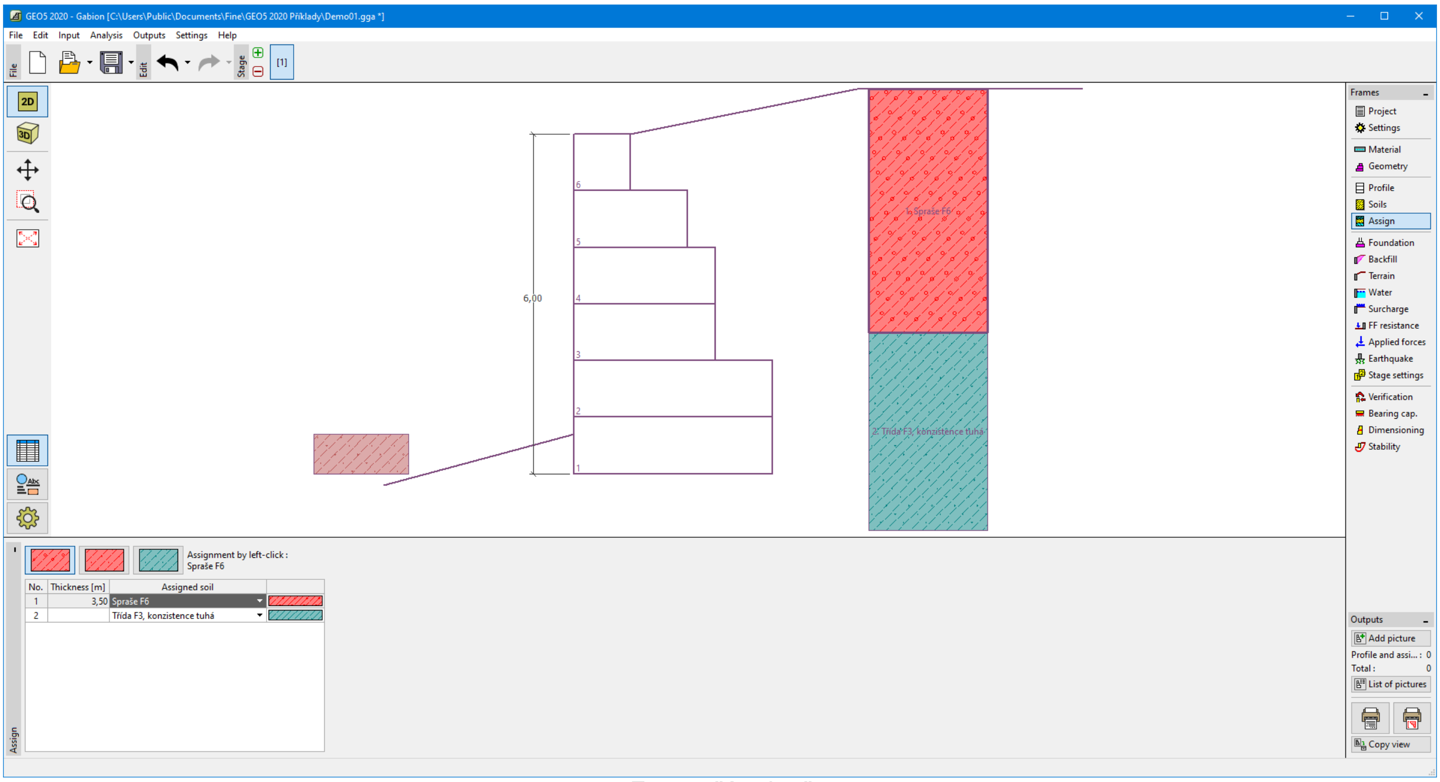


Dialog window "Add new soils" - "Basic data"

Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail herein.



Frame "Assign"

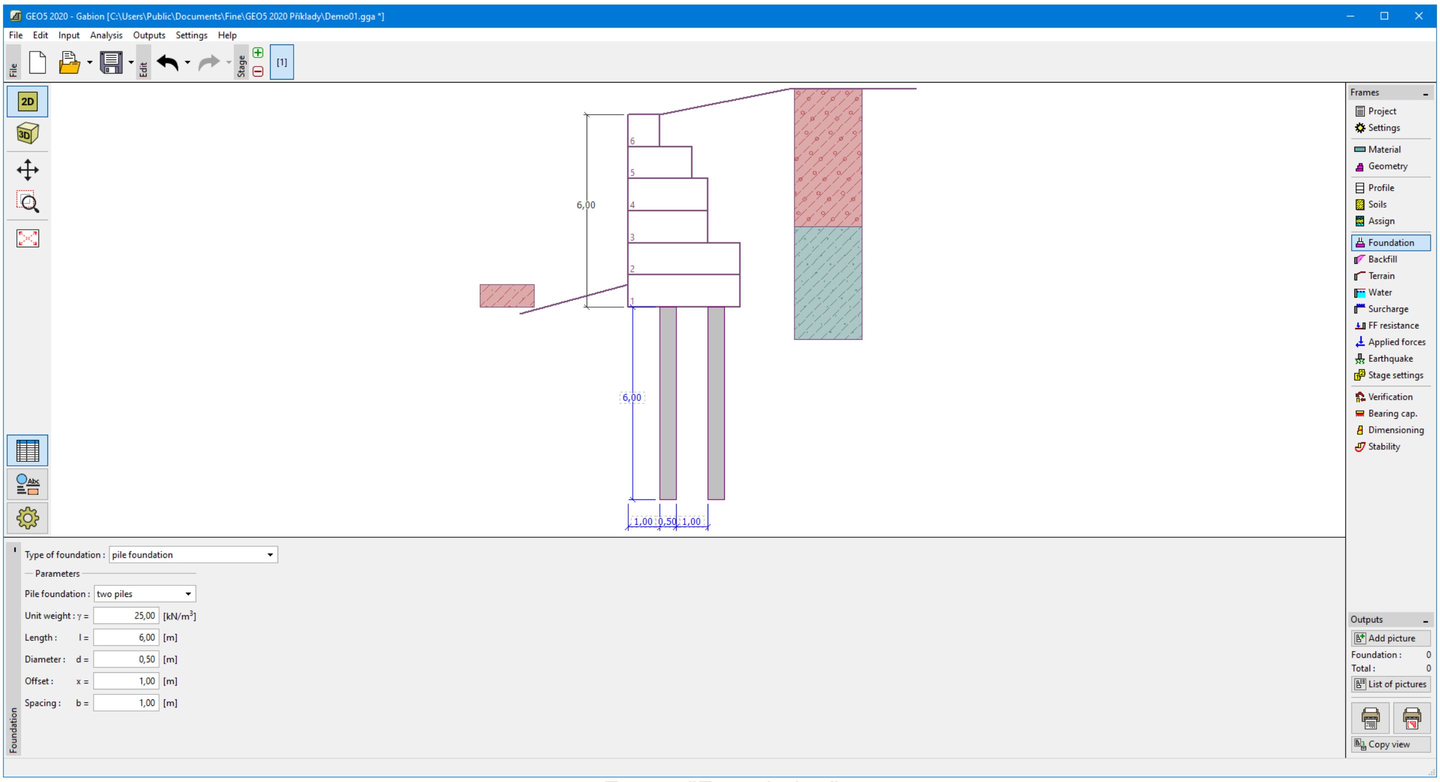
Foundation

The "**Foundation**" frame serves to specify the type of wall foundation. The following types of wall foundation are available:

- **soil from geological profile** - the wall is founded on the soil assigned from the geological profile specified in the "Profile" frame.
- **input parameters of contact base-soil** - parameters of the contact between footing bottom and structure are specified. Option "**input angle of friction base-soil**" requires inputting the friction angle ψ [°] between foundation and soil. Option "**input friction coefficient**" requires specifying the friction coefficient μ [-]. Both options require inputting the cohesion a [kPa] between foundation (base) and soil.
- **strip foundation** - strip foundation material is represented either by **soil** (input in "Soils" frame), or concrete - requires inputting the **unit weight of foundation material** γ and **parameters of contact base-soil** (friction coefficient f, cohesion c, additional resistance F).
- **pile foundation** - the wall can be founded on one row of piles or two rows of piles, respectively.

Strip foundation and **pile foundation** can be adopted for the wall foundation only if the type wall with **straight footing bottom without jump** is selected in the "Geometry" frame. The geometry of the wall foundation (**strip foundation**, **pile foundation**) can be modified either in the frame by entering specific values into the inputting fields or on the desktop with the help of active dimensions.

The input data introduced in this frame influence the actual wall analysis (check for slip) and further the bearing capacity of foundation soil.

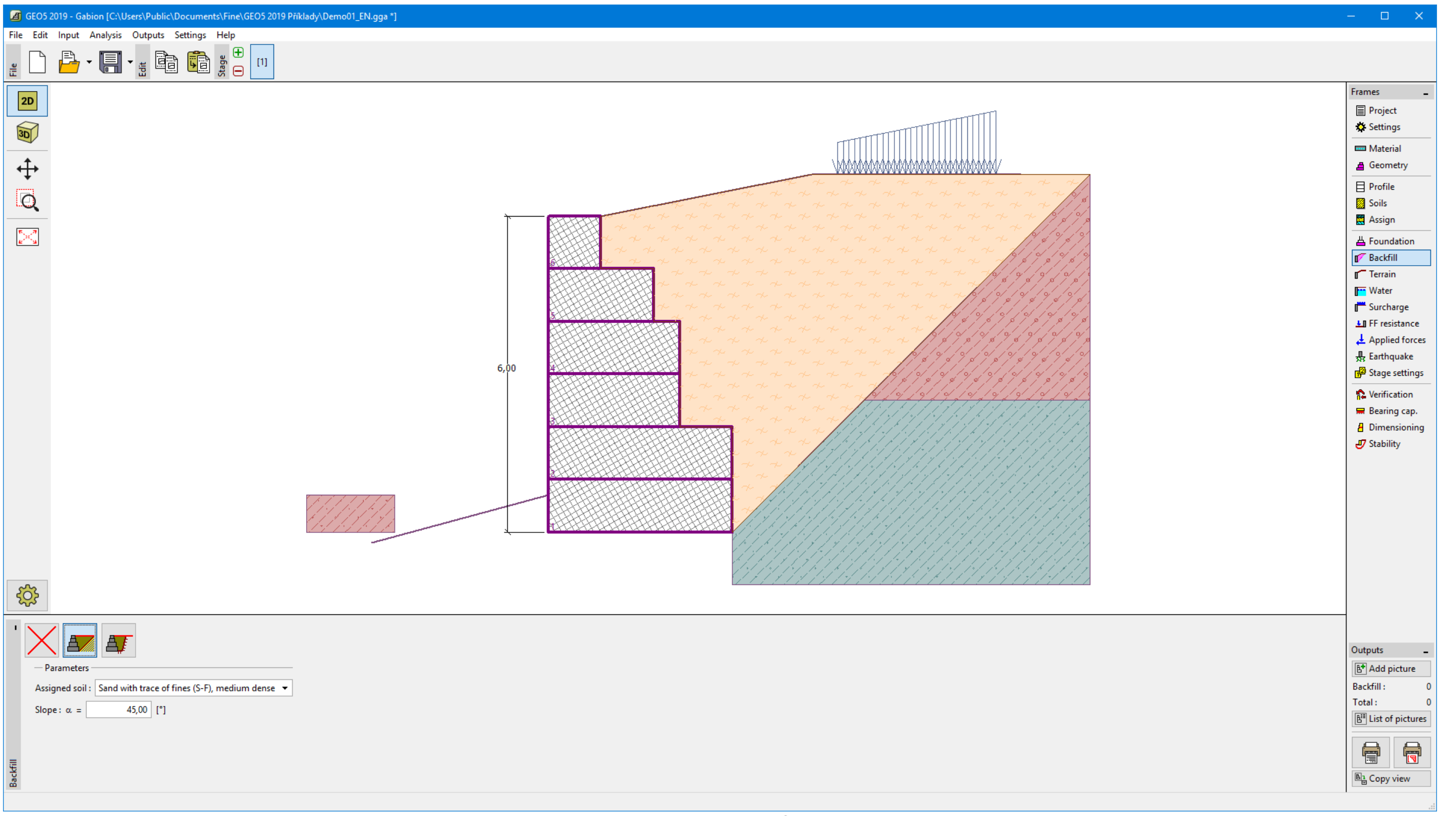


Frame "Foundation"

Backfill

The frame "**Backfill**" allows the selection of backfills behind the structure.

Analysis of earth pressures with the influence of backfill is described in the theoretical part of the help: "Influence of Backfill".



Frame "Backfill"

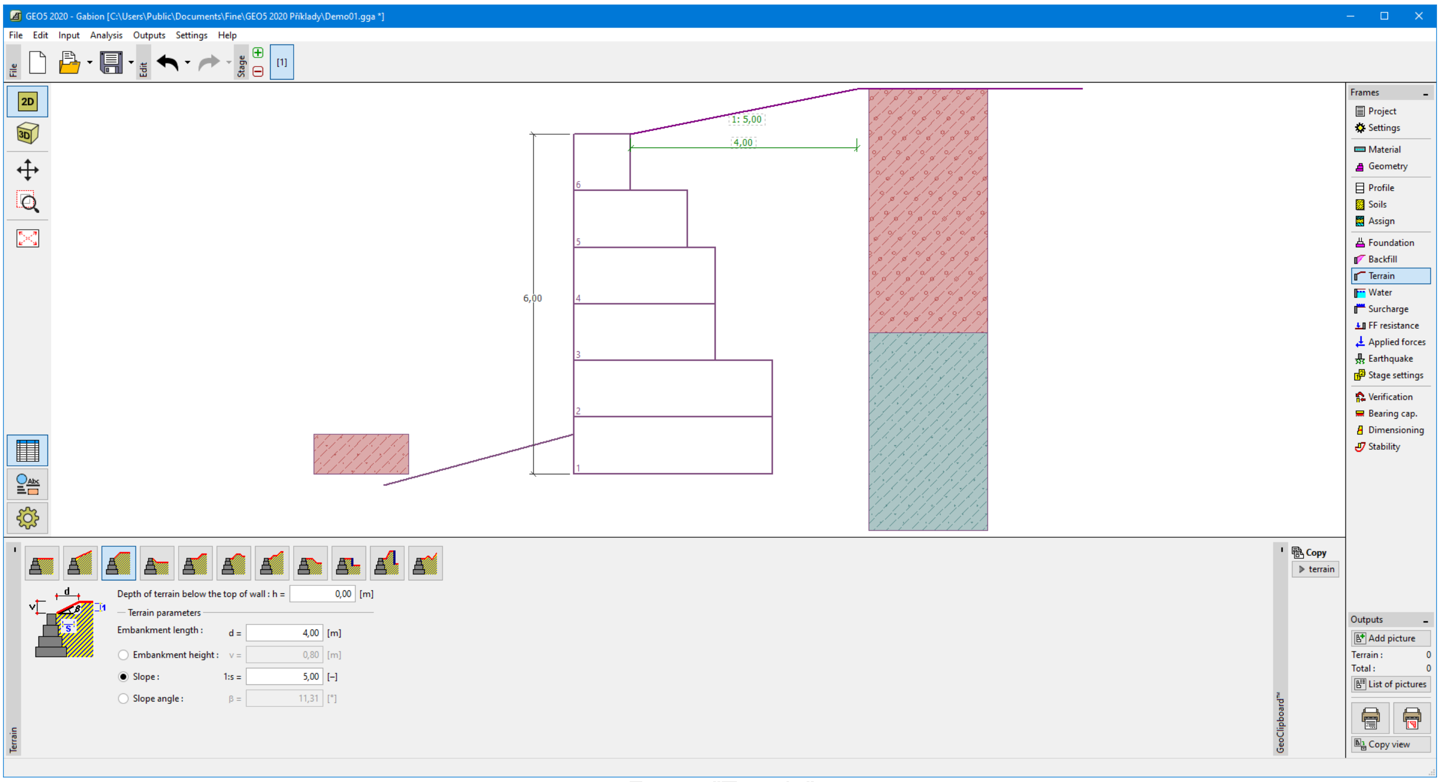
Terrain

The "**Terrain**" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by

inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case, the frame contains a table with a list of terrain points. The first point with coordinates [0; 0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "Distribution of earth pressures for broken terrain".



Frame "Terrain"

Water

The "**Water**" frame allows, by pressing the button, for selecting the type of water. The selected type with a graphic hint of input values is displayed in the left part of the frame. Water parameters (h_1 , h_2 ...) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

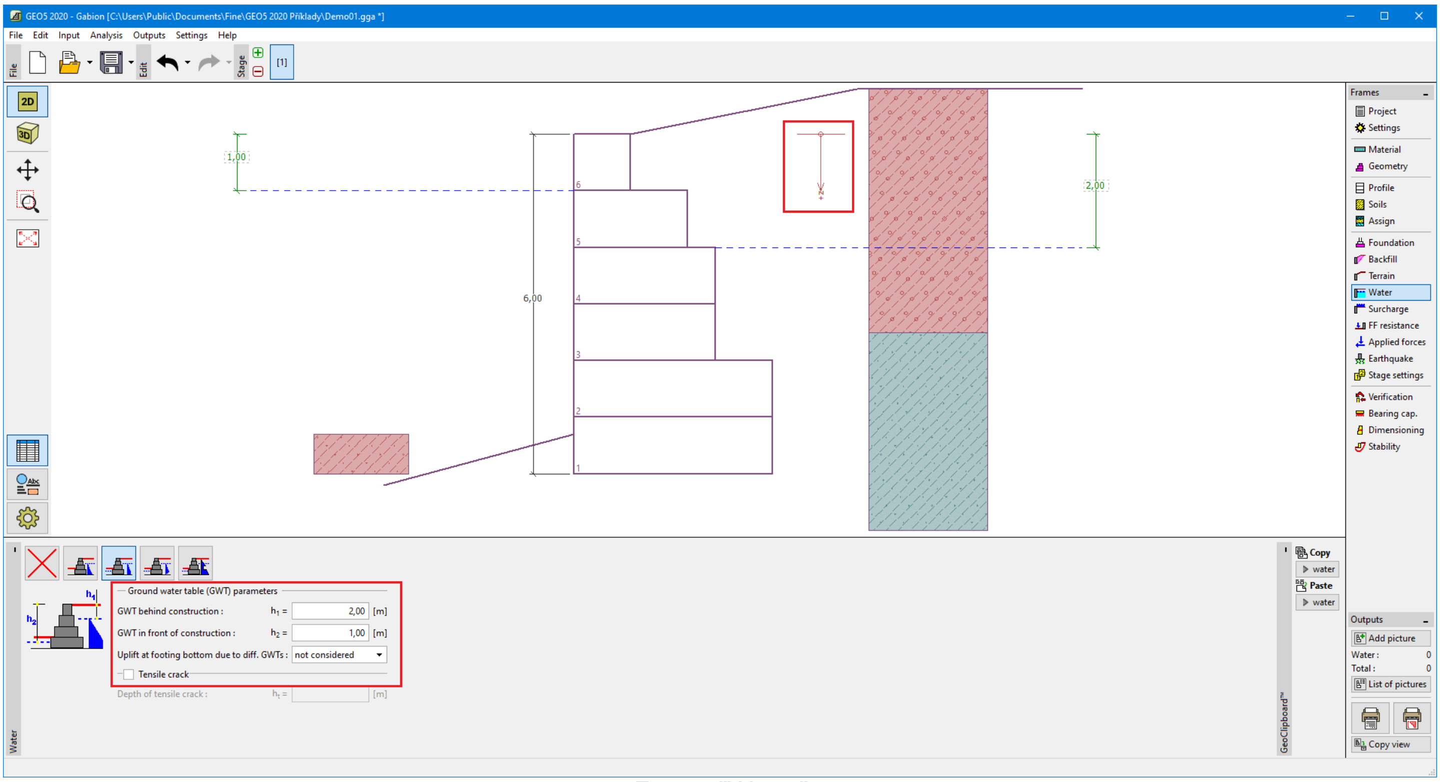
The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be linear, parabolic, or it may not be considered at all. When verifying the wall, the uplift pressure in foundation joint due to different water tables is introduced in terms of a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tabs "**In front of structure**" and "**Behind structure**" appear with tables. The table contains the values of pore pressure in front of the structure or behind it.

The groundwater table can also be specified **above the structure** or earth profile, respectively - in such a case, the depth of water is input with a negative value.

Analysis of earth pressures with the influence of water is described in the theoretical part of the help, chapter "Influence of water".

The program further allows for specifying a depth of tensile cracks filled with water.



Frame "Water"

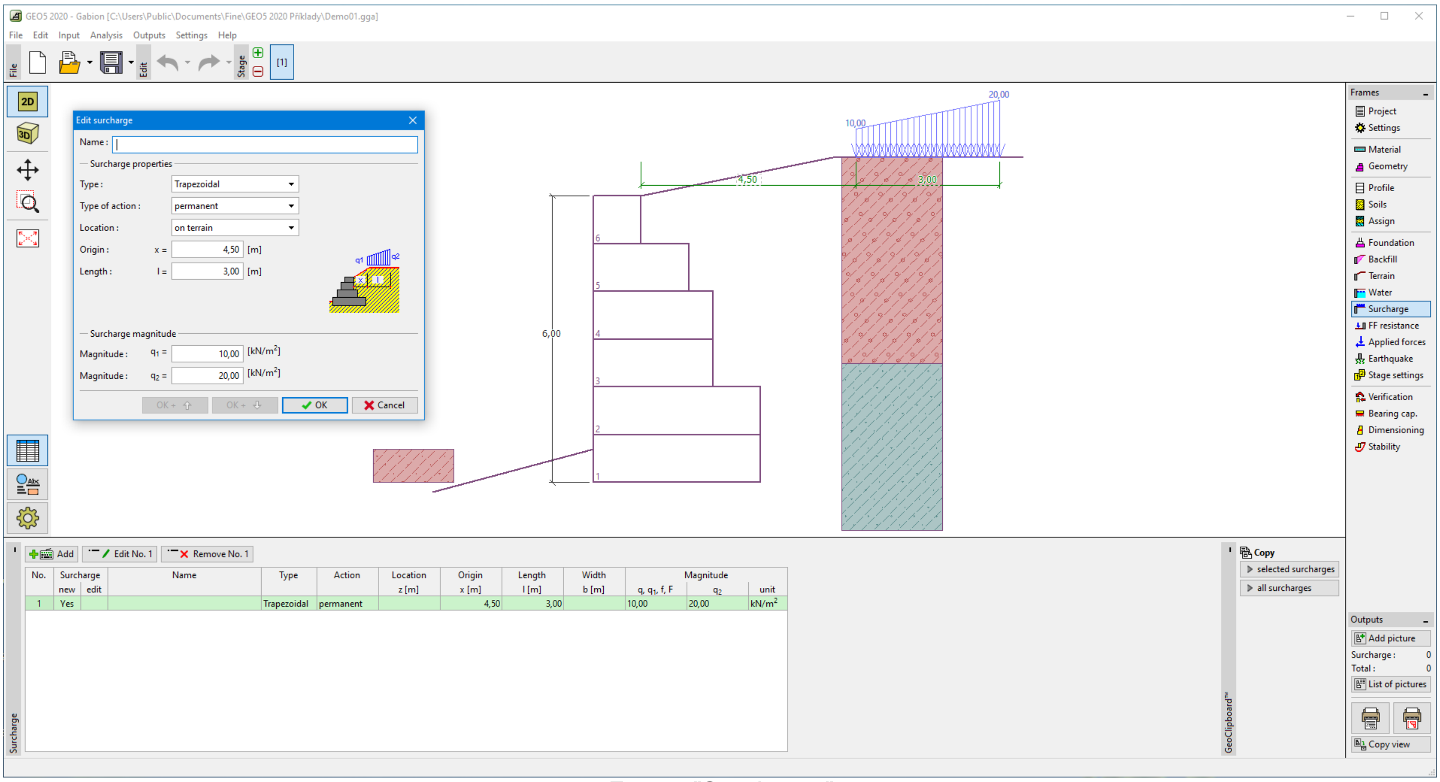
Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The z -coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case when the surcharge is found out of the terrain the program prompts an error message before calculation.

Either **permanent**, **variable,** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with a favorable effect is not considered in the analysis.

Analysis of earth pressures due to surcharges is described in the theoretical part of the help, chapter "Influence of surcharge".



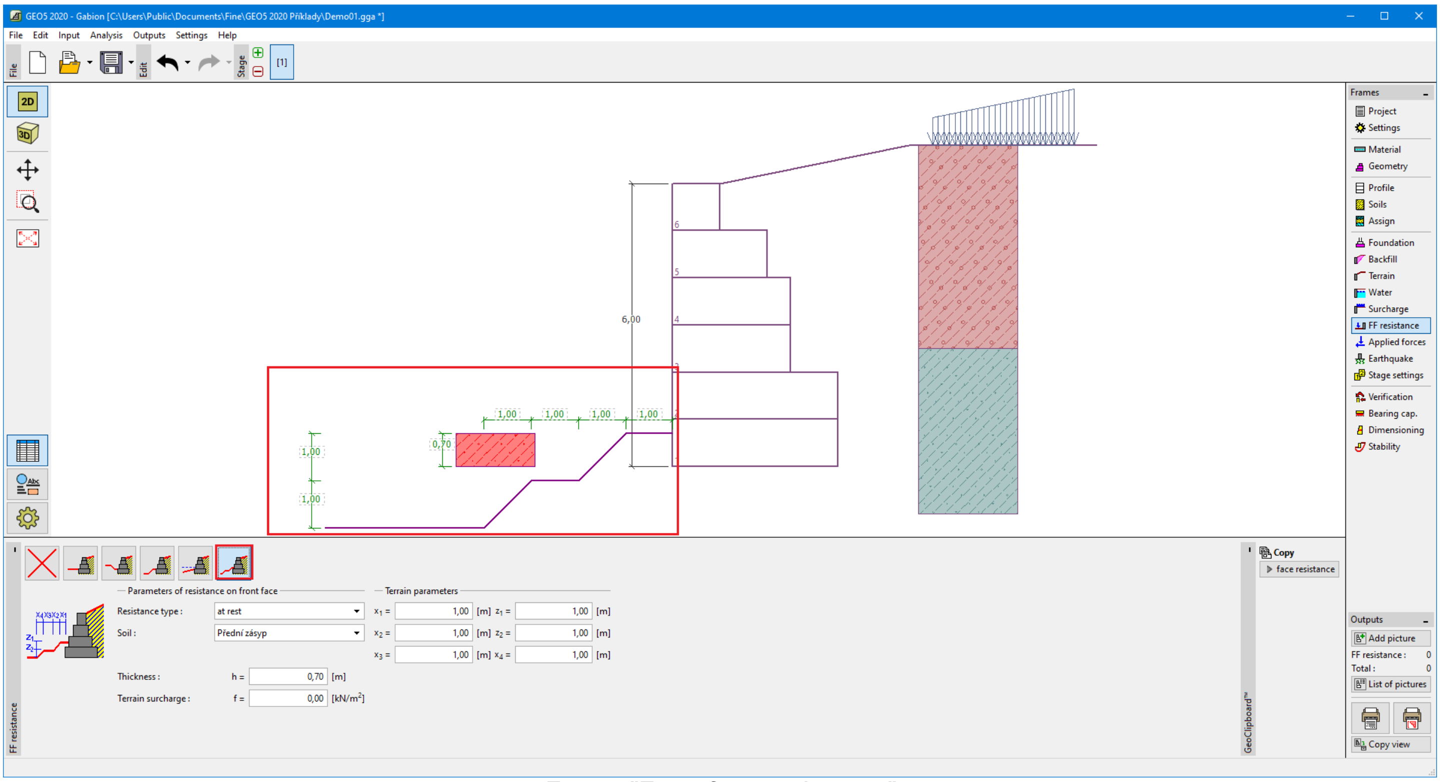
Frame "Surcharge"

Front Face Resistance

The "**Front face resistance**" frame allows us to specify the terrain shape in front of the structure and parameters of front face resistance. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop using active dimensions.

Combo lists in the frame allow selecting the type of resistance and a soil (from the soils introduced in the "Soils" frame). The terrain surcharge magnitude and soil thickness (above the wall lowest points) can also be specified.

The resistance on a structure front face can be specified as the pressure at rest, passive pressure, or reduced passive pressure. The resulting force due to the reduced passive pressure is calculated as a resultant force caused by pressure at rest and passive pressure multiplied by corresponding coefficients.



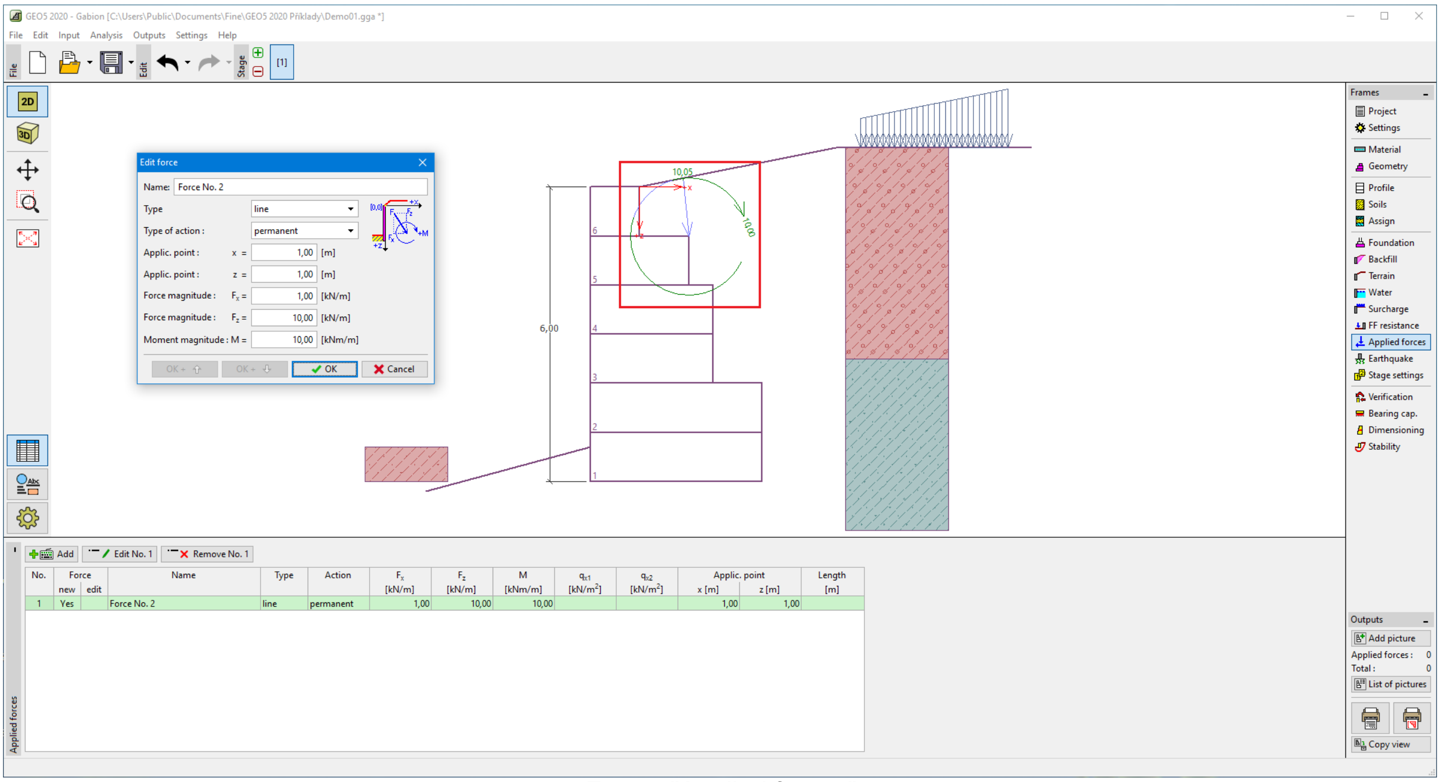
Frame "Front face resistance"

Applied Forces

The "**Applied forces**" frame contains a table with a list of forces acting on a structure. Adding forces is performed in the "**New force**" dialog window. The input forces can also be edited on the desktop with the help of active objects.

Applied forces represent an additional load on the structure of the wall, sheeting or MSE wall. We can model such as an anchoring crash barrier, crash vehicle, load from billboards and hoardings etc. The program does not modify the applied forces in the calculation except multiplying them with the corresponding coefficients according to the selected type of load (EN1997, LRFD).

External load acting on the terrain is necessary to define as a surcharge.



Frame "Applied forces"

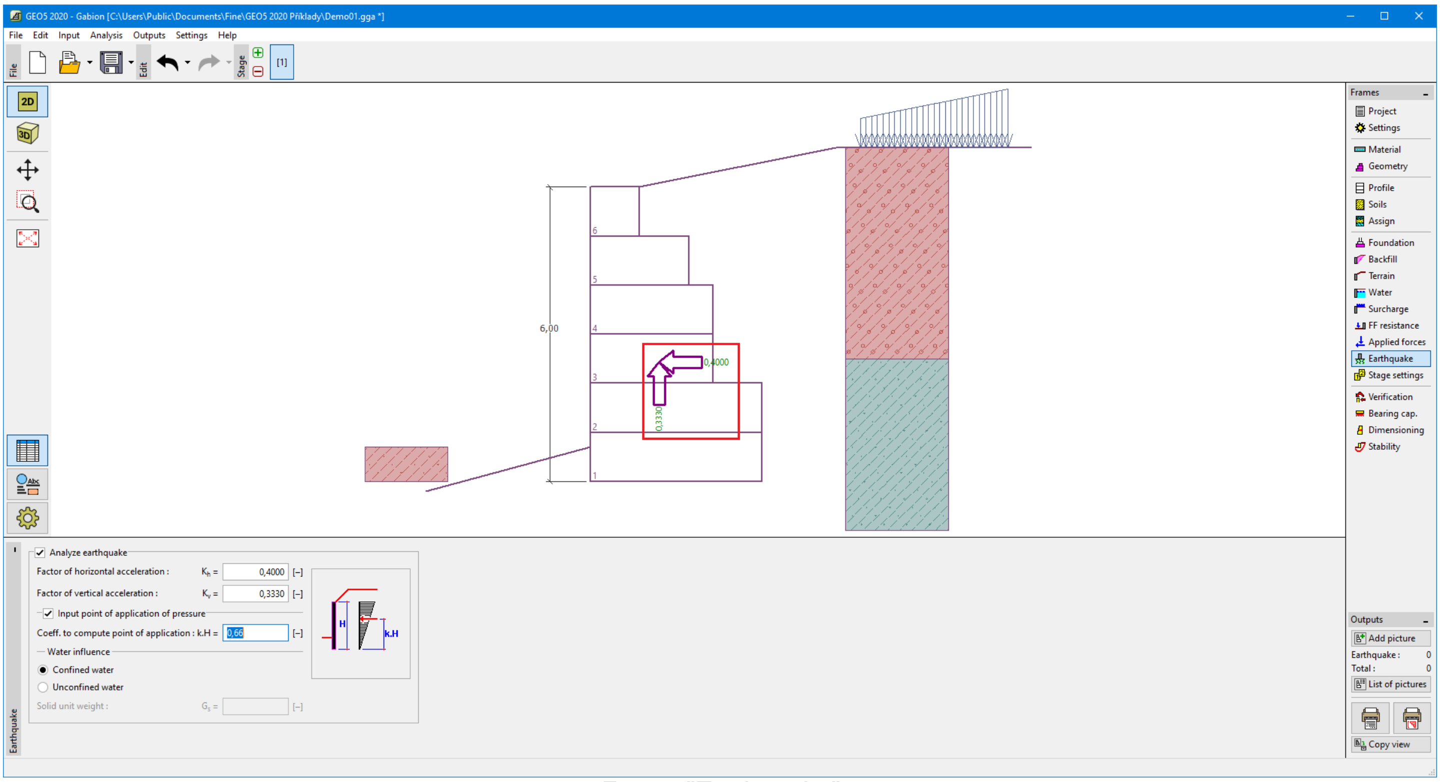
Earthquake

The "**Earthquake**" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements, the coefficients k_h and k_v can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for an earthquake is described in the theoretical part of the help, chapter "Influence earthquake".

For the LRFD Verification methodology, it is possible to define coefficients for seismic combinations according to the AASHTO.



Frame "Earthquake"

Stage Settings

The frame "**Stage settings**" serves to input settings valid for a given construction stage.

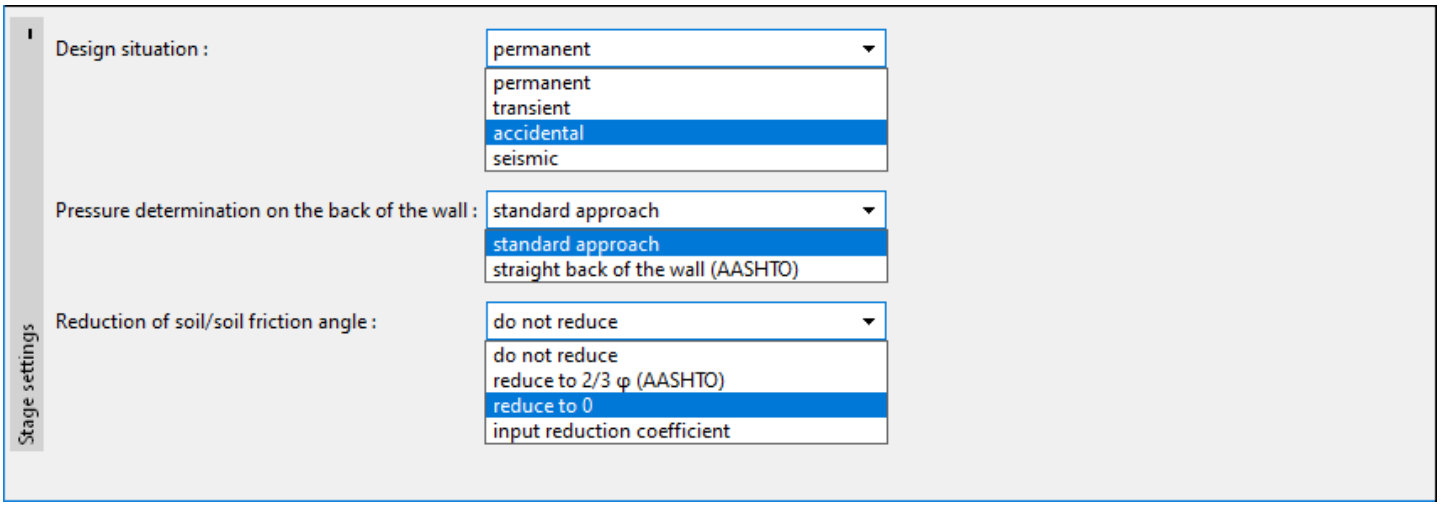
The selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology.

It is also possible to consider the simplified straight back of the wall for pressure calculation.

The reduction of soil/soil friction angle can be considered in one of following ways:

- **do not reduce**
- **reduce to 2/3φ (AASHTO)**
- **reduce to 0**
- **input reduction coefficient**



Frame "Stage settings"

Verification

The frame "**Verification**" shows the analysis results. Several computations can be carried out for a single task.

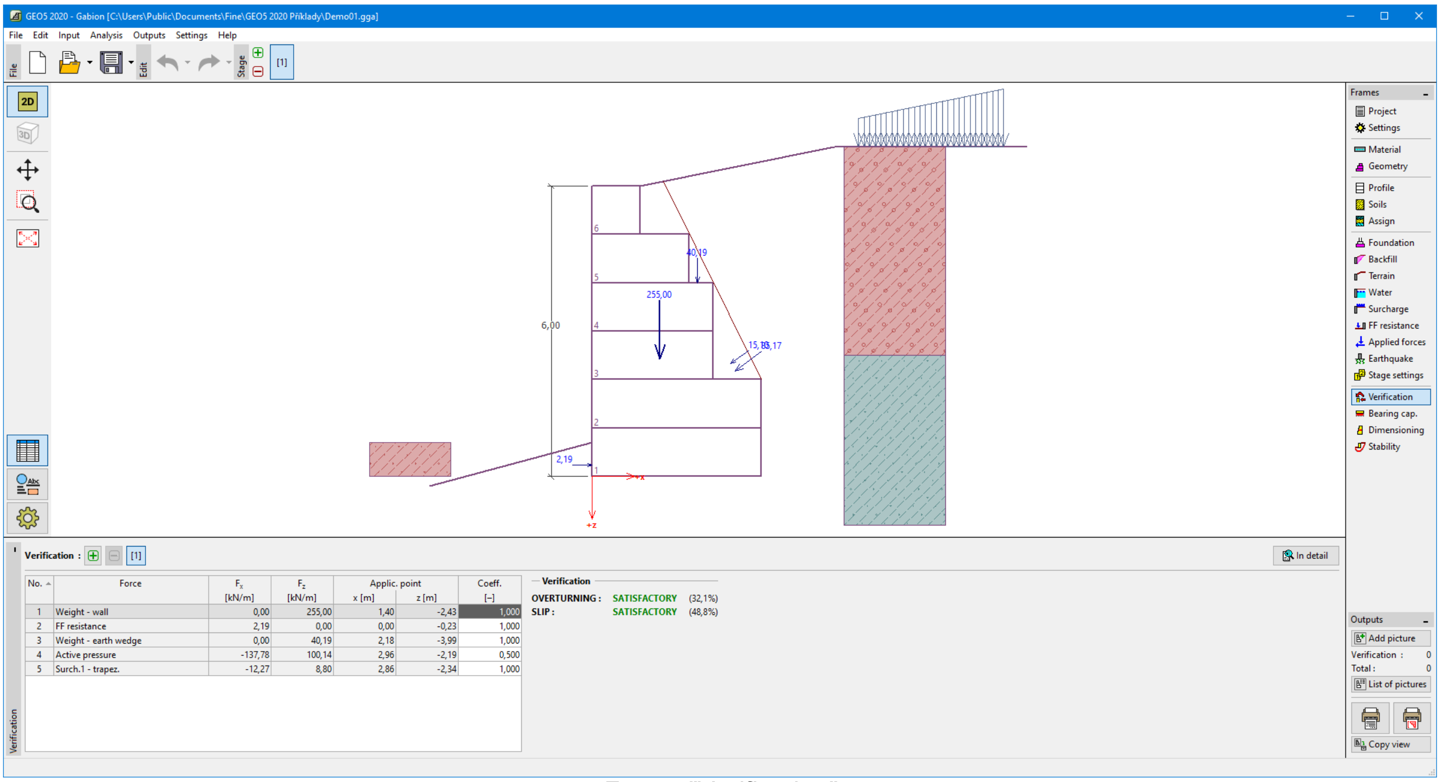
The frame appearance is adjusted based on the selected verification methodology.

- Verification according to the factor of safety or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows for specifying whether the load acting on a structure is considered as the secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in this case, the last column is not displayed.

The procedure for wall verification is described in the theoretical part of the help.

The computed forces are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against **overturning and translation**. The "**In detail**" button opens the dialog window, which contains a detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Verification"

Bearing Capacity

The "**Bearing capacity**" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the frame "Verification". The programs "**Spread footing**", "**Spread footing CPT**", "**Pile**" and "**Pile group**" then consider all verifications as load cases. In the program "**Pile CPT**", just normal load is used.

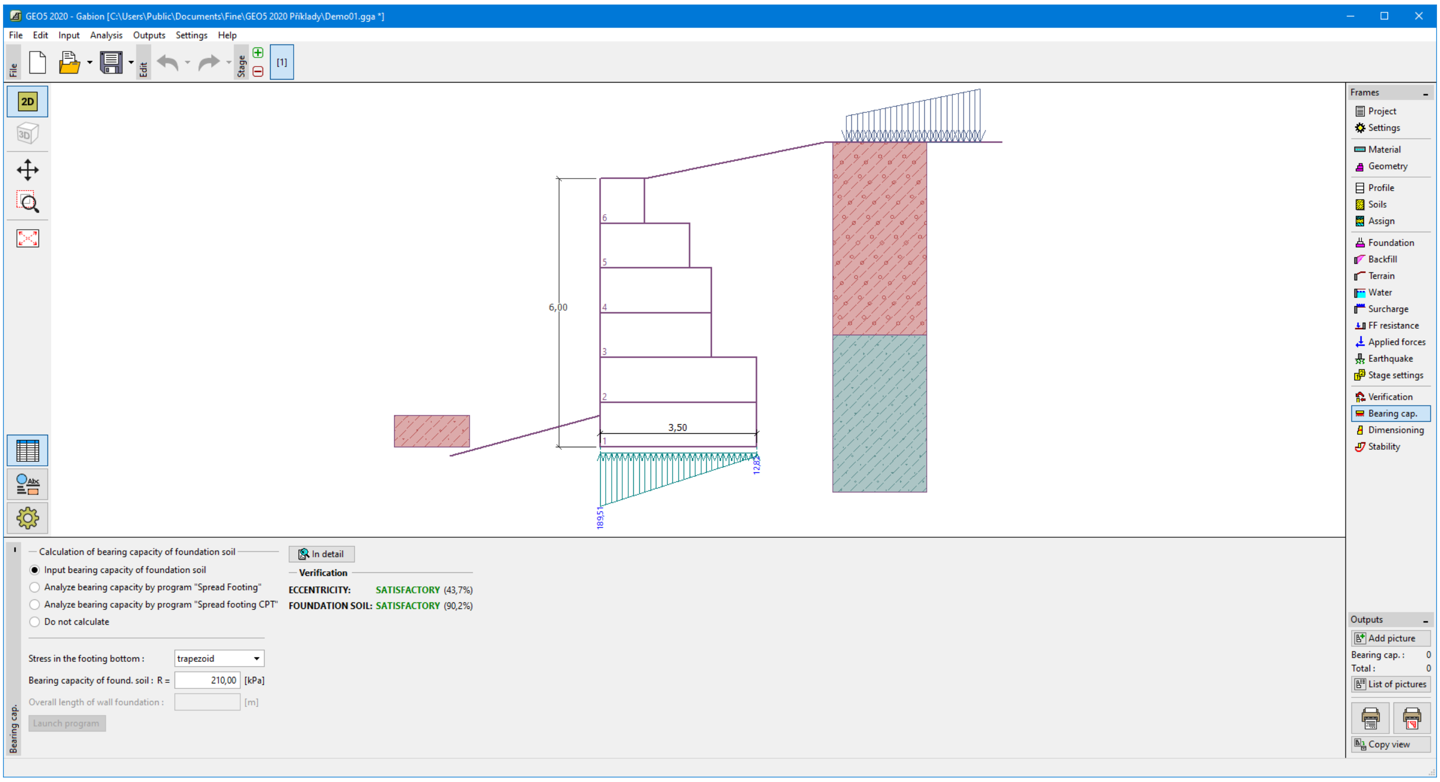
The frame contains the following analysis options:

- **Insert bearing capacity of foundation soil** – The foundation soil bearing capacity is input. The **eccentricity** and bearing capacity analysis results are displayed in the right part of the frame. The "**In detail**" button opens a dialog window that displays a detailed listing of the results.
- **Analyze bearing capacity by program "Spread footing"** – Pressing the "**Run program Spread footing**" button opens the program "**Spread footing**" which allows us to calculate the soil bearing capacity or settlement and rotation of the footing. Pressing the "**OK**" button leaves the analysis mode - the results and all plots are transferred into the program "**Gabion**". The "**Spread footing**" program must be installed for the button to be active. The overall length of the wall foundation is input.
- **Analyze bearing capacity by program "Spread footing CPT"** – The procedure is identical as if calculating soil bearing capacity by the **"Spread footing"** program.
- **Analyze bearing capacity by program "Pile"** – The procedure is identical as if calculating soil bearing capacity by the **"Spread footing"** program. The "**Run program Pile**" is available if the wall has a pile foundation (frame "Foundation"). Pile spacing s is input.

- **Analyze bearing capacity by program "Pile CPT"** – The procedure is identical as if calculating soil bearing capacity by the "**Pile**" program.
- **Analyze bearing capacity by program "Pile group"** – The procedure is identical as if calculating soil bearing capacity by the "**Spread footing**" program. The "**Run program Pile group**" is available if the wall has pile foundation with more then one pile (frame "Foundation"). Pile spacing s , the overall number of pile rows n and loading length l are input.
- **Do not calculate (pile footing)** – The foundation soil bearing capacity is not calculated.

The program allows us to specify a shape of stress in the footing bottom.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Bearing capacity"

Dimensioning

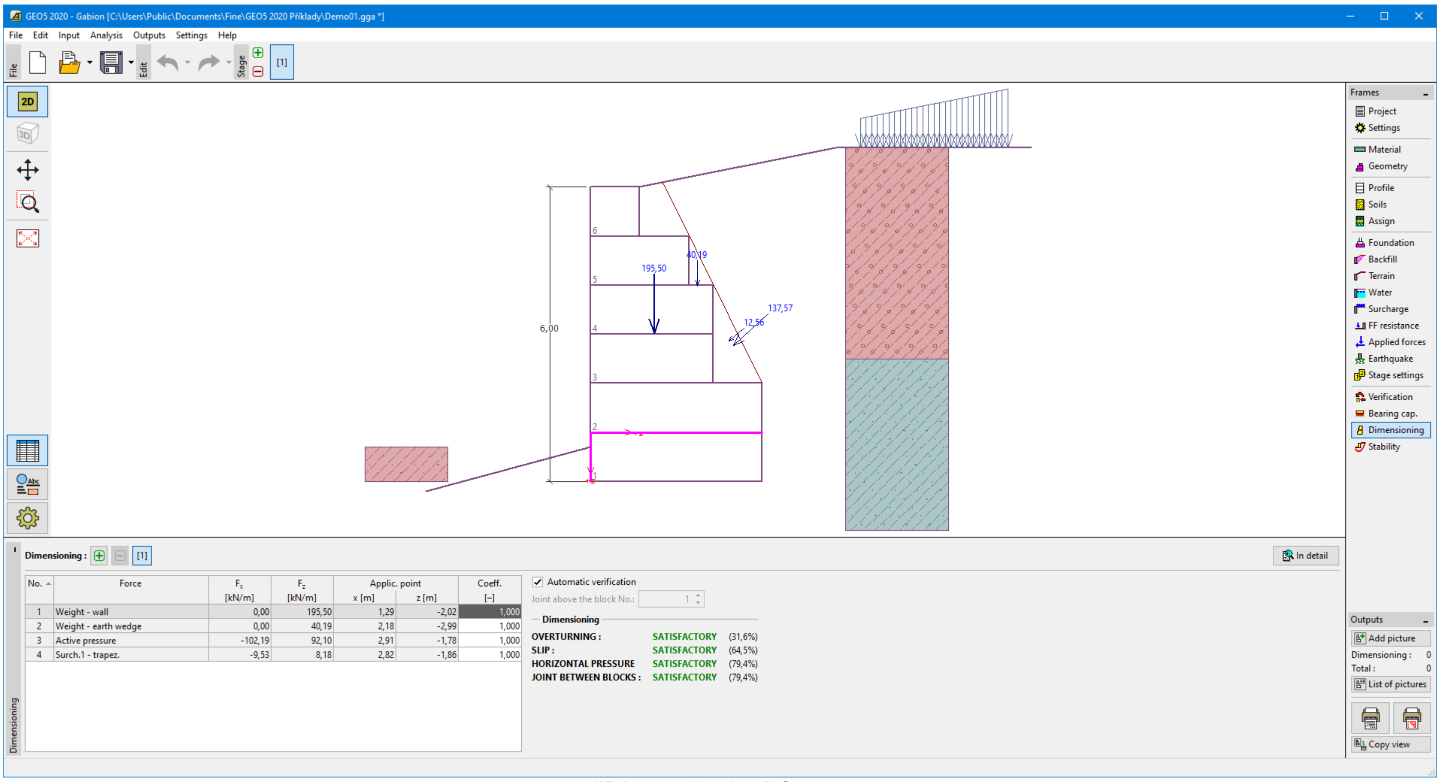
The "**Dimensioning**" frame allows us to verify individual joints of gabion blocks. The "**Joint above block No.**" field serves to select the desired joint subjected to verification analysis. The verification against overturning, translation, for side pressure and joint between blocks is performed.

The frame appearance is adjusted based on the selected verification methodology.

- Verification according to the factor of safety or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows for specifying whether the load acting on a structure is considered as the secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in this case, the last column is not displayed.

Several computations for various cross-sections can be carried out. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "**In detail**" button opens the dialog window that contains a detailed listing of the dimensioning results.

Visualization of results can be adjusted in the frame "Drawing Settings".

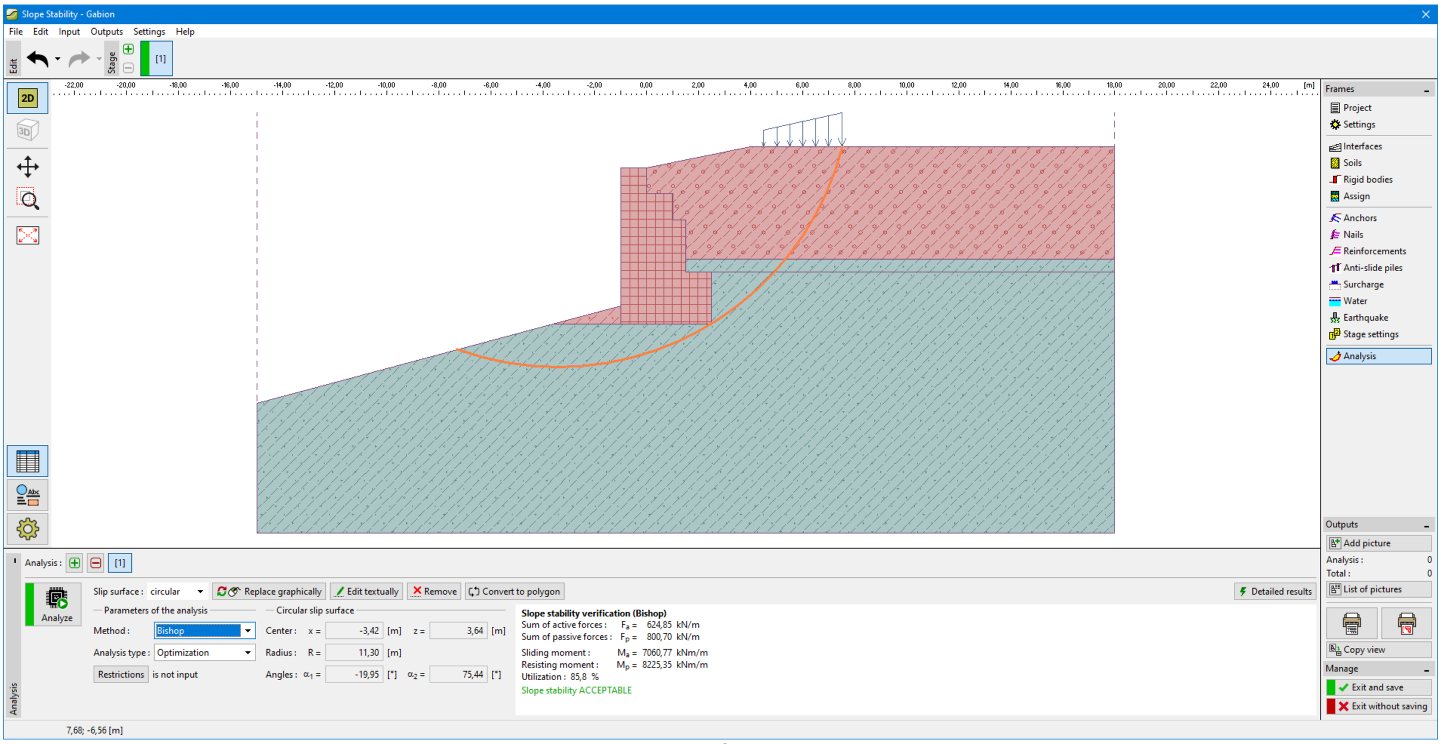


"Dimensioning" frame

Stability

Pressing the "**Stability**" button launches the "**Slope stability**" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program "**Slope stability**" is installed.

After completing all analyses press the "**OK**" button to leave the program - all data are then carried over to the analysis protocol of the "**Gabion**" program.



Frame "Stability"

Program Abutment

This program is used to design abutments including wings. It allows us to check the abutment for overturning, translation, bearing capacity of foundation soil, and dimensioning of decisive or reinforced concrete sections (including wings).

The help in the "Abutment" program includes the following topics:

- The input of data into individual frames:

Project	Settings	Geometric Section	Wings	Geometry Plane View	Footing Steps	Material
Profile	Soils	Load - LC	Assign	Foundation	Backfill	Terrain
Water	Surcharge	Front Face	Applied Forces	Earthquake	Stage Settings	Verification
		Resistance				
Bearing Capacity	Dimensioning	Stability				

- Standards and analysis methods

- Theory for analysis in the "**Abutment**" program:

Stress in Soil Body	Earth Pressures	Analysis of Walls	Analysis of Foundation Bearing Capacity	Dimensioning of Concrete Structures

- Outputs

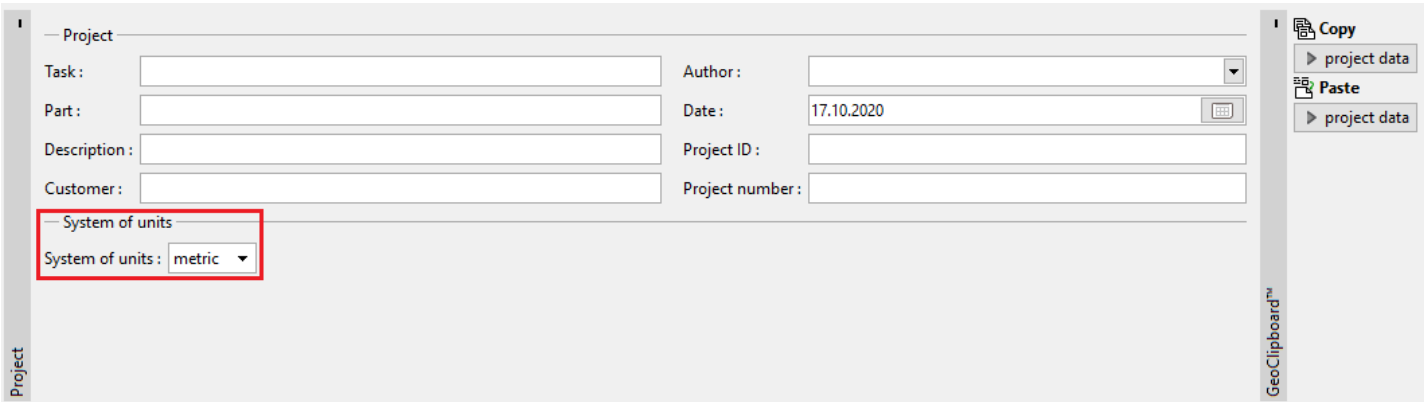
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The frame **"Project"** is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The frame "**Settings**" allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

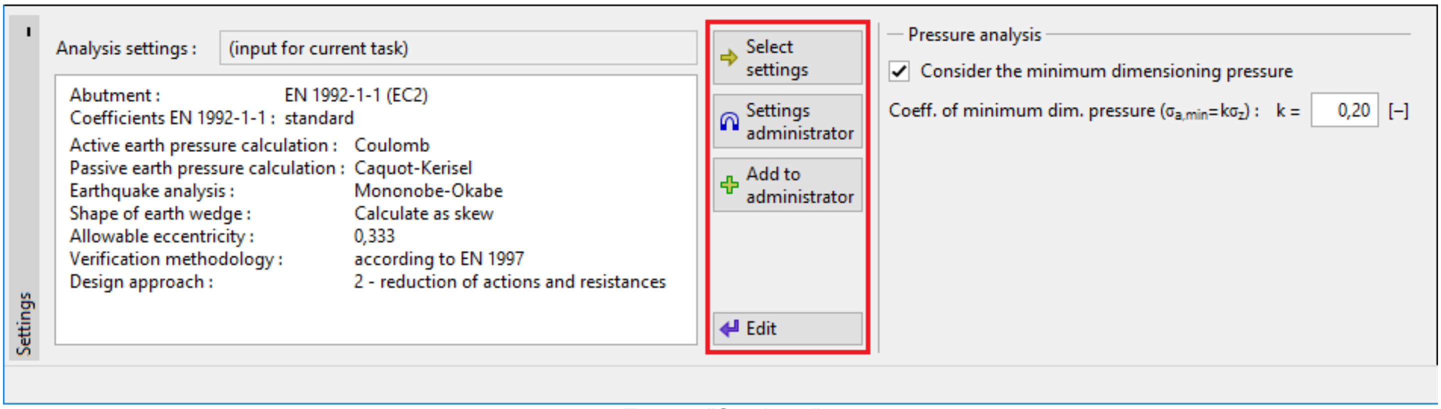
The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

The program allows us to specify a value of the minimum dimensioning pressure (by checking the option "**Consider the minimum dimensioning pressure**").

Settings of analysis parameters are performed in the "Materials and standards" and "Wall analysis" tabs.



Frame "Settings"

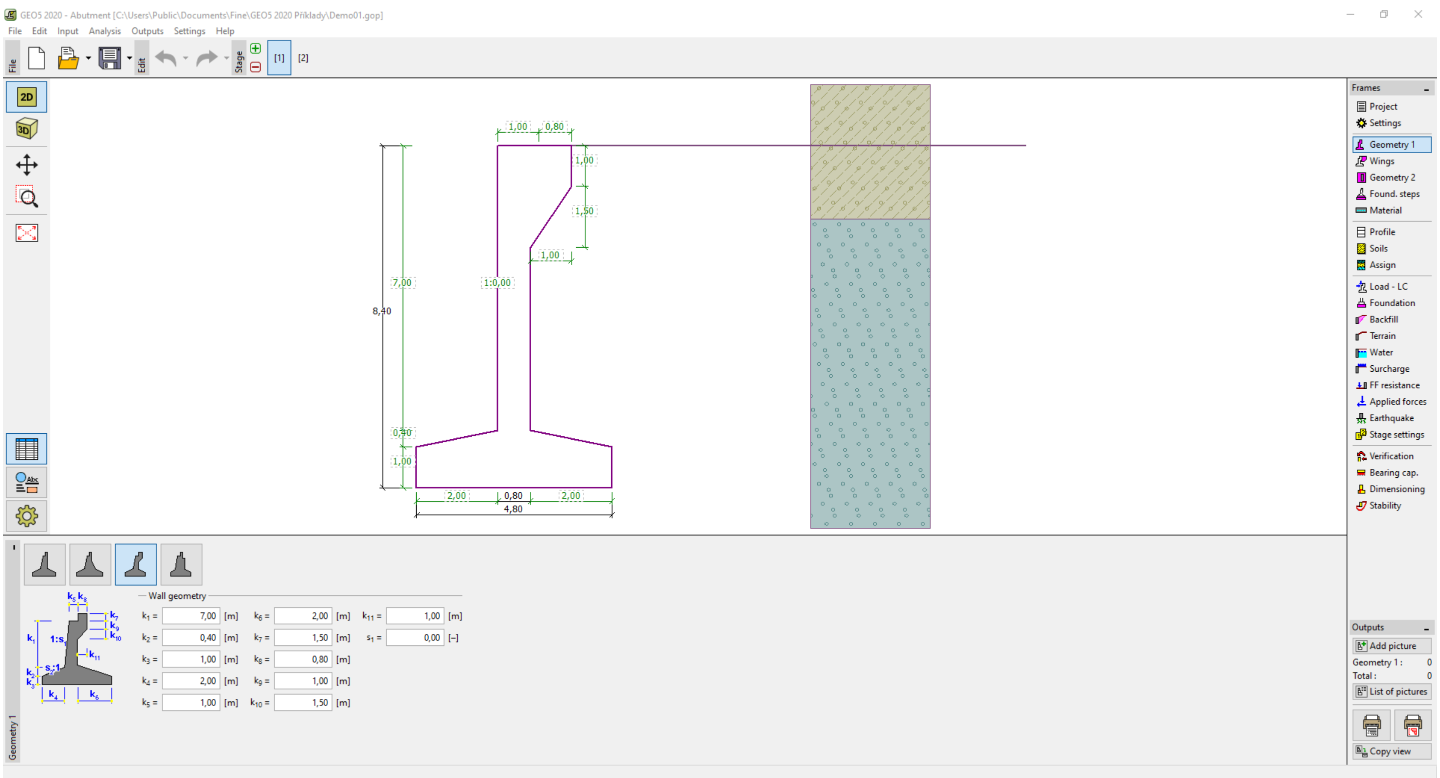
Geometric Section

The "**Geometric section**" frame allows us to select the shape of bridge abutment. The selected shape with a graphic hint appears in the left part of the frame. The shape of a wall can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

In case the structure is composed of inclined segments it is required to enter the ratio of sides of an inclined segment $1{:}x$. **The straight structure** is specified by entering the value zero.

The frame serves to specify the final shape of abutment including the closure wall. The abutment can be verified also for the construction state (without the closure wall) based on the choice in the "Load - LC" frame. The abutment length and the length of abutment foundation is specified in the "Geometry plane view" frame.

The program makes it possible to export the geometry of a structure in the *.DXF format.



"Geometric section" frame

Wings

The "**Wings**" frame allows us to input the bridge wings dimensions. The wings can be either symmetrical or unsymmetrical. Assuming unsymmetrical wings requires inputting the right and left wing dimensions separately. The screen always displays the currently input wing - only the left wing is then visualized in the remaining frames.

The "Geometry plane view" frame can also be used to input or edit the wing thicknesses and lengths.

The Wing-abutment joint cross-section can also be verified in the "Dimensioning" frame. The load due to the moment is considered. The entire wing is loaded by **active pressure** developed behind the wall. The "**Dimensioning**" dialog window serves to input the magnitude of **surface surcharge** to determine the wing pressure. The surcharge specified in the "Surcharge" frame is then not taken into account and the terrain behind the wing is considered as flat. The resulting moment applied to the joint is obtained by multiplying the overall magnitude of soil pressure acting on the wall surface and by the difference of centroids of the pressure resultant and the joint.

The length of cross-section used for dimensioning is considered by default as the wing height - a different length of wing-abutment joint can also be specified after selecting the option "**Reduce for dimensioning**".

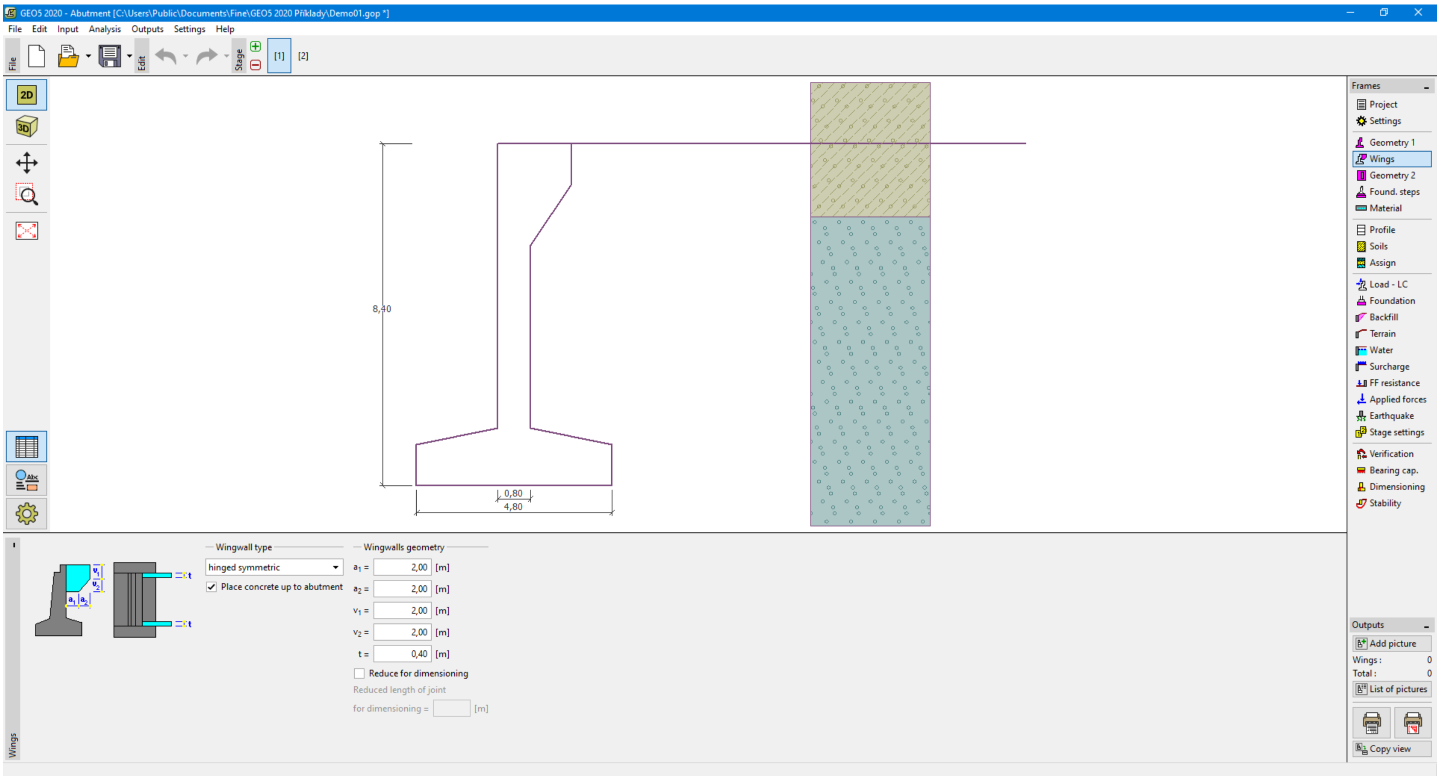
When using prolonged wing walls, it is possible to input dimensions of the foundation below the wall. Such foundation jumps are reflected in the analysis by computing a fictitious width of the foundation as:

$$d_{fict} = \frac{A_{tot}}{S}$$

where:

A_{tot}	-	the overall area of foundation including all jumps
S	-	length of abutment foundation
d_{fict}	-	fictitious width of the foundation for verification analysis

The foundation is then, in a simplified way, considered as being rectangular and yet on the safe side.



"Wings" frame

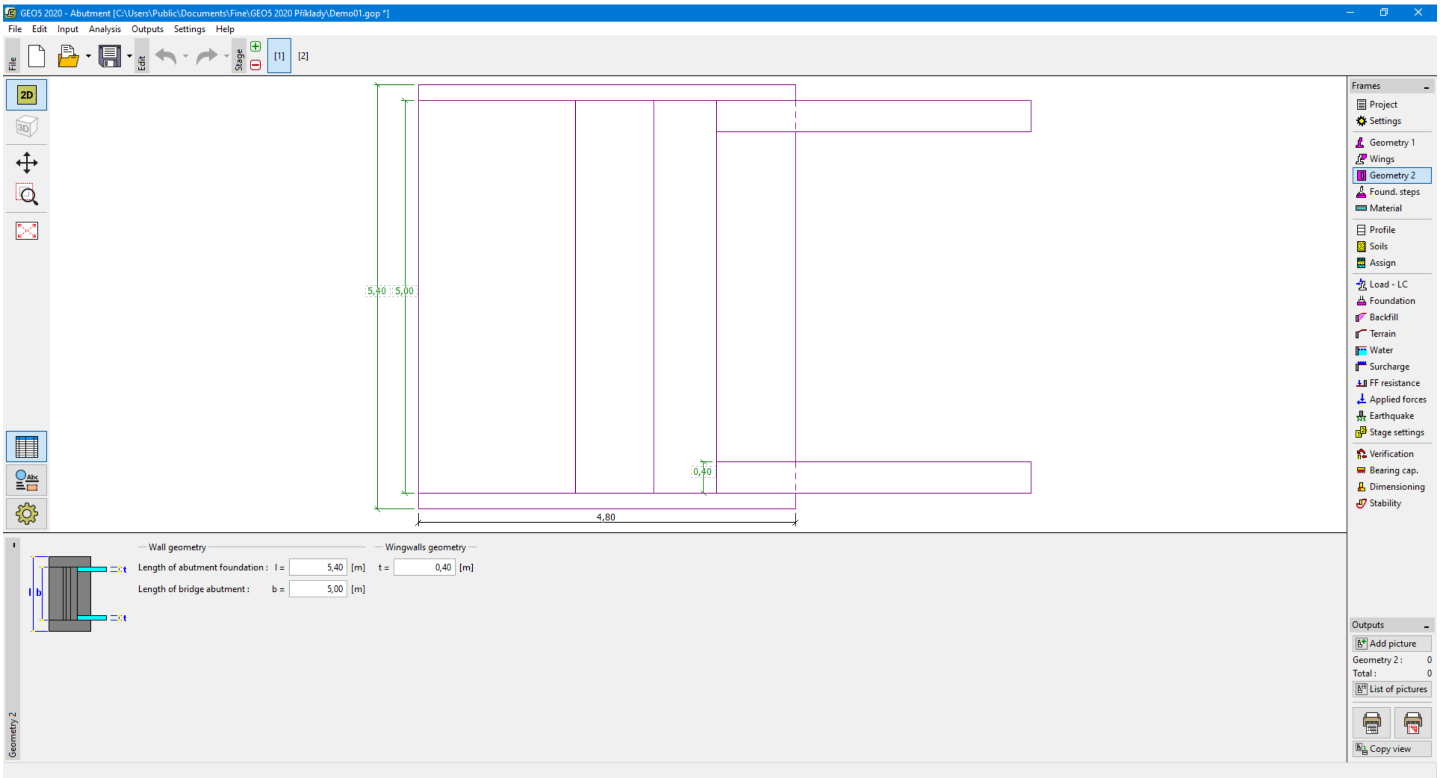
Geometry Plane View

The "**Geometry plane view**" frame allows us to input the abutment length, length of abutment foundation and also dimensions of abutment wings.

Dimensions can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

For details on **the effect of abutment dimensions** on verification analysis, we refer the reader to the "Calculating of abutment forces" section.

The program makes it possible to export the geometry of a structure in the *.DXF format.

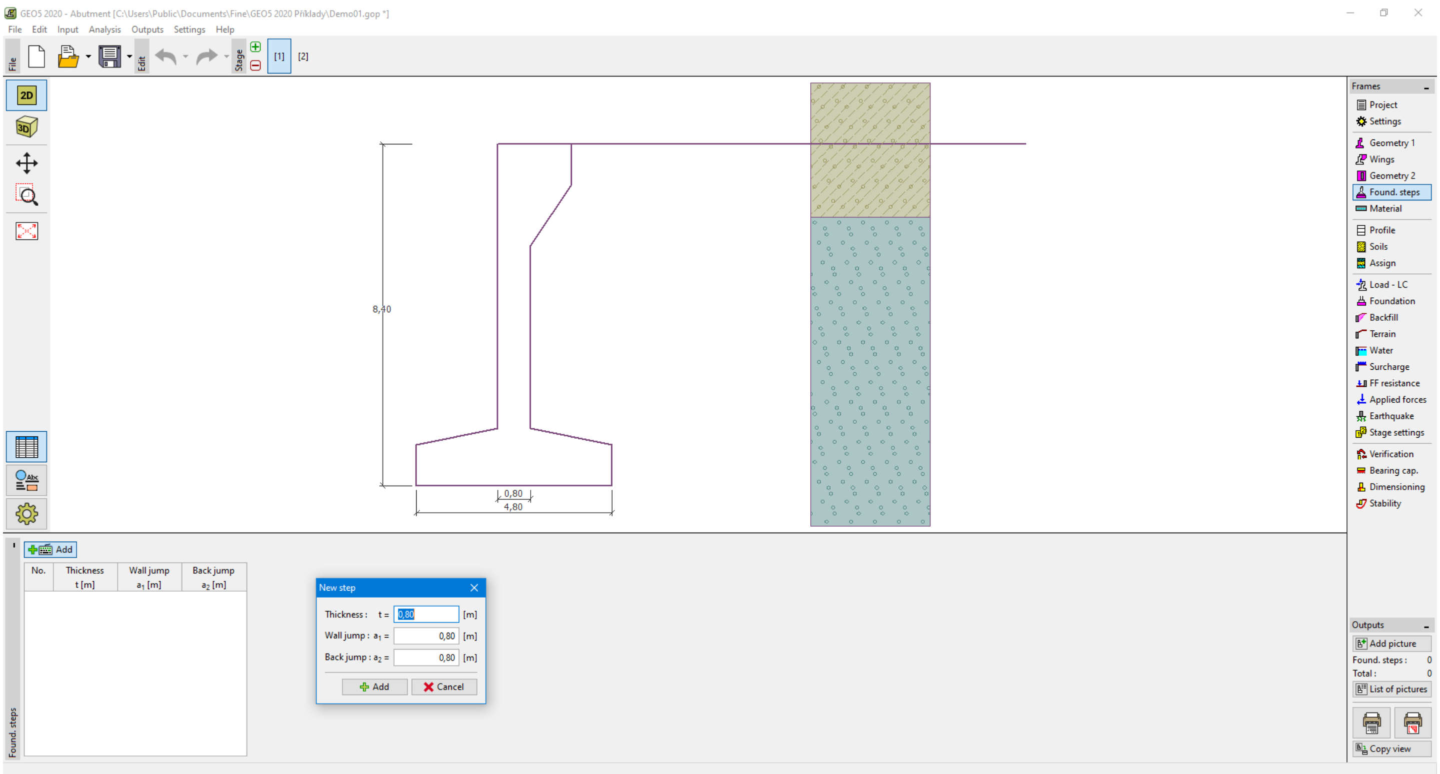


"Geometry plane view" frame

Footing Steps

The "**Footing steps**" frame serves to input the steps of the foundation below abutment. This option allows us to specify additional shapes of the bridge abutment.

Adding the foundation step is performed in the "**New step**" dialog window. Input footing steps can be edited on the desktop with the help of active dimensions or active objects, respectively.



"Footing steps" frame

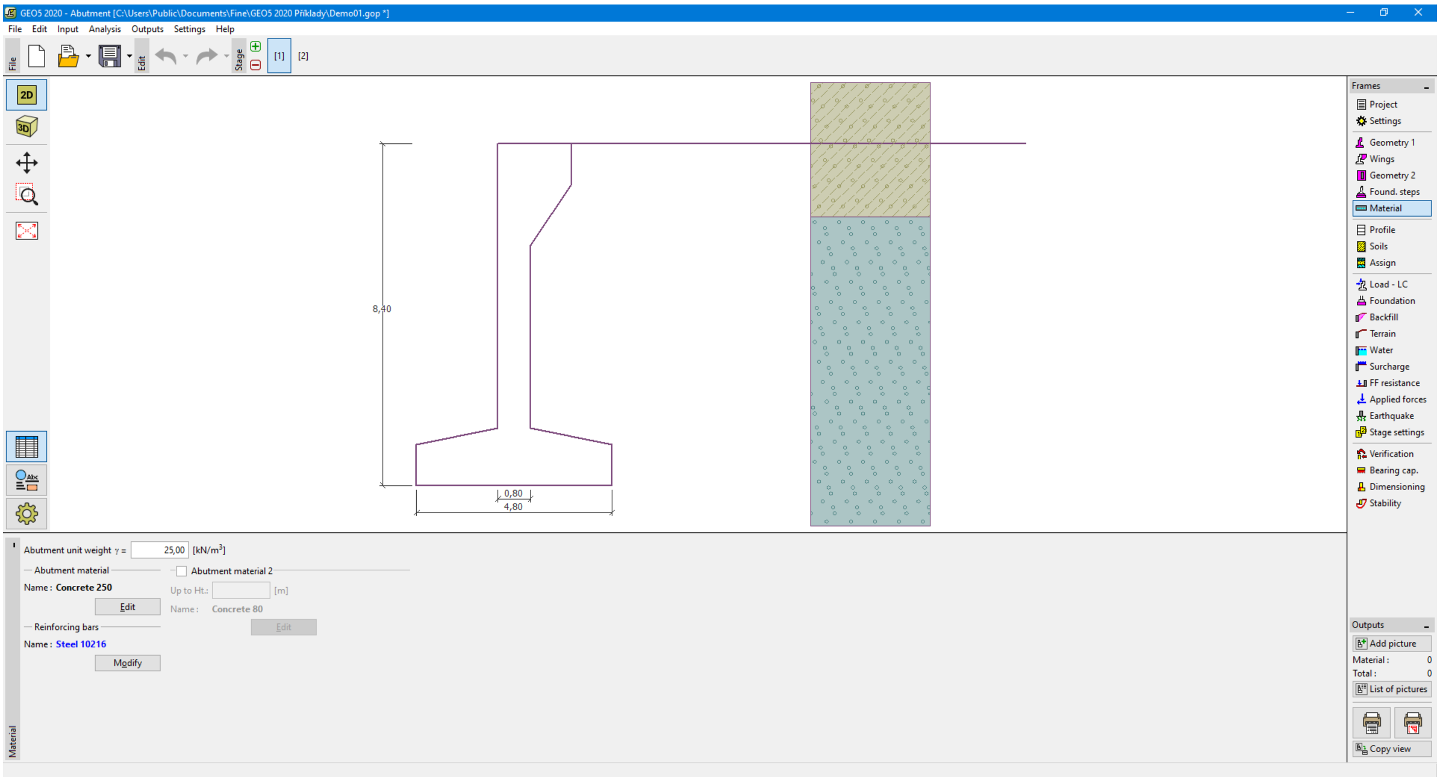
Material

The frame **"Material"** allows us to enter material parameters. The input field in the upper part of the frame serves to specify the **structure unit weight**.

Two options can be used for setting the parameters of the material:

- The "**Catalog**" button opens the "**Catalog of materials**" dialog window, where the required material can be selected.
- The "**User-defined**" button opens the "**Editor of material**", which allows us to input the specification of material parameters manually by the user.

The catalogs content depends on the selection of standard for the design of concrete structures set in the "Materials and standards" tab. The input field in the upper part of the frame serves to specify the **abutment unit weight**.



Frame "Material"

Profile

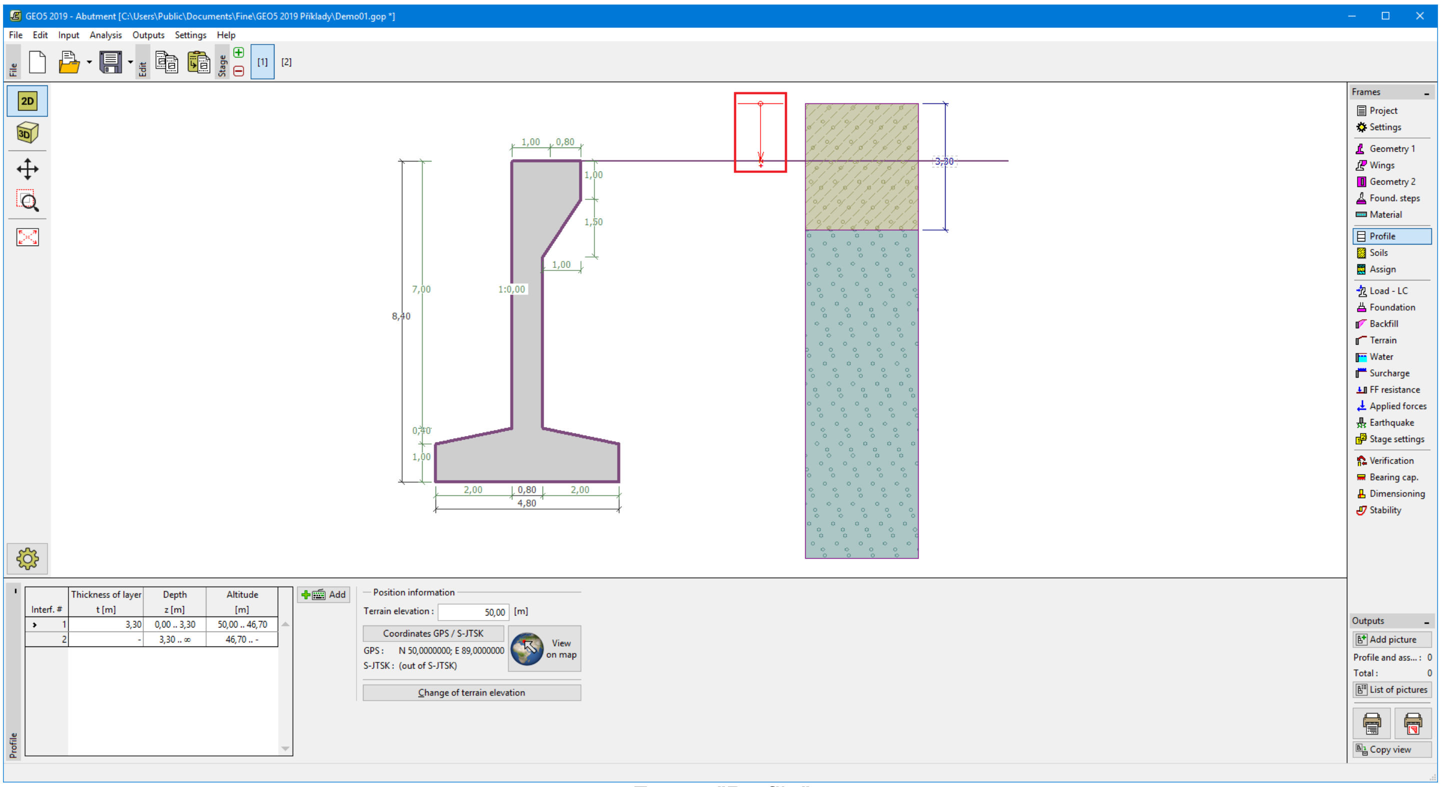
The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.



Frame "Profile"

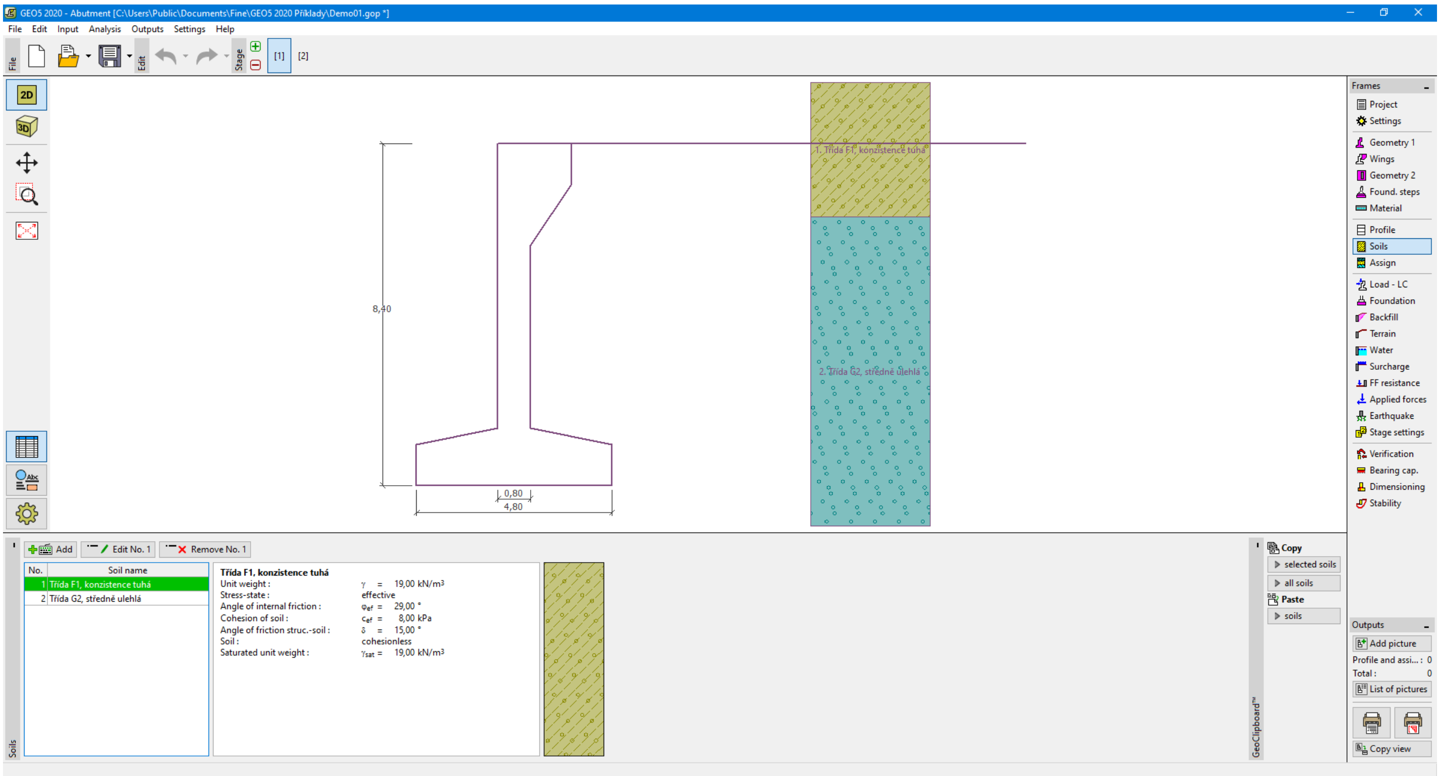
Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

Basic Data

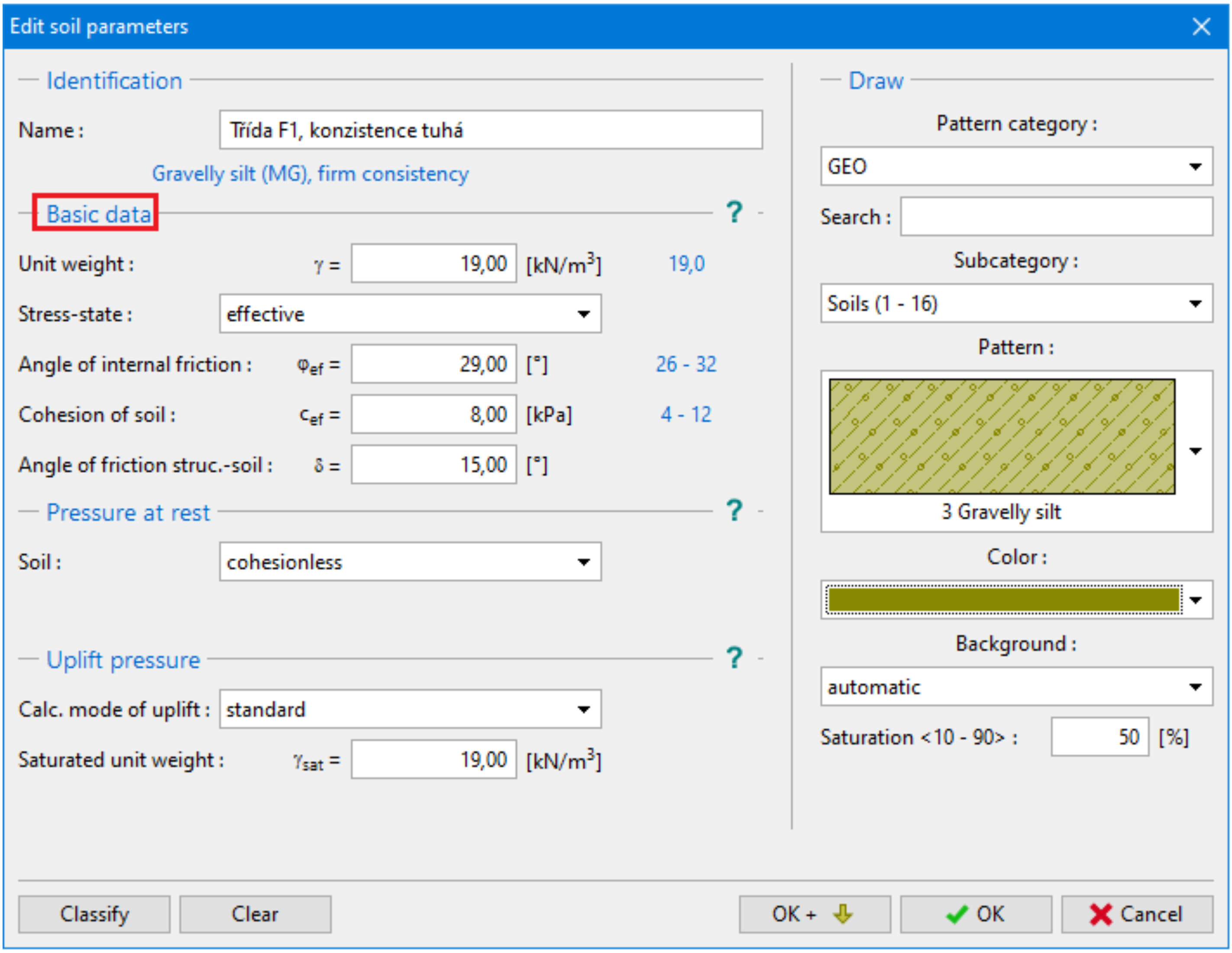
This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction, and cohesion**. The particular values are obtained from a geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit the built-in database of soils, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually.

Either **effective or total** parameters of the angle of internal friction and cohesion are specified depending on the setting in the "**Stress analysis**" combo list. Whether to use effective or total parameters depends primarily on the type of soil, type of load, structure duration, and water conditions.

For effective stress further needs to specify the angle of internal friction between the soil and structure, which depends on the structure material and the type of soil. Possible values of this parameter are listed in the table of recommended values.

For total stress further needs to specify the adhesion of soil to the structure face a .

The associated theory is described in detail in chapter "Earth pressures".

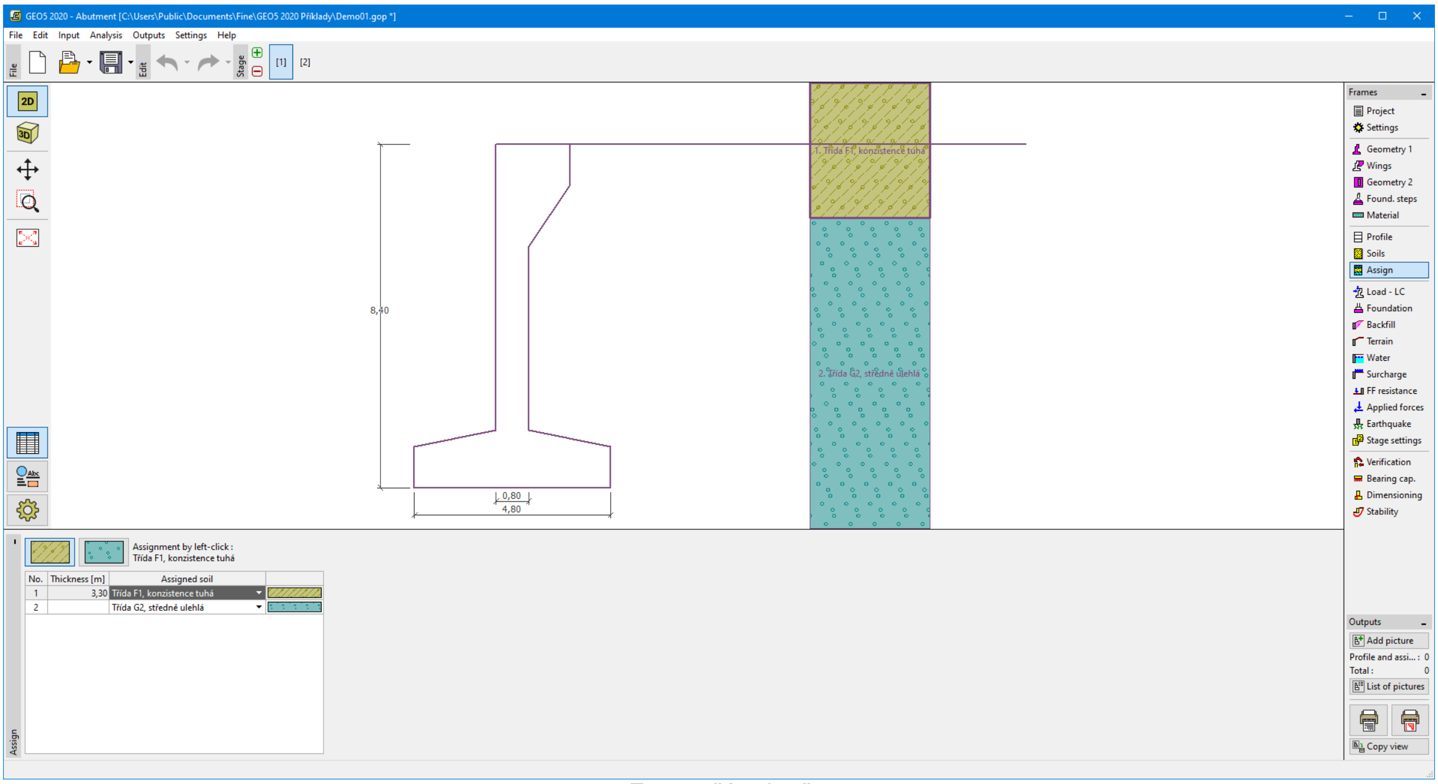


Dialog window "Add new soils" - "Basic data"

Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail herein.



Frame "Assign"

Load - LC

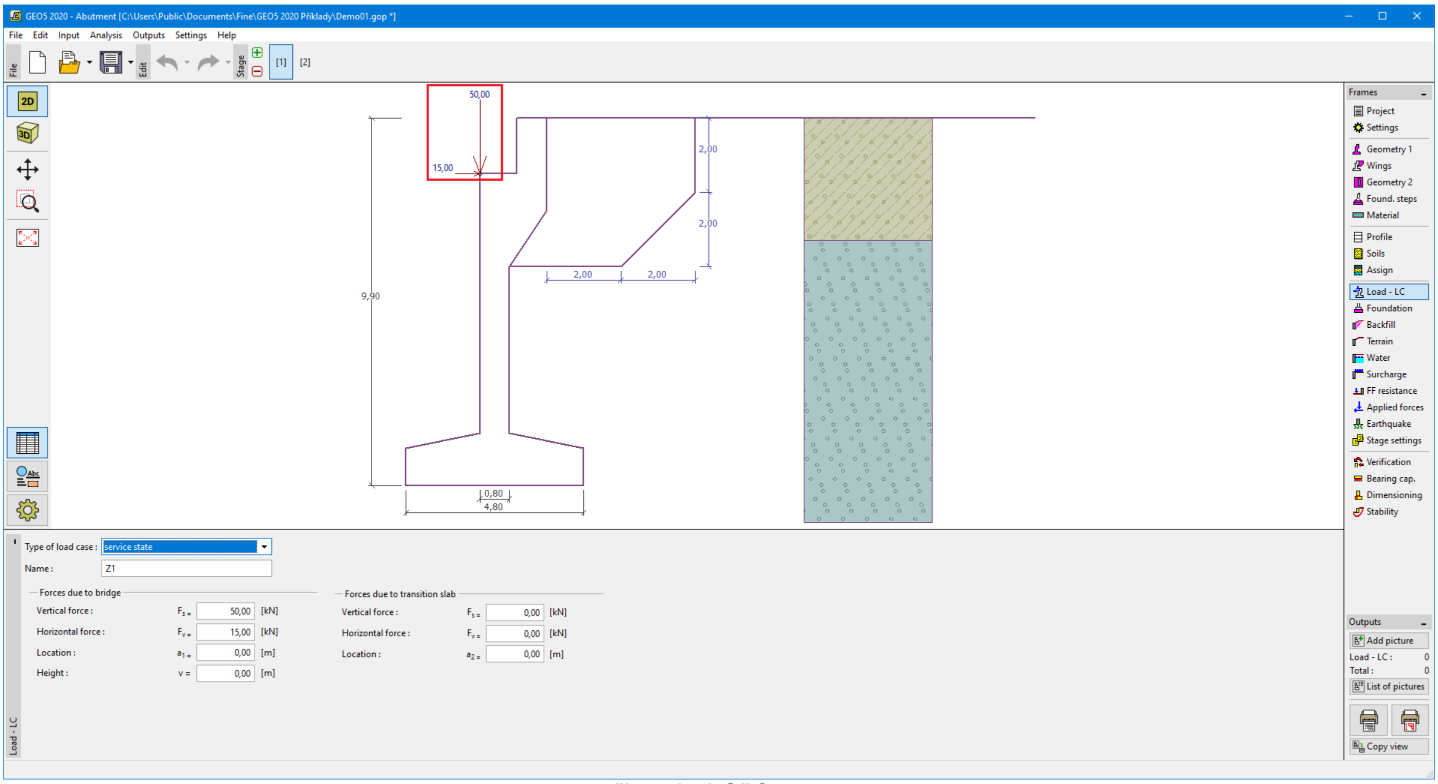
The "**Load - LC**" frame serves to specify individual **load cases** (construction, service) and the load caused by the bridge and transition slab. Verification and dimensioning analyses of the entire bridge abutment or only its part are carried out according to the specified type of LC.

When performing the analysis according to EN 1997 or LRFD, the input load due to **bridge and transition slab is not INCREASED** by any partial factors. The input forces must be determined **in accordance** with the corresponding standards (EN 1990, EN 1991).

No load is specified in the case of **construction state** and the abutment is verified in a given stage of construction without a closure wall and bridge wings.

In the case of **service state,** the abutment is loaded by the **bridge** and **transition slab**, the entire abutment is verified.

For abutment verification, it appears advantageous to exploit the stage of construction and specify in individual stages different load cases (e.g. construction state, service state without live load, service state with all loads). Individual stages then allow inputing different loads, surcharges, terrain shapes, type of pressure analysis (active, at rest), design coefficients, etc.



"Load - LC" frame

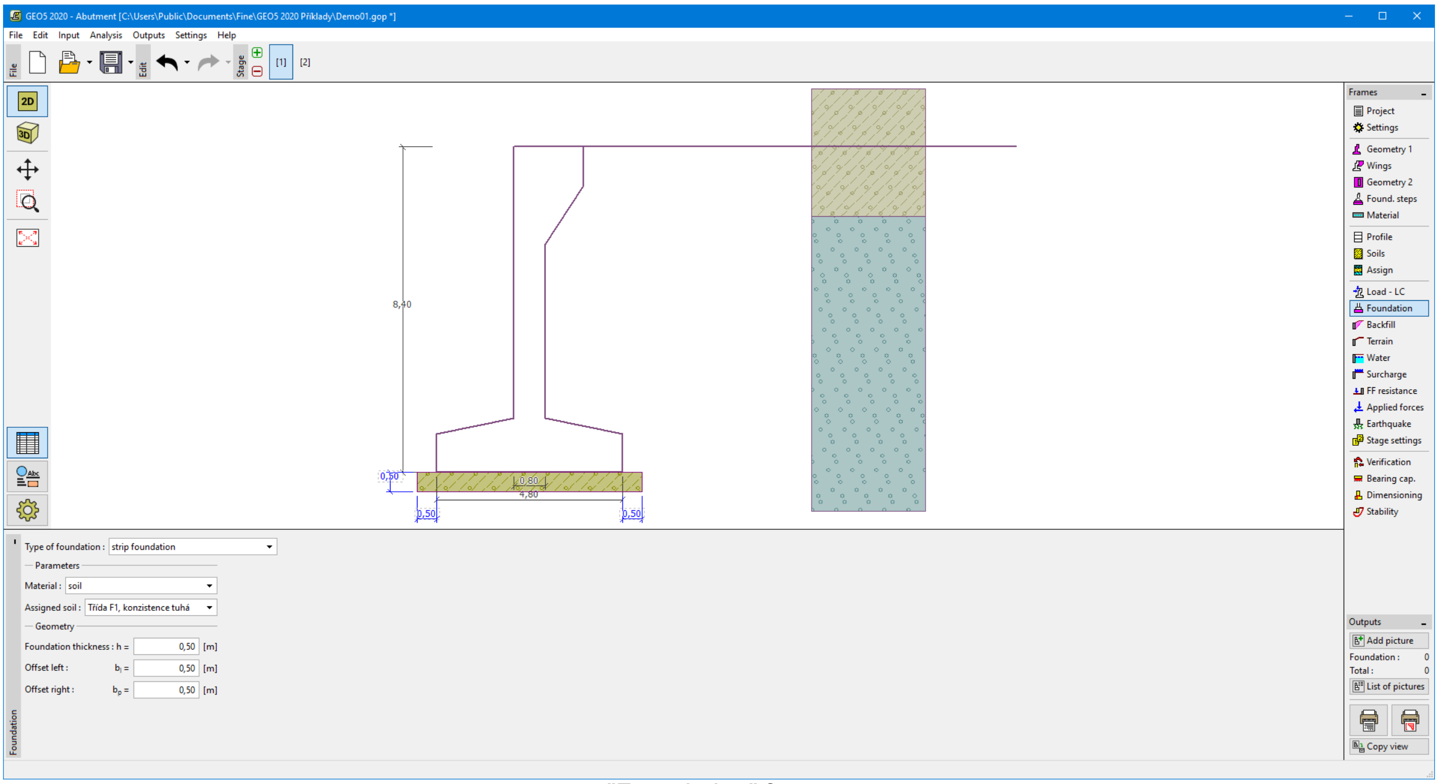
Foundation

The "**Foundation**" frame serves to specify the type of wall foundation. The following types of wall foundation are available:

- **soil from geological profile** - the wall is founded on the soil assigned from the geological profile specified in the "Profile" frame.
- **input parameters of contact base-soil** - parameters of the contact between footing bottom and structure are specified. Option "**input angle of friction base-soil**" requires inputting the friction angle ψ [°] between foundation and soil. Option "**input friction coefficient**" requires specifying the friction coefficient μ [-]. Both options require inputting the cohesion a [kPa] between foundation (base) and soil.
- **strip foundation** - strip foundation material is represented either by **soil** (input in "Soils" frame), or concrete - requires inputting the **unit weight of foundation material** γ and **parameters of contact base-soil** (friction coefficient f, cohesion c, additional resistance F).
- **pile foundation** - the wall can be founded on one row of piles or two rows of piles, respectively.

The geometry of the wall foundation (**strip foundation**, **pile foundation**) can be modified either in the frame by entering specific values into the inputting fields or on the desktop with the help of active dimensions.

The input data introduced in this frame influence the actual wall analysis (check for slip) and further the bearing capacity of foundation soil.

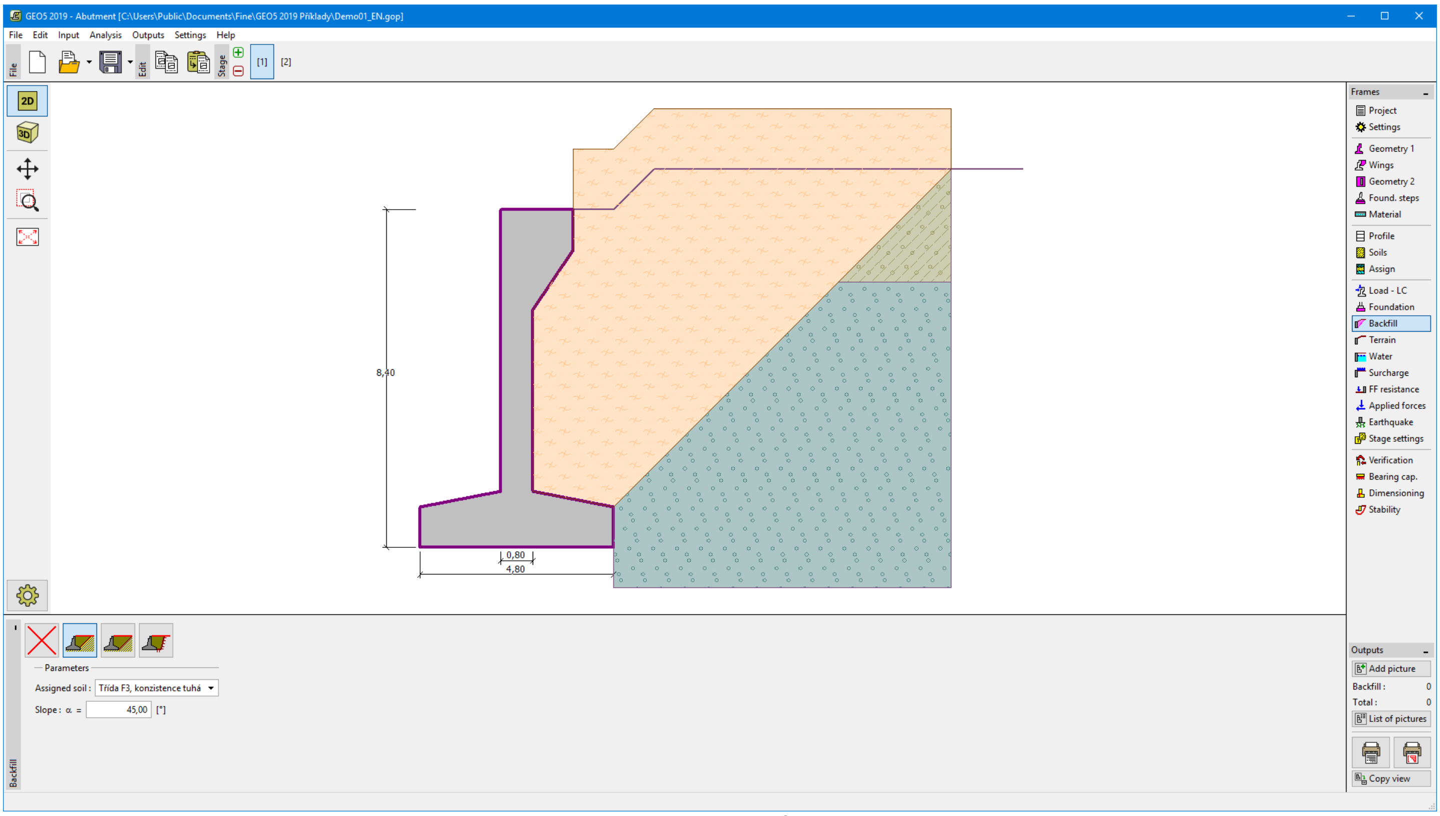


"Foundation" frame

Backfill

The frame "**Backfill**" allows the selection of backfills behind the structure.

Analysis of earth pressures with the influence of backfill is described in the theoretical part of the help: "Influence of Backfill".



Frame "Backfill"

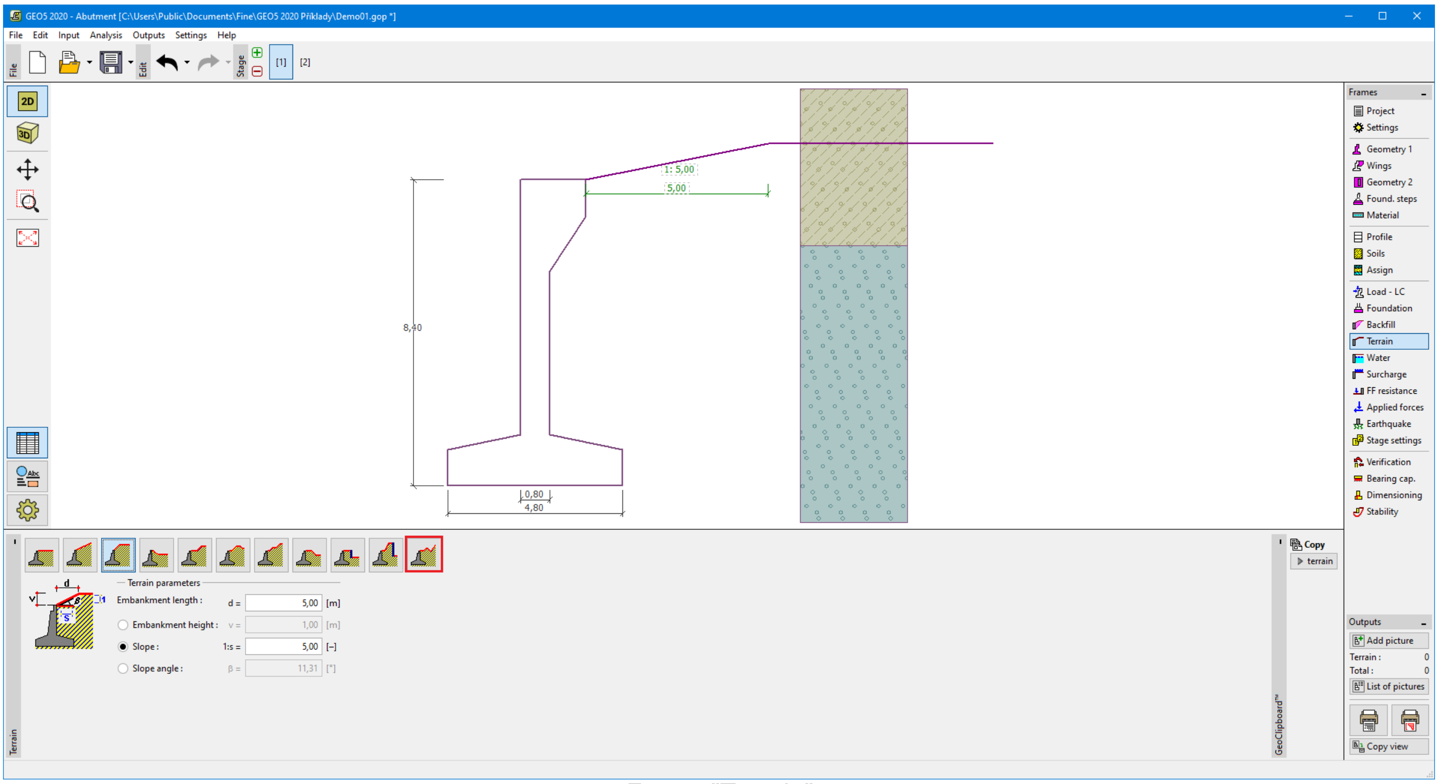
Terrain

The "**Terrain**" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by

inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case, the frame contains a table with a list of terrain points. The first point with coordinates [0; 0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "Distribution of earth pressures for broken terrain".



Frame "Terrain"

Water

The "**Water**" frame allows, by pressing the button, for selecting the type of water. The selected type with a graphic hint of input values is displayed in the left part of the frame. Water parameters (h_1, h_2 ...) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

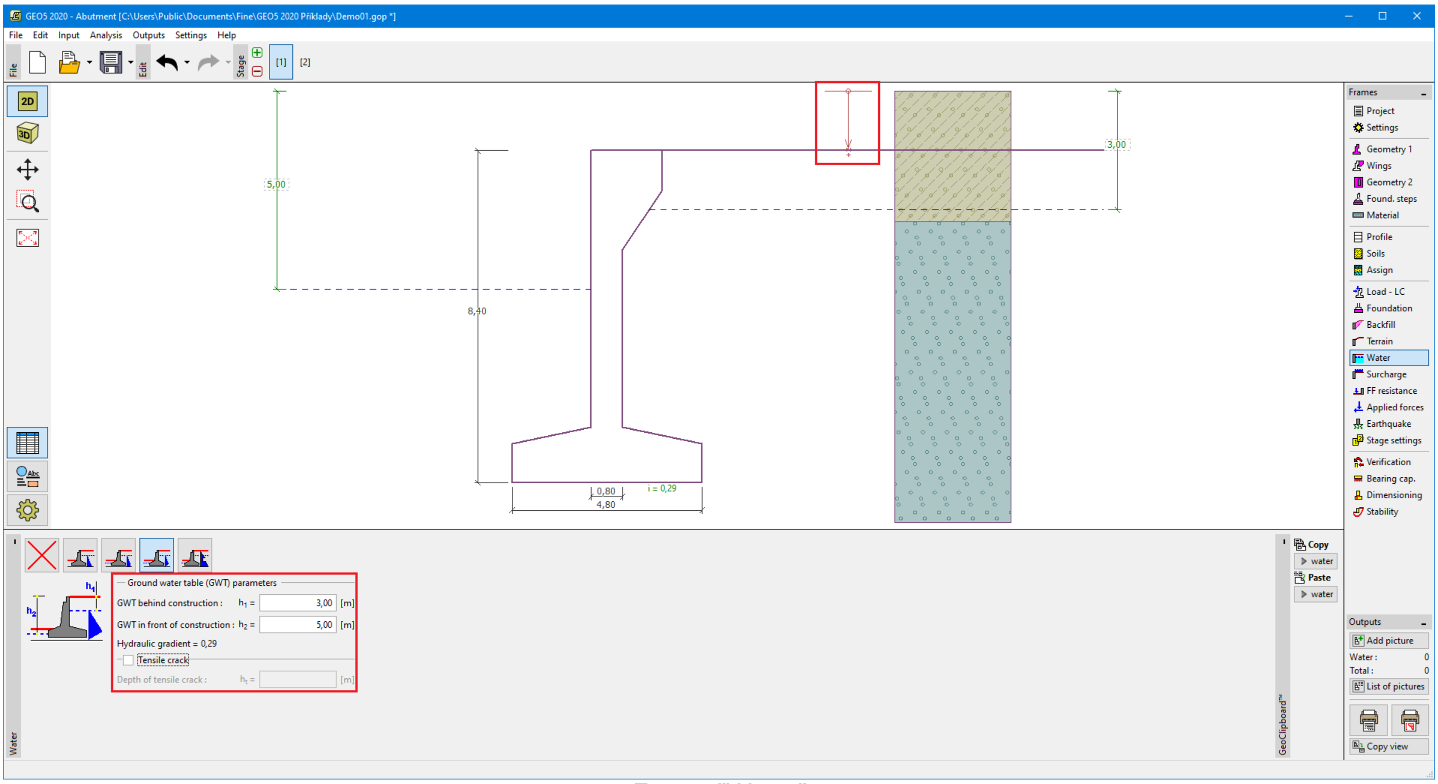
The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be linear, parabolic, or it may not be considered at all. When verifying the wall, the uplift pressure in the base of footing due to the different water tables is introduced as a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tabs "**In front of structure**" and "**Behind structure**" appear with tables. The table contains the values of pore pressure in front of the structure or behind it.

The groundwater table can also be specified **above the structure** or earth profile, respectively - in such a case, the depth of water is input with a negative value.

Analysis of earth pressures with the influence of water is described in the theoretical part of the help, chapter "Influence of water".

The program further allows for specifying a depth of tensile cracks filled with water.



Frame "Water"

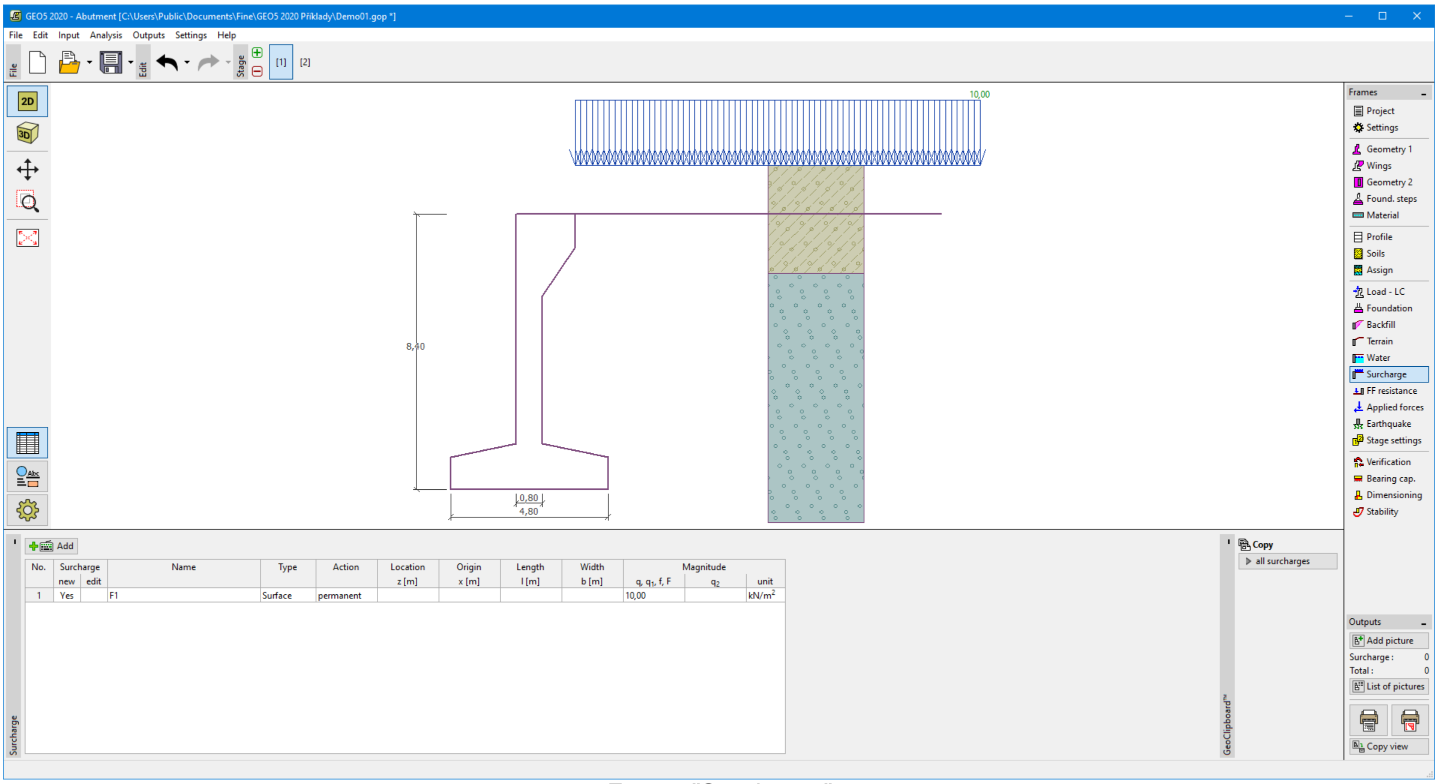
Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The z -coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case when the surcharge is found out of the terrain the program prompts an error message before calculation.

Either **permanent**, **variable,** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with a favorable effect is not considered in the analysis.

Analysis of earth pressures due to surcharges is described in the theoretical part of the help, chapter "Influence of surcharge".



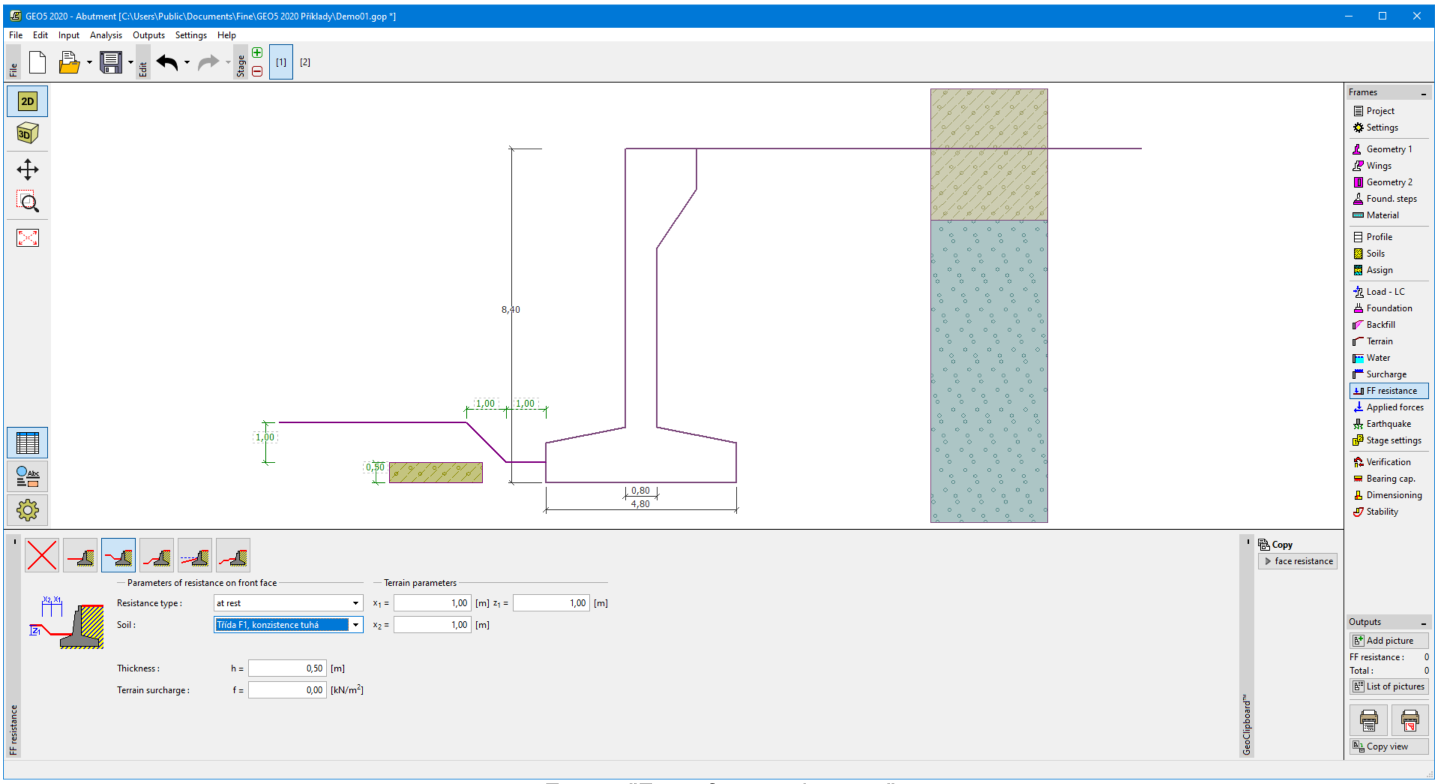
Frame "Surcharge"

Front Face Resistance

The "**Front face resistance**" frame allows us to specify the terrain shape in front of the structure and parameters of front face resistance. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop using active dimensions.

Combo lists in the frame allow selecting the type of resistance and a soil (from the soils introduced in the "Soils" frame). The terrain surcharge magnitude and soil thickness (above the wall lowest points) can also be specified.

The resistance on a structure front face can be specified as the pressure at rest, passive pressure, or reduced passive pressure. The resulting force due to the reduced passive pressure is calculated as a resultant force caused by pressure at rest and passive pressure multiplied by corresponding coefficients.



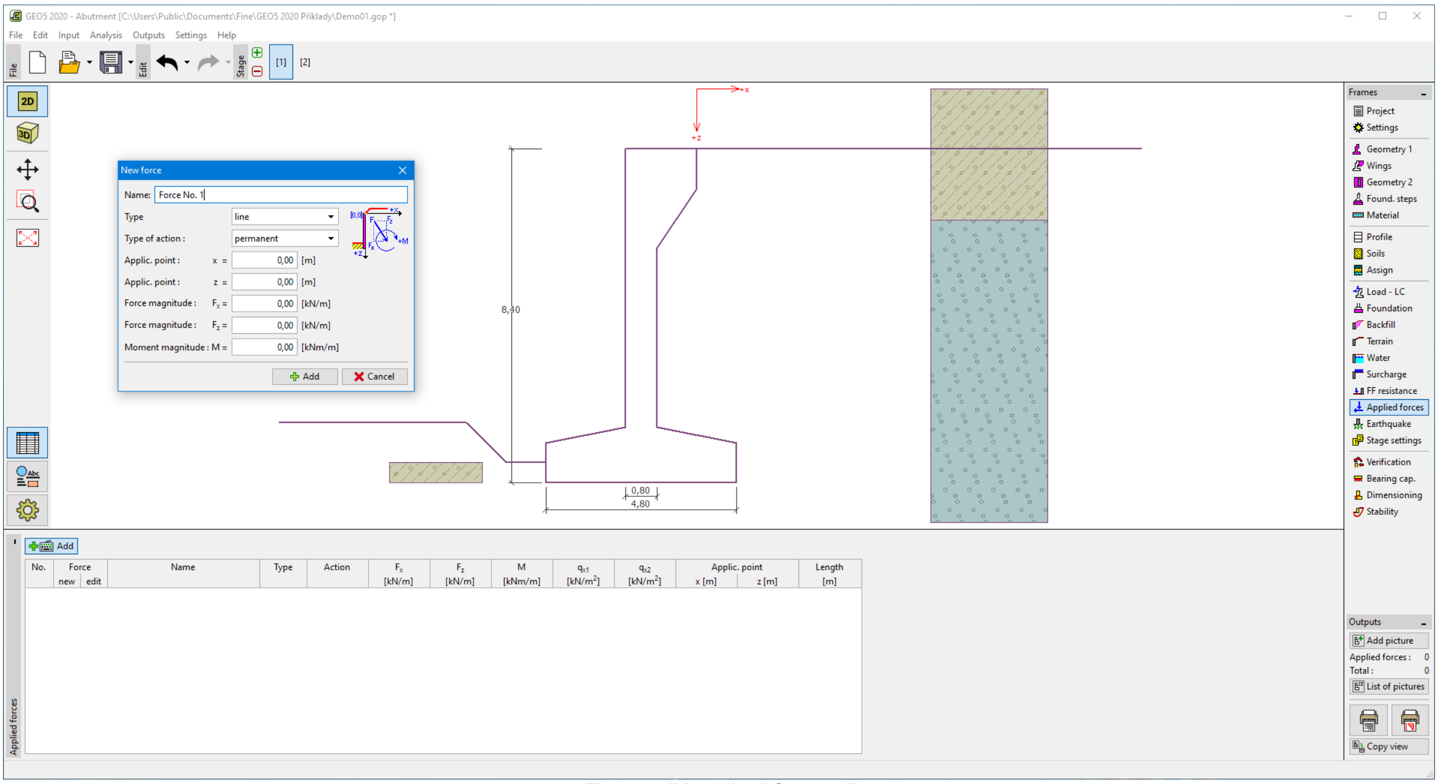
Frame "Front face resistance"

Applied Forces

The "**Applied forces**" frame contains a table with a list of forces acting on a structure. Adding forces is performed in the "**New force**" dialog window. The input forces can also be edited on the desktop with the help of active objects.

Applied forces represent an additional load on the structure of the wall, sheeting or MSE wall. We can model such as an anchoring crash barrier, crash vehicle, load from billboards and hoardings etc. The program does not modify the applied forces in the calculation except multiplying them with the corresponding coefficients according to the selected type of load (EN1997, LRFD).

External load acting on the terrain is necessary to define as a surcharge.



Frame "Applied forces"

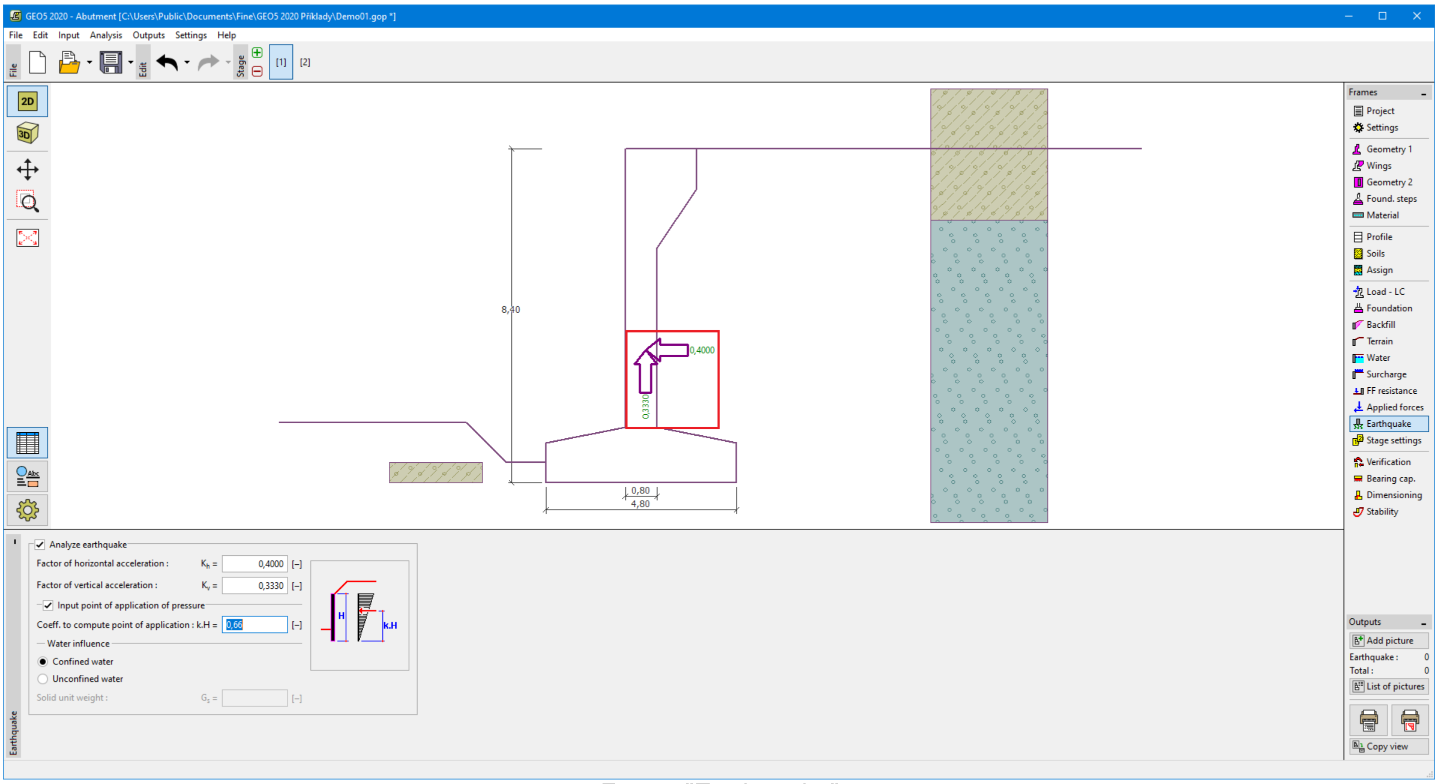
Earthquake

The "**Earthquake**" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements, the coefficients k_h and k_v can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for an earthquake is described in the theoretical part of the help, chapter "Influence earthquake".

For the LRFD Verification methodology, it is possible to define coefficients for seismic combinations according to the AASHTO.



Frame "Earthquake"

Stage Settings

The frame "**Stage settings**" serves to input settings valid for a given construction stage.

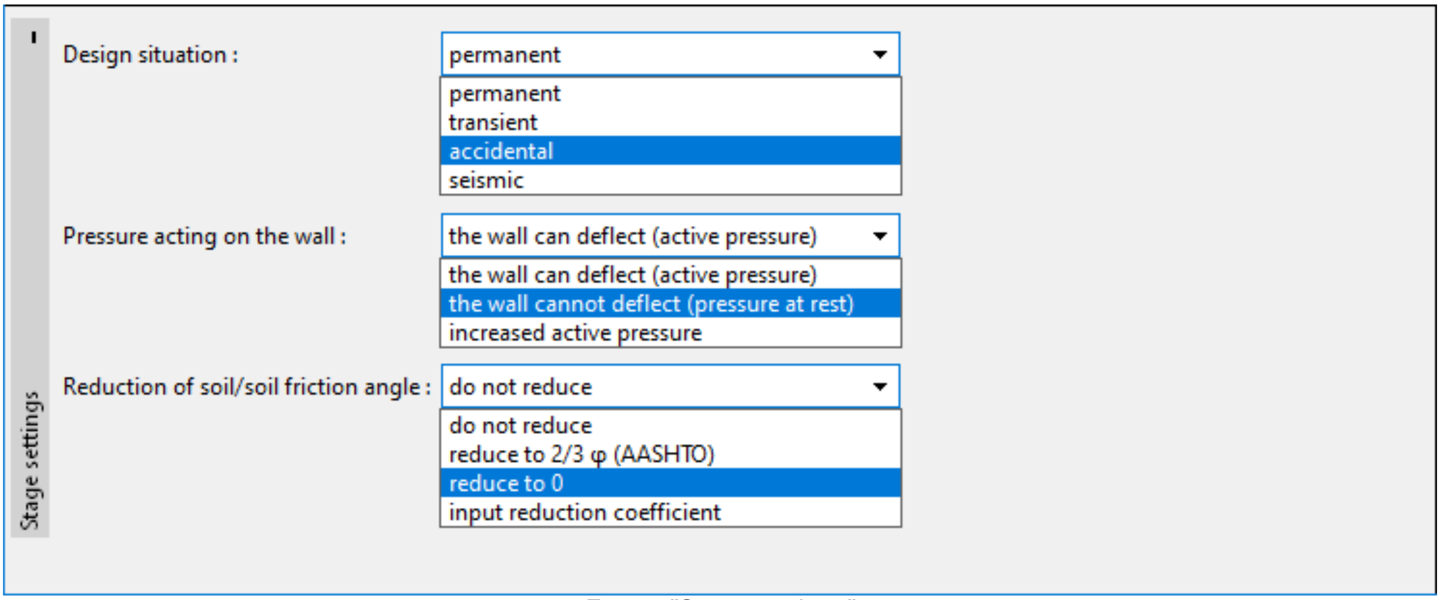
The selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology.

Next, the frame serves to specify the type of pressure acting on a wall based on the allowable wall deformation. Providing the wall is free to move, an active pressure is assumed, otherwise, a pressure at rest is used. The third option enables to load the wall by increased active pressure.

The reduction of soil/soil friction angle can be considered in one of following ways:

- **do not reduce**
- **reduce to 2/3φ (AASHTO)**
- **reduce to 0**
- **input reduction coefficient**



Frame "Stage settings"

Verification

The frame "**Verification**" shows the analysis results. Several computations can be carried out for a single task.

The frame appearance is adjusted based on the selected verification methodology.

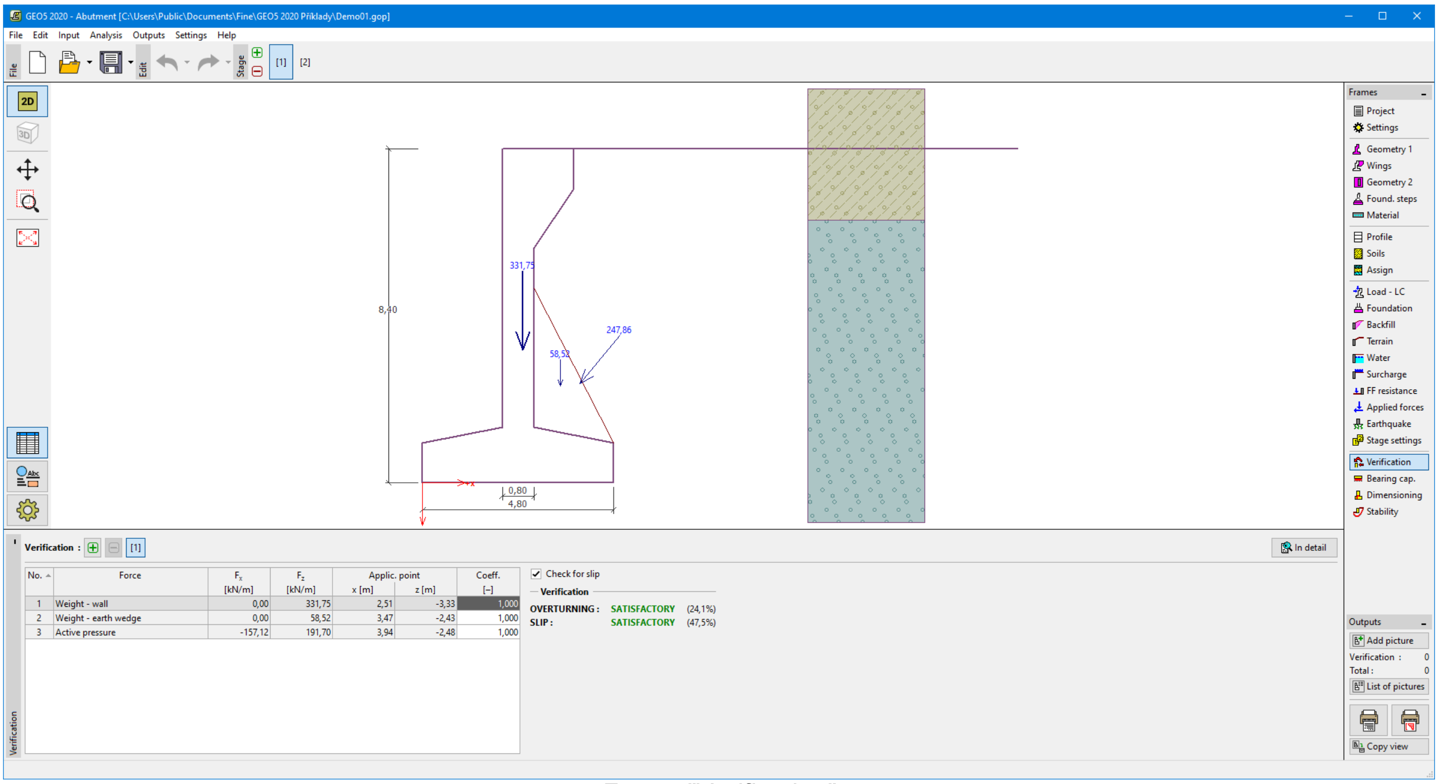
- Verification according to the factor of safety or the theory of limit states - the last column **F** in the table allows for inputting the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows for specifying whether the load acting on a structure is considered as the secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in this case, the last column is not displayed.

The wall is loaded either by active pressure or pressure at rest depending on input in the frame "Stage settings".

The procedure for wall verification is described in the theoretical part of the help.

The computed forces are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against **overturning** and **translation**. The "**In detail**" button opens the dialog window, which contains a detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Verification"

Bearing Capacity

The "**Bearing capacity**" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the frame "Verification". The programs "**Spread footing**", "**Spread footing CPT**", "**Pile**" and "**Pile group**" then consider all verifications as load cases. In the program "**Pile CPT**", just normal load is used.

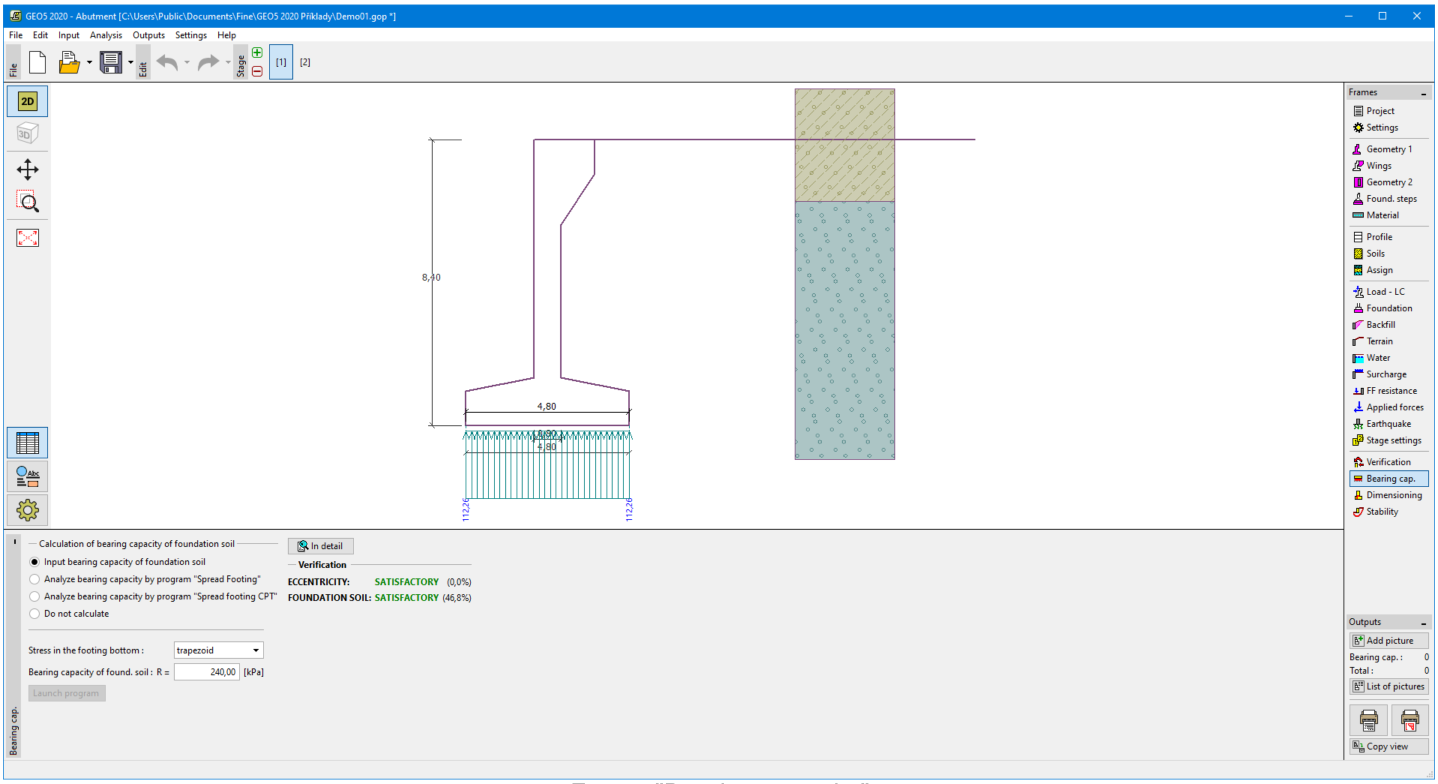
The frame contains the following analysis options:

- **Insert bearing capacity of foundation soil** — The foundation soil bearing capacity is input. The **eccentricity** and bearing capacity analysis results are displayed in the right part of the frame. The "**In detail**" button opens a dialog window that displays detailed listing of the results.
- **Analyze bearing capacity by program "Spread footing"** — Pressing the "**Run program Spread footing**" button opens the program "**Spread footing**" which allows us to calculate the soil bearing capacity or settlement and rotation of the footing. Pressing the "**OK**" button leaves the analysis mode - the results and all plots are transferred into the program "**Abutment**". The "**Spread footing**" program must be installed for the button to be active. The overall length of the wall foundation is put in.

- **Analyze bearing capacity by program "Spread footing CPT"** The procedure is identical as if calculating soil bearing capacity by the **"Spread footing"** program.
- **Analyze bearing capacity by program "Pile"** The procedure is identical as if calculating soil bearing capacity by the "**Spread footing**" program. The "**Run program Pile**" is available if the wall has a pile foundation (frame "Foundation"). Pile spacing s is input.
- **Analyze bearing capacity by program "Pile CPT"** The procedure is identical as if calculating soil bearing capacity by the "**Pile**" program.
- **Analyze bearing capacity by program "Pile group"** The procedure is identical as if calculating soil bearing capacity by the "**Spread footing**" program. The "**Run program Pile group**" is available if the wall has pile foundation with more then one pile (frame "Foundation"). Pile spacing s , the overall number of pile rows n and loading length l are input.
- **Do not calculate (pile footing)** The foundation soil bearing capacity is not calculated.

The program allows us to specify a shape of stress in the footing bottom.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Bearing capacity"

Dimensioning

The "**Dimensioning**" frame serves to design and verify the reinforcement of abutment cross-section - the cross-section subjected to dimensioning is selected in the combo list. The table shows the abutment forces.

Offer of cross-sections that can be verified depends on the selected load case (construction, service). The following cross-sections are available for both the construction and service state:

- **Wall stem verification**
- **Construction joint verification** - **depth** of construction joint from construction top edge is specified
- **Wall jump verification**

The service state makes also possible to verify:

- **Verification of closure wall**
- **Verification wing - abutment -** the surface surcharge due to terrain is input, for actual analysis we refer to the "Wings" section

The frame appearance is adjusted based on the selected verification methodology.

- Verification according to the factor of safety or the theory of limit states - the last column in the table allows us to input the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows us to specify whether the load acting on a structure is considered as the secondary one. This is explained in more detail in the "Load combinations" section.
- Analysis according to LRFD - in this case, the last column is not displayed.

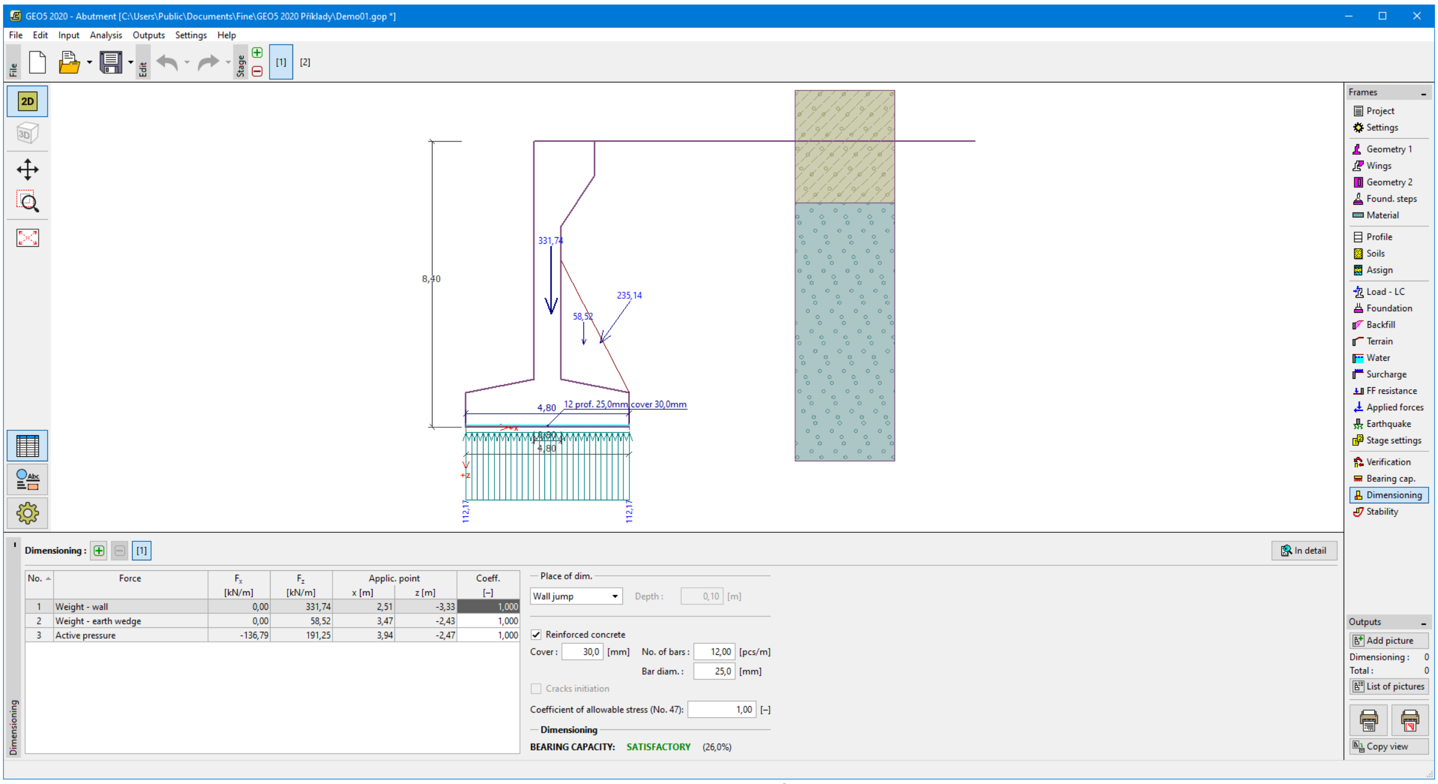
The abutment is loaded either by active pressure or pressure at rest depending on the input specified in the frame "Stage settings", an active earth pressure is used when analyzing wing walls.

The way to calculate the internal forces in cross-sections is described in the theoretical part of the help.

Dimensioning of the reinforced concrete is performed according to the standard set in the "Materials and standards" tab. Verification analysis based on the standard CSN 73 6206 **"Design of concrete and steel reinforced concrete bridge structures"** is described herein.

Several computations for various cross-sections can be carried out. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "**In detail**" button opens the dialog window that contains a detailed listing of the dimensioning results.

Visualization of results can be adjusted in the "Drawing Settings" frame.

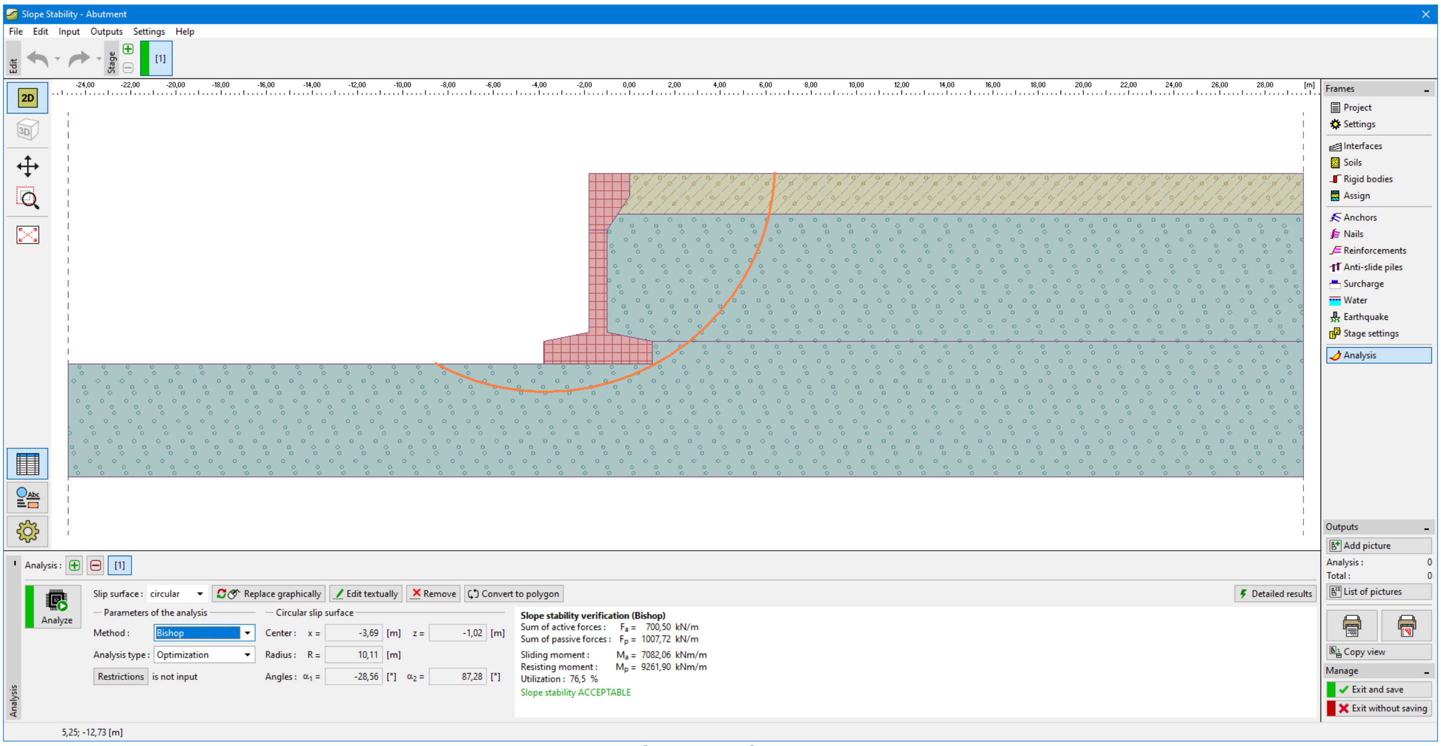


"Dimensioning" frame

Stability

Pressing the "**Stability**" button launches the "**Slope stability**" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the "**Slope stability**" program is installed.

After completing all analyses, press the "**OK**" button to leave the program - all data are then carried over to the analysis protocol of the "**Abutment**" program.



"Stability" frame

Program Nailed Slope

This program analyses nailed walls and slopes of various shapes.

The help in the program "Nailed Slope" includes the following topics:

- The input of data into individual frames:

Project	Settings	Geometry	Types of Nails	Geometry of Nails	Sections	Material
Mesh Types	Profile	Soils	Assign	Terrain	Water	Surcharge
Earthquake	Stage Settings	Internal Stability	Verification	Bearing Capacity	Dimensioning - concrete cover	Dimensioning - mesh
Stability						

- Standards and analysis methods

- Theory for analysis in the program "**Nailed Slope**":

Stress in Soil Body	Earth Pressures	Analysis of Walls	Nailed Slope	Dimensioning of Concrete Structures

- Outputs

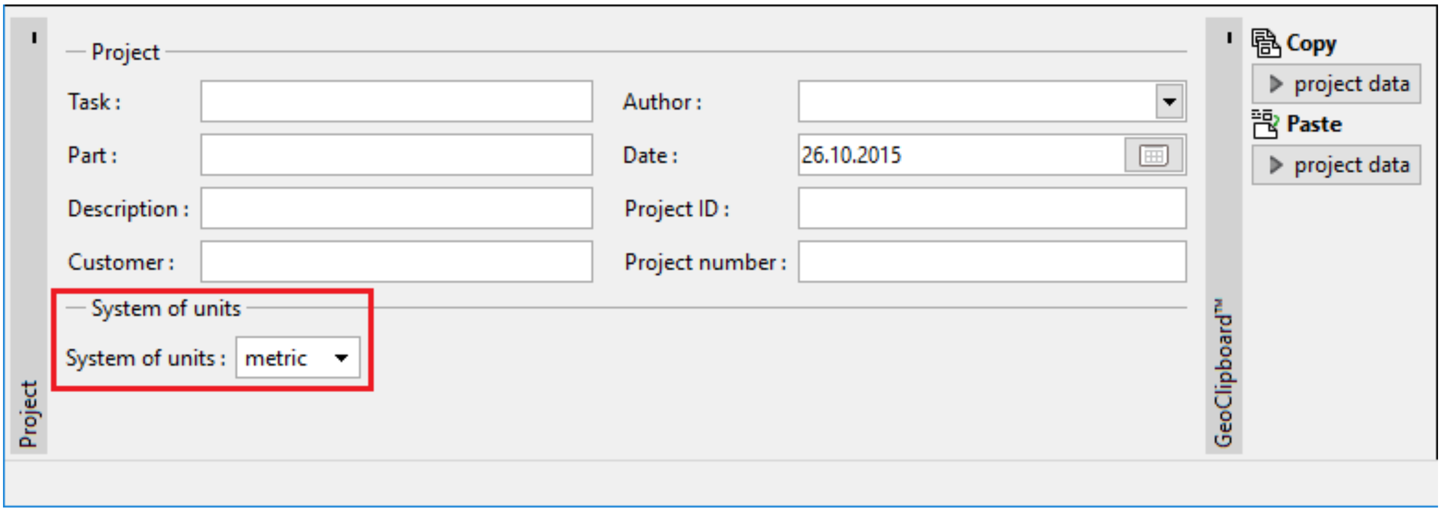
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The **"Project"** frame is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The frame "**Settings**" allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

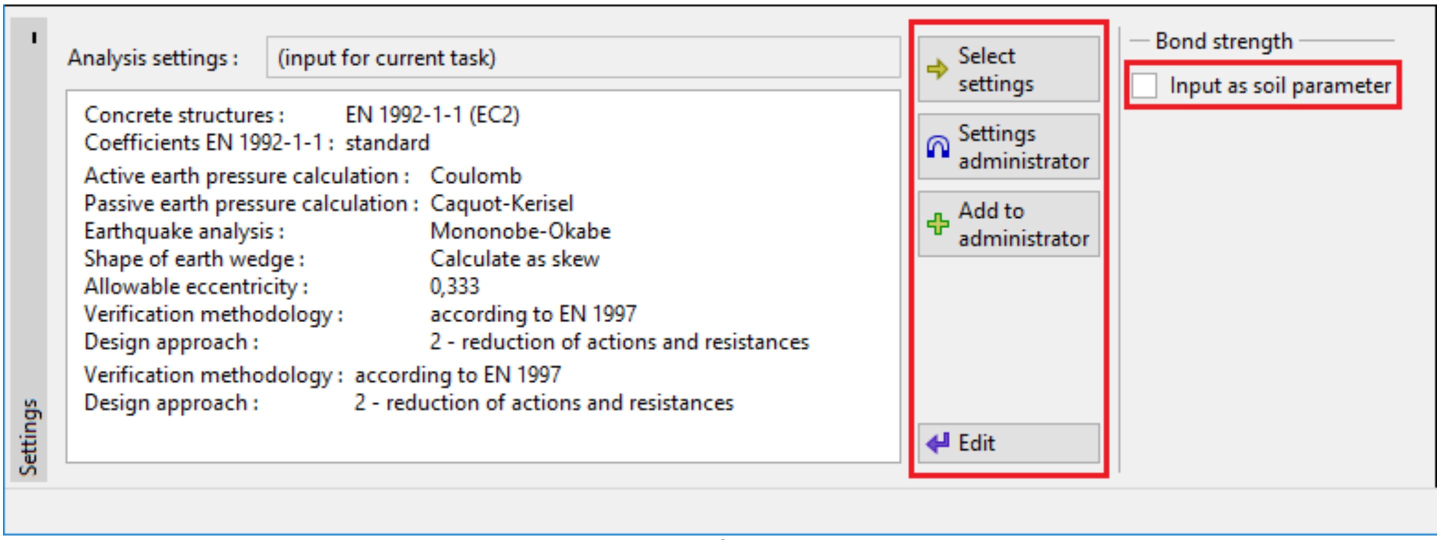
The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

Settings of analysis parameters are performed in the "Materials and standards", "Wall analysis" and "Stability analisis" tabs.

When calculating the pull-out resistance of nail, bond strength can be inserted as a soil parameter.



Frame "Settings"

Geometry

The "**Geometry**" frame allows us to select the type of cover:

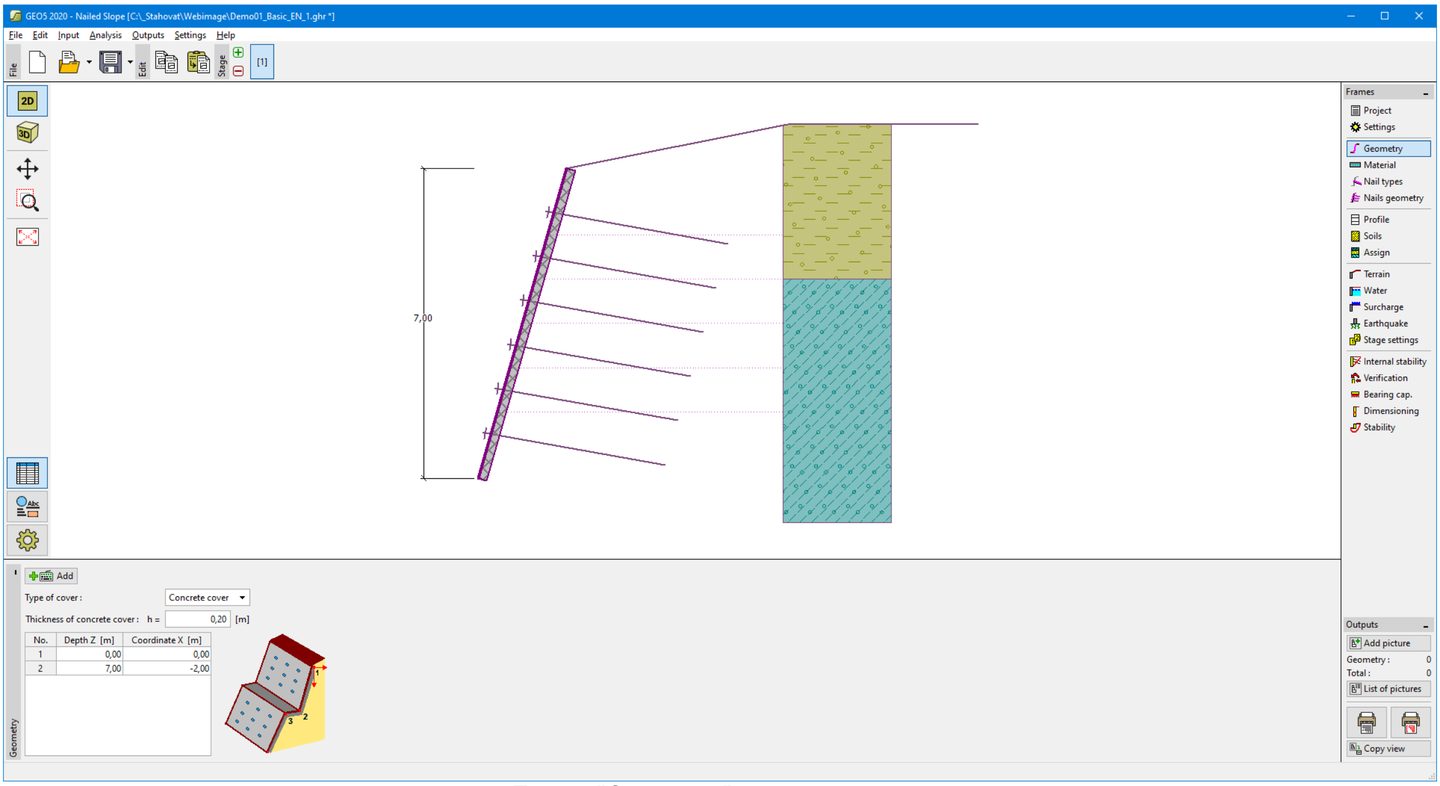
- **Concrete cover -** concrete cover thickness is input
- **Mesh -** thickness and parameters of weathered layer are input (angle of internal friction, cohession and unit weight)

The frame also contains a table with a list of the structure front face points.

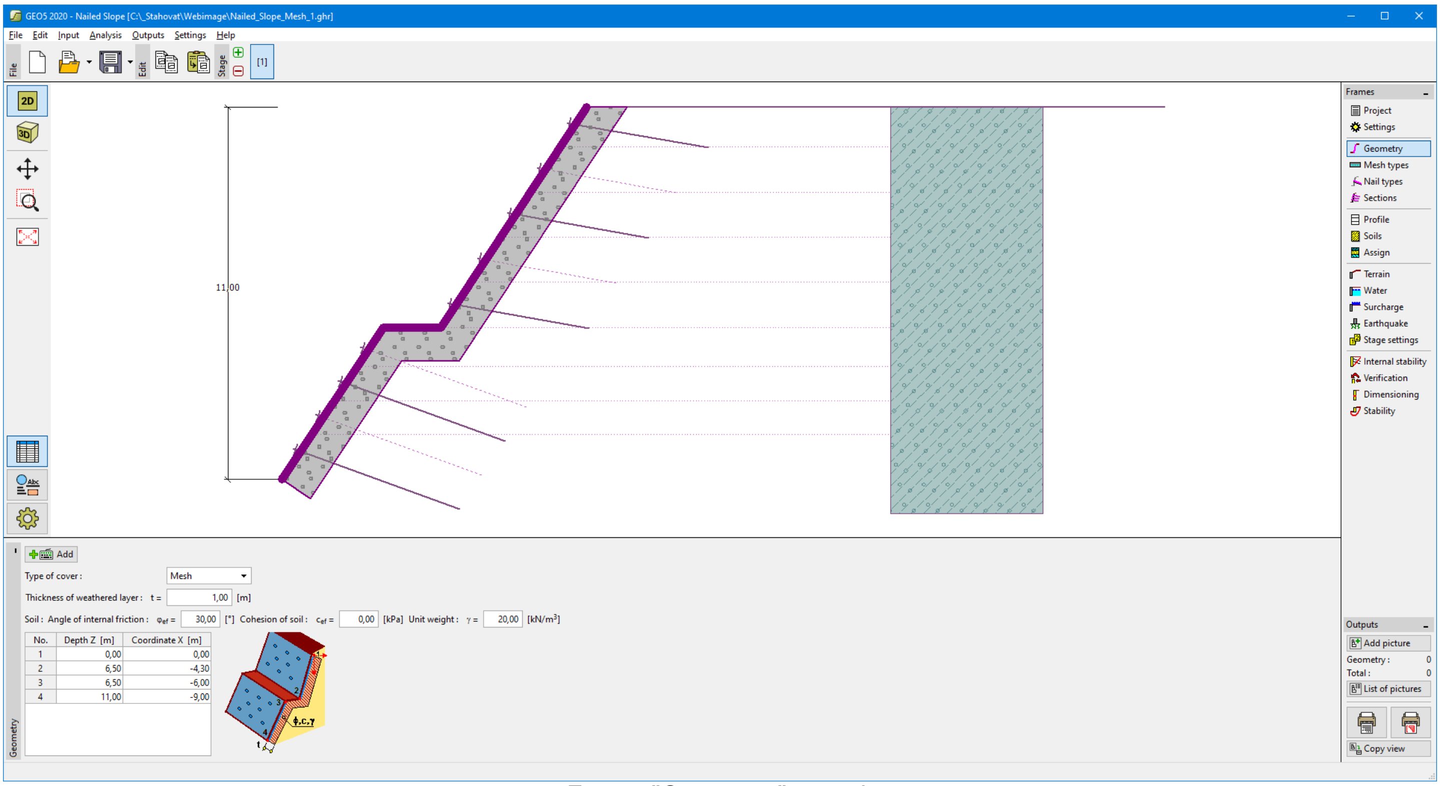
The input points can also be edited on the desktop with the help of active objects - double-clicking an already input point then opens a dialog window for its editing.

One needs to specify depth (z -coordinate from the top point of a structure - positive direction is assumed downwards) and an x -coordinate (negative direction is assumed to the left, no overhang of a structure is allowed.

The program makes it possible to export the structure geometry in the *.DXF format.



Frame "Geometry" - concrete cover



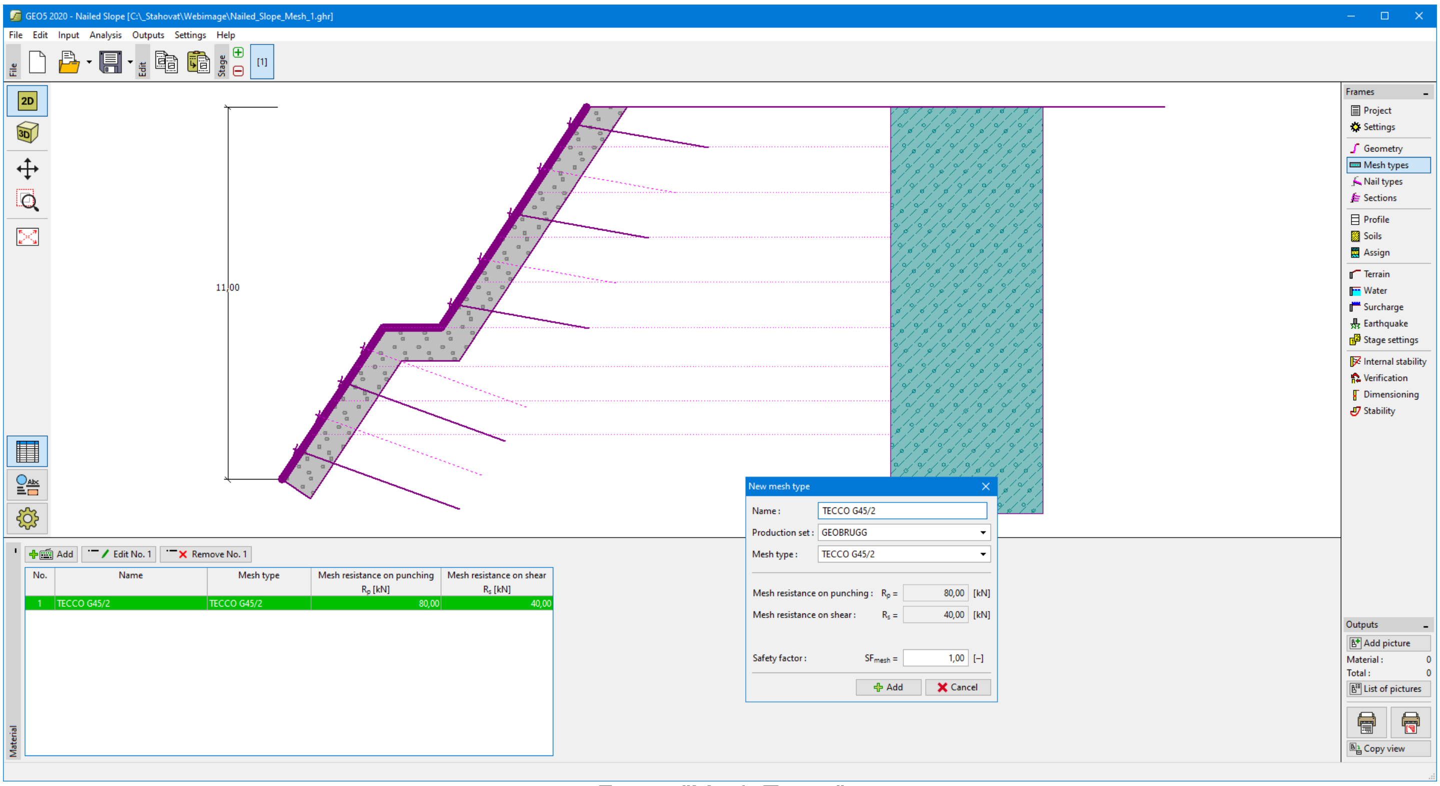
Frame "Geometry" - mesh

Mesh Types

The "**Mesh Types**" frame serves to specify mesh types in a table. The strength parameters of the mesh can be either input or taken from the database.

The table lists the following input data:

- **Mesh resistance on punching**
- **Mesh resistance on shear**



Frame "Mesh Types"

Note:

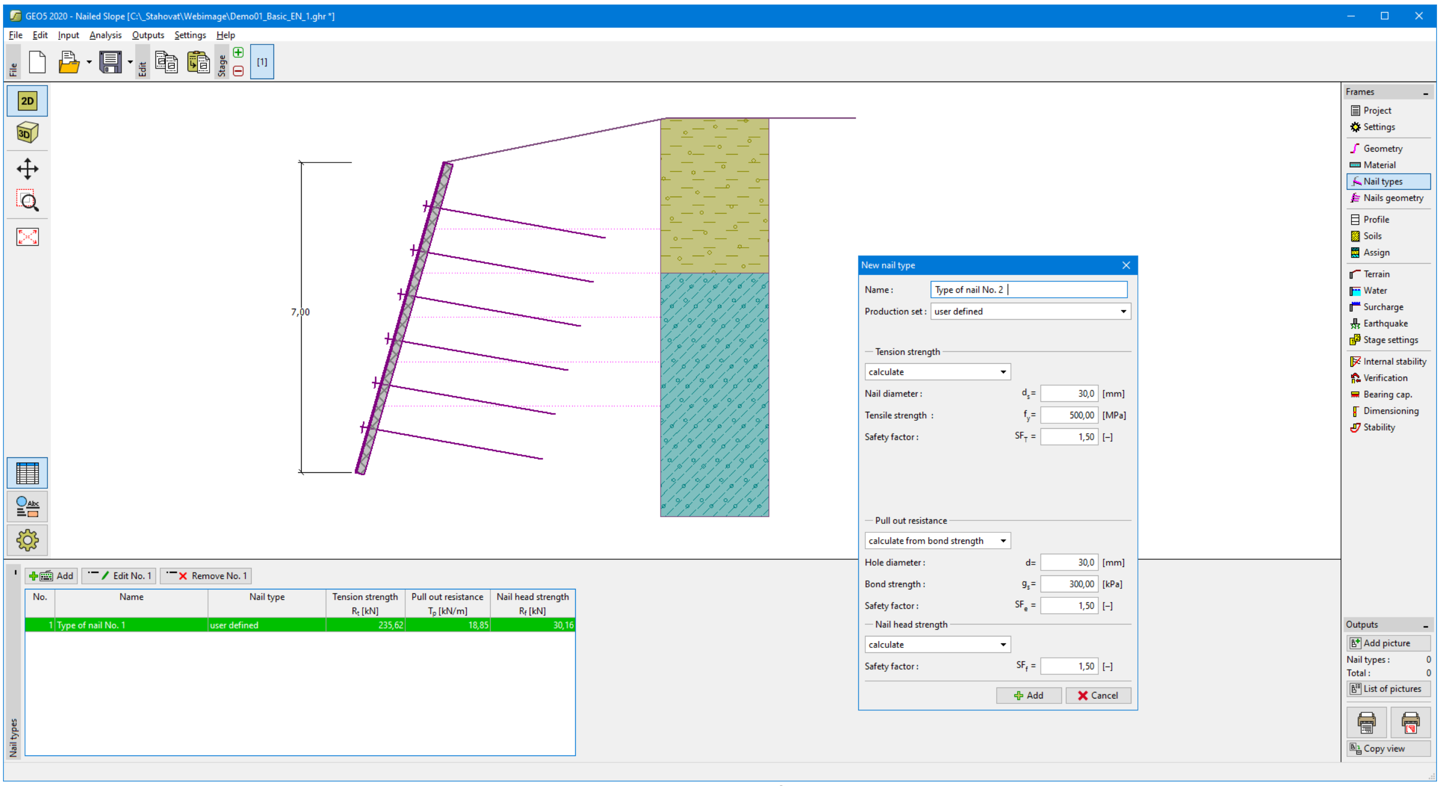
The database mesh strengths are given by manufacturers. The user should know how the values were determined and also use appropriate safety factors.

Types of nails

The "**Types of nails**" frame serves to specify a nail type in a table. The strength parameters of nails can be either **input** by the user or determined by the program from the input data.

The table lists the following input data; either input or computed:

- **nail head strength**
- **nail tensile strength**
- **pull - out resistance** per 1 ***m*** (1 ***ft***).



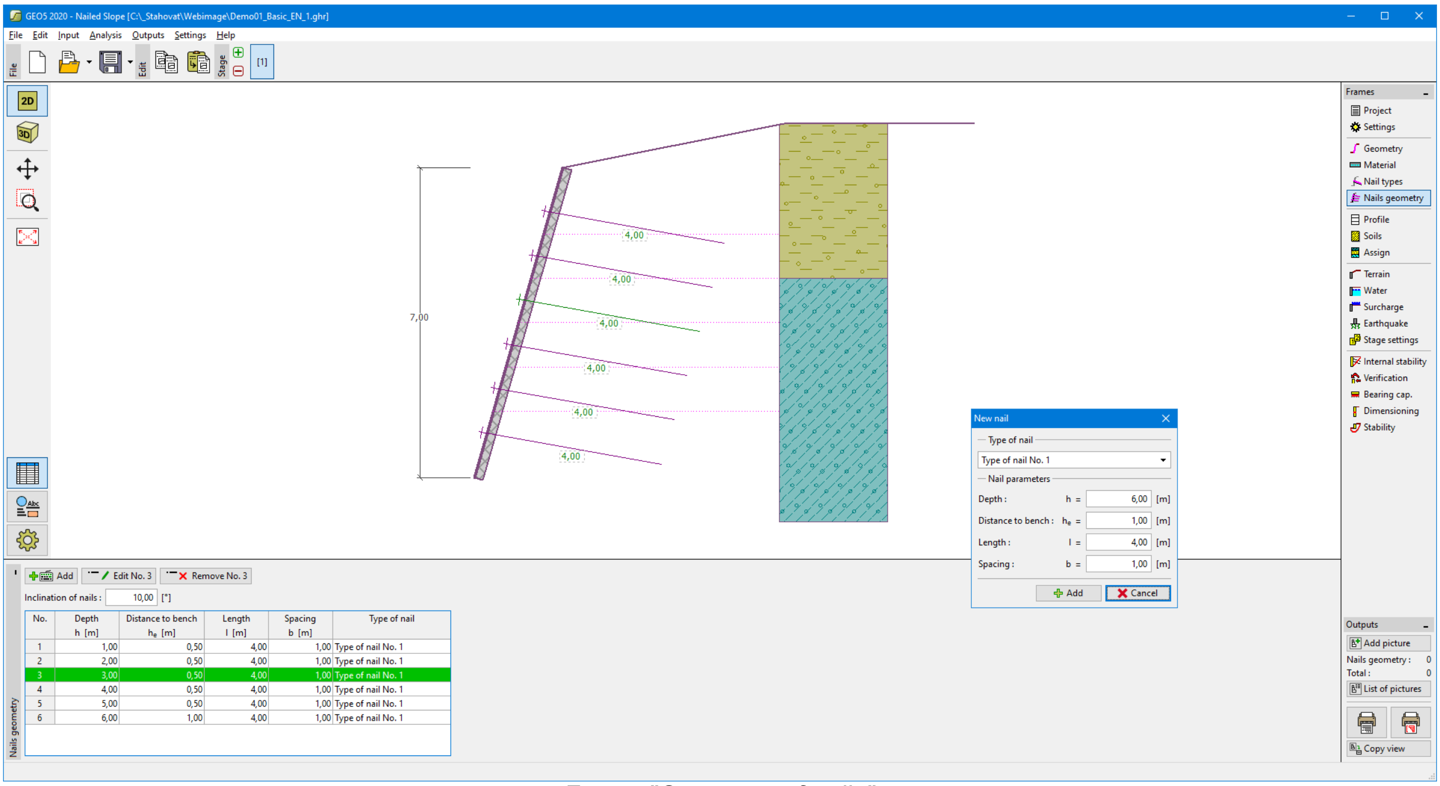
Frame "Types of nails"

Geometry of Nails

The "**Geometry of nails**" frame contains a table with a list of input nails. Adding nails is performed in the "**New nail**" dialog window. The input nails can also be edited on the desktop with the help of active objects.

The user is required to specify the nail depth, **depth of a bench from a given nail** (the next nail must be introduced below the bench of the upper nail), nail length, its diameter, and distance.

The inclination of nails is regarded from the horizontal line in a clockwise direction and is constant for all nails.

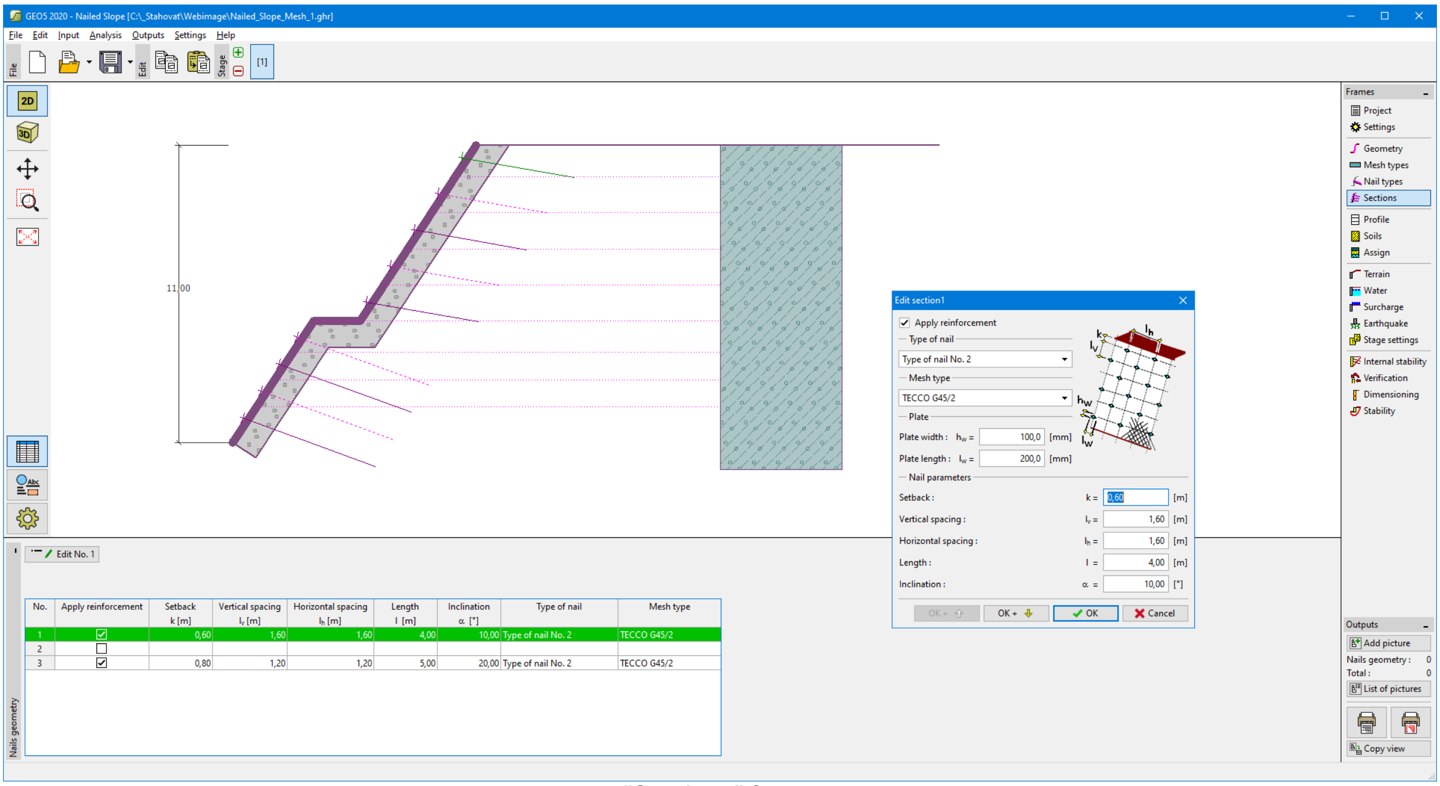


Frame "Geometry of nails"

Sections

The "**Sections**" frame contains a table with a list of sections. Sections are generated automatically according to geometry input.

For the reinforced section, the user is required to choose the nail type and mesh type from the list. Further, it is necessary to input plate length and width, offset, vertical and horizontal spacing, nail length, and nail inclination.



"Sections" frame

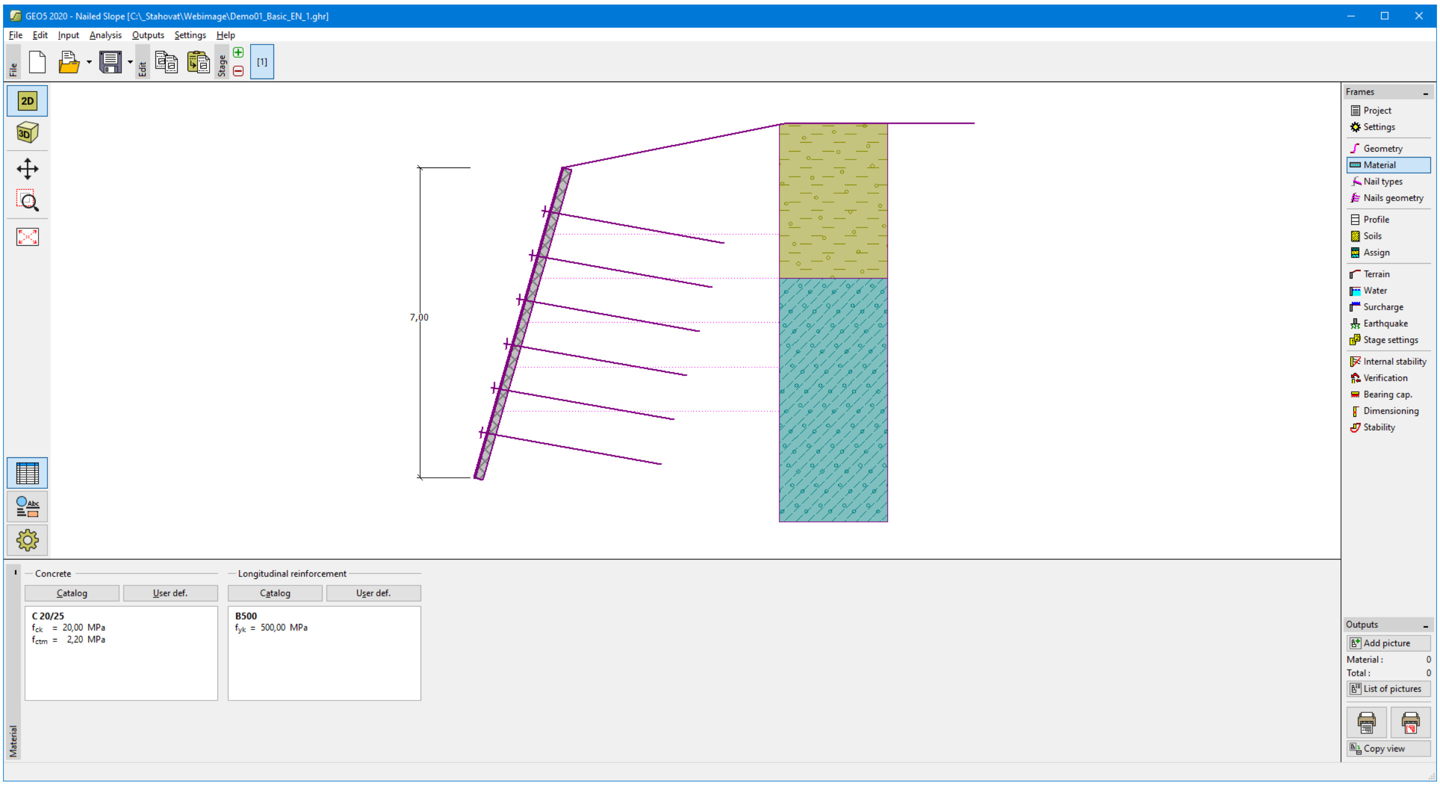
Material

The "**Material**" frame allows for the selection of material parameters for concrete and longitudinal steel reinforcements.

Two options are available when selecting the material type:

- The "**Catalog**" button opens the "**Material catalog**" dialog window (for concrete or steel reinforcements), the list of materials then serves to select the desired material.
- The "**Own**" button opens the "**Concrete**" dialog window (for concrete) or the "**Reinforcing steel bars**" dialog window (for longitudinal steel reinforcements), which allows for manual specification of material parameters.

The catalogs content depends on the selection of standard for the design of concrete structures set in the "Materials and standards" tab.



Frame "Material"

Profile

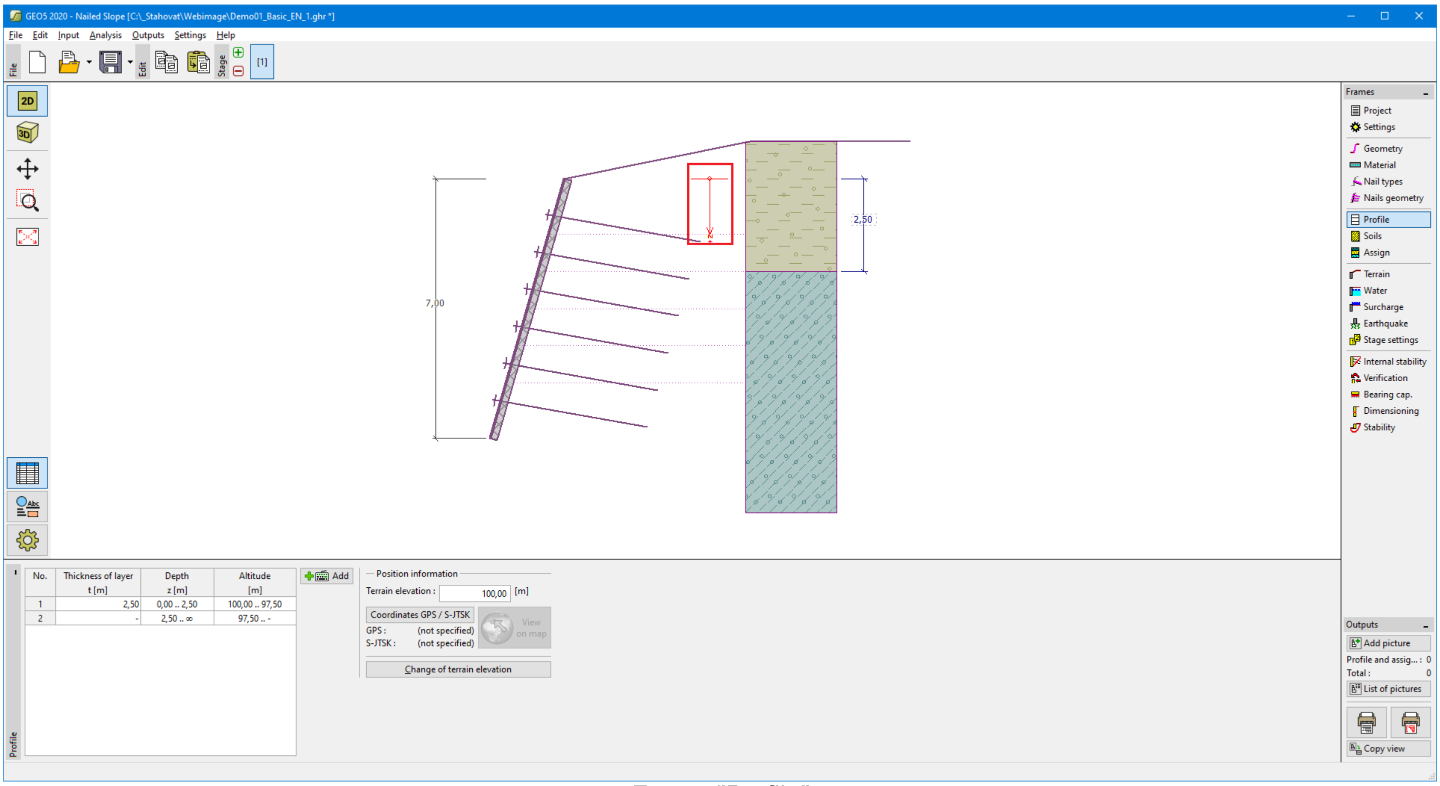
The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.



Frame "Profile"

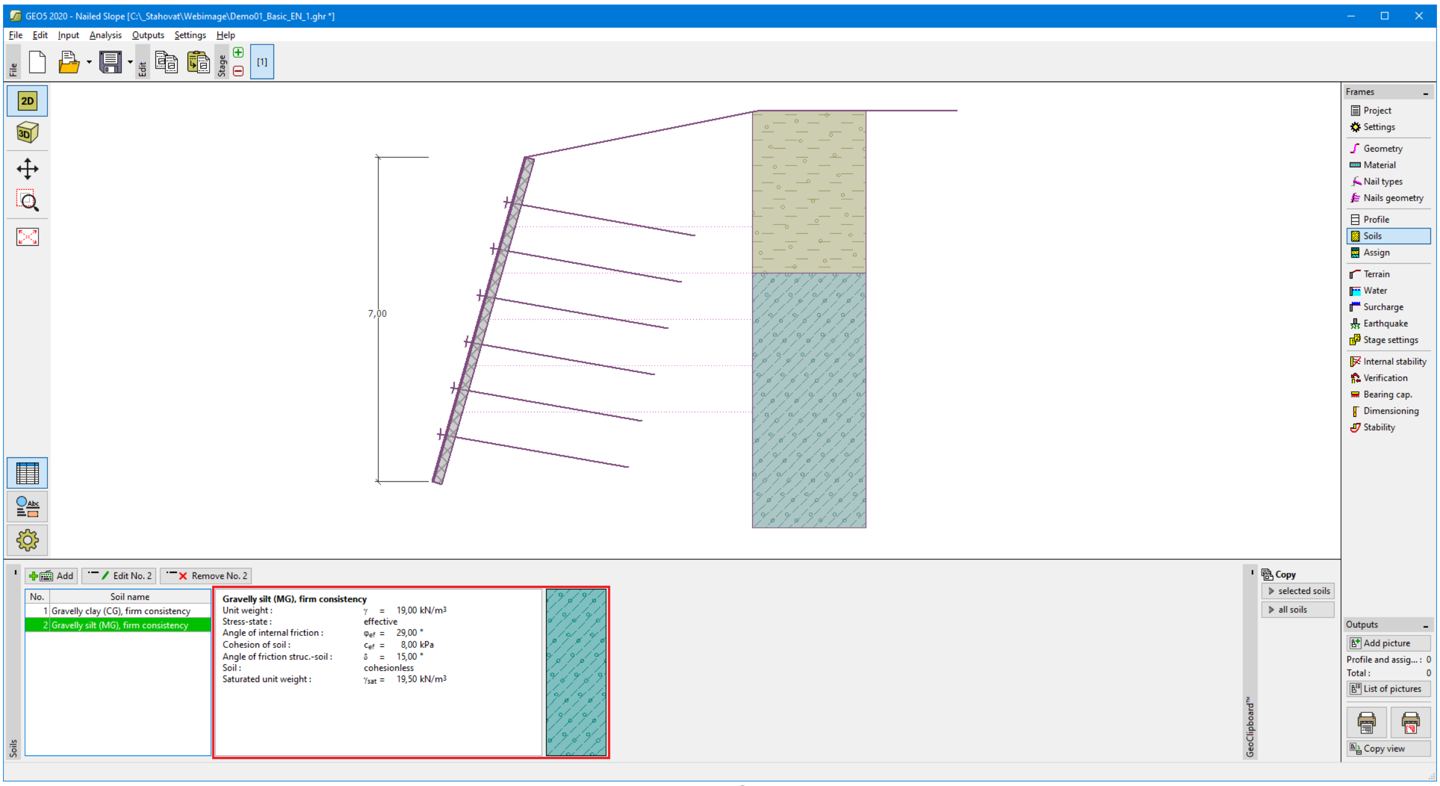
Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Earth pressure at rest", "Uplift pressure" and "Bond strength" (see frame "Settings").

Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

Basic Data

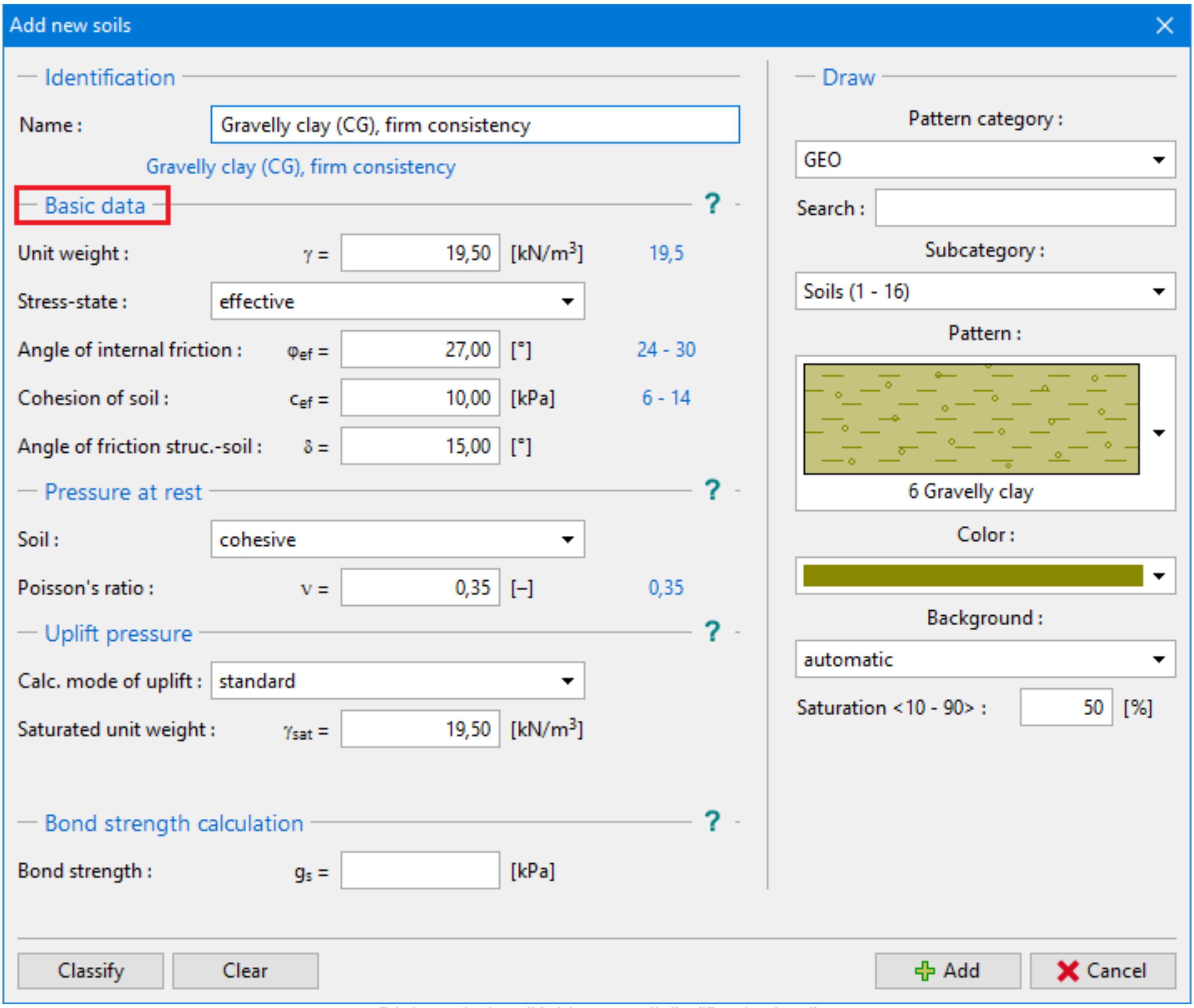
This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction, and cohesion**. The particular values are obtained from a geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit the built-in database of soils, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually.

Either **effective or total** parameters of the angle of internal friction and cohesion are specified depending on the setting in the "**Stress analysis**" combo list. Whether to use effective or total parameters depends primarily on the type of soil, type of load, structure duration, and water conditions.

For effective stress further needs to specify the angle of internal friction between the soil and structure, which depends on the structure material and the type of soil. Possible values of this parameter are listed in the table of recommended values.

For total stress further needs to specify the adhesion of soil to the structure face a .

The associated theory is described in detail in the chapter "Earth pressures".

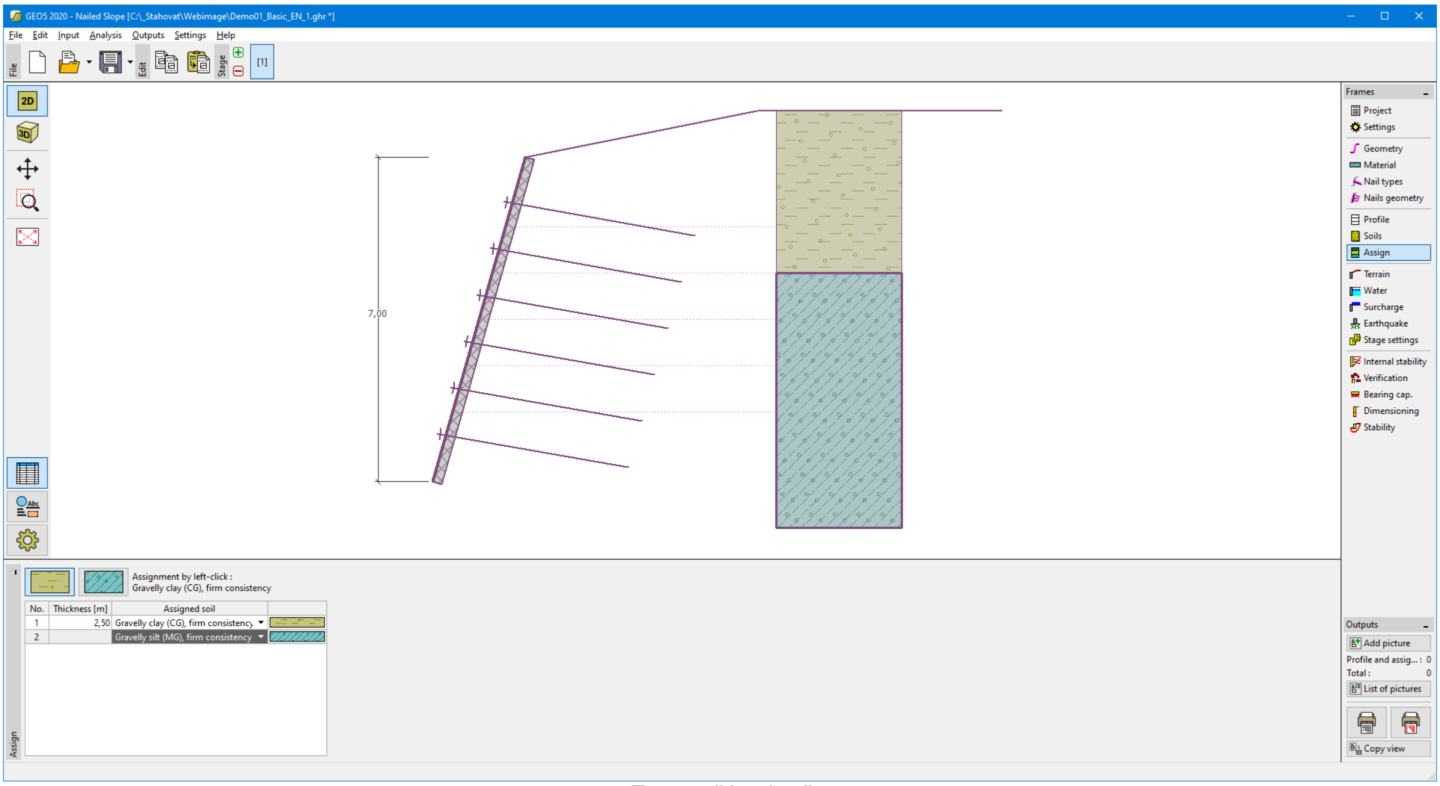


Dialog window "Add new soils" - "Basic data"

Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail herein.



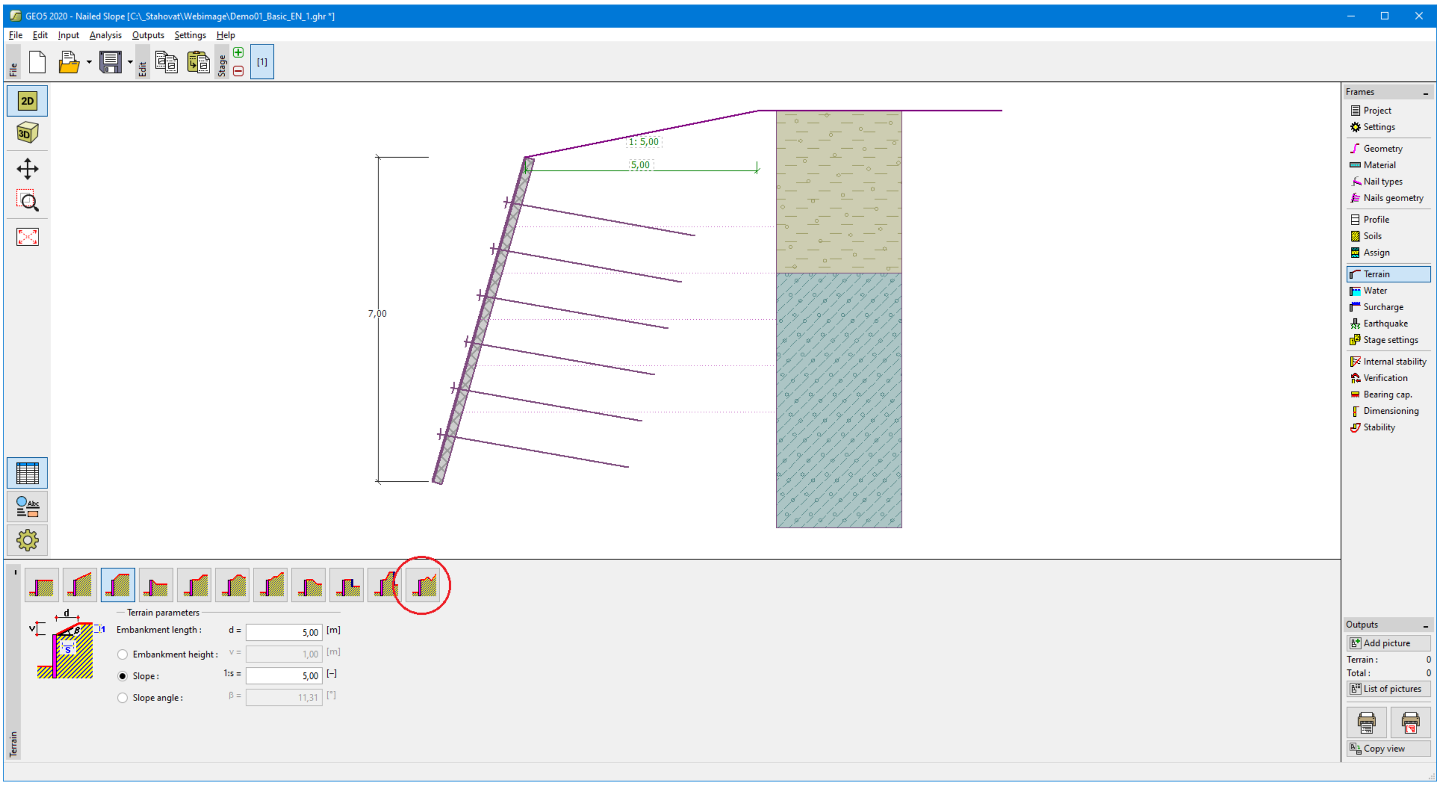
Frame "Assign"

Terrain

The "**Terrain**" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case, the frame contains a table with a list of terrain points. The first point with coordinates [0; 0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "Distribution of earth pressures for broken terrain".



Frame "Terrain"

Water

The "**Water**" frame allows, by pressing the button, for selecting the type of water. The selected type with a graphic hint of input values is displayed in the left part of the frame. Water parameters (h_1, h_2...) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

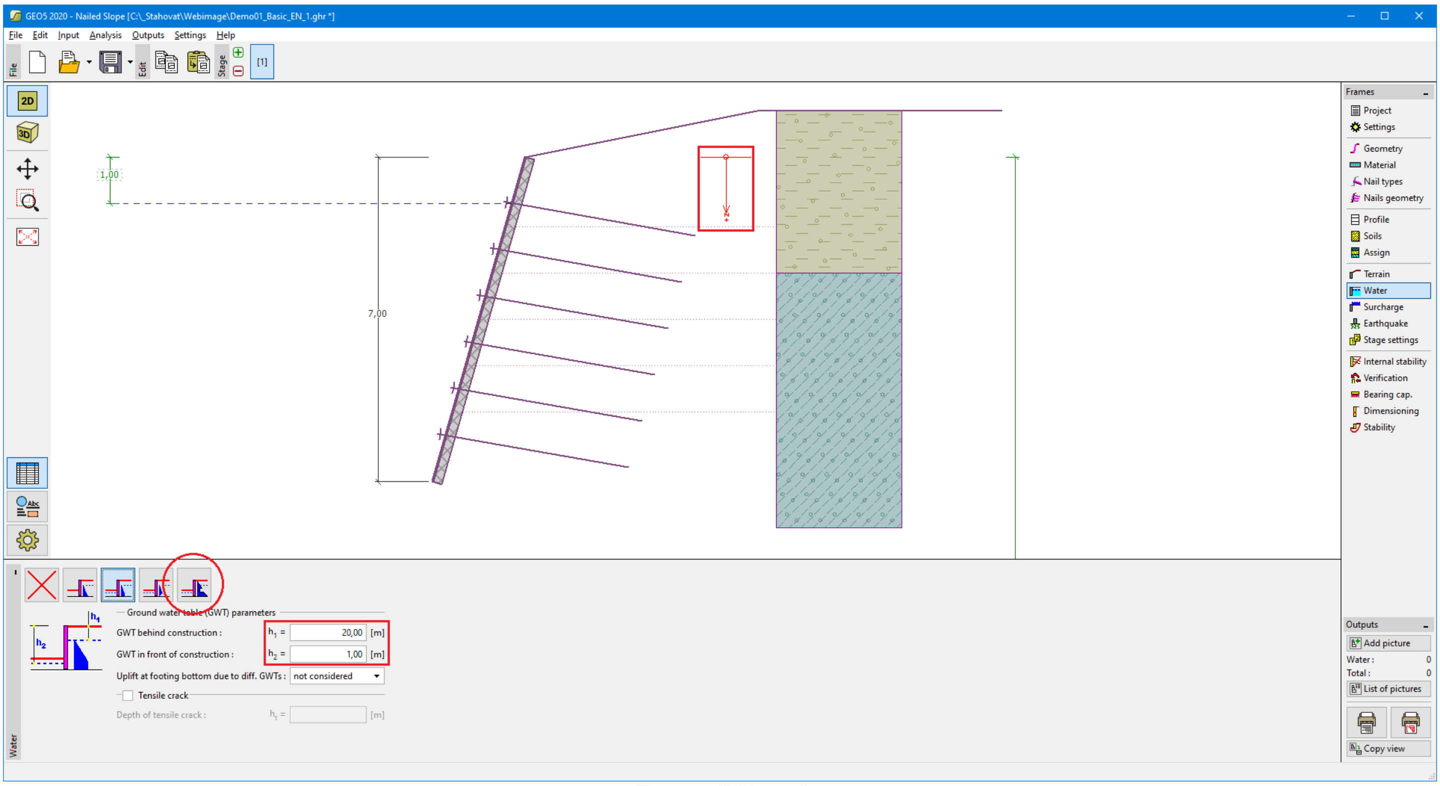
The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be linear, parabolic, or it may not be considered at all. When verifying the wall, the uplift pressure in the base of footing due to the different water tables is introduced as a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tabs "**In front of structure**" and "**Behind structure**" appear with tables. The table contains the values of pore pressure in front of the structure or behind it.

The groundwater table can also be specified **above the structure** or earth profile, respectively - in such a case, the depth of water is input with a negative value.

Analysis of earth pressures with the influence of water is described in the theoretical part of the help, chapter "Influence of water".

The program further allows for specifying a depth of tensile cracks filled with water.



Frame "Water"

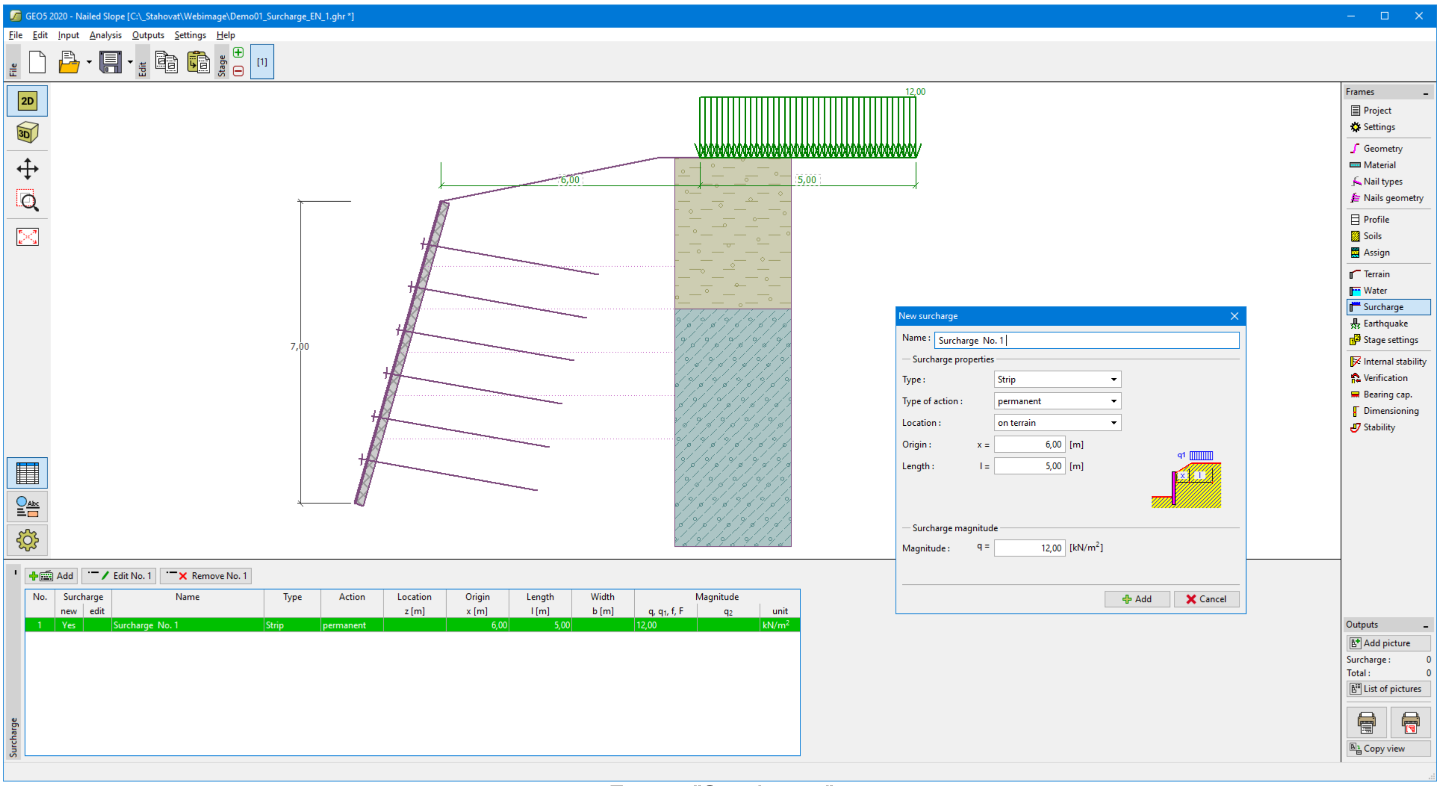
Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The z -coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case when the surcharge is found out of the terrain the program prompts an error message before calculation.

Either **permanent**, **variable,** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with a favorable effect is not considered in the analysis.

Analysis of earth pressures due to surcharges is described in the theoretical part of the help, chapter "Influence of surcharge".



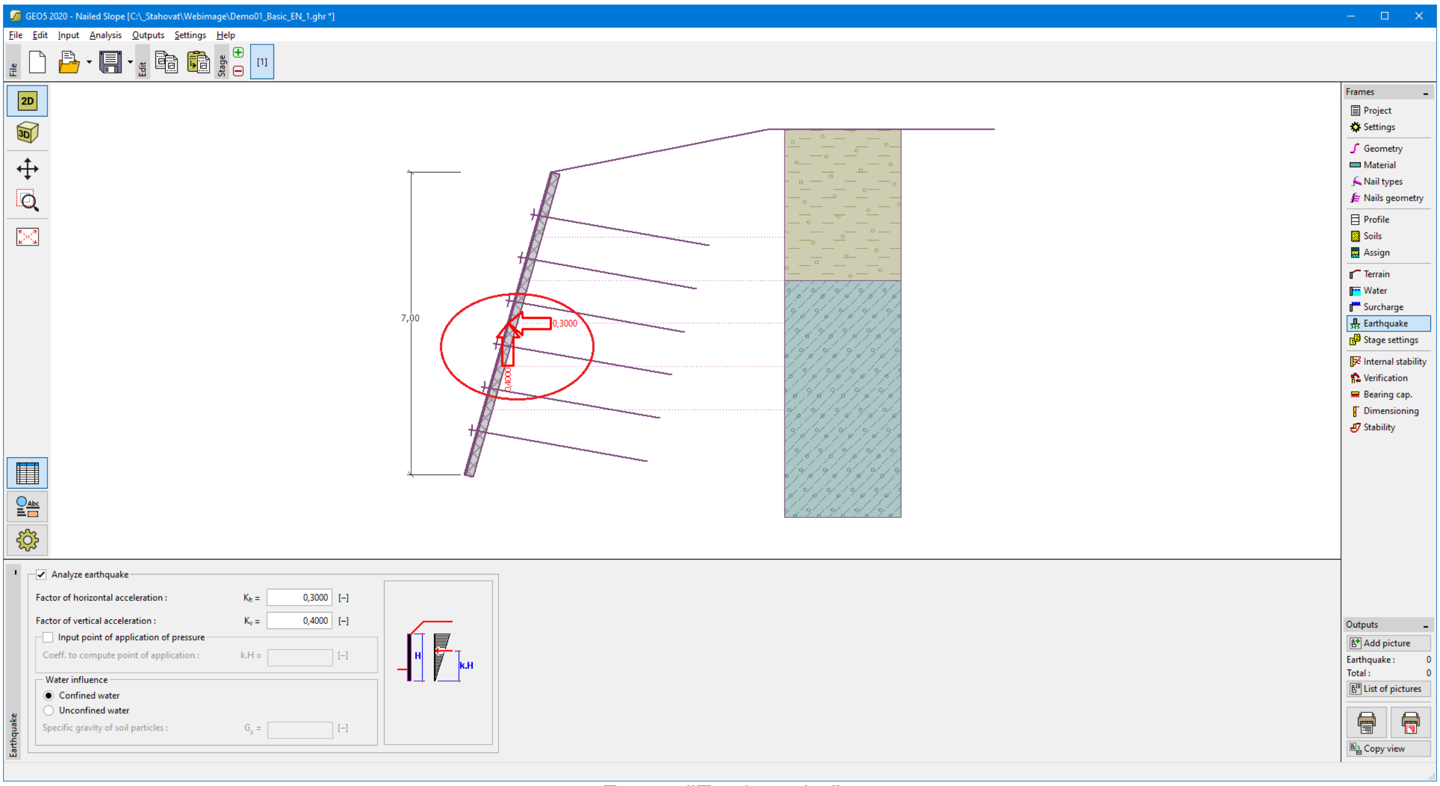
Frame "Surcharge"

Earthquake

The "**Earthquake**" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements, the coefficients k_h and k_v can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for an earthquake is described in the theoretical part of the help, chapter "Influence earthquake".



Frame "Earthquake"

Stage Settings

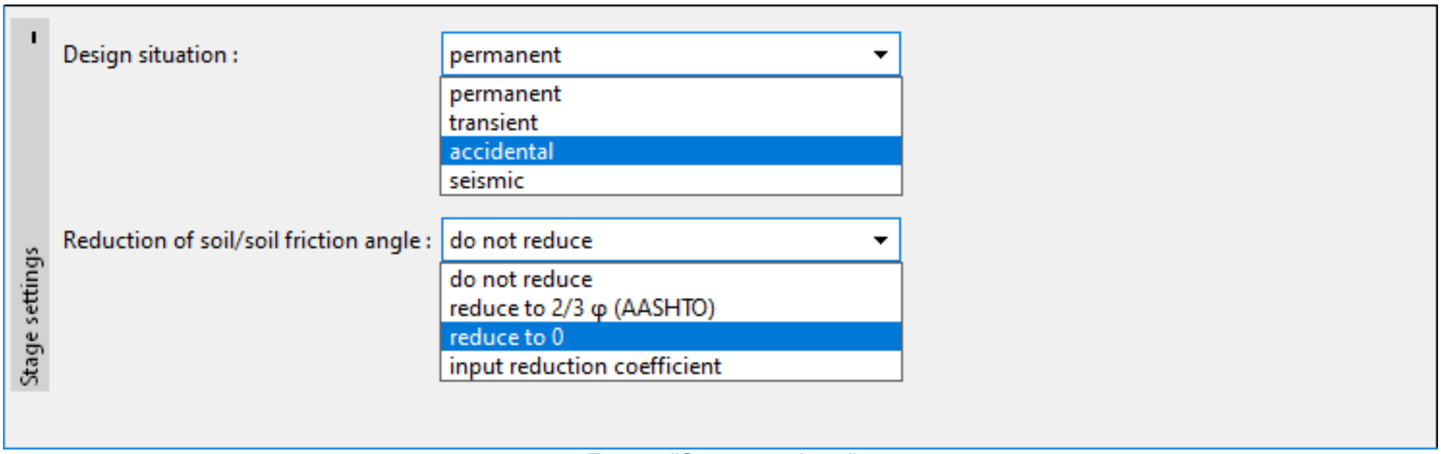
The frame "**Stage settings**" serves to input settings valid for a given construction stage.

The selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology.

The reduction of soil/soil friction angle can be considered in one of following ways:

- **do not reduce**
- **reduce to 2/3φ (AASHTO)**
- **reduce to 0**
- **input reduction coefficient**



Frame "Stage settings"

Internal Stability

The frame allows for the verification of internal stability of a structure assuming either **plane** or **broken slip surface**.

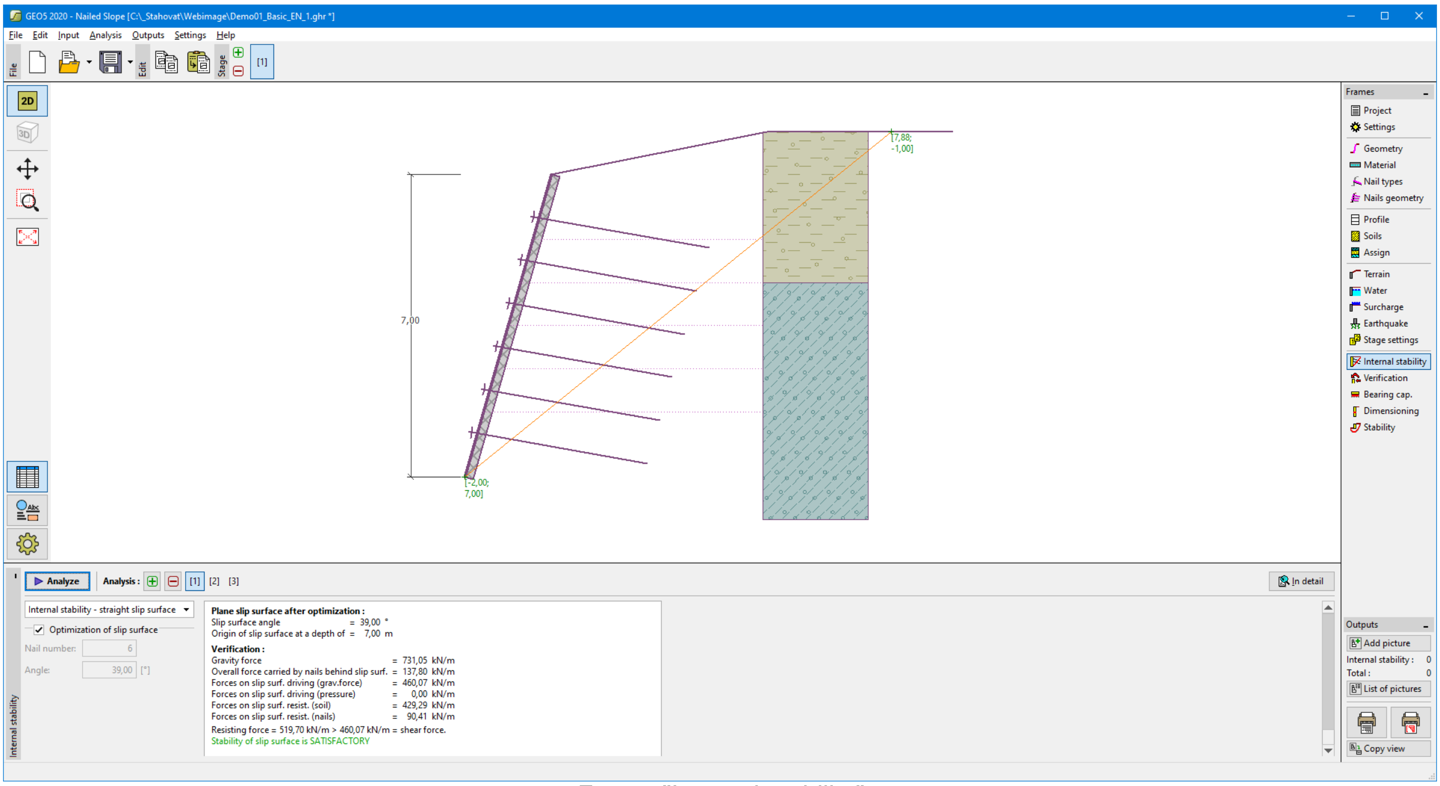
The verification of internal stability is performed:

- According to EN 1997 (the actual verification is performed according to the theory of limit states).
- According to LRFD (the actual verification is performed according to the theory of limit states).
- Verification according to the factor of safety or the theory of limit states (verification is performed depending on the setting in the "Stability analysis" tab)

Individual steps of the verification procedure are described herein.

This frame also allows for the verification of nails bearing capacity.

To determine the nail force, in this frame is defined reduction coefficient of active earth pressure k_n .



Frame "Internal stability"

Verification

The frame "**Verification**" shows the analysis results. Several computations can be carried out for a single task.

The frame appearance is adjusted based on the selected verification methodology.

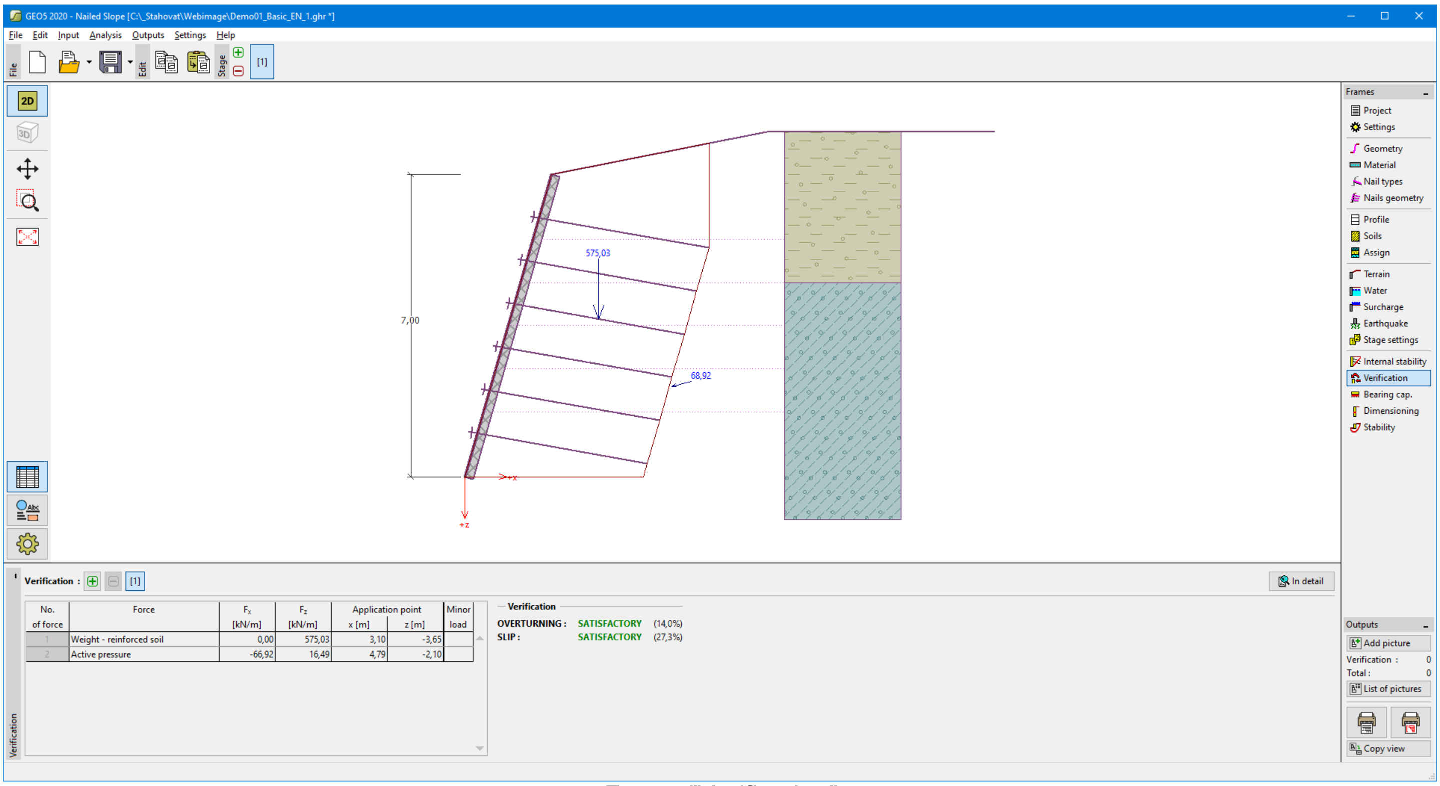
- Verification according to the factor of safety or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows for specifying whether the load acting on a structure is considered as the secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in this case, the last column is not displayed.

To verify the external stability a **fictitious structure** (wall) is created and further subjected to the verification analysis. A fictitious wall consists of the structure front face, a line connecting end points of individual nails, a vertical line constructed from the end point of the first nail up to the terrain depth and from the end point of the last nail up to the structure depth (thus the bottom edge of a fictitious structure is always horizontal). The wall points that cause a concave curvature of the structure back face are automatically excluded by the program. The structure is loaded by an active earth pressure.

The procedure for wall verification is described in the theoretical part of the help.

The computed forces are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against **overturning and translation**. The "**In detail**" button opens the dialog window, which contains a detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Verification"

Bearing Capacity

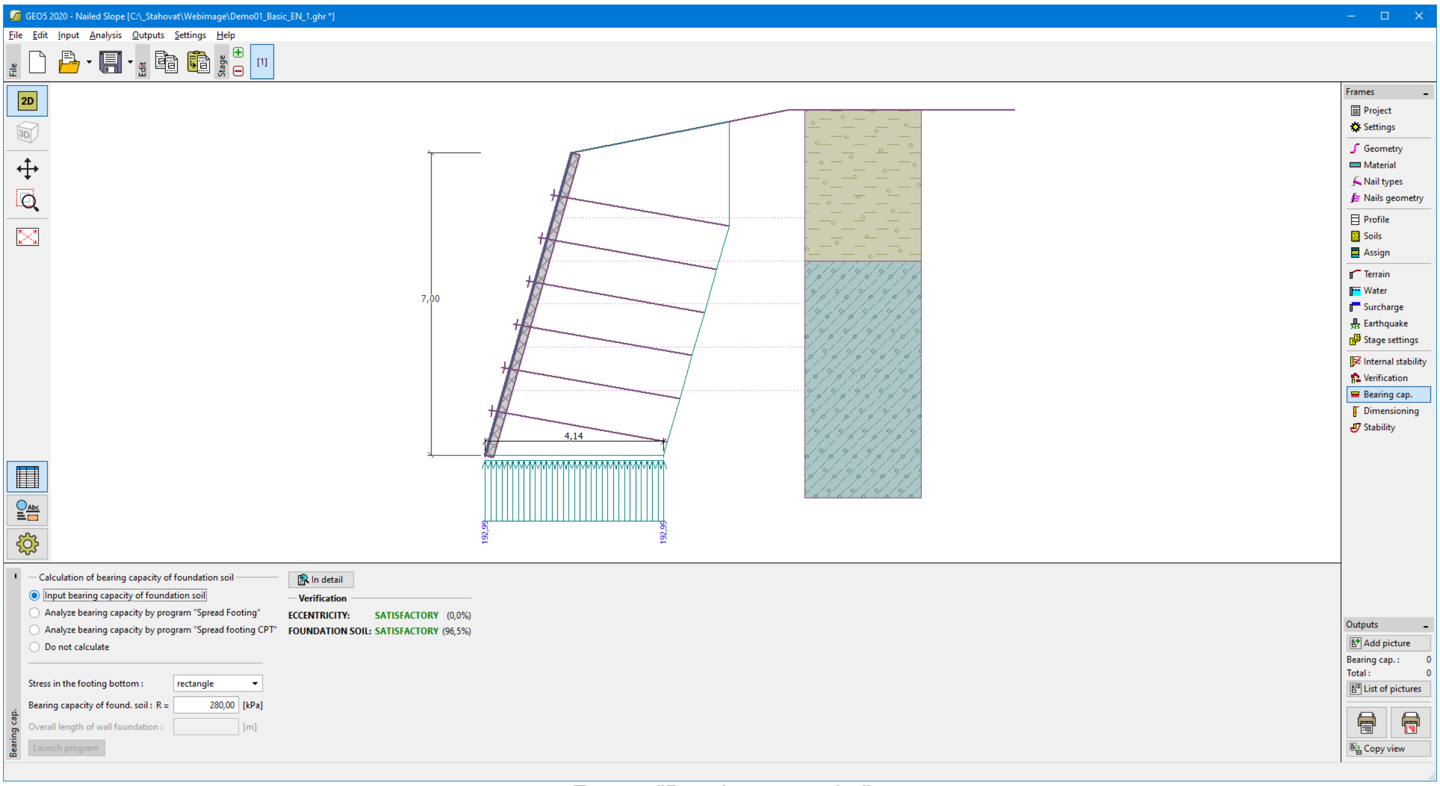
The "**Bearing capacity**" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the frame "Verification". The programs "**Spread footing**" and "**Spread footing CPT**" then consider all verifications as load cases.

Three basic analysis options are available in the frame:

- **Input the foundation soil bearing capacity** – The input field serves to specify the foundation soil bearing capacity. The results of verification analysis of soil for **eccentricity** and bearing capacity are displayed in the right part of the frame. The "**In detail**" button opens the dialog window that displays detailed listing of the results of verification analysis of foundation soil bearing capacity.
- **Compute the foundation soil bearing capacity using the program "Spread footing"** – Pressing the "**Run program Spread footing**" button starts the program "**Spread footing**" that allows for computing the soil bearing capacity or settlement and rotation of a footing. Pressing the "**OK**" button leaves the analysis regime - the results and all plots are copied to the program "**Nailed Slope**". The program "**Spread footing**" must be installed for the button to be active. The overall length of the wall foundation is input.
- **Analyze bearing capacity by program "Spread footing CPT"** – The procedure is identical as if calculating soil bearing capacity by the "**Spread footing**" program.
- **Do not compute (pile footing)** – The foundation soil bearing capacity is not computed.

The program allows us to specify a shape of stress in the footing bottom.

Visualization of results can be adjusted in the frame "Drawing Settings".



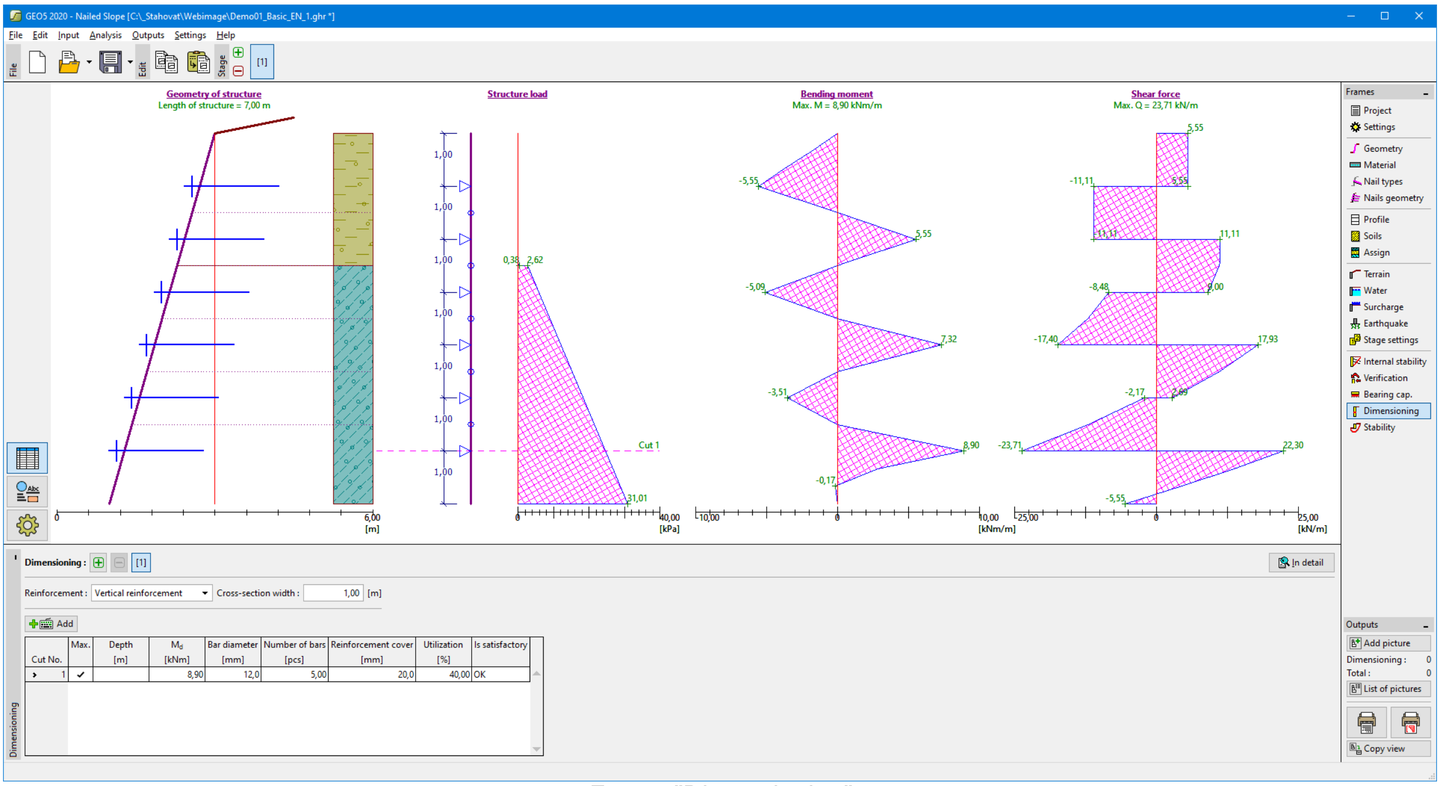
Frame "Bearing capacity"

Dimensioning - Concrete Cover

The "**Dimensioning**" frame allows for the design and verification of the reinforcement of the concrete cover. The upper part of the frame serves to choose whether the **vertical or horizontal reinforcement** and its location will be verified. The program then determines internal forces on the selected section.

The table in the bottom part of the frame serves to specify locations for the verification of the designed reinforcement depending on the input standard for dimensioning of reinforced concrete (the standard is specified in the "Materials and standards" tab). A cross-section is loaded by the bending moment in a given point. An number of the **tensile reinforcement** in the cross-section is input. If the moment is negative, the designed reinforcement is placed at the structure front face and if it is negative, then at the structure back face.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Dimensioning"

Dimensioning - Mesh

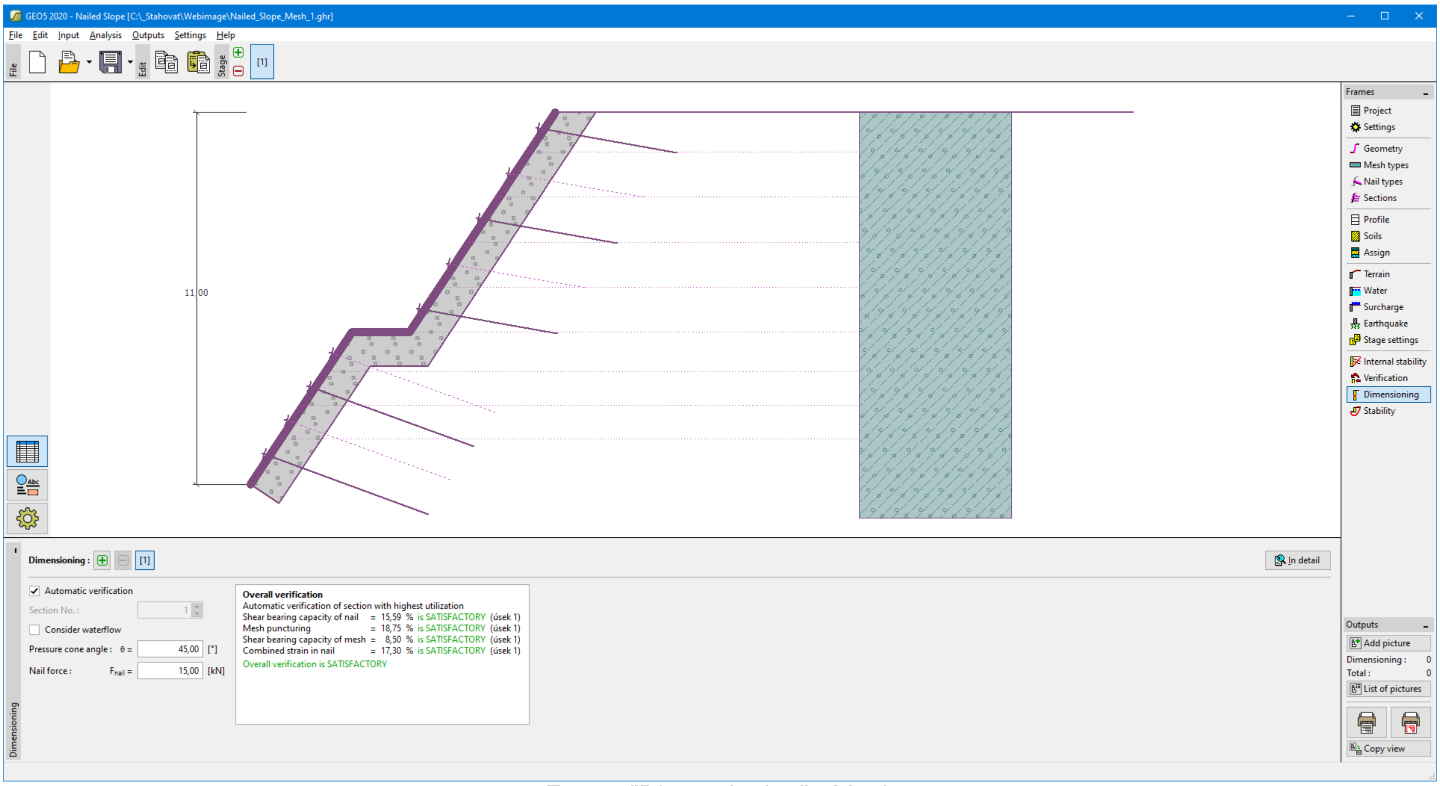
The "**Dimensioning**" frame verifies the designed structure. Several computations for one various situations can be carried out.

Verification can be done automatically for all slope sections or for each section separately.

The **nail force** is input and directly checked for bearing capacity.

It is possible to consider water flow in the weathered layer. Further, it is necessary to input pressure cone angle.

The results are displayed on the desktop. The "**In detail**" button opens the dialog window with detailed results.

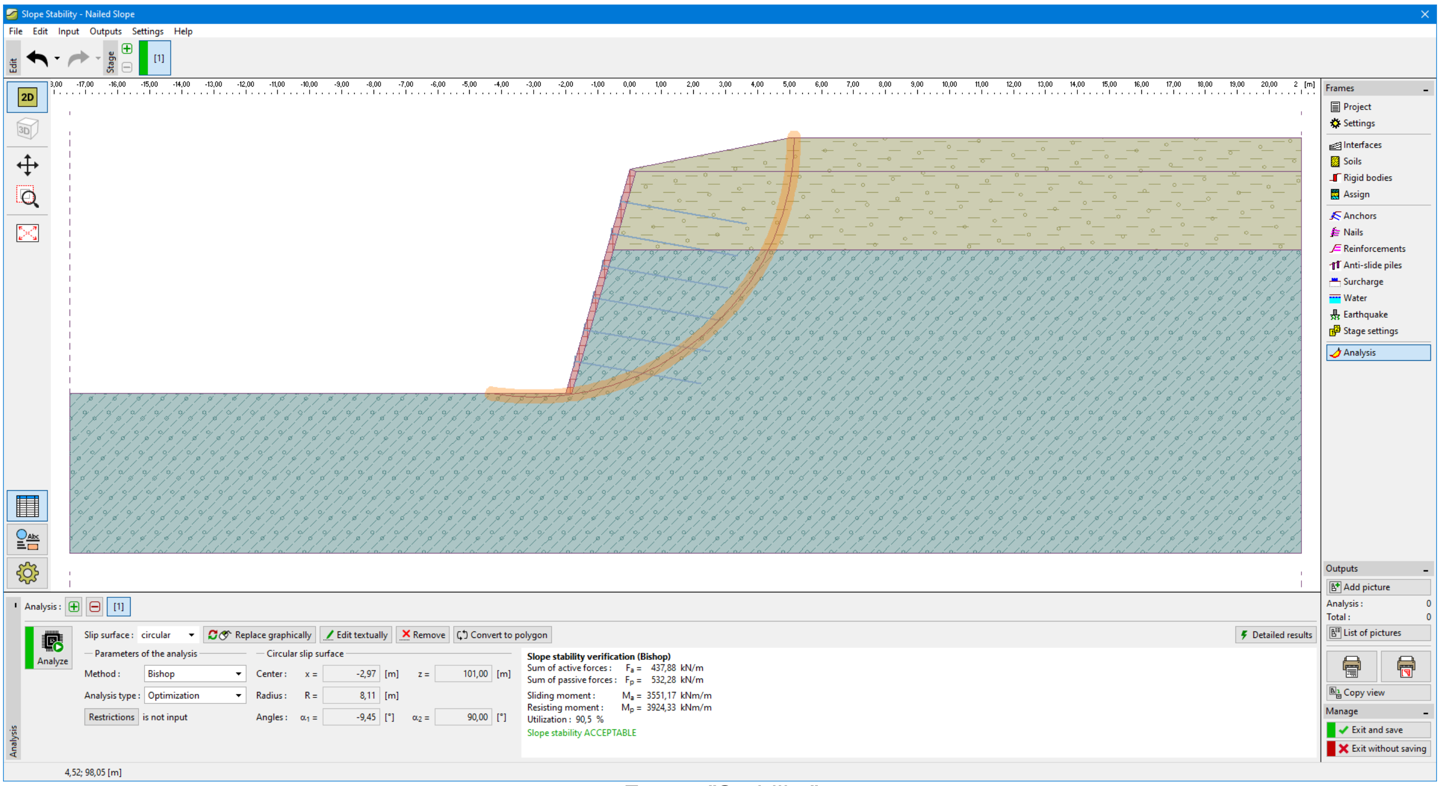


Frame "Dimensioning" - Mesh

Stability

Pressing the "**Stability**" button launches the "**Slope stability**" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program "**Slope stability**" is installed.

After completing all analyses press the "**OK**" button to leave the program - all data are then carried over to the analysis protocol of the "**Nailed Slope**" program.



Frame "Stability"

Program Redi-Rock Wall

This program is used for design and verification of retaining walls, slopes, and MSE walls. The structure of the wall is made of prefabricated Redi-Rock blocks.

The program also allows us to create a 3D model of the entire wall (in detail - Engineering Manual No. 39).

The help in the program "Redi-Rock Wall" includes the following topics:

- The input of data into individual frames:

Project	Settings	Blocks	Geometry	Footing	Profile	Soils
Assign	Backfill	Types of Reinforcements	Reinforcements	Terrain	Water	Surcharge
Front Face Resistance	Applied Forces	Earthquake	Stage Settings			

Analysis - Gravity Wall				
Verification	Dimensioning	Bearing Capacity		
Analysis - MSE Wall				
Verification	Dimensioning	Bearing Capacity	Slip on Georeinforcement	Internal Stability
Stability				

Modeling the entire wall	
Wall Geometry	Cross-section

- Standards and analysis methods

- Theory for analysis in the program "**Redi-Rock Wall**":

  Stress in Soil Body   Earth Pressures   Analysis of Walls   Dimensioning of Concrete Structures

- Outputs

- General information about the work in the User Environment of GEO5 programs

- Common

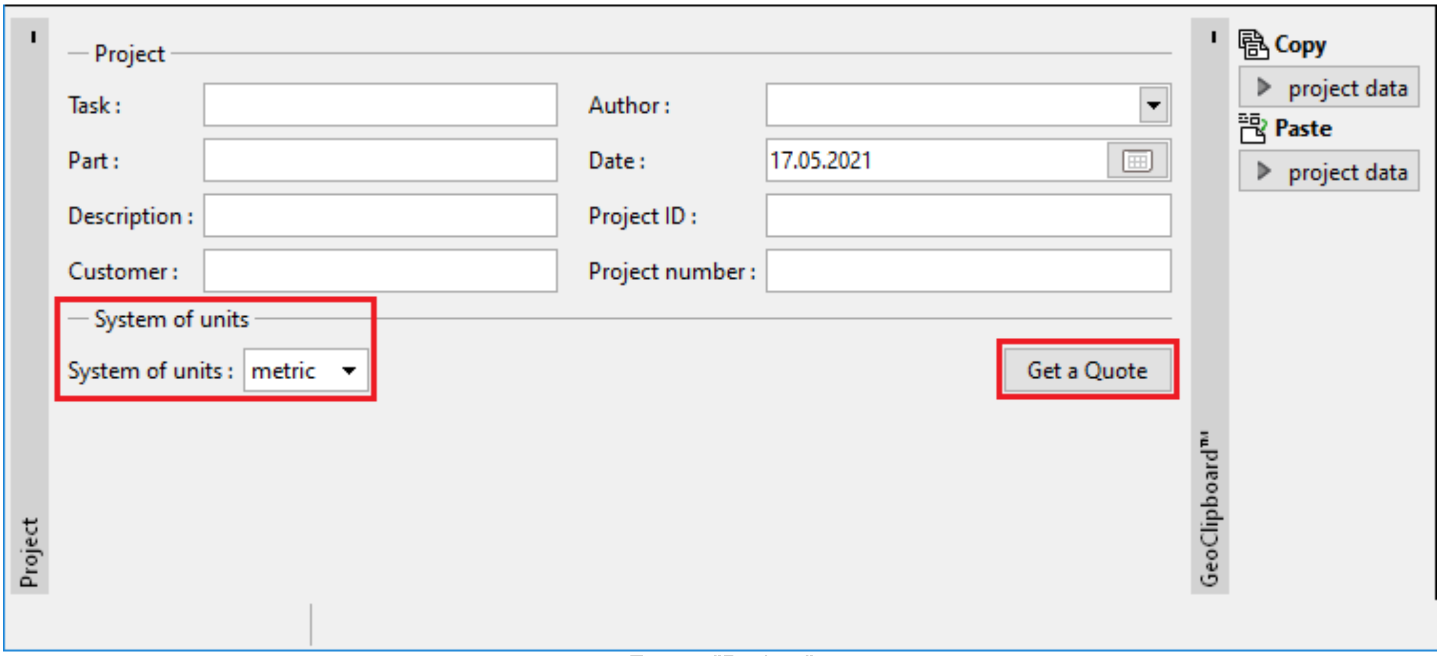
Project

The "**Project**" frame is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial).

The website of the manufacturer (https://www.redi-rock.com) can be opened by "**Get a Quote**" button.

Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The "**Settings**" frame allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

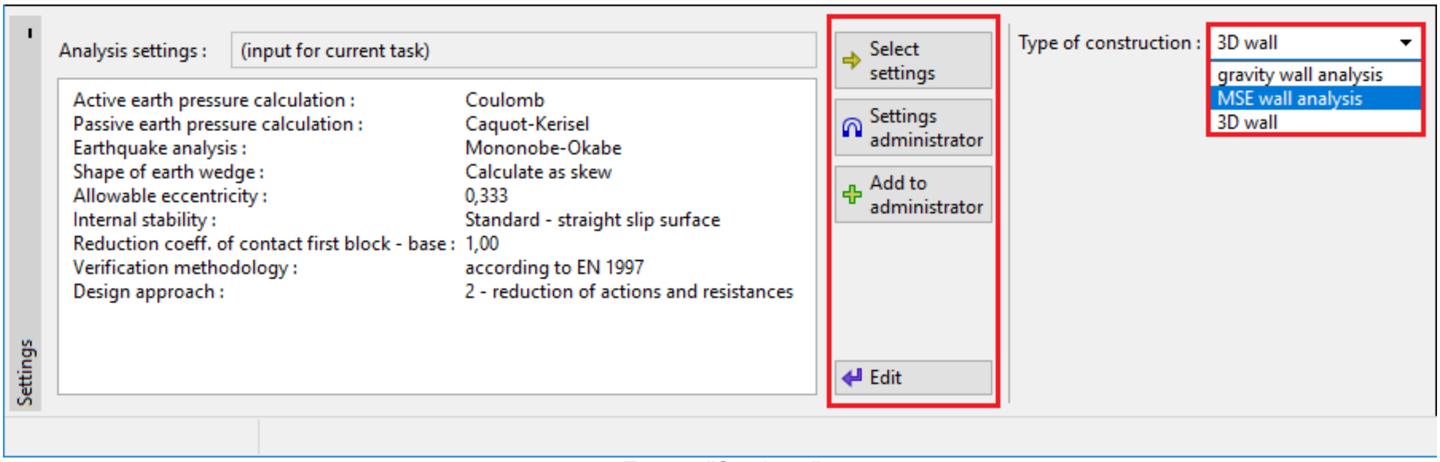
The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

Settings of analysis parameters are performed in the "Materials and standards" and "Wall analysis" tabs.

The type of structure has to be selected in this frame.

- Gravity Wall
- MSE Wall
- 3D wall (in detail - Engineering Manual No. 39).

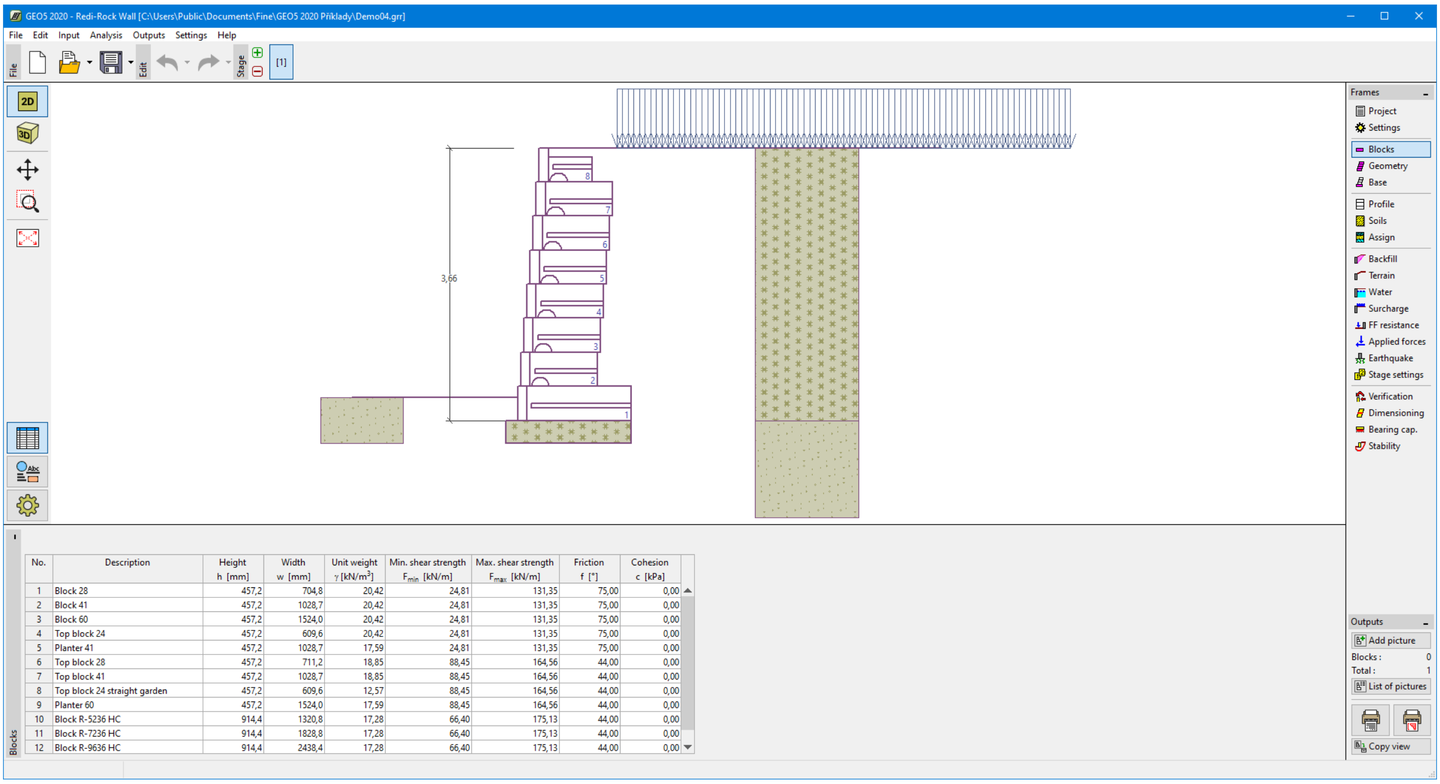


Frame "Settings"

Blocks

In the "**Blocks**" frame, the prefabricated blocks of the manufacturer are shown in the table.

Some parameters of blocks (unit weight, minimum and maximum shear strength, friction between blocks) can be edited, the dimensions of blocks cannot be changed.



Frame "Blocks"

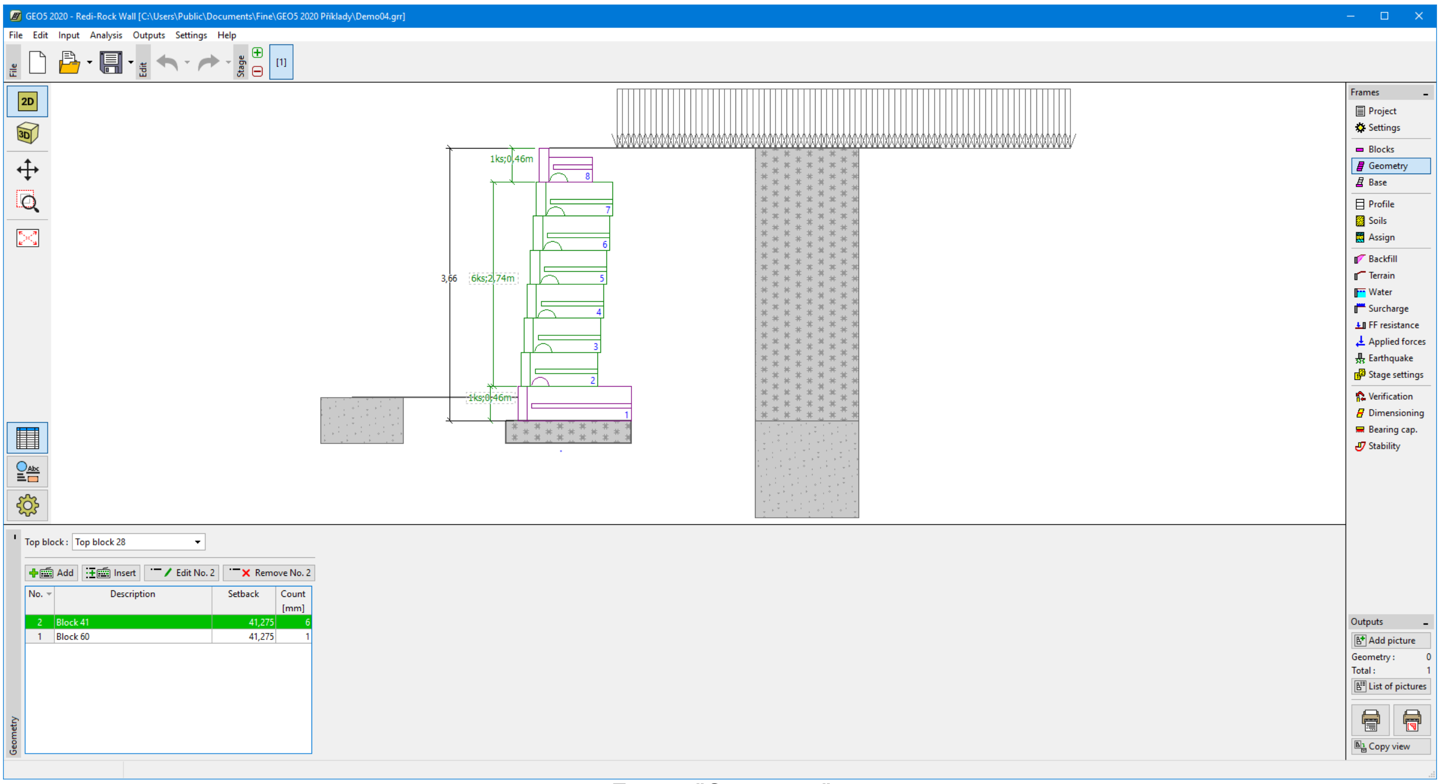
Geometry

The "**Geometry**" frame contains a table with a list of input structural precast units (blocks) of a wall (the lowest block is labeled as No. 1). Adding blocks is performed in the "**New block**" dialog window.

A group of blocks that is defined by a number of blocks and setbacks between them.

The program allows us to add (insert) another group between two already existing blocks of a structure. Inserting a new group is performed in the "**Insert**" dialog window that complies with the **"Add"** dialog window. The inserted block is ordered such to proceed the currently selected block of a structure.

The program allows us to export the geometry of a structure in the *.DXF format.



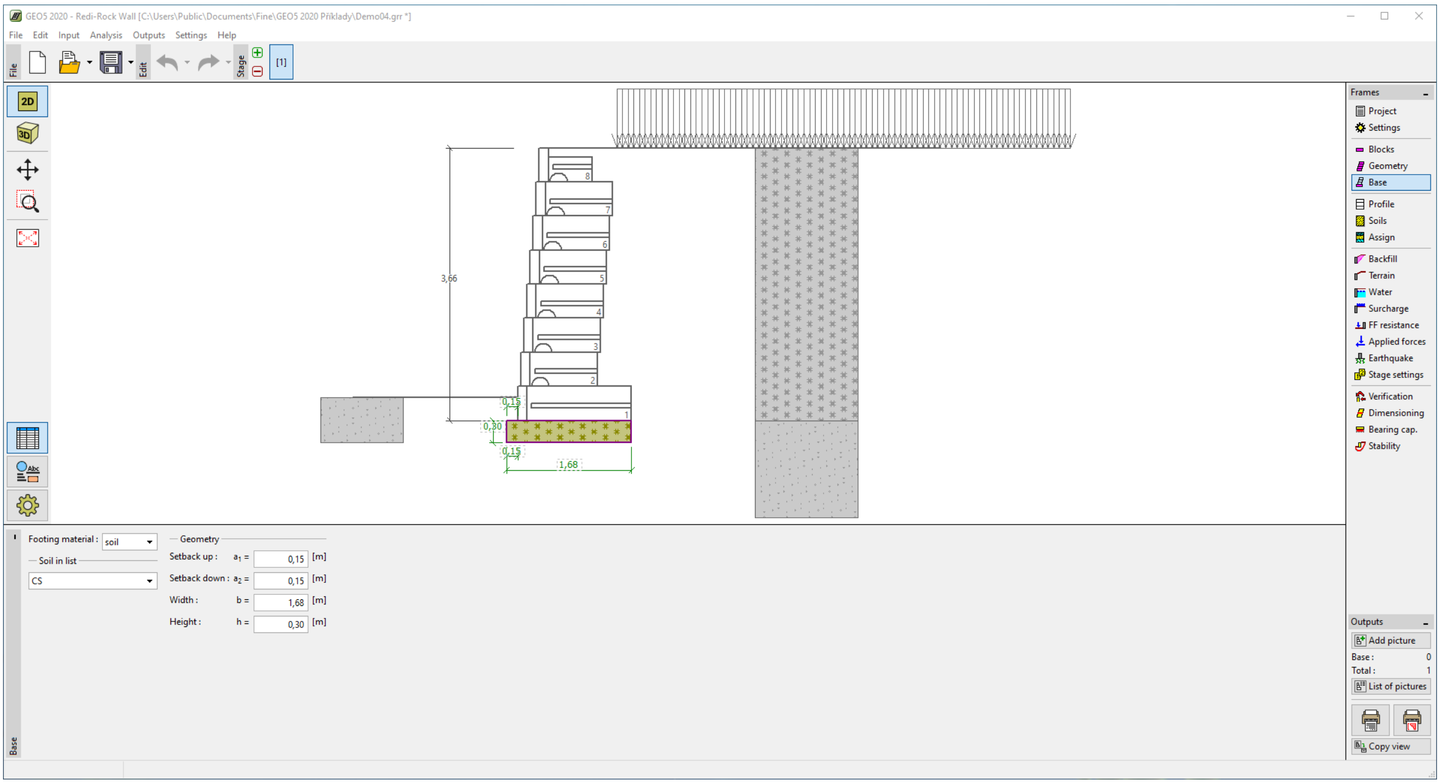
Frame "Geometry"

Footing

In The "**Footing**" frame, it is necessary to select the material of footing (soil, concrete) and input its dimensions. In the case of concrete footing, the unit weight, shear cub (key) bearing capacity and friction between a concrete footing and the first block must be also specified.

A restriction according to Fig. 4-4, page 73 of the NCMA manual is neither automatically enforced nor checked by the program.

The bearing pad must be introduced into the program such that it complies with the design criteria.



Frame "Footing"

Profile

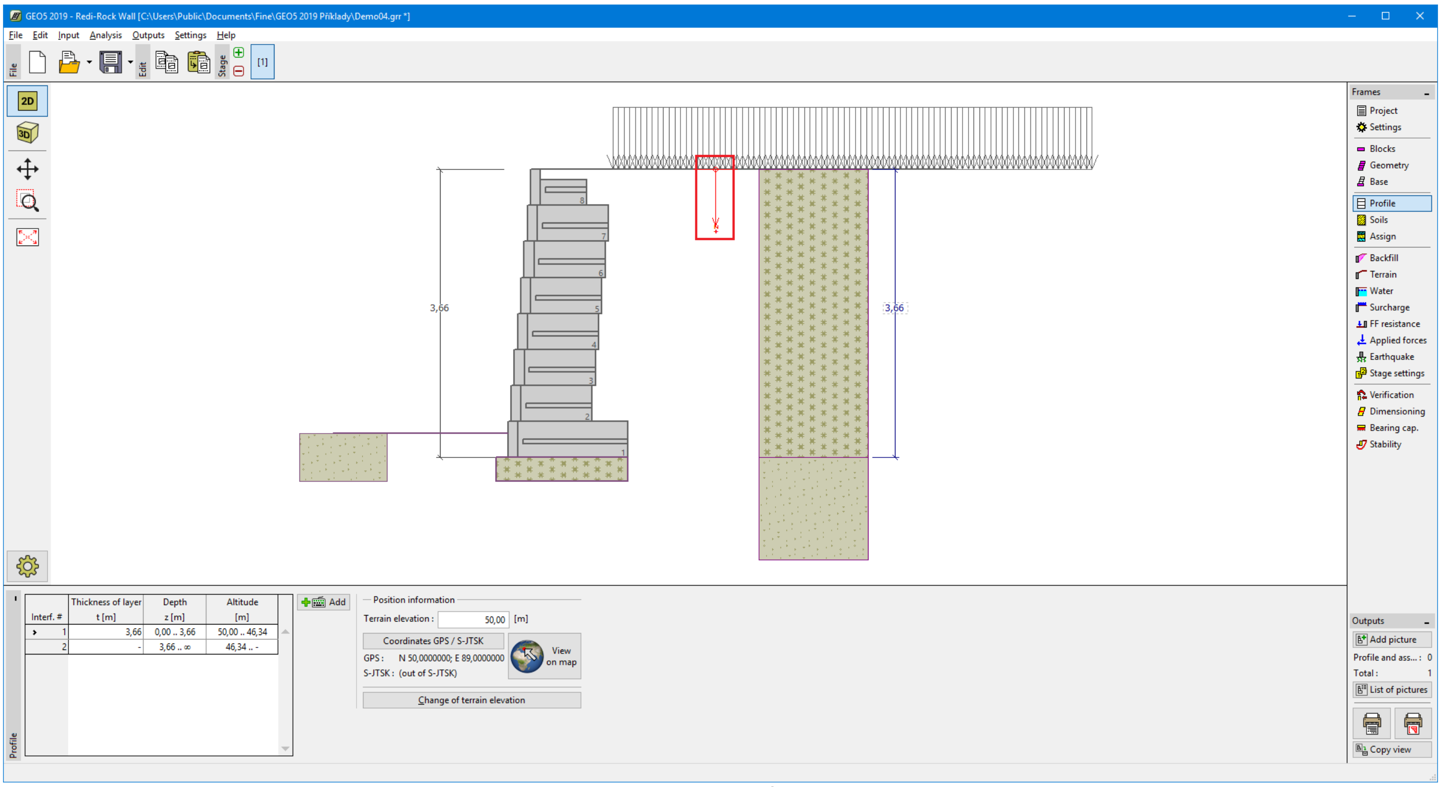
The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.



Frame "Profile"

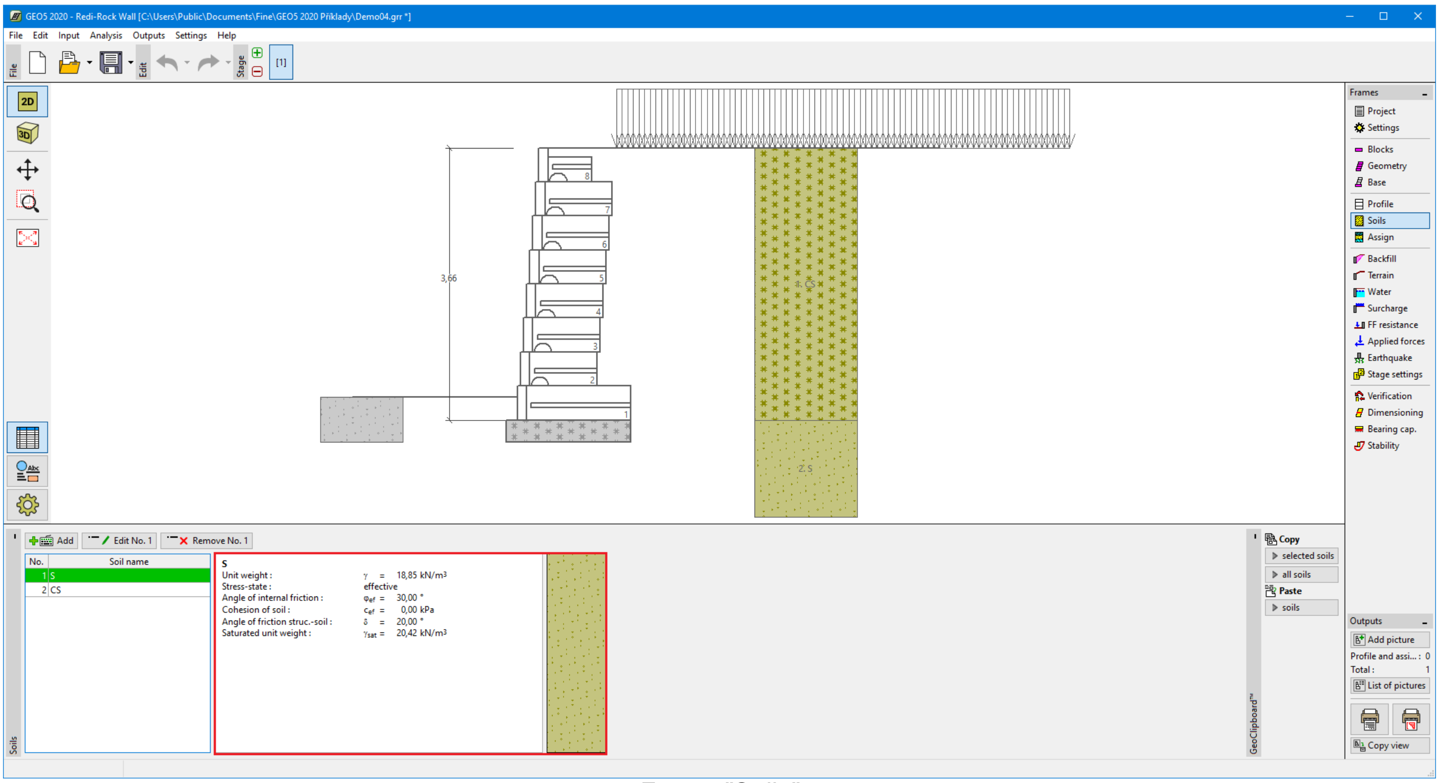
Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame.

Adding a soil is performed in the "Add soil" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

Basic data

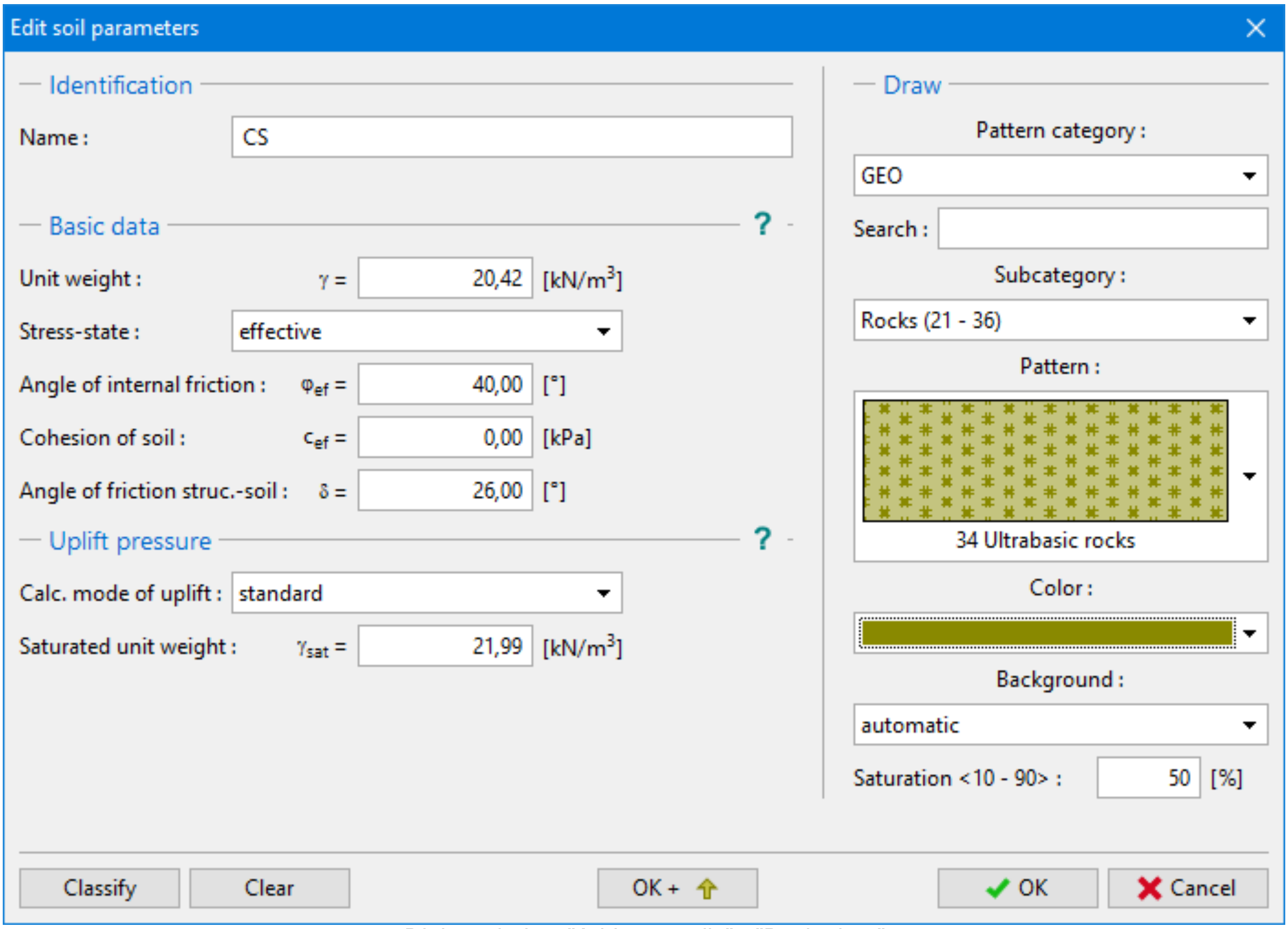
This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction, and cohesion**. The particular values are obtained from a geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit the built-in database of soils, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually.

Either **effective or total** parameters of the angle of internal friction and cohesion are specified depending on the setting in the "**Stress analysis**" combo list. Whether to use effective or total parameters depends primarily on the type of soil, type of load, structure duration, and water conditions.

For effective stress further needs to specify the angle of internal friction between the soil and structure, which depends on the structure material and the type of soil. Possible values of this parameter are listed in the table of recommended values.

For total stress further needs to specify the adhesion of soil to the structure face a .

The associated theory is described in detail in the chapter "Earth pressures".

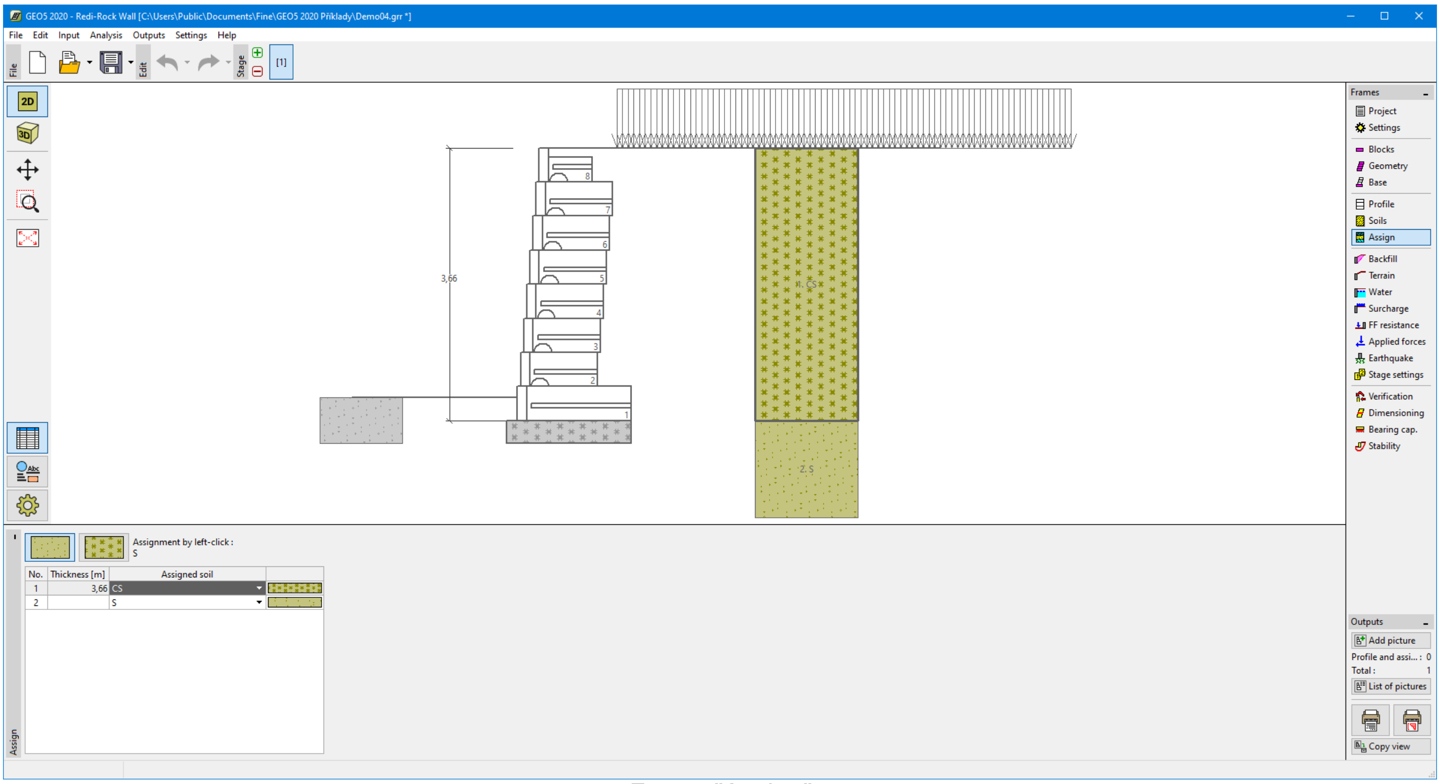


Dialog window "Add new soils" - "Basic data"

Assign

The "**Assign**" frame contains a list of profile layers and assigned soils. The list of soils is graphically represented using buttons in the bar above the table or is accessible from a combo list for each profile layer.

The procedure to assign soil to a layer is described in detail herein.

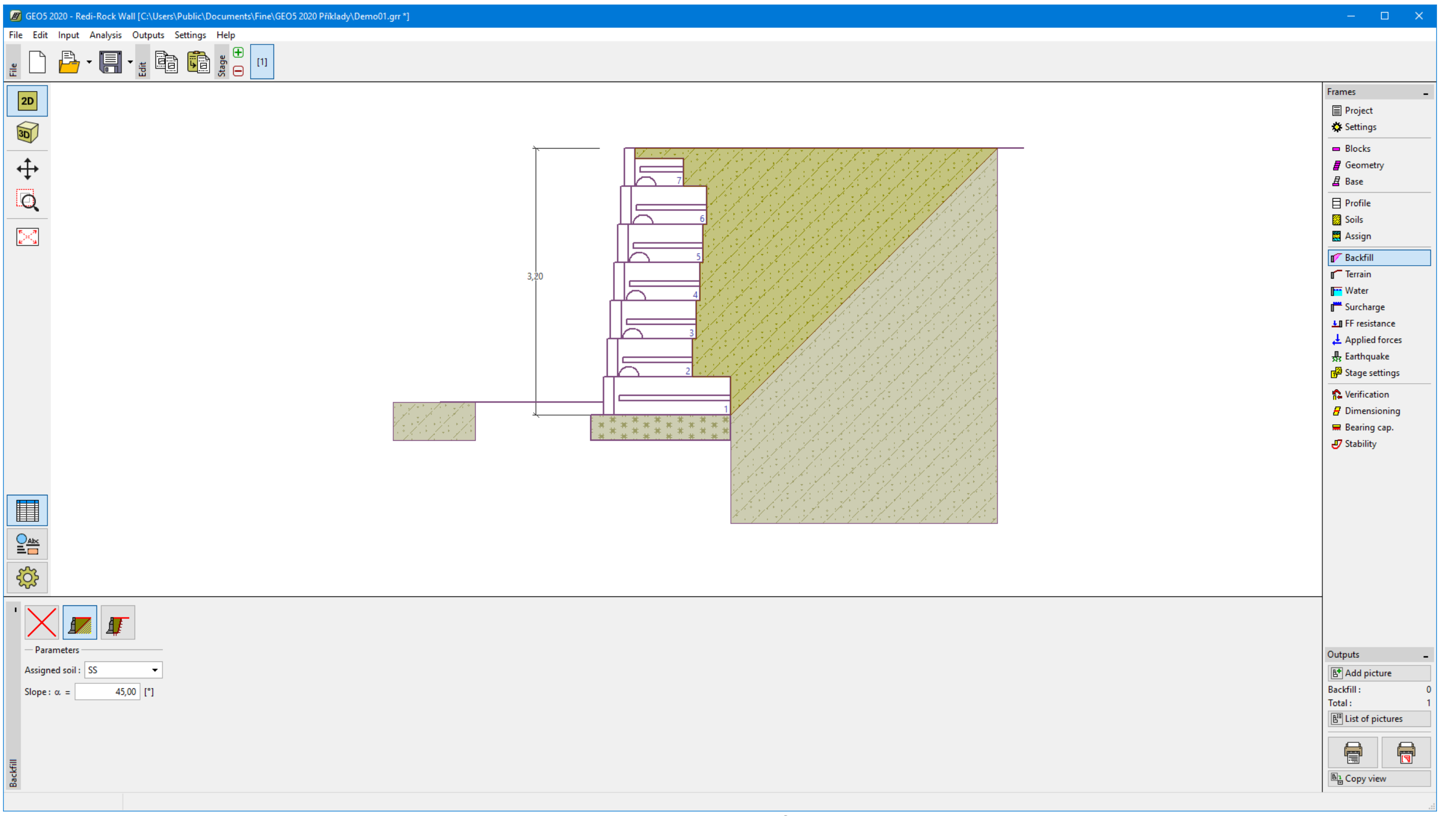


Frame "Assign"

Backfill

The frame "**Backfill**" allows the selection of backfills behind the structure.

Analysis of earth pressures with the influence of backfill is described in the theoretical part of the help: "Influence of Backfill".



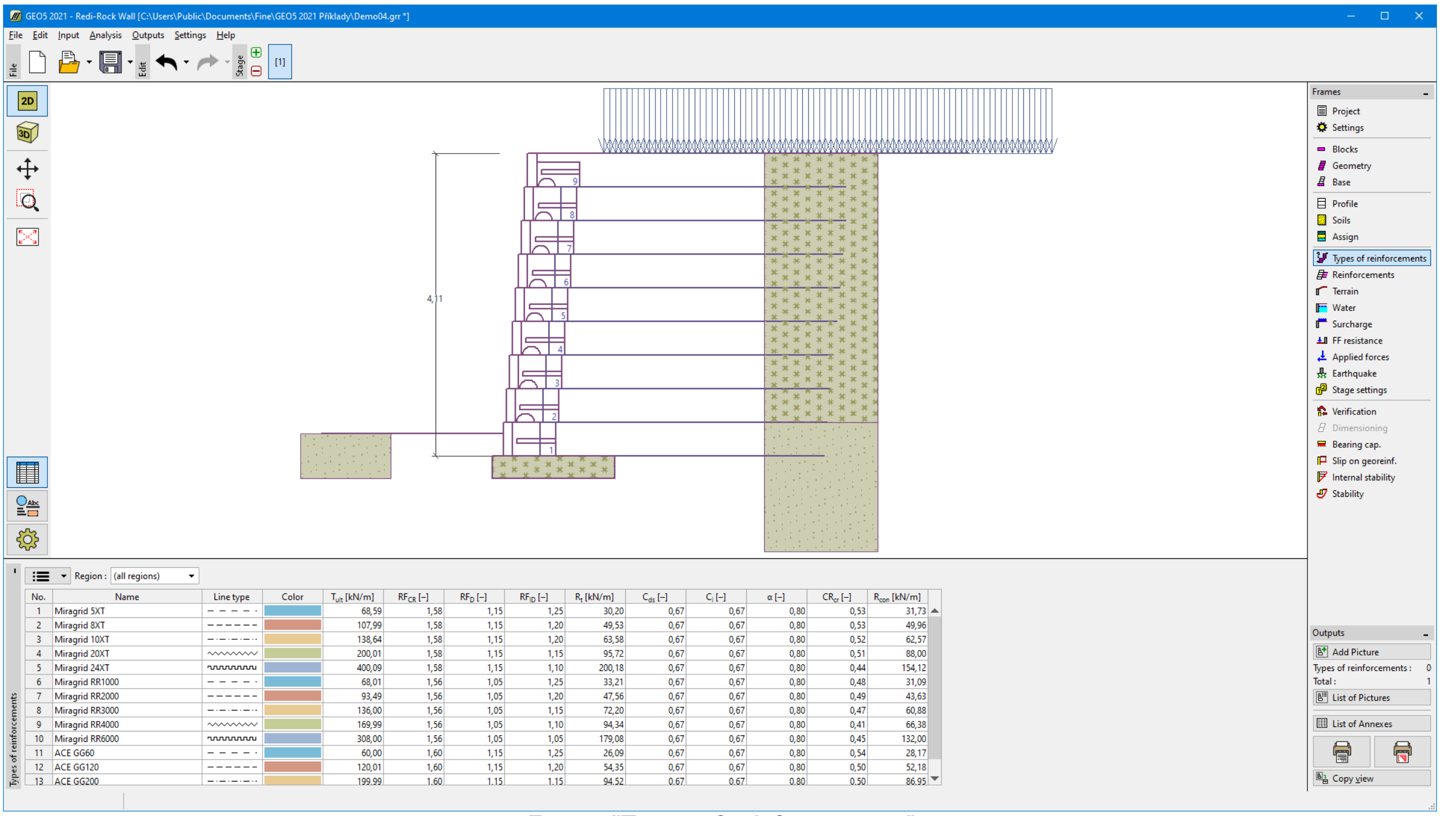
Frame "Backfill"

Types of Reinforcements

The "**Types of reinforcements**" frame contains a table with the list of input geo-reinforcements and their parameters (short-term bearing capacity, long-term bearing capacity, and coefficients of interaction).

Editing reinforcement is carried in the "Edit type of reinforcement" dialog window.

The "**Types of reinforcements**" frame is available just in case of MSE Wall calculation. The type of structure is selected in the frame "Settings".



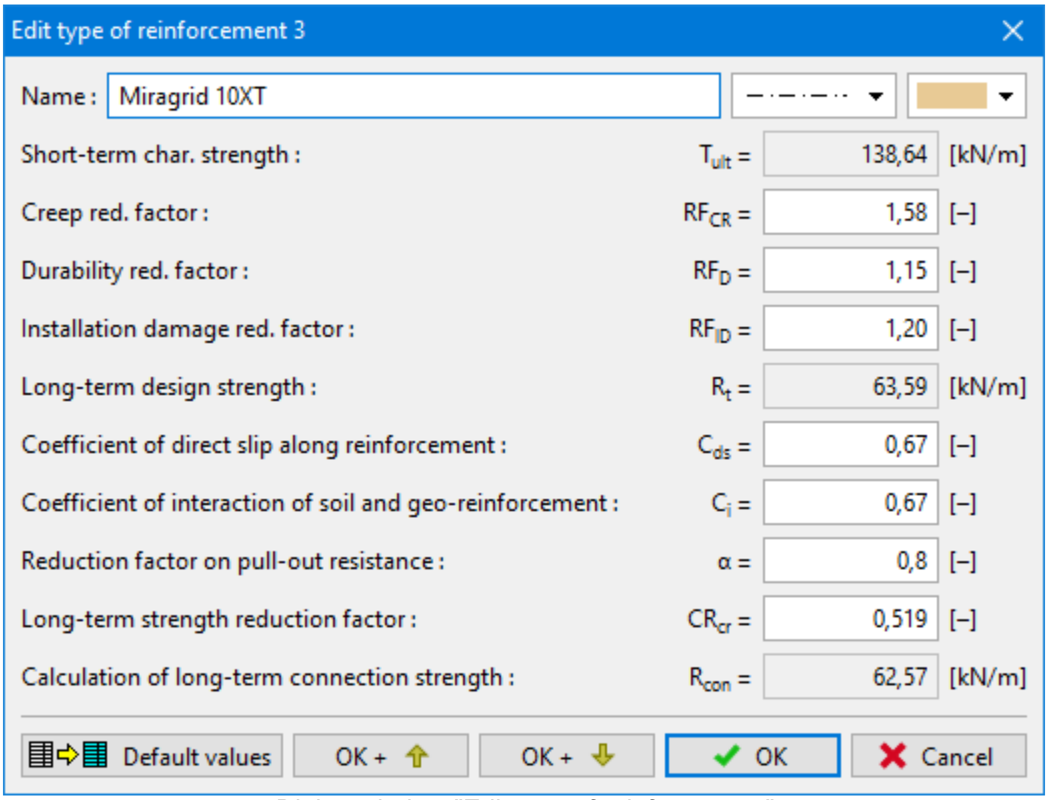
Frame "Types of reinforcements"

Edit Type of Reinforcement

The **"Edit type of reinforcement"** dialog window contains following items:

Short-term characteristic strength	• the value cannot be adjusted (given by the manufacturer)
Reduction factors	• the values of factors reducing a short-term tensile strength
Long-term design strength	• calculated value of a long-term tensile strength
Coefficient for slip resistance	• coefficient of **direct slip along reinforcement**
Coefficients for pull out resistance	• coefficient of **interaction of soil and geo-reinforcement**, and **scale correction factor**
Long-term strength reduction factor	• the value of factor reducing the **connection strength**
Connection strength	• connection strength is calculated according to: $R_{con} = T_{ult} \, . \, CR_{cr} \, / \, RF_d$

The **"Default values"** button at the bottom of the dialog window sets all coefficients to their original values.



Dialog window "Edit type of reinforcement"

Reinforcements

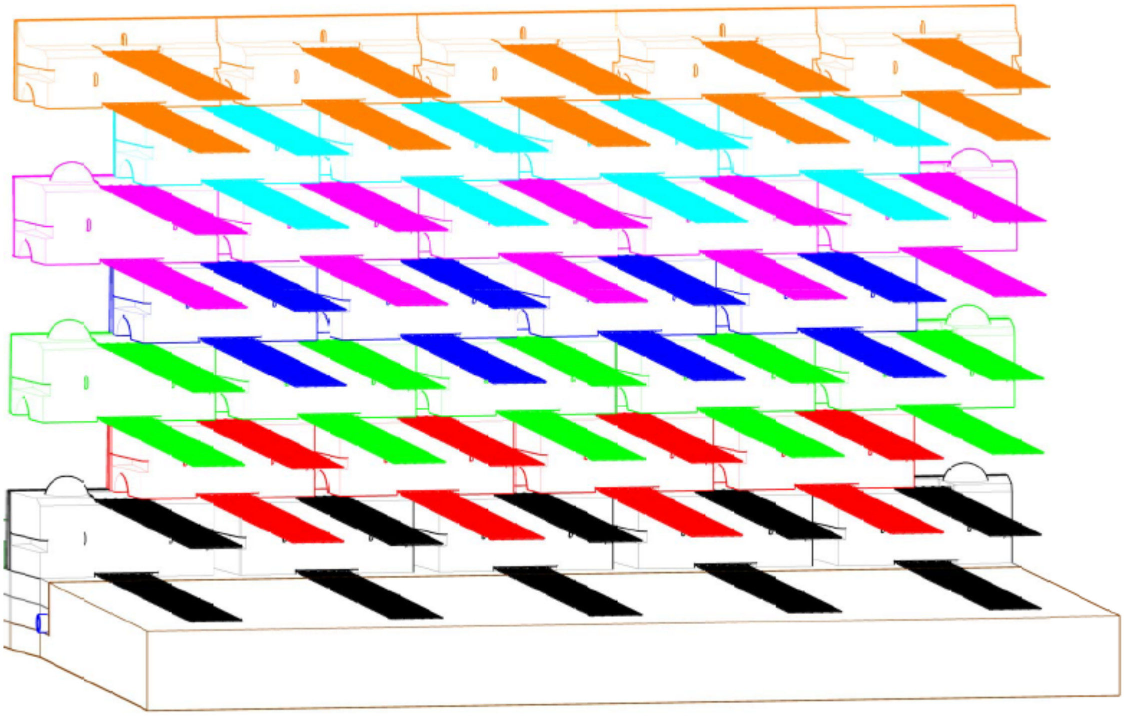
In the "**Reinforcement**" frame, parameters of reinforcement are specified.

Input mode

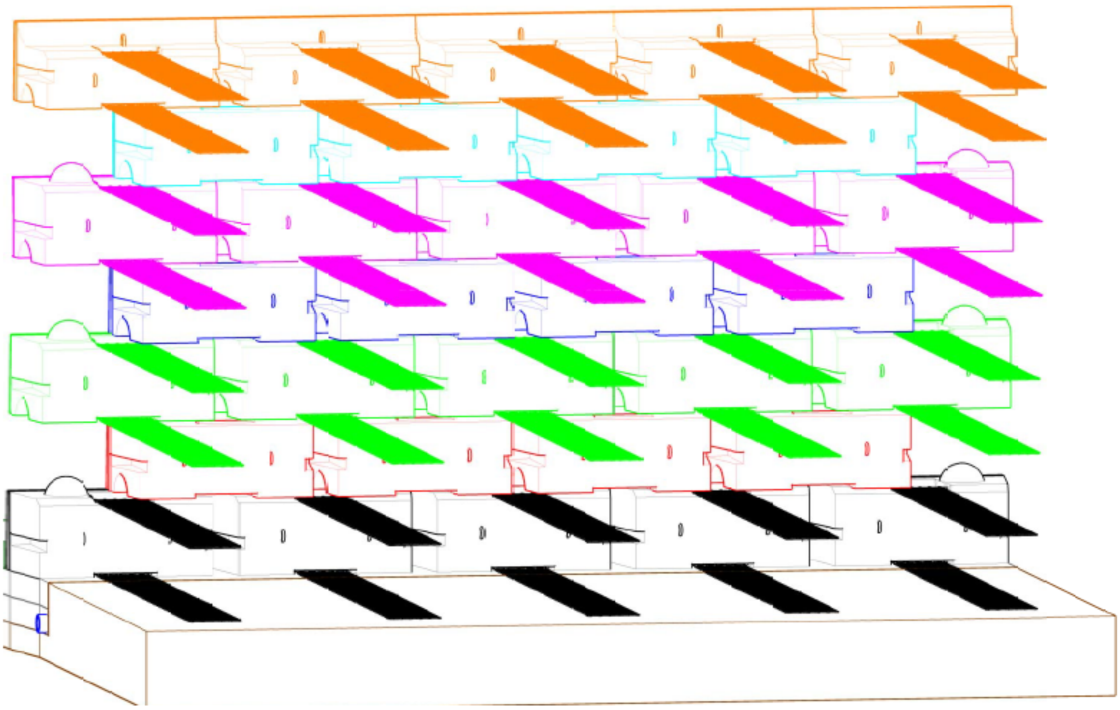
- 1 reinforcement type - the wall is reinforced by one type of reinforcement. The **type of reinforcement** is selected in the combo list. The reinforcements are generated from the bottom row of blocks to the row, which is selected in the combo list.
- 2 reinforcement type - the wall is reinforced by two types of geo-reinforcement. There is different reinforcement in the upper and bottom parts of the wall. The input is the same as in the case of one type of reinforcement.
- table of reinforcements - the reinforcements are specified for each row of blocks

Reinf. installation

- in every row of blocks (50%) - the reinforcements are placed in every block

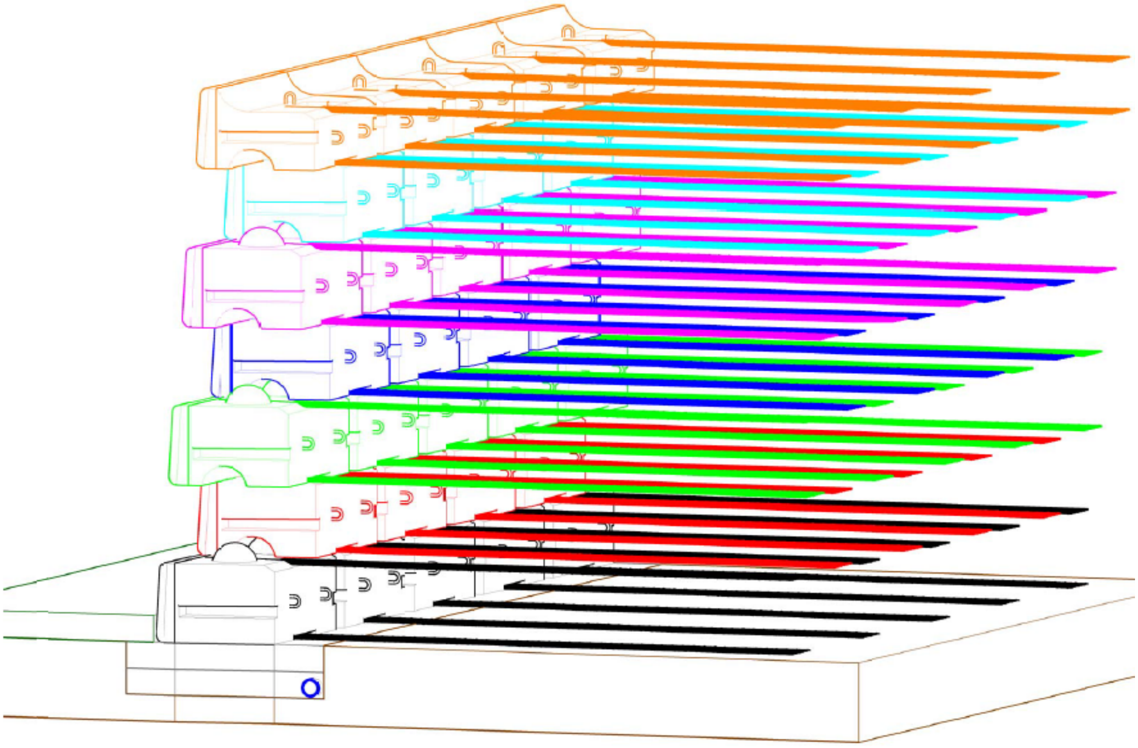


- in every other row of blocks (25%) - the reinforcements are placed in every other block

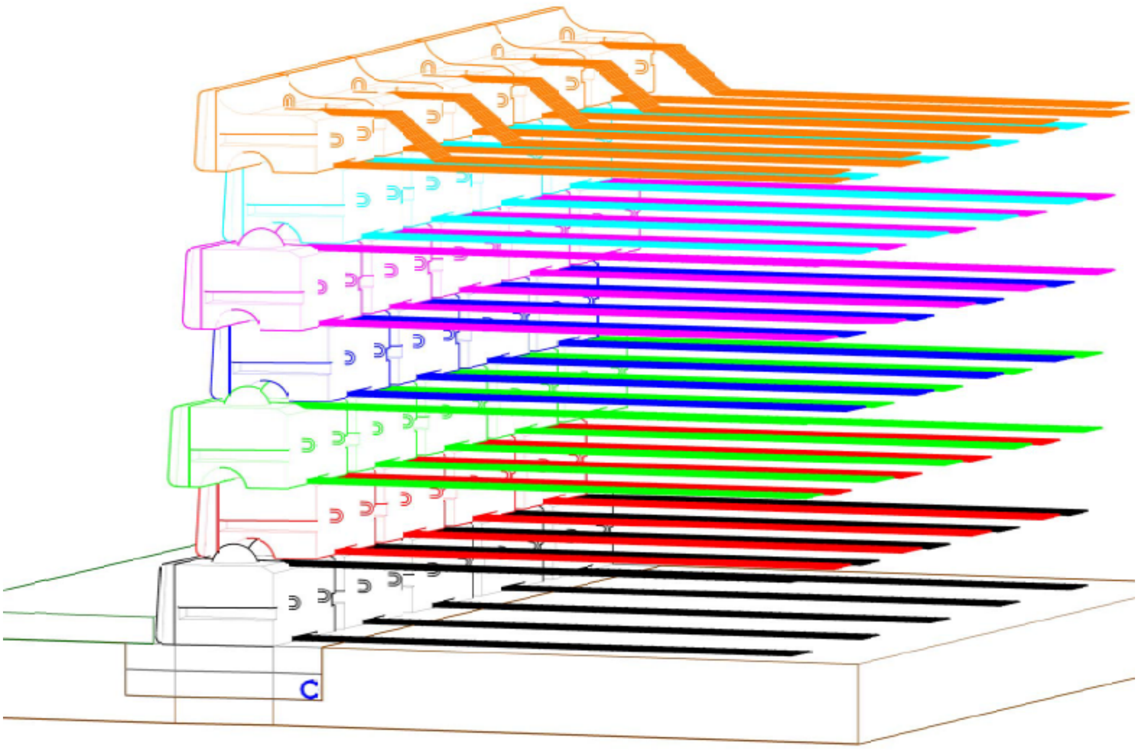


Top reinforcement

- straight (25%) - the reinforcements in the upper row are placed in every other block



- anchored tail - upper part of reinforcements in the upper row of blocks is directed in the angle to its bottom

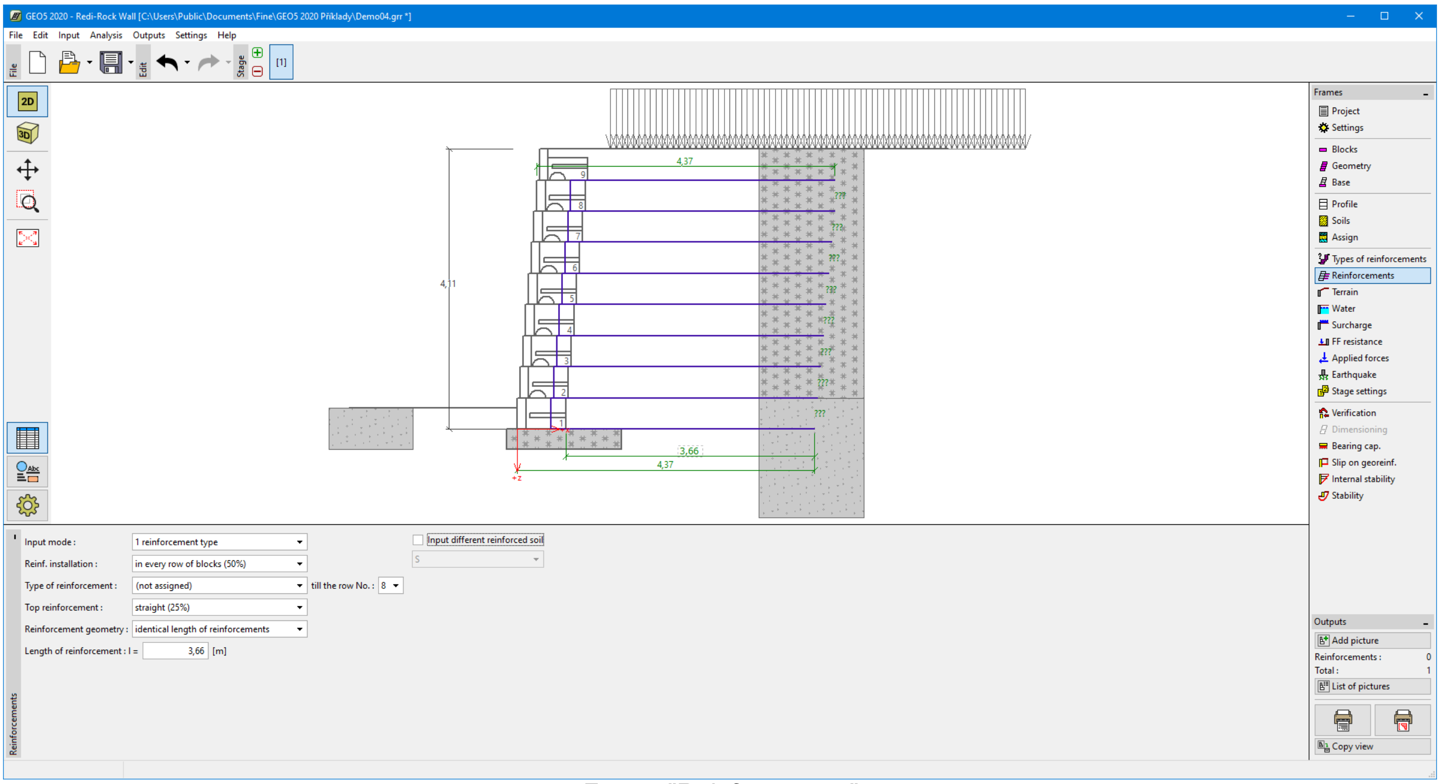


Reinforcement geometry

- identical length of reinforcements - all reinforcements have same length l
- the identical ending of reinforcements - all reinforcements end in the same distance from the wall l_k

If the soil between the reinforcements is different then soil assigned to the geological profile, then the program allows us to specify this soil by checking the option "**Input different reinforced soil**". Subsequently, in combo list user selects the type of soil (the combo list contains soils introduced in the frame "Soils").

The "**Types of reinforcements**" frame is available just in case of MSE Wall calculation. The type of structure is selected in the frame "Settings".



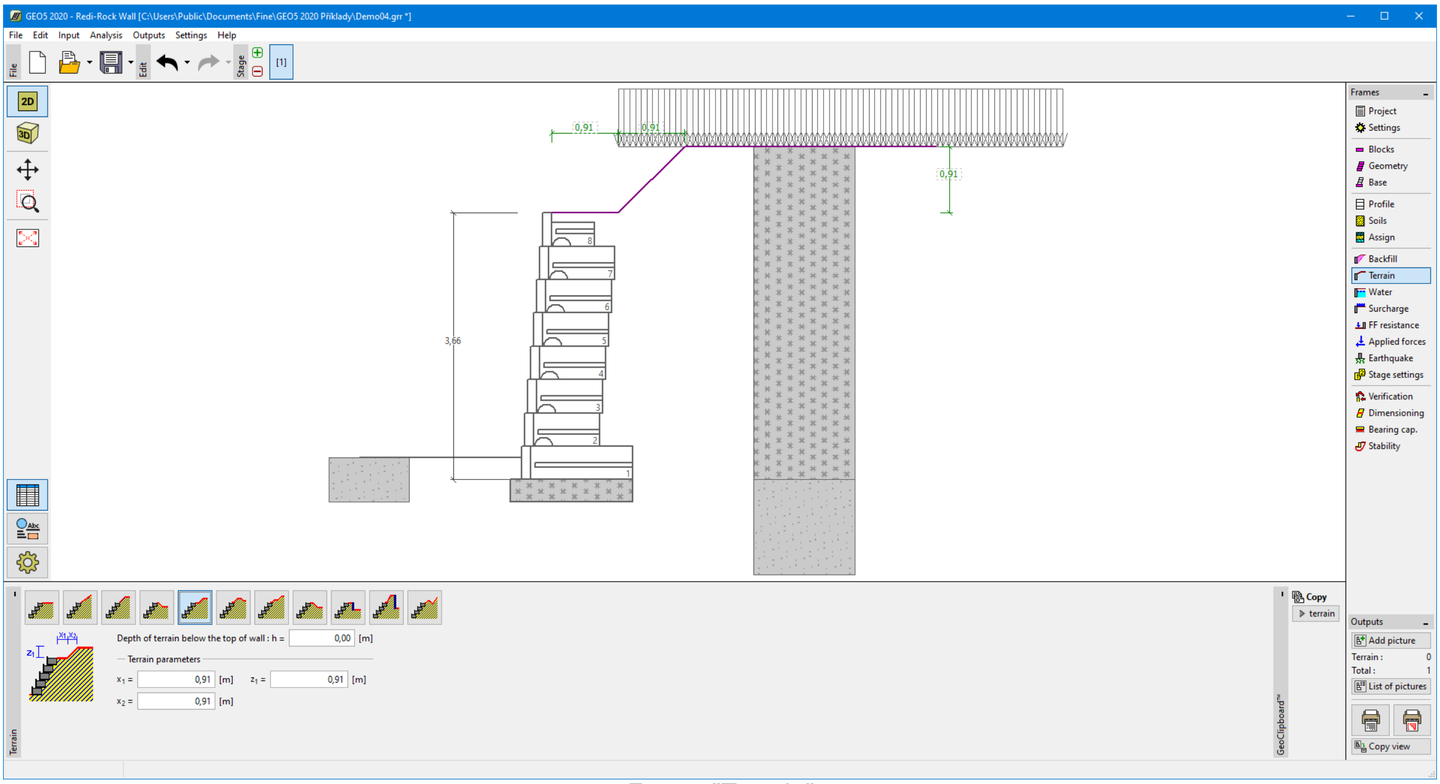
Frame "Reinforcement"

Terrain

The "**Terrain**" frame allows, by pressing the button, to specify the terrain shape. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case, the frame contains a table with a list of terrain points. The first point with coordinates [0; 0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "Distribution of earth pressures for broken terrain".



Frame "Terrain"

Water

The "**Water**" frame allows, by pressing the button, for selecting the type of water. The selected type with a graphic hint of input values is displayed in the left part of the frame. Water parameters (h_1, h_2 ...) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

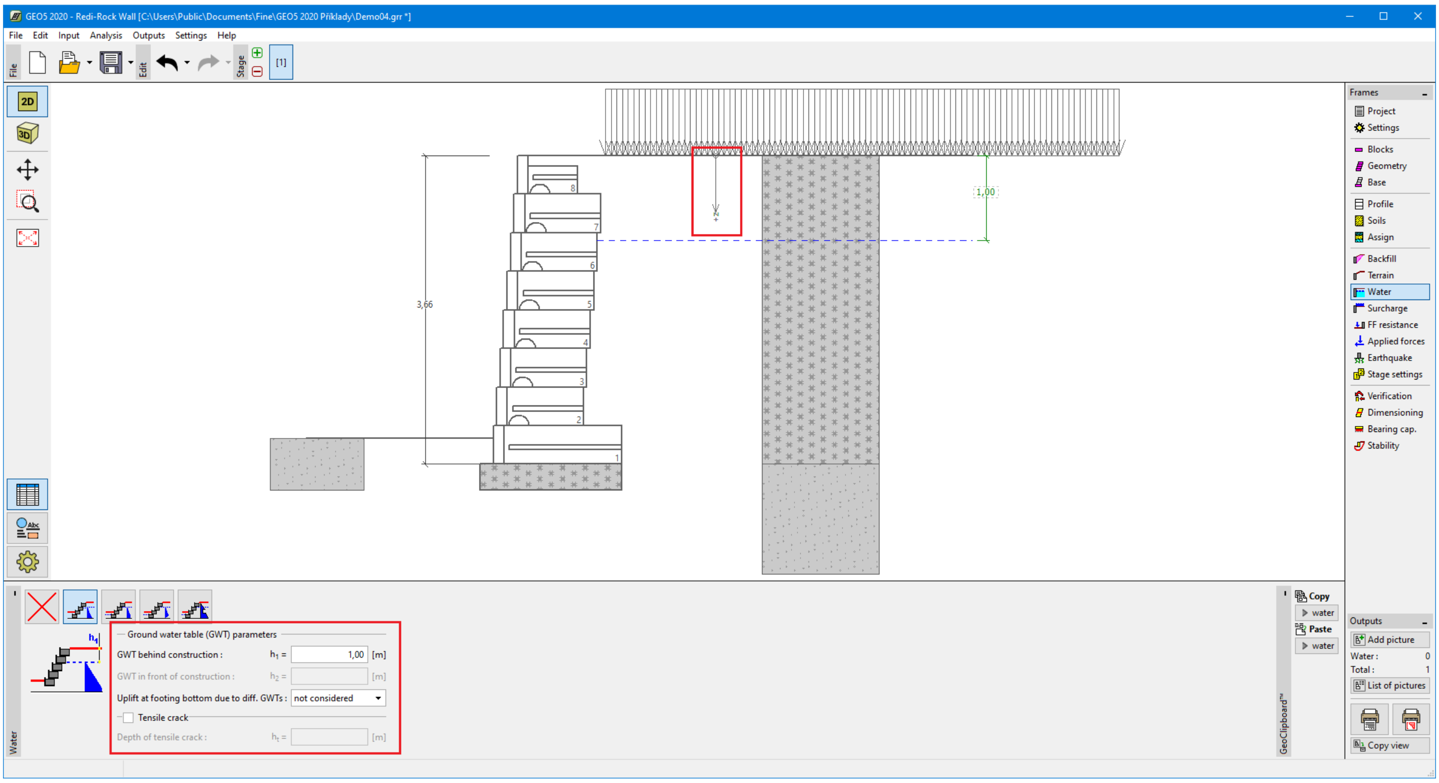
The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be linear, parabolic, or it may not be considered at all. When verifying the wall, the uplift pressure in foundation joint due to different water tables is introduced in terms of a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tabs "**In front of structure**" and "**Behind structure**" appear with tables. The table contains the values of pore pressure in front of the structure or behind it.

The groundwater table can also be specified **above the structure** or earth profile, respectively - in such a case, the depth of water is input with a negative value.

Analysis of earth pressures with the influence of water is described in the theoretical part of the help, chapter "Influence of water".

The program further allows for specifying a depth of tensile cracks filled with water.



Frame "Water"

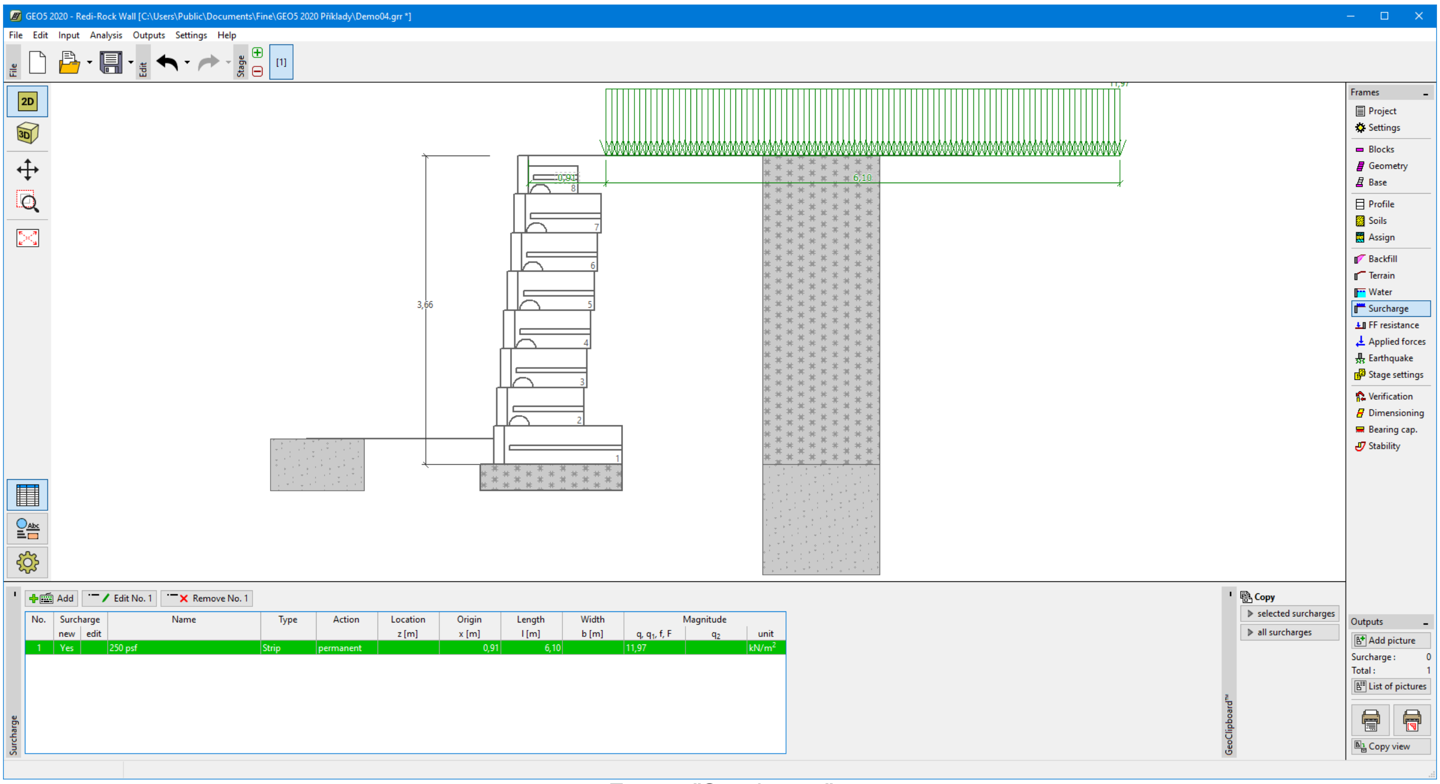
Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The z -coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case when the surcharge is found out of the terrain the program prompts an error message before calculation.

Either **permanent**, **variable,** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with a favorable effect is not considered in the analysis.

Analysis of earth pressures due to surcharges is described in the theoretical part of the help, chapter "Influence of surcharge".



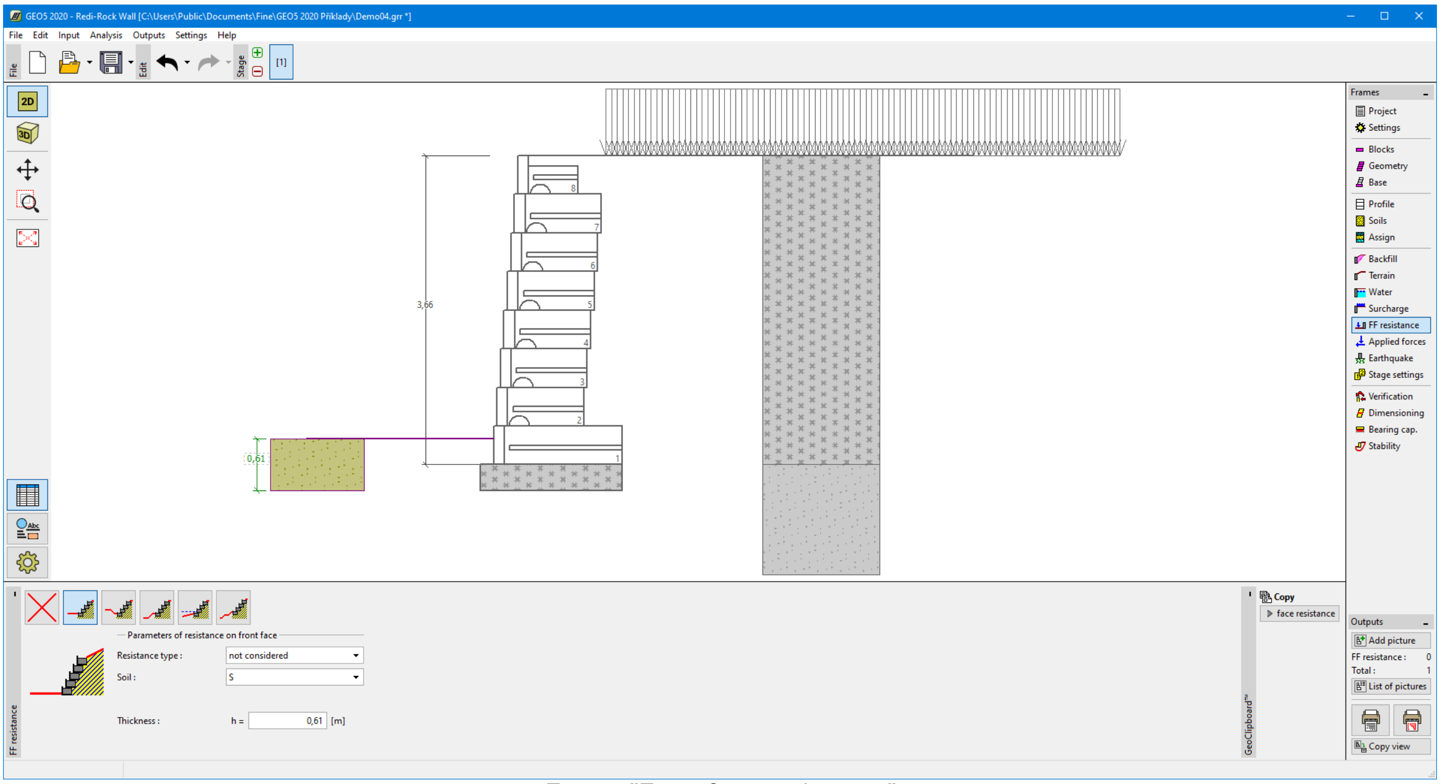
Frame "Surcharge"

Front Face Resistance

The "**Front face resistance**" frame allows us to specify the terrain shape in front of the structure and parameters of front face resistance. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop using active dimensions.

Combo lists in the frame allow selecting the type of resistance and a soil (from the soils introduced in the "Soils" frame). The terrain surcharge magnitude and soil thickness (above the wall lowest points) can also be specified.

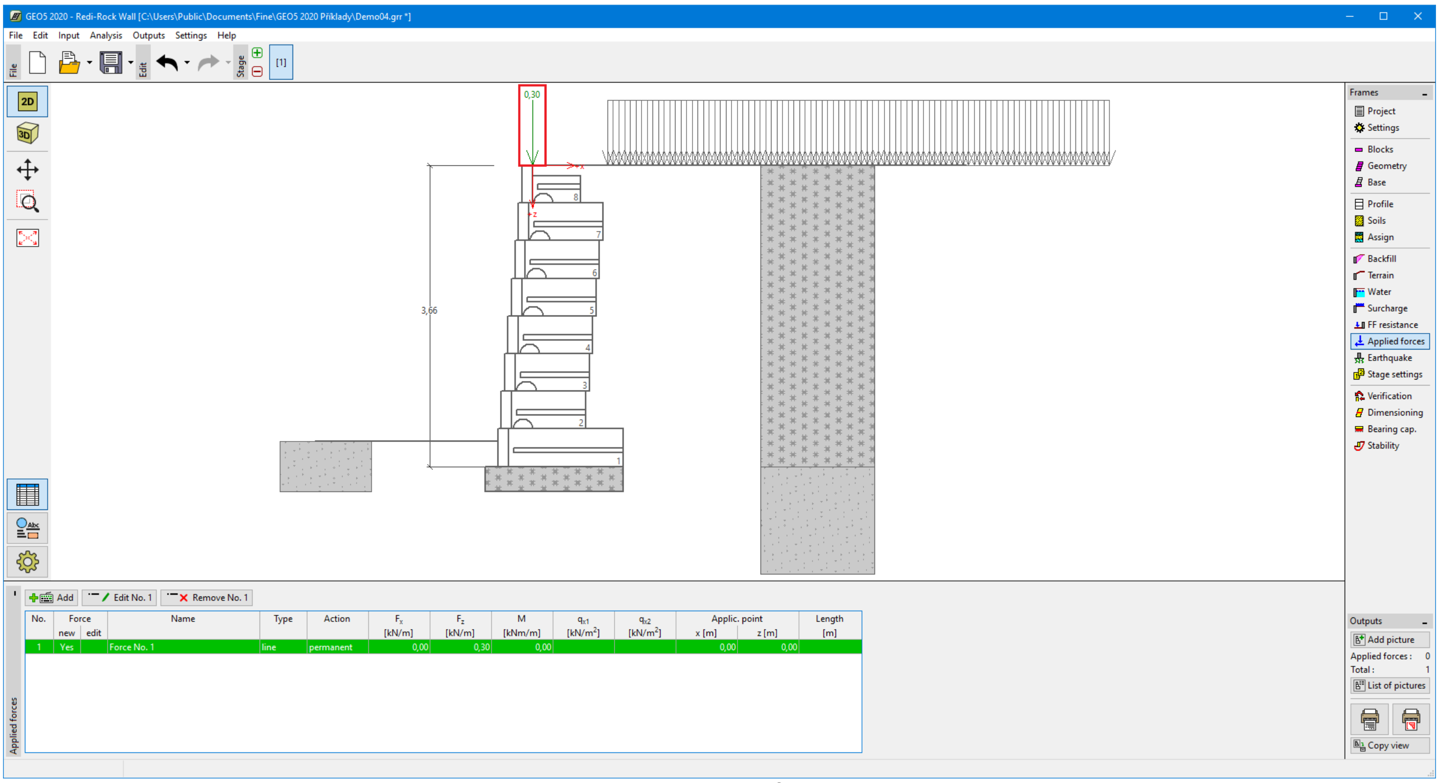
The resistance on a structure front face can be specified as the pressure at rest, passive pressure, or reduced passive pressure. The resulting force due to the reduced passive pressure is calculated as a resultant force caused by pressure at rest and passive pressure multiplied by corresponding coefficients.



Frame "Front face resistance"

Applied Forces

The "**Applied forces**" frame contains a table with a list of forces acting on a structure. Adding forces is performed in the "**New force**" dialog window. The input forces can also be edited on the desktop with the help of active objects.



Frame "Applied forces"

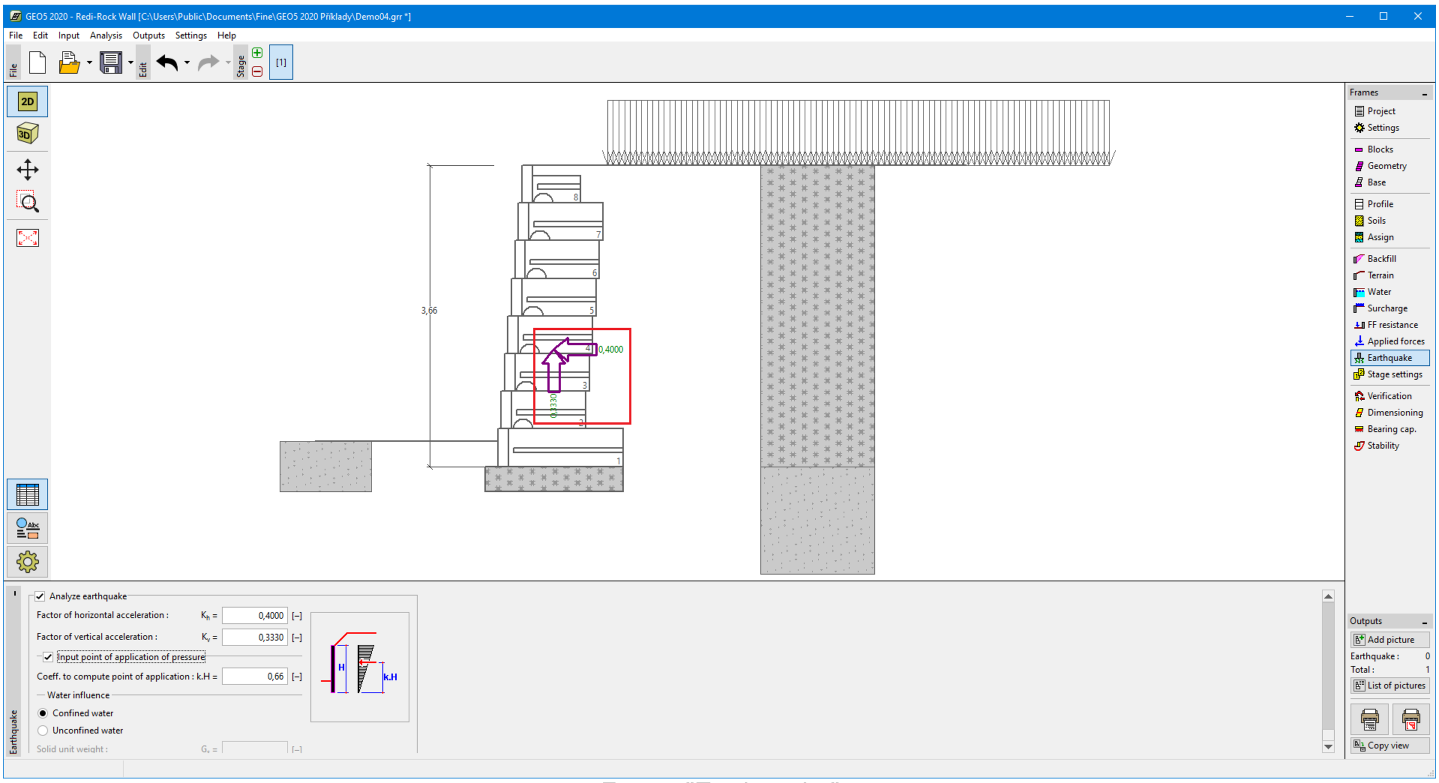
Earthquake

The "**Earthquake**" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements, the coefficients k_h and k_v can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for an earthquake is described in the theoretical part of the help, chapter "Influence earthquake".

For the LRFD Verification methodology, it is possible to define coefficients for seismic combinations according to the AASHTO.



Frame "Earthquake"

Stage Settings

The frame "**Stage settings**" serves to input settings valid for a given construction stage.

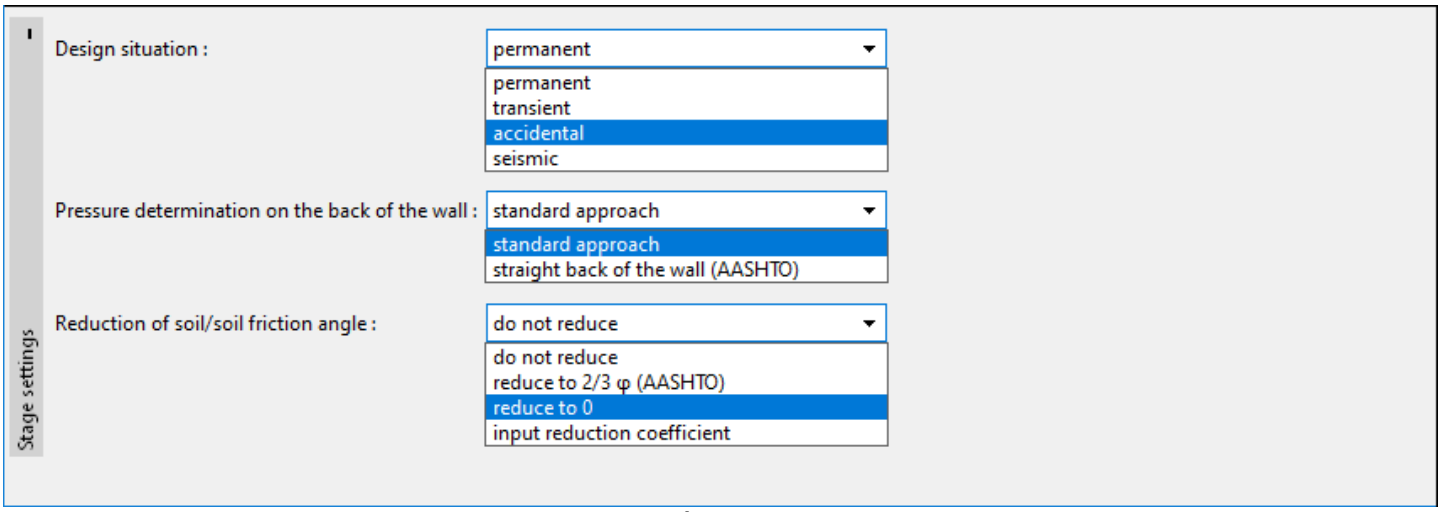
The selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology.

It is also possible to consider the simplified straight back of the wall for pressure calculation.

The reduction of soil/soil friction angle can be considered in one of following ways:

- **do not reduce**
- **reduce to 2/3φ (AASHTO)**
- **reduce to 0**
- **input reduction coefficient**

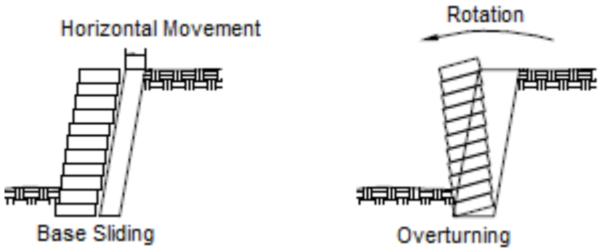


Frame "Stage settings"

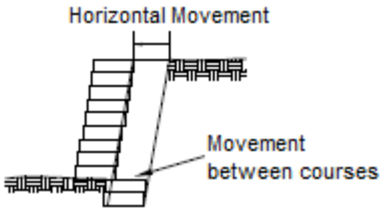
Analysis - Gravity Wall

Verification of **Gravity wall** made of RediRock blocks is performed in these frames:

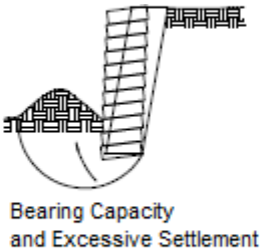
- Verification - verification of a wall against **overturning** and **slip**



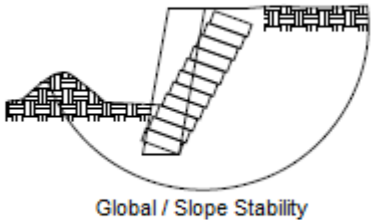
- Dimensioning - verification of joints between individual blocks of a wall against **overturning** and **slip**



- Bearing capacity - verification of the foundation soil bearing capacity



- Stability - verification of the overall stability of the analyzed structure



Pictures from: *Design Manual for Segmental Retaining Walls (NCMA, 3rd Edition)*

Verification

The frame "**Verification**" shows the analysis results. Several computations can be carried out for a single task.

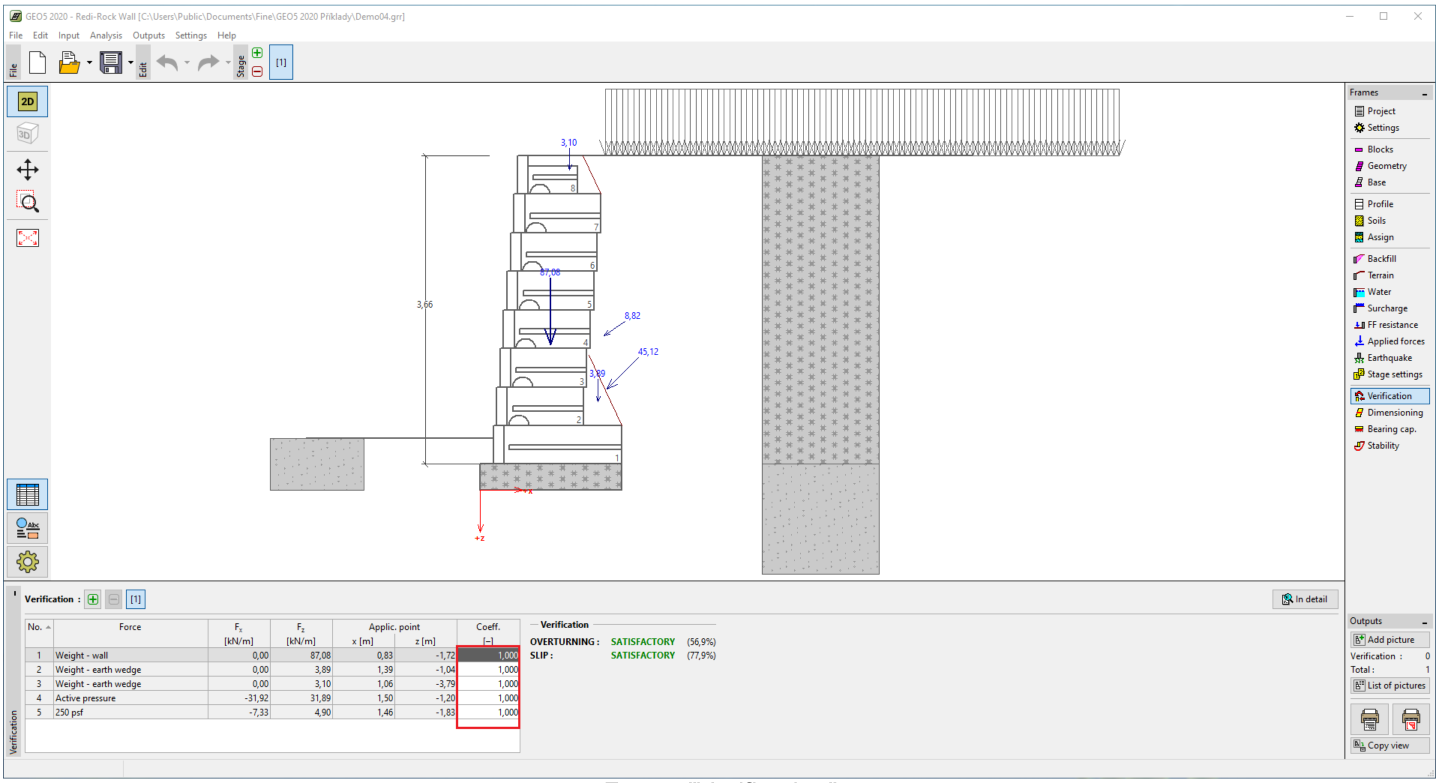
The frame appearance is adjusted based on the selected verification methodology.

- Verification according to the factor of safety or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows for specifying whether the load acting on a structure is considered as the secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in this case, the last column is not displayed.

The procedure for wall verification is described in the theoretical part of the help.

The computed forces are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against **overturning** and **translation**. The "**In detail**" button opens the dialog window, which contains a detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the "Drawing Settings" frame.



Frame "Verification"

Dimensioning

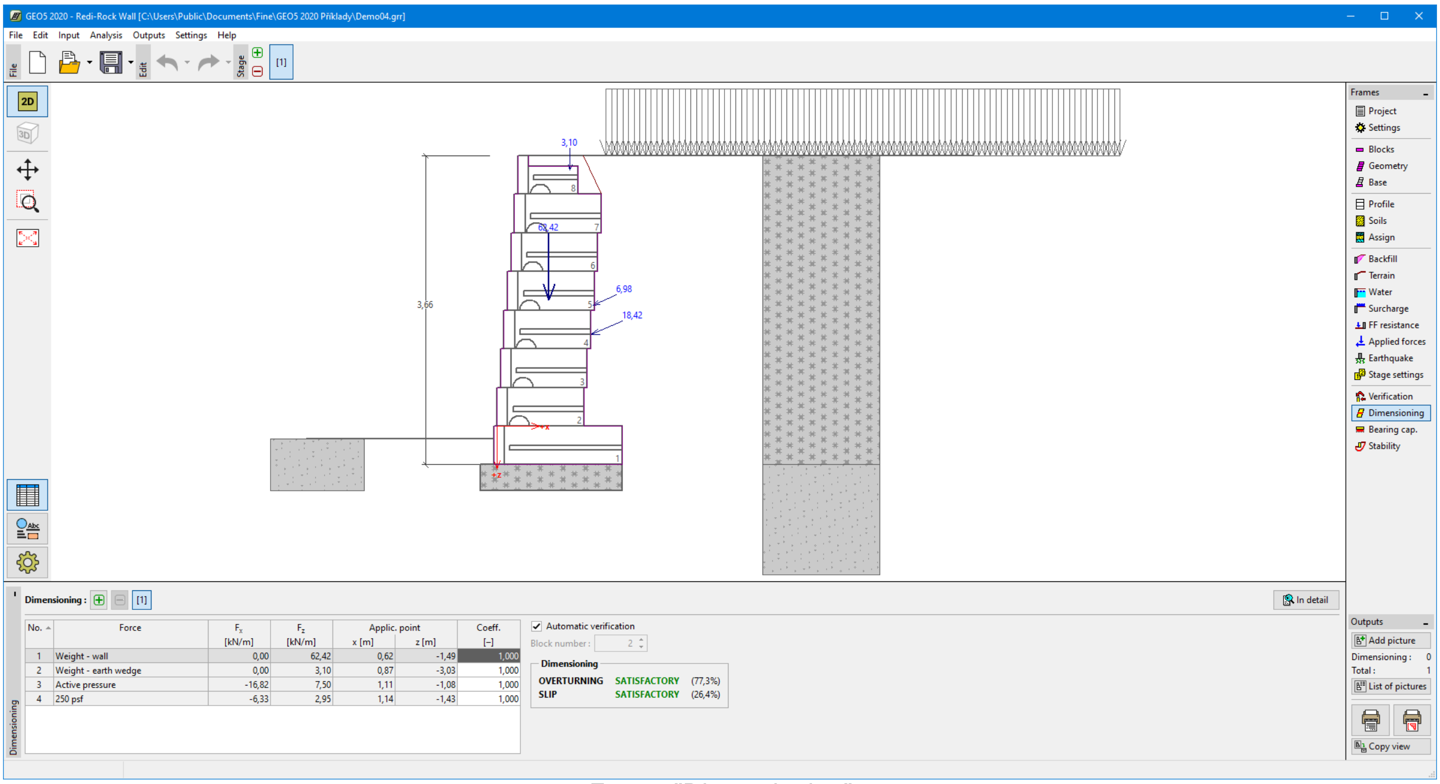
The "**Dimensioning**" frame allows us to verify joints between individual blocks of a wall against **overturning** and **slip**. The "**Automatic verification**" checks the most critical joint above the block or it is possible to perform verification for each joint.

The frame appearance is adjusted based on the selected verification methodology.

- Verification according to the factor of safety or the theory of limit states - the last column in the table allows us to input the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows us to specify whether the load acting on a structure is considered as the secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in this case, the last column is not displayed.

Several computations for various cross-sections can be carried out. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "**In detail**" button opens a dialog window that contains a detailed listing of the dimensioning results.

Visualization of results can be adjusted in the "Drawing Settings" frame.



Frame "Dimensioning"

Bearing Capacity

The "**Bearing capacity**" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the frame "Verification". The programs "**Spread footing**" and "**Spread footing CPT**" then consider all verifications as load cases.

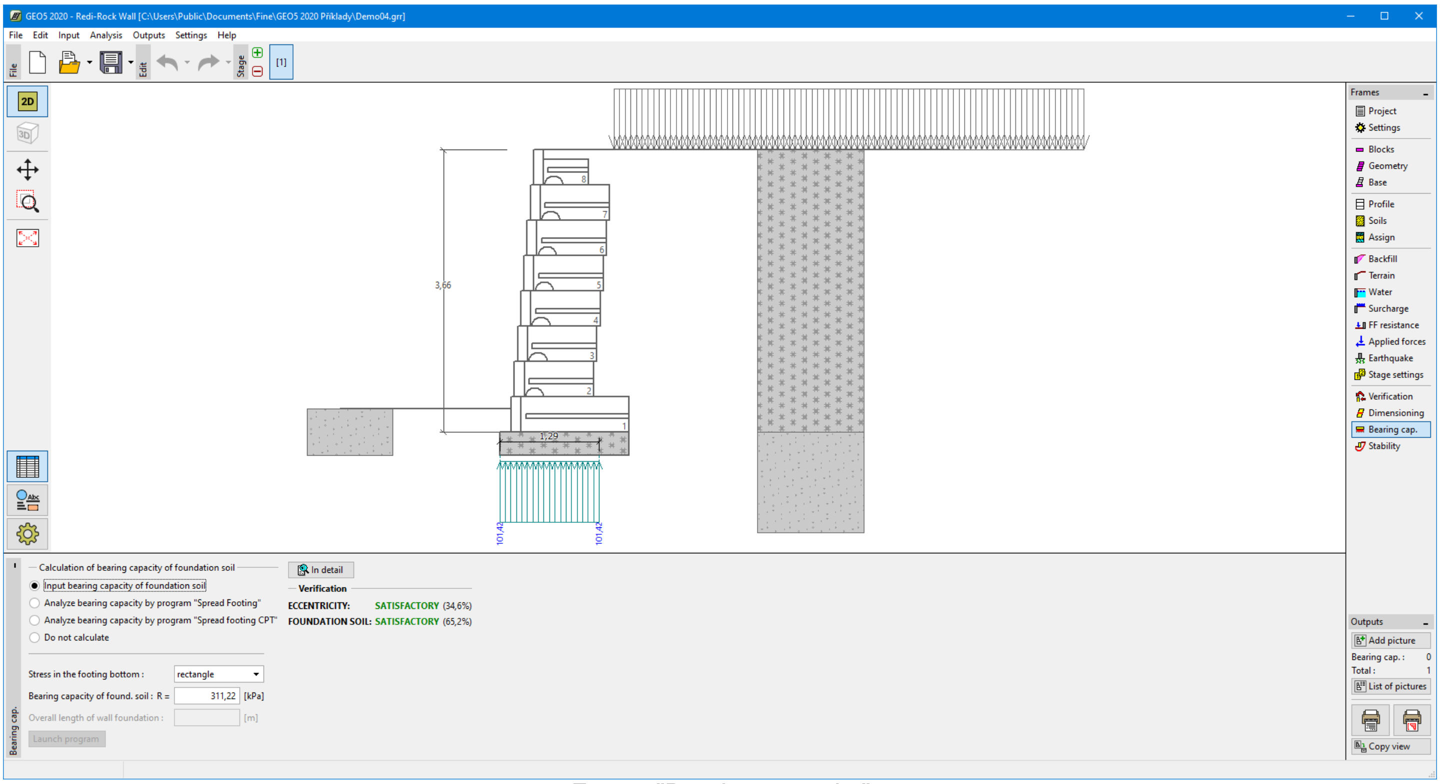
A restriction according to Fig. 4-4, page 73 of NCMA manual is neither automatically enforced nor checked by the program. The bearing pad must be introduced into the program so it complies with the design criteria.

Three basic analysis options are available in the frame:

- **Input the foundation soil bearing capacity** — The input field allows us to specify the foundation soil bearing capacity. The results of verification analysis of soil for **eccentricity** and bearing capacity are displayed in the right part of the frame. The "**In detail**" button opens the dialog window that displays detailed listing of the results of verification analysis of foundation soil bearing capacity.
- **Compute the foundation soil bearing capacity using the program "Spread footing"** — Pressing the "**Run Spread footing**" button starts the program "**Spread footing**" that allows for computing the soil bearing capacity or settlement and rotation of a footing. Pressing the "**OK**" button leaves the analysis regime - the results and all plots are copied to the program "**Redi Rock wall**". The program "**Spread footing**" must be installed for the button to be active. The overall length of the wall foundation is input.
- **Analyze bearing capacity by program "Spread footing CPT"** — The procedure is identical as if calculating soil bearing capacity by the "**Spread footing**" program.
- **Do not compute (pile footing)** — The foundation soil bearing capacity is not computed.

The program allows us to specify a shape of stress in the footing bottom.

Visualization of results can be adjusted in the "Drawing Settings" frame.

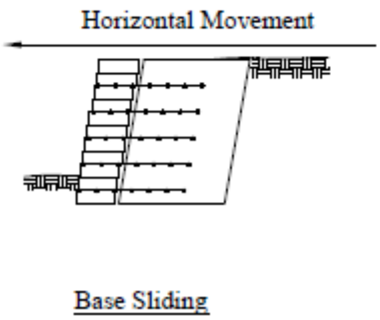


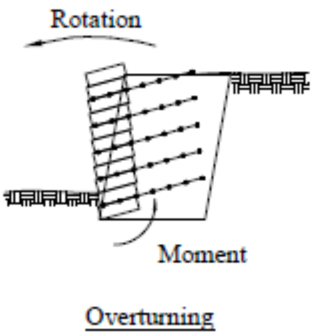
Frame "Bearing capacity"

Analysis - MSE Wall

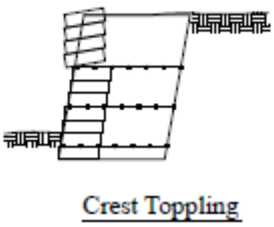
Verification of retaining **MSE Wall** made of Redi-Rock blocks and Mirafi reinforcements are performed in the following frames:

- Verification - verification of a wall against **overturning** and **slip**

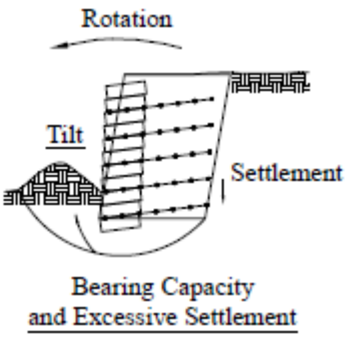




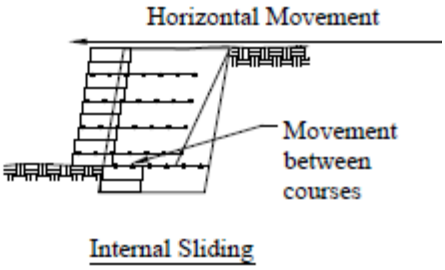
- Dimensioning - verification of joints between individual blocks of a wall against **overturning** and **slip**



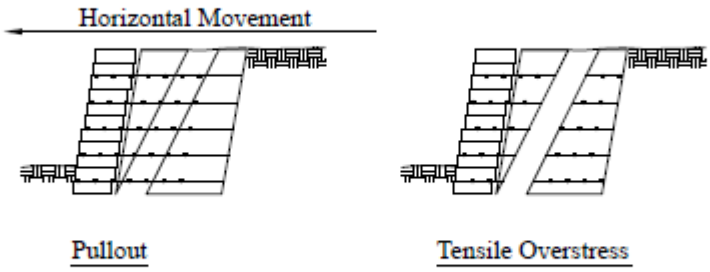
- Bearing capacity - verification of the foundation soil bearing capacity



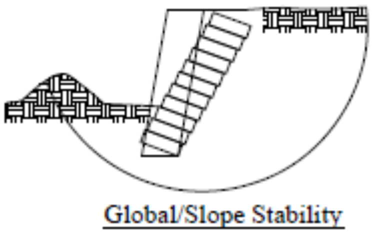
- Slip on georeinforcement - slip verification of reinforced soil block along a geo-reinforcement



- Internal stability - verification of tensile and pull-out resistance of geo-reinforcement



- Stability - verification of the overall stability of the analyzed structure



Pictures from: *Design Manual for Segmental Retaining Walls (NCMA, 3rd Edition)*

Verification

The frame "**Verification**" shows the analysis results. Several computations can be carried out for a single task.

The frame appearance is adjusted based on the selected verification methodology.

- Verification according to the factor of safety or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows for specifying whether the load acting on a structure is considered as the secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in this case, the last column is not displayed.

The procedure for wall verification is described in the theoretical part of the help.

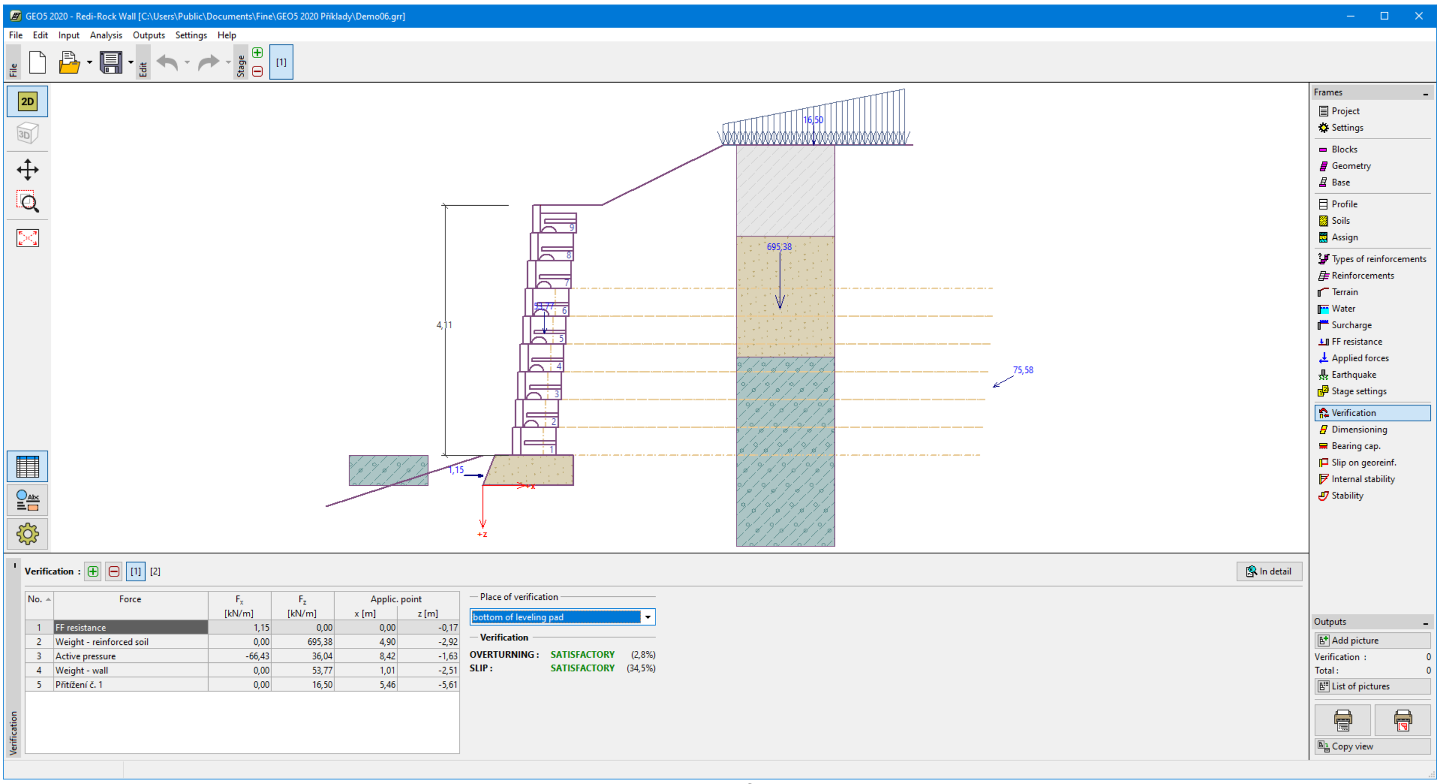
The computed forces are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against **overturning** and **translation**. The "**In detail**" button opens the dialog window, which contains a detailed listing of the results of verification analysis.

Verification can be performed in the place:

- bottom of leveling pad

- bottom of blocks

Visualization of results can be adjusted in the "Drawing Settings" frame.



Frame "Verification"

Dimensioning

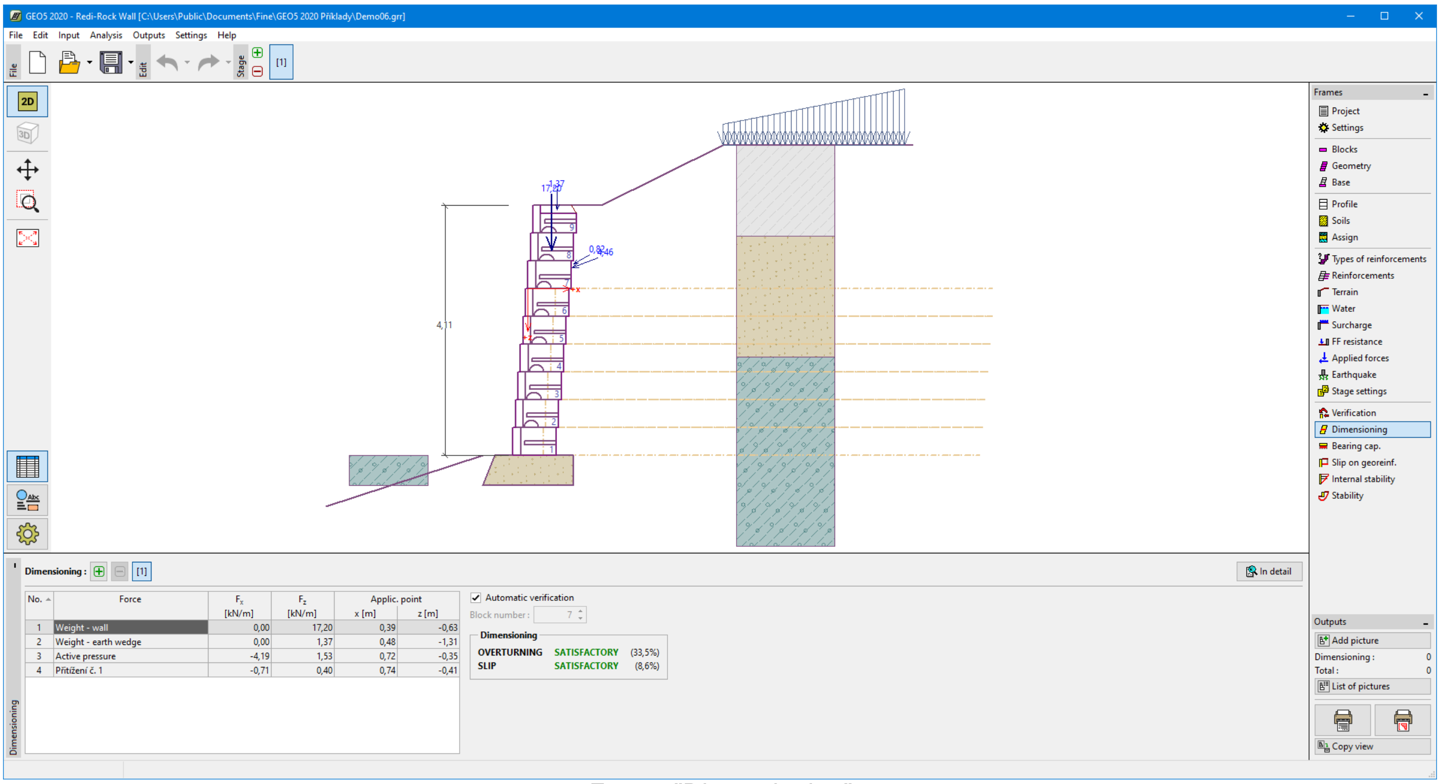
The "**Dimensioning**" frame allows us to verify joints between individual blocks of a wall against **overturning** and **slip**. Only the joints which are not reinforced by geo-reinforcements can be verified. The "**Automatic verification**" checks the most critical joint above the block or it is possible to perform verification for each joint.

The frame appearance is adjusted based on the selected verification methodology.

- Verification according to the factor of safety or the theory of limit states - the last column in the table allows us to input the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows us to specify whether the load acting on a structure is considered as the secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in this case, the last column is not displayed.

Several computations for various cross-sections can be carried out. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "**In detail**" button opens a dialog window that contains a detailed listing of the dimensioning results.

Visualization of results can be adjusted in the "Drawing Settings" frame.



Frame "Dimensioning"

Bearing Capacity

The "**Bearing capacity**" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the frame "Verification". The programs "**Spread footing**" and "**Spread footing CPT**" then consider all verifications as load cases.

A restriction according to Fig. 4-4, page 73 of the NCMA manual is neither automatically enforced nor checked by the program. The bearing pad must be introduced into the program so it complies with the design criteria.

Three basic analysis options are available in the frame:

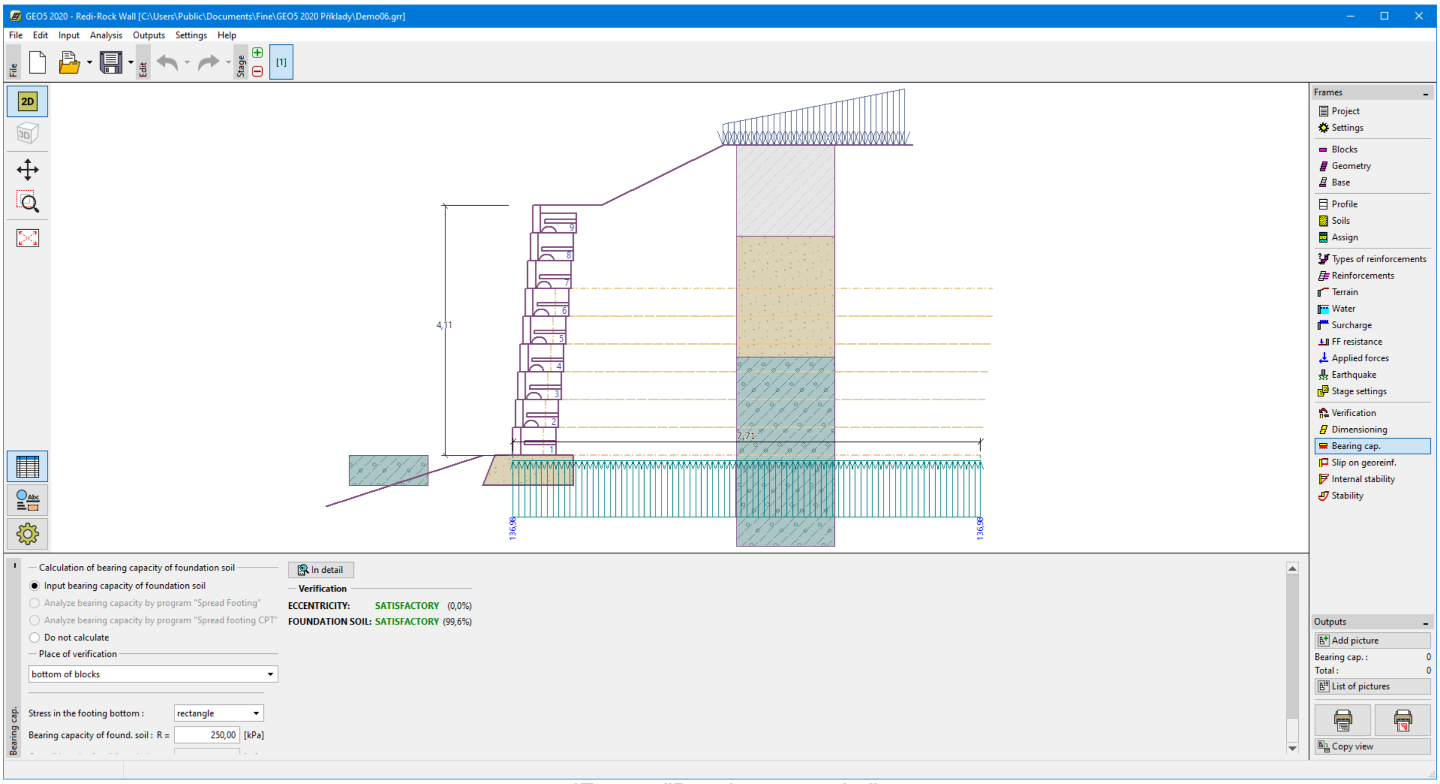
- **Input the foundation soil bearing capacity** – The input field allows us to specify the foundation soil bearing capacity. The results of verification analysis of soil for **eccentricity** and bearing capacity are displayed in the right part of the frame. The "**In detail**" button opens the dialog window that displays detailed listing of the results of verification analysis of foundation soil bearing capacity.
- **Compute the foundation soil bearing capacity using the program "Spread footing"** – Pressing the "**Run Spread footing"** button starts the program "**Spread footing**" that allows for computing the soil bearing capacity or settlement and rotation of a footing. Pressing the "**OK**" button leaves the analysis regime - the results and all plots are copied to the program "**Redi Rock wall**". The program "**Spread footing**" must be installed for the button to be active. The overall length of the wall foundation is input.
- **Analyze bearing capacity by program "Spread footing CPT"** – The procedure is identical as if calculating soil bearing capacity by the "**Spread footing**" program.
- **Do not compute (pile footing)** – The foundation soil bearing capacity is not computed.

The program allows us to specify a shape of stress in the footing bottom.

Verification can be performed in the place:

- bottom of leveling pad
- bottom of blocks

Visualization of results can be adjusted in the "Drawing Settings" frame.



dFrame "Bearing capacity"

Slip on Georeinforcement

The "**Slip on georeinforcement**" frame enables to verify a slip of the reinforced soil block along a geo-reinforcement checking the field "**Reinforcement number**". Selecting the option "**Automatic verification**" provides verification of the most critical reinforcement. The **reinforced soil block** is bounded by the wall front face, the checked geo-reinforcement, a vertical line passing through the geo-reinforcement end point and terrain. The reinforced soil block is loaded by an active earth pressure and by stabilizing forces due to geo-reinforcements exceeding the boundary of the reinforced block.

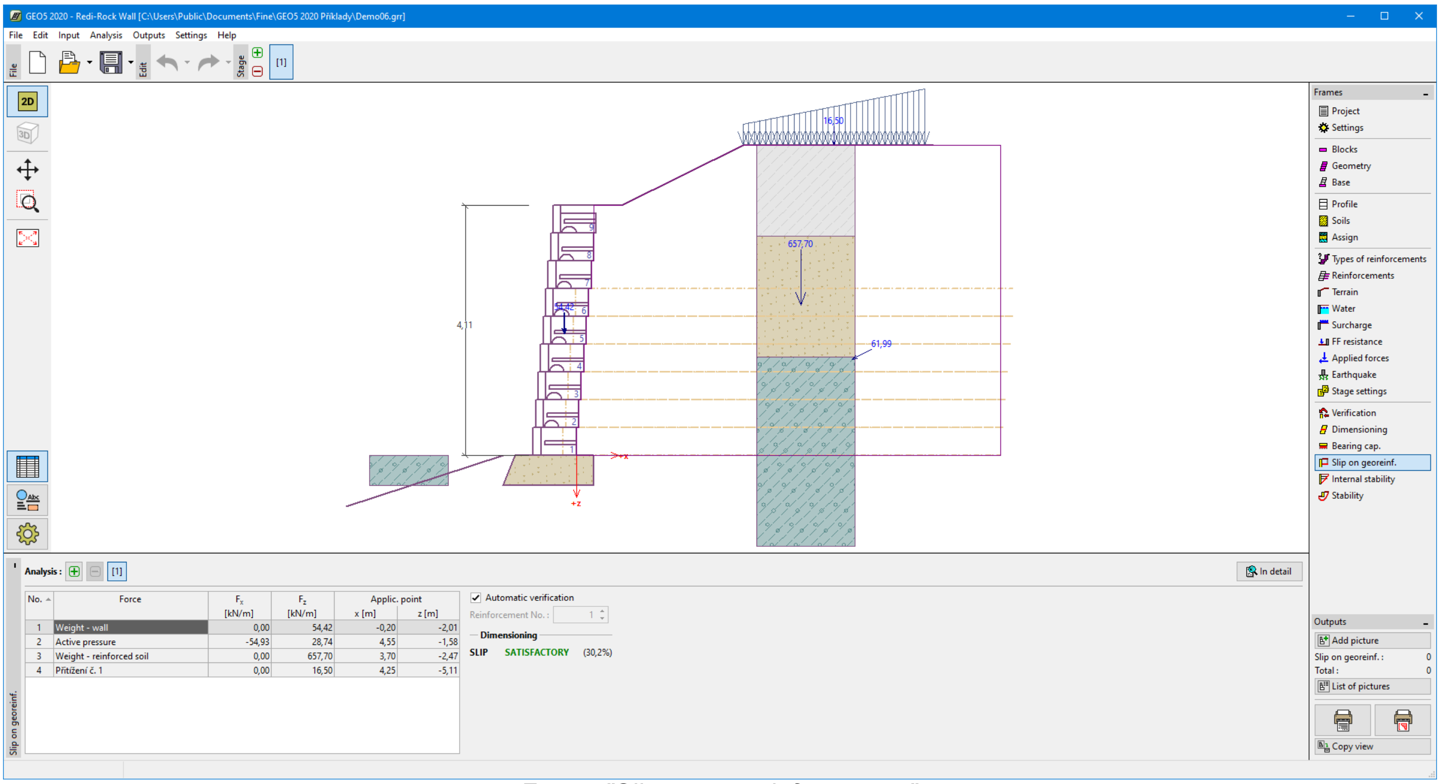
The solution procedure of slip on georeinforcement is described in the theoretical part of the help.

The frame appearance is adjusted based on the selected verification methodology:

- Verification according to the factor of safety or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiply the calculated forces.
- Analysis according to EN 1997 - the last column in the table allows for specifying whether the load acting on a structure is considered as the secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in this case, the last column is not displayed.

The frame enables us to perform more verification analyses of individual geo-reinforcements. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "**In detail**" button opens the dialog window, which contains a detailed listing of the results of internal stability.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Slip on georeinforcement"

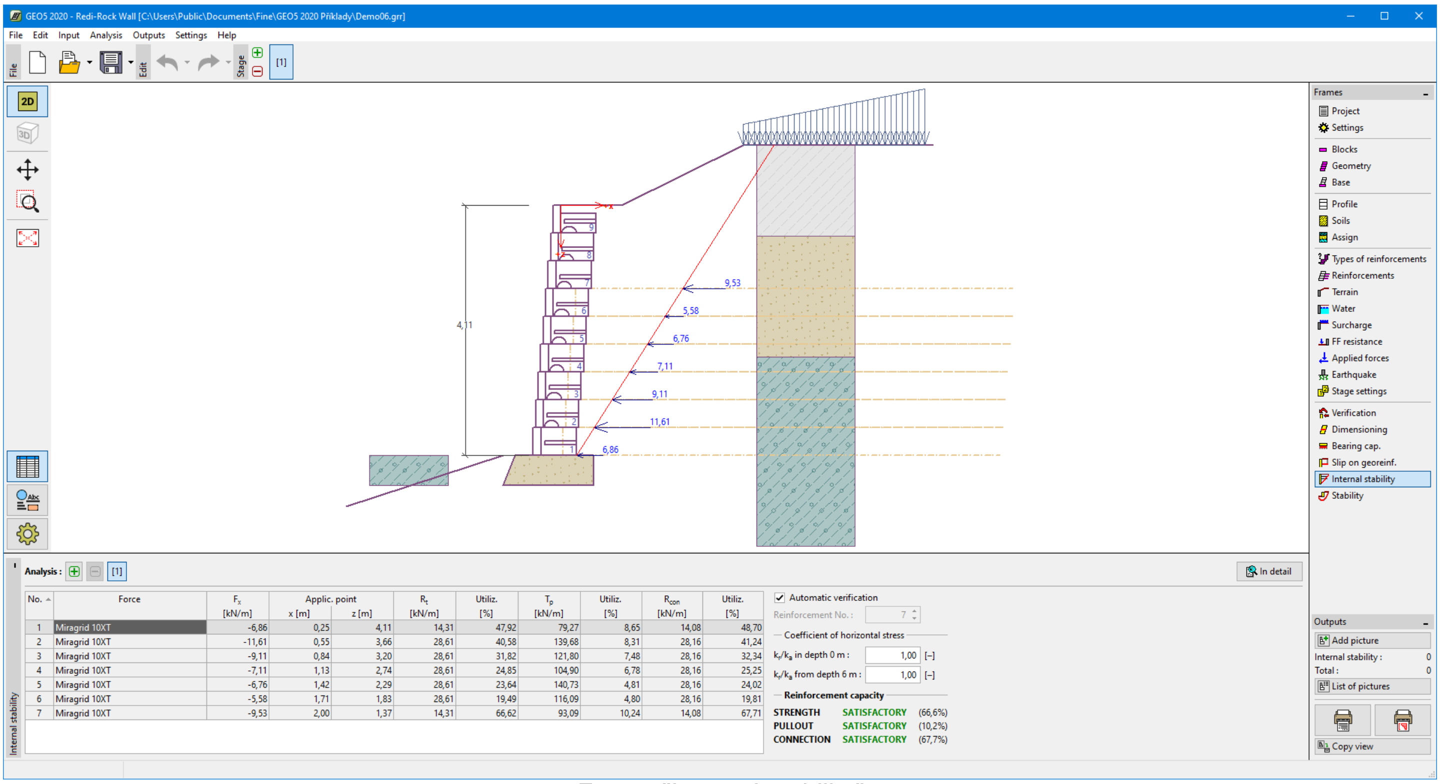
Internal Stability

The "**Internal stability**" frame enables check strength of geo-reinforcement, bearing capacity for **pull-out** from the soil, and strength of connections. Checking the field "**Reinforcement number**" yields verification for individual reinforcements only. Selecting the option "**Automatic verification**" provides verification of all reinforcements. The result for the most critical reinforcement is displayed on the right part of the desktop. The solution procedure of internal stability is described in the theoretical part of the help.

In the table, forces caused by an active earth pressure acting on the front face of the wall in individual geo-reinforcements and points of application of these forces are shown. Further, bearing capacity of geo-reinforcements for **tensile strength** R_t , **pull-out resistance** T_p , connection bearing capacity $R_{con},$ and resulting utilization is shown for each geo-reinforcement.

The frame enables us to perform more analyses of individual geo-reinforcements. The calculated forces are displayed on the desktop are automatically updated with every change of input data. The "**In detail**" button opens the dialog window, which contains a detailed listing of the results of reinforcement bearing capacity.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Internal stability"

Modeling the Entire Wall

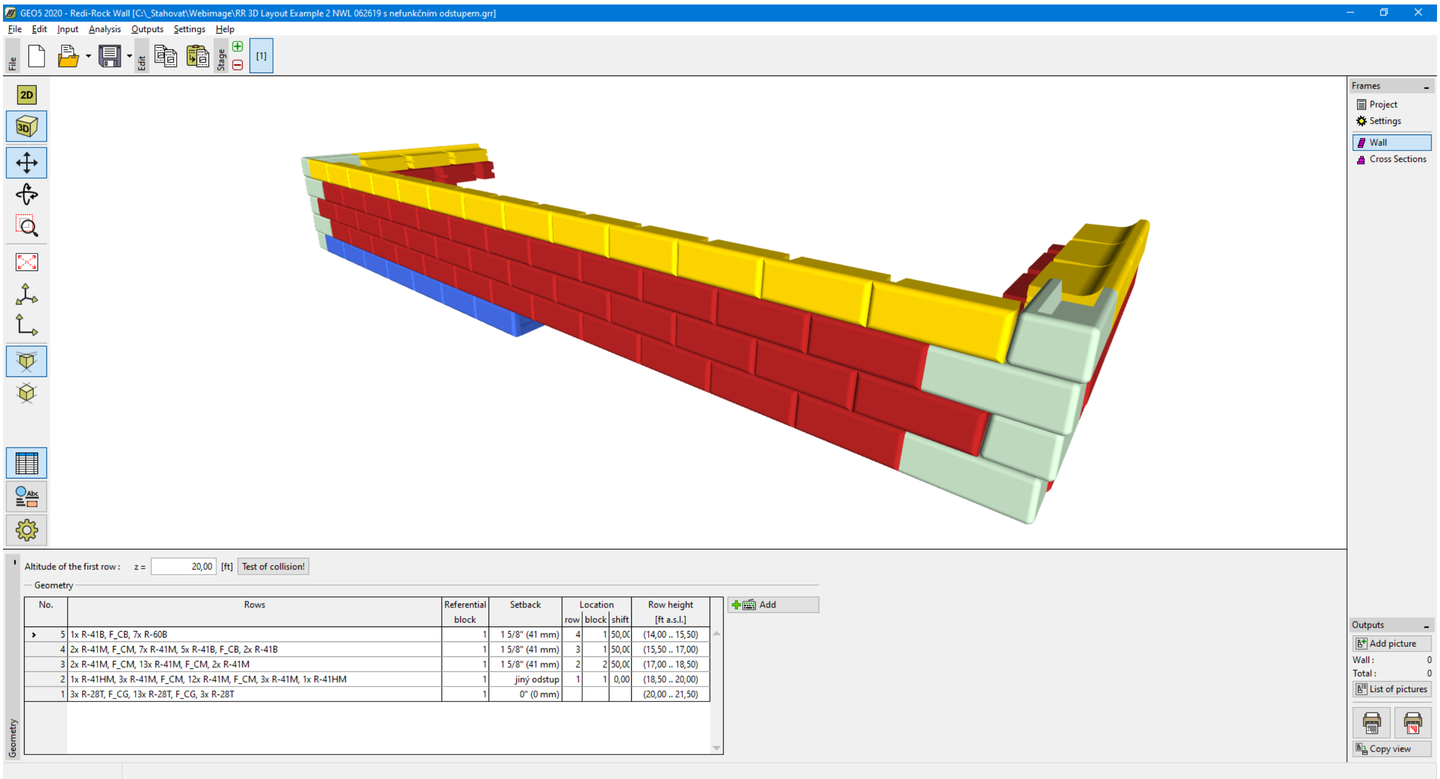
In the **"3D Wall"** mode of program (selected in the Settings frame) it is possible to:

- create the wall
- add annotations and descriptions to the wall drawing using the Annotations
- display cross-sections of the walls
- copy cross-sections data for analyses as gravity or MSE walls.

Modeling the entire wall is described in detail in Engineering Manual No. 39.

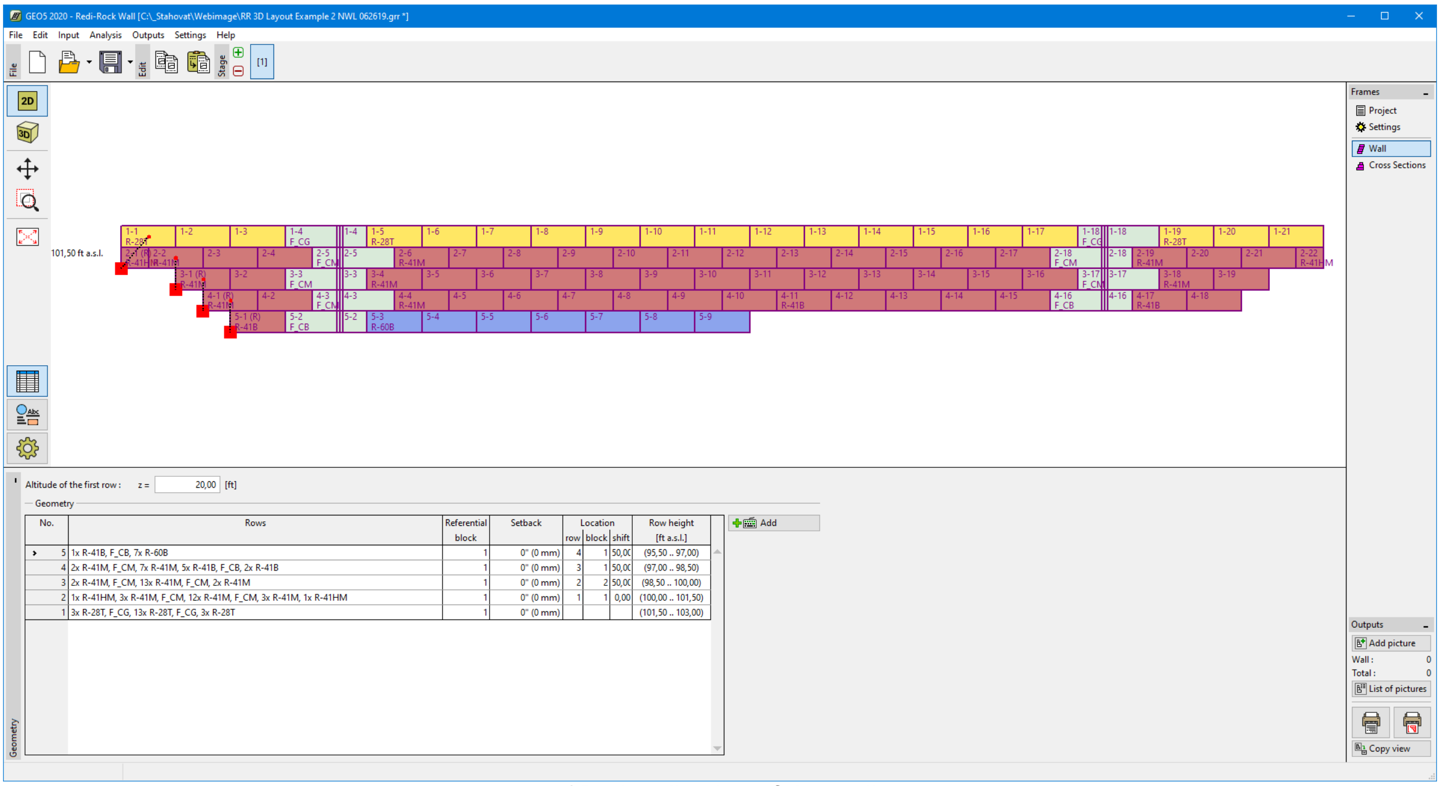
2D or 3D wall visualization options can be selected on the left control bar.

The 3D visualization is very clear but it is not possible to add annotations.



3D visualization of the wall

The 2D visualization allows for describing the structure, dimensioning, describing the blocks. It is, however, much more complicated to imagine the wall. 3D shape of the wall is unfolded into 2D shape, a triple line shows the corner. The corner blocks are rendered from both sides.



2D visualization of the wall

Wall Geometry

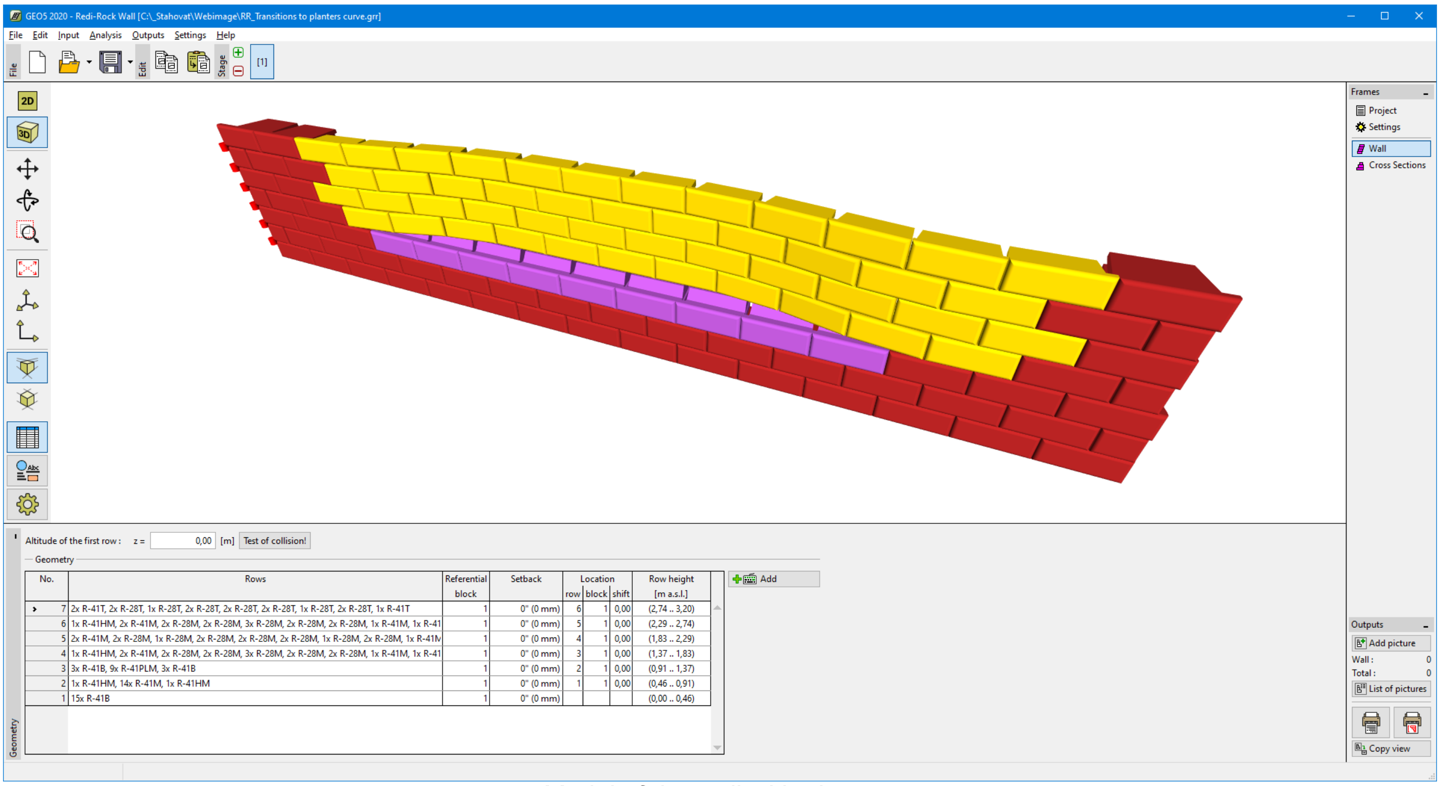
The wall configuration is defined in the **"Wall Geometry"** frame.

The wall blocks allow creating of many wall shapes including corners, arcs, setbacks, spaces between blocks.

The function of the program is similar to **creating a wall from the Lego building kit!**:

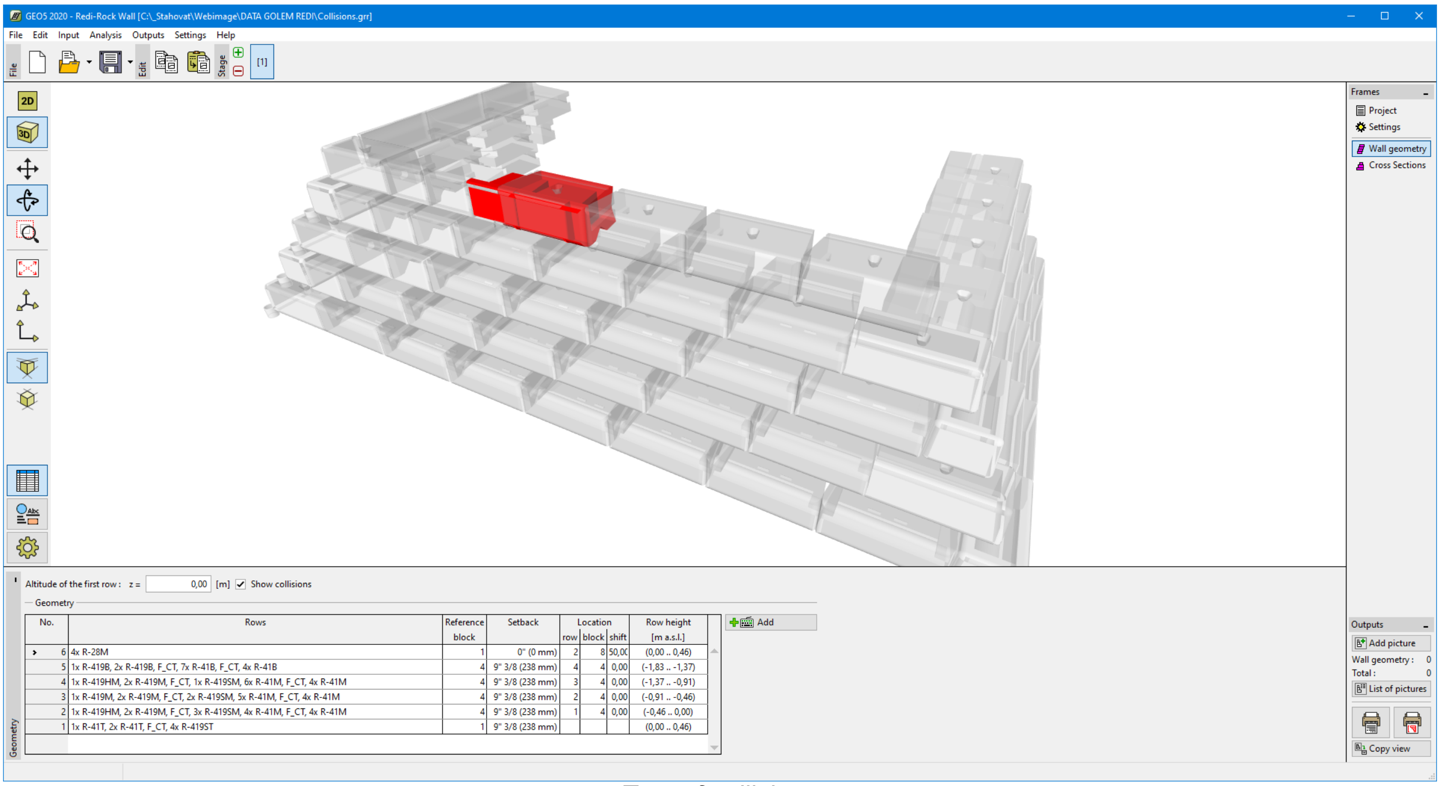
- First, we create a starting row of the wall. The row is created by adding blocks, and **groups of blocks,** one next to another.
- We create subsequent rows of the wall and connect them to an existing row of the wall. A new row can be connected to the original row from the above or below.

Rows of the wall are displayed in the table. Note that the table only allows rows to be deleted starting **from the last row entered to the first**.



Model of the wall with planters

By pressing the **"Test of collisions"** button it is possible to verify whether the wall is entered correctly and no collisions occur between the blocks. The program colors the blocks with light gray, and collisions with red.



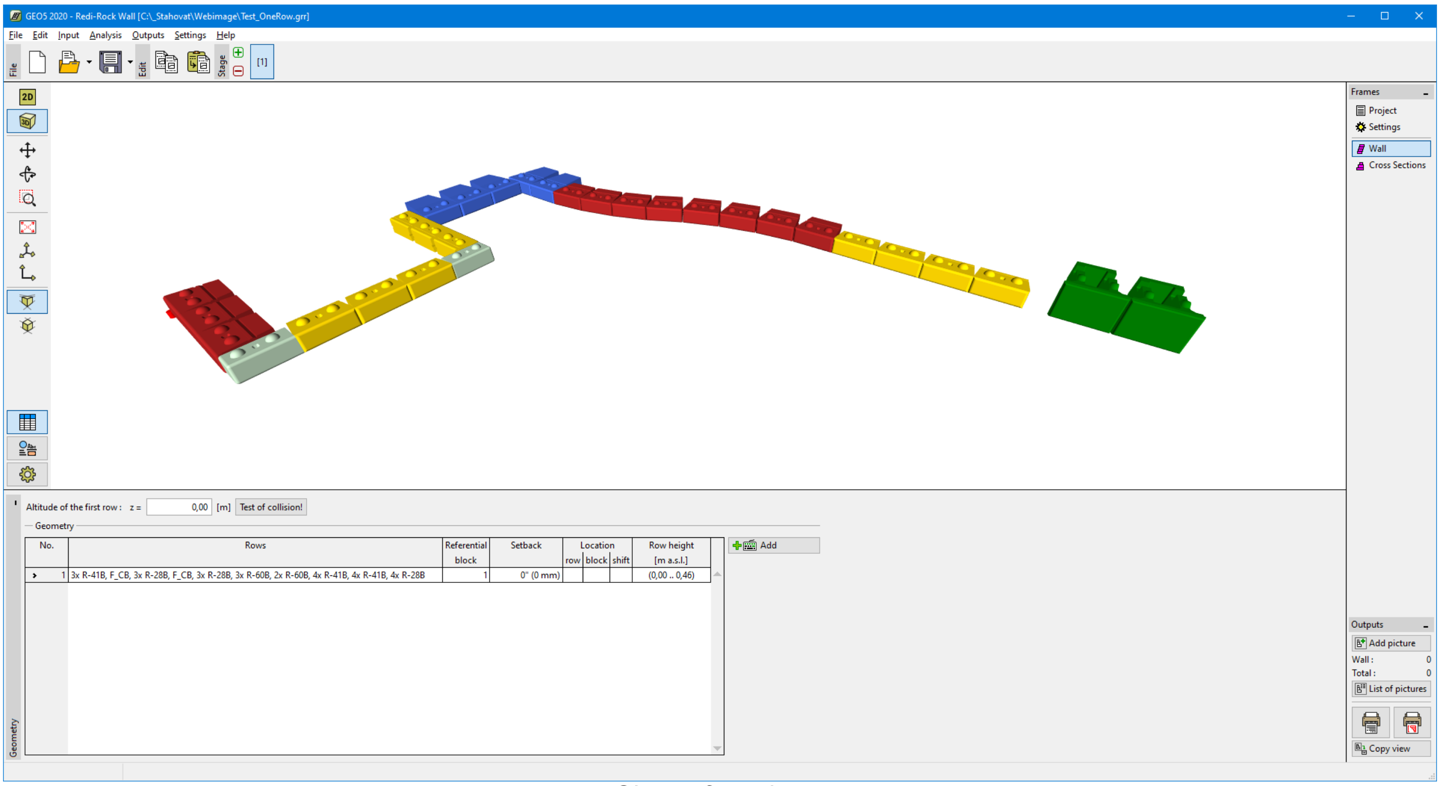
Test of collisions

Modeling the Row

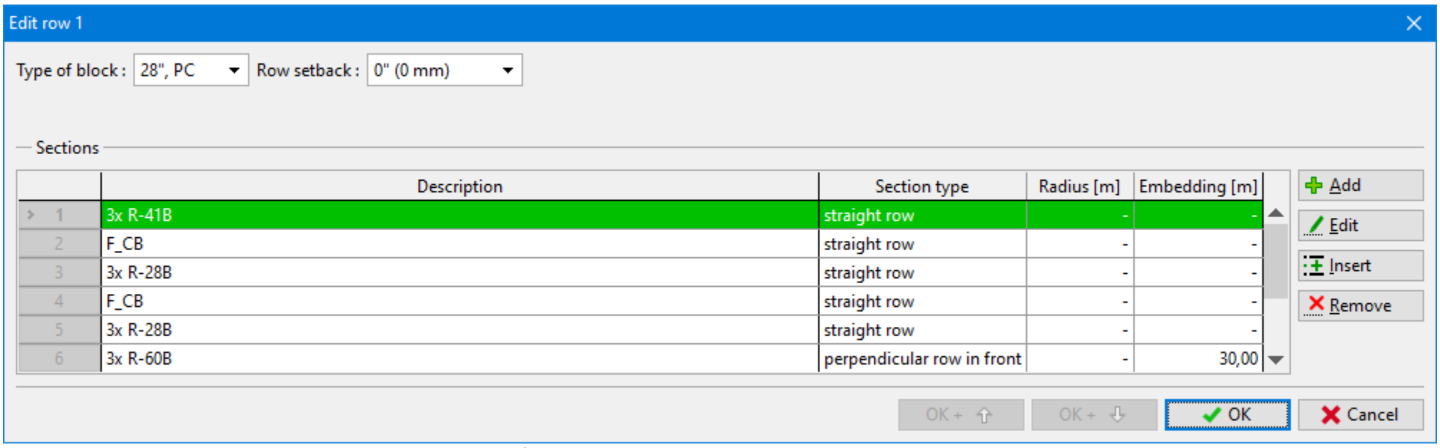
The row is modeled by the piling of blocks one behind the other.

It is necessary to define:

- **Reference block** - it is used for placing of rows on the other. It is not defined the first row.
- **Type of blocks** and **row setback** - it describes the location of knobs on defined row and **setback of other rows**. If there are blocks with various setbacks in one row, it is necessary to define **"different setback**" and setbacks are entered when block groups are defined.
- **Block groups** - block groups are displayed in the table. The groups are set one after another, each block continues the other.



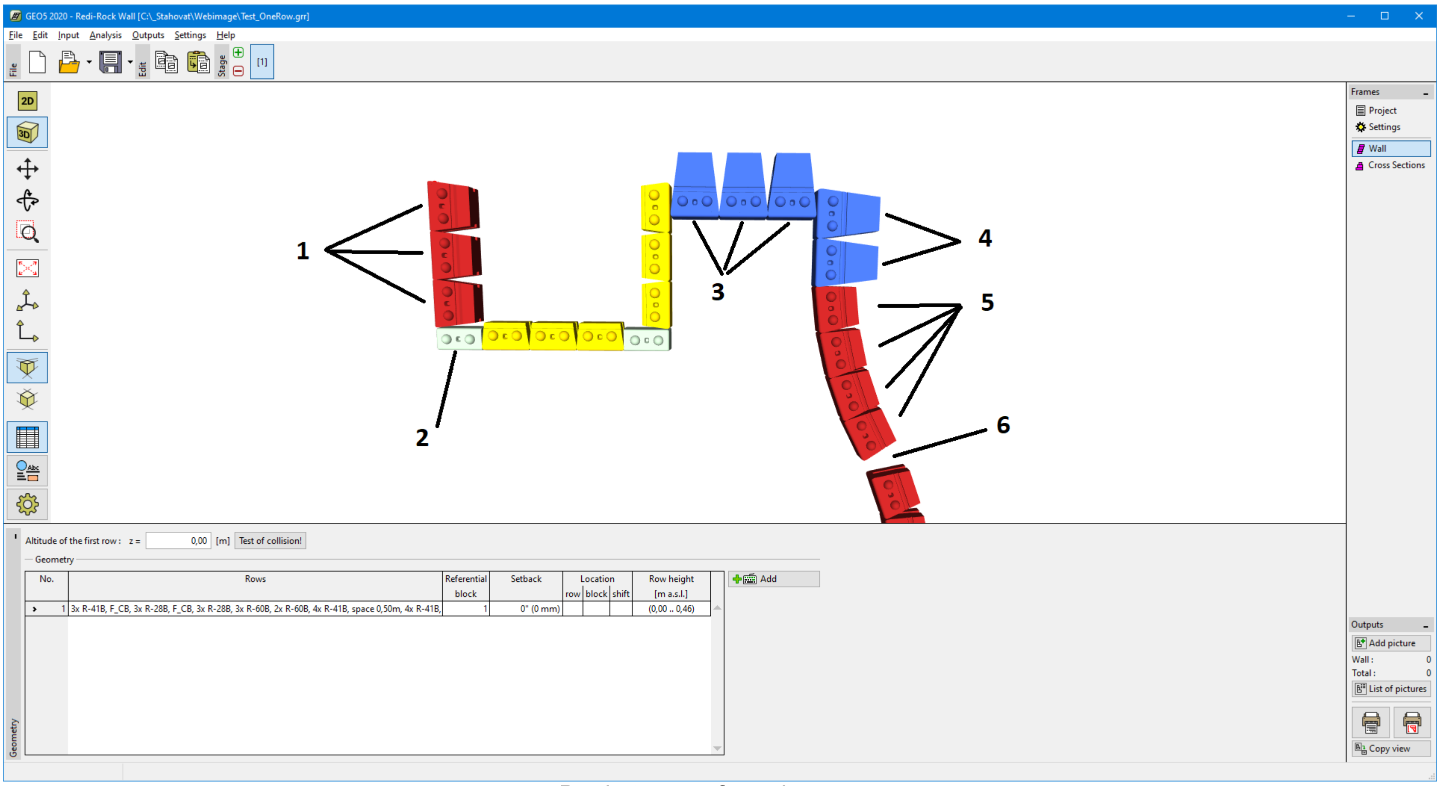
Show of row input



Display of individual block groups when editing a row

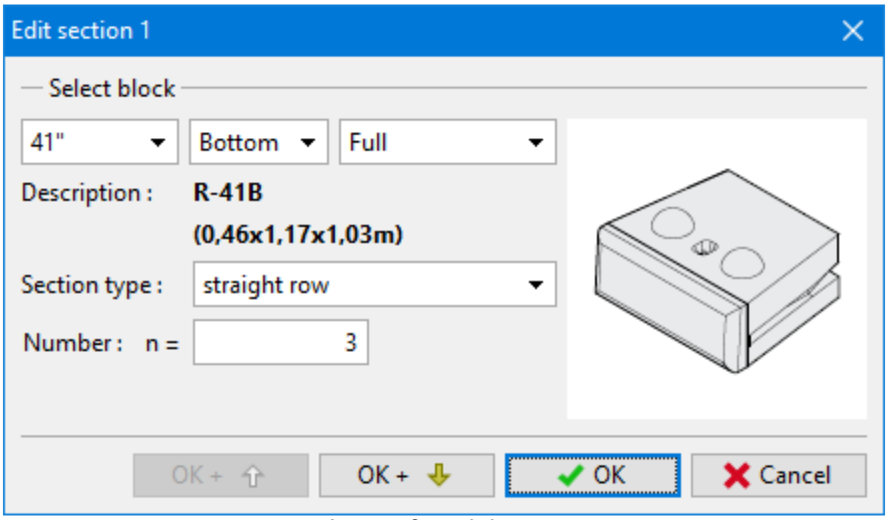
Block Groups

The shape of each wall row is defined by block groups. It is possible to define straight sections, curves, corners and also spaces between blocks.



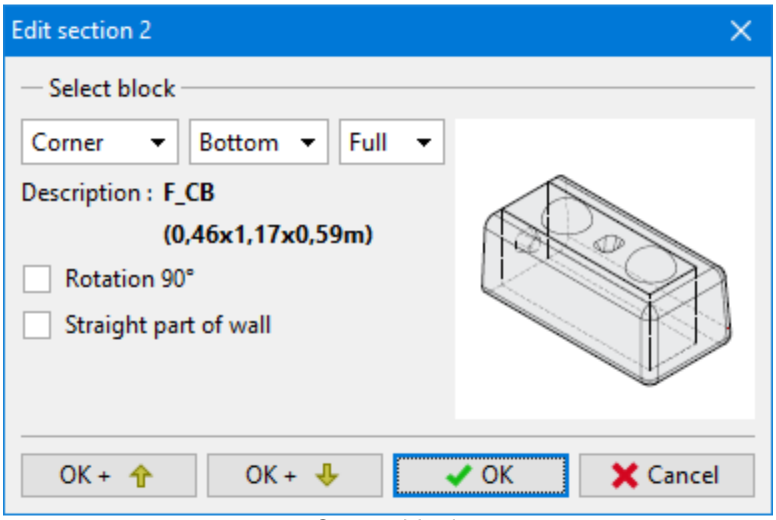
Basic types of section types

Type 1 - **straight section**



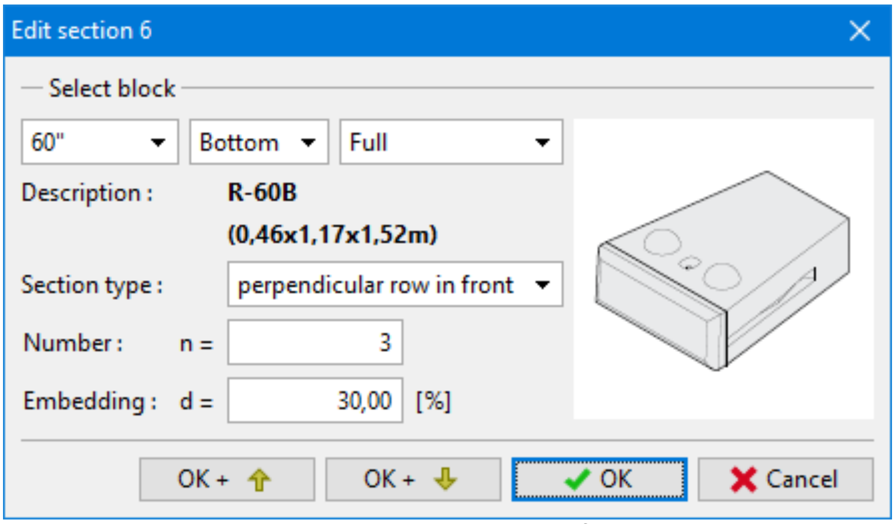
Input of straight row

Type 2 - **corner block** - If the "**straight part of wall**" option is not checked, it forms a corner automatically. It can be placed perpendicular or parallel to the original row.



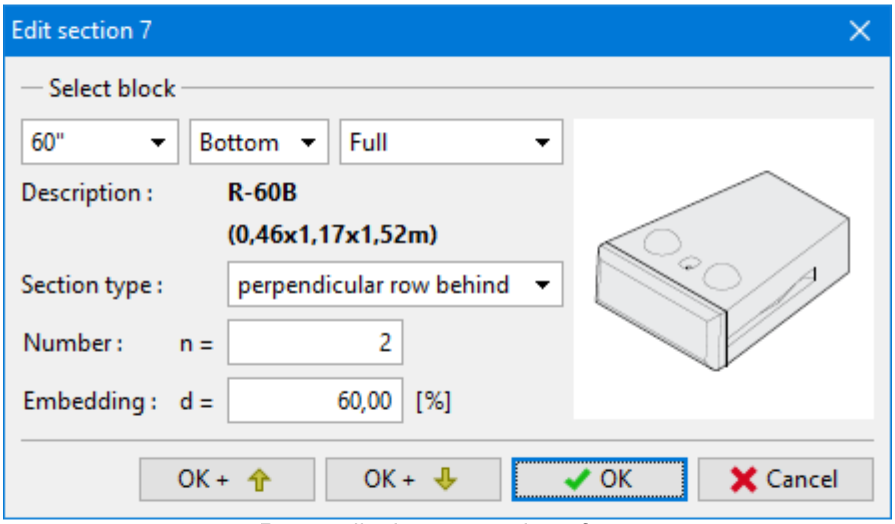
Corner block

Type 3 - piling of a **perpendicular row in front** of previous block group.



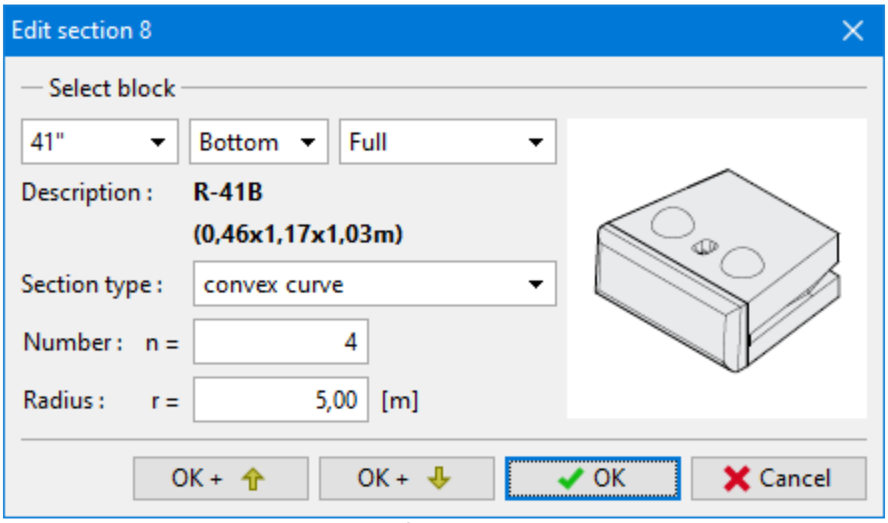
Perpendicular connection of row

Type 4 - piling of a **perpendicular row behind** of previous block group.



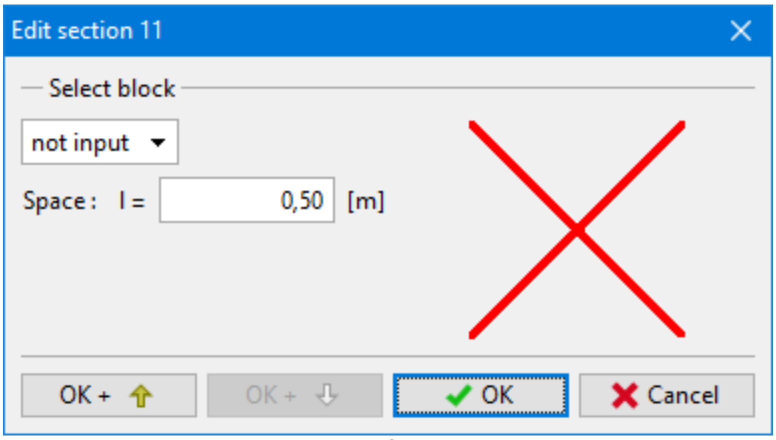
Perpendicular connection of row

Type 5 - input block group as a **curve** - it can be convex or concave.



Editing of curved section

Type 6 - creating of spaces **between blocks** - if the block is "**not input**", it is possible define space between blocks.



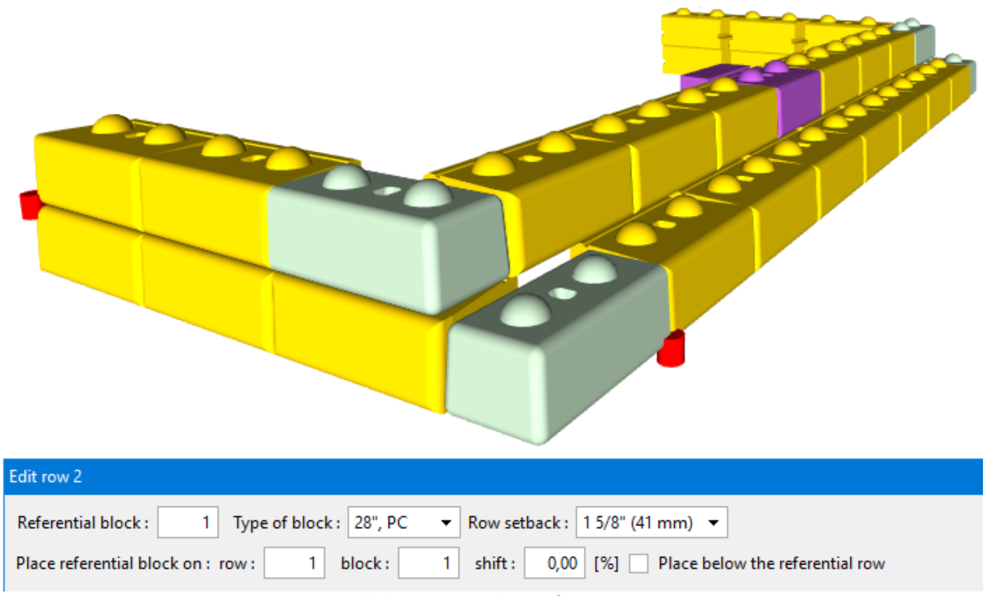
Input of space

Modeling of Wall from Rows

Individual rows are connected by **reference points**. For the show of workflow, we use modeling of the wall in the picture. The wall has setback of blocks 41 mm - in the Redi-Rock materials, it is described as "Outside corner shortback solution".

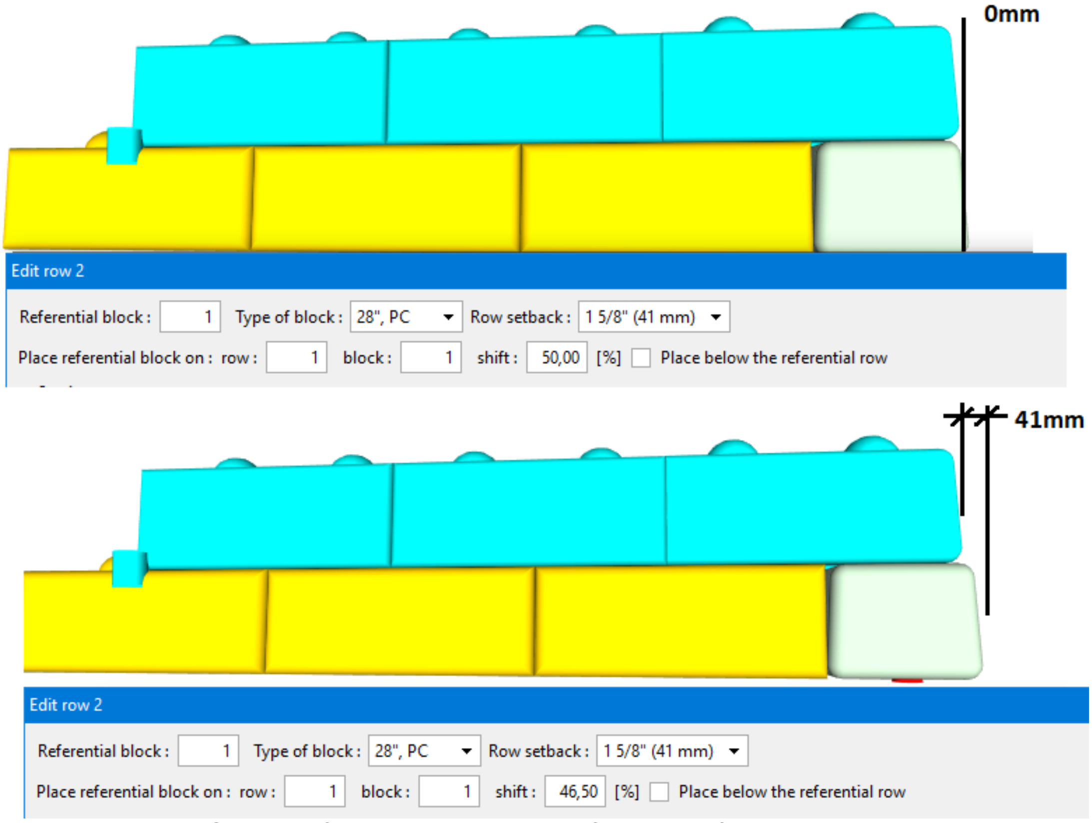
We define the first, basic line with **41 mm setback**.

We enter the second row and let the reference block as 1 and place it on block 1 of the first row. We see the result in the picture. It is necessary to shift upper row to achieve correct corner modeling.



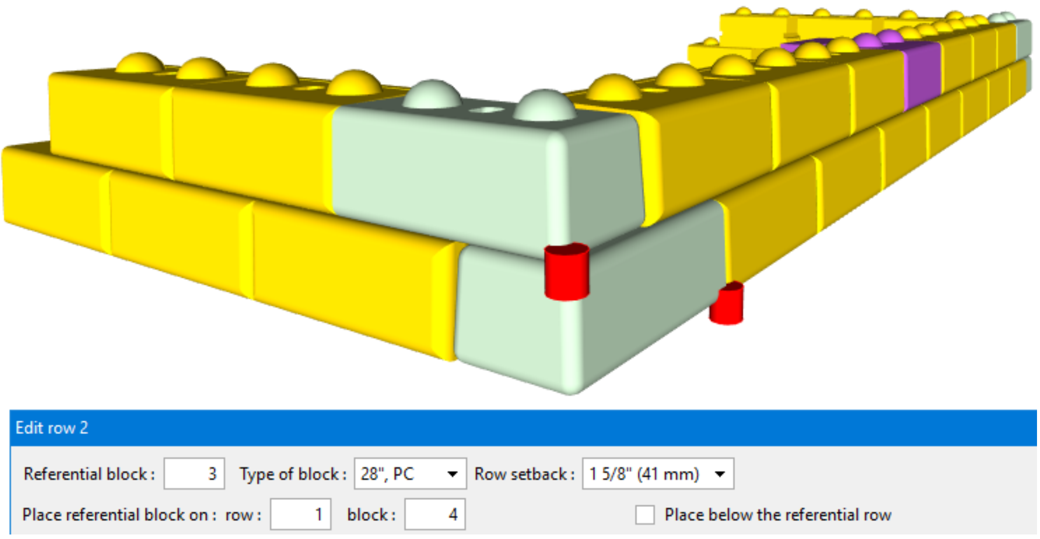
Wrong placing of rows

We change placing of reference block by shifting to 50% of bottom block. The result seems to be correct, but it is not. The **defined** 41 mm **setback is automatically set for reference block**, so both sides of wall are modeled correctly - but front of the wall is vertical without setbacks. The correct value of the shift is about 41 mm smaller - it is necessary to reduce shift percentage for 41//1172 (3,5 %) to **46,5 %**. This way we ensure **correct setback for front of wall**.



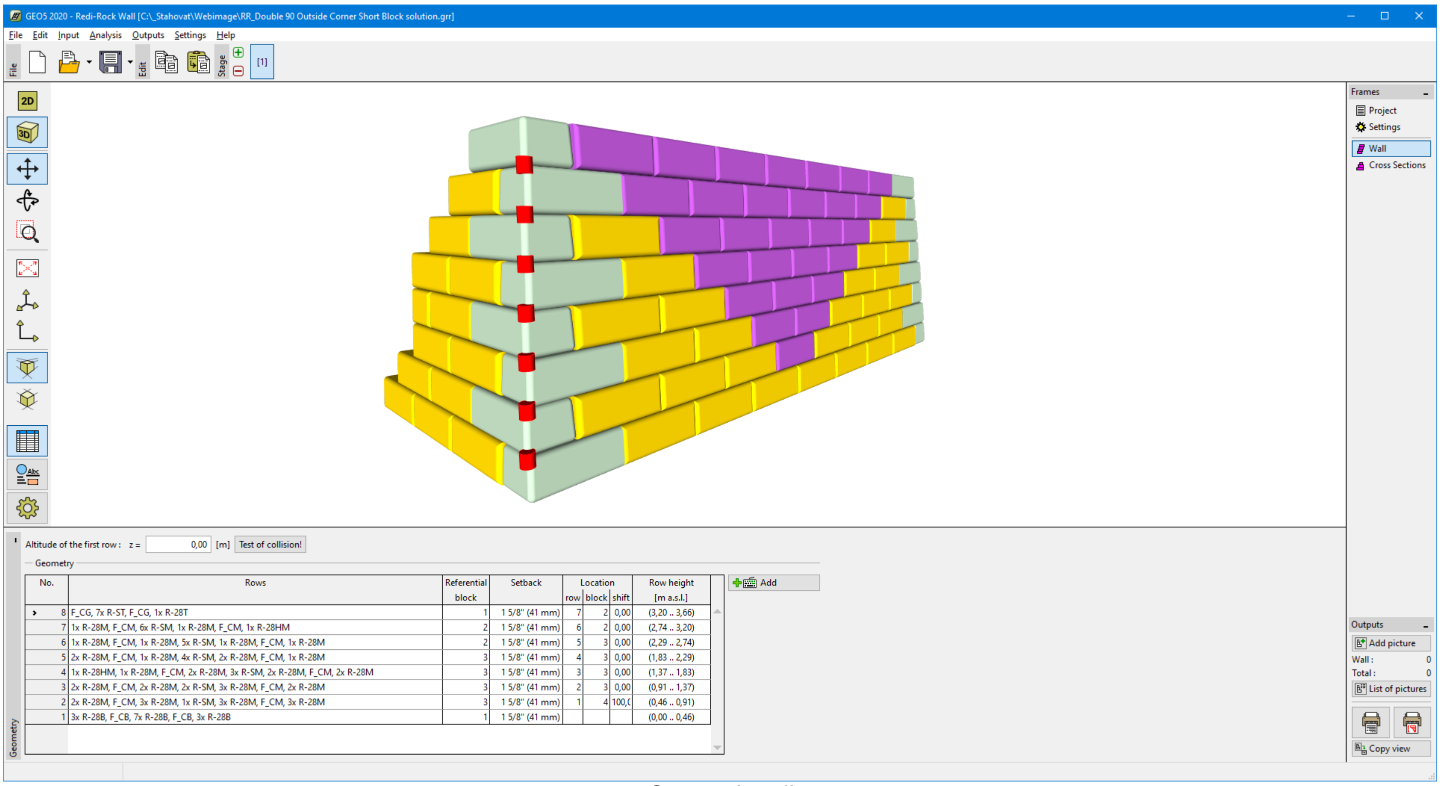
Change of row location using shift on the reference block

This process is a little bit complicated for real work - for this reason the program allows us to place **reference point on the wall corners**. In this case, the **setbacks are calculated automatically in both directions**.



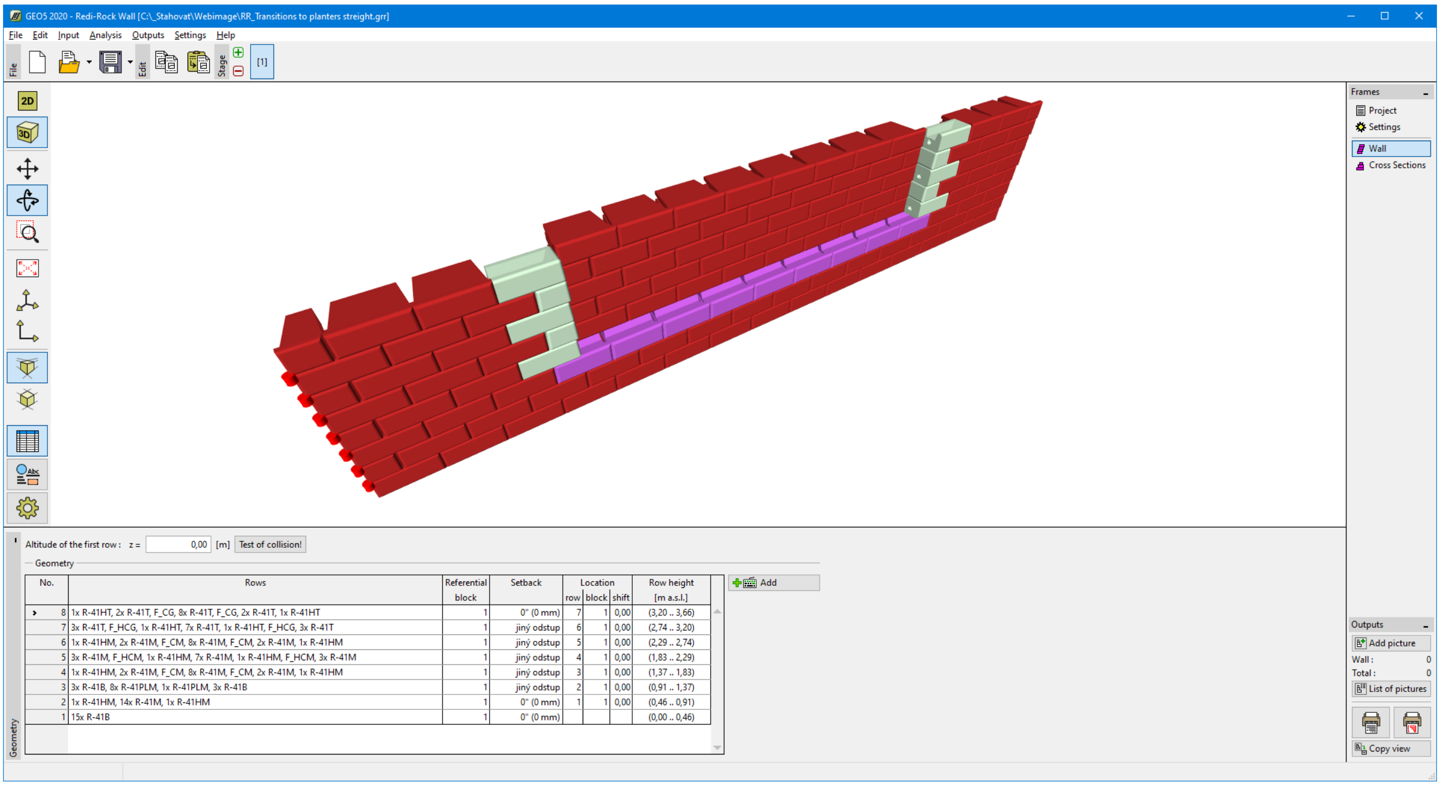
This way we add **further rows of wall**. We make sure that:

- a corner block is selected as the reference block of the upper row
- the row is placed on the corner block of the previous row



Created wall

Usually, the **setback** is the same for the whole length of the wall, but it is also possible to change it along the wall length. It can be used when more complicated walls are modeled - for example, wall with planters.



Wall with planters and straight setback

Cross Sections

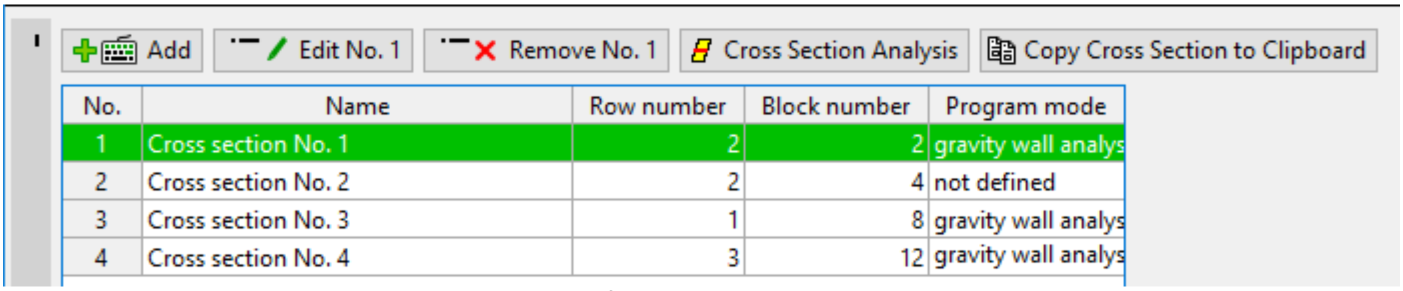
In the "**Cross-sections**" frame, it is possible to define and display an arbitrary number of wall cross-sections.

The section passes through the **middle of the selected block**, which is defined by row and its number.

The created cross-sections are automatically sorted according to the **analysis model**:

- Analysis - Gravity Wall

- Analysis - MSE Wall
- **not defined**



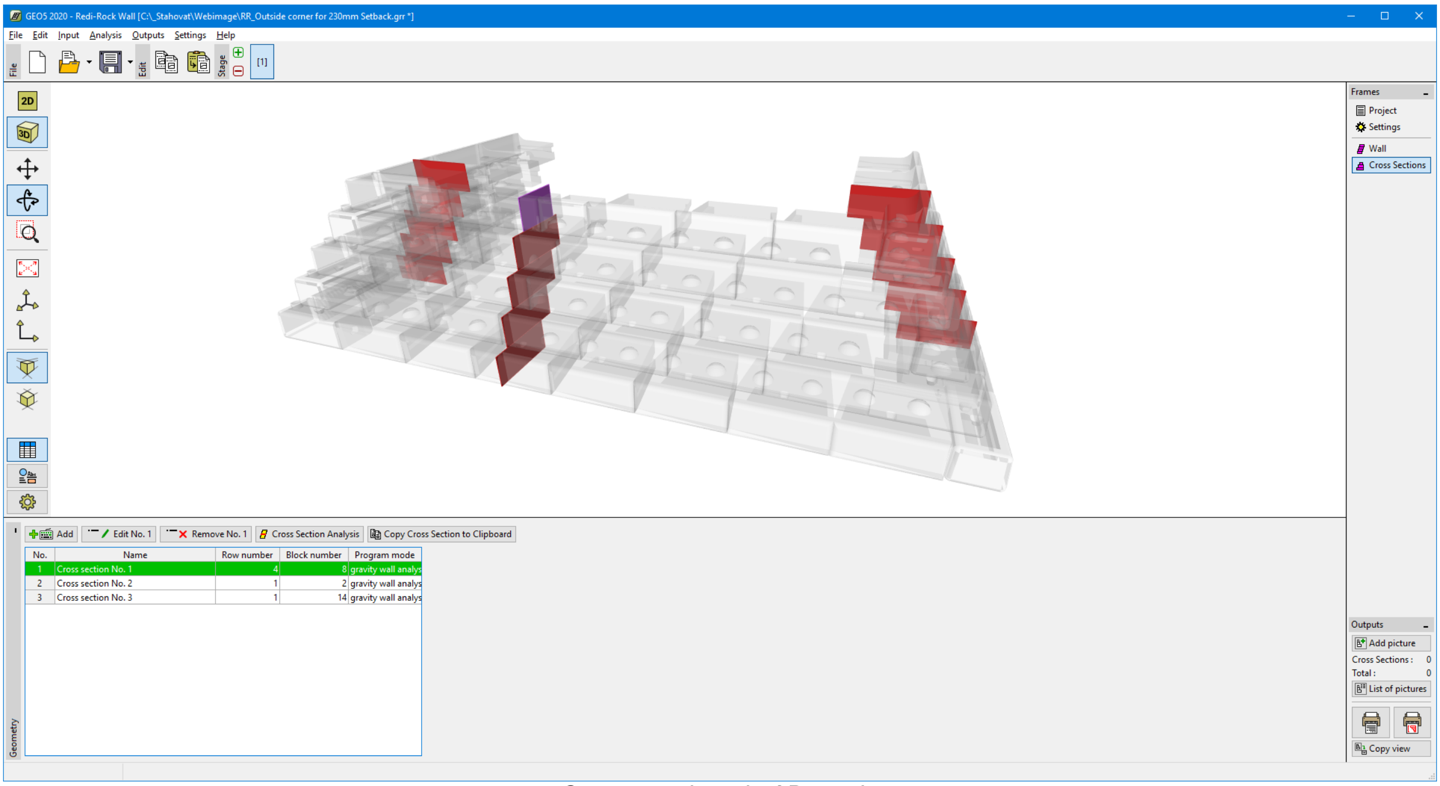
No.	Name	Row number	Block number	Program mode
1	Cross section No. 1	2	2	gravity wall analys
2	Cross section No. 2	2	4	not defined
3	Cross section No. 3	1	8	gravity wall analys
4	Cross section No. 4	3	12	gravity wall analys

Table of created cross-sections

By using "**Cross-section analysis**" button, it is possible to switch program to "**Gravity Wall Analysis**" or "**MSE Wall Analysis**" mode and export the **selected cross-section**.

The selected cross-section can also be **copied to Clipboard** and **pasted to another relation** of Redi-Rock Wall program.

Cross-sections in 2D mode

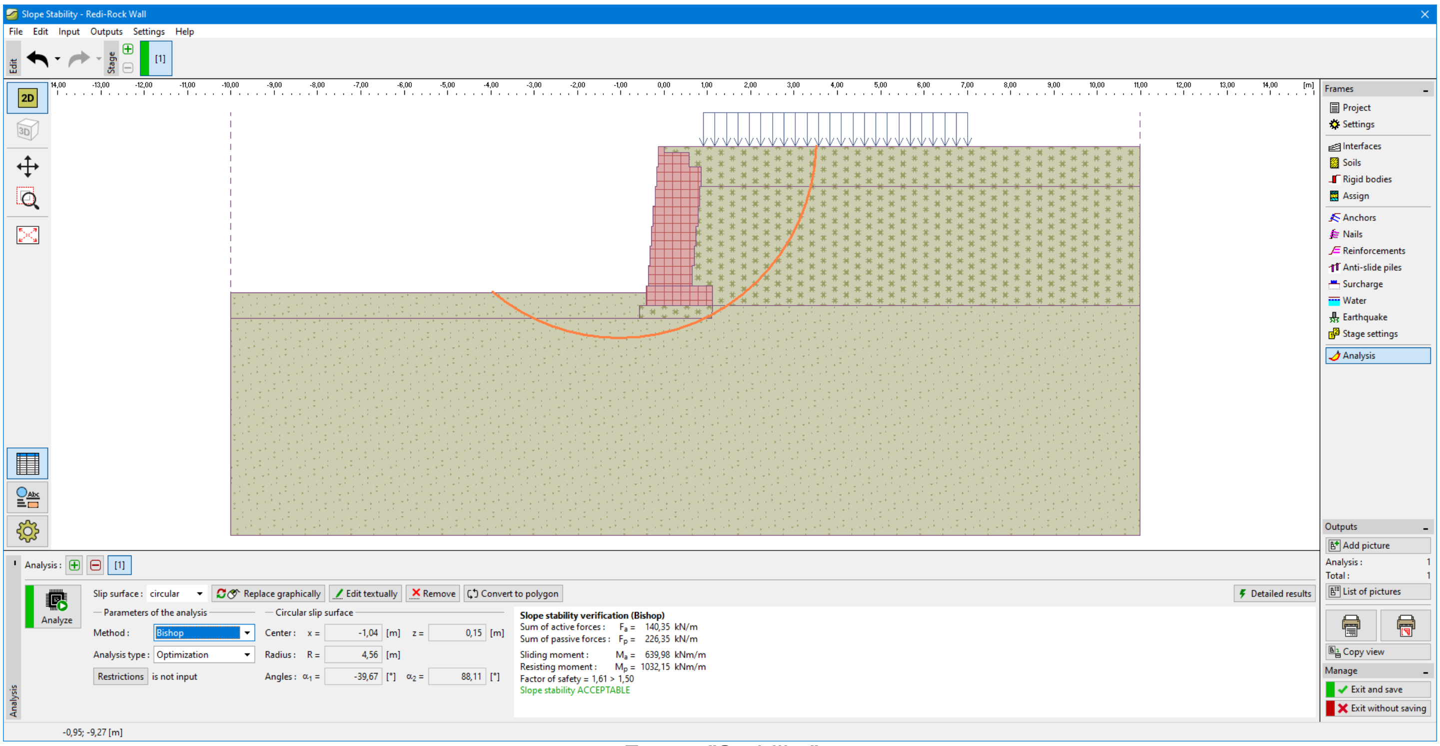


Cross-sections in 3D mode

Stability

Pressing the "**Stability**" button launches the "**Slope stability**" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program "**Slope stability**" is installed.

After completing all analyses press the "**OK**" button to leave the program - all data are then carried over to the analysis protocol of the "**Redi Rock wall**" program.



Frame "Stability"

Program Sheeting Design

This program is used for the fast design of non-anchored and preliminary design of anchored retaining walls. The results are required embedment lengths, internal forces on the structure and forces in anchors. "**Sheeting Design**" provides only a preliminary analysis.

The final analysis of multiplied anchored walls should be provided by the "Sheeting Check" program (elasto-plastic nonlinear method).

The help in the program "Sheeting Design" includes the following topics:

- The input of data into individual frames:

Project	Settings	Profile	Soils	Assign	Geometry	Material
Anchors	Props	Supports	Pressure Determination	Terrain	Water	Surcharge
Applied Forces	Earthquake	Stage Settings	Analysis	Dimensioning	Stability	

- Standards and analysis methods

- Theory for analysis in the program "**Sheeting Design**":

Stress in Soil Body	Earth Pressures	Sheeting Design	Braced Sheeting	Dimensioning of Concrete Structures	Dimensioning of Steel Cross-sections
Timber Cross Section Verification	Plastic Sheet Pile Verification	Soil Mix Verification			

- Outputs

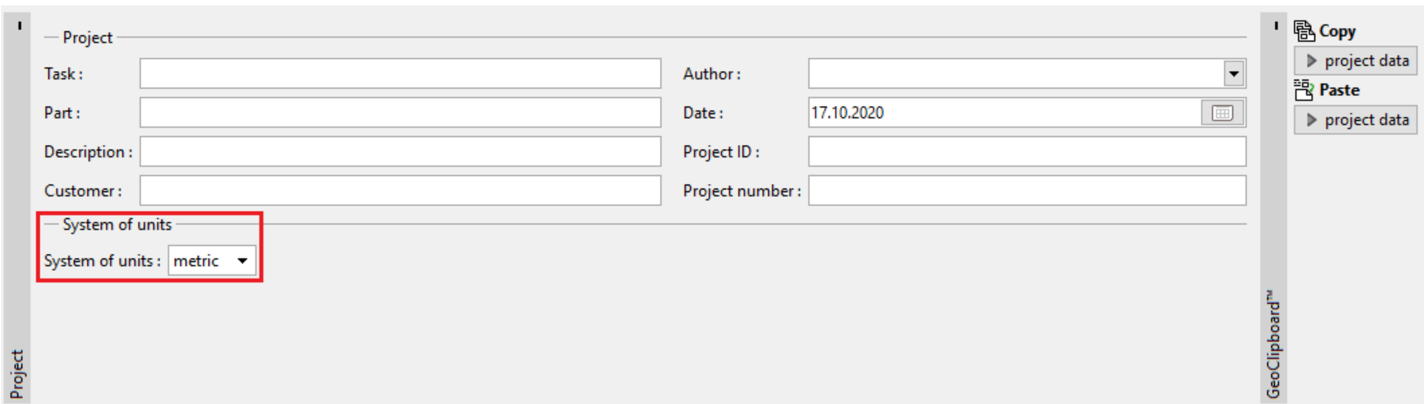
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The **"Project"** frame is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The "**Settings**" frame allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

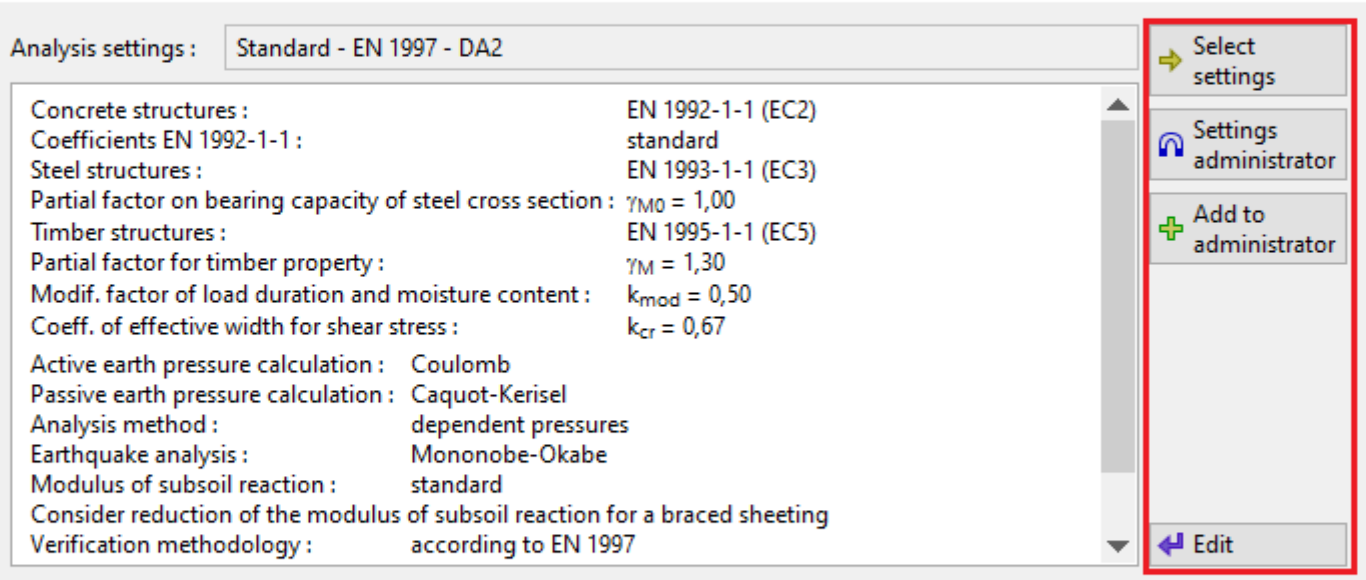
The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

Settings of analysis parameters are performed in the "Materials and standards" and the "Pressure Analysis" tabs.



Frame "Settings"

Profile

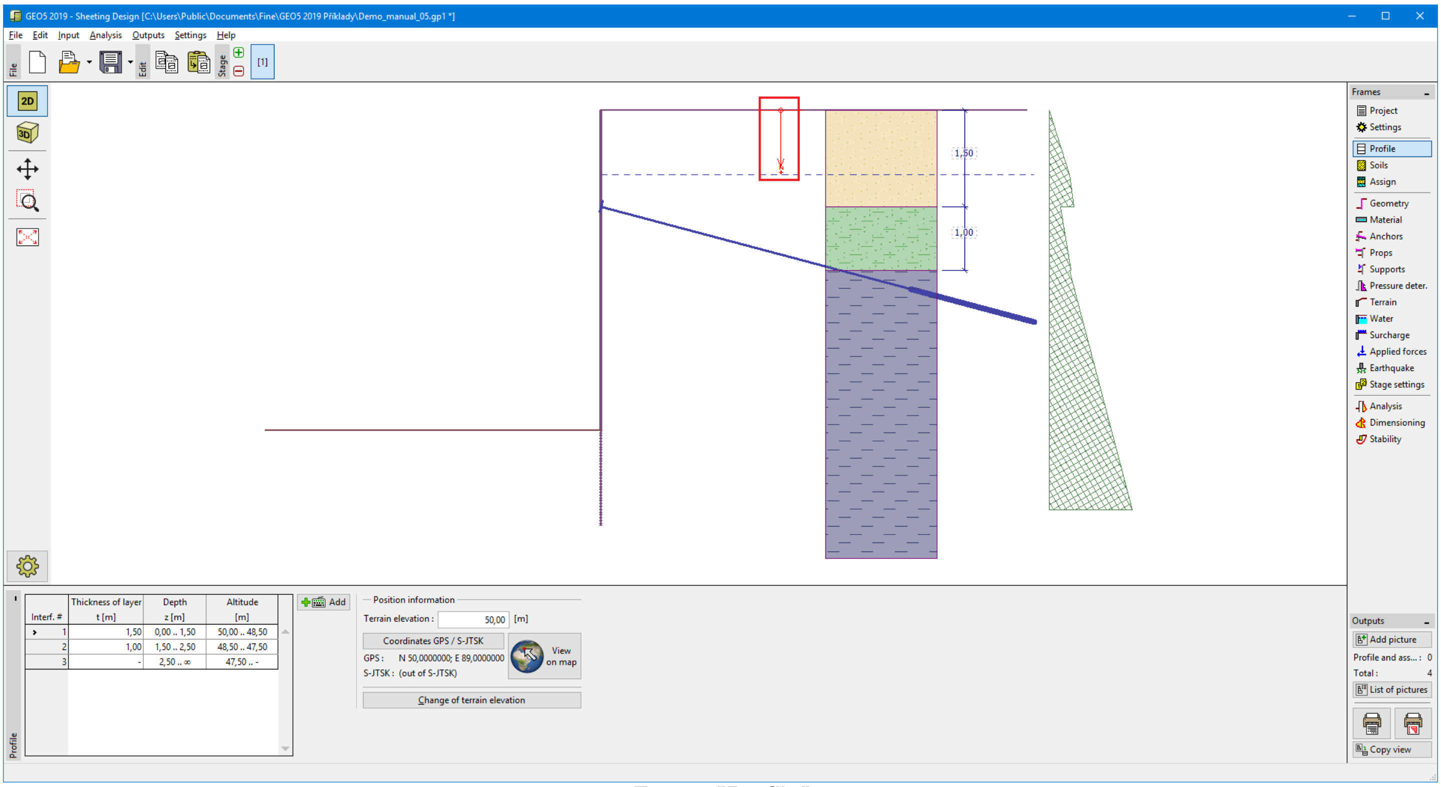
The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.



Frame "Profile"

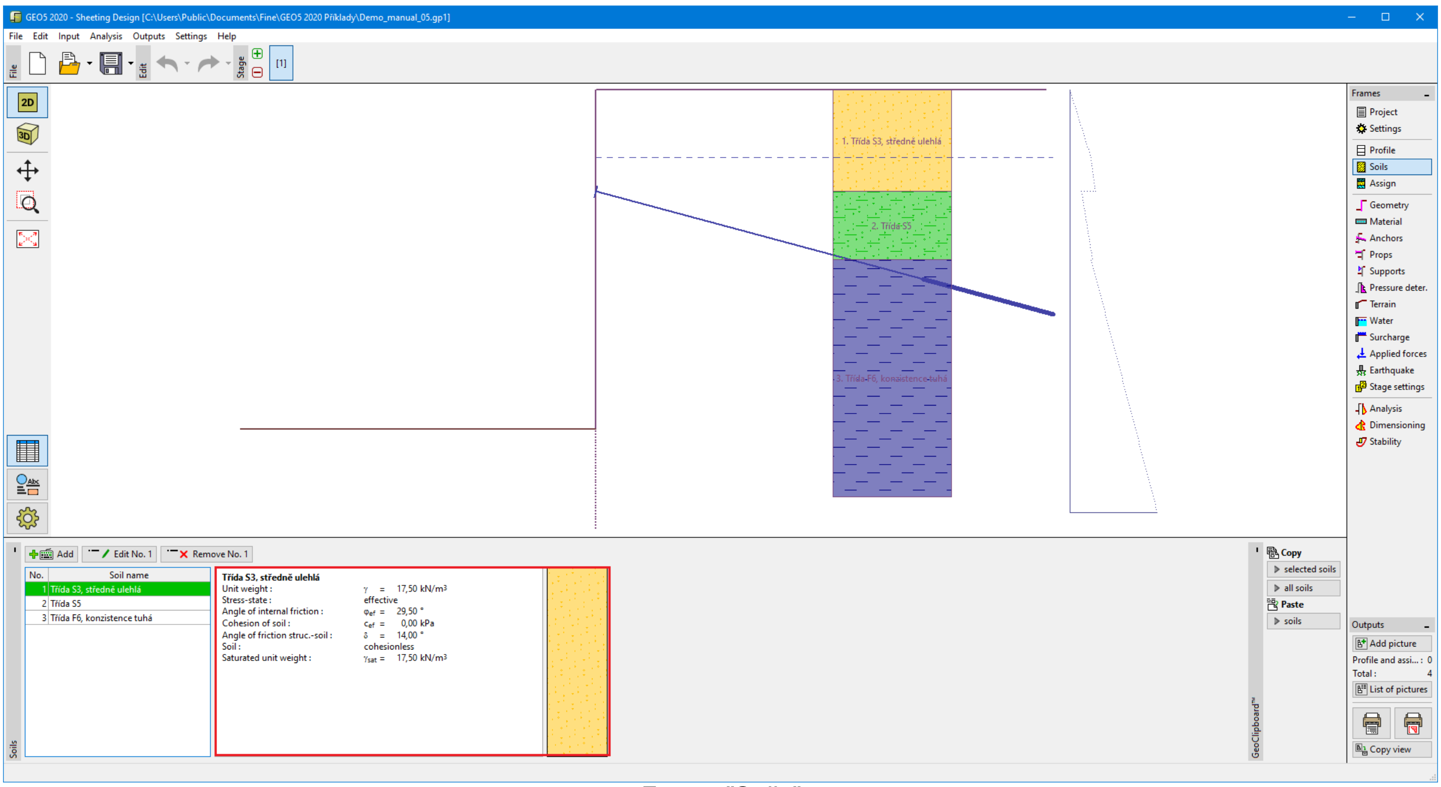
Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

Basic Data

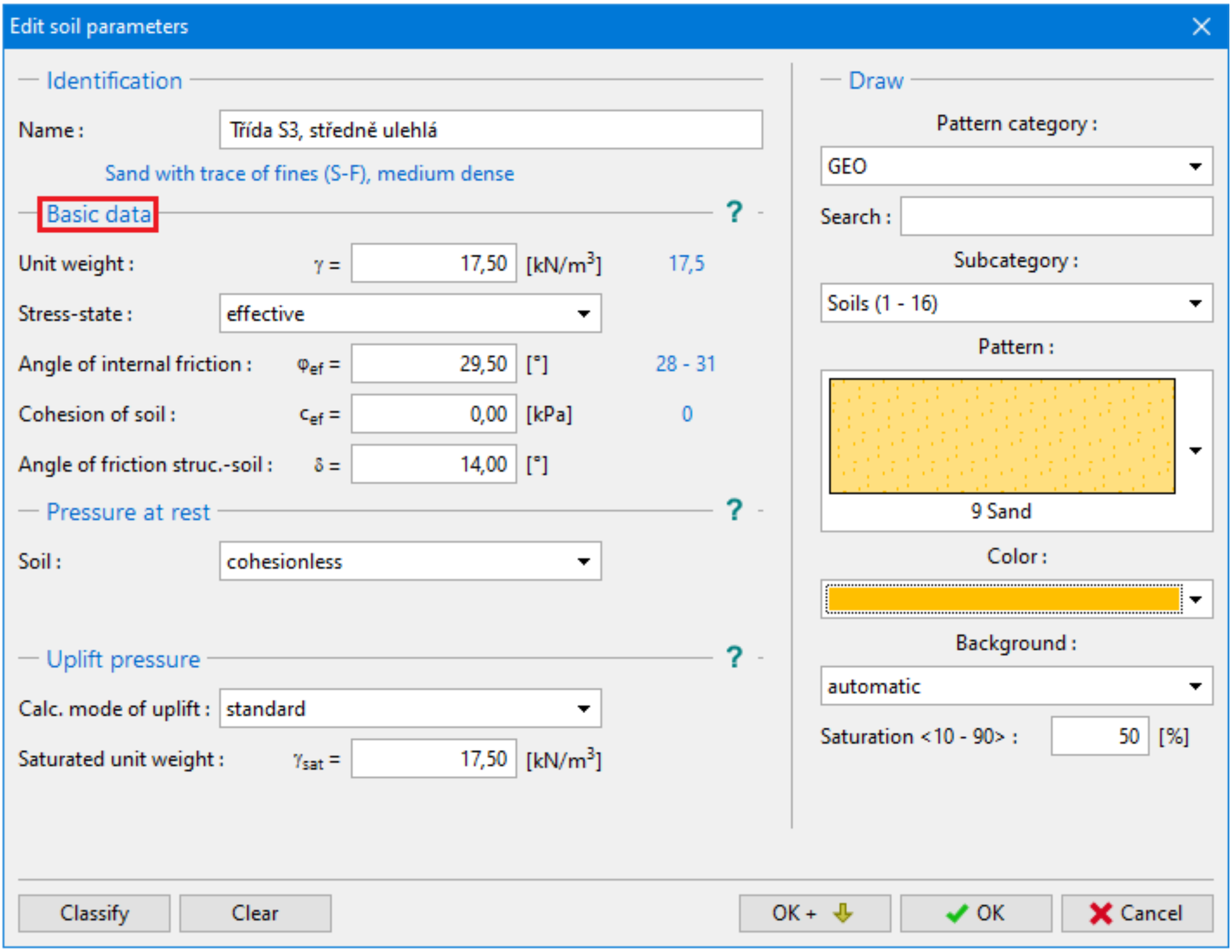
This part of the window allows us to introduce basic parameters of the soils - **unit weight, angle of internal friction, and cohesion**. The particular values are obtained from a geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit the built-in soils database, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the built-in database, these parameters must be defined manually.

Either **effective or total** parameters of the angle of internal friction and cohesion are specified depending on the settings in the "**Stress analysis**" combo list. Whether to use effective or total parameters depends primarily on the type of soil and load, structure duration, and water conditions.

For effective stress, it is further needed to specify the angle of internal friction between the soil and the structure, which depends on the structure material and type of soil. Possible values of this parameter are listed in the table of recommended values.

For total stress, it is further needed to specify the adhesion of soil to the structure face a .

The associated theory is described in detail in the chapter "Earth pressures".

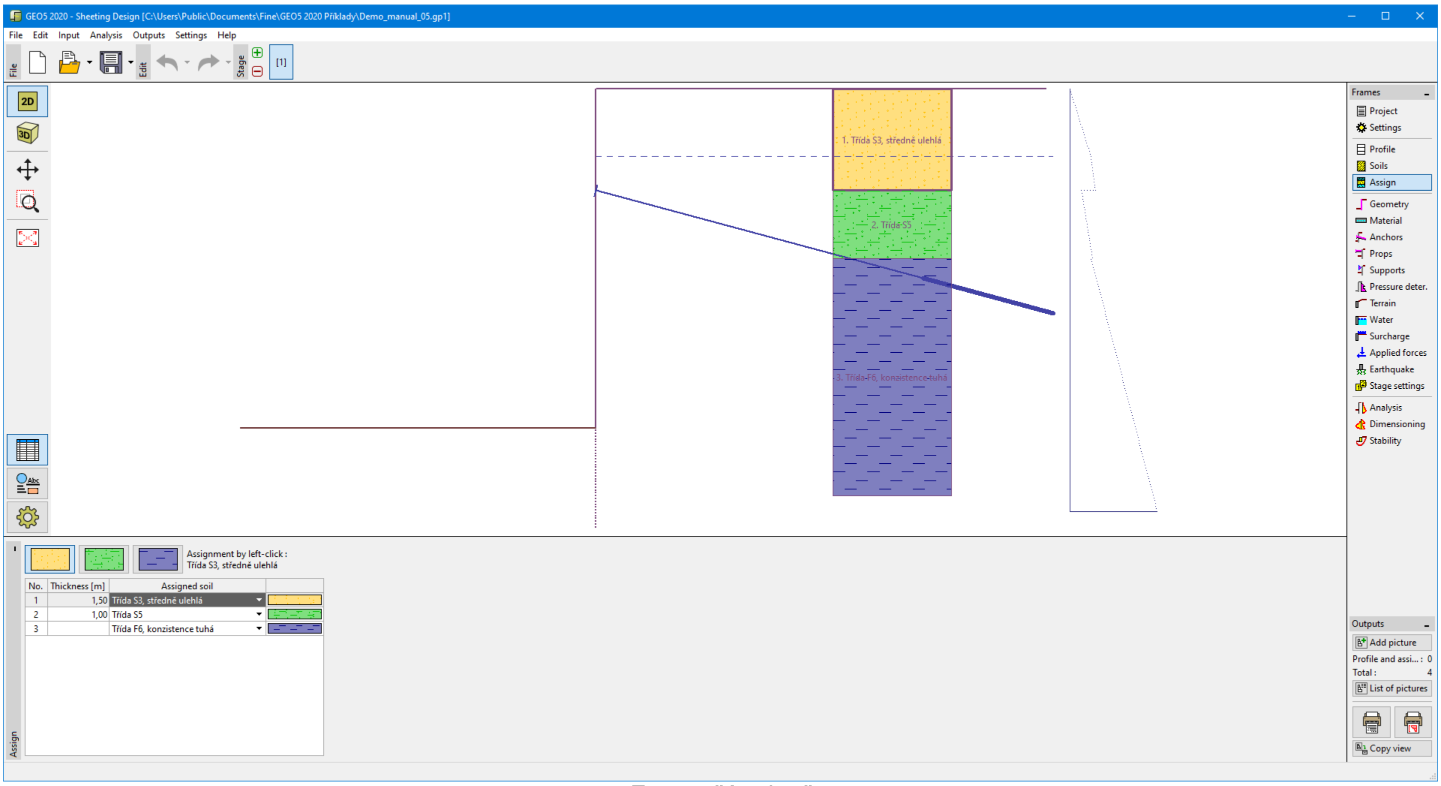


Dialog window "Add new soils" - "Basic data"

Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail herein.



Frame "Assign"

Geometry

The "**Geometry**" frame is used to specify the depth of a construction ditch and shape of the terrain in front The selected shape with a graphic hint appears in the left part of the frame. The dimensions of a structure can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

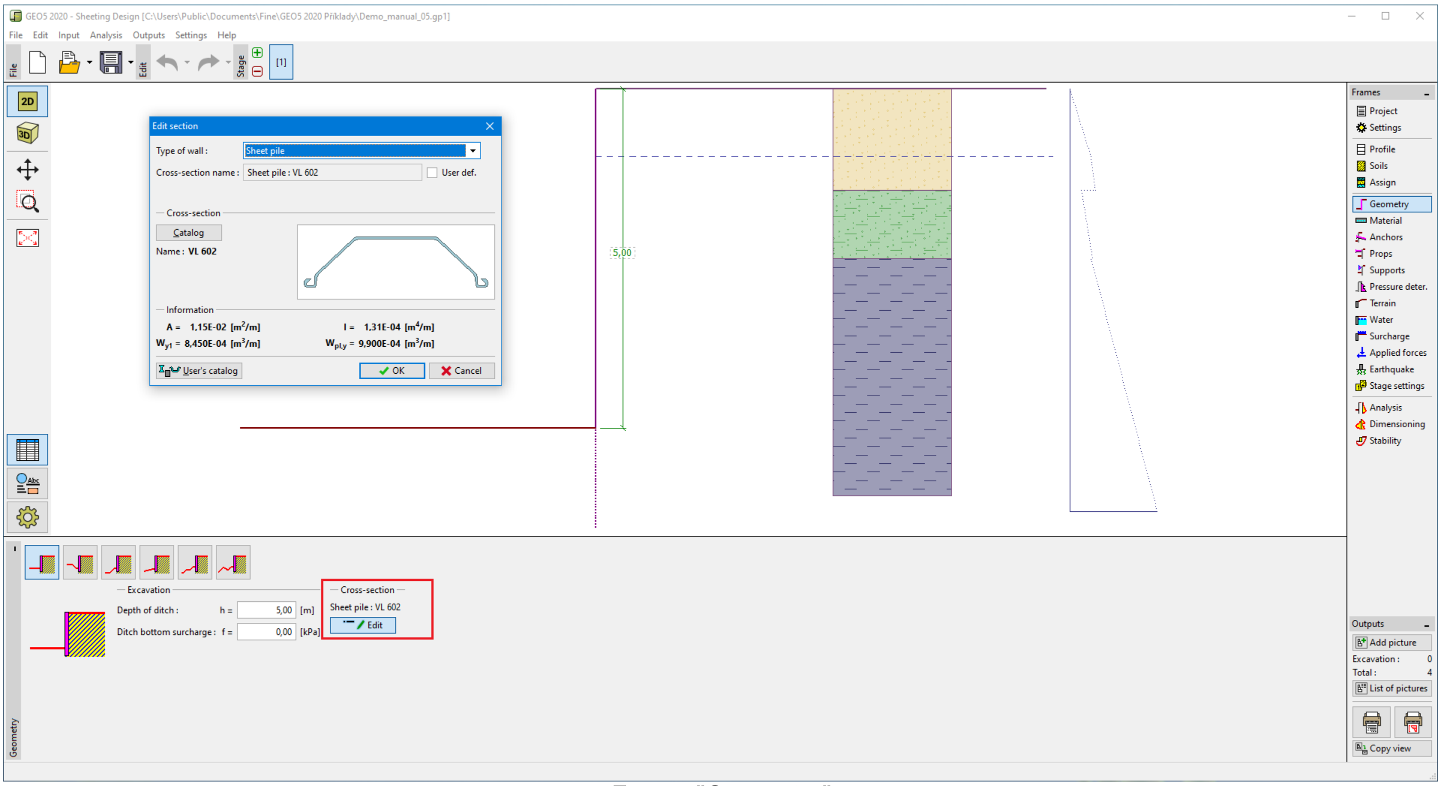
The frame can be further used to input surcharge of a construction ditch bottom.

The cross-section type is also entered in this frame. When loading data from older version "**Sheeting Design**" program, the cross-section is entered as "**not input**". Editing cross-section is performed using the "**Edit**" button.

The cross-section type does not affect the analysis of internal forces.

Results of the assessment of cross-section are shown in the frame "Dimensioning".

The program makes it possible to export the geometry of a structure in the *.DXF format.



Frame "Geometry"

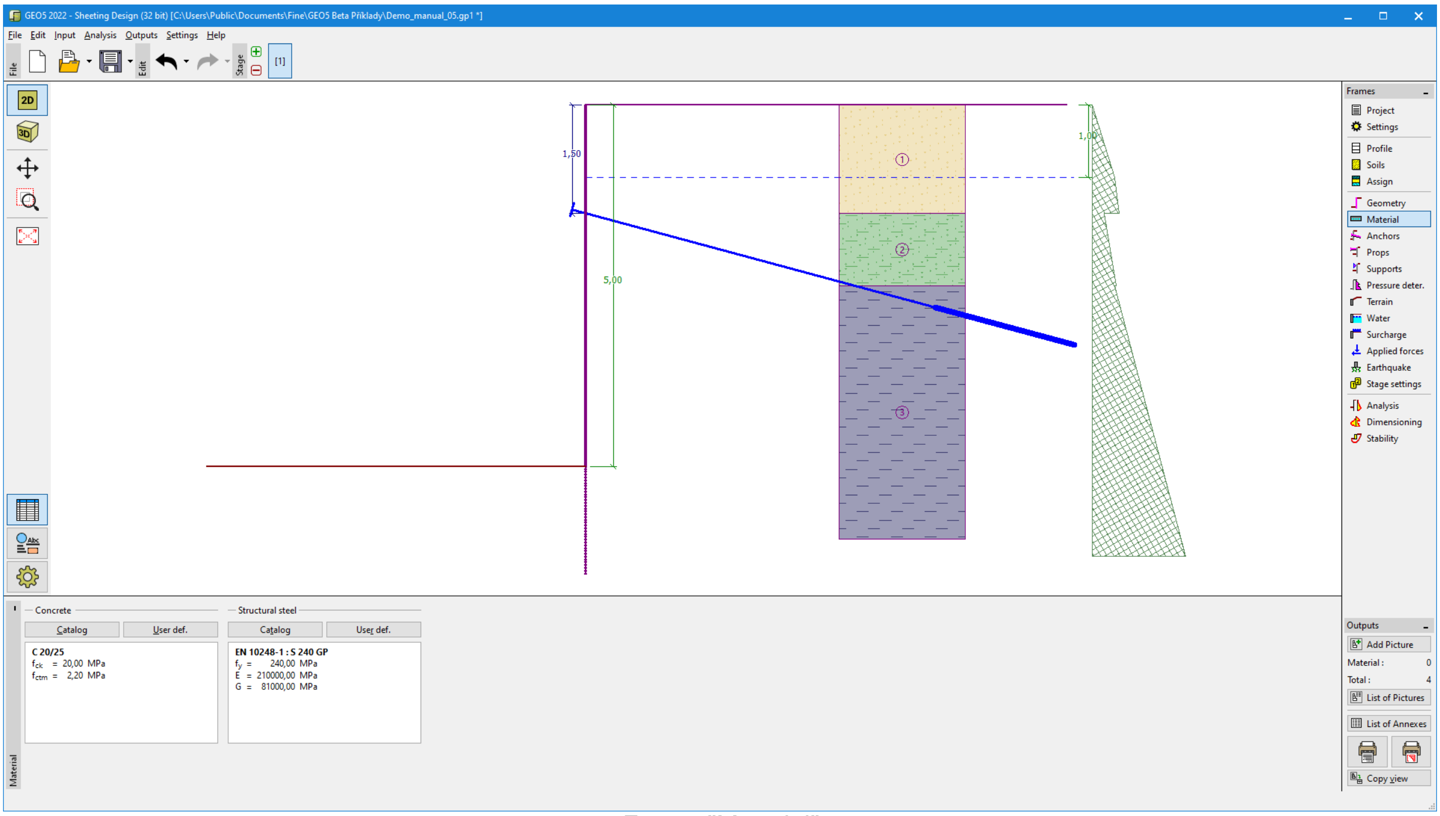
Material

The frame **"Material"** allows us to enter material parameters. The appearance of the frame varies according to the selected material (concrete, steel, timber, plastic, soil mix) in the frame "Geometry".

Two options can be used for setting the parameters of the material:

- The "**Catalog**" button opens the "**Catalog of materials**" dialog window, where the required material can be selected.
- The "**User-defined**" button opens the "**Editor of material**", which allows us to input the specification of material parameters manually by the user.

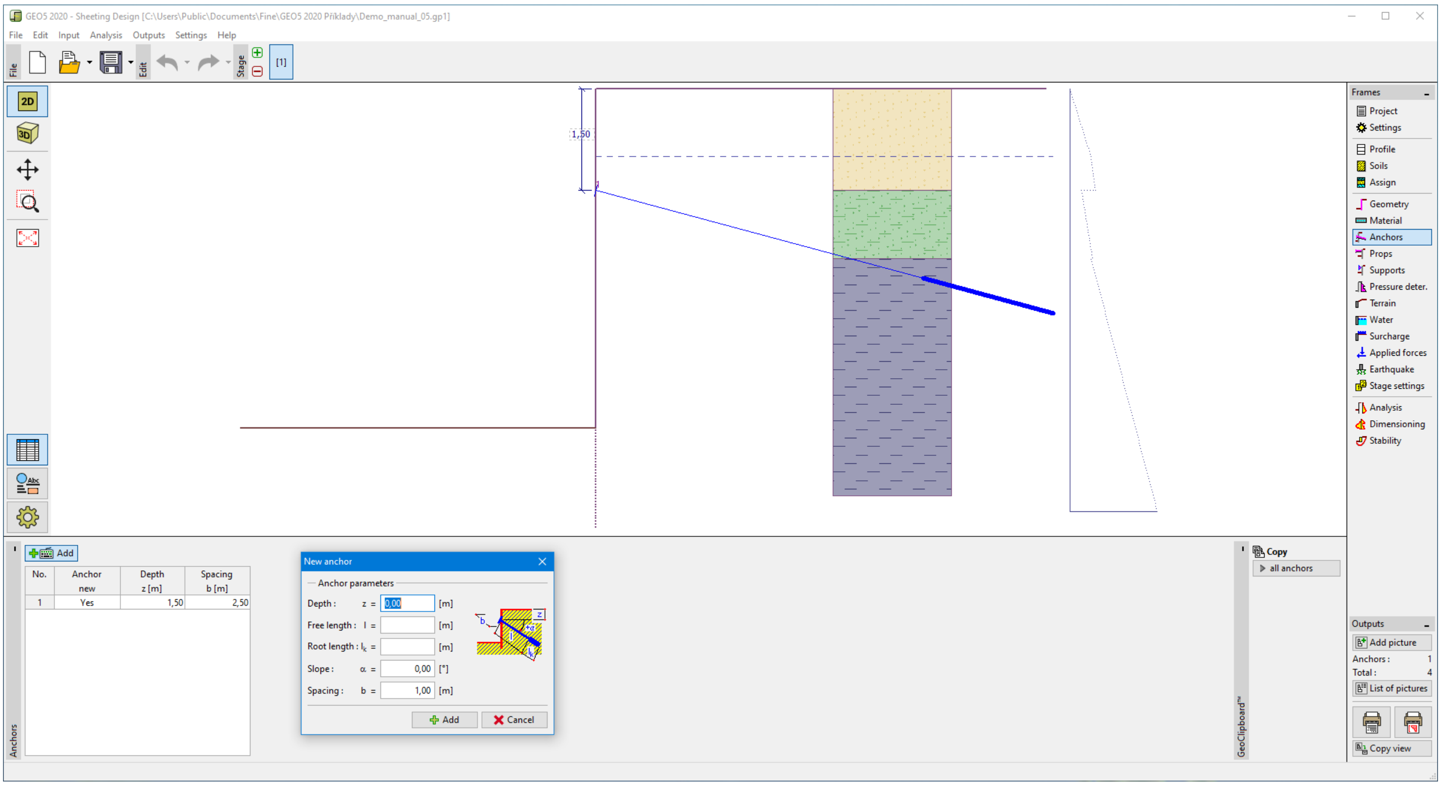
The content of catalogs depends on the selection of relevant standard for the **dimensioning of concrete, steel**, or **timber** or structures set in the "Materials and standards" tab.



Frame "Material"

Anchors

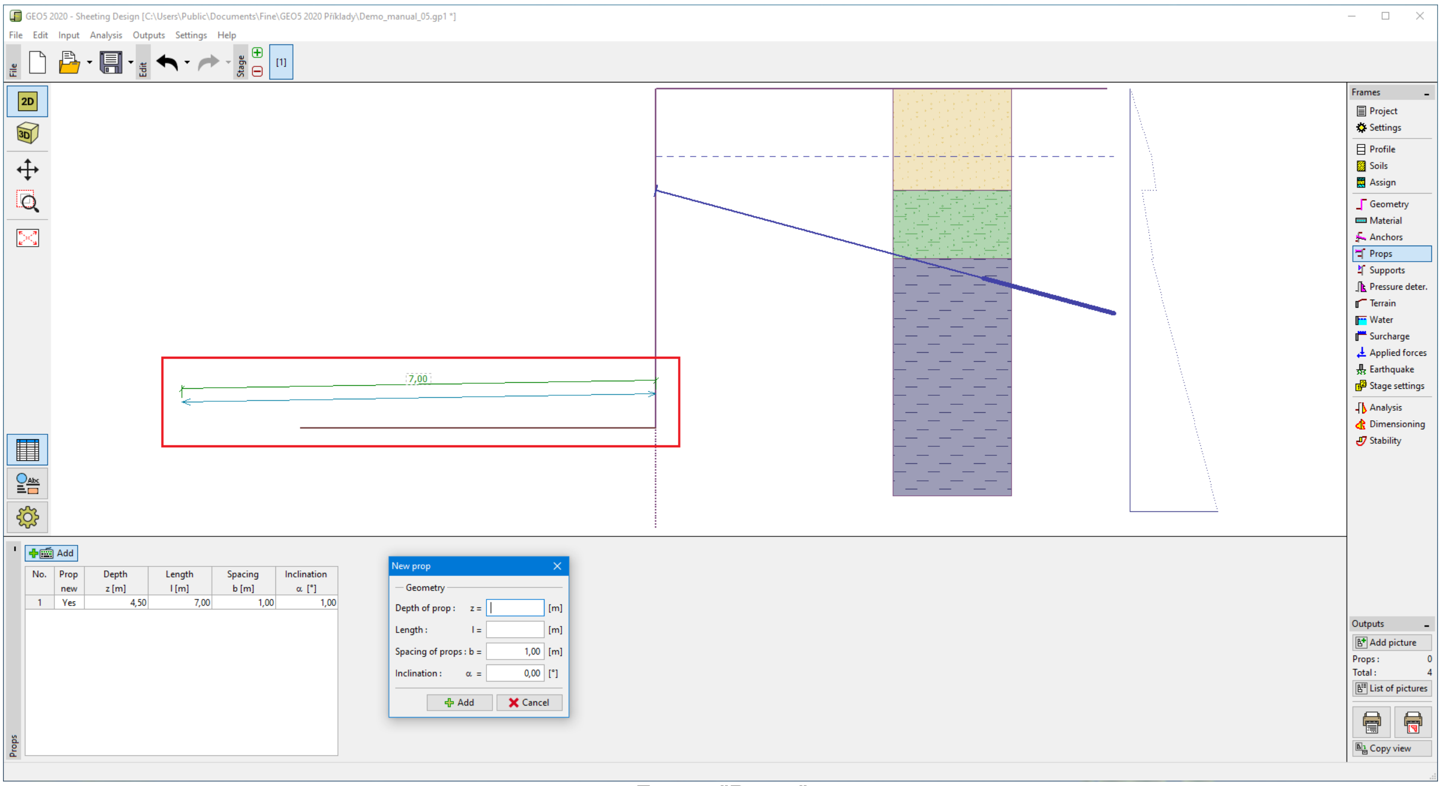
The "**Anchors**" frame contains a table with a list of input anchors. Adding anchors is performed in the "**New anchor**" dialog window. The input anchors can be edited on the desktop with the help of active objects.



Frame "Anchors"

Props

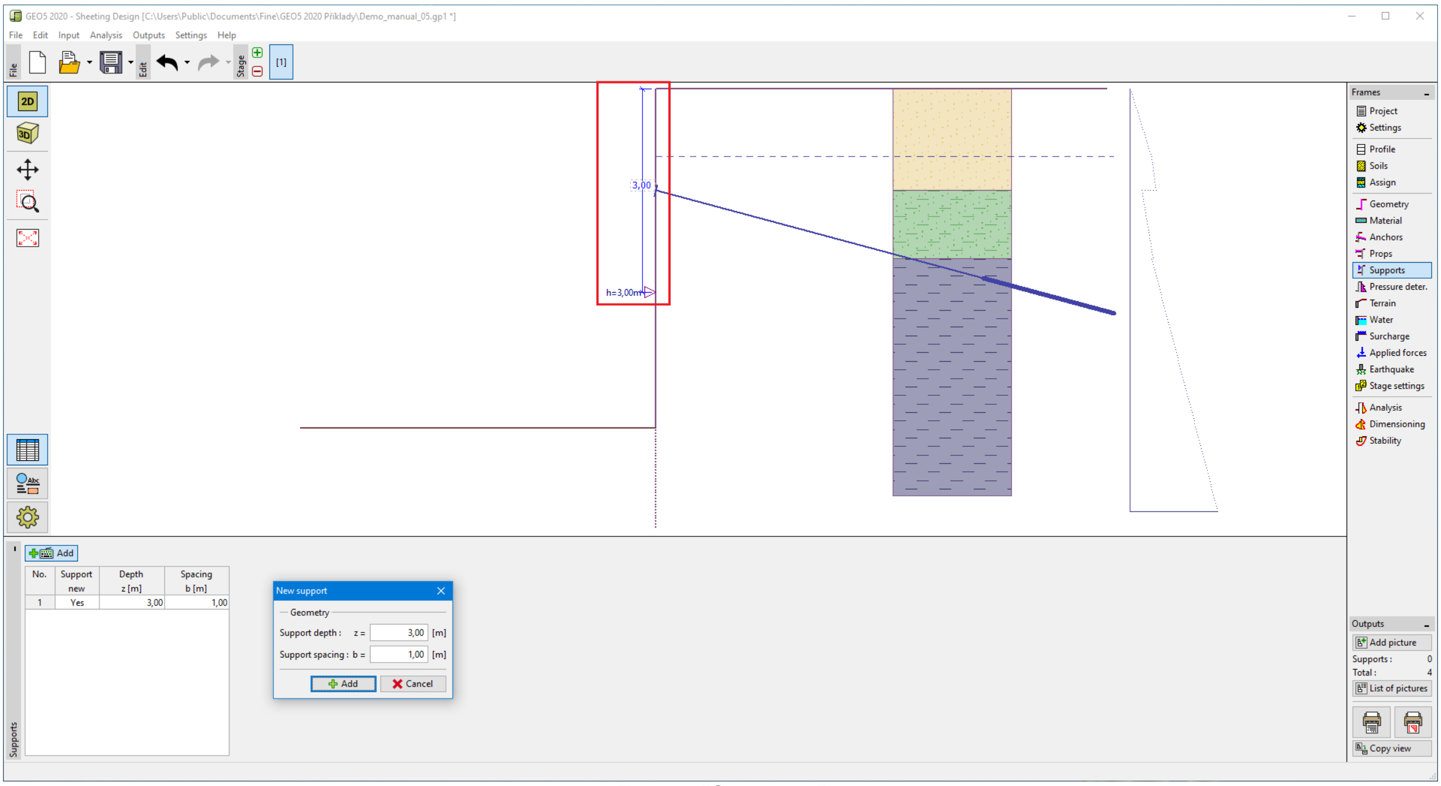
The "**Props**" frame contains a table with a list of input props. Adding props is performed in the "**New prop**" dialog window. The input props can also be edited on the desktop with the help of active dimensions or active objects, respectively.



Frame "Props"

Supports

The "**Supports**" frame contains a table with a list of input supports. Adding supports is performed in the "**New support**" dialog window. The input supports can also be edited on the desktop with the help of active dimensions or active objects, respectively.

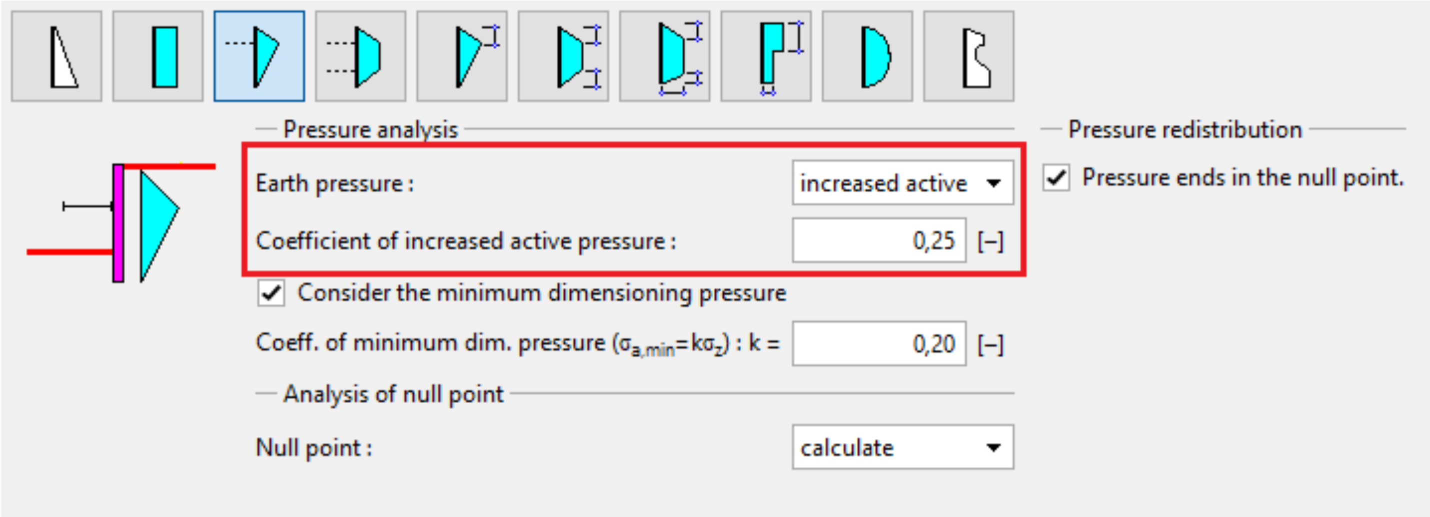


Frame "Supports"

Pressure Determination

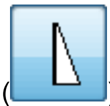
The "**Pressure Determination**" frame is used to specify the earth pressures acting on the wall.

The type of pressure acting on the wall can be specified as **active**, at **rest**, or **increased active -** according to the allowable wall deformation. It is also possible to define minimum dimensioning pressure, which is required by many standards.



Input of increased active pressure

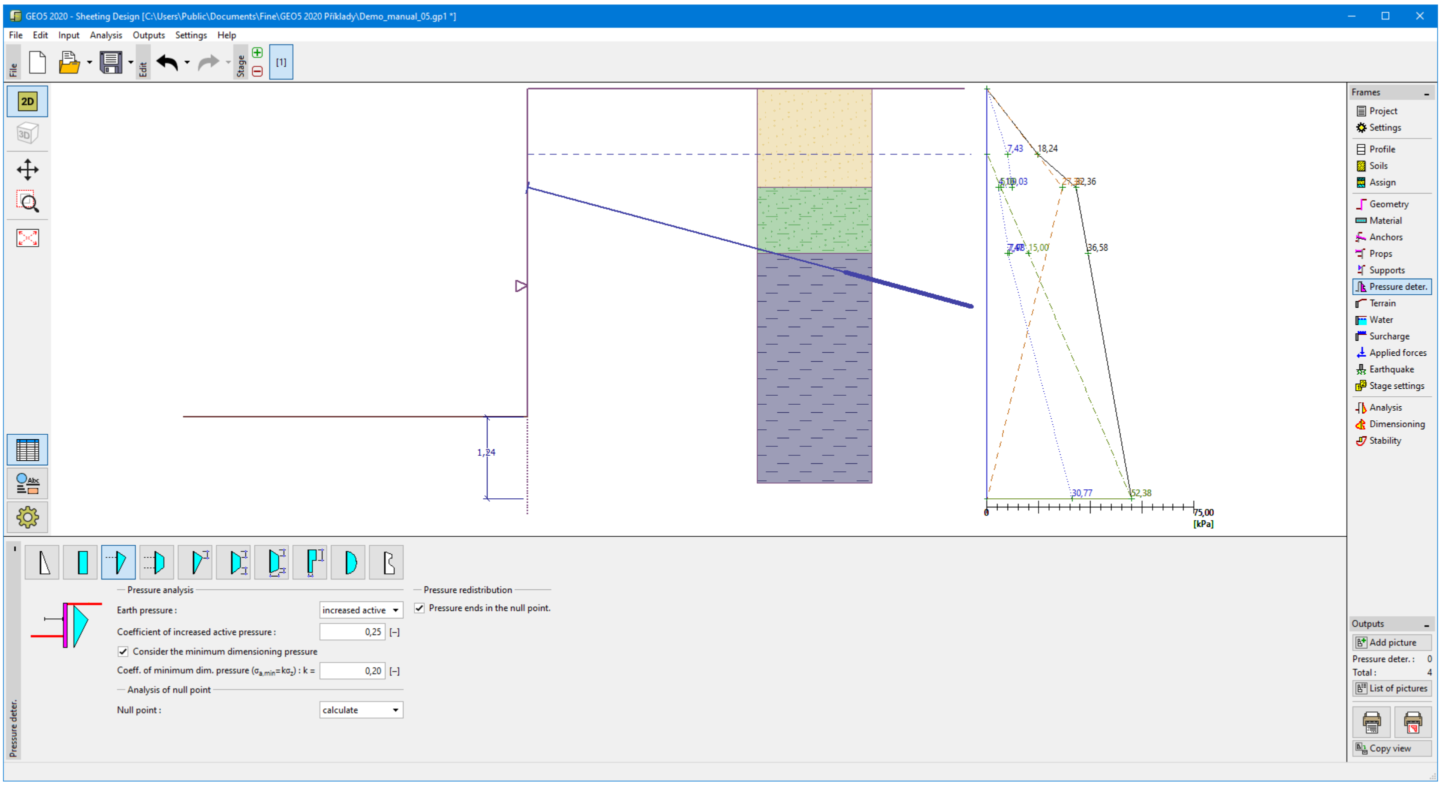
The upper bar with buttons determines whether the calculated earth pressure is redistributed.

Option 1 is used for **non-anchored walls** () - there is no redistribution of pressure.

For anchored and strutted walls, it is appropriate to use a **redistribution of earth pressures**. Additional options (buttons) are for various possibilities of pressure redistribution.

The last option () allows us to **input any distribution** of final earth pressure manually - this pressure does not depend on input geological profile.

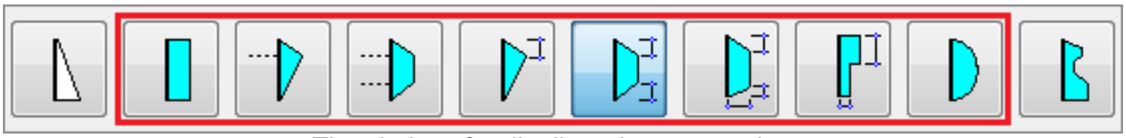
The program allows us to enter a position of a **null point**, but it is recommended to let it calculate automatically.



Frame "Pressure determination"

Redistribution of Earth Pressures

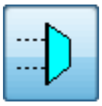
The program allows us to choose the most common **shapes of redistributed pressure**.

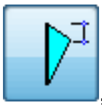
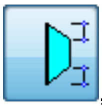
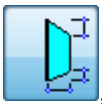


The choice of redistributed pressure shape

Practices in using redistributions are very different by country. The program contains both classical schemes of redistribution, then shapes according to the standards EAB, AASHTO.

The program is designed to be completely universal and the user can redistribute the pressure according to his habits.

For selections ,  the program calculates a position of maximal pressure automatically according to the location of anchors (props). This variant is the easiest for entering, and it is also recommended.

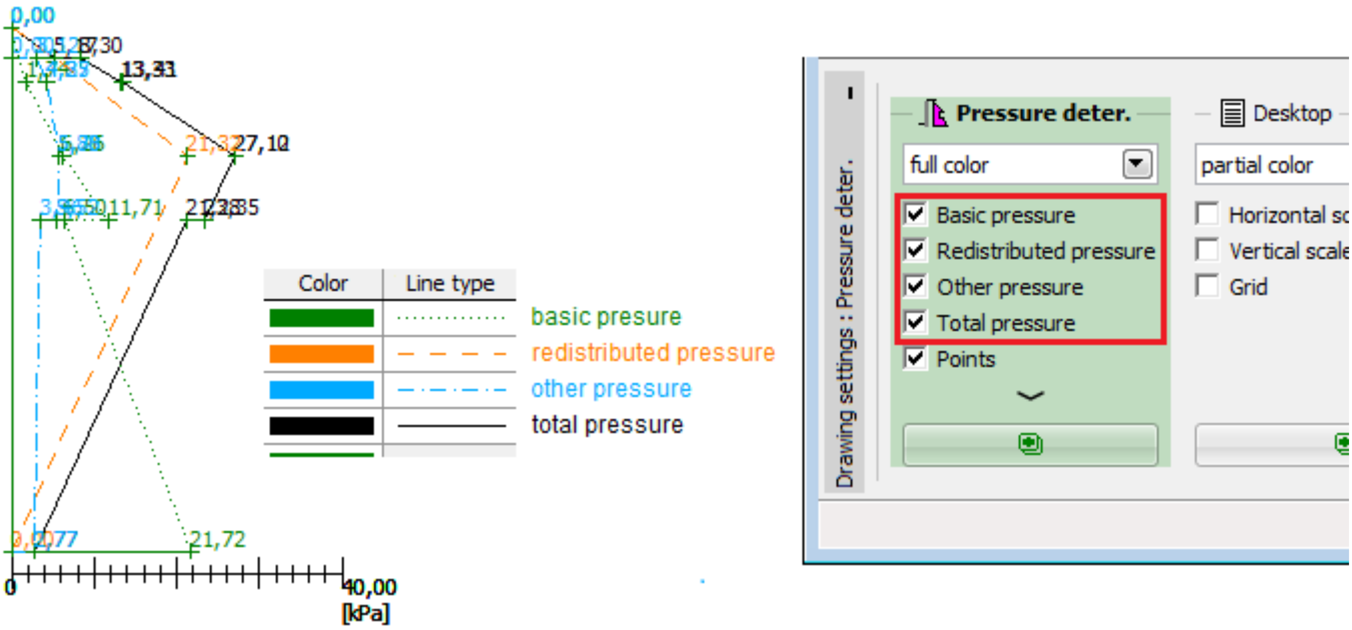
For selections , , ,  it is necessary to enter coefficients (x_1 to x_6), which determine the final shape of redistributed pressure.

The user can also specify, whether redistributed pressure acts to the **null point** (standard procedure), or to the ditch bottom (EAB).

The procedure of calculating the pressure on the structure is as follows:

- the position of a **null point** is calculated (the magnitude of passive pressure in front of structure is the same as a magnitude of earth pressure behind the wall - the total sum of the pressure at this point is zero)
- **basic pressure** behind the structure is calculated (active, at rest, **increased active**) to the depth of null point. The minimum dimensioning pressure can be considered.
- **redistributed pressure** with the same area as the area of basic pressure is calculated. Only a basic pressure (calculated from the geological profile) is redistributed. The pressure from surcharge or water pressure are not redistributed.
- the pressure from surcharge, water or earthquake is added to the redistributed pressure (**total pressure**)

All pressures are drawn on the desktop:



Graphical representation of pressures

Visualization of pressures can be activated in the frame "Drawing settings". Settings of colors and types of lines can be edited in the dialog window "Drawing Styles Administrator".

Literature:

EAB - 2012 Ernst & Sohn A Wiley Company, 5.Auflage, Empfehlungen des Arbeitskreises "Baugruben".

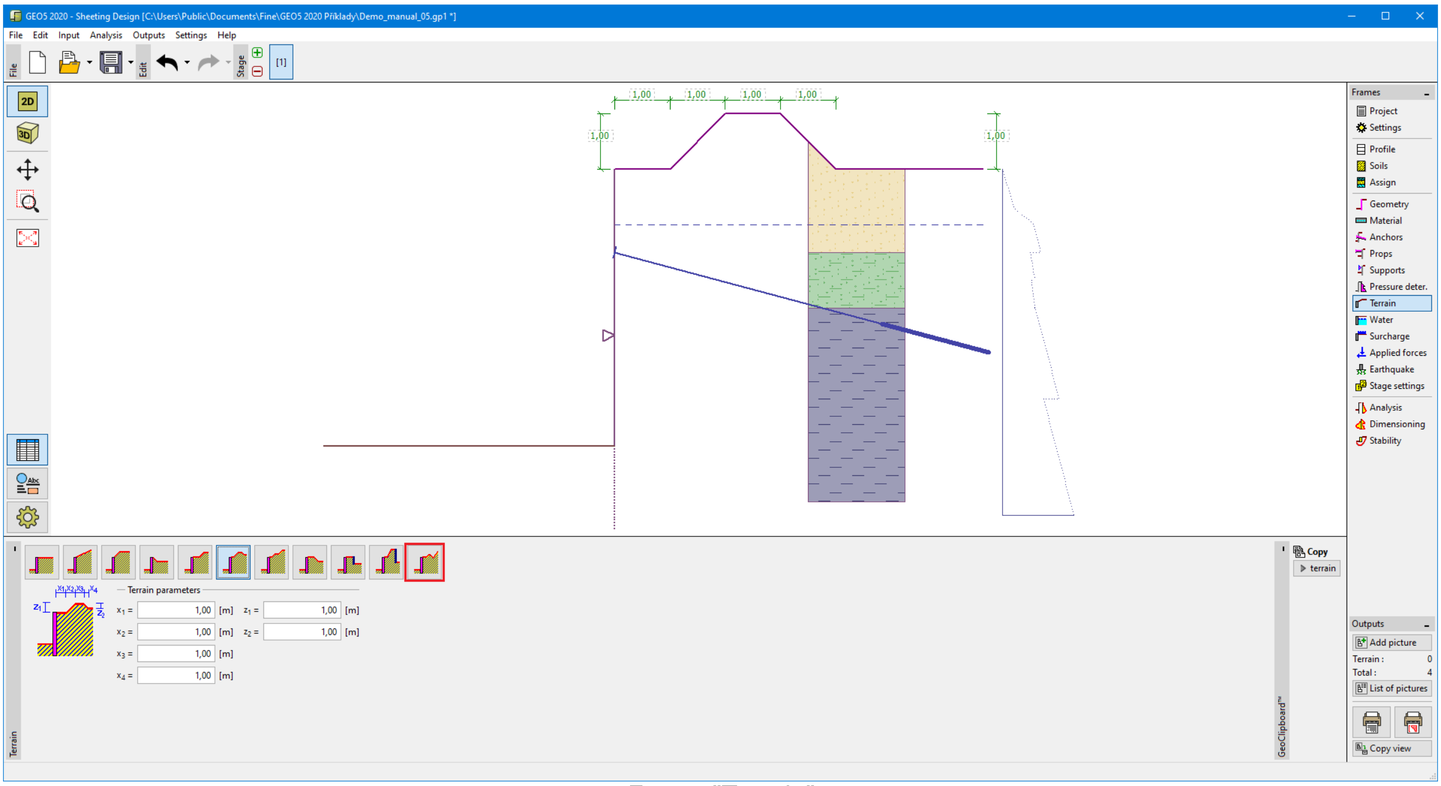
AASHTO LRFD Bridge Design Specifications, Seventh Edition, 2014.

Terrain

The "**Terrain**" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case, the frame contains a table with a list of terrain points. The first point with coordinates [0; 0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "Distribution of earth pressures for broken terrain".



Frame "Terrain"

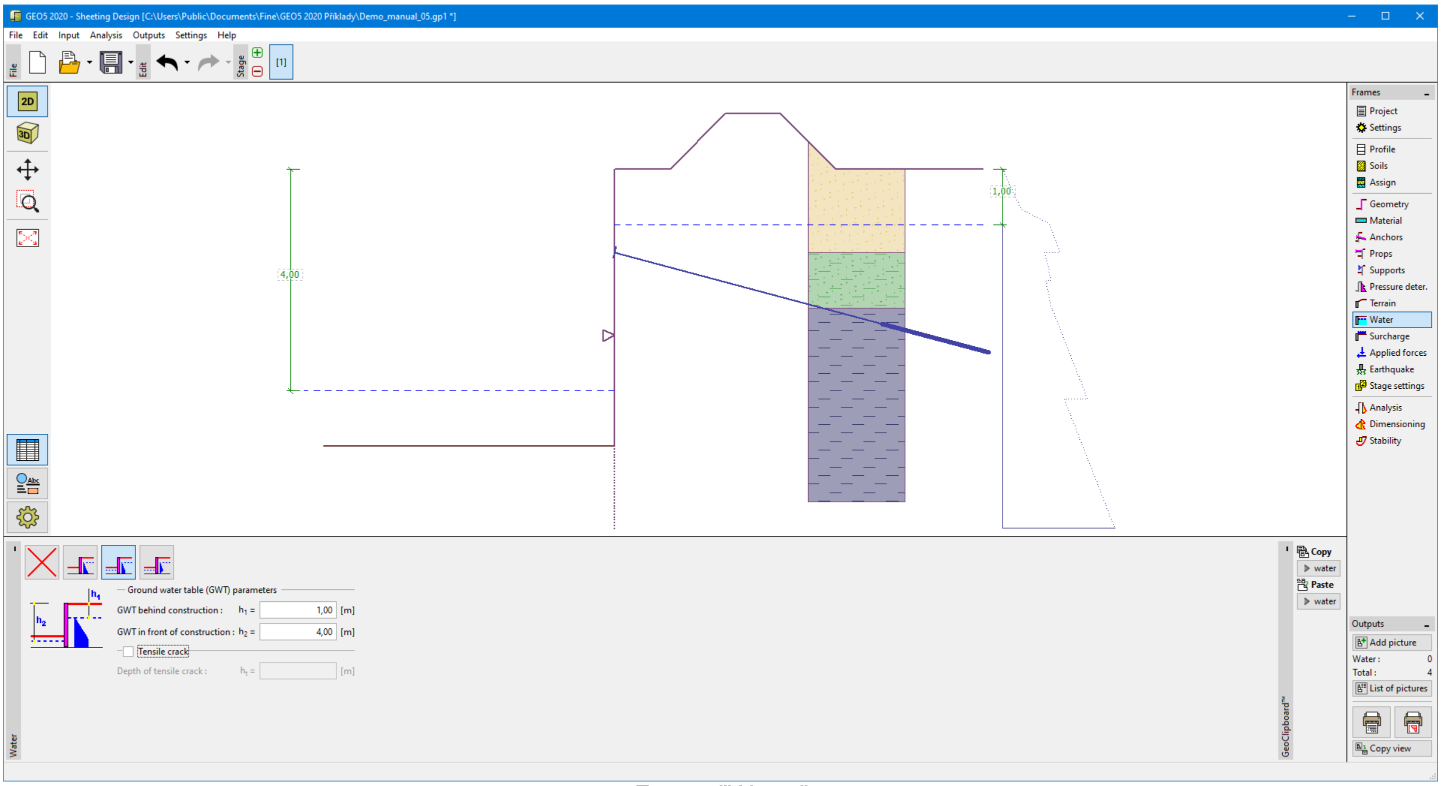
Water

The "**Water**" frame allows, by pressing the button, for selecting the type of water. The selected type with a graphic hint of input values is displayed in the left part of the frame. Water parameters (h_1 , h_2 ...) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The groundwater table can also be specified **above the structure** or earth profile, respectively - in such a case, the depth of water is input with a negative value.

Analysis of earth pressures with the influence of water is described in the theoretical part of the help, chapter "Influence of water".

The program further allows for specifying a depth of tensile cracks filled with water.



Frame "Water"

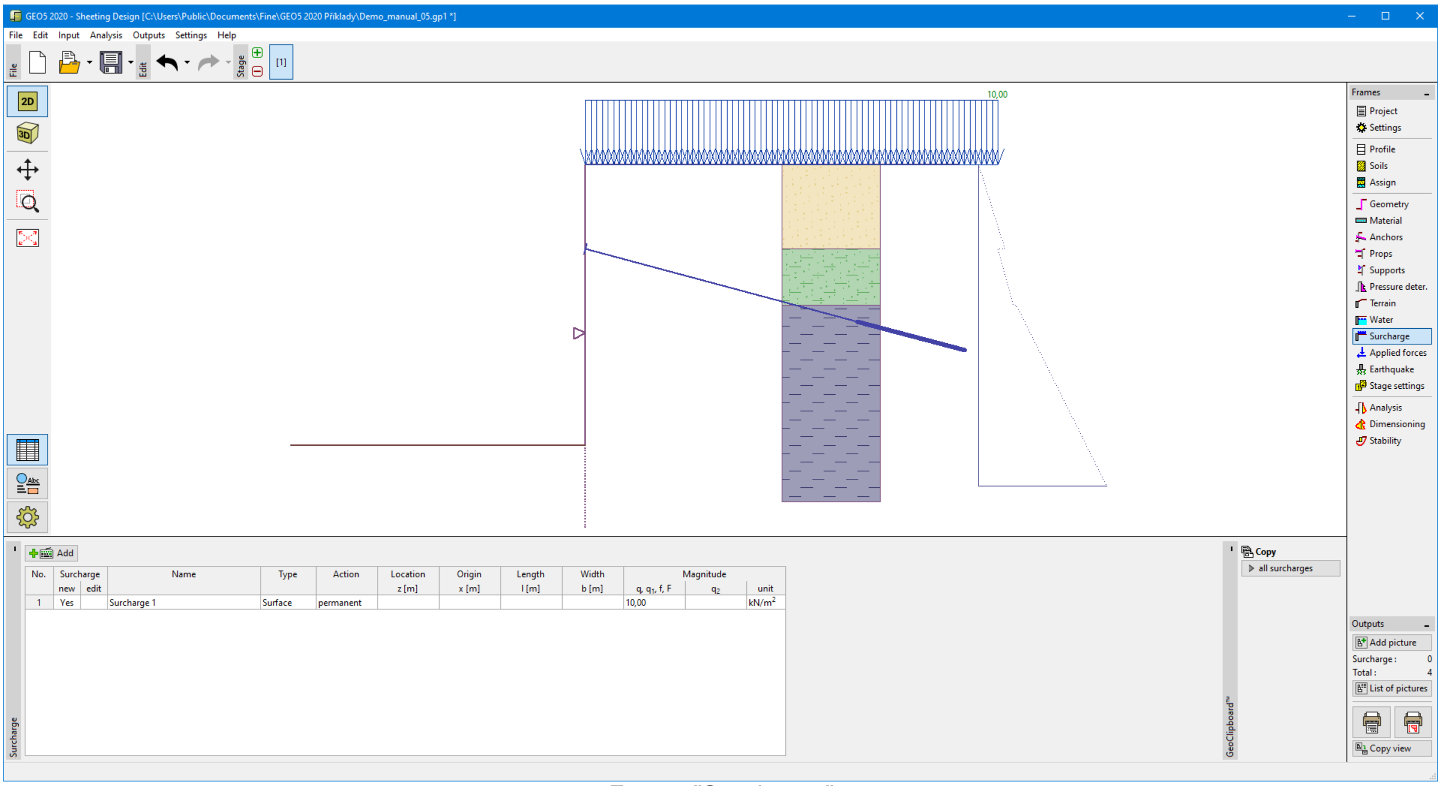
Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The z -coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case when the surcharge is found out of the terrain the program prompts an error message before calculation.

Either **permanent**, **variable,** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with a favorable effect is not considered in the analysis.

Analysis of earth pressures due to surcharge is described in the theoretical part of the help, chapter "Influence of surcharge".



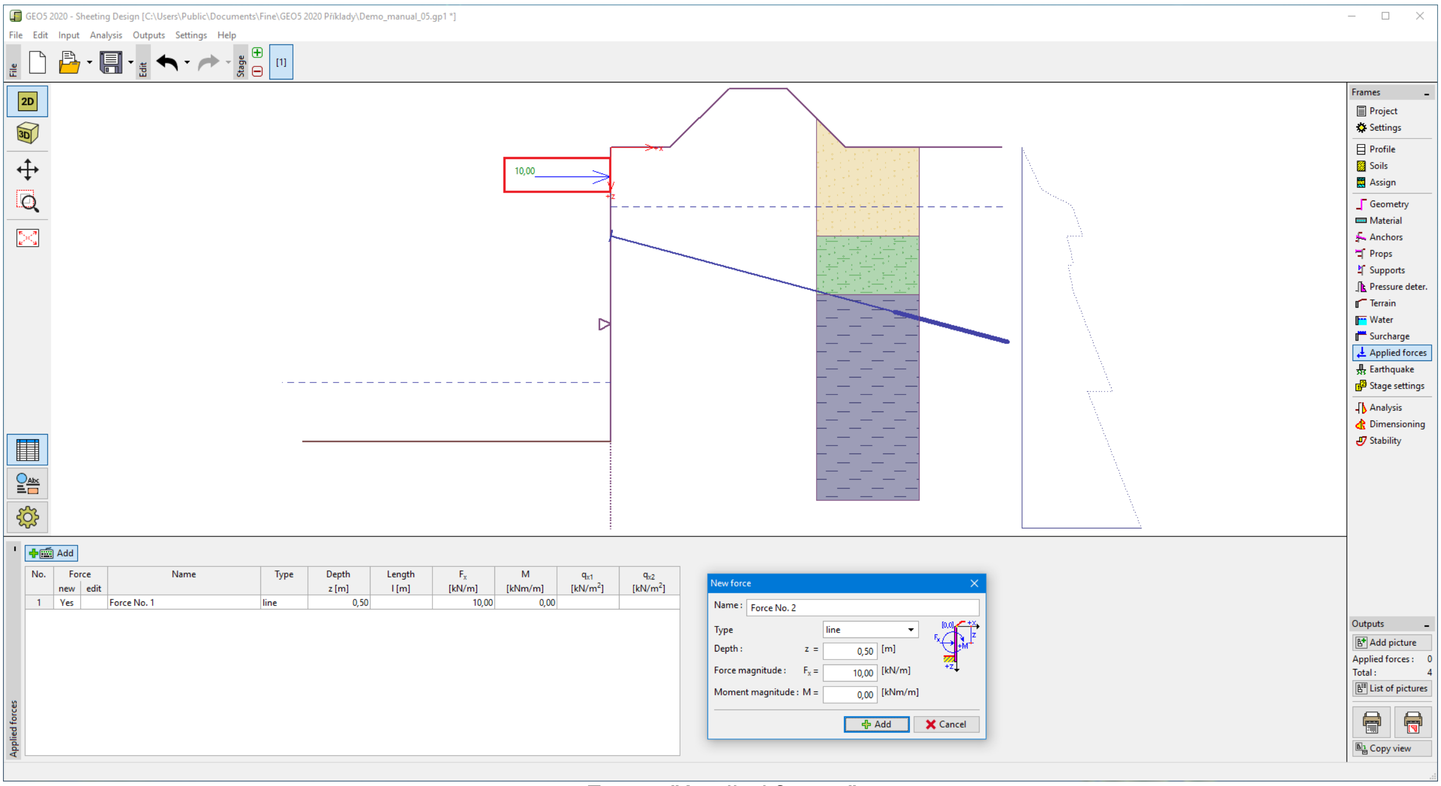
Frame "Surcharge"

Applied Forces

The "**Applied forces**" frame contains a table with a list of forces acting on a structure. Adding forces is performed in the "**New force**" dialog window. The input forces can also be edited on the desktop with the help of active objects.

Applied forces represent an additional load on the structure of the wall, sheeting or MSE wall. We can model such as an anchoring crash barrier, crash vehicle, load from billboards and hoardings etc. The program does not modify the applied forces in the calculation except multiplying them with the corresponding coefficients according to the selected type of load (EN1997, LRFD).

External load acting on the terrain is necessary to define as a surcharge.



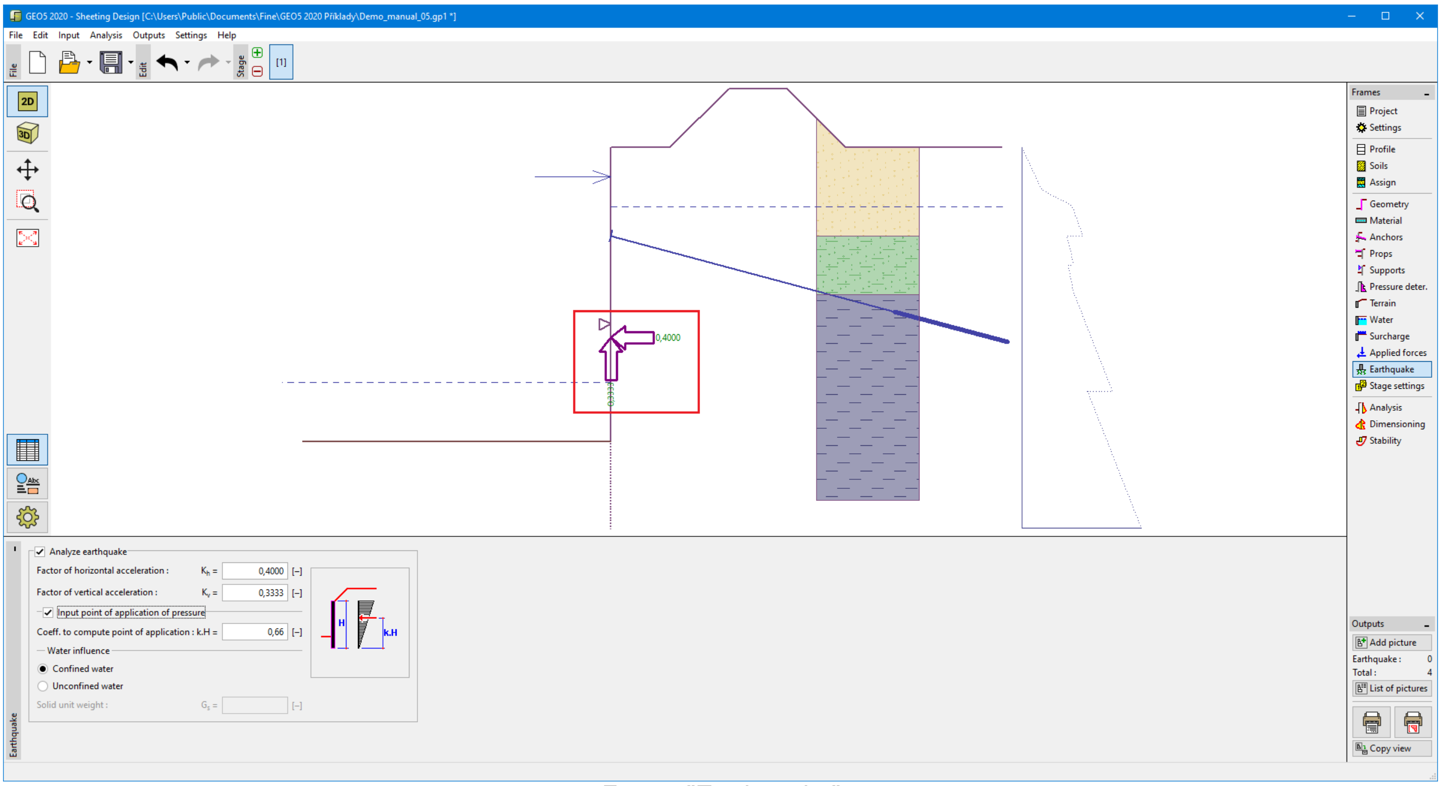
Frame "Applied forces"

Earthquake

The "**Earthquake**" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements, the coefficients k_h and k_v can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for an earthquake is described in the theoretical part of the help, chapter "Influence earthquake".



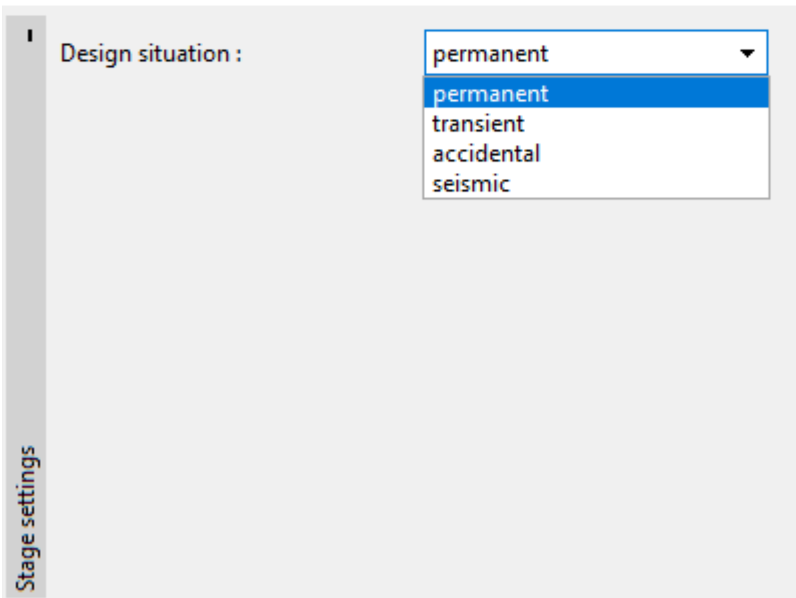
Frame "Earthquake"

Stage Settings

The frame "**Stage settings**" serves to input settings valid for a given construction stage.

The selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology.



Frame "Stage settings"

Analysis

The "**Analysis**" frame displays the analysis results.

The frame has two variants. The first variant applies to a non-anchored wall (sheet pile) and the second one to an anchored (strutted) wall.

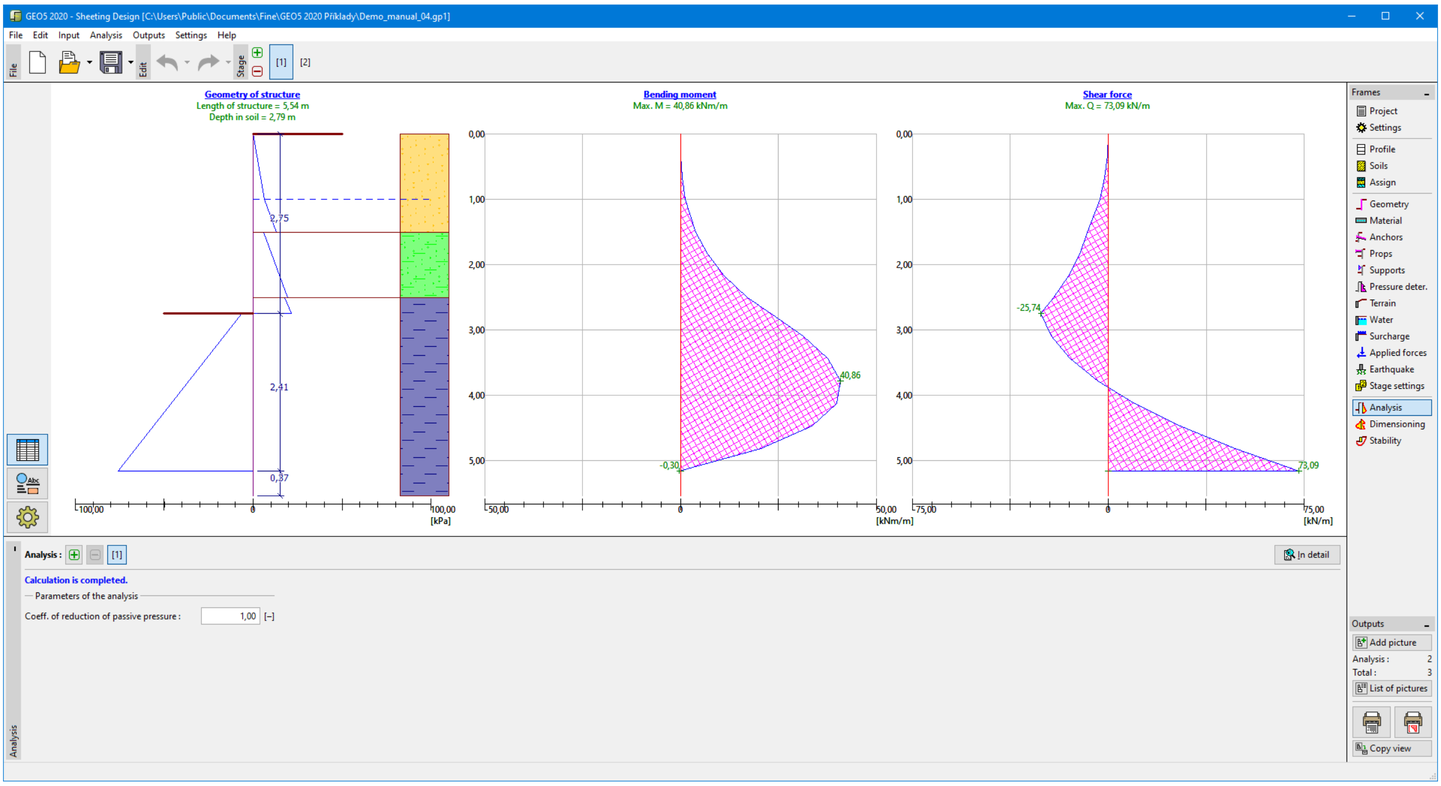
It is possible to enter a coefficient of reduction of passive earth pressure for both variants.

Using the verification methodology Limit States or Safety Factor, it is possible to specify factor of safety for a non-anchored wall.

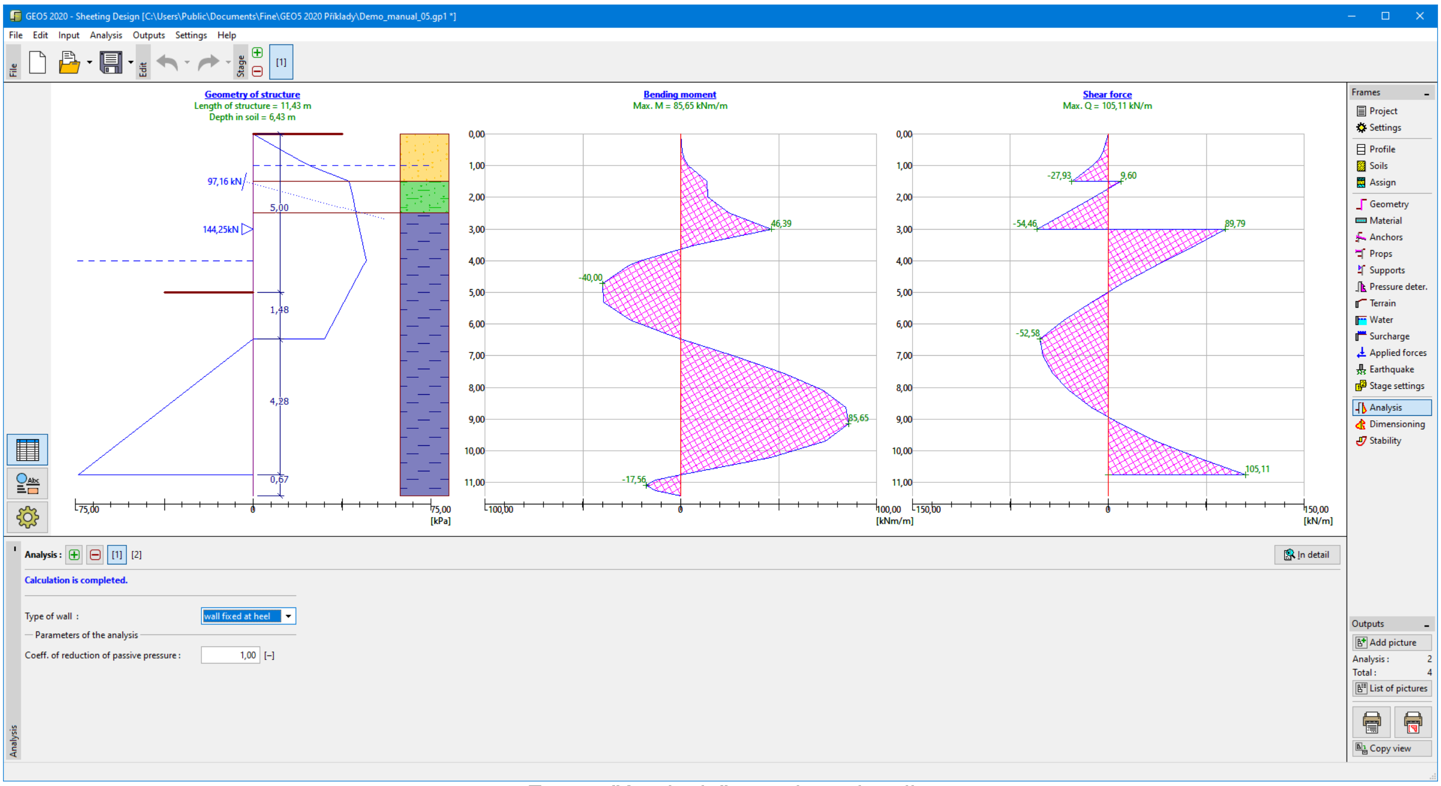
A type of heel support (fixed, free) is specified for an **anchored** wall.

When performing the analysis according to EN 1997 or LRFD the design coefficients are introduced in the "Pressure Analysis" tab. Providing the analysis according to Design approach 1, it is necessary to also enter the combination number.

The analysis results are displayed on the desktop. Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Analysis" - non-anchored wall



Frame "Analysis" - anchored wall

Dimensioning

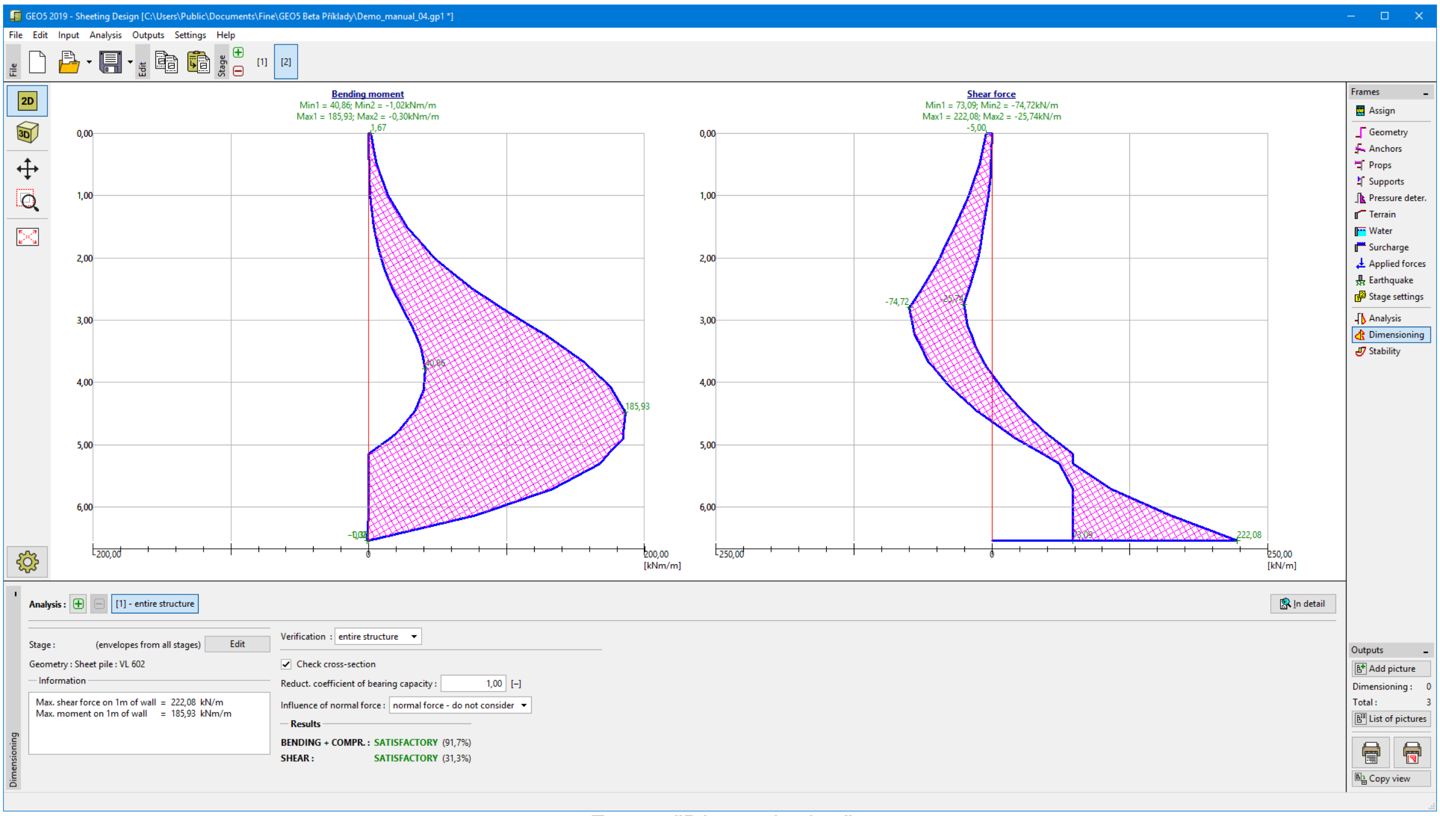
In the frame **"Dimensioning"**, it is possible to display an envelope of internal forces and displacements from all analyses and stages of constructions. Normally, the envelope is constructed from the results of all construction stages, however, it can only be created from the **selected stages**. The **"Modify"** button opens the dialog window **"Construction stage selection"**, where it is possible to select the constructions stages that are used to generate the current envelope (by pressing corresponding buttons).

The maximum values of calculated internal forces (bending moments and shear forces) are displayed at the bottom part of the frame.

The program allows us to check reinforced concrete, steel, timber and plastic cross-sections (by checking the option "**Check cross-section**"). For a more detailed design of reinforcement in concrete cross-section, it is possible to divide the structure into sections which are then assessed separately.

When checking the cross-section, it is possible to input the **reduction coefficient of bearing capacity**, which multiplies values of internal forces. For dimensioning of steel cross-sections, it is possible to assume **influence of normal force** in these ways:

- **normal forces - do not consider**: program doesn't consider influence of normal force
- **normal forces - input**: user-defined value of normal force N $[kN/m, lbf/ft]$



Frame "Dimensioning"

Stability

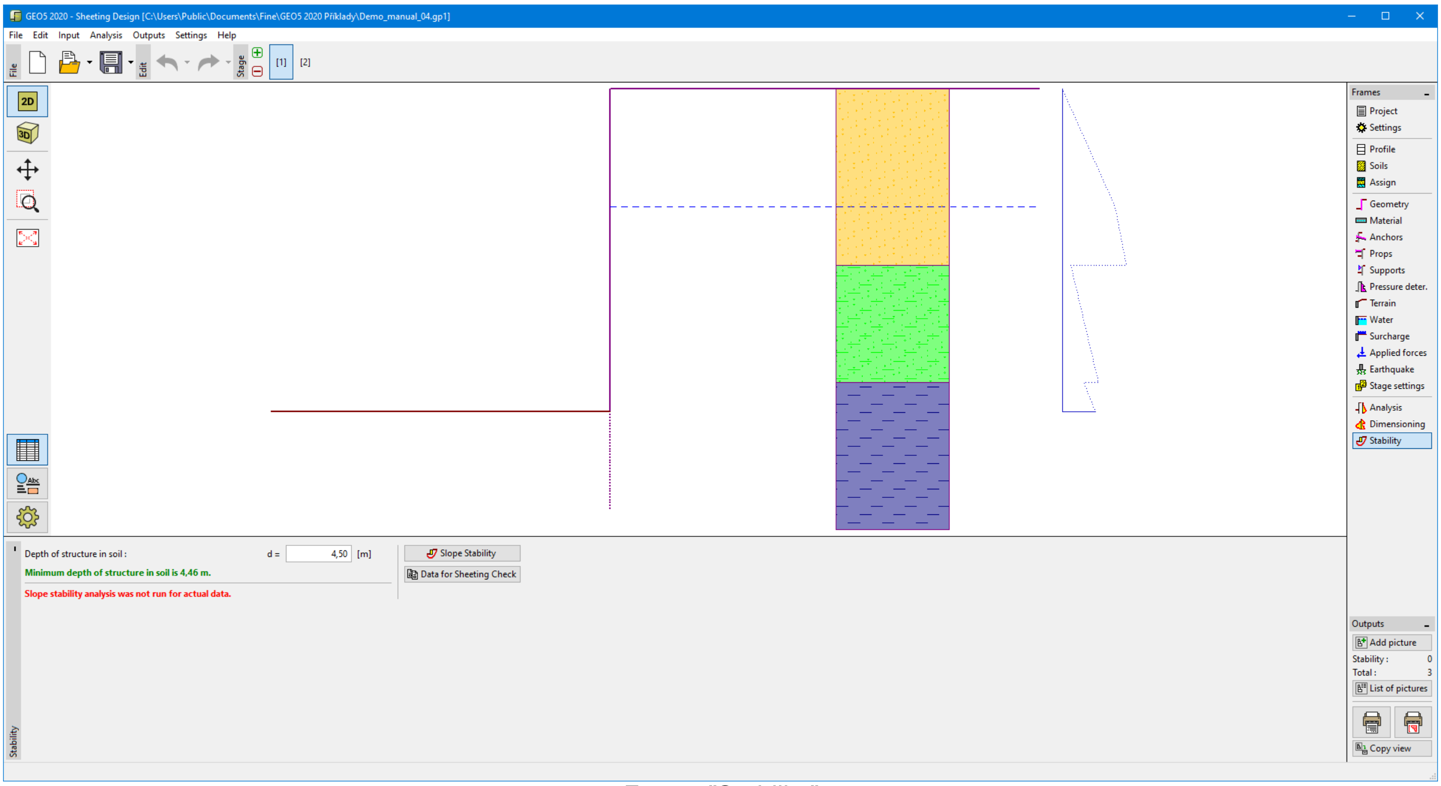
The "**Stability**" frame serves to export designed wall to other programs - "**Slope Stability**" (verification of global stability) and "**Sheeting Check**" (calculation of deformations and modeling of the real behavior of the structure, assessment of anchors and anchor system).

In the frame is input the **Depth of the structure in soil** , which is the same for all construction stages. For anchored walls the corresponding "**Anchor force**" should be specified.

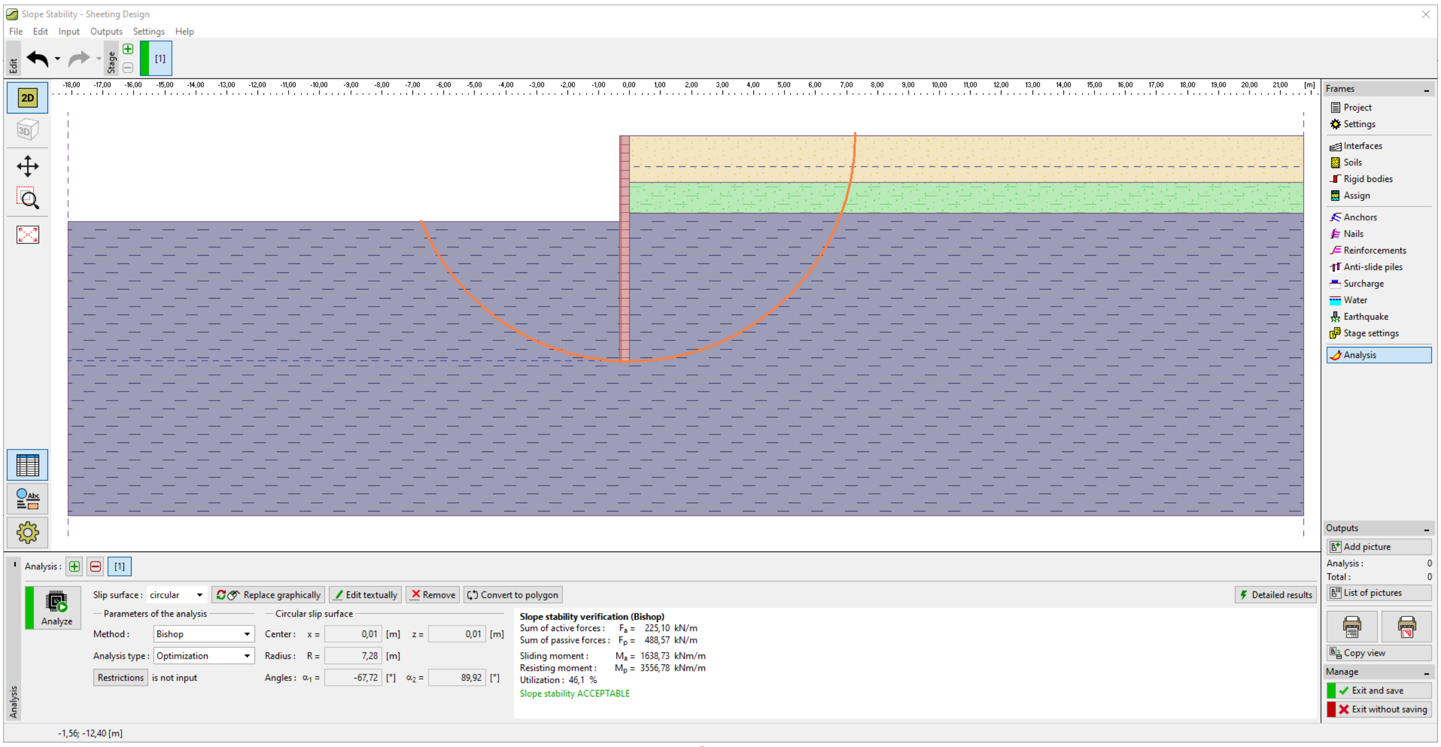
The program displays **minimum or recommended depth of structure in soil** - the depth based on results from all analysis from all construction stages. If the input depth is greater than the calculated necessary depth, information is displayed green, otherwise red.

The "**Slope Stability**" button launches the "**Slope stability**" program and all necessary data are transferred into it. The program allows us to check the overall stability of structure. The button is available only if the program "**Slope stability**" is installed. After performing calculations and leaving by "**OK**" button, the results are transferred to the protocol of the calculation into the program "**Sheeting design**".

By pressing the "**Data for Sheeting Check**" button all data are copied to the clipboard. These data can be paste into the program "**Sheeting Check**" as a new task.



Frame "Stability"



Program "Slope stability"

Program Sheeting Check

This program is used to analyze deep excavations and retaining structures by the method of elasto-plastic non-linear analysis i.e. the magnitude of pressures acting upon a structure depends on its deformation. It models the real behavior of the structure during the construction process and determines internal forces and deformations.

Preliminary design of wall dimensions, internal forces, and anchor loads can be performed by the "Sheeting Design" program.

The help in the program "Sheeting Check" includes the following topics:

- The input of data into individual frames:

Project	Settings	Profile	Modulus Kh	Pressuremeter Tests (PMT)	Dilatometric Tests (DMT)	Soils
Geometry	Material	Pressure Determination	Assign	Excavation	Terrain	Water
Surcharge	Applied Forces	Anchors	Props	Supports	Earthquake	Stage Settings
Analysis	Internal Stability	External Stability	Heave Failure	Dimensioning	Anchors Verification	Lagging
Soil Mix	Walers					

- Standards and analysis methods

- Theory for analysis in the program "**Sheeting Check**":

Stress in Soil Body	Earth Pressures	Sheeting Check	Dimensioning of Concrete Structures	Dimensioning of Steel Cross-sections	Timber Cross Section Verification	Plastic Sheet Pile Verification	Soil Mix Verification

- Outputs

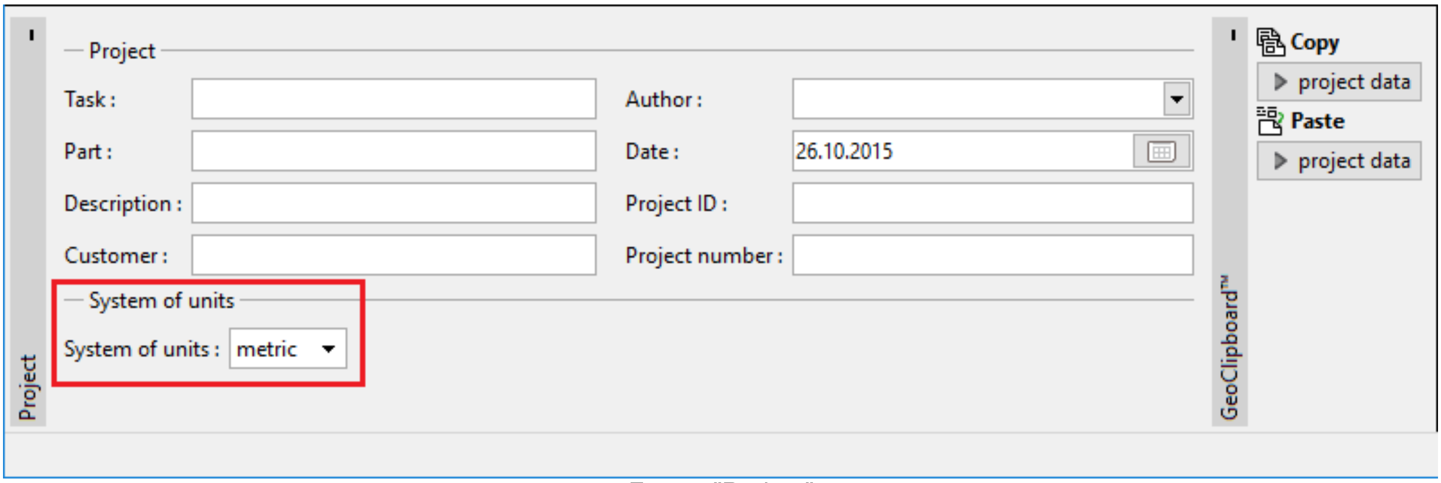
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The **"Project"**frame is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The "**Settings**" frame allows introduconmg of the basic settings, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select settings**" button allows us to choose an already created setting from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

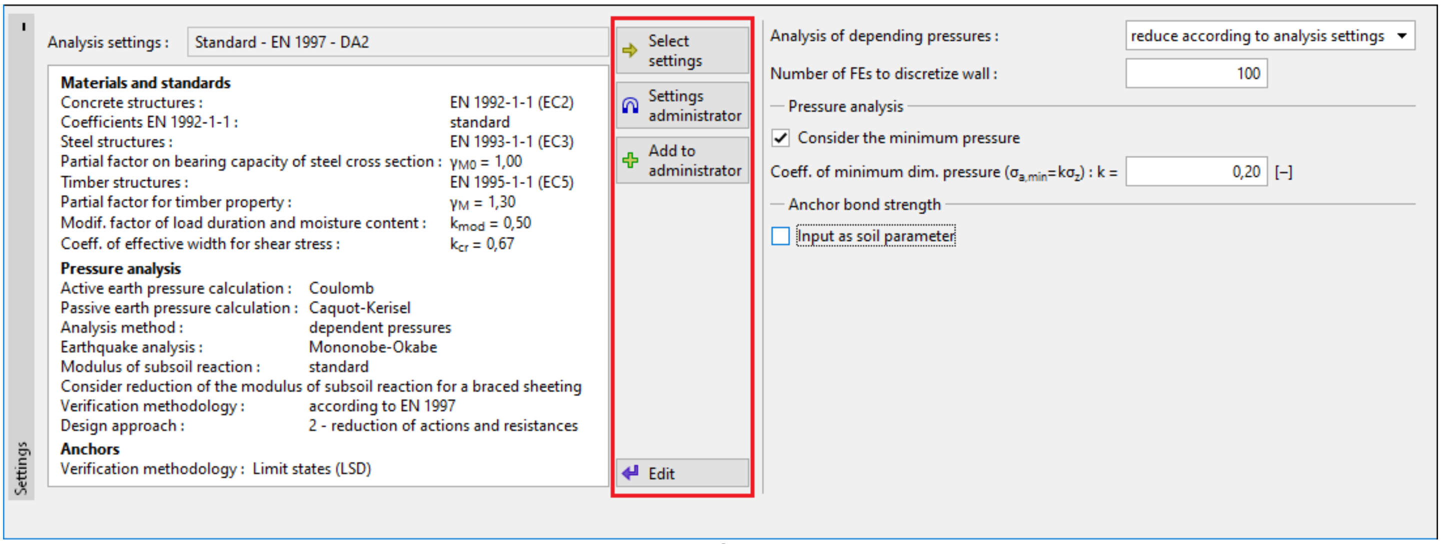
The "**Input for the current task**" setting is usually created when importing older data.

Settings of analysis parameters are performed in the "Materials and standards" and the "Pressure Analysis" tabs.

When performing analysis according to EN 1997 or according to the theory of limit states, the program enables to set whether to reduce the soil parameters for the calculation of limit pressures. For correct modeling of a real behavior of the structure, we recommend not reducing them.

The frame allows the user to specify the subdivision of a wall into finite elements (by default, the number of elements equals 100) and specify whether the structure is loaded by the minimum dimensioning pressure.

When calculating the pull-out resistance of anchor, bond strength can be inserted as a soil parameter.



Frame "Settings"

Profile

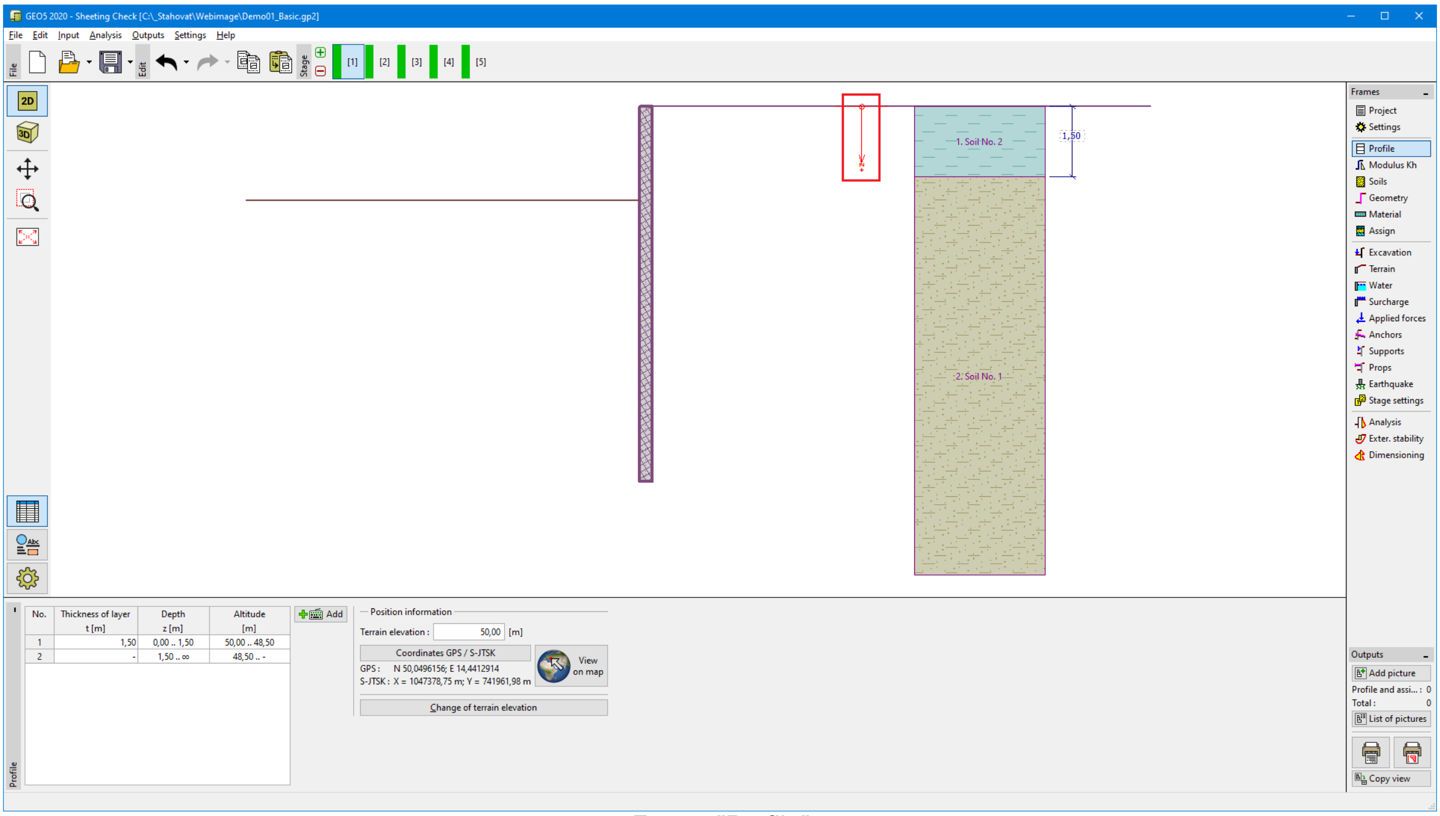
The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.



Frame "Profile"

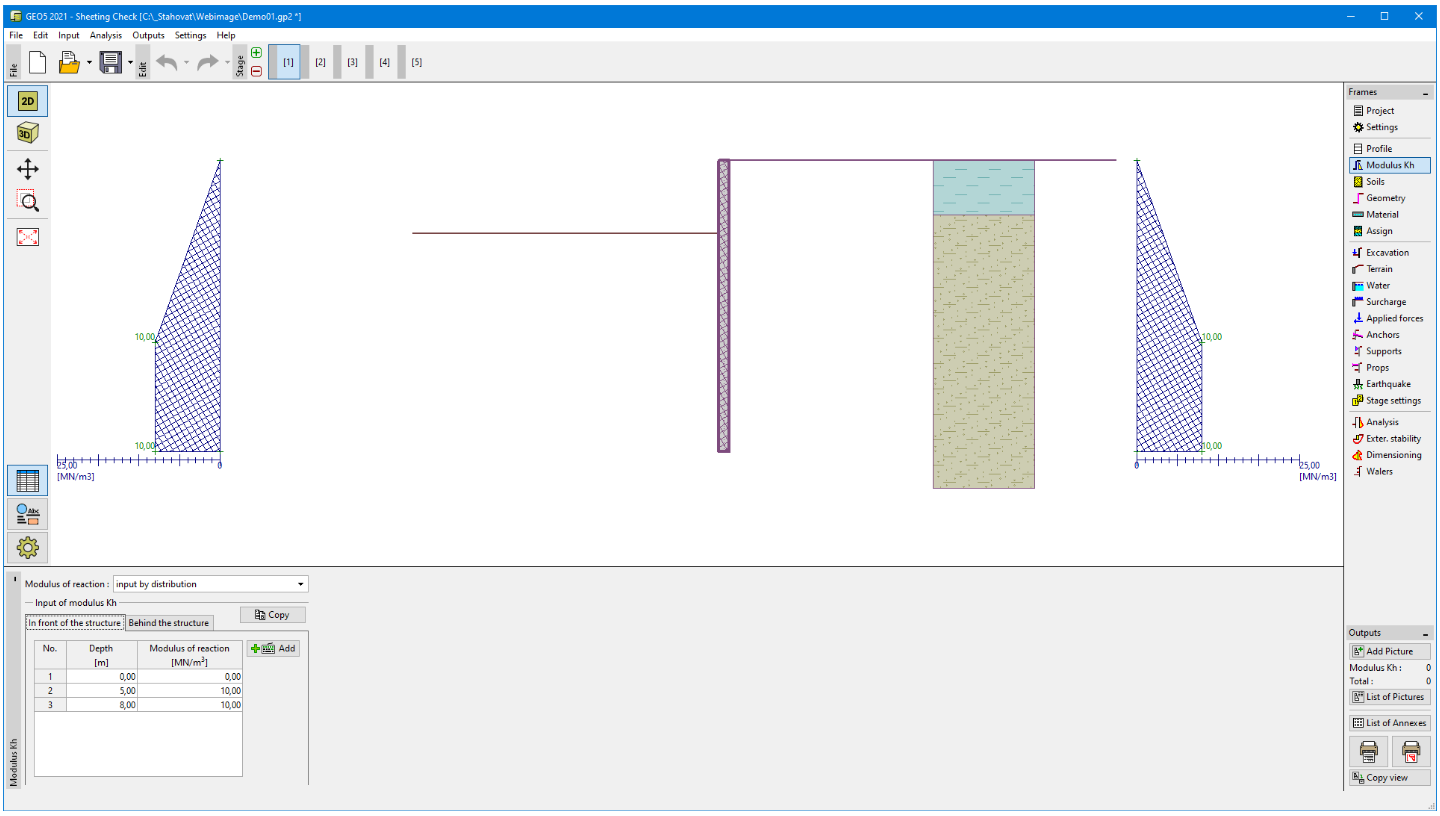
Modulus Kh

This frame serves to specify a type of analysis for calculation of the modulus of subsoil reaction, which is an important input parameter when analyzing a sheeting structure by using the method of dependent pressures.

The way of calculation of the modulus of subsoil reaction k_h is selected in the "Settings" frame (in the "**Edit current settings**" dialog window in the "Pressure Analysis" tab).

The frame can take different forms depending on the selected method of calculation:

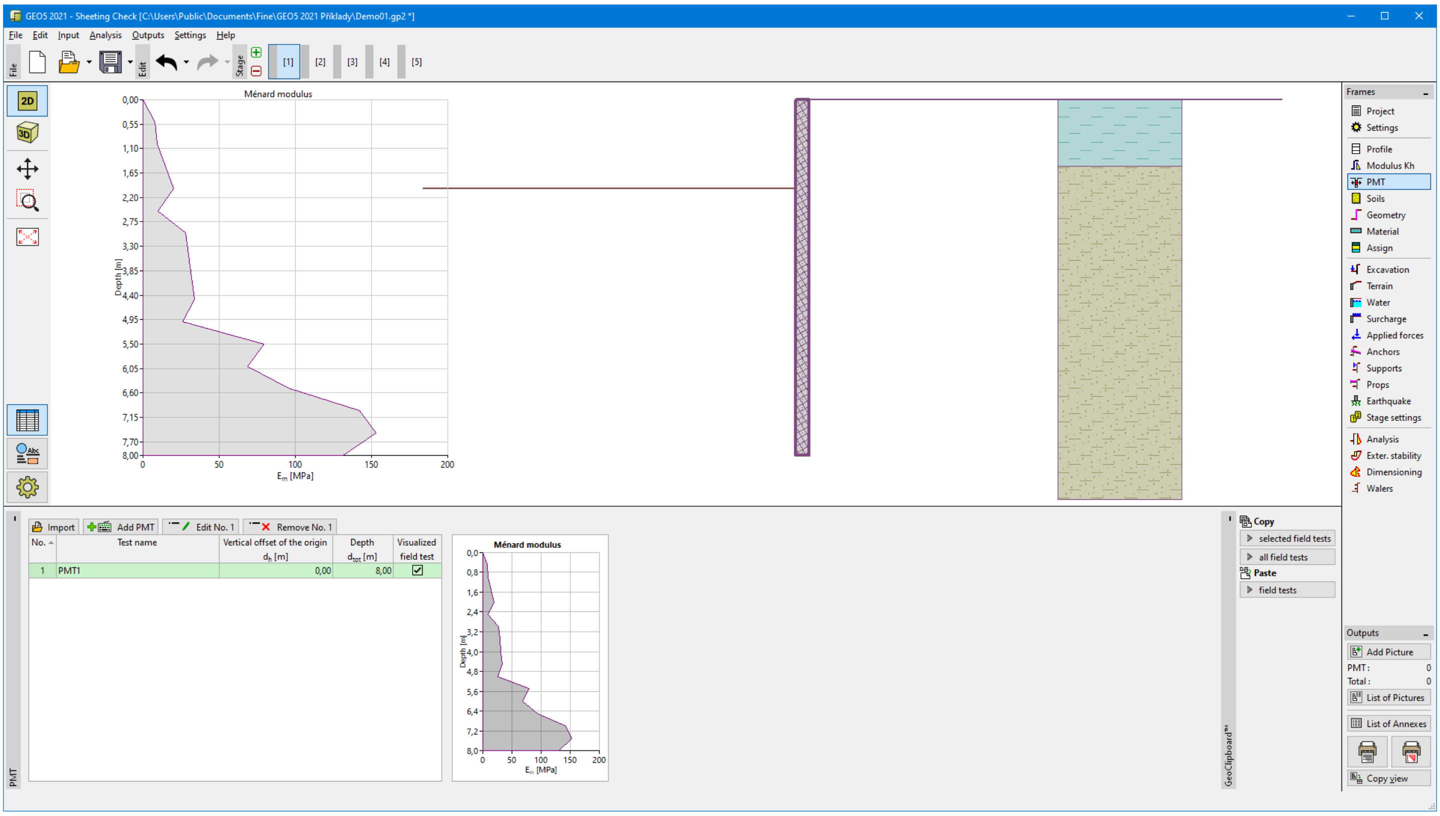
- **standard** (option "analyze – Schmitt", "analyze – Chadeisson", "manual iteration", or "automatic iteration")
- **input** (selecting the option "**Input by distribution**" opens a table in the frame that allows us to input the values of the modulus of subsoil reaction k_h both in front of and behind the structure. For option "**Input as a soil parameter**" the modulus k_h is specified in the "Soils" frame, where the modulus of subsoil reaction is considered either as **linear**, or as **nonlinear - curve**)
- **pressuremeter PMT** (modulus of subsoil reaction k_h is calculated from pressuremeter test, or input as a parameter of soil in the "Soils" frame. Then there is specified method of calculation - according to NF P 94-282 or according to Menard)
- **Chinese standards** (for "m" method is defined horizontal displacement at the ditch bottom v_b [mm] and magnitude of modulus A [MN/m^3], or option input as a parameter of soil – "c" method, "k" or "m" method)



Frame "Modulus k_h"

PMTs

The frame "PMT" contains a table with a list of pressuremeter tests (PMT).



Frame "PMT"

The results of pressuremeter tests (PMT) can be imported into the program by inserting the file in different formats (eg. ***.TXT**, ***.CSV**, ***.XLSX**, ***.ODS**).

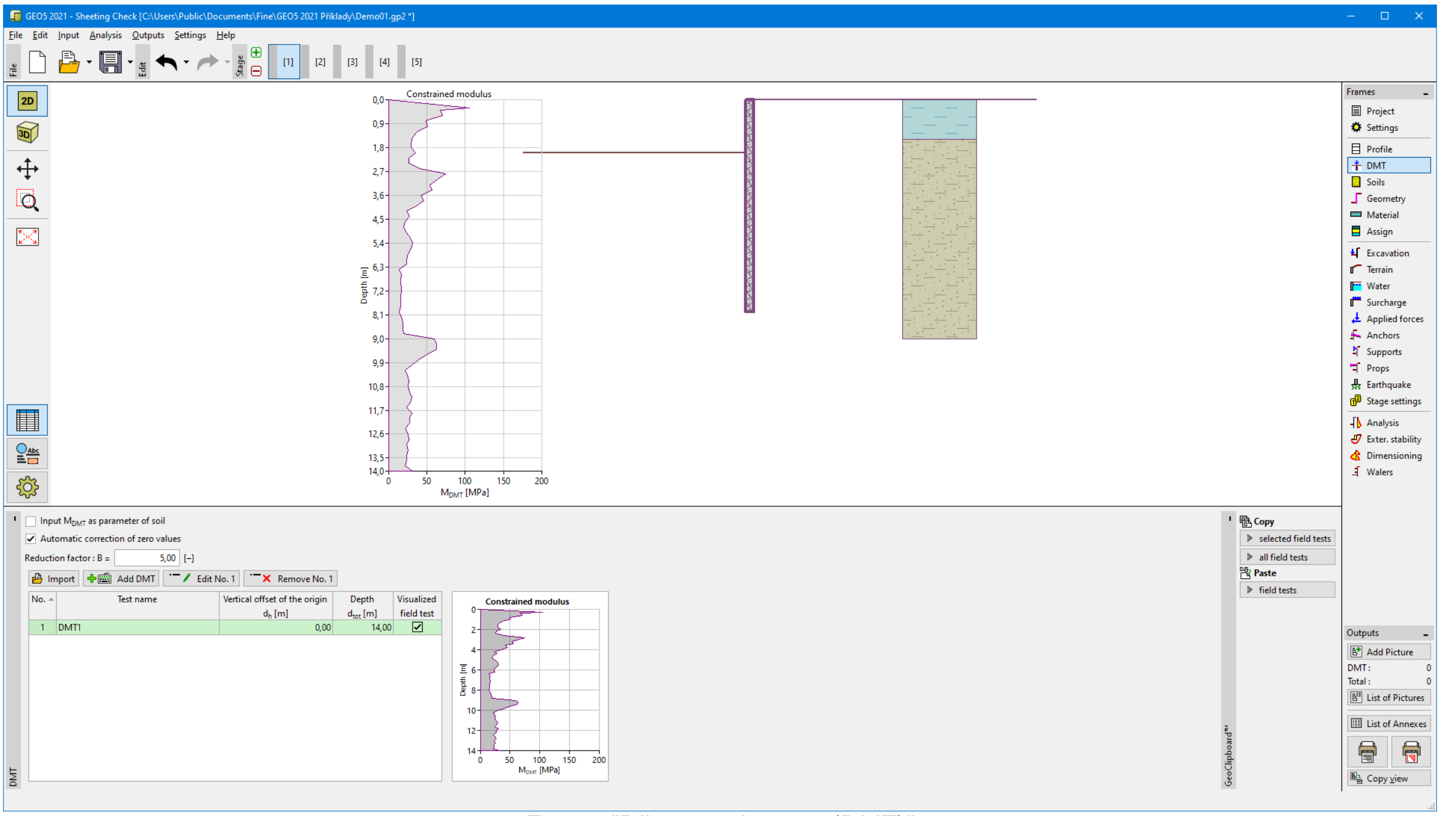
Data of PMTs can be copied within "**Micropile**", "**Sheeting Check**", "**Anti-Slide Pile**", "**Spread Footing CPT**" and "**Stratigraphy**" programs using "GeoClipboard".

Note: The frame is accessible only in case when an option "pressuremeter PMT" is selected for the determination of subsoil reaction modulus in the "Settings" frame (the "Pressure Analysis" tab).

DMT

The frame "**DMT**" serves to input the way of introducing of constrained soil modulus into the program - either as a parameter of soil (by checking the option "**Input M_{DMT} as a soil parameter**"), or by importing of a dilatometric tests (DMT).

This frame contains a table with a list of the input values of dilatometric tests (DMT).



Frame "Dilatometric tests (DMT)"

If the zero value of constrained soil modulus M_{DMT} is measured, then the program makes an automatic correction. The arithmetic average of the next upper and lower non-zero value of M_{DMT} is considered instead of zero value in the calculation.

It is also necessary to enter a coefficient of reduction B .

The results of the dilatometric test (DMT) can be imported into the program by inserting the file in format **UNI** (***.uni**).

Data of DMTs can be copied within "**Spread Footing**", "**Sheeting Check**", "**Anti-Slide Pile**" and "**Stratigraphy**" programs using "GeoClipboard".

Note: The frame is accessible only in case when an option "dilatometric DMT" is selected for the determination of subsoil reaction modulus in the "Settings" frame (the "Pressure Analysis" tab).

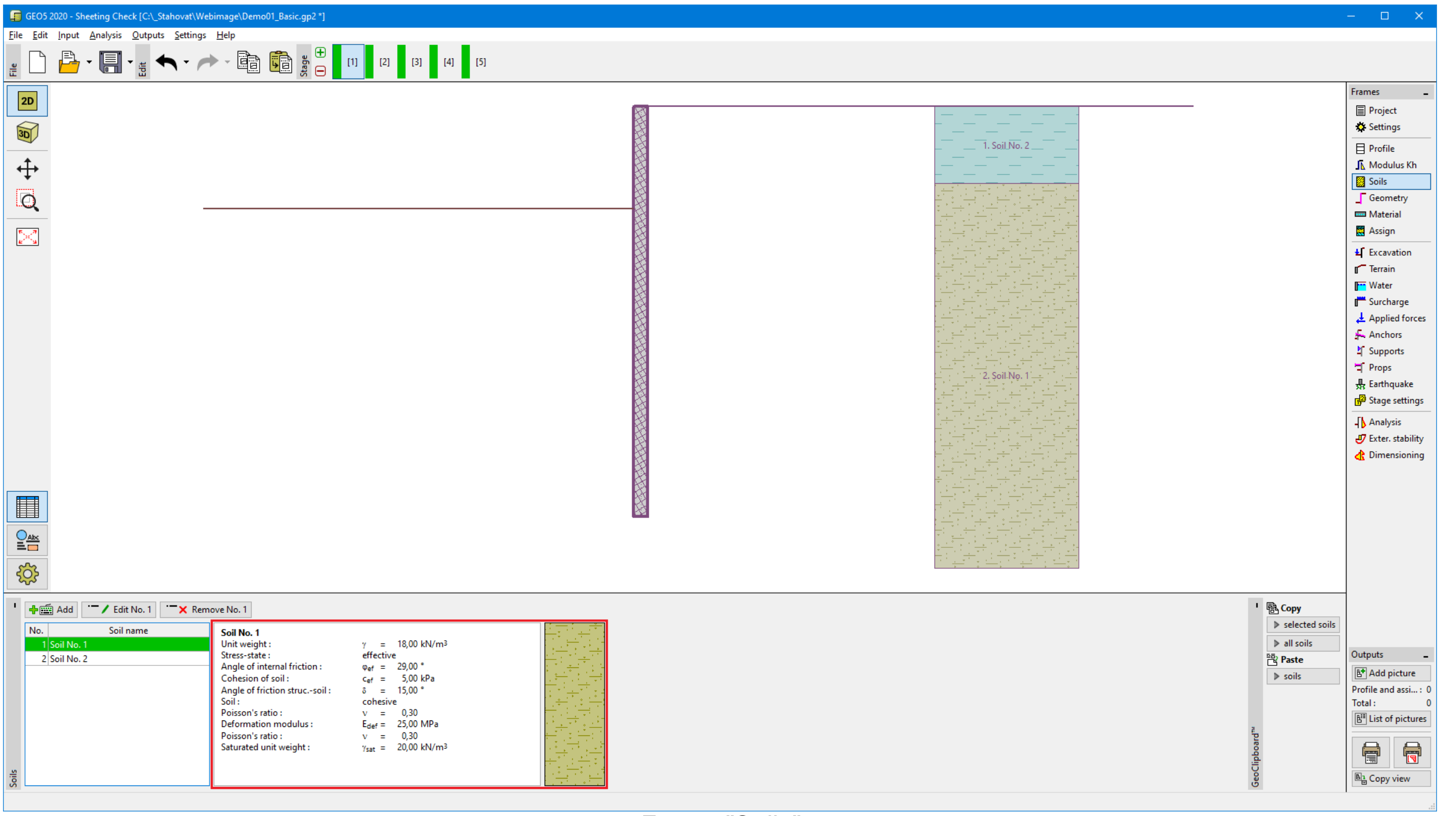
Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Earth pressure at rest", "Uplift pressure" and "Modulus of subsoil reaction".

Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

Basic Data

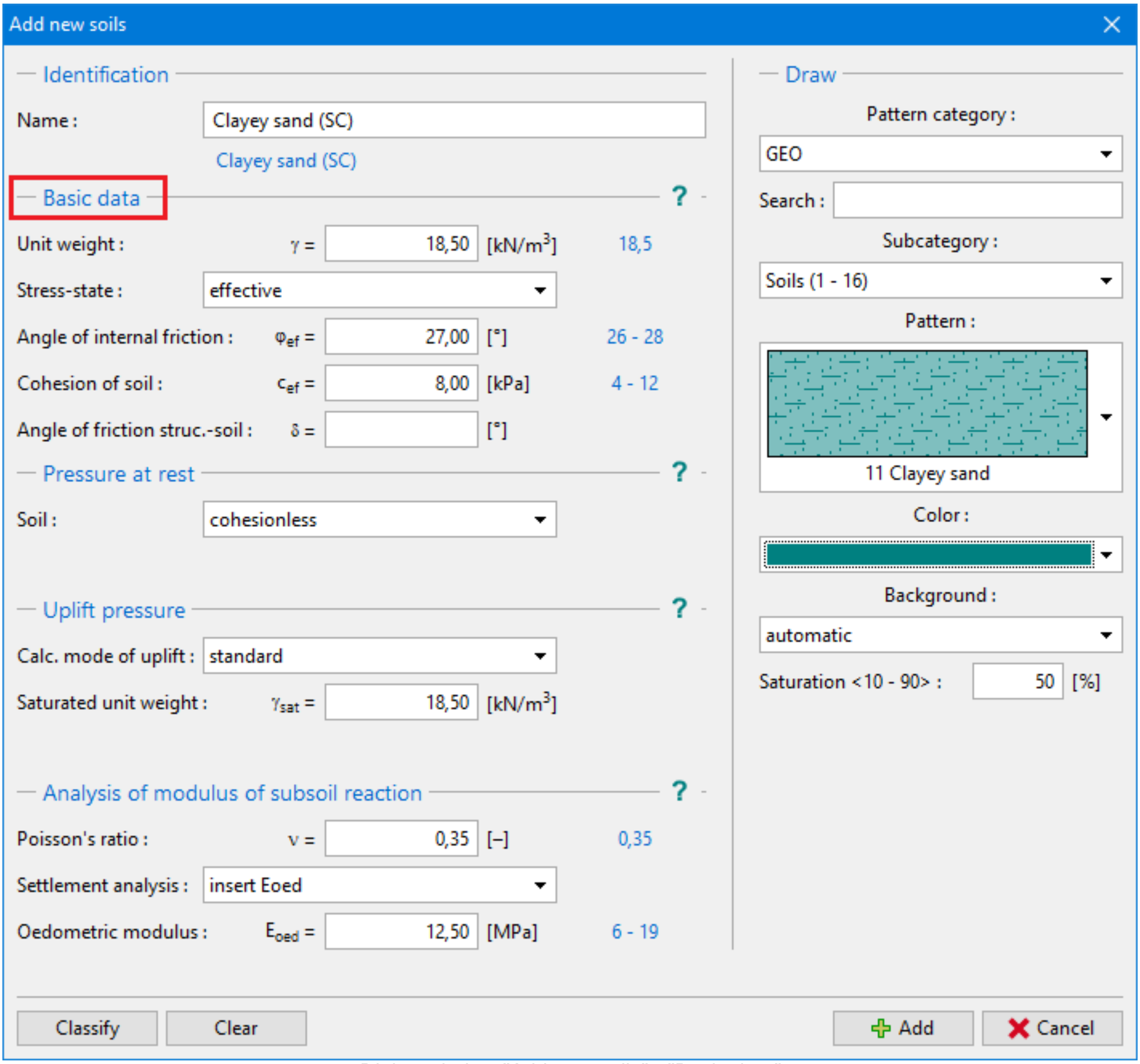
This part of the window allows us to introduce basic parameters of the soils - **unit weight, angle of internal friction, and cohesion**. The particular values are obtained from a geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit the built-in soils database, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the built-in database, these parameters must be defined manually.

Either **effective** or **total** parameters of the angle of internal friction and cohesion are specified depending on the settings in the "**Stress analysis**" combo list. Whether to use effective or total parameters depends primarily on the type of soil and load, structure duration, and water conditions.

For effective stress, it is further needed to specify the angle of internal friction between the soil and the structure, which depends on the structure material and type of soil. Possible values of this parameter are listed in the table of recommended values.

For total stress, it is further needed to specify the adhesion of soil to the structure face ***a*** .

The associated theory is described in detail in the chapter "Earth pressures".



Dialog window "Add new soils" - "Basic data"

Geometry

The "**Geometry**" frame allows us to select the **type of structure**:

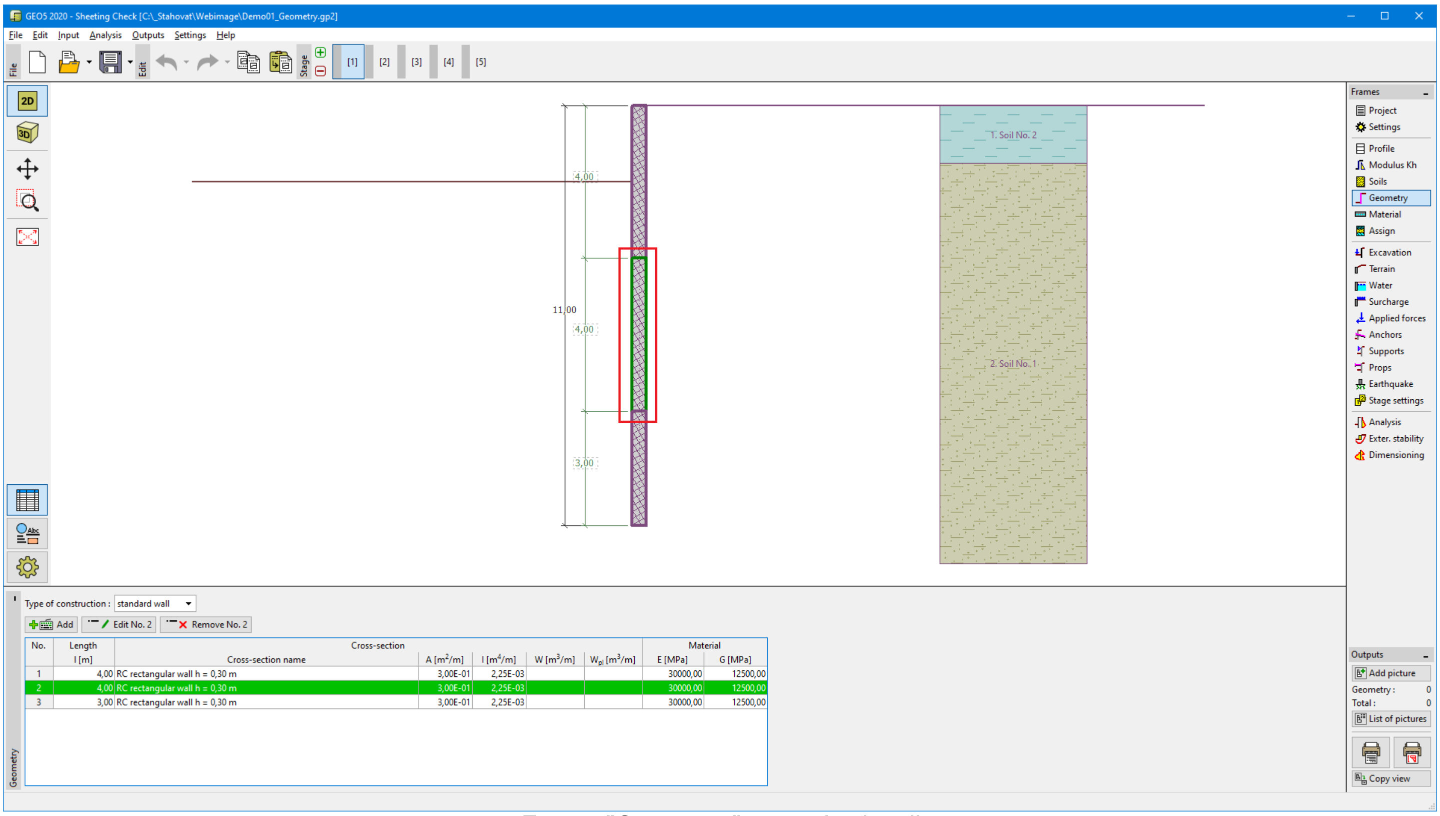
- **standard wall**
- **double-row pile wall** - depth of connecting beam h_1 and final excavation depth h are input

The frame also contains a table with a list of input structural sections forming the sheeting structure. For each section the table stores its cross-sectional characteristics (A - area, I - Moment of inertia - these variables are always expressed with respect to 1 m run of structure length) and material characteristics (E - Modulus of elasticity, G - Shear modulus). For combined sections, the modules E and G correspond to the material to which the section is converted (typically concrete or steel).

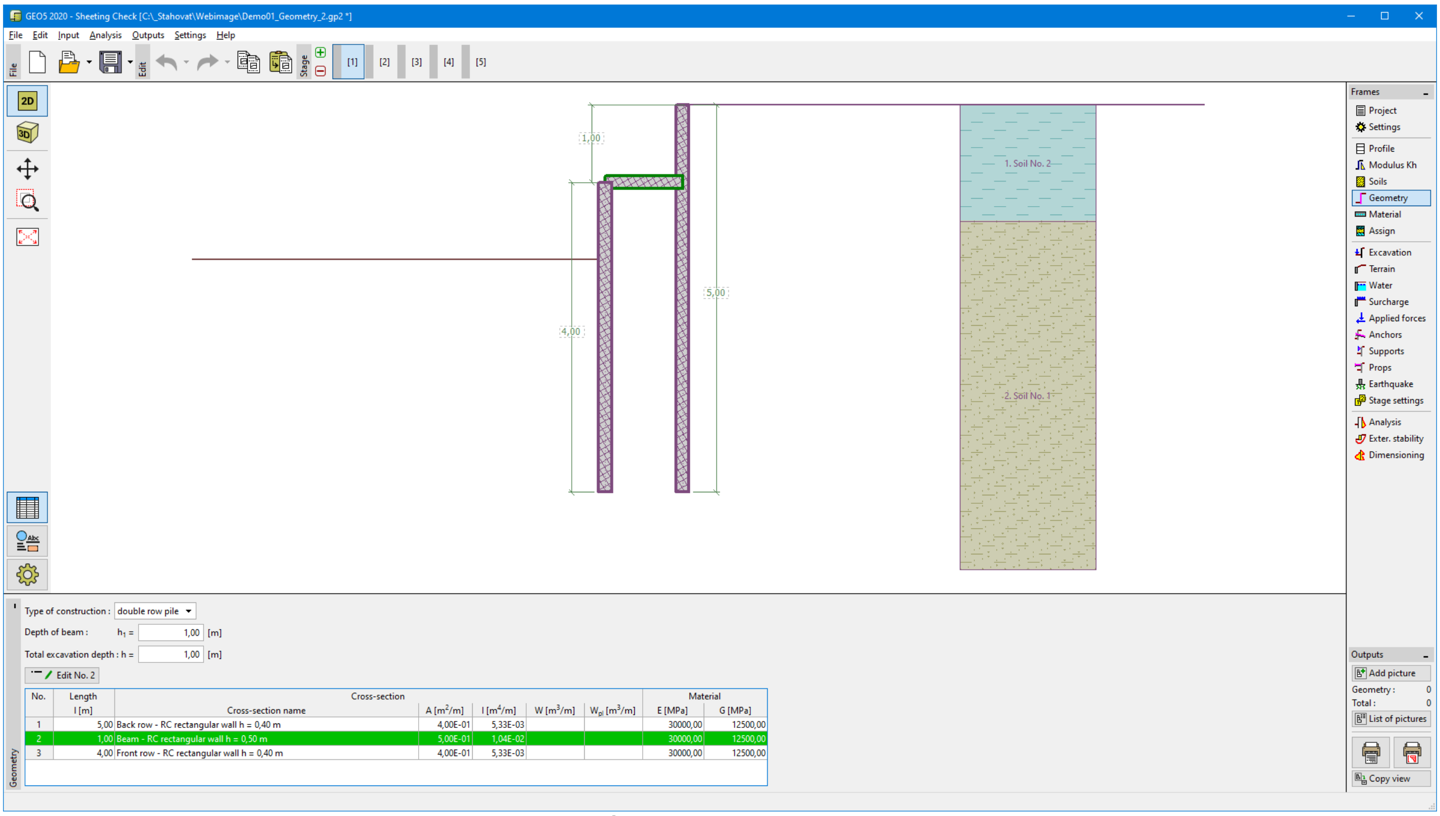
Adding sections is performed in the "New section" dialog window.

The input sections can be further edited on the desktop with the help of active objects - double-click on a structure opens a dialog window with a given section.

The program makes it possible to export the geometry of a structure in the *.DXF format.



Frame "Geometry" - standard wall



Frame "Geometry" - double row pile

Material

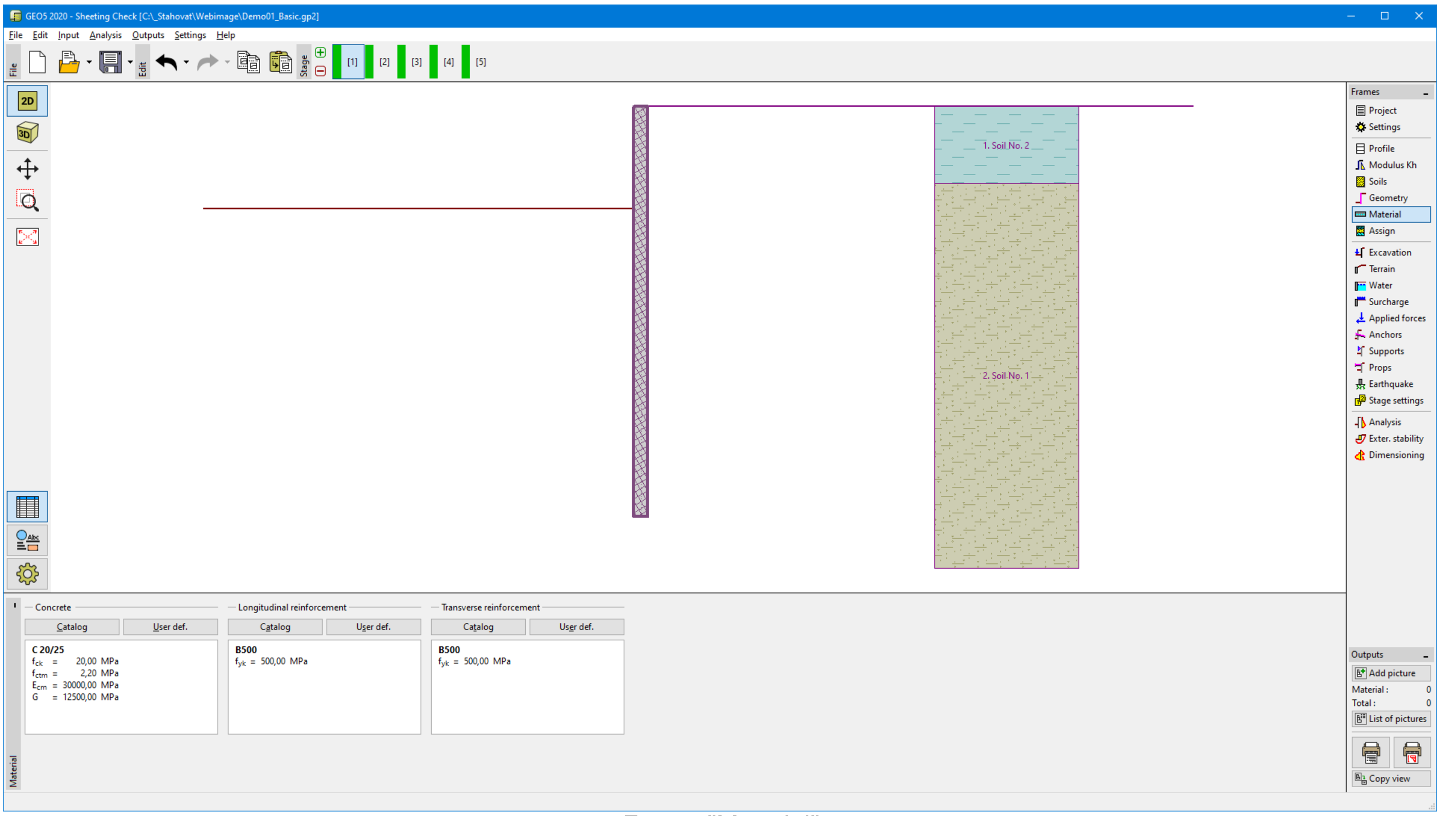
The frame **"Material"** allows us to enter material parameters. The appearance of the frame varies according to the selected material (concrete, steel, timber, plastic, soil mix) in the frame "Geometry".

Two options can be used for setting the parameters of the material:

- The "**Catalog**" button opens the "**Catalog of materials**" dialog window, where the required material can be selected.

- The "**User-defined**" button opens the "**Editor of material**", which allows us to input the specification of material parameters manually by the user.

The content of catalogs depends on the selection of relevant standard for the **dimensioning of concrete, steel**, or **timber** set in the "Materials and standards" tab.



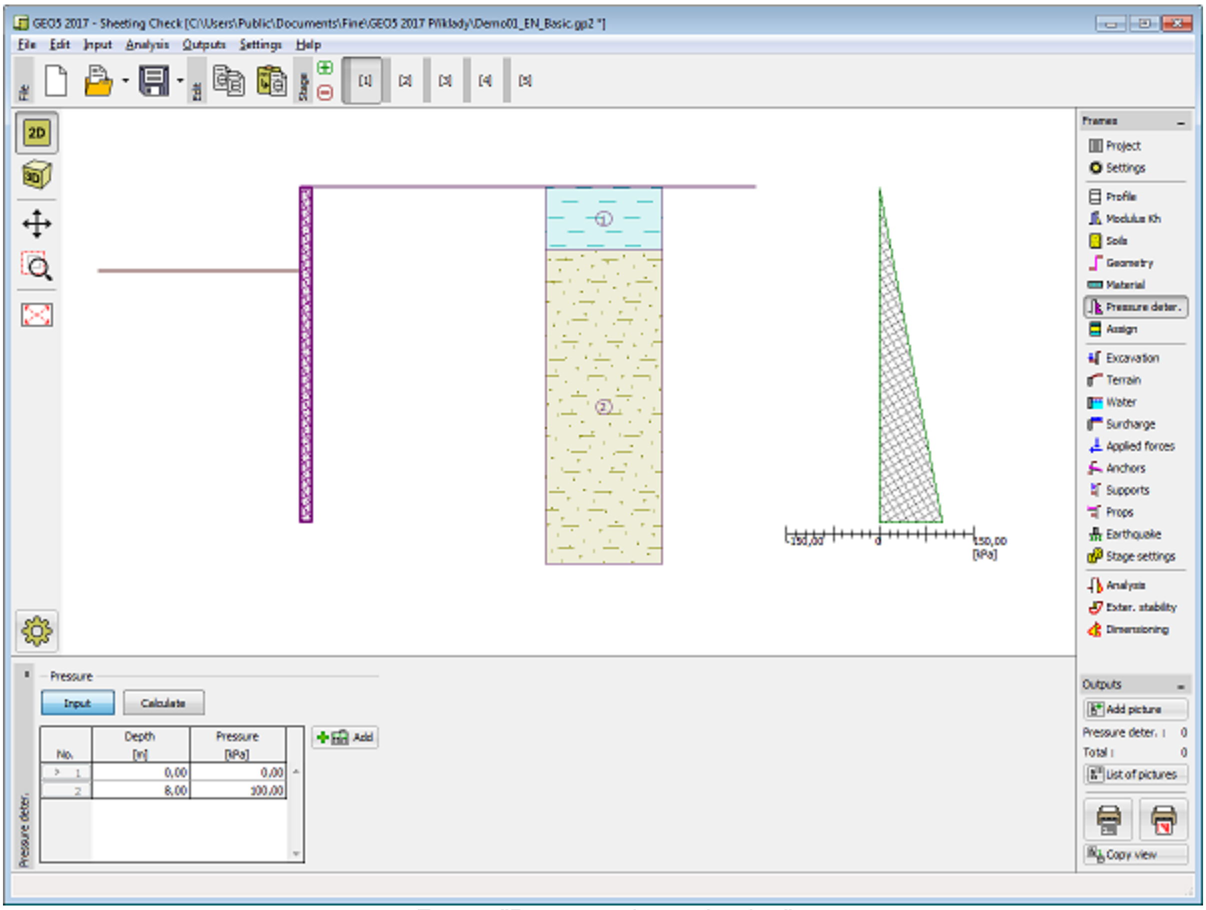
Frame "Material"

Pressure Determination

The frame "**Pressure determination**" allows us to input the values of earth pressure behind sheeting structure manually (by pressing the "**Input**" button). The frame is accessible only in case, when in the "Settings" frame is selected an option "**JGJ 120-2012**" (the "Pressure Analysis" tab).

This frame contains a table with a list of the input values of earth pressure. Adding of these values is performed in the "**New point**" dialog window, where is specified the depth of point [m] and the value of earth pressure [kPa].

It is possible to calculate the values of active earth pressure, or earth pressure at rest automatically (by pressing the "**Calculate**" button).

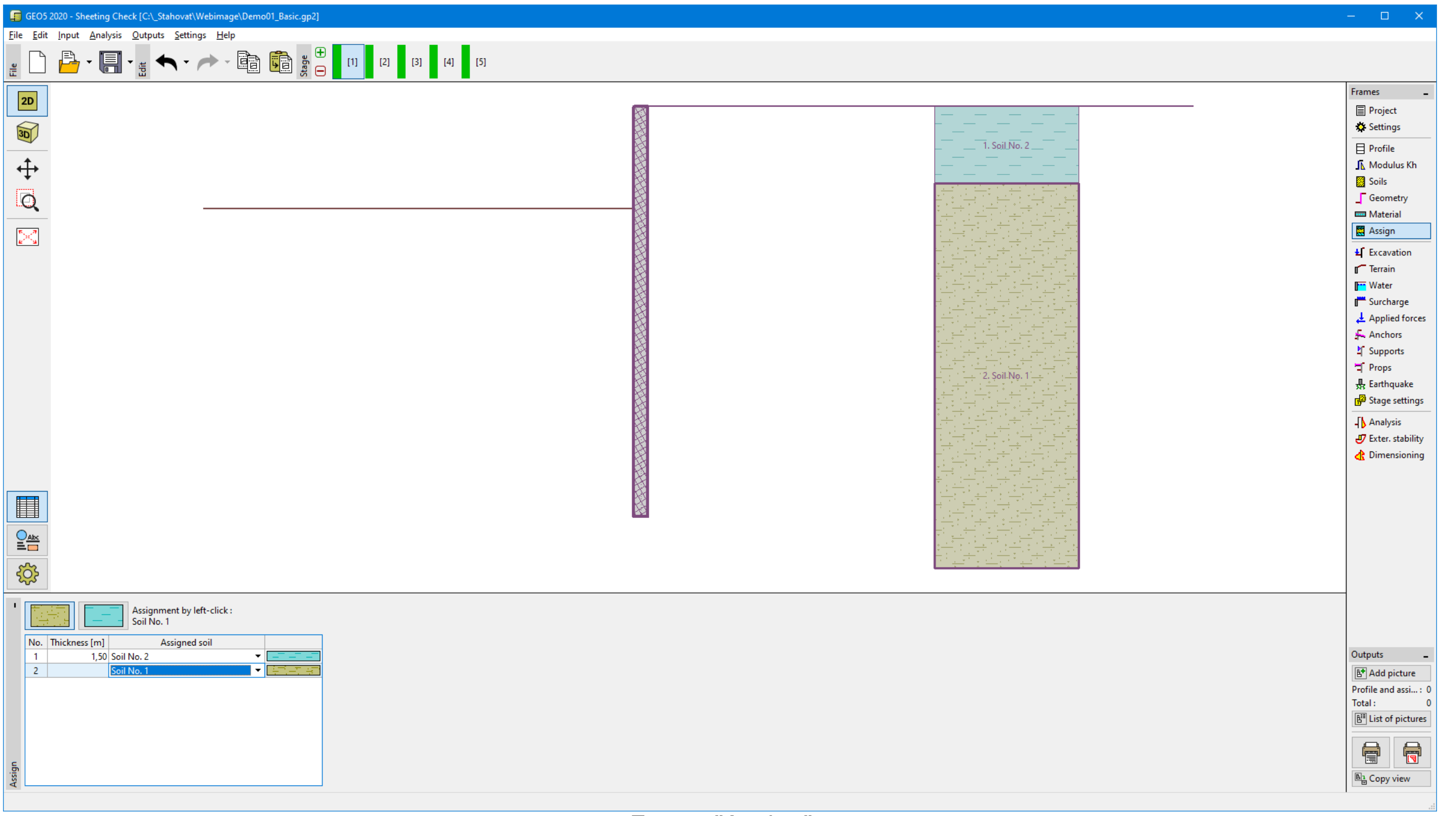


Frame "Pressure determination"

Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail herein.



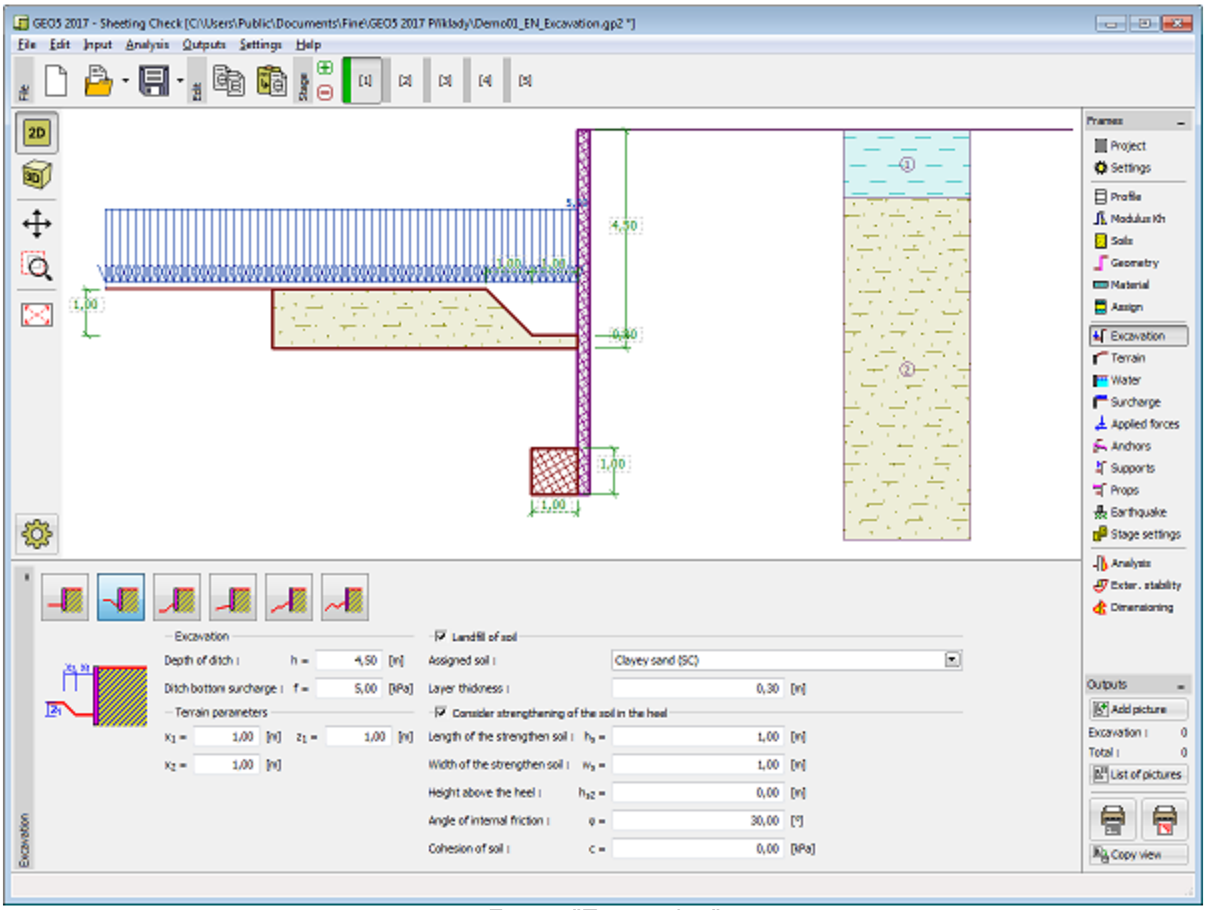
Frame "Assign"

Excavation

The "**Excavation**" frame serves to input the depth of a construction ditch h $[m]$ and by pressing the button to select the shape of the ditch base in front of the sheeting structure. The selected shape with a graphic hint appears in the left part of the frame. The dimensions of a structure can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The frame also allows us to specify ditch bottom surcharge or a thickness of layer of landfill of soil below the ditch bottom (the soil can be selected from a combo list containing soils input in the frame "Soils"). When introducing the landfill of soil with braced sheeting, it is assumed that there is a sheeted structure in the location of landfill of soil, i.e., all pressures are acting on the entire width of a structure as above the construction ditch base.

In this frame, it is possible to input **strengthening of the soil** at the heel of sheeting structure. The principle of calculation is described in more detail herein.



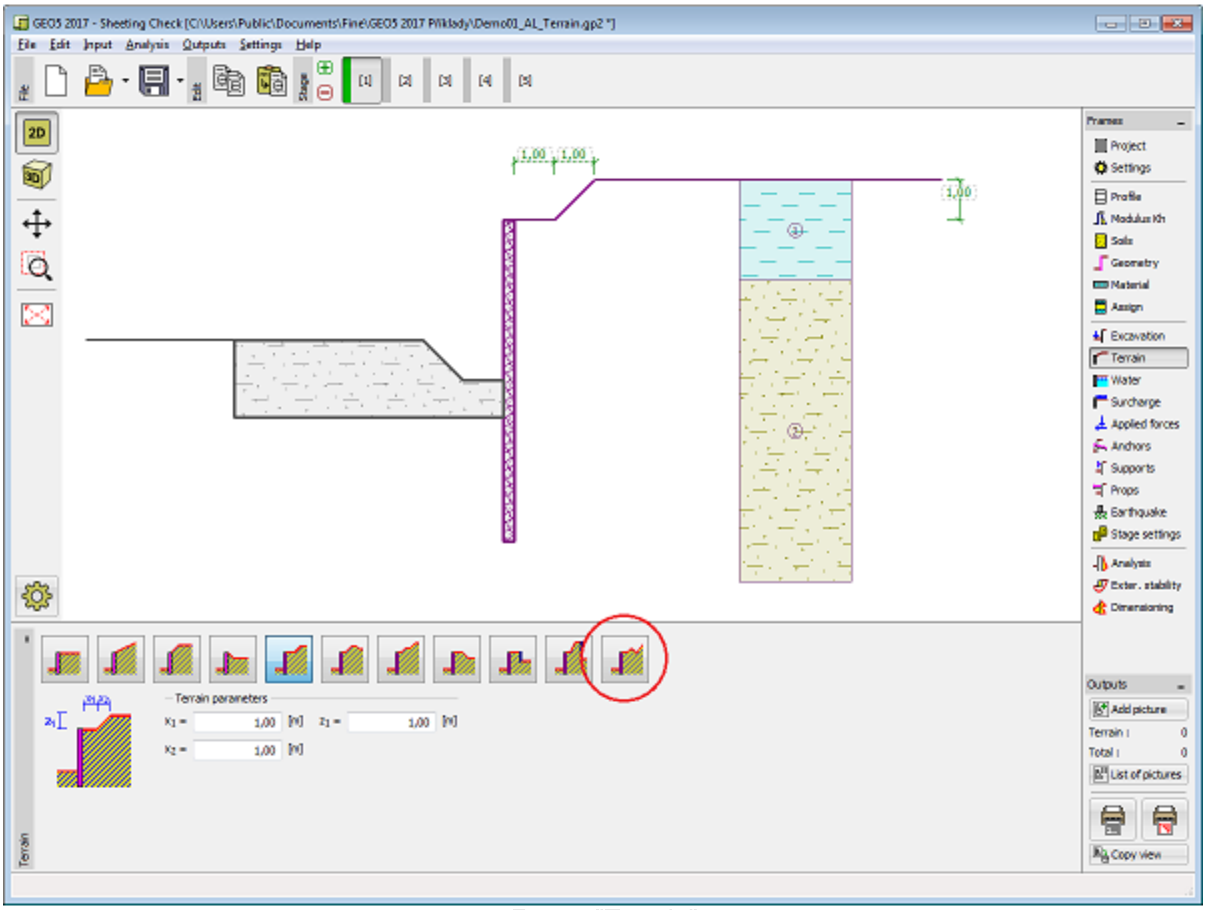
Frame "Excavation"

Terrain

The "**Terrain**" frame allows us to specify shape of the terrain by pressing the button. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case, the frame contains a table with a list of terrain points. The first point with coordinates [0; 0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "Distribution of earth pressures for broken terrain".



Frame "Terrain"

Water

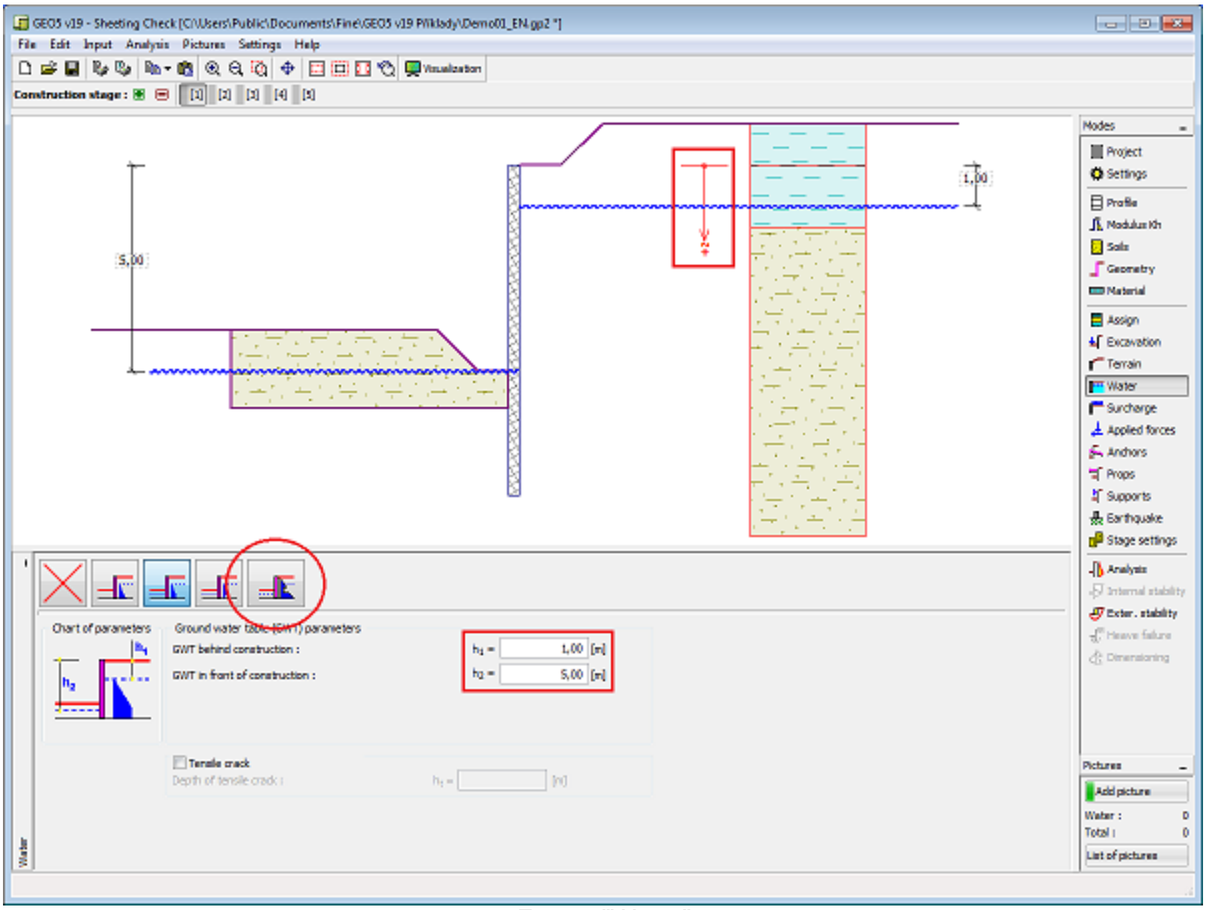
The "**Water**" frame allows us to select the type of water by pressing the button. The selected type with a graphic hint of input values is displayed in the left part of the frame. Parameters of water (h_1 , h_2 etc.) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option is a manual input of pore pressure both in front of and behind the structure. Two tabs "**In front of structure**" and "**Behind structure**" appear with tables. The table contains the values of pore pressure in front of the structure or behind it.

The groundwater table (GWT) can also be specified **above the structure** or earth profile, respectively - in such a case, the depth of water is input with a negative value.

Analysis of earth pressures with the influence of water is described in the theoretical part of the help, chapter "Influence of water".

The program further allows us to specify a depth of tensile crack h_t filled with water.



Frame "Water"

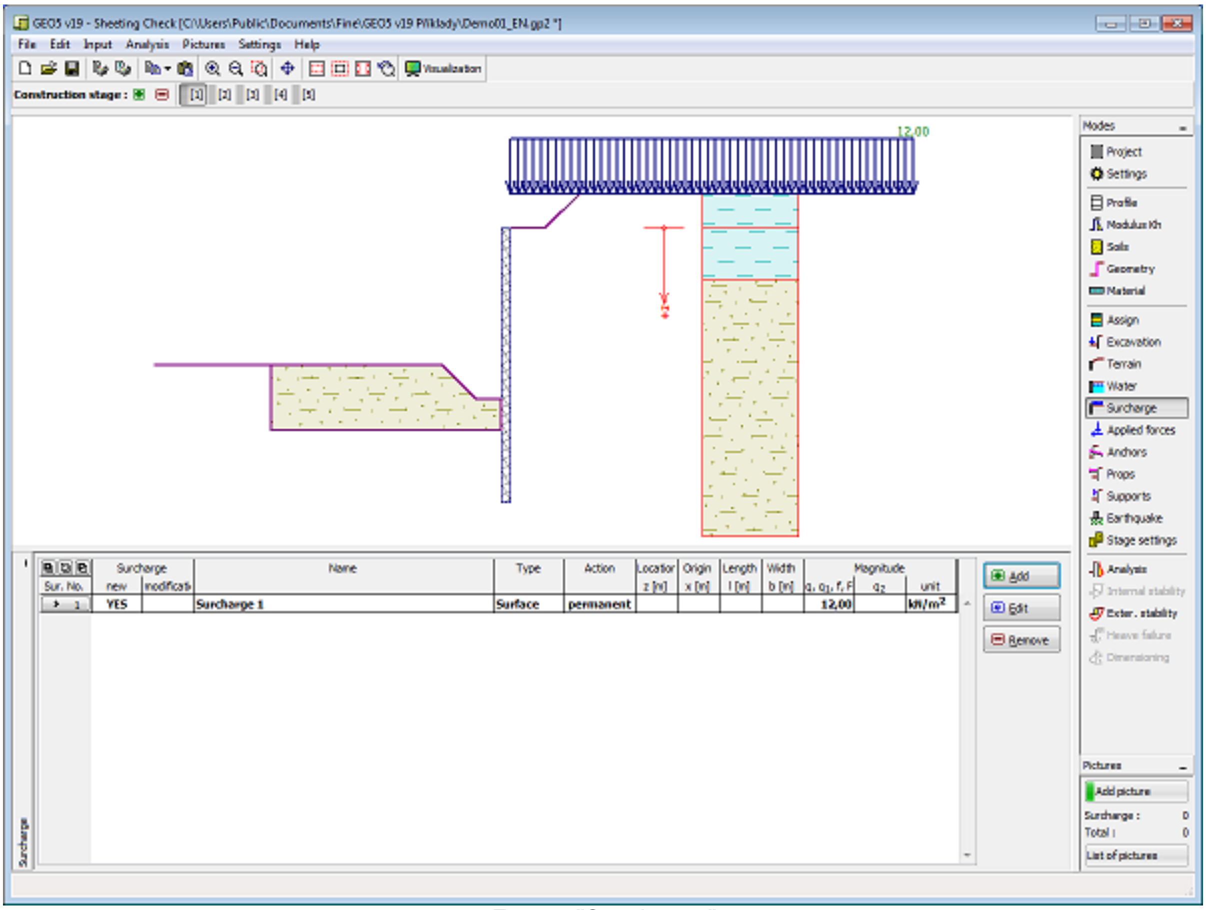
Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The z -coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case when the surcharge is found out of the terrain the program prompts an error message before calculation.

Either **permanent**, **variable** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with a favorable effect is not considered in the analysis.

Analysis of earth pressures due to surcharge is described in the theoretical part of the help, chapter "Influence of surcharge".



Frame "Surcharge"

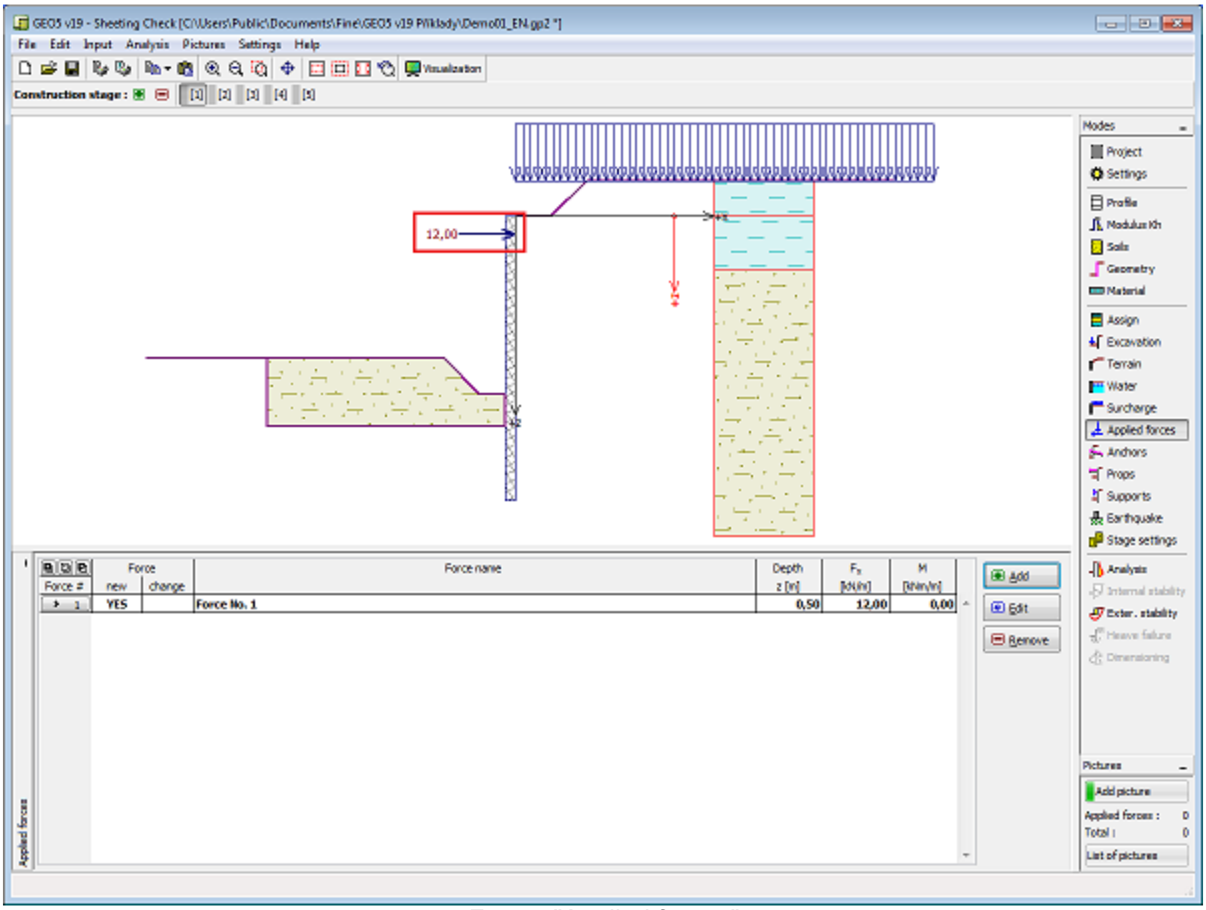
Applied Forces

The "**Applied forces**" frame contains a table with a list of forces acting on a structure. Adding forces is performed in the "**New force**" dialog window. The input forces can also be edited on the desktop with the help of active objects.

Applied forces represent an additional load on the structure of the wall, braced sheeting or MSE wall. We can model such as an anchoring crash barrier, crash vehicle, load from billboards and hoardings etc. The program does not modify the applied forces in the calculation except multiplying them with the corresponding coefficients according to the selected type of load (EN1997, LRFD).

The "vertical on wall" type of force is used only for the analysis of vertical bearing capacity. The effect of load on the wall is not considered.

External load acting on the terrain is necessary to define as a a surcharge.



Frame "Applied forces"

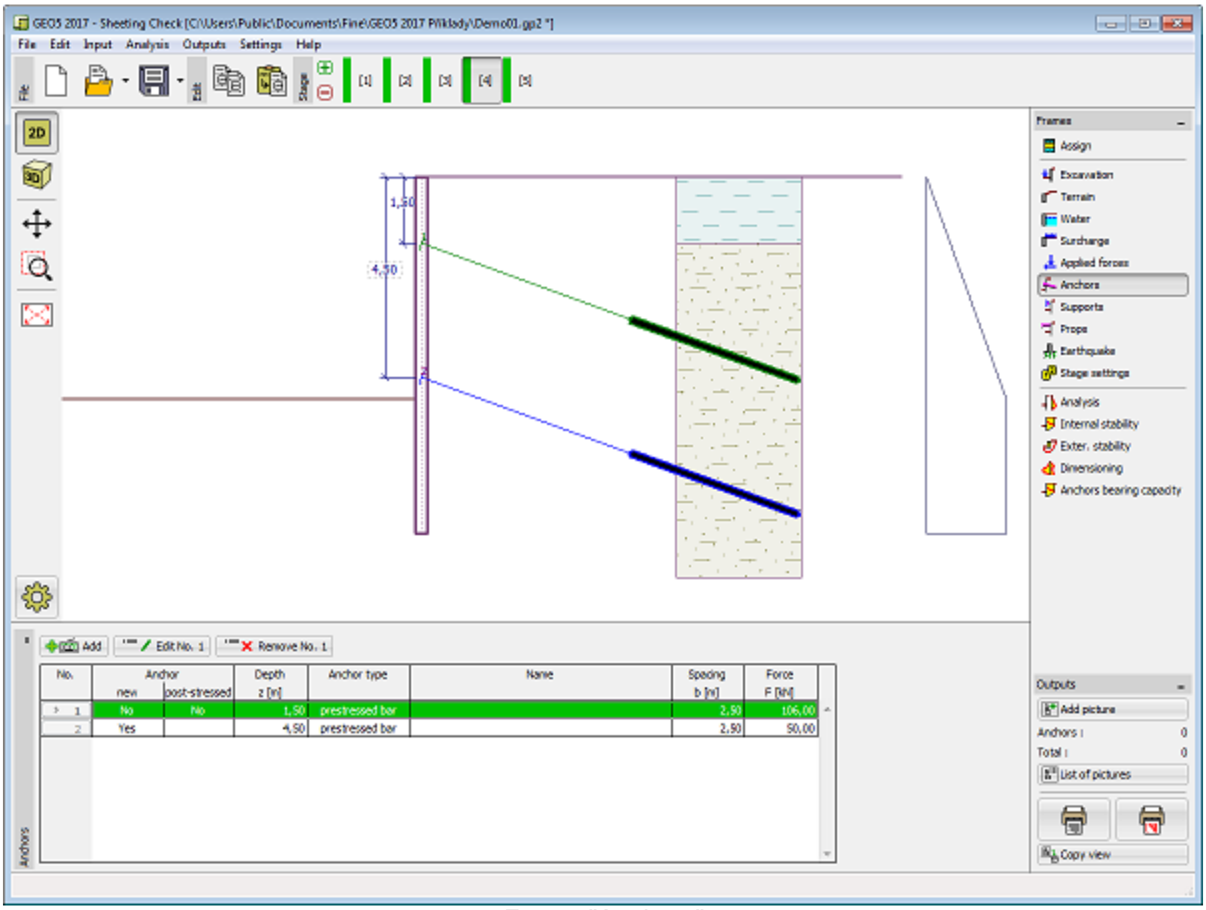
Anchors

The "**Anchors**" frame contains a table with a list of input anchors. Adding anchors is performed in the "**New anchor**" dialog window. The input anchors can be edited on the desktop with the help of active objects.

The anchor is automatically placed on already **deformed structure** (displacement is obtained from the previous construction stage).

Anchors can be entered as pre-stressed (not specified, prestressed bars, strand) and non-prestressed (helical, non-prestressed bars, deadman). The **pre-stressed anchor stiffness** becomes effective in next stages of construction. Due to the displacement of the sheeting structure the forces in anchors are changed according to deformation.

In **subsequent stages** the anchor cannot be changed, it's only possible to **post-stress** the anchor.



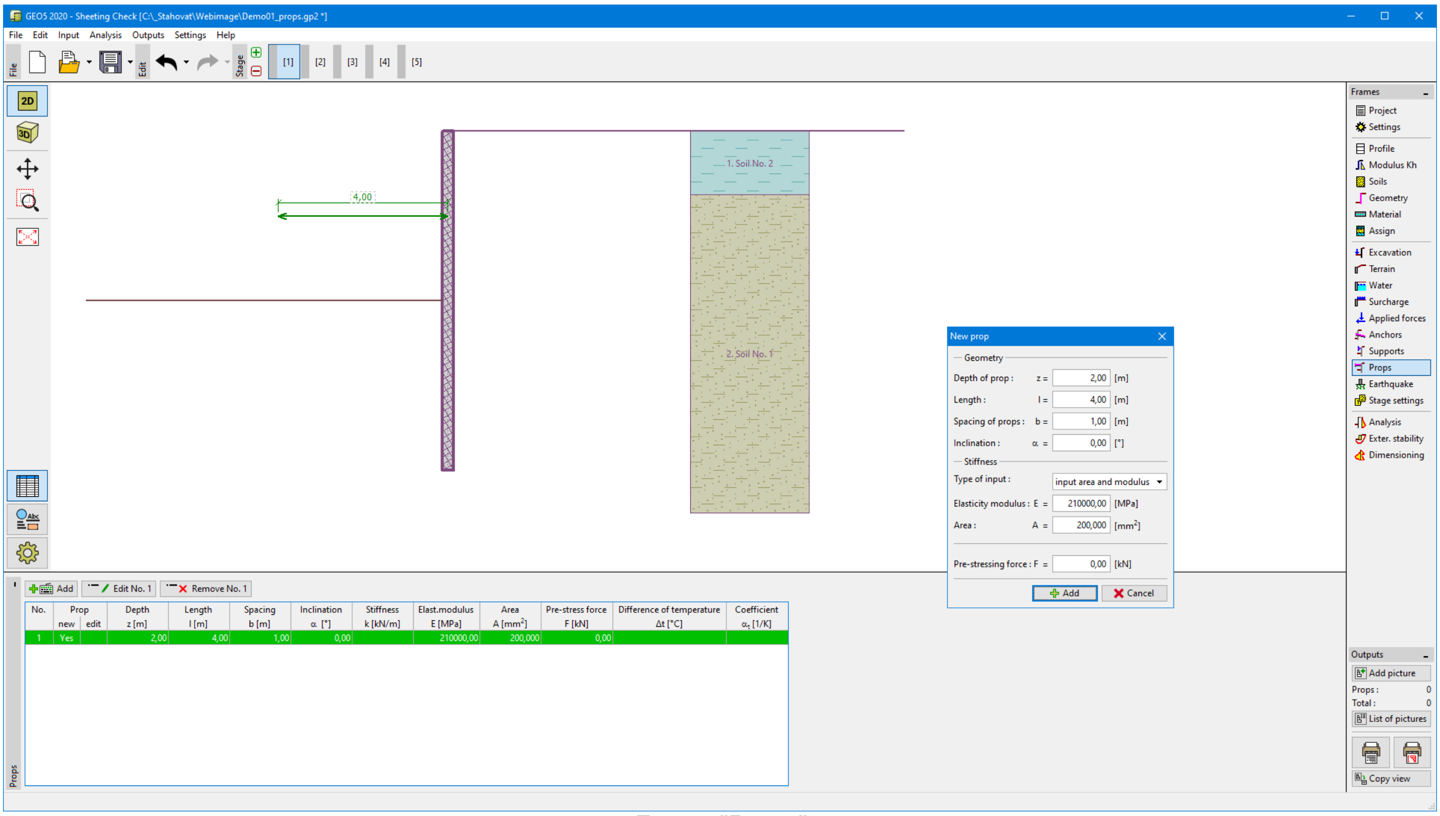
Frame "Anchors"

Props

The "**Props**" frame contains a table with a list of input props. Adding props is performed in the "**New prop**" dialog window. The input props can also be edited on the desktop with the help of active dimensions or by active objects.

The prop is introduced on already **deformed structure** automatically (obtained from the previous stage of construction). In subsequent stages, it is only possible to change the stiffness of props or to introduce temperature load. In the analysis, props are modeled in the same way as anchors but with the initial force equal to zero.

Note: The program doesn't check the prop bearing capacity neither for compression or for buckling.

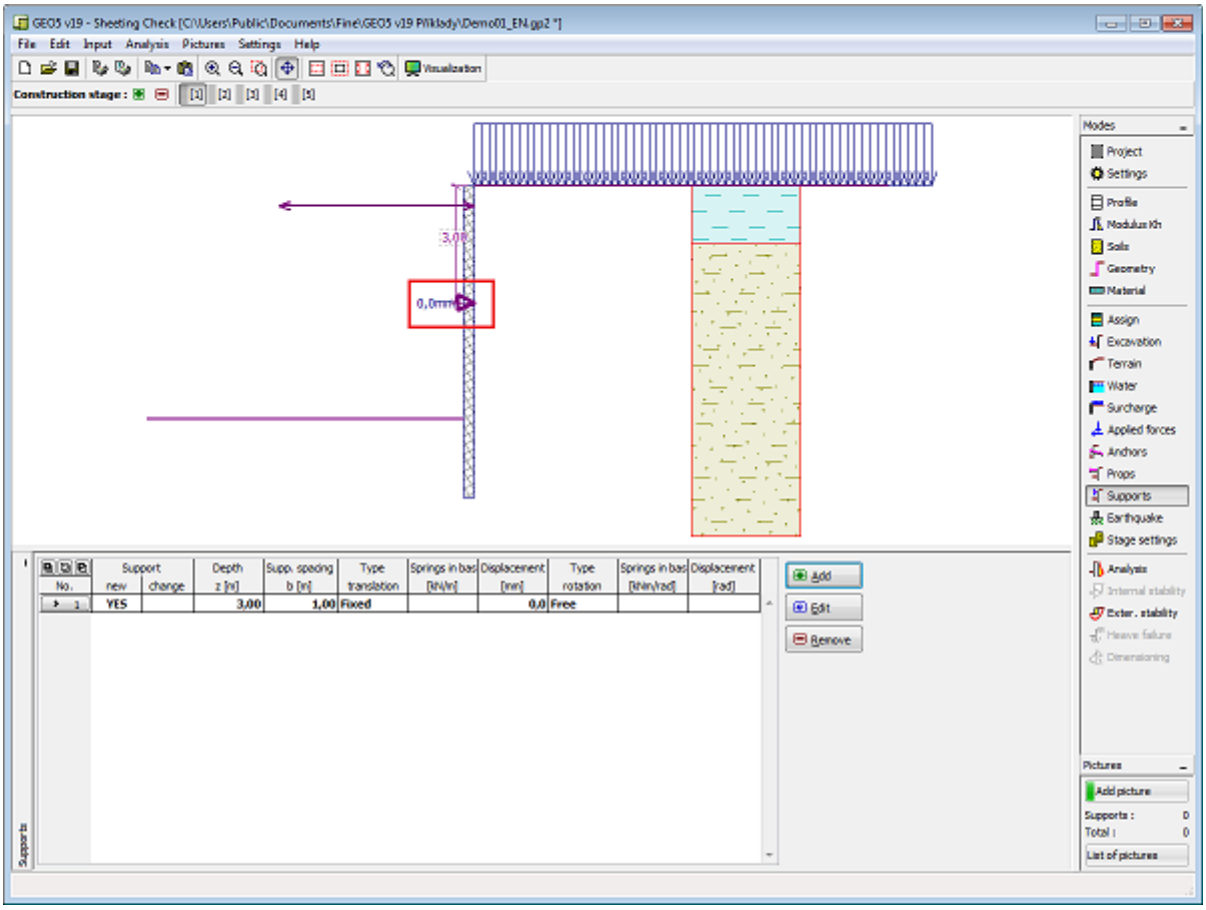


Frame "Props"

Supports

The "**Supports**" frame contains a table with a list of input supports. Adding supports is performed in the "**New support**" dialog window. The input supports can also be edited on the desktop with the help of active dimensions or active objects, respectively.

The support is input on already **deformed structure** automatically (obtained from the previous stage of construction). In subsequent stages, the supports can no longer be edited; it is only possible to input forced displacement of supports.



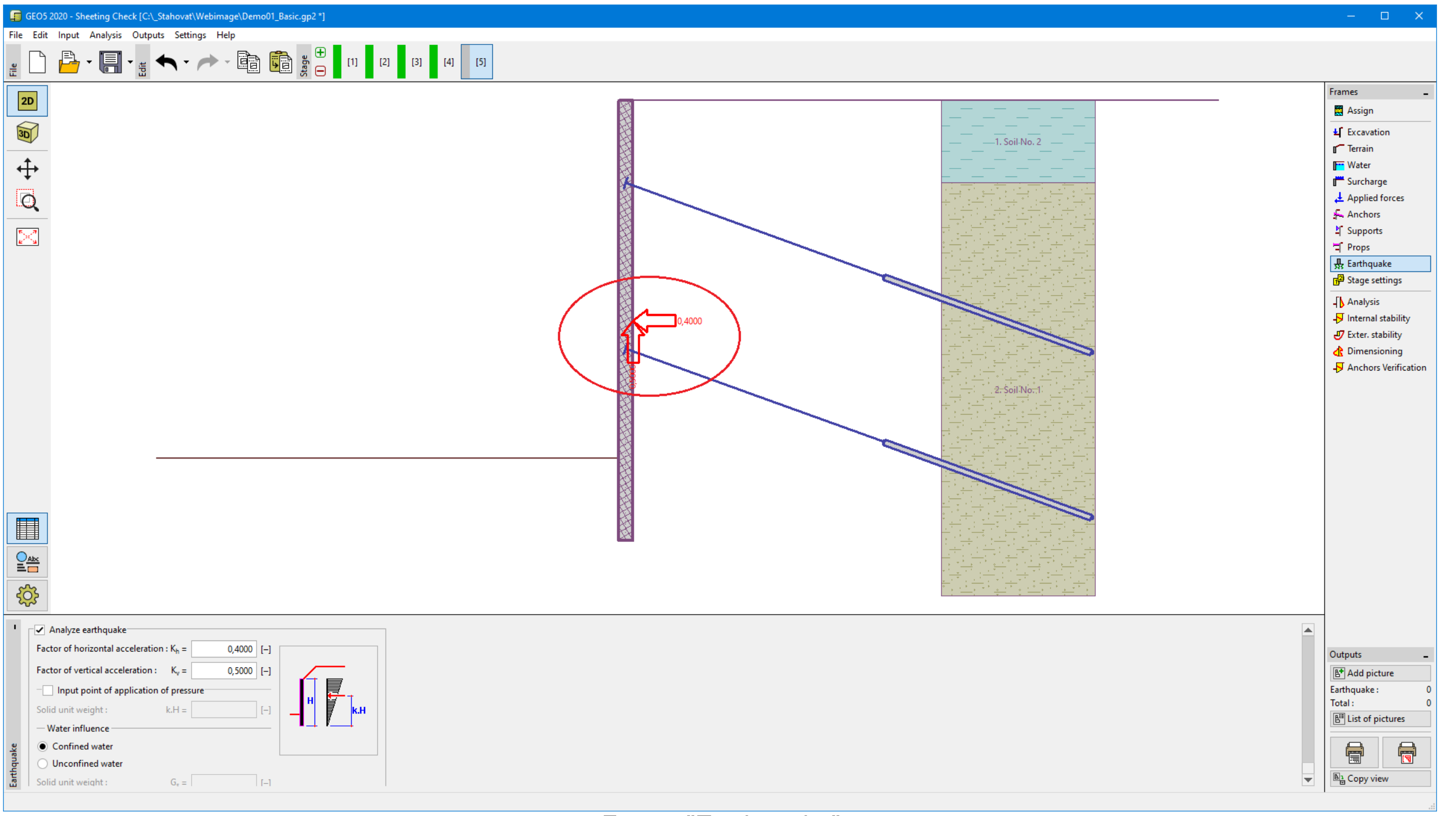
Frame "Supports"

Earthquake

The "**Earthquake**" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements, the coefficients k_h and k_v can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for an earthquake is described in the theoretical part of the help in the chapter "Influence earthquake".



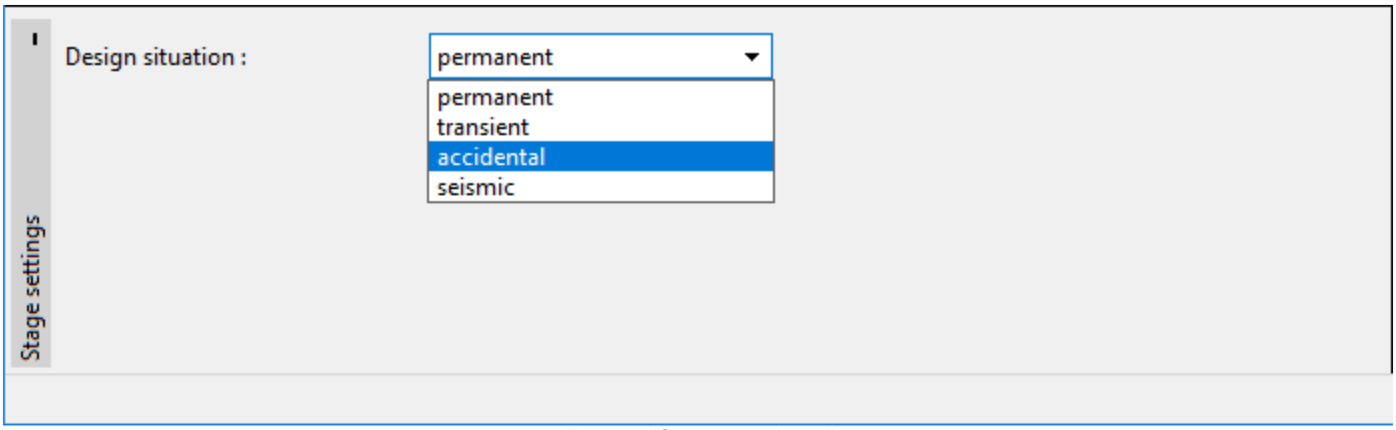
Frame "Earthquake"

Stage Settings

The frame "**Stage settings**" serves to input settings valid for a given construction stage.

The selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology.



Frame "Stage settings"

Analysis

The frame "**Analysis**" displays the analysis results. The frame contains six buttons to show them:

K_h + pressures

Variation of the modulus of subsoil reaction is displayed in the left part of the desktop (by default, a blue color with hatching). In a method of depending pressures, some of the springs (values of modules of subsoil reaction) are removed from the analysis (spring stiffness is equal to zero). If all springs are out, the analysis **may fail to converge.** The program exists without finding a solution. An error message appears in the bottom part of the frame. It is necessary to change **the construction** - e.g., add an anchor, change a depth of excavation, improve soil parameters, etc.

Some construction stages display (by default, a yellow dotted line is assumed) deformation at the onset of mobilization of the earth pressure at rest - this is a complementary information showing plastic deformation of a structure.

Distributions of limiting pressures (by default, a green dashed line is assumed) are presented in the right part of the window (passive pressure, pressure at rest and active pressure). The **actual pressure acting on a structure** is plotted in a solid blue line.

Both **deformed** (by default, a solid red color is assumed) and undeformed structure appears in the right part of the desktop. Forces and displacements developed in anchors, supports and props are also shown.

Internal forces

Plot of a structure with forces acting in anchors, reactions and deformations of supports, and props appear in the left part of the desktop. Distributions of bending moment and shear force are then plotted on the right.

Displacement + Stress

Plot of a structure with forces acting in anchors, reactions and deformations of supports, and props appear in the left part of the desktop. The deformed shape of a structure with overall pressure acting on a sheeting structure is then plotted on the right.

Terrain settlement

Plot of a structure is displayed in the left part of the desktop. Terrain settlement is plotted on the right. Method for calculation of terrain settlement is selected in the upper part of frame.

Utilization of Passive Pressure

Plot of a structure is displayed in the left part of the desktop. Distributions of limit pressures and actual pressure acting on a structure are presented in the right part of the window. Safety factor for utilization of passive pressure is inputted in the upper part of the frame.

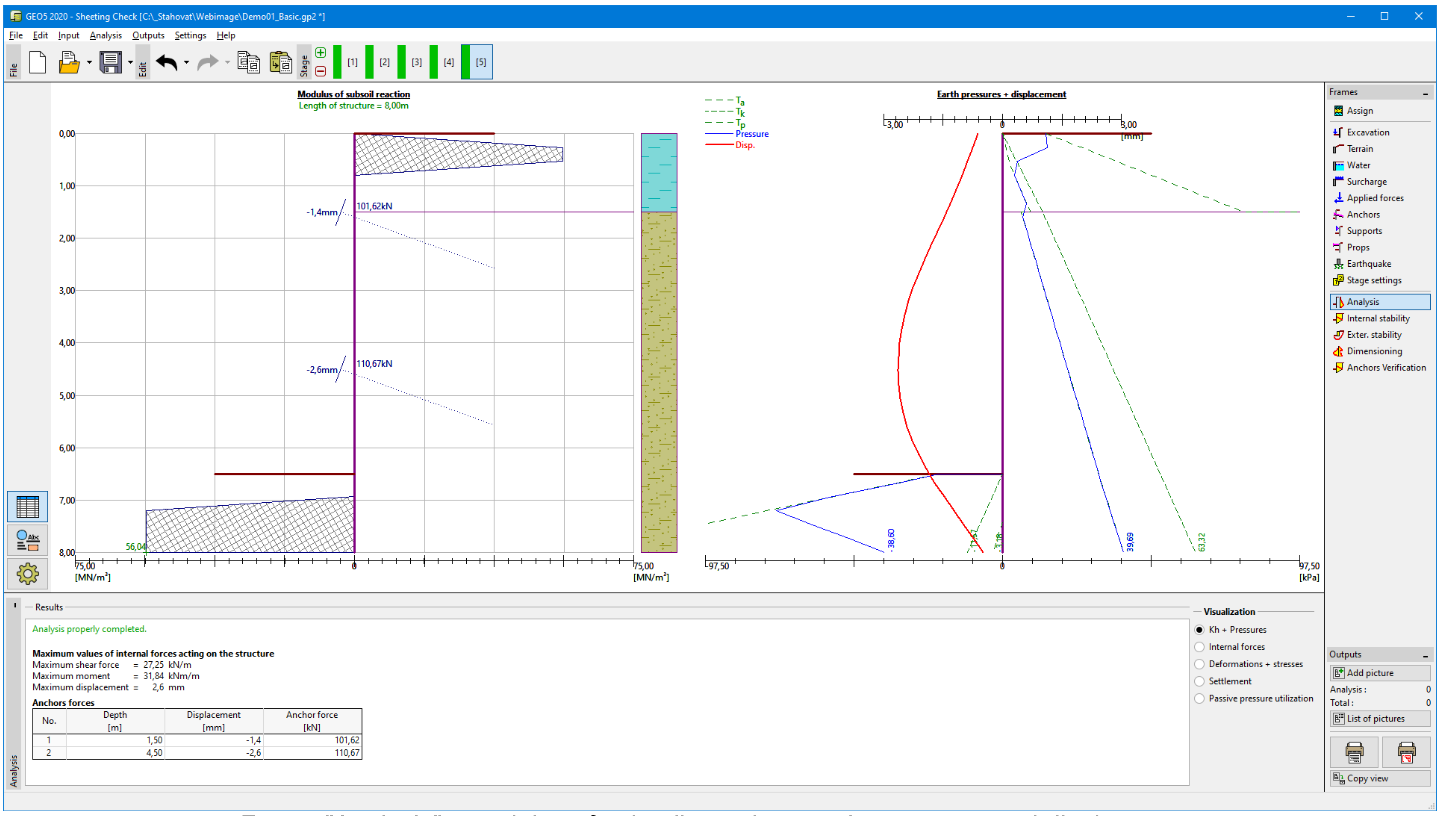
Vertical capacity

Plot of a structure is displayed in the left part of the desktop. Distributions of limit pressures and actual pressure acting on a structure are presented in the right part of the window. Parameters for vertical bearing capacity are inputted in the upper part of the frame.

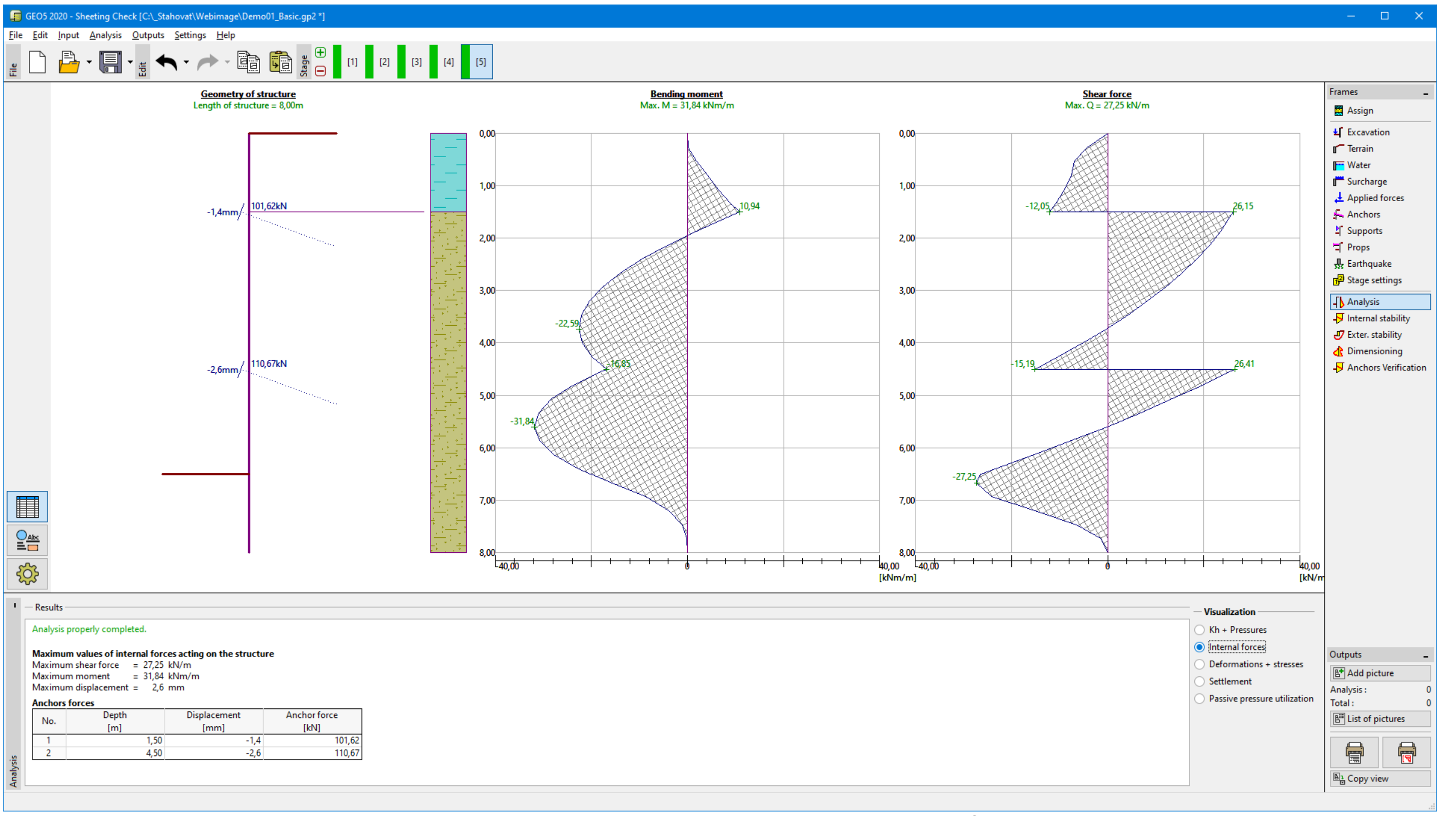
Providing the modulus of subsoil reaction is found by iteration it is necessary to check the **course of manual iteration** in the dialog window "**Iteration**". Details are provided in the theoretical part of the help, chapter "Modulus of subsoil reaction determined by iteration".

In the case of double-row wall, it is possible to show internal forces, displacement, and stress on the front or back row.

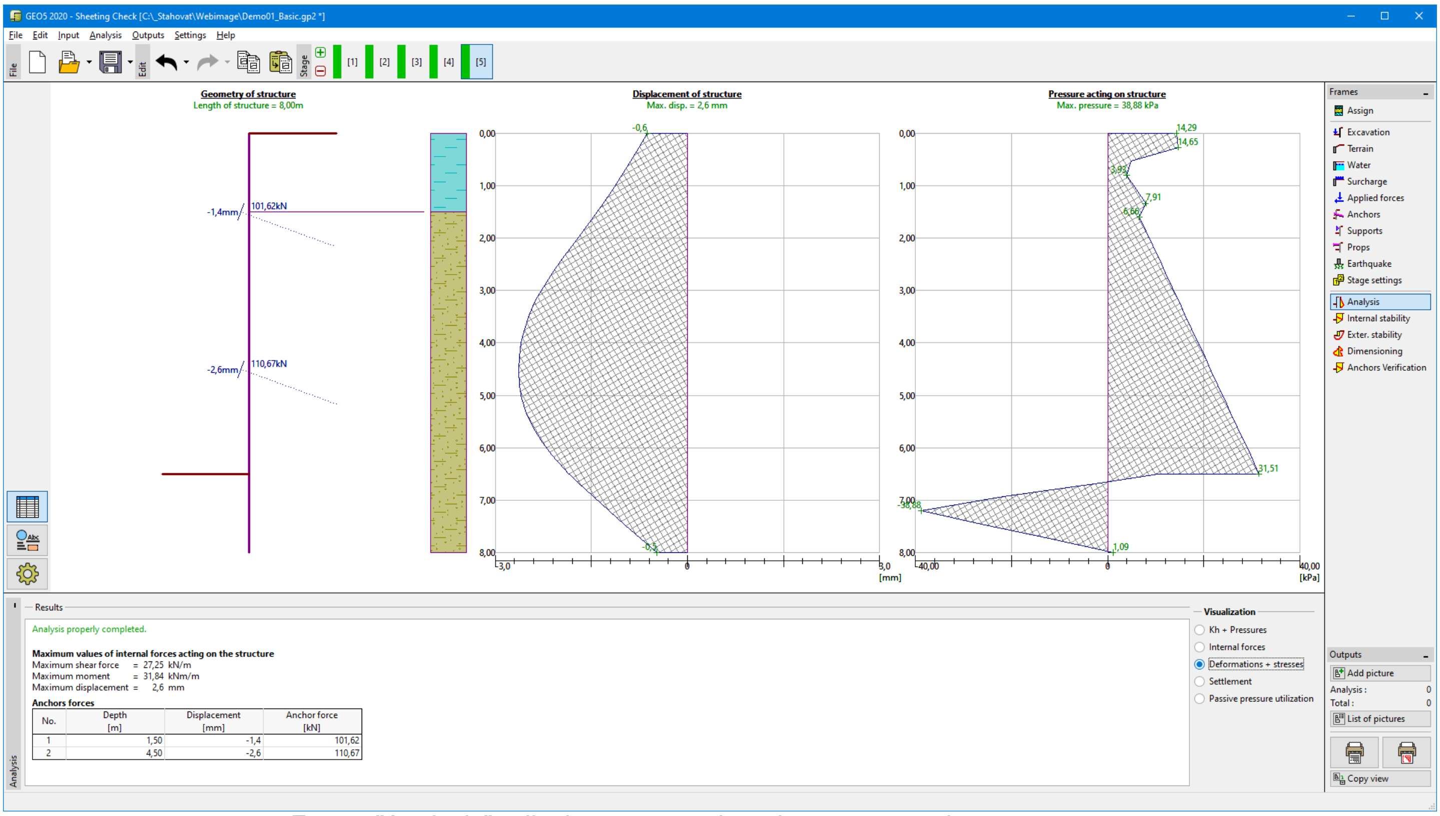
Visualization of results can be adjusted in the frame "Drawing Settings".



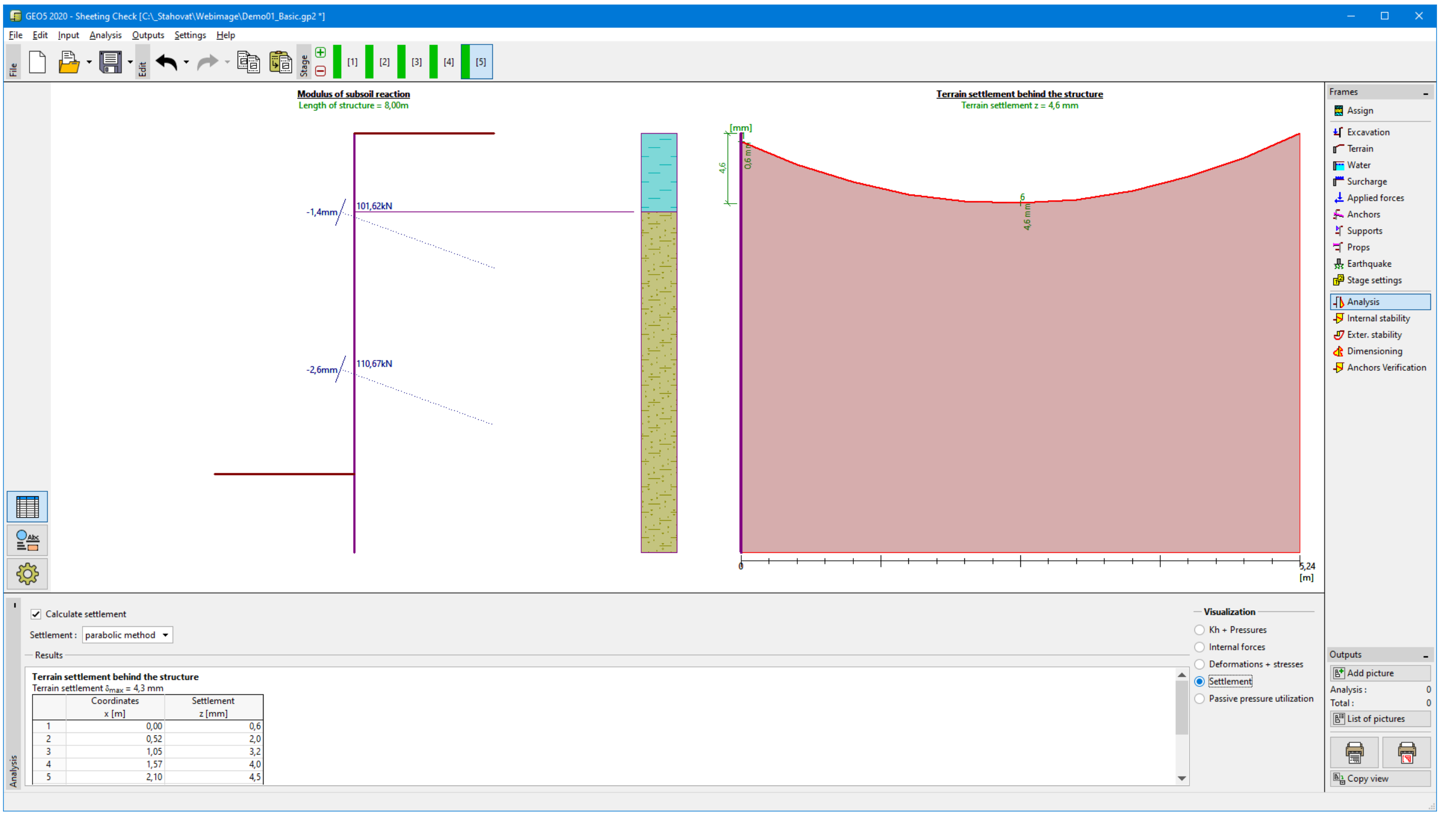
Frame "Analysis" - modulus of subsoil reaction, earth pressures and displacement



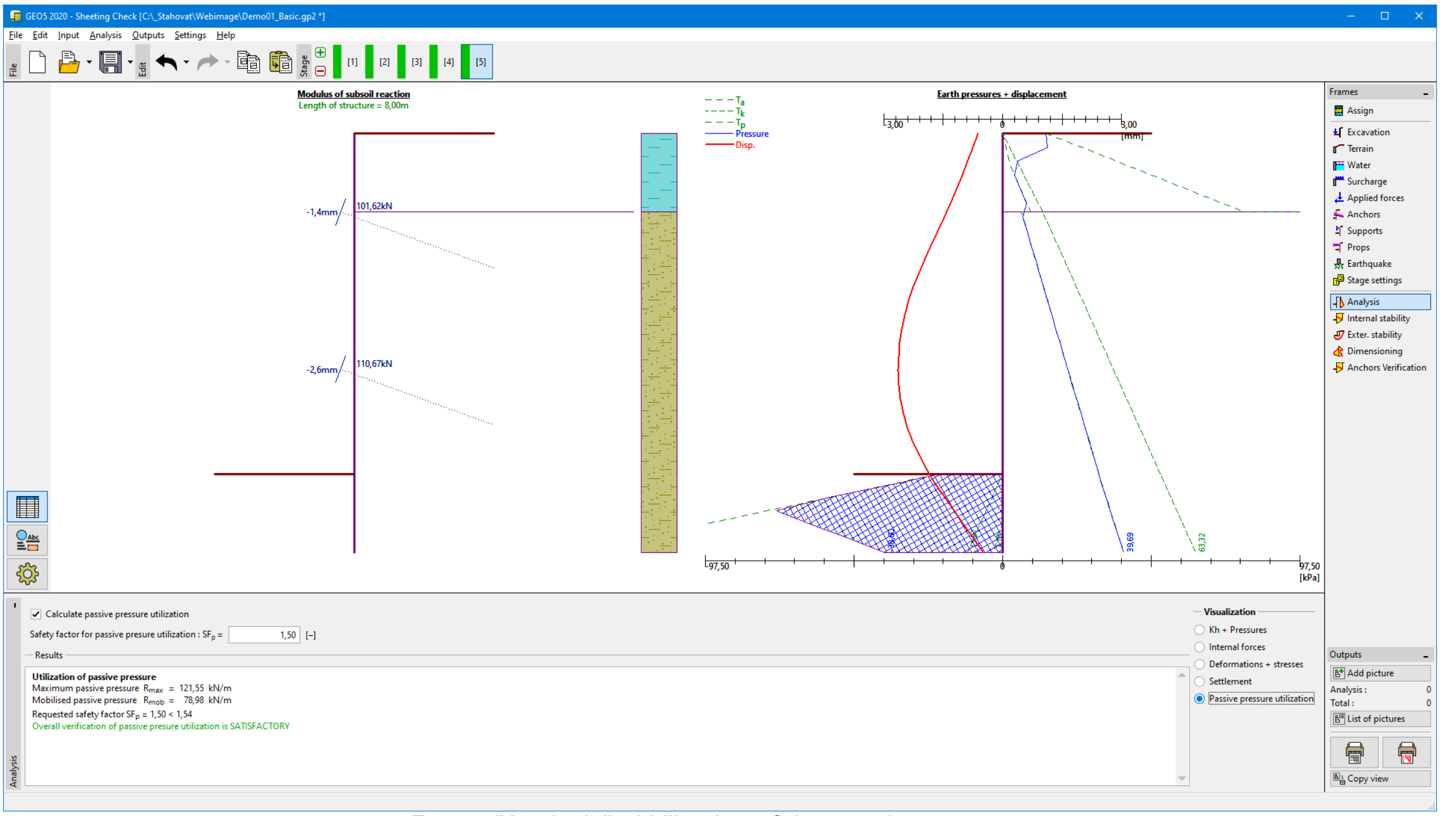
Frame "Analysis" - bending moment and shear force



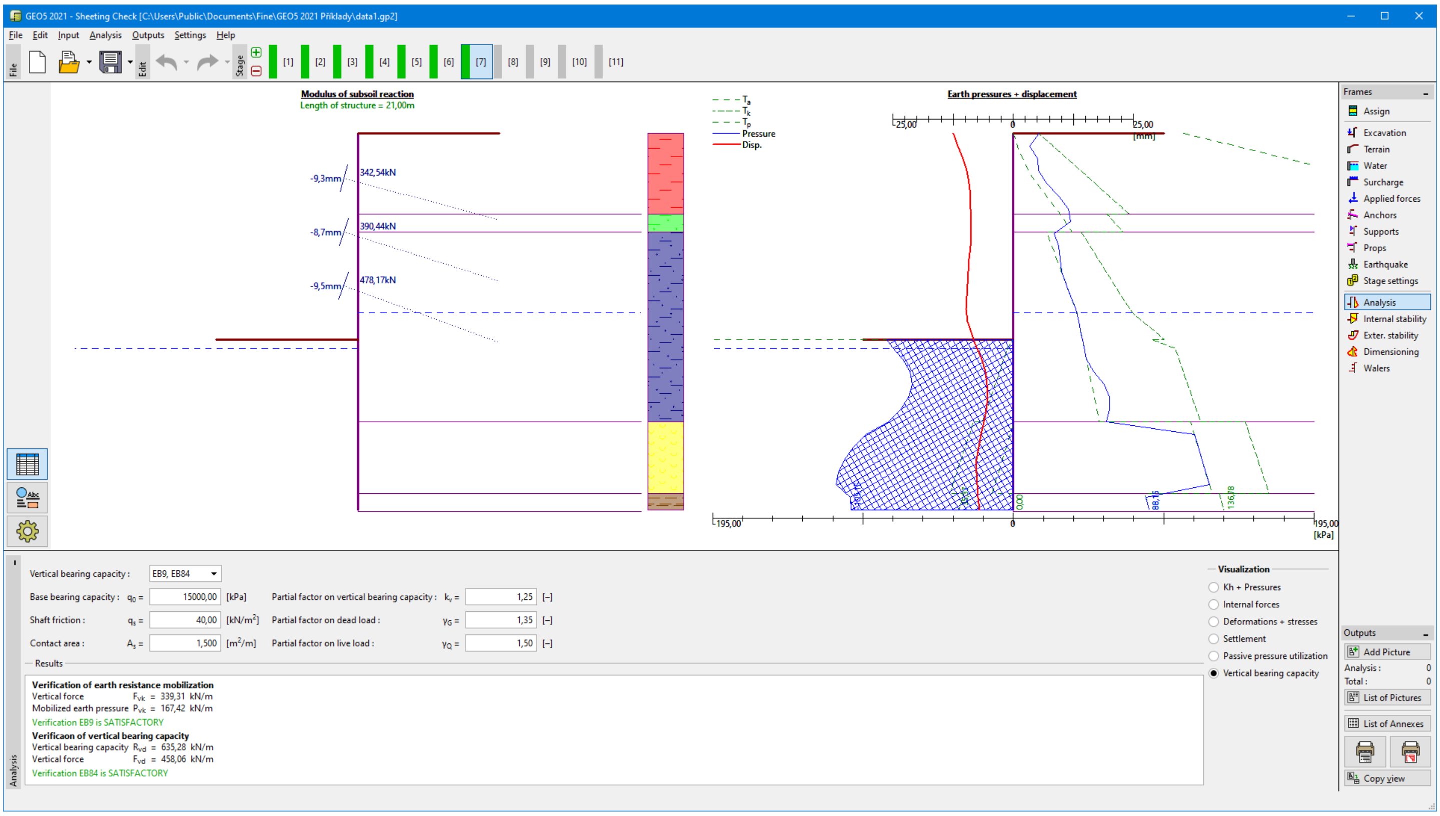
Frame "Analysis" - displacement and earth pressure acting on a structure



Frame "Analysis" - terrain settlement



Frame "Analysis" - Utilization of the passive pressure

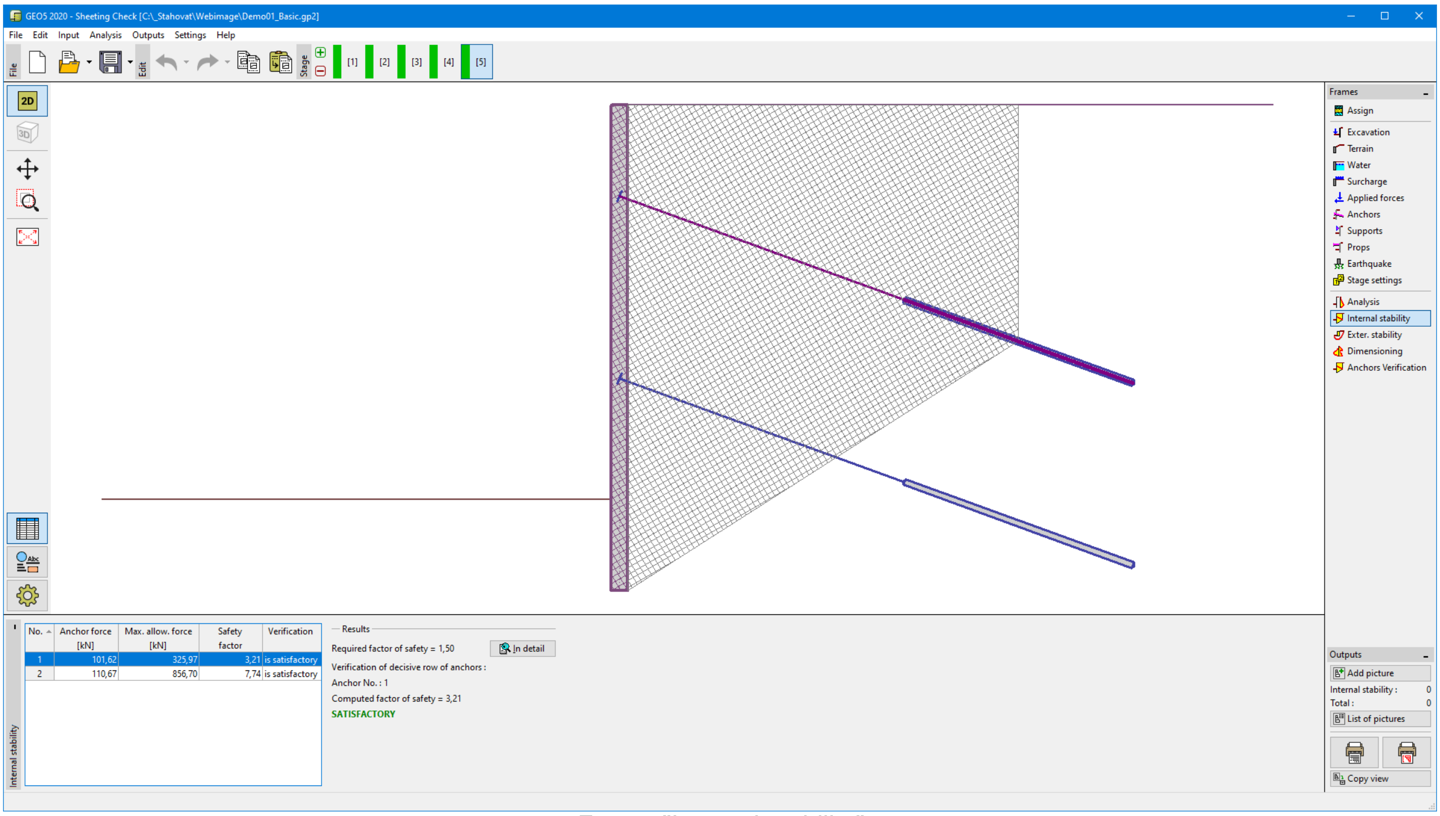


Frame "Analysis" - Vertical bearing capacity

Internal Stability

This frame serves to check the internal stability of anchors - the frame is therefore accessible only in stages, in which the anchors are introduced. For each row of anchors, the table shows input **anchor forces** andthe **maximum allowable forces** in each anchor.

The overall check for the most stressed row of anchors is displayed in the right part of the frame.

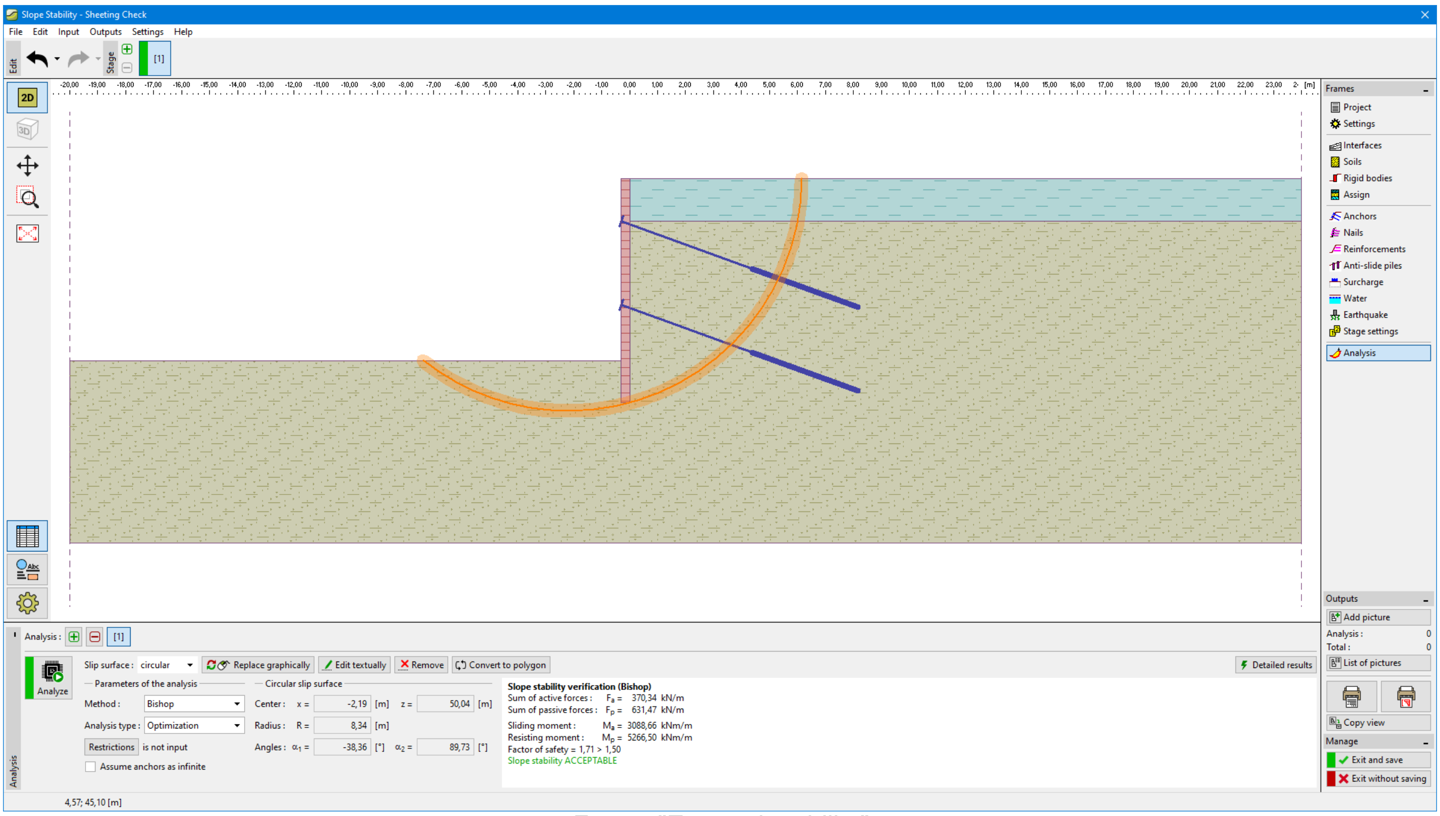


Frame "Internal stability"

External Stability

By pressing the "**External stability**" button launches the "**Slope stability**" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program "**Slope stability**" is installed.

After completing all analyses, press the "**Exit and save**" button to leave the program - all data are then carried over to the analysis protocol of the "**Sheeting check**" program.

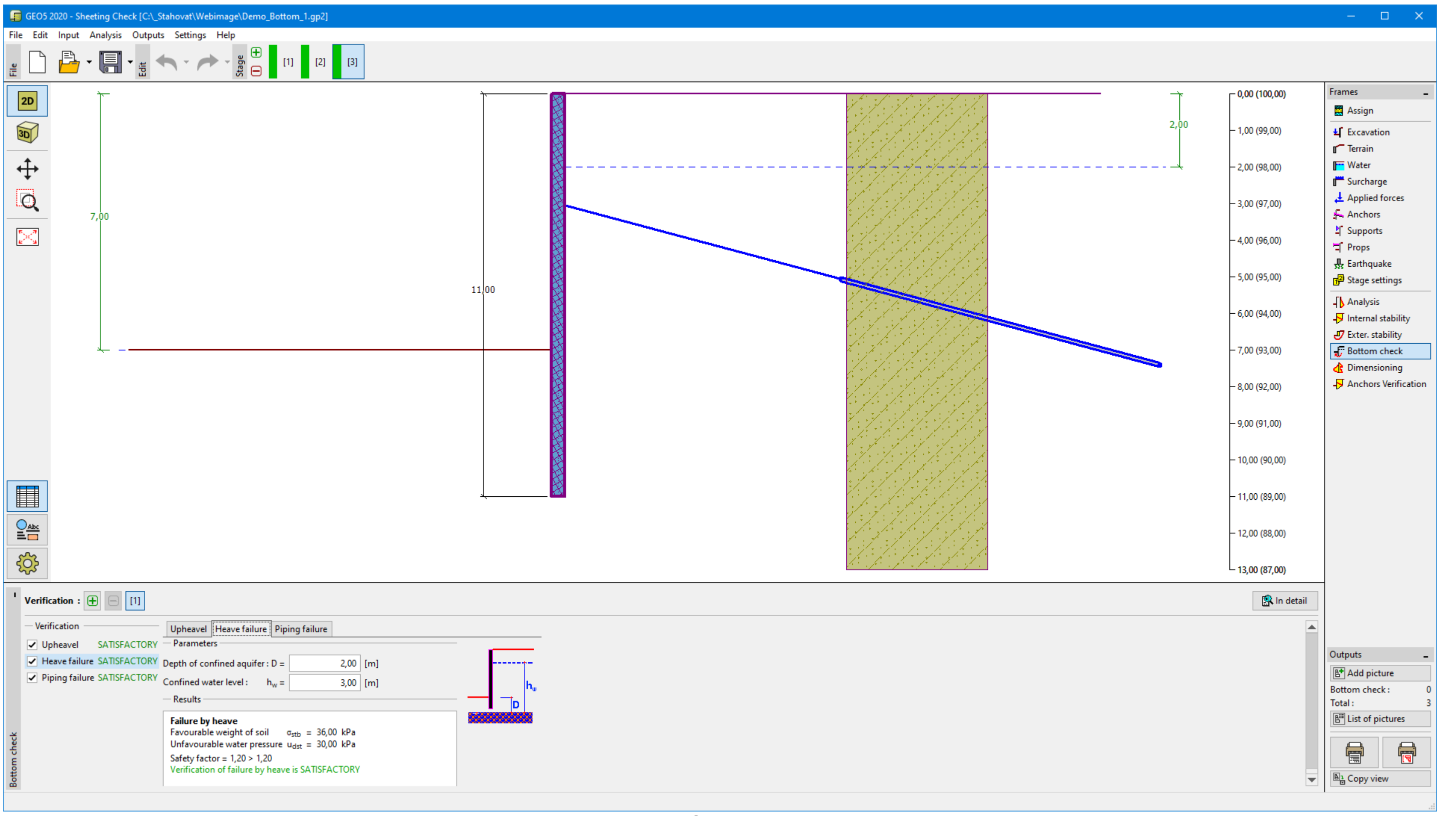


Frame "External stability"

Bottom Check

The "**Bottom Check**" frame serves to check the failure by heave and failure by piping. The frame is accessible only in case, where the influence of water is considered as the "hydrodynamic pressure" (the base of a structure is in permeable subsoil, which allows free water flow below the structure).

In the case of analysis according to Chinese Standards, the program allows us to do these verifications for all GWT types.



"Bottom Check" Frame

Dimensioning

In the frame "**Dimensioning**", it is possible to display an envelope of internal forces and displacements from all analyses (stages of constructions). Normally, the envelope is constructed from the results of all construction stages, however, it can only be created from the **selected stages**. The **"Modify"** button opens the dialog window **"Construction stage selection"**, where it is possible to select the constructions stages that are used to generate the current envelope (by pressing corresponding buttons).

The maximum values of calculated internal forces (bending moments and shear forces) and the magnitude of displacement are displayed at the bottom part of the frame.

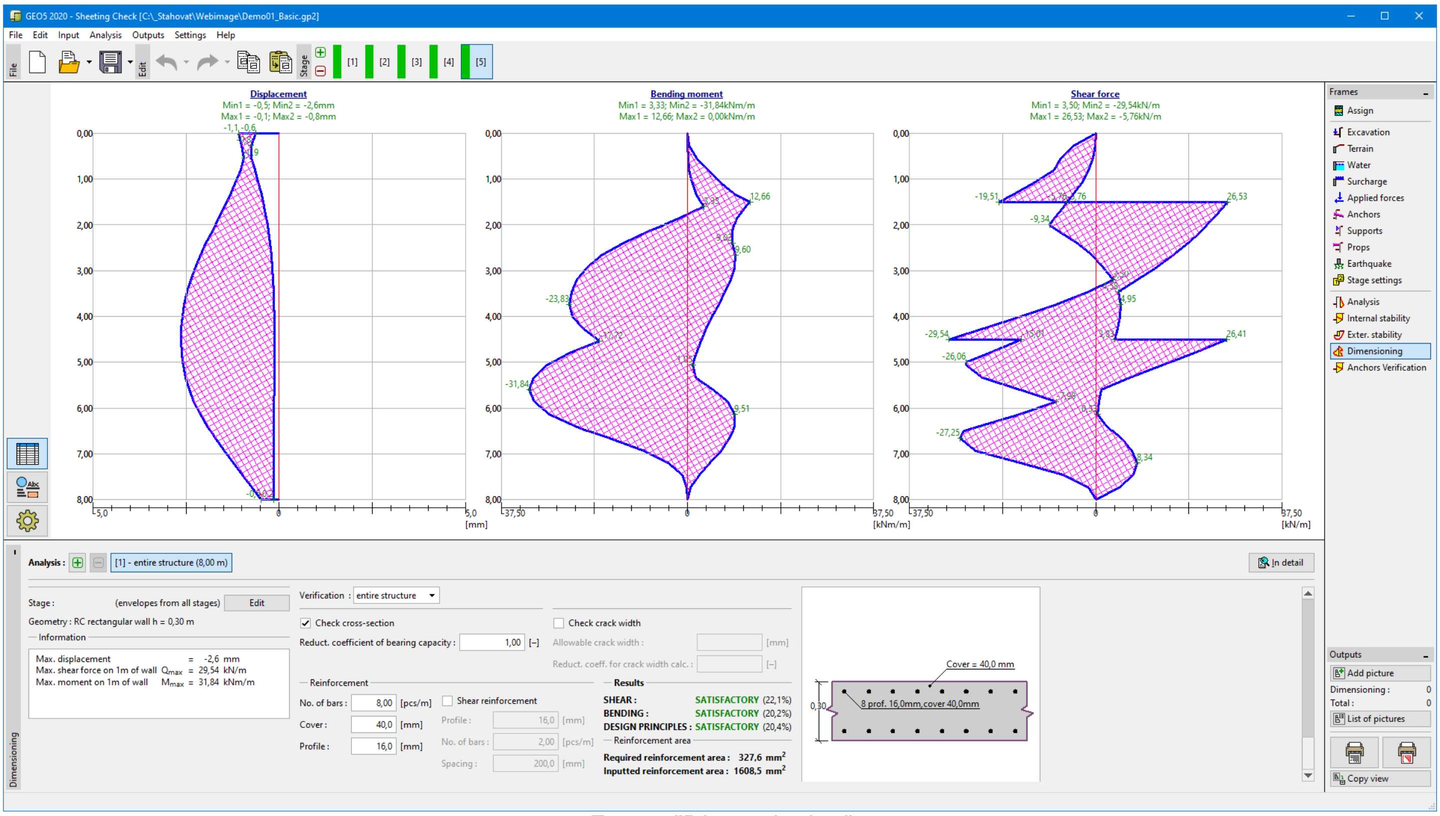
The program allows us to dimension reinforced concrete, steel and timber, plastic, or combined cross-sections (by checking the option "**Check cross-section**"). For a more detailed design of reinforcement in concrete cross-section, it is possible to divide the structure into sections which are then assessed separately.

When checking the cross-section, it is possible to input the **reduction coefficient of bearing capacity**, which reduces the overall bearing capacity of a cross-section. When performing the analysis **with the reduction of earth pressures** this coefficient should be considered 1.0. For analysis **without earth pressures reduction** (to ensure a realistic behavior of a sheeting structure), it is necessary to increase the calculated forces, by adopting a coefficient greater than 1.0 (For EN 1997 is the value in interval 1.35 - 1.5).

For dimensioning of steel cross-sections, it is possible to assume **influence of normal force** in these ways:

- **normal forces - do not consider**: program doesn't consider influence of normal force.
- **normal forces - from nearest anchor**: program assumes maximum value of local normal force near the anchor as $N = F*\sin\alpha$, where α represents slope of the anchor
- **normal forces - sum of all anchors**: program adds influence of normal force from input anchors as the sum of influence of all anchors.
- **normal forces - input**: user-defined value of normal force N

The frame allows us to perform a larger number of analyses pro dimensioning of a cross-section. The **"In detail"** button at the right part of the frame opens the "**Dimensioning**" dialog window to show detailed results.



Frame "Dimensioning"

Anchors Verification

The "**Anchors verification**" frame contains a table with a list of input anchors.

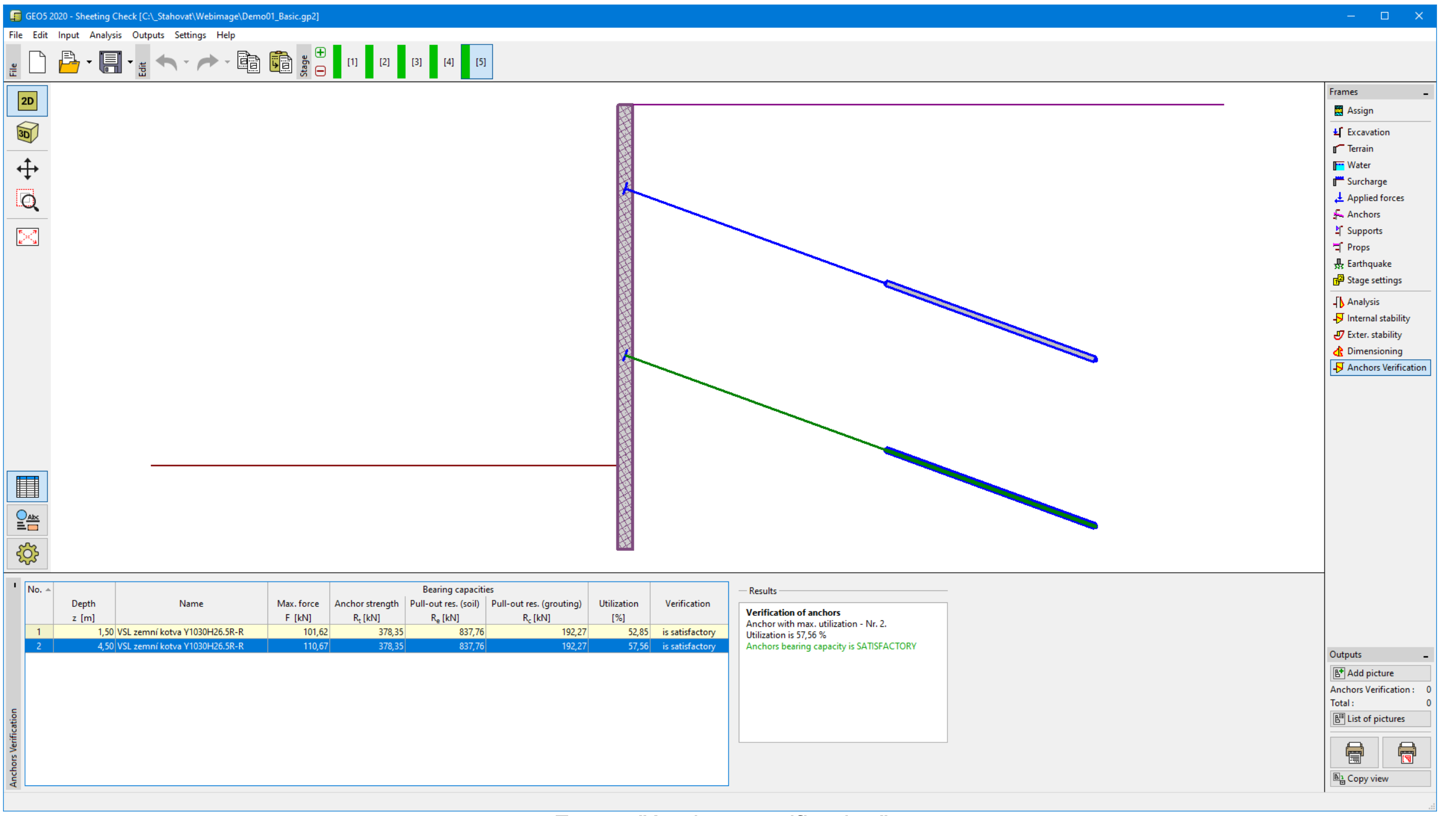
The **overall bearing capacity** of anchors is verified in this frame.

The anchors can be check for three different **types of failure**.

- Strength of anchor R_t
- Pull-out resistance (soil) R_e
- Pull-out resistance (grouting) R_c

Computed **bearing capacities** of anchors **are reduced** by corresponding safety factor or reduction coefficient, which are defined in the frame "Settings", tab "Anchors". The verification of anchor is satisfactory, when the maximum force in the anchor P_{max} is lower than all computed bearing capacities.

$$\min\left(\frac{R_t}{SF_t};\frac{R_e}{SF_e};\frac{R_c}{SF_c}\right) \geq P_{max}$$



Frame "Anchors verification"

Lagging

The frame "**Lagging**" serves to design and verify the lagging. This frame is available for the soldier pile walls only.

The program allows us to dimension reinforced concrete and timber cross-sections (by checking the option "**Check cross-section**").

The "**Edit Lagging**" dialog window (the "**Material, Section**" button) allows us for selection of **material** (timber, concrete) and **cross-section** (rectangle, circle). The "**Catalog**" and "**User. defined**" buttons are used to set the material properties in the same way as in the "Material" frame.

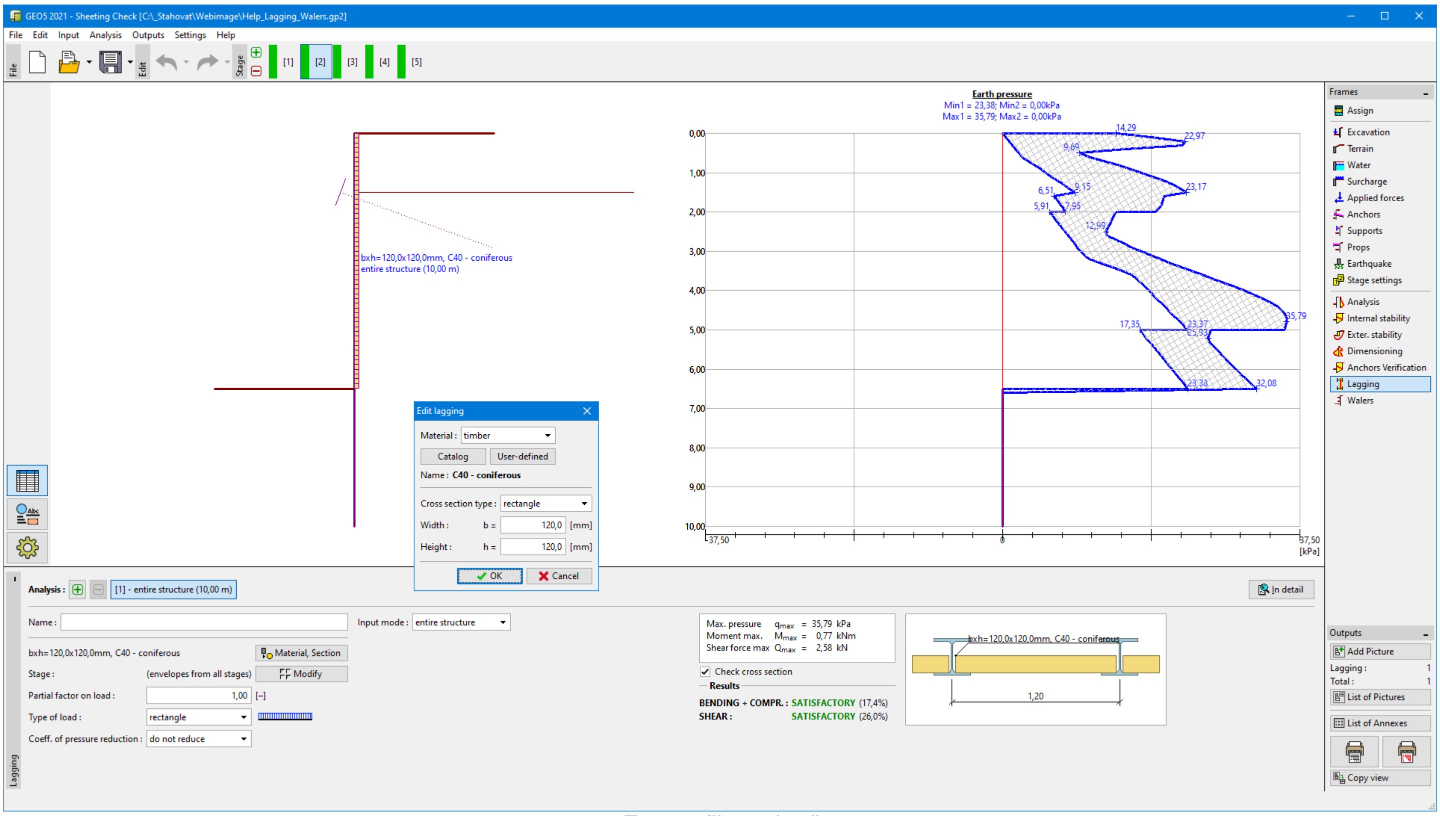
In the frame, it is possible to display an envelope of pressures acting on the wall from all analyses (stages of constructions). Normally, the envelope is constructed from the results of all construction stages. The **"Modify"** button opens the dialog window **"Construction stage selection"**, where it is possible to select the construction stages that are used to generate the current envelope (by pressing corresponding buttons).

The frame further allows for selecting the **load type** (rectangle, triangle) and **coeff. of pressure reduction** acting on the lagging. When checking the cross-section, it is possible to input the **partial factor on load**, which multiplies the values of internal forces.

The values of calculated internal forces (bending moments and shear forces) and the maximum pressure are displayed in the right part of the frame.

The frame allows us to perform a larger number of analyses for the dimensioning of a cross-section. For a more detailed design of reinforcement in concrete cross-section, it is possible to divide the structure into sections which are then assessed separately.

The **"In detail**" button at the right part of the frame opens the "**Dimensioning**" dialog window to show detailed results.



Frame "Lagging"

Soil Mix

The frame "**Soil Mix**" serves to verify the Soil Mix. This frame is available only for the soil mix wall with steel section.

In the right part, the envelope of pressures acting on the wall is displayed for all analyses. The envelope is constructed from the results of all construction stages. The **"Modify"** button opens the dialog window **"Construction stage selection"**, where it is possible to select the construction stages that are used to generate the current envelope (by pressing corresponding buttons).

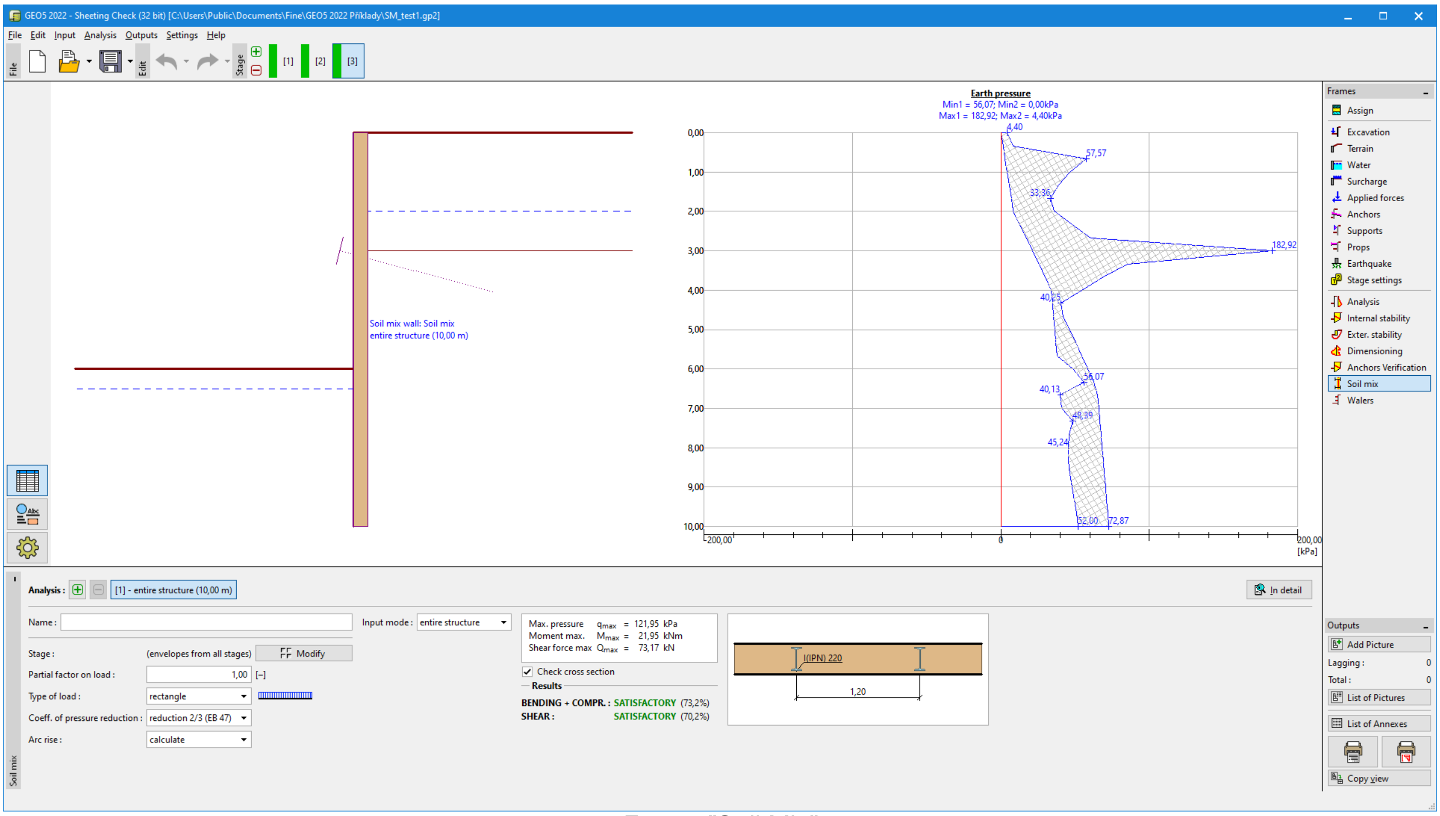
The frame further allows for selecting the **load type** (rectangle, triangle), arc rise, and **coefficient of pressure reduction** acting on the wall.

The values of calculated **internal forces (bending moments and shear forces) and the maximum pressure** are displayed in the right part of the frame. Their calculation is the same as for lagging.

Verification can be done for the **whole wall height**, or only to the **depth of the excavation**. In the case of a combined wall with more sections, it is necessary to select a specific section.

The **"In detail"** button at the right part of the frame opens the dialog window with detailed results.

The frame allows us to perform a larger number of analyses.



Frame "Soil Mix"

Walers

This frame serves to design and verify the waler for selected anchor or prop.

The program allows us to dimension reinforced concrete and steel cross-sections (by checking the option "**Check cross-section**"). The frame allows us to perform a larger number of analyses for dimensioning of a cross-section.

The "**Edit Waler**" dialog window (the "**Material, Section**" button) allows us for a selection of **material** (steel, concrete) and **cross-section**. The "**Catalog**" and "**User. defined**" buttons are used to set the material properties in the same way as in the "Material" frame.

In the case of the steel waler, the **cross-section type** (I-section, 2xI-section, or 2xU-section) is selected in the "**Catalog of profiles**" (the "**Catalog**" button) or in the "**Cross-section editor**" (the "**Welded**" button).

The **waler rotation** can be considered according to the **anchor (prop)** or the **wall**.

The **"Modify"** button opens the dialog window **"Construction stage selection"**, where it is possible to select the construction stages used to determine the maximum force.

When checking the cross-section, it is possible to input the **partial factor on load**, which multiplies the values of internal forces. For dimensioning of steel cross-sections, it is possible to assume the **influence of normal force.**

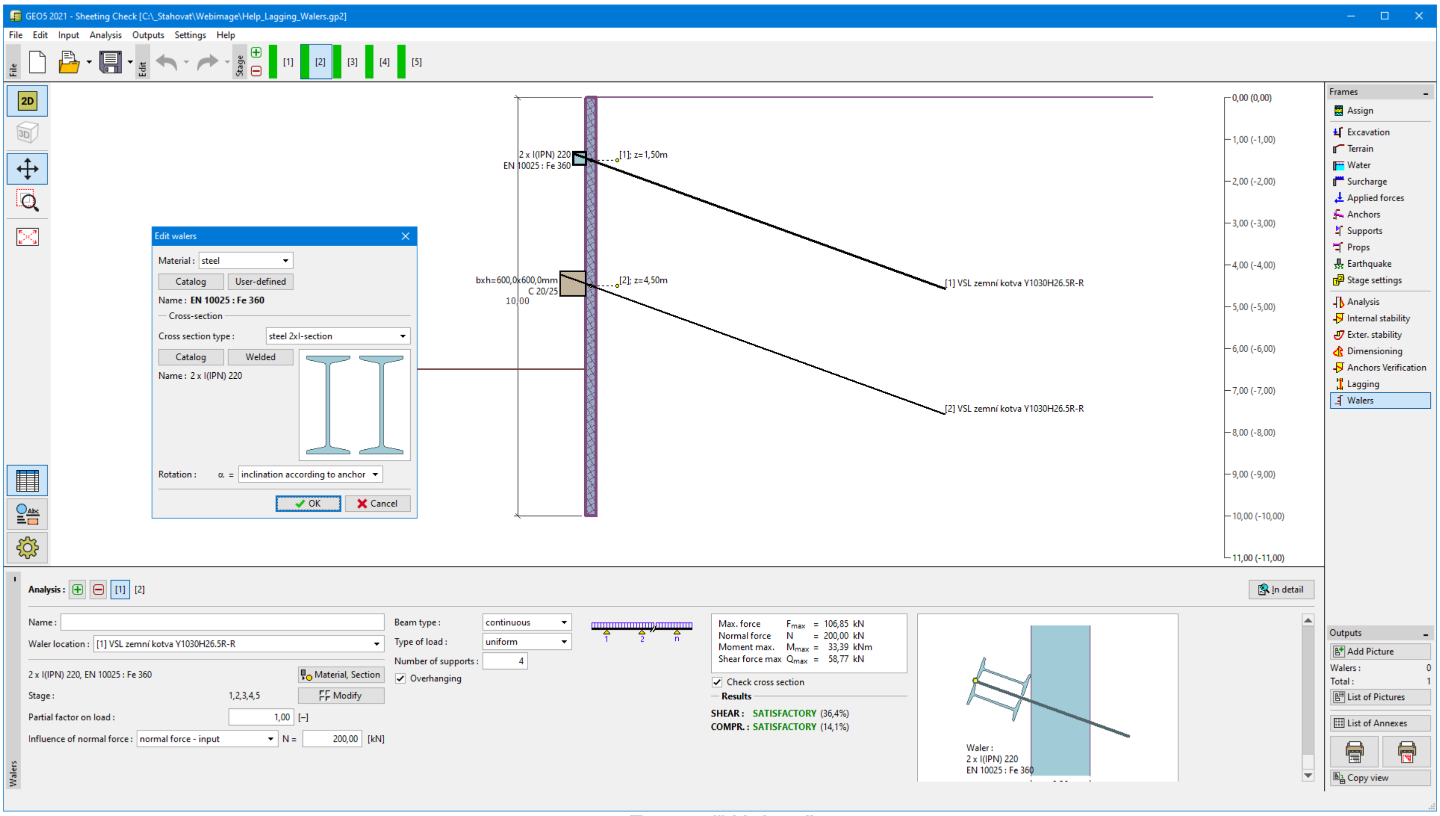
The frame further allows for selecting the **static scheme**:

- **Beam type** (simply supported, continuous)
- **Load type** (point, uniform)
- **Number of supports**
- **Overhangs**

For a **simply supported beam** loaded by **concentrated load**, the support distance L is input.

The values of calculated internal forces (bending moment and shear force) and the maximum force are displayed at the bottom part of the frame.

The **"In detail"** button at the right part of the frame opens the "**Dimensioning**" dialog window to show detailed results.



Frame "Walers"

Program Anti-Slide Pile

This program is used for the design of pile walls stabilizing slope movement or increasing the safety factor of the slope. The first analysis should be done in the Slope Stability program, where the active and passive forces acting on the pile wall are computed. Next, the data from the "Slope stability" program are transferred into the Anti-Slide Pile program, where other analyses are performed (determination of internal forces on pile, pile deformation, and dimensioning of pile reinforcement).

The help in the program "Anti-Slide Pile" includes the following topics:

- The input of data into individual frames:

Project	Settings	Profile	Modulus Kh	Pressuremeter Tests (PMT)	Dilatometric Tests (DMT)	Soils
Geometry	Material	Pressure Determination	Rock	Assign	Front Face	Terrain
Water	Surcharge	Applied Forces	Anchors	Supports	Earthquake	Stage Settings
Analysis	Dimensioning	Anchors Verification	Lagging	Walers		

- Standards and analysis methods

- Theory for analysis in the program "**Anti-Slide Pile**":

Stress in Soil Body	Earth Pressures	Sheeting Check	Anti-Slide Pile	Dimensioning of Concrete Structures	Dimensioning of Steel Cross-sections

- Outputs

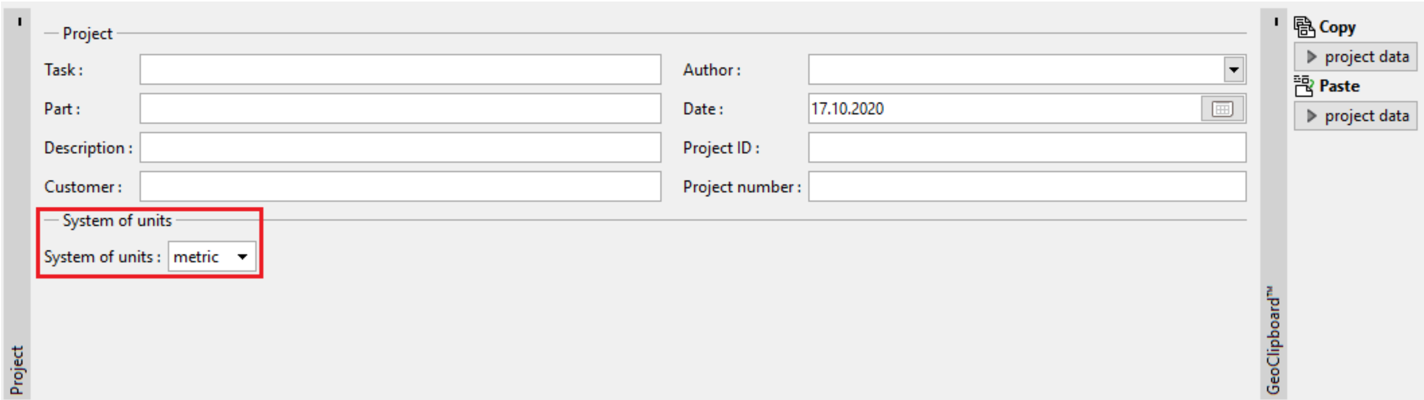
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The **"Project"** frame is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The "**Settings**" frame allows us to introduce the basic settings, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

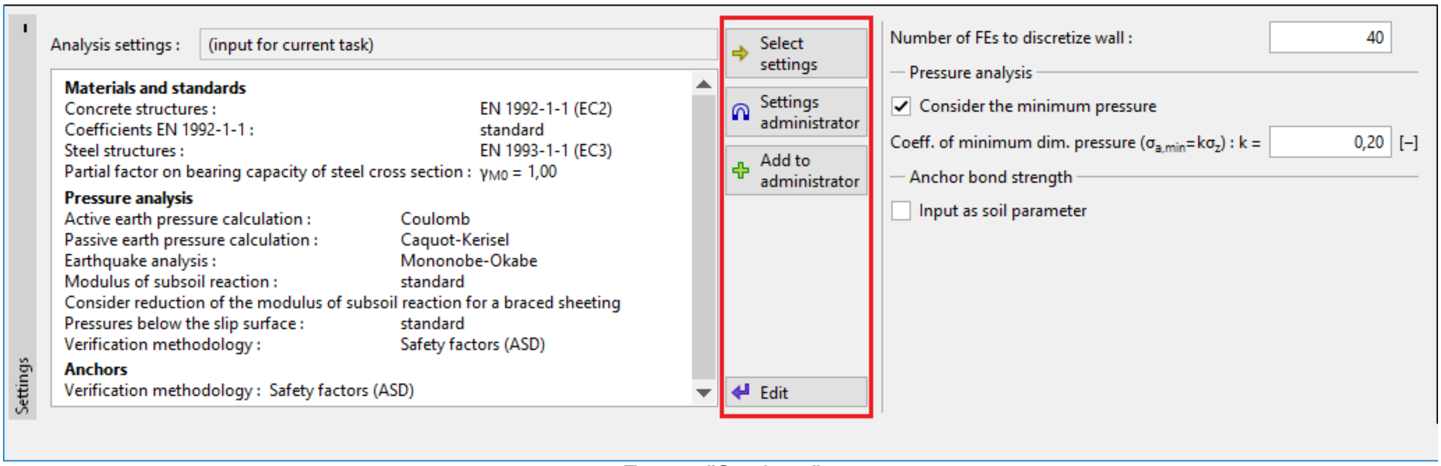
The "**Input for the current task**" setting is usually created when importing older data.

Settings of analysis parameters are performed in the "Materials and standards" and the "Pressure Analysis" tabs.

When performing analysis according to EN 1997 or according to the theory of limit states, the program enables to set whether to reduce the soil parameters for the calculation of limit pressures. For correct modeling of a real behavior of the structure, we recommend not reducing them.

The frame allows the user to specify the subdivision of a wall into finite elements (by default, the number of elements equals 100) and specify whether the structure is loaded by the minimum dimensioning pressure.

When calculating the pull-out resistance of anchor, bond strength can be inserted as a soil parameter.



Frame "Settings"

Profile

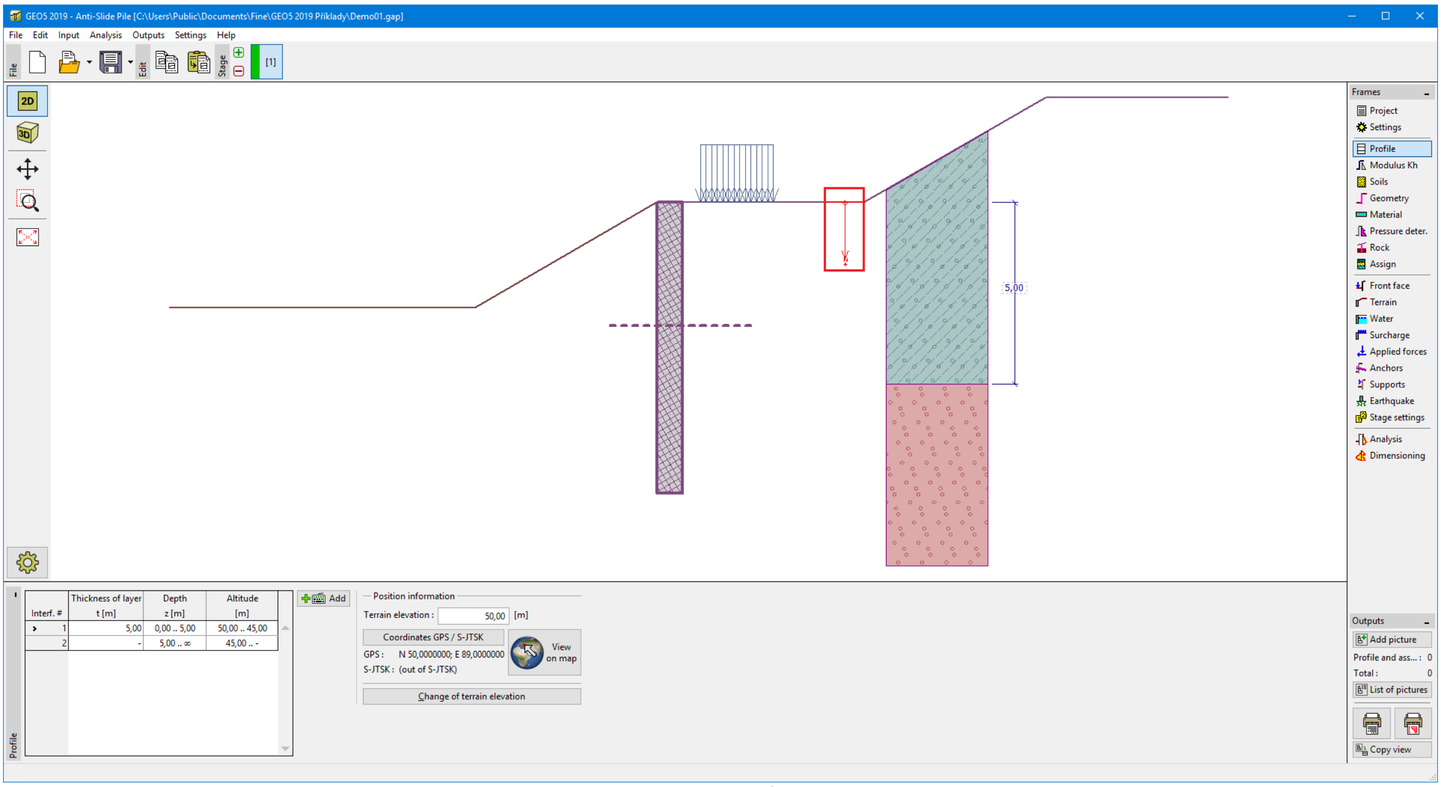
The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.



Frame "Profile"

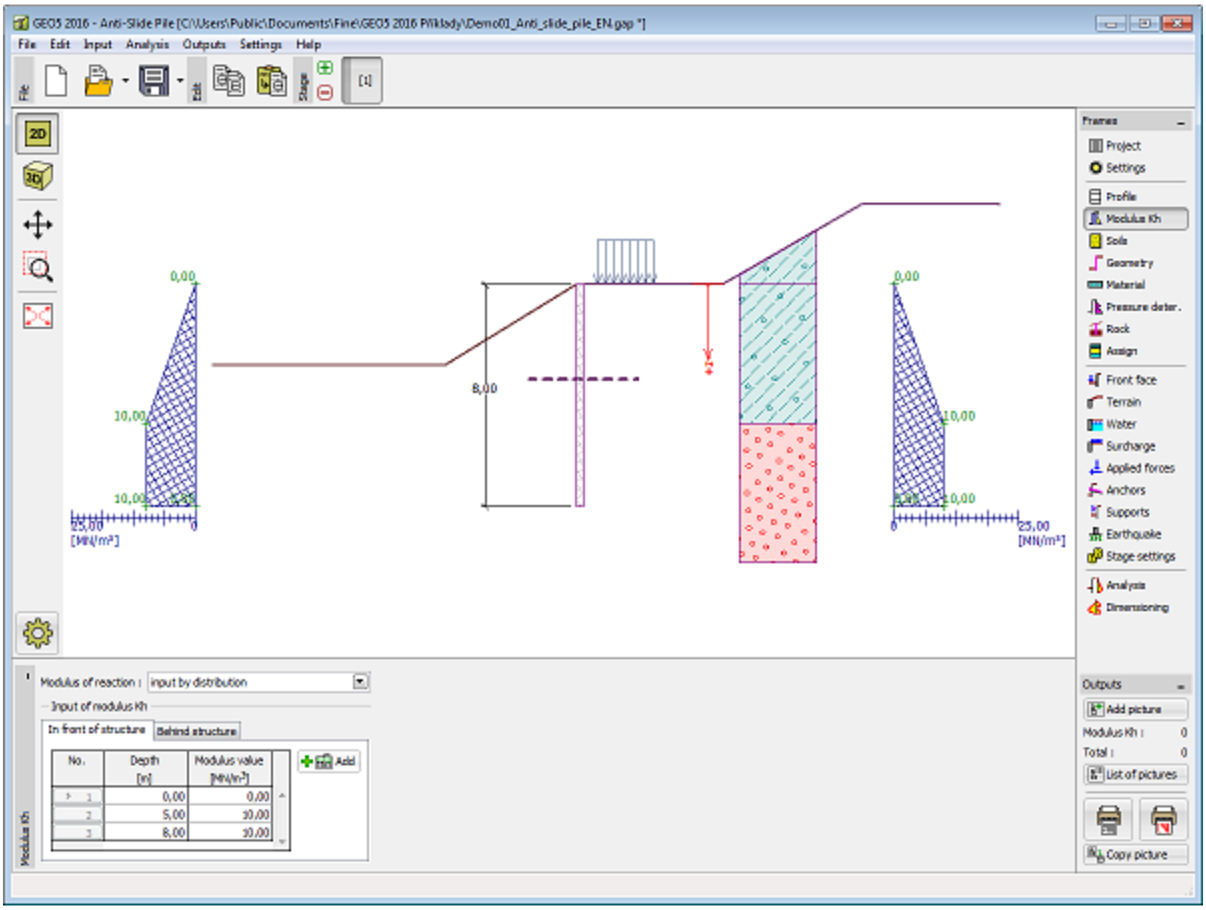
Modulus Kh

This frame serves to specify a type of analysis for calculation of the modulus of subsoil reaction, which is an important input parameter when analyzing a sheeting structure by using the method of dependent pressures.

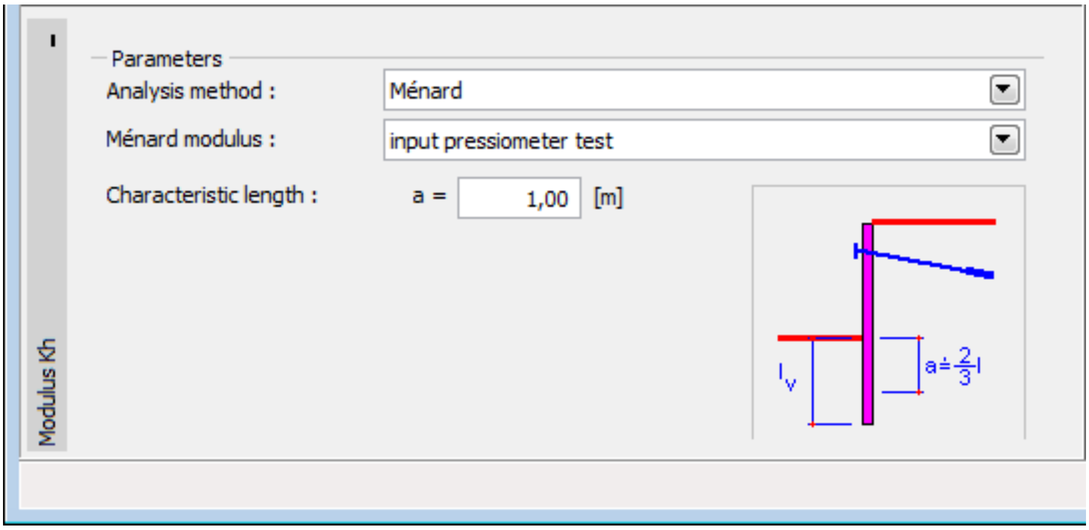
The way of calculation of the modulus of subsoil reaction k_h is selected in the "Settings" frame (in the "**Edit current settings**" dialog window in the "Pressure Analysis" tab).

The frame can take different forms depending on the selected method of calculation:

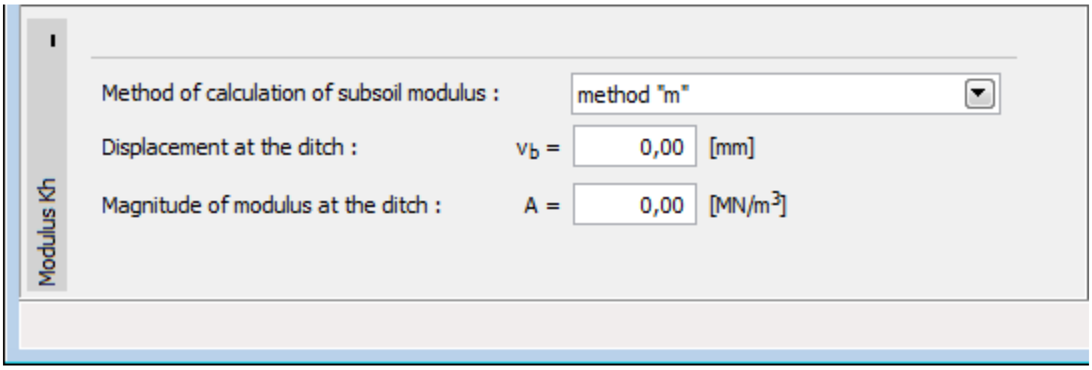
- **standard** (option "analyze – Schmitt", "analyze – Chadeisson", "manual iteration", or "automatic iteration")
- **input** (selecting the option "**Input by distribution**" opens a table in the frame that allows us to input the values of the modulus of subsoil reaction k_h both in front of and behind the structure. For option "**Input as a soil parameter**" the modulus k_h is specified in the "Soils" frame, where the modulus of subsoil reaction is considered either as **linear**, or as **nonlinear - curve**)
- **pressuremeter PMT** (modulus of subsoil reaction k_h is calculated from pressuremeter test, or input as a parameter of soil in the "Soils" frame. Then there is specified method of calculation - according to NF P 94-282 or according to Menard)
- **Chinese standards** (for "m" method is defined horizontal displacement at the ditch bottom v_b $[mm]$ and magnitude of modulus A $[MN/m^3]$, or option input as a parameter of soil – "c" method, "k" or "m" method)



Frame "Modulus k_h" - selection "input by distribution"



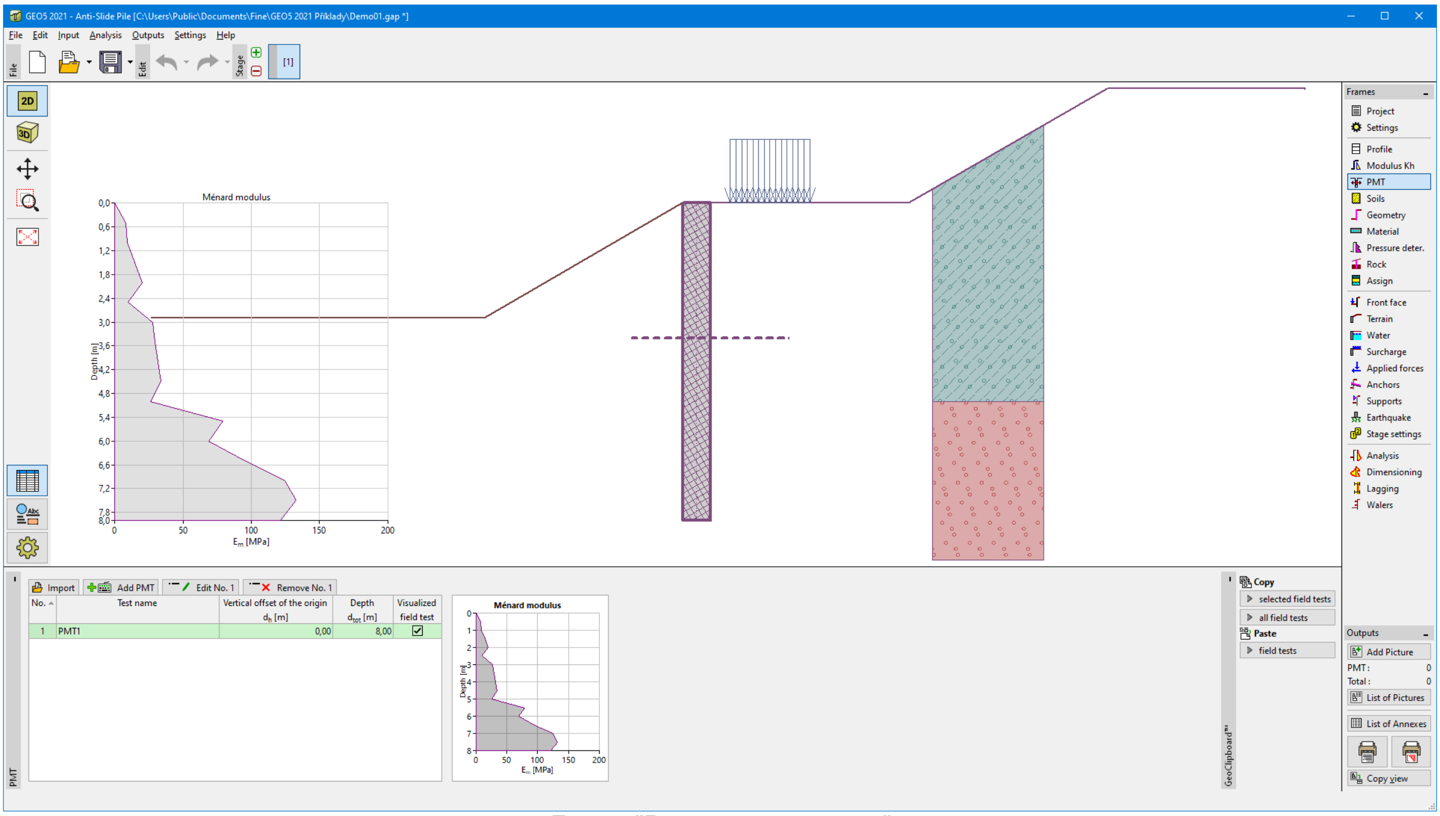
Frame "Modulus k_h" - selection "pressuremeter Ménard"



Frame "Modulus k_h" - selection "Chinese standards"

PMTs

The "**PMTs**" frame contains a table with a list of pressuremeter tests (PMT).



Frame "Pressuremeter tests"

The results of pressuremeter tests (PMT) can be imported into the program by inserting the file in different formats (eg. ***.TXT, *.CSV, *.XLSX, *.ODS**).

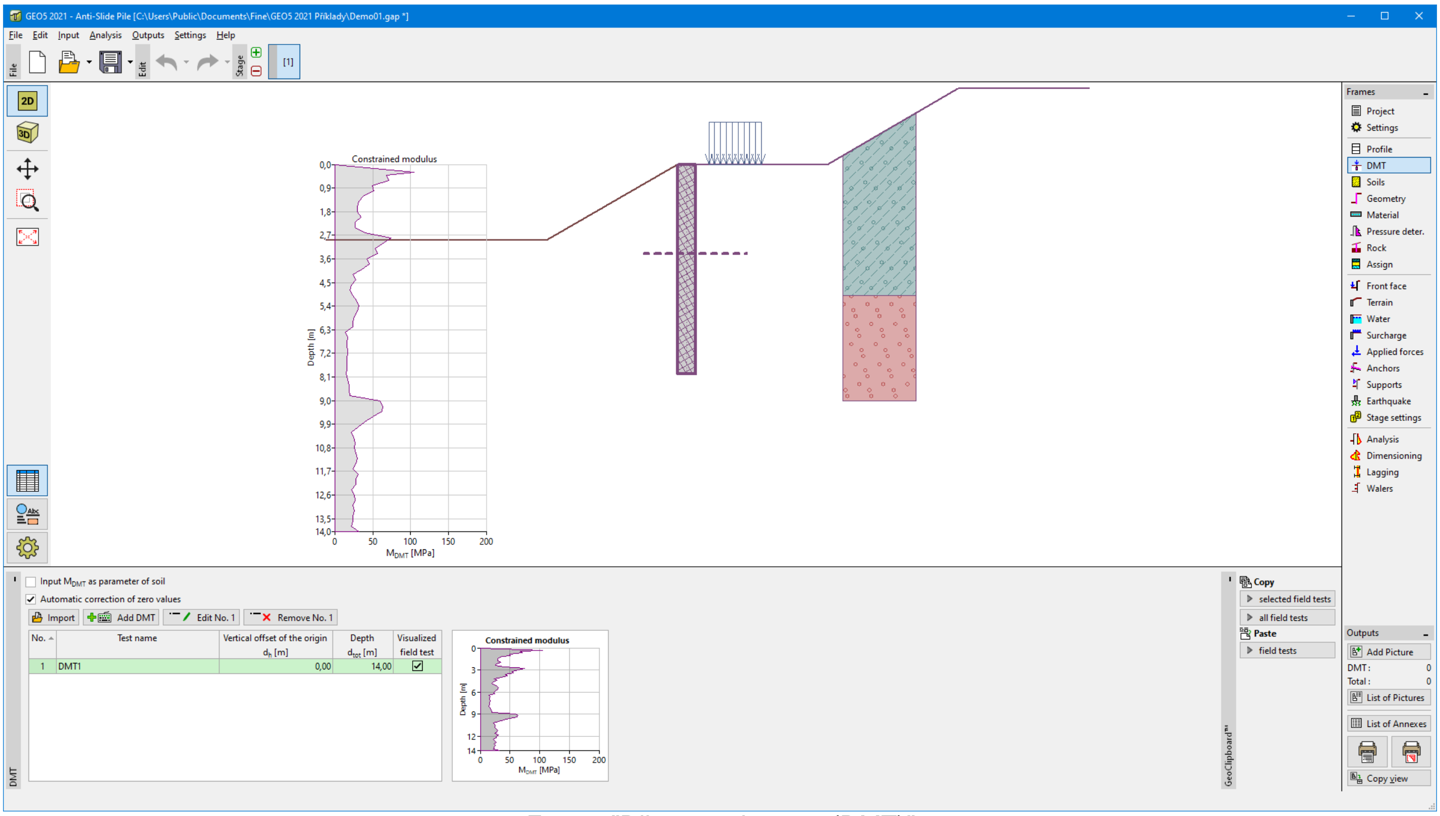
Data of PMTs can be copied within "**Micropile**", "**Sheeting Check**", "**Anti-Slide Pile**", "**Spread Footing CPT**", and "**Stratigraphy**" programs using "GeoClipboard".

Note: The frame is accessible only in case when an option "pressuremeter PMT" is selected for the determination of subsoil reaction modulus in the "Settings" frame (the "Pressure Analysis" tab).

DMT

The "**DMT**" frame serves to input the way of introducing of constrained soil modulus into the program - either as a parameter of soil (by checking the option "**Input M_{DMT} as a soil parameter**"), or by importing of a dilatometric tests (DMT).

This frame contains a table with a list of the input values of dilatometric tests (DMT).



Frame "Dilatometric tests (DMT)"

If the zero value of constrained soil modulus M_{DMT} is measured, then the program makes an automatic correction. The arithmetic average of the next upper and lower non-zero value of M_{DMT} is considered instead of zero value in the calculation.

It is also necessary to enter a coefficient of reduction B .

The results of dilatometric test (DMT) can be imported into the program by inserting the file in format **UNI** (***.uni**).Data of DMTs can be copied within "**Spread Footing**", "**Sheeting Check**", "**Anti-Slide Pile**" and "**Stratigraphy**" programs using "GeoClipboard".

Note: The frame is accessible only in case when an option "dilatometric DMT" is selected for determination of subsoil reaction modulus in the "Settings" frame (the "Pressure Analysis" tab).

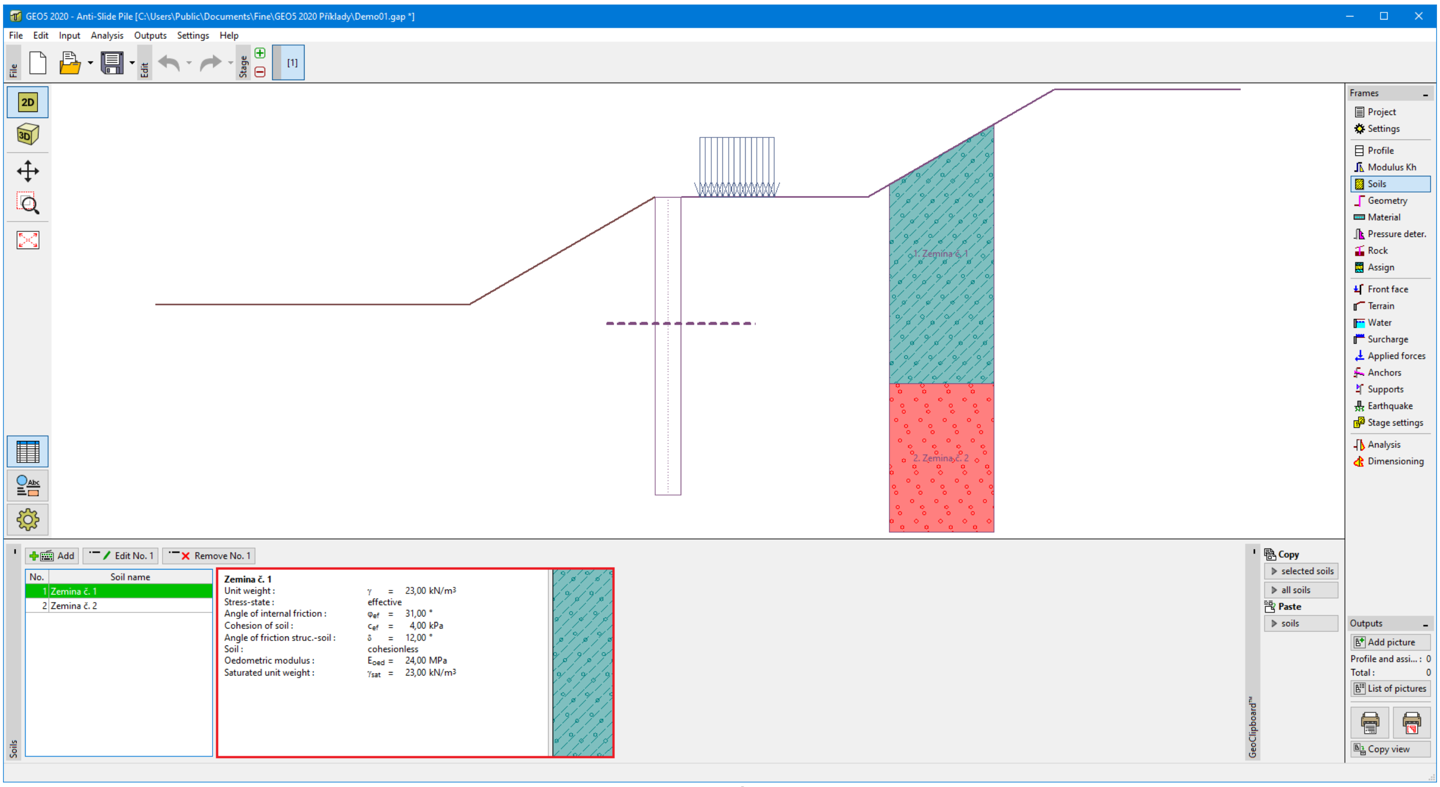
Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Earth pressure at rest", "Uplift pressure" and "Modulus of subsoil reaction".

Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

Basic Data

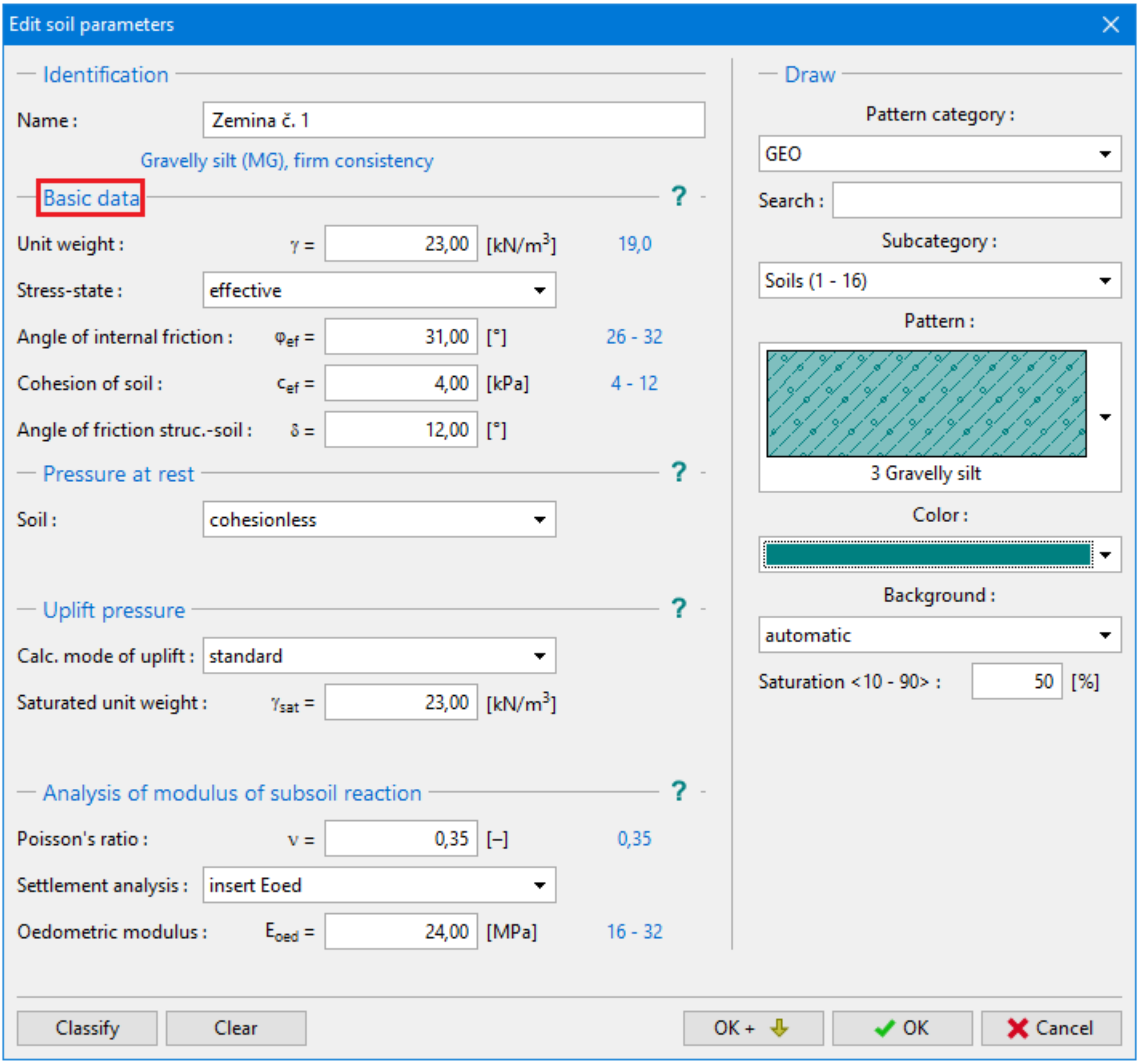
This part of the window allows us to introduce basic parameters of the soils - **unit weight, angle of internal friction, and cohesion**. The particular values are obtained from a geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit the built-in soils database, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the built-in database, these parameters must be defined manually.

Either **effective** or **total** parameters of the angle of internal friction and cohesion are specified depending on the settings in the "**Stress analysis**" combo list. Whether to use effective or total parameters depends primarily on the type of soil and load, structure duration, and water conditions.

For effective stress, it is further needed to specify the angle of internal friction between the soil and the structure, which depends on the structure material and type of soil. Possible values of this parameter are listed in the table of recommended values.

For total stress, it is further needed to specify the adhesion of soil to the structure face a .

The associated theory is described in detail in the chapter "Earth pressures".



Dialog window "Add new soils" - "Basic data"

Geometry

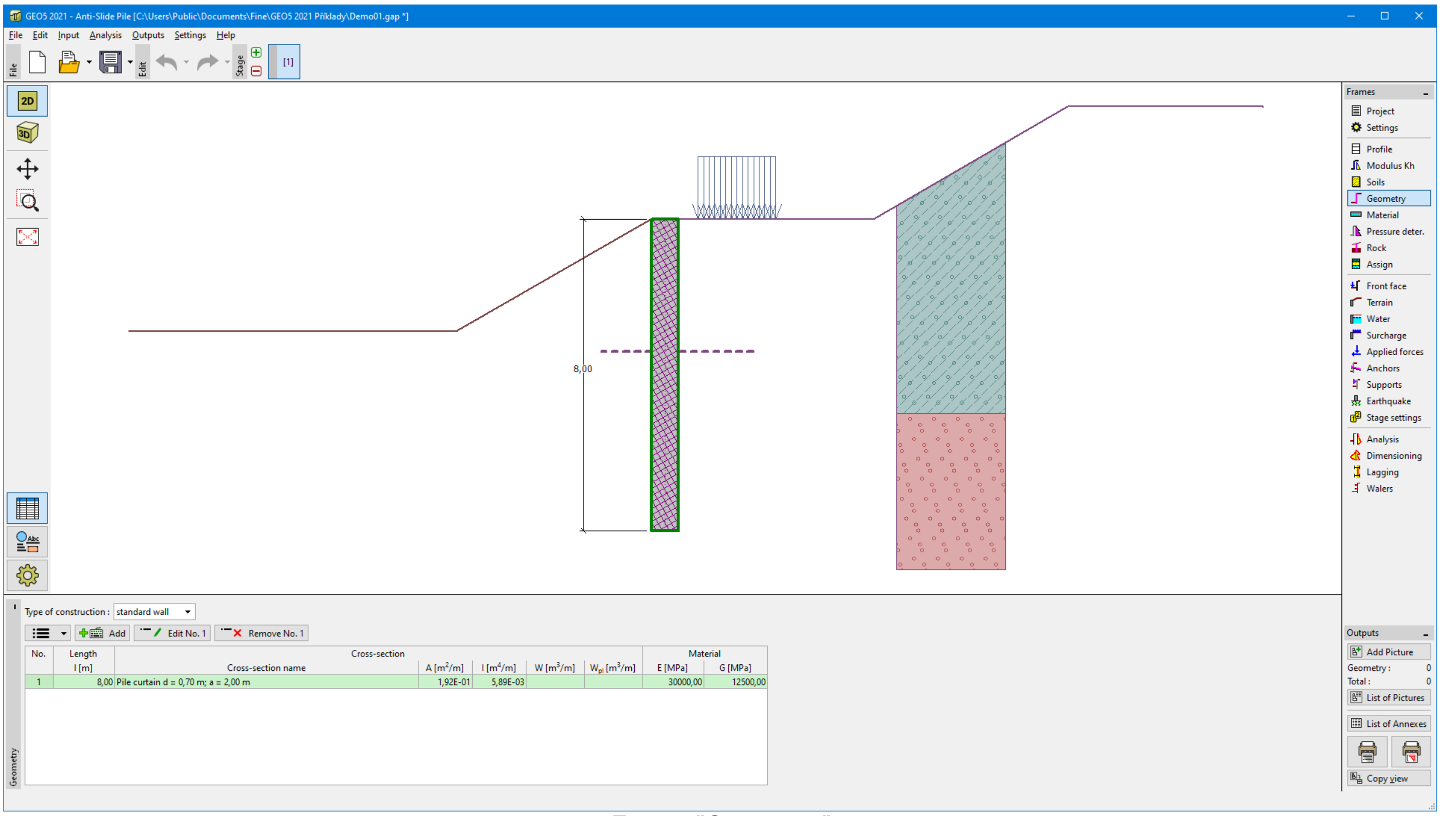
The "**Geometry**" frame allows us to select the **type of structure**:

- **standard wall**
- **double-row pile wall** - depth of connecting beam h_1 and final excavation depth h are input

The frame also contains a table with a list of input structural sections forming the anti-slide pile. For each section the table stores its cross-sectional characteristics (A - area, I - Moment of inertia - these variables are always expressed with respect to 1 m run of structure length) and material characteristics (E - Modulus of elasticity, G - Shear modulus).

Adding sections is performed in the "**New section**" dialog window.

The input sections can be further edited on the desktop with the help of active objects - double-click on a structure opens a dialog window with a given section.



Frame "Geometry"

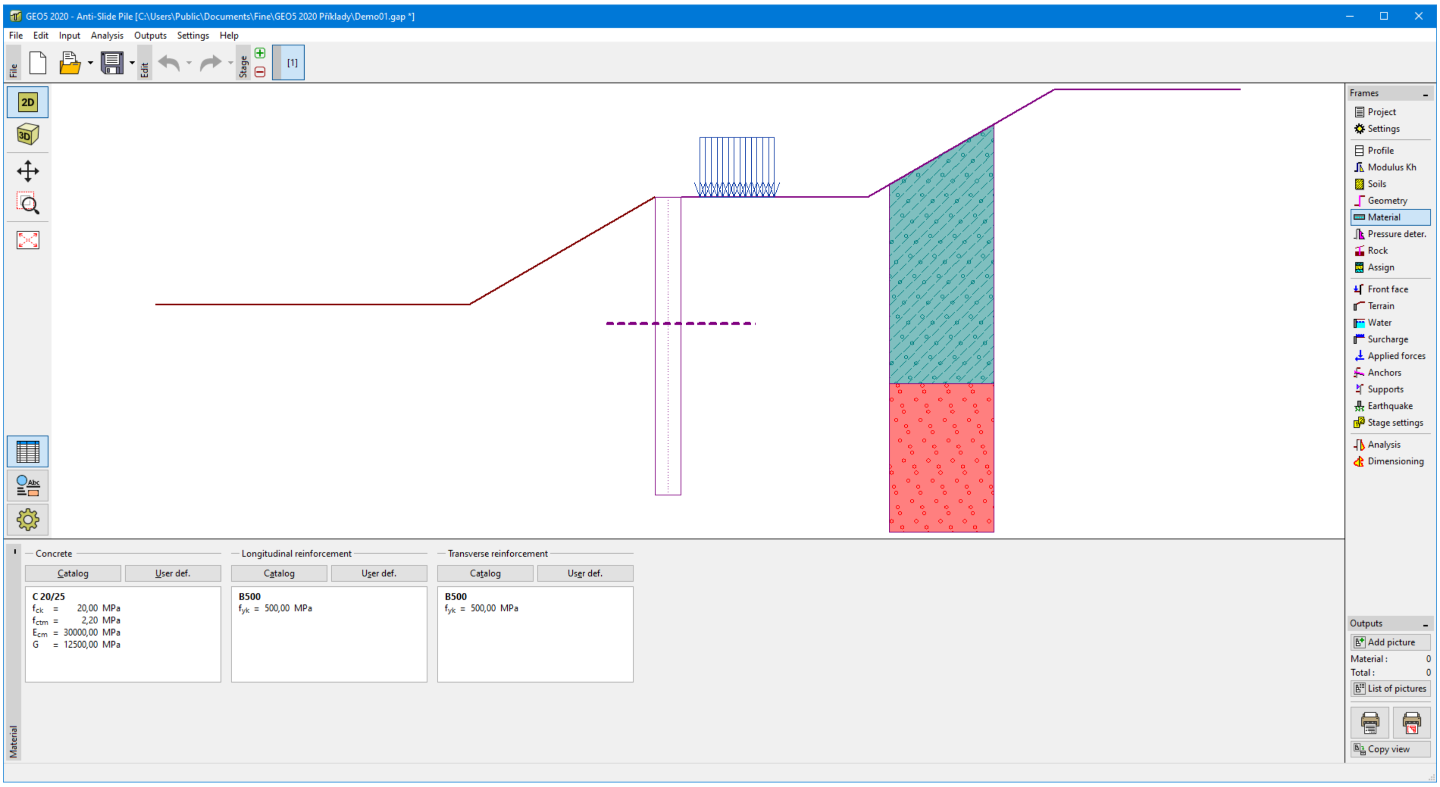
Material

The frame **"Material"** allows us to enter material parameters. The appearance of the frame varies according to the selected material (**concrete, steel, timber**) in the frame "Geometry".

Two options can be used for setting the parameters of the material:

- The "**Catalog**" button opens the "**Catalog of materials**" dialog window, where the required material can be selected.
- The "**User-defined**" button opens the "**Editor of material**", which allows us to input the specification of material parameters manually by the user.

The content of catalogs depends on the selection of relevant standard for the dimensioning of concrete, steel, timber or plastic structures set in the "Materials and standards" tab.



Frame "Material"

Pressure Determination

The frame **"Pressure determination"** allows inputting the **depth of the slip surface** and **load on the anti-slide pile** above the slip surface.

Active horizontal force acts **behind the pile,** and it is defined as follows:

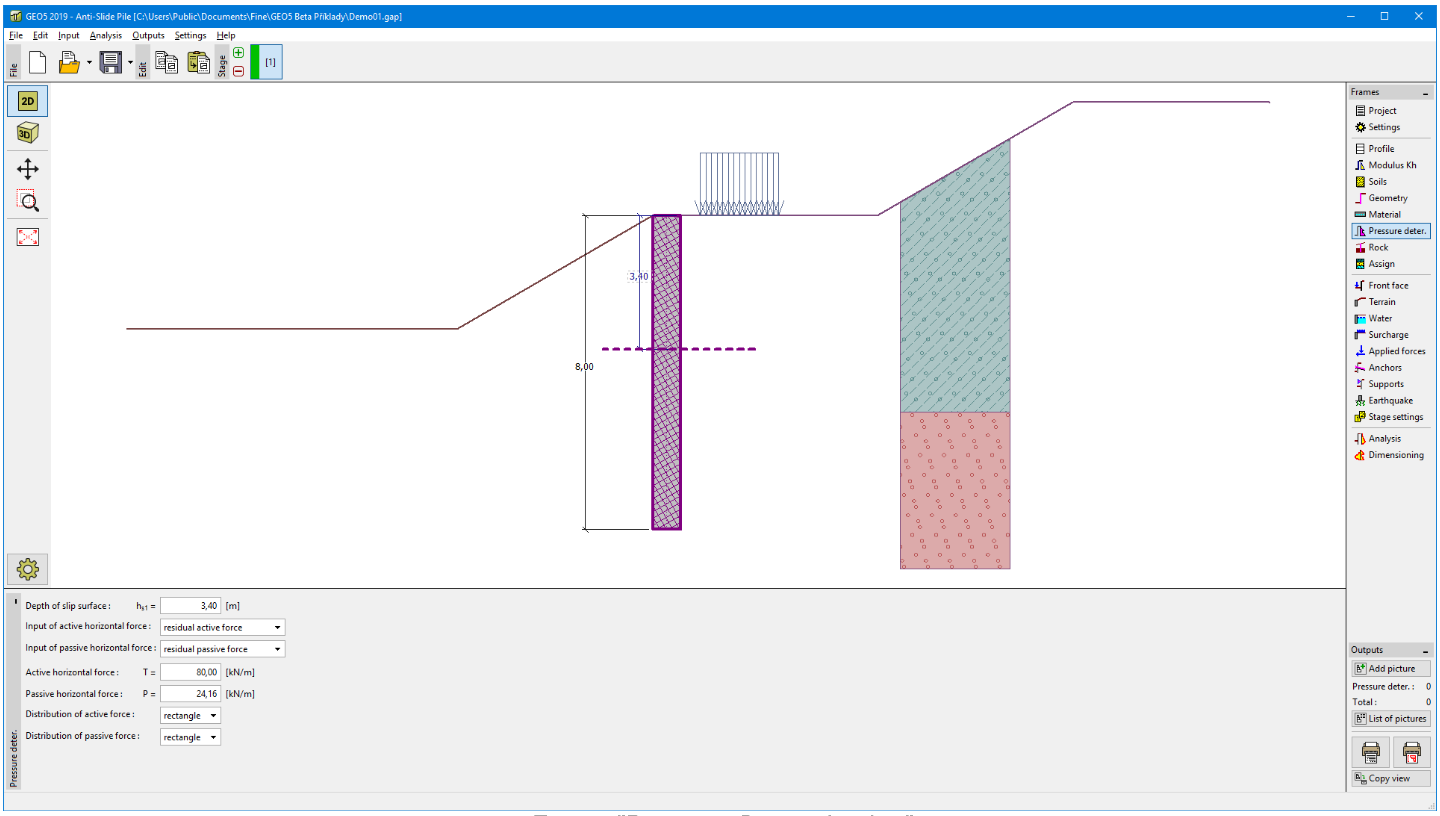
- **residual active force** - the magnitude can be changed in the subsuquent stages
- active pressure
- **input pressure distribution** - the **overall** pressure on the pile above the slip surface is defined in the table, passive horizontal force is not defined anymore

Passive horizontal force acts **in front of the pile,** and it is defined as follows:

- **residual passive force** - the magnitude can be changed in the subsuquent stages or automatically calculated according to the excavation in front of the pile
- springs

Active and passive residual forces can be calculated in the "**Slope stability**" program and transferred into the program "**Anti-Slide Pile**". Next, the shape of the pressure distribution must be specified. Program does **NOT INCREASE** the input passive and active force by any partial factor during the analysis - it is necessary to determine them according to required standards and rules.

If the forces are obtained from the **Slope stability** program, they correspond with the way of analysis set in the program. For example, when an analysis is done according to Eurocode **EN 1997-1**, the received values are already design values of forces.



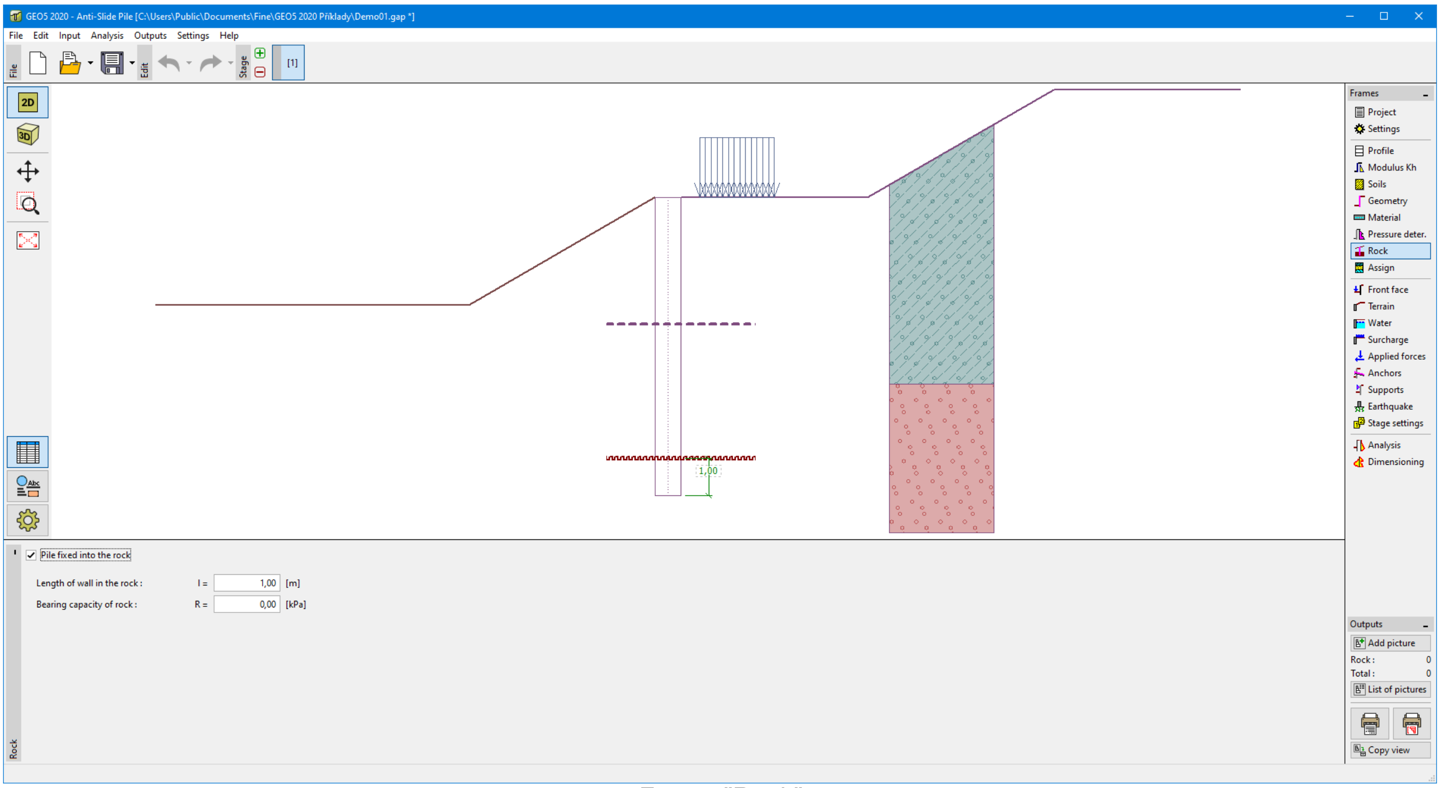
Frame "Pressure Determination"

Rock

The frame **"Rock"** allows us to input the length of the pile fixed into rock l $[m]$ and rock bearing capacity R $[MPa]$.

The rock is only modeled as elastic material, and no passive limit pressure is considered - contact stress can reach any values. In the frame "Analysis" the program checks if **maximum stress** does not exceed **design rock bearing capacity**. The active pressure of the rock is not considered.

Note: In many cases, the slip surface follows the rock subsoil. This case should be always investigated.

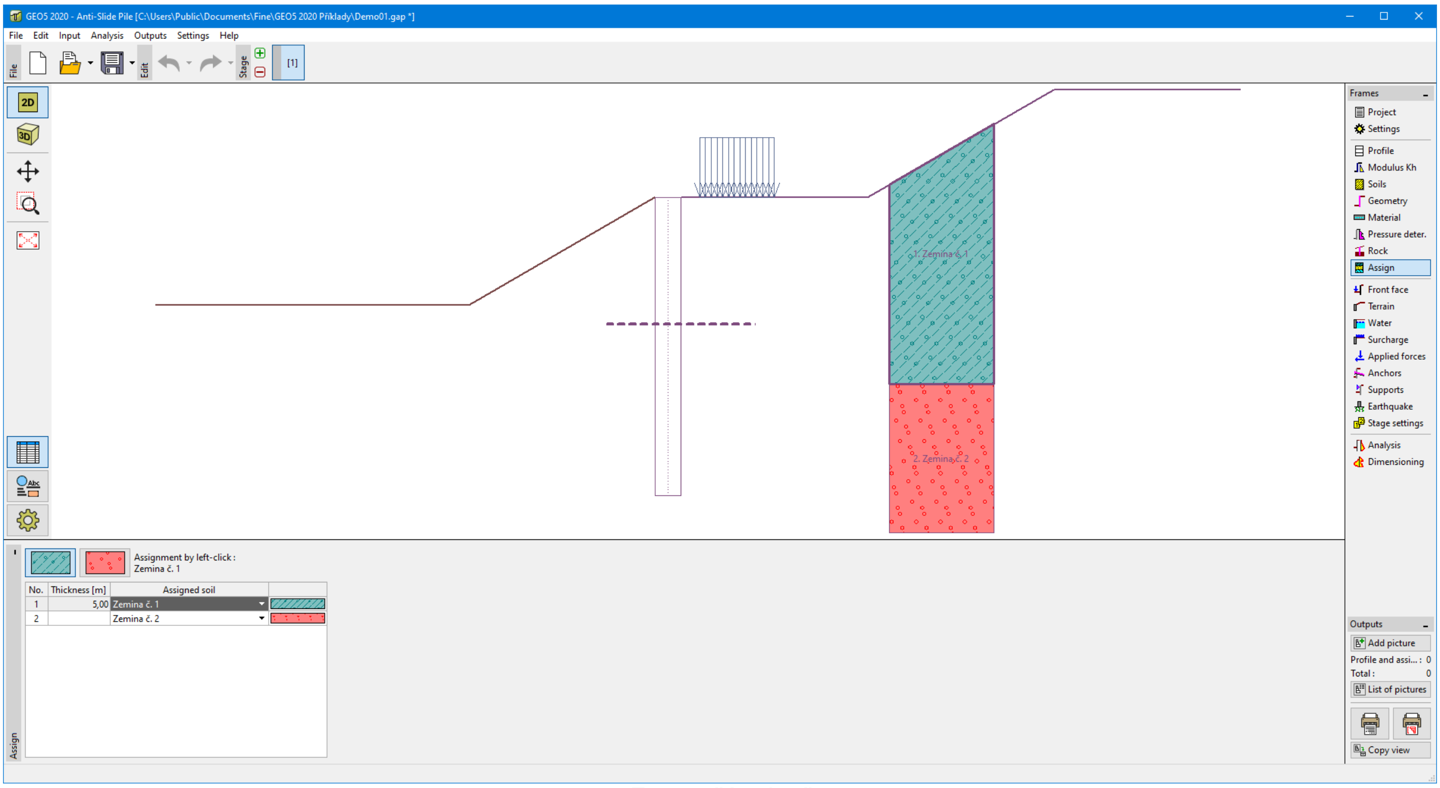


Frame "Rock"

Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail herein.

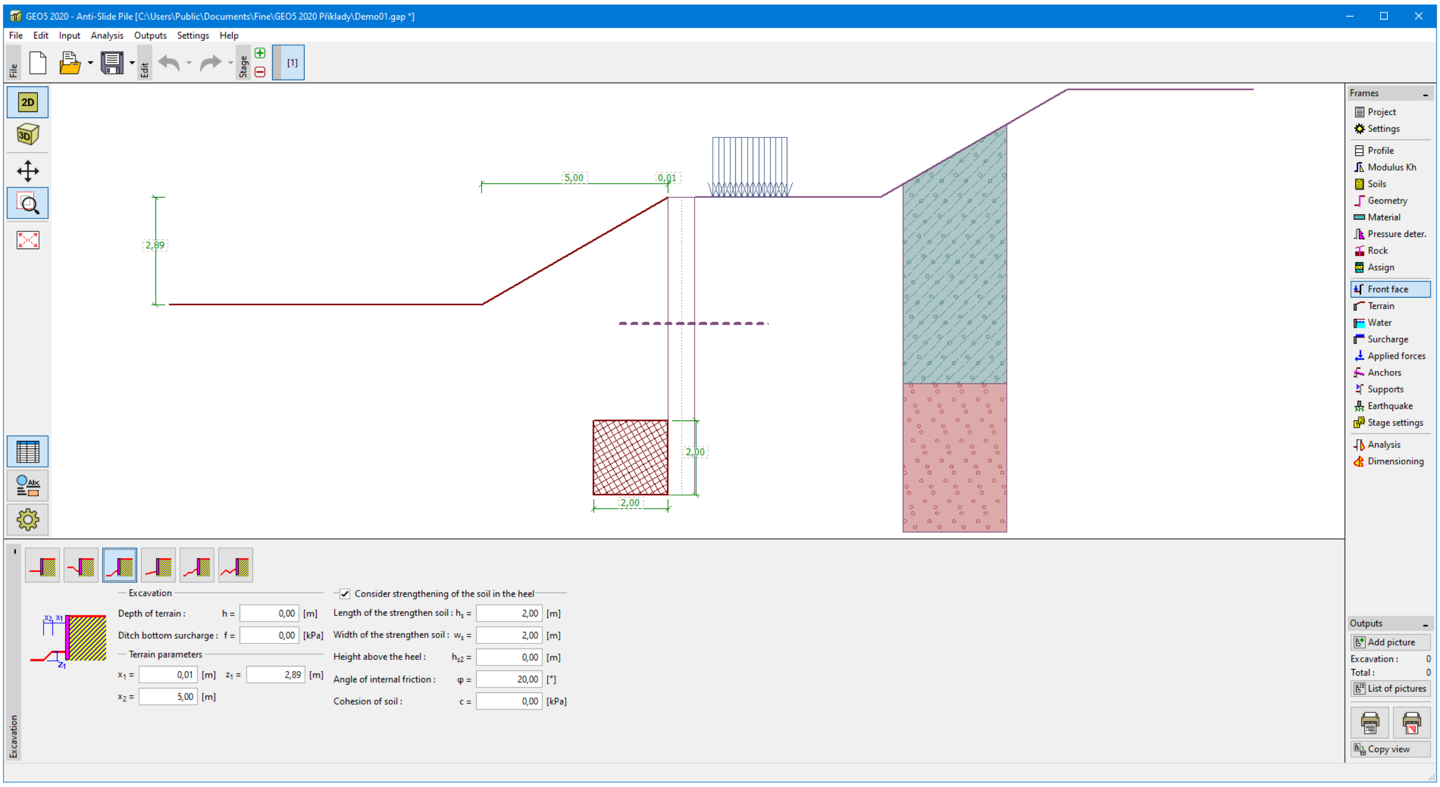


Frame "Assign"

Front Face

The "**Front Face**" frame serves to input the shape of the terrain in front of the structure. The selected shape with a graphical hint appears in the left part of the frame. The dimensions of a structure can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

In this frame, it is possible to input **strenghtening of the soil** at heel of the piles. The principle of calculation is described in more detail herein.



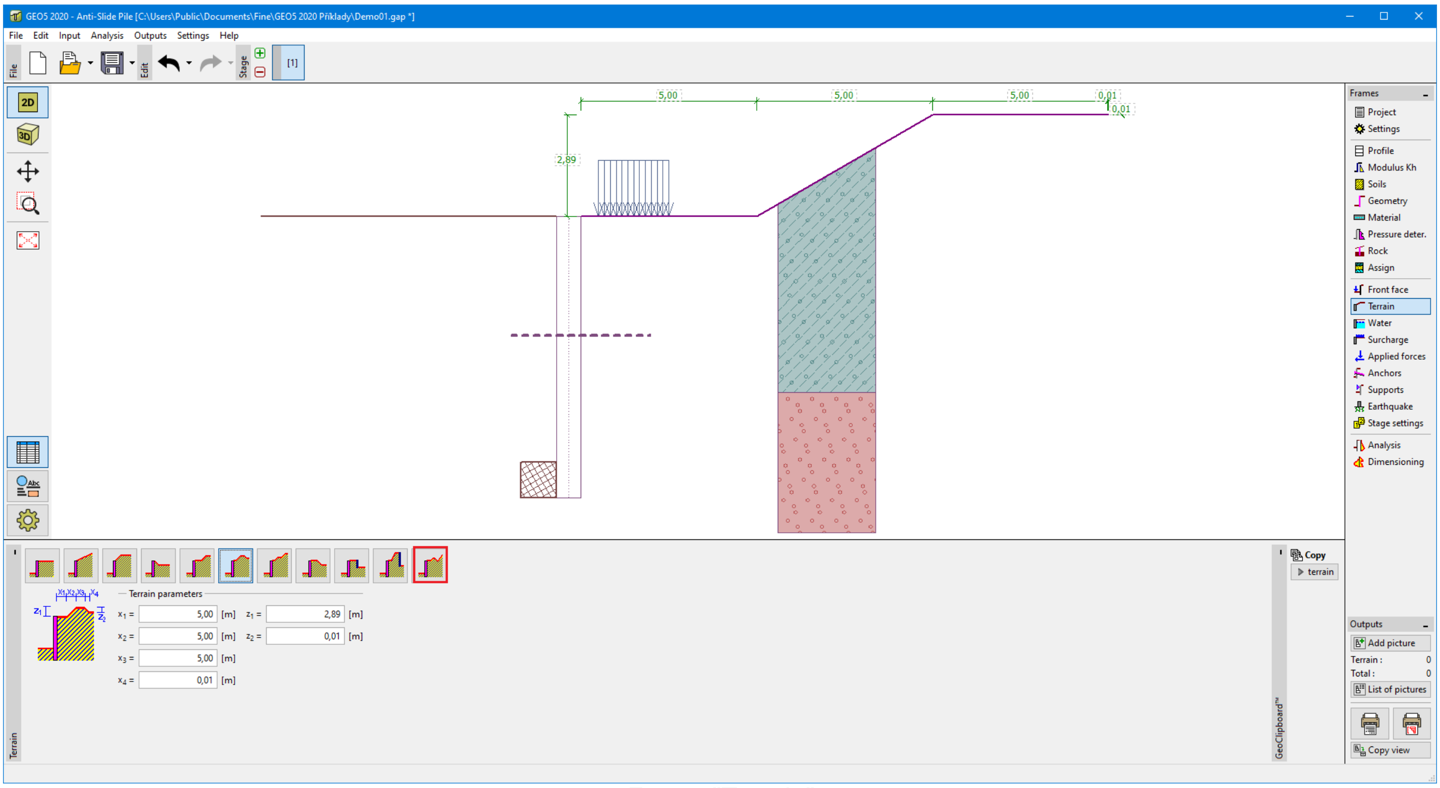
Frame "Front Face"

Terrain

The "**Terrain**" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case, the frame contains a table with a list of terrain points. The first point with coordinates [0,0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "Distribution of earth pressures for broken terrain".

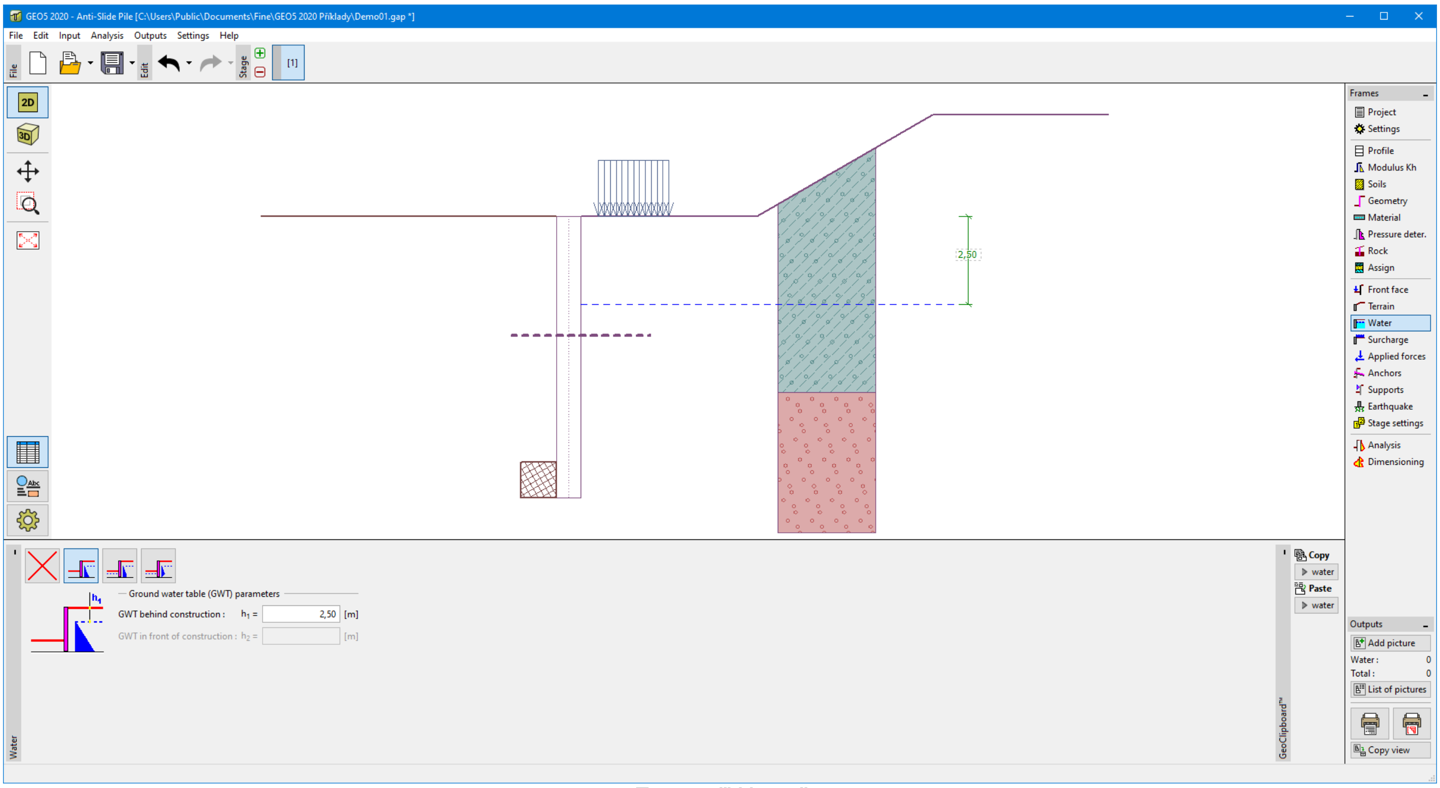


Frame "Terrain"

Water

The "**Water**" frame allows, by pressing the button, for selecting the type of water. The selected type with a graphic hint of input values is displayed in the left part of the frame. Water parameters (h_1, h_2 ...) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

Analysis of earth pressures with the influence of water is described in the theoretical part of the help, chapter "Influence of water".



Frame "Water"

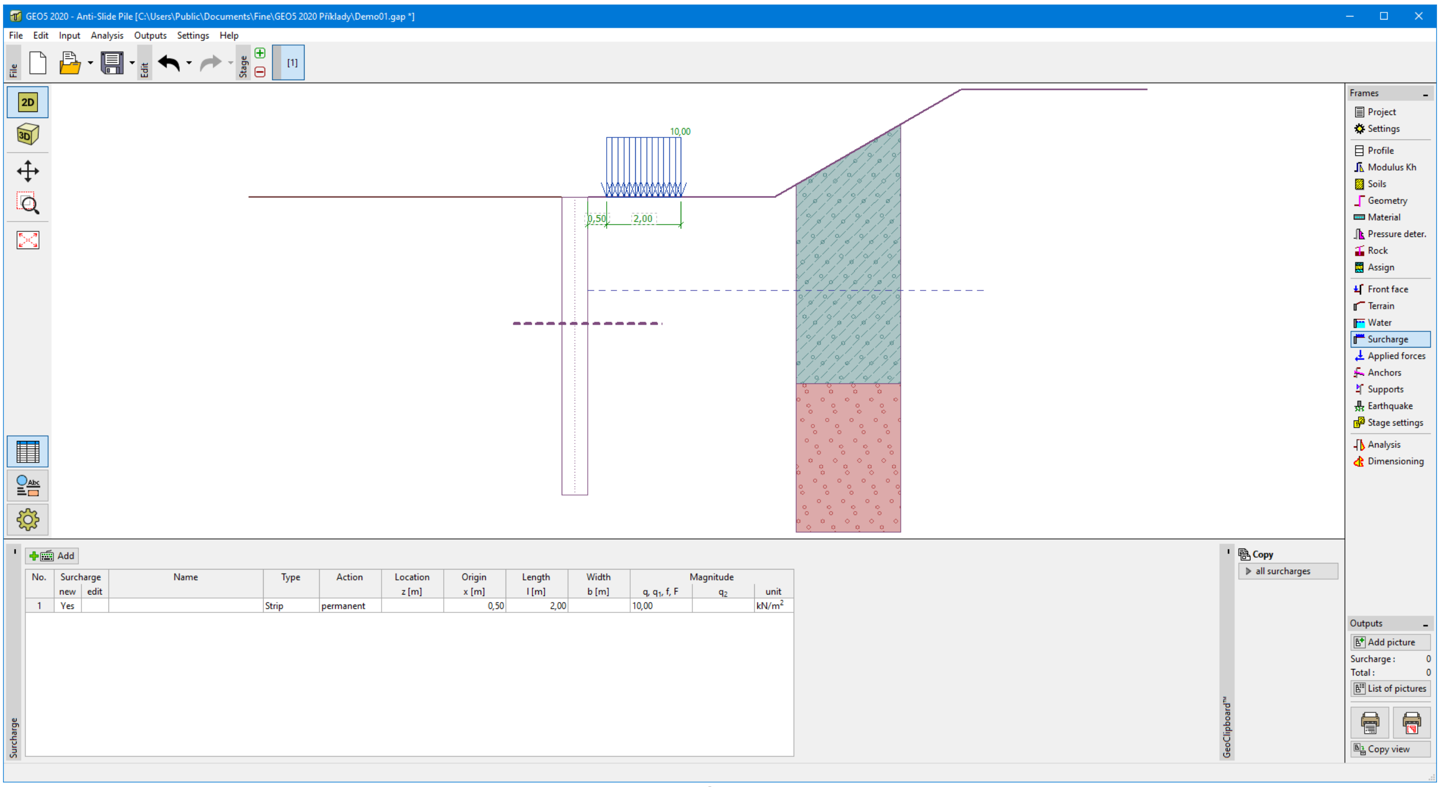
Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding (editing) surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The z -coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth.

Either **permanent**, **variable** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with a favorable effect is not considered in the analysis.

Analysis of earth pressures due to surcharges is described in the theoretical part of the help, chapter "Influence of surcharge".



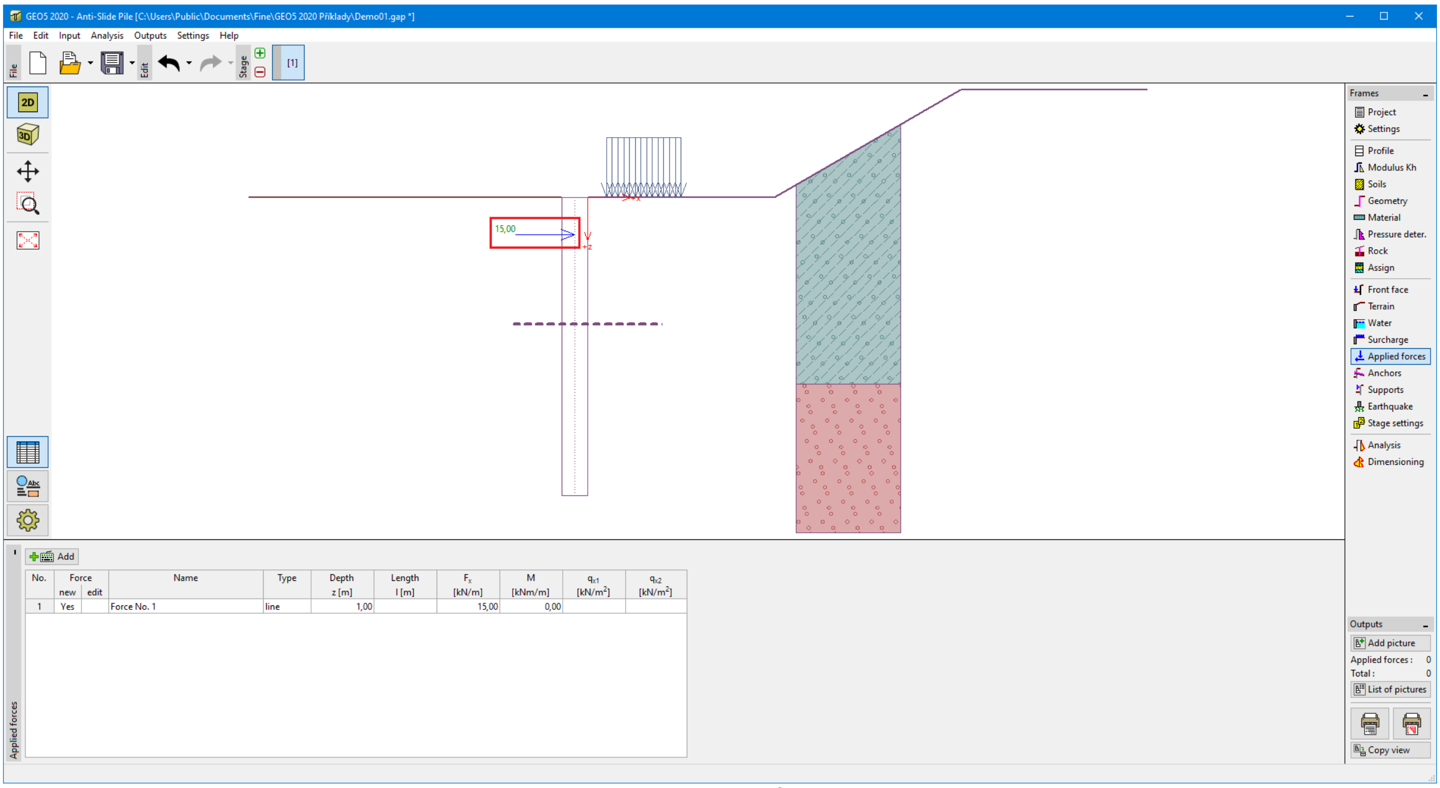
Frame "Surcharge"

Applied Forces

The "**Applied forces**" frame contains a table with a list of forces acting on a structure. Adding (editing) forces is performed in the "**New force**" dialog window. The input forces can also be edited on the desktop with the help of active objects.

Applied forces represent an additional load on the structure of the wall, braced sheeting or MSE wall. We can model such as an anchoring crash barrier, crash vehicle, load from billboards and hoardings etc. The program does not modify the applied forces in the calculation except multiplying them with the corresponding coefficients according to the selected type of load (EN1997, LRFD).

External load acting on the terrain is necessary to define as a surcharge.



Frame "Applied forces"

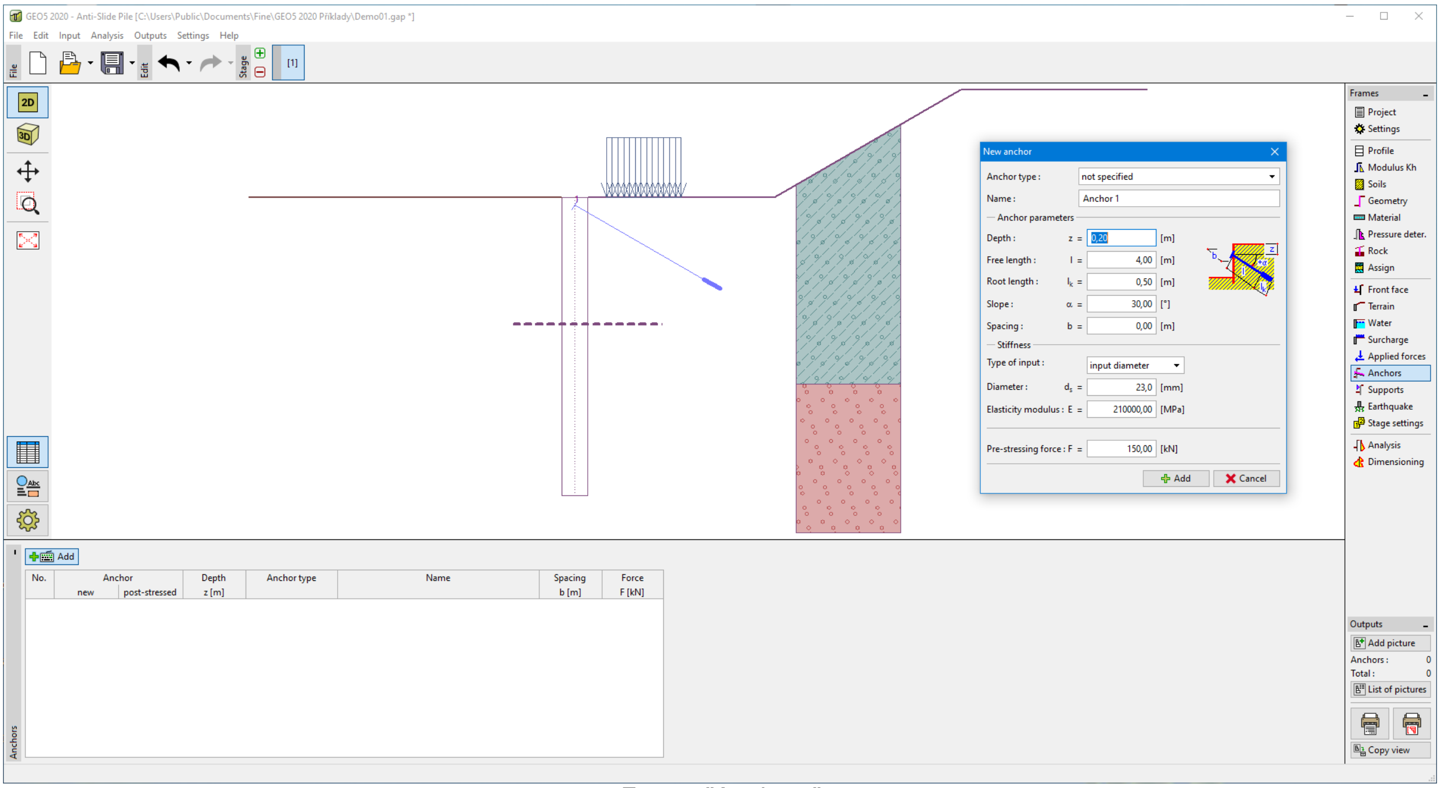
Anchors

The "**Anchors**" frame contains a table with a list of input anchors. Adding (editing) anchors is performed in the "**New anchor**" dialog window. The input anchors can be edited on the desktop with the help of active objects.

The anchor is automatically placed on already **deformed structure** (displacement is obtained from the previous construction stage).

Anchors can be entered as pre-stressed (undefined, prestressed bars, strand) and non-prestressed (helical, non-prestressed bars, deadman). The **pre-stressed anchor stiffness** becomes effective in next stages of construction. Due to the displacement of the structure the forces in anchors are changed according to its deformation.

In **subsequent stages** the anchor cannot be changed, it's only possible to **post-stress** the anchor.



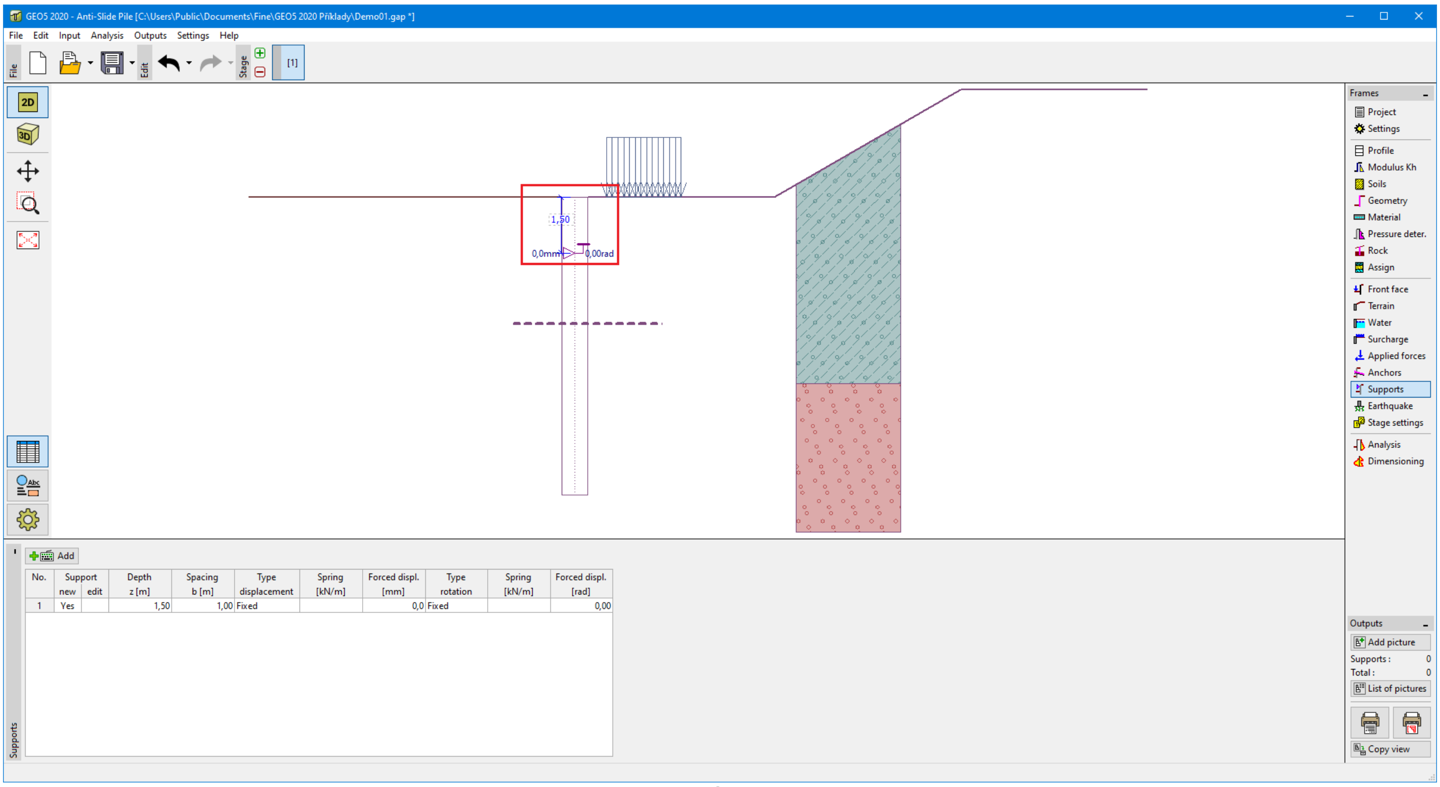
Frame "Anchors"

Supports

The "**Supports**" frame contains a table with a list of input supports. Adding (editing) supports is performed in the "**New support (Edit support)**" dialog window. The input supports can also be edited on the desktop with the help of active dimensions or by active objects.

The support is automatically placed on already **deformed structure** (displacement is obtained from the previous construction stage).

In subsequent stages the supports can no longer be edited, it is only possible to input forced displacement of supports.



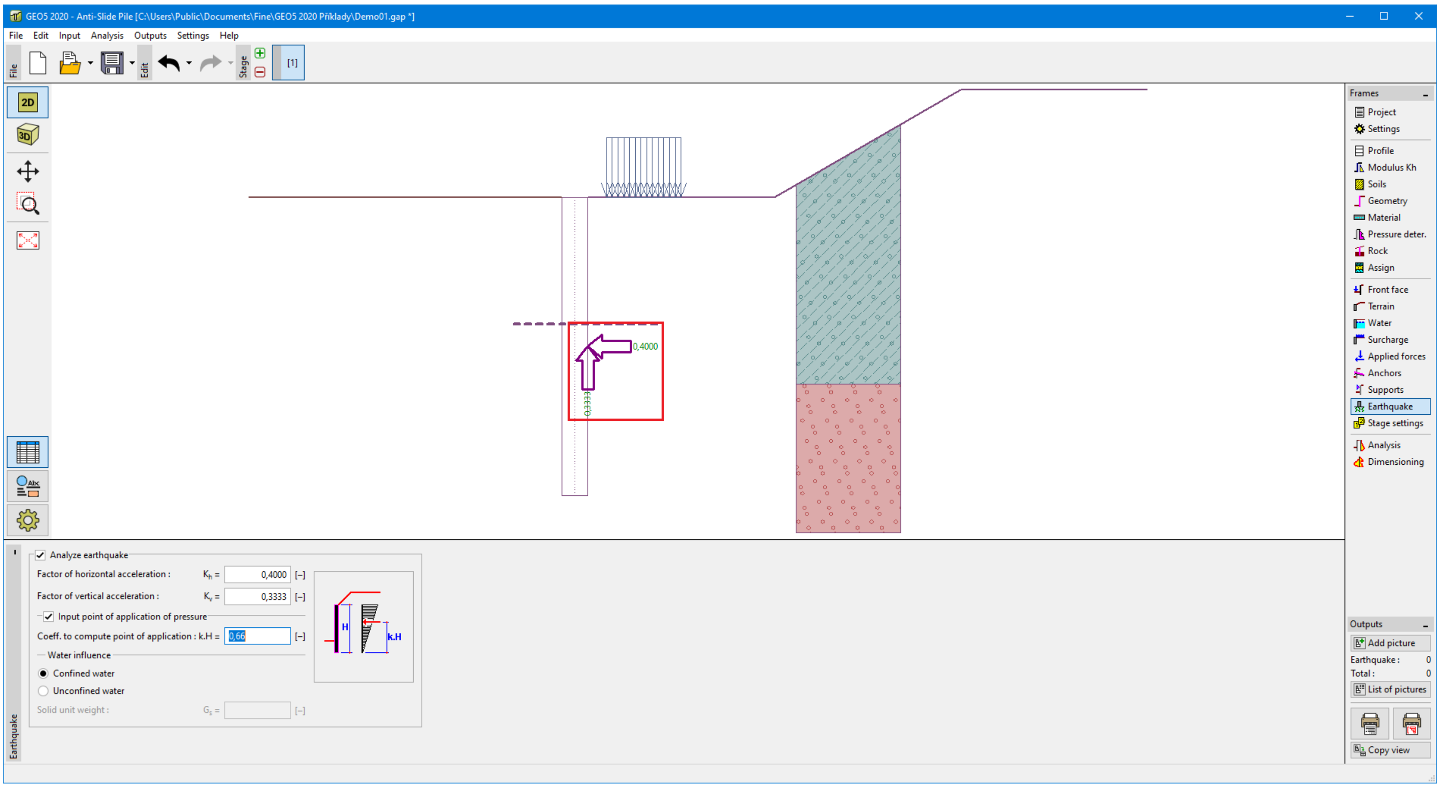
Frame "Supports"

Earthquake

The "**Earthquake**" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements, the coefficients k_h and k_v can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for an earthquake is described in the theoretical part of the help in the chapter "Influence earthquake".



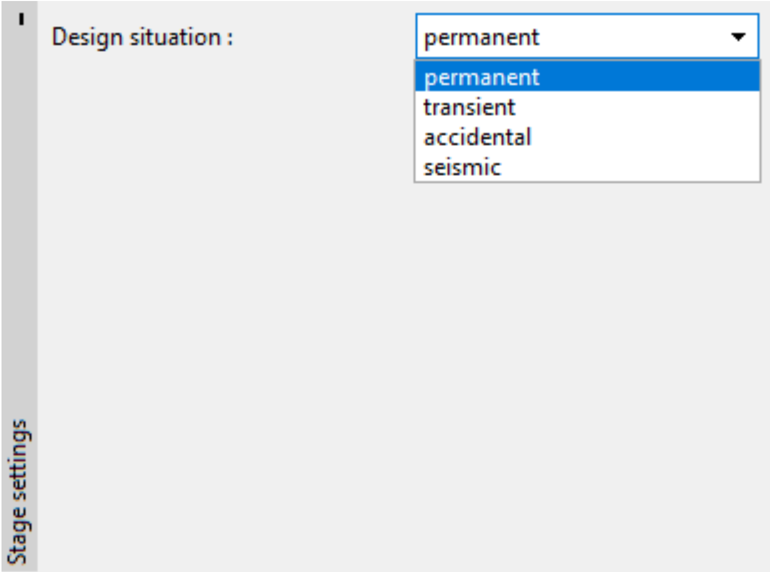
Frame "Earthquake"

Stage Settings

The frame "**Stage settings**" serves to input settings valid for a given construction stage.

The selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology.



Frame "Stage settings"

Analysis

The frame "**Analysis**" displays the analysis results. The frame contains five buttons to show them:

K_h + pressures

Variation of the modulus of subsoil reaction is displayed in the left part of the desktop (by default, a blue color with hatching). In a method of depending pressures, some of the springs (values of modules of subsoil reaction) are removed from the analysis (spring stiffness is equal to zero). If all springs are out, the analysis **may fail to converge.** The program exists without finding a solution. An error message appears in the bottom part of the frame. It is necessary to change **the construction** - e.g., add an anchor, change a depth of excavation, improve soil parameters, etc.

Distributions of limiting pressures (by default, a green dashed line is assumed) are presented in the right part of the window (passive pressure, pressure at rest and active pressure). The **actual pressure acting on a structure** is plotted in a solid blue line.

Both **deformed** (by default, a solid red color is assumed) and undeformed structure appears in the right part of the desktop. Forces and displacements developed in anchors, supports and props are also shown.

Internal forces

Plot of a structure with forces acting in anchors, reactions and deformations of supports, and props appear in the left part of the desktop. Distributions of bending moment and shear force are then plotted on the right.

Displacement + Stress

Plot of a structure with forces acting in anchors, reactions and deformations of supports, and props appear in the left part of the desktop. The deformed shape of a structure with overall pressure acting on a sheeting structure is then plotted on the right.

Terrain settlement

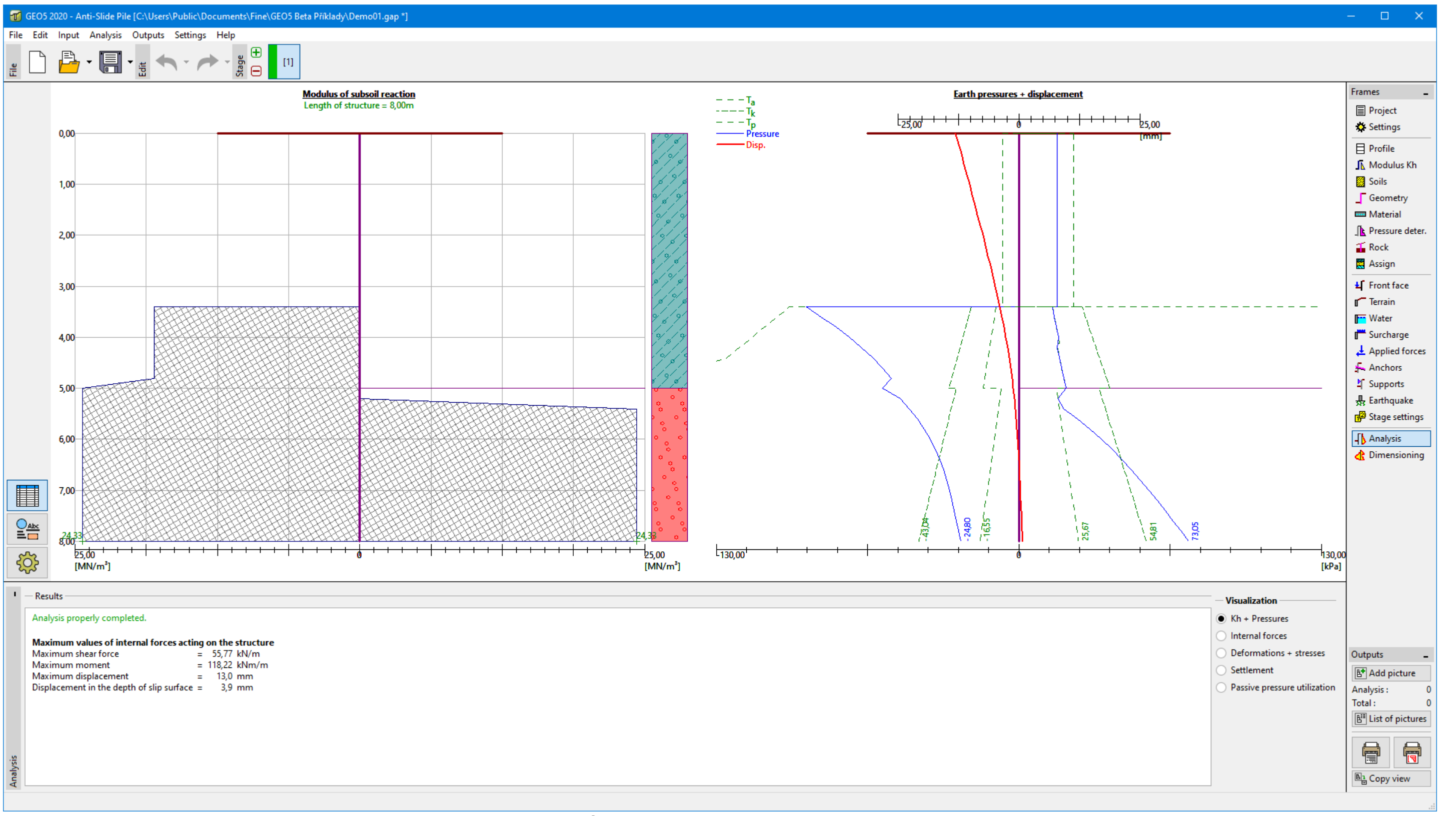
Plot of a structure is displayed in the left part of the desktop. Terrain settlement is plotted on the right. Method for calculation of terrain settlement is selected in the upper part of frame.

Utilization of Passive Pressure

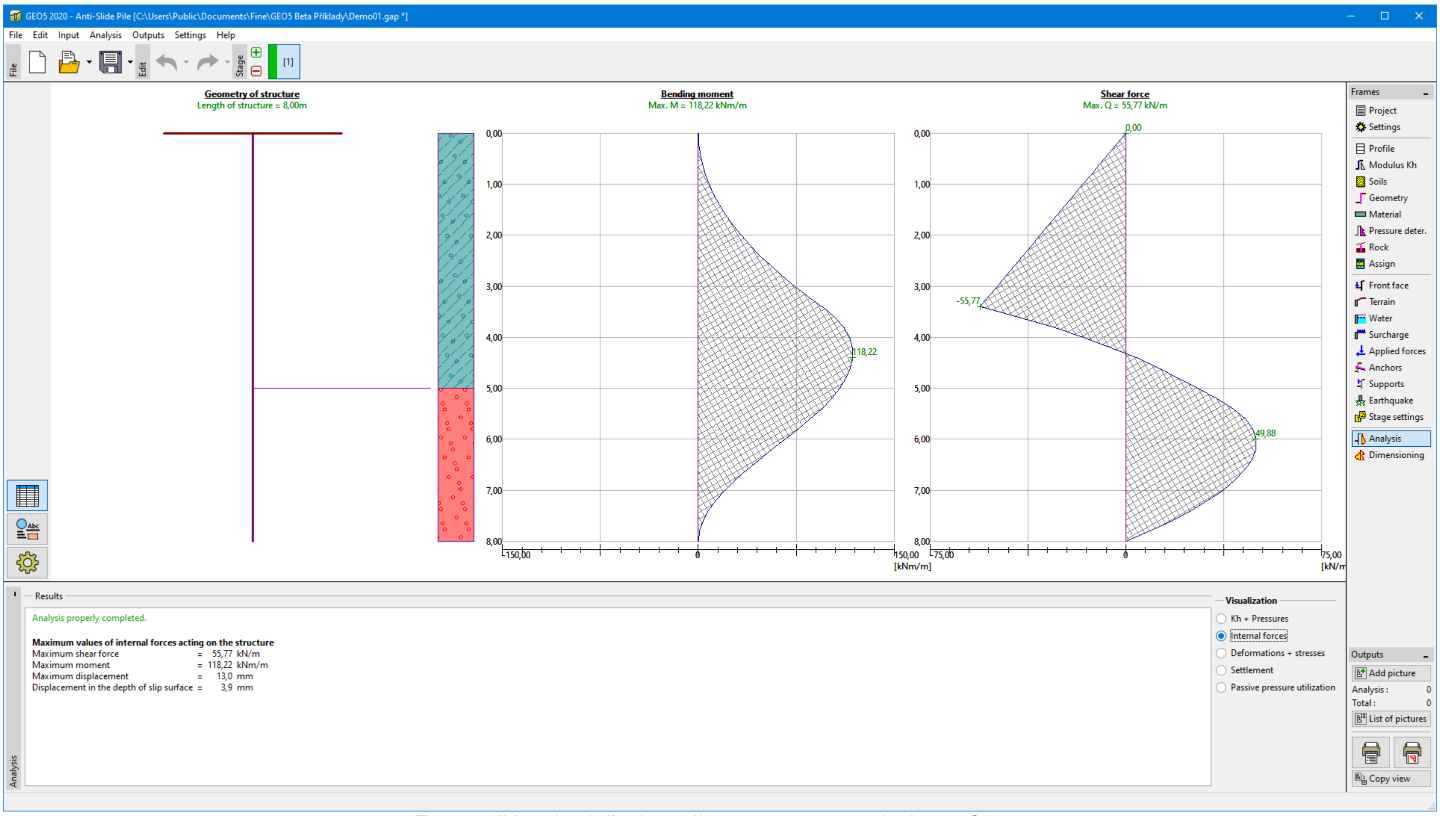
Plot of a structure is displayed in the left part of the desktop. Distributions of limit pressures and actual pressure acting on a structure are presented in the right part of the window. Safety factor for utilization of passive pressure is inputted in the upper part of the frame.

Providing the modulus of subsoil reaction is found by iteration it is necessary to check the **course of manual iteration** in the dialog window "**Iteration**". Details are provided in the theoretical part of the help, chapter "Modulus of subsoil reaction determined by iteration".

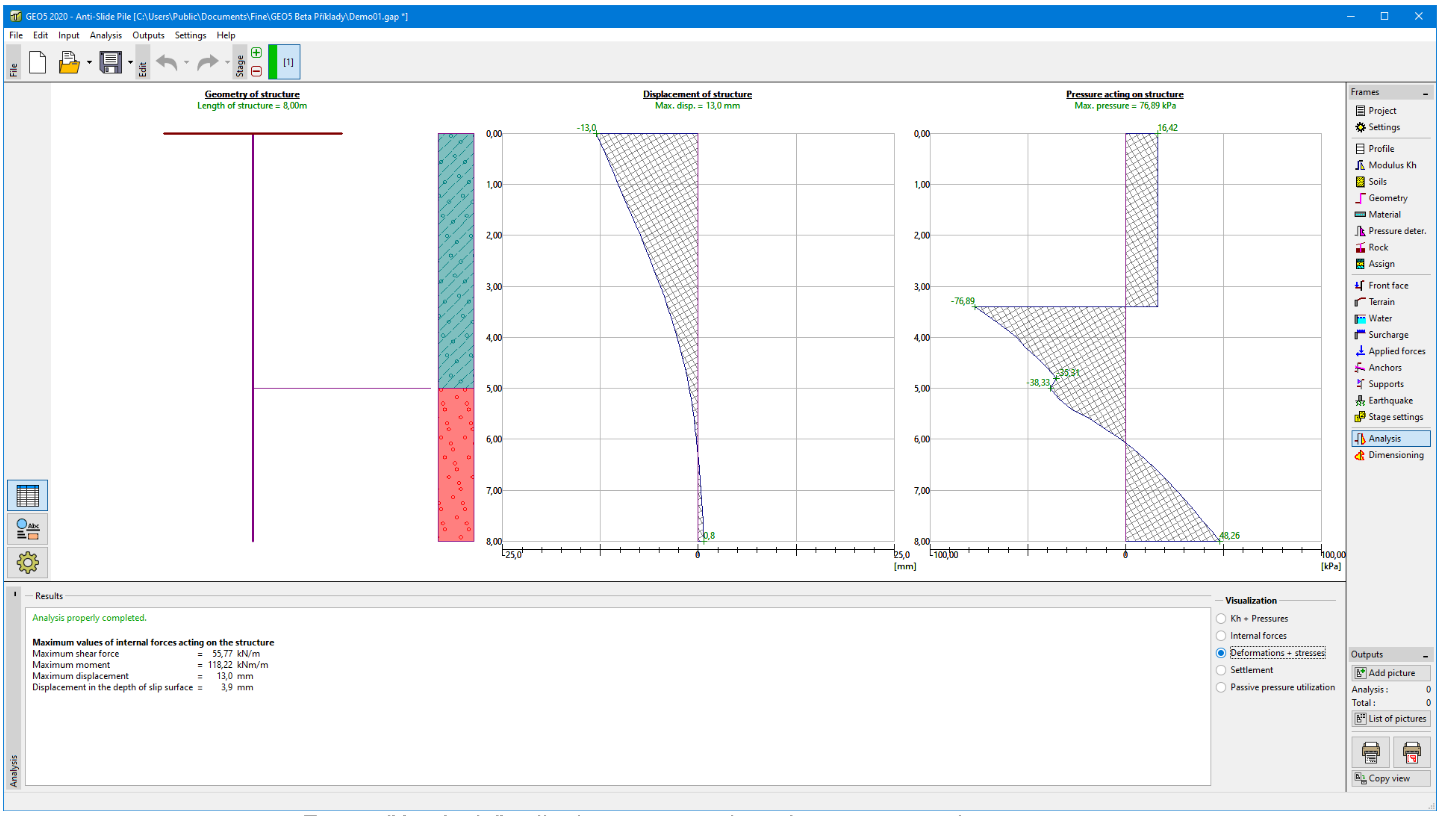
In the case of double-row wall, it is possible to show internal forces, displacement, and stress on the front or back row. Visualization of results can be adjusted in the frame "Drawing Settings".



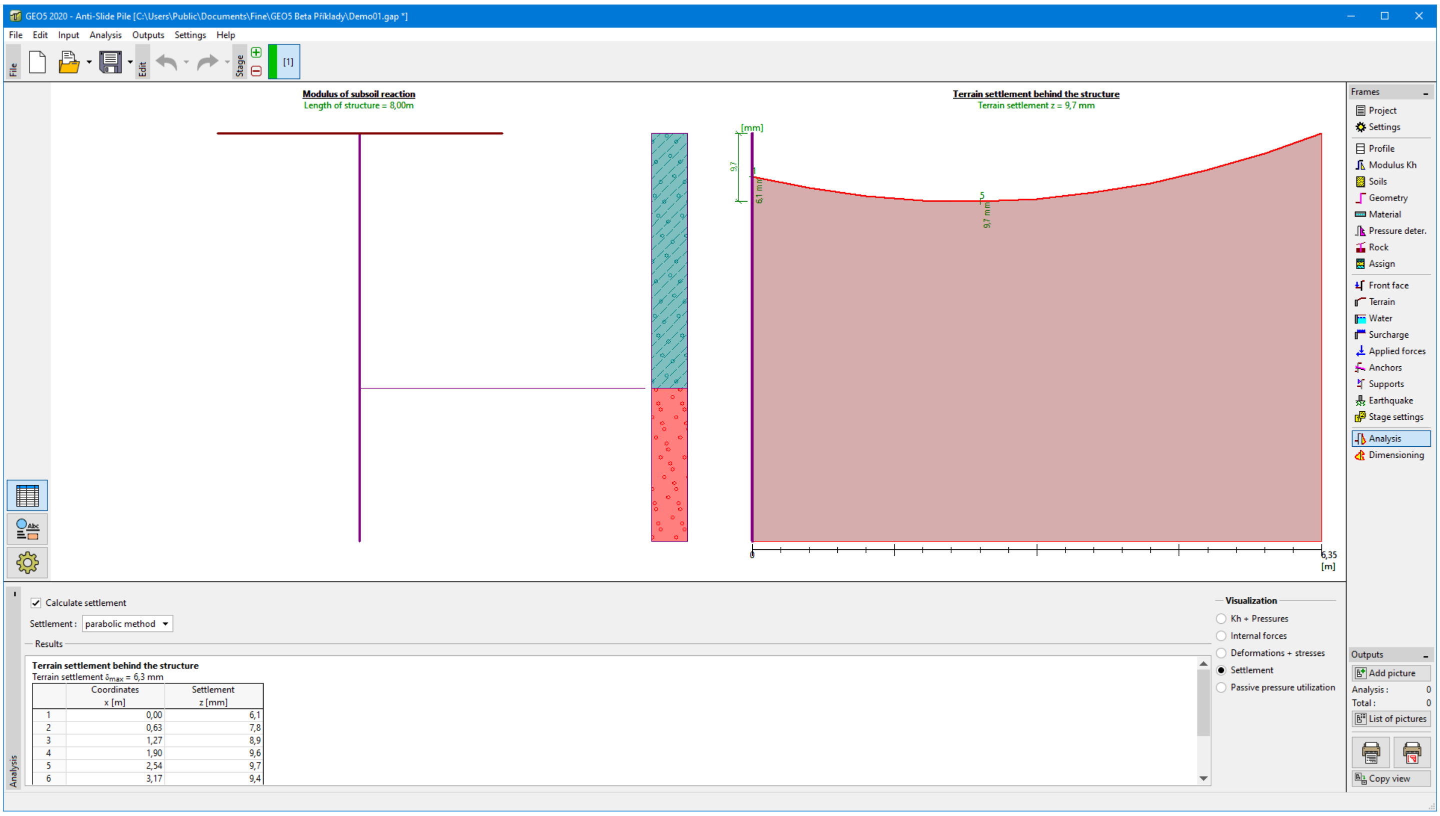
Frame "Analysis" - modulus of subsoil reaction, earth pressures and displacement



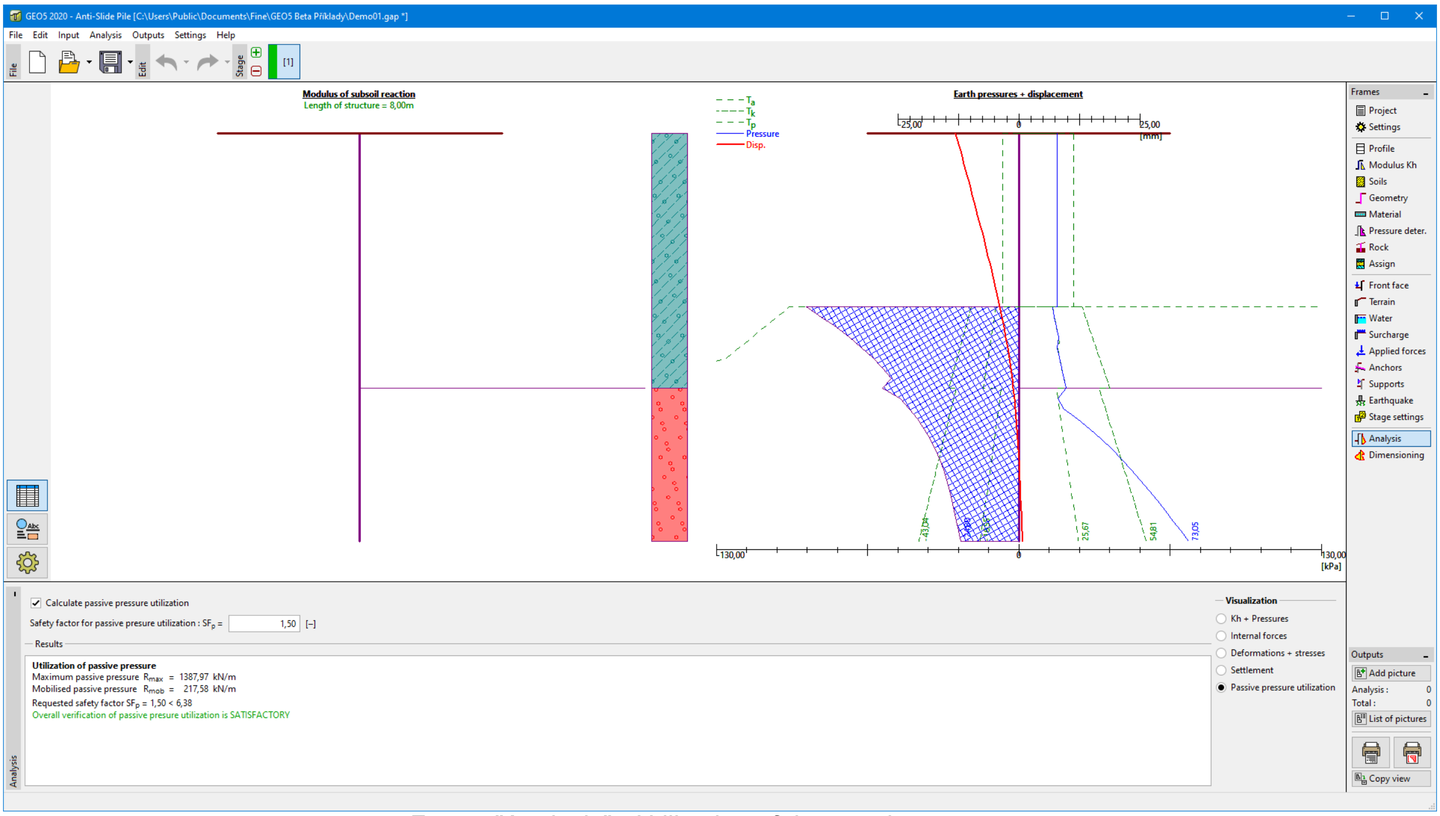
Frame "Analysis" - bending moment and shear force



Frame "Analysis" - displacement and earth pressure acting on structure



Frame "Analysis" - terrain settlement



Frame "Analysis" - Utilization of the passive pressure

Dimensioning

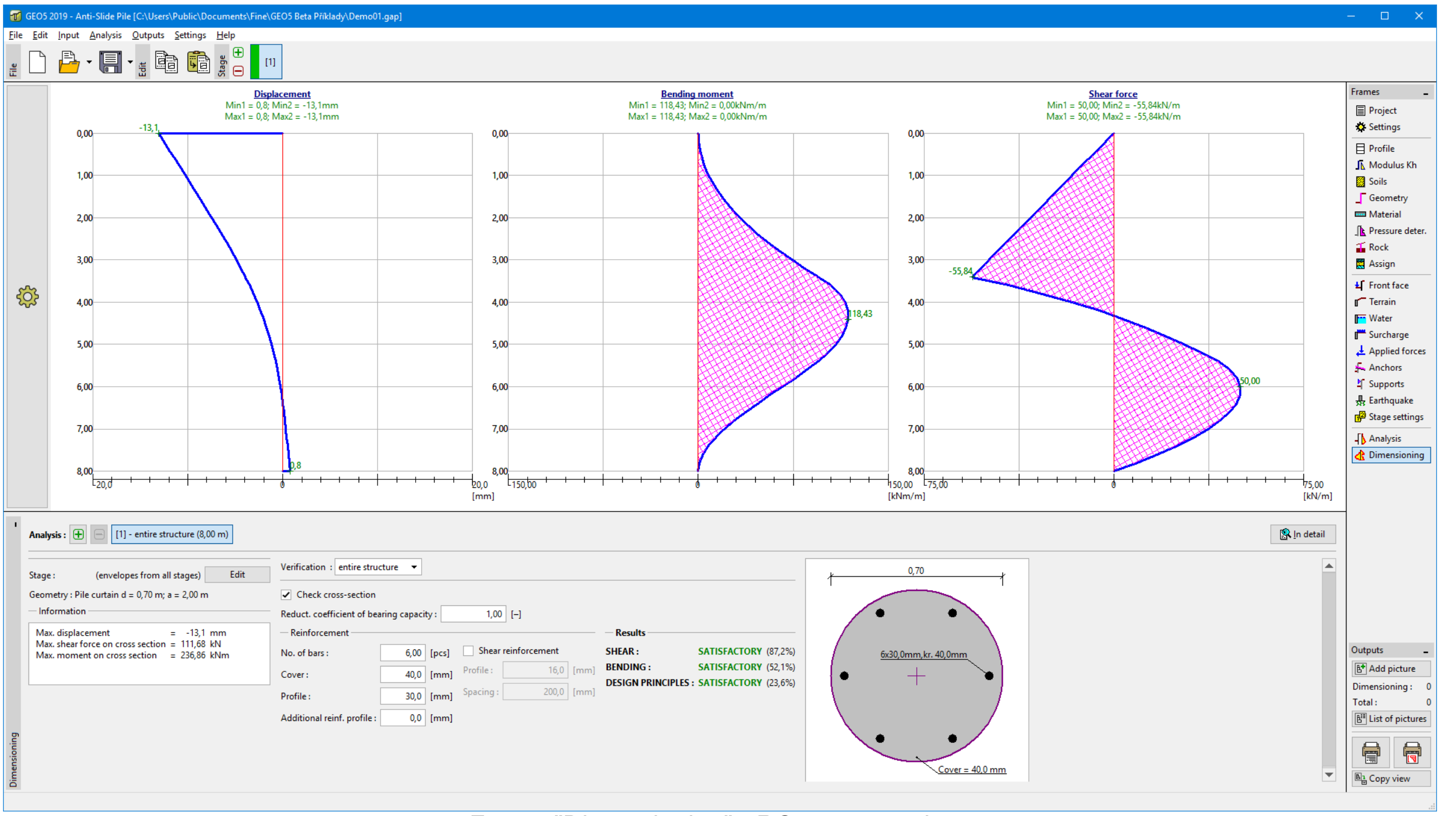
In the frame "**Dimensioning**", it is possible to display an envelope of internal forces and displacements from all analyses (stages of constructions). Normally, the envelope is constructed from the results of all construction stages, however, it can only be created from the **selected stages**. The "**Modify**" button opens the dialog window "**Construction stage selection**", where it is possible to select the constructions stages that are used to generate the current envelope (by pressing corresponding buttons).

The maximum values of calculated internal forces (bending moments and shear forces) and the magnitude of displacement are displayed at the bottom part of the frame.

The program allows us to dimension steel-reinforced concrete and steel cross-sections (by checking the option "**Check cross-section**"). For a more detailed design of reinforcement in concrete cross-section, it is possible to divide the structure into sections which are then assessed separately. When checking the cross-section, it is possible to introduce the **reduction coefficient of bearing capacity**, which reduces the overall bearing capacity of a cross-section. The magnitude of this coefficient depends on the way, how active and passive forces (frame "Pressure determination") were computed. If these values are design values (already increased by partial factors), this coefficient should be 1.0 - if not, this coefficient should be higher then 1.0. (For EN 1997 is this value in interval 1.35 - 1.5).

The frame allows us to perform a larger number of analyses pro dimensioning of a cross-section. The **"In detail**" button at the right part of the frame opens the "**Dimensioning**" dialog window to show detailed results.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Dimensioning" - RC cross-sections

Anchors Verification

The "**Anchors verification**" frame contains a table with a list of input anchors.

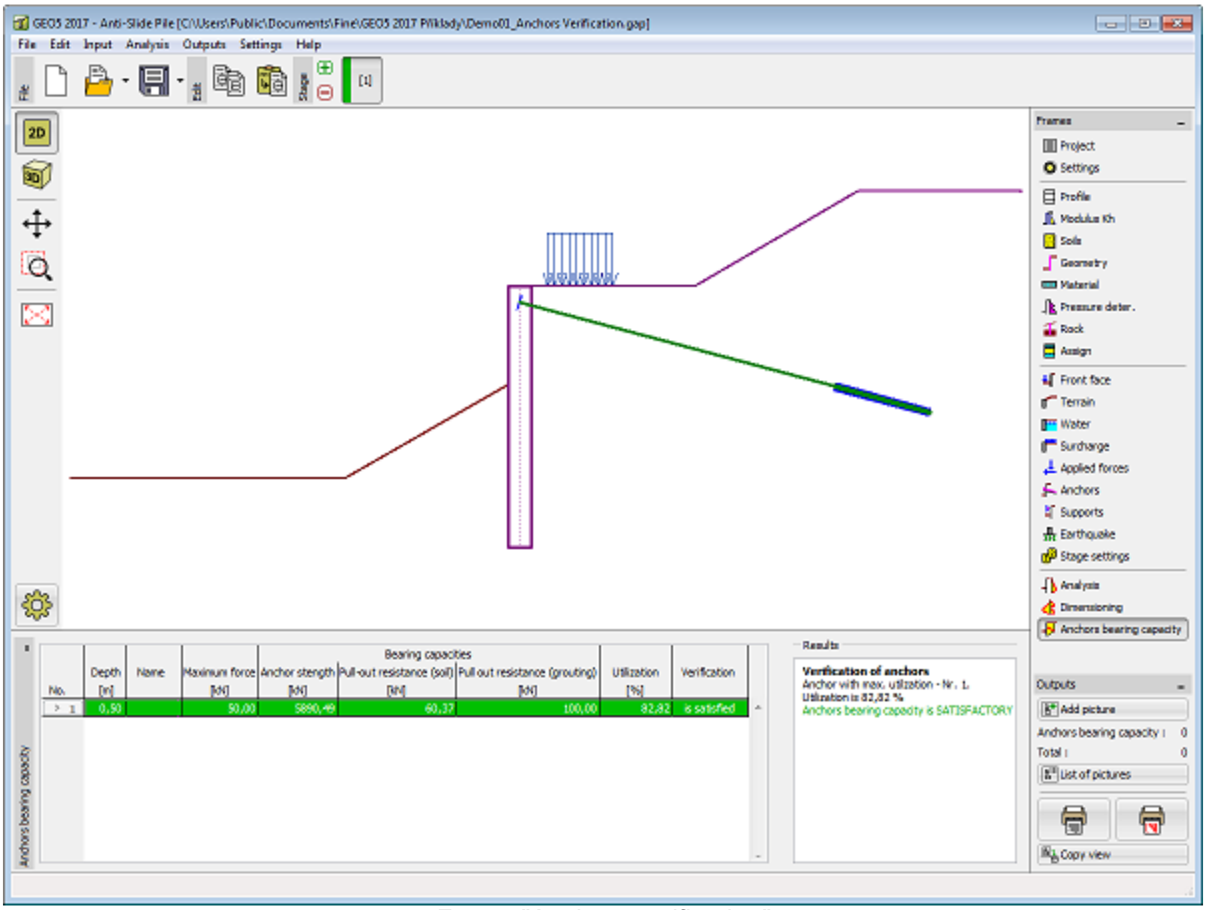
The **overall bearing capacity** of anchors is verified in this frame.

The anchors can be check for three different **types of failure**.

- Strength of anchor R_t
- Pull-out resistance (soil) R_e
- Pull-out resistance (grouting) R_c

Computed **bearing capacities** of anchors **are reduced** by corresponding safety factor or reduction coefficient, which are defined in the frame "Settings", tab "Anchors". The verification of anchor is satisfactory, when the maximum force in the anchor P_{max} is lower than all computed bearing capacities.

$$\min\left(\frac{R_t}{SF_t};\frac{R_e}{SF_e};\frac{R_c}{SF_c}\right)\geq P_{max}$$



Frame "Anchors verification"

Lagging

The frame "**Lagging**" serves to design and verify the lagging. This frame is available for the soldier pile walls only.

The program allows us to dimension reinforced concrete and timber cross-sections (by checking the option "**Check cross-section**").

The "**Edit Lagging**" dialog window (the "**Material, Section**" button) allows us for selection of **material** (timber, concrete) and **cross-section** (rectangle, circle). The "**Catalog**" and "**User. defined**" buttons are used to set the material properties in the same way as in the "Material" frame.

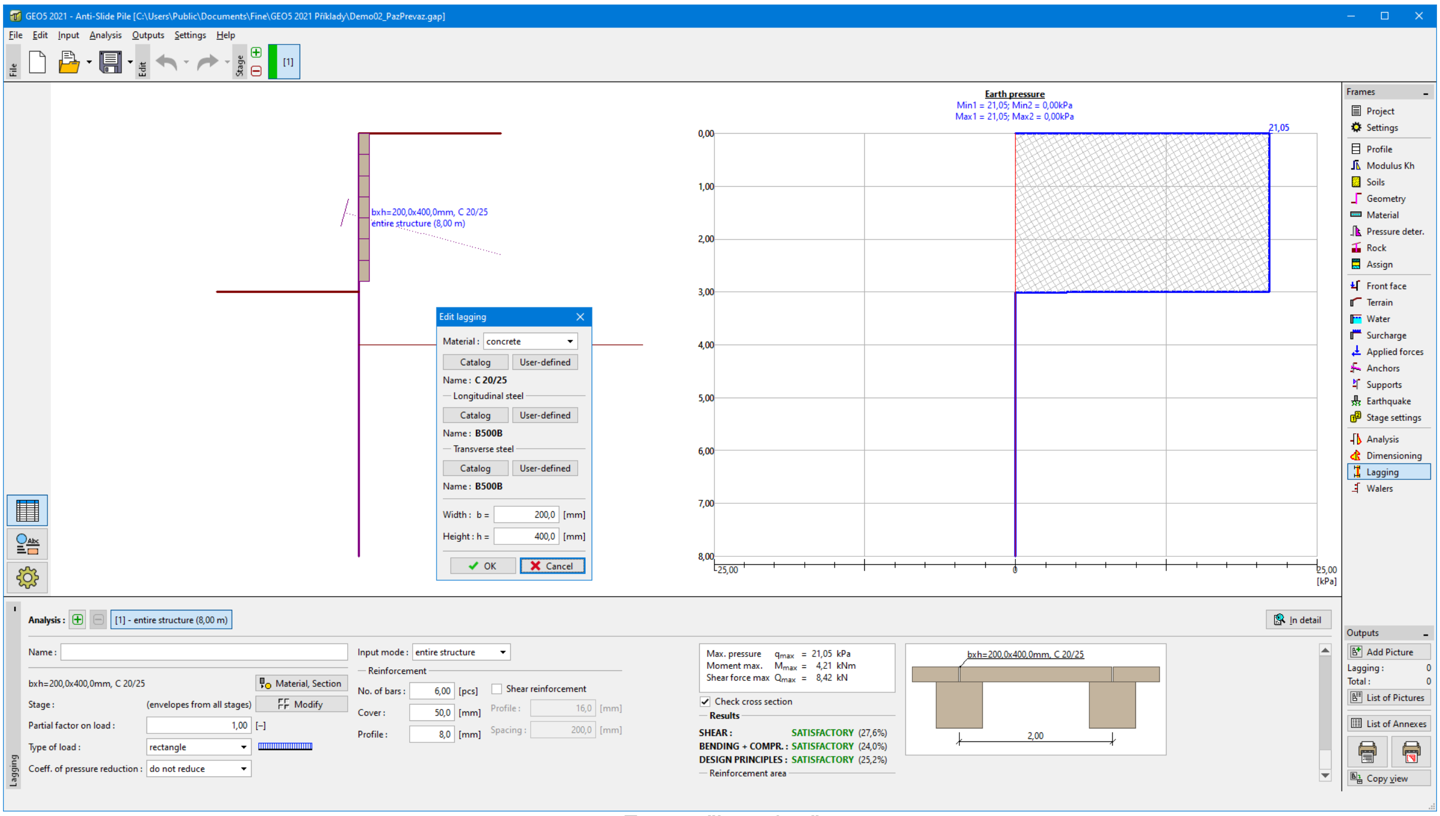
In the frame, it is possible to display an envelope of pressures acting on the wall from all analyses (stages of constructions). Normally, the envelope is constructed from the results of all construction stages. The **"Modify"** button opens the dialog window **"Construction stage selection"**, where it is possible to select the construction stages that are used to generate the current envelope (by pressing corresponding buttons).

The frame further allows for selecting the **load type** (rectangle, triangle) and **coeff. ofpressure reduction** acting on the lagging. When checking the cross-section, it is possible to input the **partial factor on load**, which multiplies the values of internal forces.

The values of calculated internal forces (bending moments and shear forces) and the maximum pressure are displayed in the right part of the frame.

The frame allows us to perform a larger number of analyses for the dimensioning of a cross-section. For a more detailed design of reinforcement in concrete cross-section, it is possible to divide the structure into sections which are then assessed separately.

The **"In detail**" button at the right part of the frame opens the "**Dimensioning**" dialog window to show detailed results.



Frame "Lagging"

Walers

This frame serves to design and verify the waler for selected anchor.

The program allows us to dimension reinforced concrete and steel cross-sections (by checking the option "**Check cross-section**"). The frame allows us to perform a larger number of analyses for dimensioning of a cross-section.

The "**Edit Waler**" dialog window (the "**Material, Section**" button) allows us for a selection of **material** (steel, concrete) and **cross-section**. The "**Catalog**" and "**User. defined**" buttons are used to set the material properties in the same way as in the "Material" frame.

In the case of the steel waler, the **cross-section type** (I-section, 2xI-section, or 2xU-section) is selected in the "**Catalog of profiles**" (the "**Catalog**" button) or in the "**Cross-section editor**" (the "**Welded**" button).

The **waler rotation** can be considered according to the **anchor** or the **wall**.

The **"Modify"** button opens the dialog window **"Construction stage selection"**, where it is possible to select the construction stages used to determine the maximum force.

When checking the cross-section, it is possible to input the **partial factor on load**, which multiplies the values of internal forces. For dimensioning of steel cross-sections, it is possible to assume the **influence of normal force.**

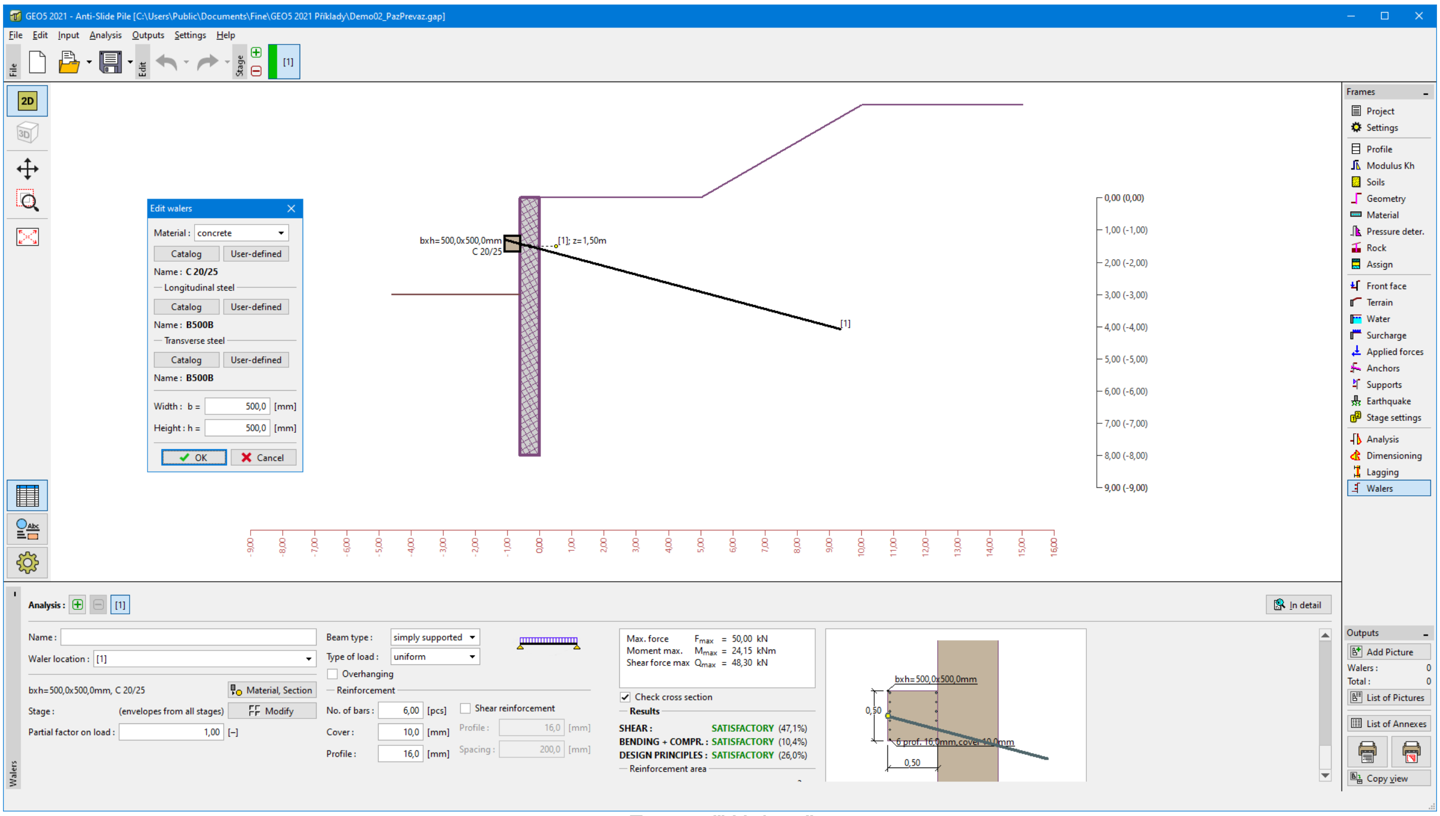
The frame further allows for selecting the **static scheme**:

- **Beam type** (simply supported, continuous)
- **Load type** (point, uniform)
- **Number of supports**
- **Overhangs**

For a **simply supported beam** loaded by **concentrated load**, the support distance L is input.

The values of calculated internal forces (bending moment and shear force) and the maximum force are displayed at the bottom part of the frame.

The **"In detail**" button at the right part of the frame opens the "**Dimensioning**" dialog window to show detailed results.



Frame "Walers"

Program Shaft

This program is used to analyze spatial earth pressures on the circular shaft and determination of internal forces on the structure.

The help in the program "Shaft" includes the following topics:

- The input of data into individual frames:

Project	Settings	Geometry	Profile	Soils	Assign
Water	Surcharge	Stage	Load	Dimensioning	
		Settings	Analysis		

- Standards and analysis methods

- Theory for analysis in the program "**Shaft**":

Geostatic Stress, Uplift Pressure	Strip Surcharge - Active Earth Pressure	Concentrated Surcharge - Active Earth Pressure	Line Surcharge - Active Earth Pressure	Concentrated Surcharge - Earth Pressure at Rest	Shaft

- Outputs

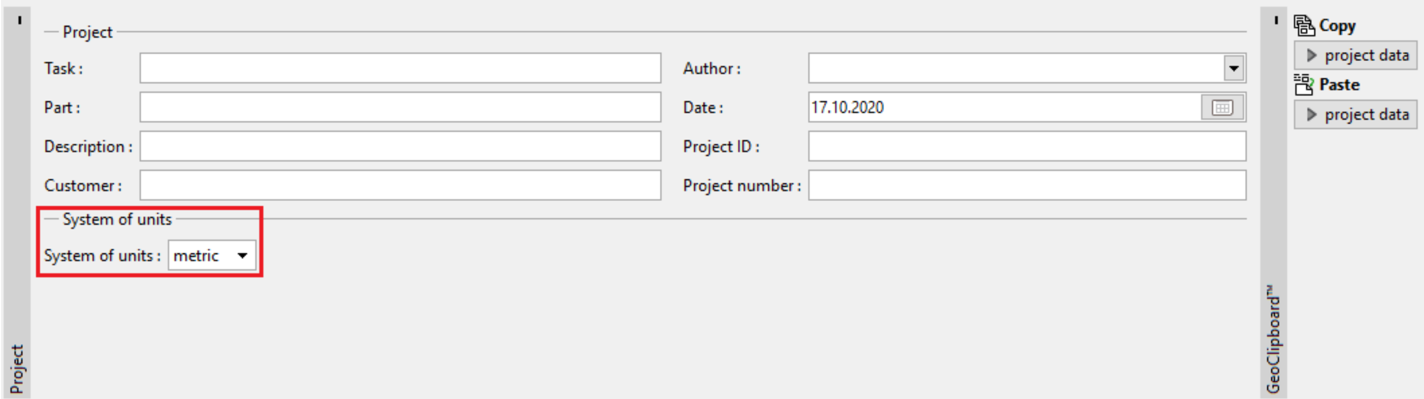
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The frame **"Project"** is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The frame "**Settings**" allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

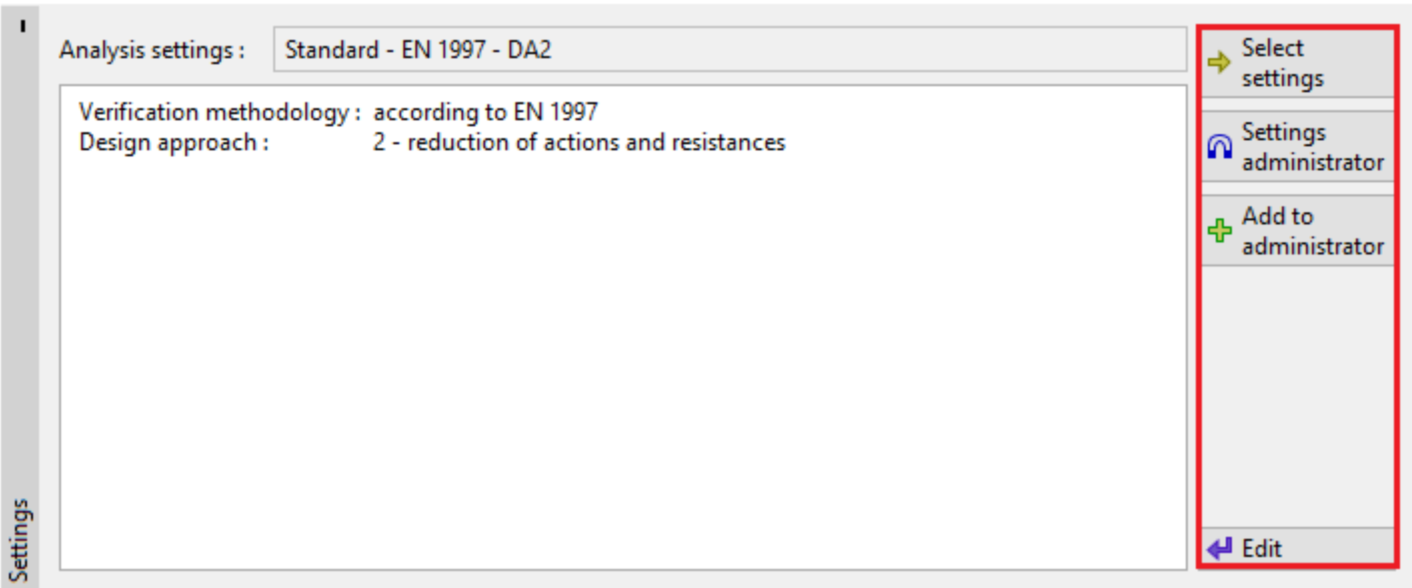
The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

Settings of analysis parameters are performed in the "Pressure Analysis" tab.



Frame "Settings"

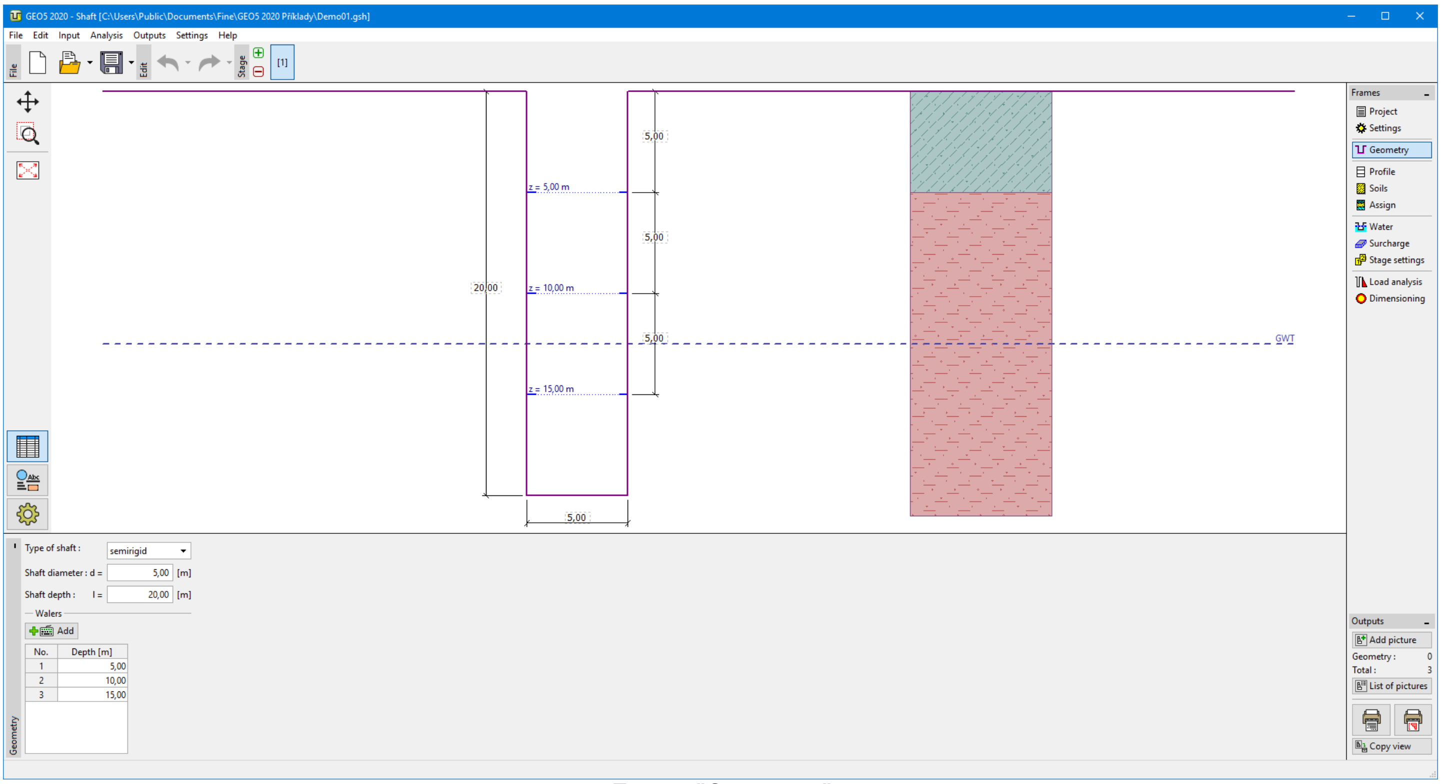
Geometry

In the frame "**Geometry**", the **shaft type** (flexible, semirigid, rigid) is selected and **shaft diameter** d, **shaft depth** l and **depth of walers**.

The frame contains a table with a list of waler depths. Adding segments is performed in the "**New waler**" dialog window.

Dimensions of the structure and depth of walers can be modified in the frame by editing values in the input fields or on the desktop by using active dimension.

The program allows us to export the geometry of a structure in *.DXF format.



Frame "Geometry"

Profile

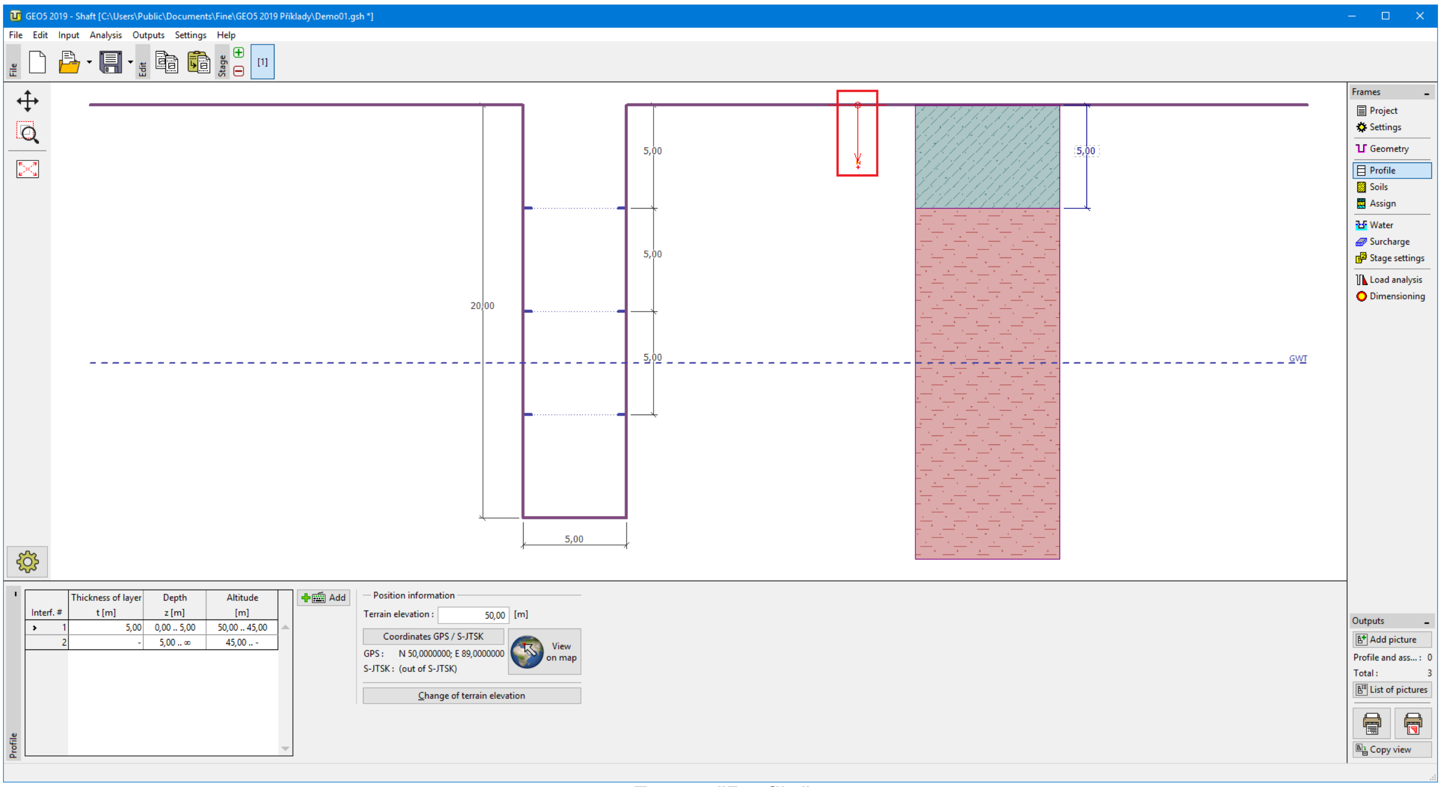
The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.



Frame "Profile"

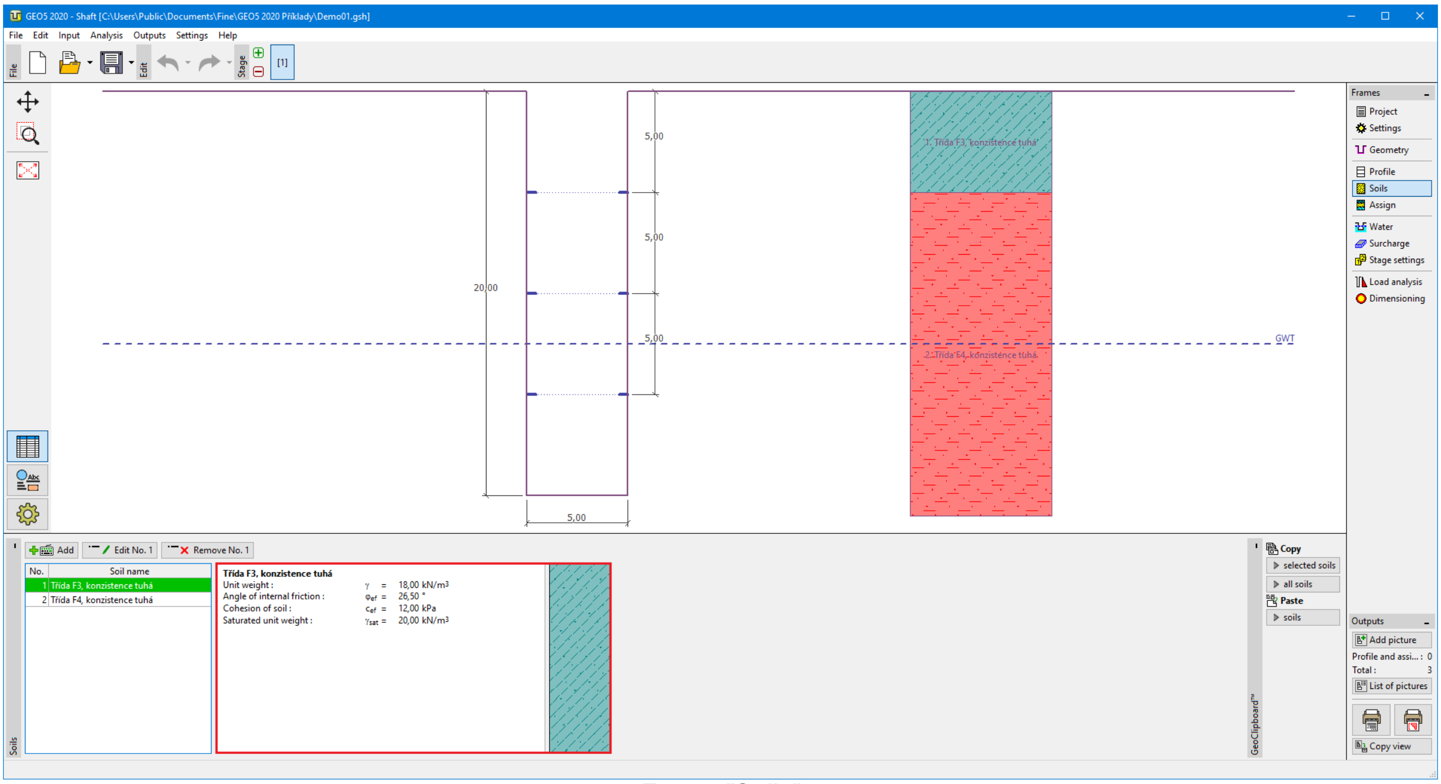
Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data" and "Uplift pressure".

Data of input soils can be copied within all GEO5 programs using "GeoClipboard".

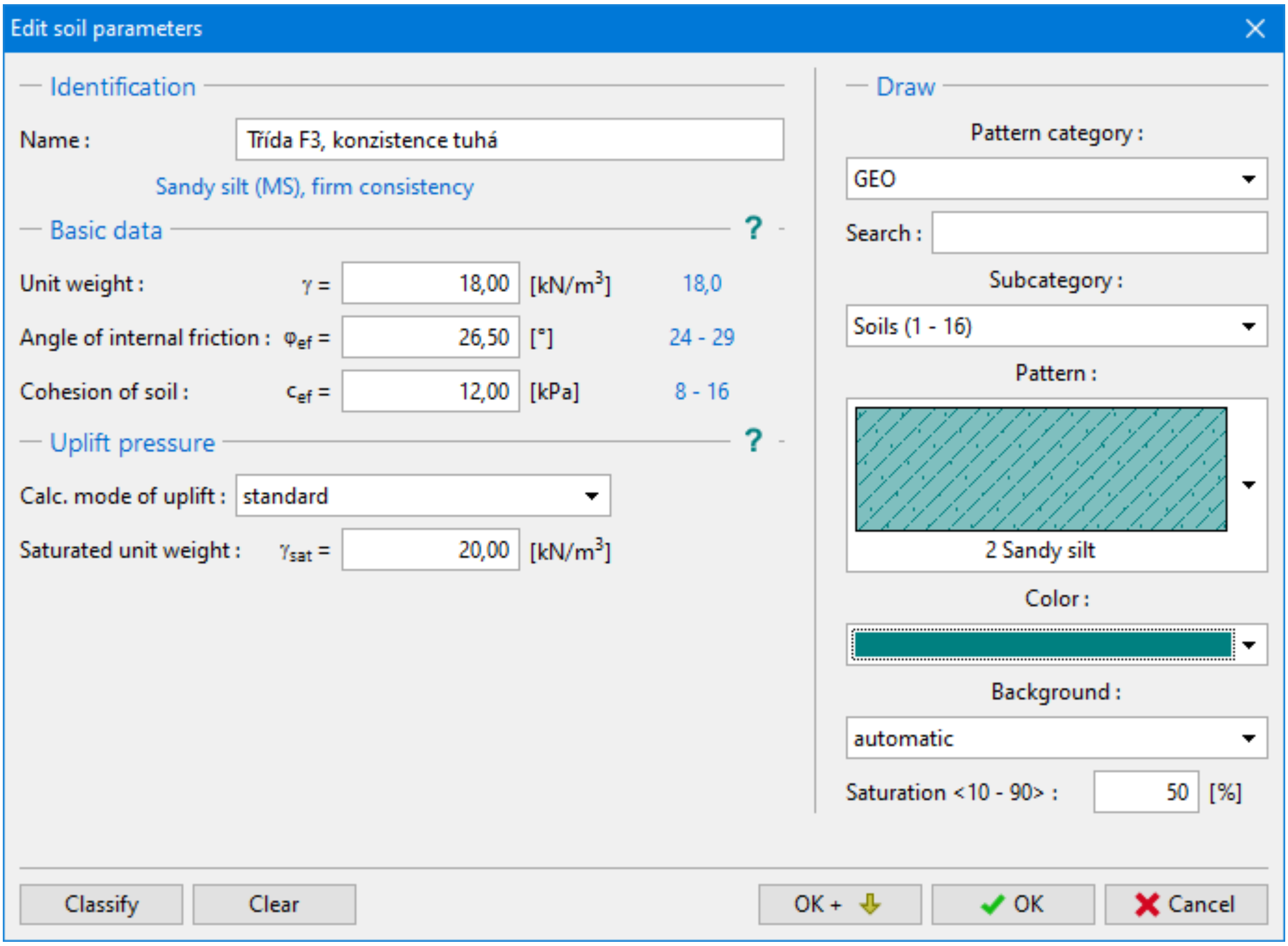


Frame "Soils"

Basic data

This part of the window introduces basic parameters of soils - **unit weight, angle of internal friction, and cohesion**. The particular values are obtained from a geotechnical survey or laboratory experiments. If these data are not available, it is possible to exploit the built-in database of soils, which contains the values of selected soil characteristics. The characteristics of rocks are not listed in the database built, these parameters must be defined manually.

The associated theory is described in detail in the chapter "Earth pressures".

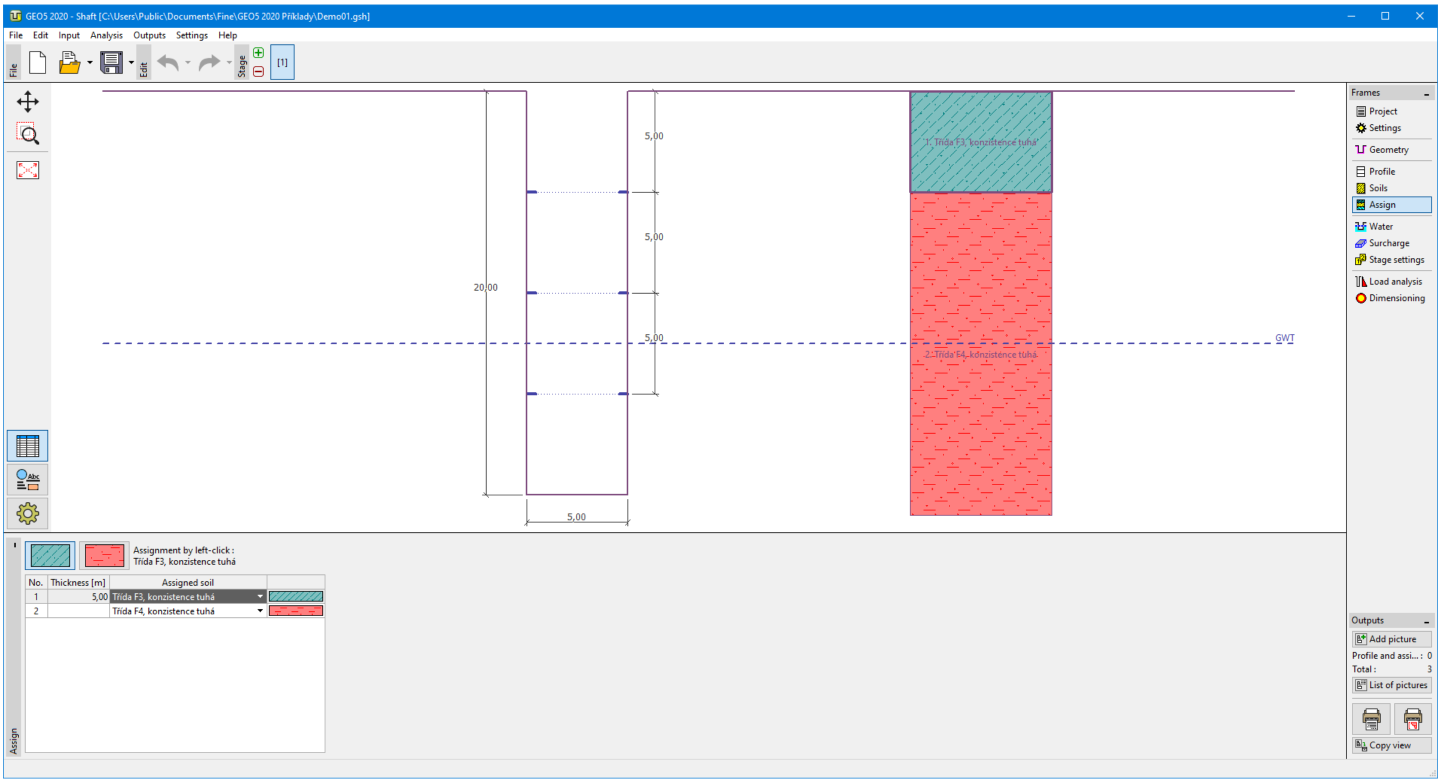


Dialog window "Add new soils" - "Basic data"

Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail herein.



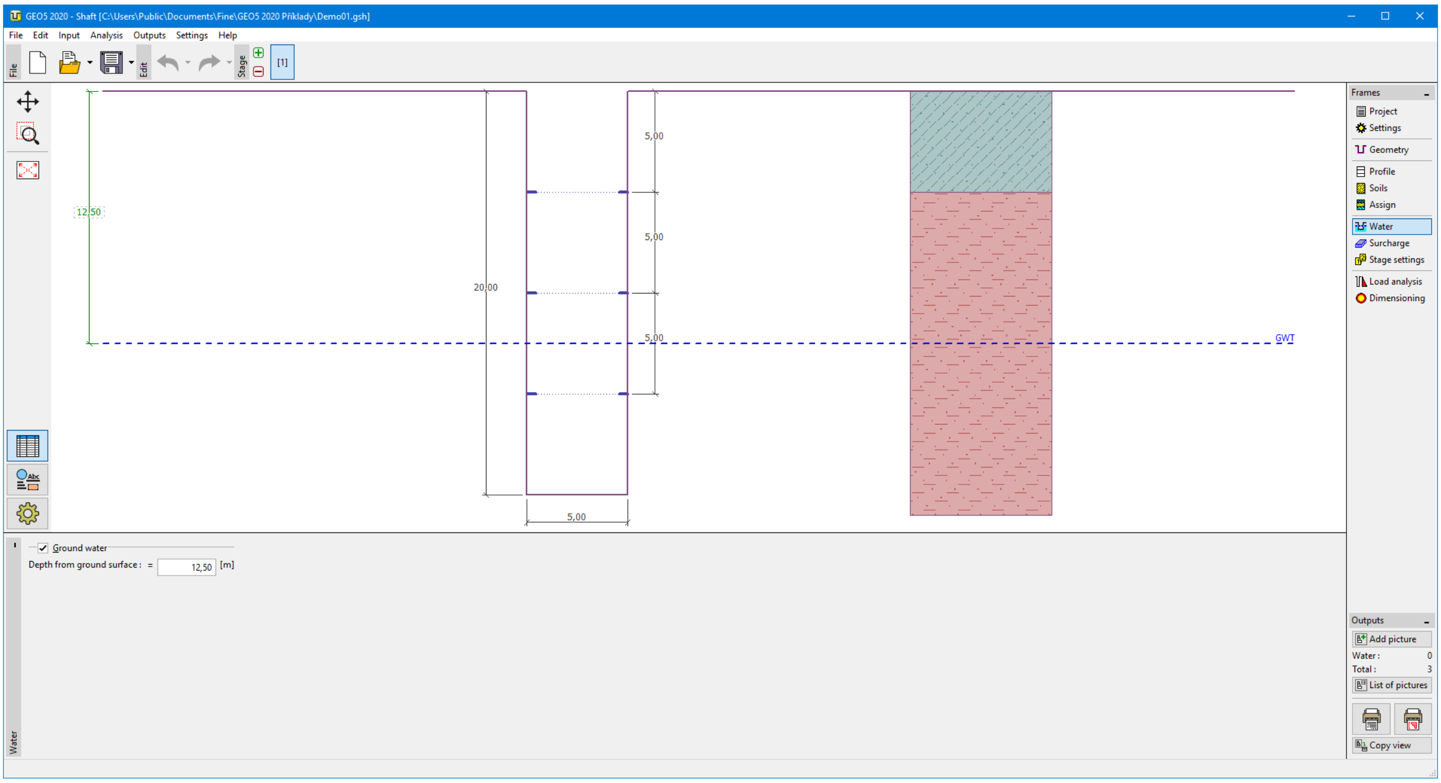
Frame "Assign"

Water

The frame "**Water**" serves to enter a **depth of groundwater table.**

The values can be edited either in the frame by entering values into particular fields, or on the desktop with the help of active dimensions.

The **GWT** changes the geostatic stress in the soil profile.



Frame "Water"

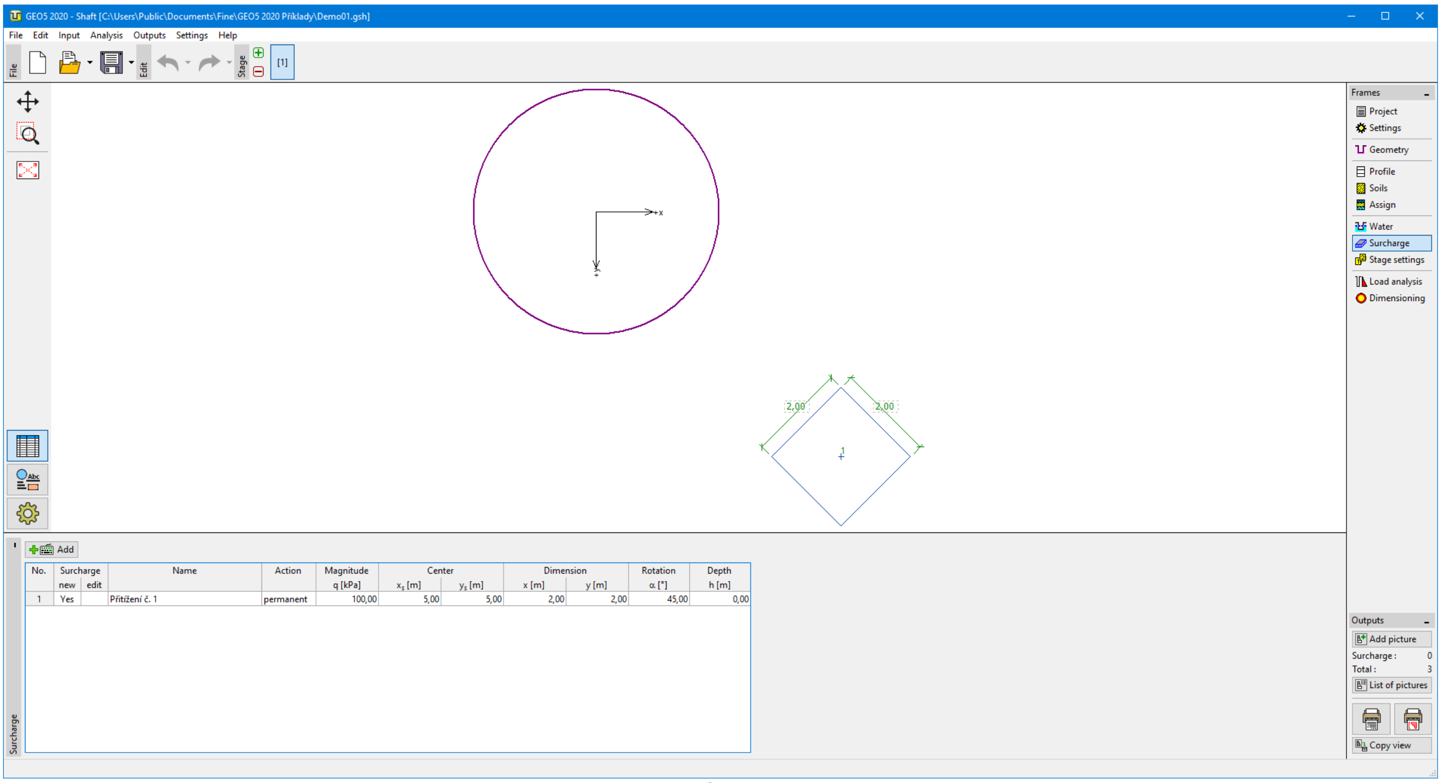
Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding surcharge is performed in the "**New**

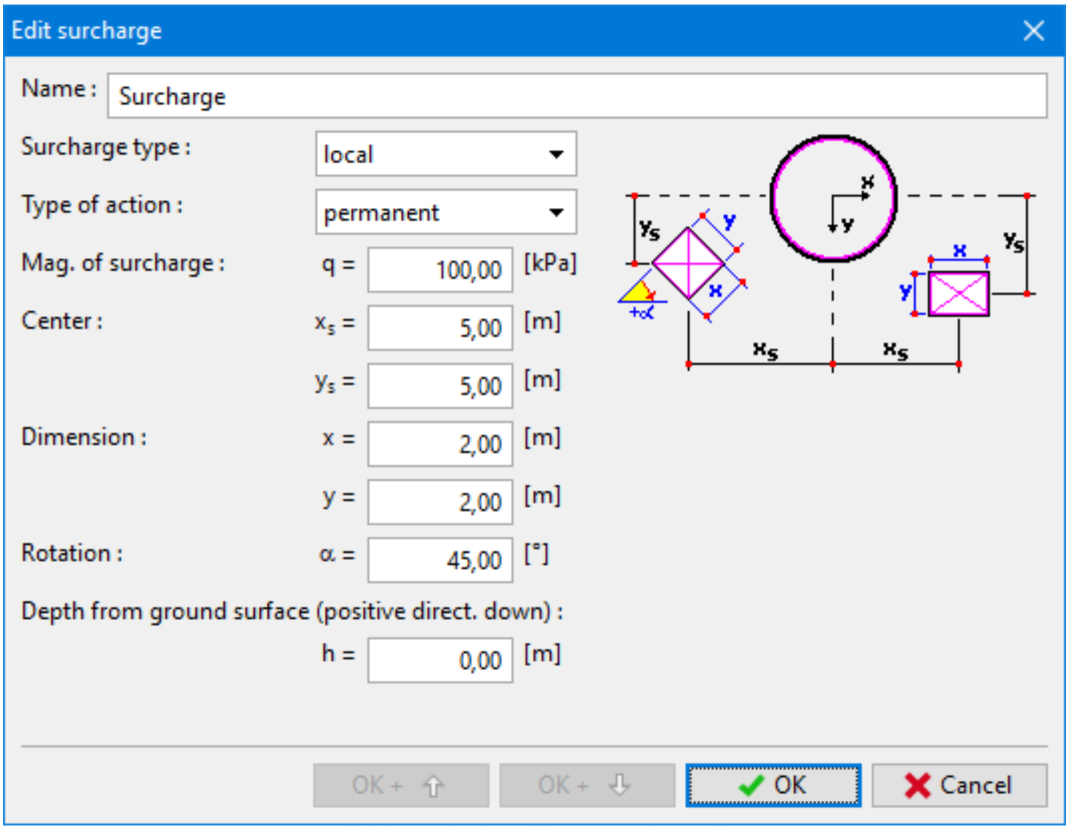
surcharge" dialog window. The input surcharges can also be edited on the desktop with the help of active objects.

It is possible to input surcharge as **permanent**, **variable** or **accidental**. Type of surcharge is either **surface** or **local**. The final effect is multiplied by corresponding verification coefficient according to the type of surcharge.

In case of considering surcharge in a different depth then on surface (for example foundation of surrounding buildings), depth ***h*** under the surface is input (positive direction downwards).



Frame "Surcharge"



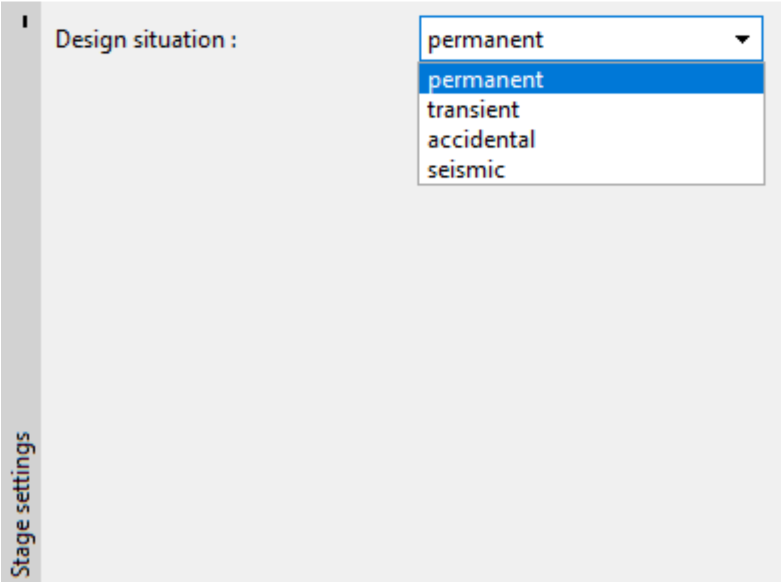
Dialog window "New surcharge"

Stage Settings

The frame "**Stage settings**" serves to input settings valid for a given construction stage.

The selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology.



Frame "Stage settings"

Load Analysis

The frame **"Load analysis"** allows us to determine the final load in the **vertical direction**, final load on the **walers,** or in the **input depth** (for shafts without walers). The program calculates all partial loads, multiplies them by corresponding partial factors, and shows the final load on the screen.

Calculated uniform earth pressure can be modified (in the compliance with DIN or SNIP standards) by reduction coefficient, so the former **"circular"** load is changed to **"elliptical"**. It is possible to specify the way of modification (increase and decrease the load, only decrease the load) and the value of the **reduction coefficient** (recommended value is 25 *%*).

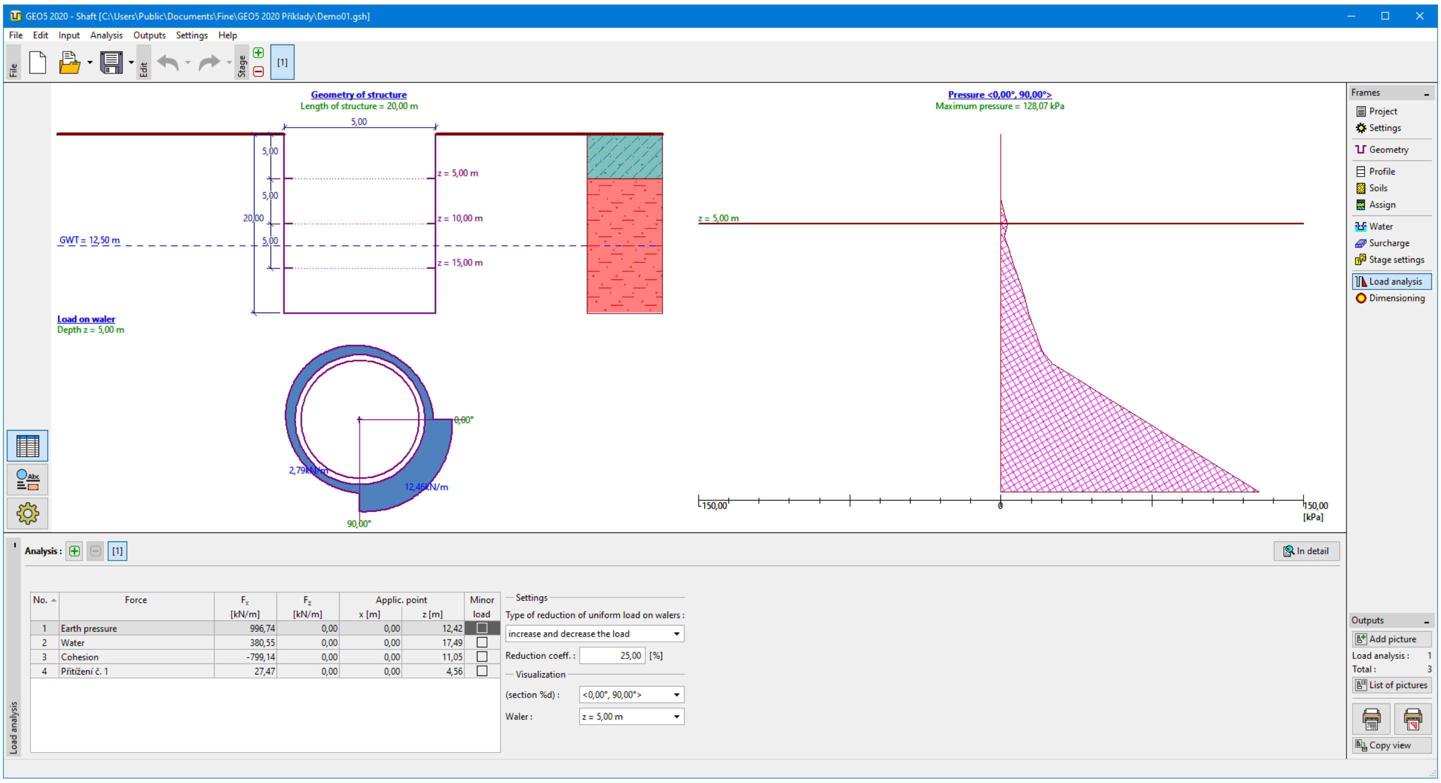
Computed load is the input for analysis of internal forces in the frame "Dimensioning". The program automatically computes the load on all walers or in input depth (shaft without walers).

Several computations can be carried out for a single task.This is very useful for determination of combinations of load cases - in the frame "Dimensioning" you can work with all here computed combinations.

The frame appearance is adjusted based on the selected verification methodology:

- Verification according to the safety factor, or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiplies the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows for specifying whether the load acting on a structure is considered as the secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in this case, the last column is not displayed.

The analysis results are displayed on the desktop and are updated immediately for an arbitrary change in input data or setting. Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Load analysis"

Dimensioning

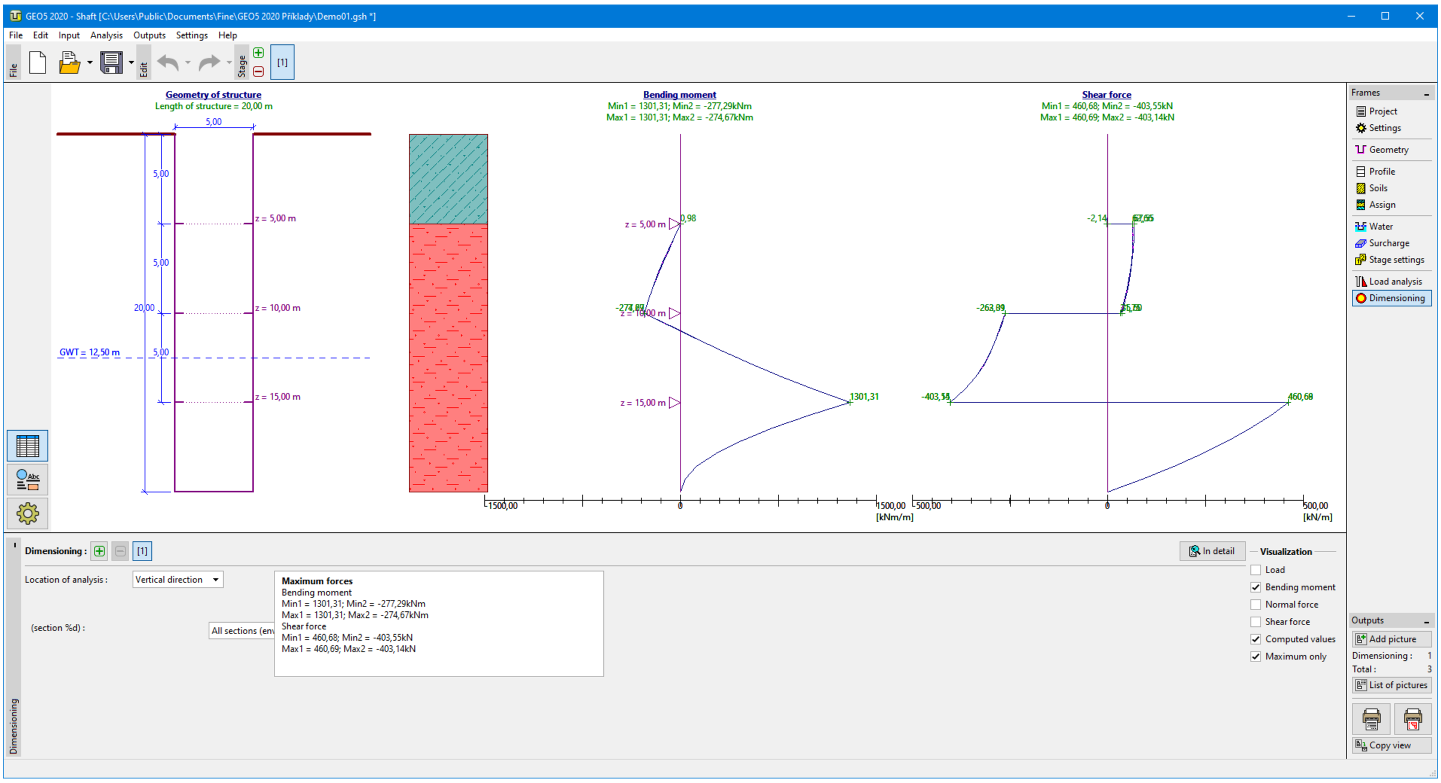
In the frame **"Dimensioning"**, the program computes the internal forces on the shaft for the load, already defined in the frame "Analysis". Two types of analysis can be performed:

- internal forces in the vertical direction
- internal forces on walers or in specified depth (for shafts without walers)

The analysis is performed for the selected load or the envelopes of internal forces computed for all specified loads (load cases).

Several computations can be carried out for a single task. Maximum values of internal forces are displayed in the output window. The "**In detail**" button opens a dialog window that contains a detailed listing of dimensioning results.

When analyzing in vertical direction, the program allows us to select a specific section (considering local surcharge is input, so more sections are available) or compute envelope of internal forces for all sections.



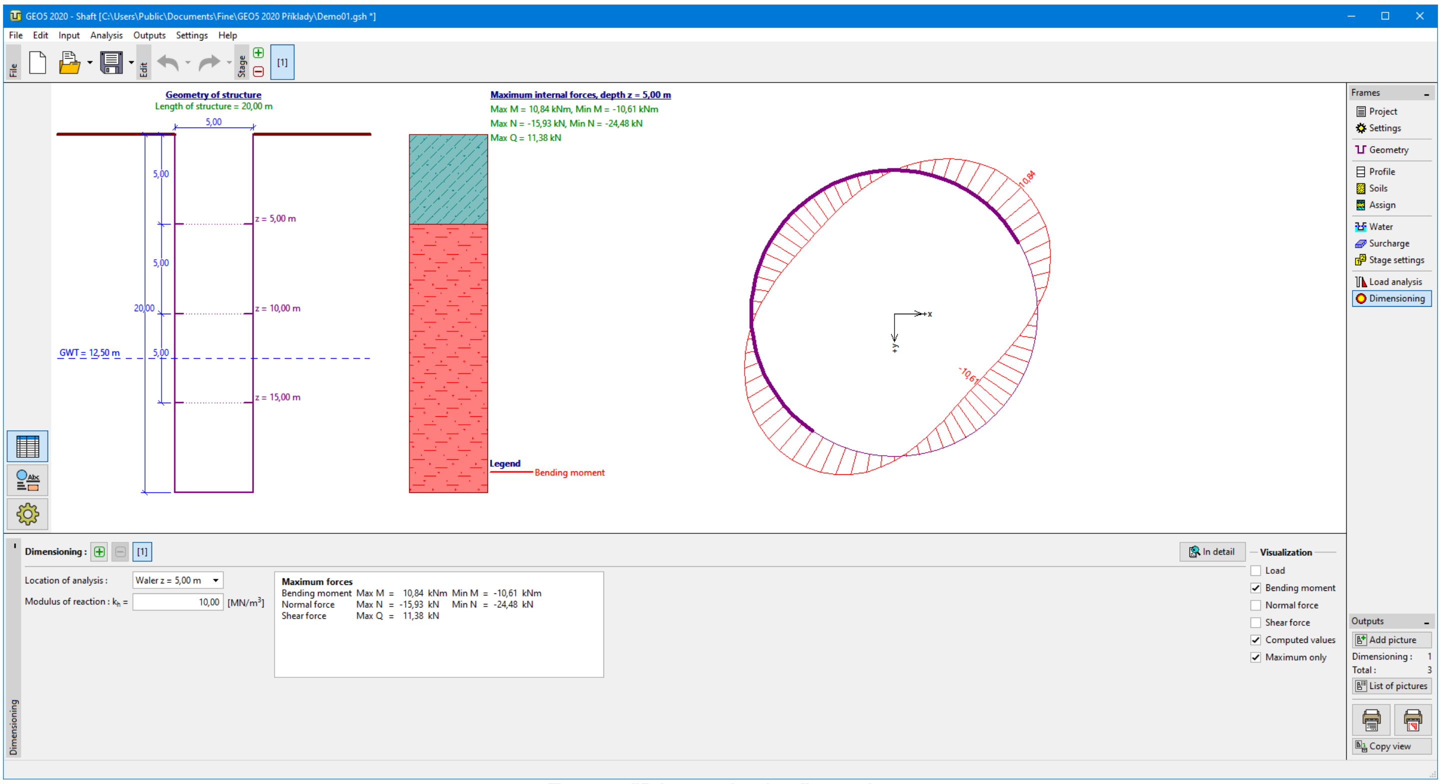
Frame "Dimensioning" - vertical direction

When analyzing in the horizontal direction (Internal forces on the waler or in input depth), the **modulus of subsoil reaction** has to be inputed for the soil in a specified depth.

The program shows the load, bending moment, and shear force - the rendering form is defined in part **"Visualization"**. The **"Maximum values"** button hides all values except the maximum values.

Part of the picture is marked with a bold line - this the part of the shaft, that deforms into the soil. In this part, the subsoil springs are considered in the analysis by the polygonal method.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Dimensioning" - waler

Program Slope Stability

This program is used to perform slope stability analysis (embankments, earth cuts, anchored retaining structures, MSE walls, etc.). The slip surface can be circular (Bishop, Fellenius/Petterson, Janbu, Morgenstern-Price, or Spencer methods) or polygonal (Sarma, Janbu, Morgenstern-Price, or Spencer methods).

The help in the "Slope Stability" program includes the following topics:

- The input of data into individual frames:

Project	Settings	Interface	Embankment	Earth Cut	Soils	Rigid Bodies
Assign	Anchors	Nails	Reinforcements	Anti-Slide Piles	Surcharge	Water
Earthquake	Stage Settings	Analysis				

- Standards and analysis methods

- Theory for analysis in the program "**Slope Stability**":

Stress in Soil Body	Anti-Slide Pile	Slope Stability	Water Flow Analysis

- Outputs

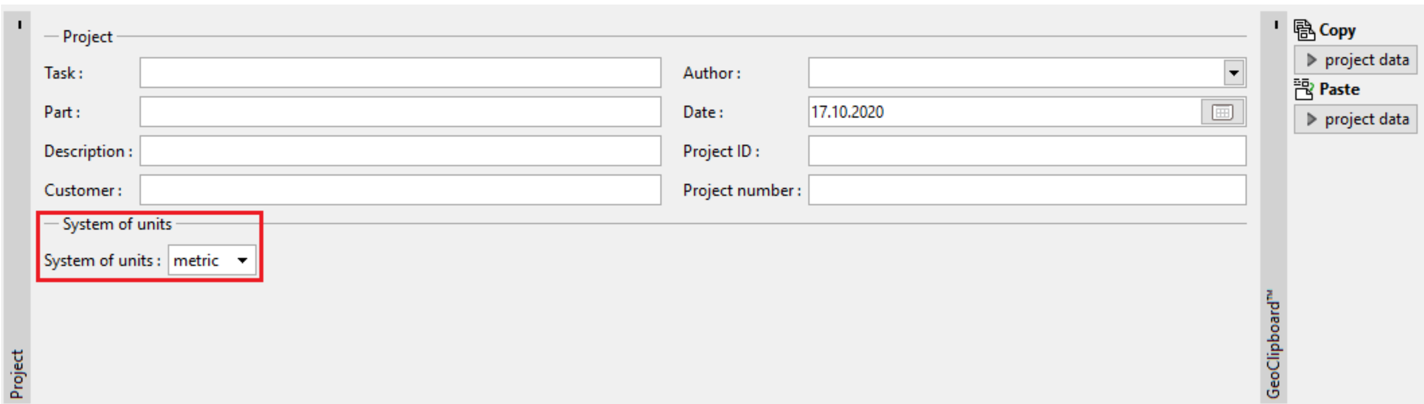
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The **"Project"** frame is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows the user to switch analysis units (metric/imperial). The project data can be copied within all GEO5 programs using "GeoClipboard".



"Project" frame

Settings

The "**Settings**" frame allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select settings**" button allows us to choose an already created setting from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying an individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

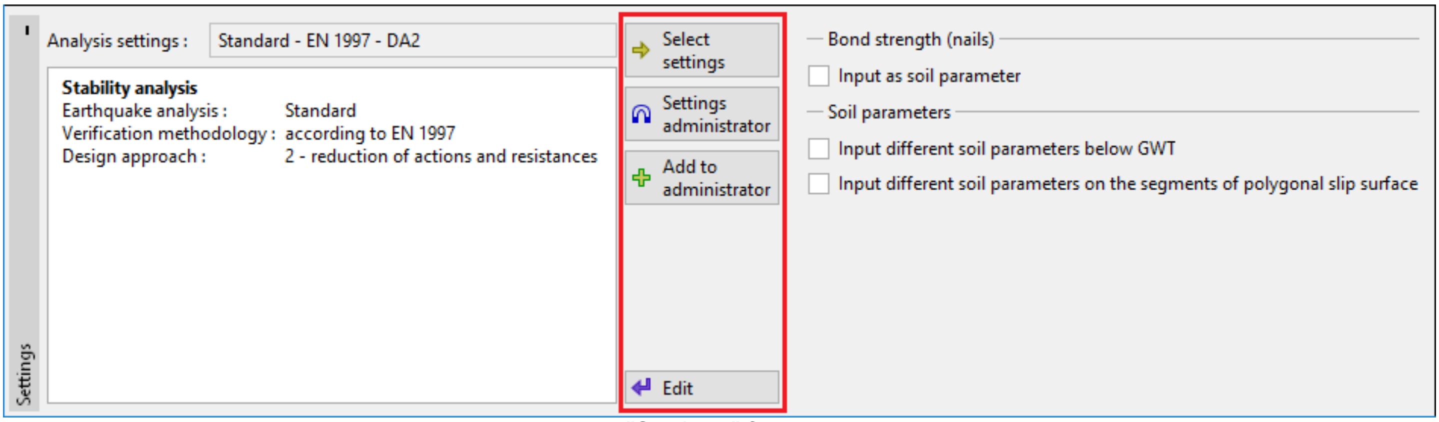
The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. In case this setting is suitable also for other tasks, we will add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

Settings of analysis parameters are performed in the "Materials and standards" and "Stability analysis" tabs.

When calculating the pull-out resistance of the nails, a bond strength can be inserted as a soil parameter.

The program allows the user to **input different soil shear parameters below the GWT** or enter them directly on the slip surface segments.

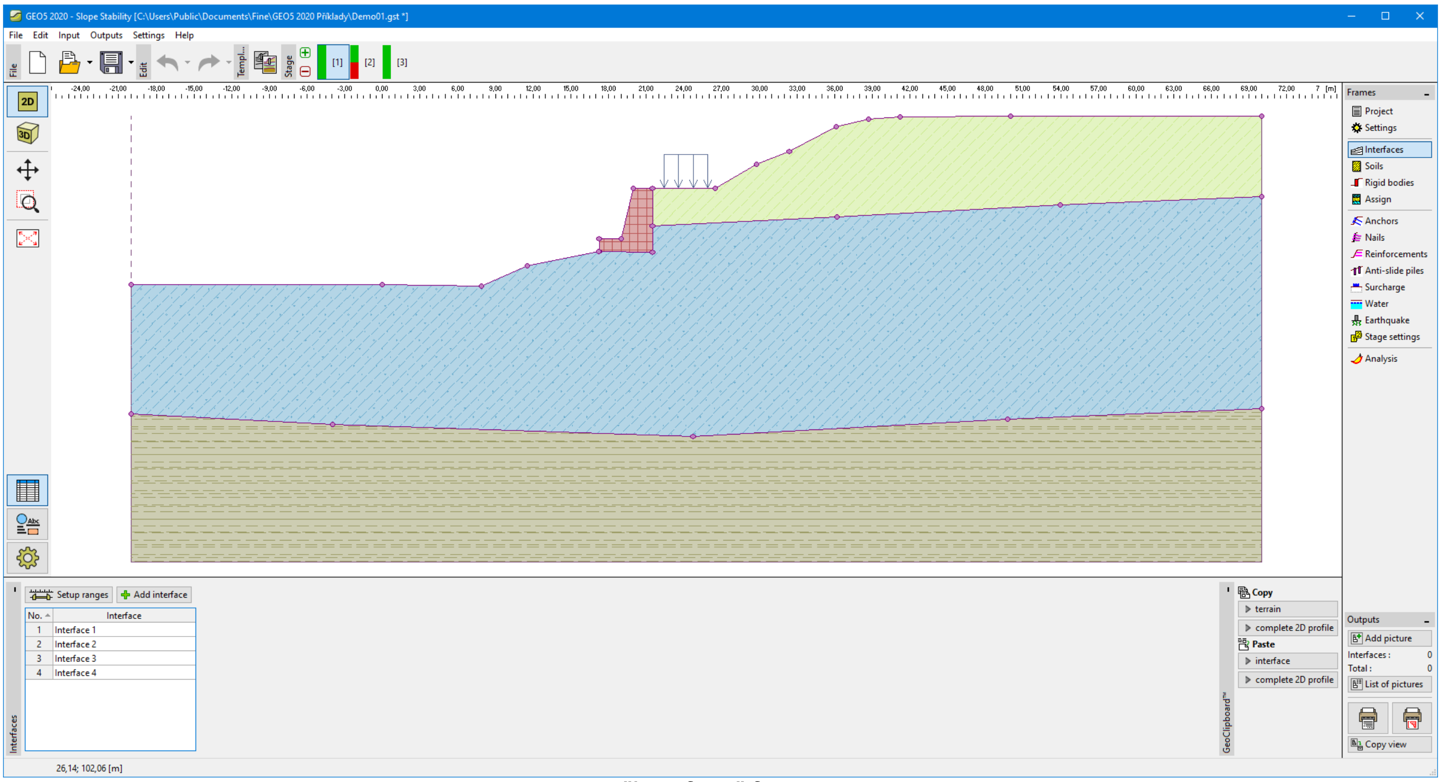


"Settings" frame

Interface

The "**Interface**" frame serves to introduce individual soil interfaces into the soil body. Detailed description of how to deal with interfaces is described herein.

The program makes it possible to import or export interfaces in the *.DXF format. Input interfaces can be copied within all 2D GEO5 programs using "GeoClipboard".



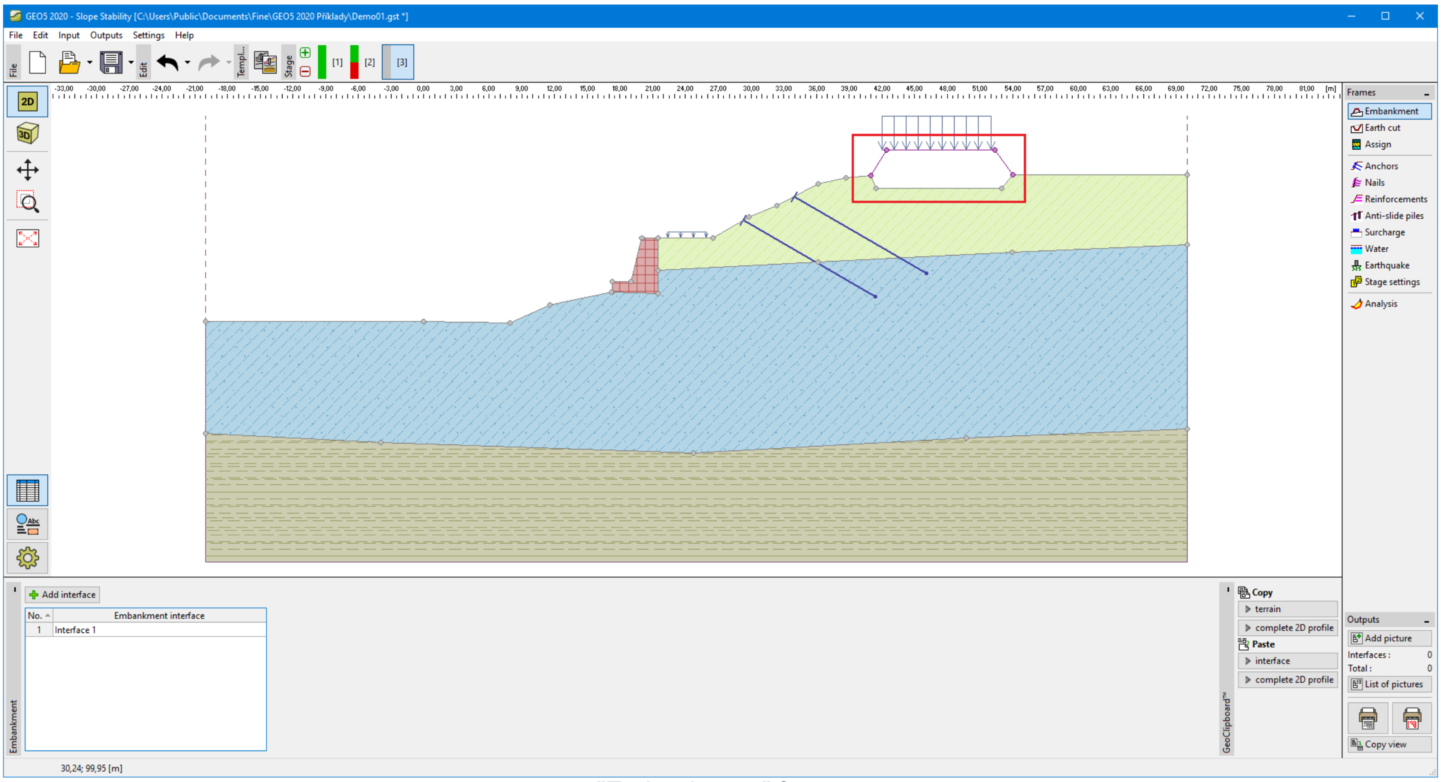
"Interface" frame

Embankment

The "**Embankment**" frame allows us to input interfaces to create an embankment above the current terrain. The frame contains a table with a list of interfaces forming the embankment. A table listing the points of a currently selected interface of the embankment is displayed in the midsection of the frame. Inputting an embankment interface follows the same steps as used for standard interfaces.

An embankment cannot be specified in the first stage of construction. An embankment cannot be built if there is an earth cut already specified in a given stage. In this case, an open cut must be removed, or the embankment must be constructed in the next construction stage.

Input interfaces of an embankment can be copied within all 2D GEO5 programs using "GeoClipboard".



"Embankment" frame

Earth Cut

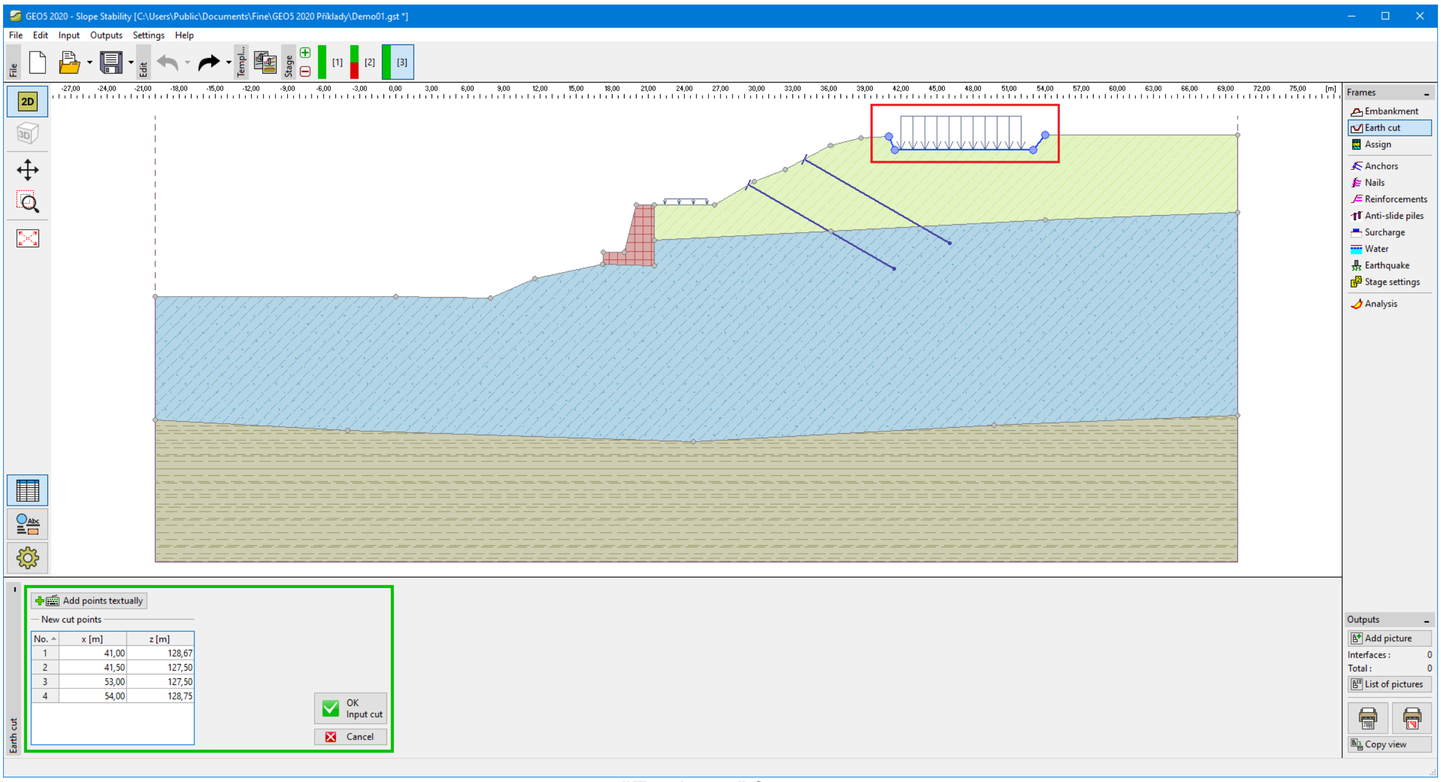
The "**Earth cut**" frame serves to specify the shape of an open cut. This function allows us to modify the terrain profile within a given stage of construction. Several **earth cuts** can be introduced at the same time. In such a case, some of the lines in the cut appear partially above the terrain.

A table listing individual interface points is displayed in the left part of the frame. Inputting an earth cut interface follows the same steps as used for standard interfaces.

An open cut cannot be specified in the first stage of construction.

An earth cut cannot be built if there is an embankment already specified in a given stage. In this case, an embankment must be removed, or the earth cut must be constructed in the next construction stage.

Input interfaces of an earth cut can be copied within all 2D GEO5 programs using "GeoClipboard".



"Earth cut" frame

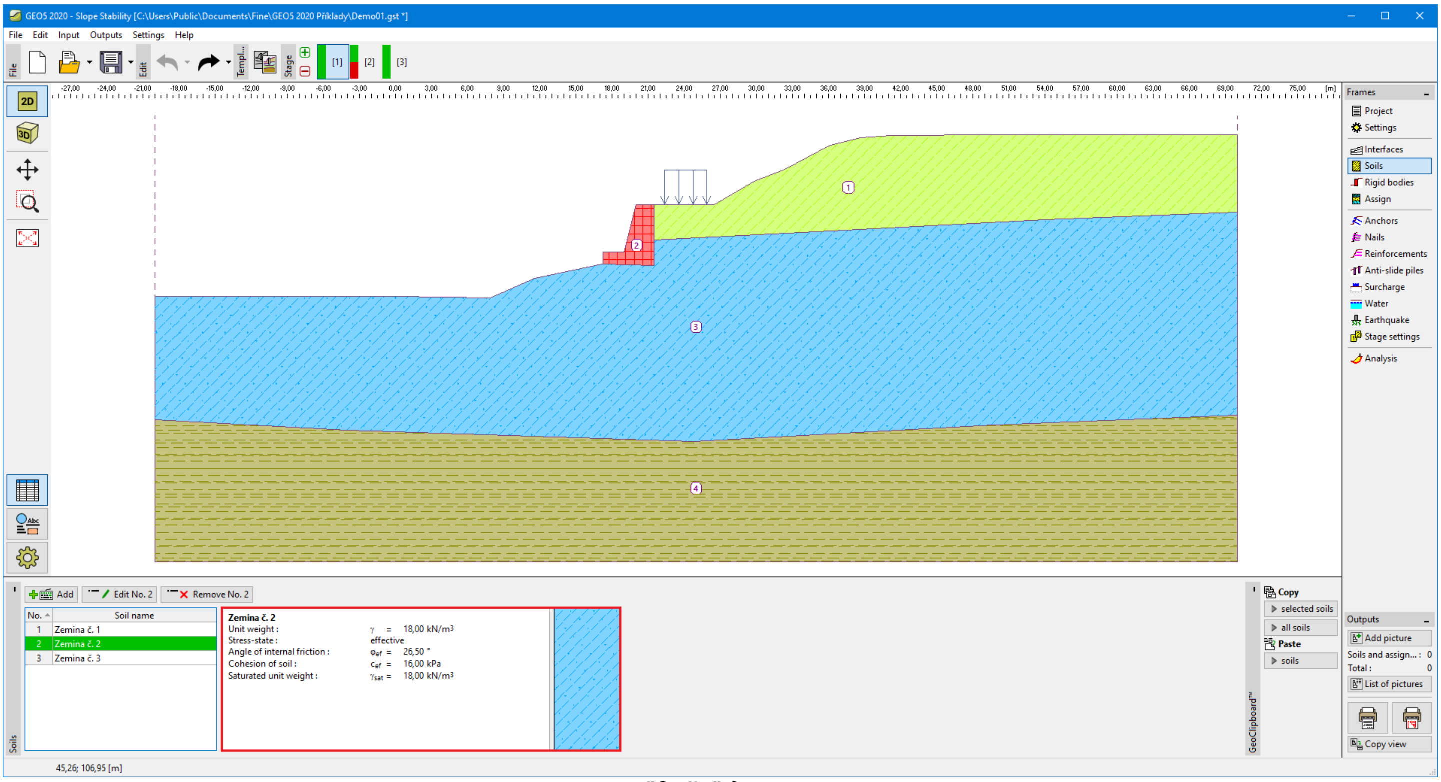
Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Uplift pressure", "Foliation", "Parameters for rapid draw down" and "Bond strength" (see the "Settings" frame). An input of parameters further depends on the selected type of analysis (effective / total stress state), which is set in the combo list.

Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



"Soils" frame

Basic Data

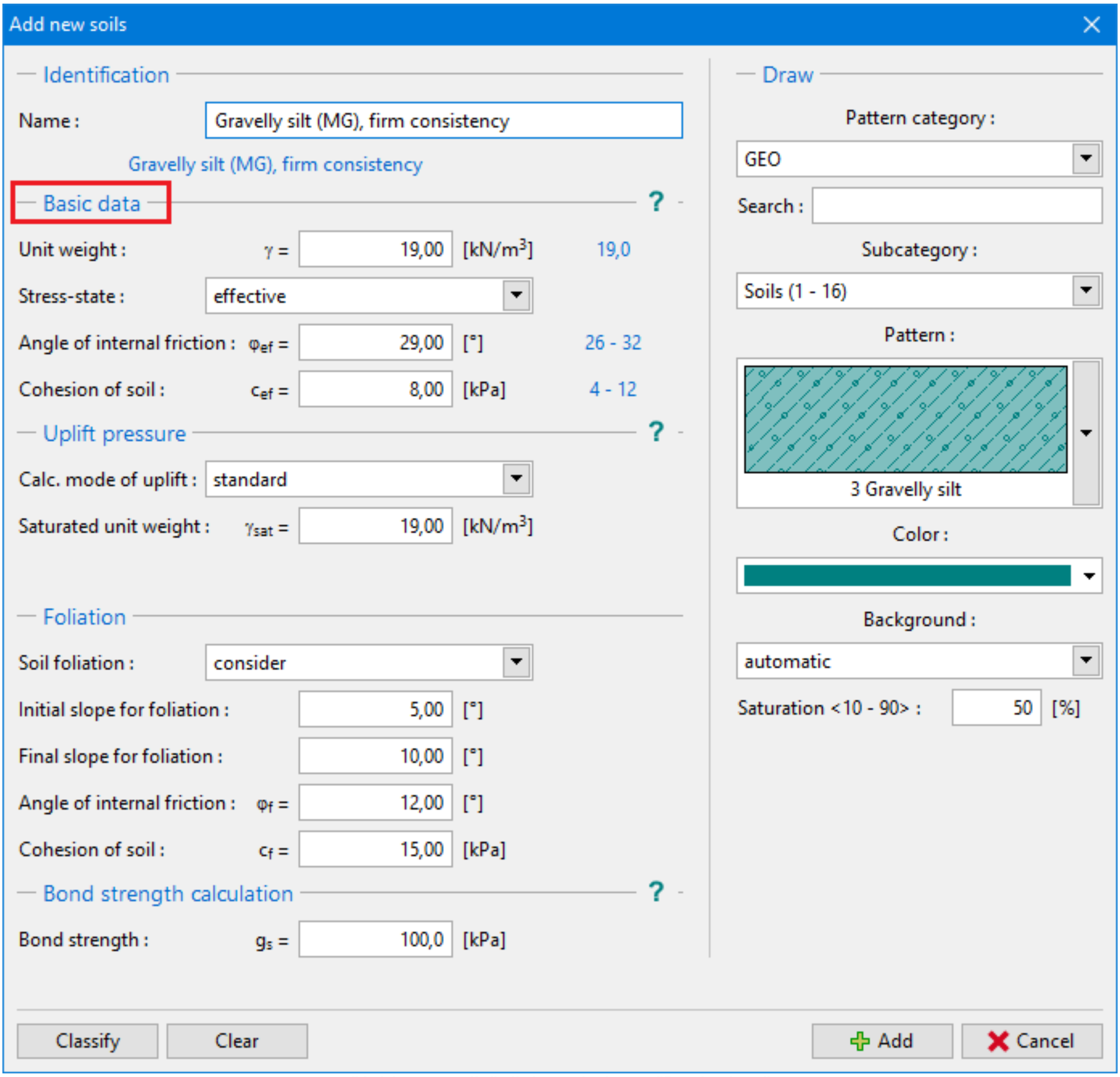
This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction, and cohesion**. The particular values are obtained from a geotechnical survey or from laboratory experiments. If this data is not available, it is possible to exploit the built-in database of soils, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the built-in database, these parameters must be defined manually. Approximate parameters of rocks are presented in the theoretical part of the Help herein.

The analysis method of slope stability differs for:

- **drained conditions:** for the slope stability calculations to determine equilibrium conditions on the slip surface (circular, polygonal), the effective stress is considered according to the equation $N{*}tg\varphi_{ef} + c_{ef}{*}l$.
- **undrained conditions:** in case of the total stress, the calculation of passive forces on the slip surface (circular, polygonal) uses the equation $c_u{*}l$.

In some countries it is customary to specify both shear strength parameters φ_u , c_u for the total stress. In this case it is necessary to specify the task as an effective stress using parameters φ_{ef}, c_{ef} in the "Slope Stability" program.

The associated theory is described in detail in the "Slope stability analysis" chapter.



"Add new soils" dialog window - "Basic data"

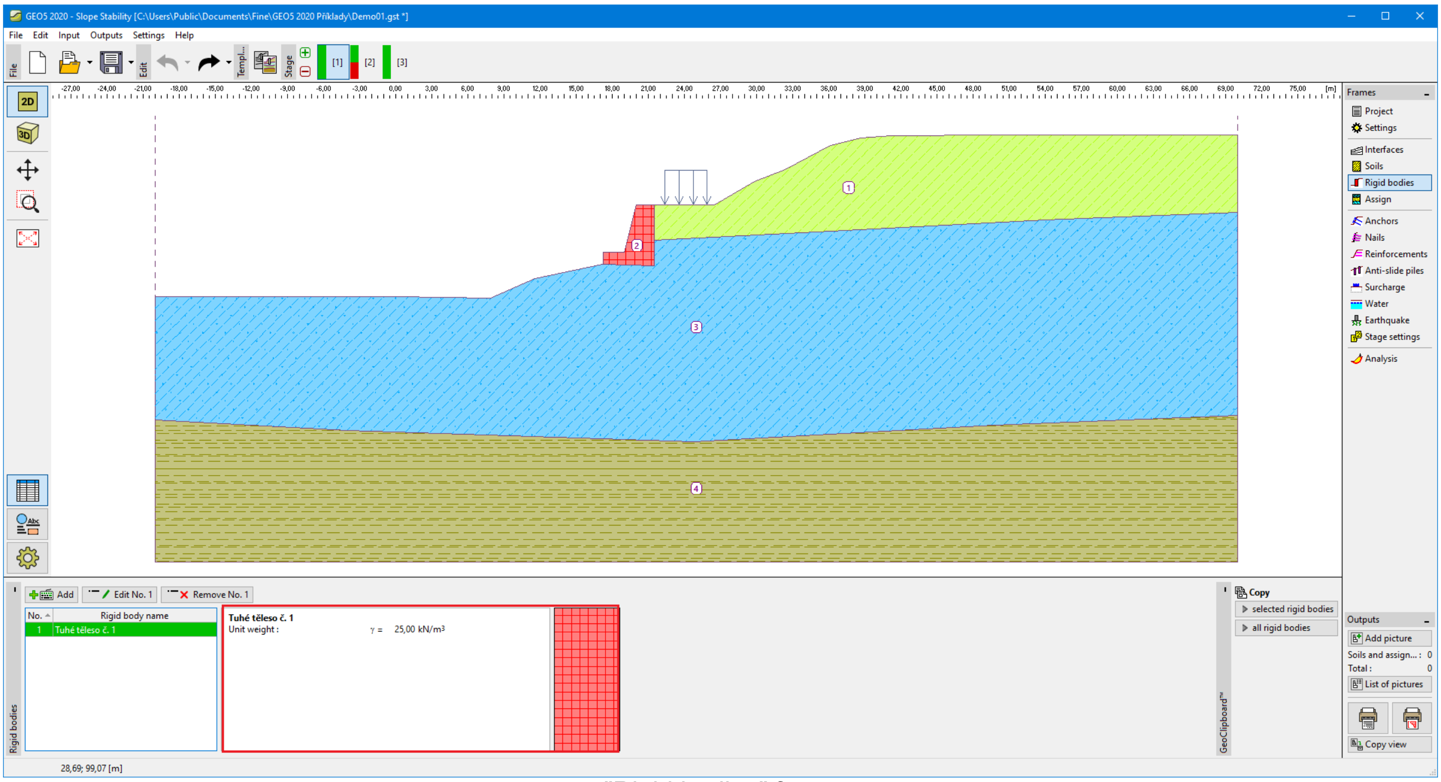
Rigid Bodies

The "**Rigid bodies**" frame contains a table with a list of rigid bodies. The rigid bodies serve to model regions with a high strength - e.g. **sheeting structures** or **rock subgrade**. This table also provides information about the currently selected rigid body displayed in the right part of the frame.

Adding rigid bodies is performed in the "**Add new rigid body**" dialog window. This window serves to input the unit weight of the rigid body material and to select color and pattern. The rigid bodies are in the "Assign" frame ordered after input soils.

Rigid bodies are introduced in the program as regions with a high strength, so they cannot be **intersected by a slip surface**. If we want the slip surface to intersect a rigid body (e.g. pile wall), it is recommended to model it as soil with a cohesion corresponding to the pile bearing capacity against slip.

Input rigid bodies can be copied within all 2D GEO5 programs using "GeoClipboard".



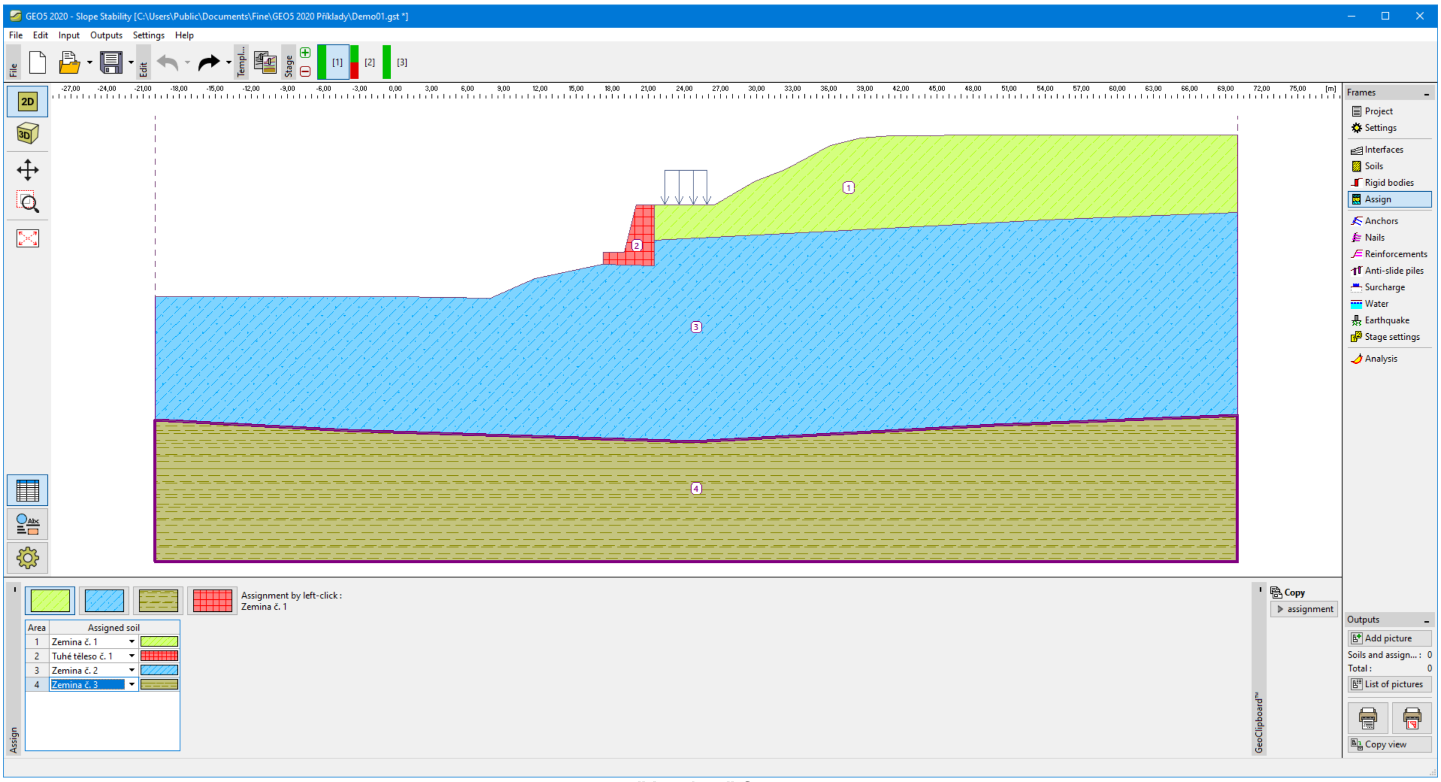
"Rigid bodies" frame

Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table or is accessible from the combo list for each layer of the profile.

The procedure to assign a soil into a layer is described in detail here.

Assignment of soils can be copied within all 2D GEO5 programs using "GeoClipboard".



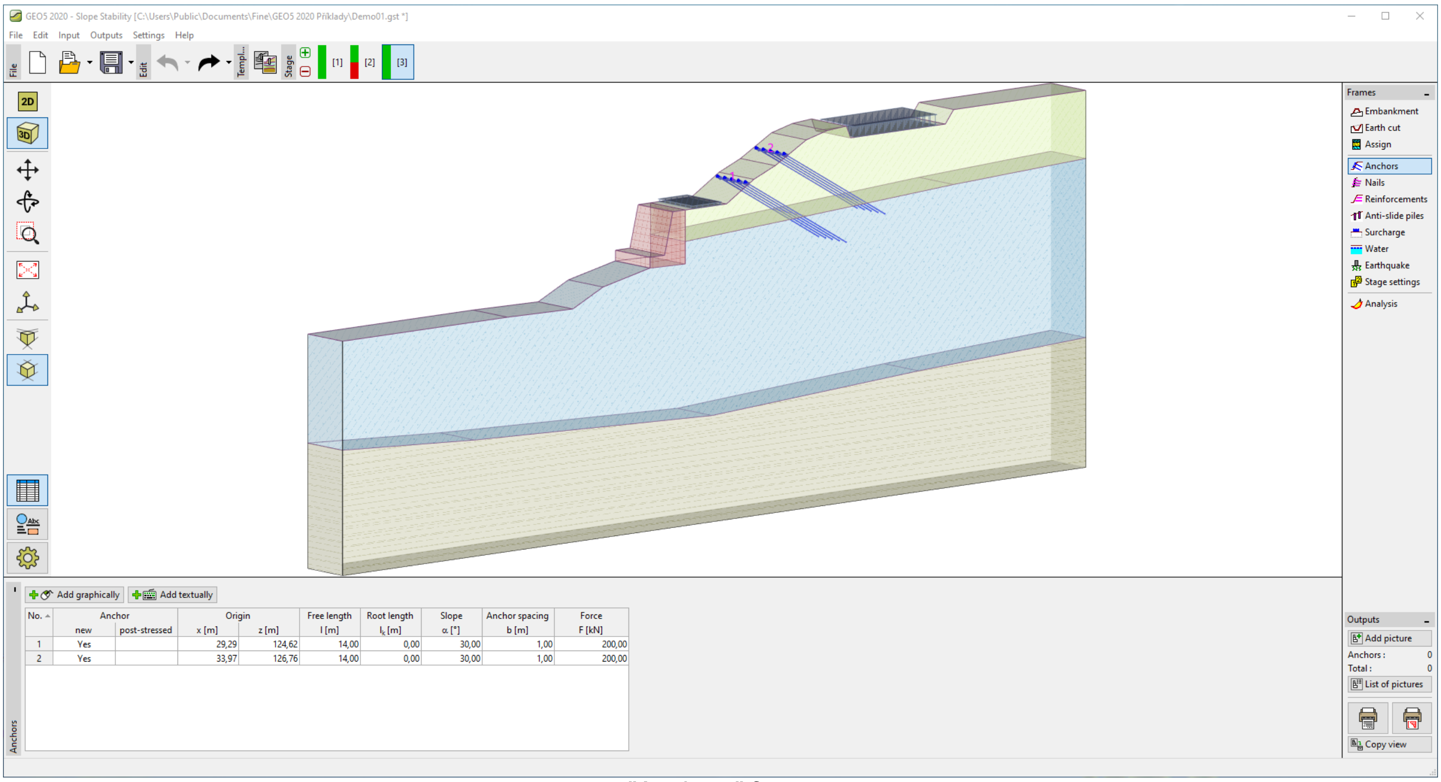
"Assign" frame

Anchors

The "**Anchors**" frame contains a table with a list of input anchors. Adding anchors is performed in the "**New anchors**" dialog window. The input anchors can be edited on the desktop with the help of active objects.

The dialog window serves to input the location of the anchor and its parameters. The starting point of the anchor is always **attached to the terrain**. Basic parameters can be modified only in the stage of construction, in which the anchor was introduced. In subsequent stages, the program allows us to modify the magnitude of an anchor pre-stress force.

The influence of anchors on the analysis is described in more detail in the theoretical part of the help.

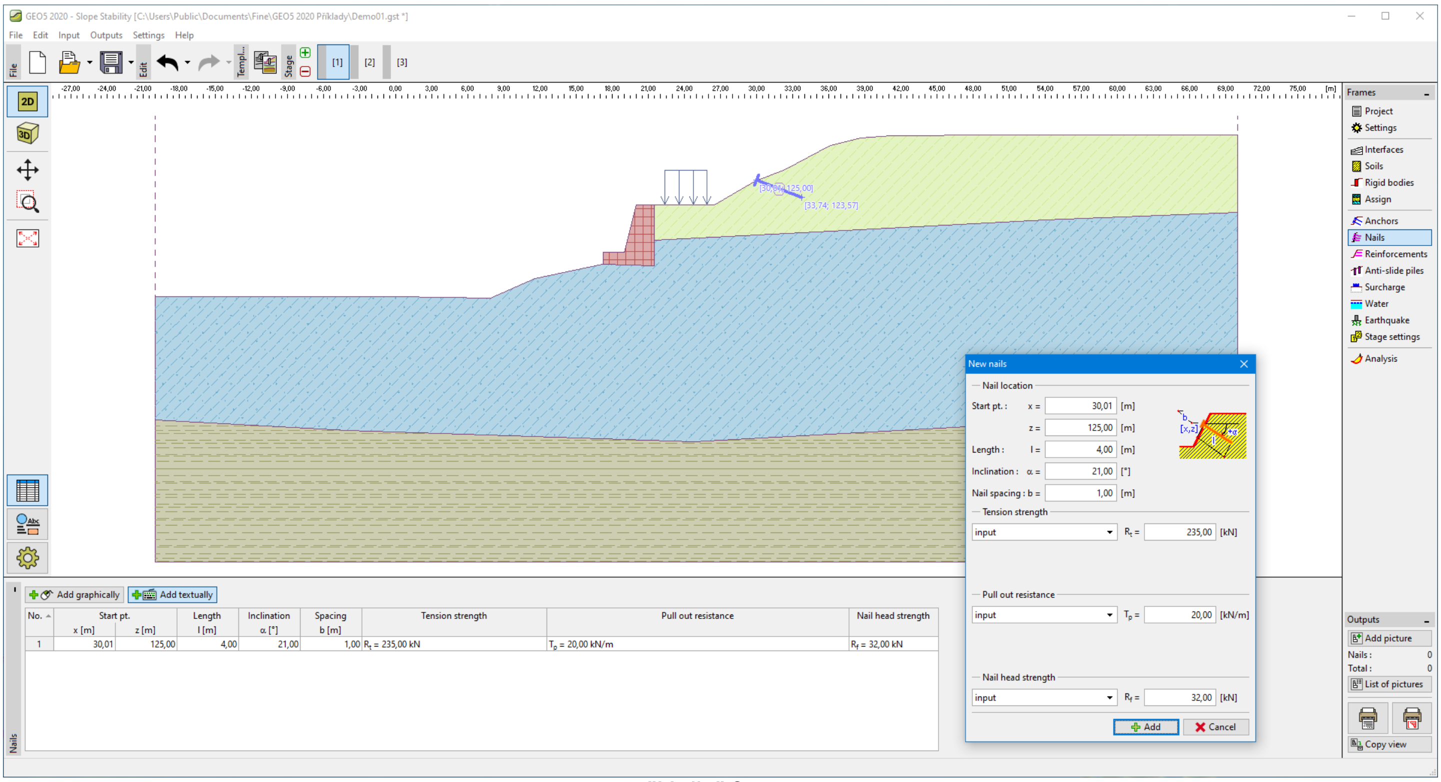


"Anchors" frame

Nails

The "**Nails**" frame contains a table with a list of input nails. Adding nails is performed in the "**New nails**" dialog window. The input nails can be edited on the desktop with the help of active objects.

The dialog window serves to input the location of nails (starting point x , z), its length l and inclination of a nail α , spacing between nails b , tension strength R_t pull out resistance T_p and nail head strength R_f. The starting point of the nail is always **attached to the terrain (ground surface)**. All input parameters can be modified in the stage of construction, in which the nail was introduced.



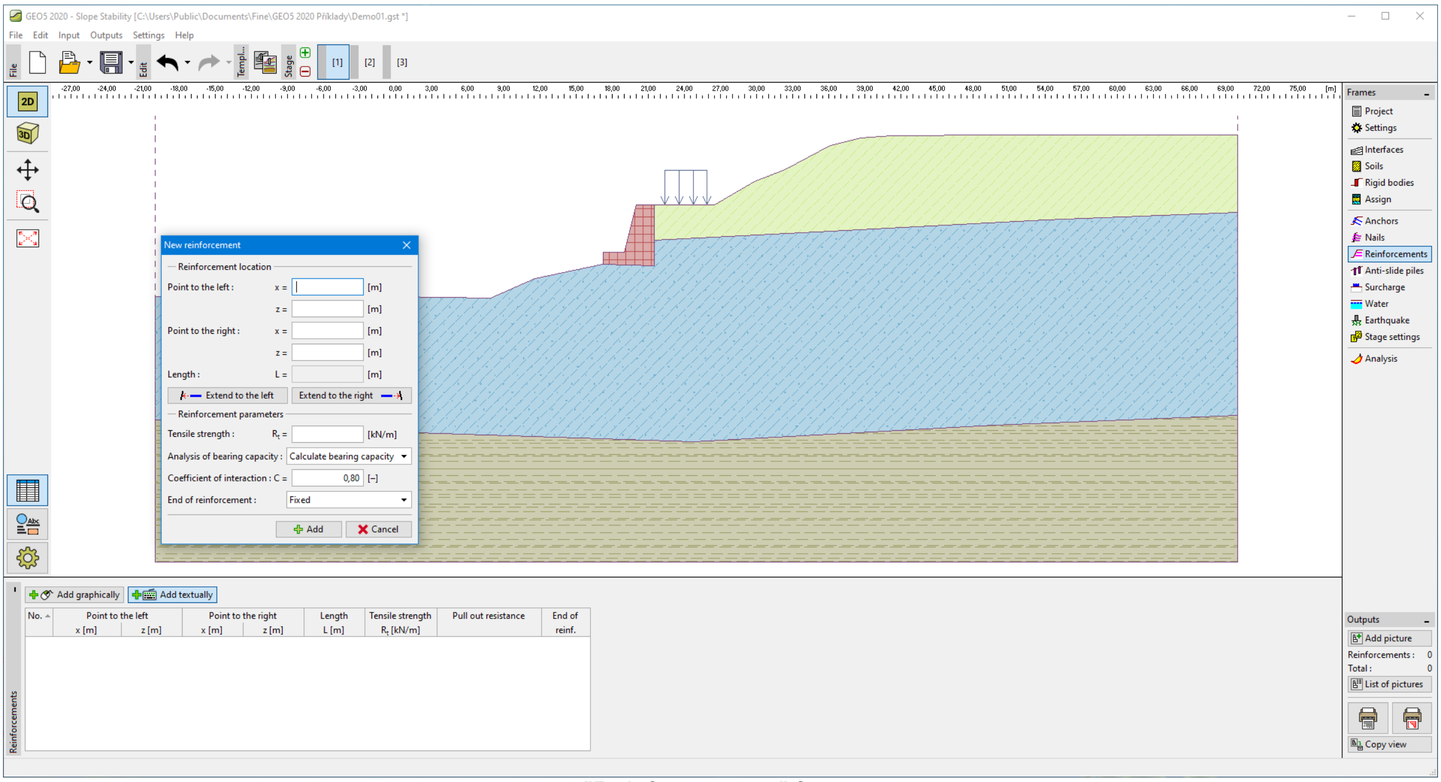
"Nails" frame

Reinforcements

The "**Reinforcements**" frame contains a table with a list of input reinforcements. Adding reinforcement is performed in the "**New reinforcement**" dialog window. The input reinforcements can be edited on the desktop with the help of active objects.

The dialog window serves to input the location of the reinforcement, anchorage length (from both left and right end), the tensile strength of reinforcement R_t and end of reinforcement (fixed or free). For the calculation of bearing capacity, one of three options in a combo list is selected: "**Calculate bearing capacity**" (coefficient of interaction C is defined), "**Input anchorage length**" l_k , or "**Input bearing capacity**" (Pull out resistance T_p is defined). All input parameters can be modified only in the stage of construction, in which the reinforcement was introduced. In subsequent stages, the geo-reinforcement can only be removed.

The influence of reinforcements in the analysis is described in more detail in the theoretical part of the help.



"Reinforcements" frame

Anti-Slide Piles

The "**Anti-slide piles**" frame contains a table with a list of input piles. Piles are input by one of two ways (the "**Add in dialog**" or "**Add graphically**" button).

The "**New pile**", or "**Modify pile properties**" dialog window serves to input the **location of the pile**.

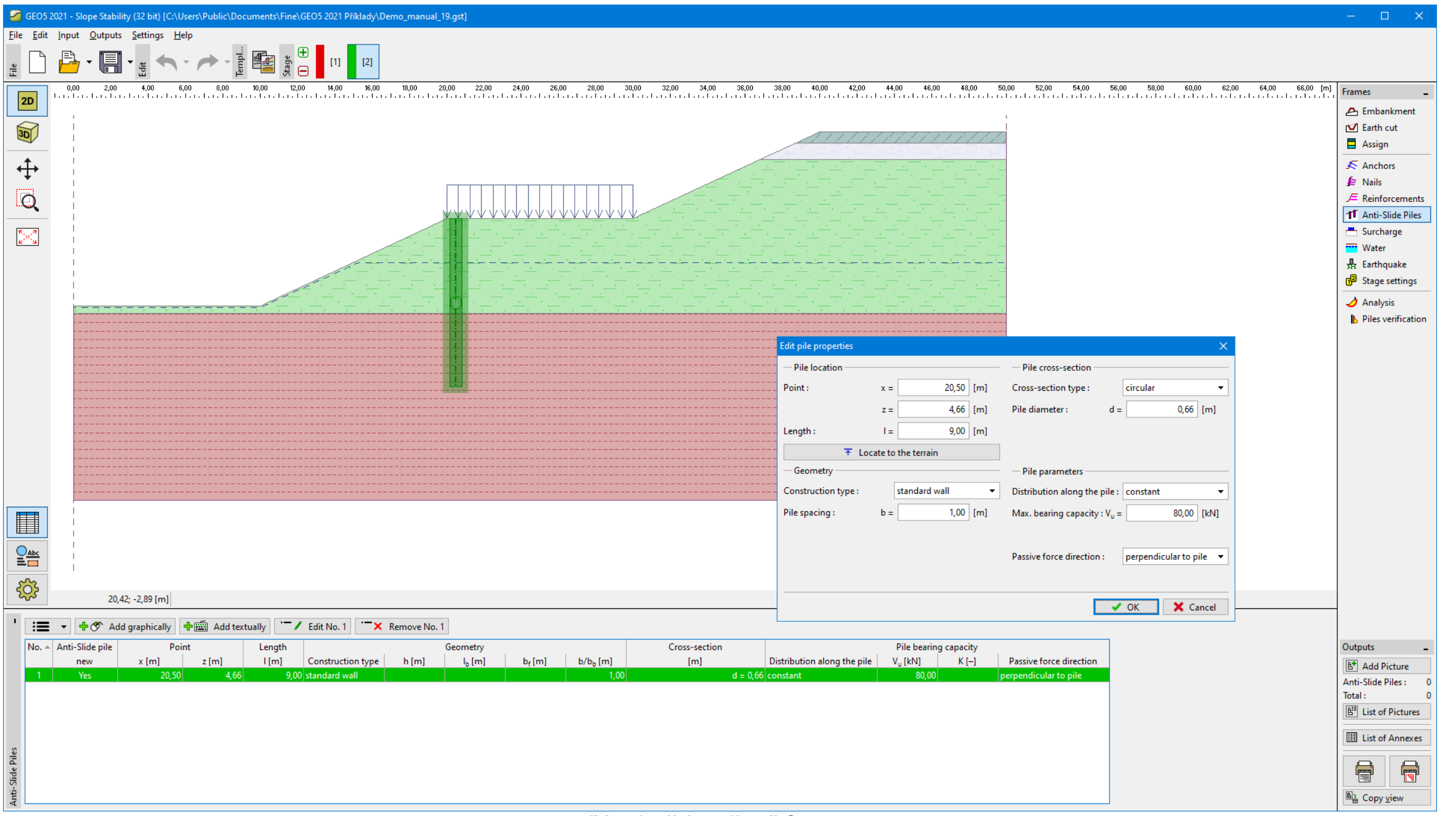
The "**Locate to the terrain**" button places the starting point of the pile head on the ground surface. Sometimes, an anti-slide pile can be located in mass directly (in this case, it's possible to analyze slope stability, but it's not possible to run the "**Anti-Slide Pile**" program).

Next, the **structure type** is selected:

- **standard wall -** the pile spacing is entered
- **double-row pile wall -** the pile spacing of front and back row, position, and length of connection beam are entered.

Then, the **pile cross-section** (circle - diameter of pile d , rectangle - dimensions s_x , s_y) and the **pile parameters** are specified - distribution of bearing capacity along the pile length (linear, constant), maximum bearing capacity V_u , gradient K and direction of passive force (perpendicular to the pile, parallel to slip surface). All input parameters can be modified in the stage of construction, in which the anti-slide pile was introduced. In subsequent stages the anti-slide pile can only be removed.

The influence of anti-slide piles on the slope stability assessment is described in more detail in the theoretical part of the help. Other calculations of the anti-slide piles (analysis of internal forces, dimensioning of reinforcement of piles) are based on the analysis of active and passive forces in the "**Anti-Slide Pile**" program.



"Anti-slide piles" frame

Surcharge

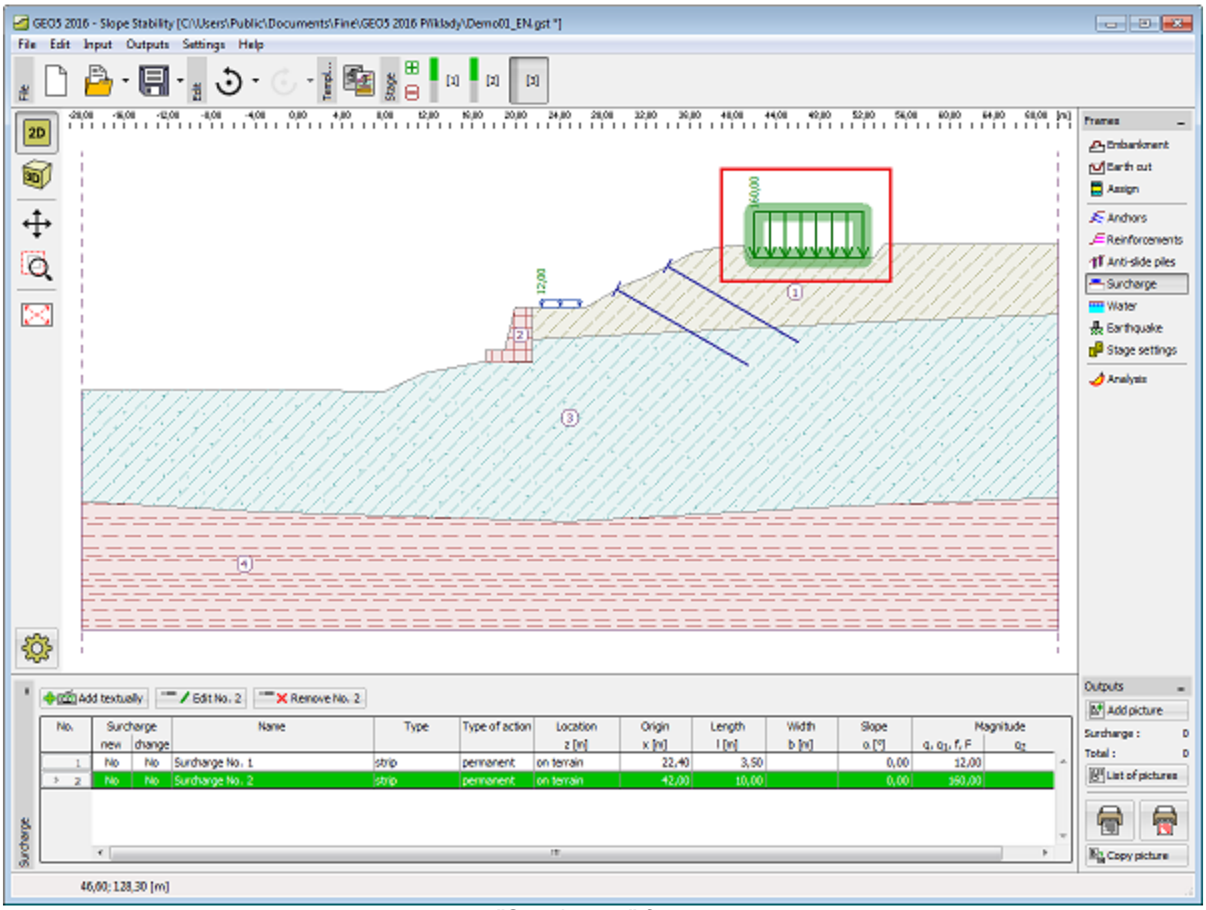
The "**Surcharge**" frame contains a table with a list of input surcharges. Adding surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active objects.

All input parameters of a surcharge can be modified in the construction stage where the surcharge was specified. Only the surcharge magnitude can be modified in all subsequent construction stages (the "**Adjust surcharge**" option).

Either **permanent**, **variable** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient. Accidental surcharge with a favorable effect is not considered in the analysis.

The Influence of surcharge on the slope stability analysis is described in the theoretical part of the help.

In this frame, it is also possible to input a special type of surcharge - the water above the terrain.



"Surcharge" frame

Water

The "**Water**" frame serves to set the type of a groundwater table. Six options to specify the type of water are available from the combo list.

Inputting the ground water table or isolines, respectively, is identical with the standard input of interfaces.

A value of coefficient R_u or pore pressure appears next to the interface in the table if introducing water using isolines of R_u - **interfaces** or **pore pressure**, respectively. Pressing the button with a blue arrow next to the input field opens the "**Coefficient Ru**" or "**Pore pressure**" dialog window to enter the desired value. It is advantageous to input all values at once using the "**OK+↑**" and "**OK+↓**". The value of a given quantity found in a specific point between two isolines is approximated by **linear interpolation** of values pertinent to given isolines. For the "**Coefficient R_u**" option, the first (the most top one) is always **identical with the terrain** - it therefore cannot be deleted.

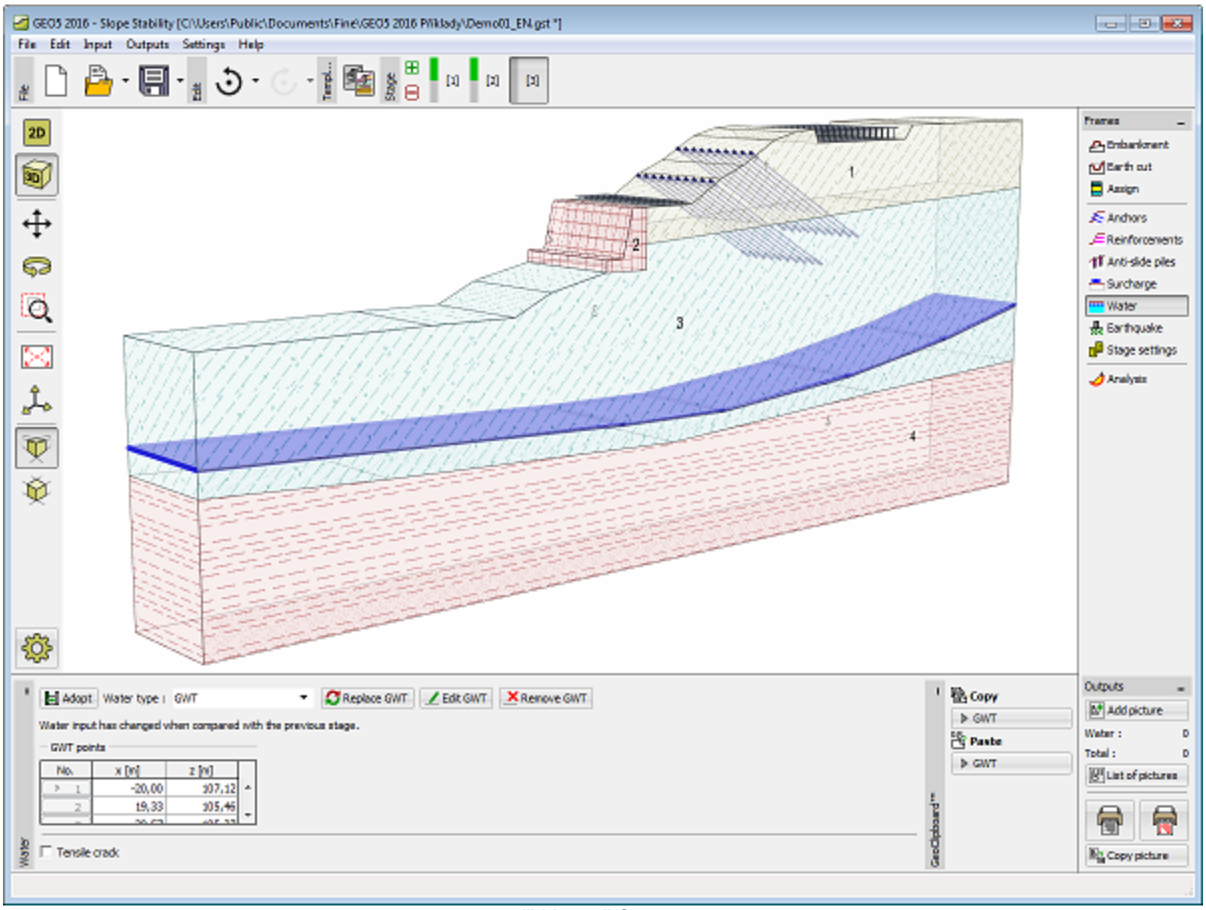
The **ground water table** (resp. **table of suction** or **original GWT**) is specified as continuous interfaces, which can be located even above the terrain.

Numerical water flow analysis is available for the users with the "**Slope Stability - Water Flow**" module purchased.

If the input data in individual stages are different, the program then allows for accepting the data from the previous construction stage of construction by pressing the "**Adopt**" button.

The program further allows for specifying a depth of tensile cracks filled with water.

Input interfaces of water can be copied within all 2D GEO5 programs using "GeoClipboard".



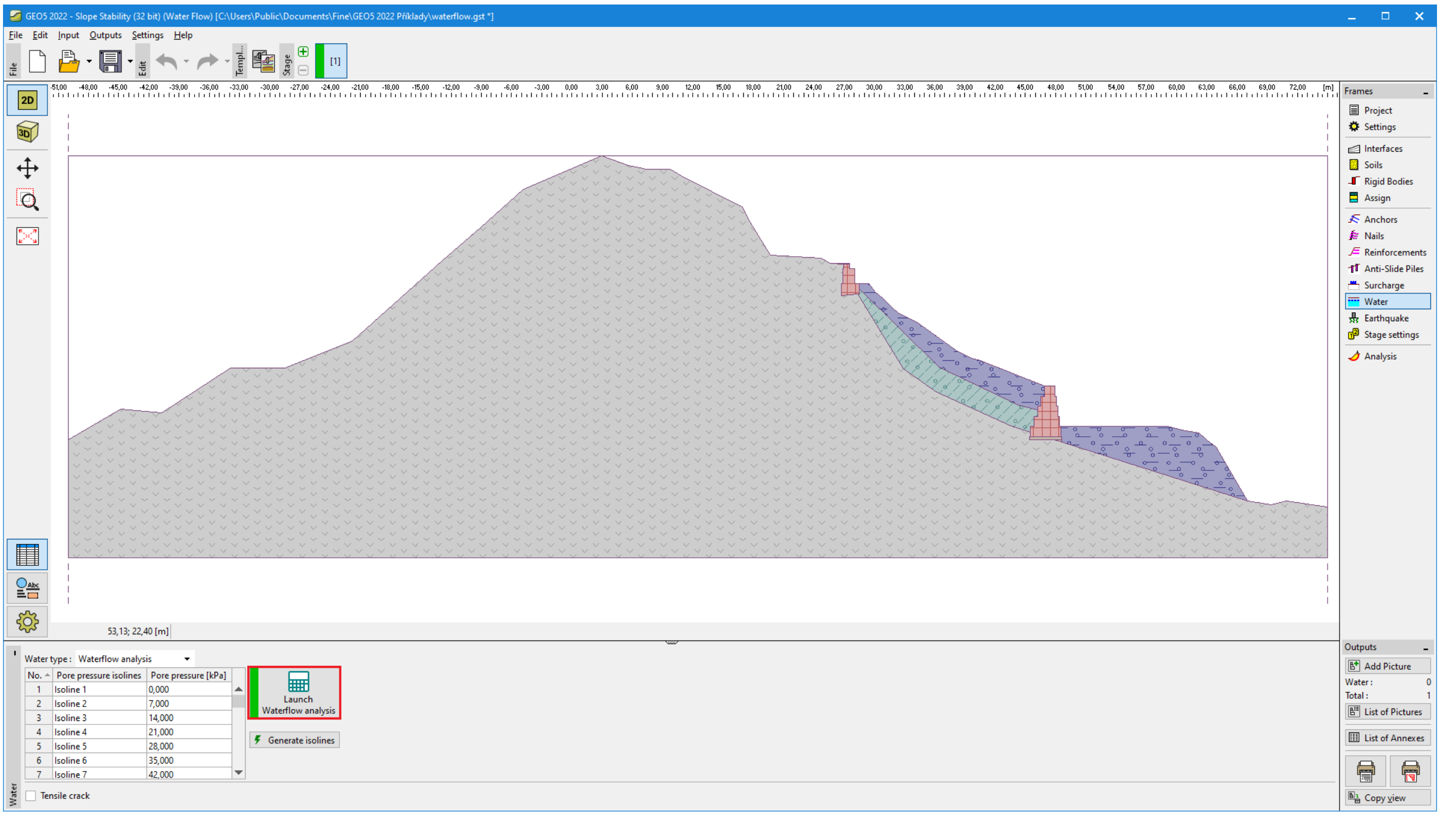
"Water" frame

The water which does not cross into the soil body at all (e.g. water tanks and reservoirs with impermeable membranes) must be modeled as a surcharge.

Water Flow Analysis

Numerical water flow analysis is available for the users with the "**Slope Stability - Water Flow**" module purchased.

The analysis is launched using the "**Launch Waterflow analysis**" button.

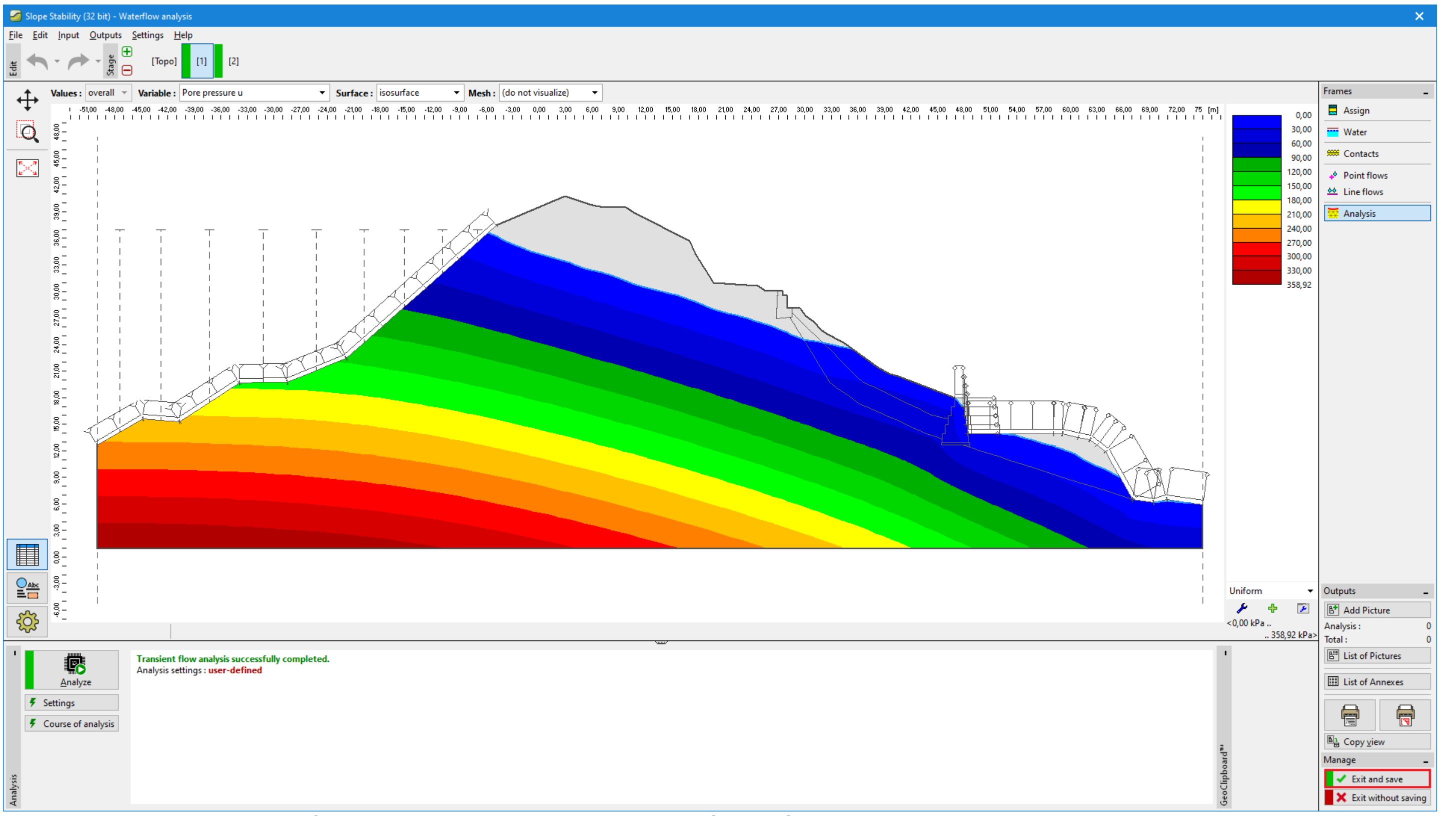


Launching of numerical Waterflow analysis

After pressing the button, the "**Slope Stability - Water Flow**" module is launched. In this module, it is necessary to generate FE mesh and **define all necessary inputs**:

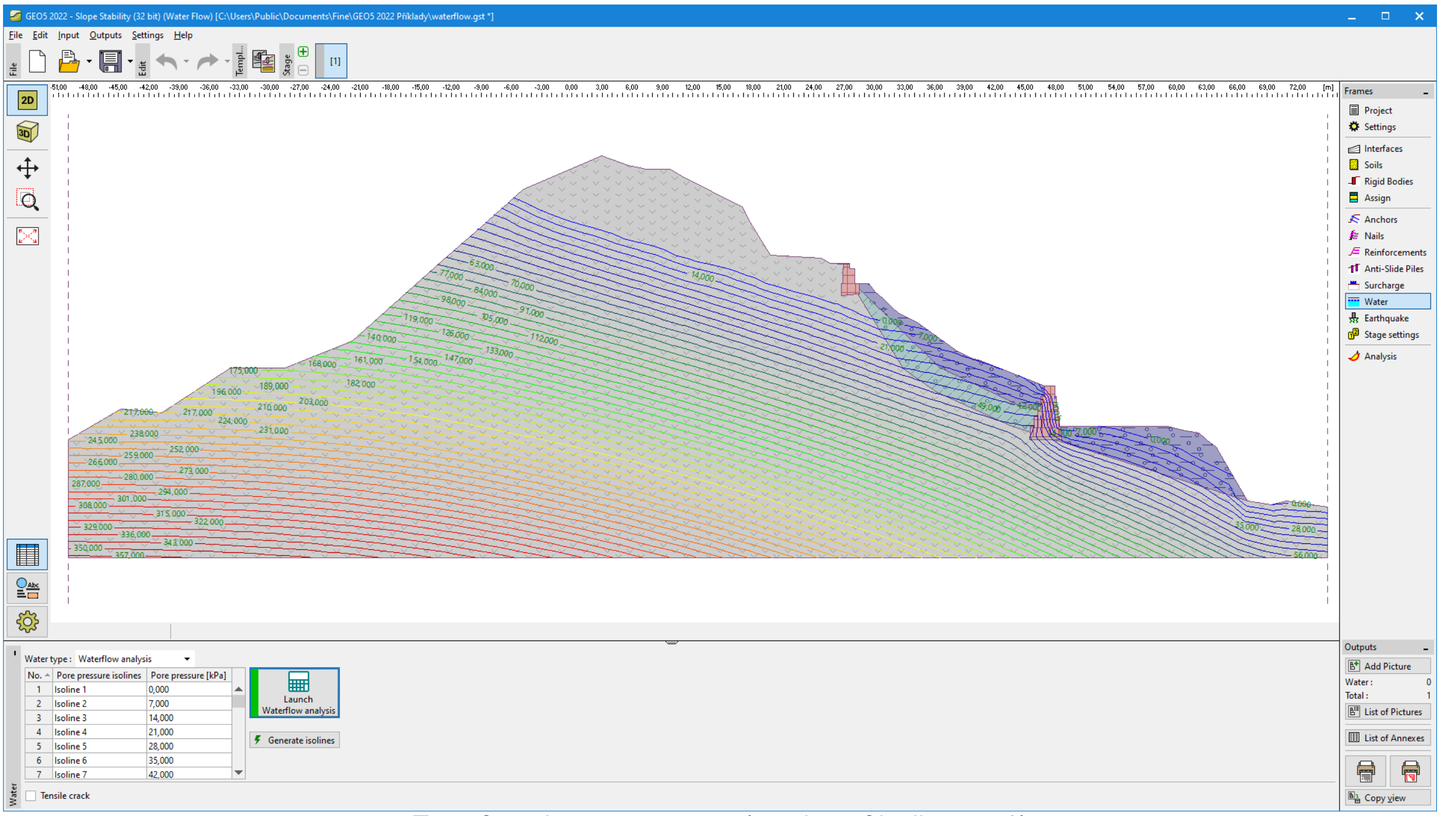
- coefficients of permeability
- parameters of flow material models
- flow boundary conditions (line, point)

The numerical calculation of the water flow results in a **pore pressure distribution** in the whole task.



Calculated pore pressures in the "Slope Stability - Water Flow" module

Using the "**Exit and save**" button, the pore pressures are transferred back to the "Slope Stability" program. The number of isolines can be changed using the "**Generate isolines**" button.



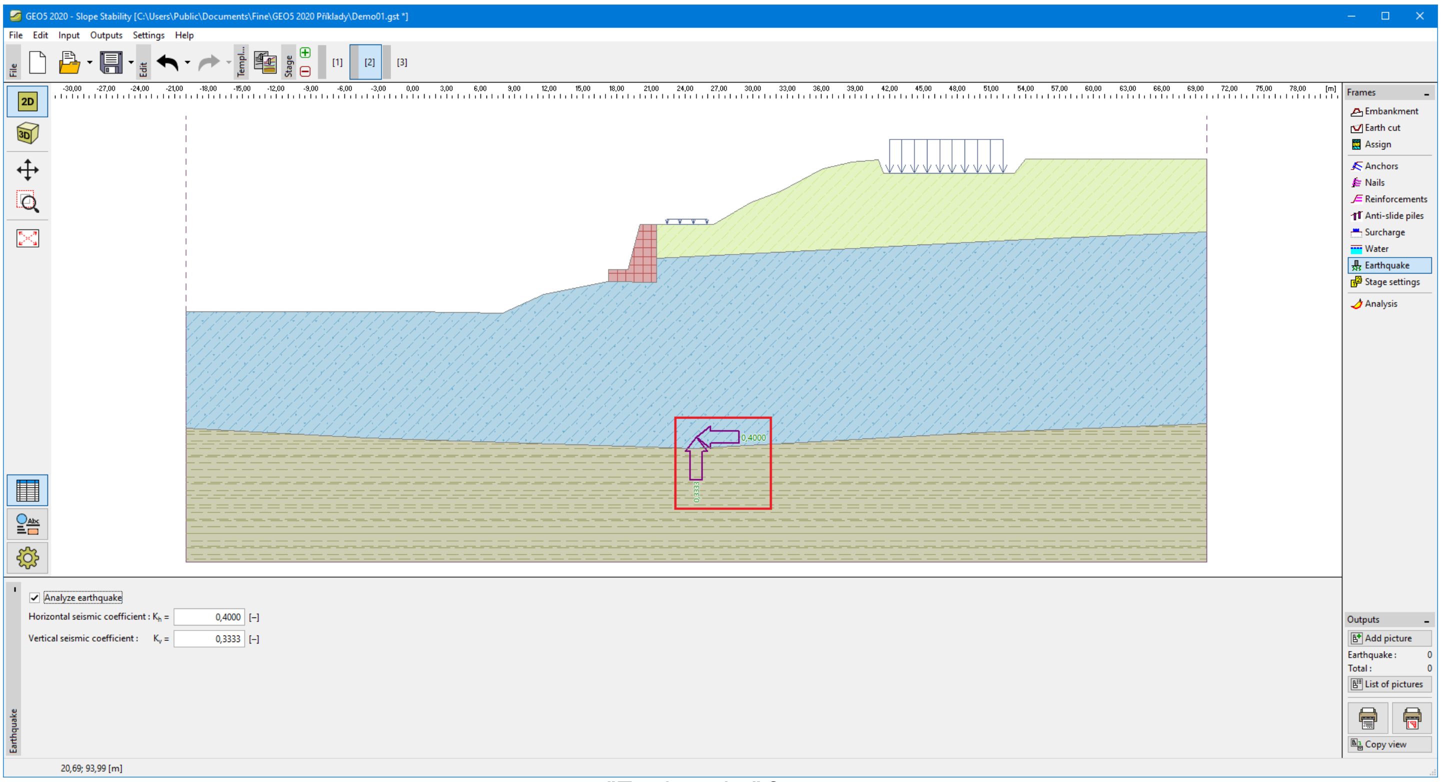
Transferred pore pressures (number of isolines - 50)

Earthquake

The "**Earthquake**" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements, the coefficients K_h and K_v can be calculated following the approach adopted from EN 1998-5.

The slope stability analysis with the earthquake influence is described in the theoretical part of the help, the "Influence of earthquake" chapter.



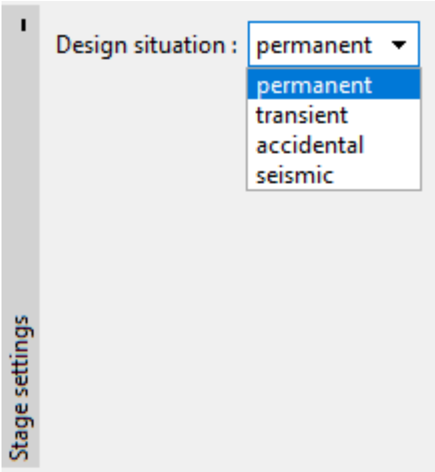
"Earthquake" frame

Stage Settings

The "**Stage settings**" frame serves to input settings valid for a given construction stage.

The selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology.



"Stage settings" frame

Analysis

The "**Analysis**" frame displays the analysis results. Several analyses can be performed for a single task.

The starting point in the slope stability analysis is the selection of the type of slip surface. The input is available from the combo list in the left top part of the frame containing two options - **circular slip surface** and **polygonal slip surface**. After introducing the slip surface, the analysis is started using the "**Analyze**" button. The analysis results appear in the right part of the frame.

The type of analysis is selected in the midsection of the frame - seven methods are available for the circular slip surface (Fellenius/Petterson, Bishop, Spencer, Janbu, Morgenstern-Price, Shahunyants or ITFM), and six methods are available for the polygonal slip surface (Sarma, Spencer, Janbu, Morgenstern-Price, Shahunyants, or ITFM). For both cases of the assumed slip surfaces it is possible to perform the analysis employing all methods at once (in such a case, however, the slip surface cannot be optimized).

The actual verification of slope stability can be performed, depending on the settings in the "Stability analysis" tab:

- Verification according to EN 1997, where the load is reduced by the partial factors and the verification is performed according to the **theory of limit states**.
- Verification according to the factor of safety
- Verification according to the theory of limit states
- Verification according to DIN1054

In the combo list the **analysis type** is selected:

- Standard
- Optimization (for circular or polygonal slip surface)
- Grid search (only for circular slip surface)

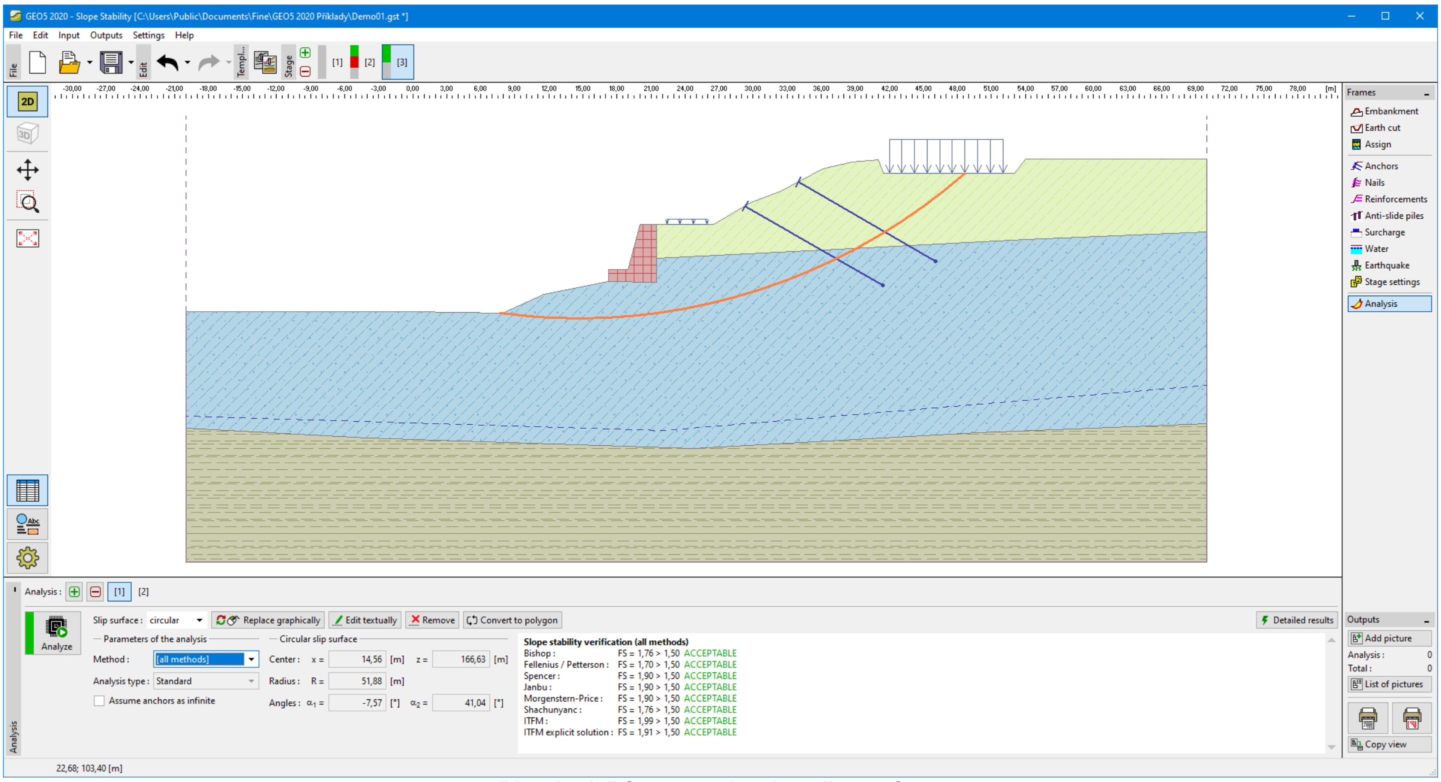
Choosing the "**Optimization**" option activates the "**Restrictions**" button - pressing this button changes the frame appearance and makes it possible to introduce restrictions on the optimization procedure.

If the "enter shear parameters of soil on the slip surface" option is selected in the "Settings" frame, the "**Parameters**" button will be displayed.

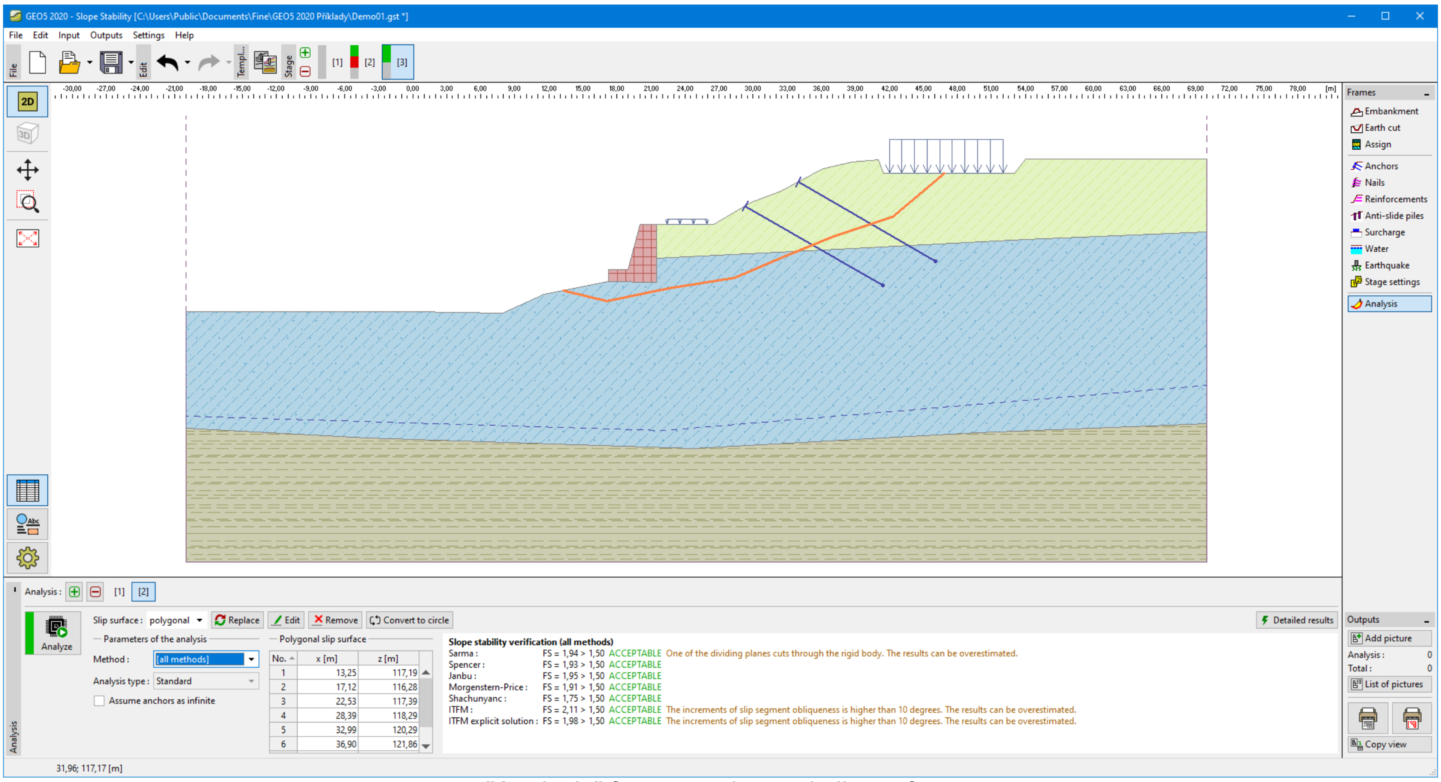
It is also possible to specify how to deal with anchors in the analysis ("**Assume anchors as infinite**" box).

The slip surface, even the optimized one, must be introduced in the frame.

The analysis results appear in the left part of the frame and the optimized slip surface on the desktop. Visualization of results can be adjusted in the frame "Drawing Setting



"Analysis" frame - circular slip surface



"Analysis" frame - polygonal slip surface

Input of Slip Surface

Select the slip surface type from the list (circular, polygonal). It is possible to input the slip surface according to its type in several ways:

- **Circular slip surface**

Graphically - after pressing the [Input graphically] button, input three points that define the circular slip surface using the left mouse button.
It is possible to change the input slip surface in the dialog window "**Circular slip surface**" after pressing the [Edit textually] button, or replace the slip surface using the mouse button by pressing [Replace graphically].

The [Cancel input] button cancels the slip surface input.

The [Convert to polygon] button converts the circular slip surface to polygonal.

The [Remove] button removes the slip surface.

Textually - by pressing the [Input textually] button, the dialog window "**Circular slip surface**" is opened, and x , y coordinates, and the diameter is input.

- **Polygonal slip surface**

  Graphically - by pressing the [Input] button on the toolbar, the slip surface input mode is turned on - while adding, the process is the same as with interface input.

  Textually - press the [Input] button and using the button above the [Add points textually] table, the "**New points**" dialog window is opened. In the dialog window, the points of the slip surface are added using the x , y coordinates.

  The [Convert to circle] button converts polygonal slip surface into circle slip surface.
  The functions of the rest of the buttons are the same as with the circular slip surface.

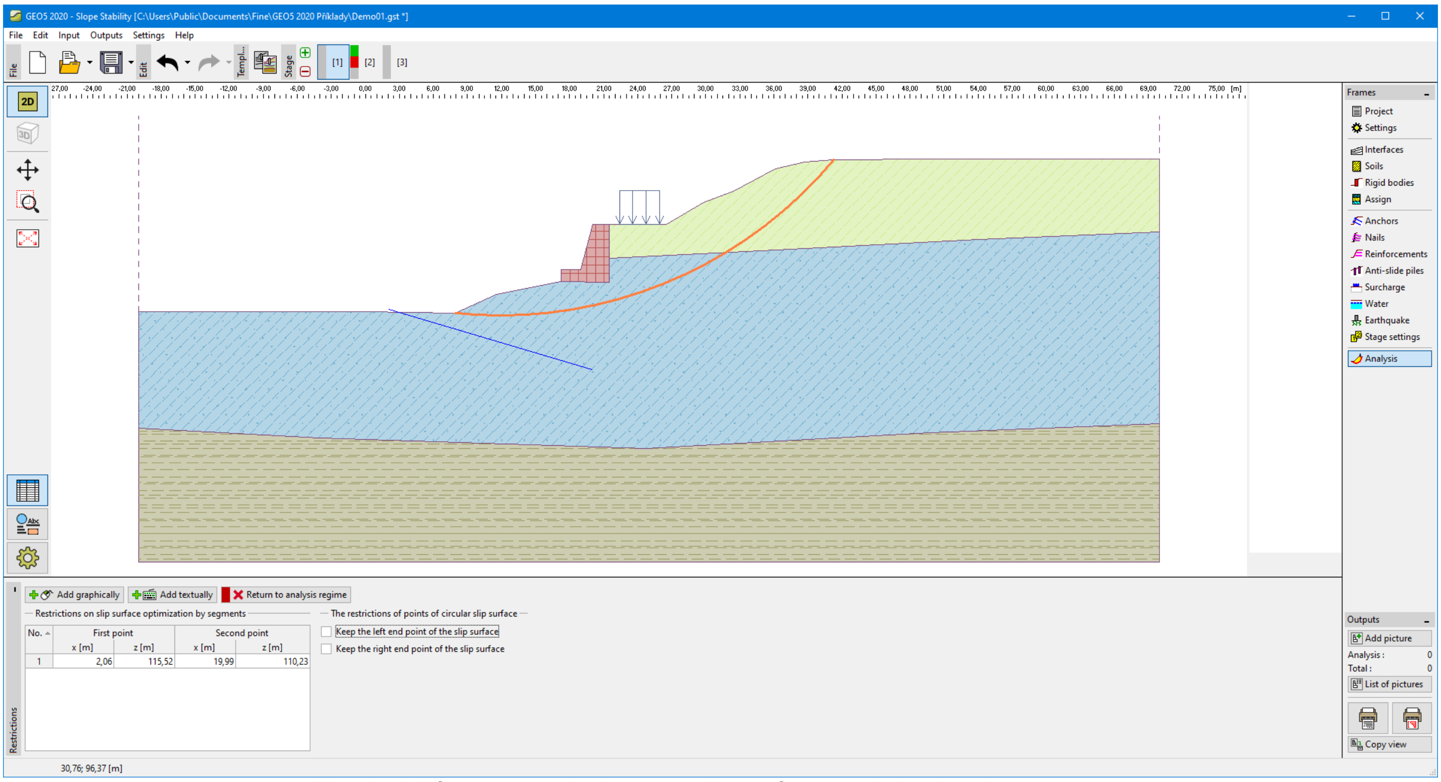
Restrictions on the Optimization Procedure

The "**Analysis**" frame allows (after pressing the "**Restrictions**") for specifying restrictions on the optimization process.

Regardless of the assumed type of slip surface (circular, polygonal), it is possible to introduce into the soil body (with the help of mouse) segments, which should not be crossed by the optimized slip surface. These segments also appear in the table in the left part of the frame.

Polygonal slip surface also allows for excluding some points from optimization, either entirely or partially, only in a specified direction. "**Keeping the point fixed**" during the optimization process is achieved by checking the box in the table with the corresponding point.

This input mode is quitted by pressing the red button "**Return to analysis**".



"Analysis" frame- restrictions on slip surface optimization by segments

Parameters on Slip Surface Segments

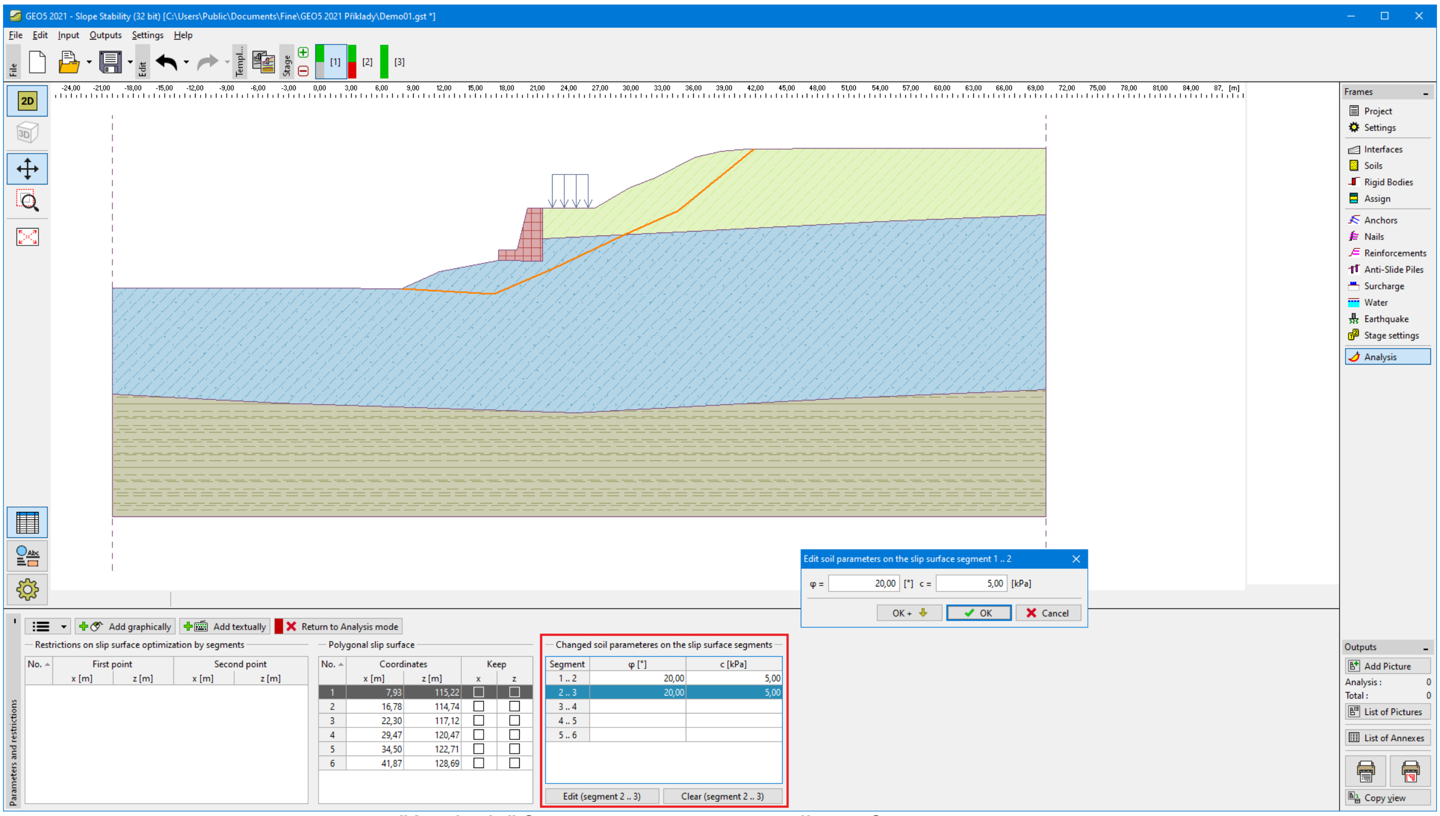
In the "**Analysis**" frame, it is possible (after pressing the "**Parameters**" button) to enter shear properties on the polygonal slip surface segments.

The table on the left side of the frame contains slip surface segments where the **angle of internal friction** φ and **cohesion** c can be entered.

If the values of φ and c are not defined on the segment, their values are taken from the soils in the profile.

These parameters can also be entered in combination with restriction on the optimization procedure. Segments with modified parameters are **held in the same position** during the optimization process.

This input mode is quitted by pressing the "**Return to** analysis" button.

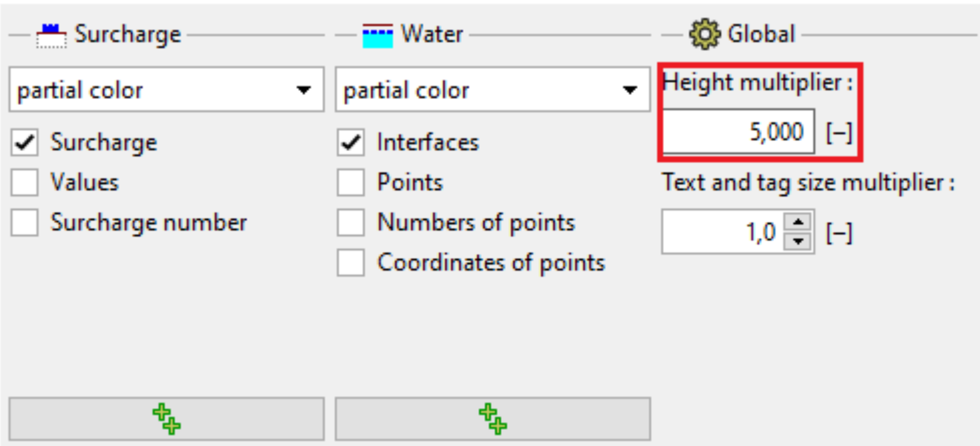


"Analysis" frame- parameters on slip surface segments

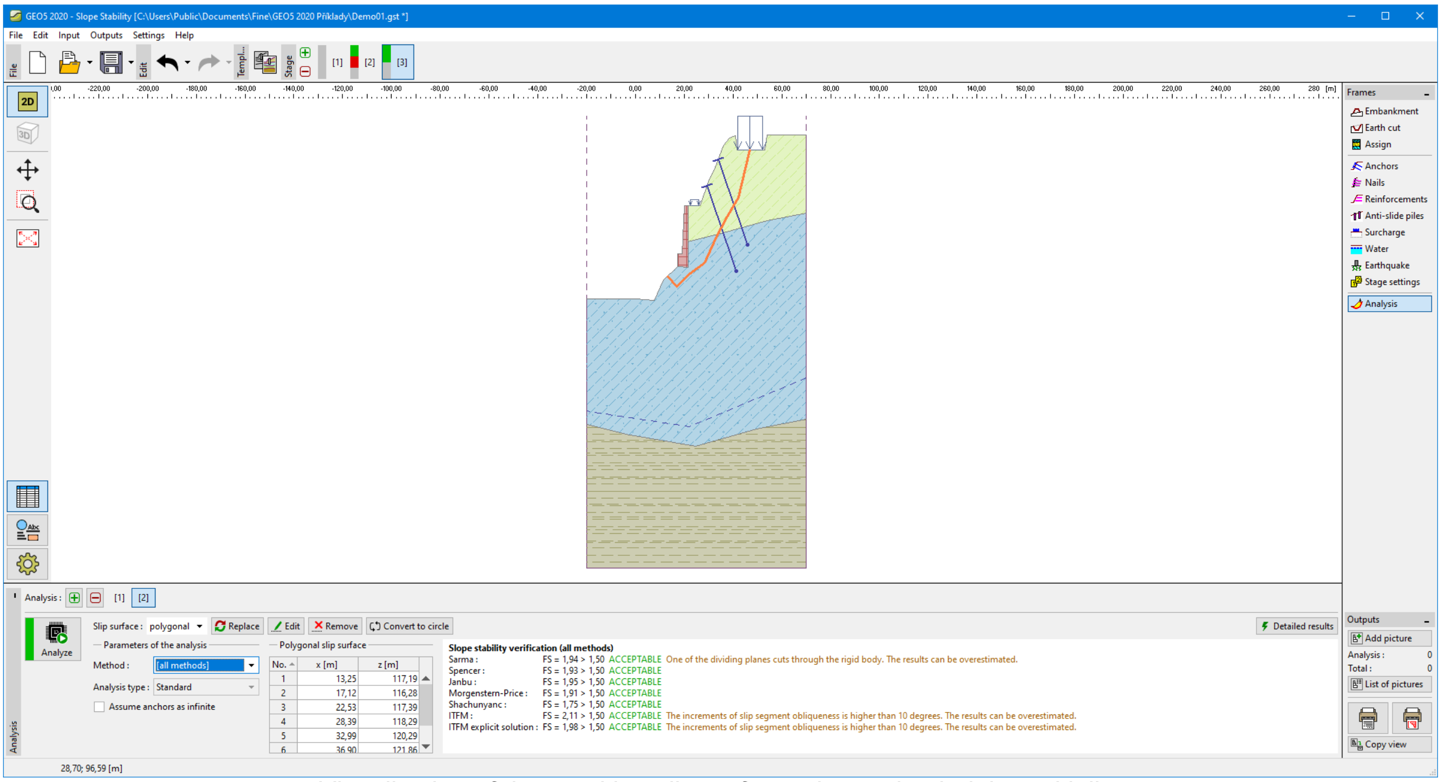
Height Multiplier

Providing the analyzed slope is too long or has a small height; the plotted slip surface might not be sufficiently visible. This problem can be solved by the selected courser scale in the vertical direction with the help of a height multiplier. The value of this multiplier is set in "Drawing Settings", "**Global 2D**" tab. Using standard setting ("**Height multiplier**" equal to one) plots undistorted structure proportional to its dimensions.

Only polygonal slip surface can be input graphically when exploiting the height multiplier option. The circular slip surface must be in such a case input manually in the "**Circular slip surface**" dialog window using the "**Input**" button.



Setting height multiplier



Visualization of the resulting slip surface when using height multiplier

Program Rock Stability

This program is used to analyze the stability of rock slopes and walls for a specified type of failure, including a planar or polygonal slip surface or rock wedge.

The help in the program "Rock Stability" includes the following topics:

- The input of data into individual frames: Project, Settings

Plane and polygonal slip surface:	Terrain	Rock	Slip Surface (Plane)	Slip Surface (Polygonal)	Water (Plane Slip Surface)	Water (Polygonal Slip Surface)
Surcharge	Applied Forces	Anchors	Earthquake	Stage Settings	Analysis (Plane Slip Surface)	Analysis (Polygonal Slip Surface)

Rock Wedge:	Geometry	Slip Surface	Parameters	Water	Surcharge	Anchors
Earthquake	Stage Settings	Analysis				

- Standards and analysis methods

- Theory for analysis in the program "**Rock Stability**":

  Rock Stability

- Outputs

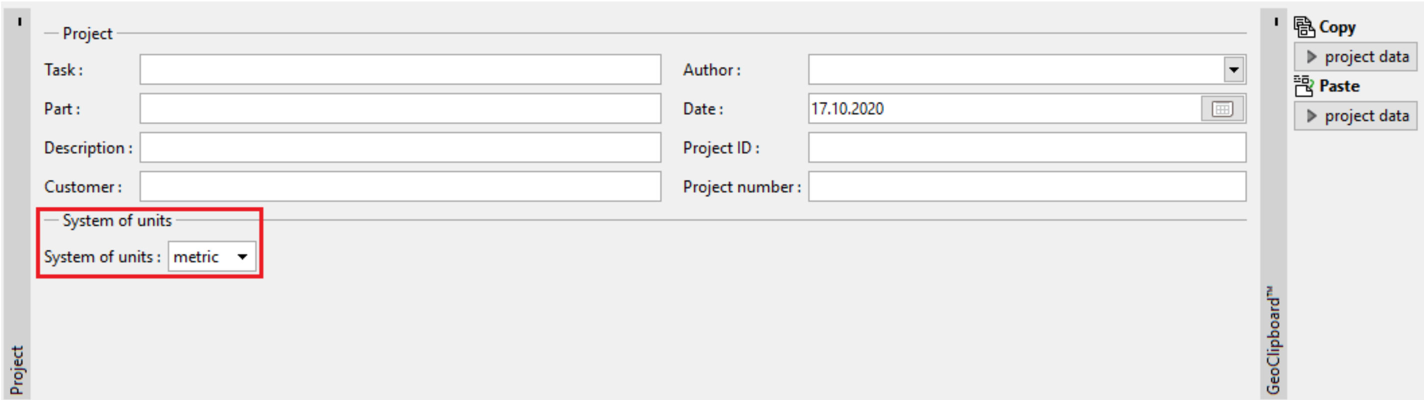
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The frame **"Project"** is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The frame "**Settings**" allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

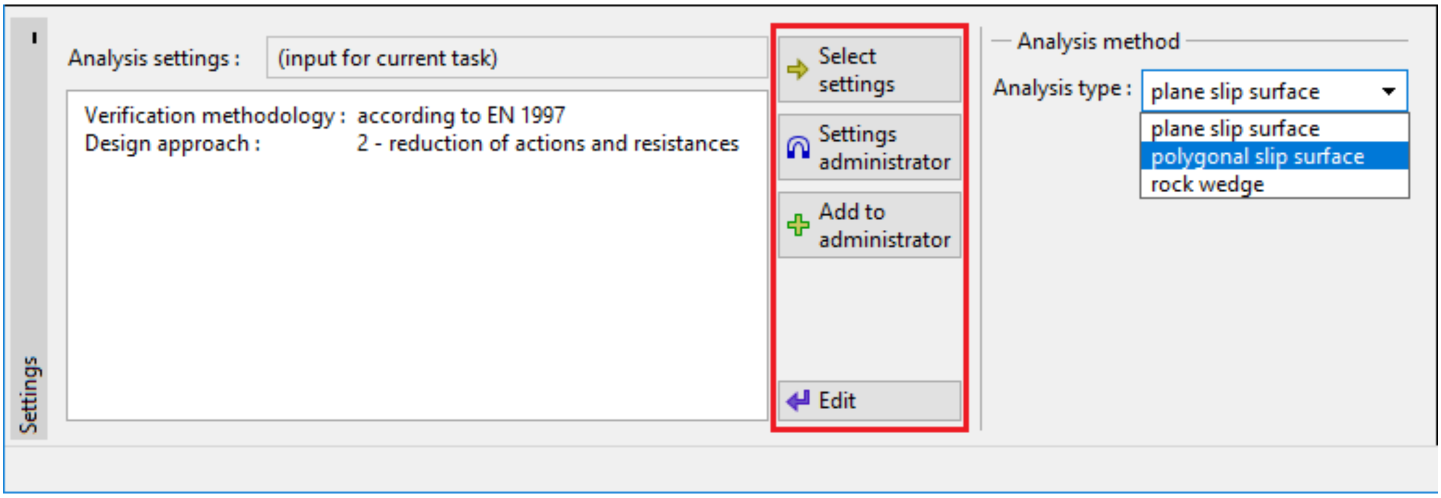
The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting.** Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

Settings of analysis parameters are performed in the "Stability analysis" tab.

The frame also allows us to select the type of slip surface:

- Plane slip surface
- Polygonal slip surface
- Rock wedge



Frame "Settings"

Terrain - Plane and Polygonal Slip Surface

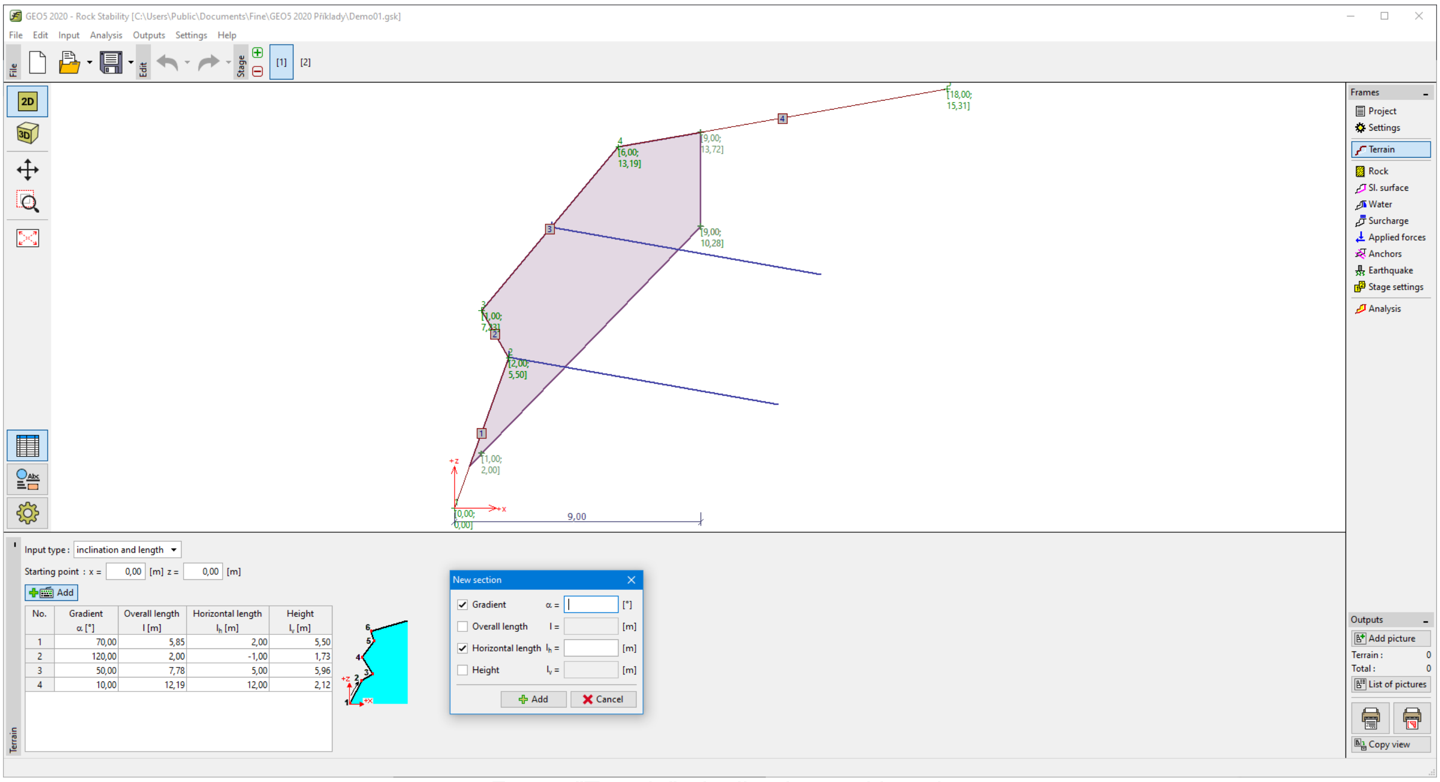
Two possibilities of terrain input are available it the frame **"Terrain"**. Terrain can be defined by sections of rock slope or it can be defined by coordinates X and Z. This frame contains a table with a list of defined sections of a rock slope or table with a list of coordinates of each terrain point. In the program, the slope is always oriented from **left to right.**

The **coordinates of the origin** - the first point of terrain followed by defined sections - are entered in the upper part of the frame if the method of definition of **sections of rock slope** (inclination and length) is used.

Adding section is performed in the **"New section"** dialog window. These sections can also be edited on the desktop with the help of active objects.

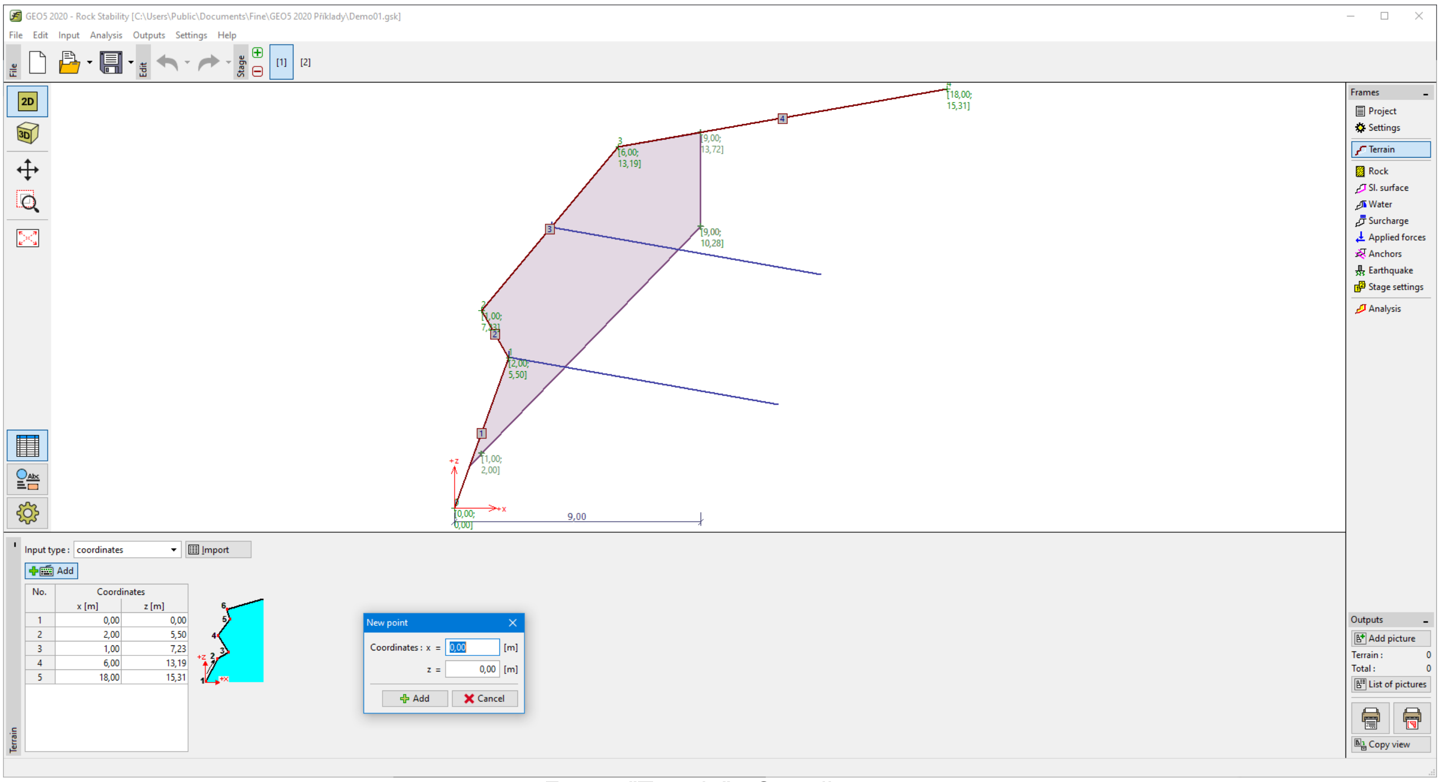
Each section can be defined by its dip, by the overall length of the section, by the horizontal length, or height of a section. Only **two selected values** are used while the others are determined by the program automatically (if more than two entry fields are checked than the input and computation are not carried out). Both vertical and horizontal sections, as well as overhangs, can be represented. Terrain must be defined from the bottom up.

In case of a proper input, the **program automatically plots the defined section** on the desktop using a dashed line so that before accepting it, it is possible to check whether the section is correctly defined.



Frame "Terrain" - Inclination and length

If you use the **coordinates** for definition of terrain, the surface of terrain must be defined from bottom up. Adding point is performed in the **"New point"** dialog window. These points can also be edited on the desktop with the help of active objects. Both vertical and horizontal sections, as well as overhangs, can be represented. The program allows importing points from table data.



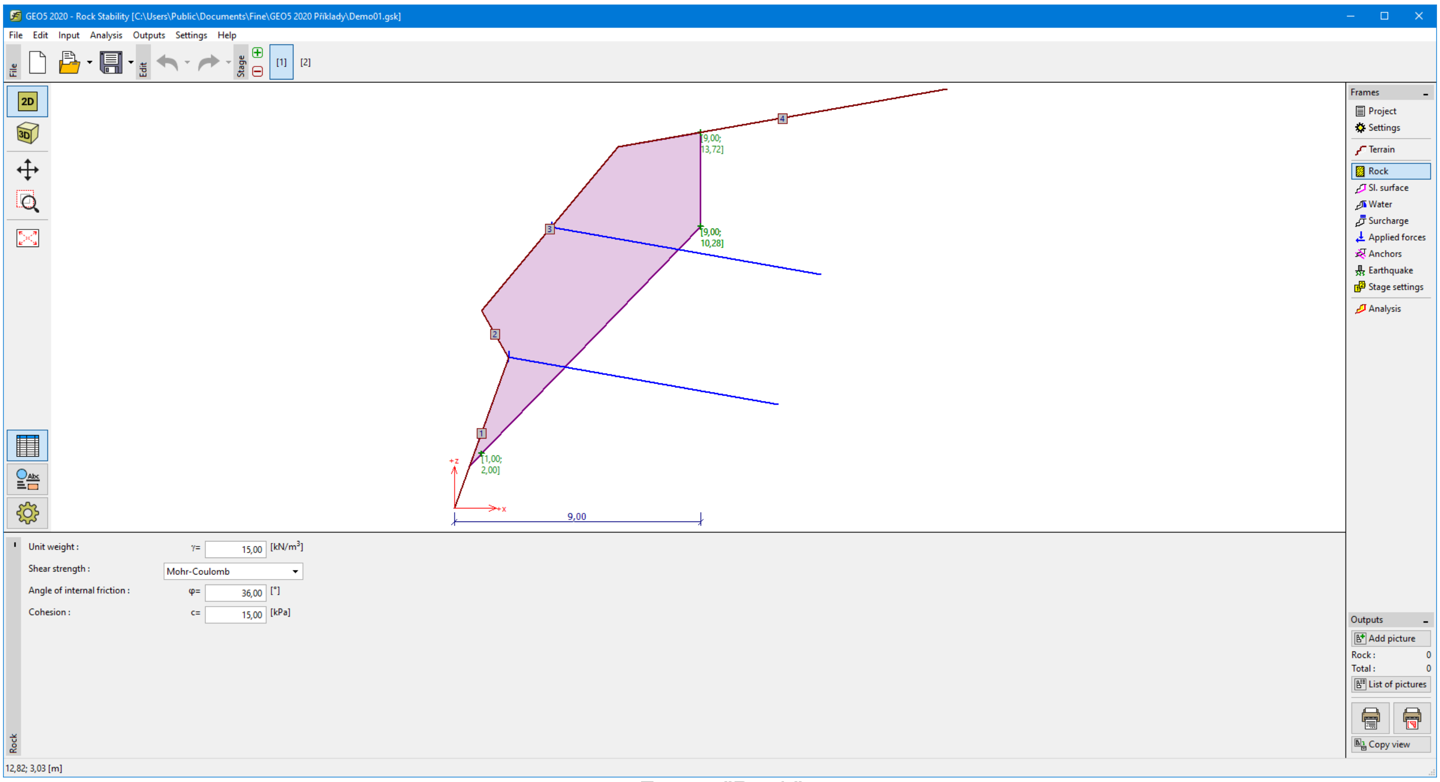
Frame "Terrain" - Coordinates

Rock

The frame "**Rock**" allows for entering the material parameters of a rock slope (depending on the type of shear strength), including the unit weight of a rock. Three types of shear strengths on a slip surface are available in the program:

- Mohr - Coulomb
- Barton - Bandis
- Hoek - Brown

Material parameters of rock are then entered based on the selected method.



Frame "Rock"

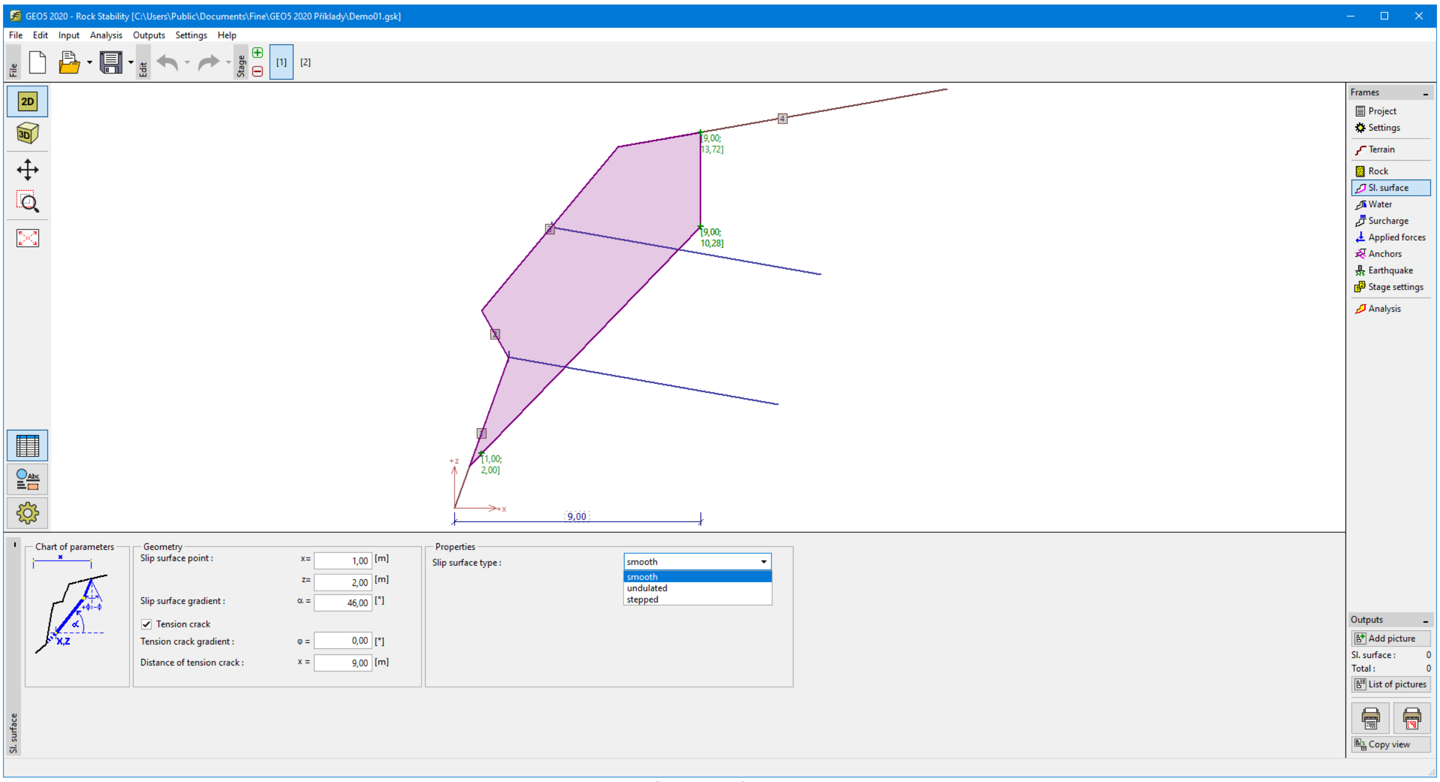
Slip Surface - Plane

The frame "**Slip surface**" serves to specify the shape and parameters of a plane slip surface. The slip surface is defined by a point in the rock body and by its gradient. The program automatically determines intersections of the slip surface with the terrain.

The program also allows for defining a **tension crack** with an arbitrary gradient (not available for stepped slip surface). The crack is defined by a horizontal distance from the origin and by its gradient.

The plane slip surface can further be labeled as smooth, undulated, or stepped.

The program makes it possible to export the geometry of a structure in the *.DXF format.



Frame "Slip surface - plane"

Slip Surface - Polygonal

The frame "**Slip surface**" contains a table with a list of defined sections of a slip surface. Adding section is performed in the "**New section**" dialog window. These sections can also be edited on the desktop with the help of active objects.

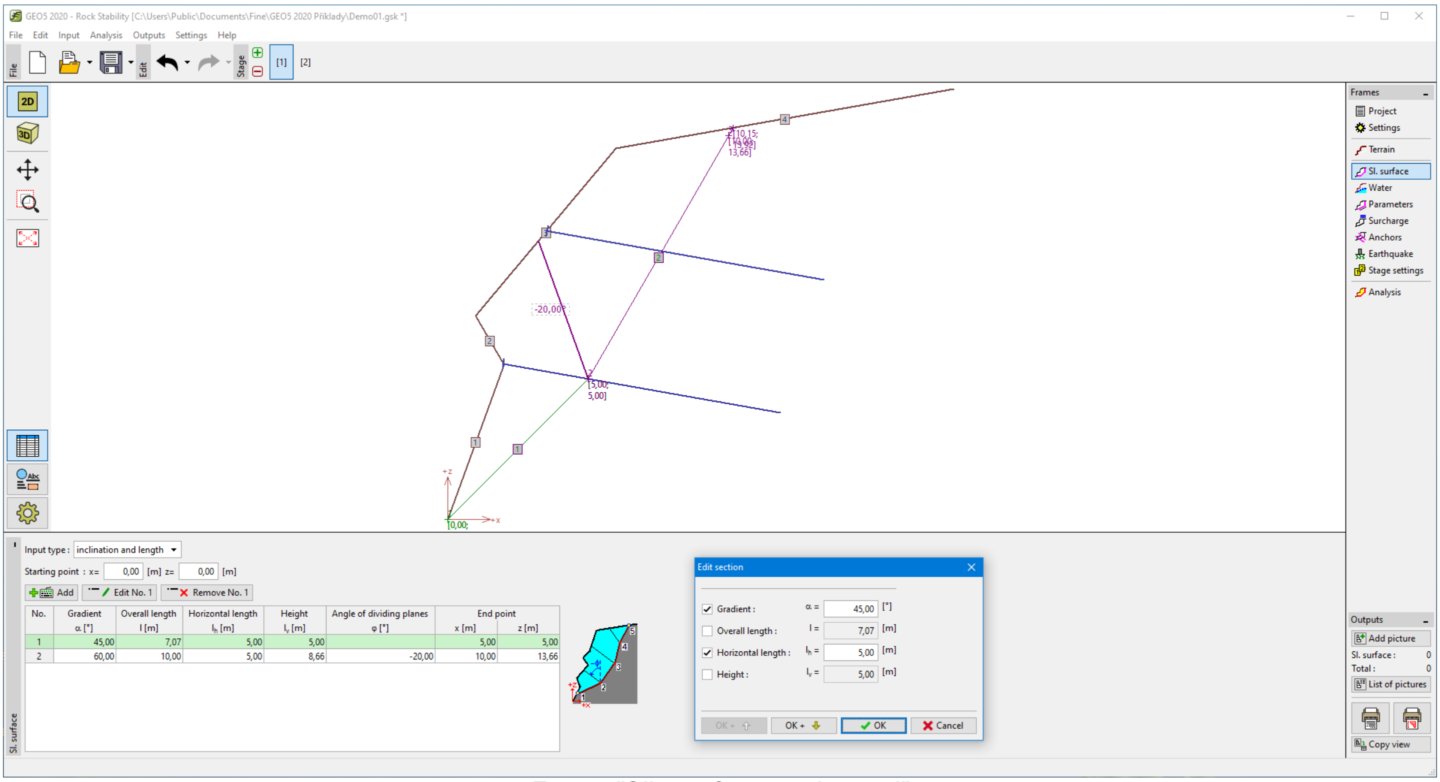
The coordinates of the slip surface **origin** - a point on the slip surface followed by other sections - are entered in the upper part of the frame. This point can be found even out of the soil body - the program then automatically calculates the intersection of slip surface with the terrain.

Individual sections of the slip surface can be defined by their dip, by the overall length of the section, by the horizontal length and height of a section of a rock slope. Only **two selected values** are used while the others are determined by the program automatically (if more than two entry fields are checked than the input and computation are not carried out). Both vertical and horizontal sections, as well as overhangs, can be represented.

In case of a proper input, the **program automatically plots the defined section** on the desktop using a dashed line so that before accepting it, it is possible to check whether the section is correctly defined.

General assumptions for the calculation of the polygonal slip surface are listed here.

The program makes it possible to export the geometry of a structure in the *.DXF format.

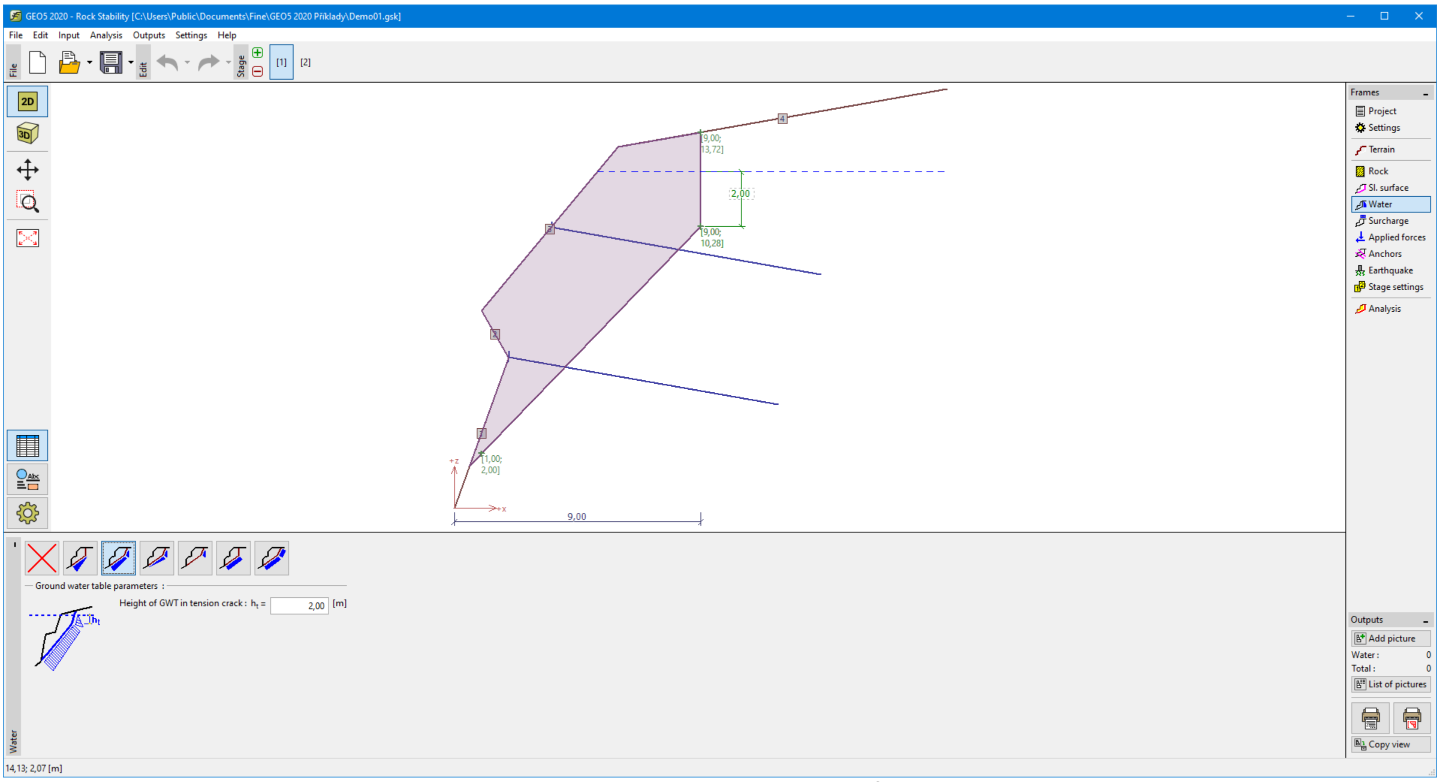


Frame "Slip surface - polygonal"

Water - Plane Slip Surface

The "**Water**" frame allows, by pressing the button, for selecting the type of water. The selected type with a graphic hint of input values is displayed in the left part of the frame. Water parameters can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

Solution procedure when accounting for water is described in the theoretical part of the help "Influence of water on slip surface".



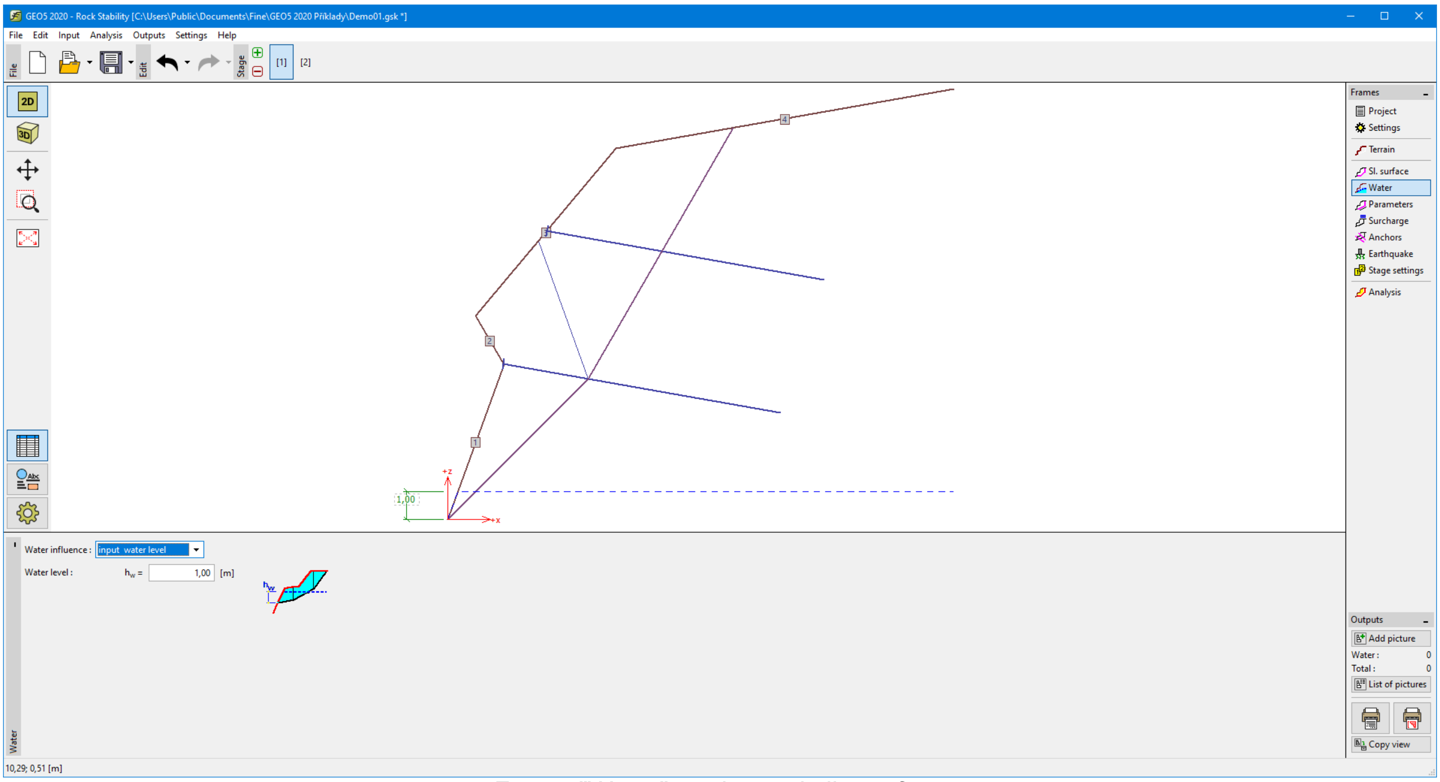
Frame "Water" - plane slip surface

Water - Polygonal Slip Surface

The "**Water**" frame serves to input the **influence of water** (not considered, input forces on blocks, input horizontal water level, input GWT). Water parameters can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

Solution procedure when accounting for water is described in the theoretical part of the help "Influence of water on polygonal slip surface".

For option "**input forces on blocks**" the forces of water acting on the slip surface F_v or forces from the water acting on the inner sliding surface U are entered in the frame "**Parameters**" (by pressing the button "**Edit**").



Frame "Water" - polygonal slip surface

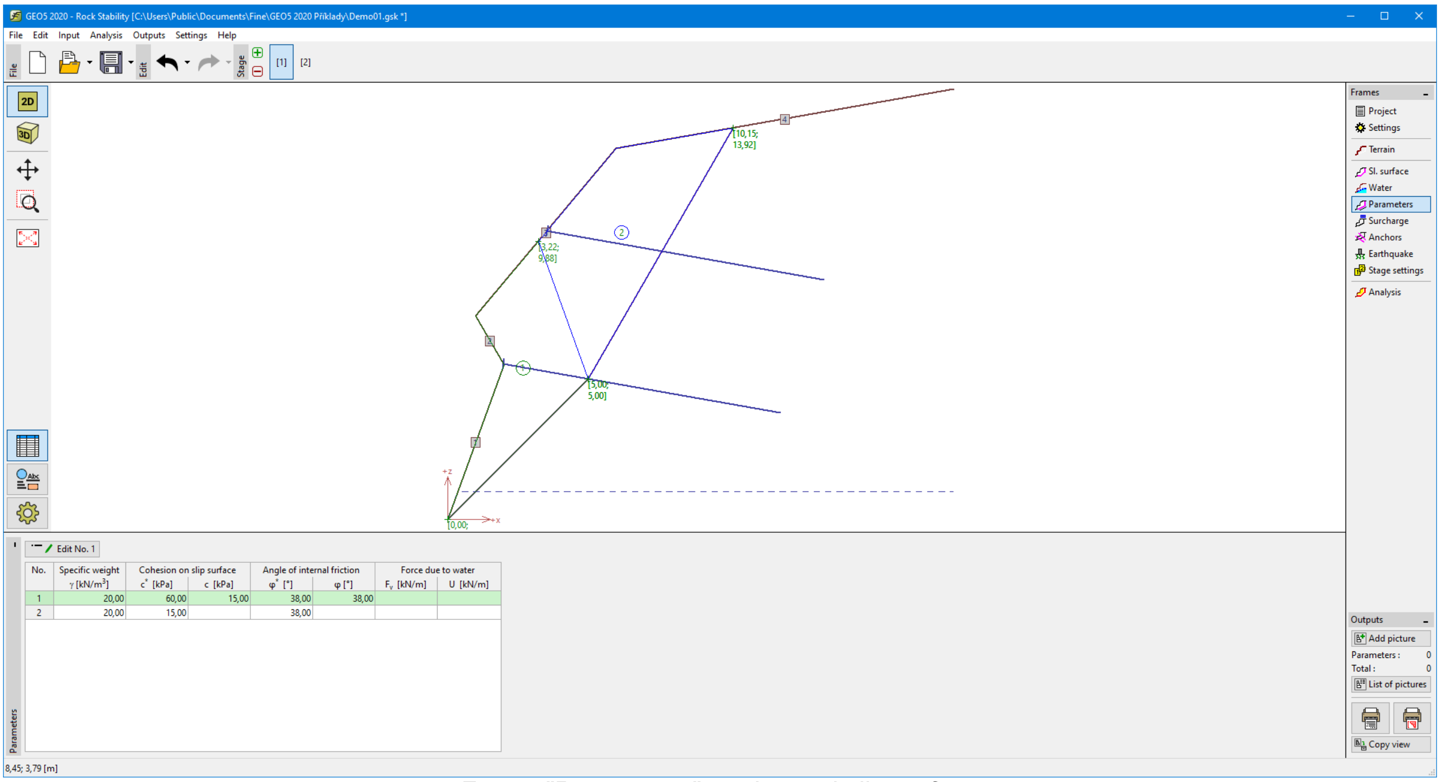
Parameters - Polygonal Slip Surface

The frame "**Parameters**" contains a table with a list of blocks, which are created by entering a polygonal slip surface. Parameters of individual blocks are edited in the "**Edit block**" dialog window. Blocks can also be edited on the desktop with the help of active objects.

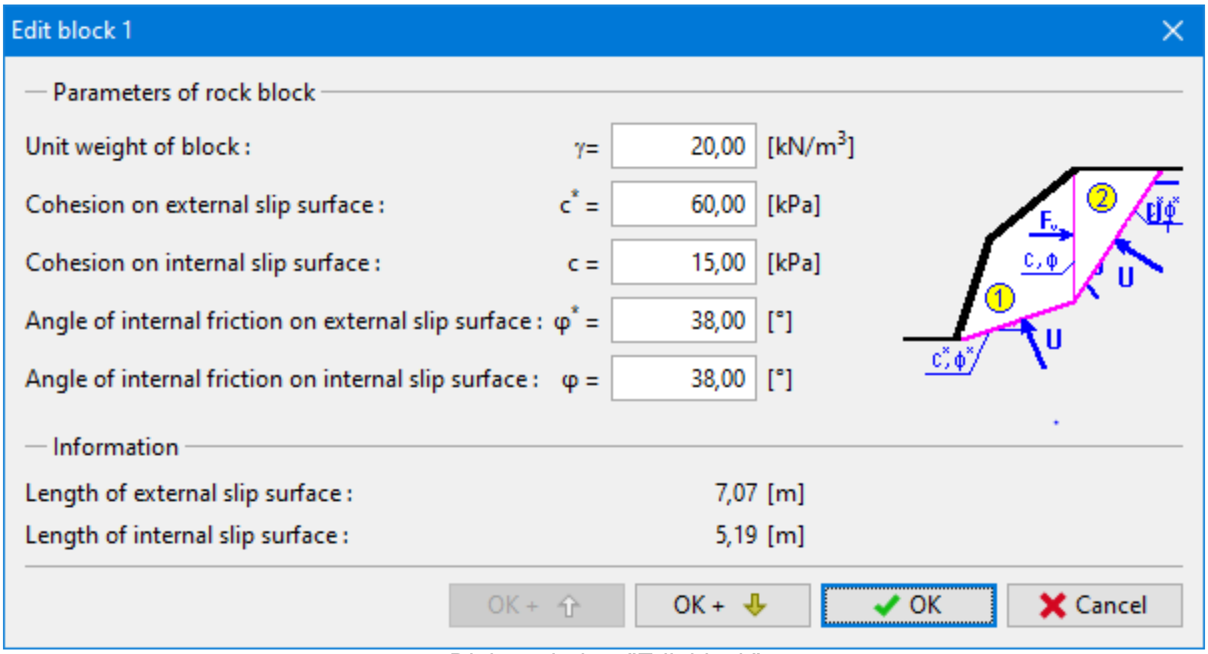
The Mohr-Coulomb strength parameters on a slip surface and in the joints separating individual blocks, including the unit weight of a rock, are specified here.

This window also serves to introduce forces due to water in rock blocks.

General assumptions for the calculation of the polygonal slip surface are listed here.



Frame "Parameters" - polygonal slip surface



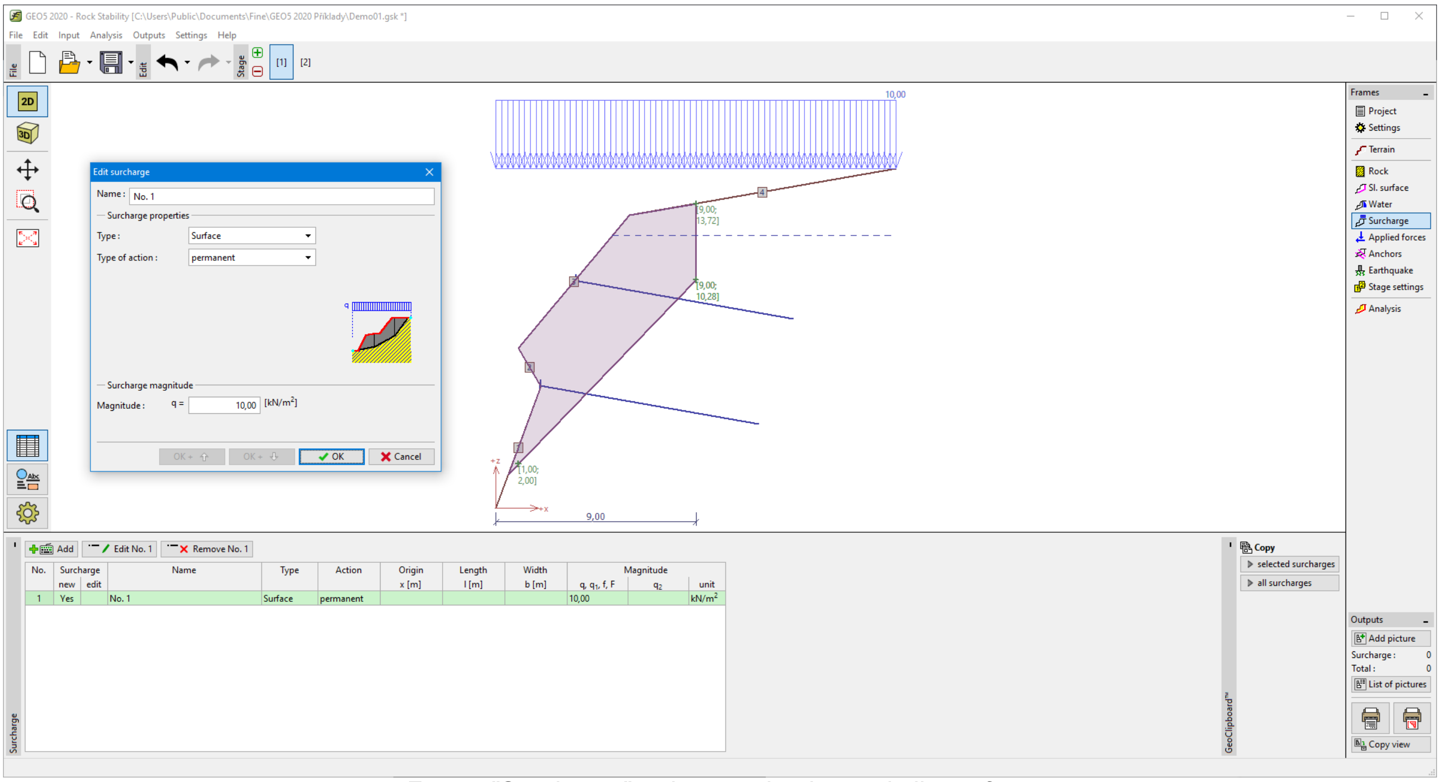
Dialog window "Edit block"

Surcharge - Plane and Polygonal Slip Surface

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or active objects, respectively.

Either **permanent**, **variable,** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with a favorable effect is not considered in the analysis.

Introducing surcharge forces into the analysis differs for a plane and a polygonal slip surface.



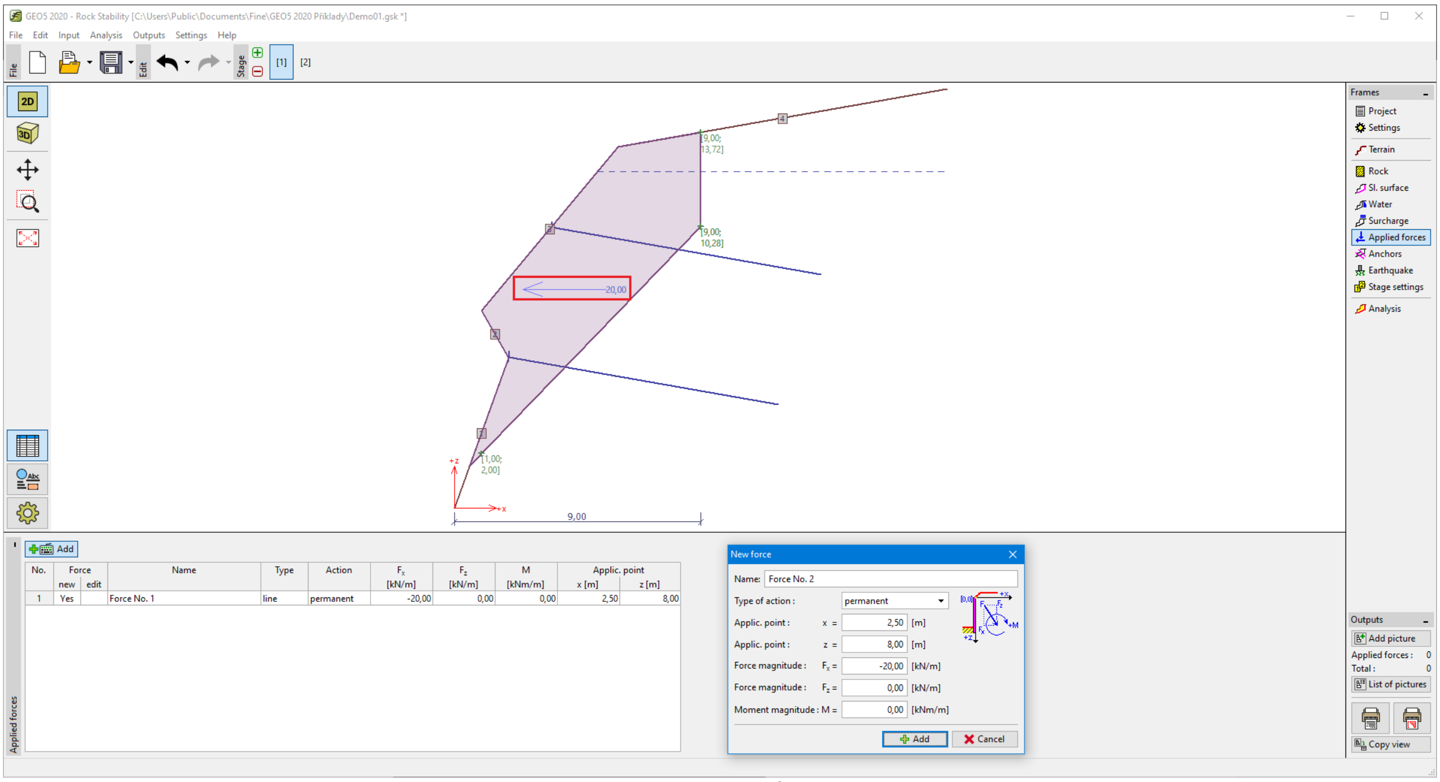
Frame "Surcharge" - plane and polygonal slip surface

Applied Forces - Plane Slip Surface

The "**Applied forces**" frame contains a table with a list of forces acting on a rock block. Adding forces is performed in the "**New force**" dialog window. The input forces can also be edited on the desktop with the help of active objects.

Applied forces represent an additional load on the rock block. We can model such as rope anchorage of bridges or pedestrian over path etc. The program does not modify the applied forces in the calculation in any way except multiplying them with the corresponding coefficients according to the selected type of load (EN1997, LRFD).

External load acting on the terrain is necessary to define as a a surcharge.



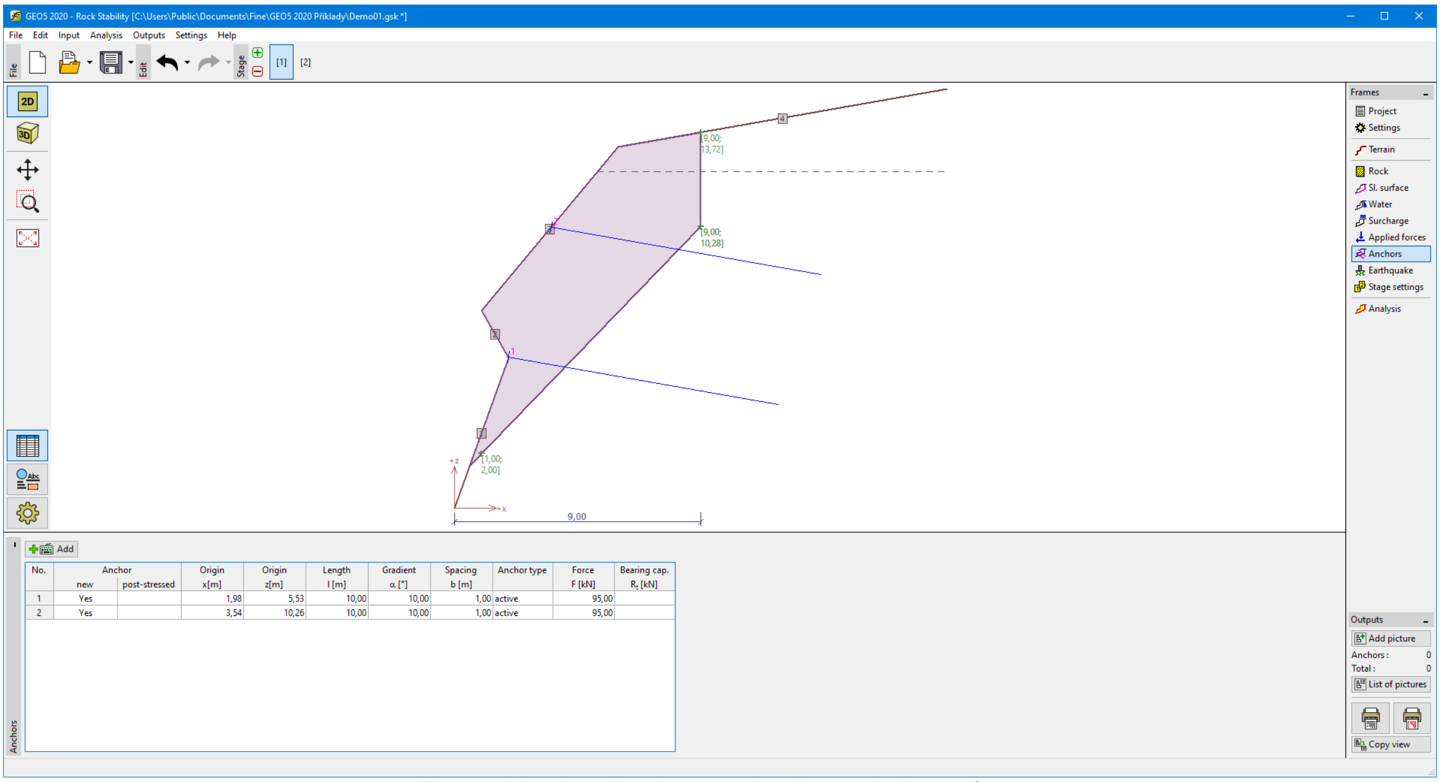
Frame "Applied forces"

Anchors - Plane and Polygonal Slip Surface

The "**Anchors**" frame contains a table with a list of input anchors. Adding anchors is performed in the "**New anchor**" dialog window. The input anchors can be edited on the desktop with the help of active objects.

The following is specified - location (origin), depth, free length, anchor slope, spacing between anchors, and anchor force. The anchor origin can automatically be **positioned on terrain** (by checking the particular entry field). All anchor parameters can be modified only in the construction stage, where it was introduced. The subsequent stages allow only adjusting the anchor force (option "**Post-stressing anchor**").

The plane slip surface allows the input of active and passive anchors. Only active anchors are allowed using the polygonal slip surface.

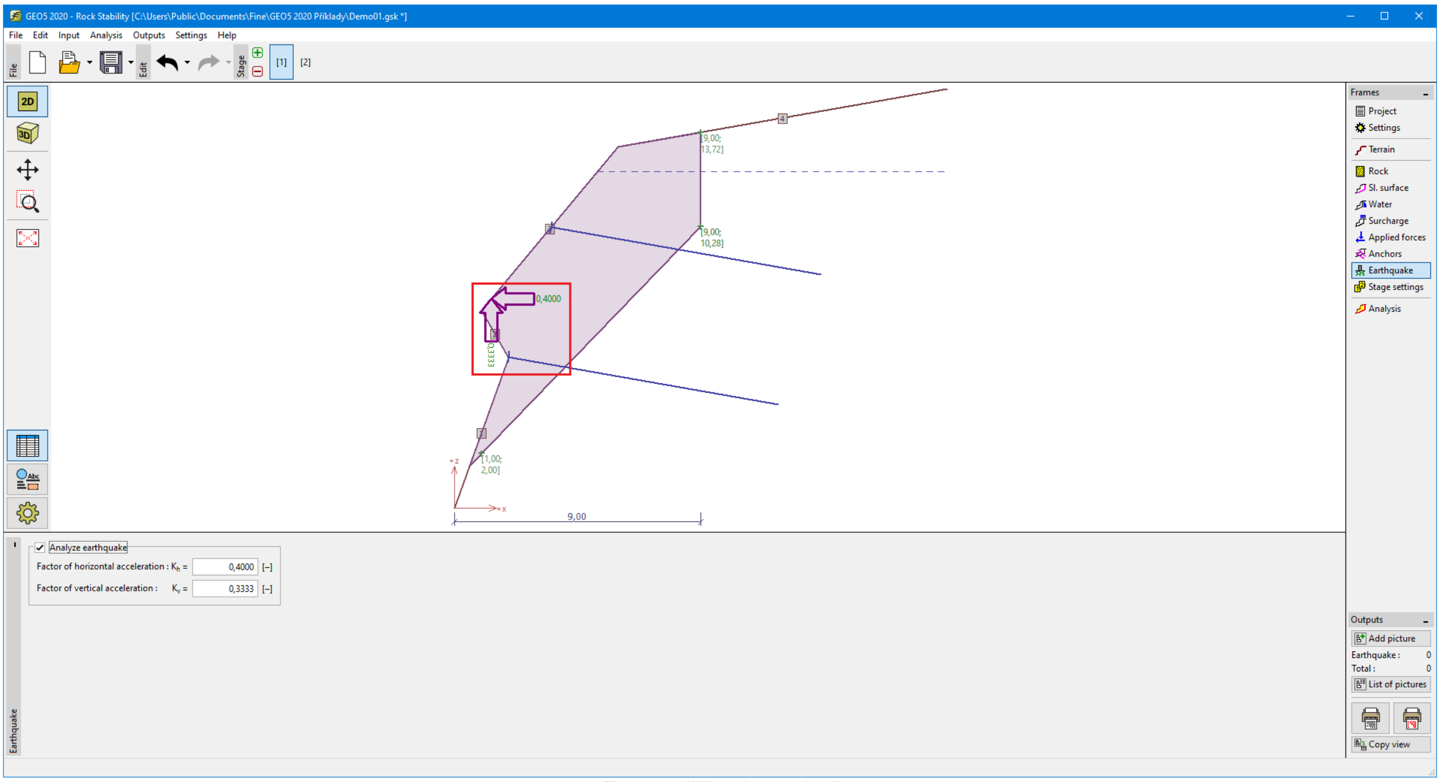


Frame "Anchors" - plane and polygonal slip surface

Earthquake

The "**Earthquake**" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

Rock slope analysis while accounting for an earthquake is described in the theoretical part of the help, chapter "Influence earthquake".



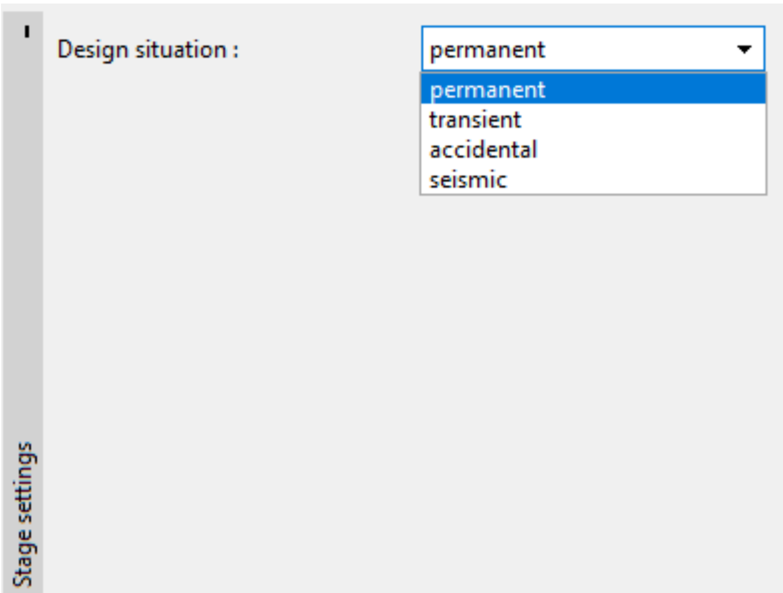
Frame "Earthquake"

Stage Settings

The frame "**Stage settings**" serves to input settings valid for a given construction stage.

The selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology.



Frame "Stage settings"

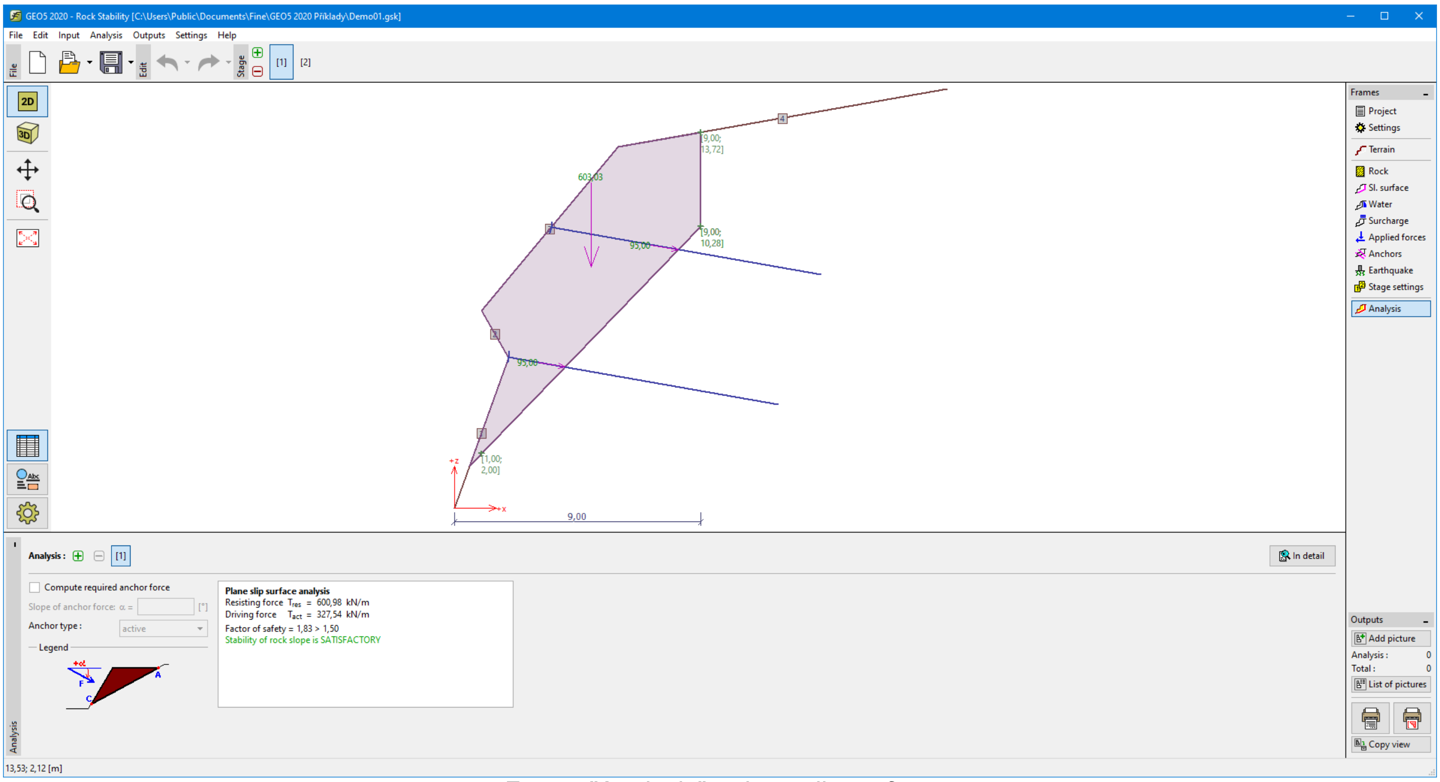
Analysis - Plane Slip Surface

The "**Analysis**" frame displays the analysis results. Several analyses can be performed for a single task.

Assessment of the rock slope for the plane slip surface can be carried out according to the selected verification methodology based on the input in the "Settings" frame. The analysis results are displayed in the frame in the bottom part of the desktop.

In this frame, the program makes it possible to determine the **anchor force needed** for obtaining the required safety factor. In such a case, the "**Compute required anchor force**" entry field must be checked and the slope of anchor force from horizontal must be entered.

Visualization of results can be adjusted in the "Drawing Settings" frame.



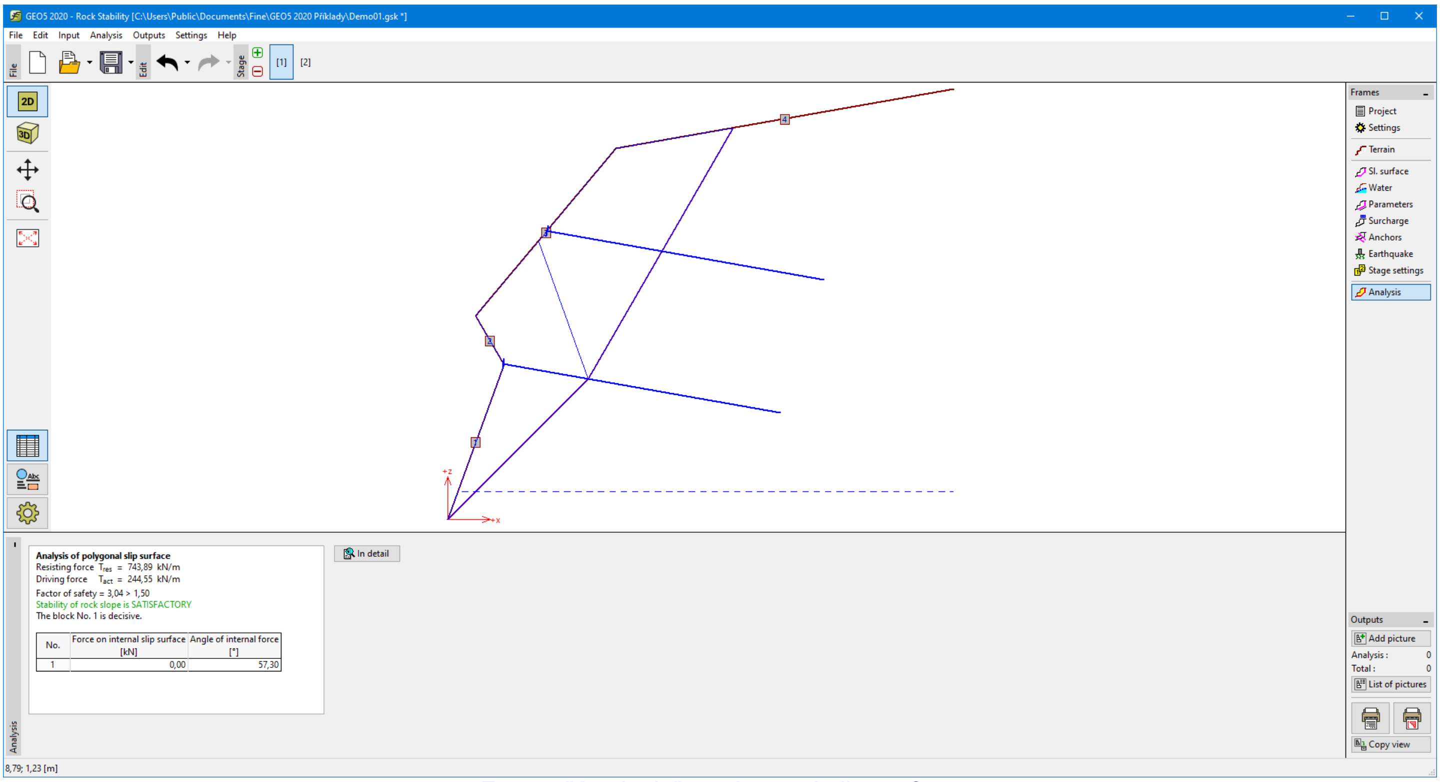
Frame "Analysis" - plane slip surface

Analysis - Polygonal Slip Surface

The "**Analysis**" frame displays the analysis results. Several analyses can be performed for a single task.

Assessment of the rock slope for the polygonal slip surface can be carried out according to the selected verification methodology based on the input in the "Settings" frame. The analysis results are displayed in the frame in the bottom part of the desktop.

Visualization of results can be adjusted in the frame "Drawing Settings" frame.



Frame "Analysis" - polygonal slip surface

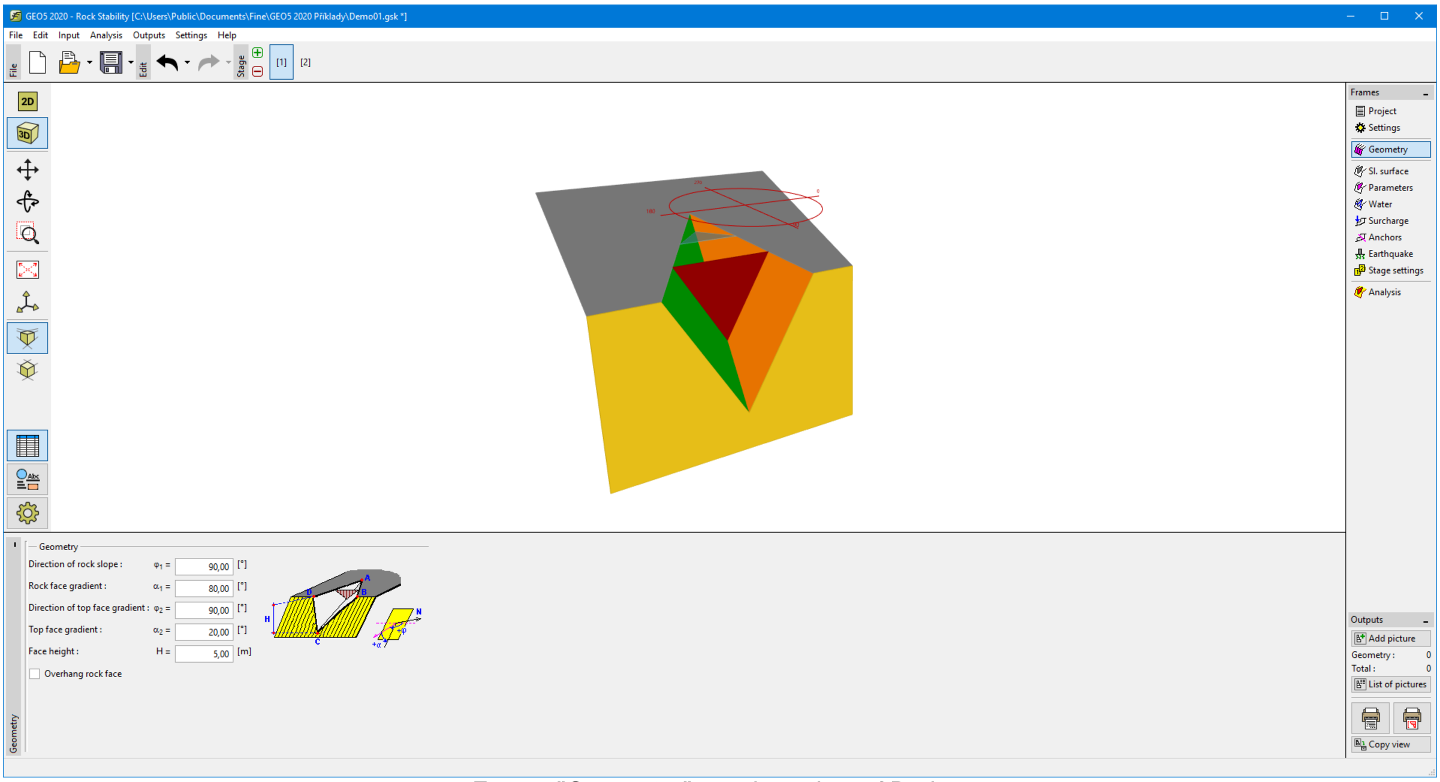
Geometry - Rock Wedge

The frame "**Geometry**" allows for entering the shape of a rock slope (earth wedge).

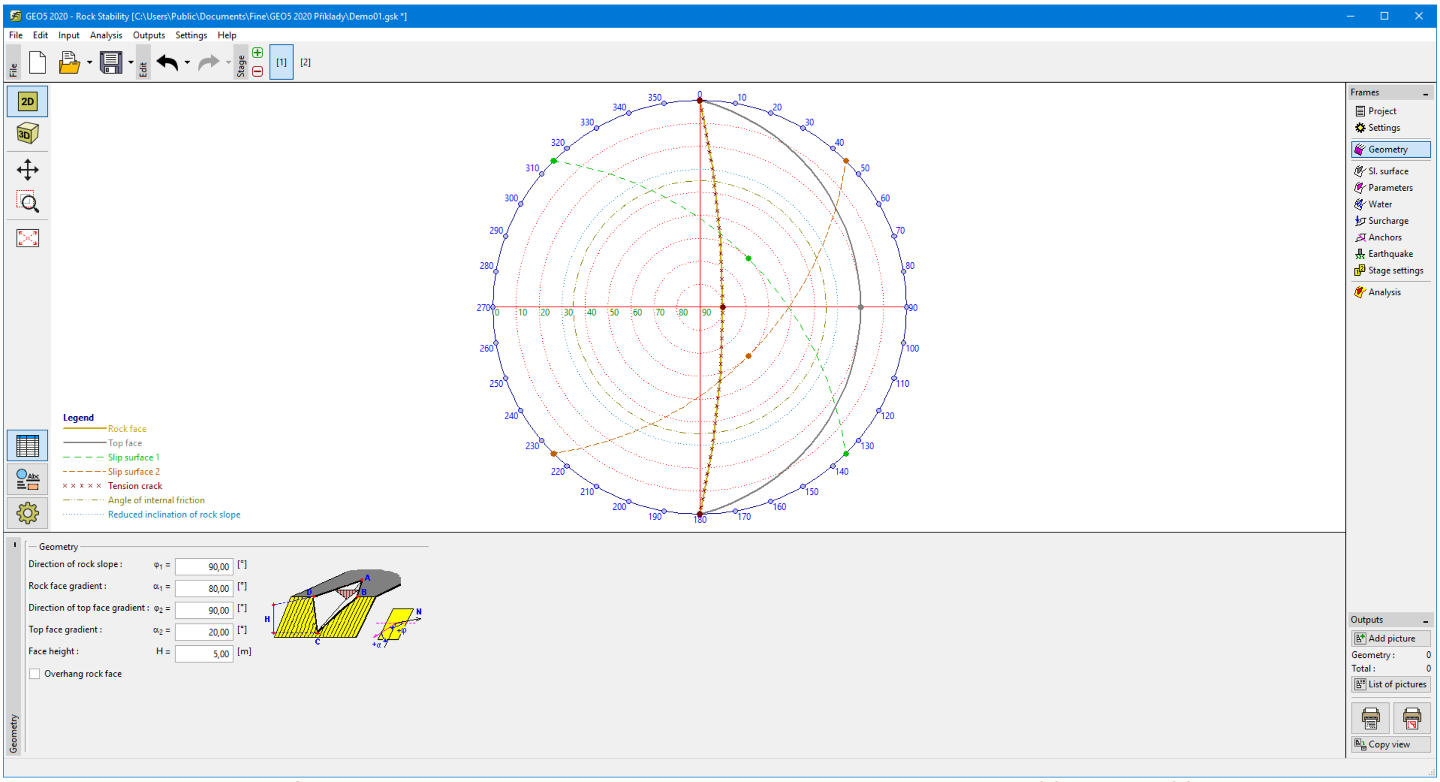
Geometry of earth wedge is defined by directions and gradients of fall lines of faces forming the wedge. Geometry of earth wedge is displayed on the desktop using a 3D view or using stereographic projection.

Pressing the button "**Overhang rock face**" can be modeled overhang rock faces.

The program makes it possible to export the geometry of a structure in the *.DXF format.



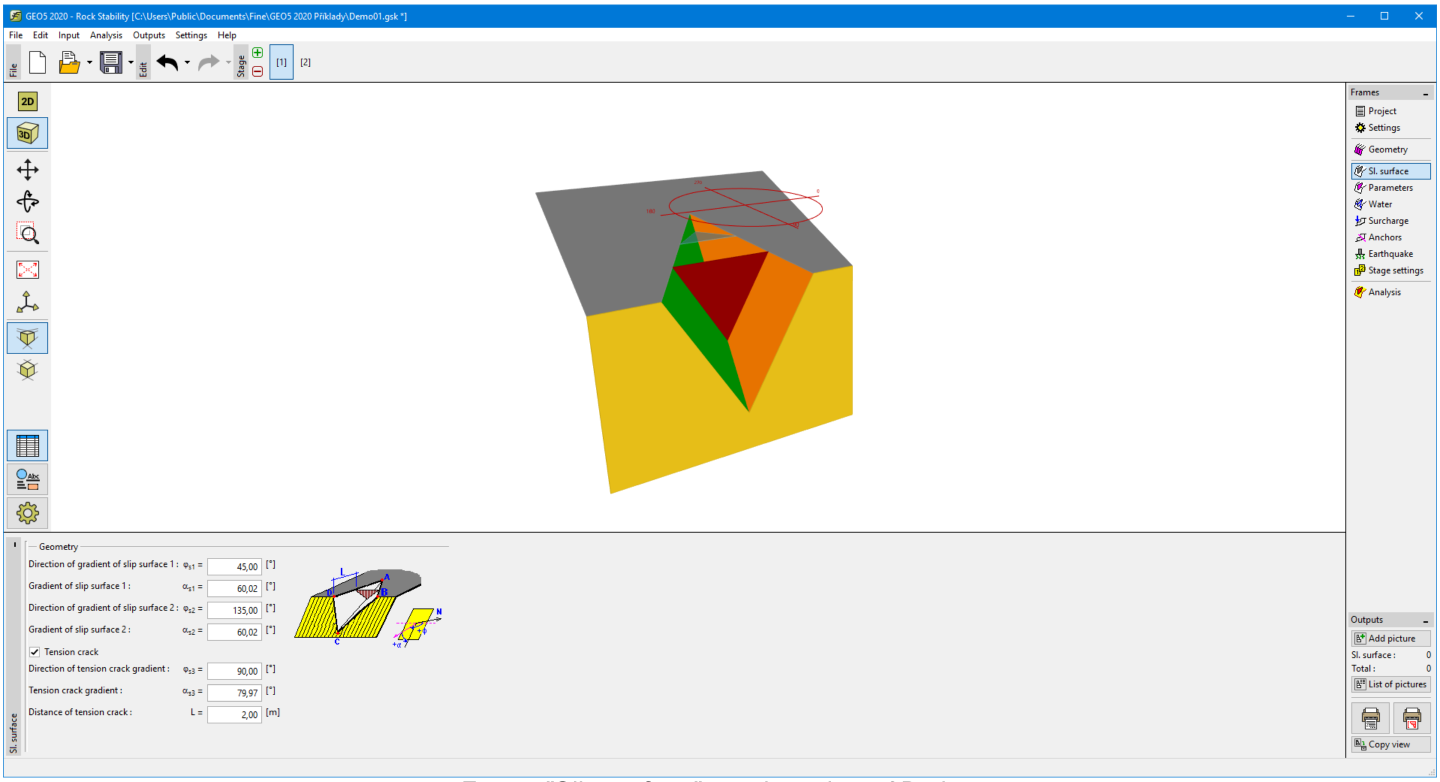
Frame "Geometry" - rock wedge - 3D view



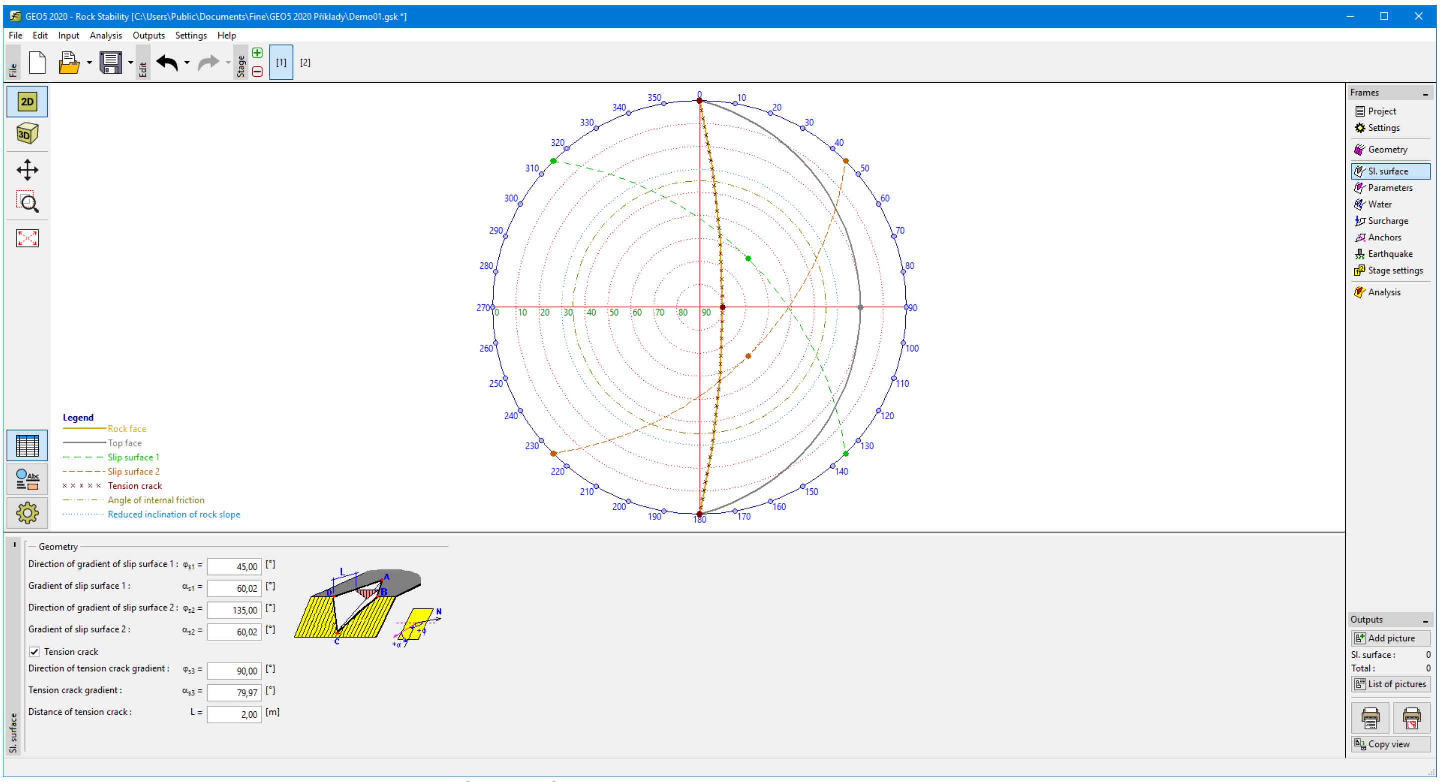
Frame "Geometry" - rock wedge - input using directions and gradients of fall lines of faces

Slip Surface - Rock Wedge

The frame "**Slip surface**" serves to enter the shape of a slip surface using directions and gradients of fall lines of faces forming the wedge. A tension crack can also be defined. Geometry of earth wedge is displayed on the desktop using a 3D view or using stereographic projection.



Frame "Slip surface" - rock wedge - 3D view

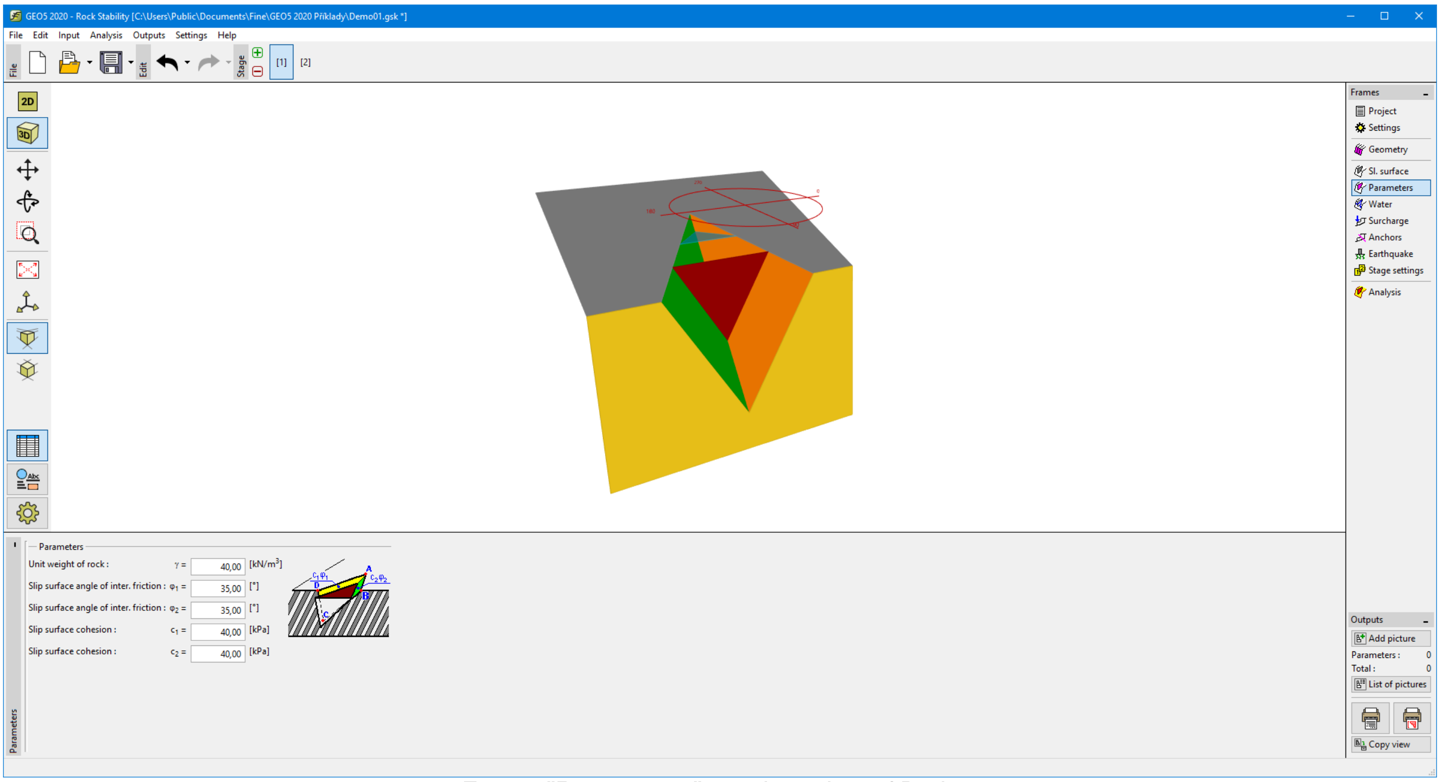


Frame "Slip surface" - rock wedge - stereographic projection

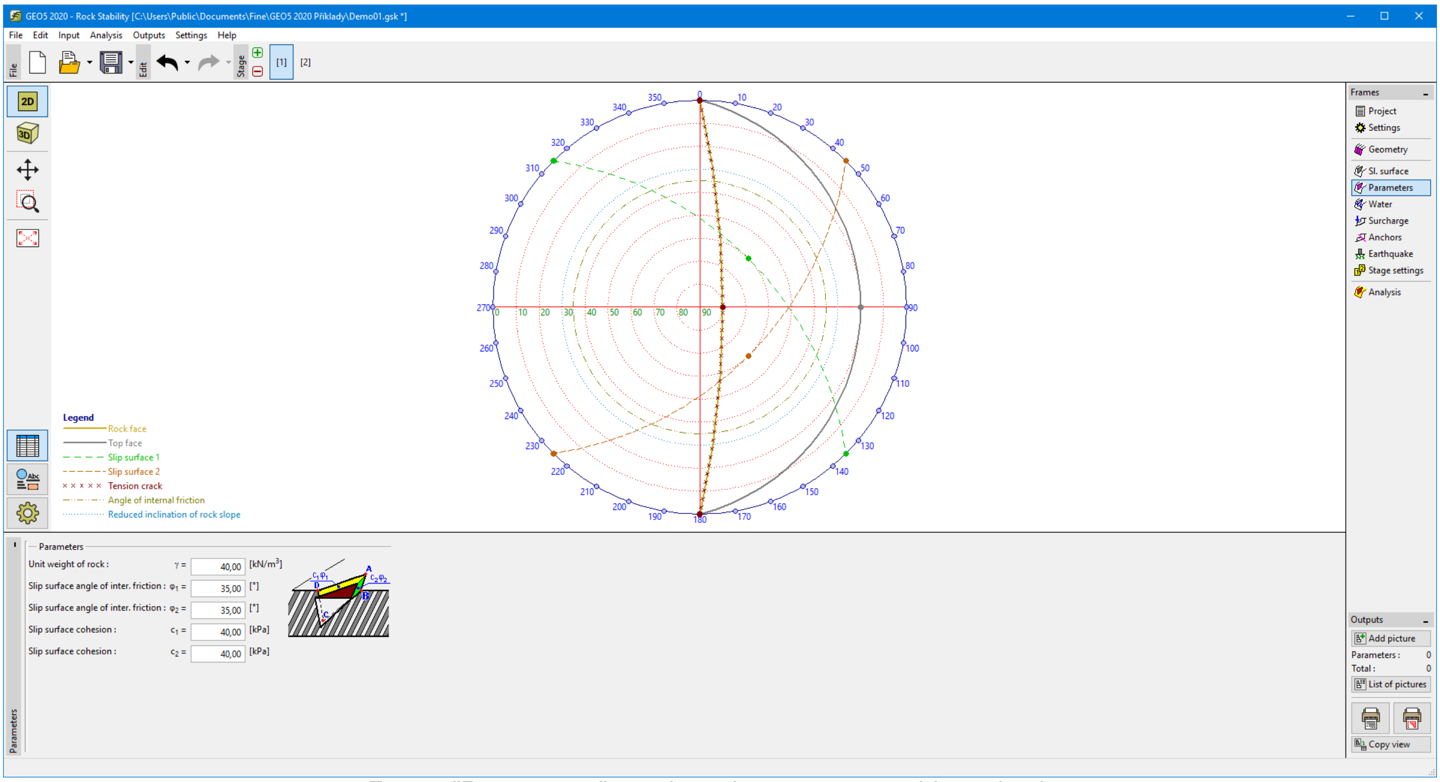
Parameters - Rock Wedge

The frame "**Parameters**" serves to enter parameters of an earth wedge. The unit weight of a rock and the Mohr-Coulomb strength parameters of slip surfaces must be specified.

Geometry of earth wedge is displayed on the desktop using a 3D view or using stereographic projection.



Frame "Parameters" - rock wedge - 3D view



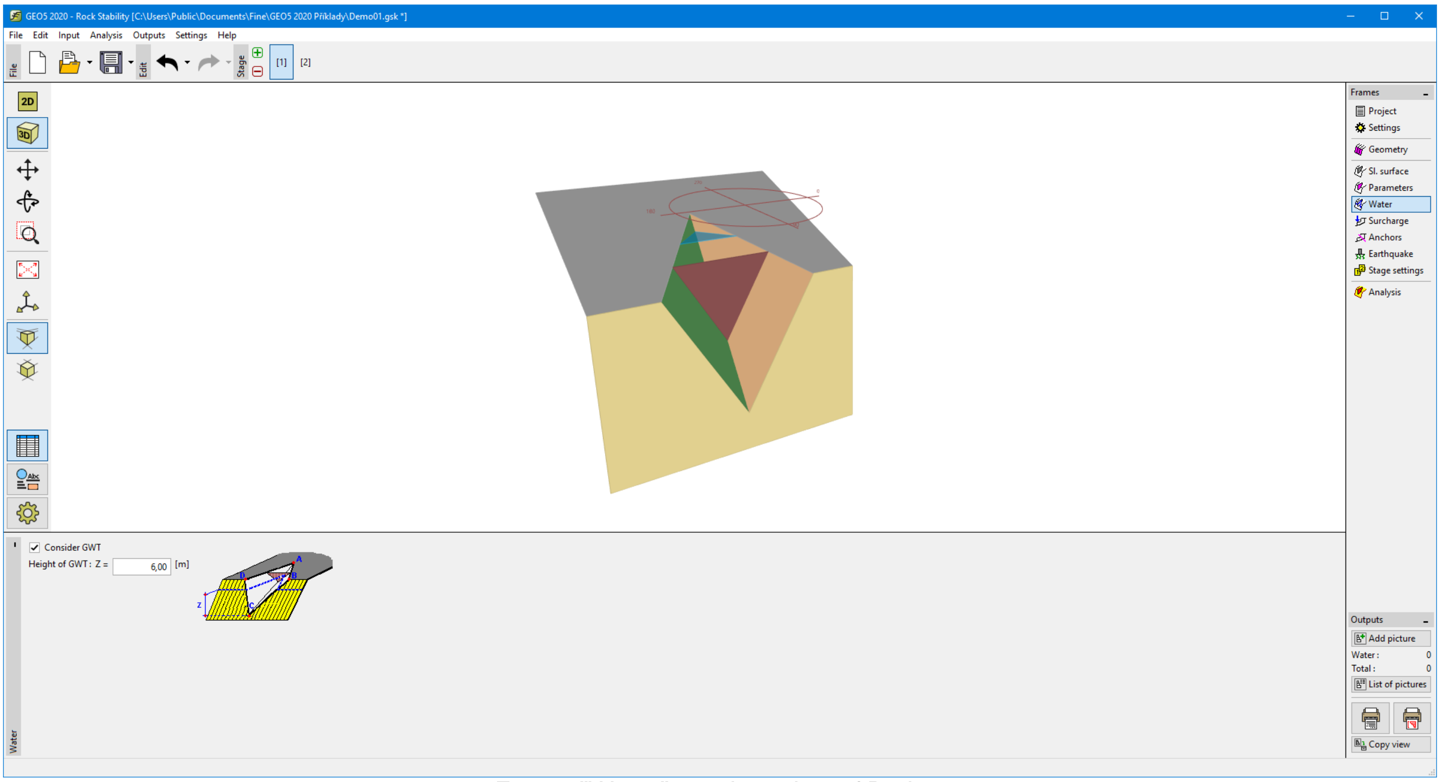
Frame "Parameters" - rock wedge - stereographic projection

Water - Rock Wedge

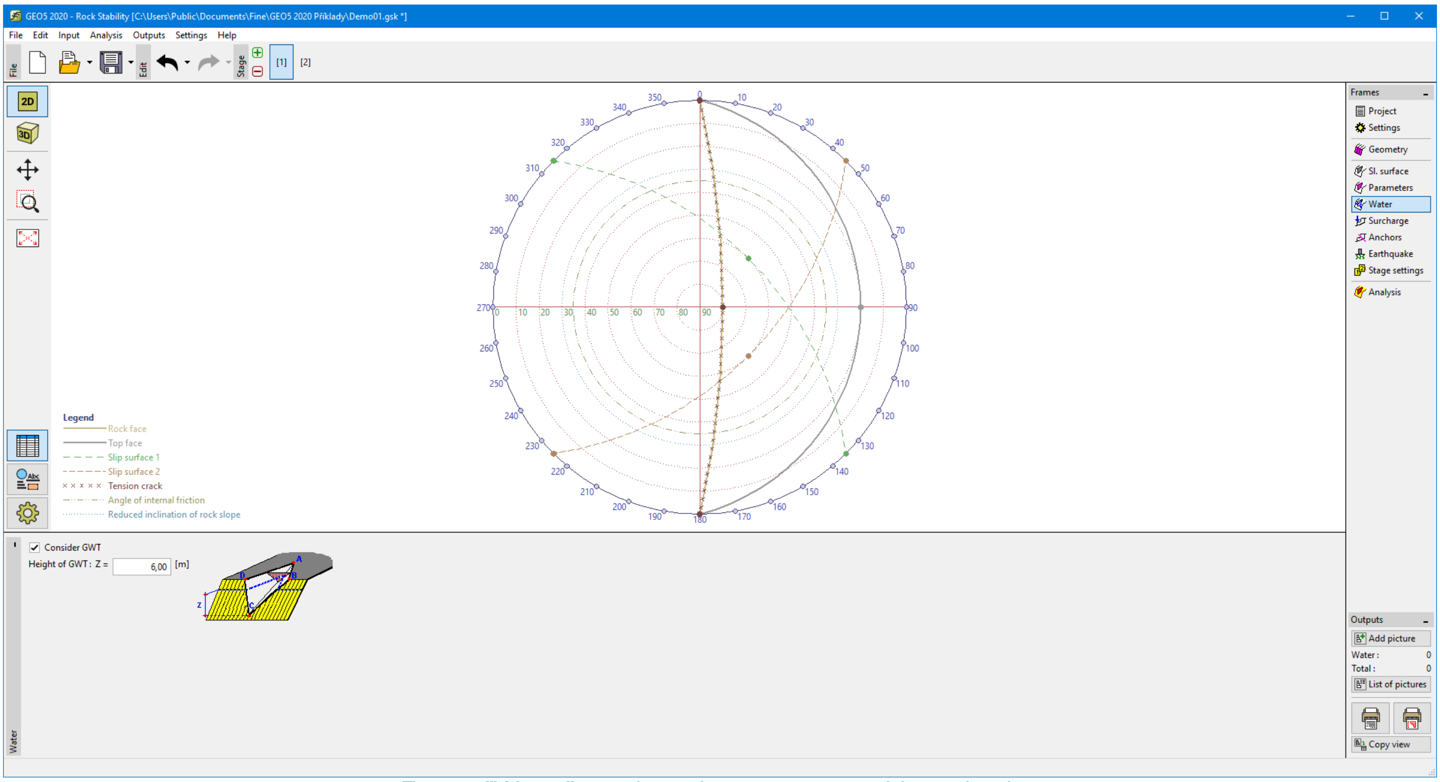
The frame "**Water**" allows for introducing water into the analysis. If the influence of water is taken into account, then checking the respective entry field opens the field for entering the height of GWT above the lowest point of an earth wedge.

Solution procedure when accounting for water is described in the theoretical part of the help "Influence of water".

Geometry of earth wedge is displayed on the desktop using a 3D view or using stereographic projection.



Frame "Water" - rock wedge - 3D view



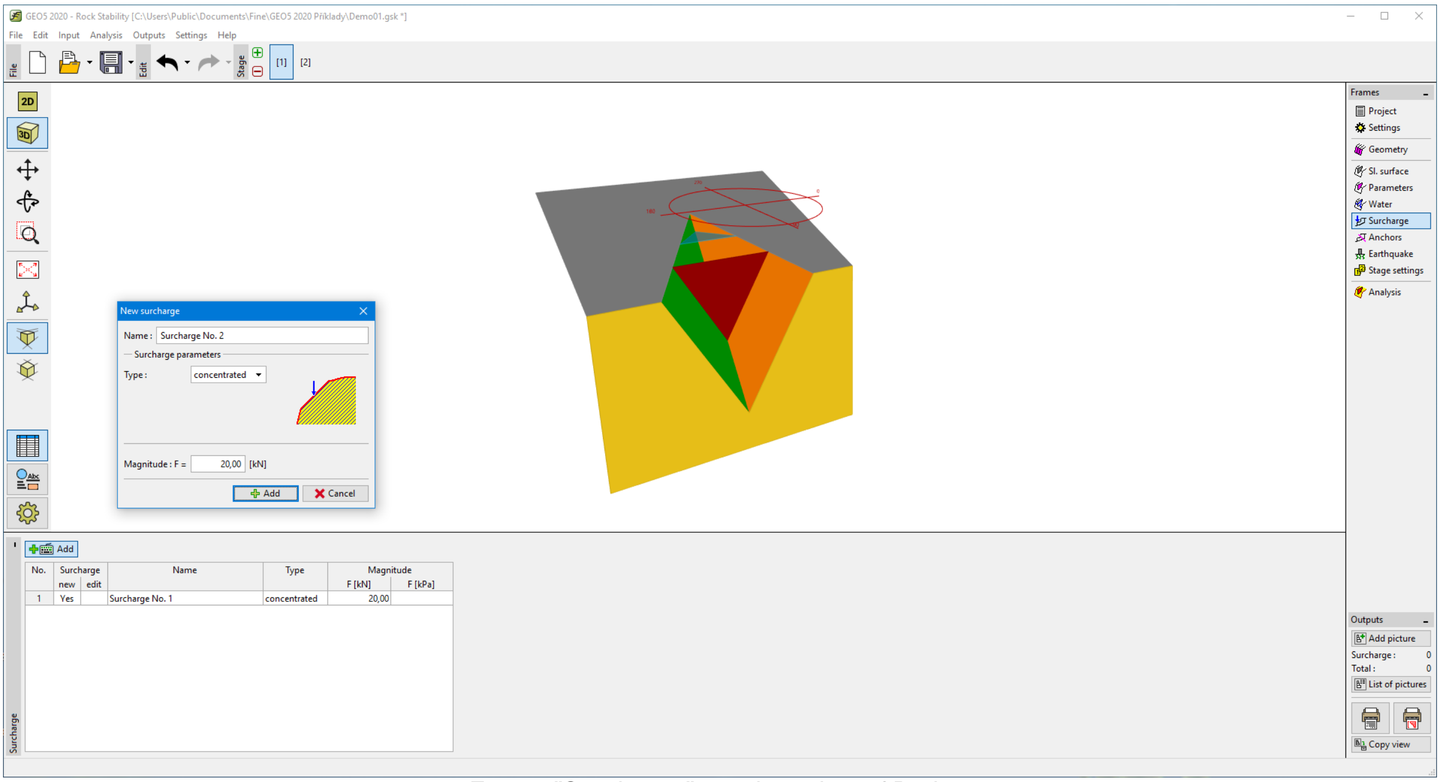
Frame "Water" - rock wedge - stereographic projection

Surcharge - Rock Wedge

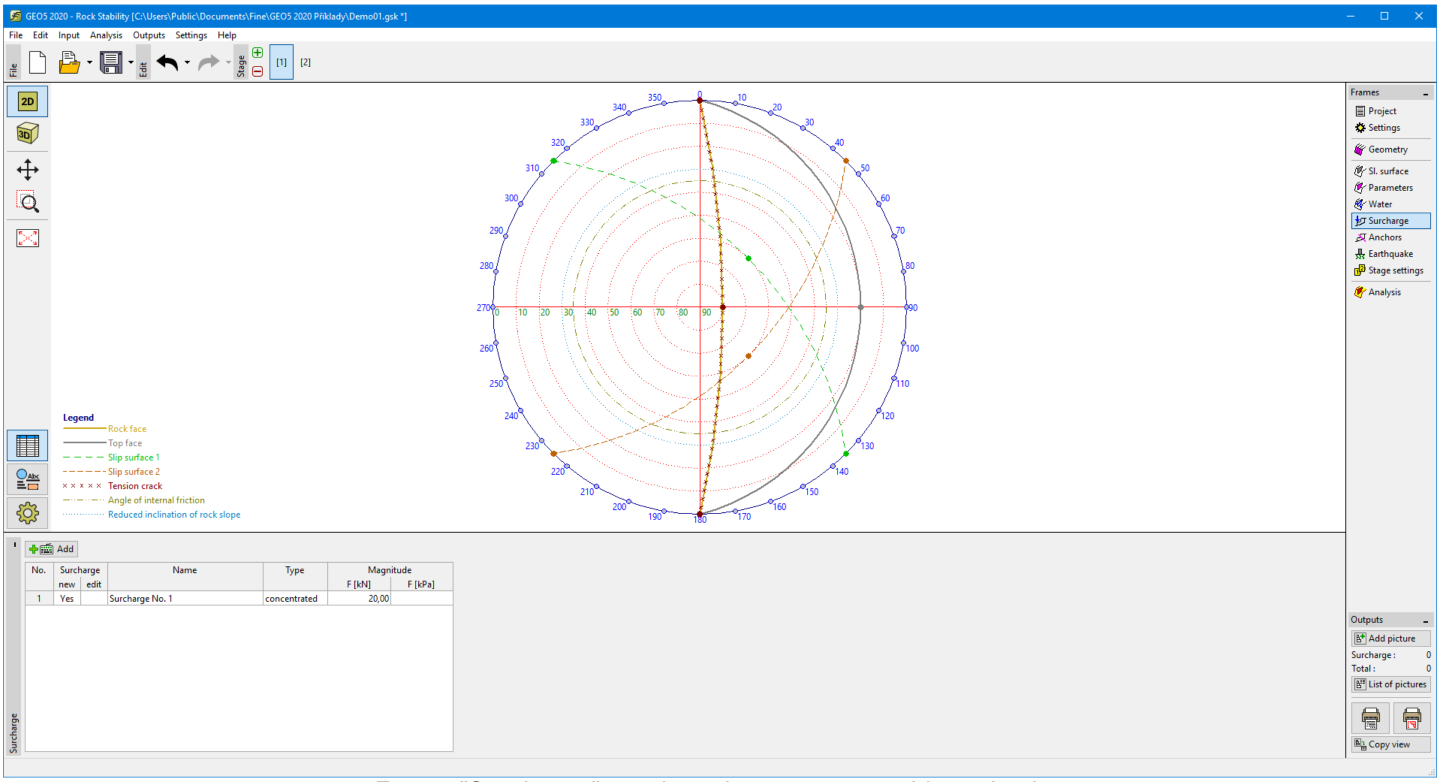
The "**Surcharge**" frame contains a table with a list of input surcharges. Adding surcharge is performed in the "**New surcharge**" dialog window.

Surcharge forces are introduced into the stability analysis of earth wedge using the resolution of forces.

Geometry of earth wedge is displayed on the desktop using a 3D view or using stereographic projection.



Frame "Surcharge" - rock wedge - 3D view



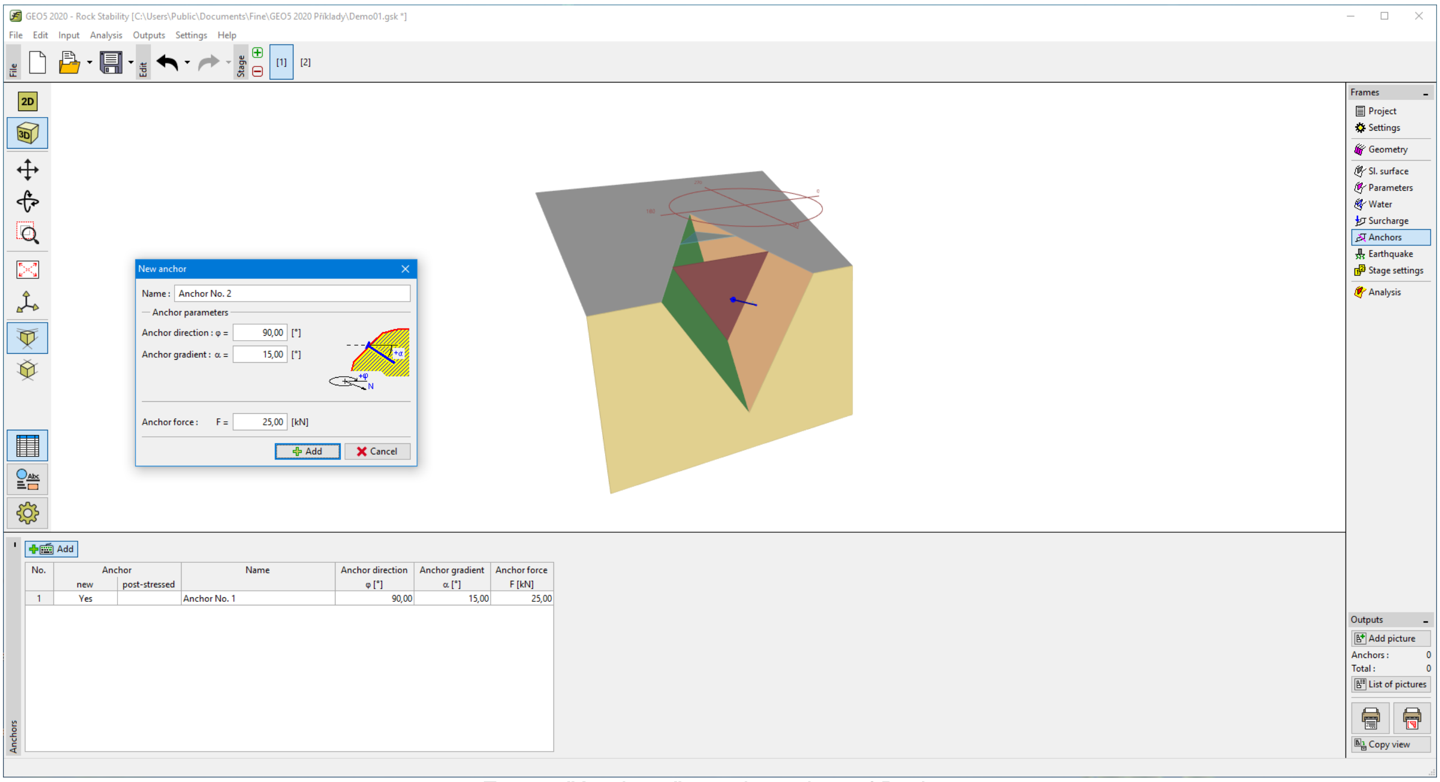
Frame "Surcharge" - rock wedge - stereographic projection

Anchors - Rock Wedge

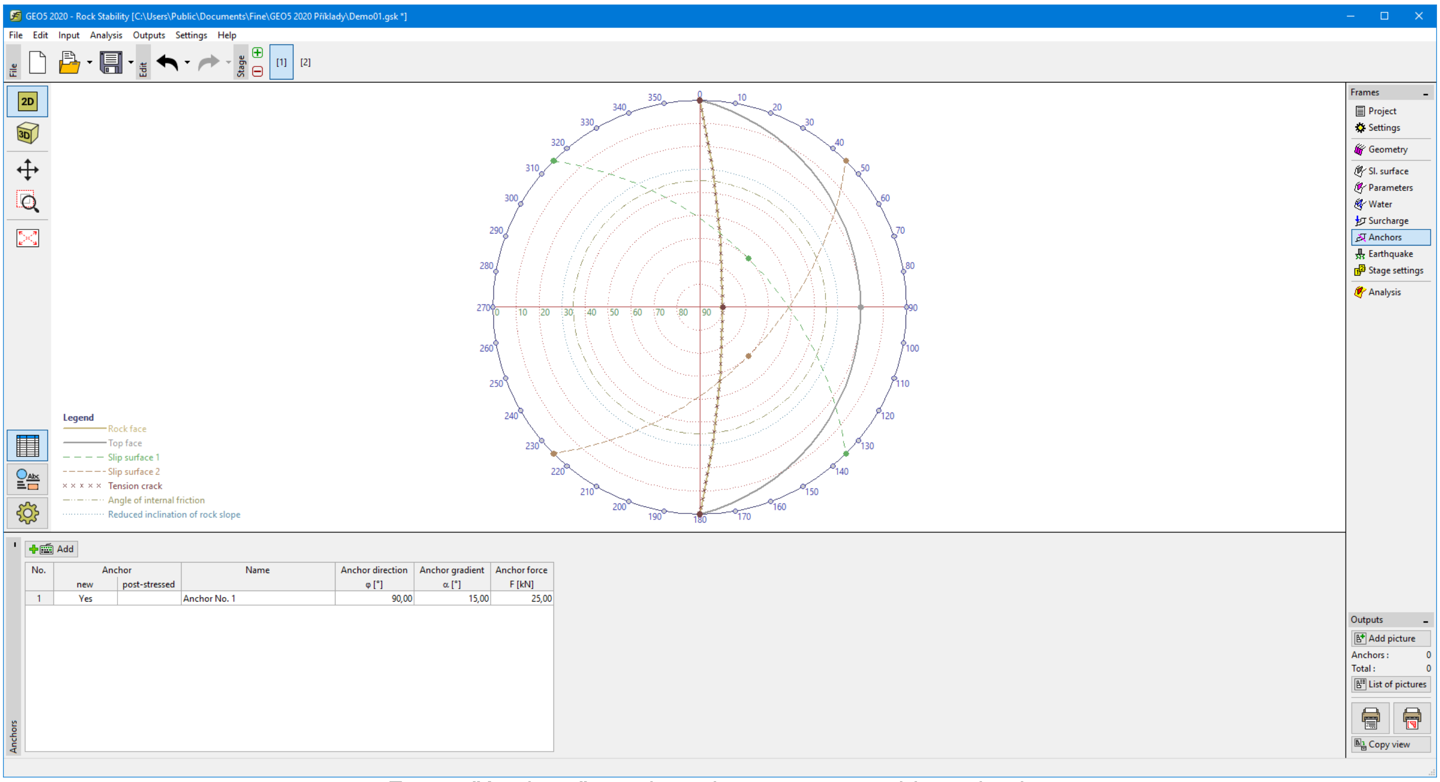
The "**Anchors**" frame contains a table with a list of input anchors. Adding anchors is performed in the "**New anchor**" dialog window.

Anchor forces are introduced into the stability analysis of earth wedge using resolution acting of forces.

Geometry of earth wedge is displayed on the desktop using a 3D view or using stereographic projection.



Frame "Anchors" - rock wedge - 3D view



Frame "Anchors" - rock wedge - stereographic projection

Analysis - Rock Wedge

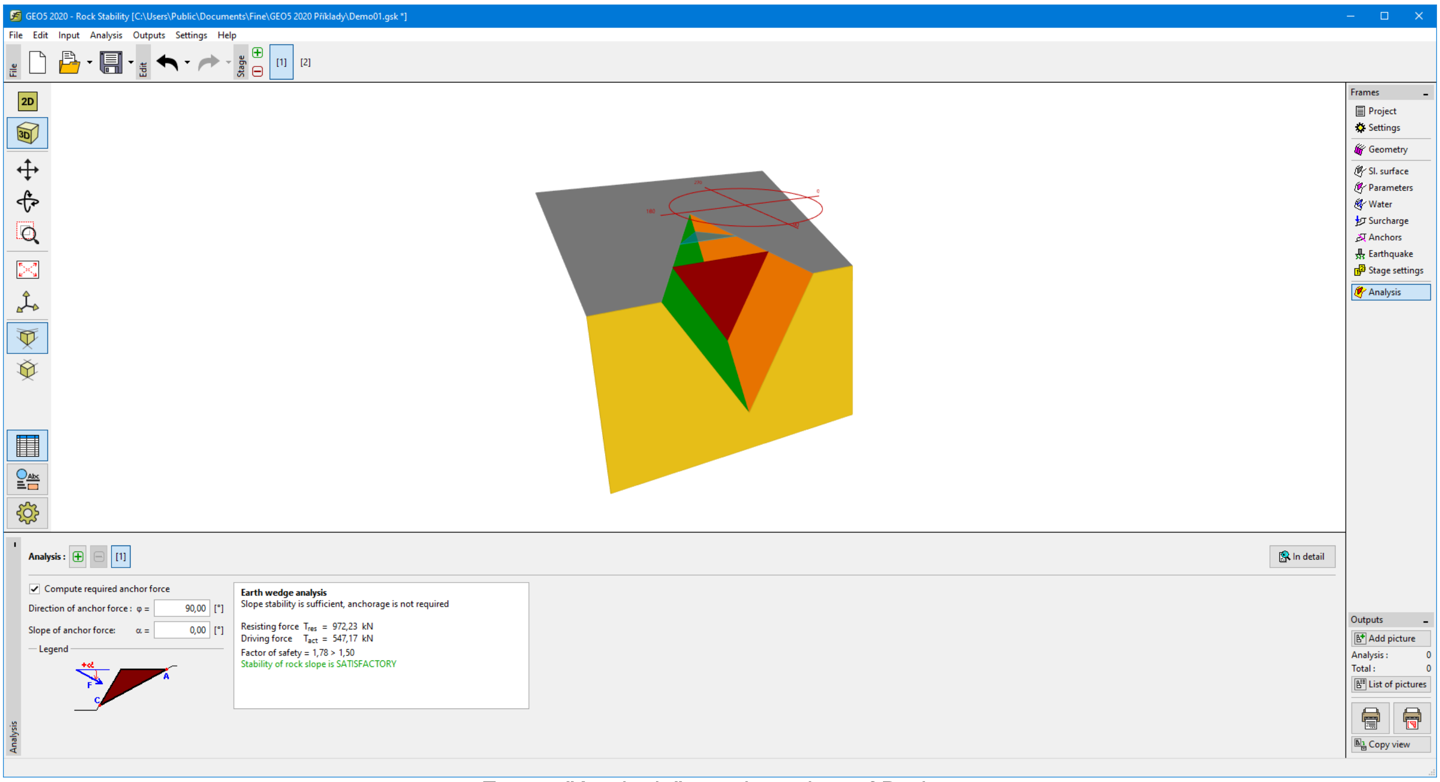
The "**Analysis**" frame displays the analysis results. Several analyses can be performed for a single task.

Assessment of the rock slope that is considered as rock wedge can be carried out according to the selected verification methodology based on the input in the "Settings" frame. The analysis results are displayed in the frame in the bottom part of the desktop.

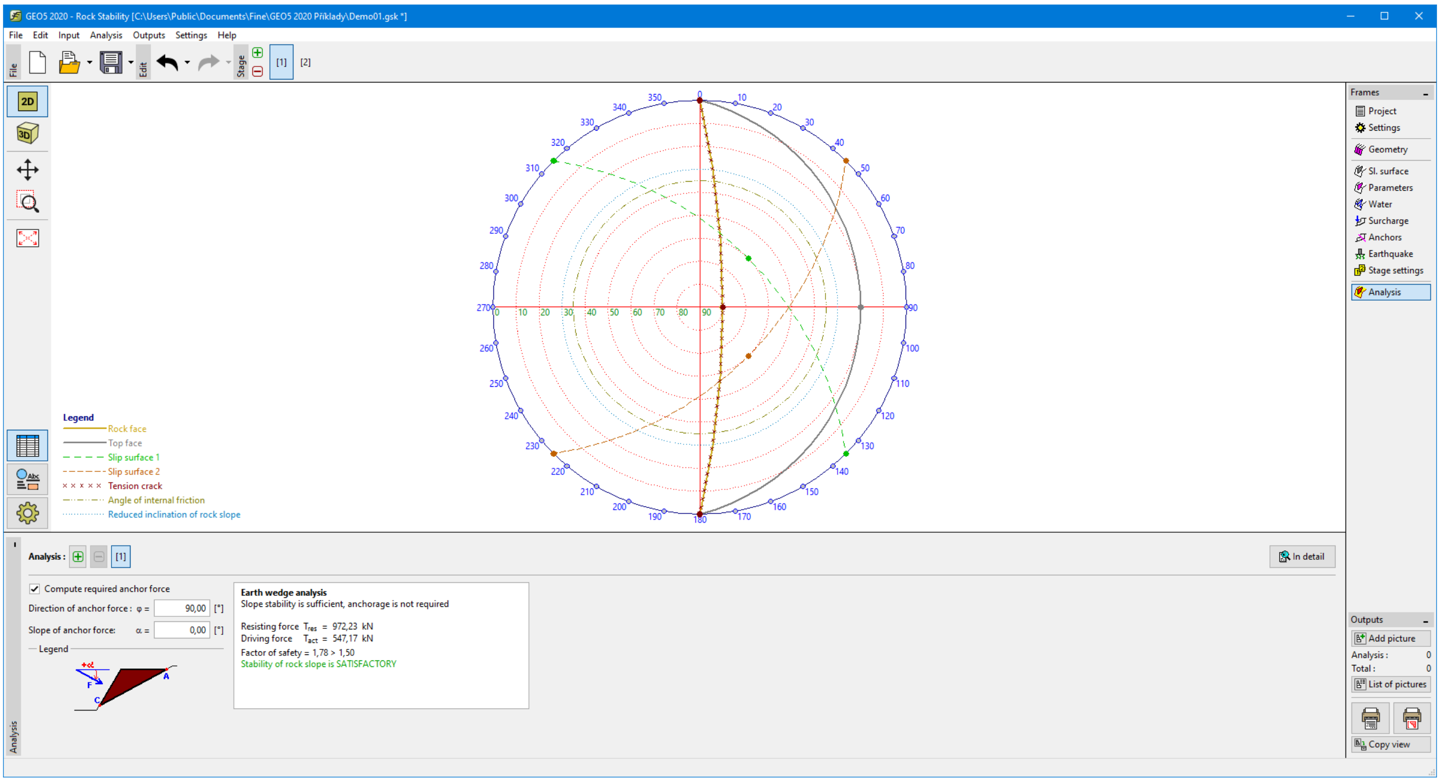
In this frame, the program makes it possible to determine the **anchor force needed** for obtaining the required safety factor. In such a case, the "**Compute required anchor force**" entry field must be checked, and the slope of anchor force from horizontal and its direction must be entered.

Geometry of earth wedge is displayed on the desktop using a 3D view or using stereographic projection.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Analysis" - rock wedge - 3D view



Frame "Analysis" - rock wedge - stereographic projection

Program MSE Wall

This program is used for verification of mechanically stabilized earth walls and segmental retaining walls reinforced by geogrids (georeinforcements).

The help in the program "MSE Wall" includes the following topics:

- The input of data into individual frames:

Project	Settings	Geometry	Material	Types of Reinforcements	Reinforcement (Blocks)	Reinforcement
Profile	Soils	Assign	Terrain	Water (Blocks)	Water	Surcharge
Front Face Resistance	Applied Forces	Earthquake	Stage Settings	Verification	Dimensioning	Bearing Capacity
Slip on Georeinforcement	Internal Stability	Global Stability	Stability			

- Standards and analysis methods

- Theory for analysis in the program "**MSE Wall**":

Stress in Soil Body	Earth Pressures	Analysis of Walls	Slope Stability	MSE Wall

- Outputs

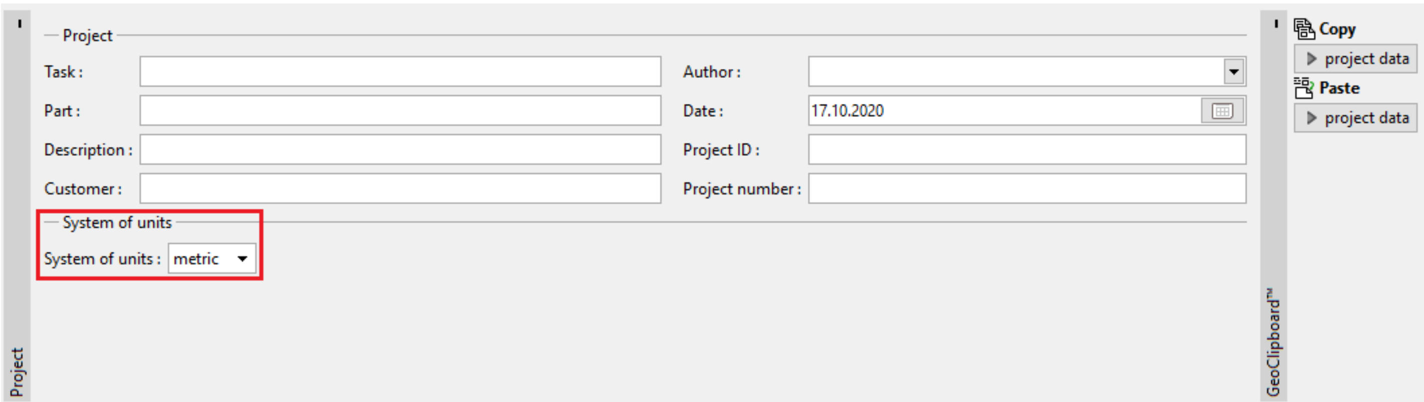
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The frame **"Project"** is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows user to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The frame "**Settings**" allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

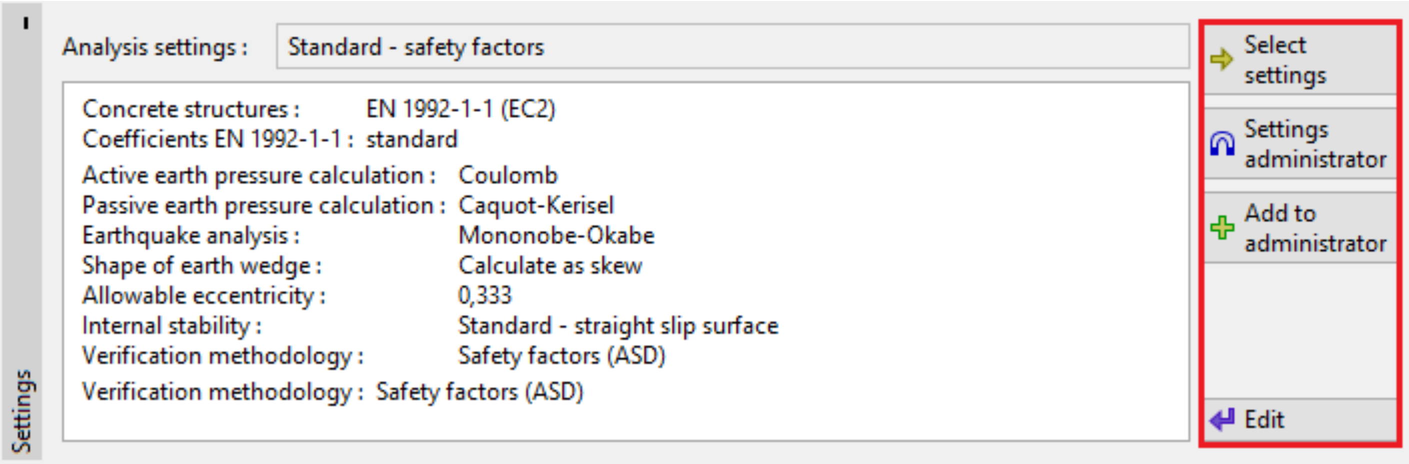
The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local**

setting. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

Settings of analysis parameters are performed in the "Materials and standards", "Wall analysis" and "Stability analysis" tabs.



Frame "Settings"

Geometry

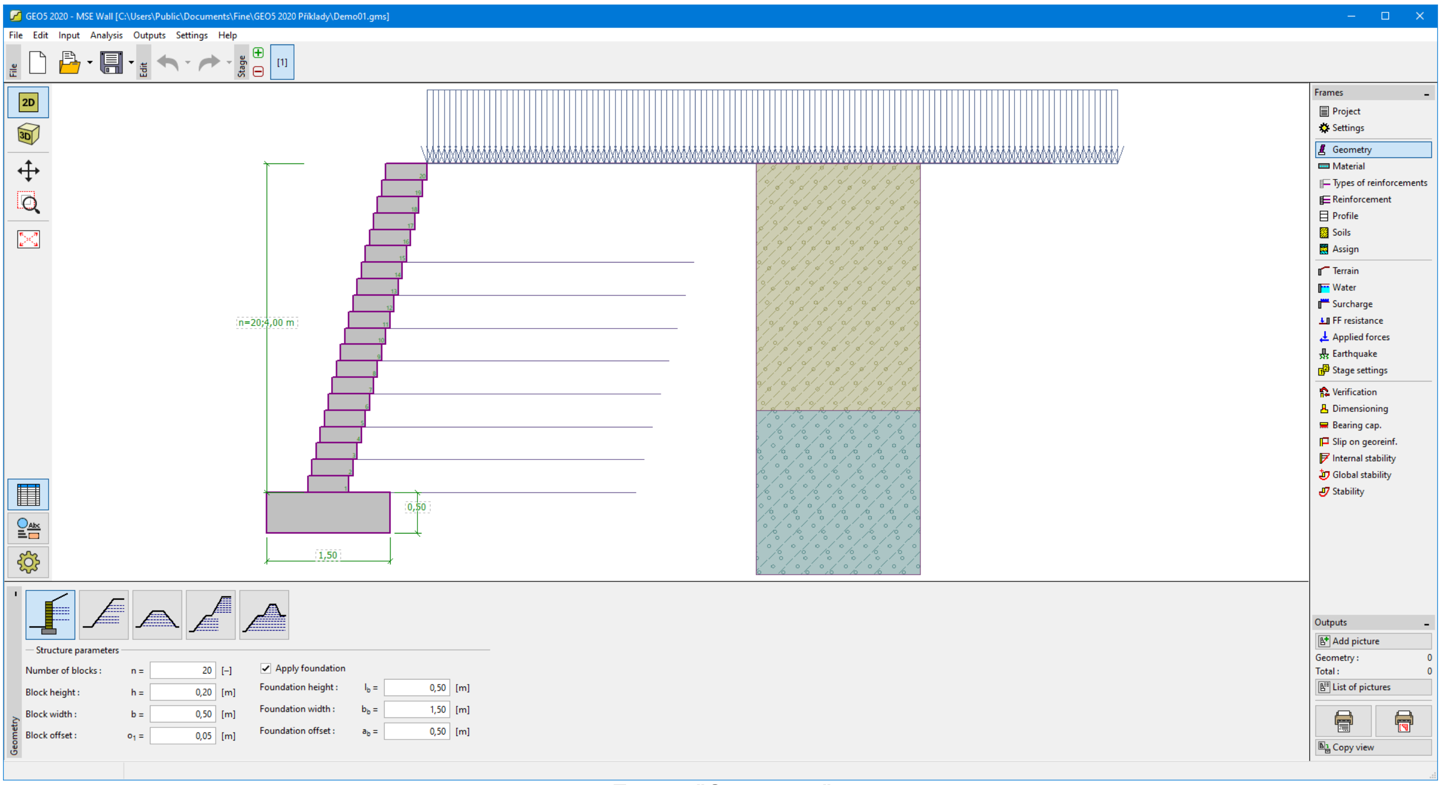
The "**Geometry**" frame allows by pressing the button for selecting the wall shape.

The shape of a wall can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The first type of geometry (wall) enables to define **foundation;** for other types (embankments) the program allows for inputting the **cover**. The selected type of geometry influences other frames and their inputting modes (water, surcharge, reinforcement). The following verification options are available for individual types of geometries:

	Geometry type	**Verification**
1	Wall with the option to define a foundation	Verification, dimensioning, bearing capacity, internal stability, reinforcement bearing capacity, global stability, slope stability
2	One-sided slope	Verification, bearing capacity, internal stability, global stability, slope stability
3	Two-sided slope	Global stability, slope stability
4	One-sided zoned by benches	Verification, bearing capacity, internal stability, global stability, slope stability
5	Two-sided zoned by benches	Global stability, slope stability

The program makes it possible to export the geometry of a structure in the *.DXF format.



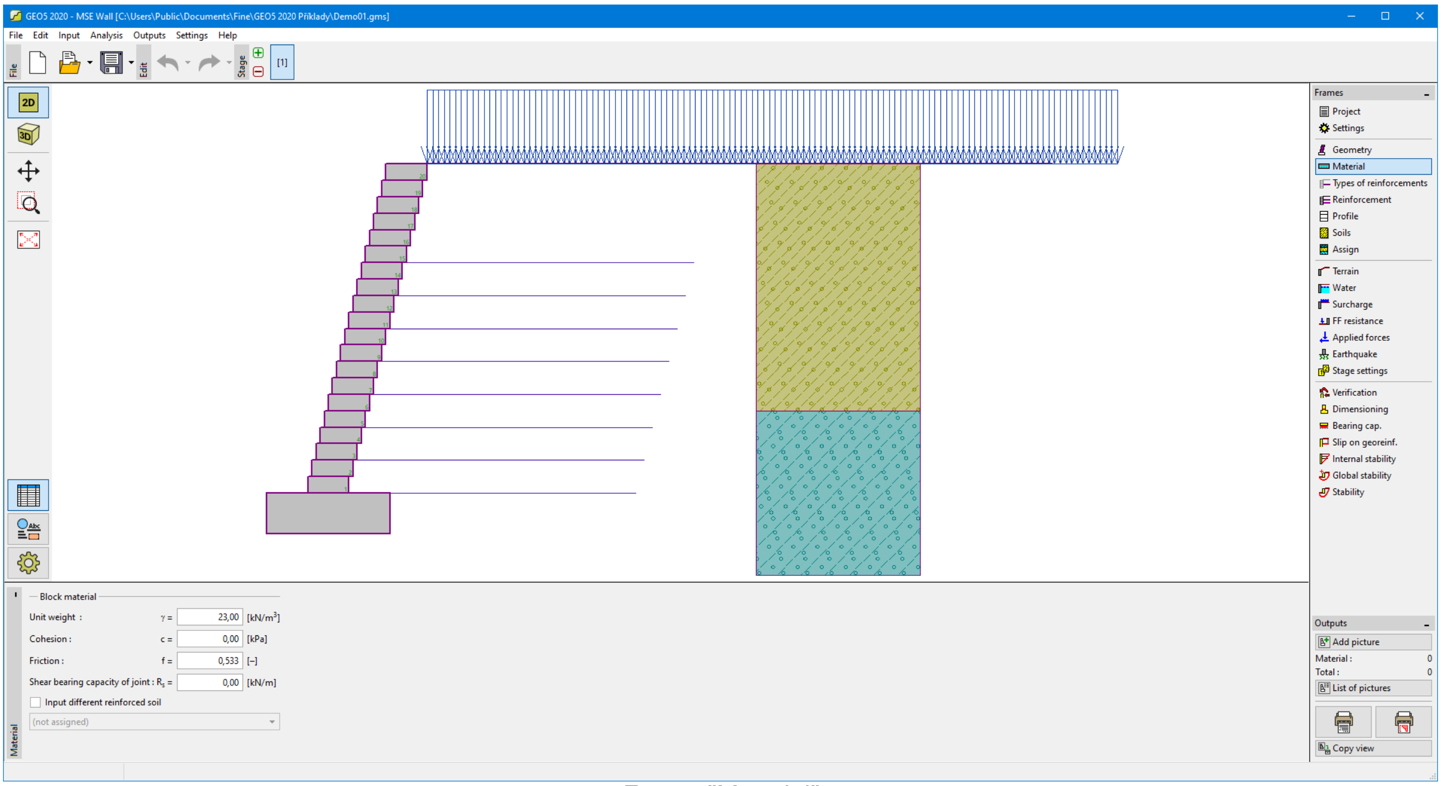
Frame "Geometry"

Material

The "**Material**" frame serves to choose parameters of the material adopted for blocks or cover. Defining materials depends on the selected type of "Geometry". The first type of geometry (structure with blocks) requires inputting the unit weight of blocks γ, cohesion c, friction f, and shear bearing capacity of joint R_S $[kN/m]$.

The other types of geometry (structure without blocks) enable to consider a cover, which requires inputting the unit weight γ and shear resistance R_S $[kPa]$.

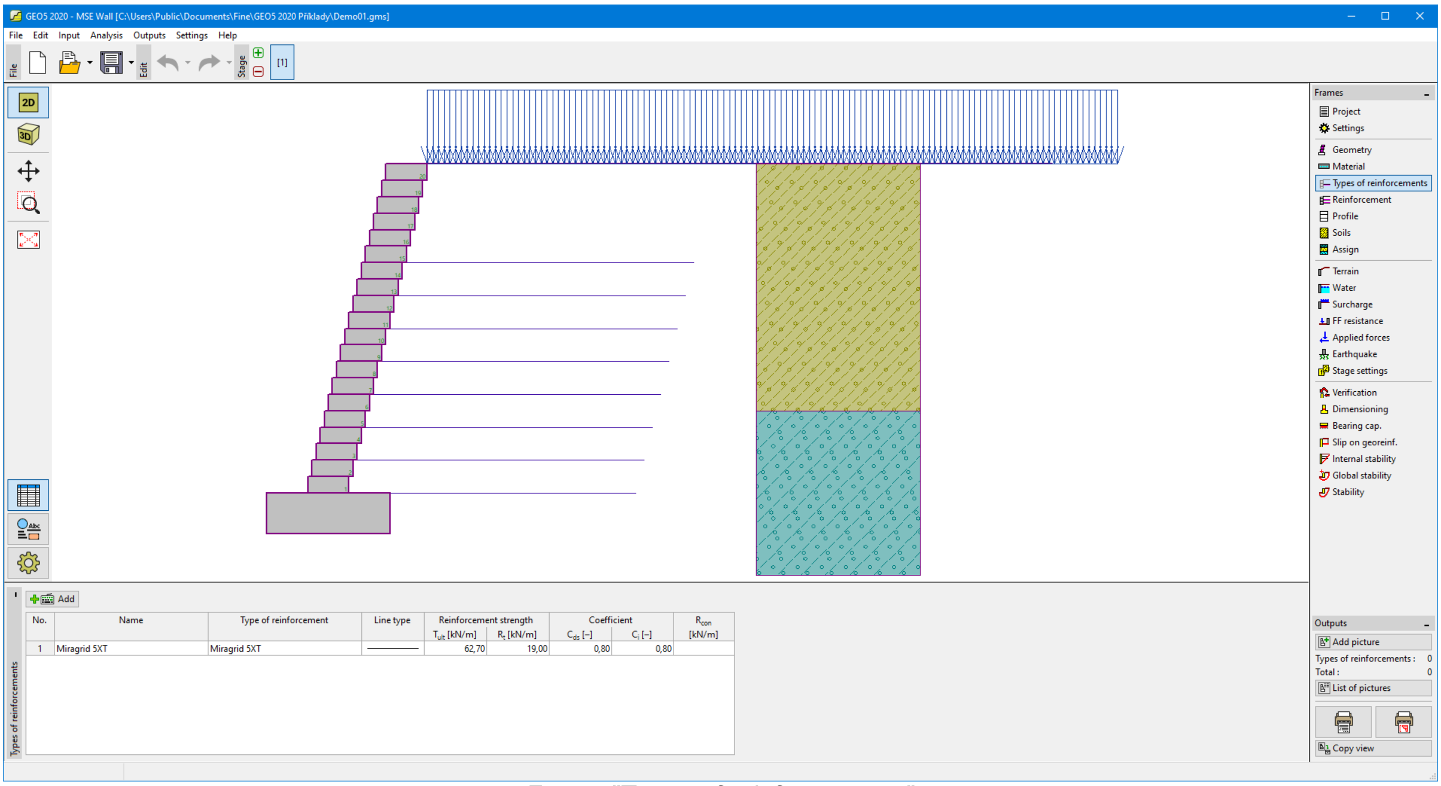
If the soil between the reinforcements is different than soil assigned to the geological profile, then the program allows us to specify this soil by checking the option "**Input different reinforced soil**". Subsequently, in combo list user selects the type of soil (the combo list contains soils introduced in the frame "Soils").



Frame "Material"

Types of Reinforcements

The "**Types of reinforcements**" frame contains a table with the list of input geo-reinforcements and their parameters (long-term bearing capacity of reinforcements and coefficients of interaction). Adding reinforcement is carried in the "New type of reinforcement" dialog window.



Frame "Types of reinforcements"

Adding and Editing Type of Reinforcement

The "**New type reinforcement (Edit type of reinforcement, Inserted type of reinforcement)**" dialog window contains the following items:

Reinforcement group and Type of reinforcement
- a combo list contains reinforcement group and individual types of reinforcements from a database, or its allows for inputting the "**user-defined**" type of reinforcement

Short-term characteristic strength
- the strength value can be changed only for reinforcements stored with the help of "User's catalog"

Analysis of long-term strength
- a combo list enables to choose the way of analysis of long-term strength: "**input reduction factors**" (direct input of factors), "**calculate reduction factors**" (factors are determined based on the selected lifetime of a reinforcement, soil Ph and grain size) or "**input strength**" (already reduced long-term strength is input)

Reduction factors
- the values of factors reducing a short-term tensile strength - can be input directly or calculated based on the selected options in combo lists (lifetime, chemistry, grain size)

Overall coefficient of model uncertainty
- the value of factor to reduce a short-term strength is input

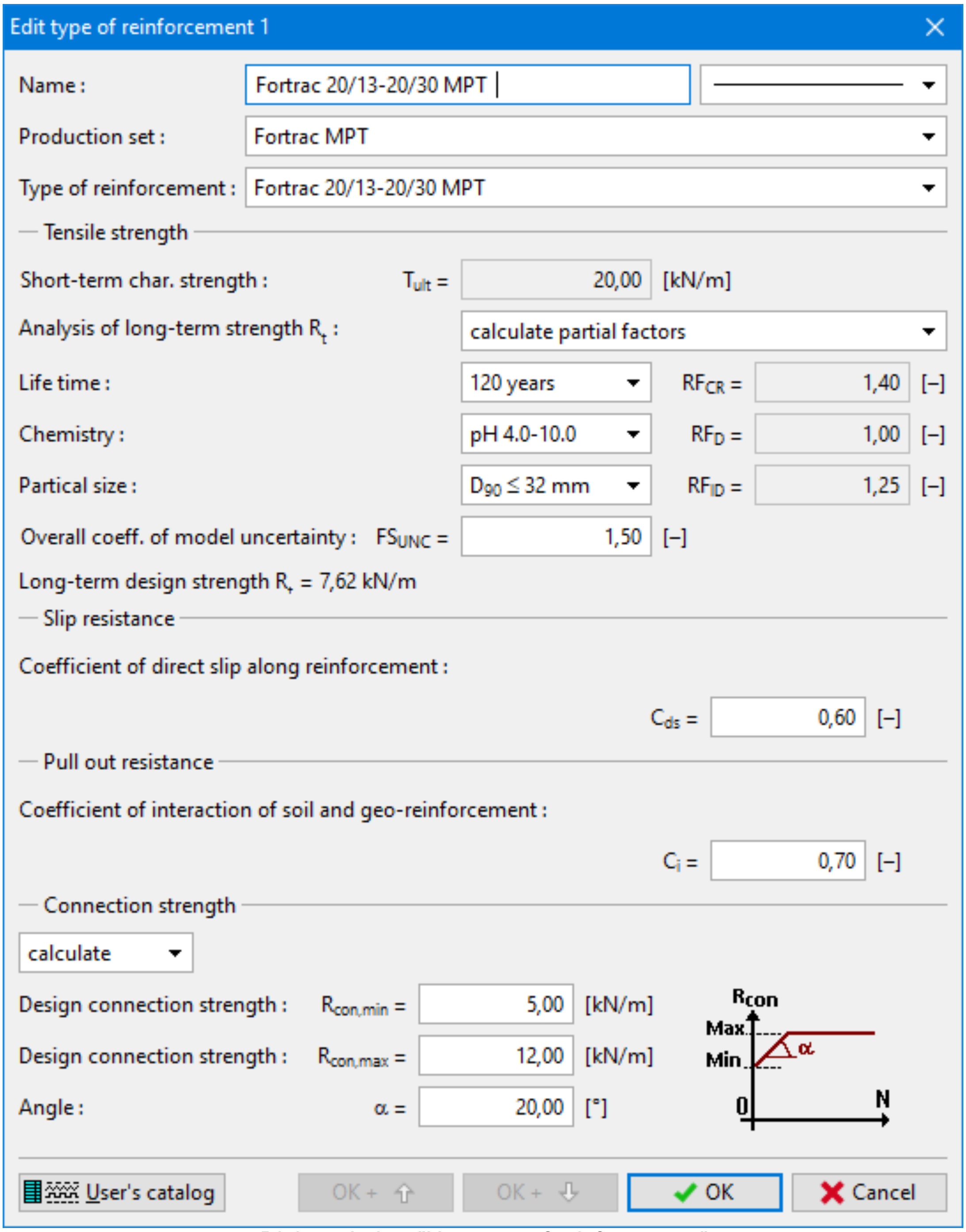
Long-term design strength
- calculated value of a long-term tensile strength

Slip resistance
- the "**Coefficient of direct slip along reinforcement**" can be input directly or calculated based on the type of soil

Pull out resistance
- the "**Coefficient of the interaction of soil and geo-reinforcement**" can be input directly or calculated based on the type of soil

Connection strength
- can be input directly or calculated based on the magnitude of normal force between blocks: $R_{con}=R_{con,min}+N.tan\alpha \leq R_{con,max}$

The "**User's catalog**" button in the bottom part of the window opens the "User's catalog" dialog window.

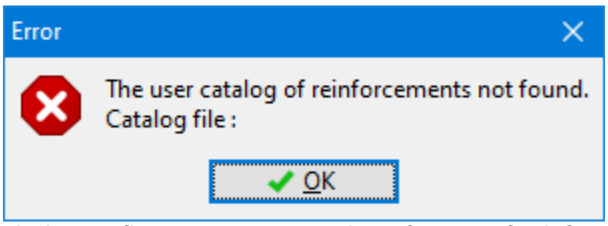


Dialog window "New type of reinforcement"

User's Catalog

The user's catalog allows us to define and store user-defined types of reinforcements and their material characteristics. At first use of the catalog (has not been yet created) the program prompts a warning message that no catalog was found. Then, pressing the button "**OK**" opens the "**Save as**" dialog window that allows for entering the catalog name and saving it into a specified location by pressing the "**Save**" button (by default, a folder used for saving the project data is assumed).

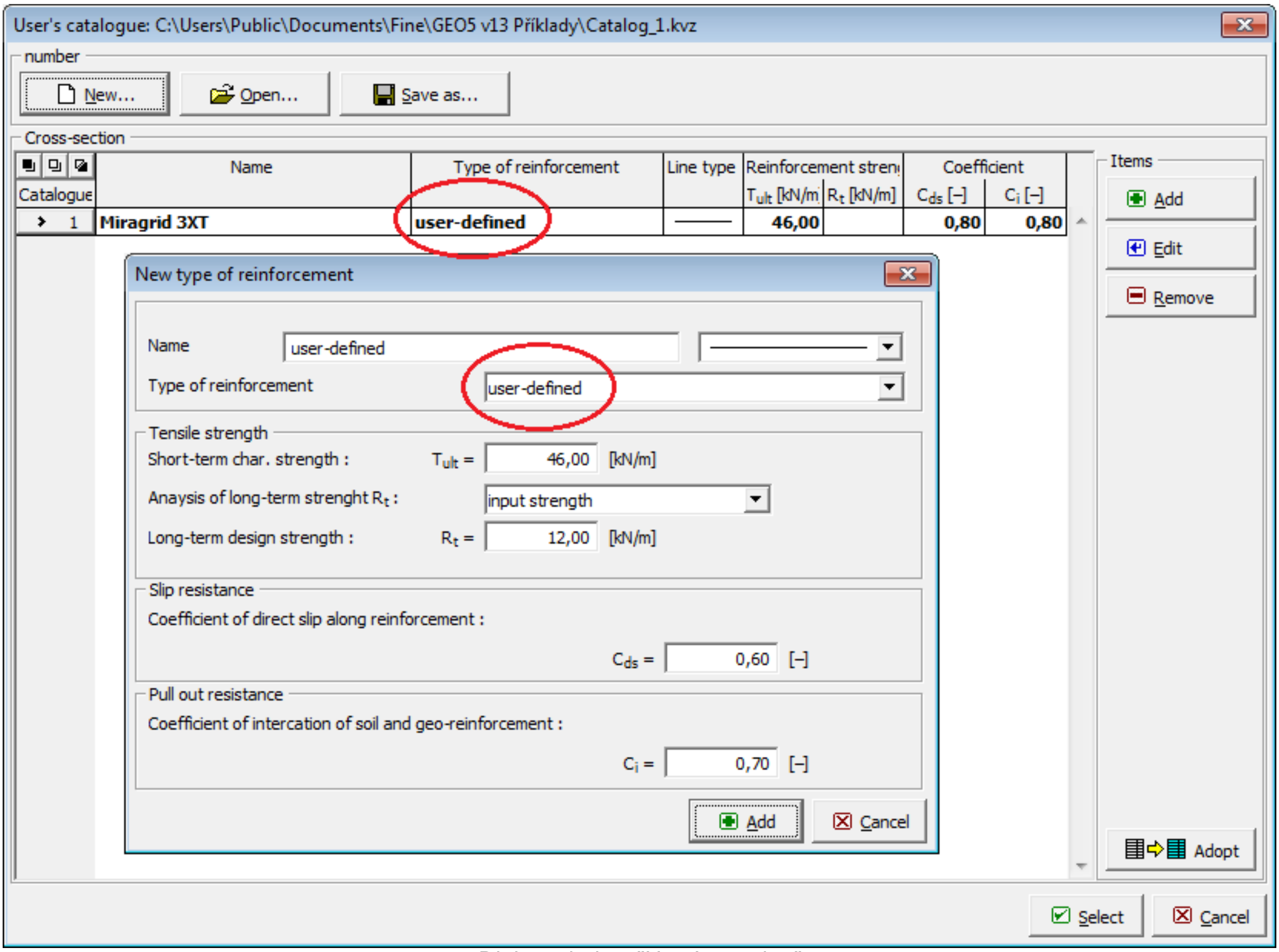
The program allows the user to create more than one catalog. The next catalog is created by pressing the "**New**" button - the program asks, whether the current catalog should be replaced (**the currently loaded catalog is not DELETED!**) and saves the new catalog under a new name. The "**Open**" button opens any user catalog.



Dialog window at first use - user catalog of types of reinforcements

The "**User catalog**" dialog window contains a table listing the User-defined reinforcements. The "**Add**" button opens the "New type of reinforcement" dialog window that allows for specifying and subsequent saving of characteristics of a new reinforcement into the catalog. Buttons "**Edit**" and "**Remove**" serve to edit individual items in the table.

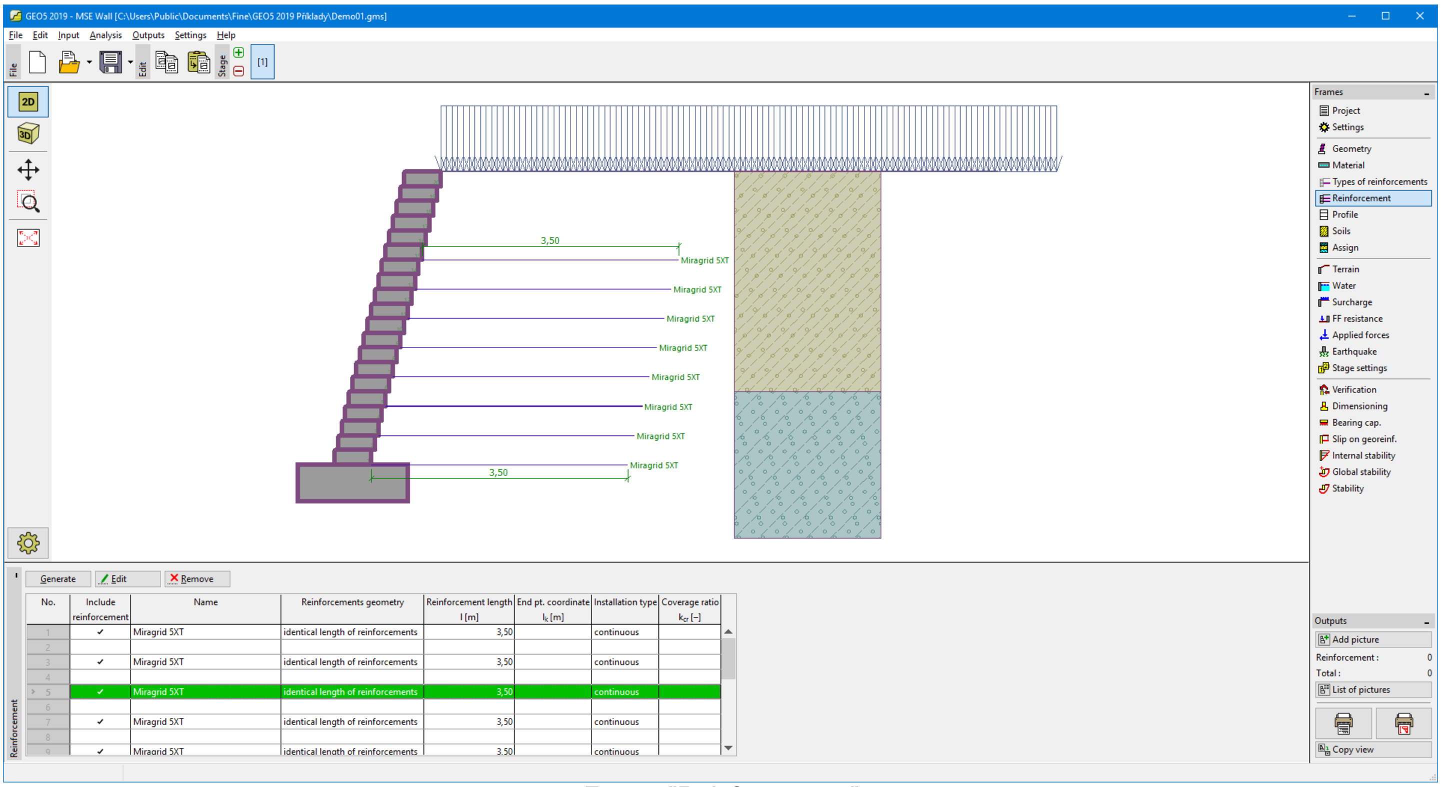
The "**Adopt**" button accepts the current reinforcement characteristics specified in the "New type of reinforcement" dialog window and opens the "**New type of reinforcement**" dialog window that allows for their modification and saving.



Dialog window "User's catalog"

Reinforcement

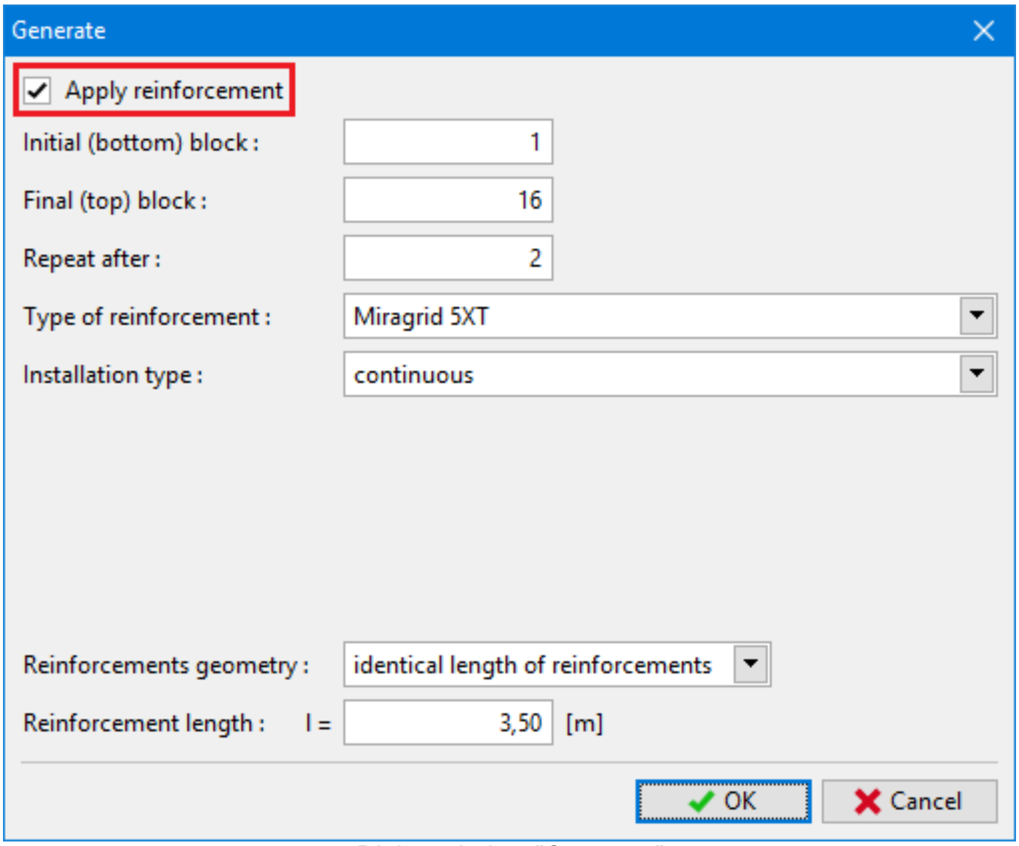
The "**Reinforcement**" frame contains a table with the list of input geo-reinforcements and their geometries.



Frame "Reinforcement"

The "**Generate**" button opens the "**Generate**" dialog window that enables to set automatic parameters of generating a group of reinforcements. Geo-reinforcements can be positioned only in joints between the blocks (checking the option "**Apply reinforcement**"). The next step is to define the type of reinforcement, installation type (continuous or strip), the initial and the last block, the number of blocks to reenter the reinforcement, reinforcement geometry (the same length of reinforcements or the same type of reinforcement finishing). The input reinforcements can also be edited on the desktop with the help of active dimensions or active objects, respectively.

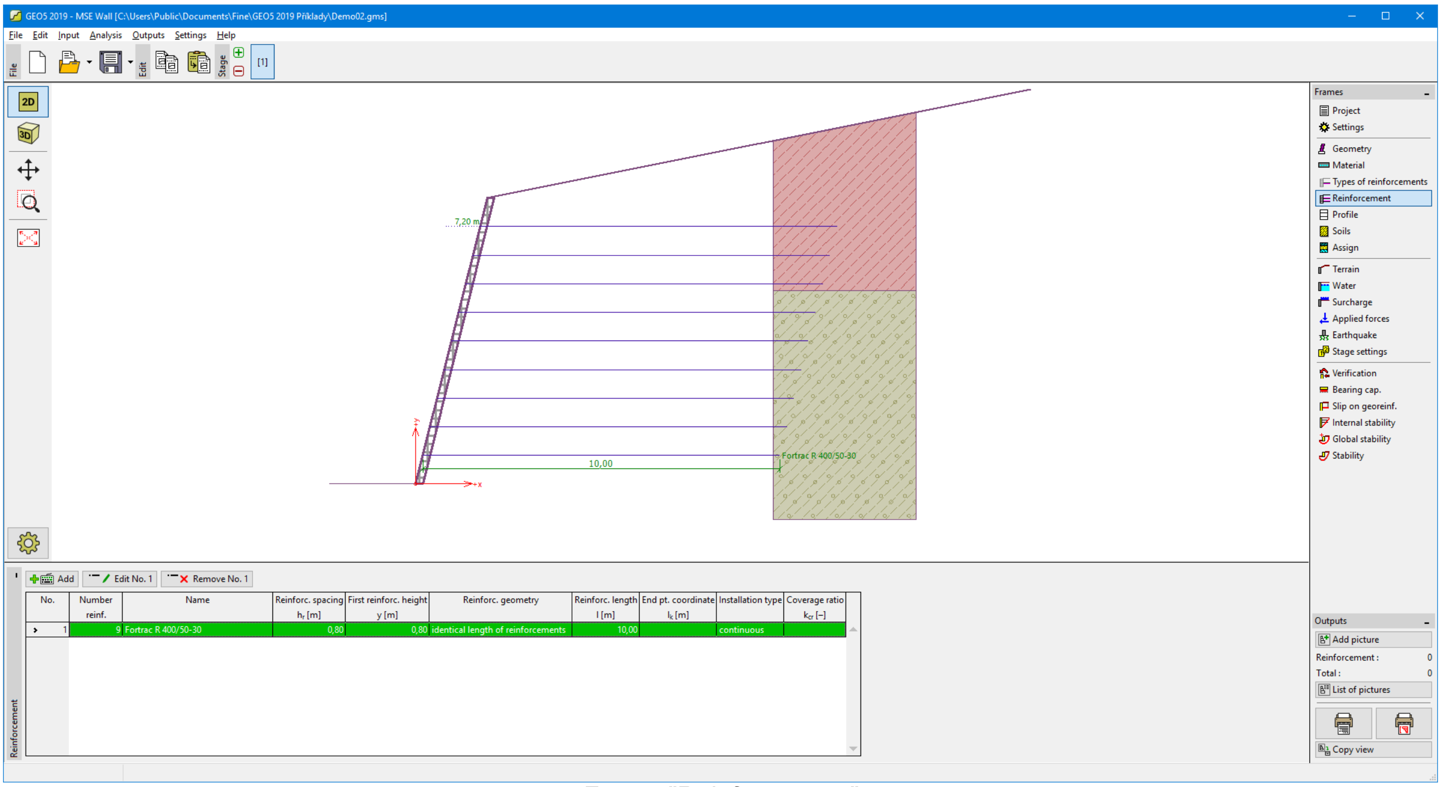
The "**Edit**" button opens the "**Edit block**" dialog window that enables to change the type of reinforcement, its geometry or to specify whether the reinforcement between the blocks is to be considered. The "**Remove**" button removes **all** geo-reinforcements.



Dialog window "Generate"

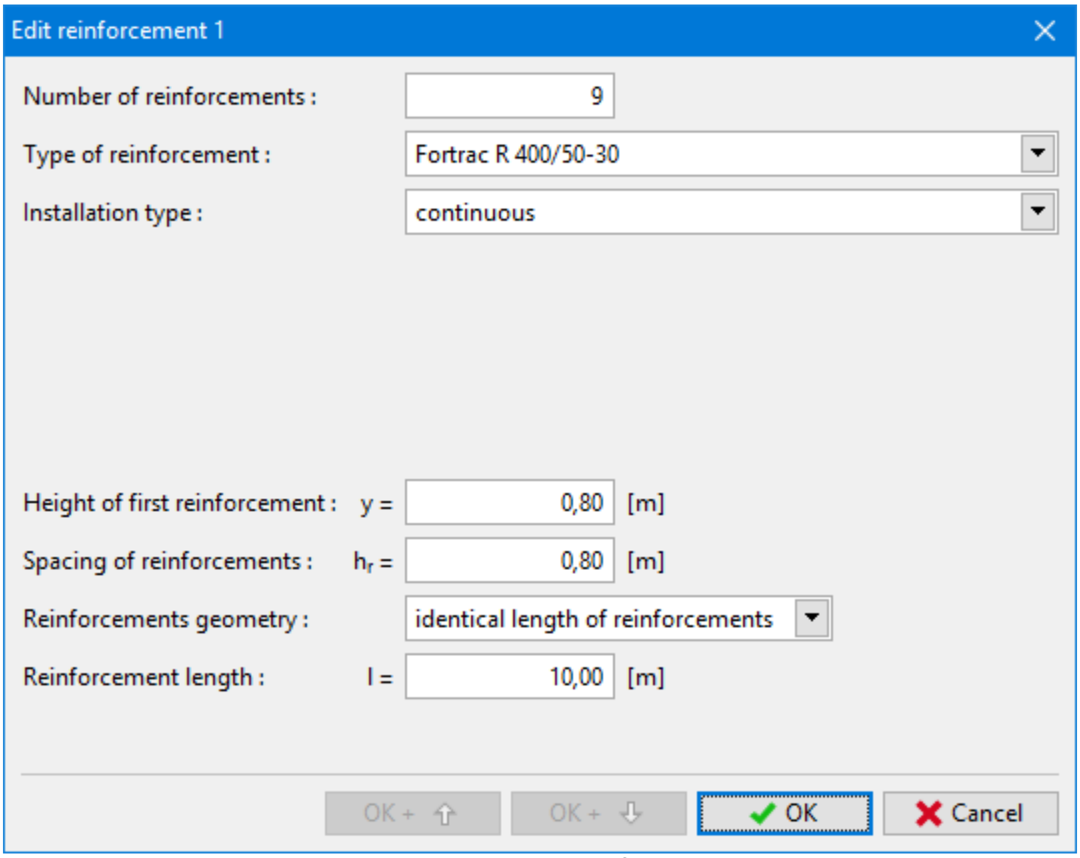
Reinforcement

The "**Reinforcement**" frame contains a table with the list of input groups of reinforcements and their geometries.



Frame "Reinforcement"

Adding groups of geo-reinforcements is performed in the "**New reinforcement**" dialog window. The input reinforcements can also be edited on the desktop with the help of active dimensions or active objects, respectively. Each input group of reinforcements requires input of the number of reinforcements and type, installation type (continuous or strip), height of the first reinforcement, reinforcement spacing, and their geometry.



Dialog window "Edit reinforcement"

Profile

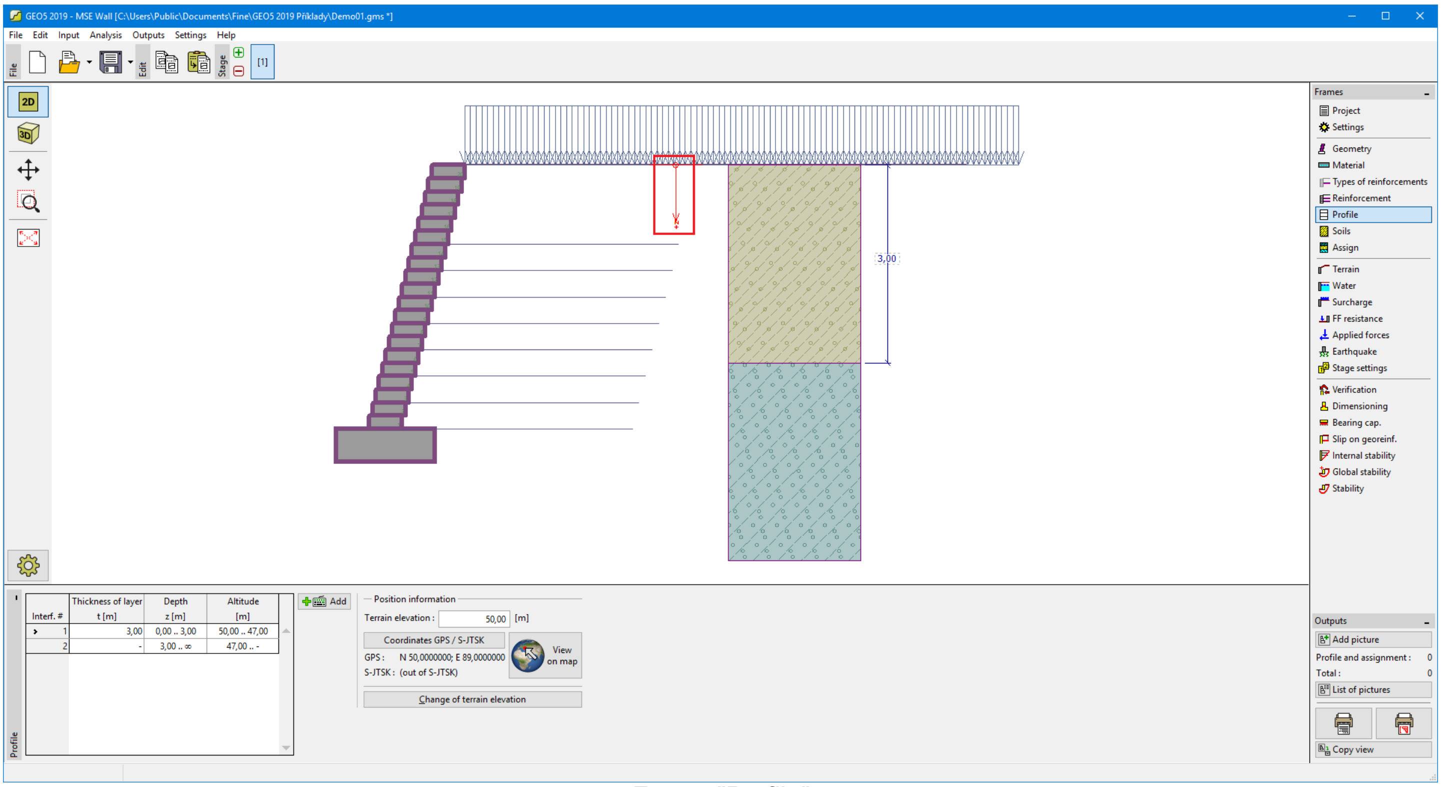
The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.



Frame "Profile"

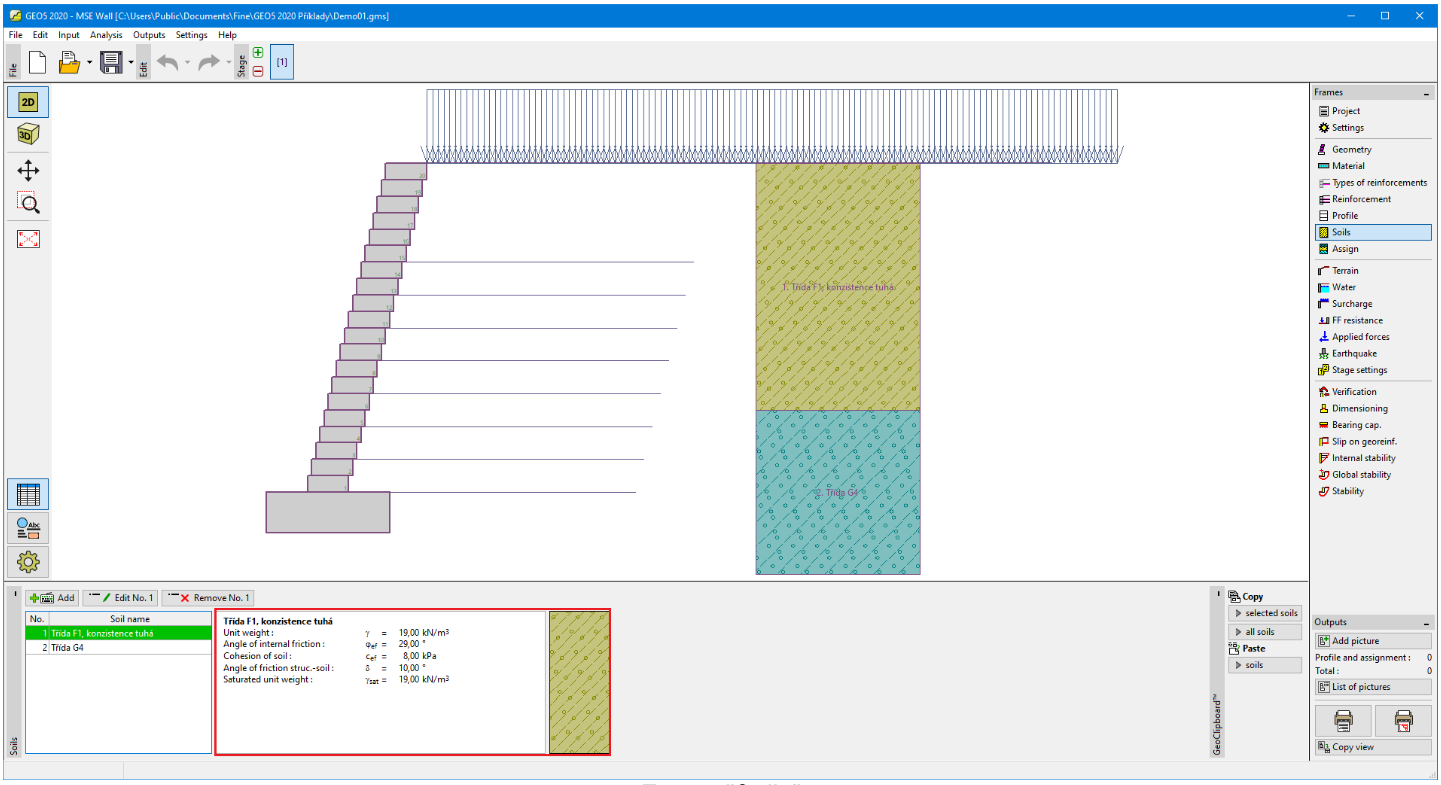
Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data" and "Uplift pressure".

Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



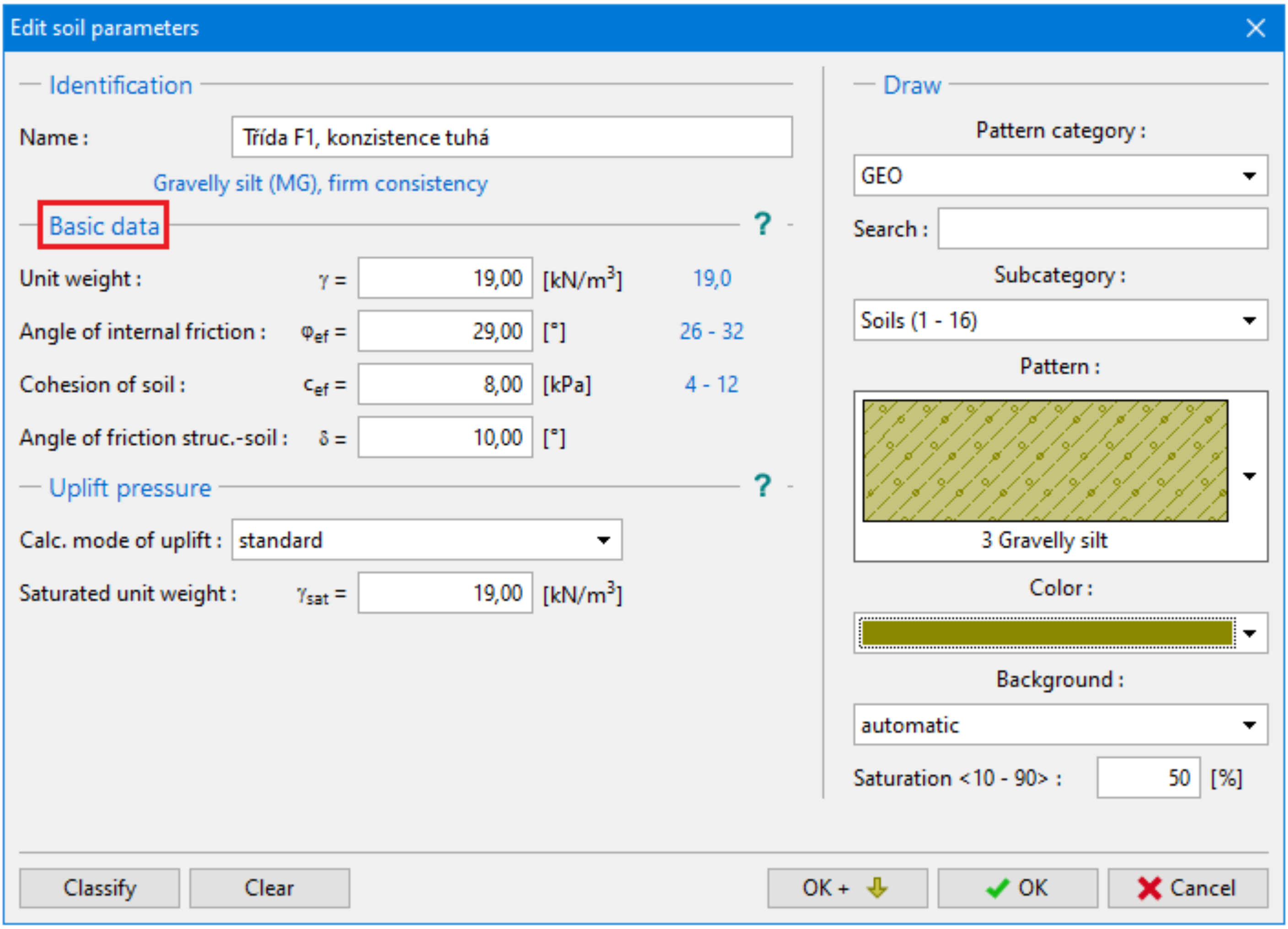
Frame "Soils"

Basic Data

This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction, and cohesion**. The particular values are obtained from a geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit the built-in database of soils, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually.

One further needs to specify the angle of internal friction between the soil and structure, which depends on the structure material and the type of soil. Possible values of this parameter are listed in the table of recommended values.

The associated theory is described in detail in the chapter "Earth pressures".

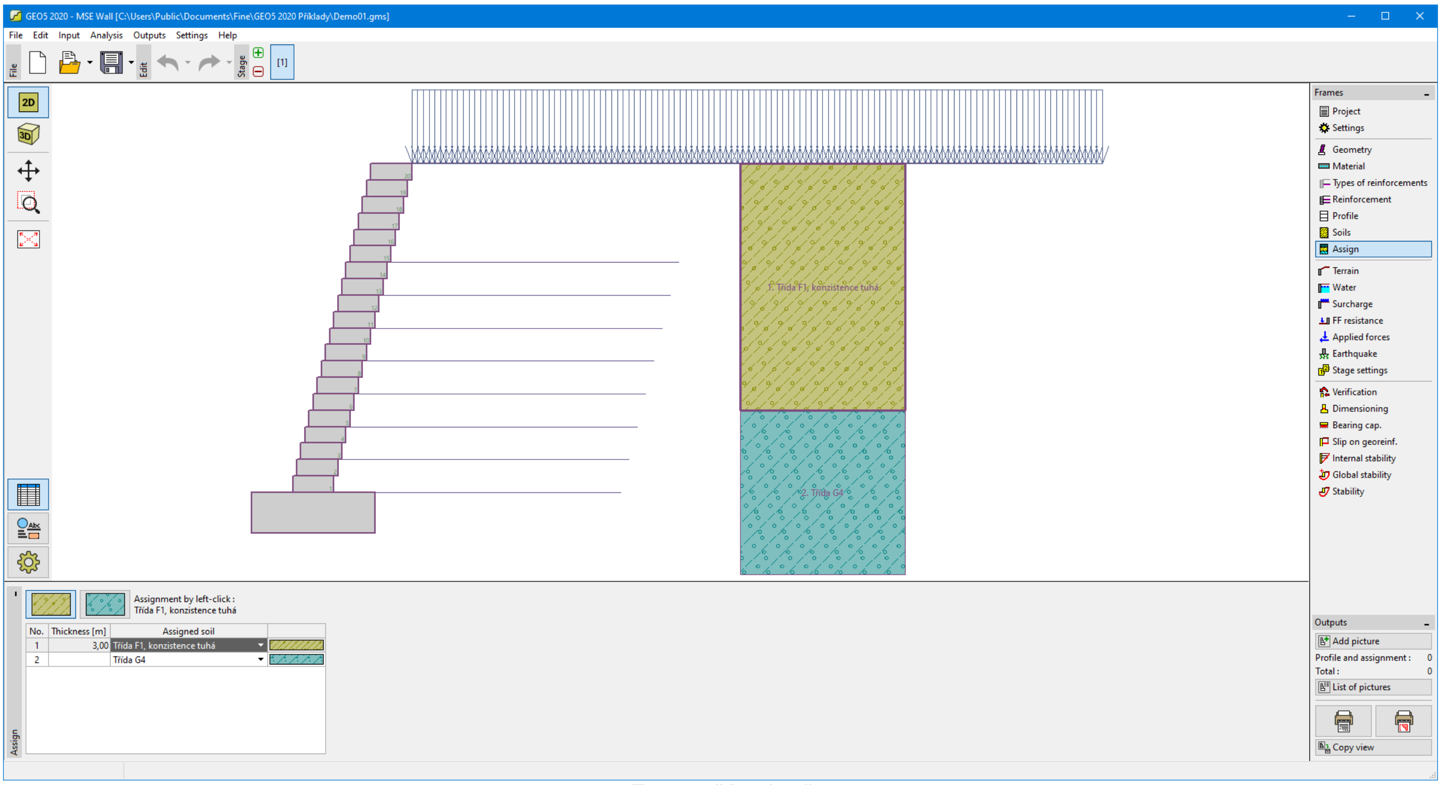


Dialog window "Add new soils" - "Basic data"

Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail herein.



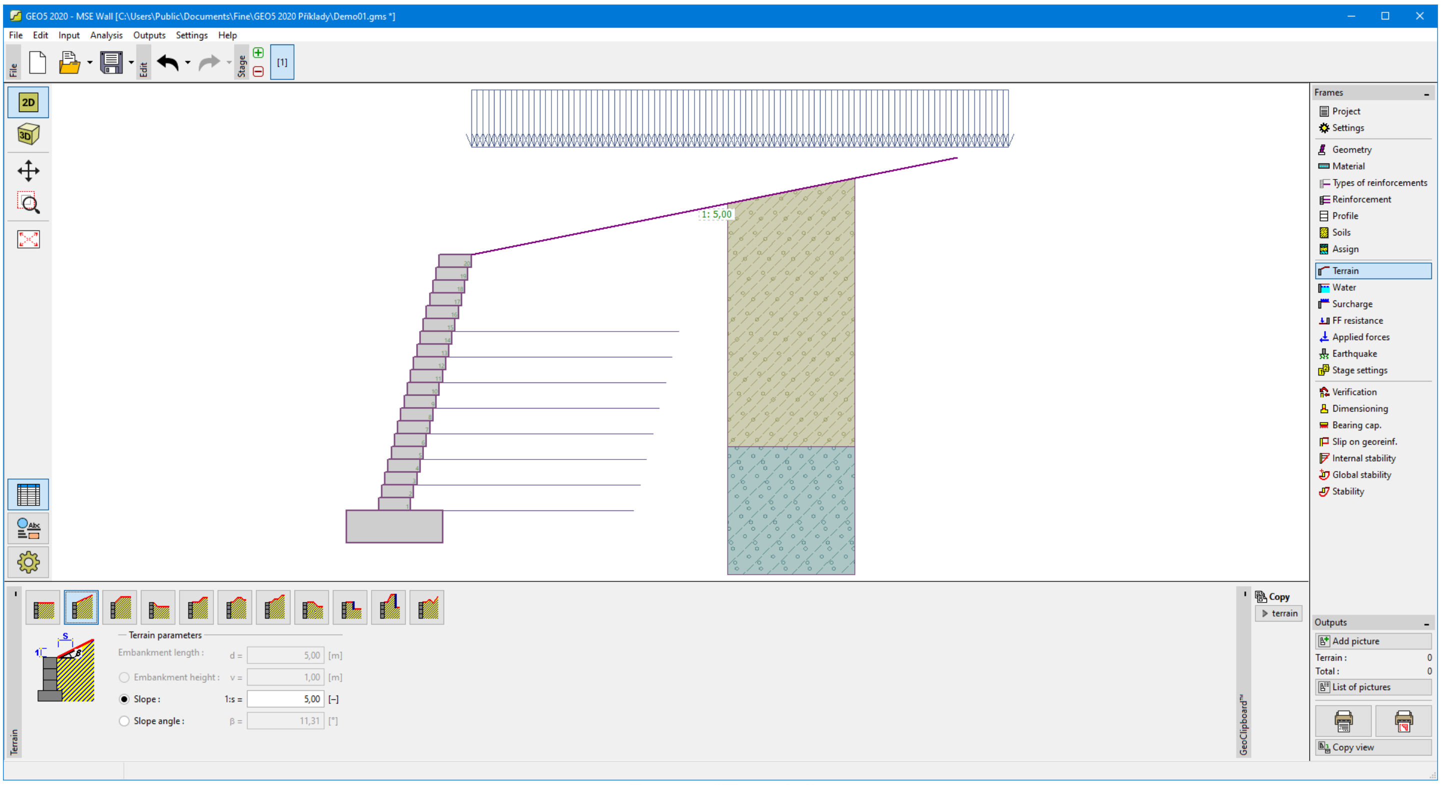
Frame "Assign"

Terrain

The "**Terrain**" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case, the frame contains a table with a list of terrain points. The first point with coordinates [0, 0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "Distribution of earth pressures for broken terrain".



Frame "Terrain"

Water

The "**Water**" frame allows, by pressing the button, for selecting the type of water. The selected type with a graphic hint of input values is displayed in the left part of the frame. Water parameters (h_1 , h_2 ...) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

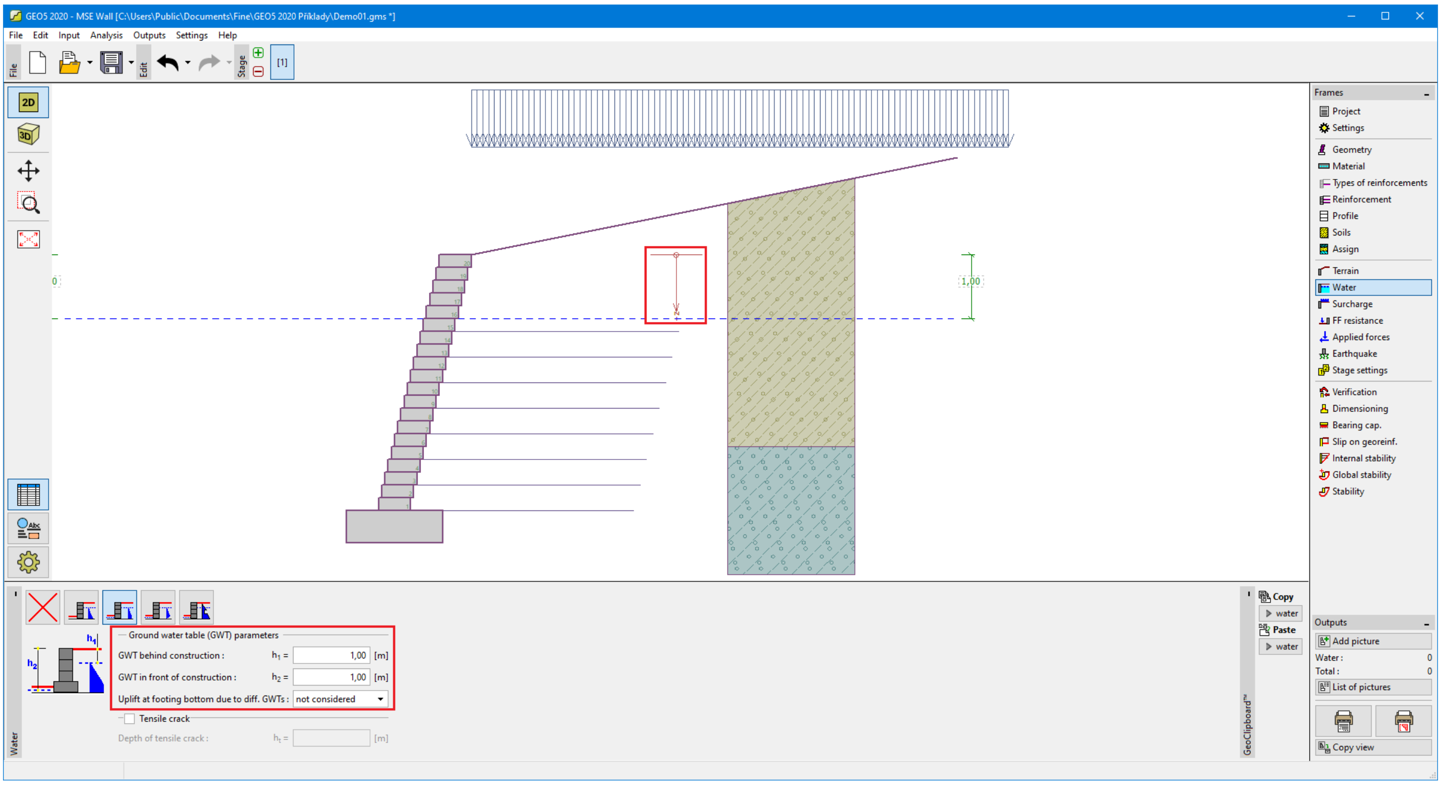
The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be linear, parabolic, or it may not be considered at all. When verifying the wall, the uplift pressure in the base of footing due to the different water tables is introduced as a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tabs "**In front of structure**" and "**Behind structure**" appear with tables. The table contains the values of pore pressure in front of the structure or behind it.

The groundwater table can also be specified **above the structure** or earth profile, respectively - in such a case, the depth of water is input with a negative value.

Analysis of earth pressures with the influence of water is described in the theoretical part of the help, chapter "Influence of water".

The program further allows for specifying a depth of tensile cracks filled with water.

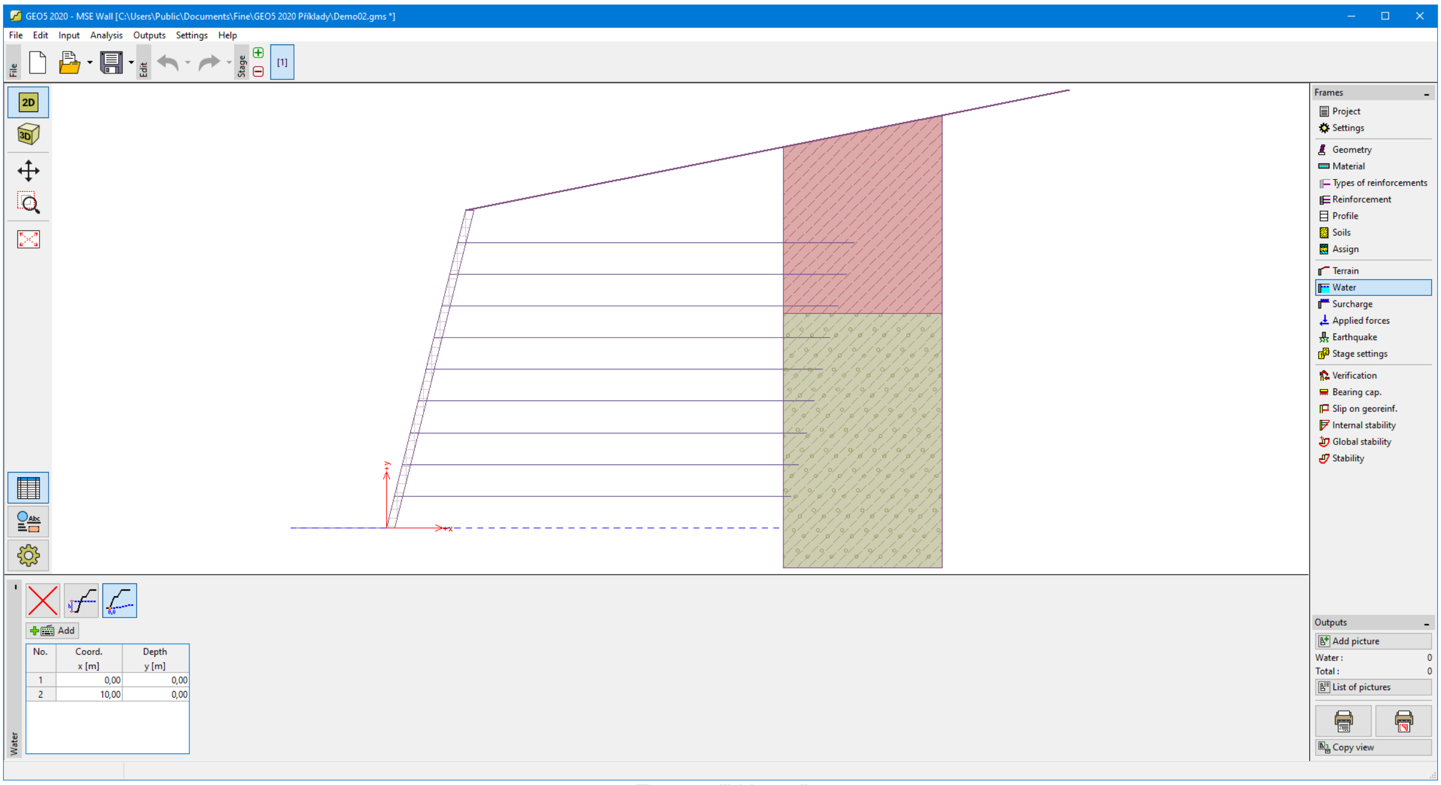


Frame "Water"

Water

The "**Water**" frame allows for selecting the type of water. The groundwater table can be specified in two ways. The first option allows us to input the height of groundwater table. The second option enables to define an arbitrary shape of the groundwater table by coordinates.

The water parameters (water table height, coordinates of points) can be edited either in the frame by inserting values into the input fields or on the desktop using the active dimensions or active blocks.



Frame "Water"

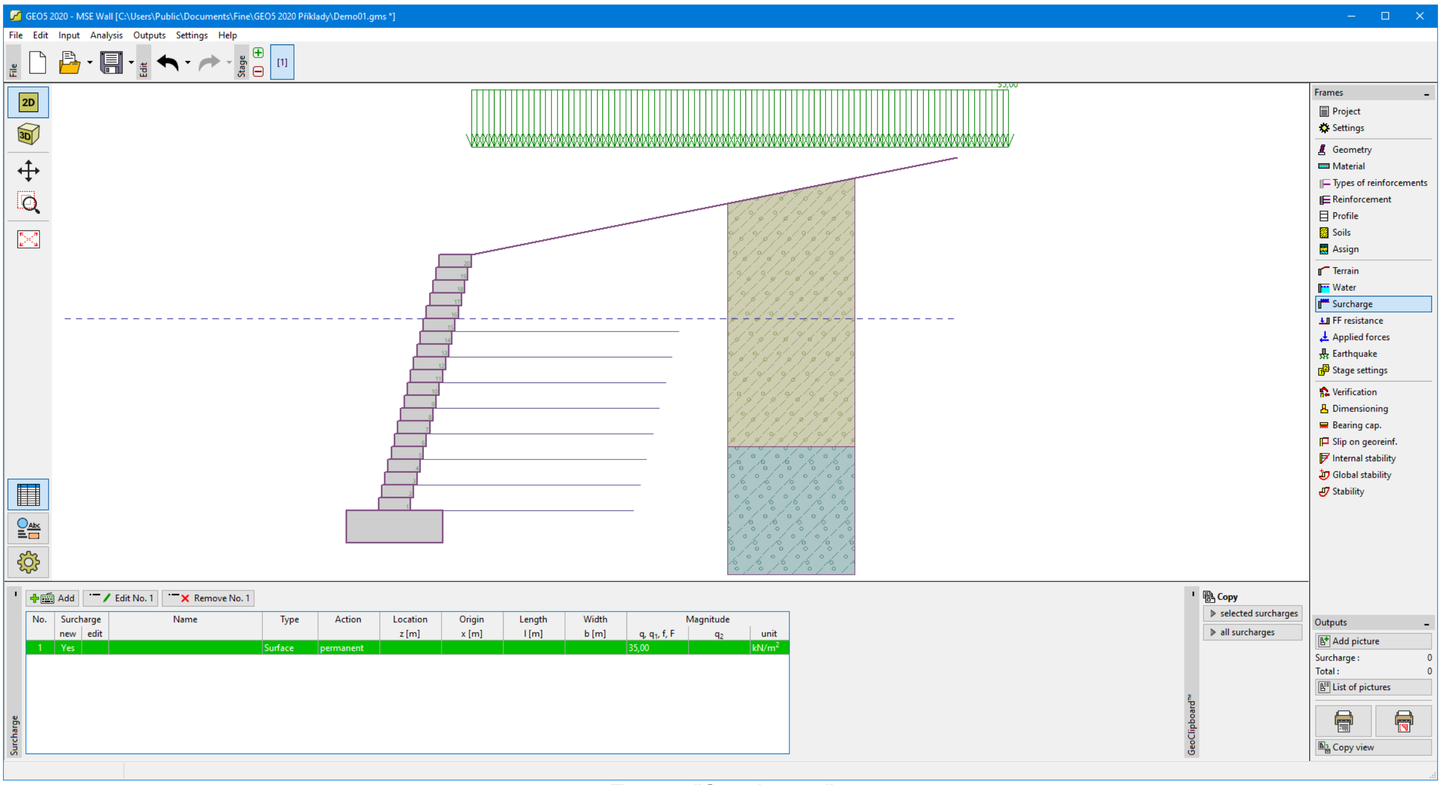
Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The z -coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case when the surcharge is found out of the terrain the program prompts an error message before calculation.

Either **permanent**, **variable,** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with a favorable effect is not considered in the analysis.

Analysis of earth pressures due to surcharges is described in the theoretical part of the help, chapter "Influence of surcharge".



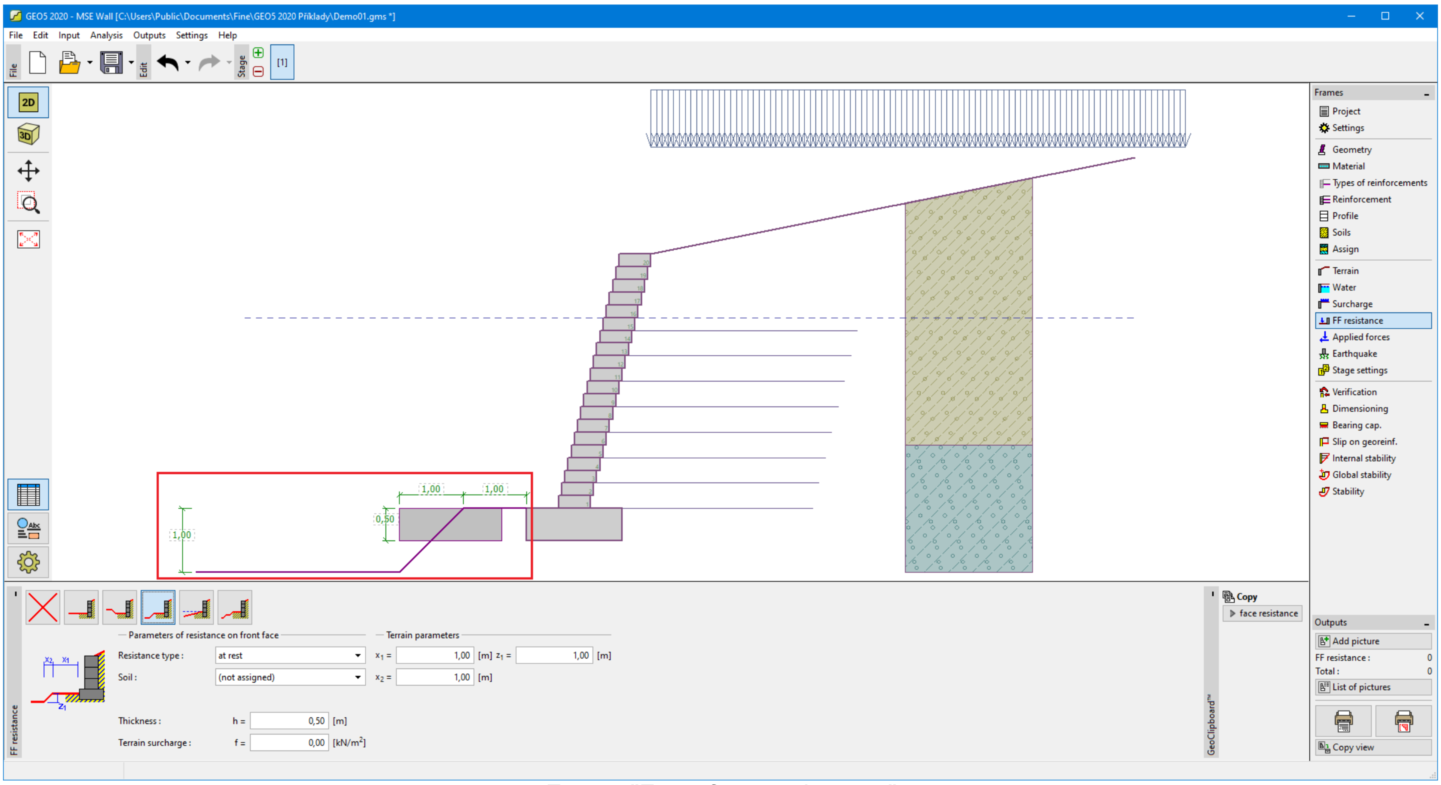
Frame "Surcharge"

Front Face Resistance

The "**Front face resistance**" frame allows us to specify the terrain shape in front of the structure and parameters of front face resistance. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop using active dimensions.

Combo lists in the frame allow selecting the type of resistance and a soil (from the soils introduced in the "Soils" frame). The terrain surcharge magnitude and soil thickness (above the wall lowest points) can also be specified.

The resistance on a structure front face can be specified as the pressure at rest, passive pressure, or reduced passive pressure. The resulting force due to the reduced passive pressure is calculated as a resultant force caused by pressure at rest and passive pressure multiplied by corresponding coefficients.



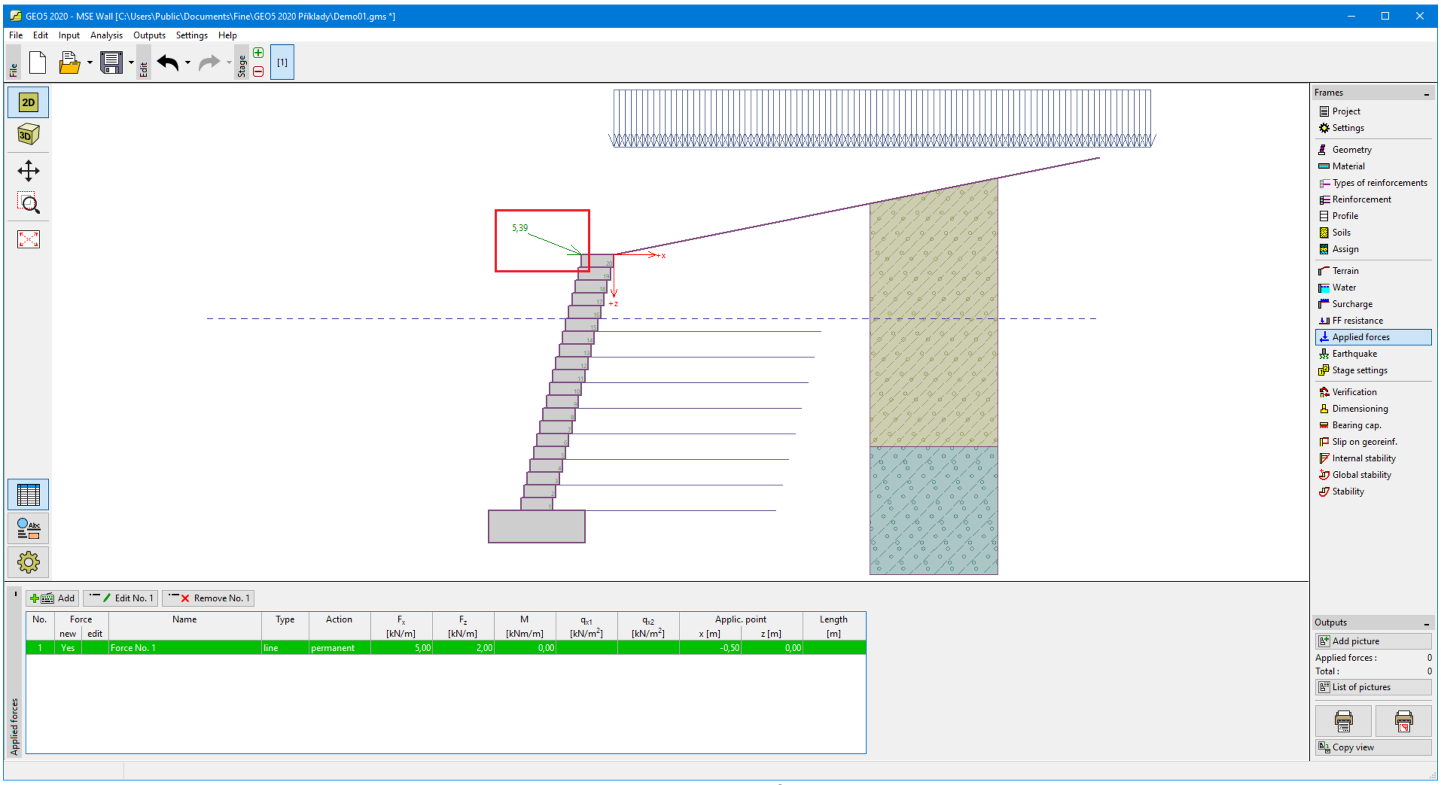
Frame "Front face resistance"

Applied Forces

The "**Applied forces**" frame contains a table with a list of forces acting on a structure. Adding forces is performed in the "**New force**" dialog window. The input forces can also be edited on the desktop with the help of active objects.

Applied forces represent an additional load on the structure of the wall, sheeting or MSE wall. We can model such as an anchoring crash barrier, crash vehicle, load from billboards and hoardings etc. The program does not modify the applied forces in the calculation except multiplying them with the corresponding coefficients according to the selected type of load (EN1997, LRFD).

External load acting on the terrain is necessary to define as a surcharge.



Frame "Applied forces"

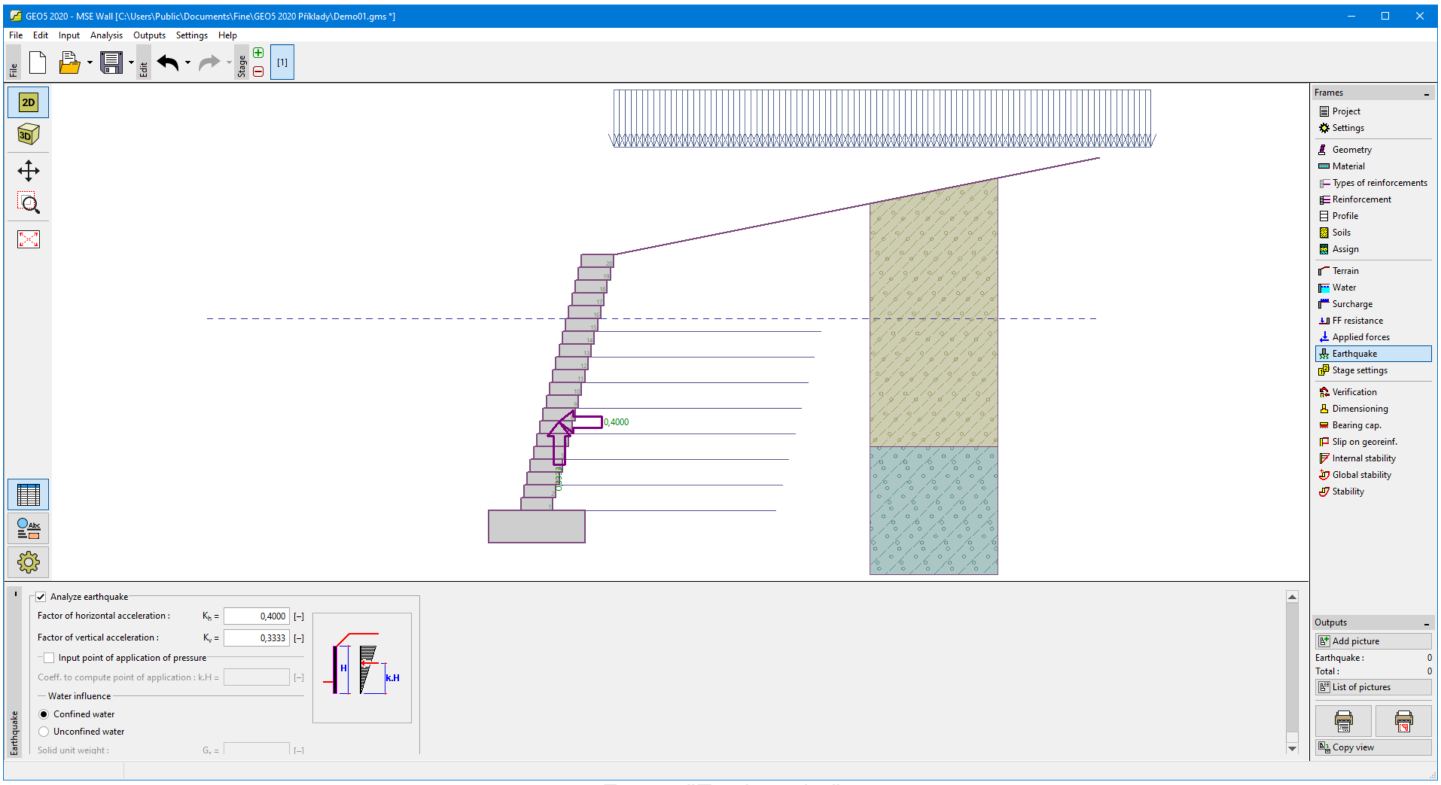
Earthquake

The "**Earthquake**" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements, the coefficients k_h and k_v can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for an earthquake is described in the theoretical part of the help, chapter "Influence earthquake".

For the LRFD Verification methodology, it is possible to define coefficients for seismic combinations according to the AASHTO.



Frame "Earthquake"

Stage Settings

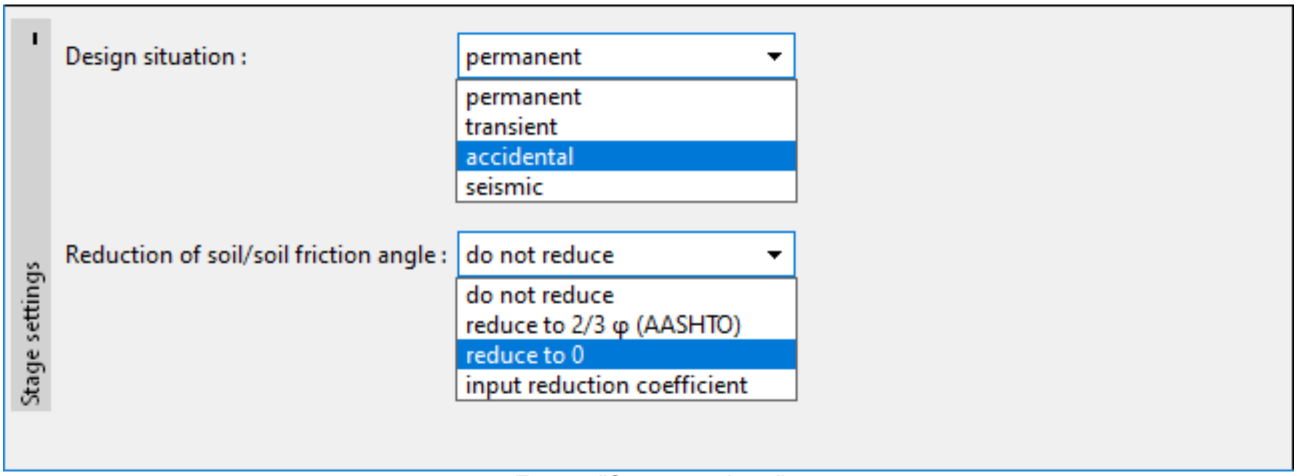
The frame "**Stage settings**" serves to input settings valid for a given construction stage.

The selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology.

The reduction of soil/soil friction angle can be considered in one of following ways:

- **do not reduce**
- **reduce to 2/3φ (AASHTO)**
- **reduce to 0**
- **input reduction coefficient**



Frame "Stage settings"

Verification

The frame "**Verification**" shows the analysis results.

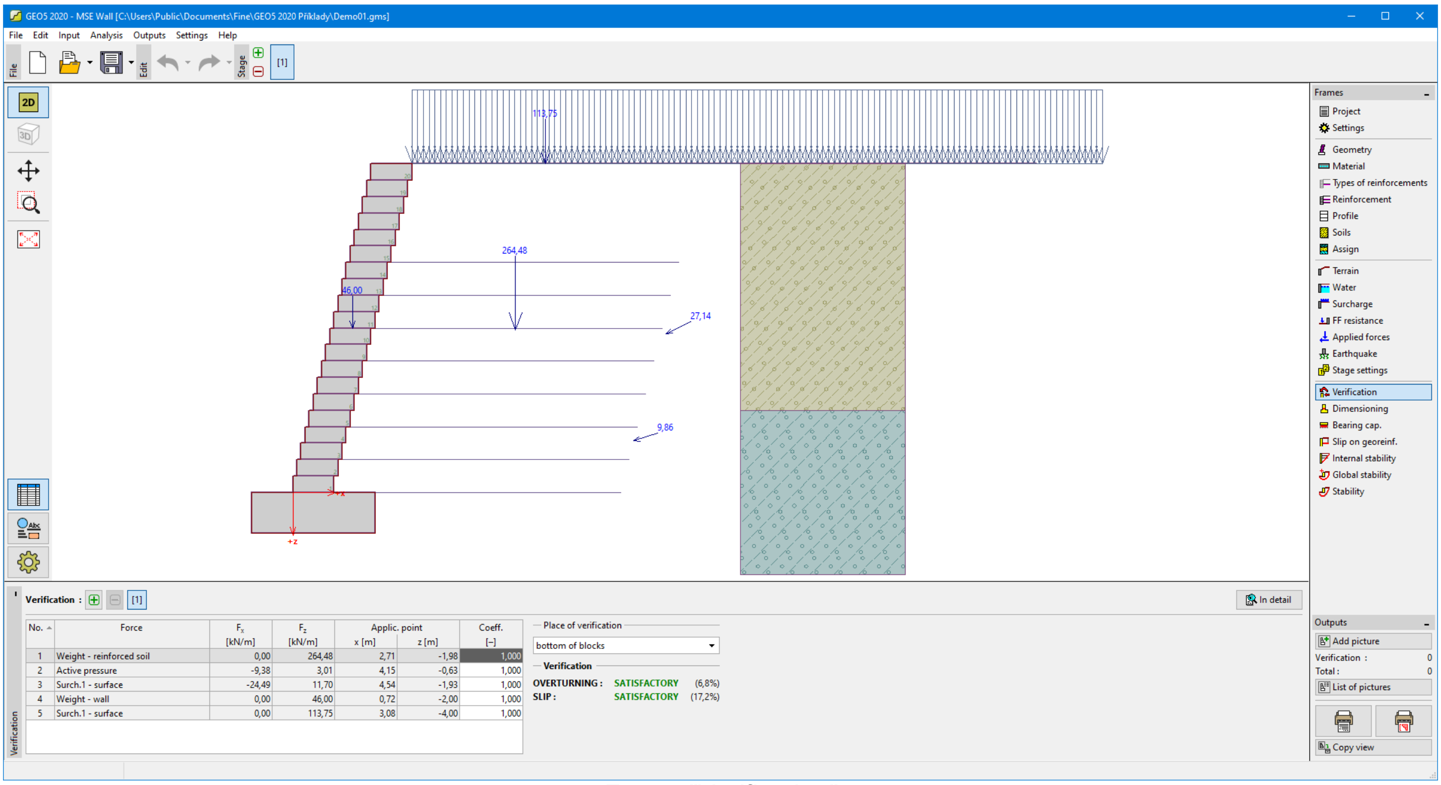
To perform verification of external stability, the program creates a **fictitious structure** (wall), which is then checked for **overturning and slip**. The fictitious wall consists of the structure front face and the end points of geo-reinforcements. An active earth pressure loads the fictitious structure. The actual verification procedure is described in the theoretical part of the help.

The frame appearance is adjusted based on the selected verification methodology:

- Verification according to the factor of safety or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows for specifying whether the load acting on a structure is considered as the secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in case the last column is not displayed.
- Analysis according to BS8006

Several computations can be carried out for a single task. The computed forces are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against **overturning and slip**. The "**In detail**" button opens the dialog window, which contains a detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Verification"

Dimensioning

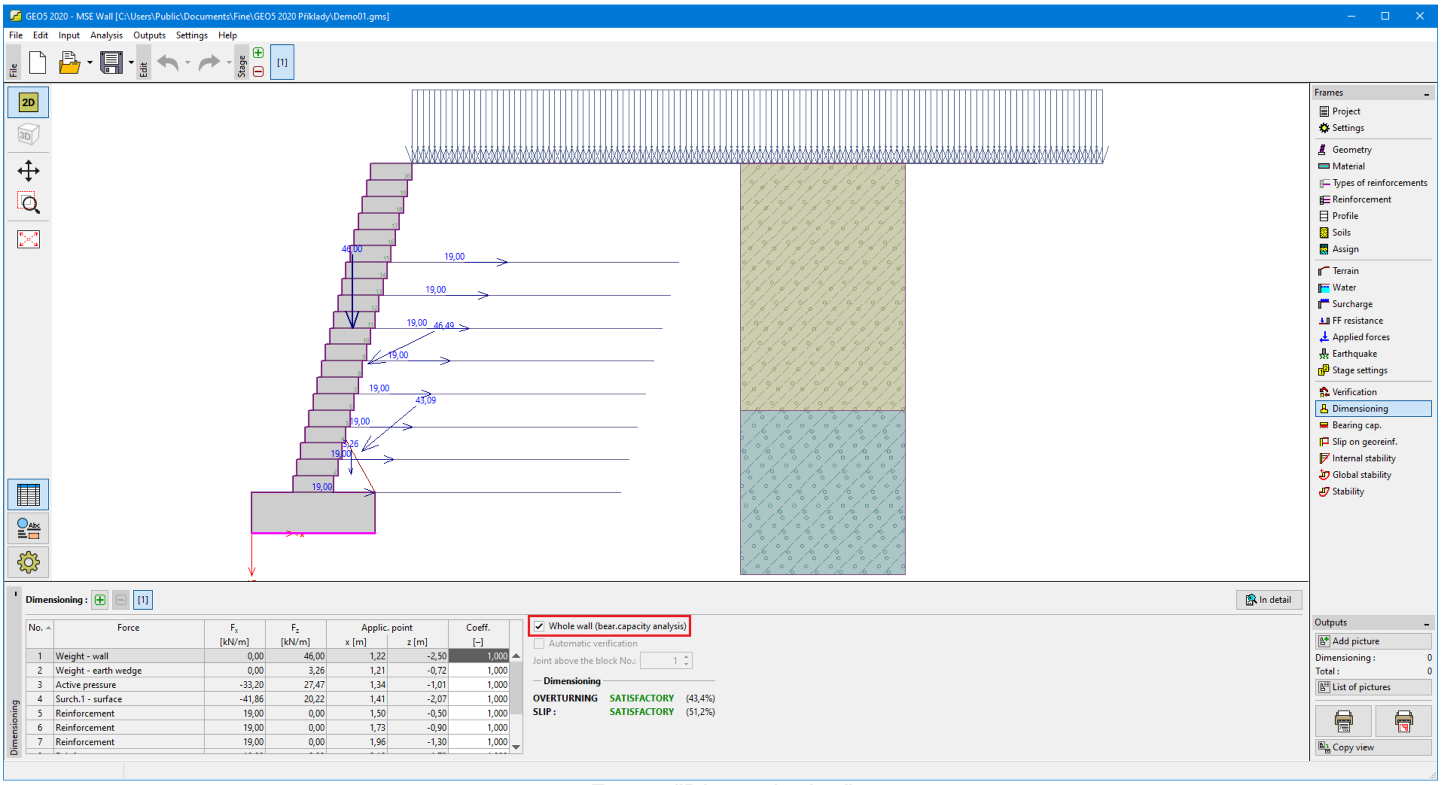
The "**Dimensioning**" frame enables to verify individual joints between blocks for **overturning** and **slip**. The option "**Entire wall**" allows for verifying the overall structure above the foundation joint as well as the foundation soil bearing capacity in the "**Bearing capacity**" frame. Checking the option "**Automatic verification**" provides verification of the most critical joint above the block. Or it is possible to input the "**Joint above block number**" to prompt the program to perform the analysis for a given joint only. The procedure for wall dimensioning is described in the theoretical part of the help.

The frame appearance is adjusted based on the selected verification methodology:

- Verification according to the factor of safety or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiply the calculated forces.
- Analysis according to EN 1997 - the last column in the table allows for specifying whether the load acting on a structure is considered as the secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in case the last column is not displayed.
- Analysis according to BS8006

The frame enables us to perform more analyses of individual joints of the wall blocks. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "**In detail**" button opens the dialog window that contains a detailed listing of the dimensioning results.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Dimensioning"

Bearing Capacity

The "**Bearing capacity**" frame displays the results of the analysis of foundation soil bearing capacity. Stress in the foundation joint (assumed constant) is determined from all forces calculated in the "Verification" frame. In case of the input foundation, the bearing capacity is determined from all forces calculated in the "Dimensioning" frame (the option "**Entire wall**" must be selected). The programs "**Spread footing**" and "**Spread footing CPT**" adopts individual verifications as load cases.

Three basic analysis options are available in the frame:

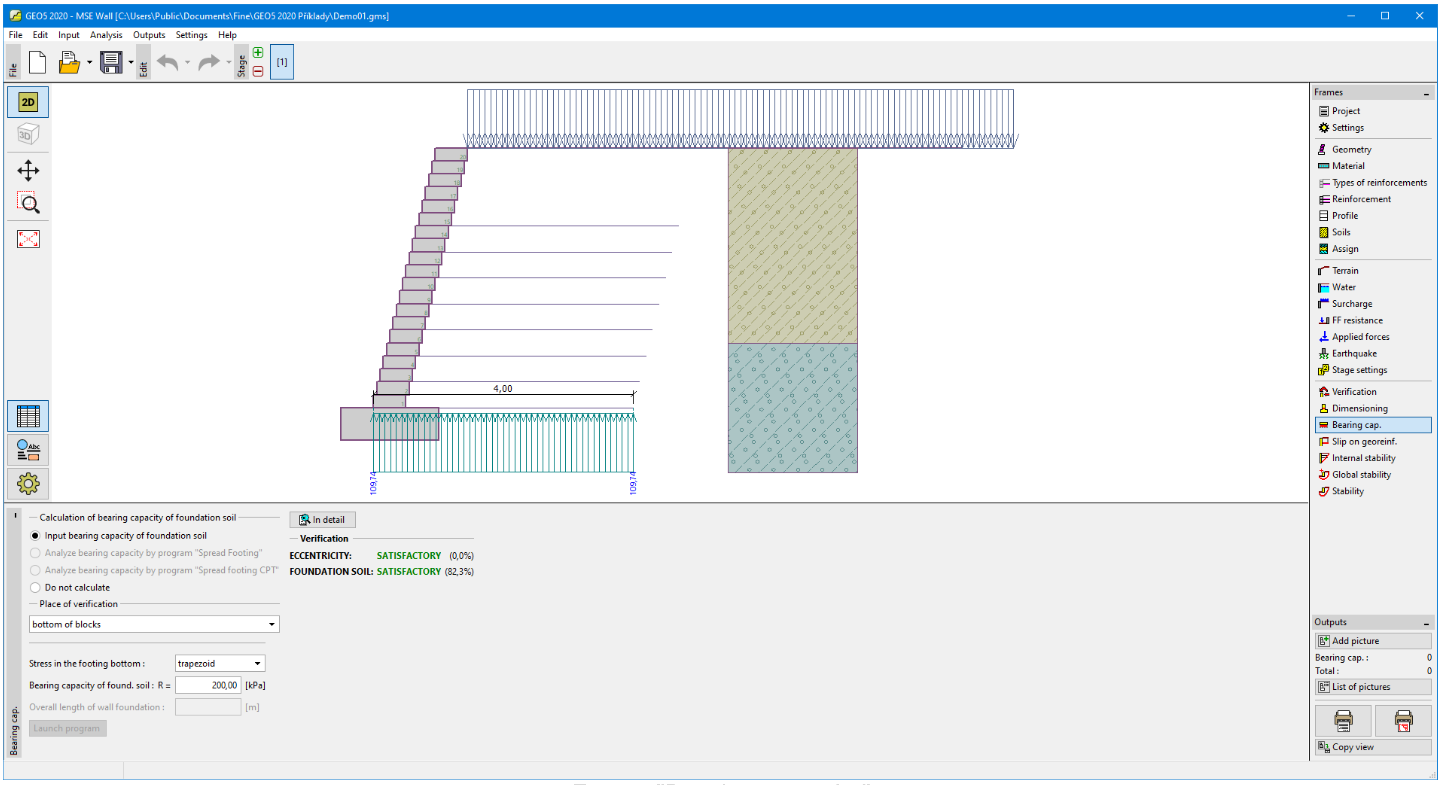
- **Insert bearing capacity of foundation soil** — The foundation soil bearing capacity is input. The **eccentricity** and bearing capacity analysis results are displayed in the right part of the frame. The "**In detail**" button opens a dialog window that displays detailed listing of the results.
- **Analyze bearing capacity by program "Spread footing"** — Pressing the "**Run program Spread footing**" button opens the program "**Spread footing**" which allows us to calculate the soil bearing capacity or settlement and rotation of the footing. Pressing the "**OK**" button leaves the analysis mode - the results and all plots are transferred into the program "**MSE Wall**". The "**Spread footing**" program must be installed for the button to be active. Thje overall length of the wall foundation is input.
- **Analyze bearing capacity by program "Spread footing CPT"** — The procedure is identical as if calculating soil bearing capacity by the **"Spread footing**" program.
- **Do not calculate (pile footing)** — The foundation soil bearing capacity is not calculated.

The program allows us to specify a shape of stress in the footing bottom.

Verification can be performed in the place:

- bottom of leveling pad
- bottom of blocks

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Bearing capacity"

Slip on Georeinforcement

The "**Slip on georeinforcement**" frame enables to verify a slip of the reinforced soil block along a geo-reinforcement checking the field "**Reinforcement number**". Selecting the option "**Automatic verification**" provides verification of the most critical reinforcement. The **reinforced soil block** is bounded by the wall front face, the checked geo-reinforcement, a vertical line passing through the geo-reinforcement end point and terrain. The reinforced soil block is loaded by an active earth pressure and by stabilizing forces due to geo-reinforcements exceeding the boundary of the reinforced block.

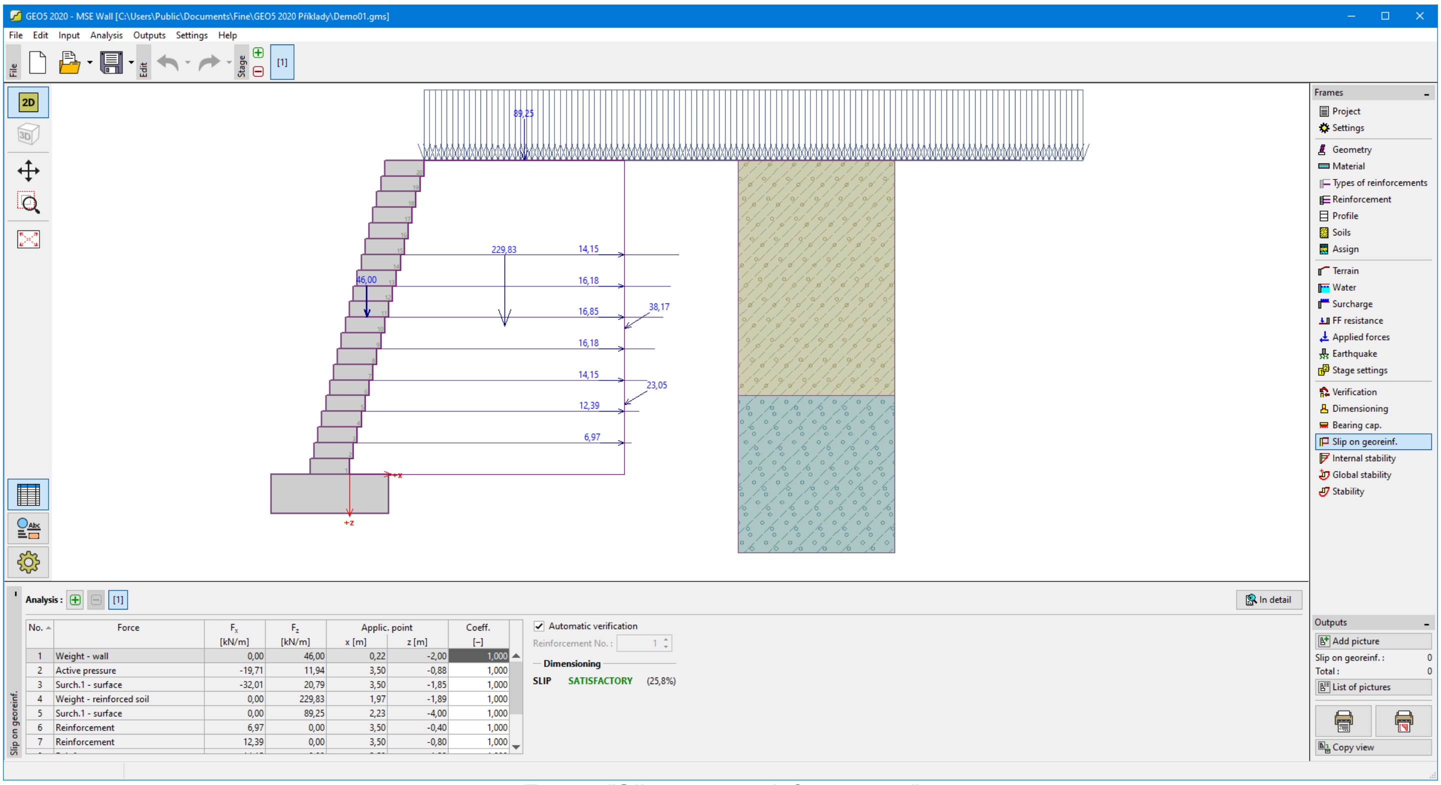
The solution procedure of slip on georeinforcement is described in the theoretical part of the help.

The frame appearance is adjusted based on the selected verification methodology:

- Verification according to the factor of safety or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiply the calculated forces.
- Analysis according to EN 1997 - the last column in the table allows for specifying whether the load acting on a structure is considered as the secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in case the last column is not displayed.
- Analysis according to BS8006

The frame enables us to perform more verification analyses of individual geo-reinforcements. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "**In detail**" button opens the dialog window, which contains a detailed listing of the results of internal stability.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Slip on georeinforcement"

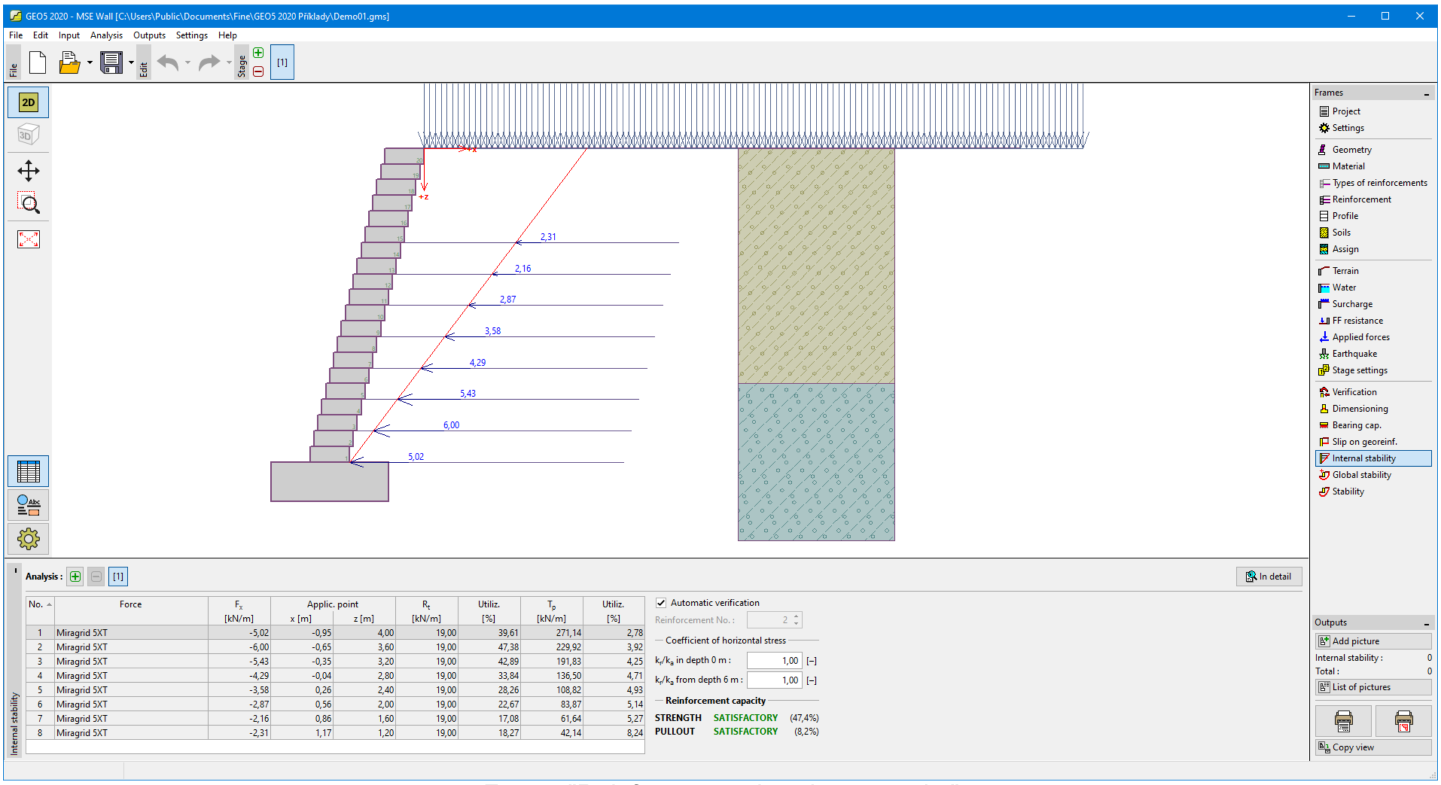
Internal Stability

The "**Internal stability**" frame enables check strength of geo-reinforcement, bearing capacity for **pull-out** from the soil, and strength of connections. Checking the field "**Reinforcement number**" yields verification for individual reinforcements only. Selecting the option "**Automatic verification**" provides verification of all reinforcements. The result for the most critical reinforcement is displayed on the right part of the desktop. The solution procedure of internal stability is described in the theoretical part of the help.

In the table,forces caused by an active earth pressure acting on the front face of the wall in individual geo-reinforcements and points of application of these forces are shown. Further, bearing capacity of geo-reinforcements for **tensile strength** R_t, **pull-out resistance** T_p and resulting utilization is shown for each geo-reinforcement.

The frame enables us to perform more analyses of individual geo-reinforcements. The calculated forces are displayed on the desktop are automatically updated with every change of input data. The "**In detail**" button opens the dialog window, which contains a detailed listing of the results of reinforcement bearing capacity.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Reinforcement bearing capacity"

Global Stability

The "**Global stability**" frame enables us to perform the slope stability analysis along a circular slip surface. It is required to input parameters of the slip surface (center and radius or 3 points input) and the analysis method (Spencer, Bishop).

By pressing the **"Substitute"** button, it is possible to input points of the slip surface using mouse on the desktop.

If the "**Optimize**" option is checked, the stability analysis is performed on the most critical slip surface. The program allows us to "keep" the end points of the slip surface (by checking the **"Keep the left end point of the slip surface"** or **"Keep the right end point of the slip surface"** option).

The "**Initial slip surface**" enables to input the circular slip surface automatically. The analysis is then performed after the "**Analyze**" button is pressed.

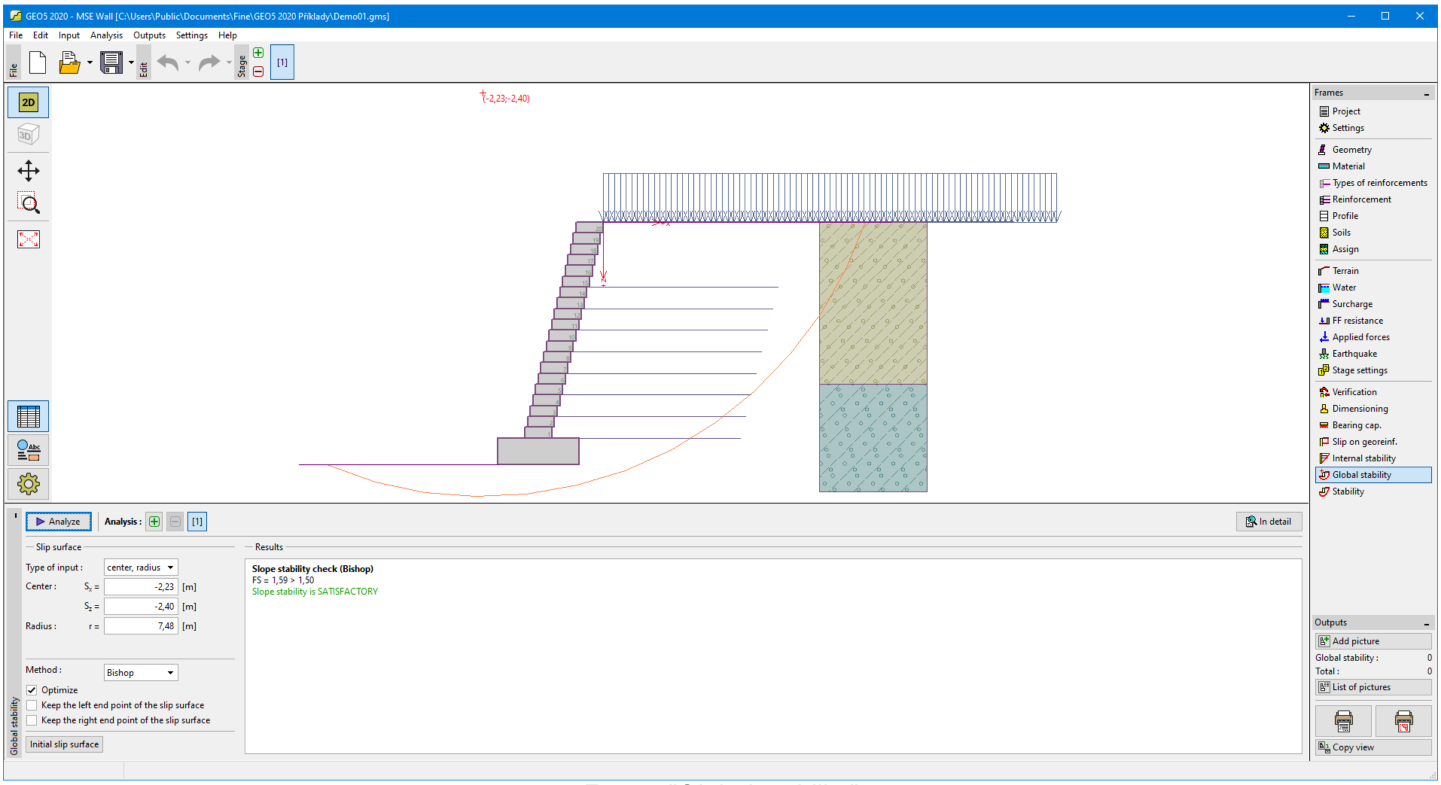
The **actual** slope stability **verification analysis** is carried out depending on the setting in the "Stability analysis" tab:

- According to EN 1997, where load is reduced by the partial factors of analysis and the verification is performed based on the theory of limit states.
- According to LRFD, the analysis is carried out similarly to the theory of limit states
- According to the **safety factor**/the theory of **limit states** depending on the setting in "Wall analysis" tab.

More analyses can be performed for a single task. The "**In detail**" button opens the dialog window, which contains a detailed listing of the results of stability analysis, i.e. parameters of the resulting slip surface and the factor of safety, alternatively utilization (for limit states).

The results are displayed in the right part of the frame, the optimized slip surface on the desktop.

Visualization of results can be adjusted in the frame "Drawing Settings".

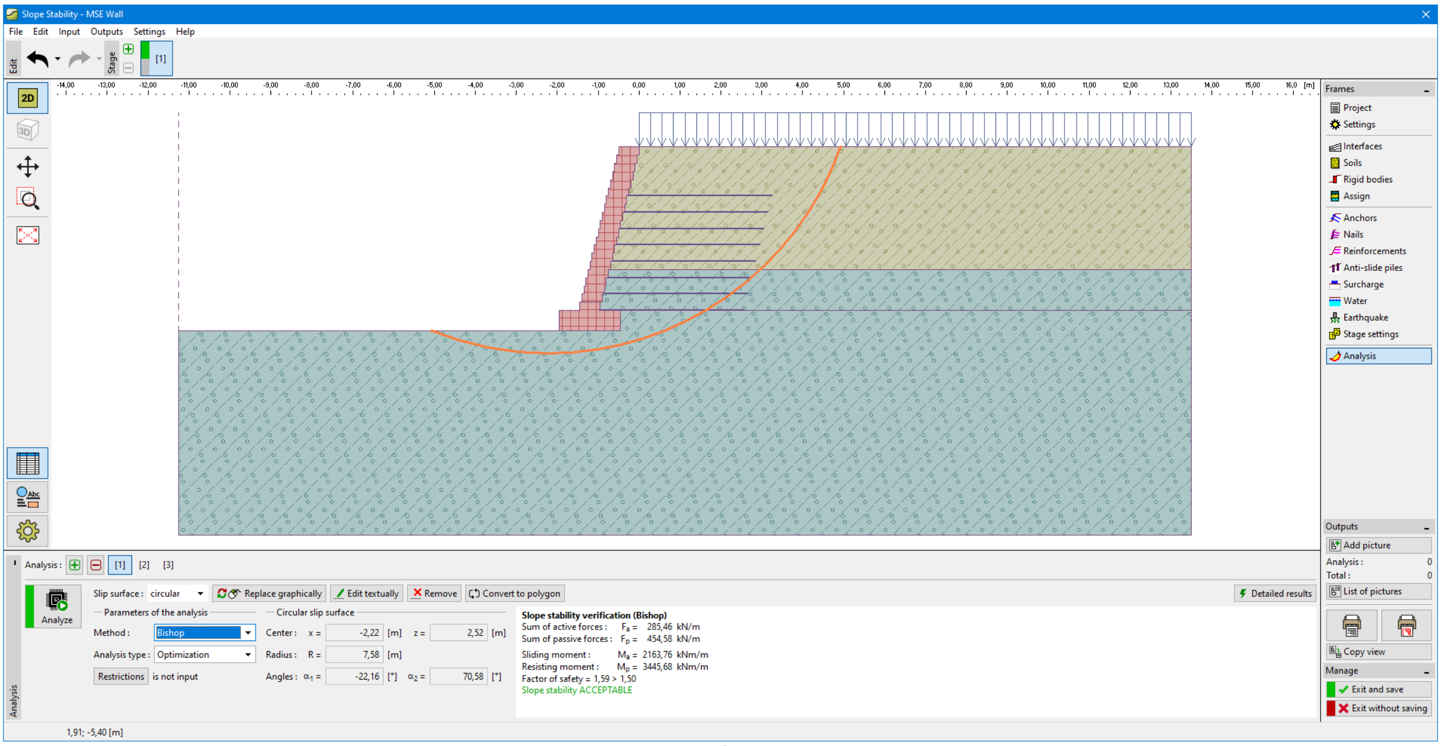


Frame "Global stability"

Stability

Pressing the "**Stability**" button launches the "**Slope stability**" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program "**Slope stability**" is installed.

After completing all analyses, press the "**OK**" button to leave the program - all data are then carried over to the analysis protocol of the "**MSE Wall**" program.



Frame "Stability"

Program Spread Footing

This program is used for the design of spread footings subject to the general load. It computes vertical and horizontal bearing capacity, settlement and rotation of a footing, and determines required longitudinal and shear reinforcement (punching).

The help in the program "Spread Footing" includes the following topics:

- The input of data into individual frames:

Project	Settings	Profile	Dilatometric Tests (DMT)	Soils	Assign	Foundation
Load	Geometry	Footing Bottom	Sand-Gravel Cushion	Material	Surcharge	Water, Incompressible Subsoil
Earthquake	Stage Settings	Bearing Capacity	Settlement and Rotation	Dimensioning		

- Standards and analysis methods

- Theory for analysis in the program "**Spread Footing**":

Stress in Soil Body	Analysis of Foundation Bearing Capacity	Settlement Analysis	Dimensioning of Concrete Structures

- Outputs

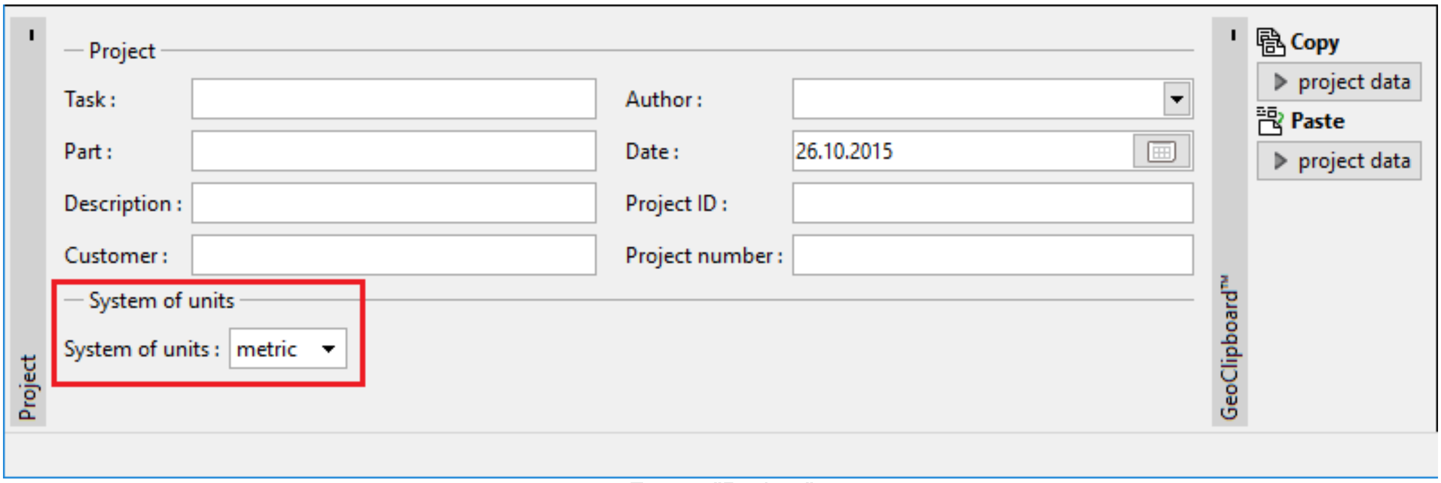
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The **"Project"**frame is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The "**Settings**" frame allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

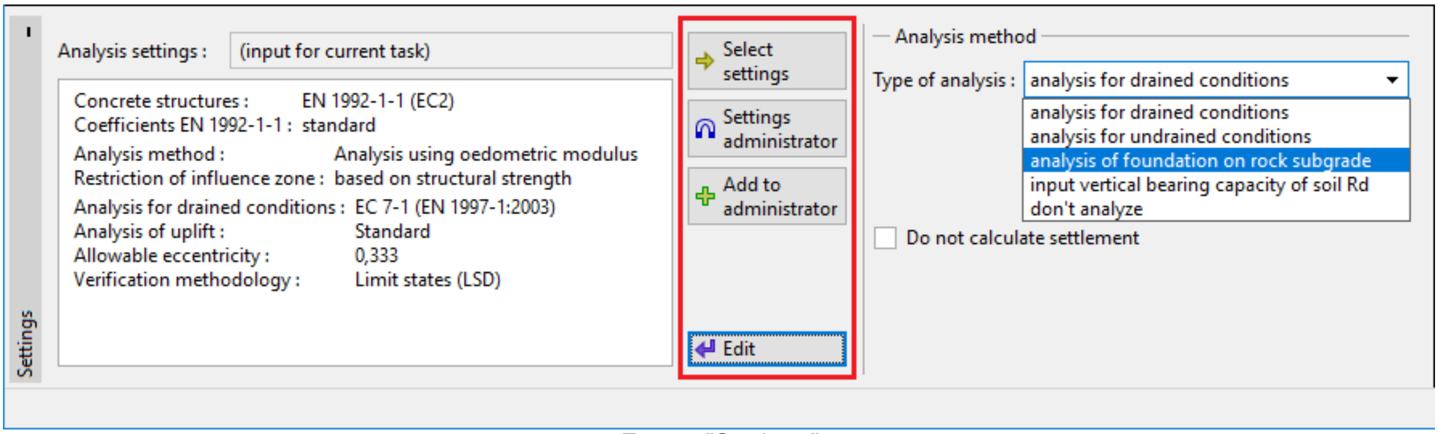
The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

Settings of analysis parameters are performed in the "Materials and standards", "Settlement" and "Spread Footing" tabs.

Four options are available to calculate the vertical bearing capacity of a spread footing:

- **analysis for drained conditions**
- **analysis for undrained conditions**
- **analysis of foundation on rock subgrade**
- **input vertical bearing capacity of soil R_d**



Frame "Settings"

Profile

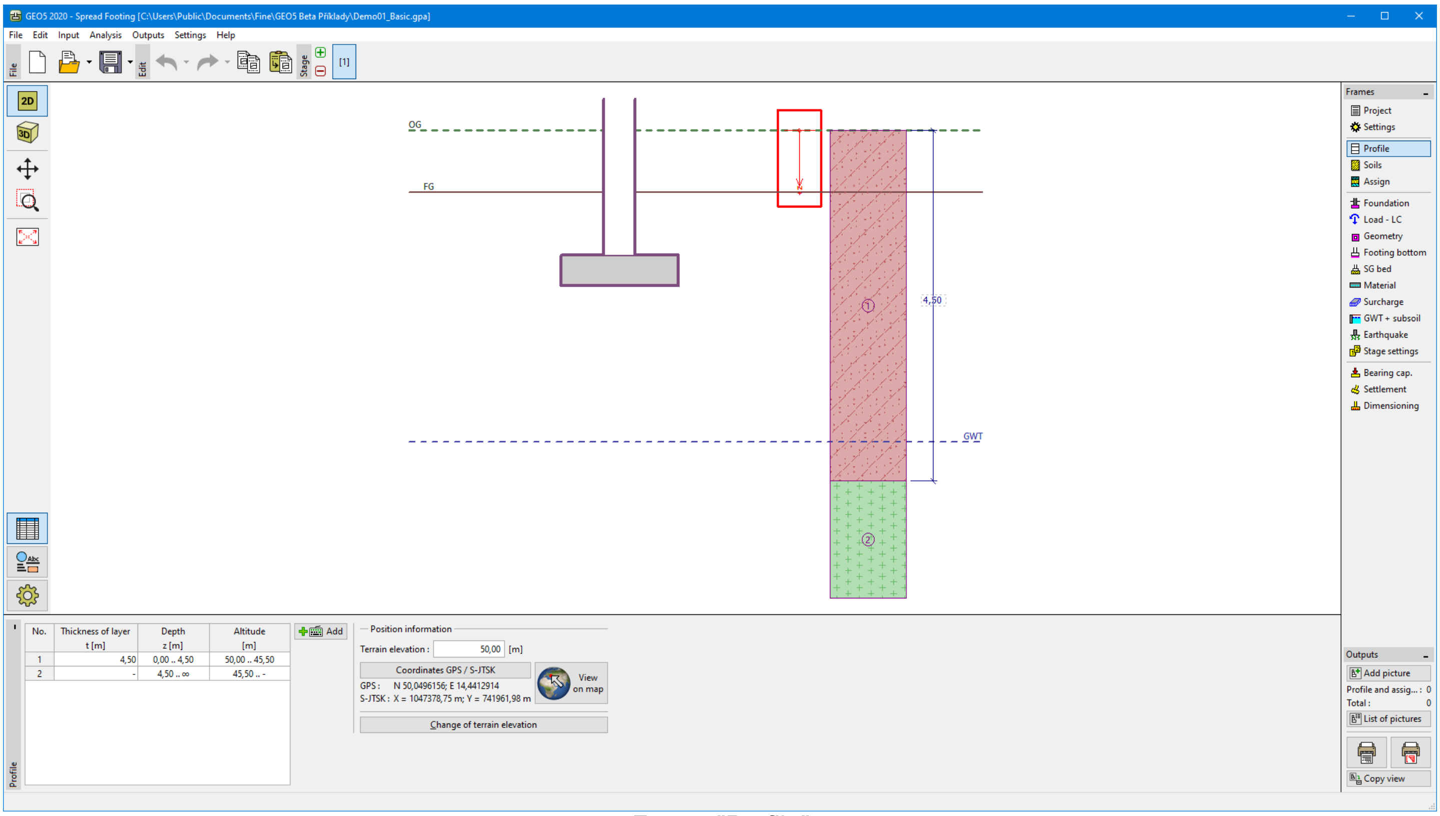
The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.

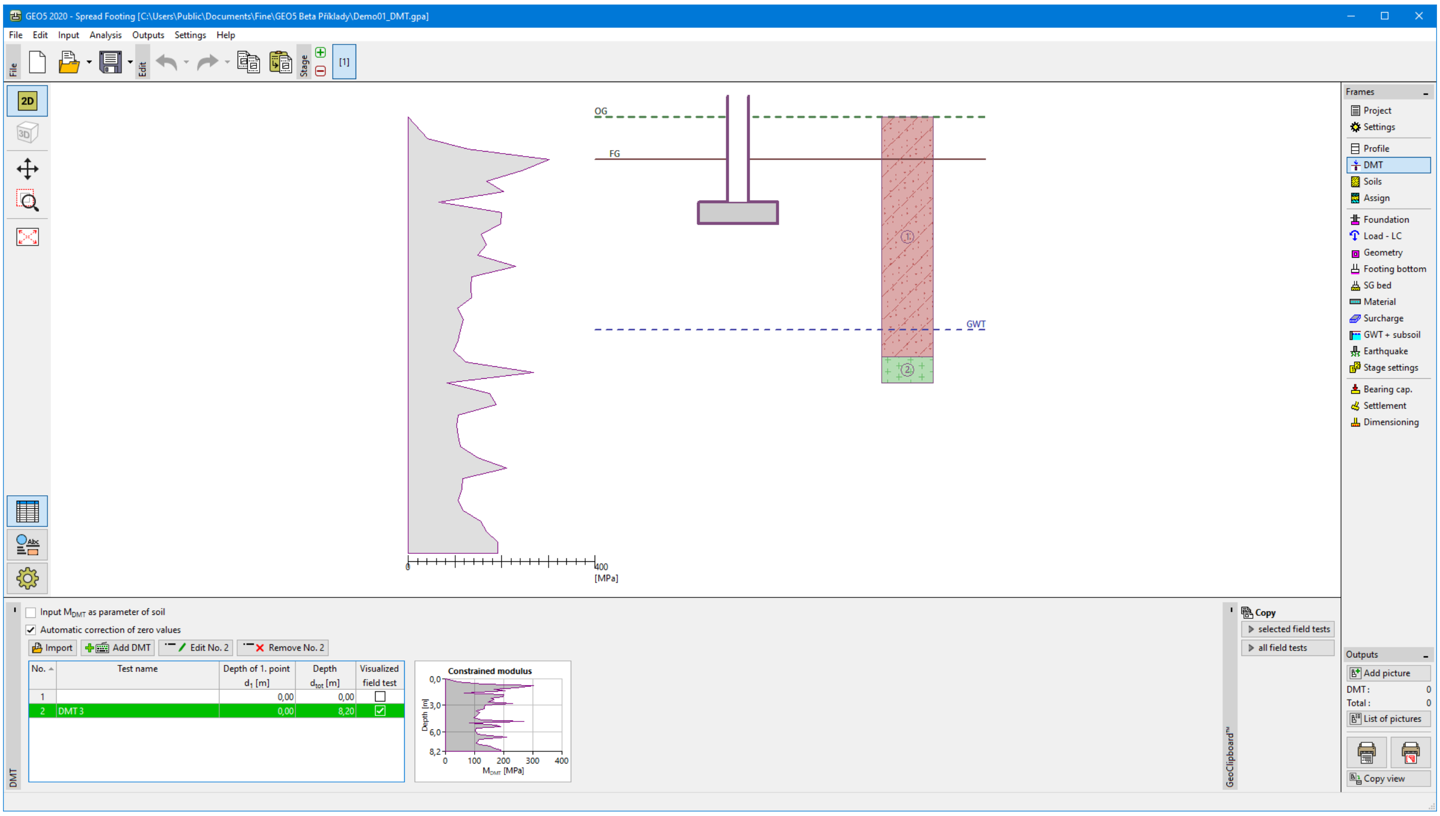


Frame "Profile"

DMT

The frame "**DMT**" serves to input the way of introducing of constrained soil modulus into the program - either as a parameter of soil (by checking the option "**Input M_{DMT} as a soil parameter**"), or by importing of a dilatometric tests (DMT).

This frame contains a table with a list of the input values of dilatometric tests (DMT).



Frame "DMT"

If during the evaluation of dilatometric test, the zero value of constrained soil modulus M_{DMT} is measured, then program allows the automatic correction of measurement errors - the arithmetic average of the next upper and lower non-zero value of M_{DMT} is considered instead of zero value in the calculation.

The results of dilatometric test (DMT) can be imported into the program by inserting the file in format **UNI** (***.uni**).

Data of DMTs can be copied within "**Spread Footing**", "**Sheeting Check**", "**Anti-Slide Pile**" and "**Stratigraphy**" programs using "GeoClipboard".

Note: The frame is accessible only in case, when method of calculation is selected as "**Dilatometric DMT**" in the "Settings" frame ("Settlement" tab).

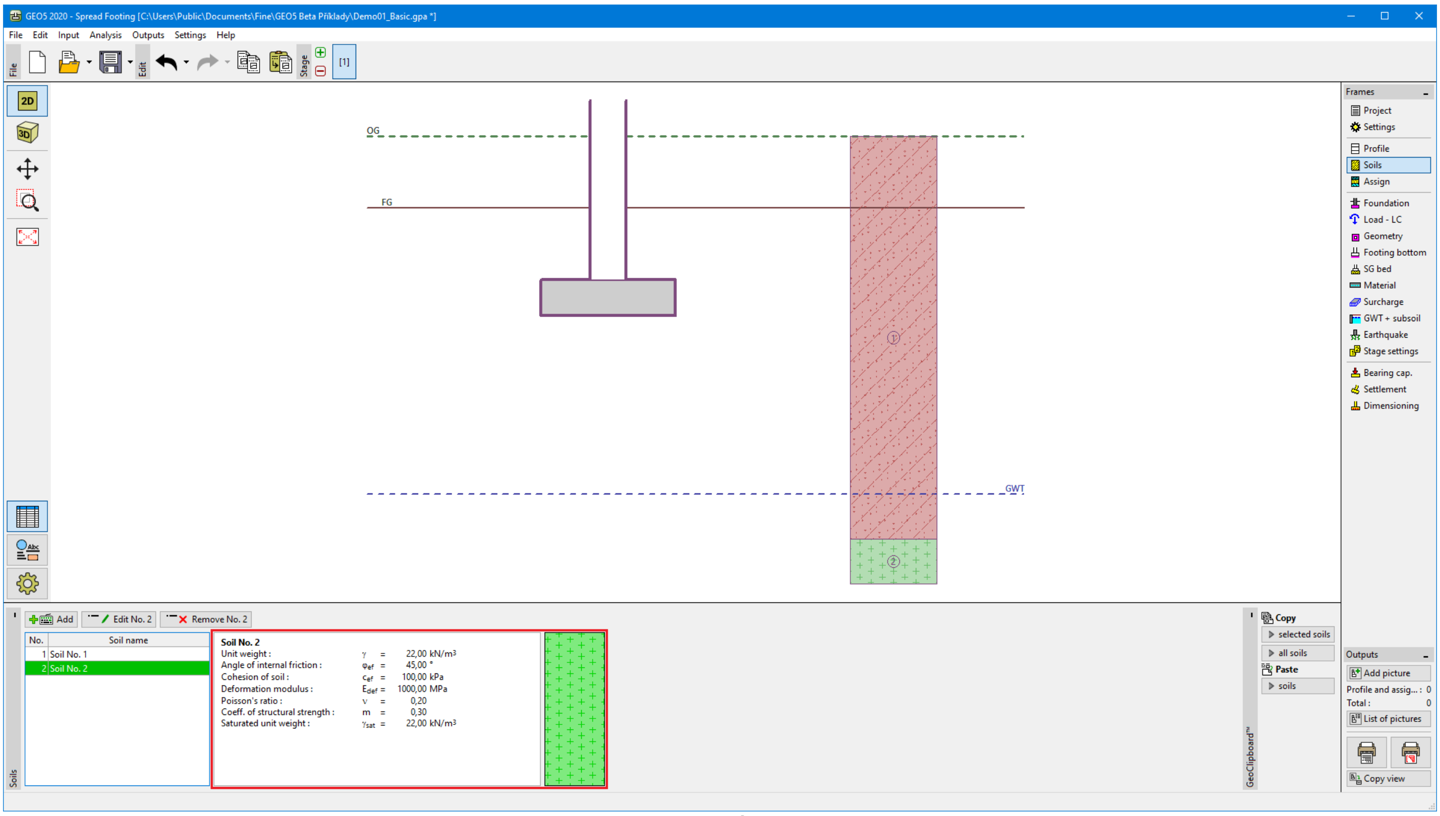
Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Uplift pressure", "Foundation bearing capacity" and "Settlement".

Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

Basic Data

This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction, and cohesion**. The particular values are obtained from a geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit the built-in database of soils, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually.

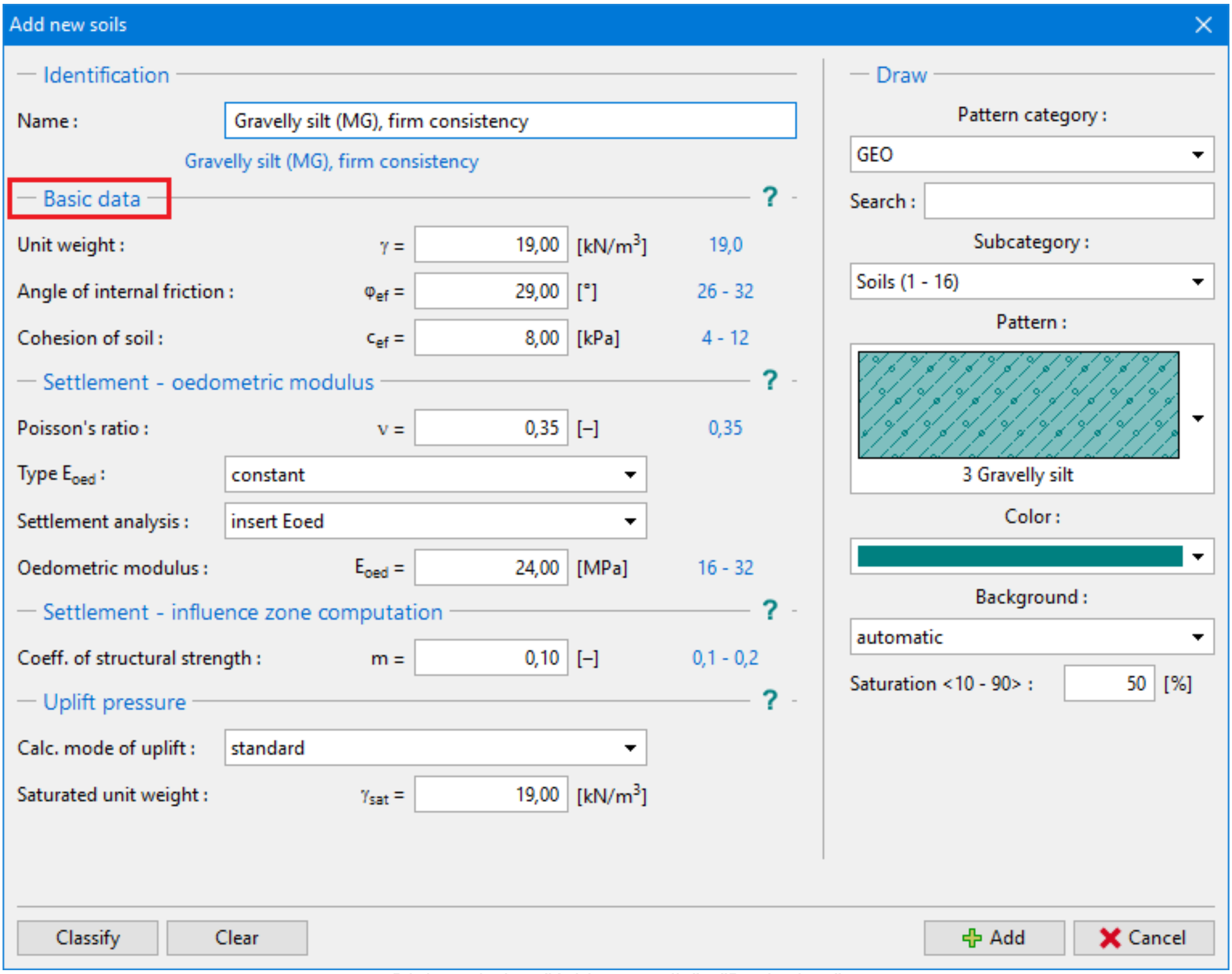
Soil parameters differ according to the analysis type and analysis method ("Settings" frame, "Spread Footing" and "Settlement" tabs).

The analysis type differs according:

- **Analysis for drained conditions:** **effective** parameters of shear strength of soil c_{ef}, φ_{ef} are used commonly.
- **Analysis for undrained conditions**: the vertical bearing capacity of foundation depends on the **undrained** shear strength of soil c_u . **Effective** angle of internal friction φ_{ef} is defined only for calculation of earth pressure to solve horizontal bearing capacity of foundation.

- **Analysis of foundation on rock subgrade:** for this analysis method program defines angle of internal friction of rock φ , compressive strength σ_c , coefficient of damage of rock D , coefficient of structural strength m_i and Geological Strength Index.

The associated theory is described in detail in the chapter "Analysis of bearing capacity of foundation".

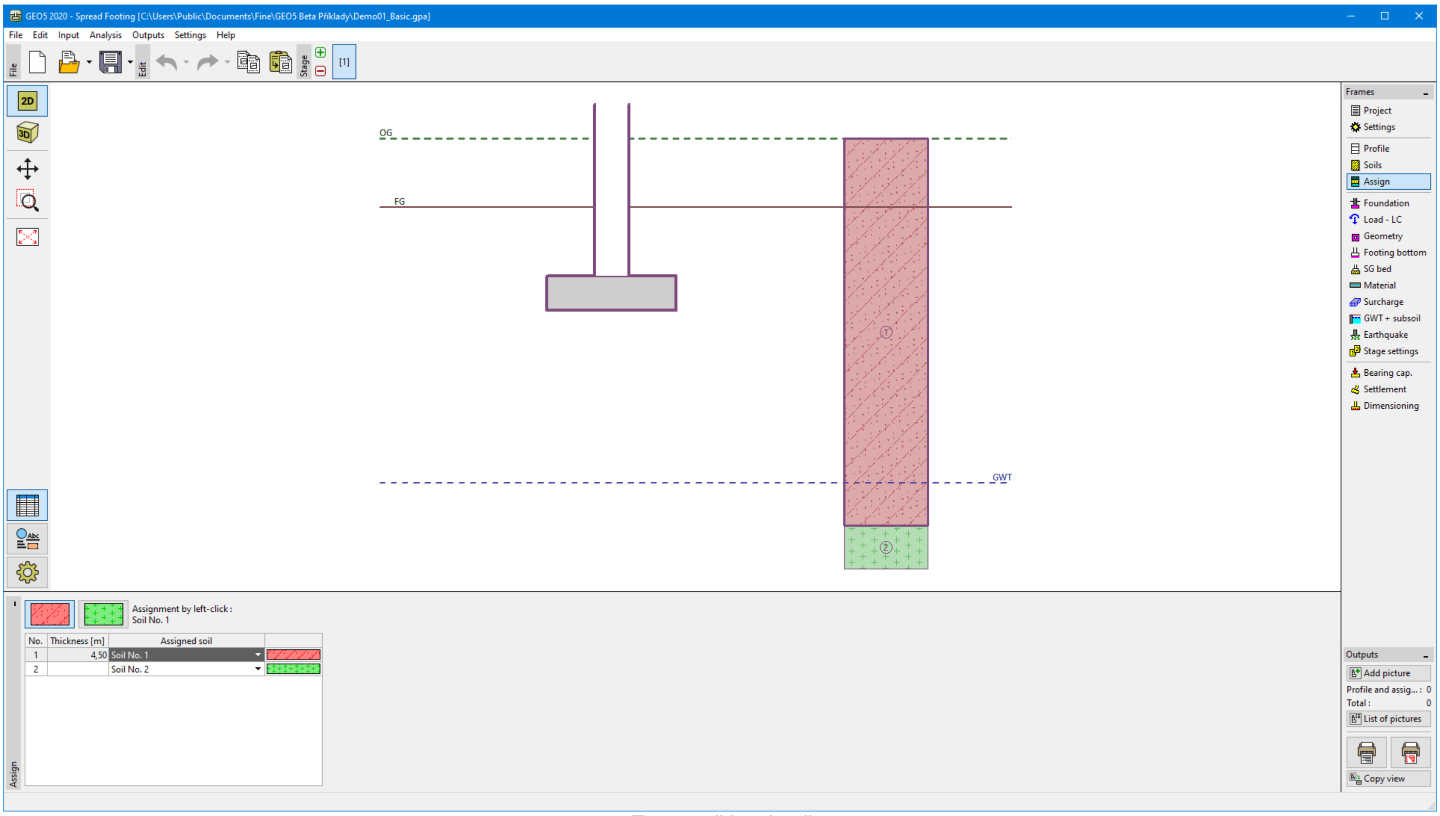


Dialog window "Add new soils" - "Basic data"

Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail herein.



Frame "Assign"

Foundation

The "**Foundation**" frame allows us to select a type of foundation. The selected type with a graphic hint of input values is displayed in the left part of the frame. The values can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The following types of foundations can be selected:

- **Centric spread footing**
- **Eccentric spread footing**
- **Strip footing**
- **Stepped centric spread footing**
- **Stepped eccentric spread footing**
- **Circular spread footing**
- **Circular stepped spread**
- **Centric spread footing with batter**
- **Eccentric spread footing with batter**
- **Centric spread footing with circular step**

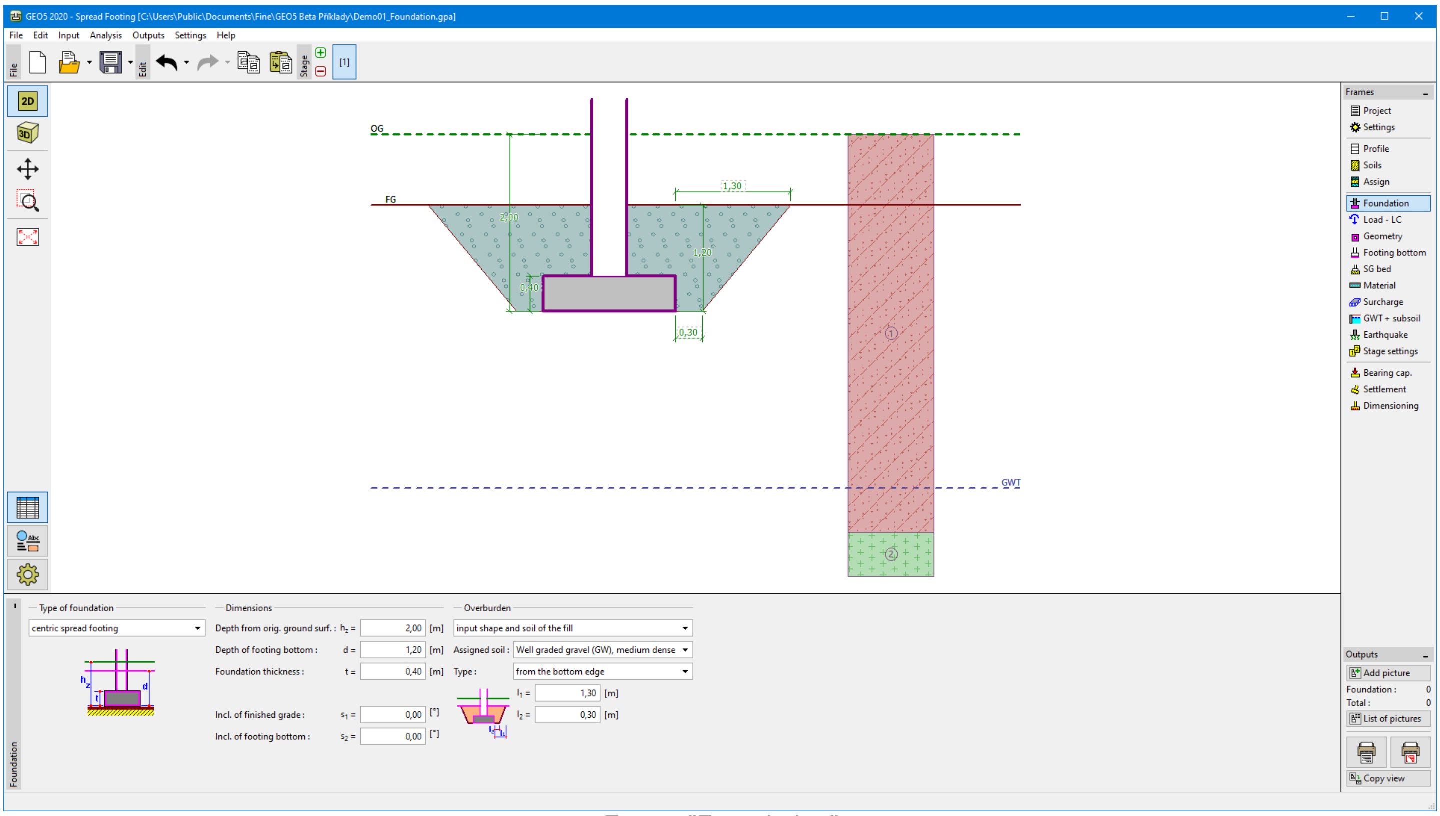
The soil profile is specified from the **original ground**. The foundation bearing capacity depends mainly on the depth **of foundation measured from the finished grade**. When the finished grade is found above the original ground, then it is required to assign the same depth to both, the finished grade and original ground, and introduce into subsoil a layer with a new made-up-ground. This frame also allows for inputting the **foundation thickness**.

The **overburden type** is chosen from the following options:

- **input unit weigh** - the overburden unit weight γ_1 is specified
- **from geological profile** - the overburden unit weight is calculated from the soils assigned in the geological profile
- **input shape and soil** - this option allows us to model the overburden shape from the **bottom** or **upperedge** of footing. The overburden unit weight is determined from the assigned soil.

Providing the analysis follows the theory of limit states its weight is multiplied by the coefficient $\gamma_{m\gamma}$ input in the "Spread Footing" tab.

For foundations with drained subsoil (type of analysis is selected in the frame "Settings") it is possible to introduce an **inclination of the finished grade and footing bottom**. In all other cases both the ground and footing bottom are horizontal.



Frame "Foundation"

Load

The "**Load**" frame contains a table with a list of input loads. Adding loads is performed in the "**New load**" dialog window. Input of individual forces follows the sign convention displayed in the right part of the dialog window.

The following types of load can be specified:

- **design load** serves to verify the foundation bearing capacity
- **service load** serves to compute the foundation settlement and rotation

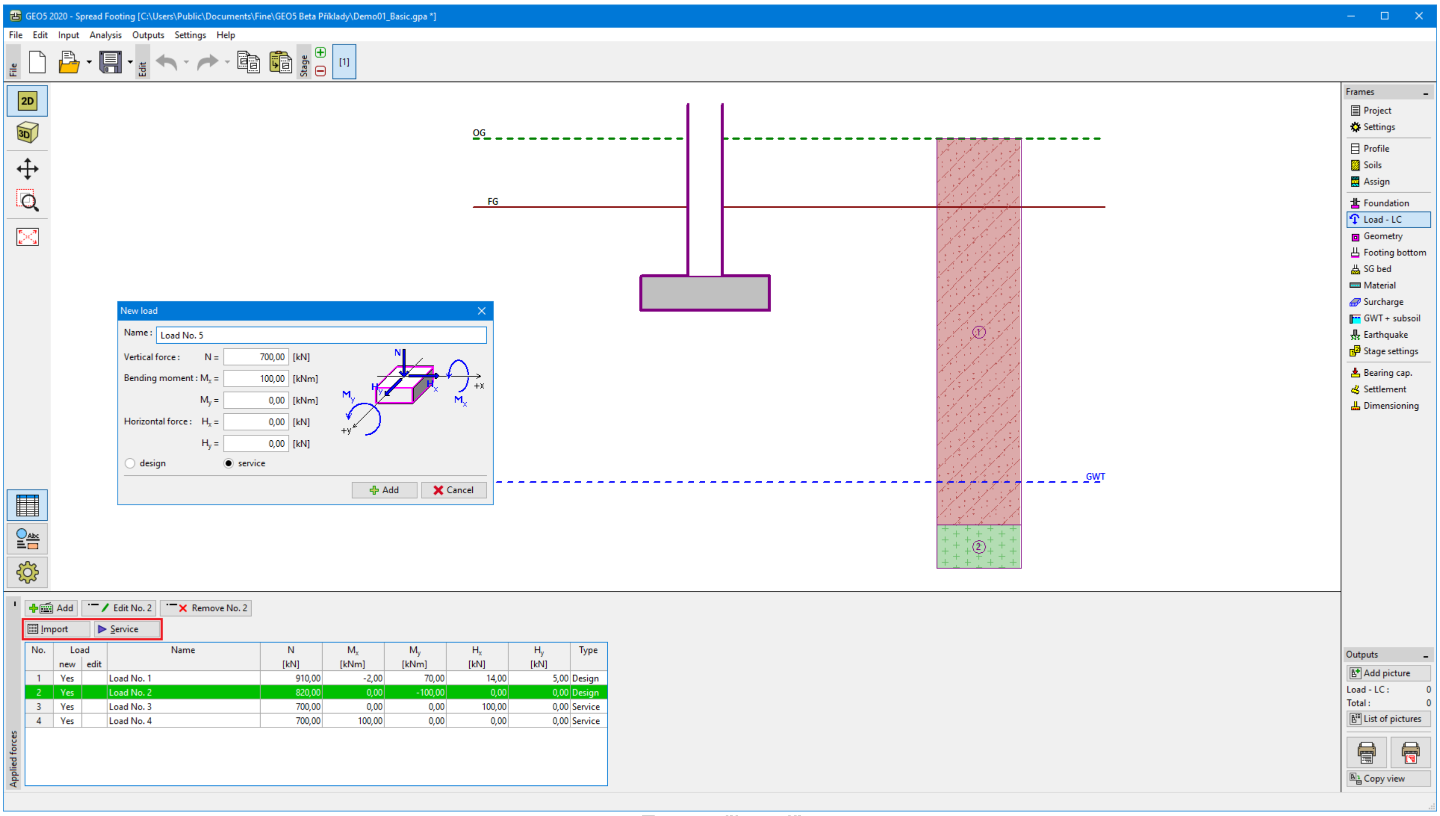
Dimensioning of reinforcements assumed for the foundation is carried out for both types of load.

When performing the analysis according **to EN 1997 or LRFD** (selected in the "Spread Footing" tab) it is assumed that the design load is determined in accordance with the corresponding standards and individual components of load are **already pre-multiplied** by corresponding partial factors - the program does not modify the input load **any further**.

The foundation is always loaded at the contact point between column and foundation. The program automatically computes the **foundation self-weight** and the **weight of overburden**.

The "**Service**" button allows for creating the service loads from the already input design loads (analysis according to the factor of safety or the theory of limit states).

The program also allows importing of load using the "**Import**" button.



Frame "Load"

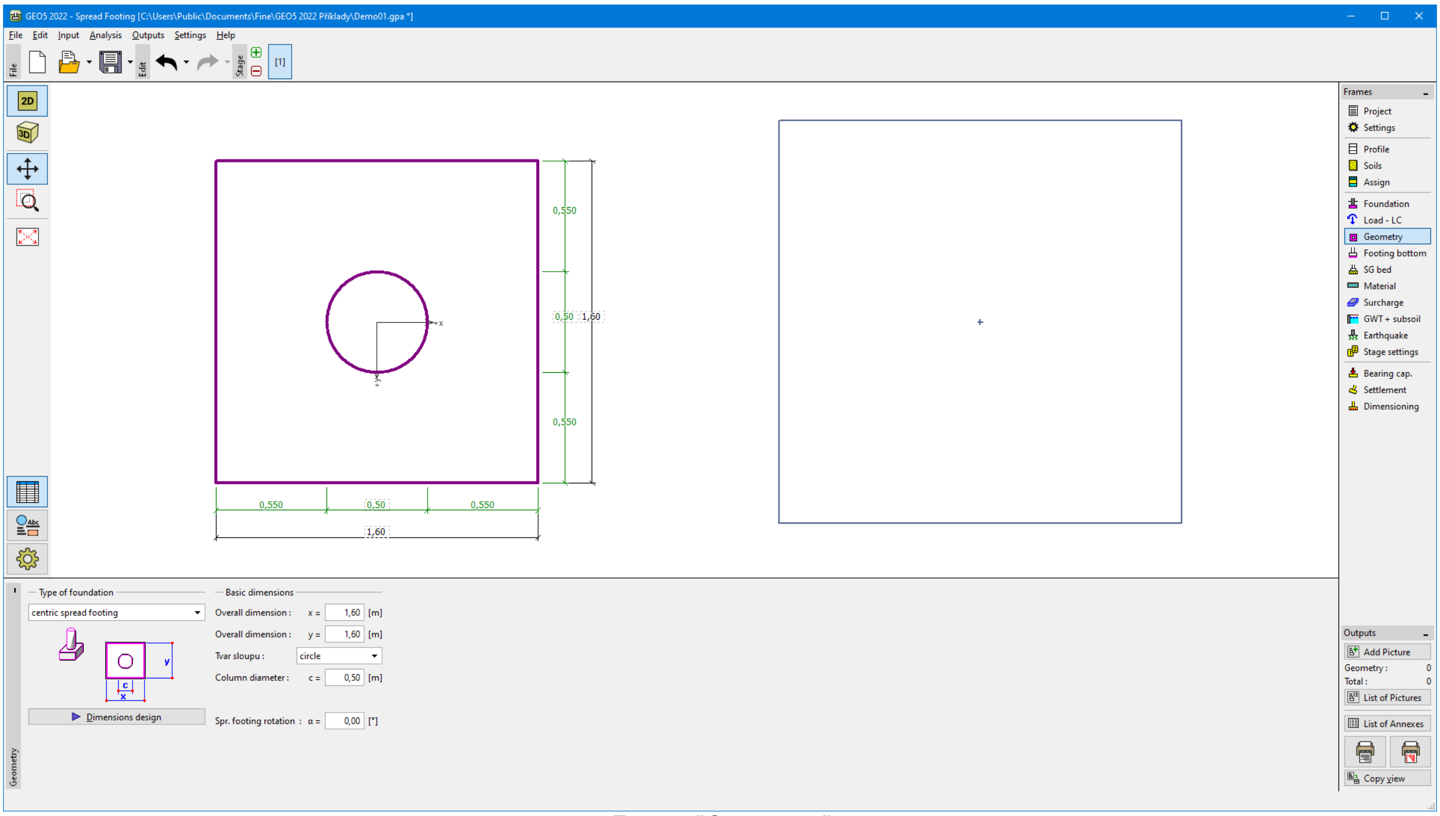
Geometry

The "**Geometry**" frame allows us to specify the foundation and column shape. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The values can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

Foundation type and its **thickness** are specified in the "Foundation" frame.

The program automatically computes the **self-weight of both foundation** and **overburden above the foundation**. The foundation self-weight is specified in the "Material" frame. Providing the analysis is carried out employing the theory of limit states the footing self-weight is multiplied by coefficients specified in the "Spread Footing" tab.

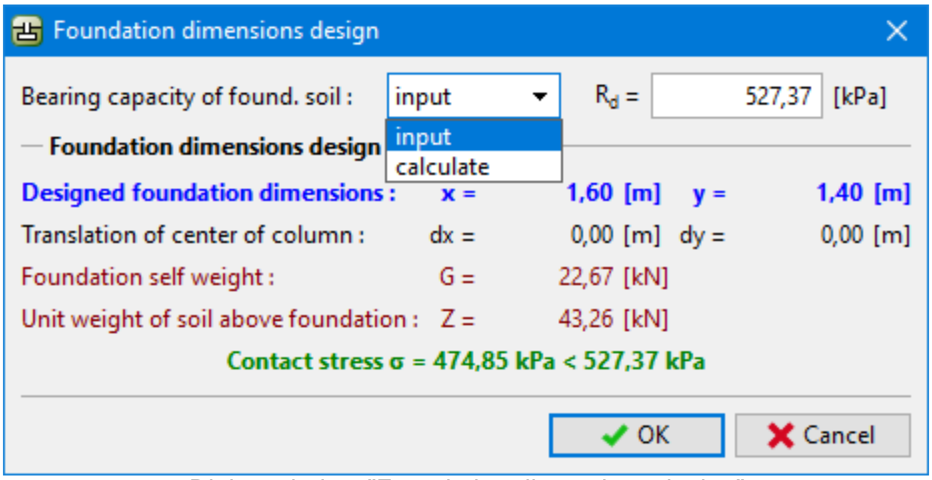
The program makes it possible to export the geometry of a structure in the *.DXF format.



Frame "Geometry"

The "**Dimensions design**" button opens the dialog window for designing the dimensions of a foundation. The dialog window allows inputting the bearing capacity of foundation soil R_d or selecting the option "**calculate**". In this case the program determines all dimensions of a foundation based on **input parameters** (soils, profile, water impact, send-gravel-cushion, setting, etc.).

While leaving the dialog window by pressing the "**OK**" button, the calculated dimensions are transfered into the "**Geometry**" frame.



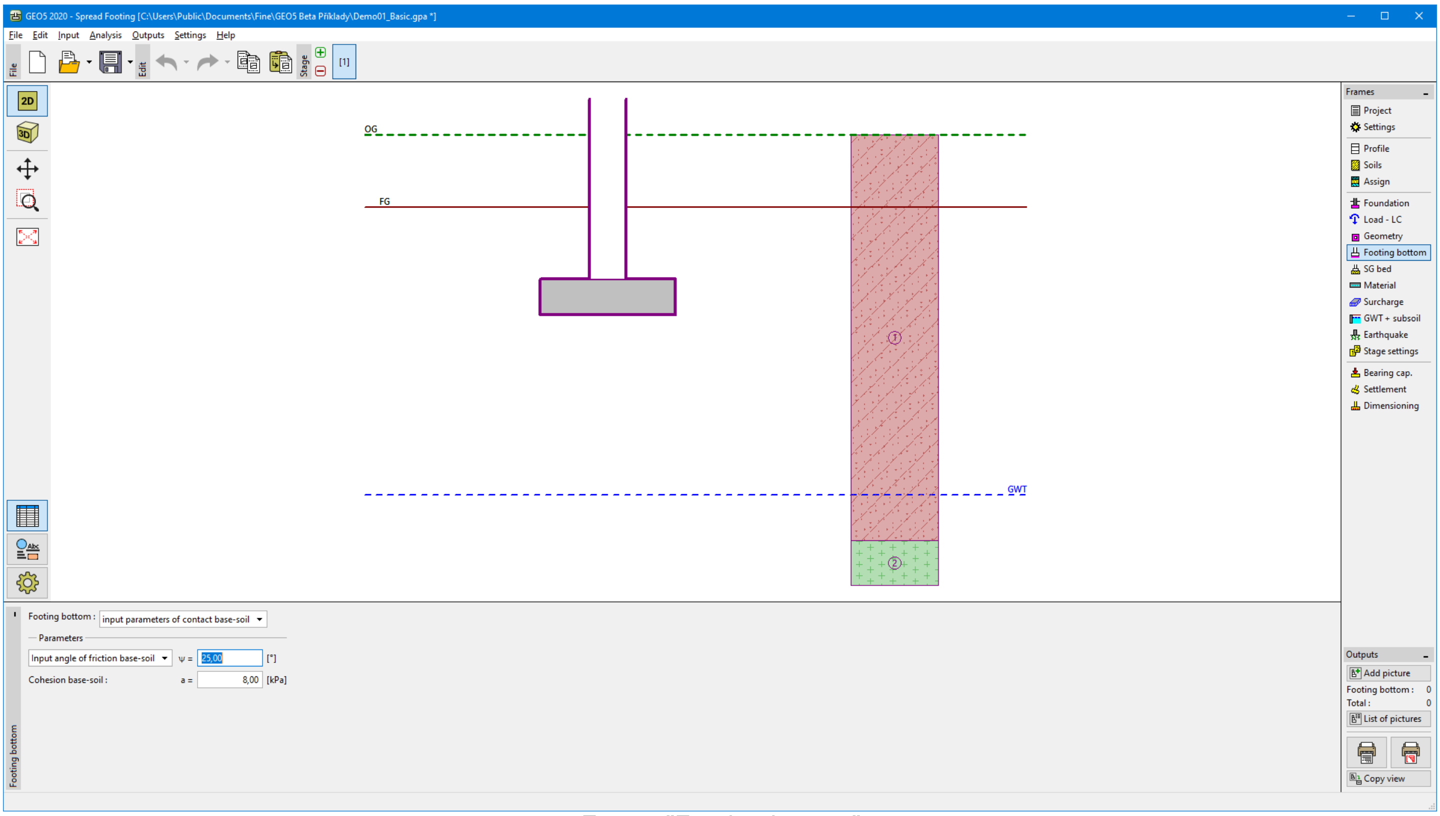
Dialog window "Foundation dimensions design"

Footing Bottom

The "**Footing bottom**" frame serves to input characteristics of the action of the footing bottom:

- **soil from geological profile** - the spread footing is founded on the soil assigned from the geological profile specified in the "Profile" frame.
- **input parameters of contact base-soil** - parameters of the contact between footing bottom and soil are specified. Option "**input angle of friction base-soil**" requires inputting the friction angle ψ [°] between foundation and soil. Option "**input friction coefficient**" requires specifying the friction coefficient μ [-]. Both options require inputting the cohesion a [kPa] between foundation (base) and soil.

The input data introduced in this frame influence the spread footing bearing capacity.



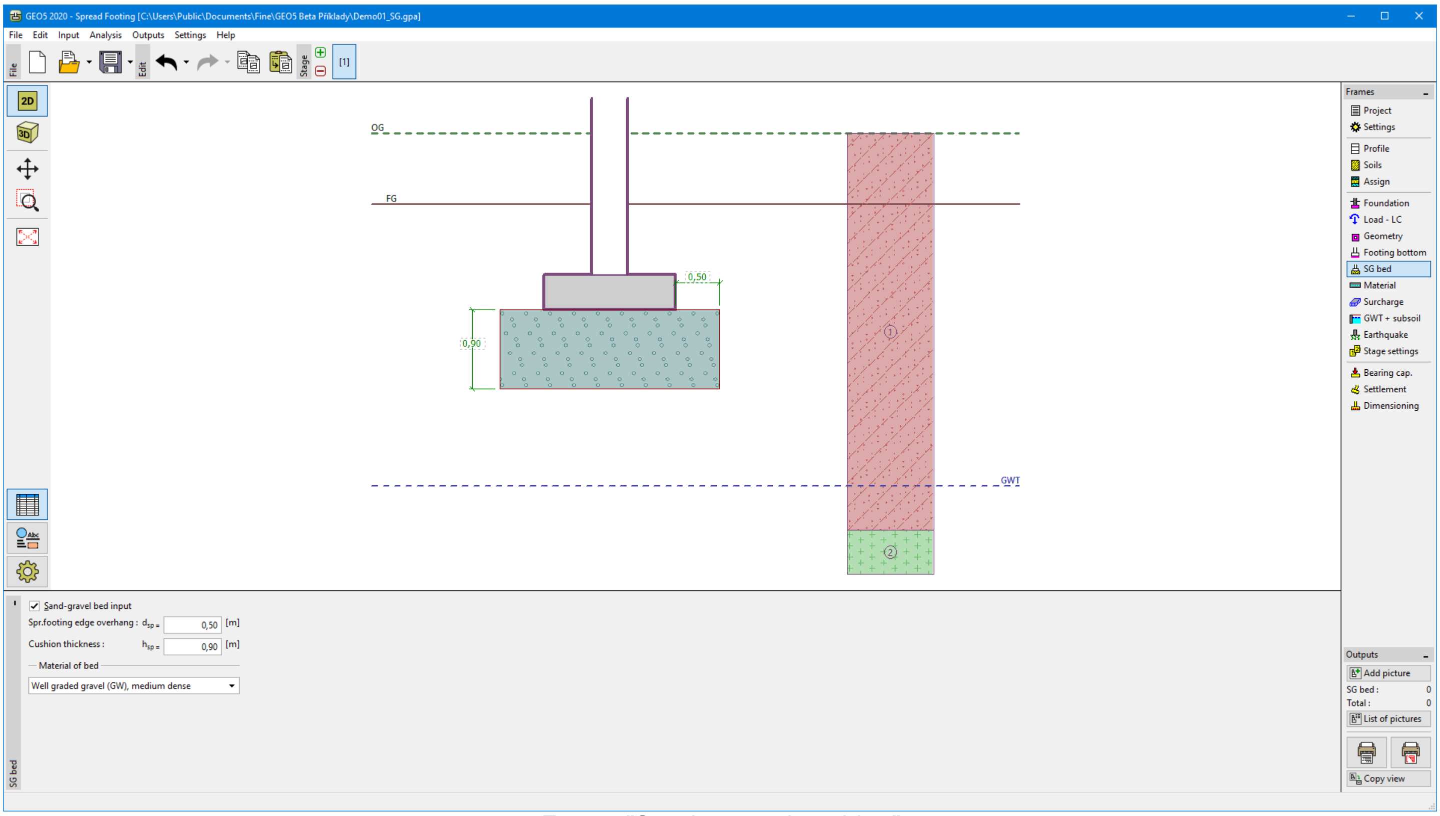
Frame "Footing bottom"

Sand-Gravel Cushion

The "**Sand-gravel cushion**" frame allows us to input parameters of the sand-gravel cushion below the foundation. The cushion thickness and overhang over the foundation edge are required. The values can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The cushion filling can be selected from a combo list that contains soils specified in the frame "Soils".

The input sand-gravel cushion influences the analysis of both the bearing capacity and settlement.



Frame "Sand - gravel cushion"

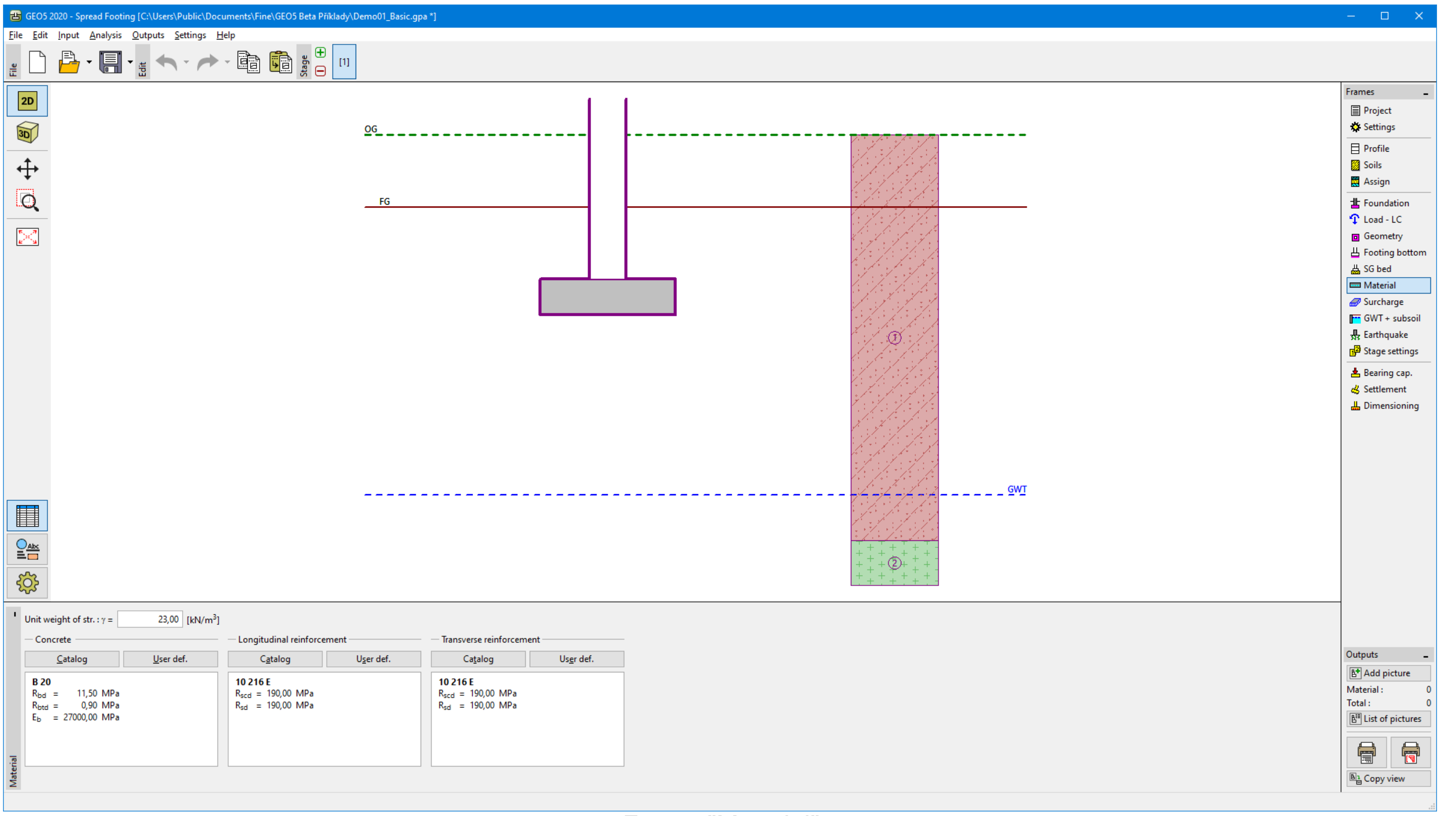
Material

The frame **"Material"** allows us to enter material parameters. The input field in the upper part of the frame allows us to specify the **unit weight of structure**.

Two options can be used for setting the parameters of the material:

- The "**Catalog**" button opens the "**Catalog of materials**" dialog window, where the required material can be selected.
- The "**User-defined**" button opens the "**Editor of material**", which allows us to input the specification of material parameters manually by the user.

The content of catalogs depends on the selection of relevant standard for the dimensioning of concrete and steel structures in the "Materials and standards" tab.



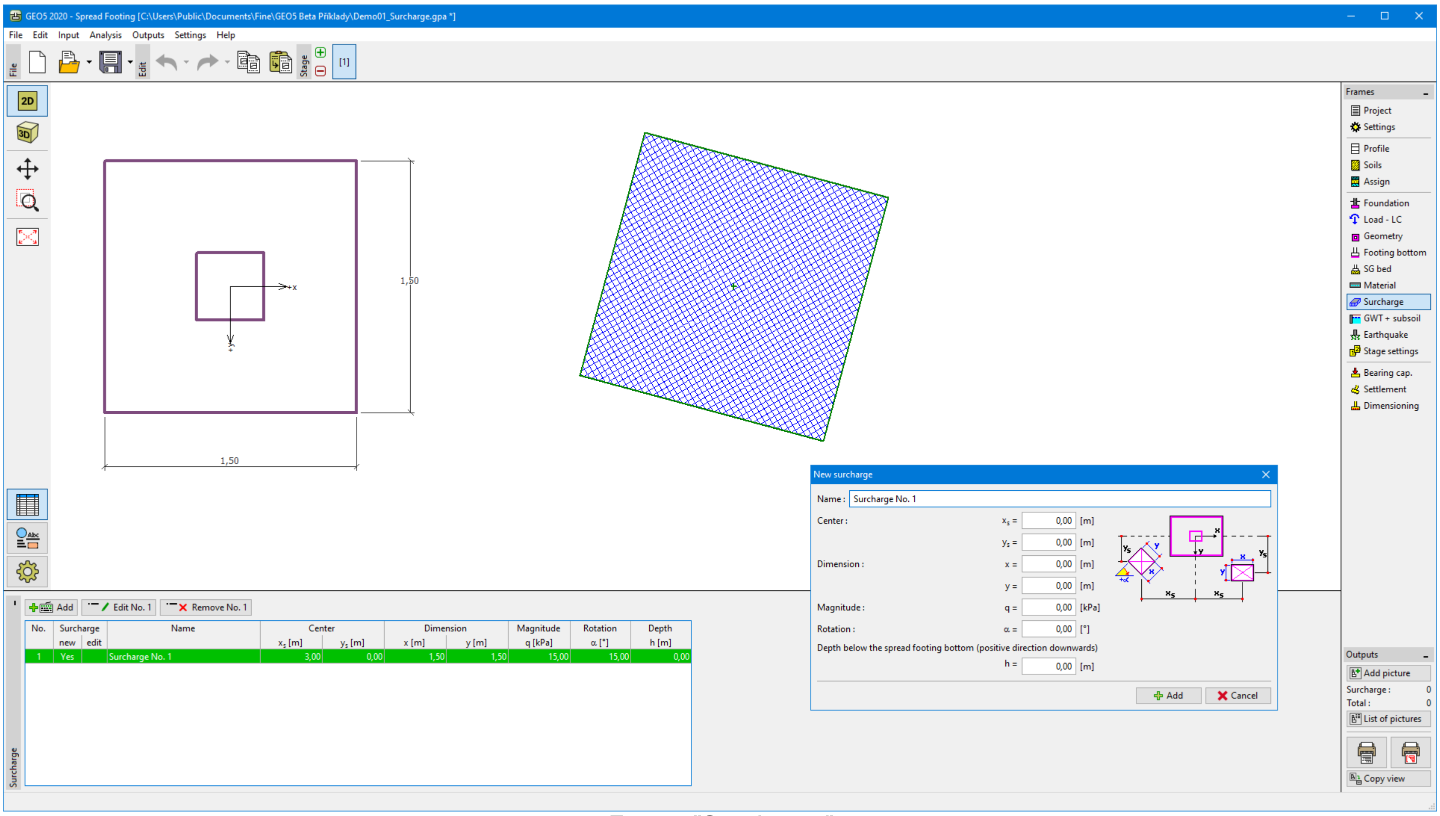
Frame "Material"

Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding surcharge is performed in the "**New surcharge**" dialog window. The values are specified according to the "**Geometry**" chart displayed in the right part of the dialog window. The input surcharges can also be edited on the desktop with the help of active objects.

The z-coordinate measured from the foundation joint of a structure is specified (positive direction downwards) when inputting the surcharge at a depth different from the depth of the foundation joint.

The surcharge is considered only when **computing settlement** and rotation of a foundation, in which case it increases the stress in the soil below foundation. When **computing the foundation bearing capacity**, the surcharge is not considered - its presence would increase the bearing capacity.



Frame "Surcharge"

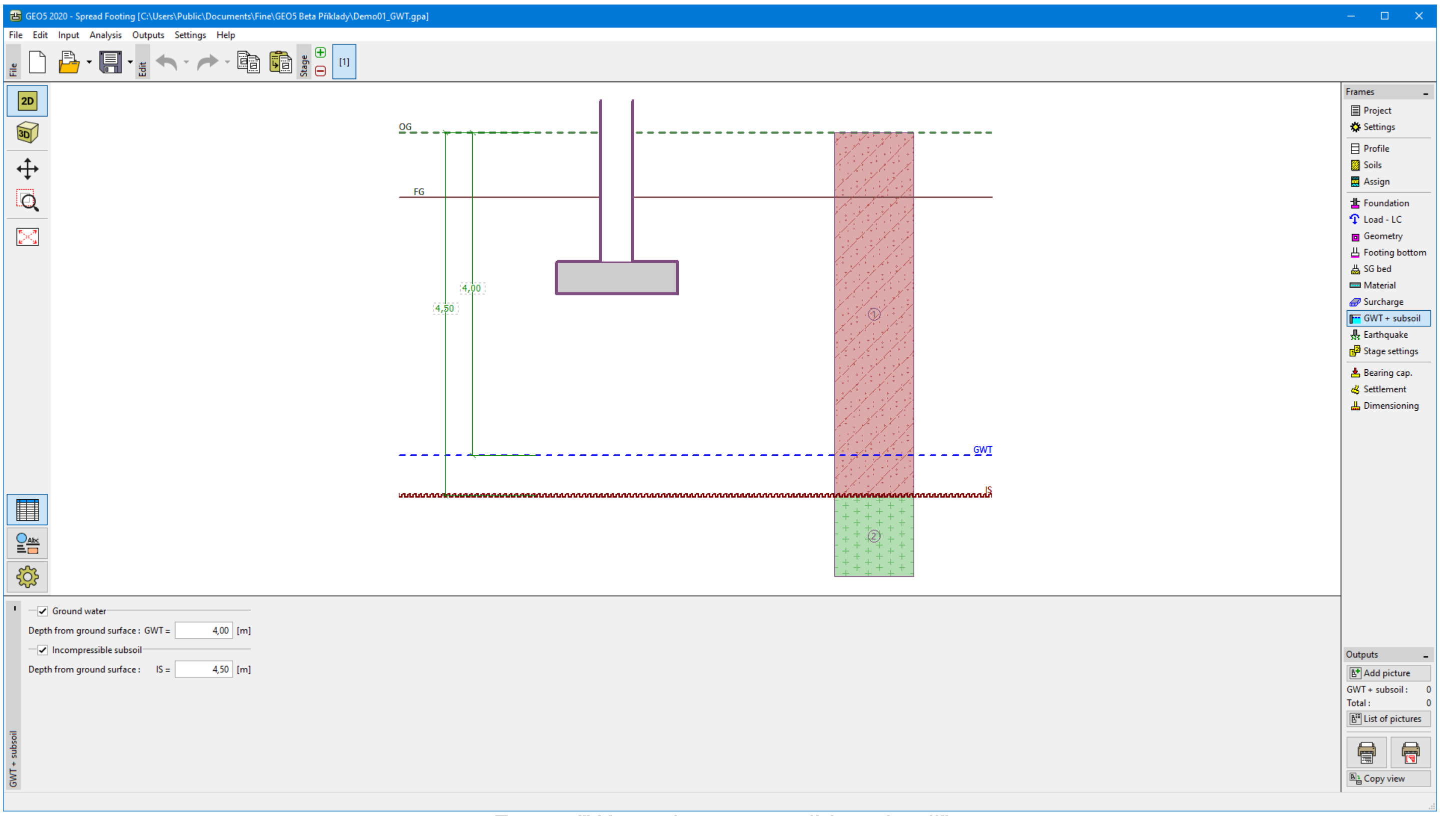
Water, Incompressible Subsoil

The "**Water + IS**" frame serves to specify the **depth of groundwater table** and level of **incompressible subsoil**.

The values can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The **GWT** changes the geostatic stress in the soil profile.

The **incompressible subsoil** cuts the **influence zone** below foundation and also influences reduction factors in settlement calculation.



Frame "Water, incompressible subsoil"

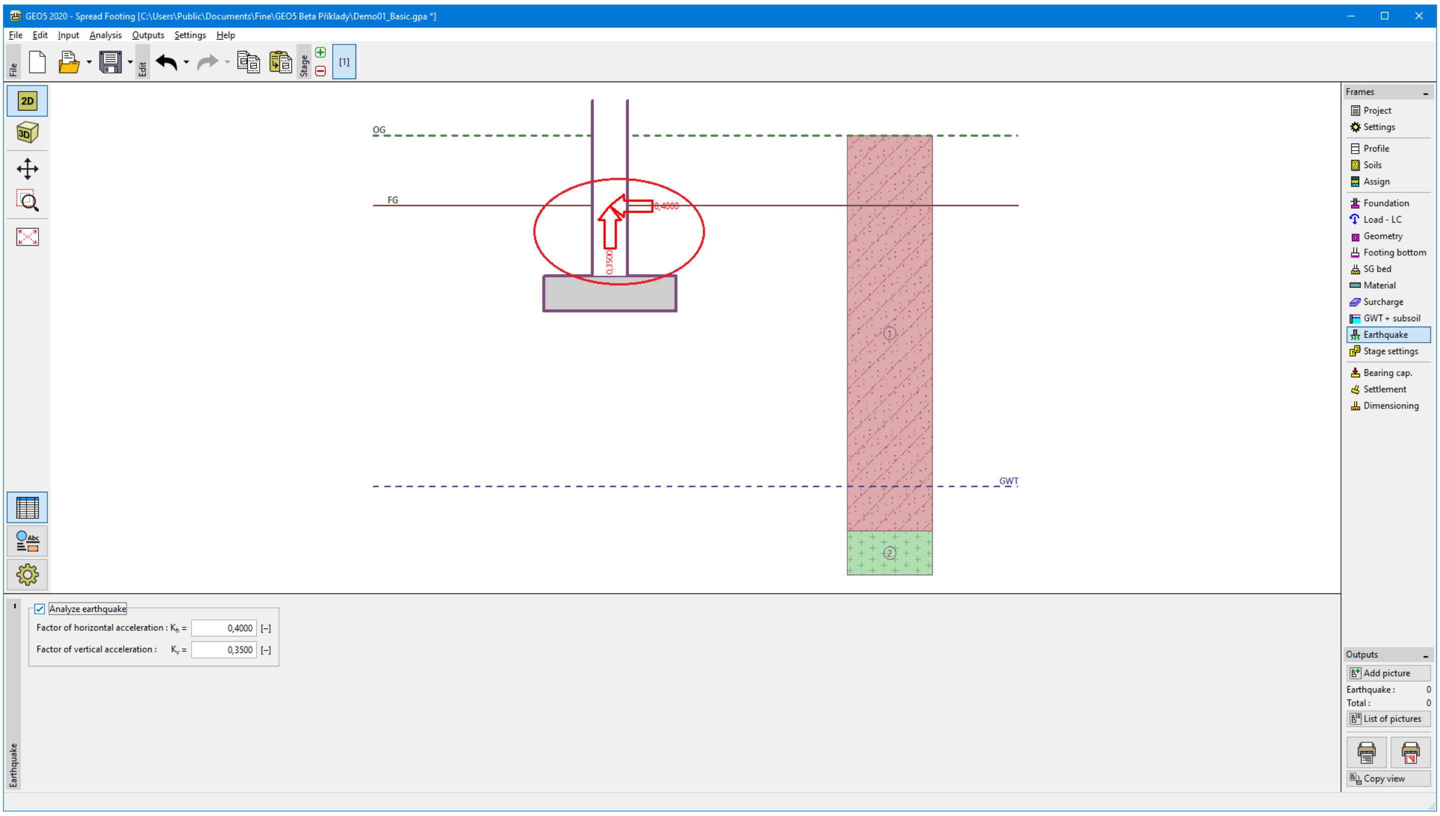
Earthquake

The "**Earthquake**" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements, the coefficients k_h and k_v can be calculated following the approach adopted from EN 1998-5.

Analysis of foundation bearing capacity while accounting for an earthquake is described in the theoretical part of the help, chapter "Analysis of Seismic Bearing Capacity".

During **seismic analysis**, the program performs the calculation of bearing capacity for both cases - with and without seismic effect. The resultant bearing capacity is lower from these two values.



Frame "Earthquake"

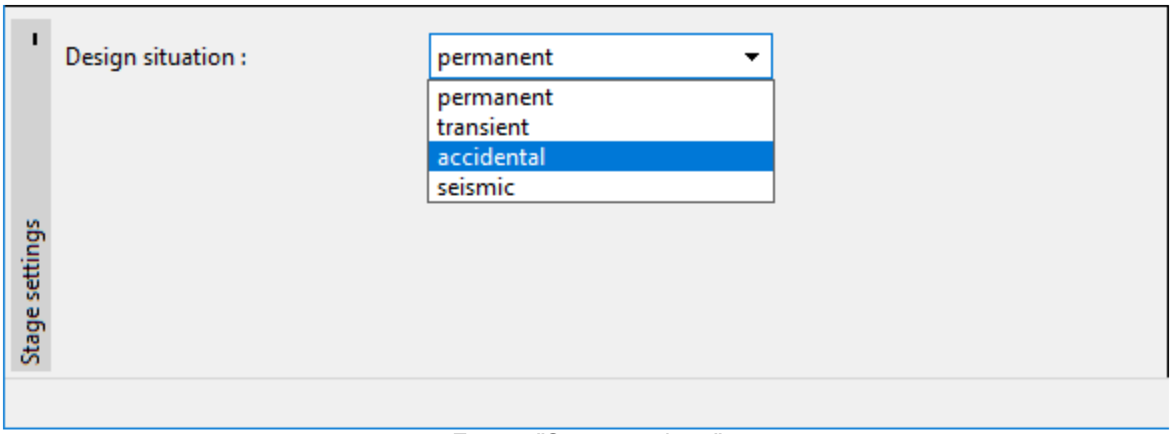
Note: The frame is accessible only in case, when **analysis for drained conditions** is selected in the "Settings" frame.

Stage Settings

The frame "**Stage settings**" serves to input settings valid for a given construction stage.

The selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology.



Frame "Stage settings"

Bearing Capacity

The "**Bearing capacity**" frame serves to verify the **vertical and horizontal bearing capacity of a footing**. More computations can be performed in the frame. The verification can be performed for individual loads, or the program finds the **most critical one** (can be selected from a combo list).

The analysis follows the theory approach selected in the "Spread Footing" tab. This tab serves to choose the verification methodology (according to EN 1997, LRFD, factor of safety, limit states).

The vertical bearing capacity analysis requires the selection of the type of contact pressure (general shape, rectangle). The shape of contact pressure is plotted in the left part of the desktop.

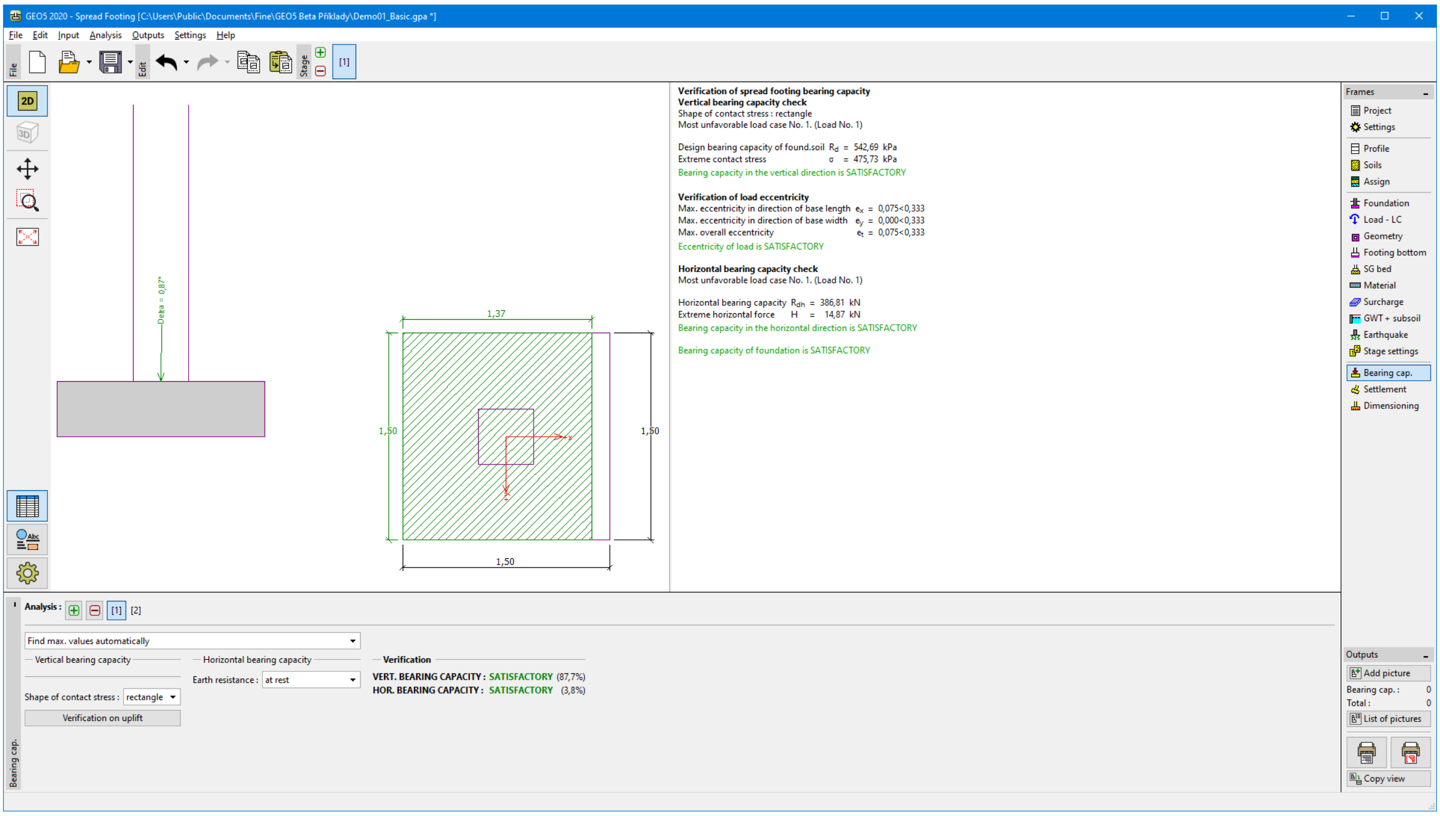
For both limit states (strength, usability) program assesses the eccentricity of the foundation. The value of the maximum allowable eccentricity of foundation e_{alw} is assumed in the frame "**Settings**" in the "Spread Footing" tab.

The horizontal bearing capacity analysis requires the selection of the type of earth resistance that can be assumed as the **pressure at rest, passive pressure,** or the **reduced passive pressure**.

The **soil parameters** (friction angle structure-soil, cohesion structure-soil) can be further **reduced** when computing the horizontal bearing capacity.

When evaluating the uplift resistance, the view of the "**Verification on uplift**" dialog window is adjusted according to the analysis method selected in the "Settings" frame.

A detailed listing of the results is displayed in the right part of the desktop. Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Bearing capacity"

Settlement and Rotation

The "**Settlement**" frame serves to compute the foundation settlement and rotation. The frame allows for more analyses. The verification can be performed for individual loads, or the program finds the **most critical one** (can be selected from combo list).

The analysis of foundation settlement and rotation is carried out according to the theory specified in the frame "Spread Footing" tab.

For both limit states (strength, usability) program assesses the eccentricity of the foundation. The value of the maximum allowable eccentricity of foundation e_{alw} is assumed in the frame "**Settings**" in the "Spread Footing" tab.

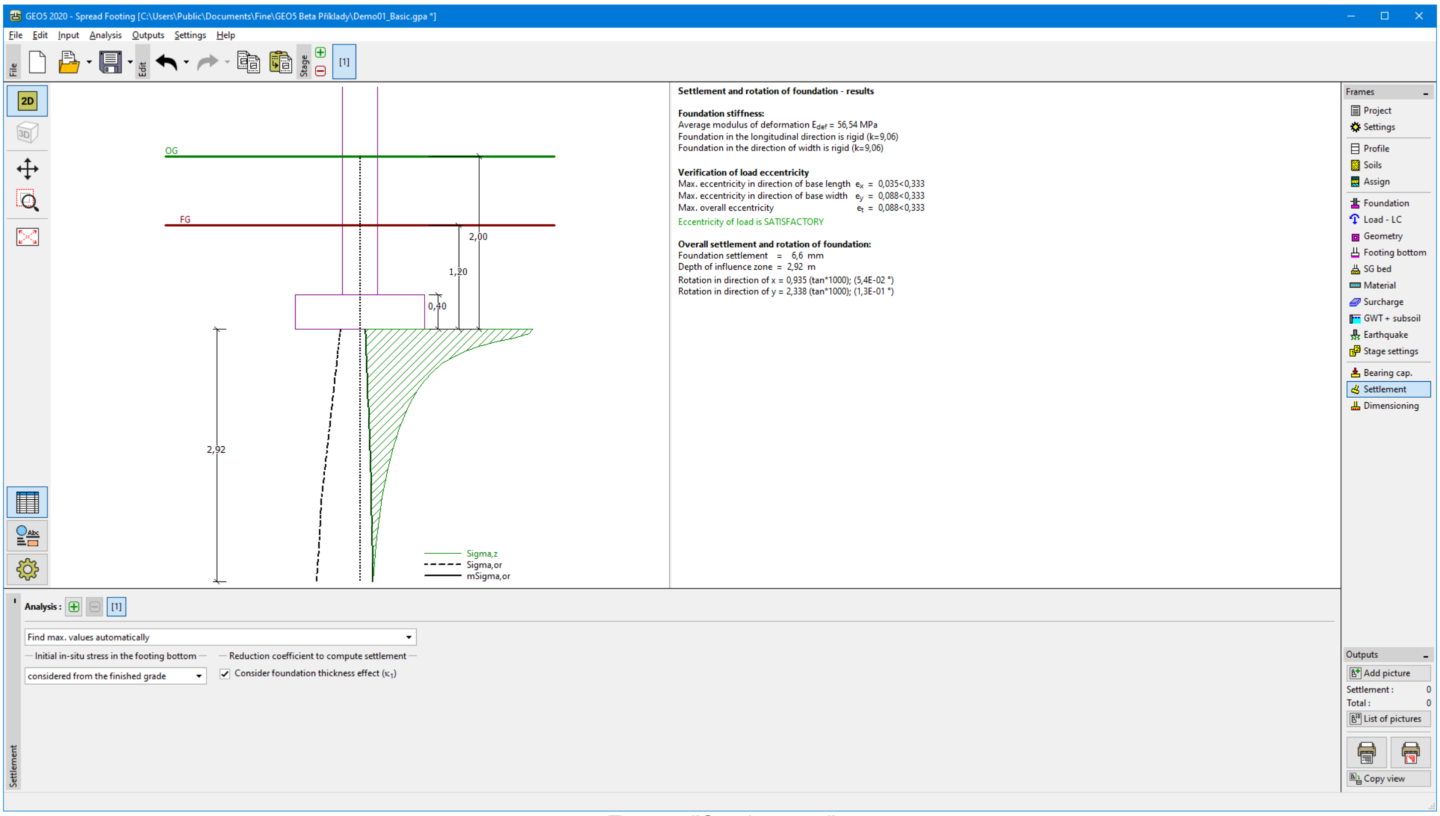
The stress in the footing bottom can be subtracted from the geostatic stress given by:

- **original ground**
- **finished grade**
- **not specified**

Distributions of the **geostatic stress** and the **stress increment** below foundation are displayed in the left part of the desktop. The label below footing represents the **depth of the deformation zone**. The stress is drawn below footing at the point with a characteristic deformation.

The frame also allows for specifying the coefficient of reduction of computation of settlement.

The detailed listing of the verification analysis results is displayed in the right part of the desktop. Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Settlement"

Dimensioning

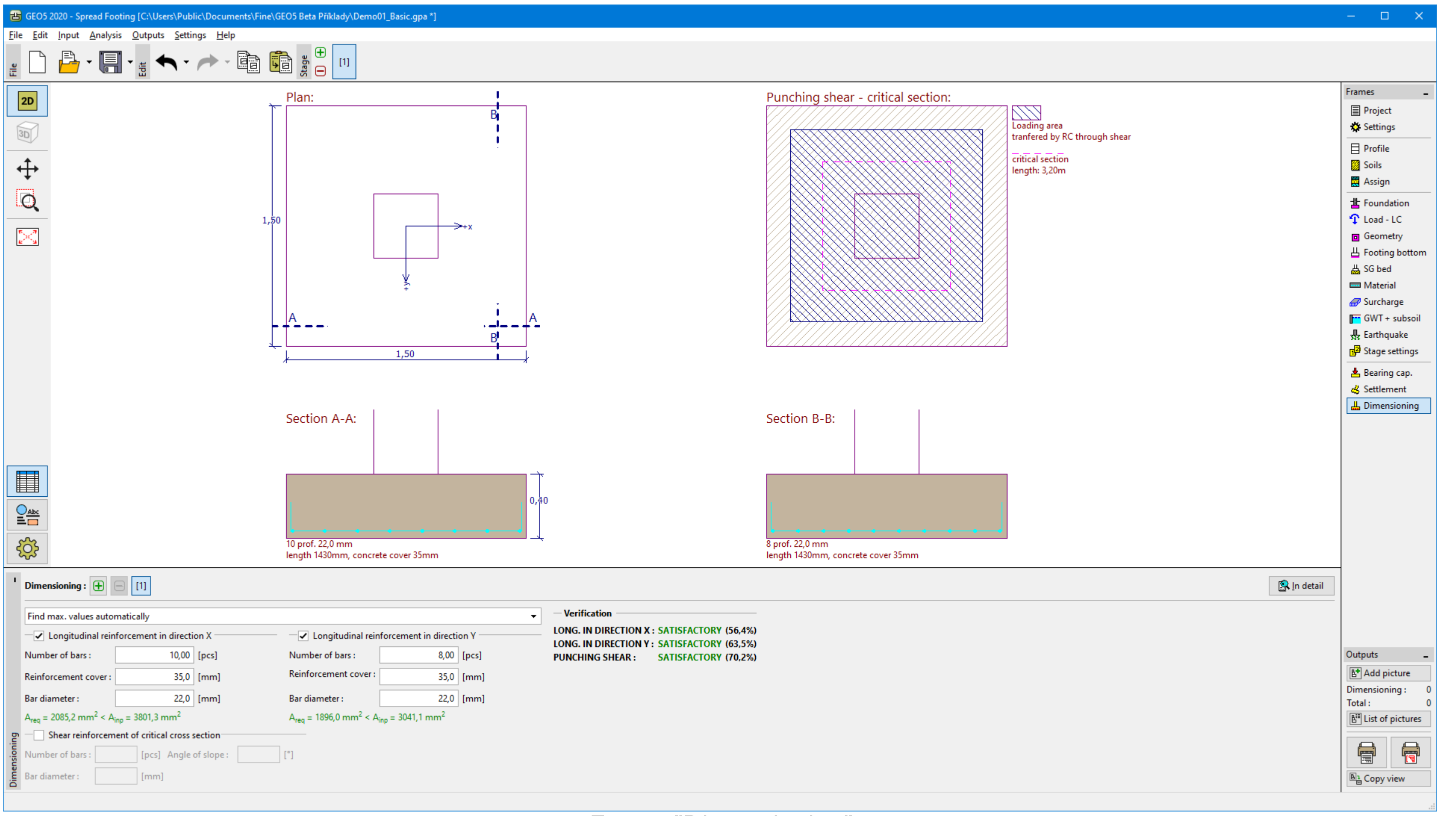
The "**Dimensioning**" frame allows for designing and verifying the longitudinal reinforcement of a foundation and also for verifying the foundation against being pushed through. The verification can be performed for individual loads, or the program finds the **most critical one** (can be selected from a combo list).

The program derives the stress in the construction joint and determines the internal forces in individual cross-sections.

Dimensioning of the reinforced concrete is performed according to the standard set in the "Materials and standards" tab.

The resulting information are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "**In detail**" button opens the dialog window that contains a detailed listing of the dimensioning results.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Dimensioning"

Program Spread Footing CPT

This program is used for the design and verification of spread footings based on data from cone penetration tests (CPT), standard penetration tests (SPT), or the Pressumeter test (PMT).

It computes vertical bearing capacity, settlement and determines required longitudinal and shear reinforcement (punching).

The help in the program "Spread Footing CPT" includes the following topics:

- The input of data into individual frames:

Project	Settings	CPT	SPT	PMT	Soil Classification	Profile
Soils	Assign	Water	Foundation	Load	Geometry	Material
Analysis	Dimensioning					

- Standards and analysis methods

- Theory for analysis in the program "**Spread Footing CPT**":

  Stress in Soil Body — Spread Footing CPT — Field Testing

- Outputs

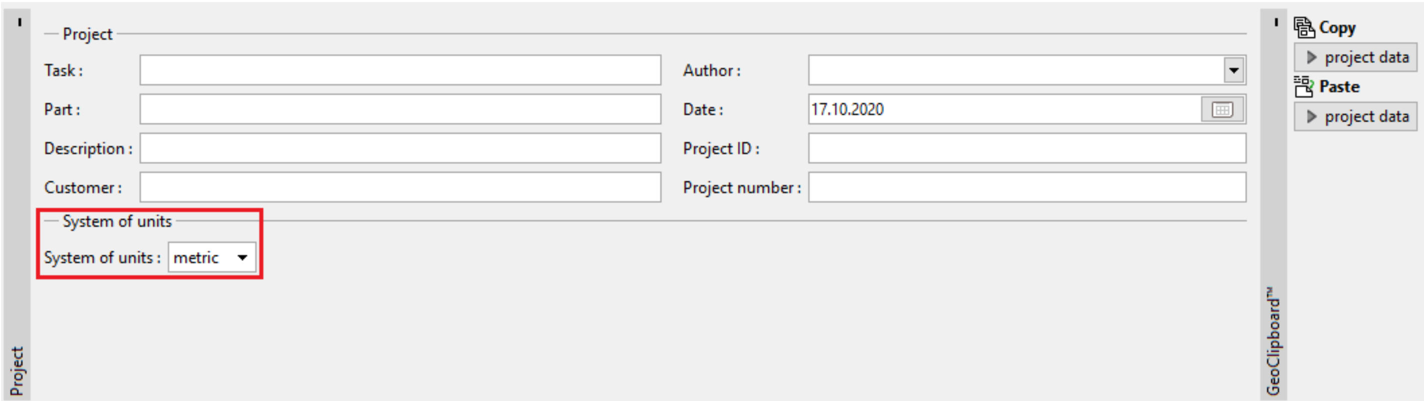
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The frame **"Project"** is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The frame "**Settings**" allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose already created settings from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

Settings of analysis parameters are performed in the "Materials and standards".

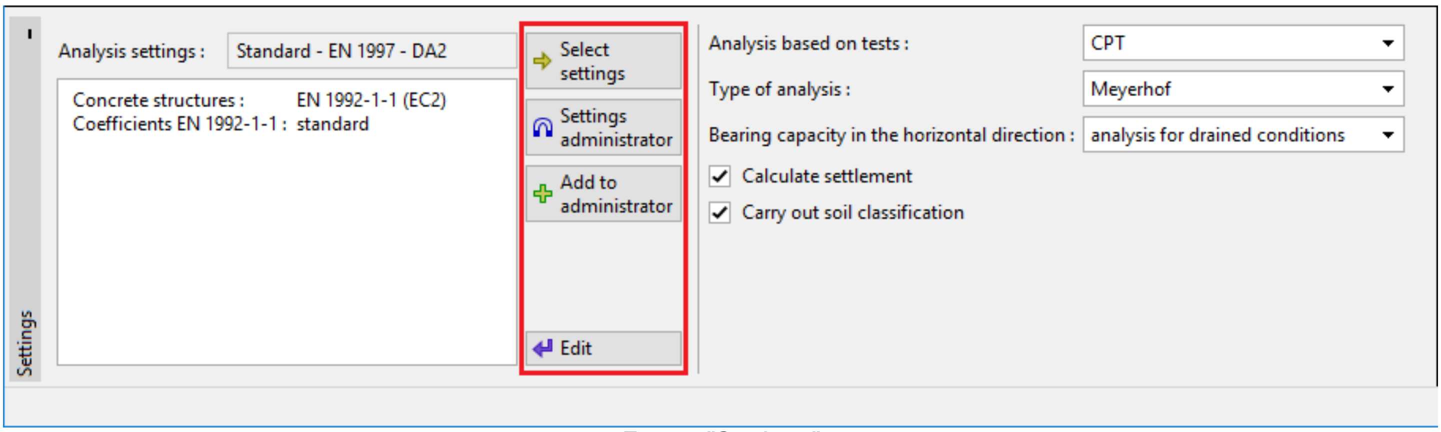
The type of field tests (CPT, SPT, PMT) used for the analysis of bearing capacity and settlement is selected in this frame. According to the selected type of field tests, the frame "CPT", "SPT" or "PMT" is displayed.

In case of CPT tests, the type of analysis for bearing capacity is also selected (Meyerhof, Schmertmann, Skempton).

In case of analysis according to the Meyerhof method (CPT or SPT), the frames "Profile", "Soils" and "Assign" are not displayed. All calculations are based just on the data from field tests.

For horizontal bearing capacity the type of analysis is selected (drained conditions, undrained conditions).

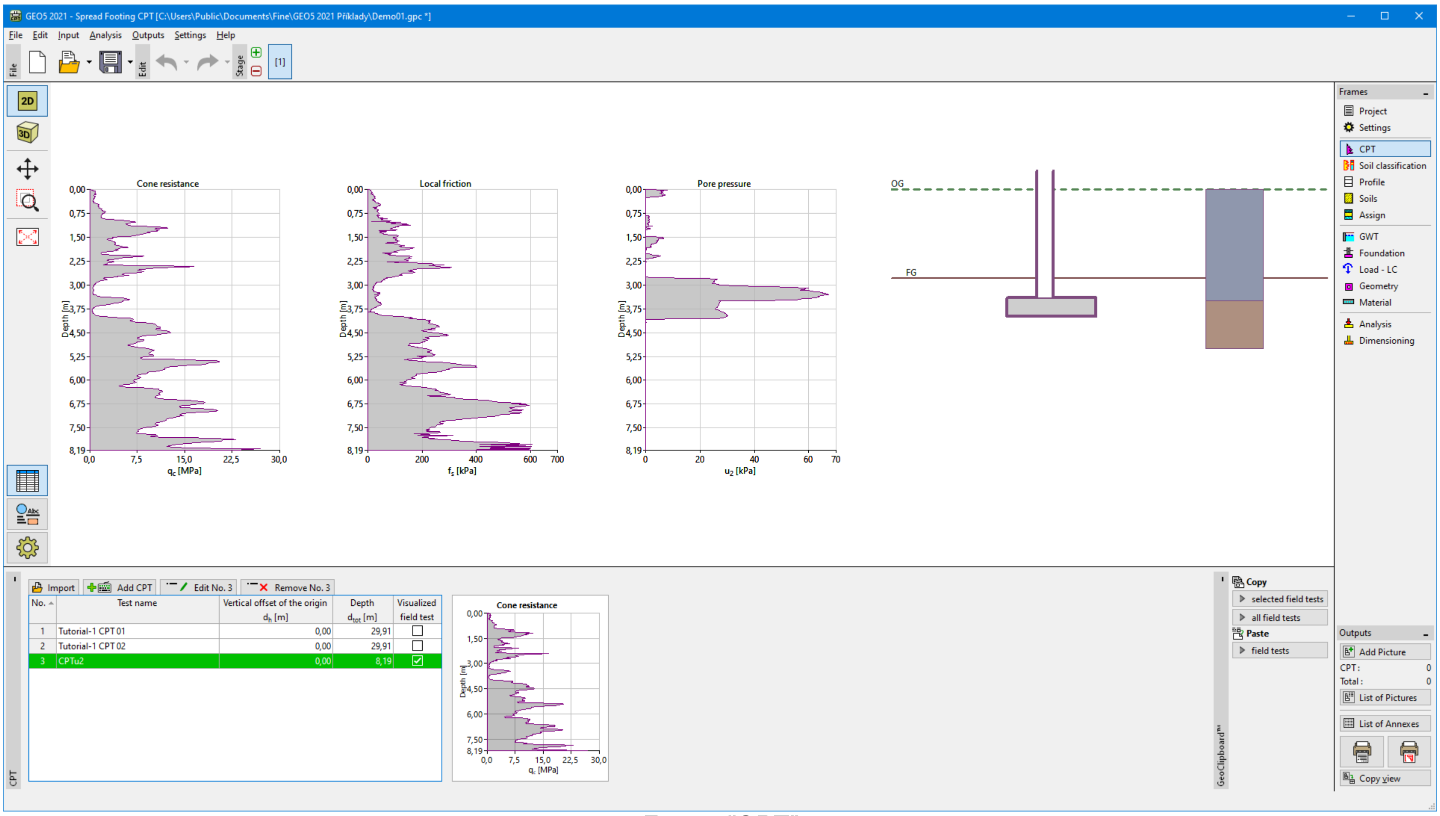
In this frame is also possibility to decide if we want to analyse spread footing settlement or carry out soil classification.



Frame "Settings"

CPT

The frame "**CPT**" contains a table with a list of cone penetration tests (CPT).



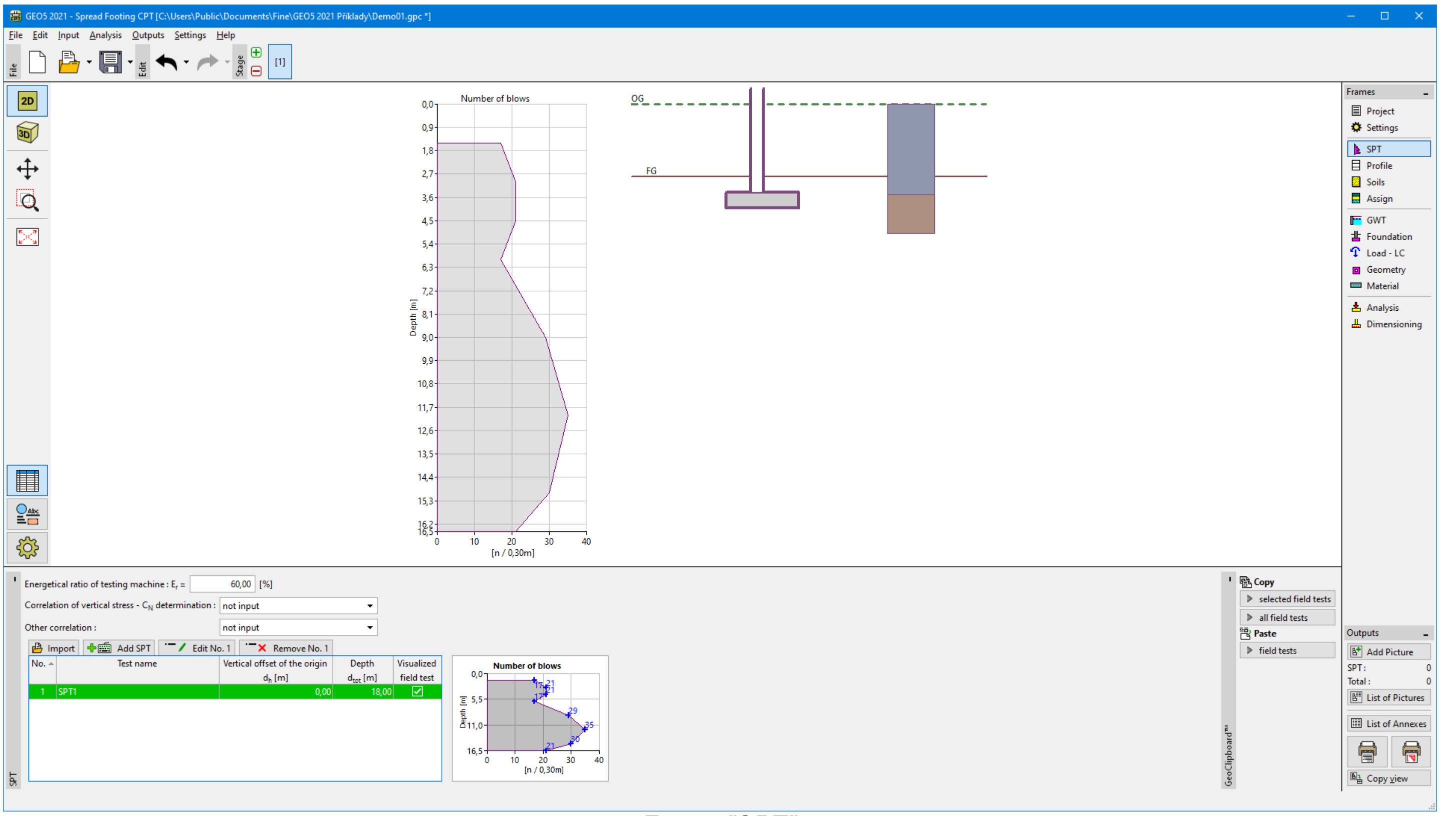
Frame "CPT"

The results of cone penetration tests (CPT) can be imported into the program in many formats (eg. ***.CPT**, ***.GEF**, ***.AGS**, ***.SPE**, ***.GRU**,***.xlsx**, ***.csv**, ***.ods**,***.txt**).

Data of CPT can be copied within "**Pile CPT**", "**Spread Footing CPT**" and "**Stratigraphy**" programs using "GeoClipboard".

SPT

The frame "**SPT**" contains a table with a list of standard penetration tests (SPT).



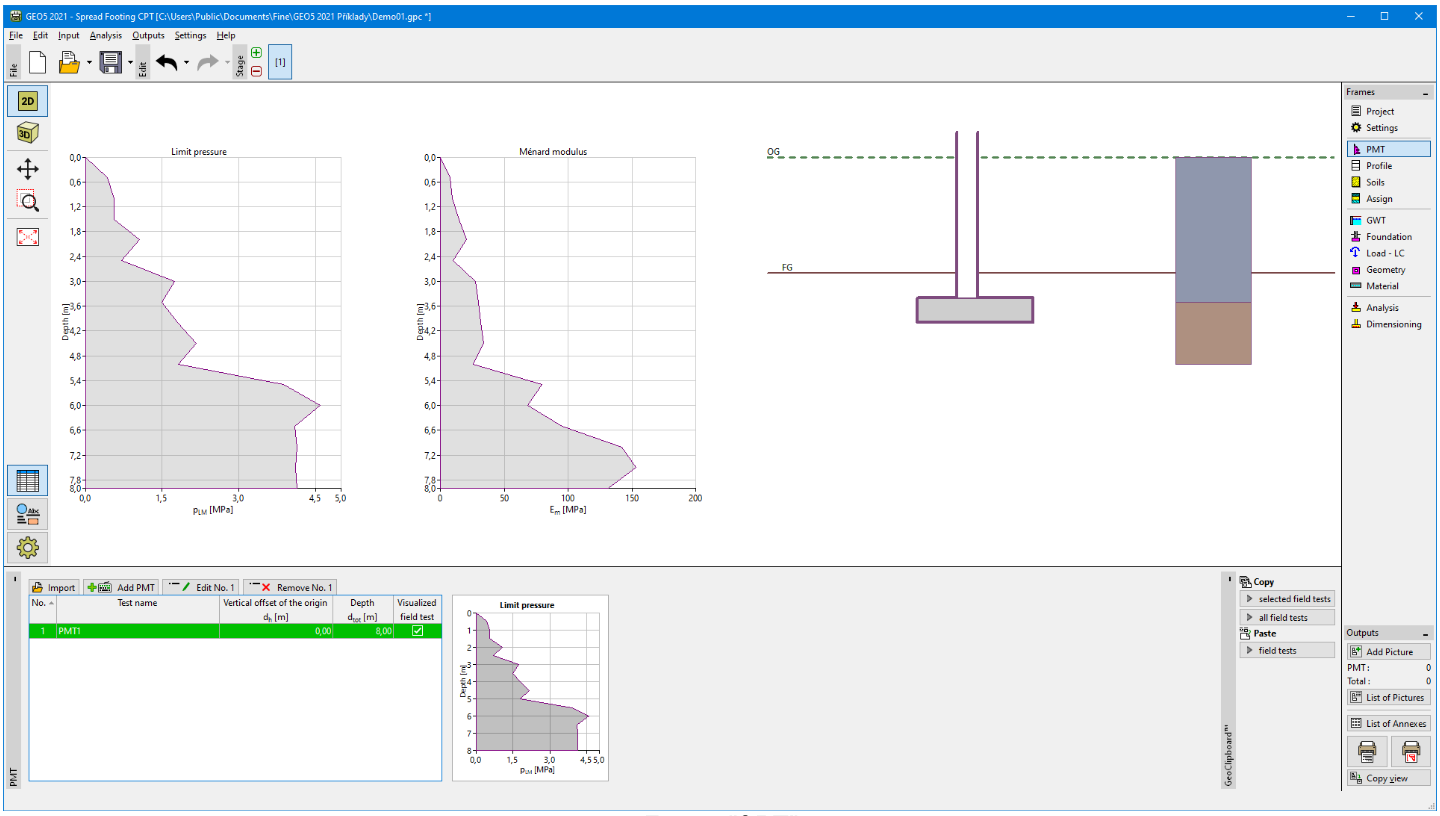
Frame "SPT"

The results of standard penetration tests (SPT) can be imported into the program by inserting the file in different formats (eg. ***.TXT**, ***.CSV**, ***.XLSX**, ***.ODS**).

Data of SPTs can be copied within "**Micropile**", "**Spread Footing CPT**" and "**Stratigraphy**" programs using "GeoClipboard".

PMTs

The frame "**PMT**" contains a table with a list of pressuremeter tests (PMT).



Frame "SPT"

The results of pressumeter tests (PMT) can be imported into the program by inserting the file in different formats (eg. ***.TXT, *.CSV, *.XLSX, *.ODS**).

Data of PMTs can be copied within "**Micropile**", "**Sheeting Check**", "**Anti-Slide Pile**" "**Spread Footing CPT**" and "**Stratigraphy**" programs using "GeoClipboard".

Soil Classification

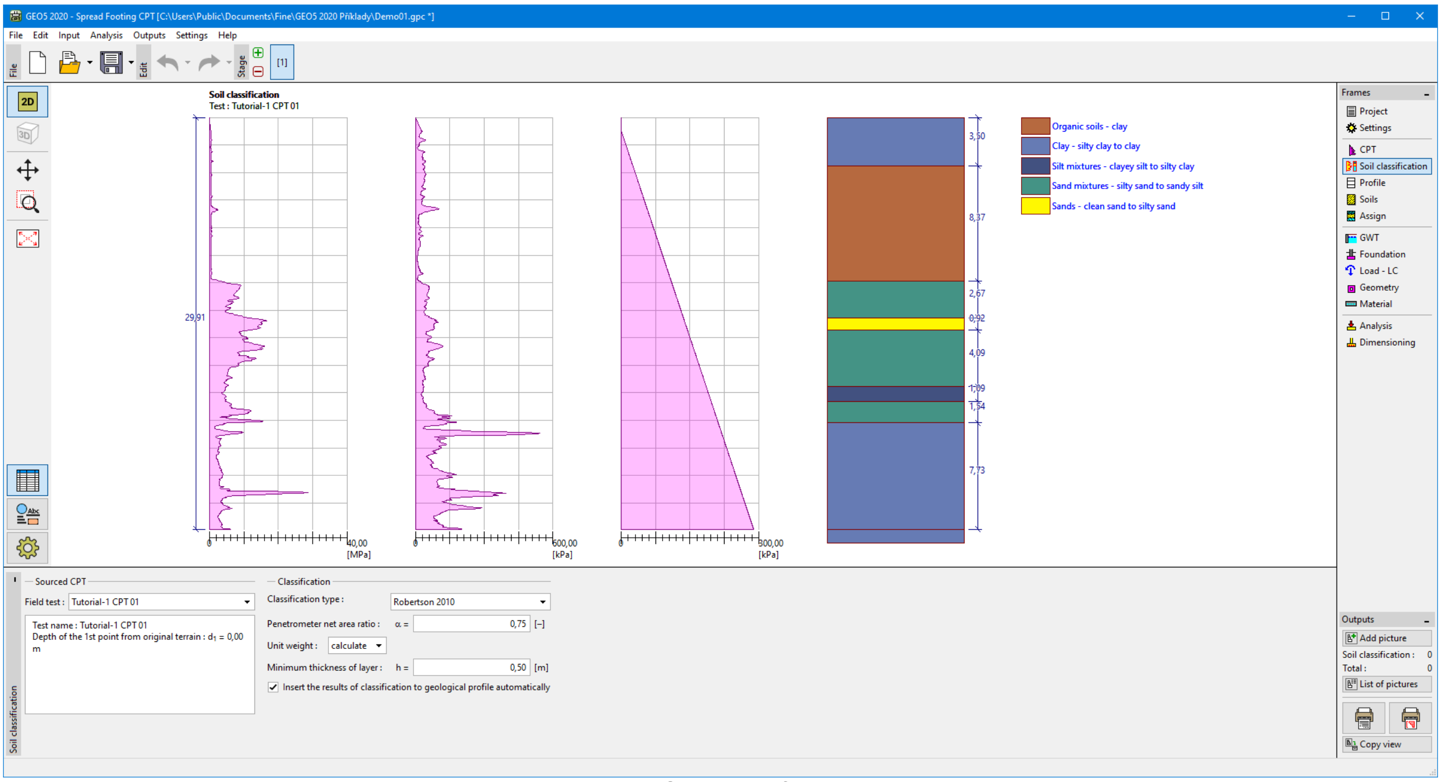
Classification of soils according to Robertson (1986 or 2010) allows us to specify soil behavior type (SBT) and other parameters directly from the results of CPT - then input parameters of soils it's not necessary to input. We recommend to check generated soil parameters before the calculation.

In the frame "**Soil classification**" is selected **test for classification** (defined in the "CPT" frame). Classification of soils is performed according to Robertson (1986 or 2010).

This frame serves to input penetrometer net area ratio α [-].

The unit weight of soil γ is possible to input with the same value for all layers of soils or it's calculated automatically from the results of CPT (for each layer separately). The frame serves to input a minimum thickness of the layer of soil h . It affects the distribution and number of individual layers of soil in the geological profile of solved task.

By checking the option "**Insert the results of classification into a geological profile automatically**" the generated geological profile assigns to the analysis automatically.



Frame "Soil classification"

Profile

The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

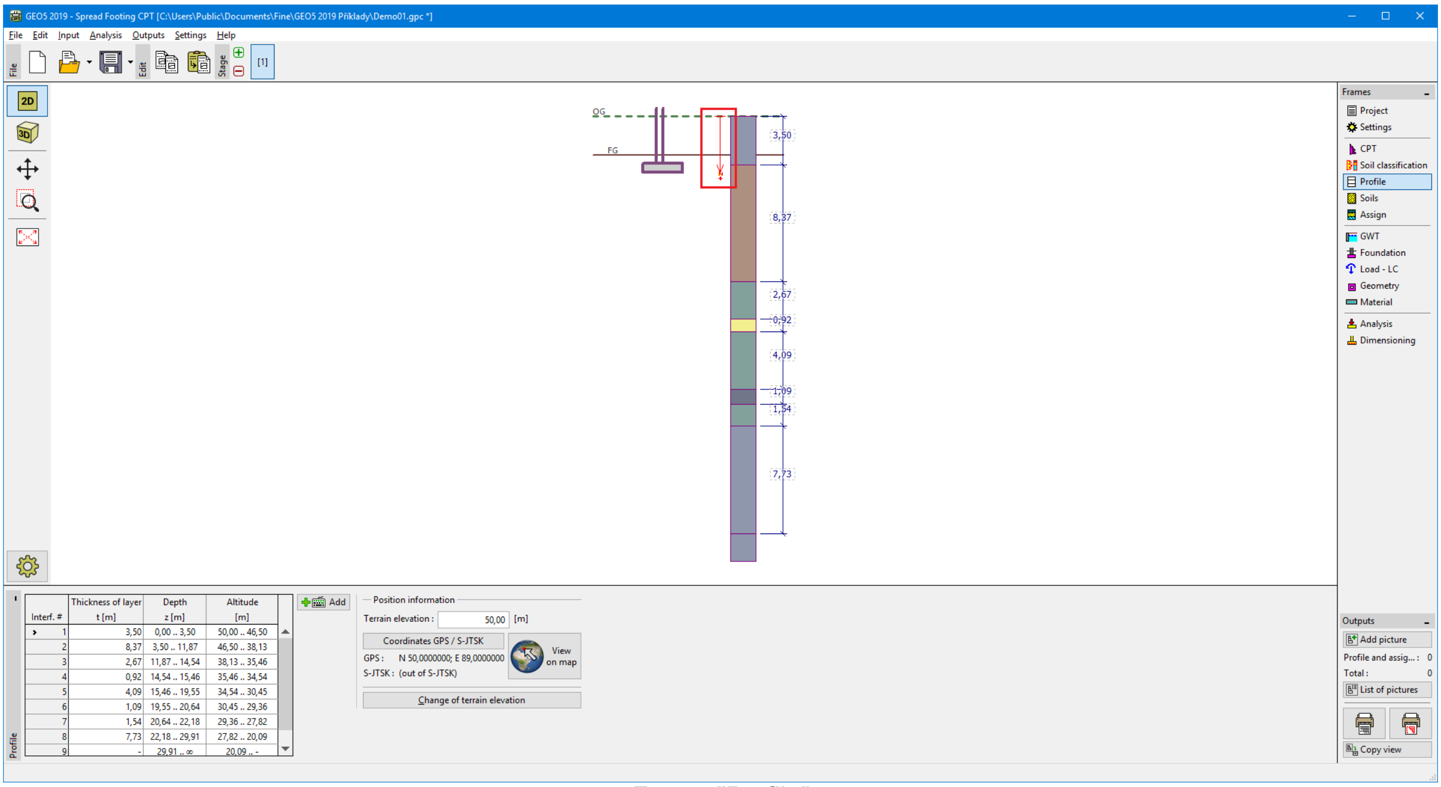
Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.

The frame "**Profile**" is not available in case of analysis according to the Meyerhof method (CPT or SPT).



Frame "Profile"

Soils

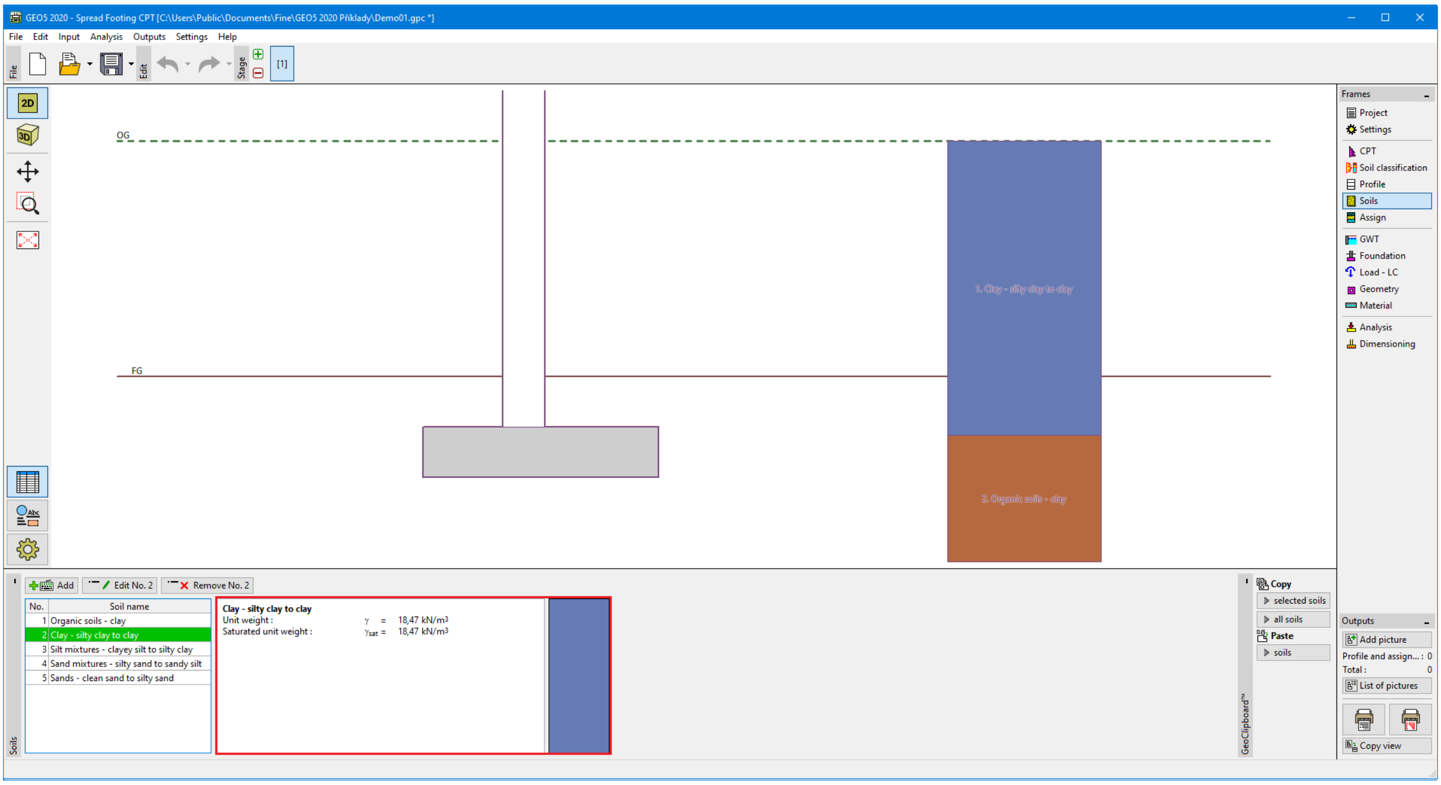
The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

In the program "**Spread Footing CPT**", just input of unit weight γ and saturated unit weight γ_{sat} is required. These characteristics are used for calculation of geostatic stress and uplift pressure.

The frame "**Profile**" is not available in case of analysis according to the Meyerhof method (CPT or SPT).

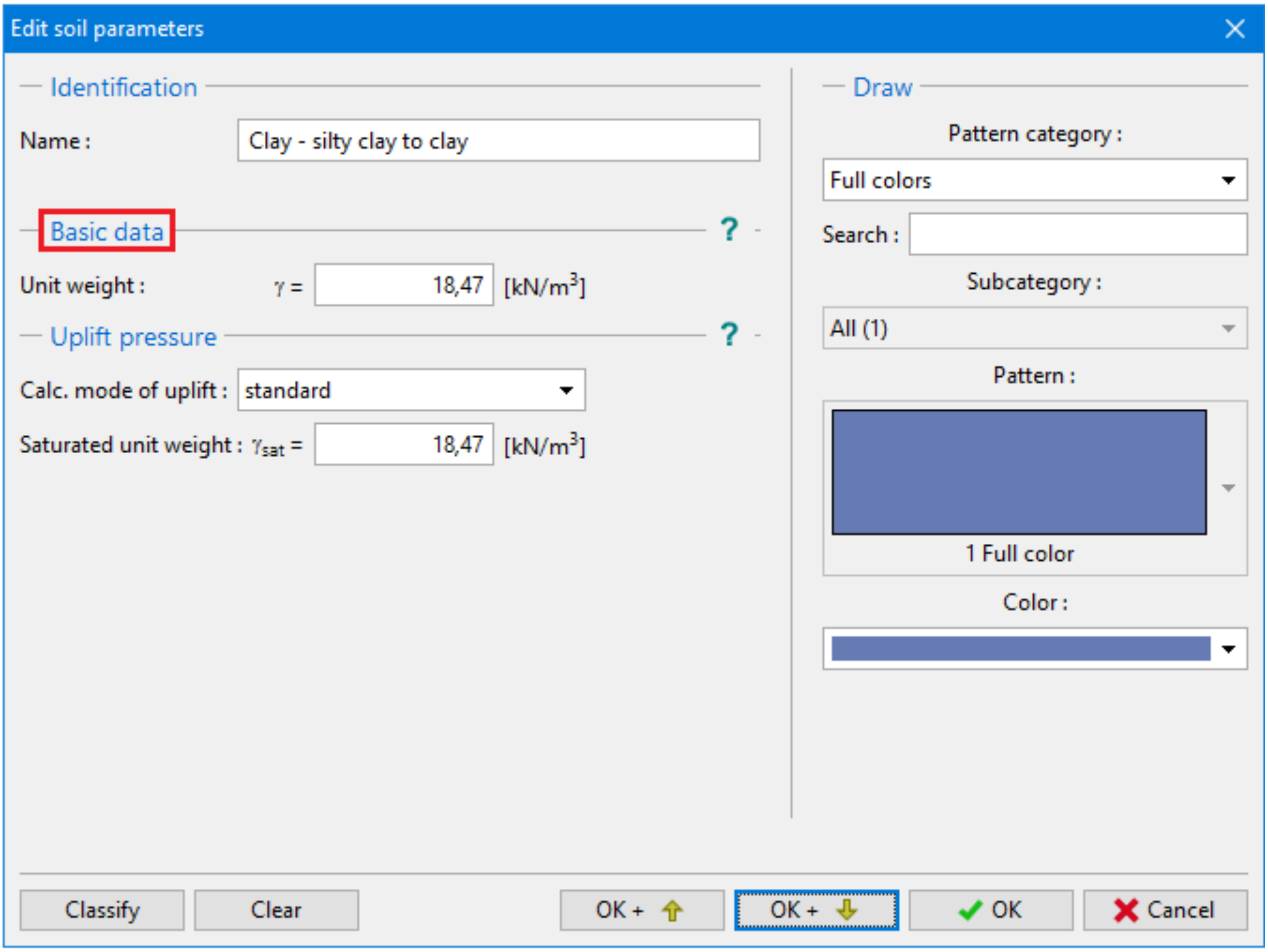
Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

Basic Data

This part of the dialog window serves to specify **the unit weight of soil**.



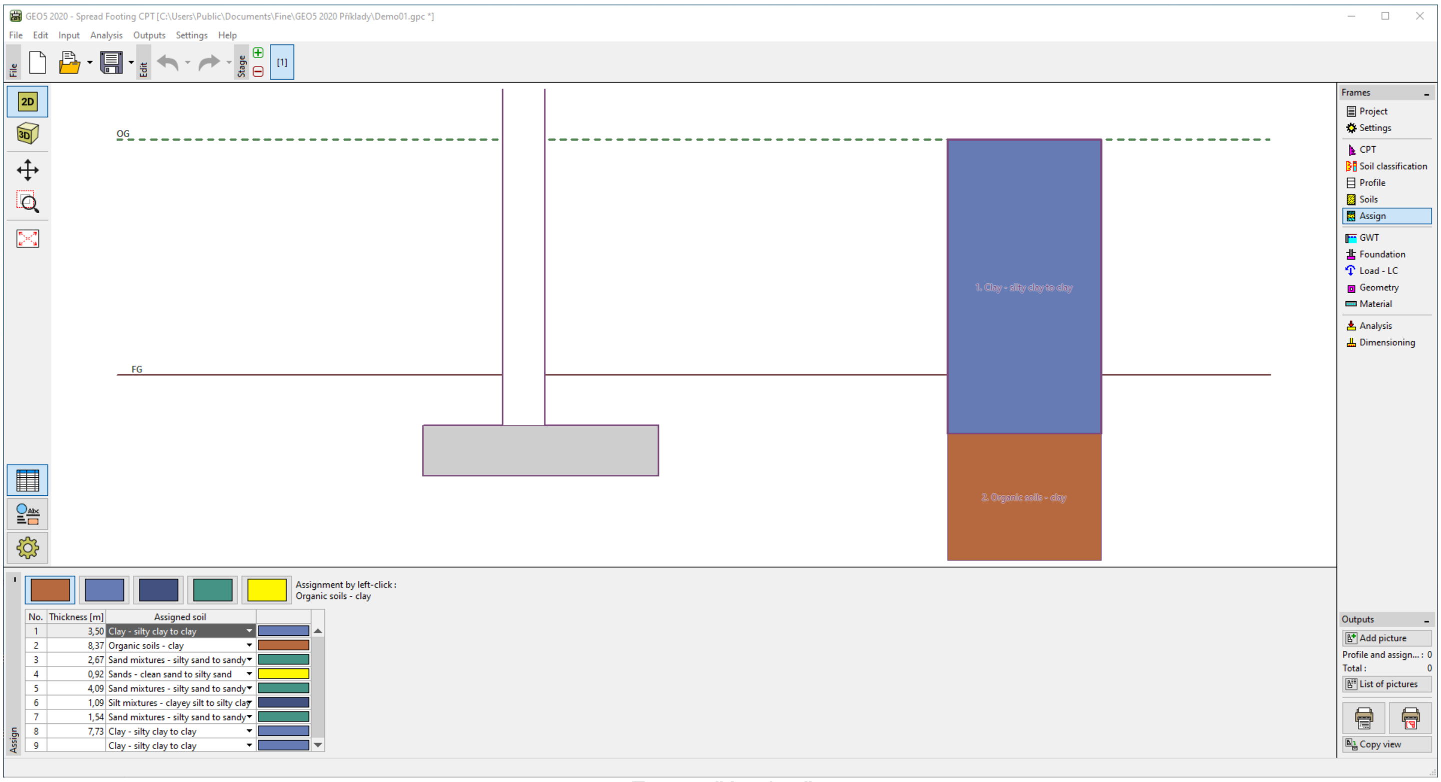
Dialog window "Add new soils" - "Basic data"

Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or it is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail herein.

The frame "**Assign**" is not available in case of analysis according to the Meyerhof method (CPT or SPT).

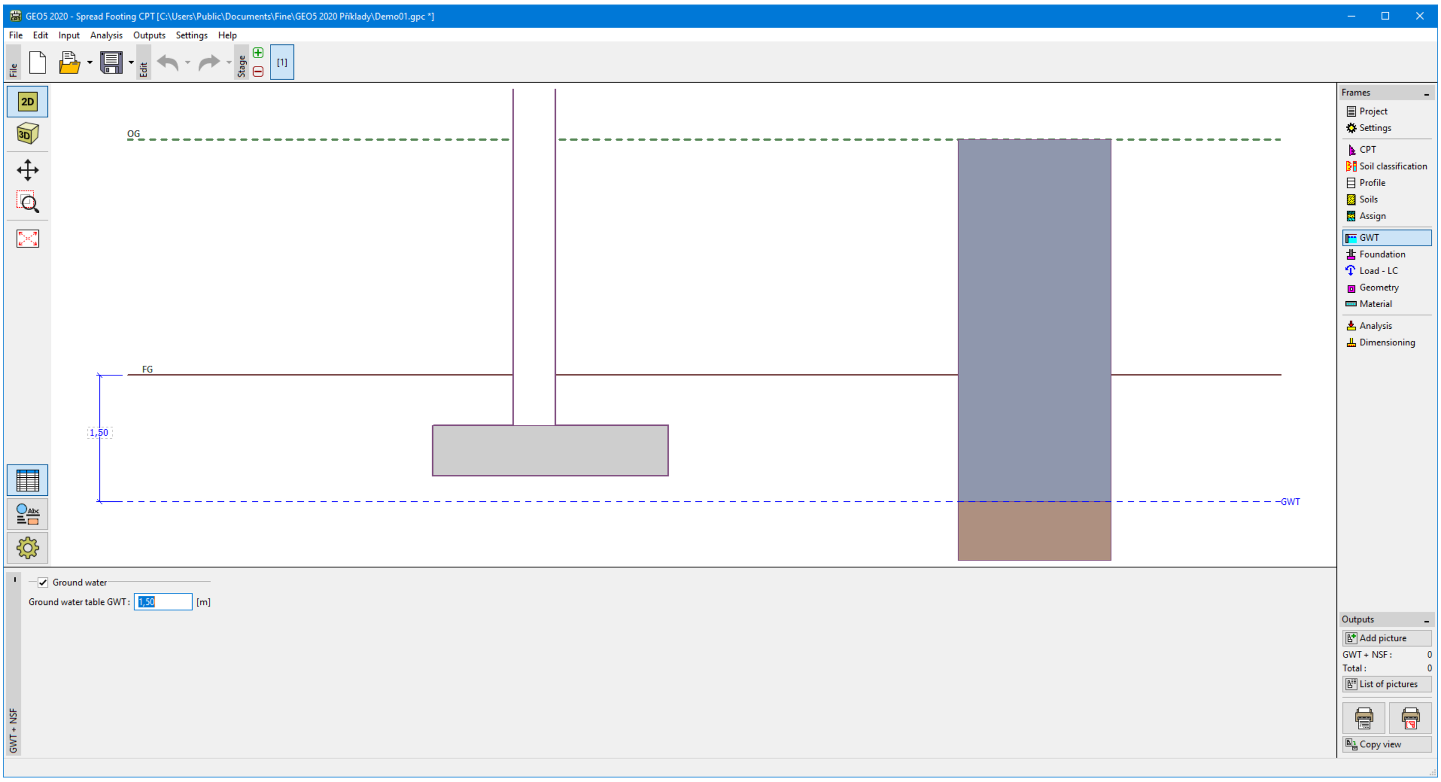


Frame "Assign"

Water

The "**GWT**" frame serves to specify the **depth of groundwater table**. The values can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The **GWT** changes the geostatic stress in the soil profile.



Frame "Water"

Foundation

The "**Foundation**" frame allows us to select a type of foundation. The selected type with a graphic hint of input values is displayed in the left part of the frame. The values can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions. The frame also serves to specify the unit weight of overburden.

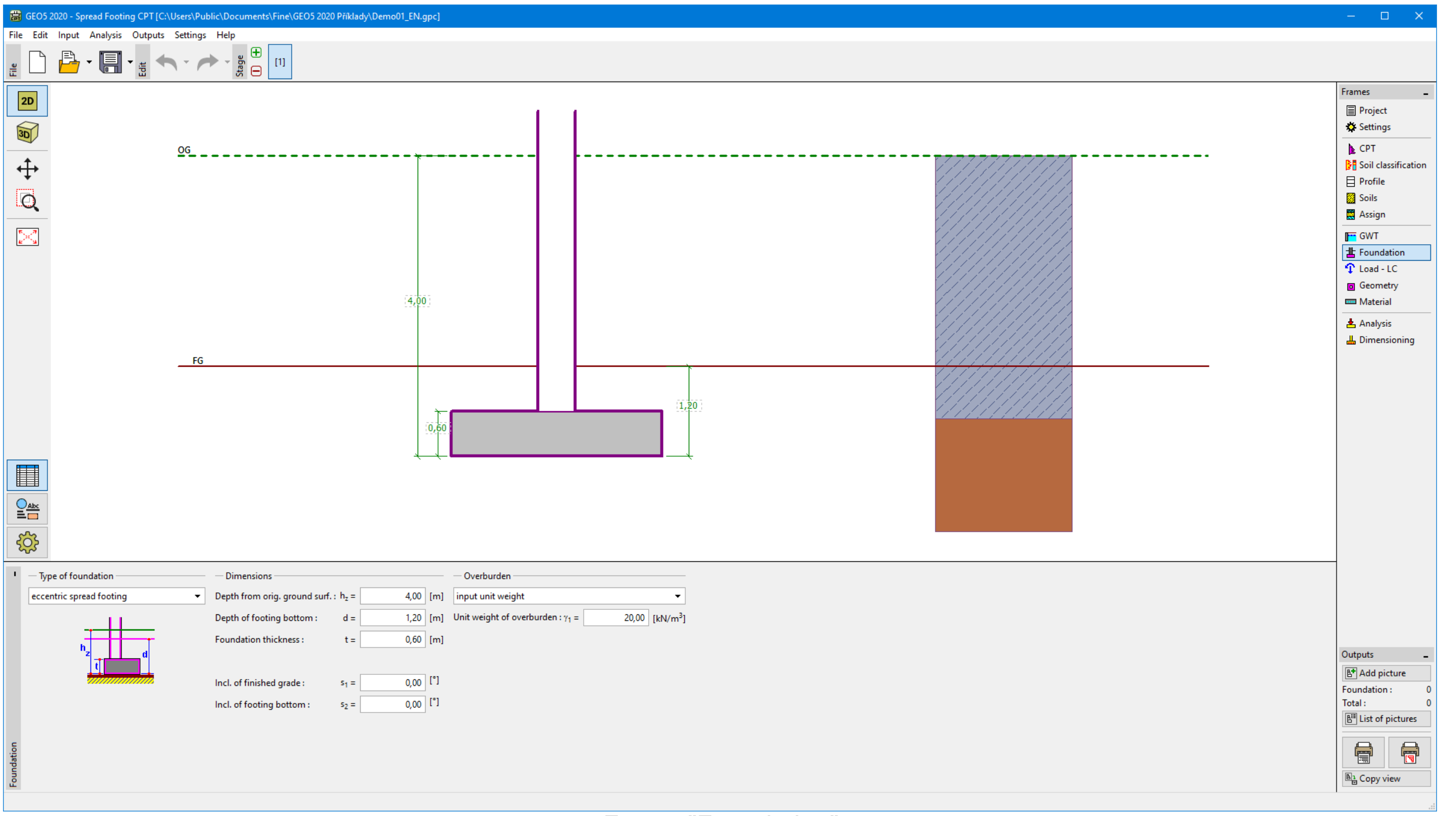
The following types of foundations can be selected:

- **Centric spread footing**
- **Eccentric spread footing**
- **Strip footing**
- **Stepped centric spread footing**
- **Stepped eccentric spread footing**
- **Circular spread footing**
- **Circular stepped spread**
- **Centric spread footing with batter**
- **Eccentric spread footing with batter**
- **Centric spread footing with circular step**

The soil profile is specified from the **original ground.** The foundation bearing capacity depends mainly on the depth **of foundation measured from the finished grade**. When the finished grade is found above the original ground, then it is required to assign the same depth to both, the finished grade and original ground, and introduce into subsoil a layer with a new made-up-ground. This frame also allows for inputting the **foundation thickness**.

The **overburden type** is chosen from the following options:

- **input unit weigh** - the overburden unit weight γ_1 is specified
- **from geological profile** - the overburden unit weight is calculated from the soils assigned in the geological profile
- **input shape and soil** - this option allows us to model the overburden shape from the **bottom** or **upperedge** of footing. The overburden unit weight is determined from the assigned soil.



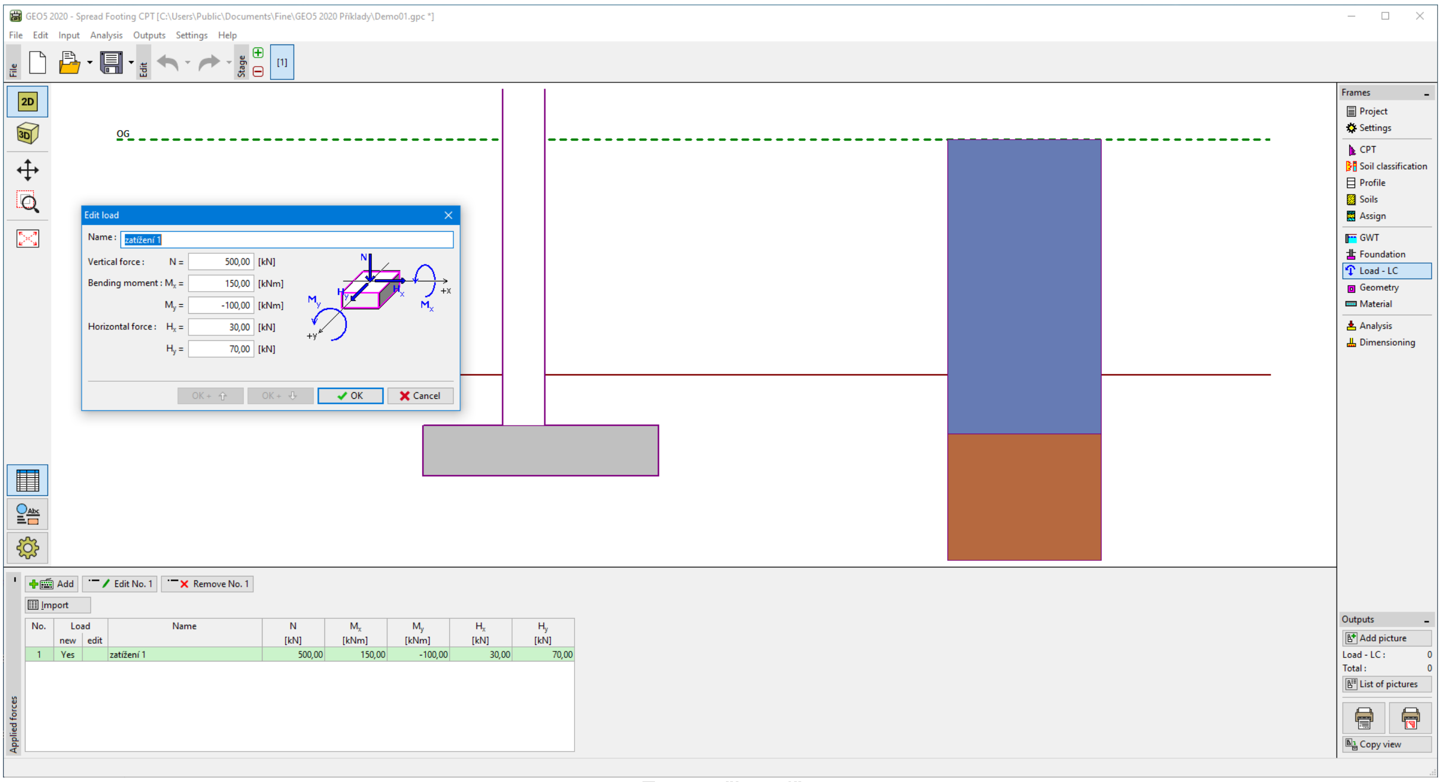
Frame "Foundation"

Load

The "**Load**" frame contains a table with a list of input loads. Adding loads is performed in the "**New load**" dialog window. Input of individual forces follows the sign convention displayed in the right part of the dialog window.

The foundation is always loaded at the contact point between column and foundation. The program automatically computes the **foundation self-weight** and the **weight of overburden**.

The program also allows importing of load using the "**Import**" button.



Frame "Load"

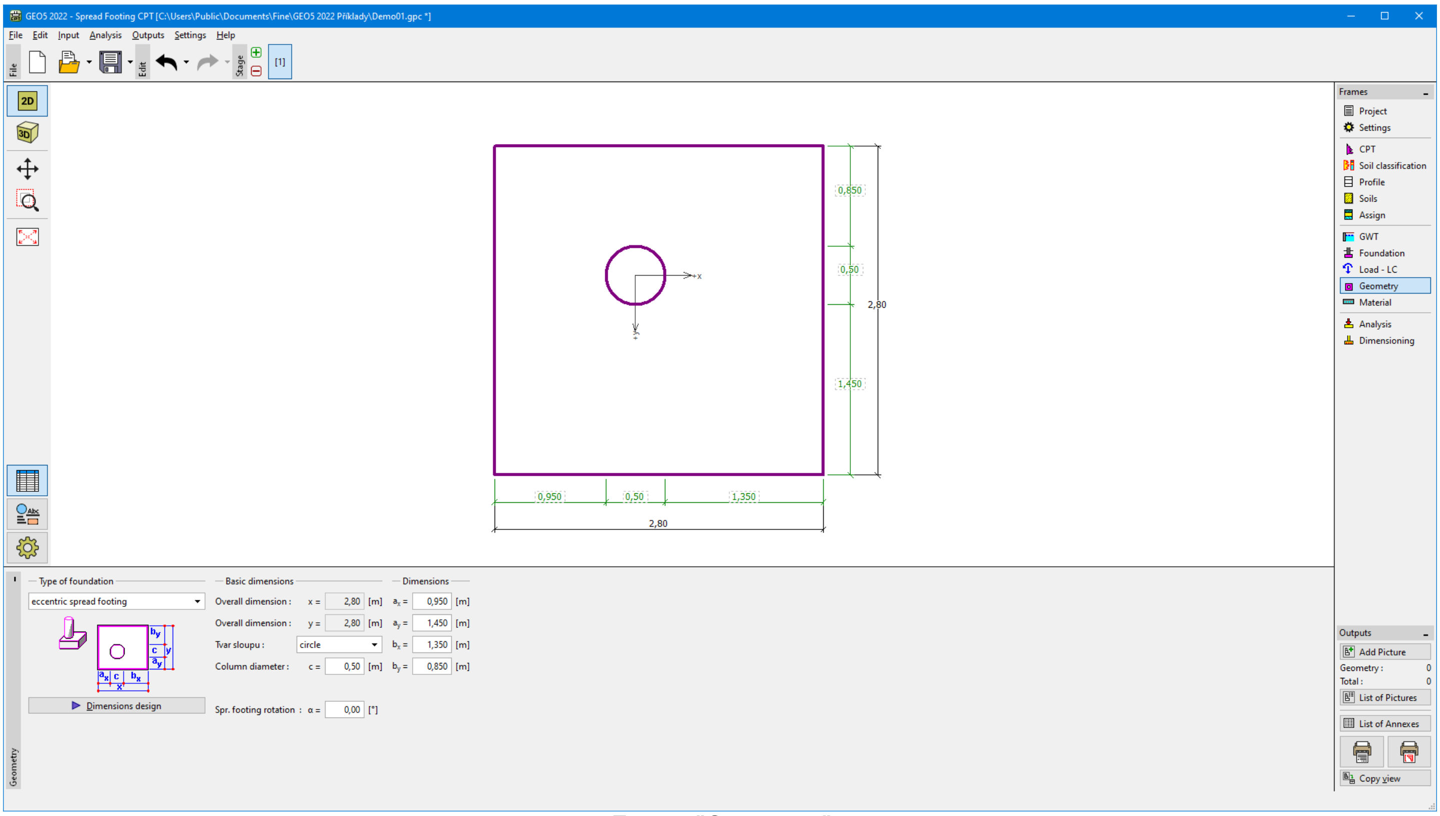
Geometry

The "**Geometry**" frame allows us to specify the foundation and column shape. The selected shape with a graphic hint of input values is displayed in the left part of the frame. The values can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

Foundation type and its **thickness** are specified in the "Foundation" frame.

The program automatically computes the **self-weight of both foundation** and **overburden above the foundation**. The foundation self-weight is specified in the "Material" frame.

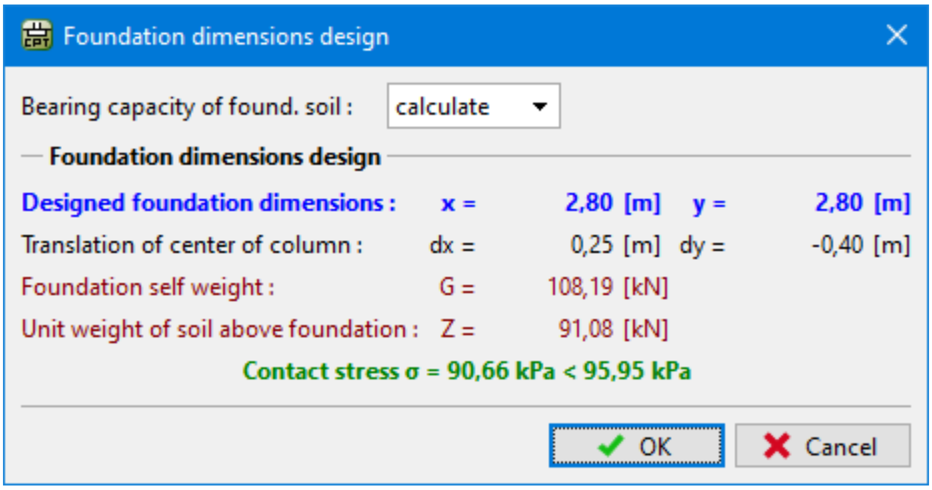
The program makes it possible to export the geometry of a structure in the *.DXF format.



Frame "Geometry"

The "**Dimensions design**" button opens the dialog window for designing the dimensions of a foundation. The dialog window allows inputting the bearing capacity of foundation soil R_d or selecting the option "**calculate**". In this case the program determines all dimensions of a foundation based on **input parameters** (soils, profile, water impact, send-gravel-cushion, setting, etc.).

While leaving the dialog window by pressing the "**OK**" button, the calculated dimensions are transfered into the "**Geometry**" frame.



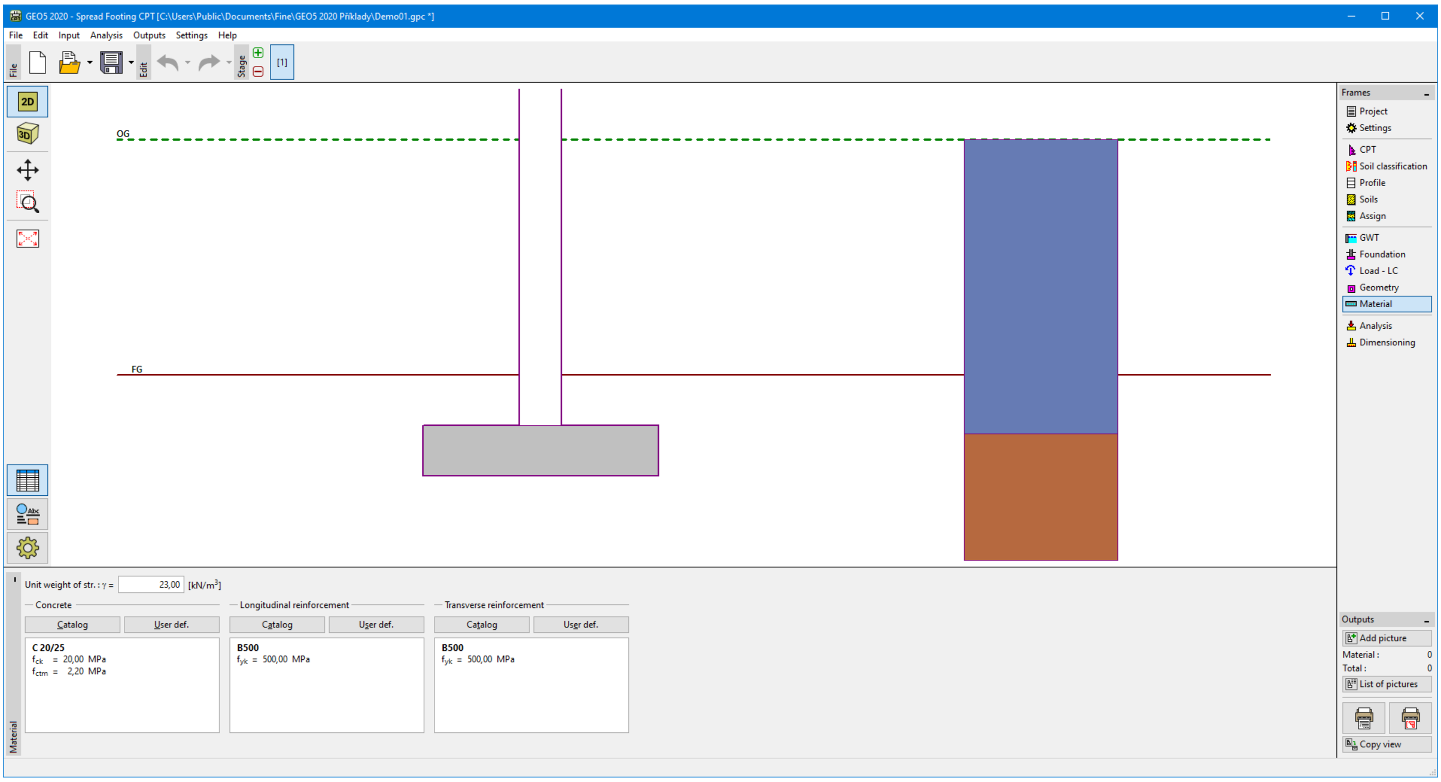
Material

The frame **"Material"** allows us to enter material parameters. The input field in the upper part of the frame allows us to specify the **unit weight of structure**.

Two options can be used for setting the parameters of the material:

- The "**Catalog**" button opens the "**Catalog of materials**" dialog window, where the required material can be selected.
- The "**User-defined**" button opens the "**Editor of material**", which allows us to input the specification of material parameters manually by the user.

The content of catalogs depends on the selection of relevant standard for the dimensioning of concrete structures in the "Materials and standards" tab.



Frame "Material"

Analysis

The "**Analysis**" frame serves to verify the vertical bearing capacity and settlement of a footing. More computations can be performed in the frame. The verification can be performed for individual loads, or the program finds the **most critical one** (can be selected from a combo list).

For both limit states (strength, usability) program assesses the eccentricity of the foundation. The value of the maximum allowable eccentricity of foundation is entered in the left part of frame.

The analysis is performed according to the **selected type of field tests** in the frame "Settings", where analysis type is also chosen in case of using CPT tests.

The analysis can be performed for the field test **with the highest utilisation**, the **average from all tests** or **selected test** from a combo list.

Further it is necessary to enter a **safety factor for bearing capacity**. It is recommended to use a $SF = 3$ for all methods using CPTs, SPTs or PMTs.

The **stress in the footing bottom used for settlement analysis** can be reduced by the geostatic stress given by:

- **original ground**
- **finished grade**
- **not specified**

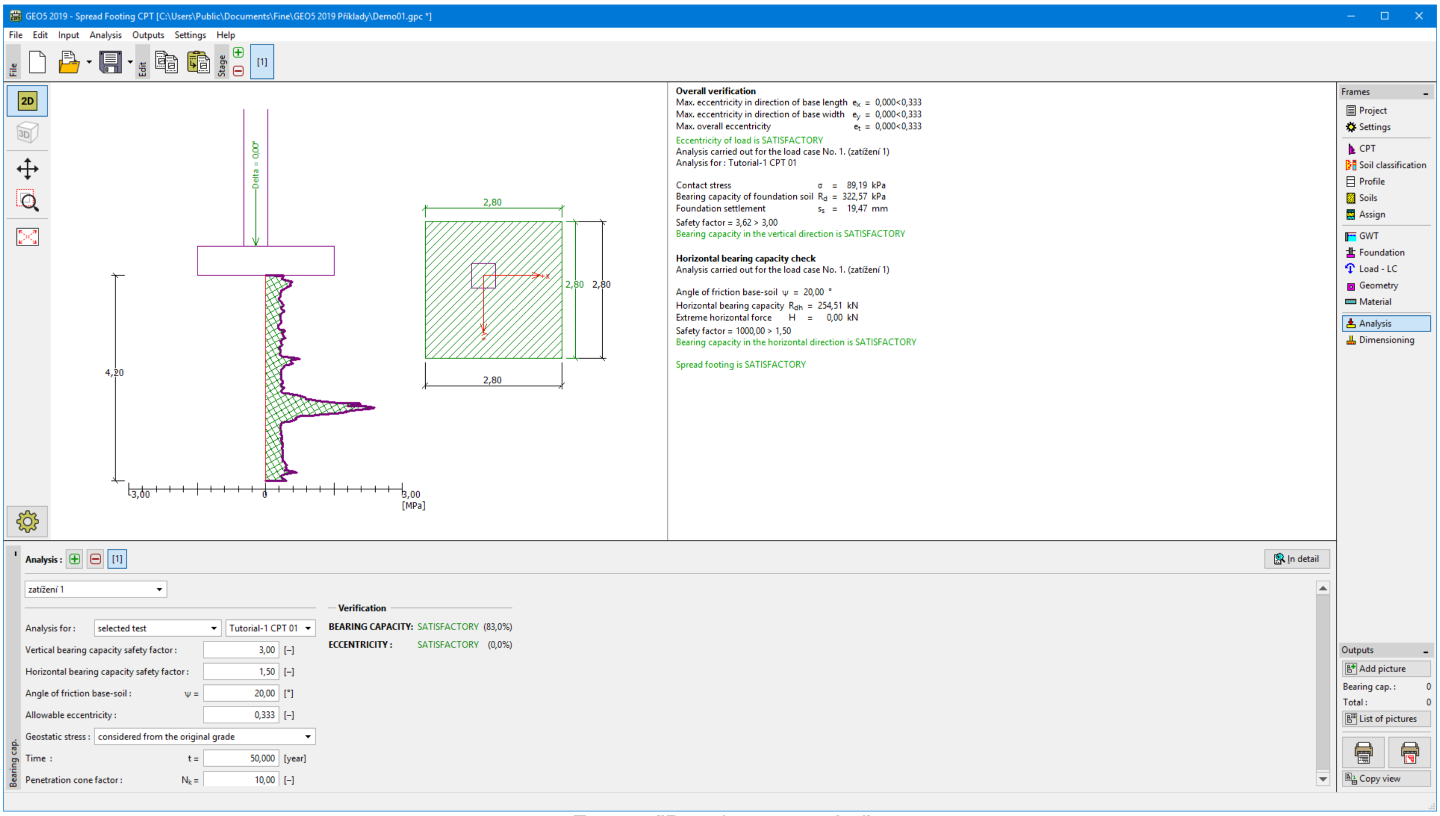
In case of inclined footing bottom or analysis according to the Schmertmann method it is also necessary to enter an angle of internal friction φ in the footing bottom.

When calculating according to the SPT tests (Meyerhof method), the correlation for saturated fine or silty sands can be used.

When calculating horizontal bearing capacity, the **factor of safety** and further data related to the type of analysis selected in the "Settings" frame are entered:

- **angle of friction** between soil and foundation ψ (analysis for drained conditions)
- **cohesion** between soil and foundation a (analysis for undrained conditions)

A detailed listing of the results is displayed in the right part of the desktop. Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Bearing capacity"

Dimensioning

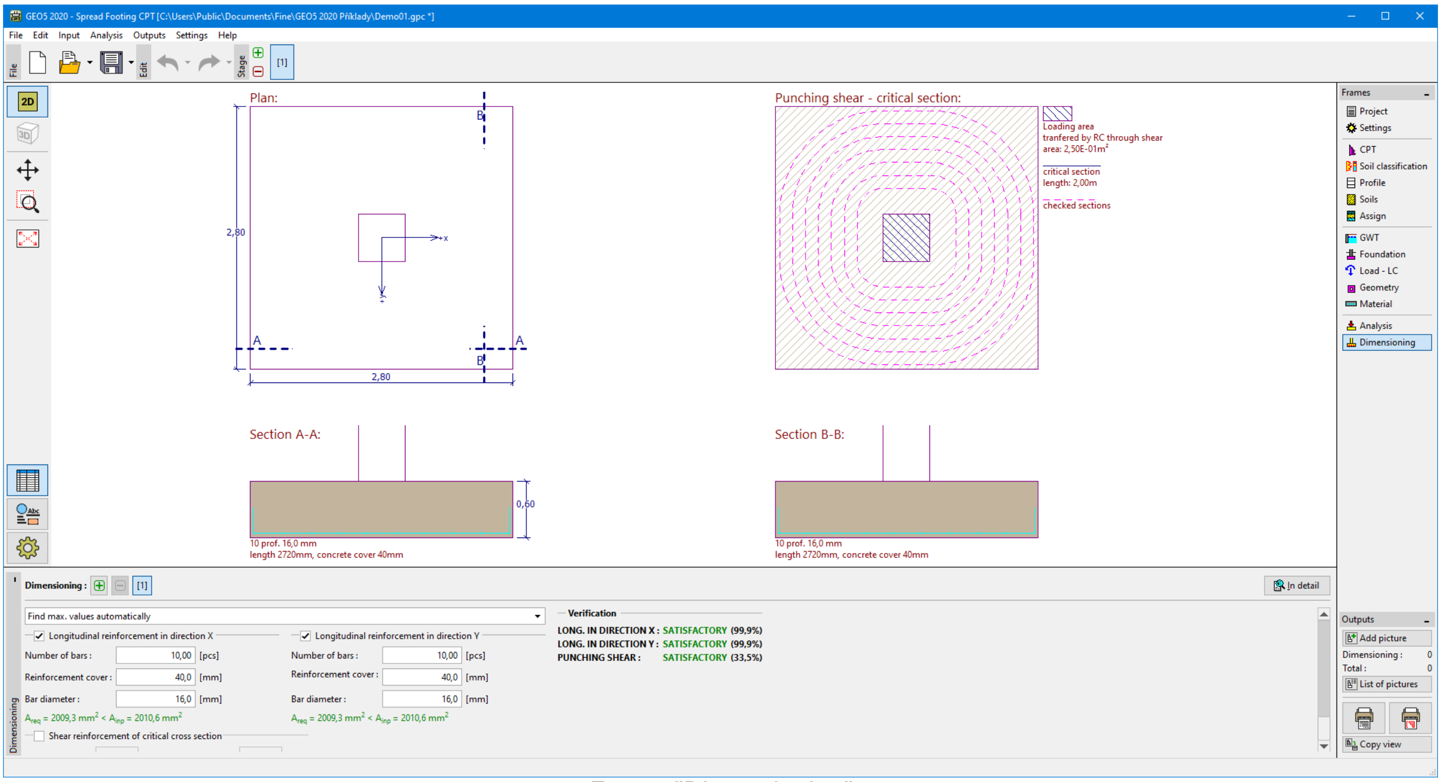
The "**Dimensioning**" frame allows for designing and verifying the longitudinal reinforcement of a foundation and also for verifying the foundation against being pushed through. The verification can be performed for individual loads, or the program finds the **most critical one** (can be selected from a combo list).

The program derives the stress in the construction joint and determines the internal forces in individual cross-sections.

Dimensioning of the reinforced concrete is performed according to the standard set in the "Materials and standards" tab.

The resulting information are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "**In detail**" button opens the dialog window that contains a detailed listing of the dimensioning results.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Dimensioning"

Program Pile

This program is used for the the analysis of the vertical bearing capacity of a single pile loaded both in tension or compression, pile settlement as well as horizontal bearing capacity of a single pile.

The help in the program "Pile" includes the following topics:

- The input of data into individual frames:

Project	Settings	Profile	Modulus of Subsoil Reaction	Soils	Assign	Load
Geometry	Material	Water, Incompressible Subsoil	Negative Skin Friction	Stage Settings	Vertical Bearing Capacity (Analytical Solution)	Vertical Bearing Capacity (Spring Method)
Settlement (Poulos)	Settlement (Masopust)	Settlement (EA-Pfähle)	Horizontal Bearing Capacity	Horizontal Bearing Capacity (Brom's Method)		

- Standards and analysis methods

- Theory for analysis in the program "**Pile**":

Stress in Soil Body	Pile Analysis	Dimensioning of Concrete Structures	Steel Cross Sections Verification	Timber Cross Section Verification

- Outputs

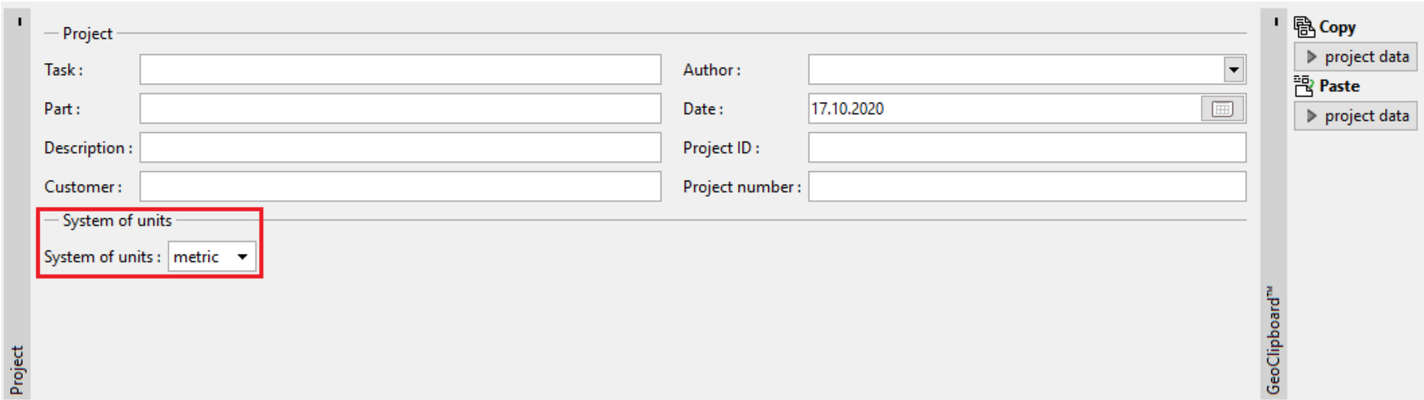
- General information about the work in the User Environment of GEO5 programs

- Common input for all program

Project

The frame **"Project"** is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The frame "**Settings**" allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

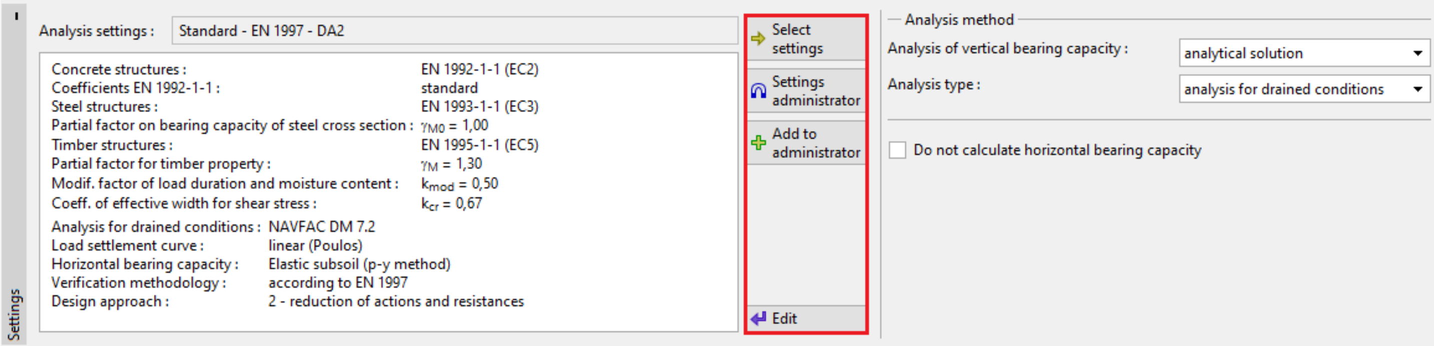
The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

Settings of analysis parameters are performed in the "Materials and standards" and "Piles" tabs.

The pile vertical bearing capacity can be found either by using the analytical solution or the spring method. Analytical solution is defined for:

- **analysis for drained conditions** (CSN 73 1002, Effective stress method, NAVFAC DM 7.2, CTE-DB SE-C)
- **analysis for undrained conditions** (Tomlinson, NAVFAC DM 7.2, CTE-DB SE-C)



Frame "Settings"

Profile

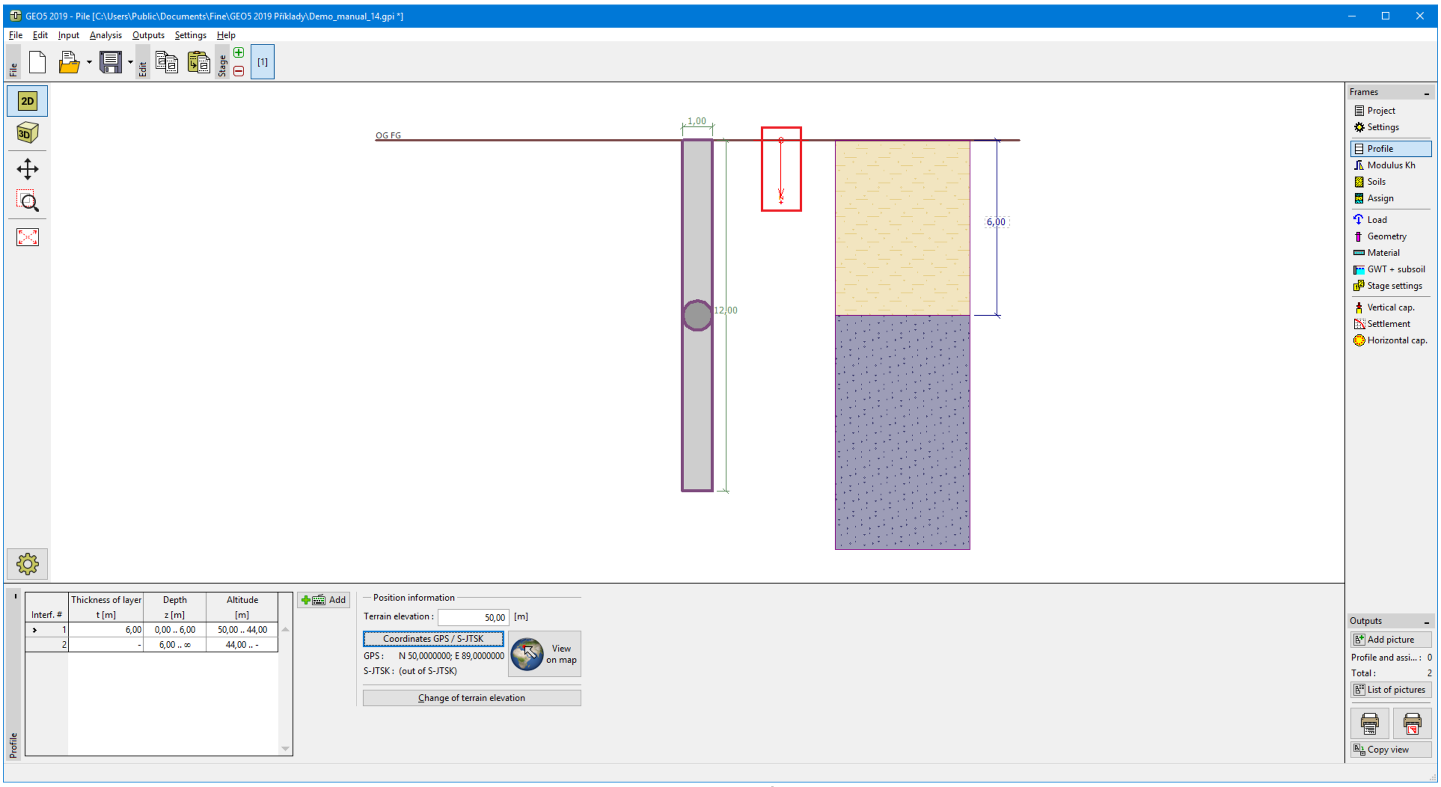
The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.

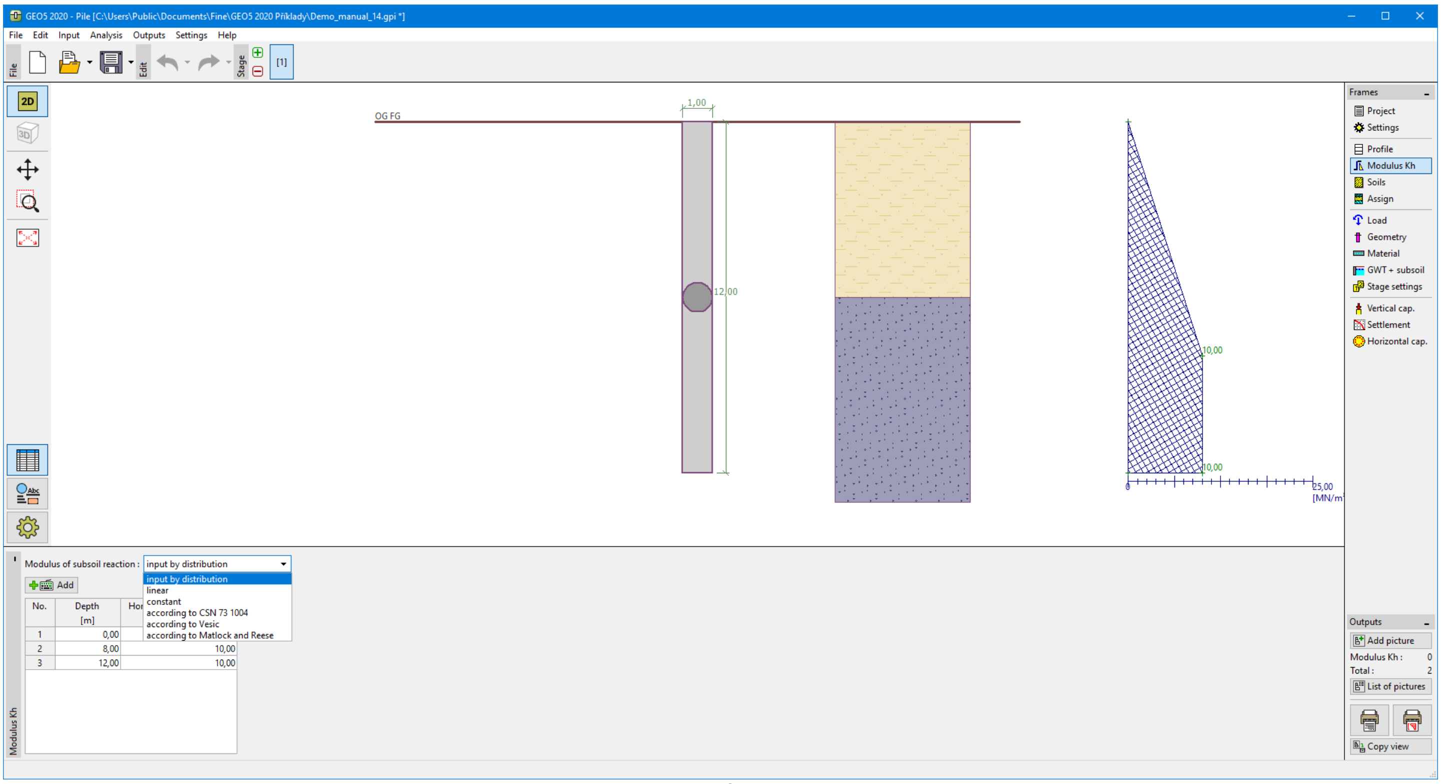


Frame "Profile"

Modulus of Subsoil Reaction

The combo list serves to select one of the **methods for the evaluation of** modulus of subsoil reaction - the required material parameters of soils are input in the frame "Soils" based on the selected method.

Selecting the option "**Input by distribution**" opens a table that allows for specifying the values of the modulus of subsoil reaction along the pile.



Frame "Modulus of subsoil reaction"

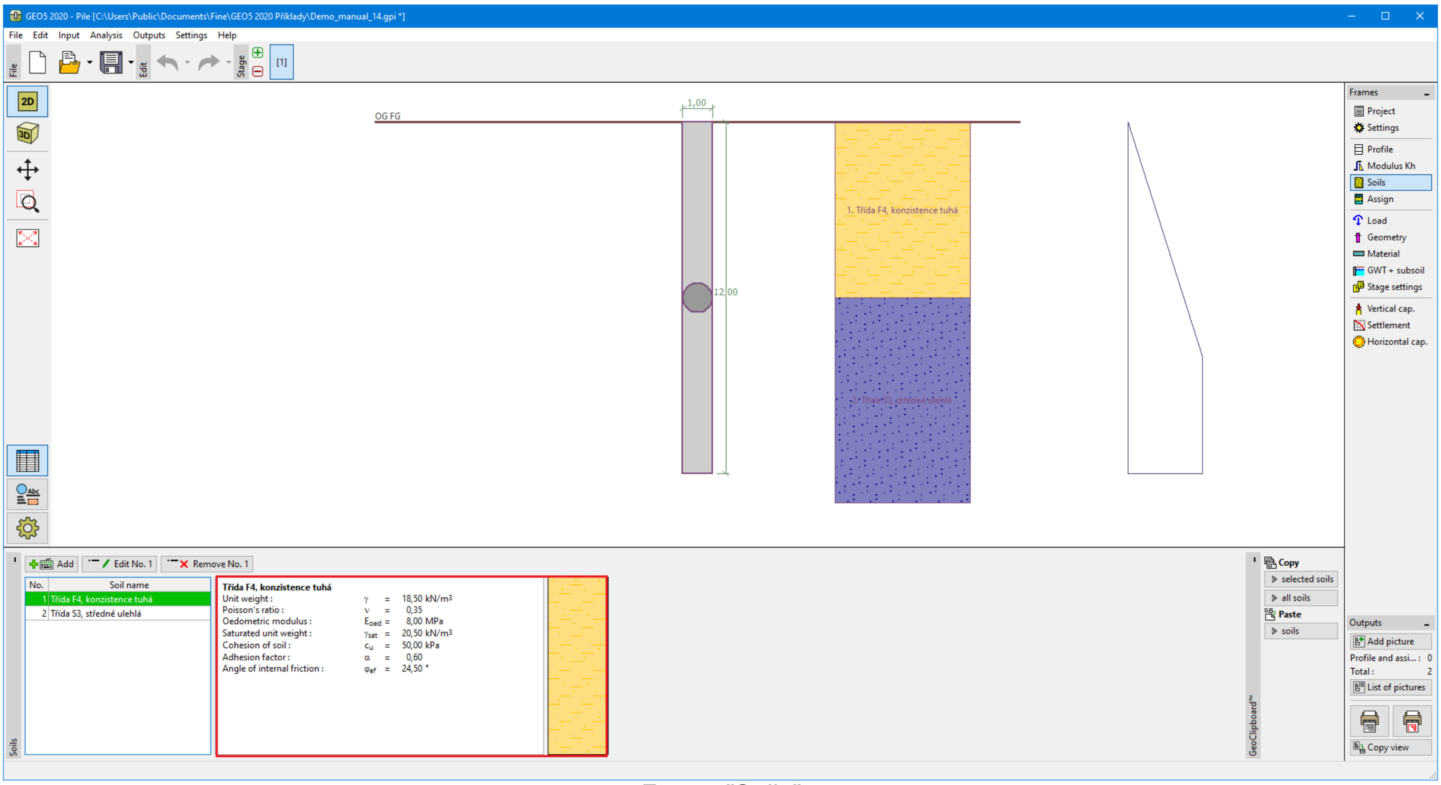
Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Uplift pressure", "Oedometric modulus", "Modulus of subsoil reaction", "Empirical coefficient of adhesion" and "Consistency coefficient". The specified soil parameters depend on the set up of modulus of subsoil reaction and selected theory of analysis specified in the "Piles" tab.

Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

Basic Data

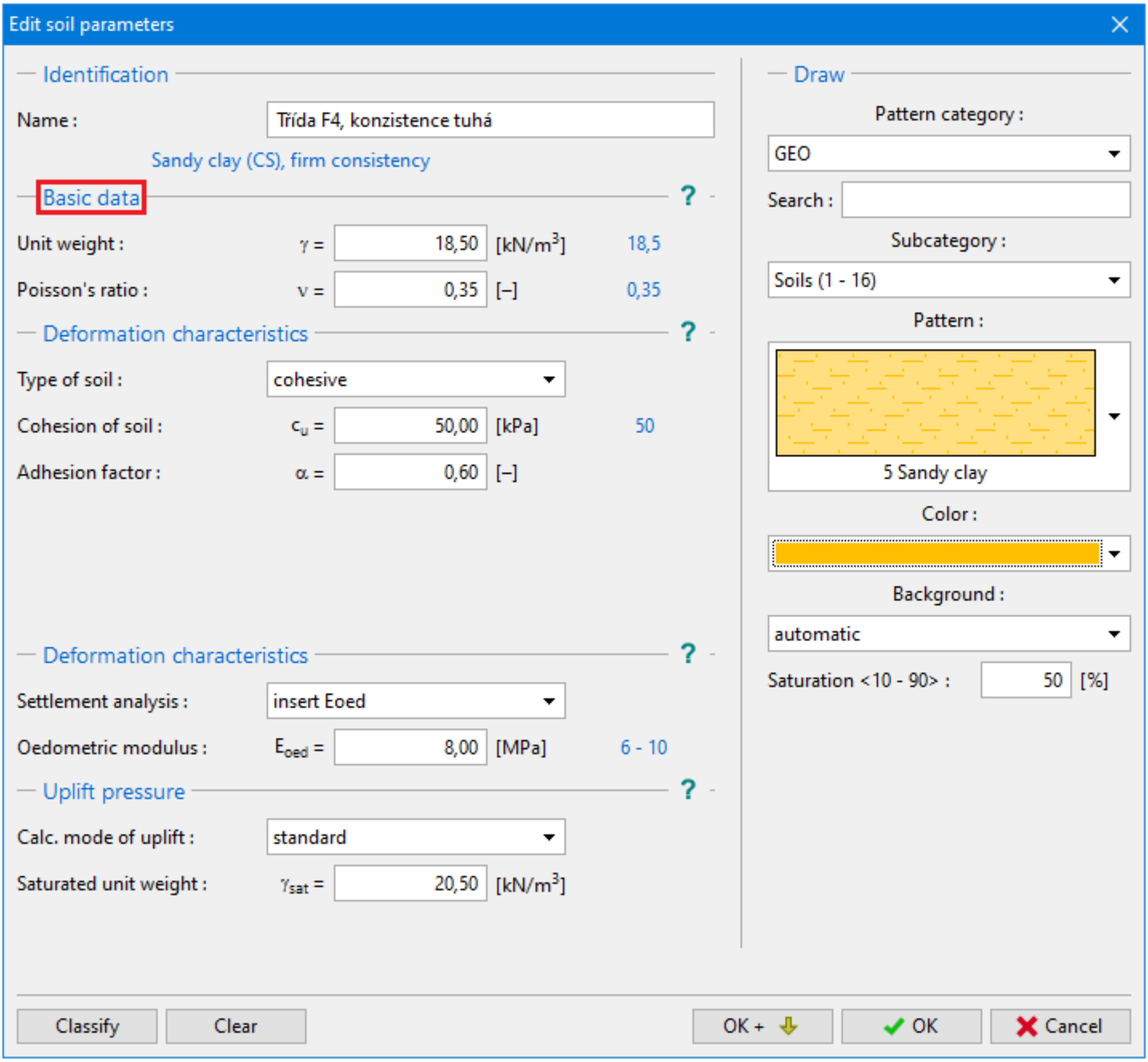
This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction, and cohesion**. The particular values are obtained from a geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit the built-in database of soils, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually.

Soil parameters differ according to the analysis type and analysis method ("Settings" frame, "Piles" tab).

The analysis type differs according:

- **Analysis for drained conditions: effective** parameters of shear strength of soil c_{ef}, φ_{ef} are used commonly ("CSN 73 1002","Effective stress").
- **Analysis for undrained conditions:** in program is defined **total** shear strength of soil c_u ("Tomlinson").
- **Method NAVFAC DM 7.2**: this method combines two types of calculation. For each layer of soil is defined, whether the soil is calculated as drained (cohesionless) or undrained (cohesive).

The associated theory is described in detail in the chapter "Pile analysis".

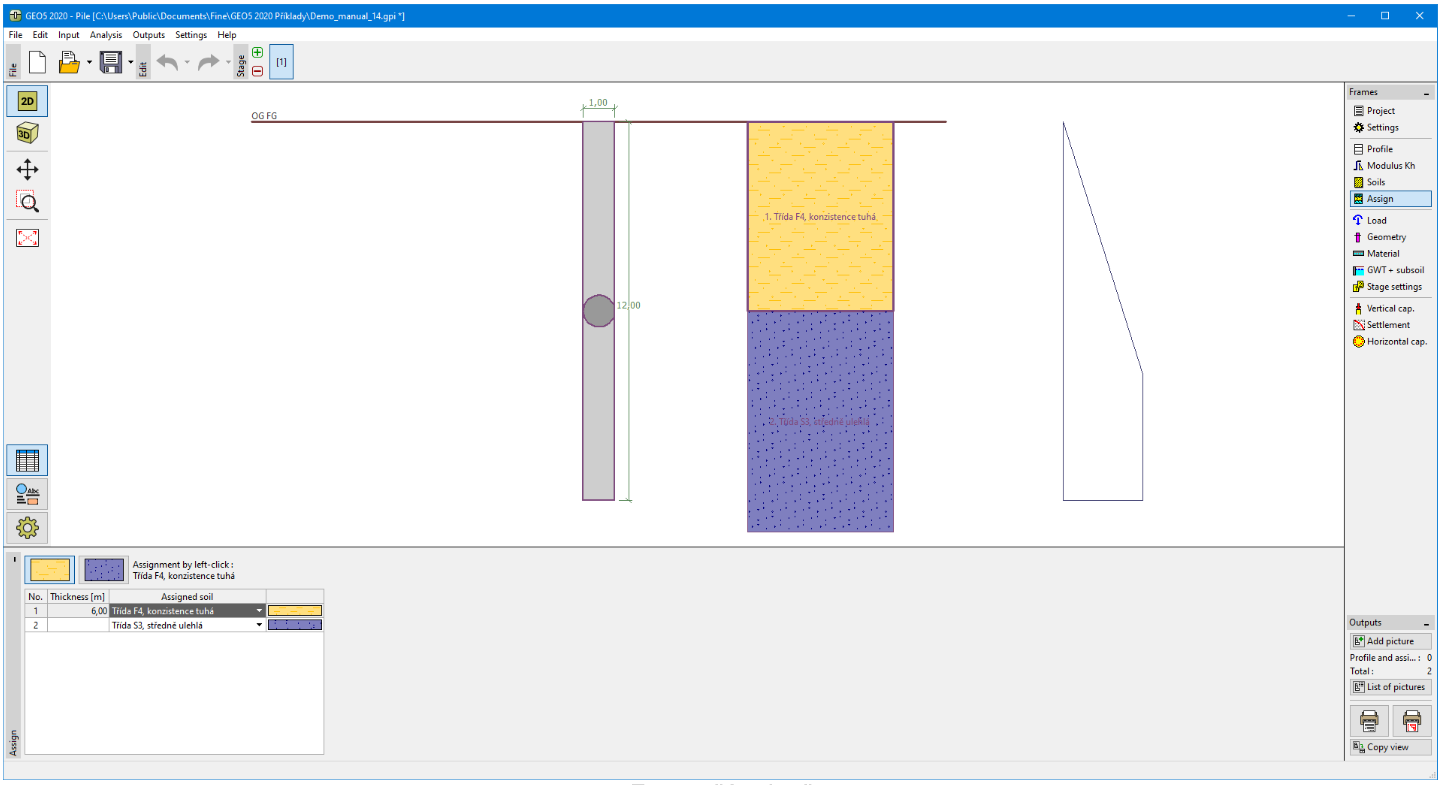


Dialog window "Add new soils" - "Basic data"

Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail herein.



Frame "Assign"

Load

The "**Load**" frame contains a table with a list of input loads. Adding load is performed in the "**New load**" dialog window. The forces are input following the sign convention displayed in the upper part of the dialog window.

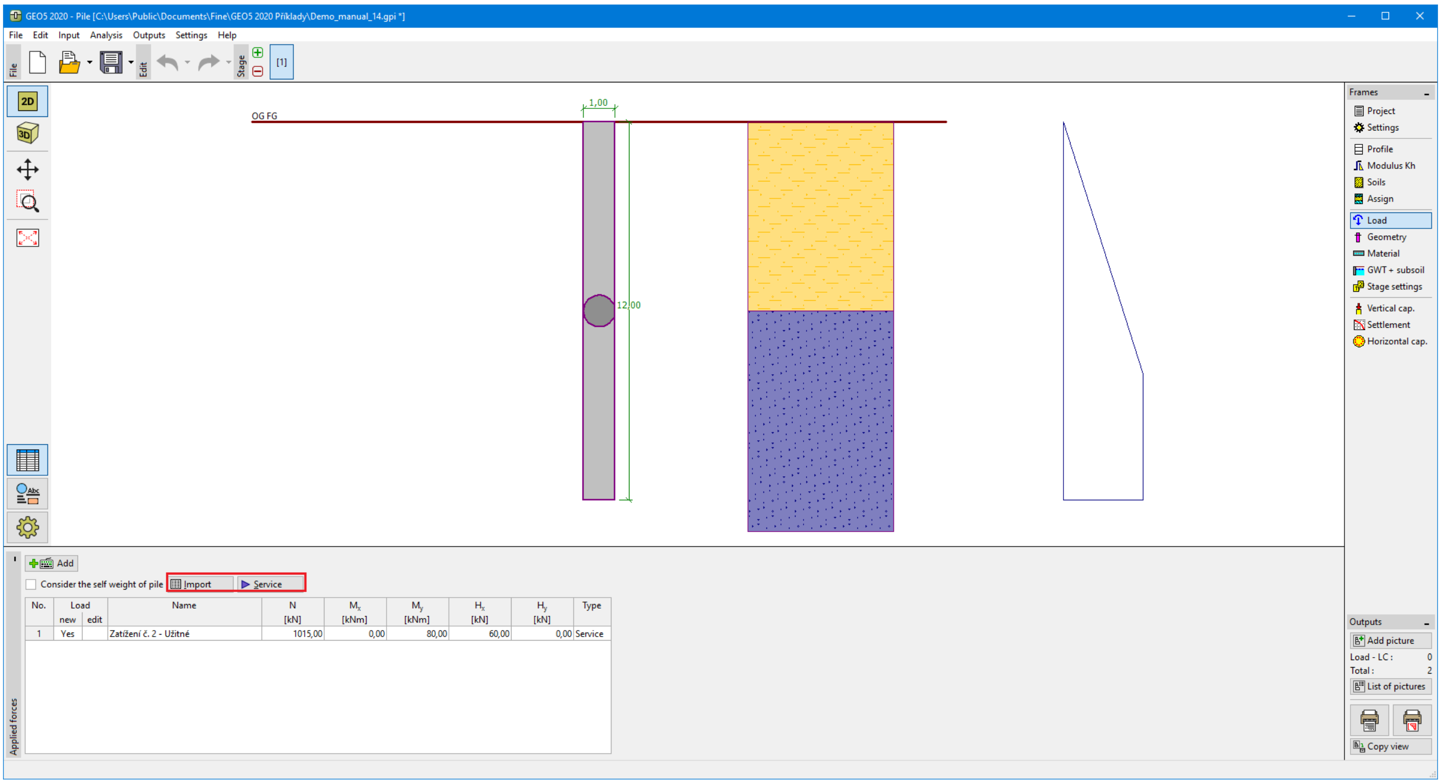
The following types of load can be specified:

- design load serves to verify the vertical and horizontal bearing capacity
- service load serves to compute the settlement of pile (Poulos, Masopust)

When performing the analysis according to **EN 1997**, **LRFD** or **EA-Pfähle** (selected in the "Piles" tab) it is assumed that the design load is determined in accordance with the corresponding standards and individual components of load are **already pre-multiplied** by corresponding partial factors - the program does not modify the input load **any further**.

The "**Service**" button allows for creating the service loads from the already input design loads (analysis according to the factor of safety or the theory of limit states).

The program also allows importing of load using the "**Import**" button.



Frame "Load"

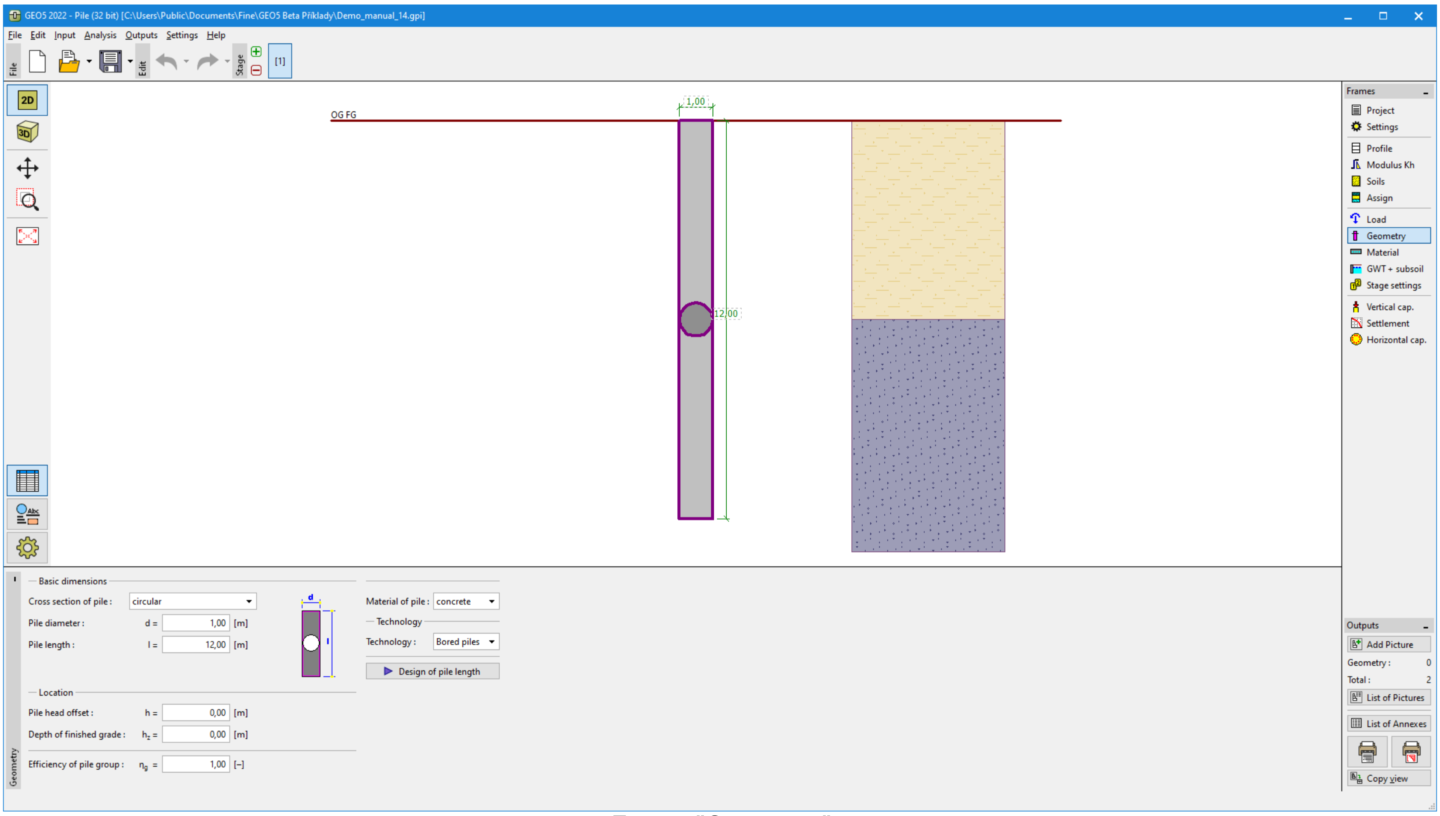
Geometry

The "**Geometry**" frame allows for specifying the **pile cross-section** (circular, circular variable, pipe, square, rectangle, I-section, user-defined...).

For the selected section type, it is necessary to select **material** (concrete, steel, wood) and **technology** (bored, driven, CFA).

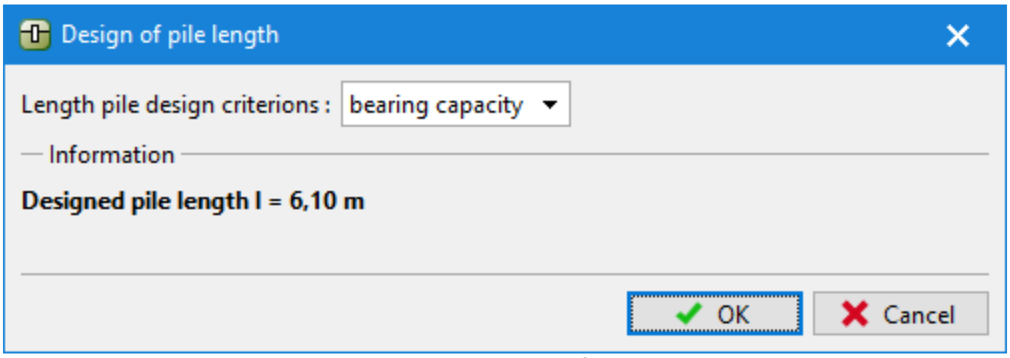
The frame also serves to specify the pile location (pile lift-out and the depth of finished grade). The pile lift out can also be negative - in such a case the pile is placed under the terrain.

The program makes it possible to export the geometry of a structure in the *.DXF format.



Frame "Geometry"

The "**Design of pile length**" button opens the dialog window for the automatic design of the pile length. The pile length can be computed according to **vertical bearing capacity** or **specified maximum settlement**. All **entered parameters** (soils, profile, water impact, pile diameter, settings, etc.) are taken into account in the calculation. The "**OK**" button confirms and transfers the computed pile length to the "**Geometry**" frame.



Dialog window "Design of pile length"

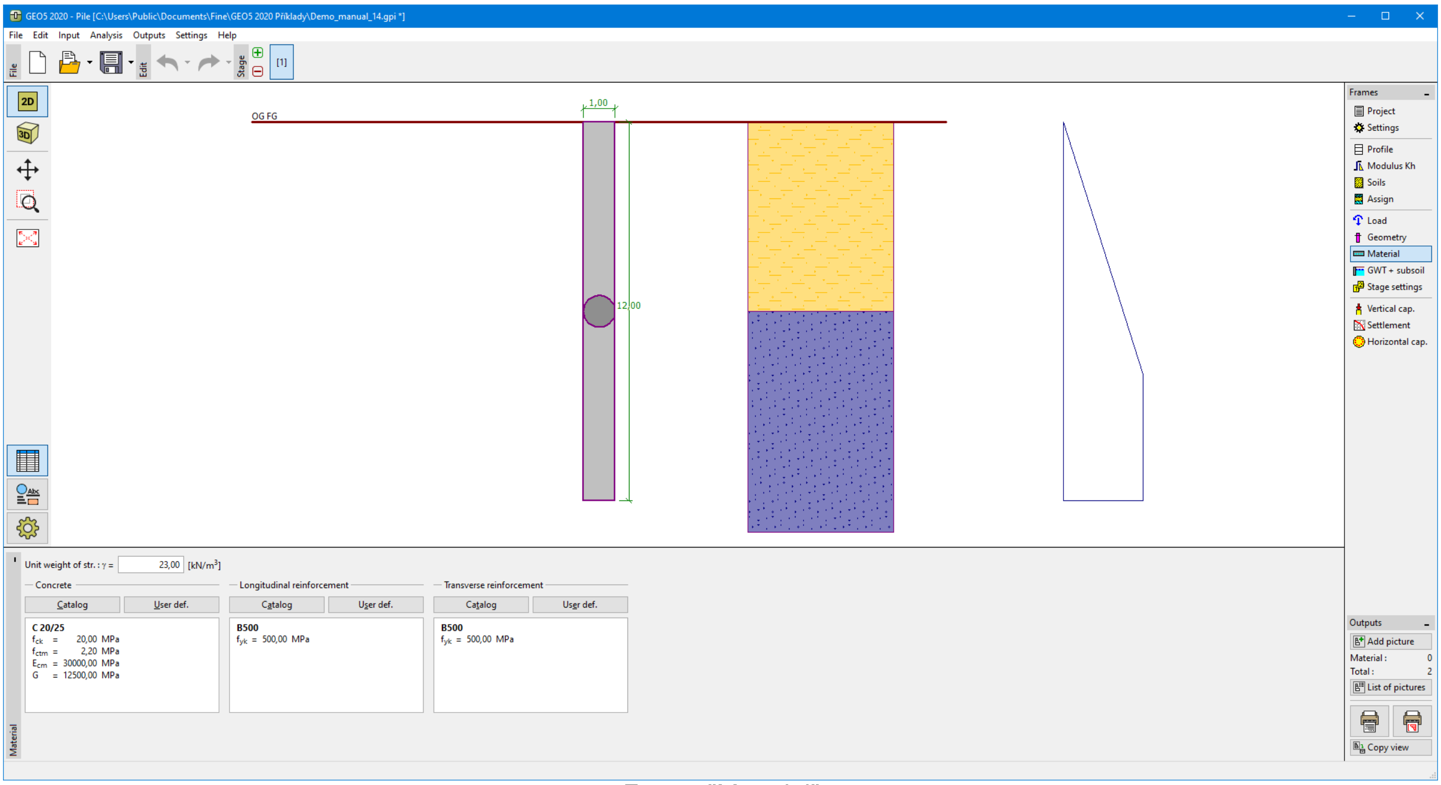
Material

The "**Material**" frame allows for specifying the material parameters. The **unit weight of a structure** is introduced in the input field in the upper part of the frame. The appearance of the frame varies according to the selected material (**concrete, steel, timber**) in the frame "Geometry".

Two options can be used for setting the parameters of the material:

- The "**Catalog**" button opens the "**Catalog of materials**" dialog window, where the required material can be selected.
- The "**User-defined**" button opens the "**Editor of material**", which allows us to input the specification of material parameters manually by the user.

The content of catalogs depends on the selection of relevant standard for the dimensioning of concrete, steel, or timber structures set in the "Materials and standards" tab.



Frame "Material"

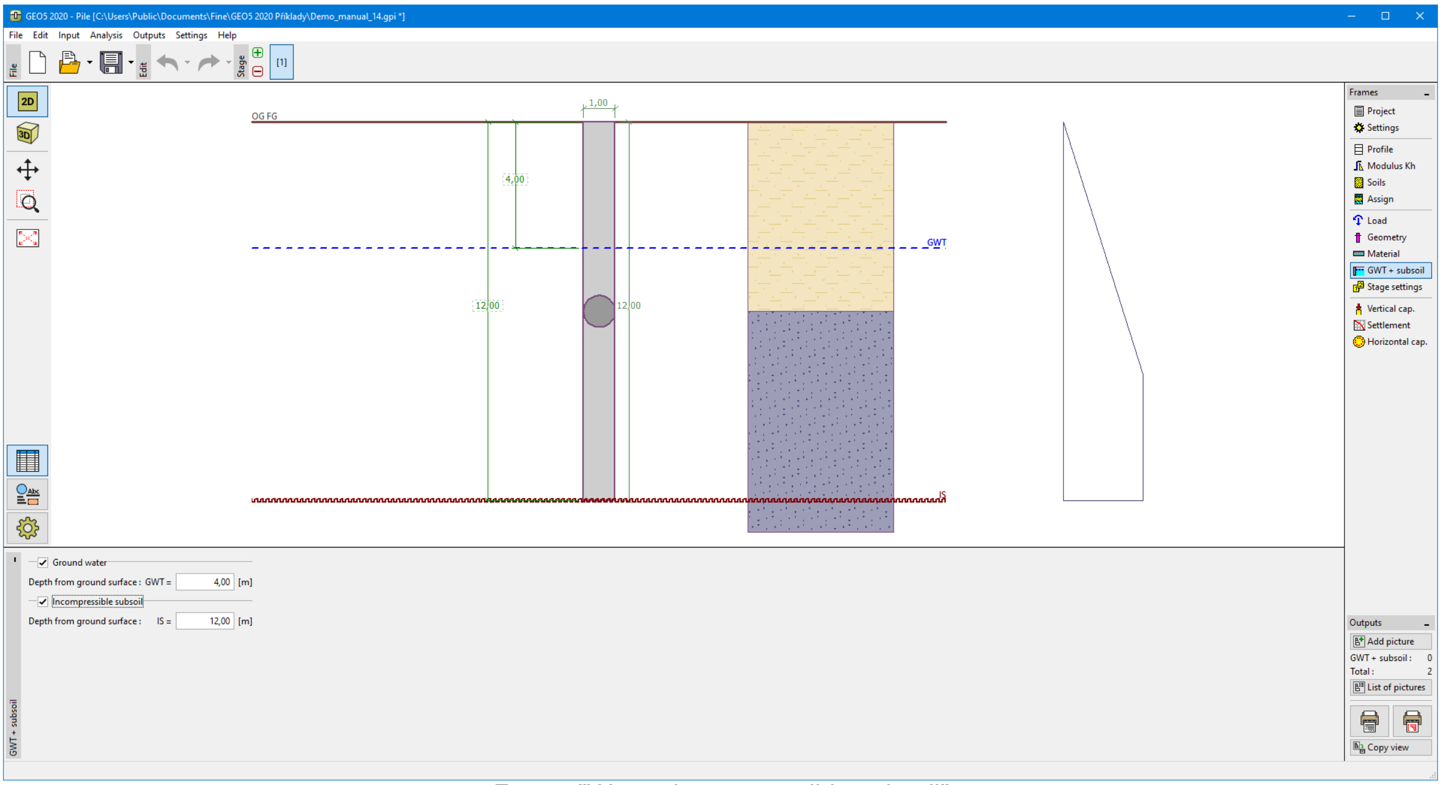
Water, Incompressible Subsoil

The "**Water + IS**" frame serves to specify the **depth of groundwater table** and level of **incompressible subsoil**.

The values can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The **GWT** changes the geostatic stress in the soil profile.

The **incompressible subsoil** cuts the **influence zone** below foundation and also influences reduction factors in settlement calculation.

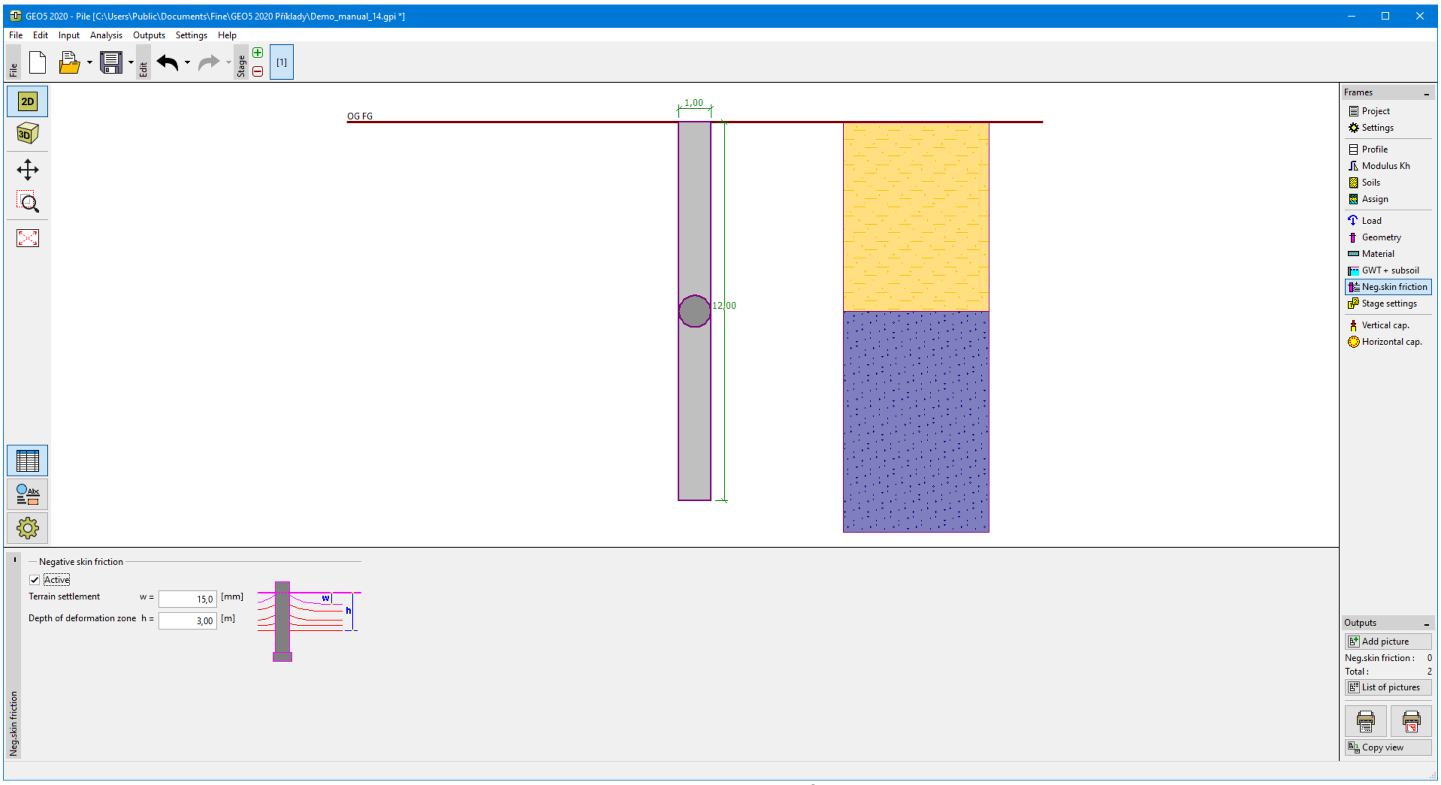


Frame "Water, incompressible subsoil"

Negative Skin Friction

The "**Negative skin friction**" frame serves to specify the settlement of surrounding terrain and the depth of influence zone. For more information on the influence of negative skin friction ,the user is referred to theoretical section.

The setting option in the frame is active only when the **spring method** is selected for the analysis in the frame "Settings".

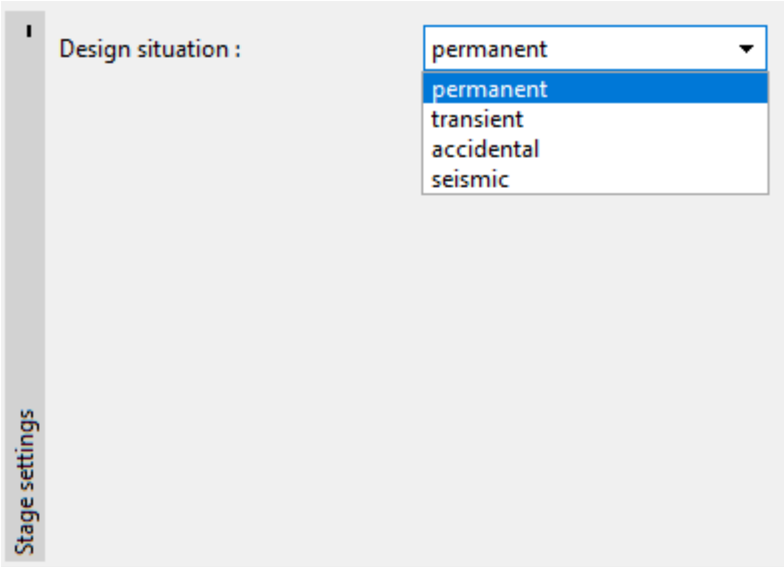


Frame "Negative skin friction"

Stage Settings

The frame "**Stage settings**" serves to input settings valid for a given construction stage.

The selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.



Frame "Stage settings"

Vertical Bearing Capacity - Analytical Solution

The "**Vertical bearing capacity**" frame serves to verify the vertical bearing capacity of a pile. Several analyses can be carried out in the frame. The verification can be performed for individual loads, or the program locates **the most critical one** (can be selected from a combo list). The frame appearance changes depending on the analysis type in the frame "Settings".

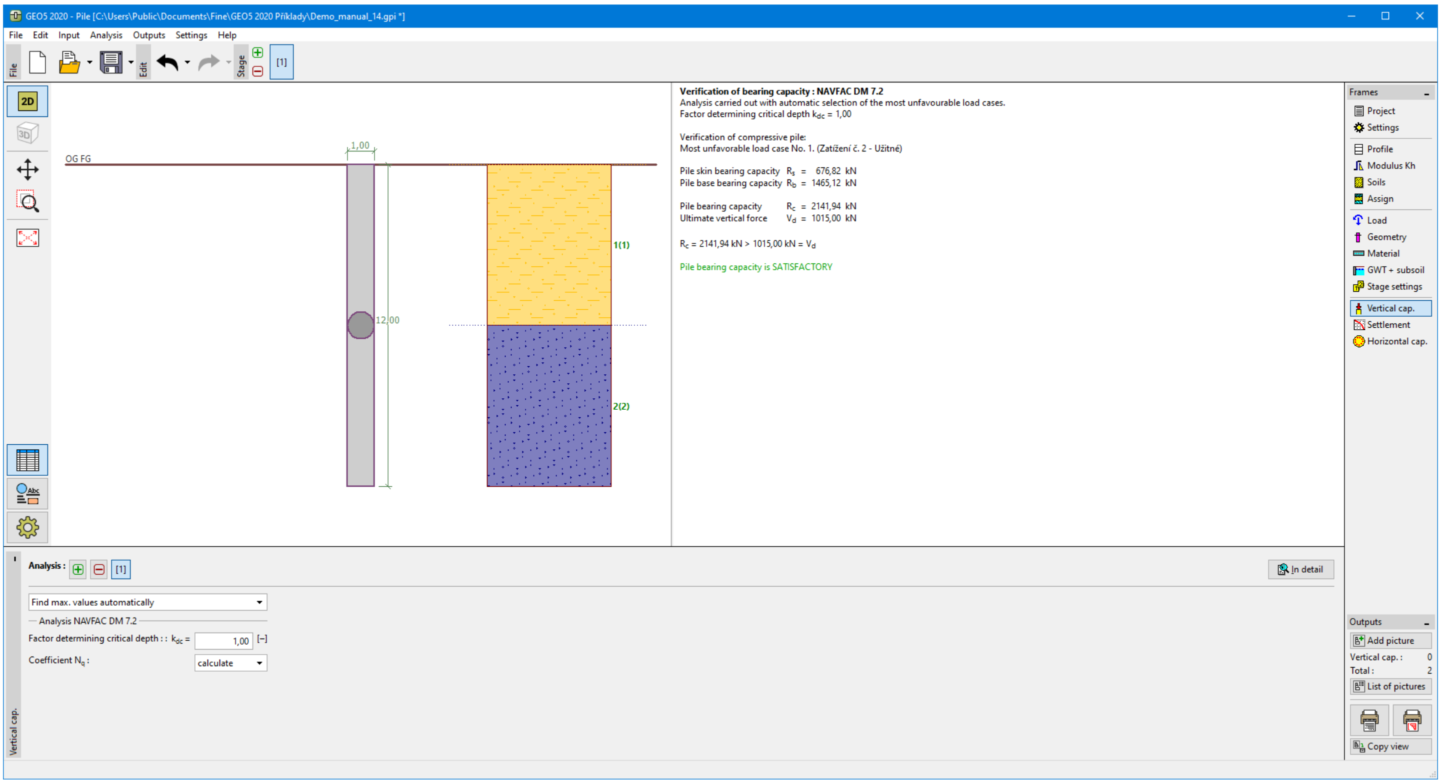
The analysis is performed based on the theory defined in the "Piles" tab. This tab serves to choose the verification methodology (EN 1997-1, factor of safety, limit states).

Calculation of the pile vertical bearing capacity by using the analytical solution is performed for:

- **drained conditions** (CSN 73 1002, Effective stress method, NAVFAC DM 7.2, CTE-DB SE-C)
- **undrained conditions** (Tomlinson, NAVFAC DM 7.2, CTE-DB SE-C)

The "**In detail**" button opens the dialog window containing detailed listing of the verification results.

The analysis results are displayed in the right part of the desktop. Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Vertical bearing capacity" - analysis based on classic theory

Vertical Bearing Capacity - Spring Method

The "**Vertical bearing capacity**" frame allows for verifying the pile **vertical bearing capacity**. The analysis is performed automatically when switching to this frame. More computations can be performed in the frame. The verification can be performed for individual loads, or the program finds the **most critical one** (can be selected from a combo list).

The analysis is performed with the help of spring method. The results are automatically updated whenever one of the analysis parameters "**Maximal deformation**", "Coefficient increasing limit skin friction due to technology" or "**Procedure determining influence zone**" is changed.

Two options are available to determine influence zone:

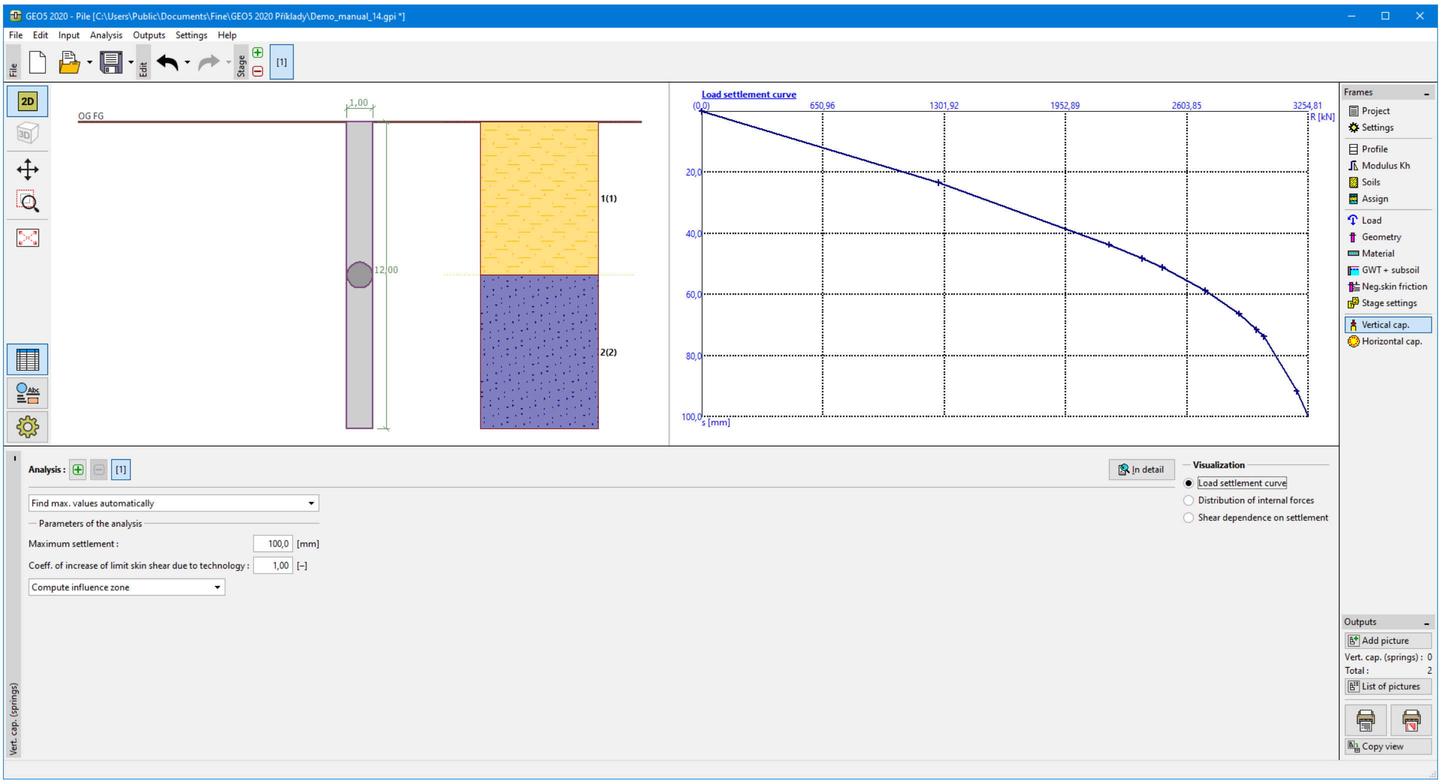
- By default, the evaluation of the depth of influence zone below the pile base follows the procedure described in the theoretical part of the help in section "Depth of influence zone". The depth of influence zone on the pile skin is determined as a k-multiple of the pile diameter. The value of k increases from 1 for zero load to the value of 2.5 when exceeding the limit skin friction.
- The second option assumes the depth of influence zone below the foot and on the skin to be set conservatively a k^{th} multiple of the pile diameter, where the value of k can be selected. During a gradual increase of pile surcharge the value of k for the depth of influence zone on the pile skin is continuously changed from 1 at the onset of load to the specified value when exceeding the limit skin friction. The value of k for the influence zone below the pile base remains constant during the analysis.

The second method, originally used in the old version GEO4, with the value of $k =$ 2.5 offers less accurate results and usually underestimates the pile bearing capacity. Therefore a new option that allows for specifying the depth of influence zone through analysis is offered and is also recommended by the default setting.

Switching between results is available in the left part of the frame (load-settlement curve, distributions of internal forces, dependence of shear on displacement). The shear-displacement relationship is derived for a given depth measured from the pile head. The results are updated whenever the depth is changed.

The "**In detail**" button opens the dialog window, which contains a detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Vertical bearing capacity" - analysis using spring method

Settlement - Linear Load-Settlement Curve (Poulos)

The "**Settlement**" frame serves to display the **pile load-settlement curve**. More computations can be performed in the frame.

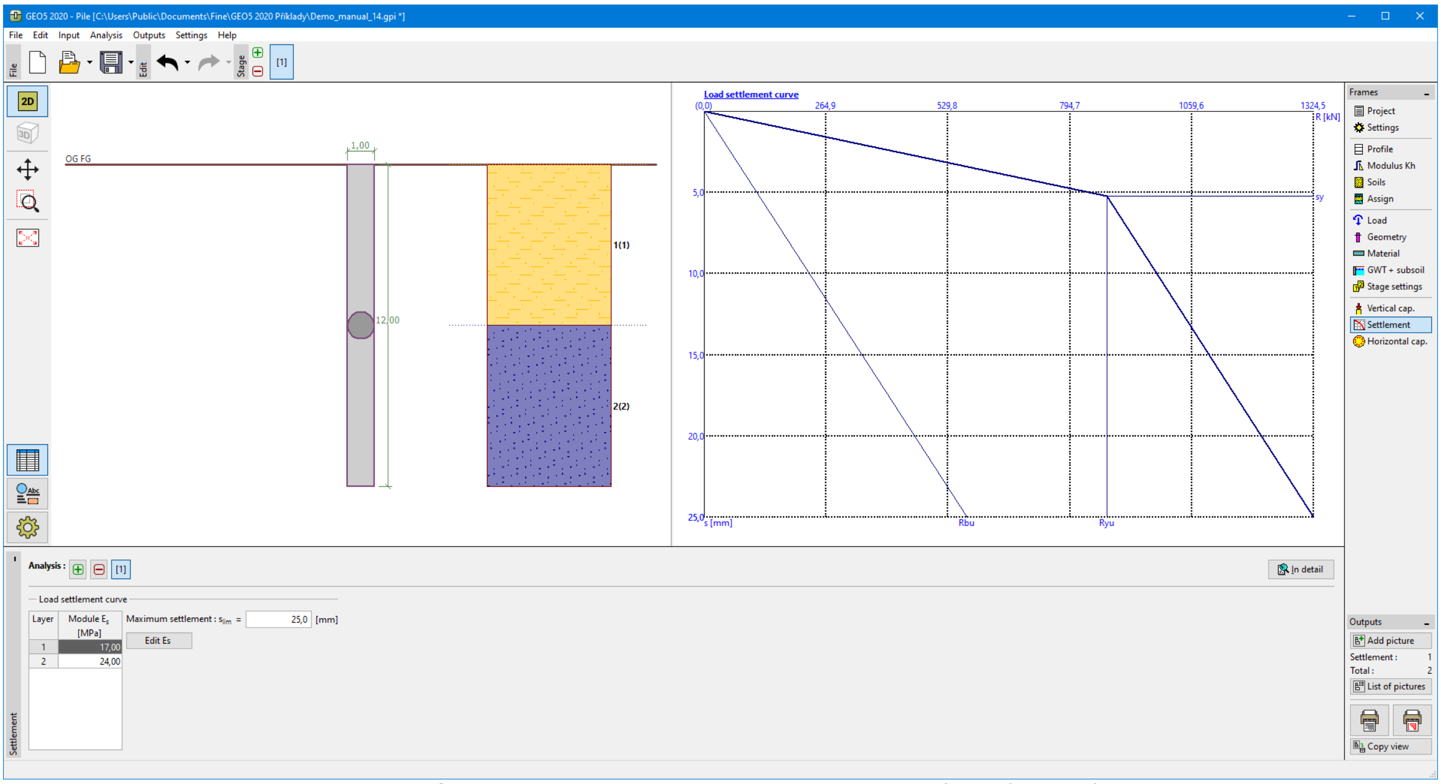
Next, it is necessary to enter the value of limit settlement. The program **constructs the load-settlement curve** of pile always such that this **limit settlement should be reached**.

The analysis is carried out according to the selected theory of settlement analysis (linear). The analysis theory is selected in the "Piles" tab. The table in the left bottom part of the frame directly allows us to specify values of the secant modulus of soil for the relevant layers of soil.

The **analysis results** are displayed in the right part of the desktop. The "**In detail**" button opens the dialog window containing detailed listing of the verification results.

The analysis results (load-settlement curve of pile) are displayed in the right part of the desktop.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Settlement" - linear load-settlement curve of pile (Poulos)

Settlement - Non-Linear Load-Settlement Curve (Masopust)

The "**Settlement**" frame serves to display the **pile load-settlement curve**. More analyses can be performed in the frame.

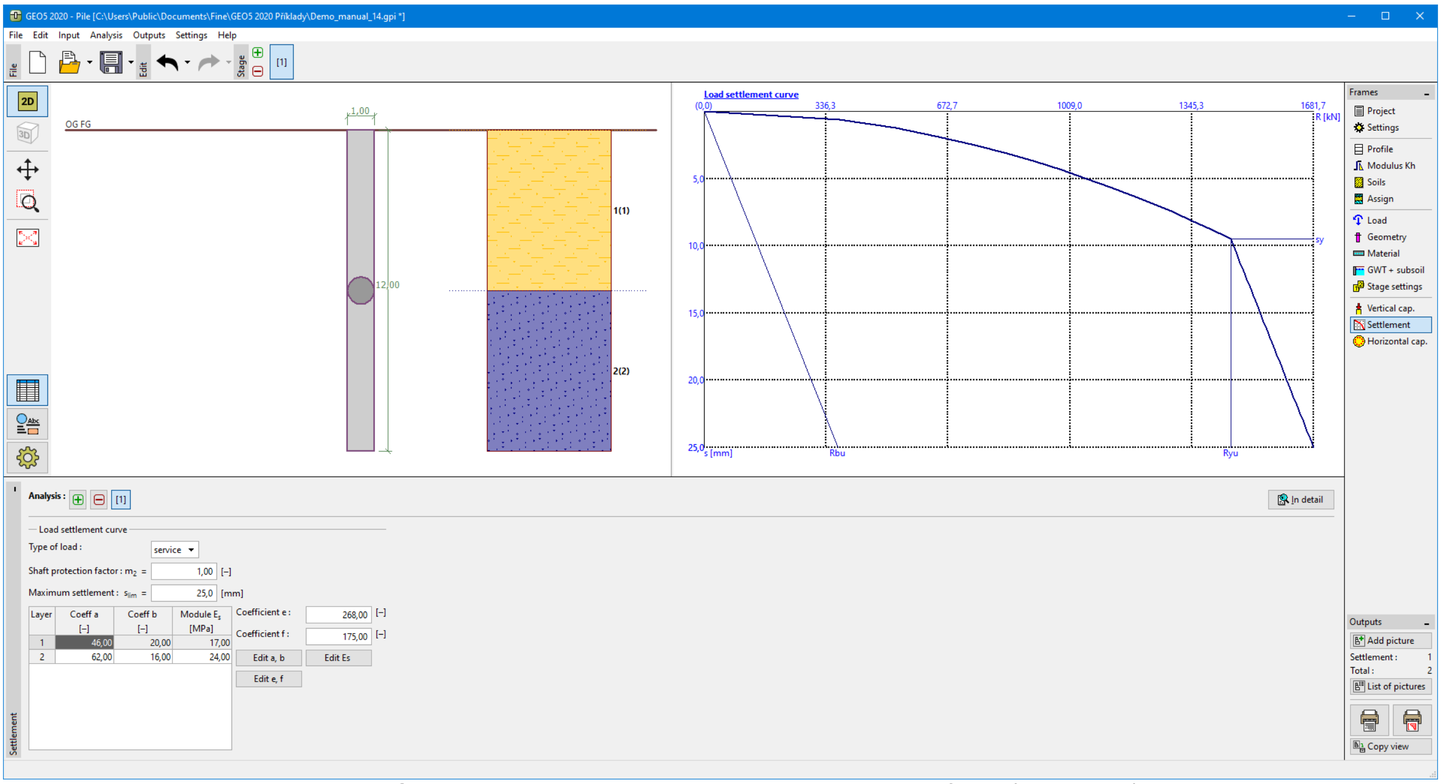
The combo list serves to choose the type of load (**design, service**). Next, it is possible to enter the coefficient of influence of pile shaft. The analysis of load-settlement of pile is always performed up to the **limit settlement** of 25 mm .

The analysis is performed according to the selected theory of settlement analysis (nonlinear). The analysis theory is selected in the "Piles" tab. The table in the bottom part of the frame directly allows with the help of the mouse for editing the defined parameters. Buttons "**Edit** a **,** b ", "**Edit** e **,** f " and "**Edit** E_s " open dialog windows with a help section for entered parameters of regression coefficients and secant modulus of soil. Pressing the "**OK**" button in a particular window **stores the input parameters** for a given layer into the table.

The **analysis results** are displayed in the right part of the desktop. The "**In detail**" button opens the dialog window containing detailed listing of the verification results.

The analysis results (load-settlement curve of pile) are displayed in the right part of the desktop.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Settlement" - nonlinear load-settlement curve of pile (Masopust)

Settlement - EA-Pfähle

The "**Settlement**" frame serves to display the **pile load-settlement curve**. More computations can be performed in the frame.

The calculation is carried out according to EA-Pfähle method. The method is selected in the "Settings" frame.

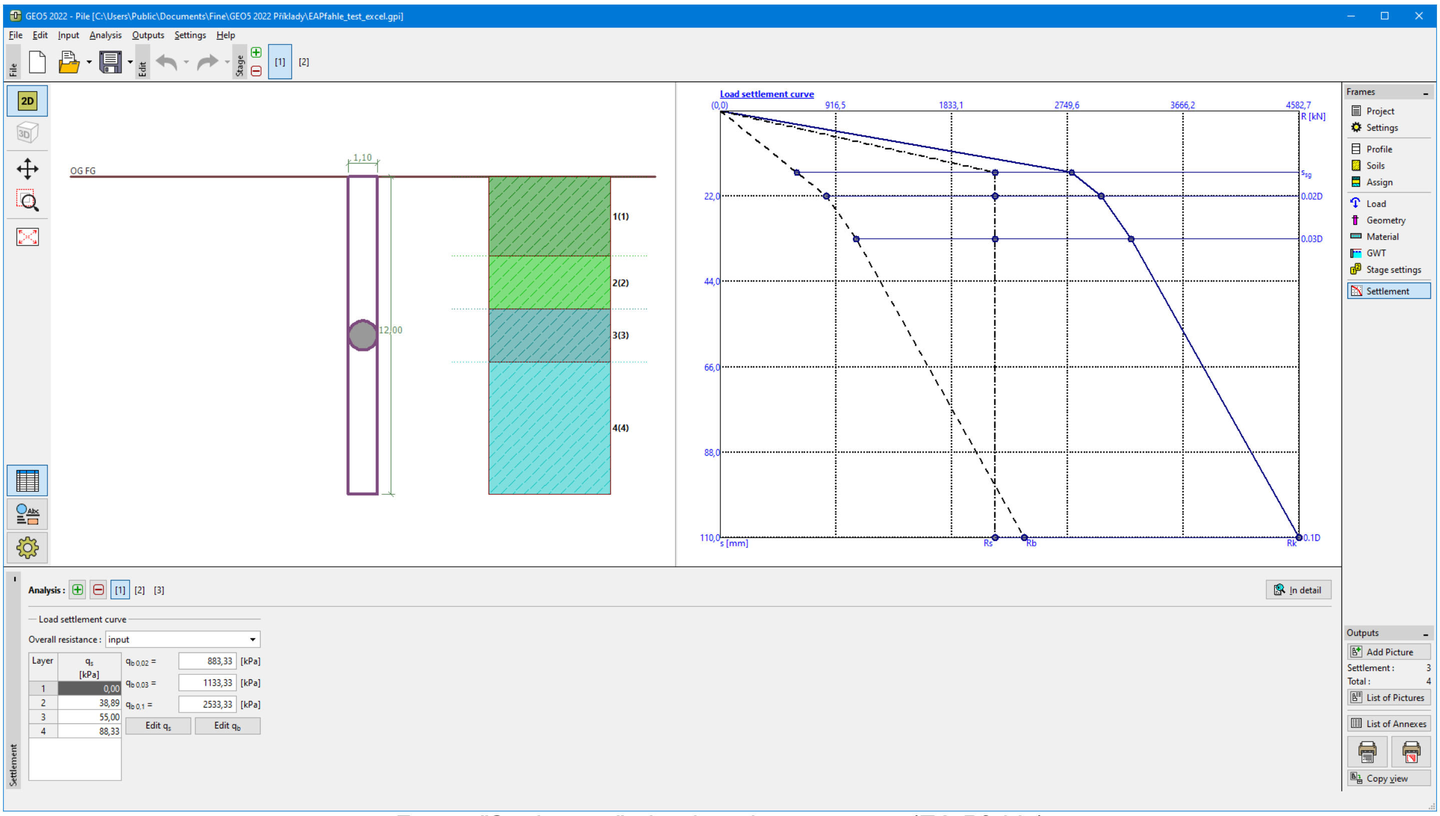
The combo list serves to choose the type of determination of the **pile shaft and base resistance**:

- **input**
- **lower values** (program automatically determines the lower values for q_s and q_b)
- **upper values** (program automatically determines the upper values for q_s and q_b)

The table in the left bottom part of the frame directly allows us to specify values of the shaft resistance for the relevant layers of soil. Buttons "**Edit** q_s " and "**Edit** q_b " open dialog windows with a help section for entered parameters of shaft resistance q_s and base resistance q_b . Pressing the "**OK**" button in a particular window **stores the input parameters** for a given layer into the table.

The **analysis results** are displayed in the right part of the desktop. The "**In detail**" button opens the dialog window containing detailed listing of the verification results.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Settlement" - load-settlement curve (EA-Pfähle)

Horizontal Bearing Capacity - Elastic Subsoil (p-y Method)

The program calculates internal forces on pile and verifies the bearing capacity of cross-section (concrete, steel, wooden).

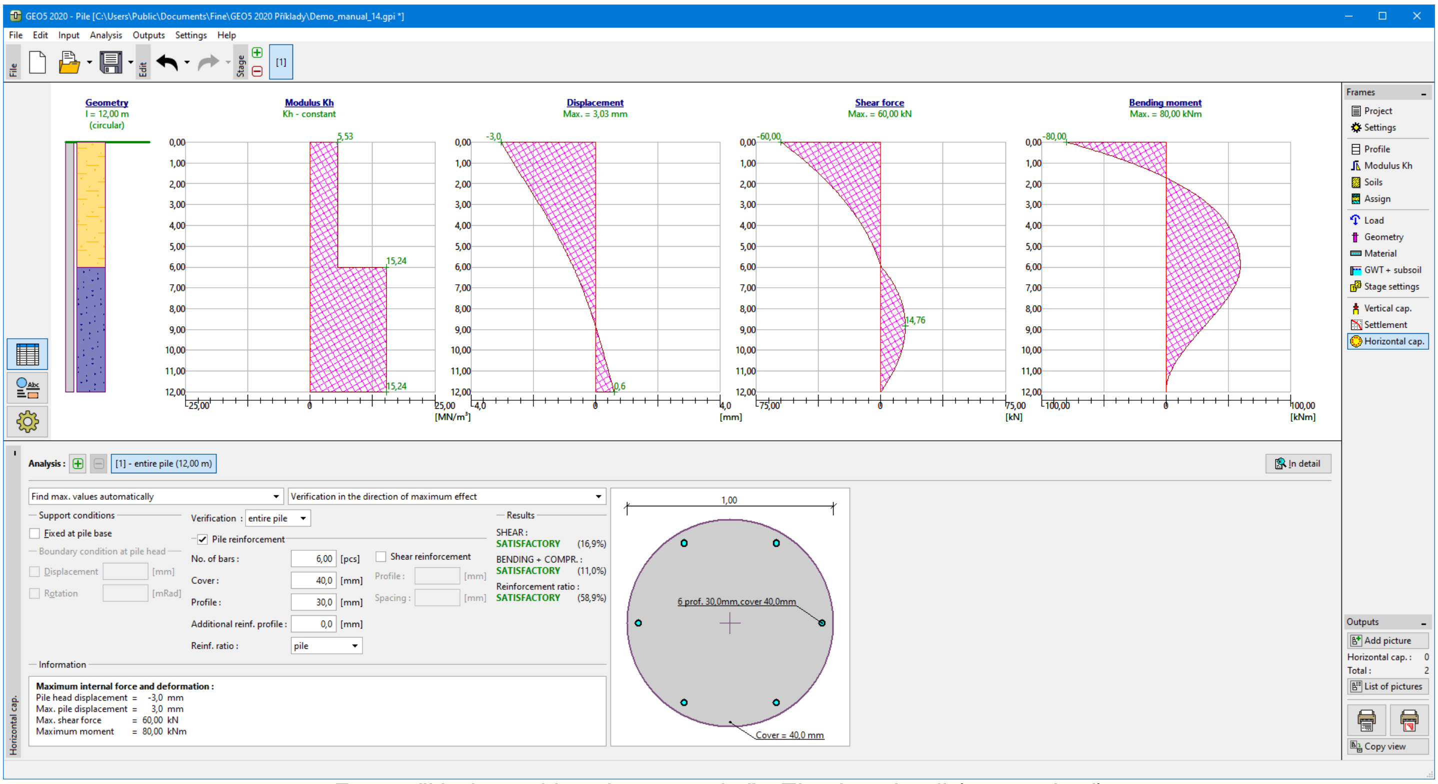
The internal forces can be carried out for:

- individual loads
- prescribed displacement
- the most critical load

Assuming the prescribed displacement type of load requires the introduction of **boundary conditions in pile head** (translation and rotation).

The **fixed end** type of boundary condition prescribed in the pile heel can be assumed for all types of load.

The combo list serves to specify the direction of pile verification (x , y , results in the most stressed direction).



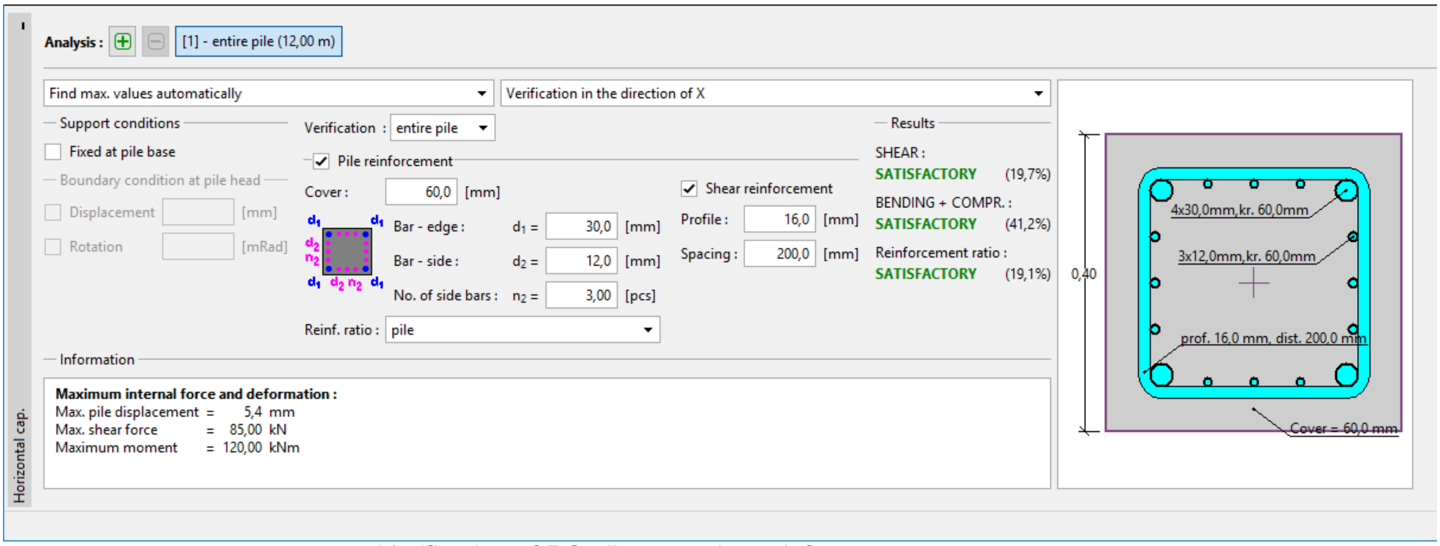
Frame "Horizontal bearing capacity" - Elastic subsoil (p-y method)

According to the selected cross-section and material the program verify a bearing capacity of cross-section. (The cross-section and material are selected in the frame "Geometry".). In some special cases (user-defined, concrete pipe) are displayed only internal forces and deformations - the verification of cross-section can not be performed.

Reinforced Concrete

Program verifies the reinforcement according to standard selected in the frame "Materials and standards" tab. The pile is verified for **shear**, **bending + compression** and **reinforcement ratio**. The reinforcement is entered using checkbox "**Pile reinfrocement**". For a more detailed design of reinforcement, the option of **dimensioning in sections** is available for circular and square piles. Each section is assessed separately.

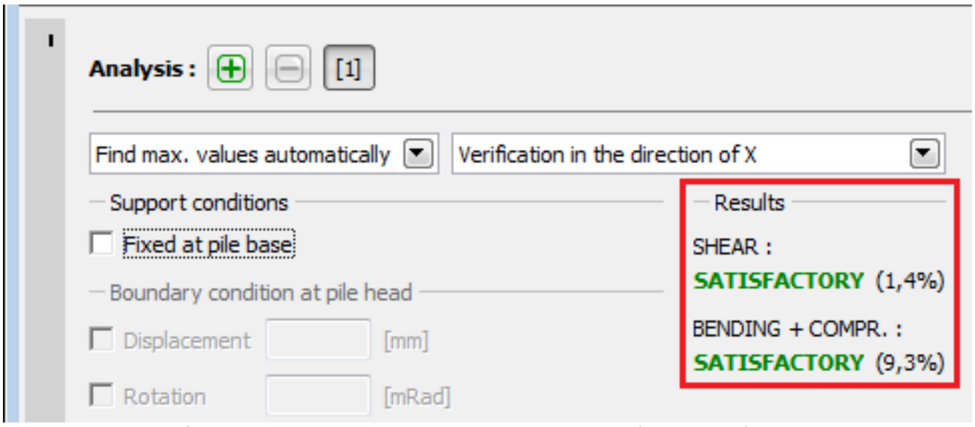
In case of circular pile **profile of reinforcement, number of profiles and concrete cover** are input. Reinforcement of square pile is according to the picture.



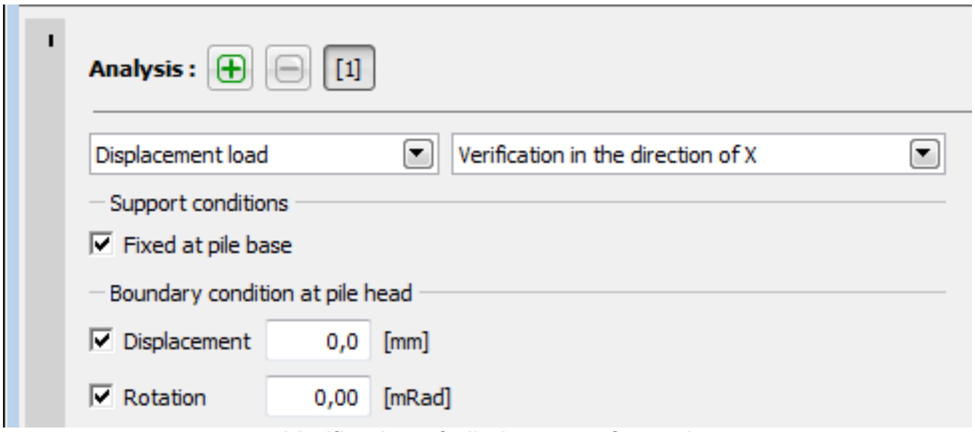
Verification of RC pile - entering reinforcement parameters

Steel, Timber

The pile is verified for **shear** and **bending + compression.**



Shear and bending + compression verification of pile



Verification of pile is not performed

The "**In detail**" button opens the dialog window that contains a detailed listing of the verification results.The analysis results are displayed on the desktop. Visualization of results can be adjusted in the frame "Drawing Settings".

Horizontal Bearing Capacity - Brom's Method

The horizontal bearing capacity of a pile is verified in the "**Horizontal bearing capacity**" frame. Several analyses can be carried out. The verification analysis can be carried out for individual loads, prescribed displacements, or the program finds the **most critical load** (can be selected from a combo list). Assuming the prescribed displacement type of load requires the introduction of **boundary conditions in pile head** (translation and rotation).

The reinforced concrete pile requires inputting the **reinforcement profile**, **number of profiles and reinforcement cover**, which influence the pile bending stiffness.

The input parameters for the analysis of pile horizontal bearing capacity are the **pile material characteristics** (modulus of elasticity and strength of a given material), **pile geometry** (pile length l and its diameter d) and also the **pile load** due to shear force and bending moment.

When adopting the analysis according to the **Broms method** the program ignores up to now input soil layers and checks the single pile horizontal bearing capacity only for the soil defined in the "**Horizontal capacity**" frame. Soil parameters are specified based on the **type of soil**:

- **cohesive** - requires inputting the undrained cohesion of soil c_u, modulus of subsoil reaction k_h, coefficient of cross-section bearing capacity γ_k and the coefficient of bearing capacity reduction γ_{Qu}.
- **cohesionless** - requires inputting the effective angle of internal friction φ, unit weight of soil γ, furthermore the coefficient of subsoil reaction n_h, coefficient of cross-section bearing capacity γ_k and the coefficient of pile bearing capacity reduction γ_{Qu}.

The frame further allows for inputting the **criteria of type of pile**:

- **standard** - in this case, the pile stiffness coefficient $\beta{*}l$, respectively $\eta{*}l$ is calculated automatically by the program.
- **user-defined** - this option allows the user to set the pile stiffness coefficient $\beta{*}l$, respectively $\eta{*}l$ to check short as well as medium piles.

Type of pile can be considered in two ways:

- **free head** - rotation at pile head is not constrained

- **restrained** - pile is constrained against rotation at its head. In such cases we typically deal with piles that are part of a planar pile grid or a pile group.

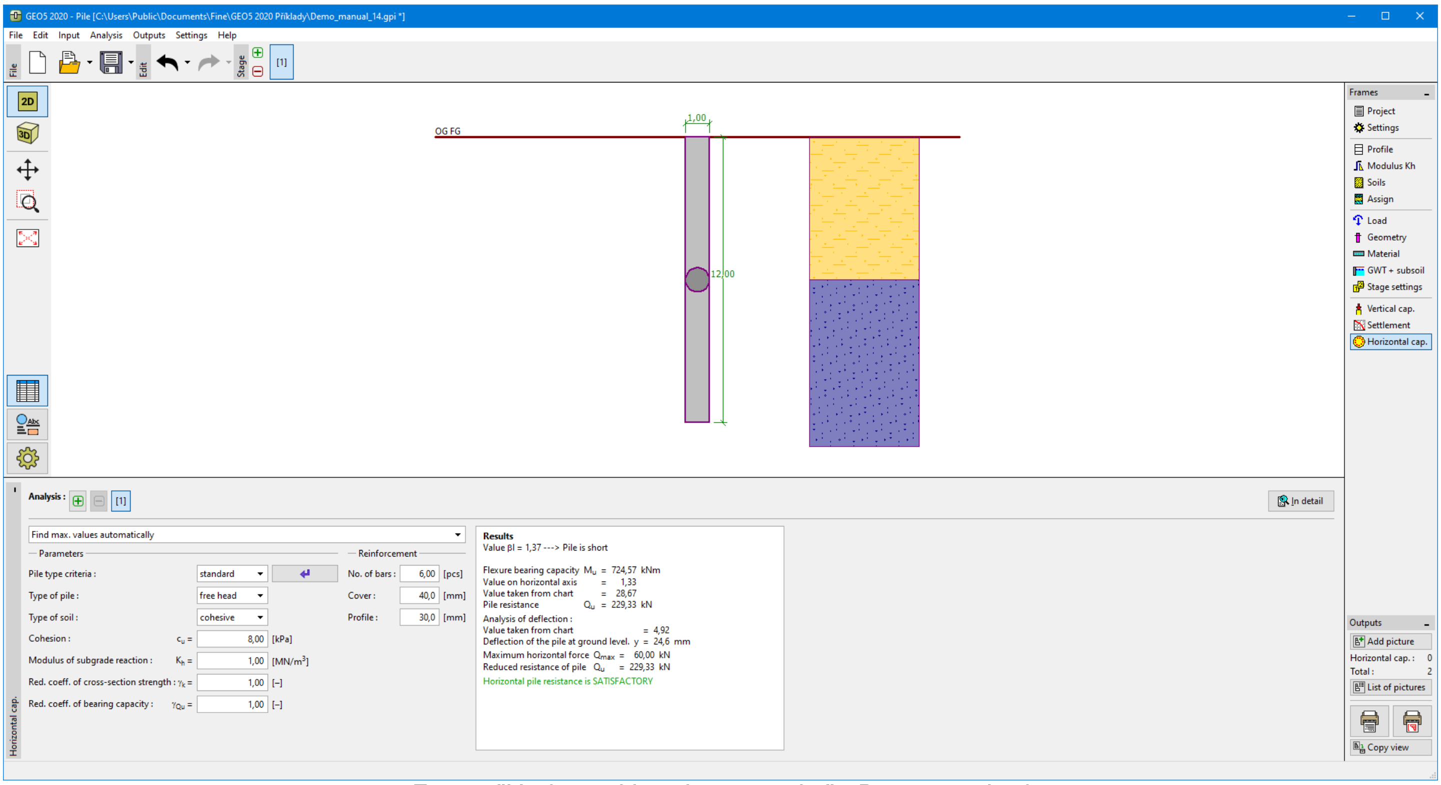
The **reduction coefficient of cross-section strength** γ_k serves to reduce the flexure bearing capacity M_u .

The **reduction coefficient of bearing capacity** γ_{Qu} serves to reduce the overall value horizontal bearing capacity of a single pile Q_u .

The result of an analysis is the horizontal bearing capacity of a single pile Q_u and displacement of a pile at the terrain surface u .

The "**In detail**" button opens the dialog window that contains a detailed listing of the verification results.

The analysis results are displayed on the desktop. Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Horizontal bearing capacity" - Broms method

Program Pile CPT

This program verifies the vertical bearing capacity and settlement of a single pile or a group of piles, based on the results provided by (static) cone penetration tests (CPT).

The help in the program "Pile CPT" includes the following topics:

- The input of data into individual frames:

Project	Settings	CPT	SPT	GWT + NSF	Soil Classification	Profile
Soils	Assign	Construction	Geometry	Bearing Capacity	Settlement	

- Standards and analysis methods

- Theory for analysis in the program "**Pile CPT**":

Stress in Soil Body	Pile CPT

- Outputs

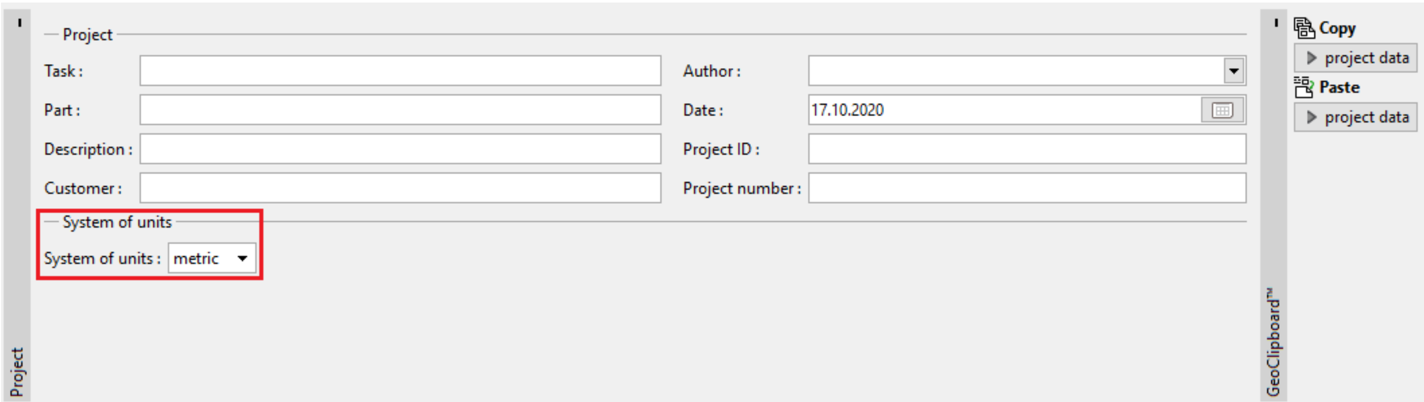
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The frame **"Project"** is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The "**Settings**" frame allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows us to view and modify an individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. If we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

Settings of analysis parameters are performed in the "Piles CPT" tab.

The type of a field test (CPT, SPT) used for the analysis of bearing capacity is selected in this frame.

For the SPT analysis the method of calculation is selected (Décourt-Quaresma, Aoki-Velloso). The other settings in the window **do not affect the calculation.**

For the CPT analysis all frame options are used - see below.

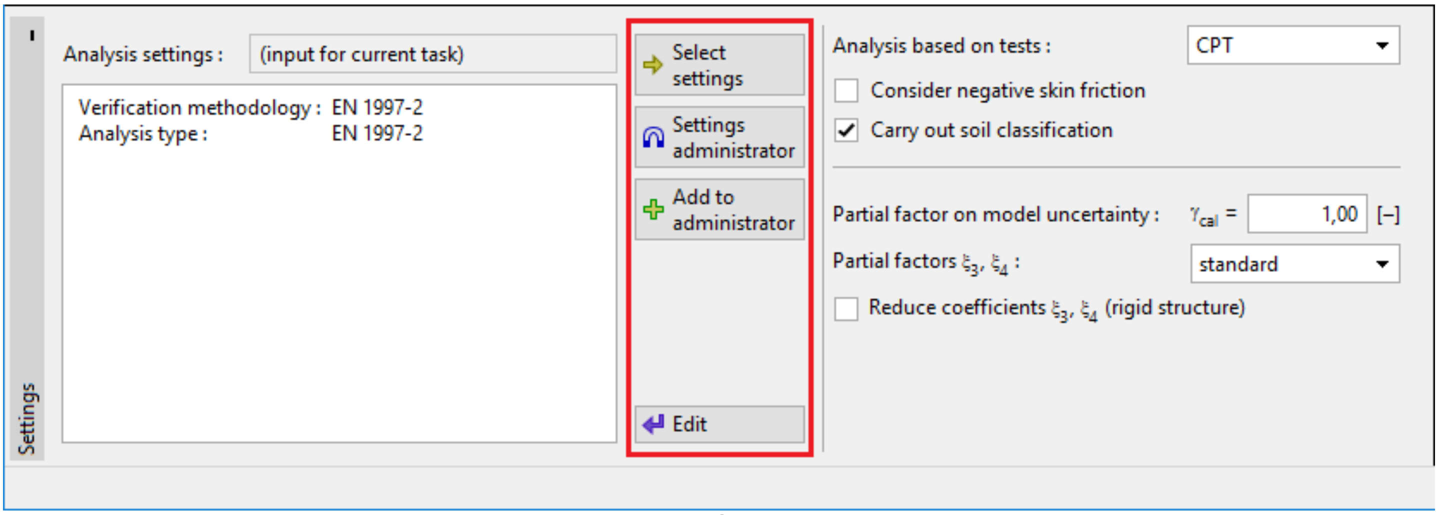
This frame also allows us to introduce negative skin friction. Parameters of the negative skin friction are defined in the frame "GWT+NSF".

This frame also allows us to introduce soil behavior type (SBT) - classification of soils, which is performed in the frame "Soil Classification".

If the analysis is performed **according to EN 1997**, other partial factors are defined:

Partial factor of model uncertainty γ_{cal} reduces vertical bearing capacity of pile.

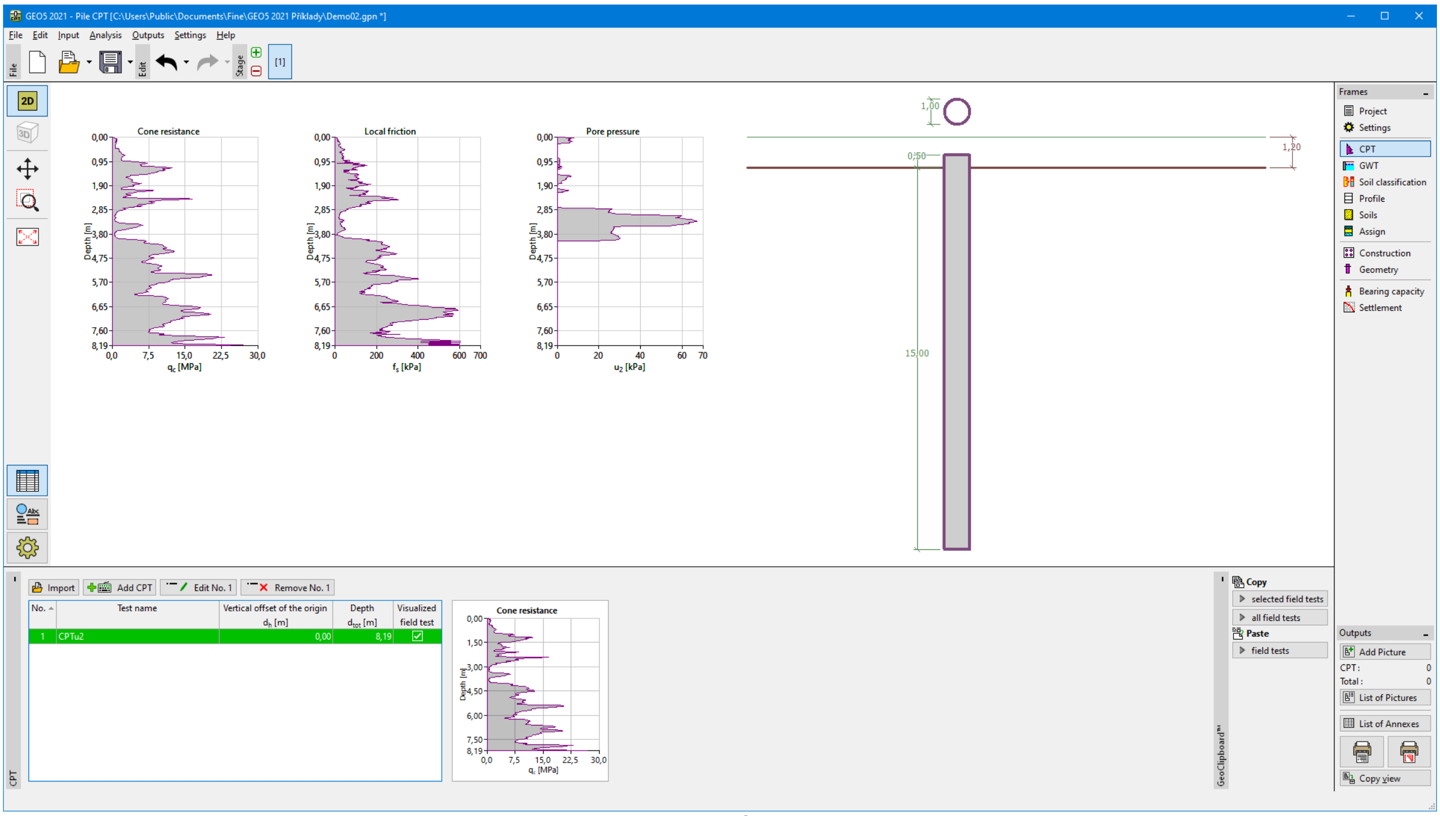
Partial factors ξ_3 and ξ_4 can be either input by the user, or program can calculate standard values depending on number of tests.



Frame "Settings"

CPT

The frame "**CPT**" contains a table with a list of cone penetration tests (CPT).



Frame "CPT"

The type and diameter of the CPT cone are inputted for the NBN EN1997-1 ANB analysis type.

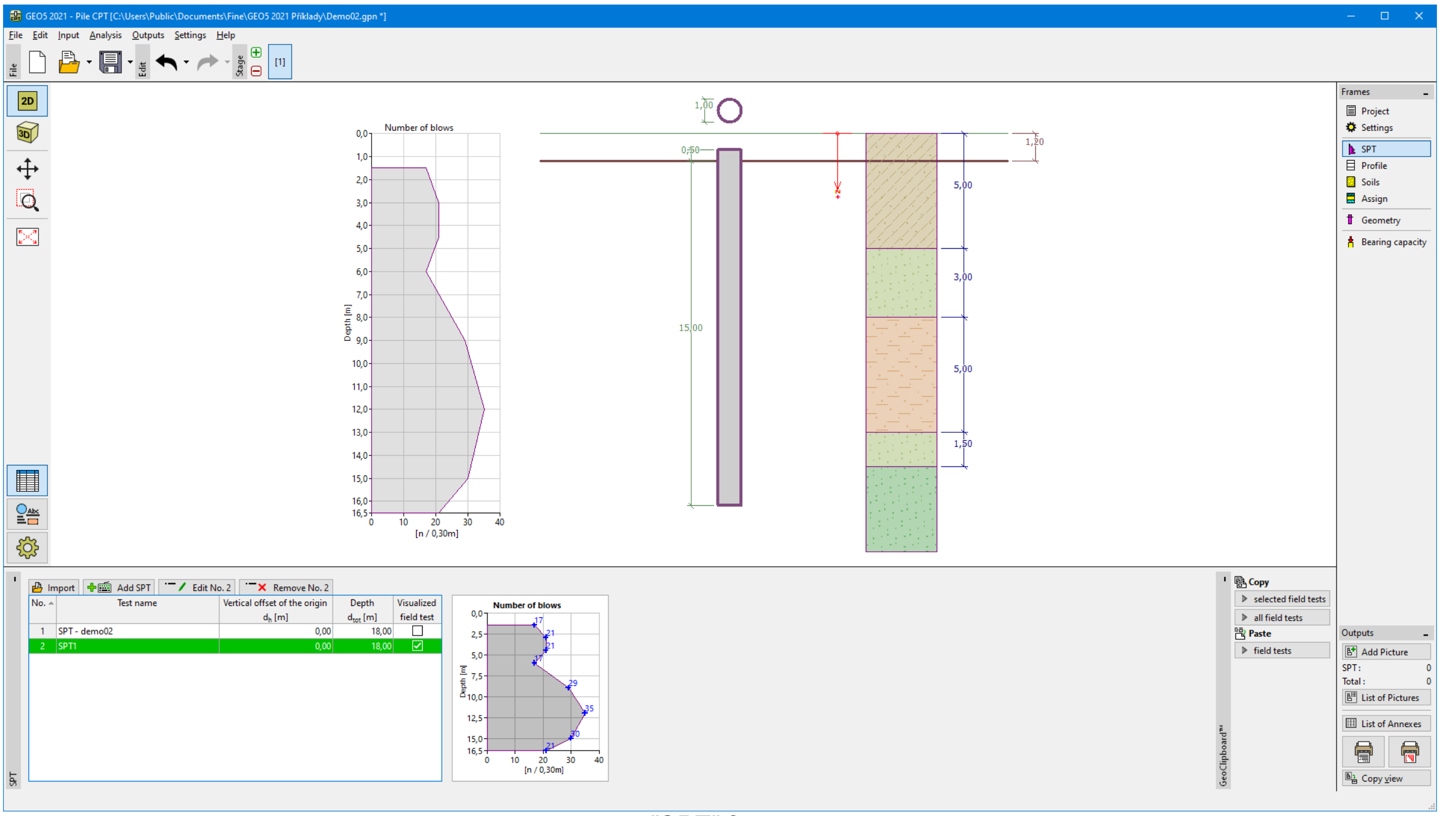
In the tertiary clay layers, the cone resistance is reduced with coefficient $\gamma_{qc} =$ 1,3 for M1/M2 cone types and $\gamma_{qc} =$ 1,15 for M4 cone types.

The results of cone penetration tests (CPT) can be imported into the program in many formats (eg. ***.CPT**, ***.GEF**, ***.AGS**, ***.SPE**, ***.GRU**,***.xlsx**, ***.csv**, ***.ods**,***.txt**).

Data of CPT can be copied within "**Pile CPT**", "**Spread Footing CPT**" and "**Stratigraphy**" programs using "GeoClipboard".

SPT

The "**SPT**" frame contains a table with the list of standard penetration tests (SPT



"SPT" frame

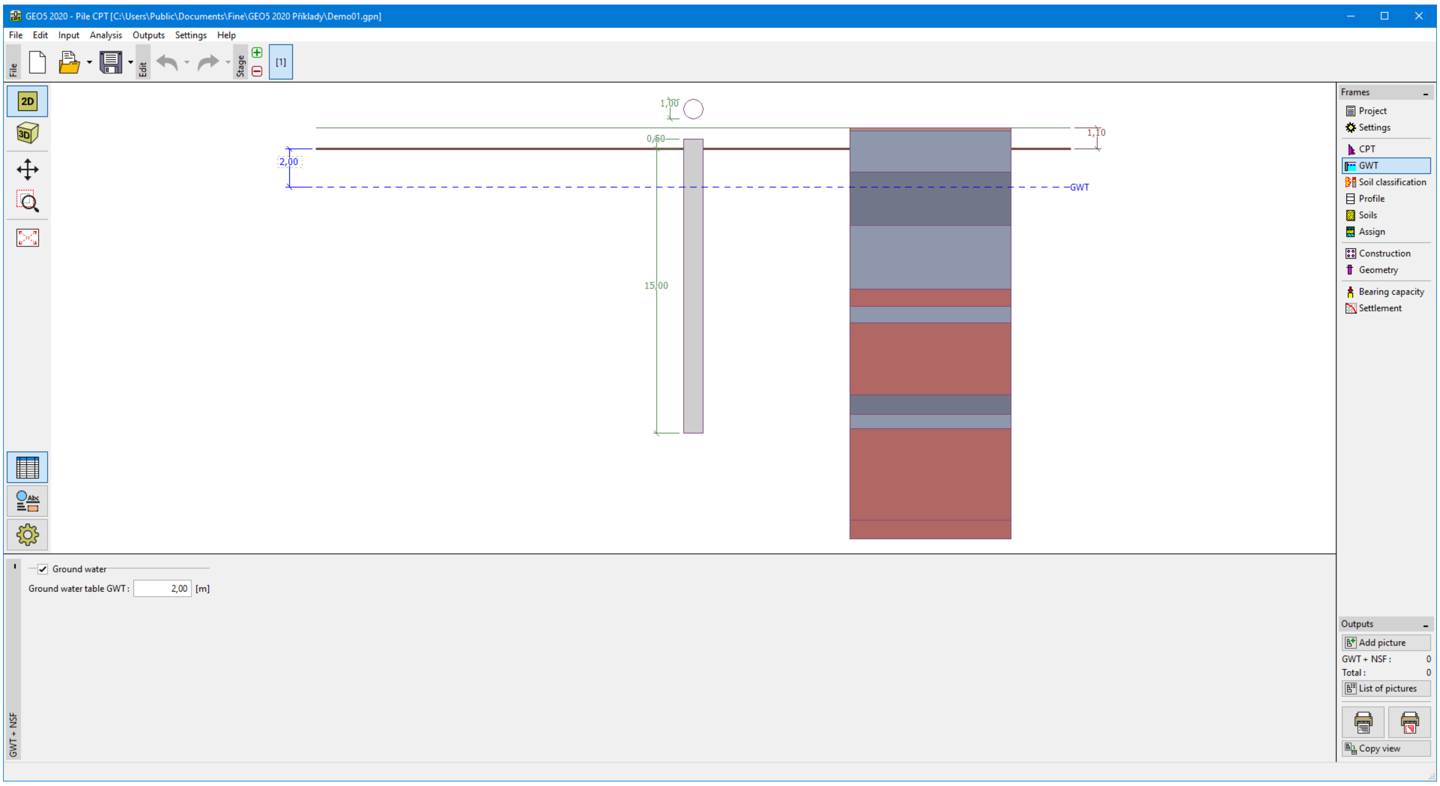
The results of standard penetration tests (SPT) can be imported into the program by inserting the file in different formats (e.g. ***.TXT**, ***.CSV**, ***.XLSX**, ***.ODS**).

Data of SPTs can be copied within "**Micropile**", "**Spread Footing CPT**", "**Pile CPT**" and "**Stratigraphy**" programs using "GeoClipboard".

GWT + NSF

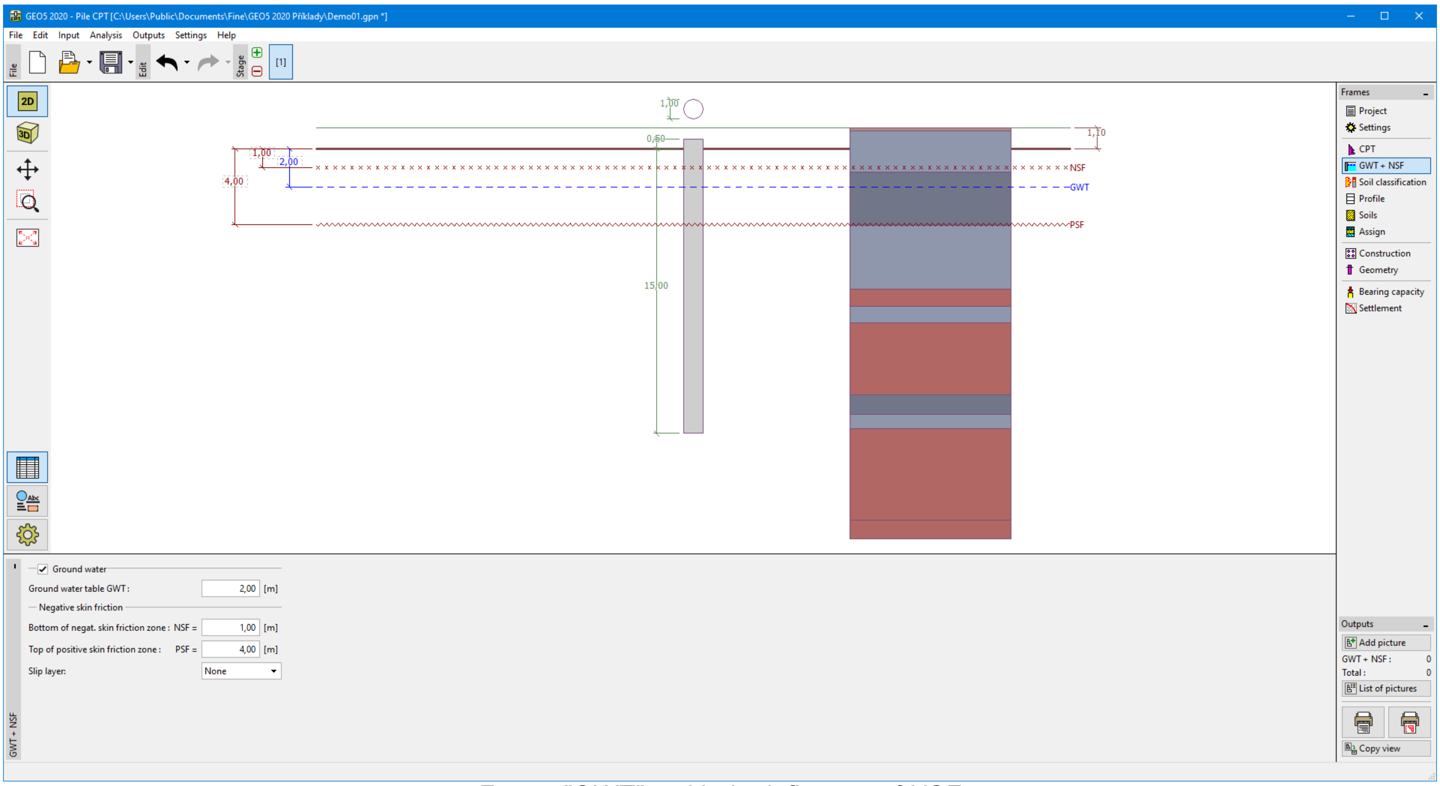
The frame "**GWT**" ("**GWT + NSF**") serves to specify the **depth of groundwater table** and the level of **incompressible subsoil**.

The values can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions. The **GWT** changes the geostatic stress in the soil profile.



Frame "GWT" - without influence of NSF

If the option **"Consider influence of negative skin friction"** is set in the frame "Settings", then it becomes possible to enter parameters of the negative skin friction using the **"GWT + NSF"** frame - boundaries of the region, where the influence of negative skin friction is considered, or sliding region and its material and cohesion.



Frame "GWT" - with the influence of NSF

Soil Classification

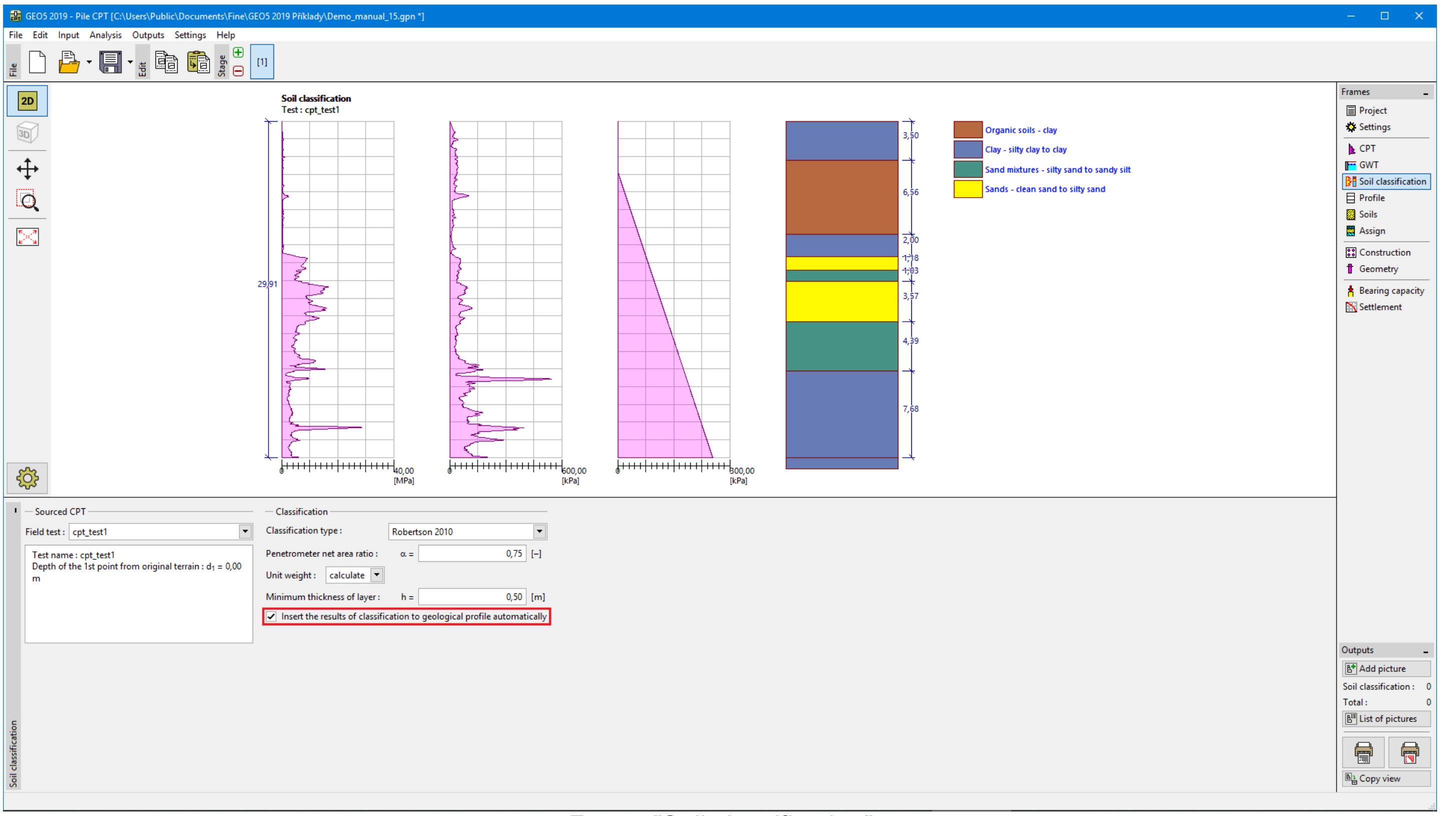
Classification of soils, according to Robertson (1986 or 2010) allows us to specify soil behavior type (SBT) and other parameters directly from the results of CPT - then input parameters of soils it's not necessary to input. We recommend to check generated soil parameters before the calculation.

In the "**Soil classification**" frame **test for classification** is selected (defined in the "CPT" frame). Classification of soils is performed according to Robertson (1986 or 2010).

This frame serves to input penetrometer net area ratio α [-].

The unit weight of soil γ is possible to input with the same value for all layers of soils or it's calculated automatically from the results of CPT (for each layer separately). The frame serves to input a minimum thickness of the layer of soil h . It affects the distribution and number of individual layers of soil in the geological profile of solved task.

By checking the option "**Insert the results of classification into a geological profile automatically**" the generated geological profile assigns to the analysis automatically.



Frame "Soil classification"

Profile

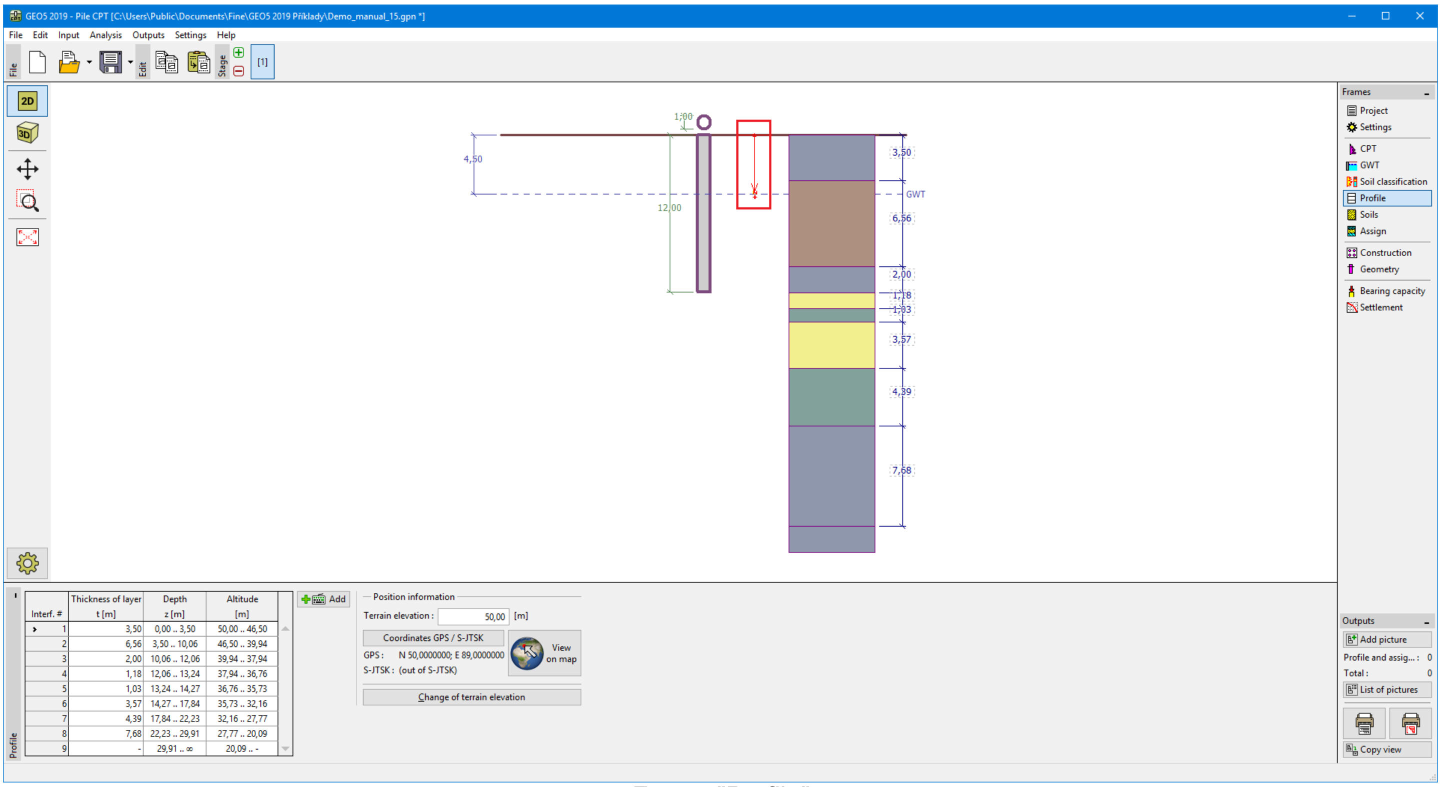
The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.



Frame "Profile"

Soils

The frame "**Soils**" contains a table with the list of input soils. The table also provides information about the currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

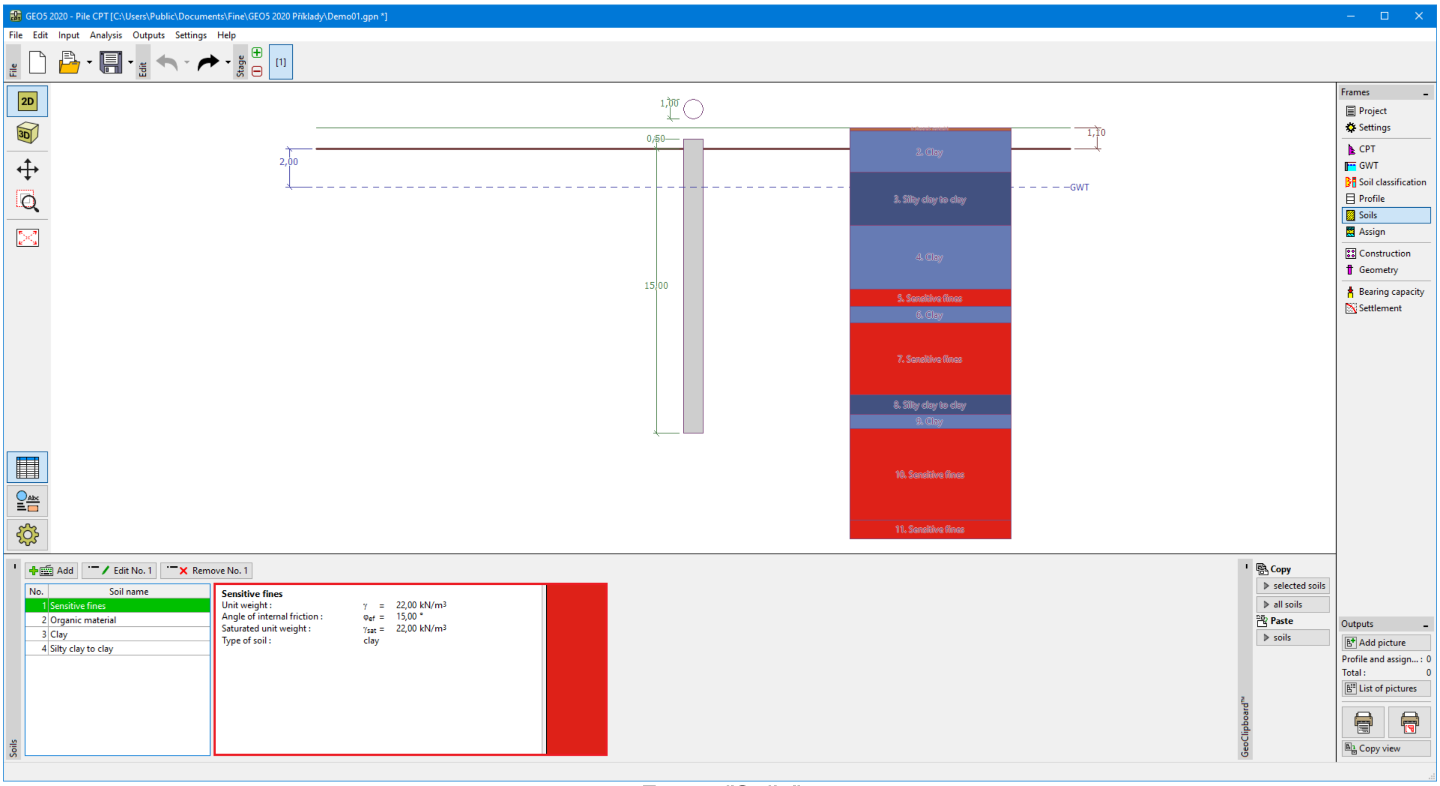
Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data" and "Uplift pressure".

These parameters depend on the theory of analysis specified in the "Piles CPT" tab.

For the SPT analysis, the calculation method is selected in the "Settings" frame.

Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



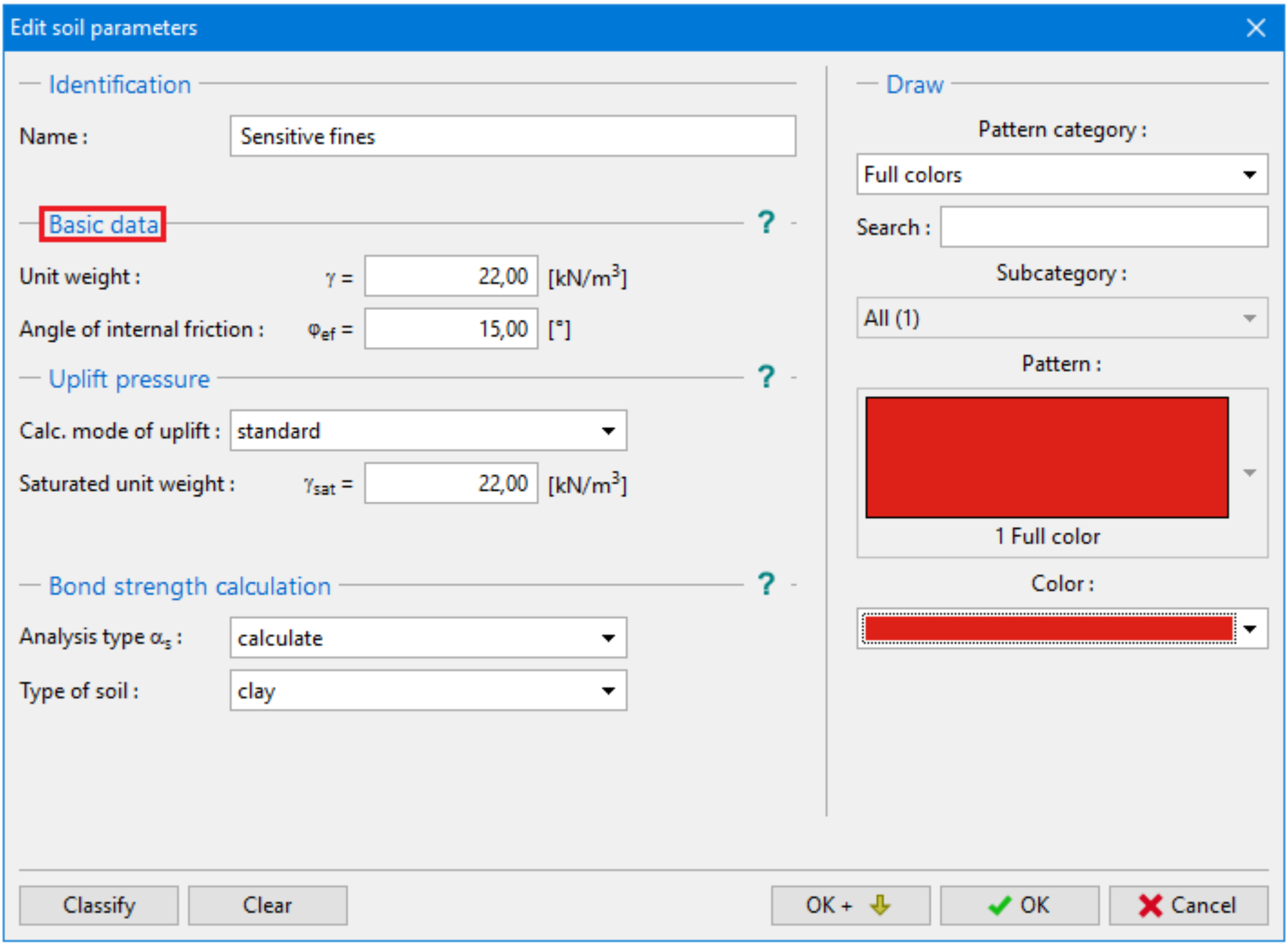
Frame "Soils"

Basic Data

This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction, and cohesion**. The particular values are obtained from a geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit the built-in database of soils, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually.

The associated theory is described in detail in the chapter "Analyses in program Pile CPT".

For calculation of shaft resistance according to the EN 1997-2, NEN 6743 and LCPC (Bustamante) further needs to specify coefficient reducing the shaft friction α_s . For coarse-grained soils - **sands** and **gravels** further needs to specify the magnitude of overconsolidation (OCR) and type of grains.

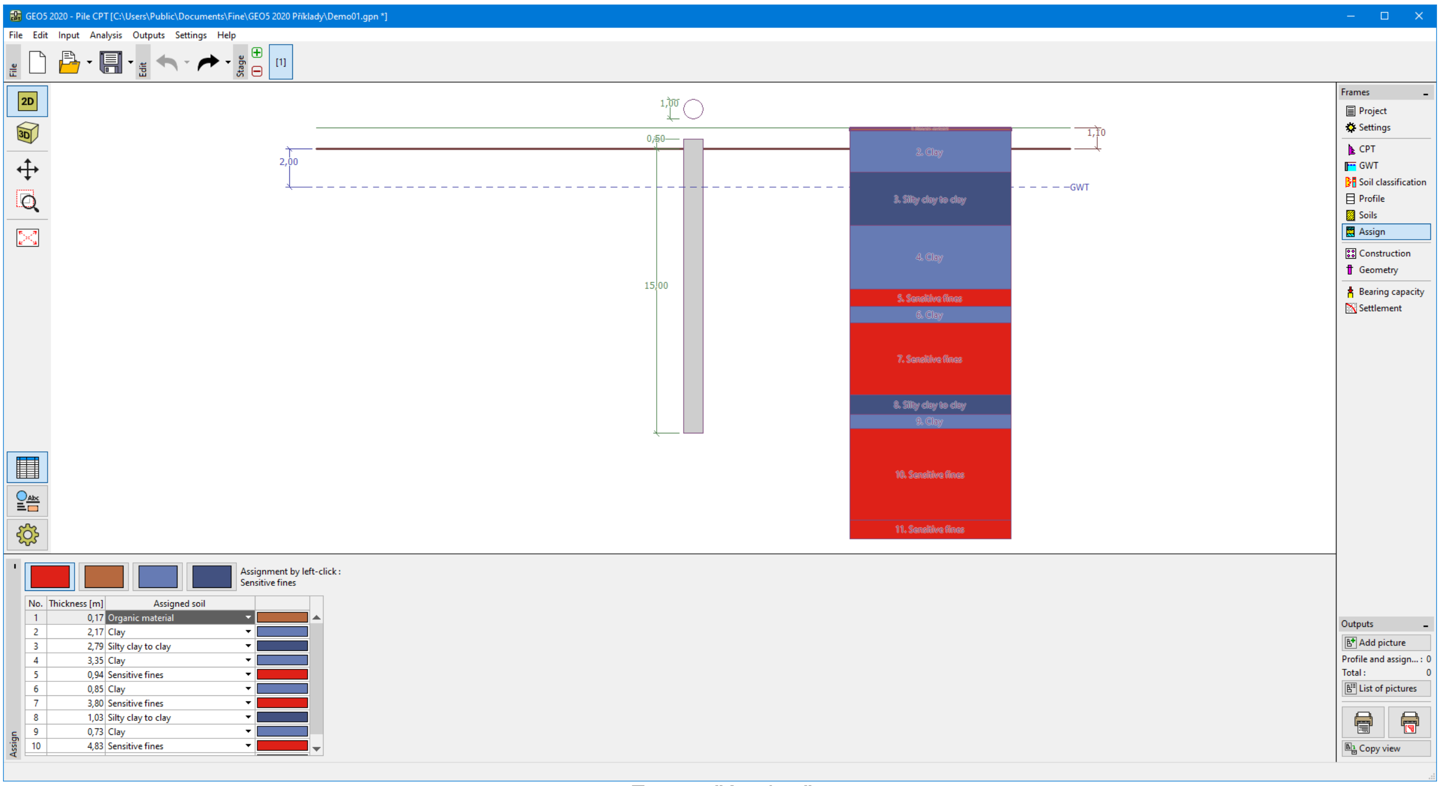


Dialog window "Add new soils" - "Basic data"

Assign

The frame "**Assign**" contains the list of layers of the profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or it is accessible from a combo list associated with each layer of the profile.

The procedure to assign a soil to a layer is described in detail herein.



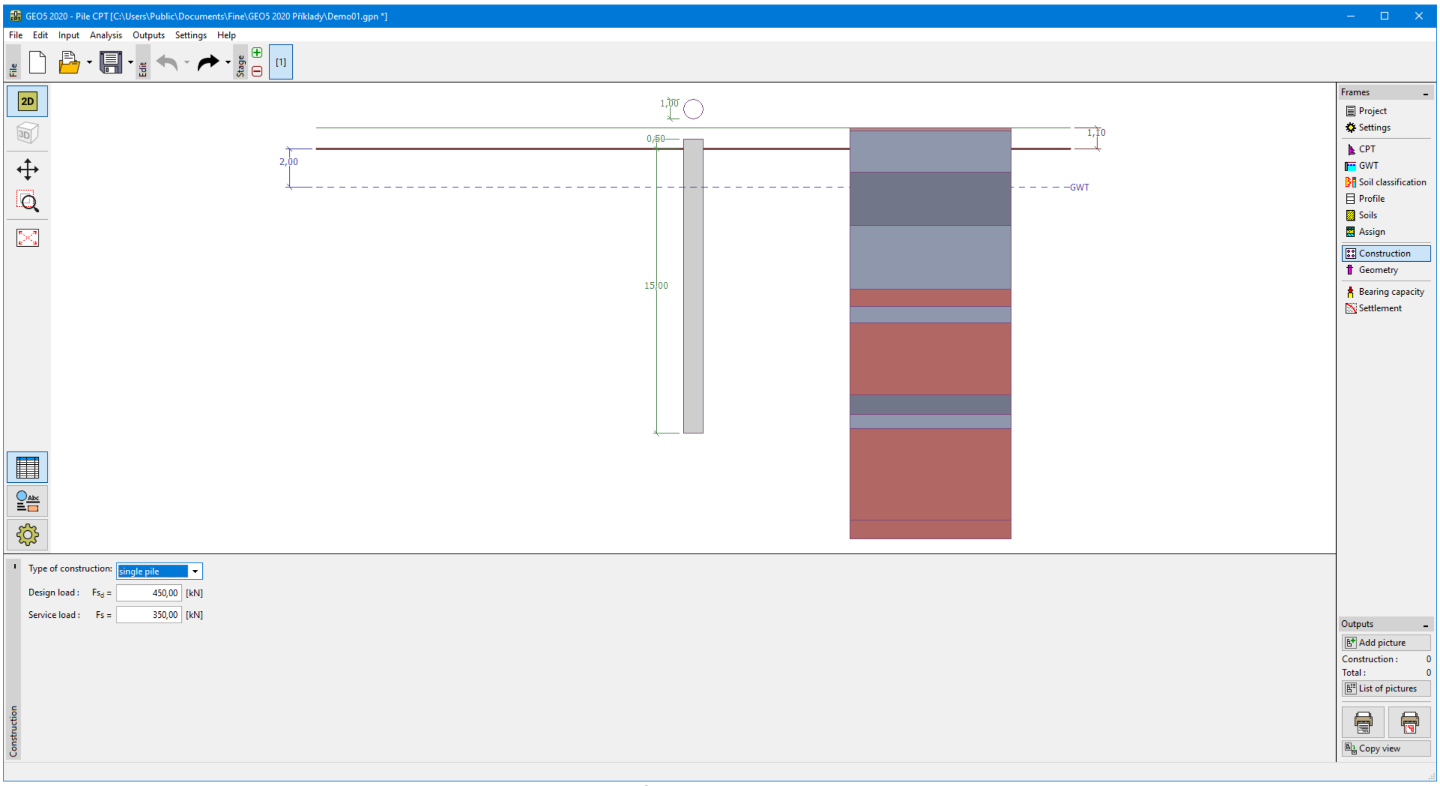
Frame "Assign"

Construction

The frame **"Construction"** serves to select the type of structure - a single pile or a group of piles. This frame also serves to input the values of surcharge - design and standard value. The design value is used to calculate the pile bearing capacity, while the standard value is used to calculate the pile settlement, for both types of load when the NEM standard is employed (state 1B and 2).

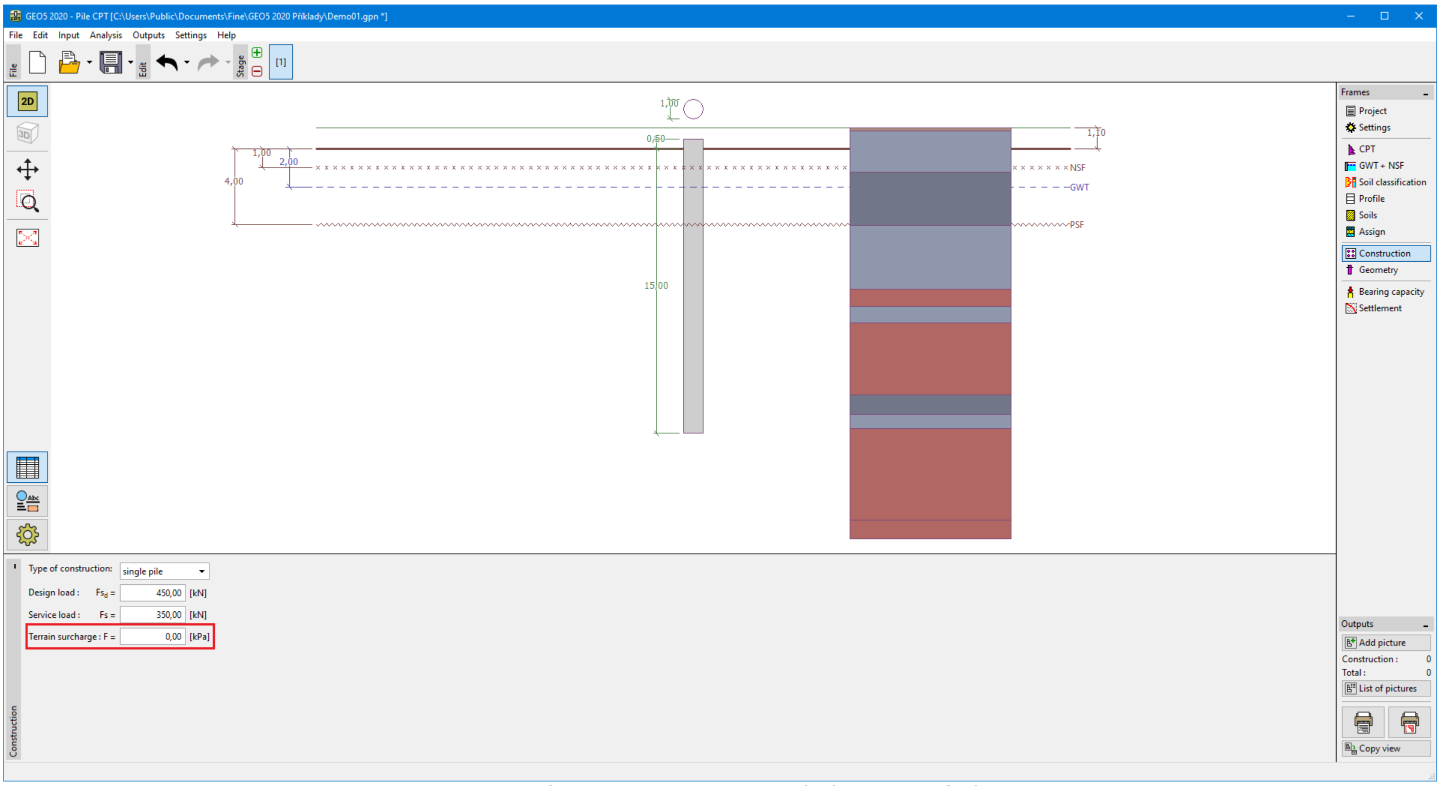
The Size of the construction site must be defined for the NBN EN1997-1 ANB analysis type. It is used for the calculation of correlation coefficients ξ_3 a ξ_4 according to the CPTs density.

The program makes it possible to export the geometry of a structure in the *.DXF format.



Frame "Structure" - single pile

If the option **"Consider influence of negative skin friction"** is set in the frame "Settings", then it is also possible to enter the **surface surcharge** using the **"Construction"** frame.

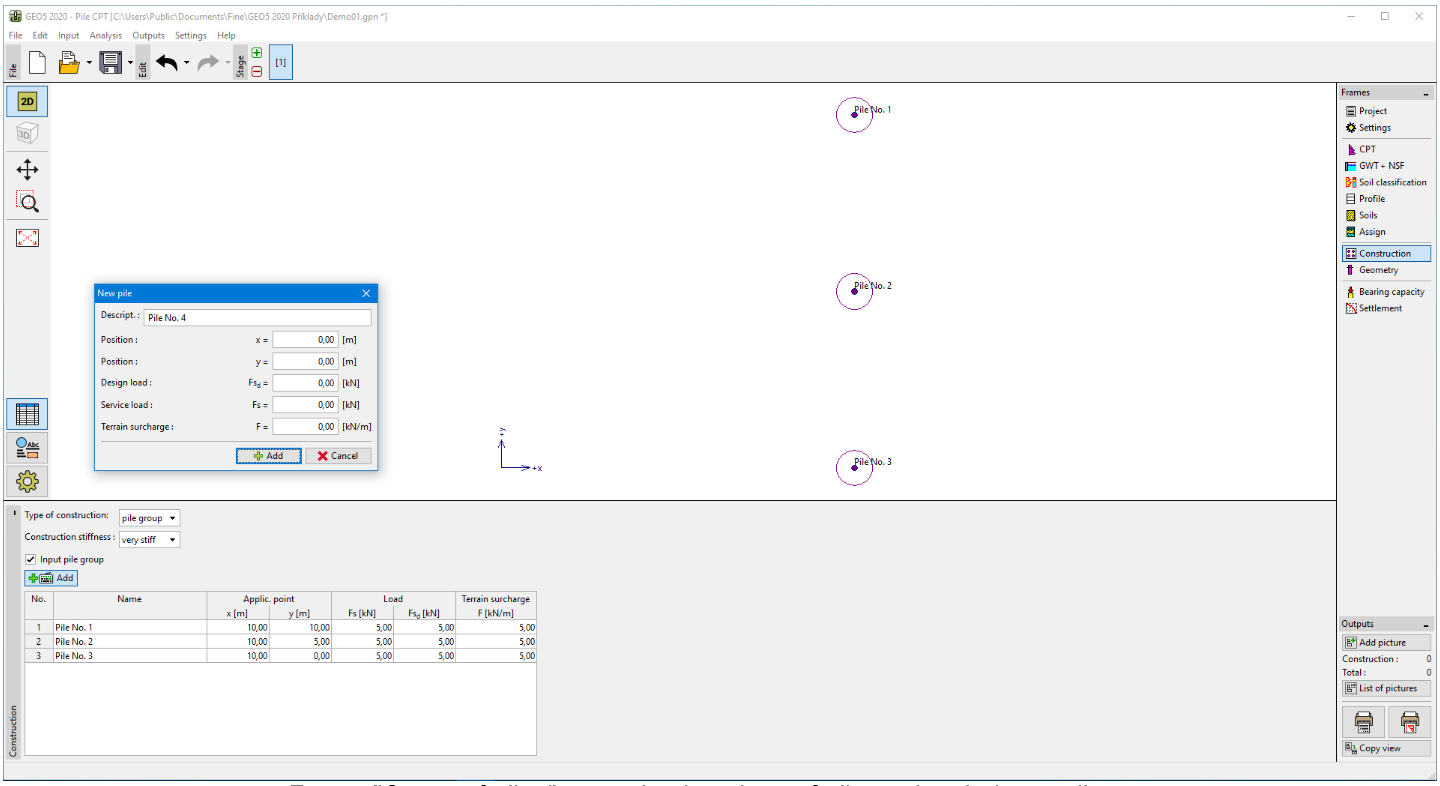


Frame Structure - single pile (influence NSF)

Group of Piles

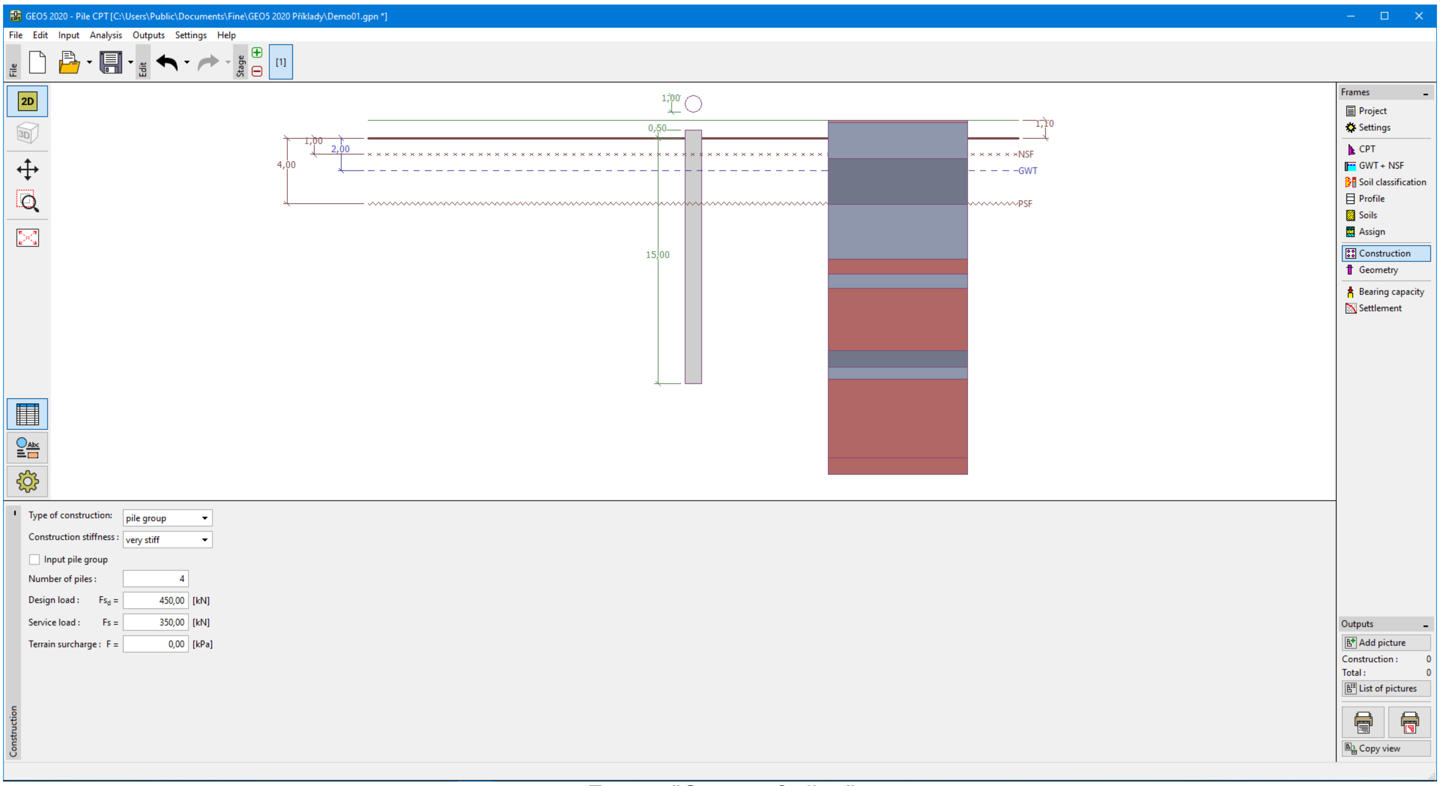
When defining a group of piles, it is necessary to input the structure stiffness, which then drives the way of analysis and verification of the structures. The basic assumption is that for a stiff structure all piles have the same settlement, while for a compliant structure each pile deforms independently. When running the analysis, according to NEN6473 this frame also serves to select the way the CPT is carried out.

For both stiff and compliant structures the program allows for defining locations of individual piles using their coordinates. In such a case the coordinates of each pile are required (in the x , y coordinate system) and the load acts on each input pile. If the option "**Consider the influence of negative skin friction**" is set in the frame "Settings", then it is also possible to enter the **surface surcharge** using the "**GWT + NSF**" frame. Adding a new pile is performed in the "**New pile**" dialog window.



Frame "Group of piles" - entering locations of piles using their coordinates

If the user does not enter the coordinates of piles locations, then their parameters are defined directly in the frame "Construction". Selecting the stiff structure allows for specifying the number of piles below the structure (the piles are then spread uniformly).



Frame "Group of piles"

Geometry

The frame "**Geometry**" serves to input the **pile cross-section** (circular, rectangular, circular with enlargement, rectangular with enlargement) and a type of the pile (cast in place screw piles, prefabricated screw pile, continuous Flight Auger - CFA). Using input fields the cross-section dimensions are then specified for the selected cross-section.

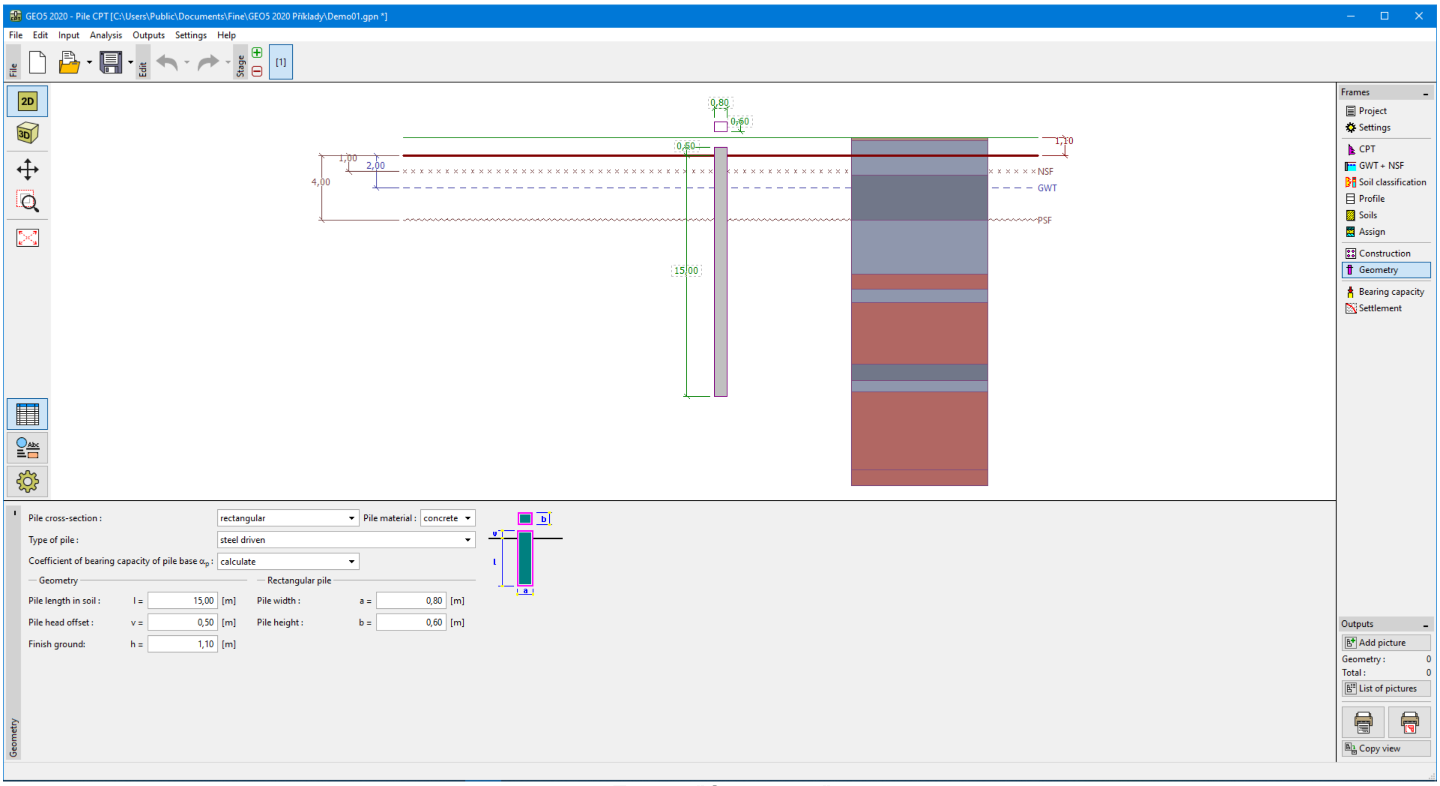
This frame also serves to input a **material of the pile** (timber, concrete, steel) a **geometry of position of the pile** (a pile length in the soil, a pile head offset and a depth of finished grade). The selected shape with a graphical hint of input values is displayed in the right part of the frame.

If the original terrain, from which the CPT tests were carried out, is excavated before piles performing, the measured values of cone resistance q_c are reduced due to a decrease of the geostatic stress.

The base bearing capacity coefficient α_p is specified in the center part of the frame. This coefficient is automatically calculated based on the selected procedure while taking into account the type of pile and the surrounding soil.

When analyzing rectangular piles the pile shape coefficient s is introduced to reduce the base bearing capacity. When analyzing piles with enlargement the expanded pile base coefficient β is introduced to adjust the expanded base bearing capacity.

The program makes it possible to export the geometry of a structure in the *.DXF format.



Frame "Geometry"

Effect of Finished Ground

If the original terrain, from which the CPT tests were carried out, is excavated before piles performing, the measured values of cone resistance q_c are reduced due to a decrease of the geostatic stress at all test points as follows:

$$q_{c,i,red} = q_{c,i} \cdot x_i$$

Where: $q_{c,i,red}$ - reduced cone resistance in the i[th] layer

$q_{c,i}$ - original cone resistance in the i[th] layer

χ_i - reduction coefficient in the i[th] layer

Reduction coefficient χ_i is determined as follows:

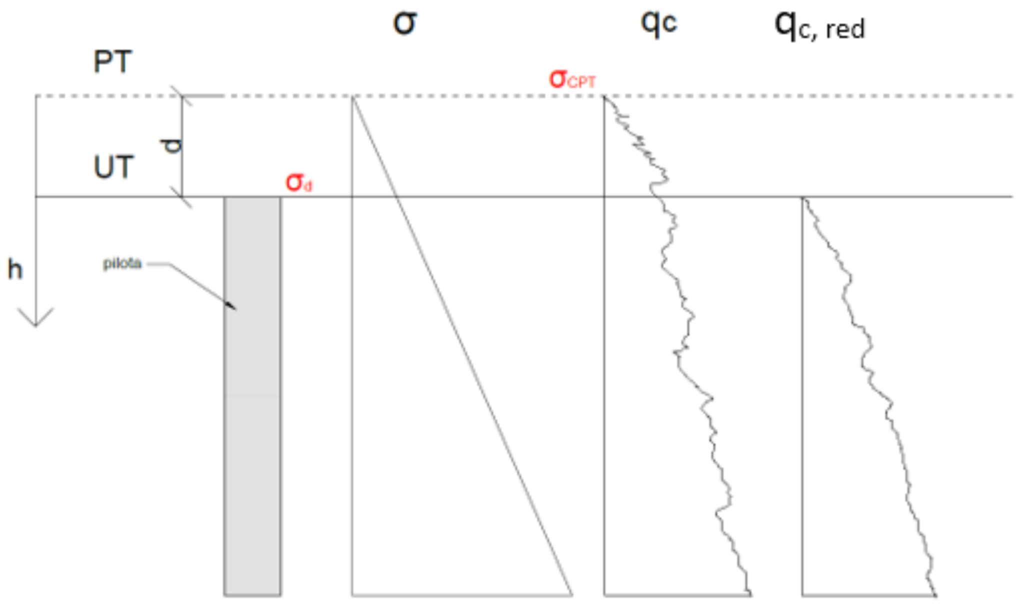
$$x_i = \frac{\sigma_{z,i} - \sigma_d + \sigma_{CPT}}{\sigma_{z,i}}$$

$$x \in (0,1)$$

Where: $\sigma_{z,i}$ - original geostatic stress in the i[th] layer

σ_d - original geostatic stress in the place of finished ground (in the depth d)

σ_{CPT} - original geostatic stress in the place of first point of CPT test



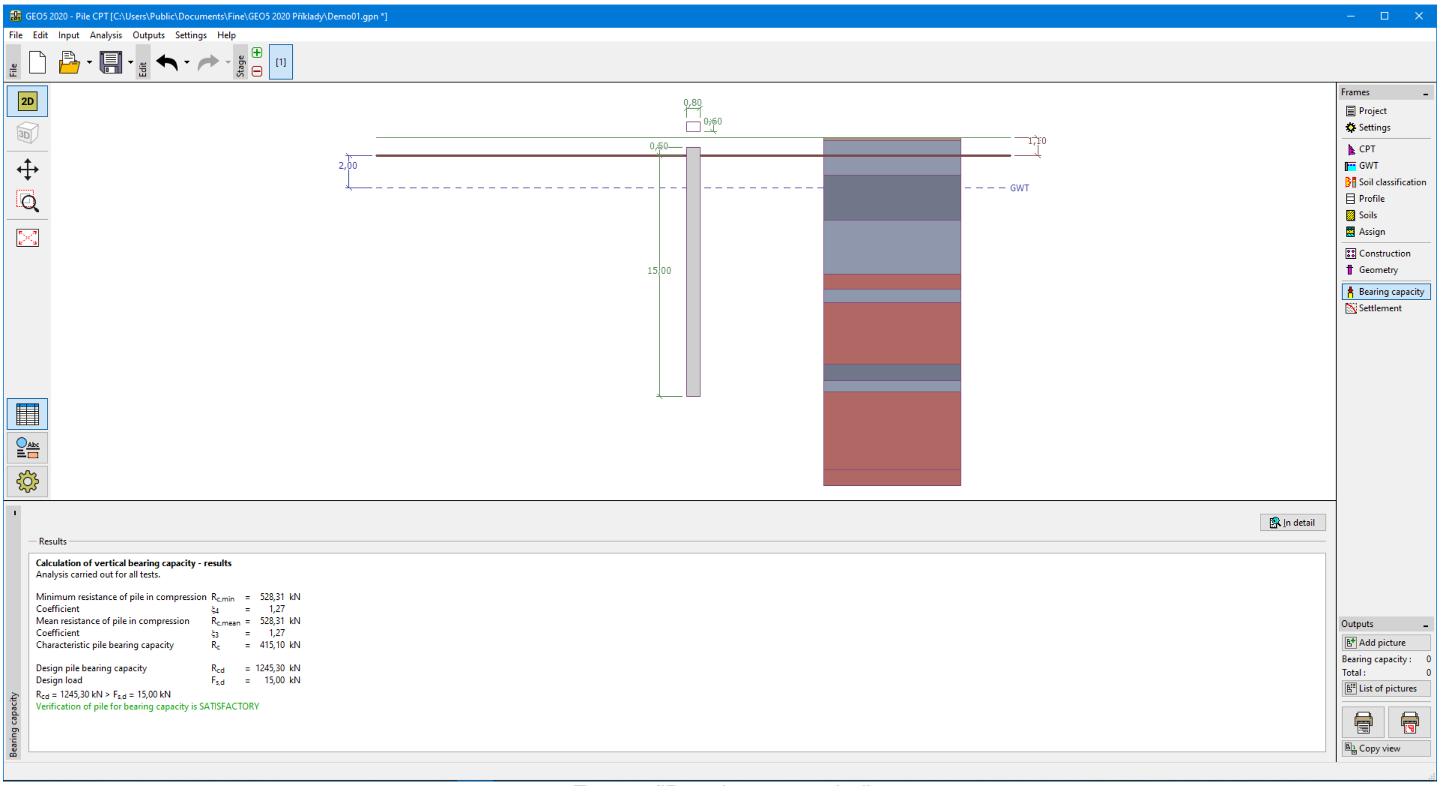
Cone resistance reduction due to effect of finished ground

In case, that CPT test were carried out on the finished ground, cone resistance values are not reduced.

Bearing Capacity - CPT

The "**Bearing capacity**" frame serves to verify the pile **vertical bearing capacity**. The analysis results are plotted in the right bottom part of the frame. The "**In details**" button opens the dialog window, which contains a detailed printout of results from the verification analysis.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Bearing capacity"

Bearing Capacity - SPT

The "**Bearing capacity**" frame serves to verify the pile **vertical bearing capacity**. The program calculates vertical bearing capacity according to the method selected in the Settings frame:

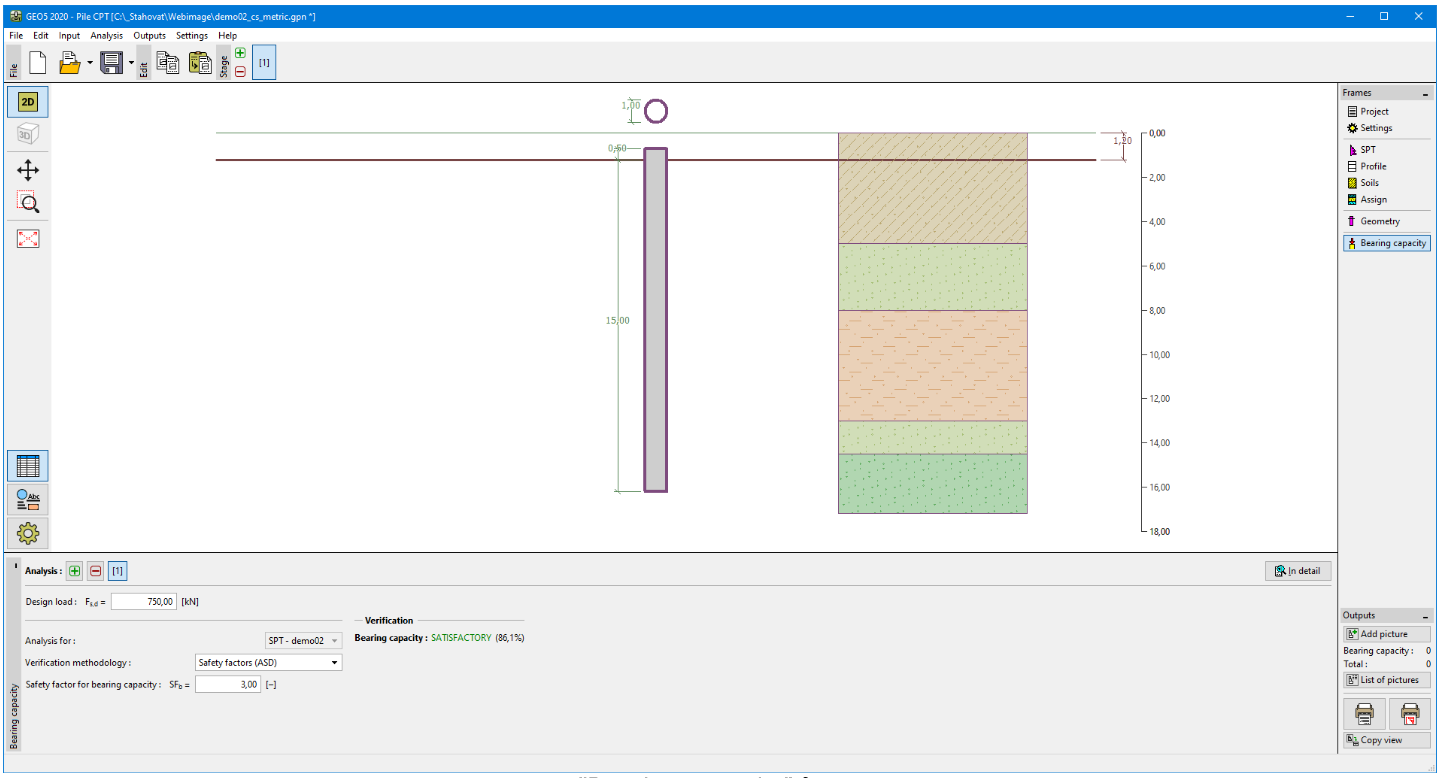
- Décourt-Quaresma
- Aoki-Velloso

The **design value of the load** should be input.

The analysis can be performed for the field test **with the highest utilization**, the **average from all tests** or a **selected test** from the combo list.

Further, it is necessary to select verification methodology according to safety factors or limit states. Required **safety factor** or **base and shaft reduction coefficients** are input in this frame.

The "**In details**" button opens the dialog window with detailed results.



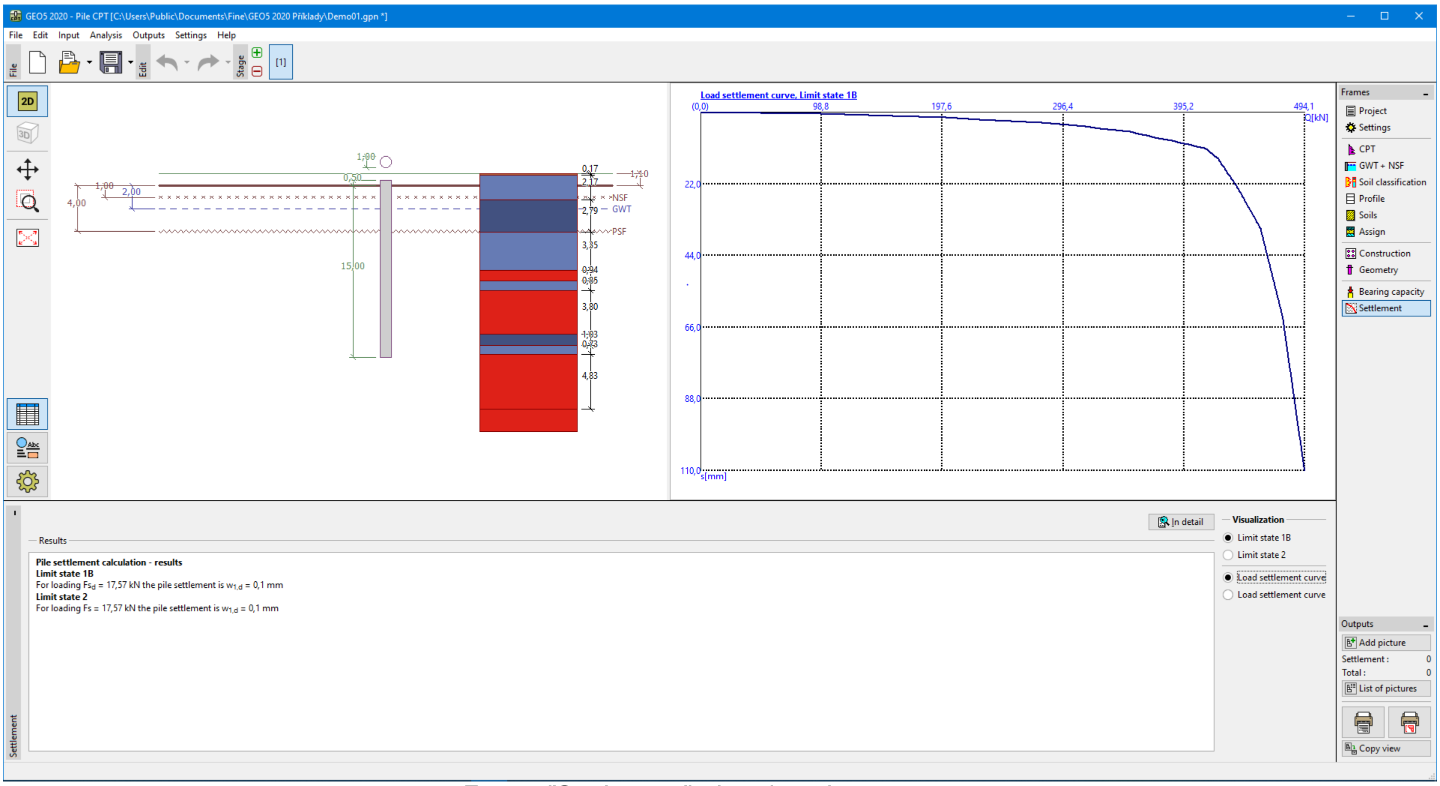
"Bearing capacity" frame

Settlement

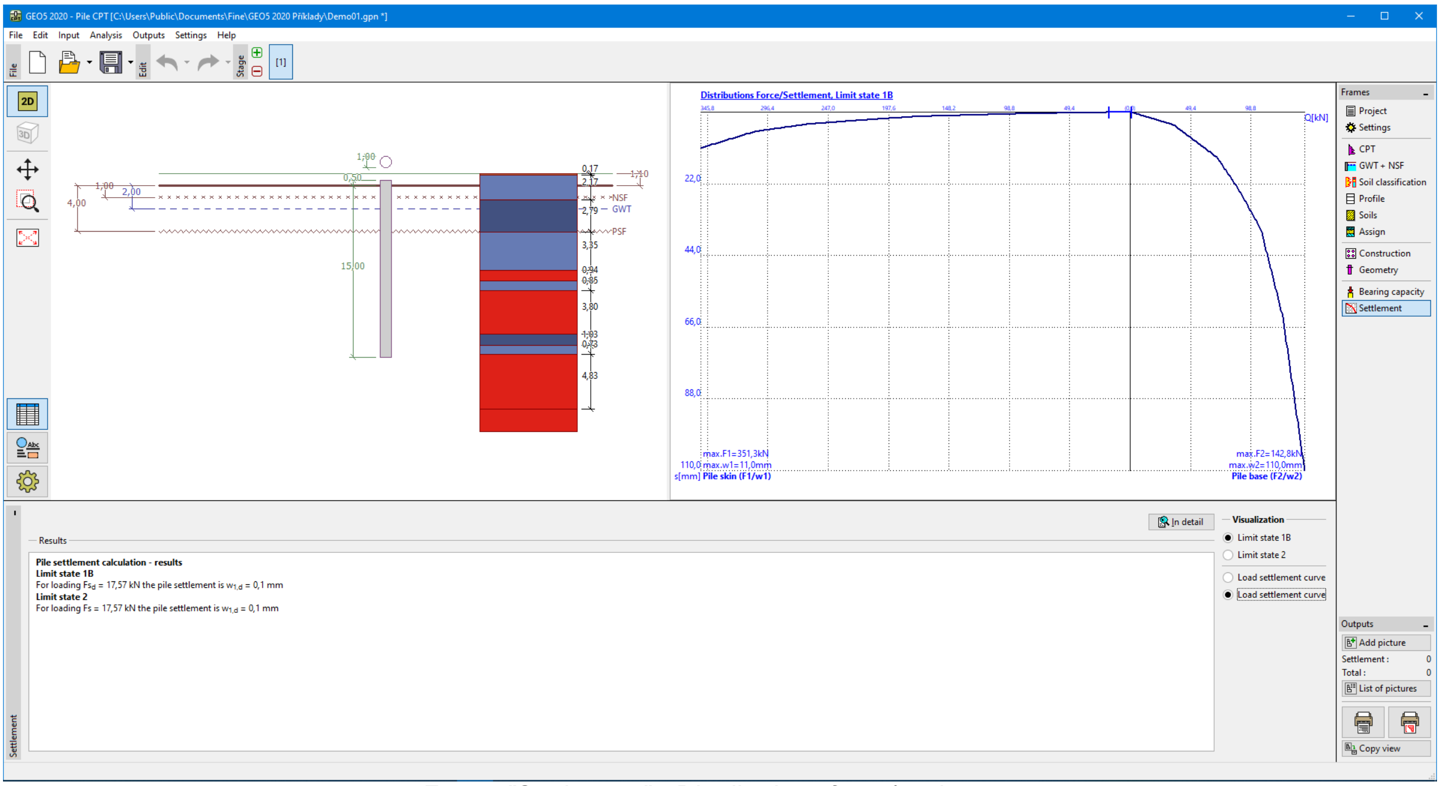
The frame "**Settlement**" serves to verify the pile settlement. The analysis results are plotted in the right bottom part of the frame. The "**In details**" button opens the dialog window, which contains a detailed printout of results from the verification analysis.

The program plots the load-settlement curve, and the load diagram (**force**/**displacement curve**) for NEN 6743 standard.

The analysis results are displayed in the top part of the frame. Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Settlement" - Load-settlement curve



Frame "Settlement" - Distributions force/settlement

Program Pile Group

This program is used to analyze a pile group (pile raft foundation with a rigid pile cap) using both spring method (FEM), or analytical solutions. Both floating piles and piles fixed into subsoil can be considered.

The help in the program "Pile Group" includes the following topics:

- The input of data into individual frames:

Project	Settings	Structure	Geometry	Material	Load	Profile
Soils	Assign	Water	Negative Skin Friction	Vertical Springs	Horizontal Modulus	Stage Settings
Vertical Bearing Capacity	Settlement (Cohesive Soil)	Settlement (Cohessionless Soil)	Analysis (Spring Method)	Dimensioning	Bearing capacity	

- Standards and analysis methods

- Theory for analysis in the program "**Pile Group**":

Stress in Soil Body	Pile Group	Dimensioning of Concrete Structures

- Outputs

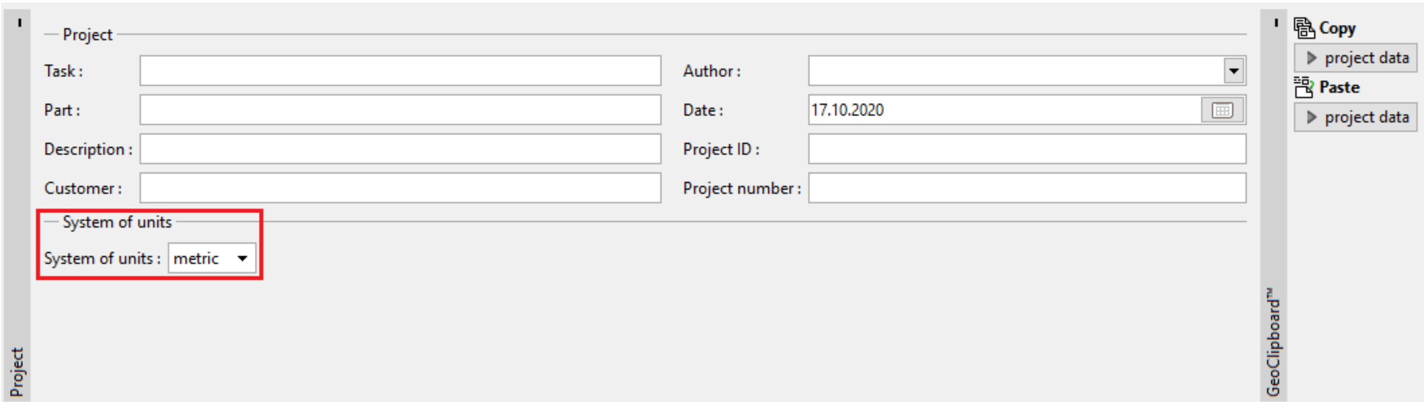
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The frame **"Project"** is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The frame "**Settings**" allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

Settings of analysis parameters are performed in the "Materials and standards" and "Pile Group" tabs.

The right part of the frame allows us to select the type of analysis - analytical solution, spring method or spring method - micropiles.

The analytical solution requires defining the type of subsoil:

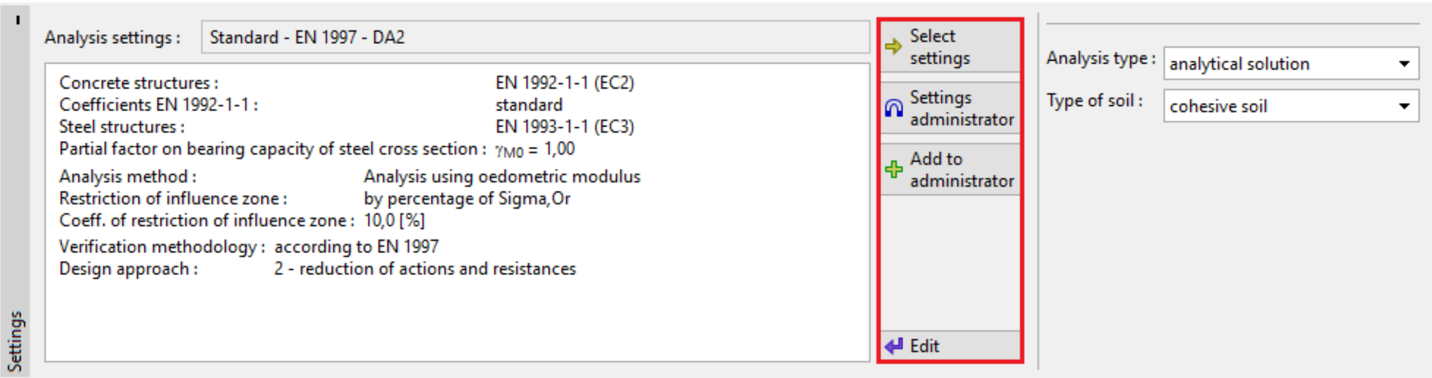
- cohesionless soil (analysis for drained conditions)
- cohesive soil (analysis for undrained conditions)

The spring method requires input of:

- type of pile (pile acts vertically)
- connection piles / pile cap
- modulus of subsoil reaction (pile acts horizontally)

The spring method - micropiles requires input of:

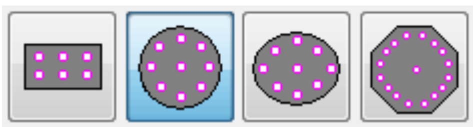
- connection piles / pile cap
- modulus of subsoil reaction (pile acts horizontally)



Frame "Settings"

Structure

The frame "**Structure**" allows us to input the **dimensions of the pile cap** according to the defined scheme, **number of piles (micropiles)**, their **diameter,** and **spacing**.



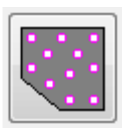
Defined pile group shapes

Piles

Individual piles in the group share the same diameter. A correct and reliable design of a pile group requires meeting the construction rules regarding:

- the number of piles in a group ($3 - 20$)
- the diameter of piles (from $0{,}3\ m$ to $4{,}0\ m$)
- the spacing of piles ($s = 1{,}5d$ to $6d$), where d is the diameter of individual piles in the group
- the overhang of a pile cap from the obverse of outer piles ($o = 0$ to $2d$)

If defined pile cap shapes are not satisfactory for the pile group geometry input, the program allows us to enter the general shape of pile group. The general shape of the pile cap is entered by coordinates of points, but it is also possible (by pressing the button **"Generate general shape"**) to generate coordinates of structure from already input predefined pile cap.



Input of general pile cap shape

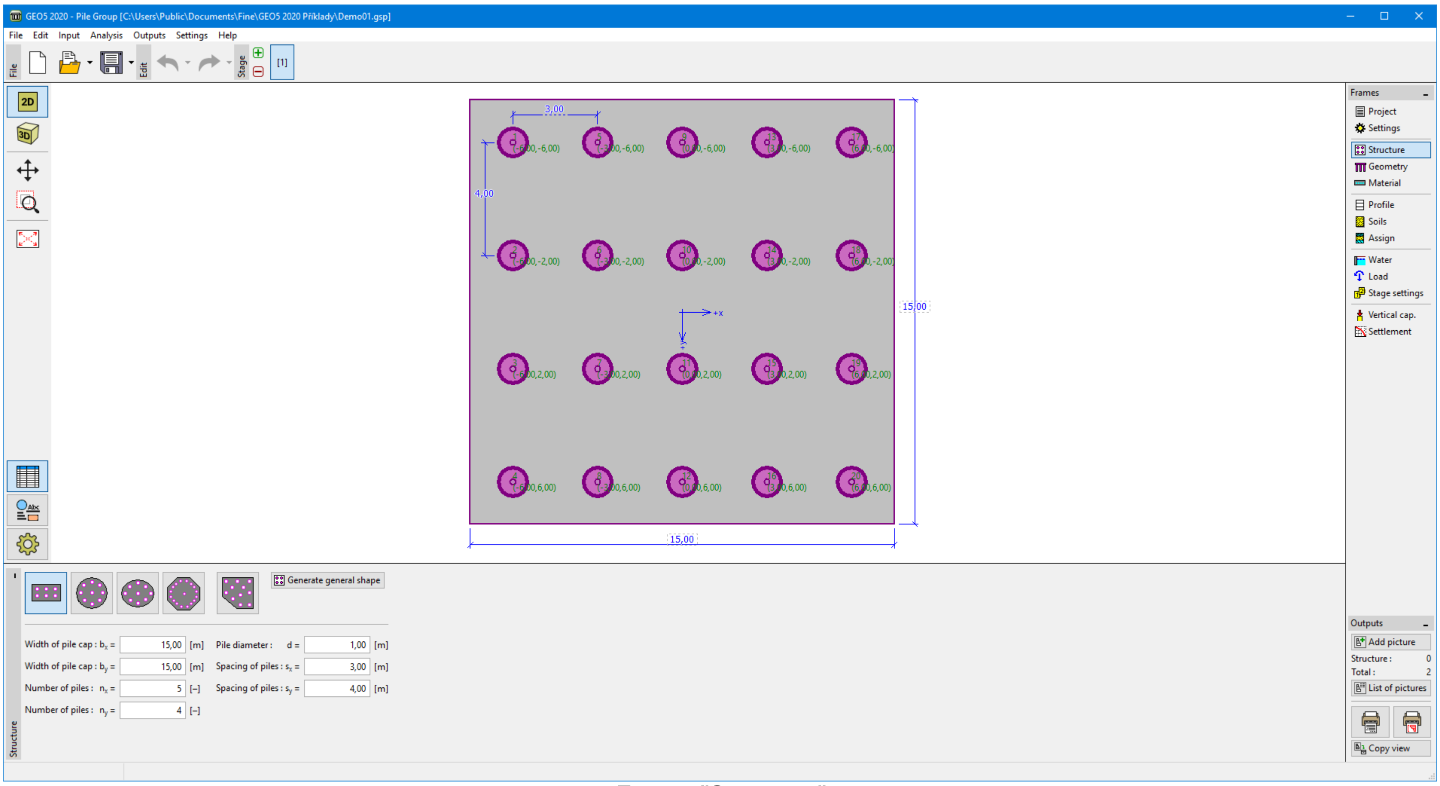
If the general shape of a pile cap is selected, it is possible to input the inclination of piles α . This option ($\alpha \neq 0°$) is available only for the spring method, not for analytical solutions. For analytical solution, all piles are considered as vertical ($\alpha = 0°$). So, for calculation of vertical bearing capacity or settlement of pile group an inclination of piles is not considered.

The program allows us to export the geometry of the structure in *.DXF format.

Micropiles

Individual micropiles in the group have the same diameter. Micropile group requires meeting the construction rules regarding:

- the number of micropiles in a group (3 - 20)



Frame "Structure"

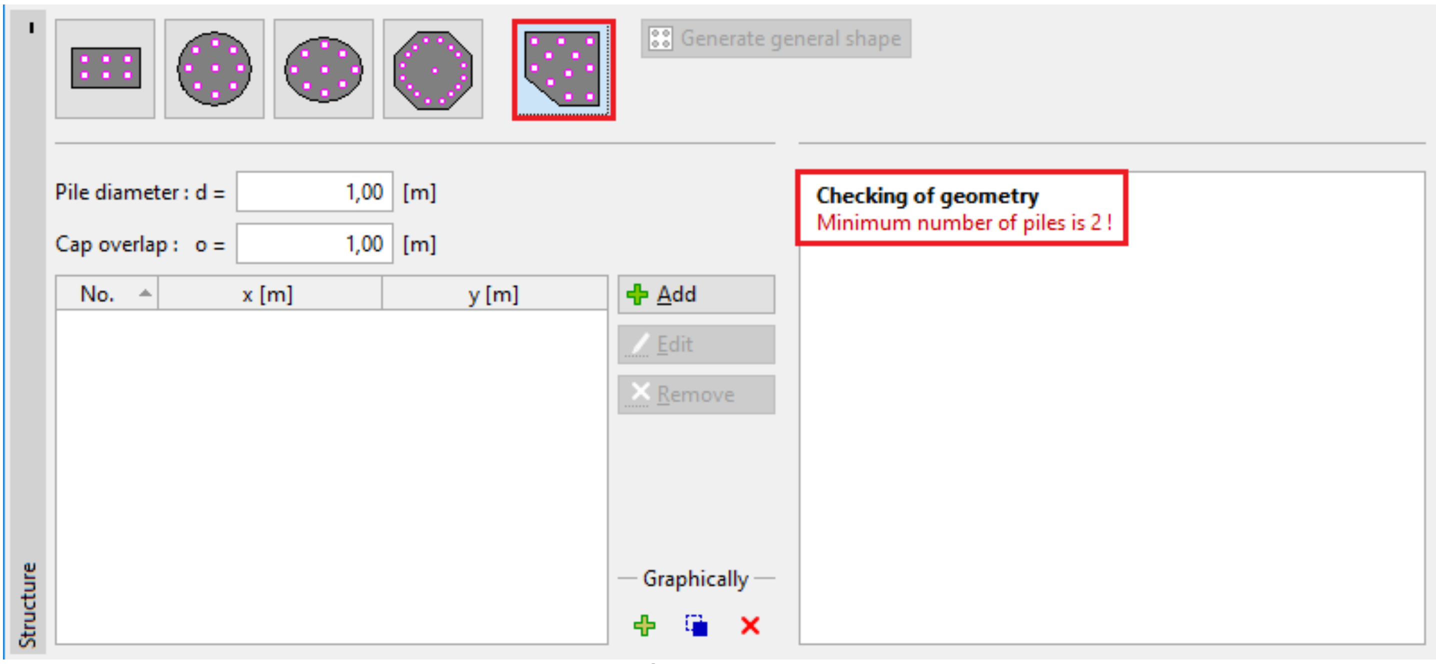
General Pile Group Shape

Input of general pile group shape in a new task

The program allows us to input general pile group shape in two ways:

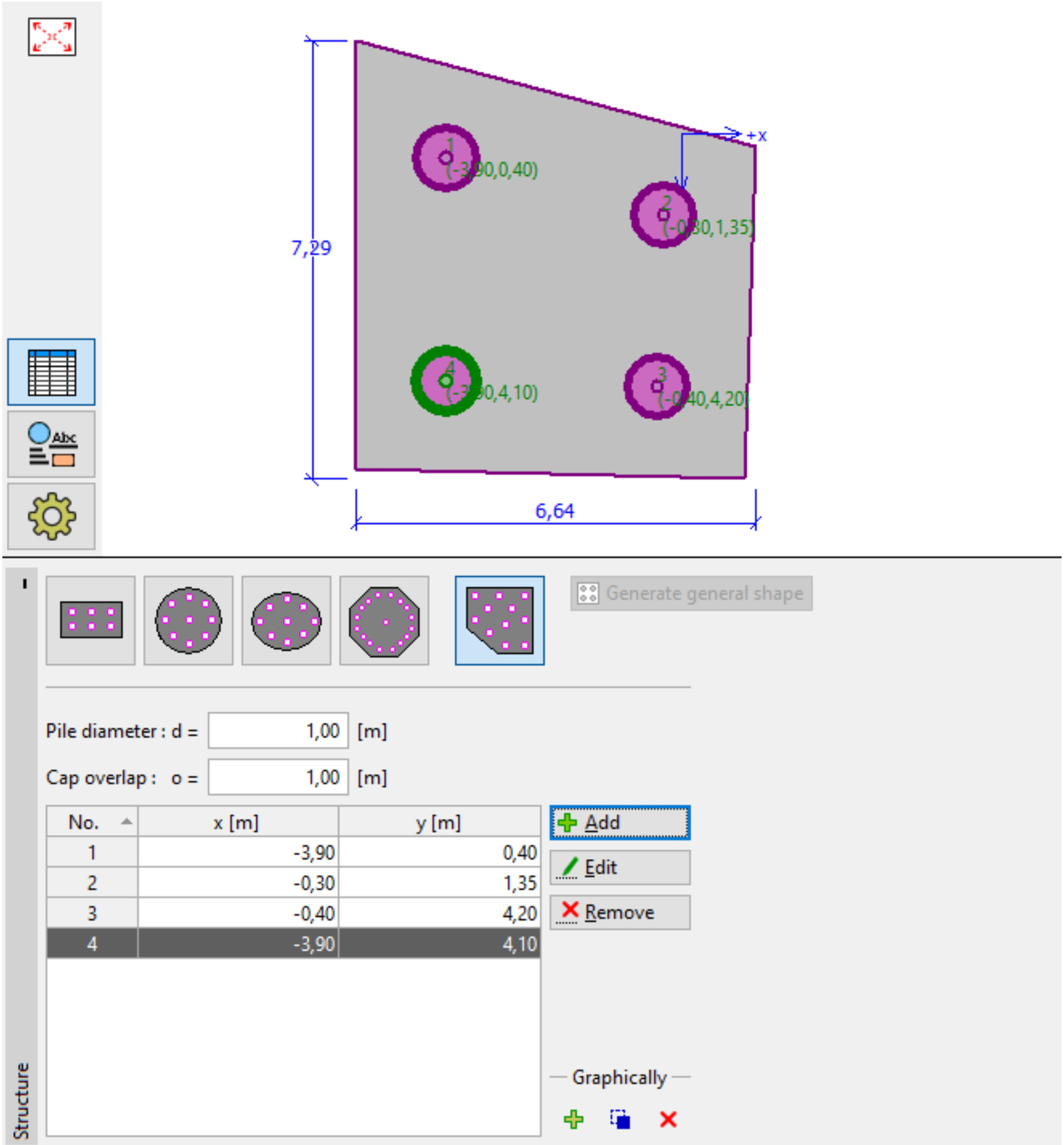
1. Input of general pile group shape using points

By pressing the icon for creating a general pile group shape on the toolbar, the program will delete the desktop. The minimum input number of piles is 3 (in case less number is input, the program will show an error message).



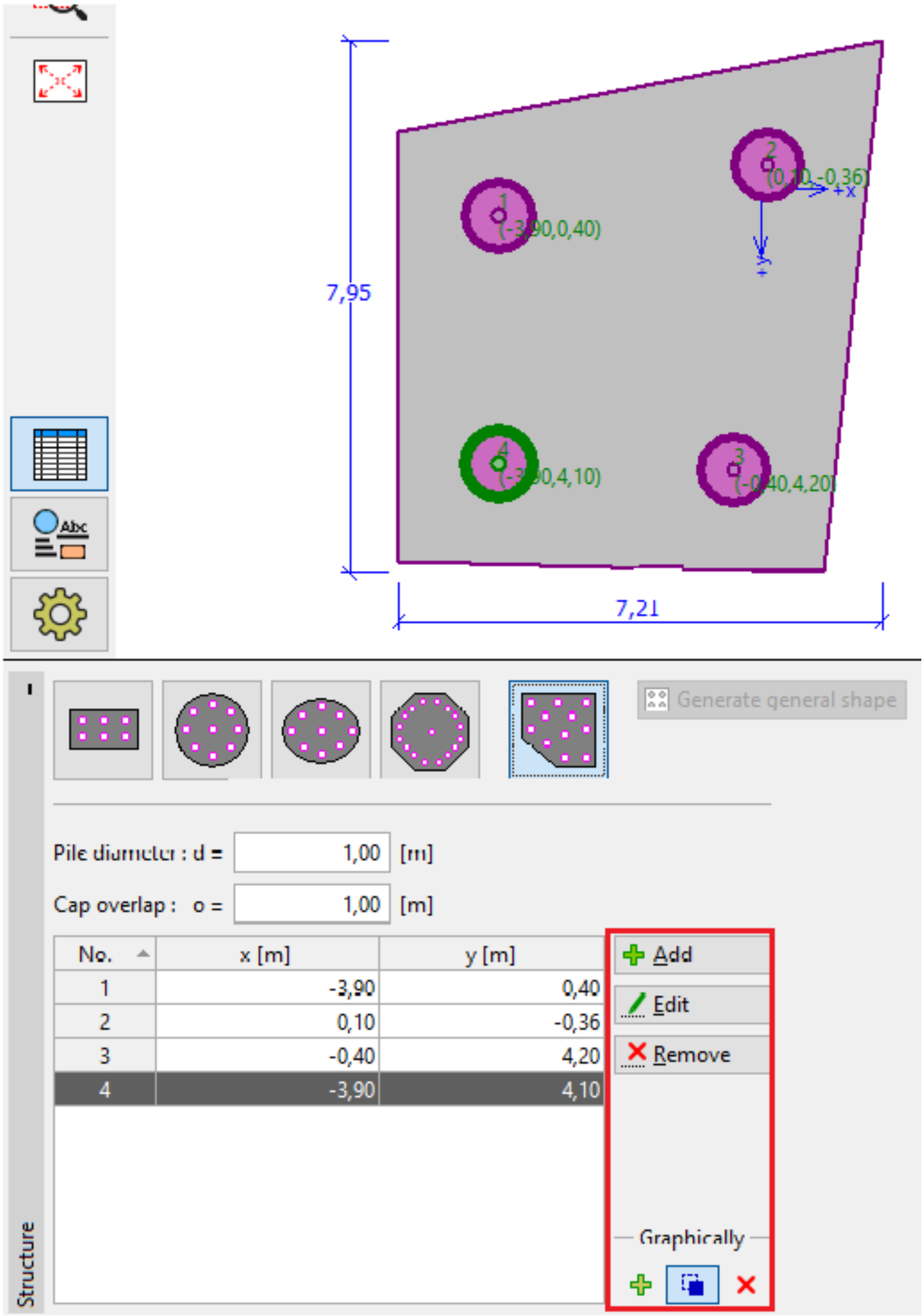
Frame "Structure" - new task

Using the "**Add**" button, which opens a dialog window "**New point**", pile center points are input (it is possible to input points by clicking on the desktop).



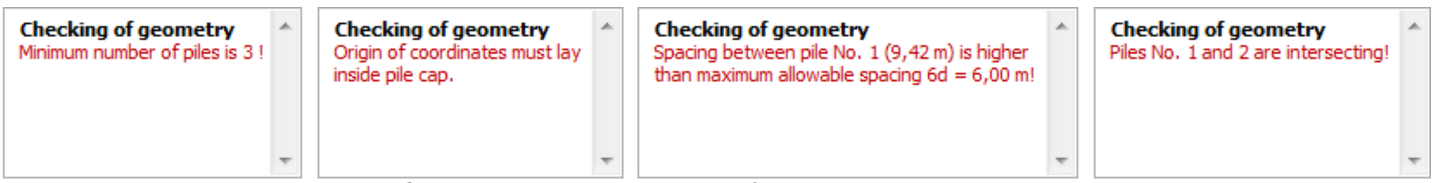
Frame "Structure" - input of general pile group shape using points

Input points are being added to the table, and it is then possible to edit them, and delete them using the desired buttons **"Edit"** and **"Remove"** or by clicking the points on the desktop in the corresponding mode. Points can be moved right on the desktop by a mouse after clicking on the special icon.



Frame "Structure" - edit points

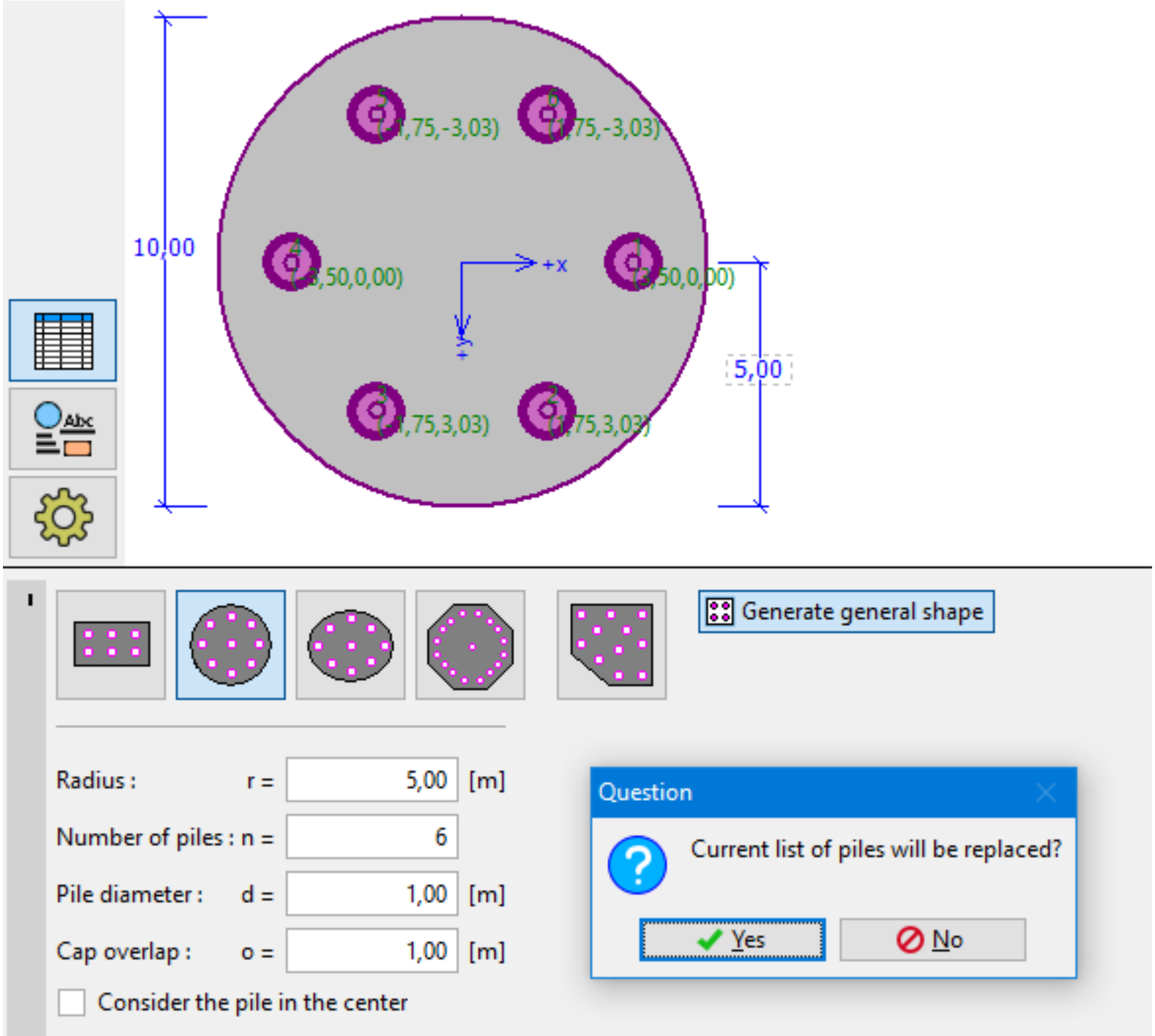
The minimum number of piles in a group is 3. In case of incorrect input (contravention of maximum alloqable spacing of piles, intersecting piles) the program checks the geometry and warns the user of an error. In that case, it is necessary to change the location of the piles.



Frame "Structure" - error messages of general pile group shape input

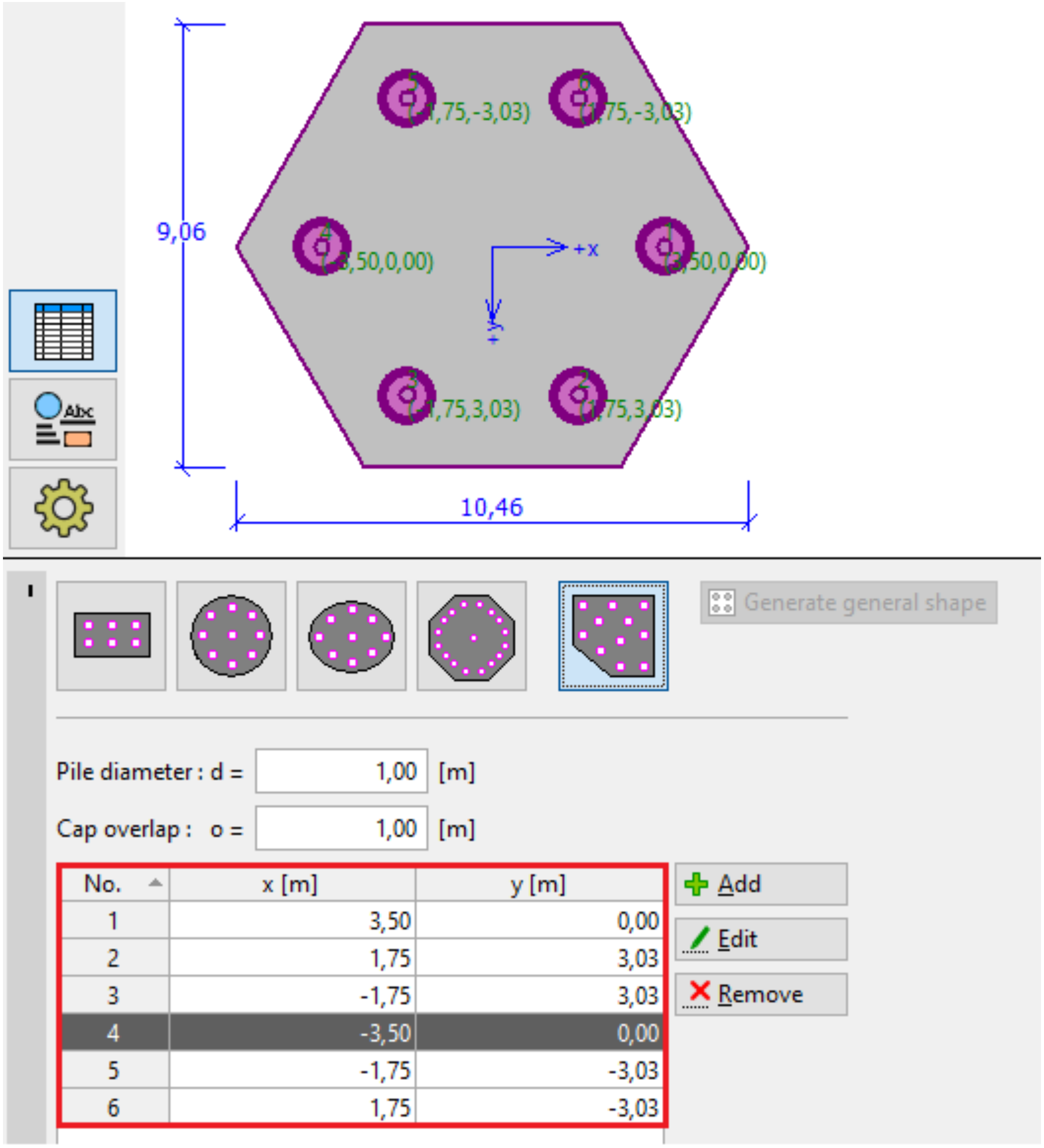
2. Input of general pile group shape using general shape generator

The structure defined by the scheme of construction and its dimensions can be taken to the general pile group shape input by pressing the "**Generate general shape**" button. It is possible to work with the newly generated points and edit the generated pile cap shape.



Frame "Structure" - input of pile group shape using general shape generator

Frame appearance is then changed as in the first case of general pile group shape input. It is possible to work with the picture of the structure as already described.



Frame "Structure" - frame appereance after point input

Geometry

The frame "**Geometry**" allows us to input the parameters of piles (micropiles). The appearance of the frame varies according to the selected analysis type in the frame "Settings" (spring method, analytical solution, spring method - micropiles).

The **analytical solution and spring method** requires input of:

- depth from ground surface
- pile head offset
- thickness of pile cap
- length of piles

The **spring method - micropiles** requires input of:

- thickness of pile cap
- length of micropiles
- diameter of root
- root length
- resistance of foundation soil - this input is very important and has a big influence on results of the analysis - the magnitude of resistance of subsoil depends on the type of subsoil, the process of building of structure and history of loading. The force N is subtracted from the entered load in all load cases.

$$N = A\,R$$

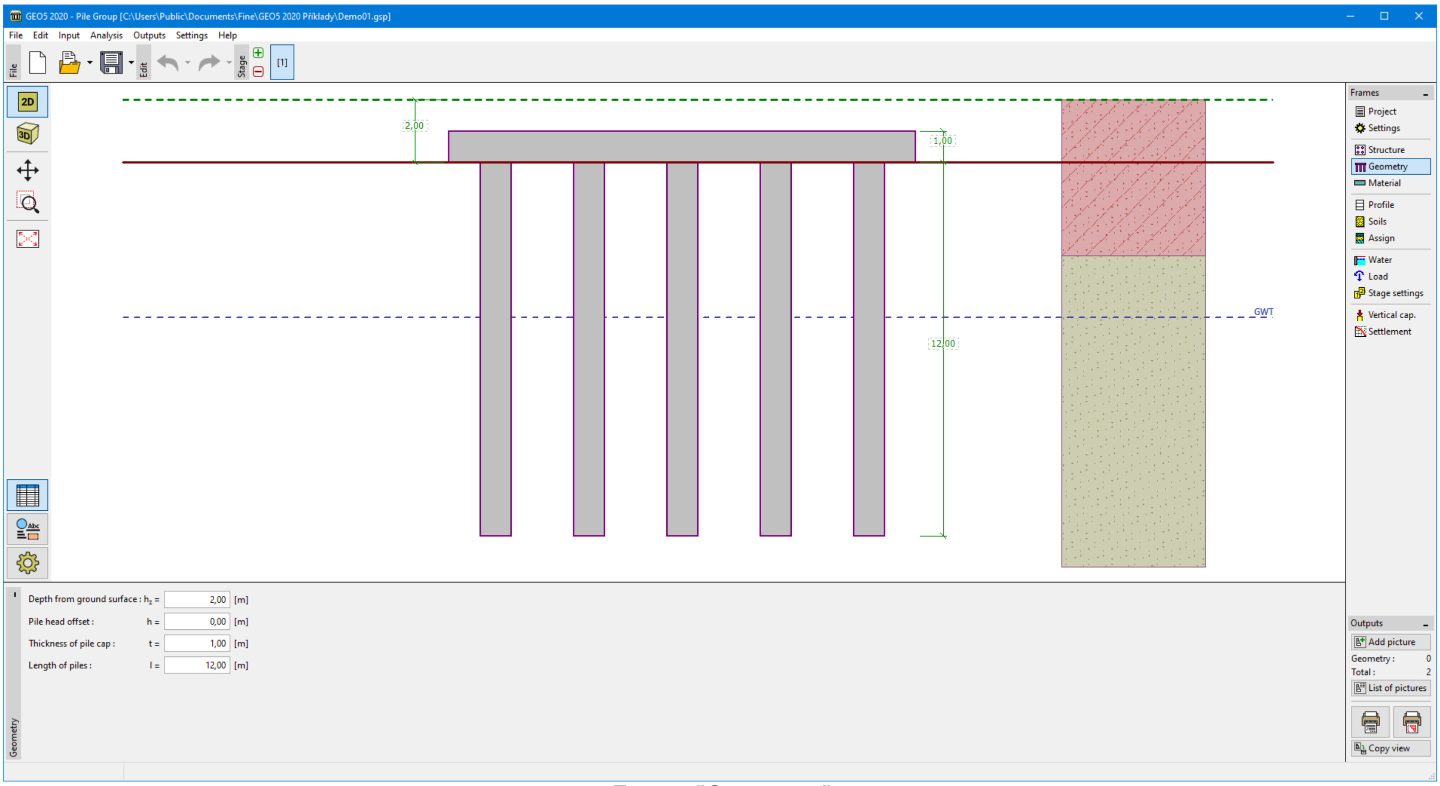
where: A - foundation area

R - entered resistance of foundation soil

Limit values of soil resistance

- entered resistance is equal to zero => all load is transferred to the micropiles, the soil under the foundation is not considered
- entered resistance is higher than total load and self-weigth of cap => no load is transferred to the micropiles and the settlement of foundation is zero.

Individual piles in the group are always of the **same length**.



Frame "Geometry"

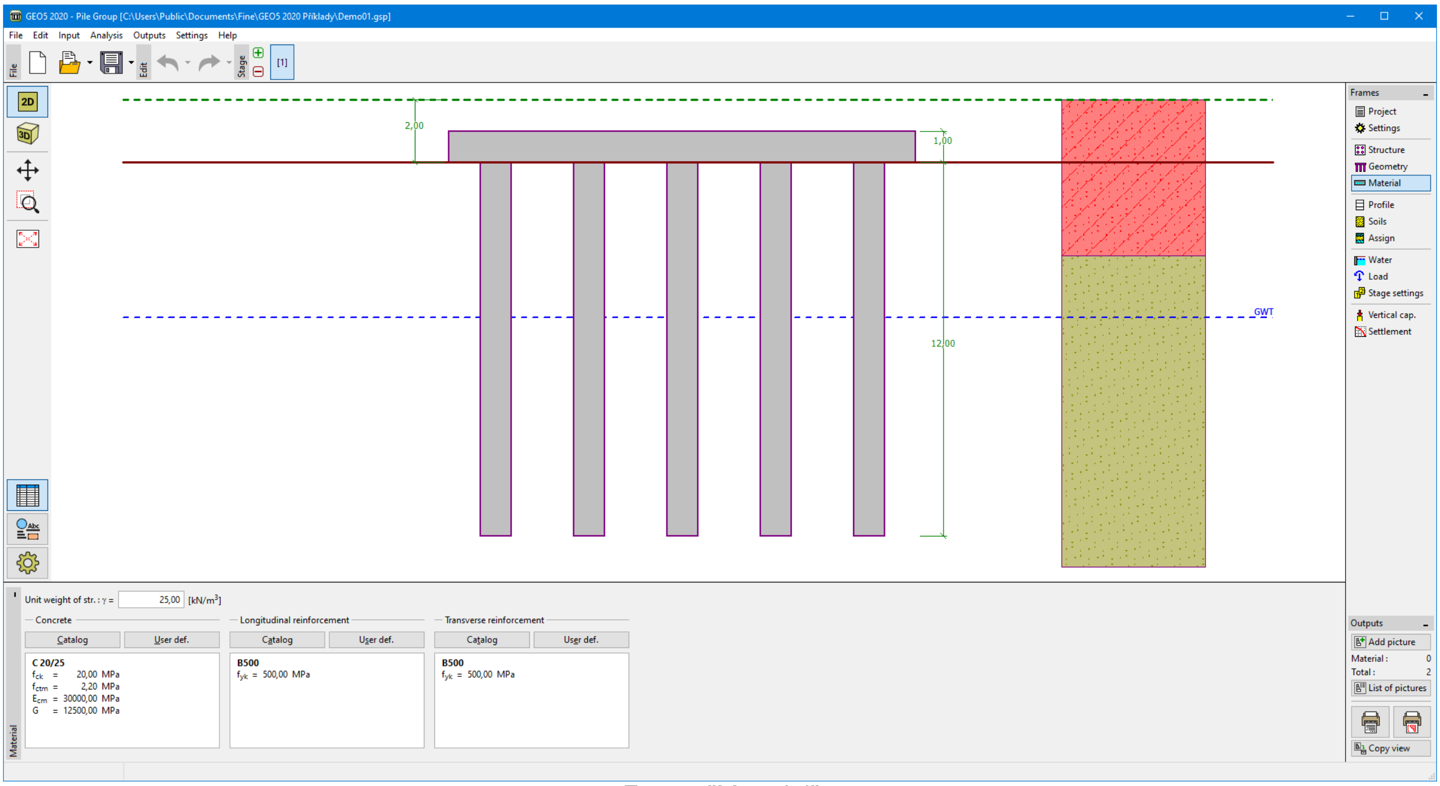
Material

The frame **"Material"** allows us to enter material parameters.

Two options can be used for setting the parameters of the material:

- The "**Catalog**" button opens the "**Catalog of materials**" dialog window, where the required material can be selected.
- The "**User-defined**" button opens the "**Editor of material**", which allows us to input the specification of material parameters manually by the user.

The content of catalogs depends on the selection of relevant standard for the dimensioning of concrete or steel structures set in the "Materials and standards" tab. The input field in the upper part of the frame allows us to specify the unit weight of structure.



Frame "Material"

Load

The frame "**Load**" contains a table with the list of input loads. Adding (editing) a load is performed in the "**New (edit) load**" dialog window. The forces are input according to the sign convention displayed in the right part of the dialog window.

The program also allows us to import a load using the "**Import**" button.

The load applied to a pile group acts at the level of the pile cap upper base at point [0,0]. This point cannot be located outside the pile cap. These values can be easily obtained from the analysis by an arbitrary program that performs static analysis.

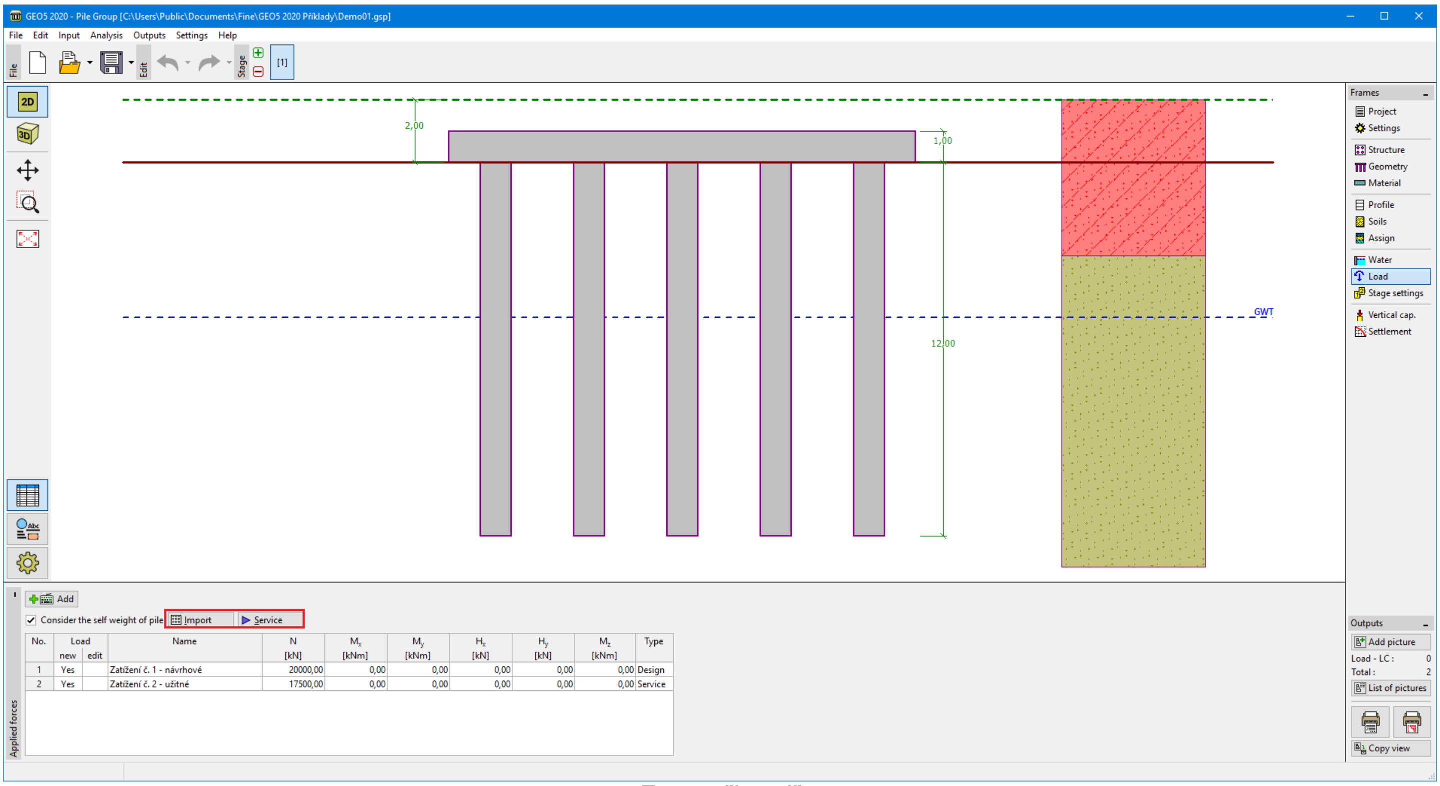
The program automatically back-calculates the **pile cap self-weight** to be added to the already existing load. This program further enables to **consider the self-weight of pile** (using the button in the left part of the desktop)

The pile cap self-weight G_{cap} is provided by:

$$G_{cap} = A_{cap} t \gamma$$

where:

A_{cap}	-	base area of pile cap $[m^2]$
t	-	thickness of pile cap $[m]$
γ	-	unit weight of structure $[kN/m^3]$



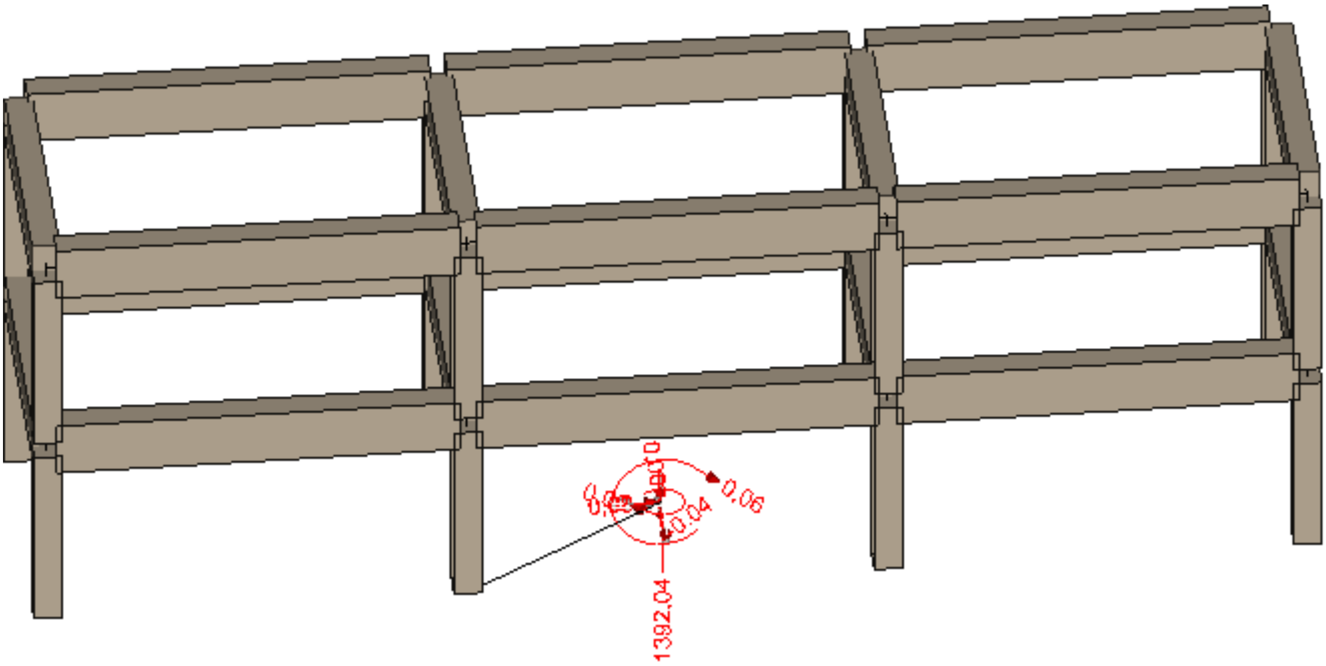
Frame "Load"

Load Acting on a Pile Group

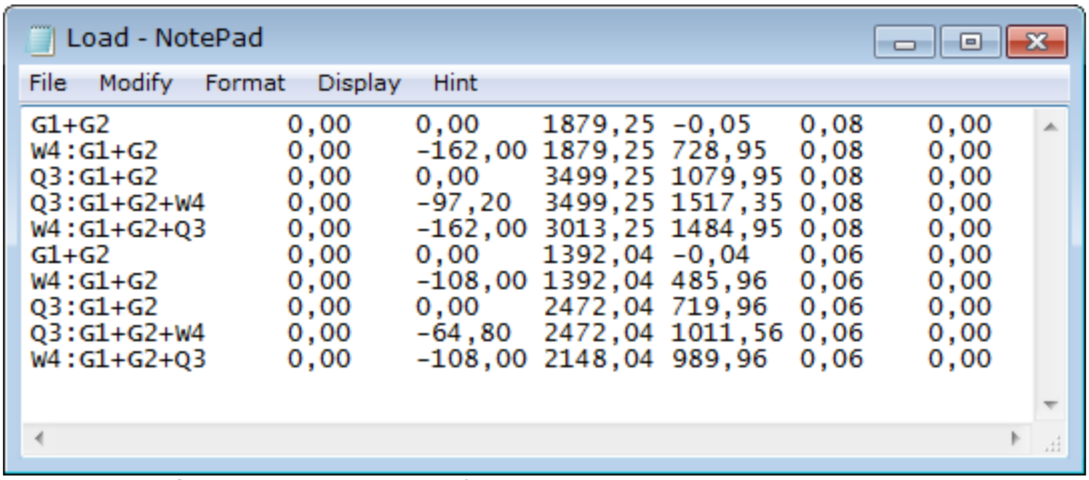
The pile group can be used to found both bridge abutments and arbitrary civil engineering structures. To determine the load acting on a pile group can be rather complicated. The load can be applied at several locations as concentrated load (column), distributed along the line (wall), or over the area. The procedure shows a simple way of determining the load at a given point by adopting an arbitrary static program.

1. We start from the model of a structure in the static program
2. Providing no joint is defined in the **center of the pile cap**, we introduce it.
3. A **fixed support in all 6 directions** is assigned to the joint (fixed, fixed, fixed, fixed, fixed, fixed)
4. If the joint is not found on the pile cap (beam model), we connect it with the actual structure (The stiffness should correspond to other elements)
5. Apart from the new joint, we remove **all boundary conditions** on the analyzed model
6. Perform analysis - the **joint reactions correspond to the load**, which is input into the "**Pile group**" program - the function for importing the load can also be utilized

Note: **internal hinges** found in the structure must be changed to fixed-end supports for the static program to find a solution.

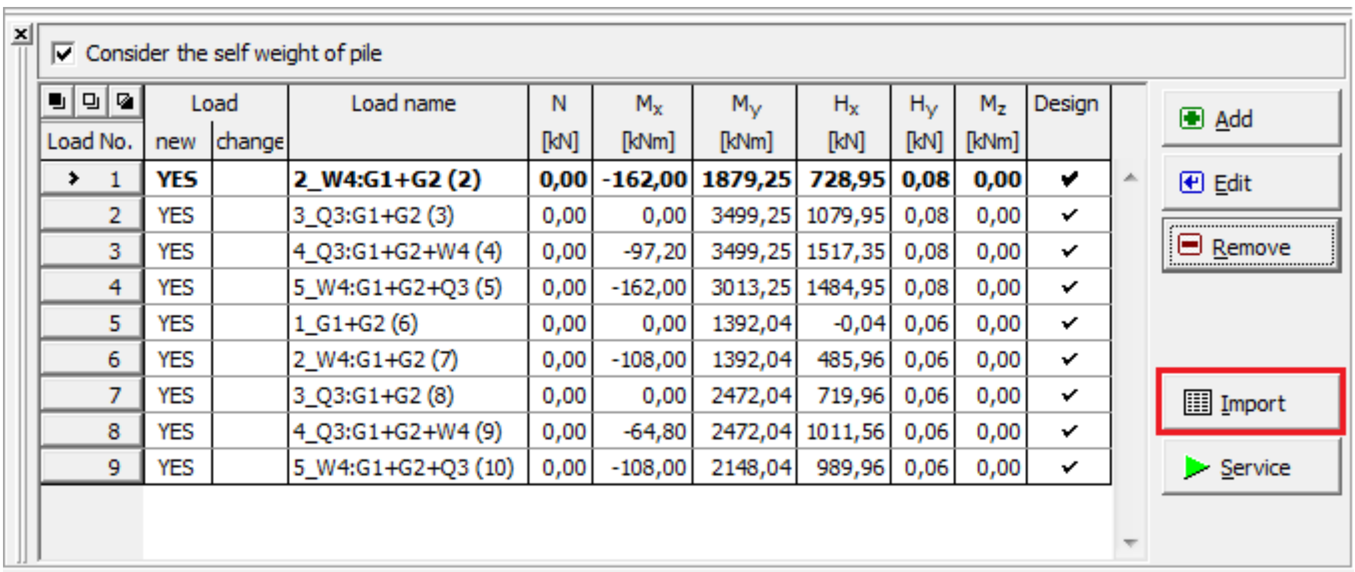


Structure with a fixed support



G1+G2	0,00	0,00	1879,25	-0,05	0,08	0,00
W4:G1+G2	0,00	-162,00	1879,25	728,95	0,08	0,00
Q3:G1+G2	0,00	0,00	3499,25	1079,95	0,08	0,00
Q3:G1+G2+W4	0,00	-97,20	3499,25	1517,35	0,08	0,00
W4:G1+G2+Q3	0,00	-162,00	3013,25	1484,95	0,08	0,00
G1+G2	0,00	0,00	1392,04	-0,04	0,06	0,00
W4:G1+G2	0,00	-108,00	1392,04	485,96	0,06	0,00
Q3:G1+G2	0,00	0,00	2472,04	719,96	0,06	0,00
Q3:G1+G2+W4	0,00	-64,80	2472,04	1011,56	0,06	0,00
W4:G1+G2+Q3	0,00	-108,00	2148,04	989,96	0,06	0,00

Support reactions ready for import into the "Pile group" program



Load No.	Load new	Load change	Load name	N [kN]	M_x [kNm]	M_y [kNm]	H_x [kN]	H_y [kN]	M_z [kNm]	Design
1	YES		2_W4:G1+G2 (2)	0,00	-162,00	1879,25	728,95	0,08	0,00	✔
2	YES		3_Q3:G1+G2 (3)	0,00	0,00	3499,25	1079,95	0,08	0,00	✔
3	YES		4_Q3:G1+G2+W4 (4)	0,00	-97,20	3499,25	1517,35	0,08	0,00	✔
4	YES		5_W4:G1+G2+Q3 (5)	0,00	-162,00	3013,25	1484,95	0,08	0,00	✔
5	YES		1_G1+G2 (6)	0,00	0,00	1392,04	-0,04	0,06	0,00	✔
6	YES		2_W4:G1+G2 (7)	0,00	-108,00	1392,04	485,96	0,06	0,00	✔
7	YES		3_Q3:G1+G2 (8)	0,00	0,00	2472,04	719,96	0,06	0,00	✔
8	YES		4_Q3:G1+G2+W4 (9)	0,00	-64,80	2472,04	1011,56	0,06	0,00	✔
9	YES		5_W4:G1+G2+Q3 (10)	0,00	-108,00	2148,04	989,96	0,06	0,00	✔

Load imported into "Pile group" in the frame "Load"

Profile

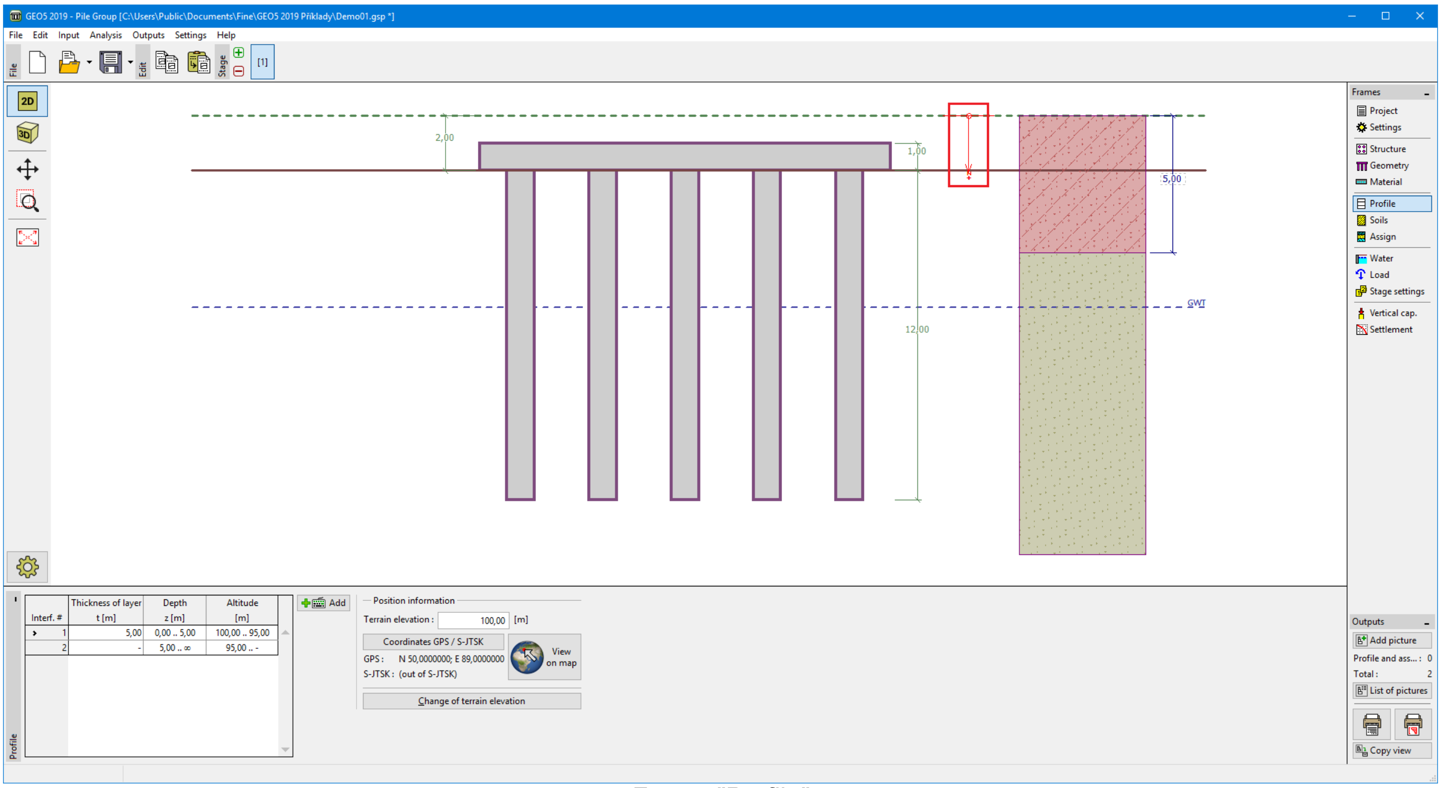
The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.



Frame "Profile"

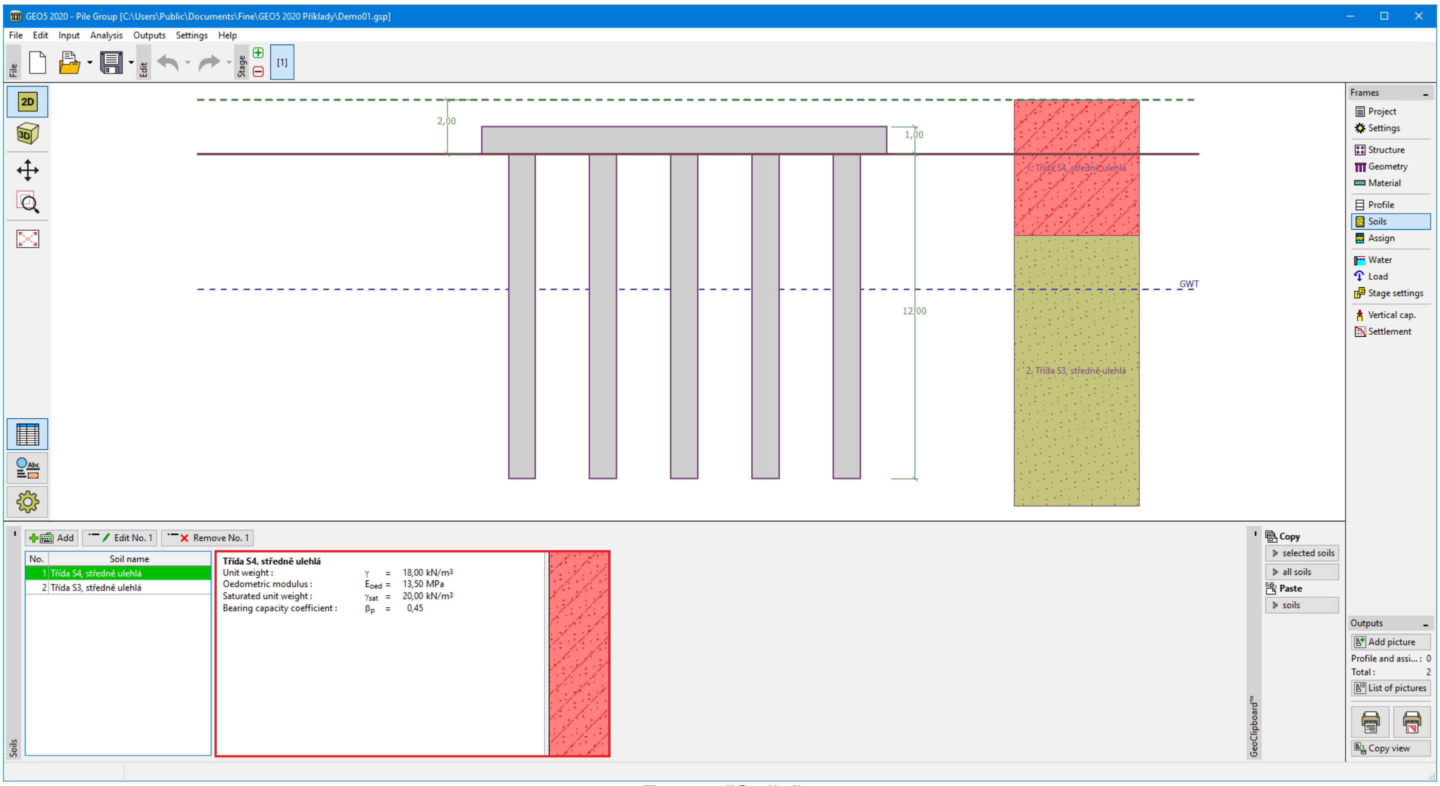
Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Uplift pressure", "Settlement", "Modulus of subsoil reaction". These parameters depend on the type of soil specified in the frame "Settings" and the theory of analysis specified in the "Pile Group" tab.

Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

Basic Data

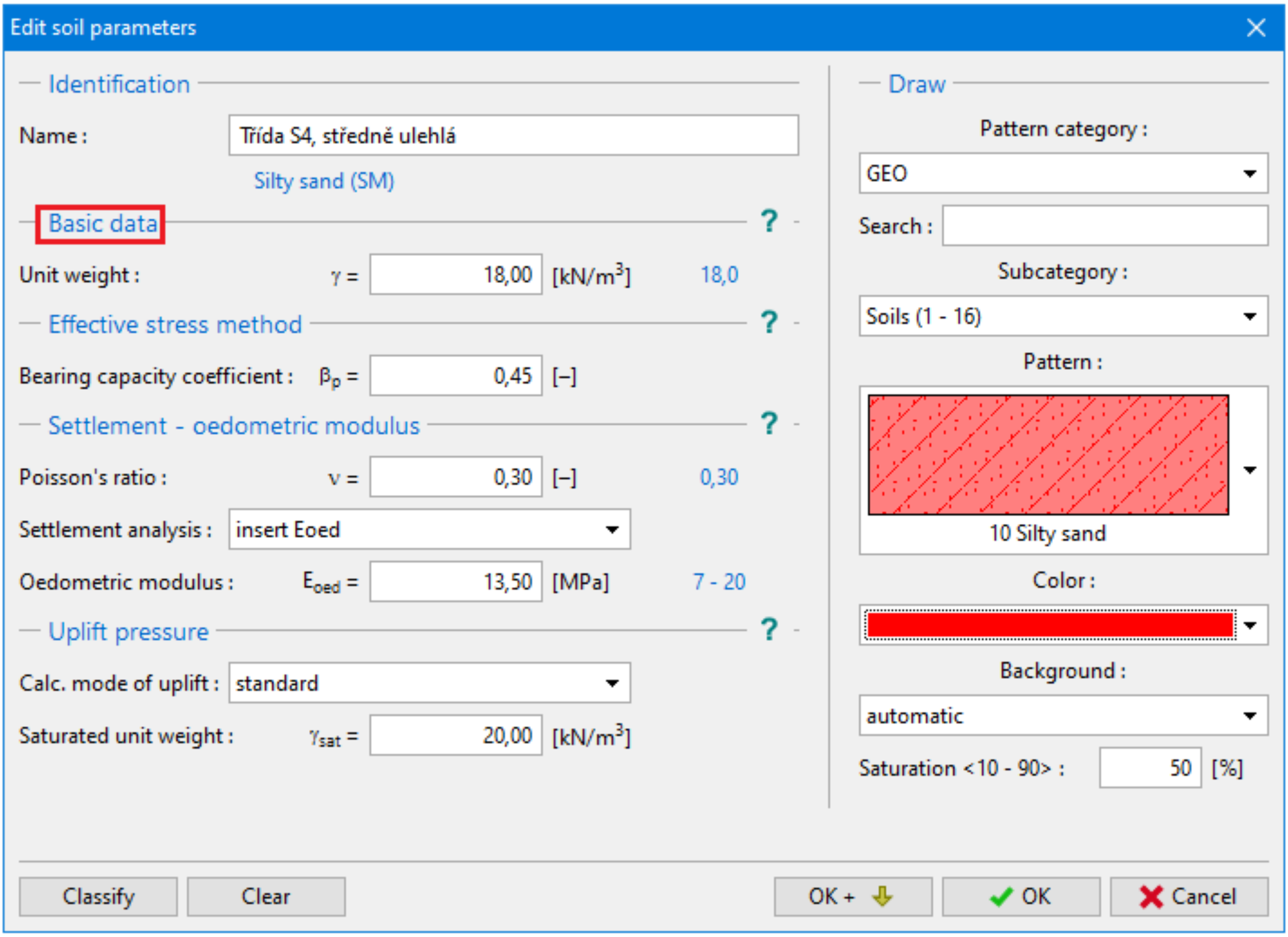
This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction, and cohesion**. The particular values are obtained from a geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit the built-in database of soils, which contains values of the selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually.

The calculation of "Pile Group" differs according to the type of subsoil:

- **cohesionless soil: effective** parameters of shear strength of soil c_{ef}, φ_{ef} are used commonly.
- **cohesive soil**: in program is defined only the value of **total** shear strength of soil c_u , which determines vertical bearing capacity of pile group (or earth block).

Additional parameters are input depending on the settings in the frame "Settings" and in the "Pile Group" tab.

The associated theory is described in detail in the chapter "Pile Group".

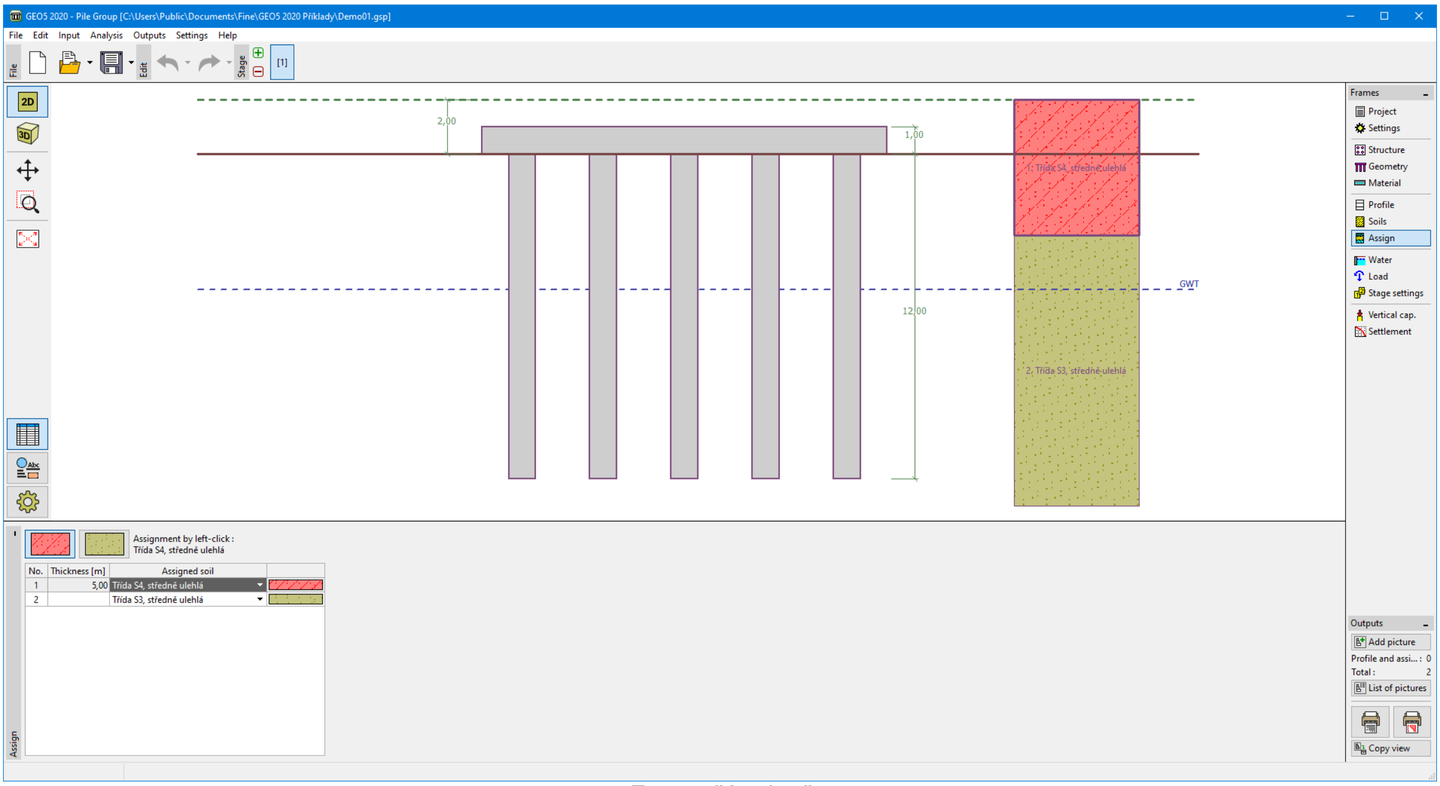


Dialog window "Add new soils" - "Basic data"

Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail herein.



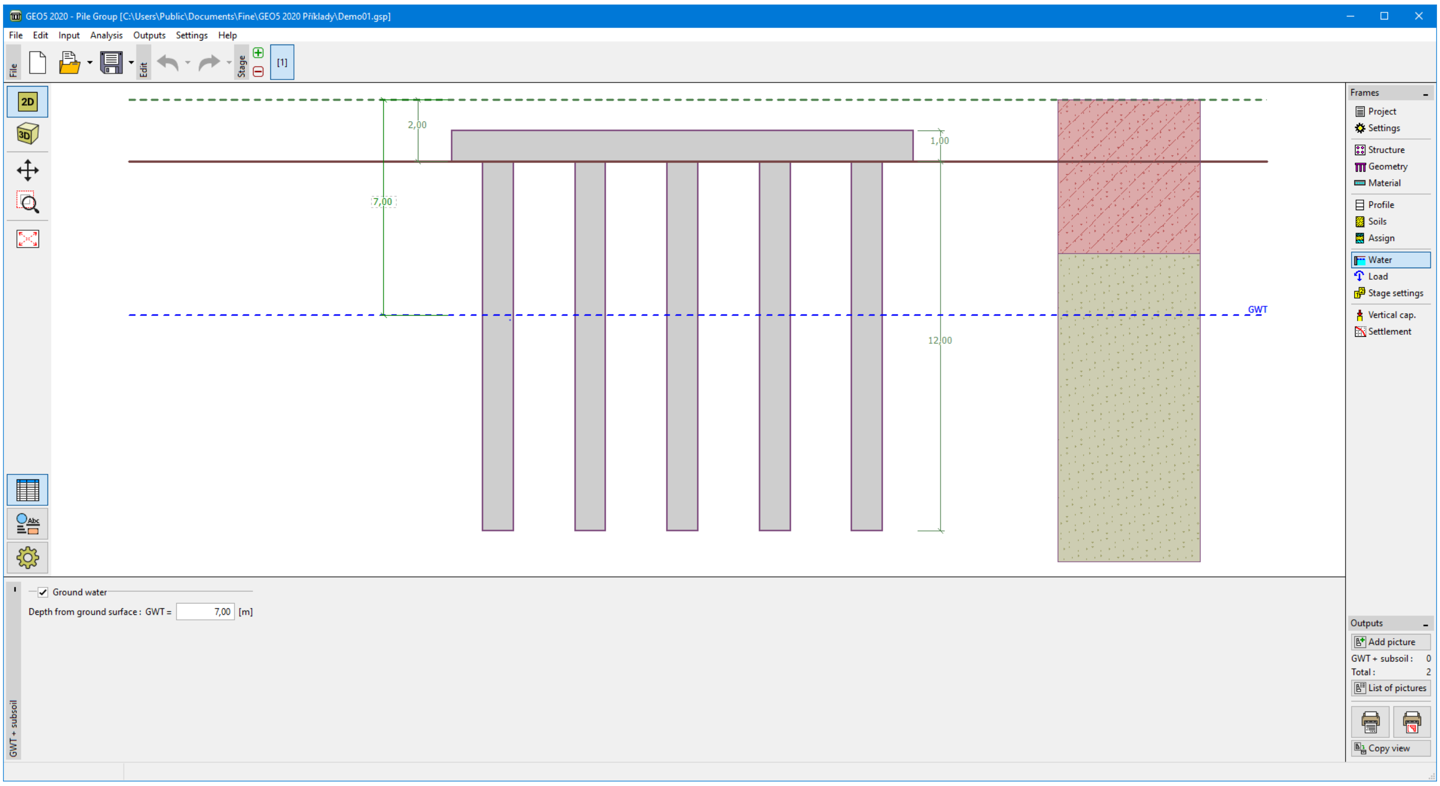
Frame "Assign"

Water

The "**Water**" frame serves to specify the **depth of groundwater table**.

The value can be edited either in the frame by inserting the value into the input field, or on the desktop with the help of active dimensions.

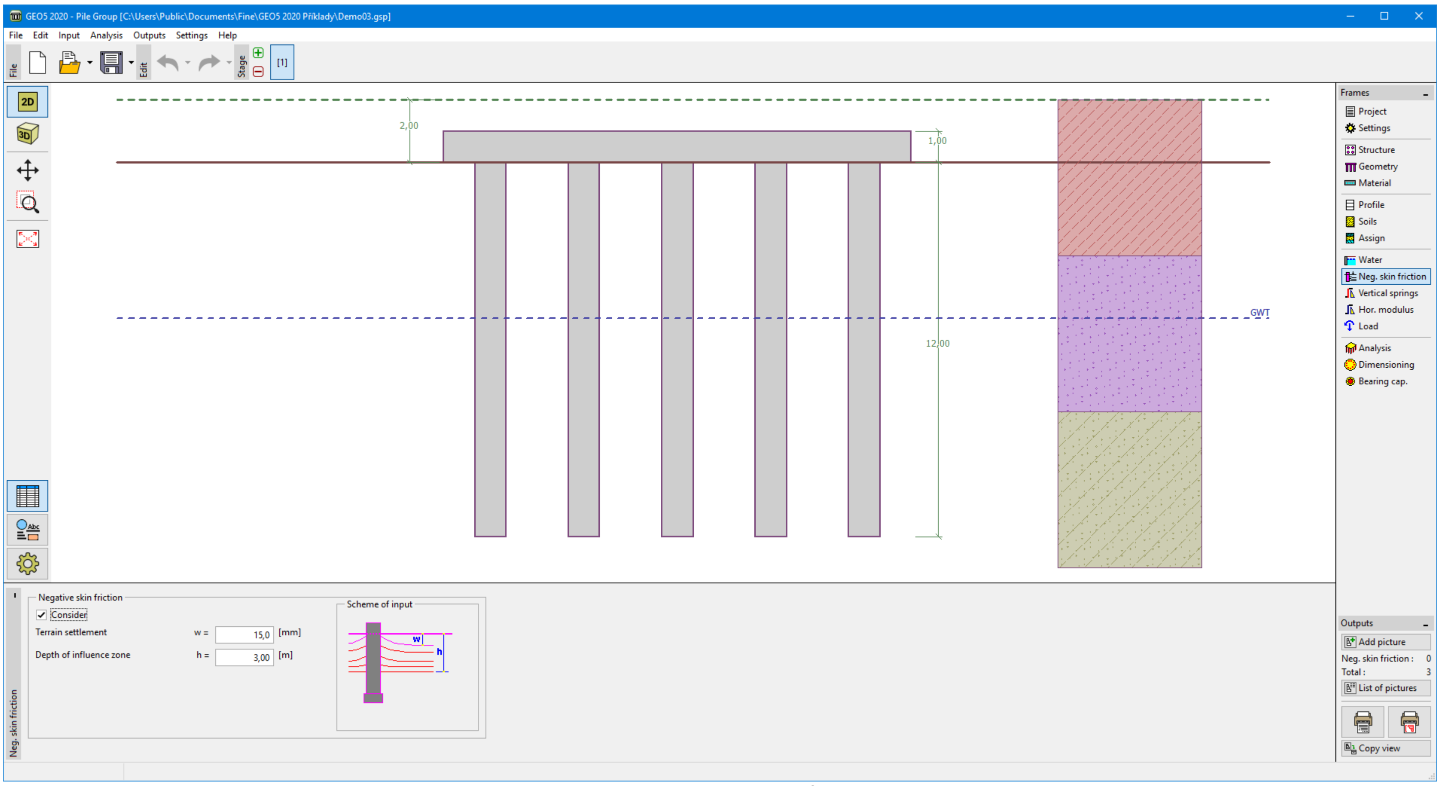
The **GWT** changes the geostatic stress in the soil profile.



Frame "Water"

Negative Skin Friction

The "**Negative skin friction**" frame serves to specify the settlement of surrounding terrain and the depth of influence zone. For more information on the influence of negative skin friction, the user is referred to theoretical section.



Frame "Negative skin friction"

Vertical Springs

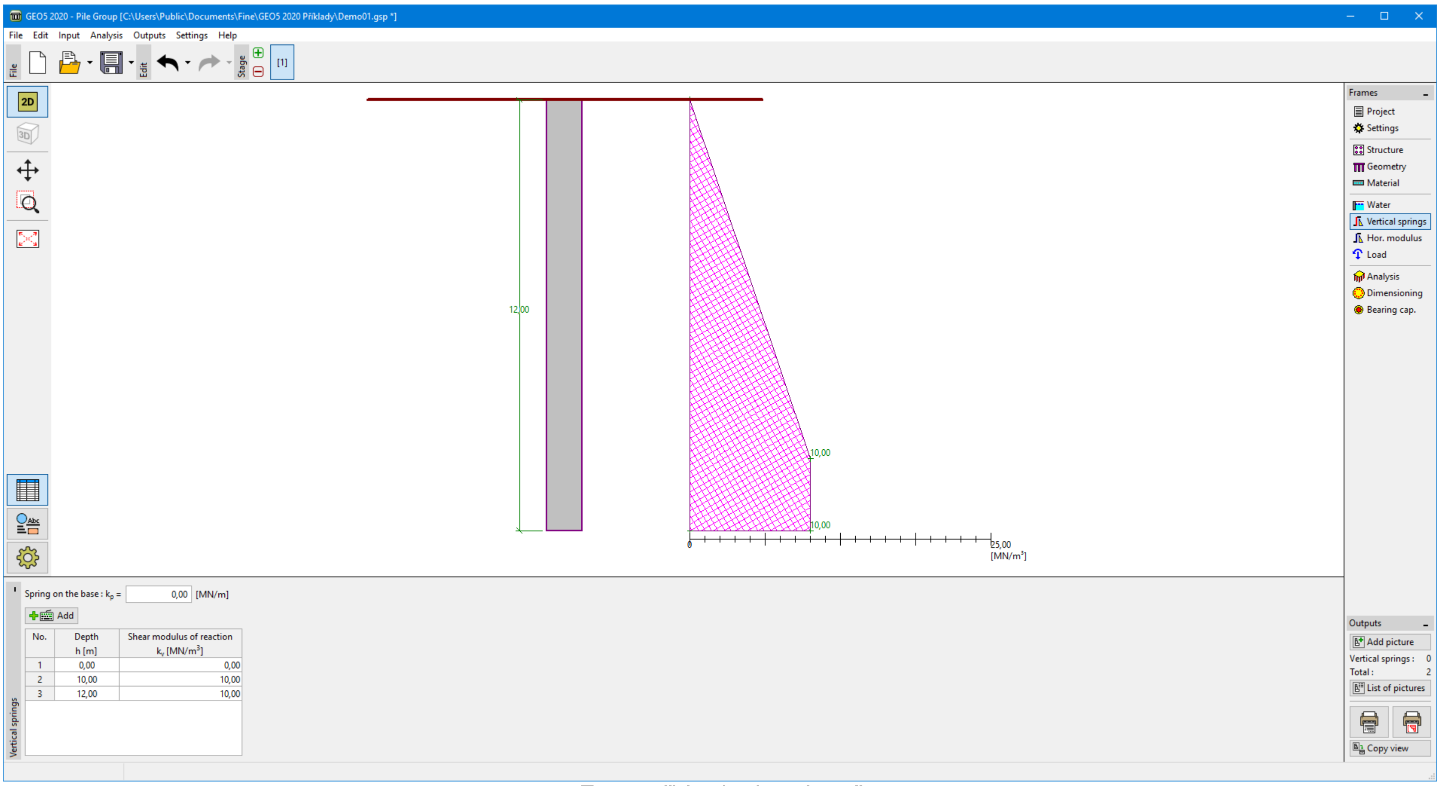
The "**Vertical springs**" frame is active only when analyzing a **floating pile**. The input springs are displayed in the table.

The option "**input the stiffness of springs**" requires inputting:

- spring at the pile base [MN/m]
- shear modulus of subsoil reaction along the pile [MN/m^3].

The input values are the same for all piles. In the analysis, the vertical stiffnesses of inner and outer piles in the group are reduced by particular coefficients.

The option "**Compute the stiffness of springs from soil parameters**" requires inputting the typical load to obtain the spring stiffnesses from calculation.

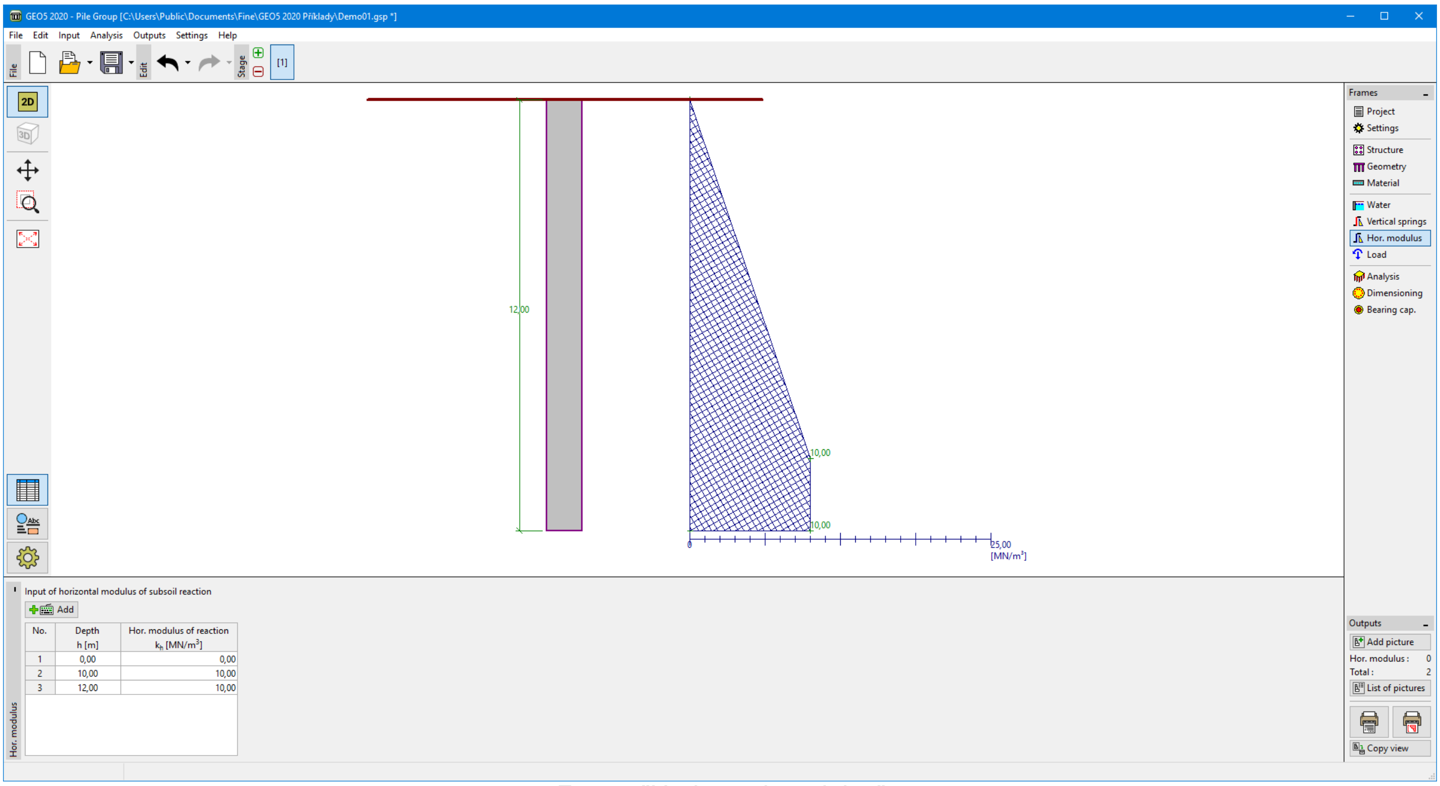


Frame "Vertical springs"

Horizontal Modulus

The"**Horizontal modulus**" frame serves to input the horizontal modulus of subsoil reaction characterizing the pile response in the horizontal direction.

The values of the modulus of subsoil reaction at a given depth of the profile are displayed in the table.

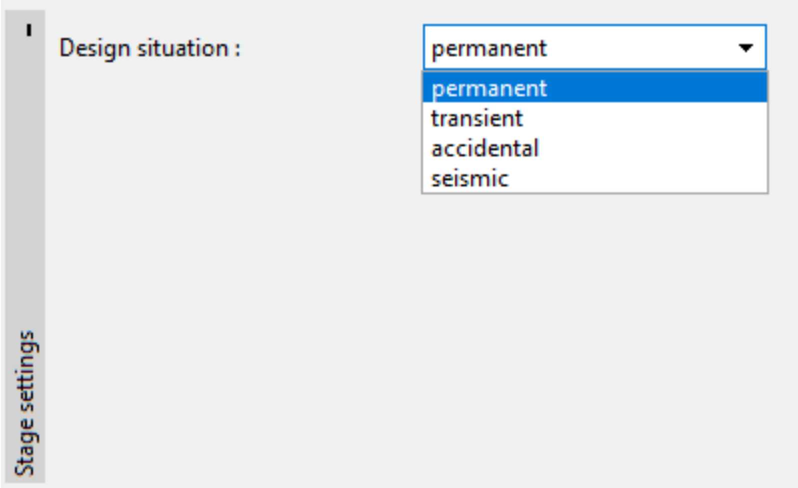


Frame "Horizontal modulus"

Stage Settings

The "**Stage settings**" frame serves to input settings valid for a given construction stage.

The selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.



Frame "Stage settings"

Vertical Bearing Capacity - Analytical Solution

The frame "**Vertical bearing capacity**" serves to verify the pile group vertical bearing capacity. Several analyses can be carried out in the frame.

The verification analysis can be carried out for individual loads or the program identifies **the most critical one** (can be selected from the combo list).

The analysis is performed according to the theory set in the frame "Settings" (analytical solution):

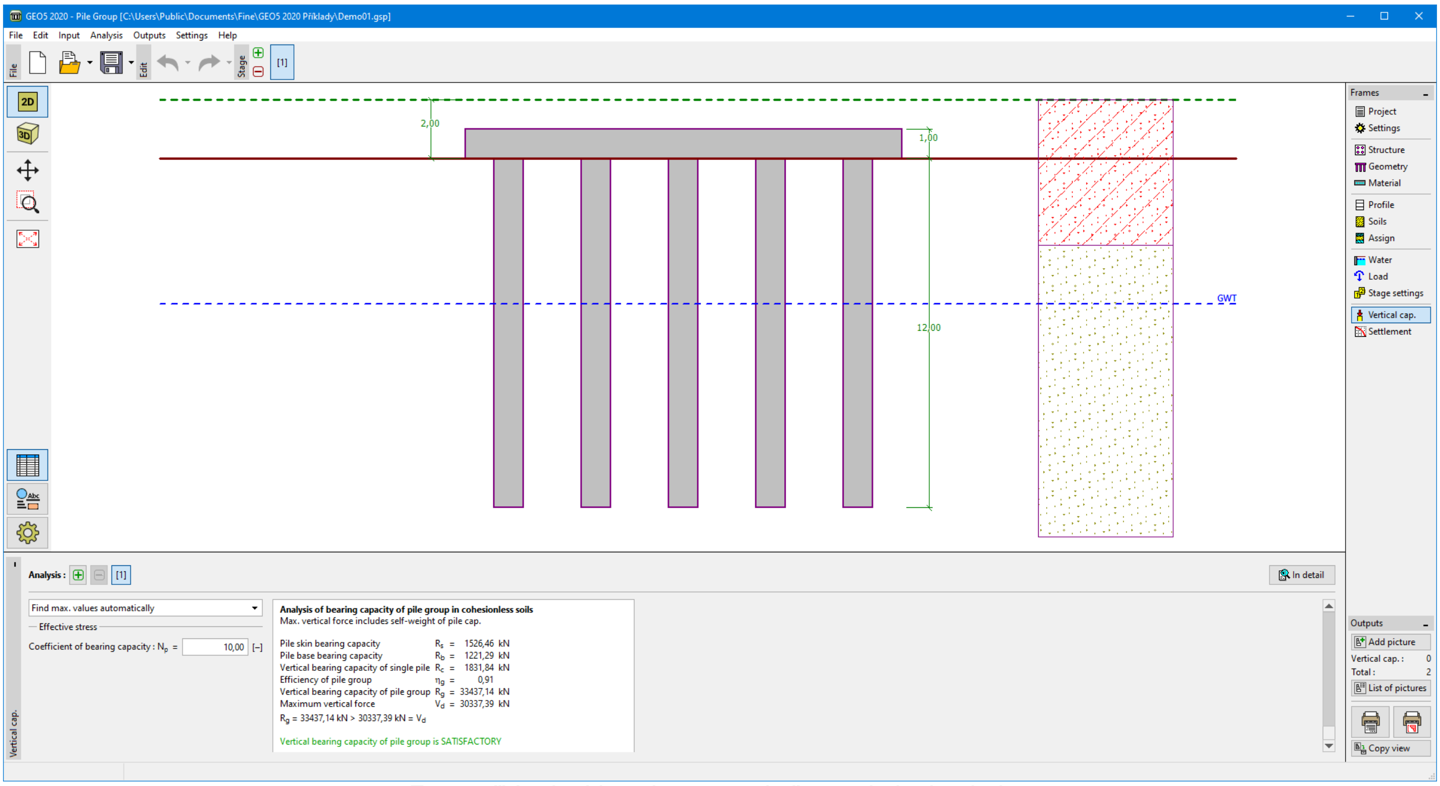
- for cohesive soil (undrained conditions) - analysis of bearing capacity of an earth block according to FHWA
- for cohesionless soil (drained conditions) - NAVFAC DM 7.2, Effective stress, CSN 73 1002

The parameters needed for the pile group analysis are introduced for individual methods in left part of the frame.

The verification analysis is carried out according to the verification methodology selected in the "Pile Group" tab (factors of safety, theory of limit states, EN 1997-1).

The "**In detail**" button opens the dialog window containing detailed listing of the verification results.

The analysis results are displayed in the right part of the desktop.

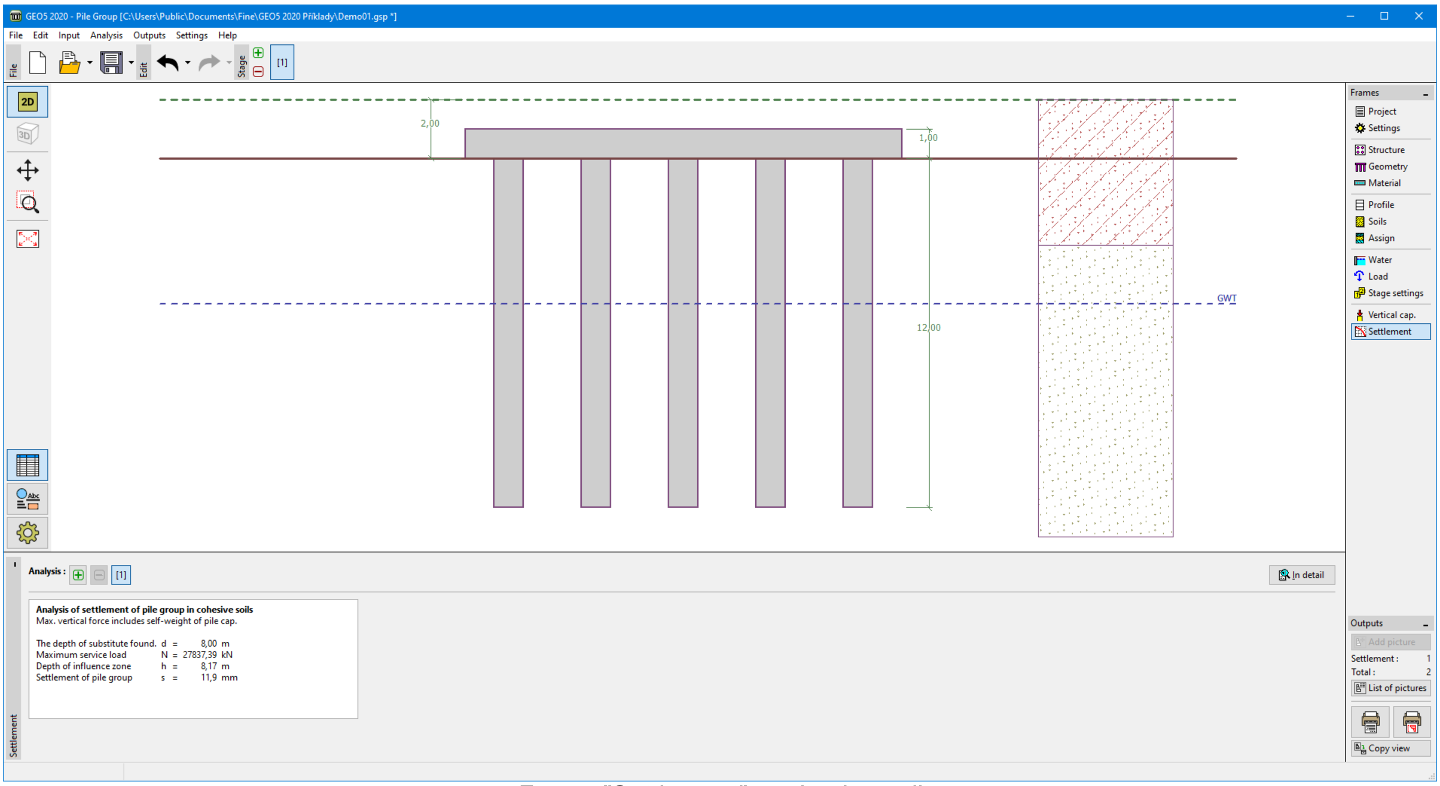


Frame "Vertical bearing capacity" - analytical solution

Settlement - Cohesive Soil

This frame serves to calculate the pile group **settlement** for cohesive soils. The analysis of settlement is performed according to the selected theory set in the "Pile Group" tab. The analysis results are displayed in the right part of the desktop.

The "**In detail**" button opens the dialog window that contains a detailed description of the results of the verification analysis.



Frame "Settlement" - cohesive soil

Settlement - Cohesionless Soil (Load-Settlement Curve)

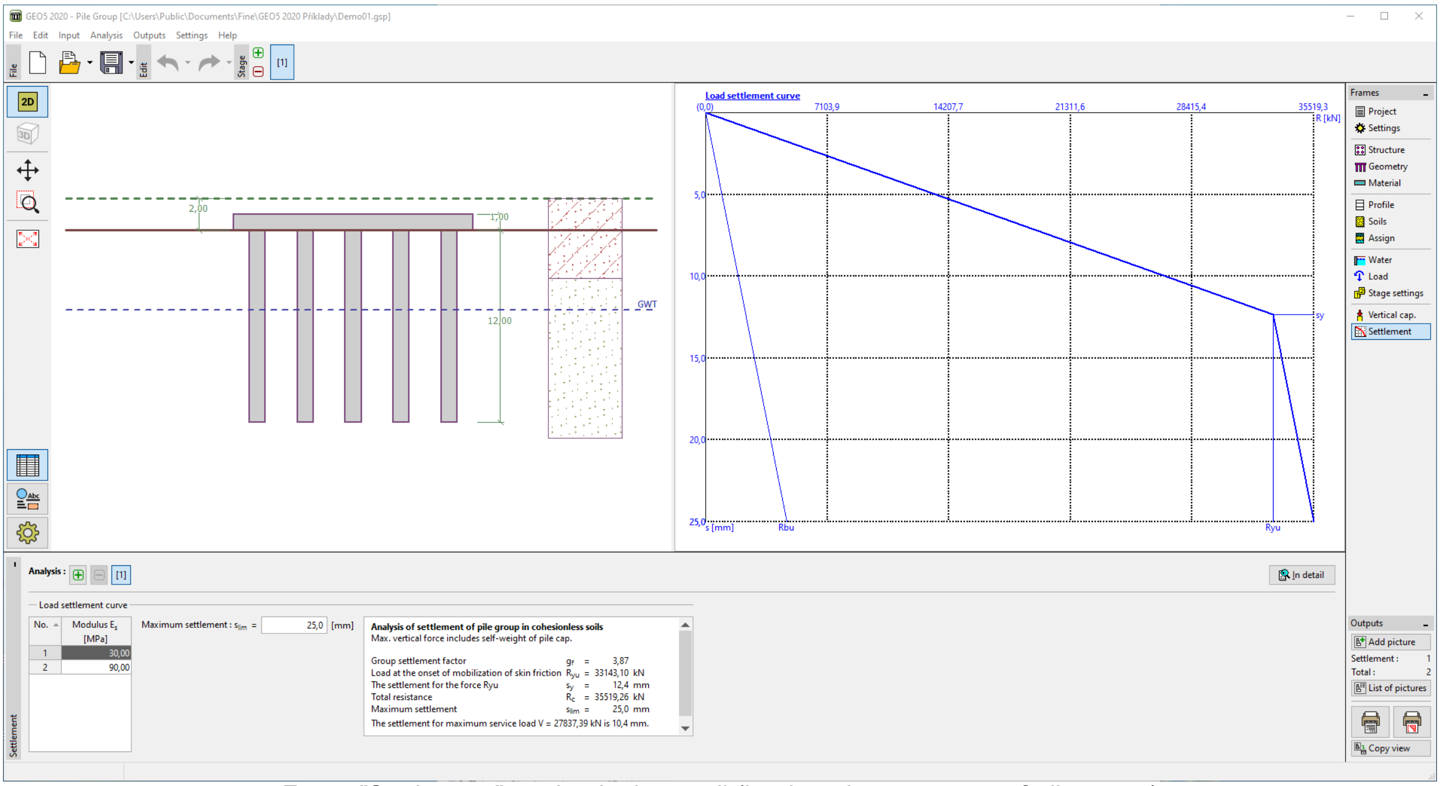
The frame "**Settlement**" displays a linear load-settlement curve for the settlement of a pile group in a cohesionless soil. Several analyses can be carried out in the frame.

The load-settlement curve of the pile group is always calculated for the **input limit settlement**.

The table in the left bottom part of the frame directly allows us to specify values of the secant modulus of soil for the relevant layers of soil.

The **analysis results** are displayed in the right part of the frame. The "**In detail**" button opens the dialog window that contains a detailed description of the results of the verification analysis.

The analysis results (load-settlement curve of pile group) are displayed in the right part of the desktop.



Frame"Settlement" - cohesionless soil (load-settlement curve of pile group)

Analysis - Spring Method

This frame serves to analyze a group of piles using the spring method. The analysis is run by pressing the "**Analysis**" button.

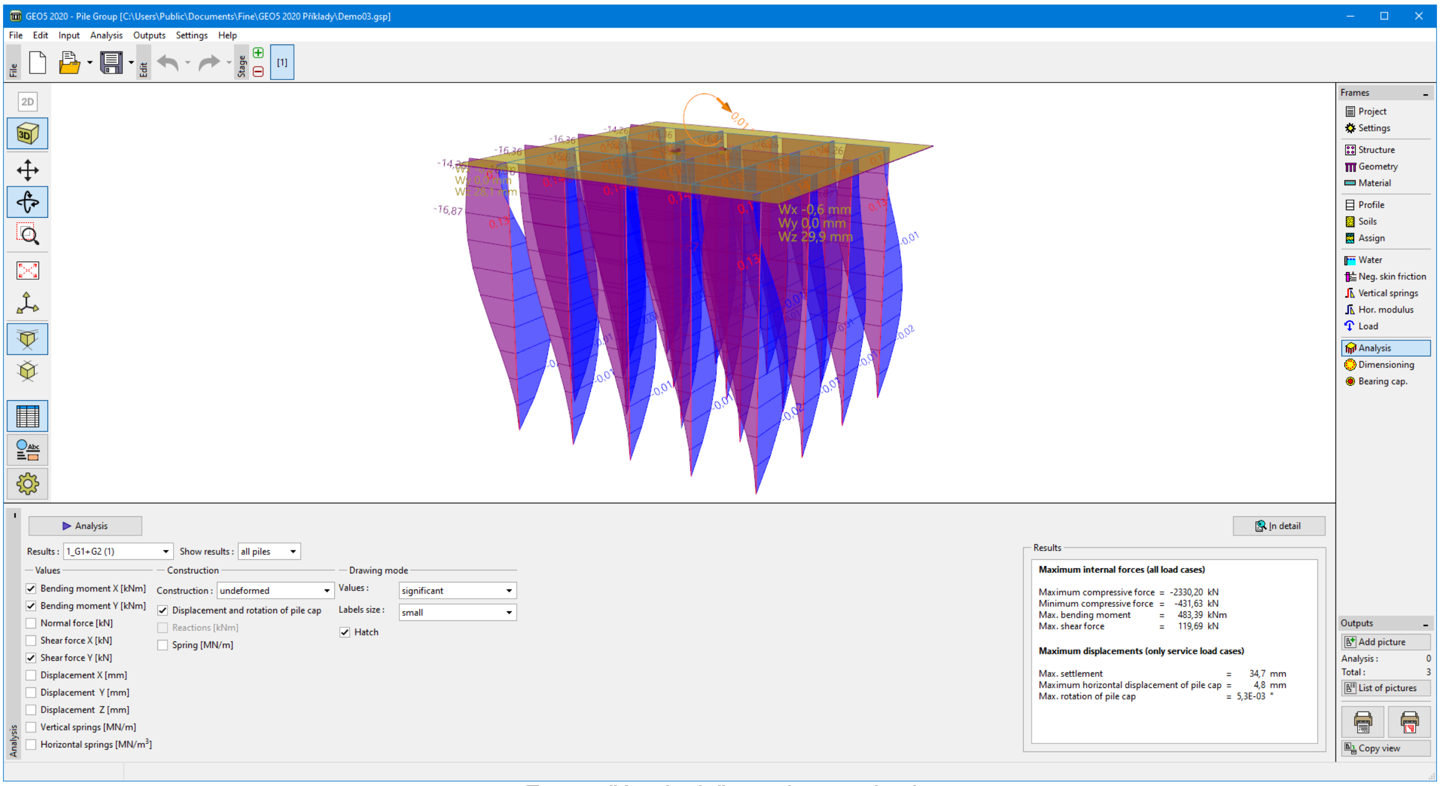
Upon performing the analysis, the results appear in the right part of the frame ("**Results**") providing information about the **maximum internal forces**, **displacements and rotations of a structure**. Displacements of the structure shown in the window are determined for service loads. The "**In detail**" button opens the dialog window that contains a detailed description of the results of the verification analysis.

The left part of the frame allows for defining the way of plotting the results on the screen:

- **Results** - the results can be displayed for individual load cases or for their envelope
- **Show results** - the results can be displayed for all piles or for individual piles only
- **Variables** - visualization of values of individual variable (moments, normal and shear forces, displacements, springs)
- **Structure** - allows for plotting a deformed structure (only undeformed structure can be displayed for envelopes of load cases), next it is possible to show the magnitudes of pile cap deflections, reactions and magnitudes of springs at the pile base.
- **Drawing mode** - defines the style of describing the results

The displayed results can be added to the "List of pictures" at any time and used in the analysis protocol.

Rotation, zooming and illumination of a structure can be adjusted with the help of the "Visualization" toolbar. Visualization of results can be adjusted in the "Drawing Settings" frame.



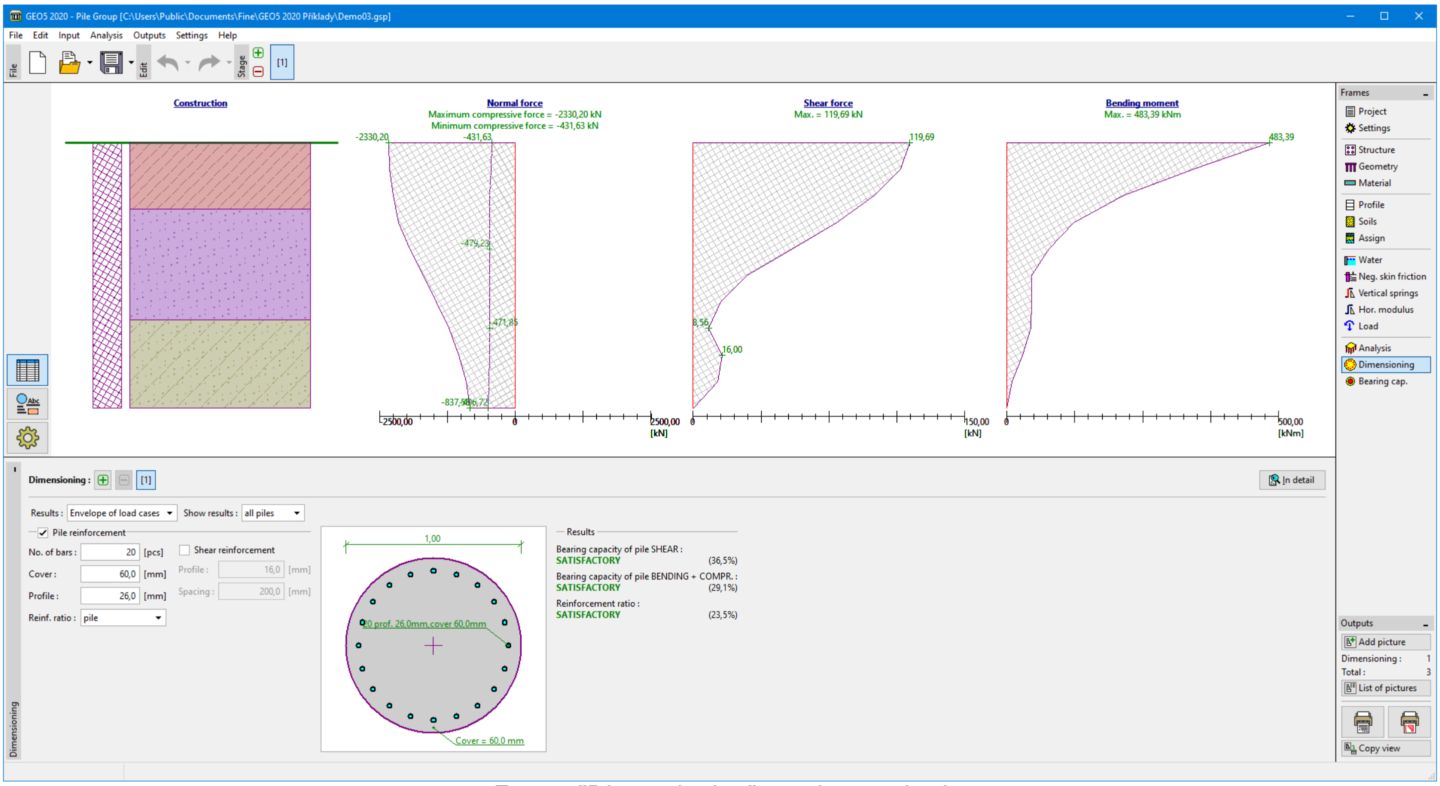
Frame "Analysis" - spring method

Dimensioning

The frame "**Dimensioning**" allows the programs to adopt the results obtained from the calculations performed in the frame "Analysis". Both the envelope of loads and individual load cases can be selected. The reinforcement can be design for the selected pile or the same reinforcement for all piles in the group can be assumed.

The verification analysis of a steel-reinforced concrete pile is carried out according to the standard selected in the "Materials and standards" tab.

Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Dimensioning" - spring method

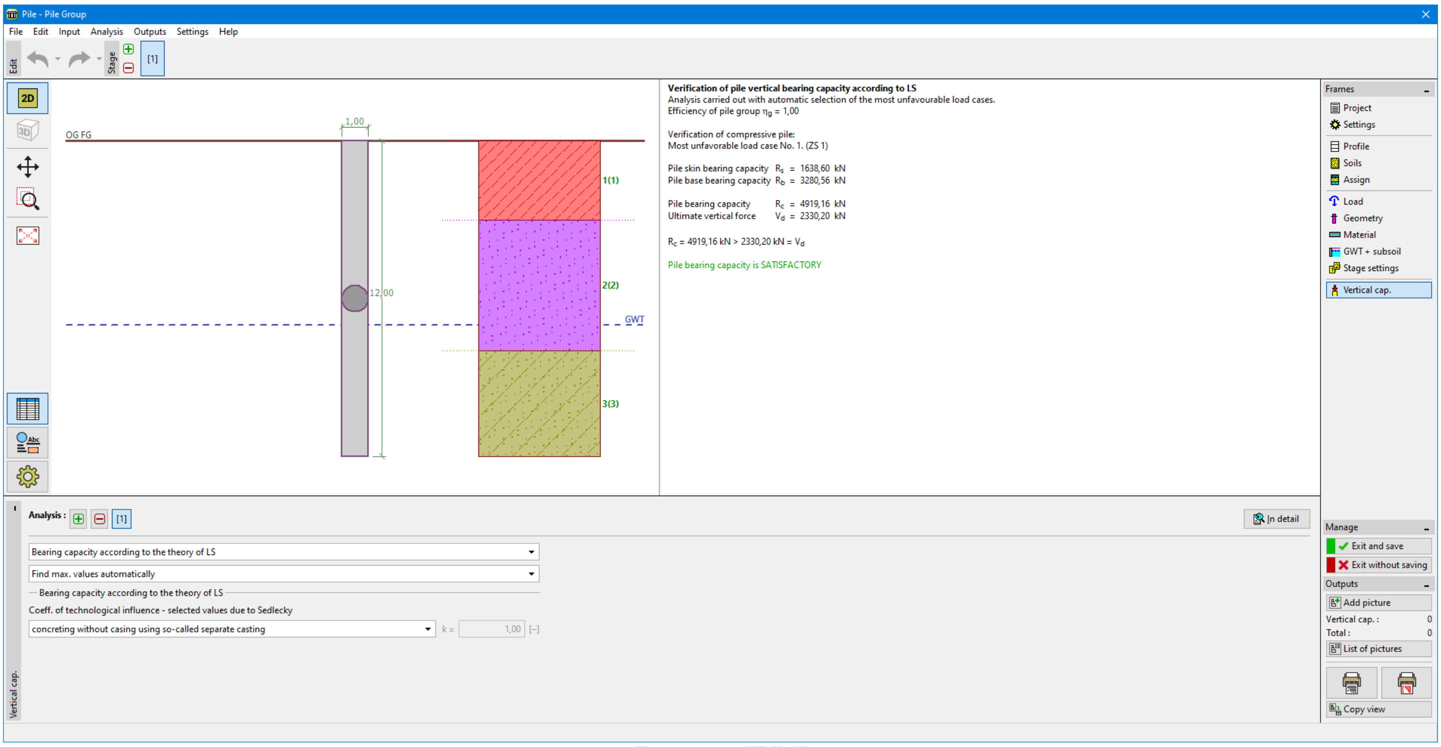
Bearing capacity

The program "**Pile**" or "**Micropile**" can be launched from the frame "**Vertical cap.**" according to the selected analysis type in the frame "Settings". (spring method, spring method - micropiles).

Pile

The "Bearing capacity" button launches the **program "Pile"**, and all necessary data are transferred into it. In this program, it is possible to verify the vertical capacity of the pile. If the program "Pile" is not installed, the button is not available.

After leaving the program by the "**Exit and save**" button, the results are transferred into the output protocol in the program "**Pile Group**".

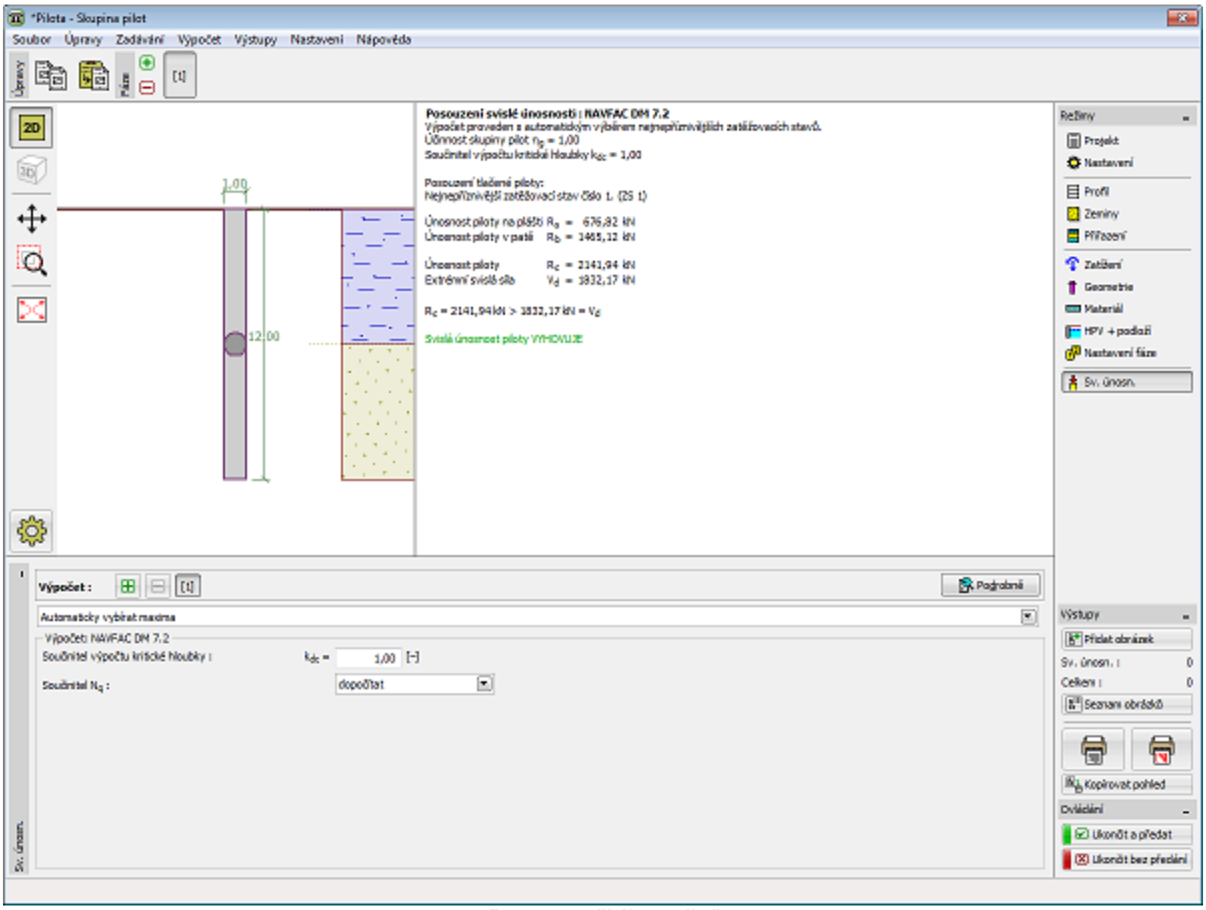


Program "Pile"

Micropile

The "Bearing capacity" button launches the **program** "**Micropile**", and all necessary data are transferred into it. In this program, it is possible to verify the vertical capacity of the pile. If the program "**Micropile**" is not installed, the button is not available.

After leaving the program by the "**Exit and save**" button, the results are transferred into the output protocol in the program "**Pile Group**".



Program "Micropile"

Program Micropile

This program is used for verification of steel tube micropiles. When calculating the micropile bearing capacity, the program verifies both the root and the shaft.

The help in the program "Micropile" includes the following topics:

- The input of data into individual frames:

Project	Settings	Profile	Soils	Geometry	Material	Assign
Load	Water	Standard Penetration Tests (SPT)	Pressuremeter Tests	Verification of Cross-section	Root Verification	

- Standards and analysis methods

- Theory for analysis in the program "**Micropile**":

Micropile	Field Testing	Steel Cross Sections Verification

- Outputs

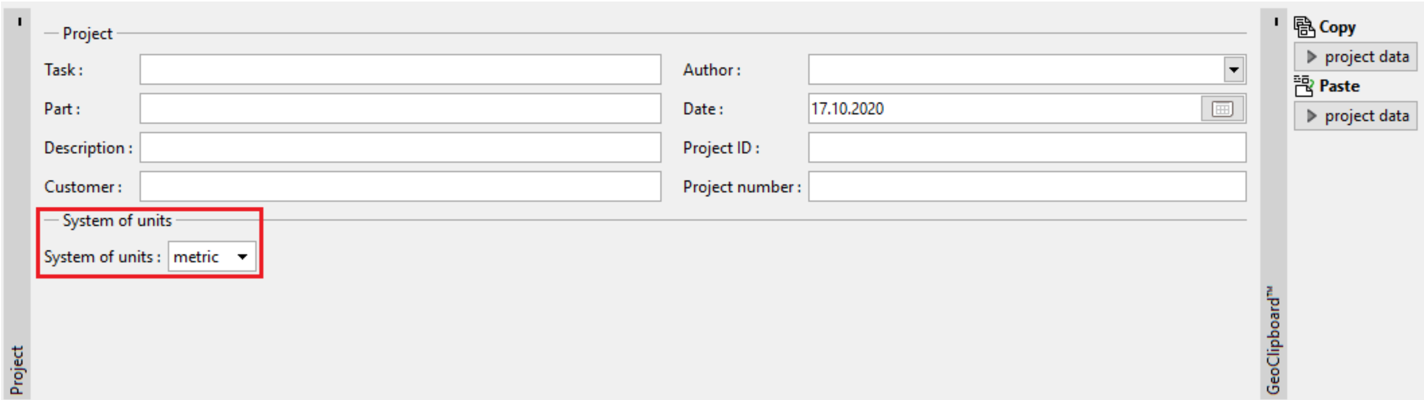
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The frame **"Project"** is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The frame "**Settings**" allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

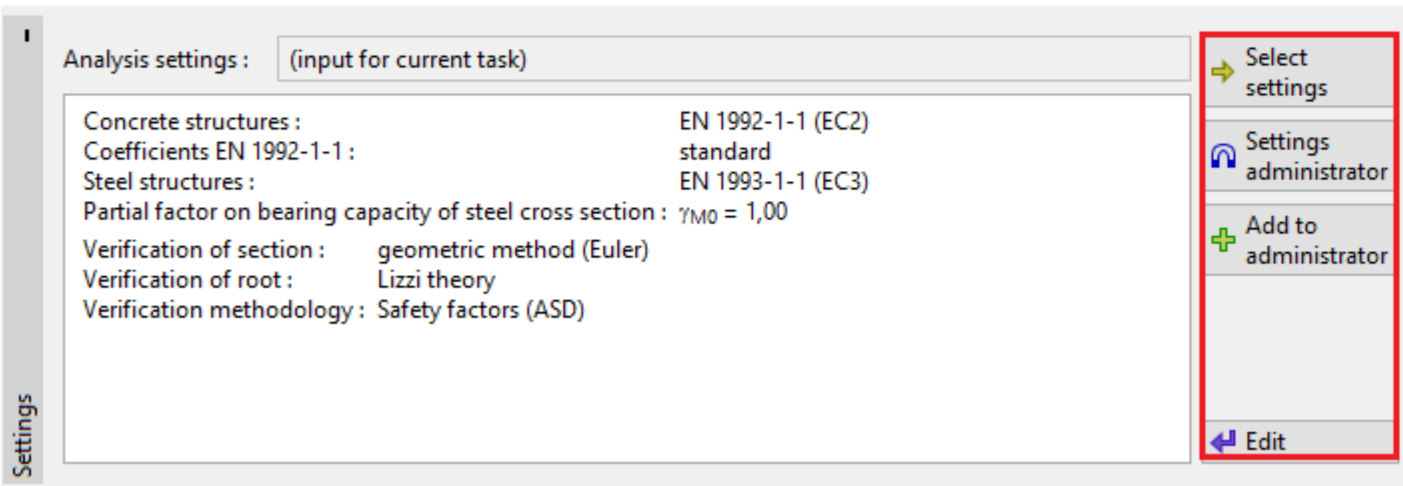
The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

Settings of analysis parameters are performed in the "Micropiles" tab.



Frame "Settings"

Profile

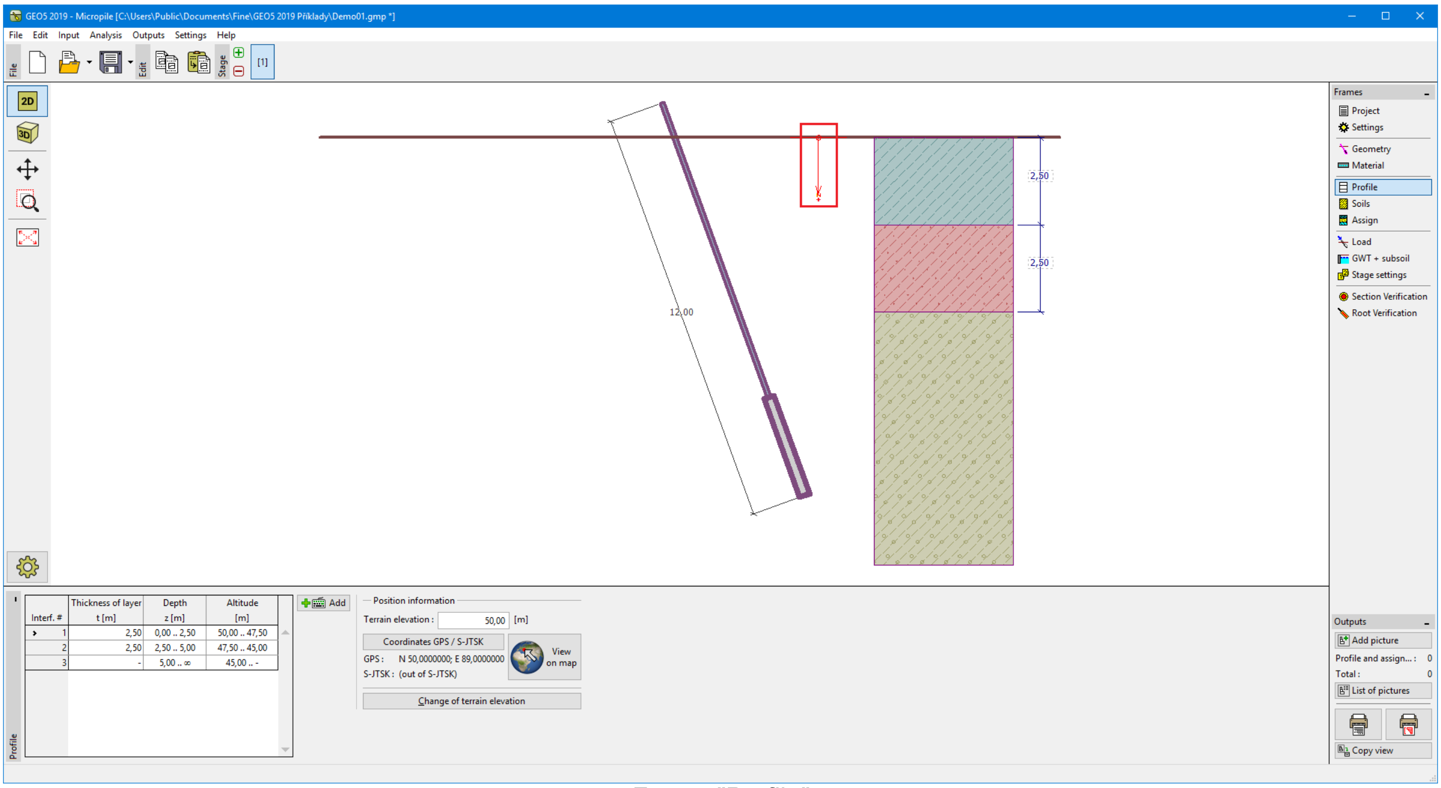
The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.



Frame "Profile"

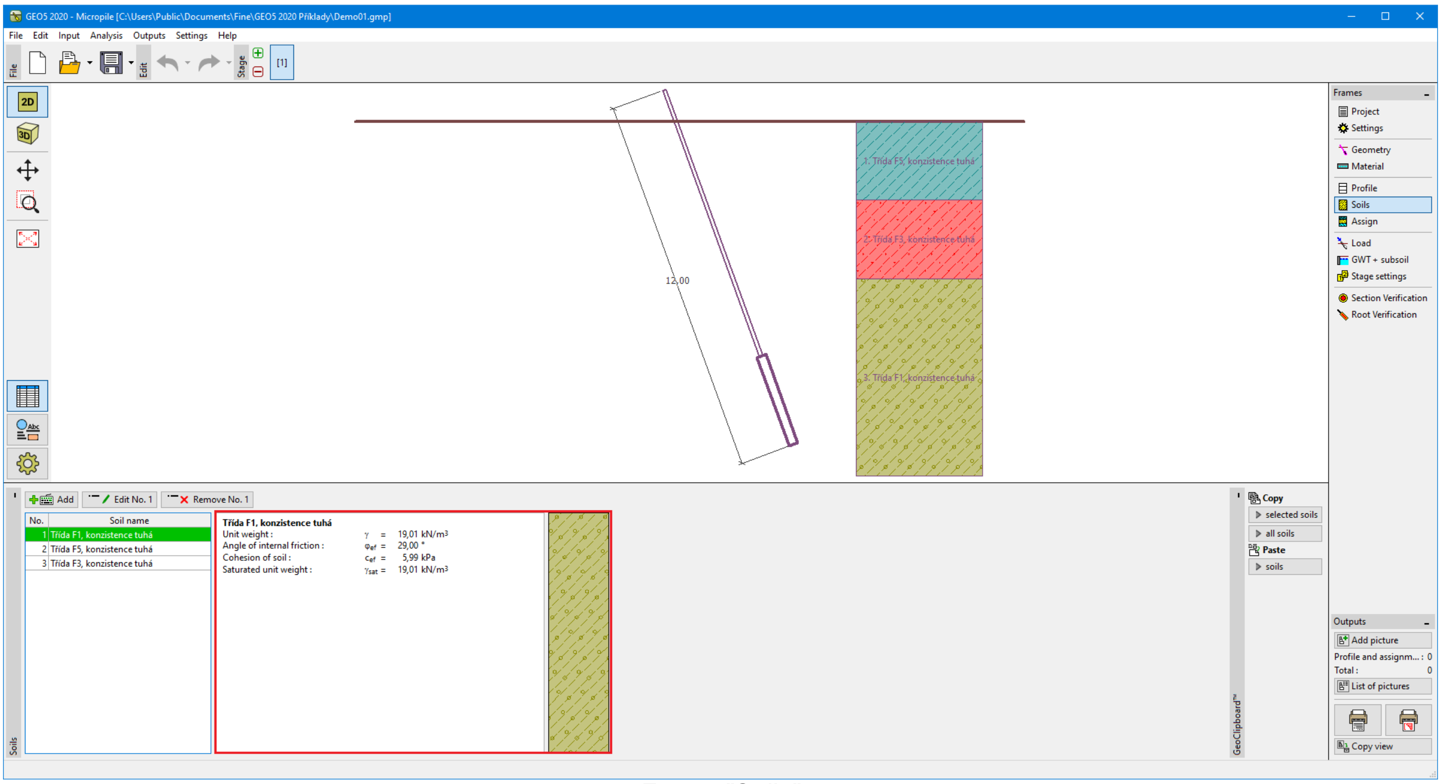
Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data" "Uplift pressure". These parameters depend on the theory of analysis specified in the "Micropiles" tab.

Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



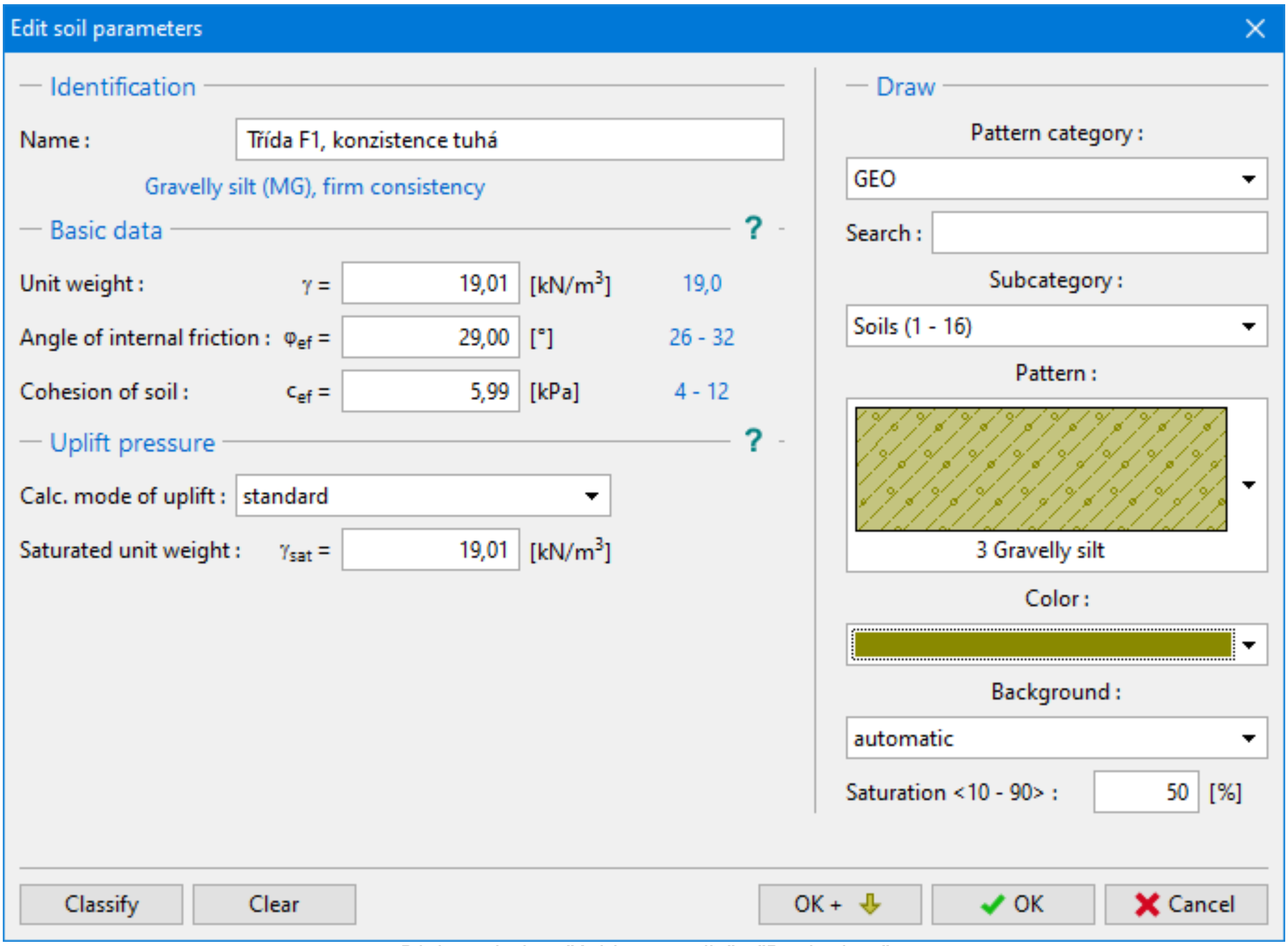
Frame "Soils"

Basic Data

This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction, and cohesion**. The particular values are obtained from a geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit the built-in database of soils, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually.

When calculating the tube bearing capacity according to Salas, moreover, enters elastic modulus E .

The associated theory is described in detail in the chapter "Micropile".



Dialog window "Add new soils" - "Basic data"

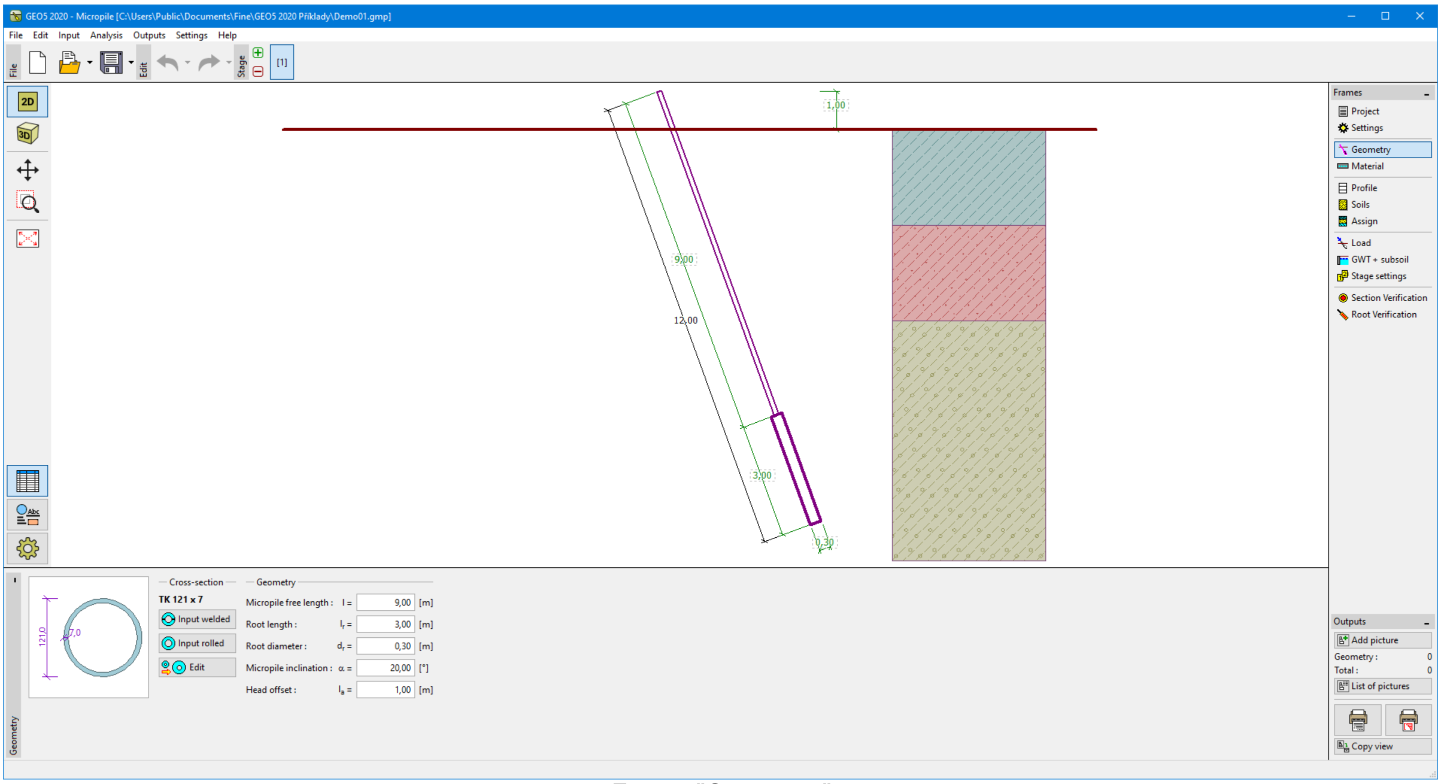
Geometry

The frame "**Geometry**" serves to input a **micropile cross-section** (welded, rolled). The selected shape with a graphical hint of input values is displayed in the left part of the frame. The micropile cross-section is selected in dialog windows opened by pressing the "**Enter welded**" "**Enter rolled**" buttons (the selection for rolled cross-sections is performed from a catalog in the dialog window). An info window, displaying a detailed description of data of the selected cross-section, can be activated in the window. The selected data can be edited after choosing the type of micropile cross-section.

The basic geometrical data are specified in the right top part of the frame:

- free length of micropile (distance between the micropile head and the origin of micropile base is considered)
- root length
- root diameter
- micropile inclination (range from $-60°$ to $60°$ measured from vertical, a positive value of an inclination angle is measured counterclockwise)
- head offset (end of micropile above terrain (range from 0 to $10\ m$).

The program makes it possible to export the geometry of a structure in the *.DXF format.



Frame "Geometry"

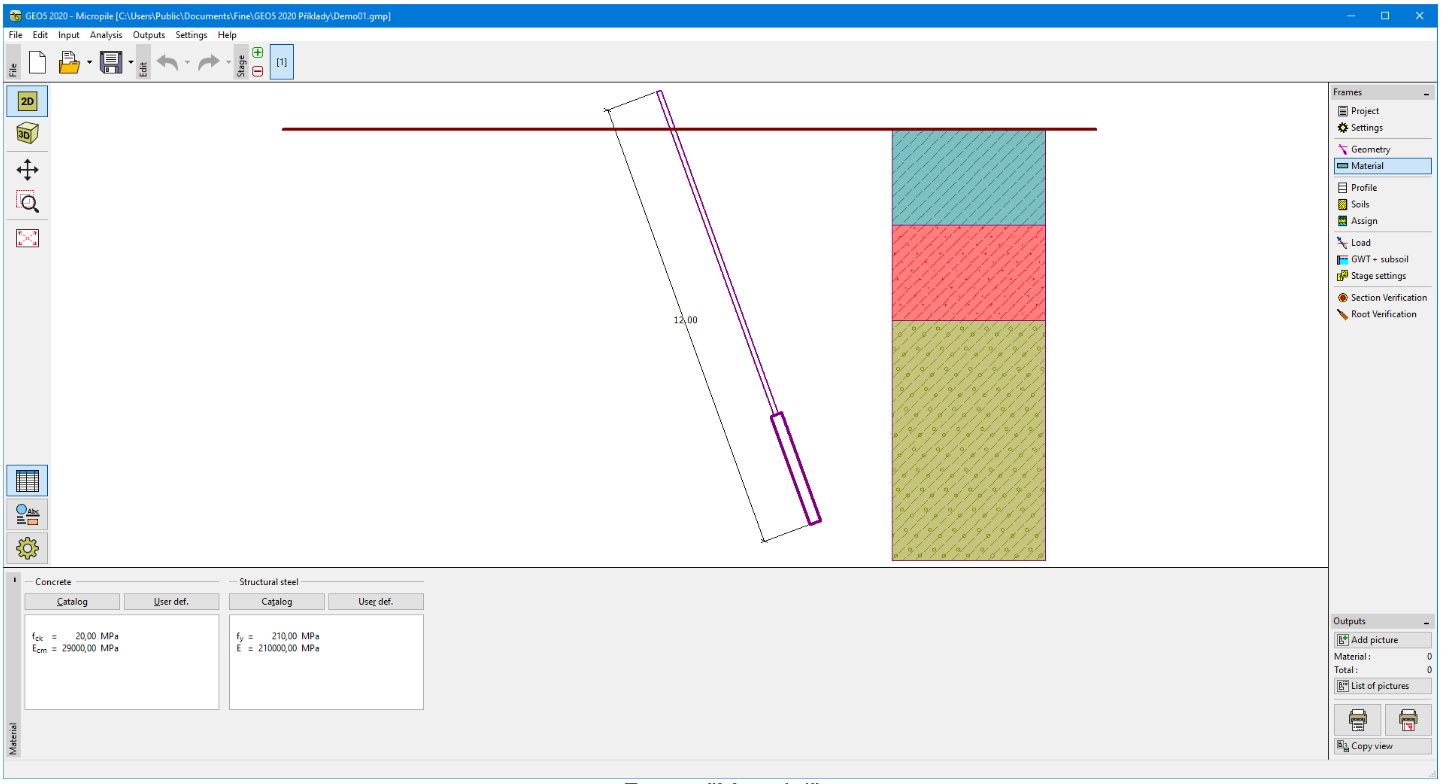
Material

The frame "**Material**" serves to specify material parameters of cement mixture and steel.

Two options are available when selecting the material type:

- The "**Catalog**" button opens the "**Catalog of materials**" dialog window, where the required material can be selected.
- The "**Own**" button opens the "**Editor of material"** which allows us to input the specification of material parameters manually by the user.

The content of catalog depends on the selection of relevant standard for the steel set in the "Materials and standards" tab.

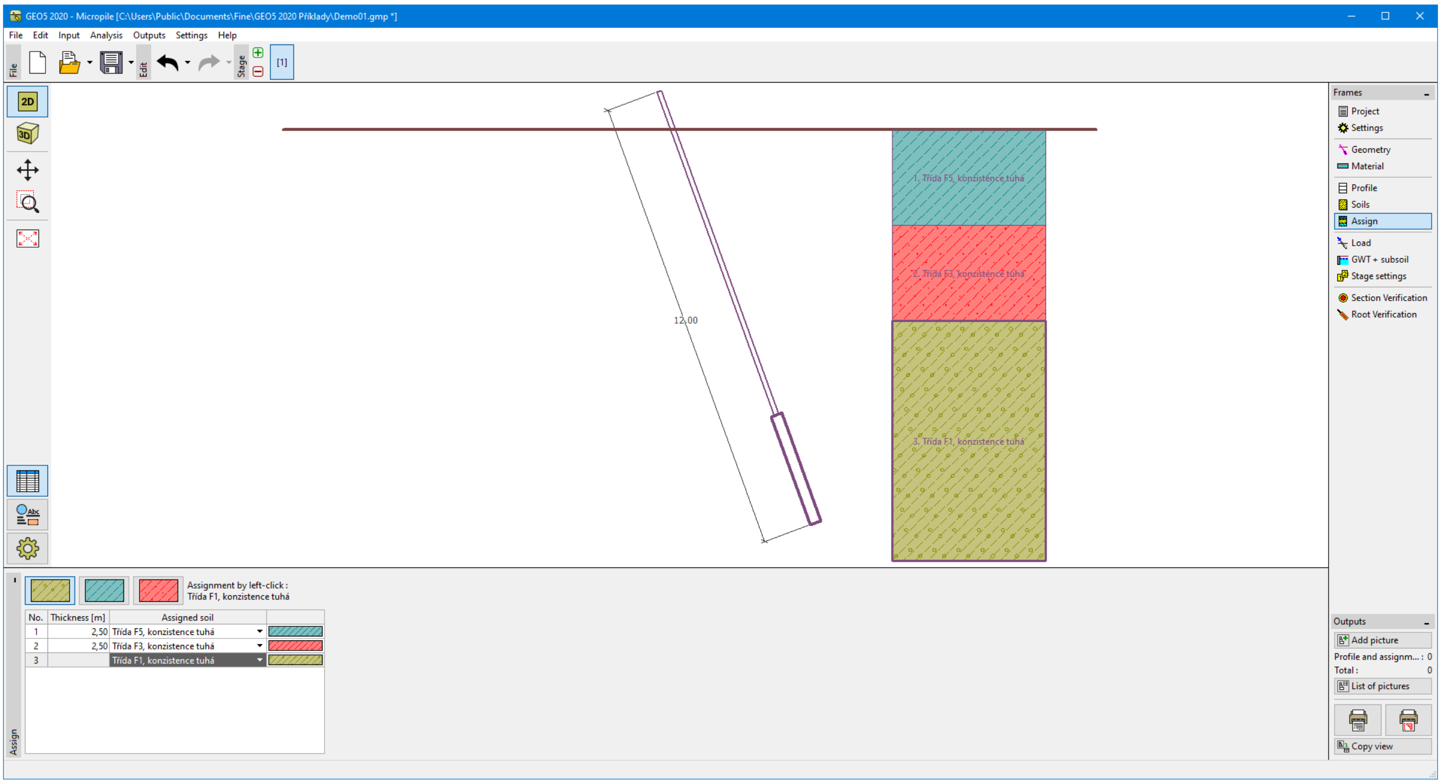


Frame "Material"

Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table or is accessible from a combo list for each layer of the profile.

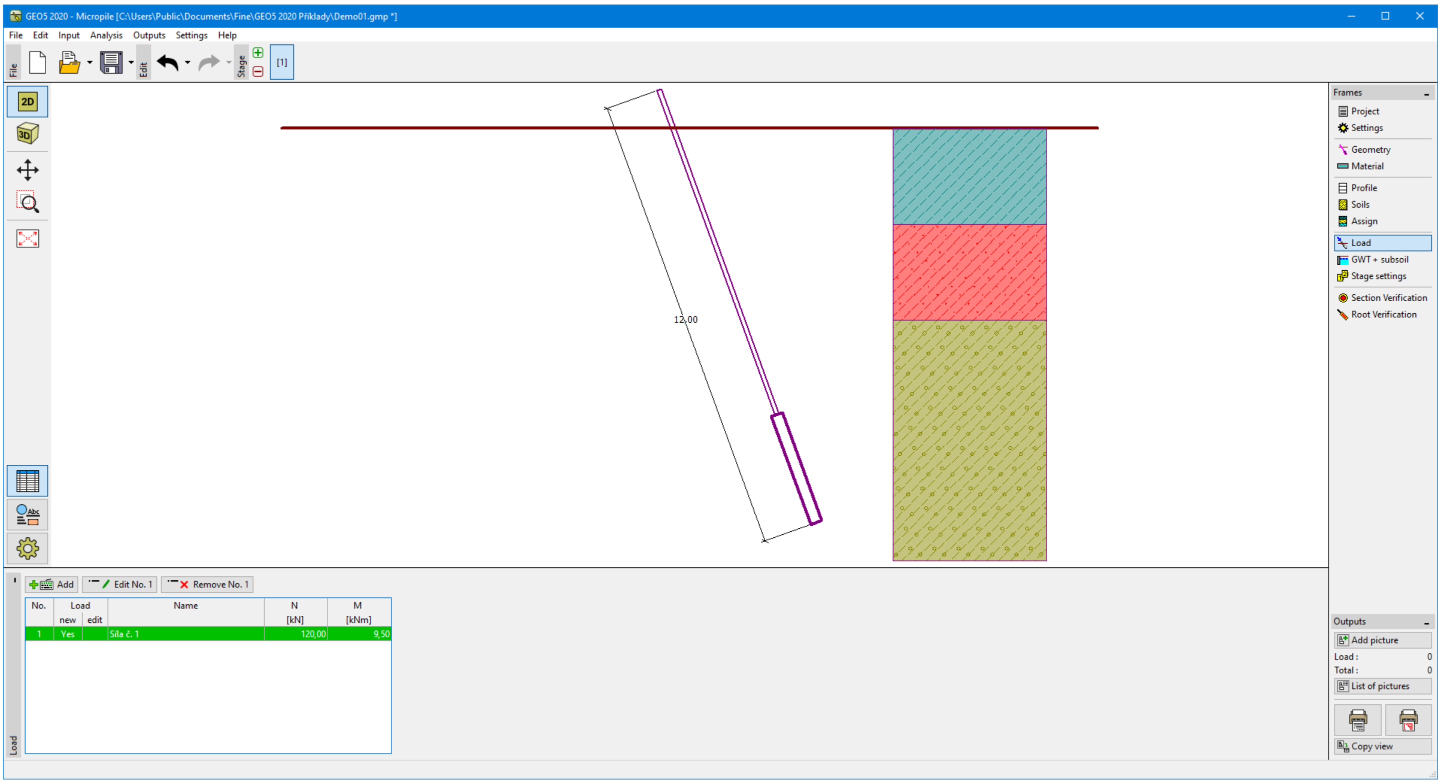
The procedure to assign soil into a layer is described in detail herein.



Frame "Assign"

Load

The "**Load**" frame contains a table with a list of input loads. Adding load is performed in the "**New load**" dialog window. Forces and moments are entered according to the sign convention displayed in the right part of the dialog window.

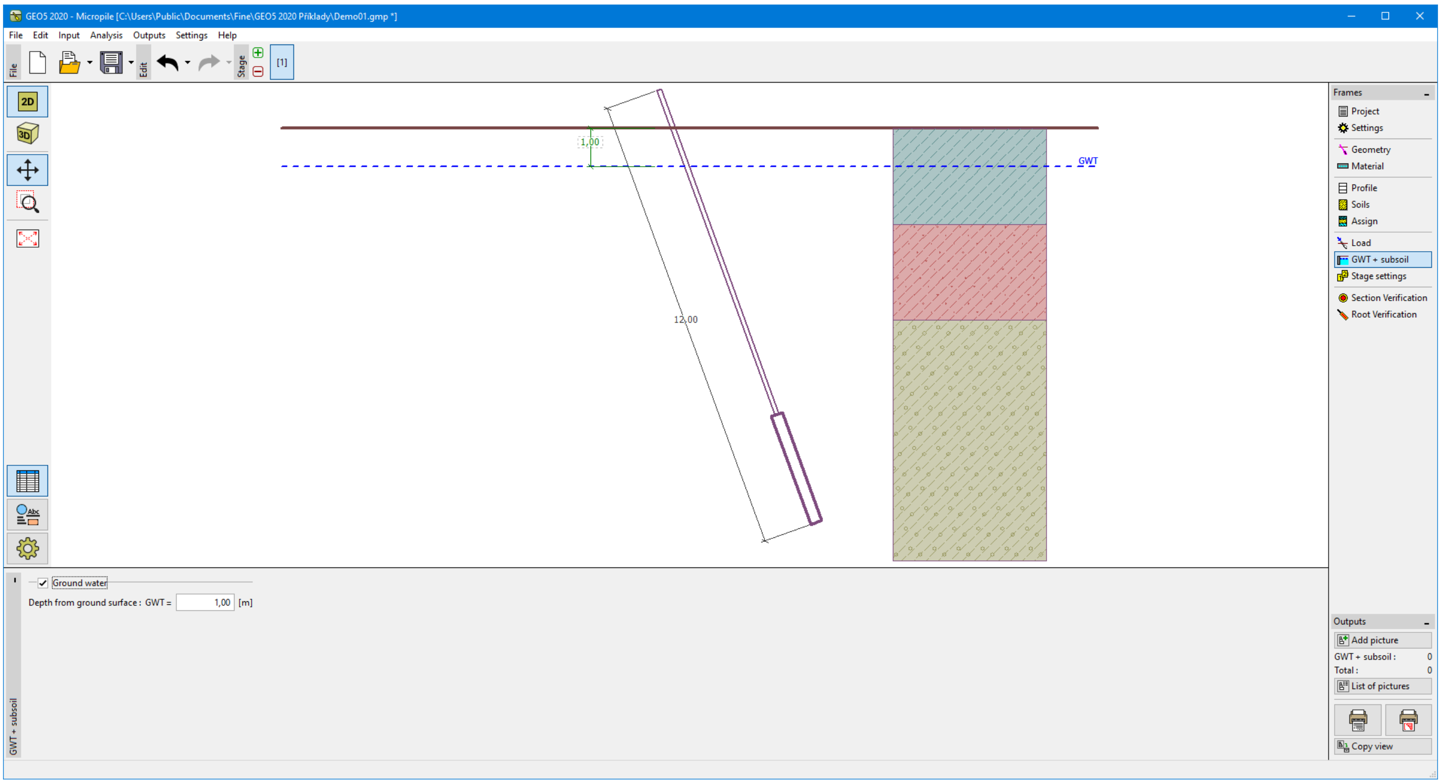


Frame "Load"

Water

The frame "**Water**" serves to enter a **depth of groundwater table.**

The values can be edited either in the frame by entering values into particular fields, or on the desktop with the help of active dimensions.

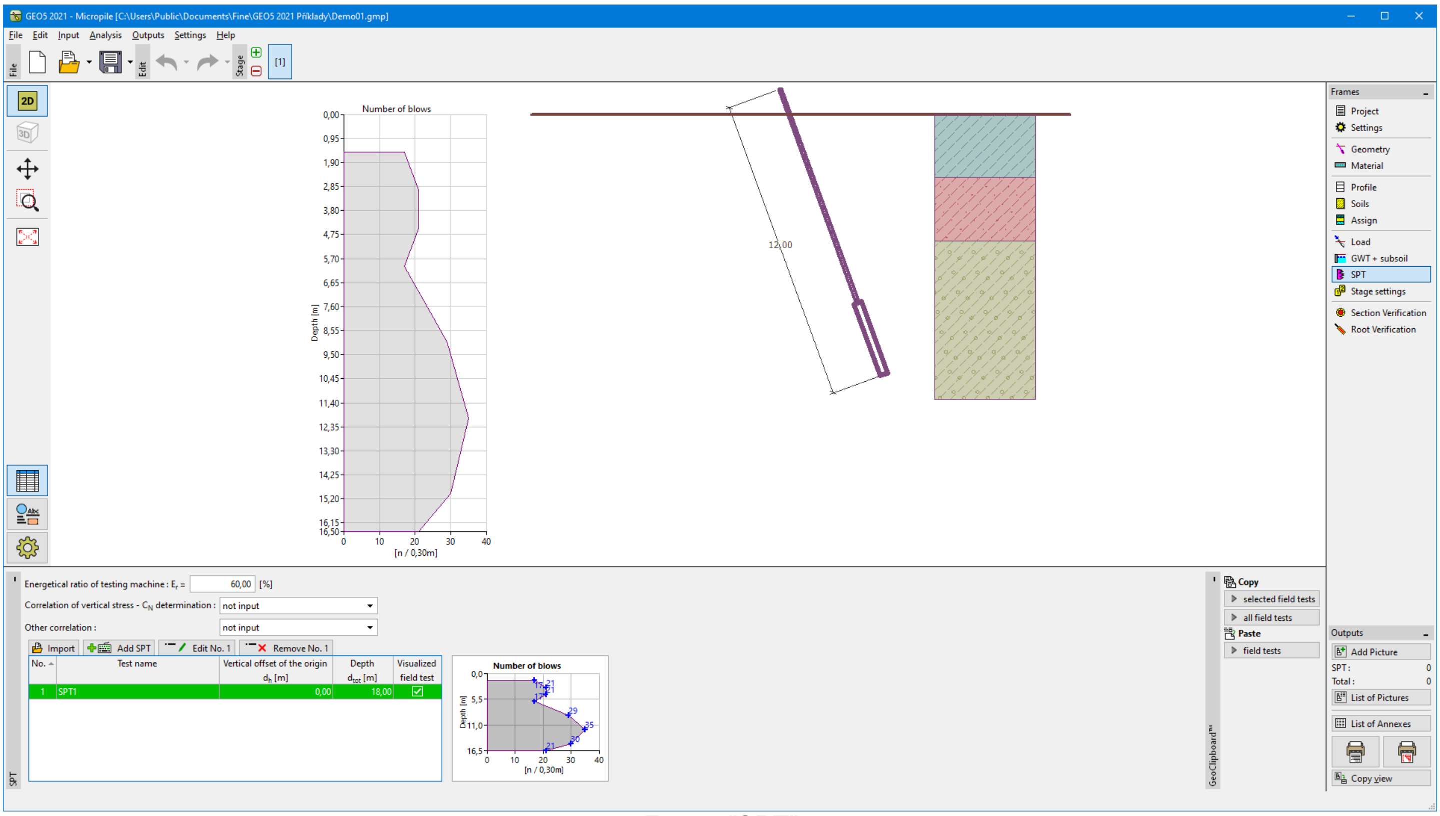


Frame "Water"

SPT

The frame "**SPT**" contains a table with a list of standard penetration tests (SPT).

The SPTs are used in the analysis according to the Bustamante method.



Frame "SPT"

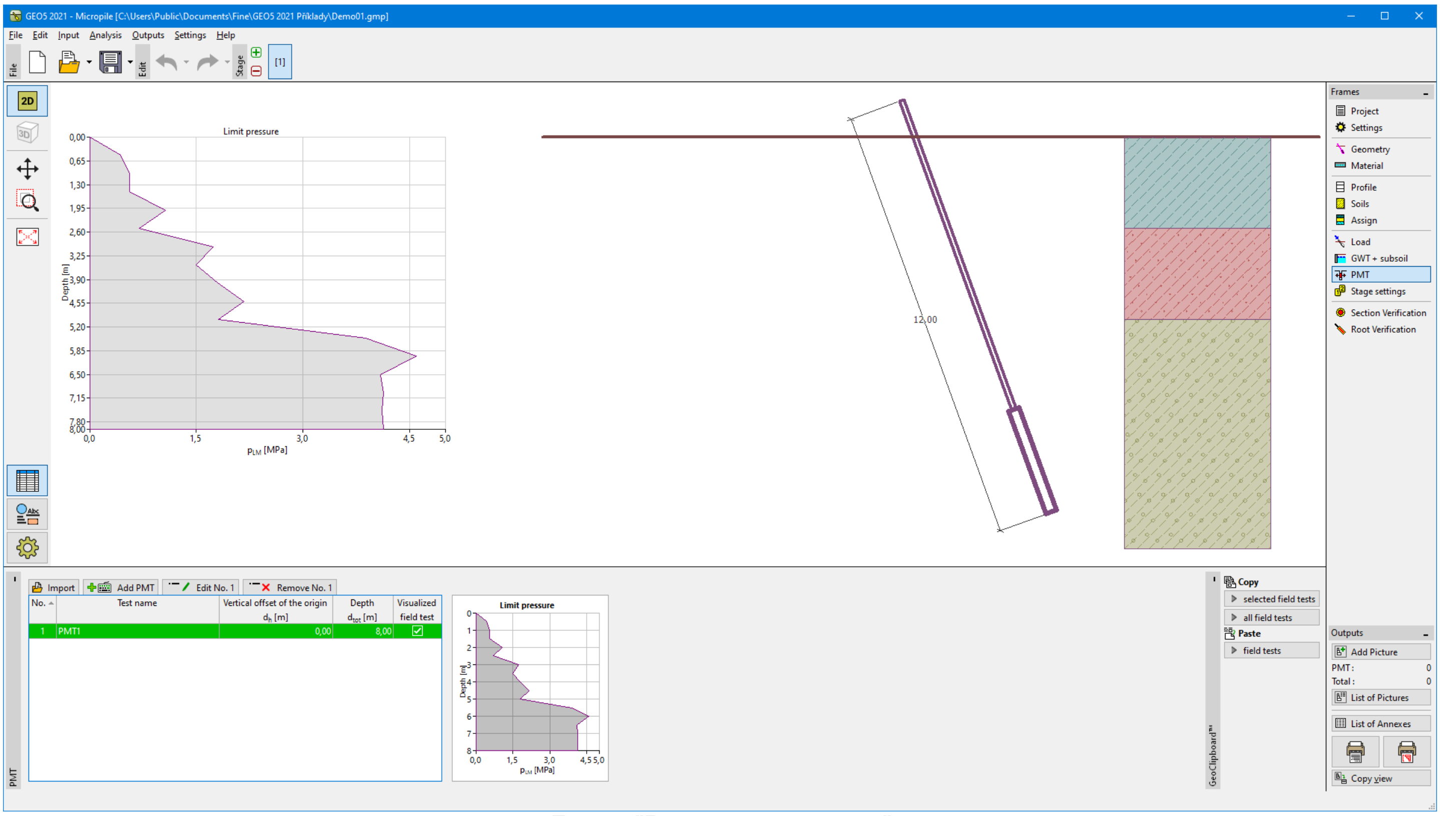
The results of standard penetration tests (SPT) can be imported into the program by inserting the file in different formats (eg. ***.TXT, *.CSV, *.XLSX, *.ODS**).

Data of SPTs can be copied within "**Micropile**", "**Spread Footing CPT**" and "**Stratigraphy**" programs using "GeoClipboard".

PMTs

The frame "**PMTs**" contains a table with a list of pressuremeter tests (PMT).

The PMTs are used in the analysis according to the Bustamante method.



Frame "Pressuremeter tests"

The results of pressuremeter tests (PMT) can be imported into the program by inserting the file in different formats (eg. ***.TXT, *.CSV, *.XLSX, *.ODS**).

Data of SPTs can be copied within "**Micropile**", "**Spread Footing CPT**" and "**Stratigraphy**" programs using "GeoClipboard".

Verification of Cross Section

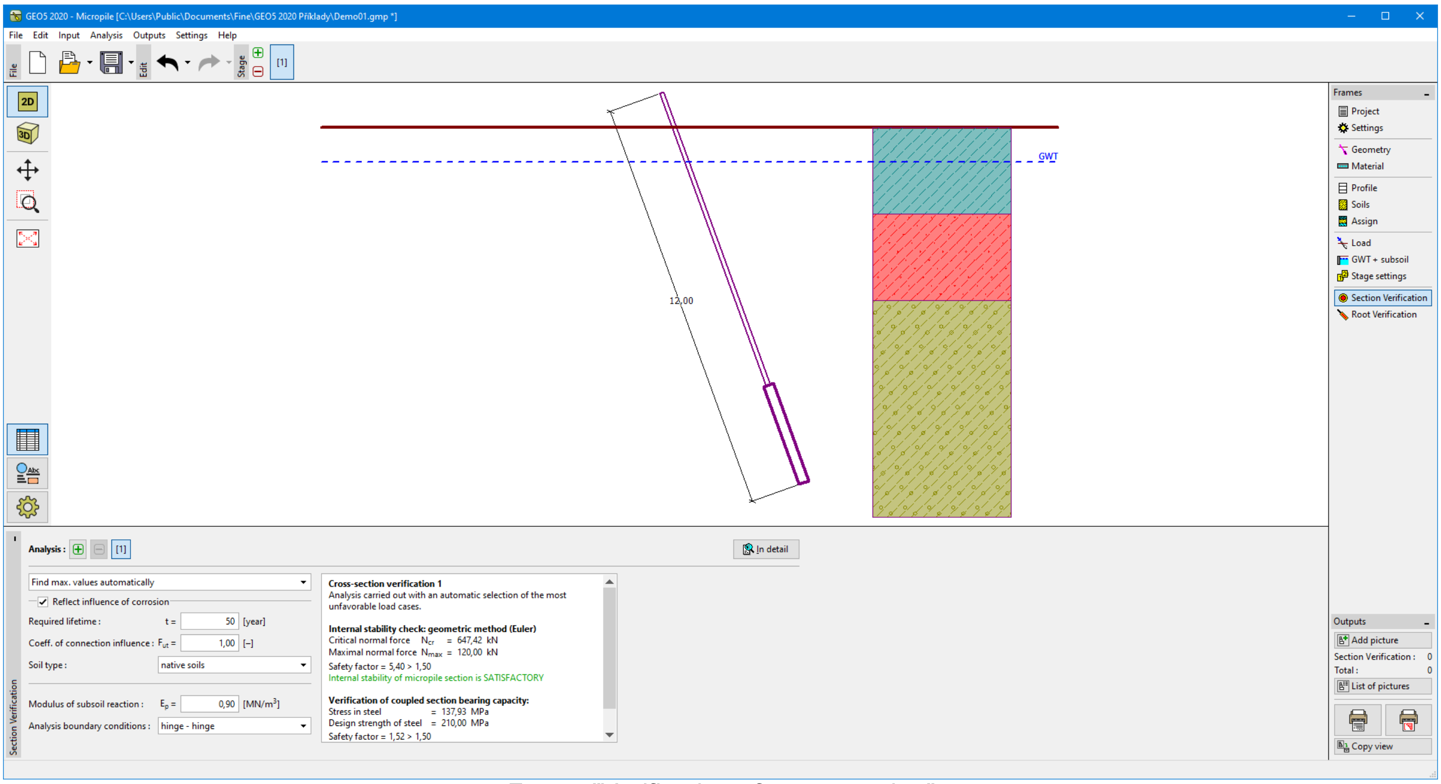
The results of the analysis of the micropile tube bearing capacity loaded either in tension or compression are displayed in the frame "**Verification of cross-section**". More computations can be carried out for a single task. The left part of the frame allows inputting the modulus of subsoil reaction and the influence of corrosion.

When calculating the tube bearing capacity (micropile cross-section) the program differentiates between a micropile loaded in tension or in compression.

In case of tension, the program determines coupled section bearing capacity (strength of cement mixture is not considered).

In the case of compression, the program examines both coupled section bearing capacity and internal stability of section, depending on the method set in the "Micropiles" tab.

The results of the verification analysis are displayed in the right part of the window. Visualization of results can be adjusted in the frame "Drawing Settings".



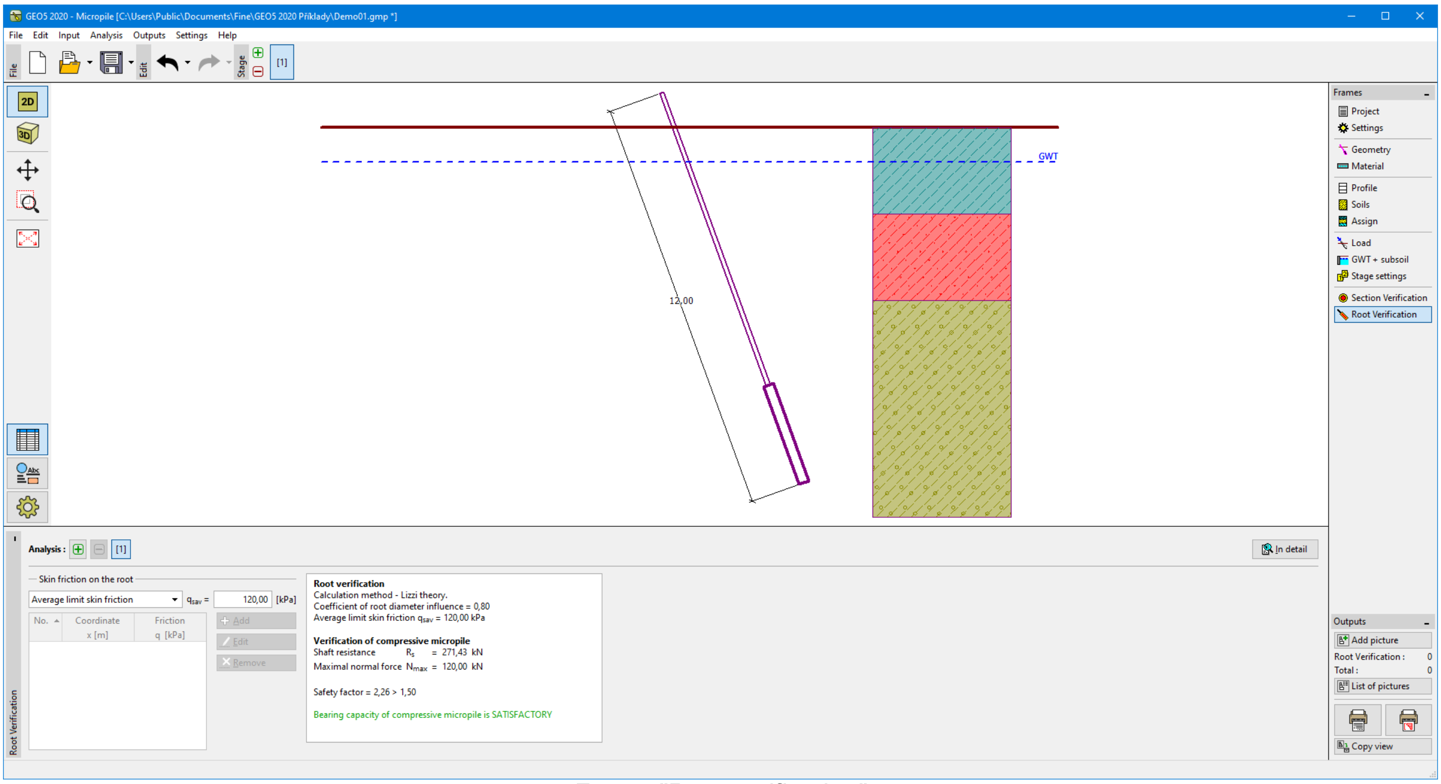
Frame "Verification of cross-section"

Root Verification

The analysis results are displayed in the "**Root verification**" frame. Several calculations can be carried out for a single task. The limit skin friction can be specified in the left part of the frame.

The procedure to examine the micropile root is described in detail herein.

The results of the verification analysis are displayed in the right part of the window. Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Root verification"

Program Slab

This program is used for the analysis of foundation mats and slabs of any shape on elastic subsoil, using the Finite Element Method.

The help in the program "Slab" includes the following topics:

- The input of data into individual frames:

Project	Settings	Joints	Lines	Macroelements	Openings	Joint Refinements
Line Refinements	Macroelement Refinements	Mesh Generation	Joint Supports	Line Supports	Beams	Internal Hinges
Macroelement Subsoils	Load Cases	Joint Loads	Line Loads	Macroelement Loads	Free Point Loads	Free Line Loads
Free Area Loads	Combination ULS	Combination SLS	Dimensioning Parameters	Macroelement Dimensioning	Analysis	Values
Distributions						

- Standards and analysis methods

- Theory for dimensioning of concrete structures

- Outputs

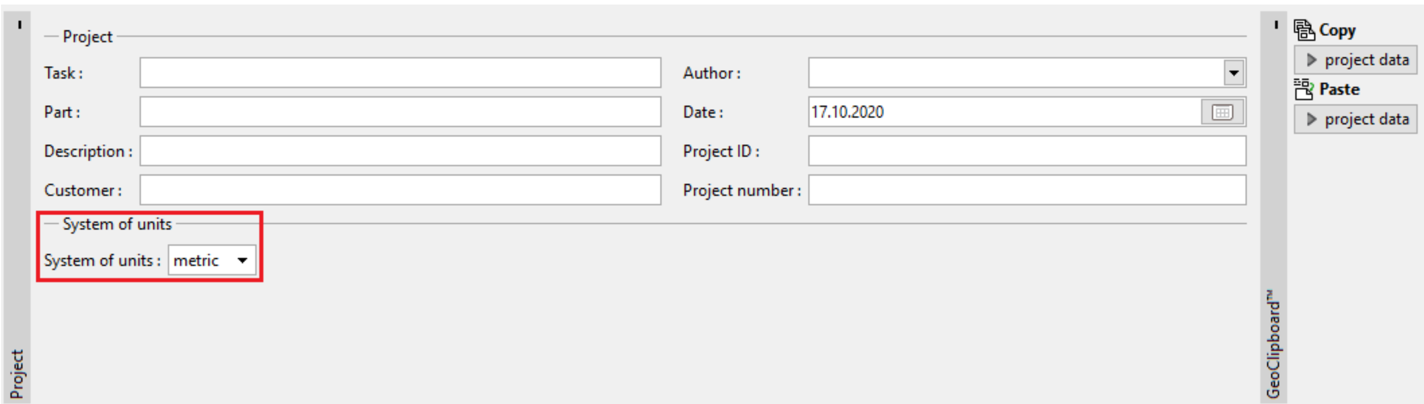
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The frame **"Project"** is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The frame "**Settings**" allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

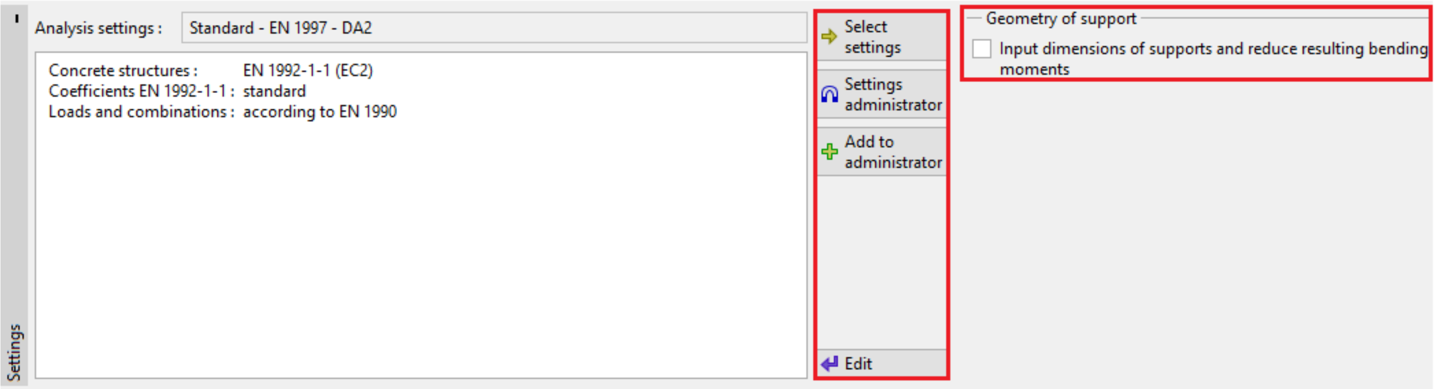
The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

Settings of analysis parameters are performed in the "Materials and standards" tab.

The program allows the user to specify support dimensions and reduce dimensioning moments.

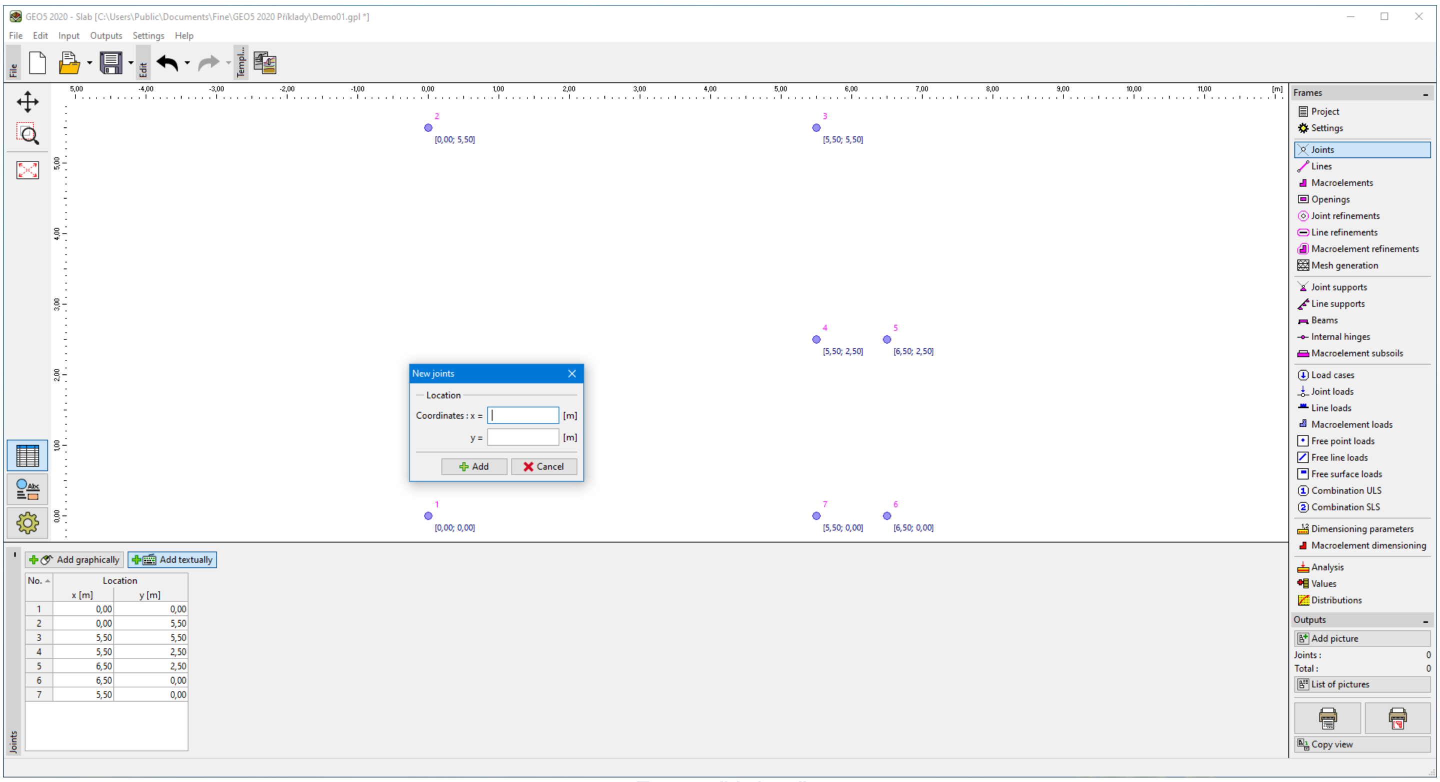


Frame "Settings"

Joints

The frame "**Joints**" contains a table with the list of input joints. Adding joints is performed in the "**New joints**" dialog window.

The input joints can also be edited on the desktop with the help of active objects.



Frame "Joints"

Lines

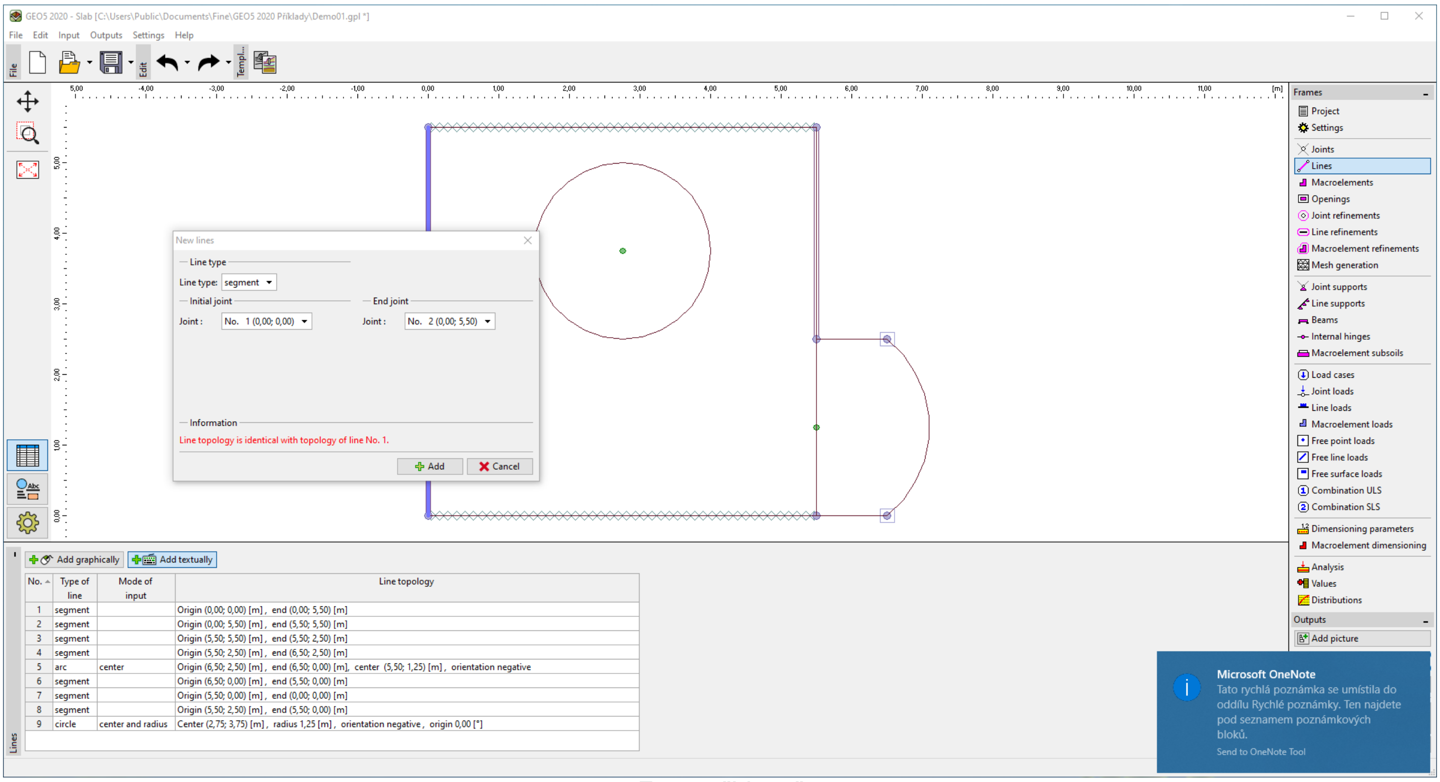
The frame "**Lines**" contains a table with the list of input lines. Adding lines is performed in the "**New lines**" dialog window.

The lines are defined **between individual points** (segments, arcs, circles) or around individual points (circles). The lines may arbitrarily cross or touch each other - intersections of input lines are identified by the program automatically when correcting the input geometry.

The type of line is selected from the combo box. The following modes are available:

- **Line type** A combo list is used to select the desired type of line (segment, arc, circle).
  - **- segment** Clicking the left mouse button on the joint introduces the line location.
  - **- arc** Use the combo list to choose a particular mode of defining an arc segment (third point, center, radius, height). Clicking the left mouse bottom on the desktop then selects points to define the arc. When selecting one of the following options - center, radius or included angle, you are further requested to select the orientation (positive, negative) from the combo list.
  - **- circle** Use the combo list to choose a particular mode of defining a circle (center and radius, three points). Clicking the left mouse button on the desktop then selects points to define the circle. The combo list is also used to select the orientation (positive, negative).

The lines can also be edited on the desktop with the help of active objects.

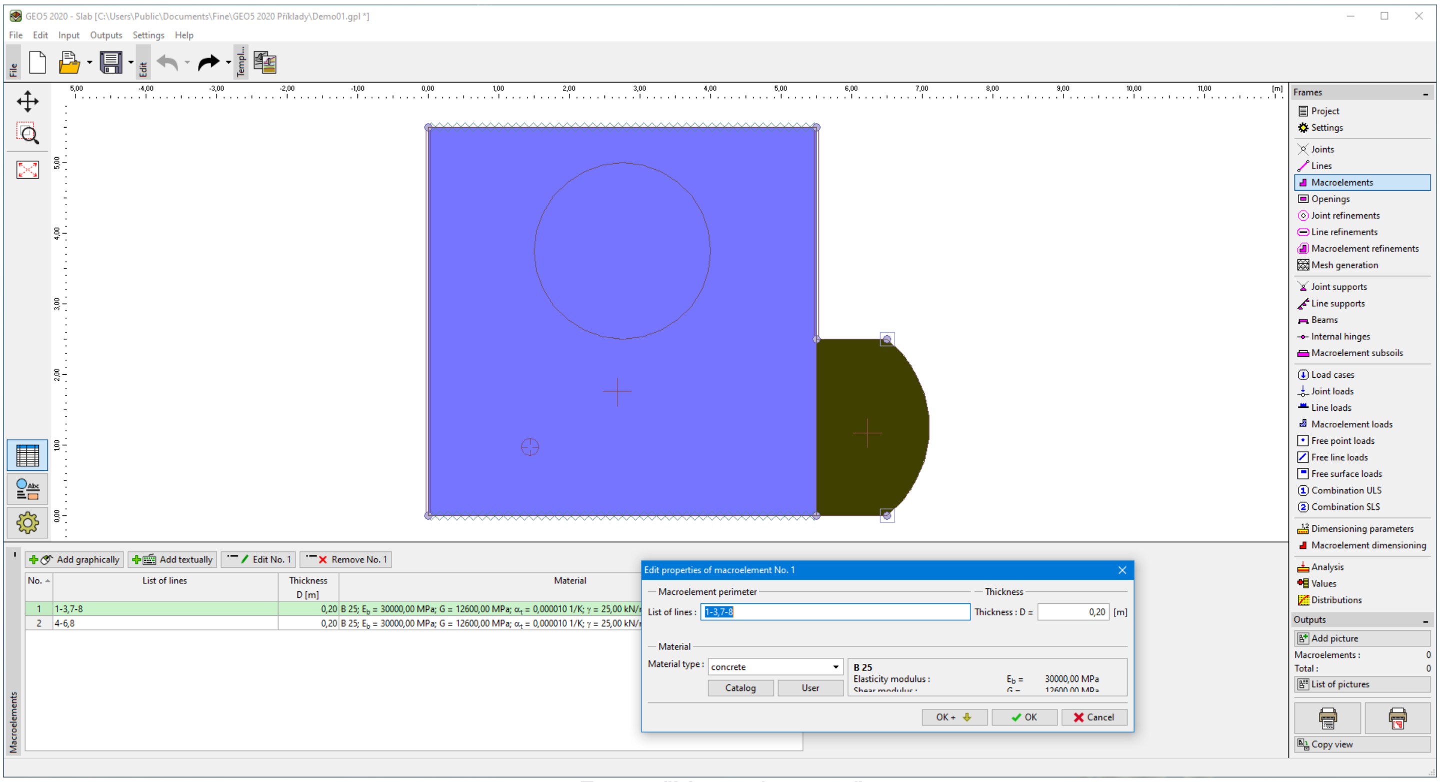


Frame "Lines"

Macroelements

The frame "**Macroelements**" contains a table with the list of input macroelements. Adding macroelements is performed in the "**New macroelements**" dialog window. The dialog window servers to input a list of lines defining the macroelement outline, its thickness and material. The macroelement material can be selected from the catalog of materials, or its material parameters can be input manually using the "**Edit material**" dialog window.

The input macroelements can also be edited on the desktop with the help of active objects.

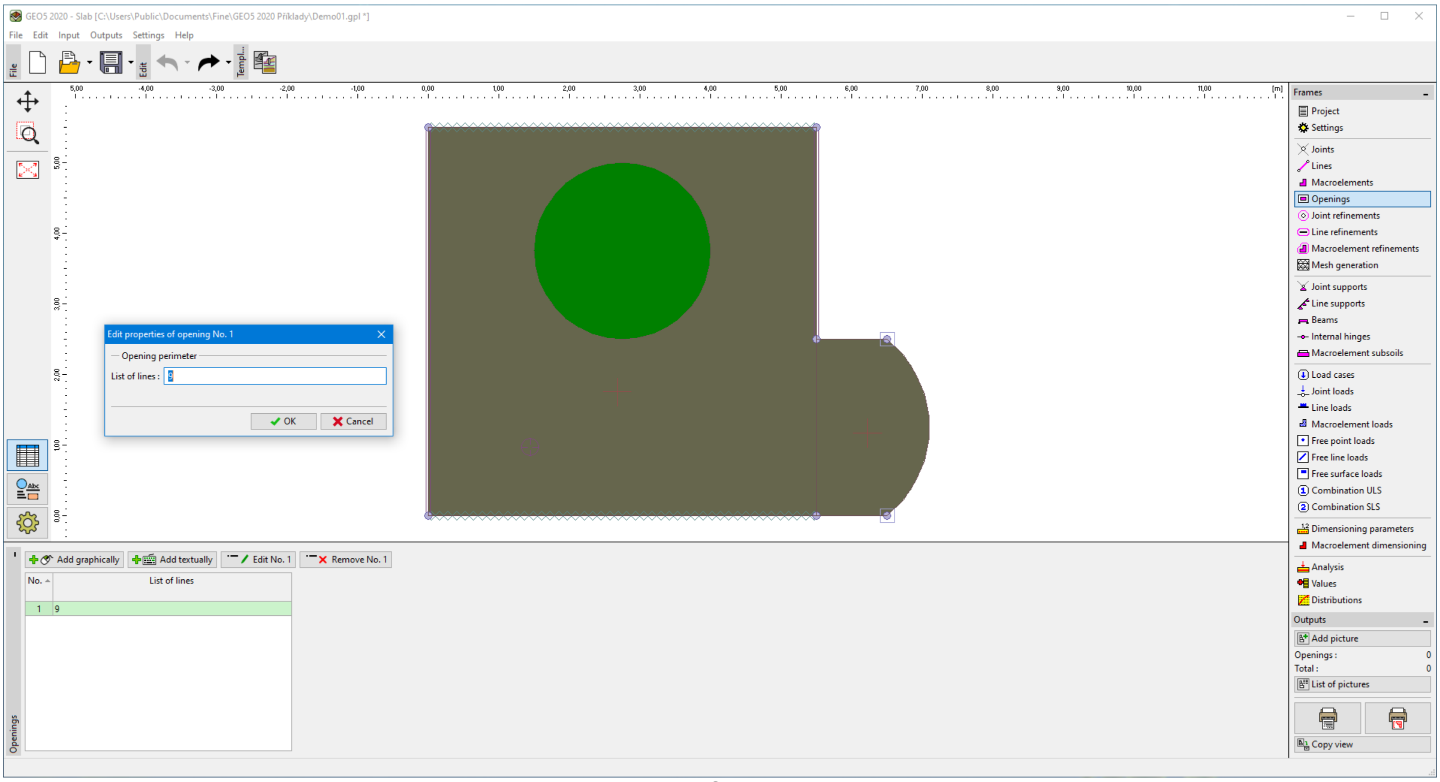


Frame "Macroelements"

Openings

The frame "**Openings**" contains a table with the list of input openings. Adding openings is performed in the "**New openings**" dialog window.

The input openings can also be edited on the desktop with the help of active objects.



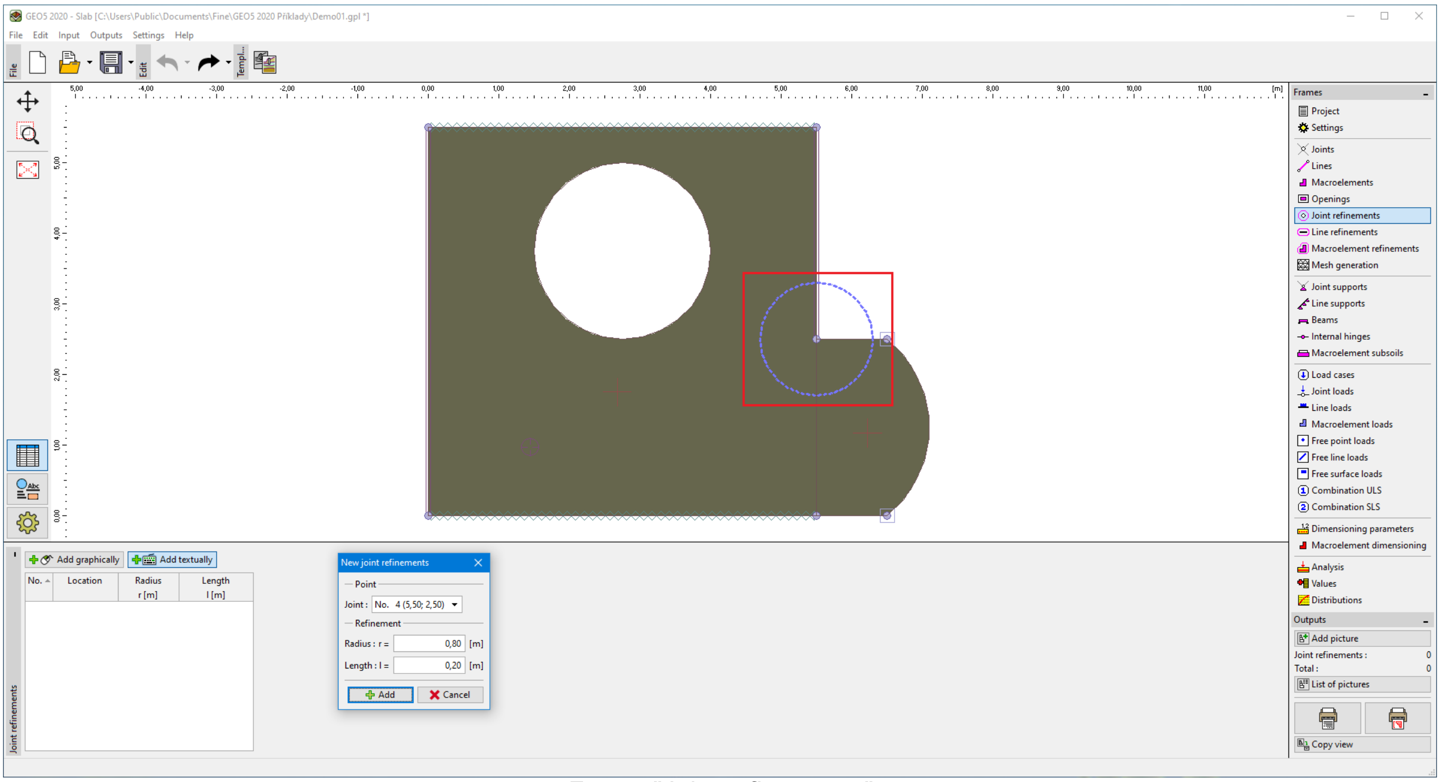
Frame "Openings"

Joint Refinements

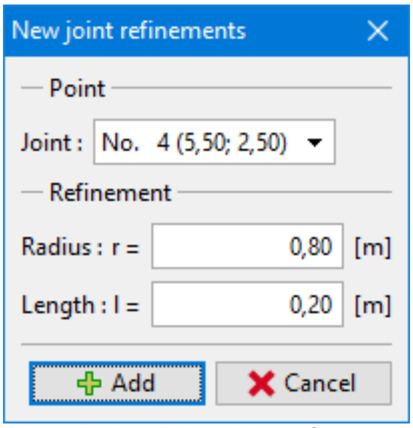
The frame "**Joint refinements**" contains a table with the list of input joint refinements. Adding joint refinements is performed in the "**New joint refinements**" dialog window.

Refining the finite element mesh around joints is an important feature, which allows us to create an appropriate finite element mesh.

The input joint refinements can also be edited on the desktop with the help of active objects.



Frame "Joint refinements"



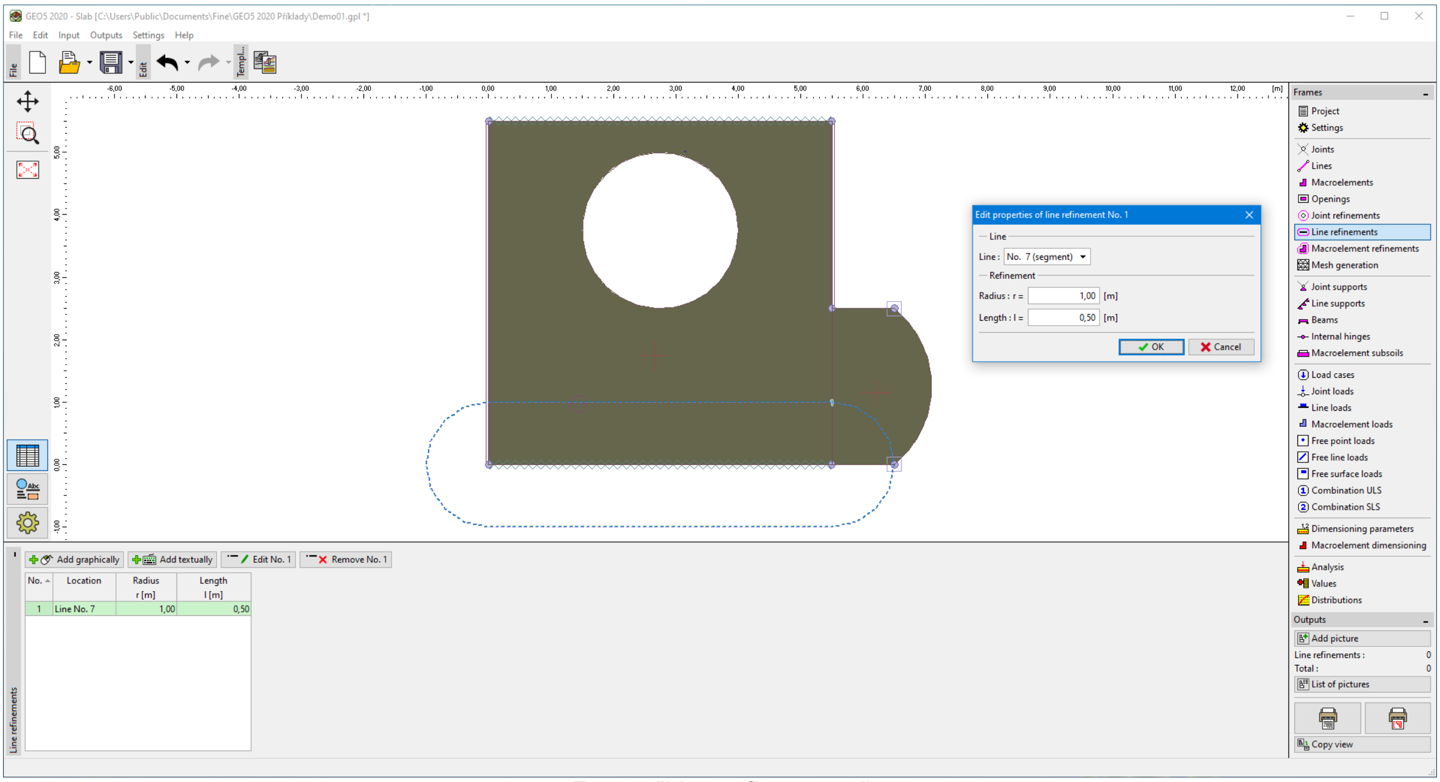
Dialog window "New joint refinements"

Line Refinements

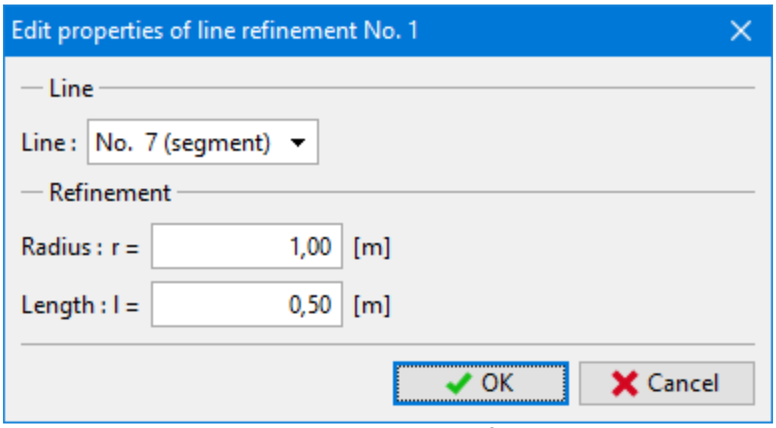
The frame "**Line refinements**" contains a table with the list of input line refinements. Adding line refinements is performed in the "**New line refinements**" dialog window.

Refining the finite element mesh around lines is an important feature, which allows us to create an appropriate finite element mesh.

The input line refinements can also be edited on the desktop with the help of active objects.



Frame "Line refinements"



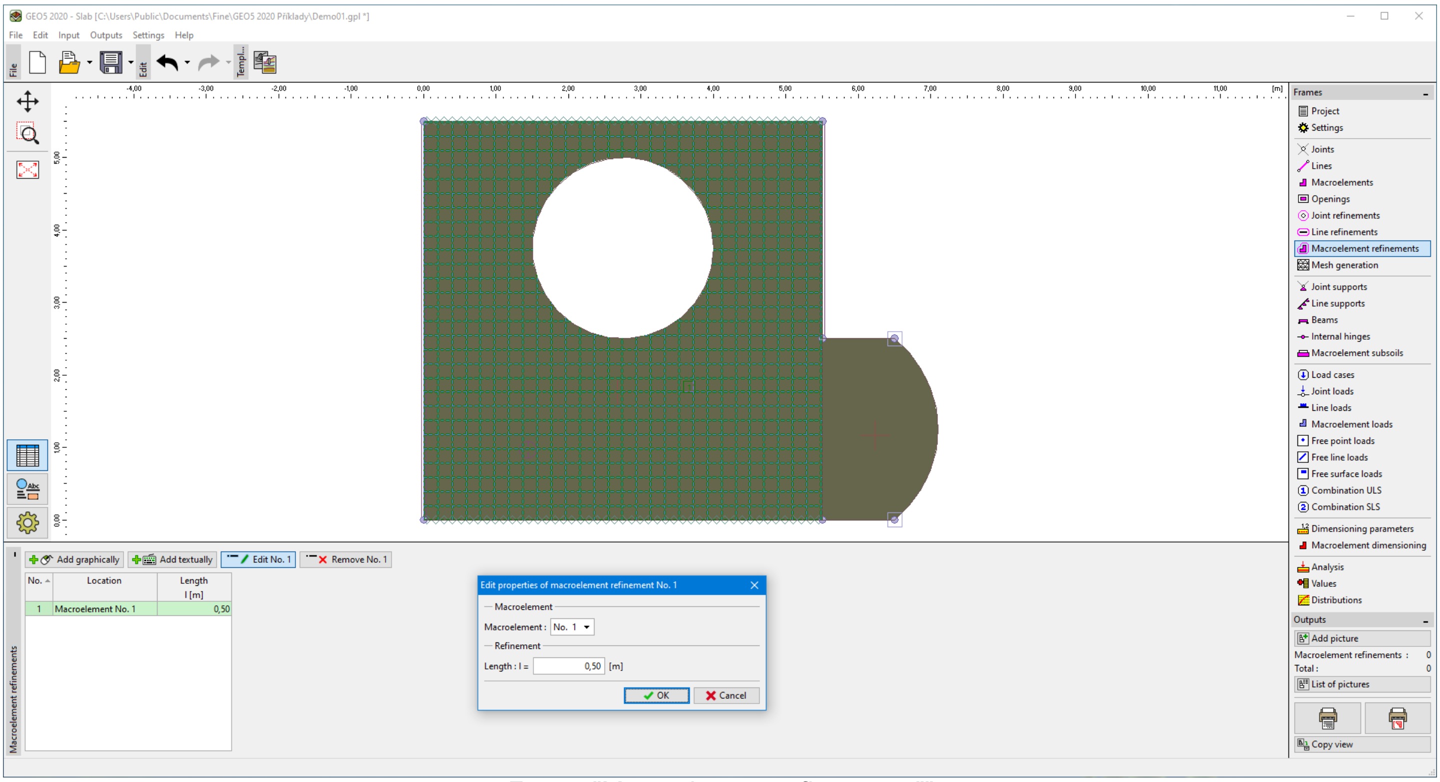
Dialog window "New line refinements"

Macroelement Refinements

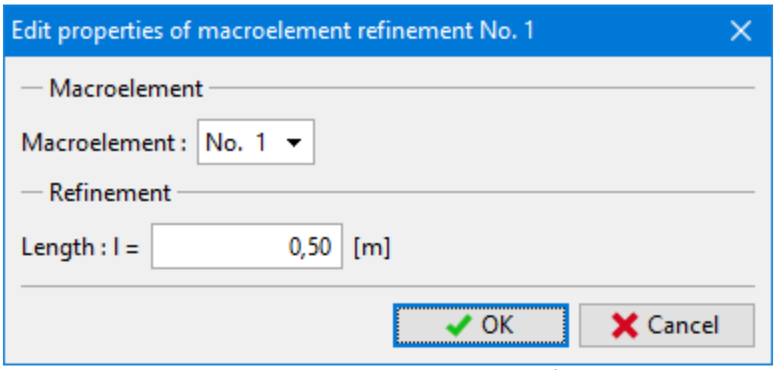
The frame "**Macroelement refinements**" contains a table with the list of input macroelement refinements. Adding macroelement refinements is performed in the "**New macroelement refinements**" dialog window.

Refining the finite element mesh of macroelements is an important feature, which allows us to create an appropriate finite element mesh.

The input macroelement refinements can also be edited on the desktop with the help of active objects.



Frame "Macroelement refinements""



Dialog window "New macroelement refinements"

Mesh Generation

The frame "**Mesh generator**" serves to define the basic setting to generate mesh (element edge length, mesh type, mesh smoothing) and to view information about the generated mesh (right part). The "**Error analysis**" button allows for visualization of error listing in the right part of the frame (list of problems the structure has).

Information about the resulting mesh, including warnings for possible weak points in the mesh, is displayed in the right bottom window.

An arbitrary part of the slab specified by lines (segments, arches and circles) can be meshed. The slab can be formed by one or more macroelements all having a constant thickness and identical material properties and may contain an arbitrary number of openings. In addition, it is possible to introduce internal points and lines which are then considered as mesh nodes and edges. The joints along lines and inside macroelements allow for mesh refinement, which is characterized by the required length of element edges in the center of the refinement and by the refinement radius. The user may choose either a purely triangular mesh or a hybrid mesh consisting of both triangular and quadrilateral elements. The meshing algorithm is based on Delaunay triangulation enhanced by several methods to modify and optimize the finite element mesh. The mesh nodes are automatically renumbered to minimize the computational effort.

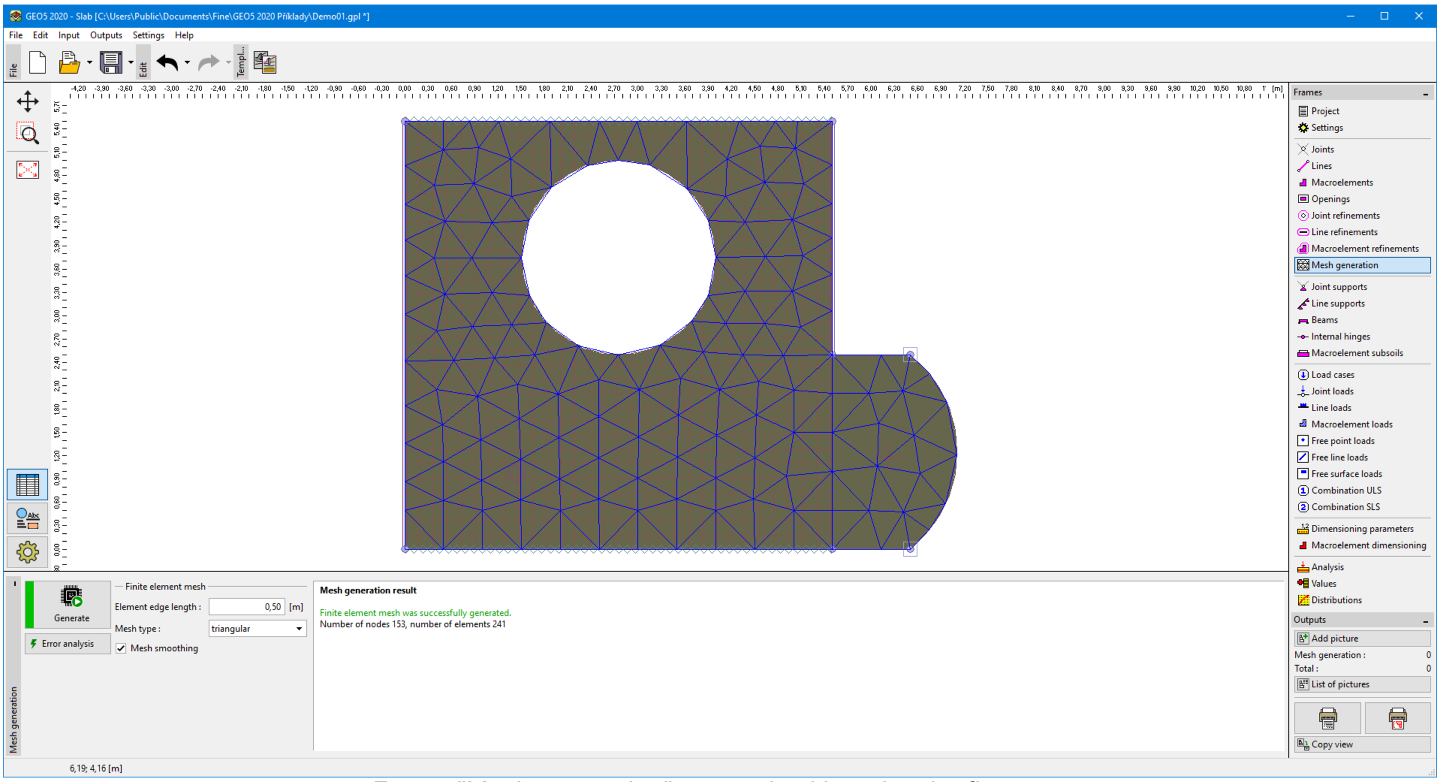
Properly generated finite element mesh is the stepping stone for obtaining accurate results. Optimal are equilateral triangular and square quadrilateral elements. The program contains a built-in automatic mesh generator considerably simplifying this task. The basic mesh density is specified in the "**Mesh generator**" window. Refining the mesh increases accuracy of the results. However, high mesh density considerably slows down both the solution and subsequent visualization of the results. The goal is thus to create an optimally refined mesh - this strongly depends on user’s experience.

Thanks to the efficiency of the mesh generator, there is no problem adjusting input parameters until obtaining an optimal mesh. The mesh quality is further maintained with the help of a built-in smoothing algorithm, which can be turned off. The actual analysis step is extremely fast even for relatively dense meshes.

The following procedure to generate the finite element mesh is recommended:

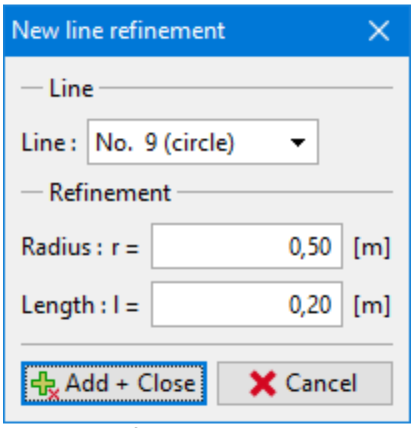
Correctly generated finite element mesh is the major step in achieving accurate and reliable results. The program FEM has an automatic mesh generator, which may substantially simplify this task. Nevertheless, **certain rules should be followed** when creating a finite element mesh:

- First, a uniform mesh linked to the slab thickness (1-5 multiple of its thickness) is generated throughout the slab.

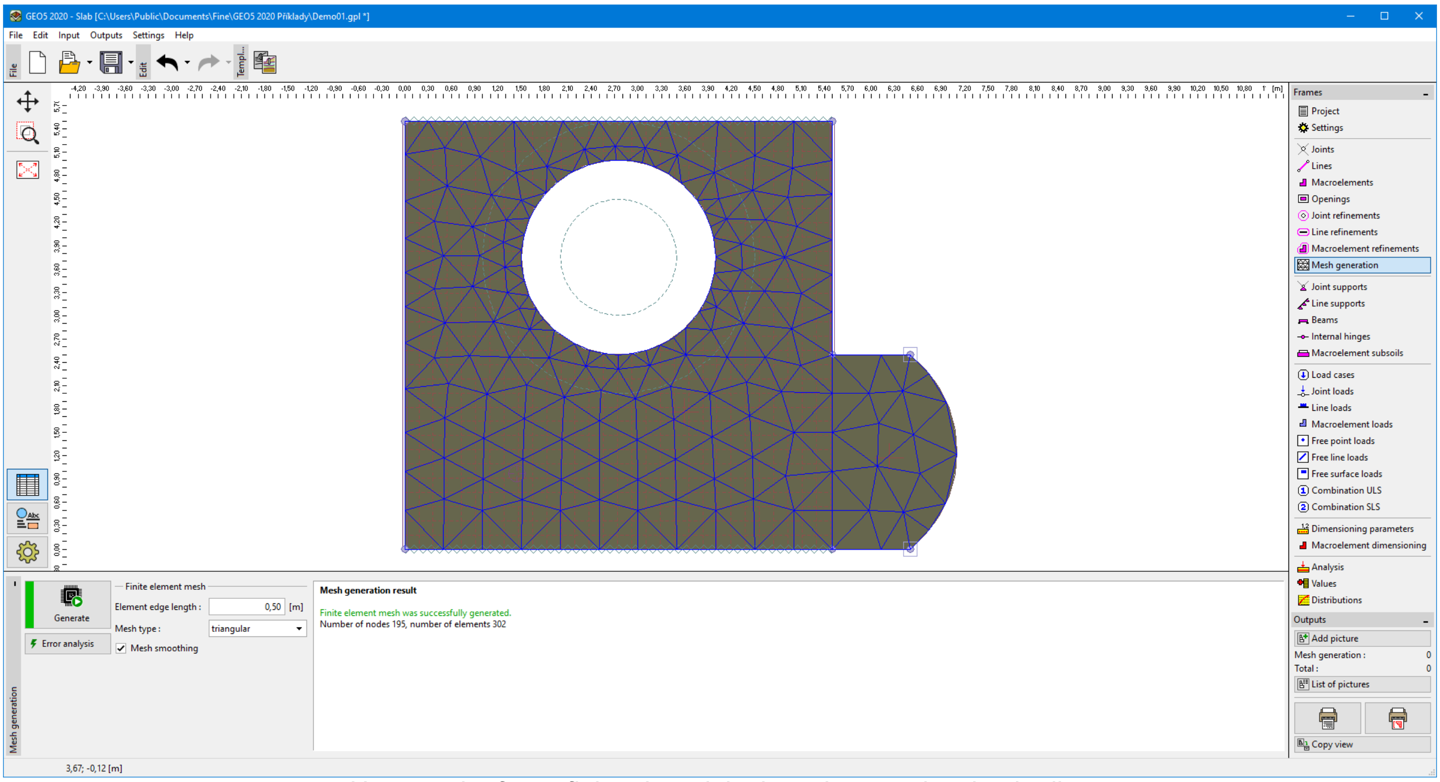


Frame "Mesh generation" - a mesh with no local refinement

- The finite element mesh should be sufficiently fine in the locations where large stress gradients are expected (point supports, corners, openings, etc.). The mesh refinement can be specified around individual joints, lines and on the macroelements. Its radius should be at least 2-3 multiple of the density assumed in the center of the refinement and both values (density, radius) should be reasonable with respect to the refinement prescribed for the neighboring regions. This assures a smooth transition between regions with different mesh densities. Singular lines should be tackled in the same way.



Defining mesh refinement around a circular line



New mesh after refining the original mesh around a circular line

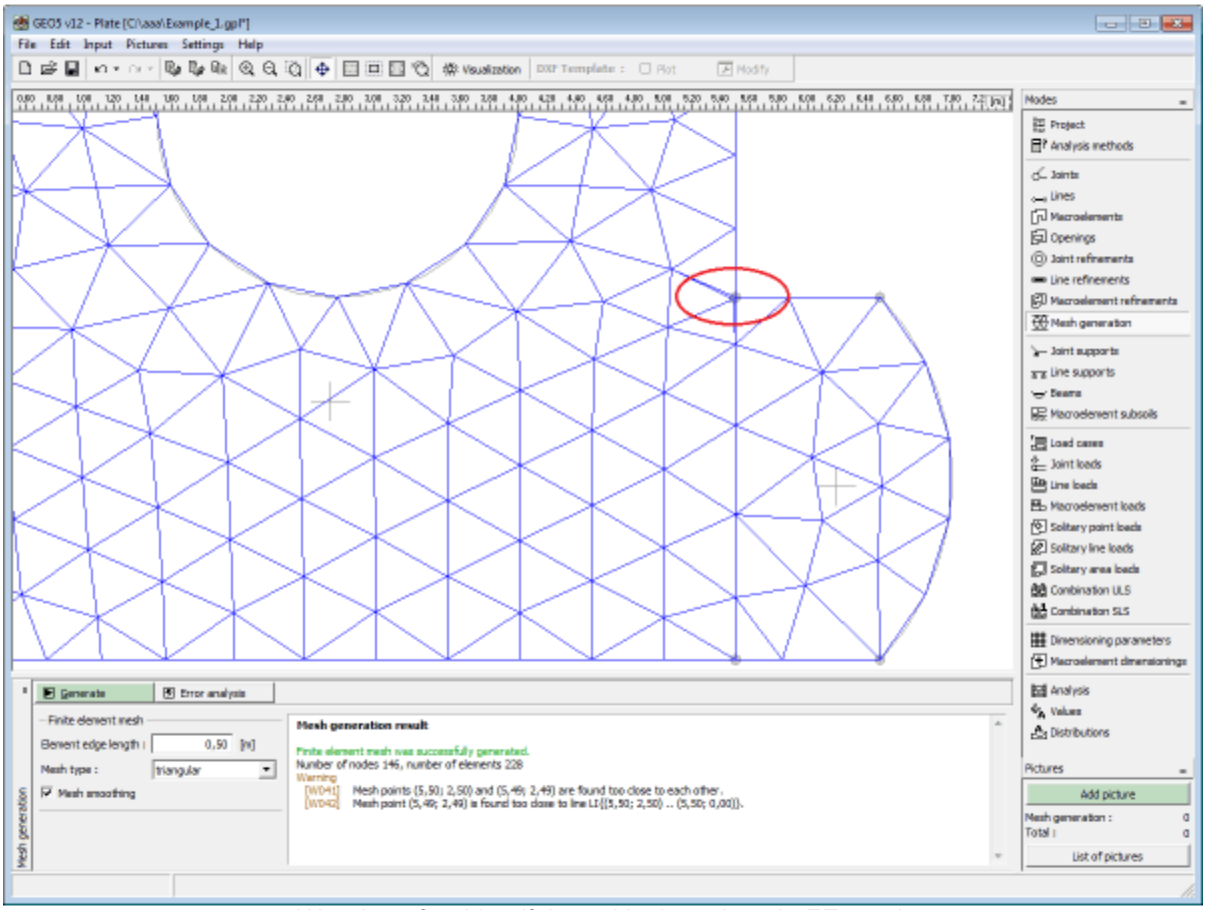
Mesh Generator Warning

In the "**Mesh generation result**" dialog window the user is prompted for possible locations on the structure that may cause problems during automatic mesh generation. When positioning the cursor on individual warnings the corresponding critical region on a structure is highlighted with a red color. The following items are checked:

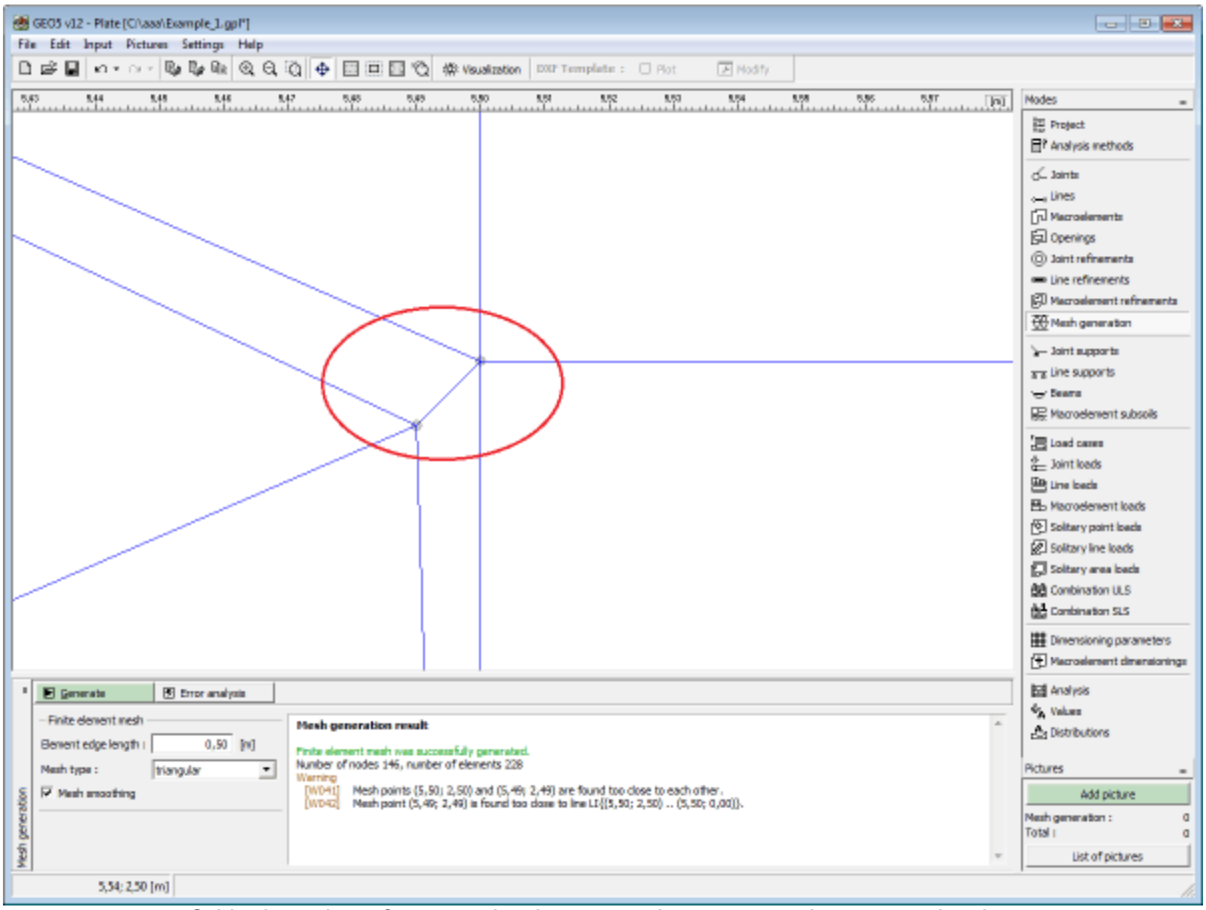
- whether the distance between two points is greater than one tenth of the required element edge length
- whether the distance between a point and a line is greater than one tenth of the element edge length
- whether the area of a region is greater than twice the element edge length
- whether points and/or lines are found inside the structure (in the soil)

These warnings suggest locations, in which the mesh generator experience problems. The following possibilities may occur:

- the mesh is not generated => this calls for a new input of geometrical data
- the mesh is generated => in this case it is up to the user to decide whether the mesh is reasonable - in any case, the warning can be further ignored and the analysis can be carried out



Warning after identifying critical sections in FE mesh



Critical section after zooming in - two points are too close to each other

Joint Supports

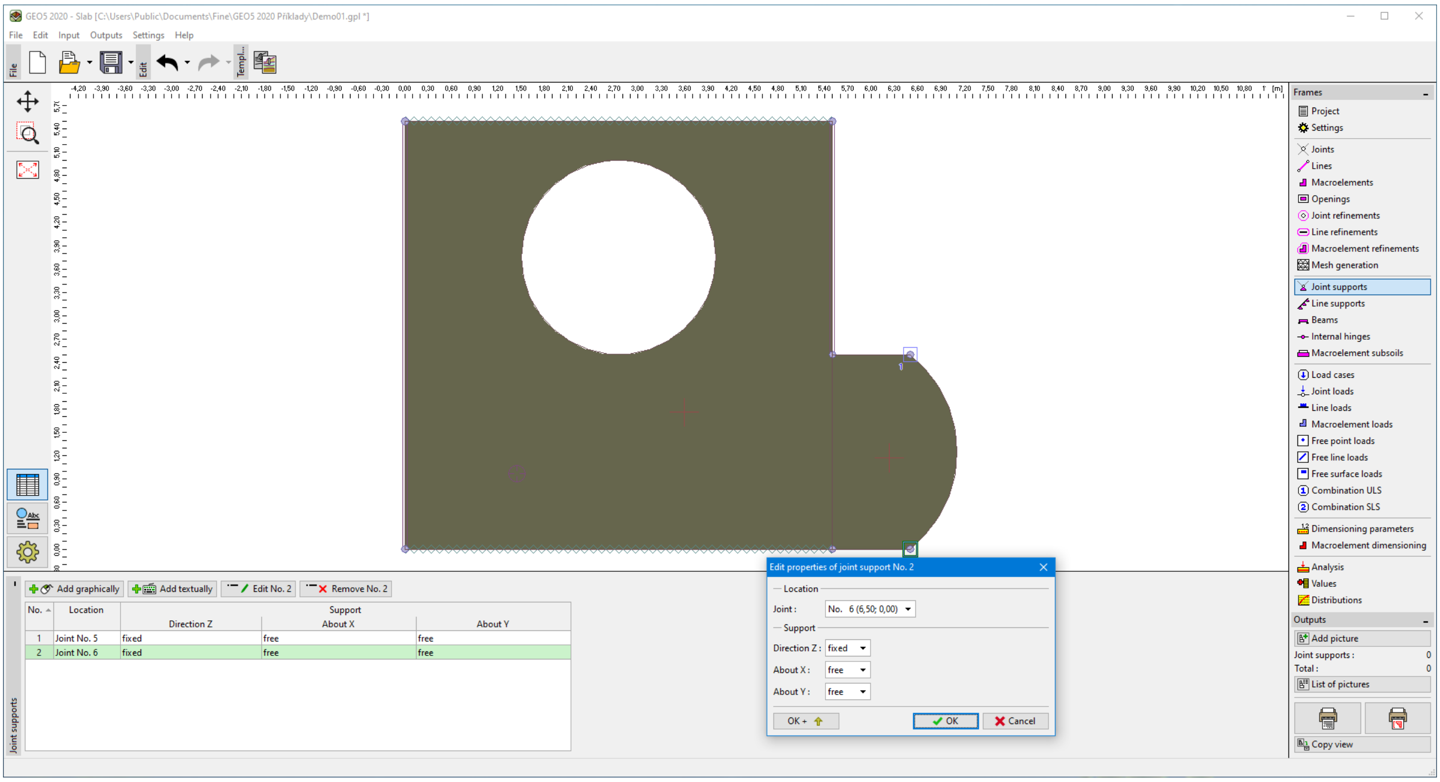
The frame "**Joint supports**" contains a table with the list of input joint supports. Adding joint supports is performed in the "**New joint supports**" dialog window.

The input joint supports can also be edited on the desktop with the help of active objects.

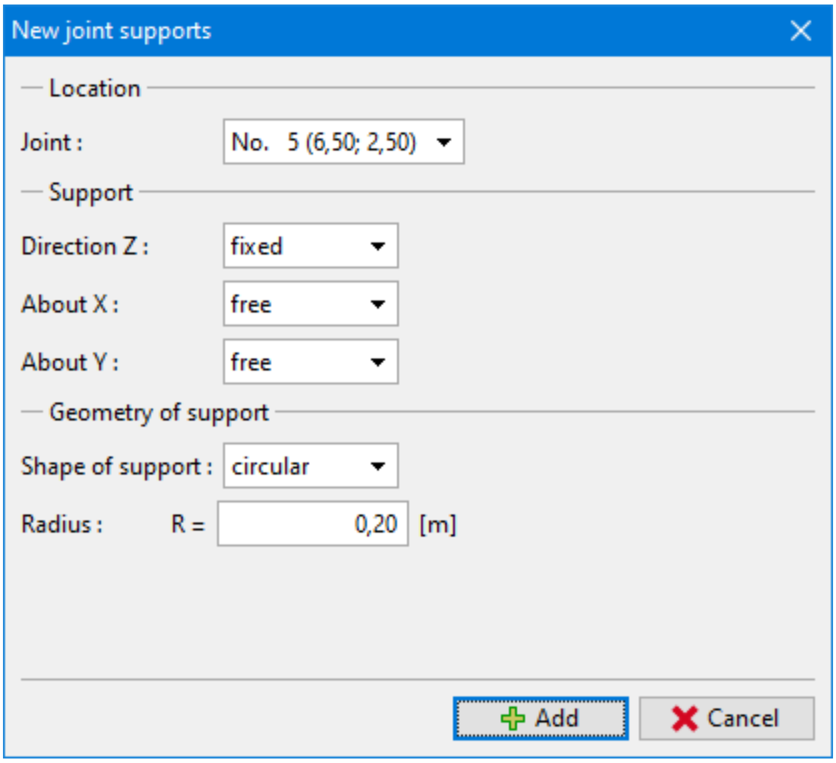
The following types of joint supports are considered:

- free
- fixed
- spring

In case of reduction of dimensioning moments, support geometry can be entered (frame "Settings"). The support geometry can be entered as circular (radius R parameter) or rectangular (parameters dimensions d_x and d_y and rotation α).



Frame "Joint supports"



Dialog window "New joint supports"

Line Supports

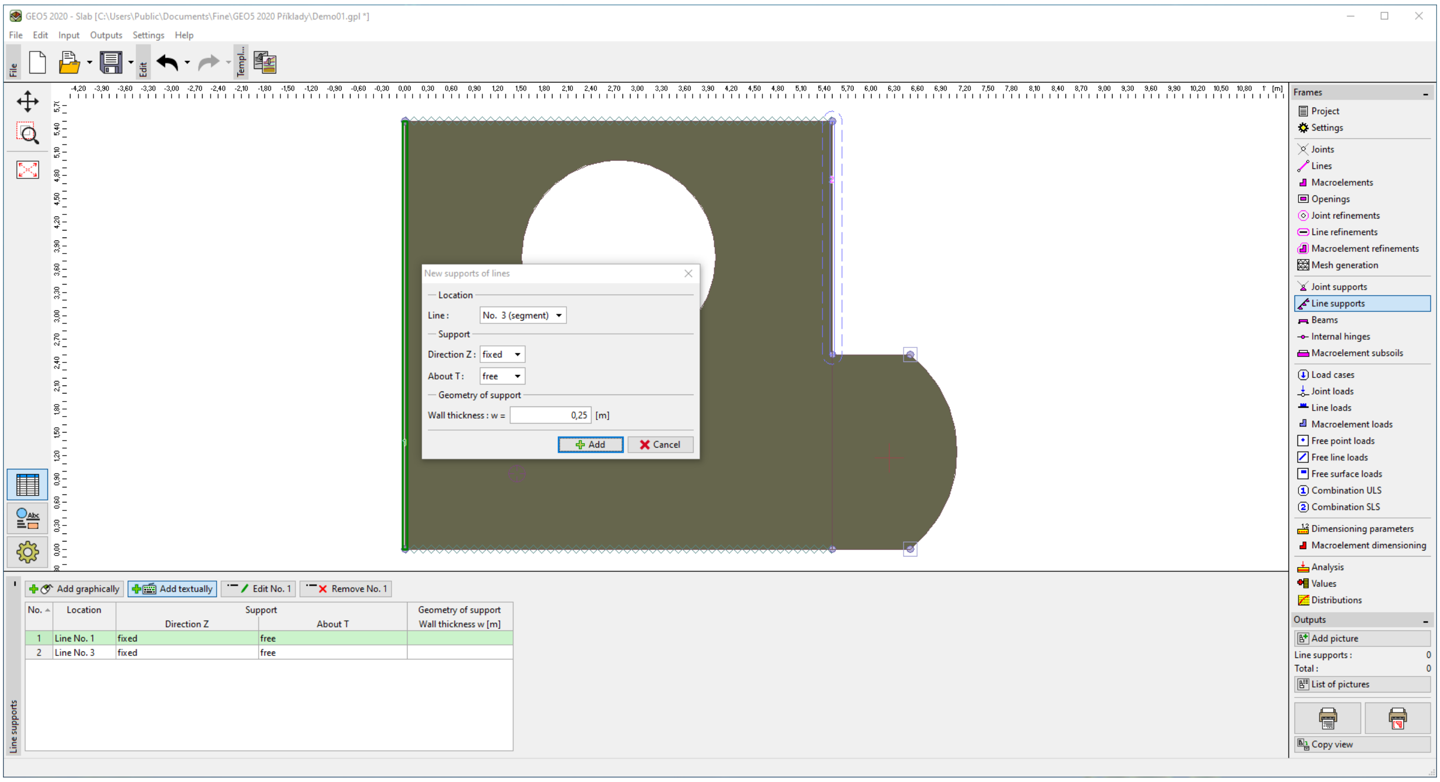
The frame "**Line supports**" contains a table with the list of input line supports. Adding line supports is performed in the "**New supports of lines**" dialog window.

The input line supports can also be edited on the desktop with the help of active objects.

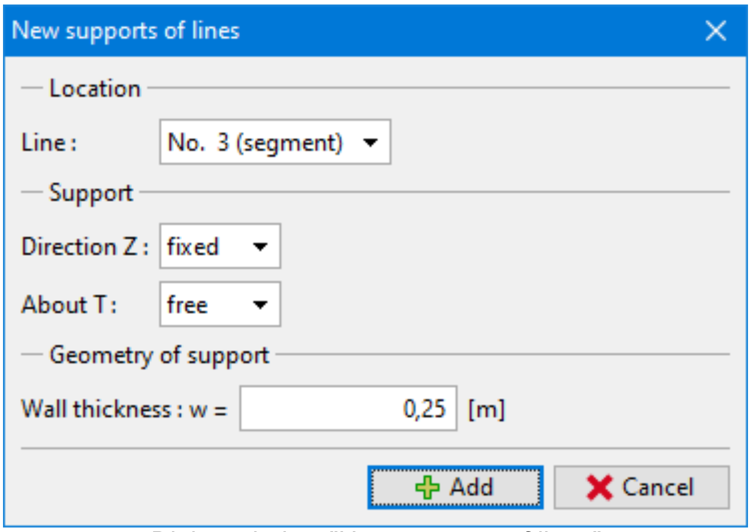
The following types of line supports are considered:

- free
- fixed
- spring

In case of reduction of dimensioning moments, support geometry can be entered (frame "Settings"). Enter the wall thickness w; the circular area is considered at the end of the wall.



Frame "Line supports"



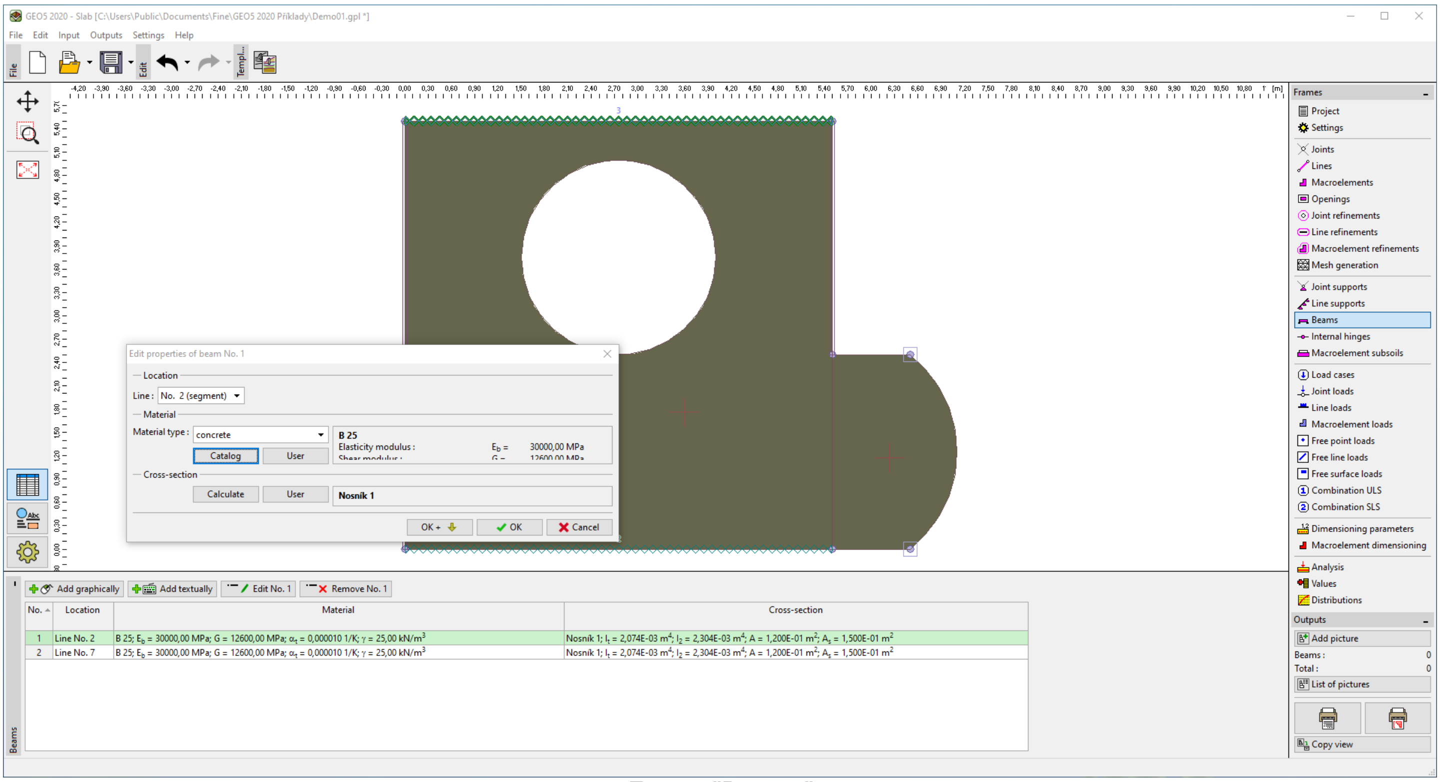
Dialog window "New supports of lines"

Beams

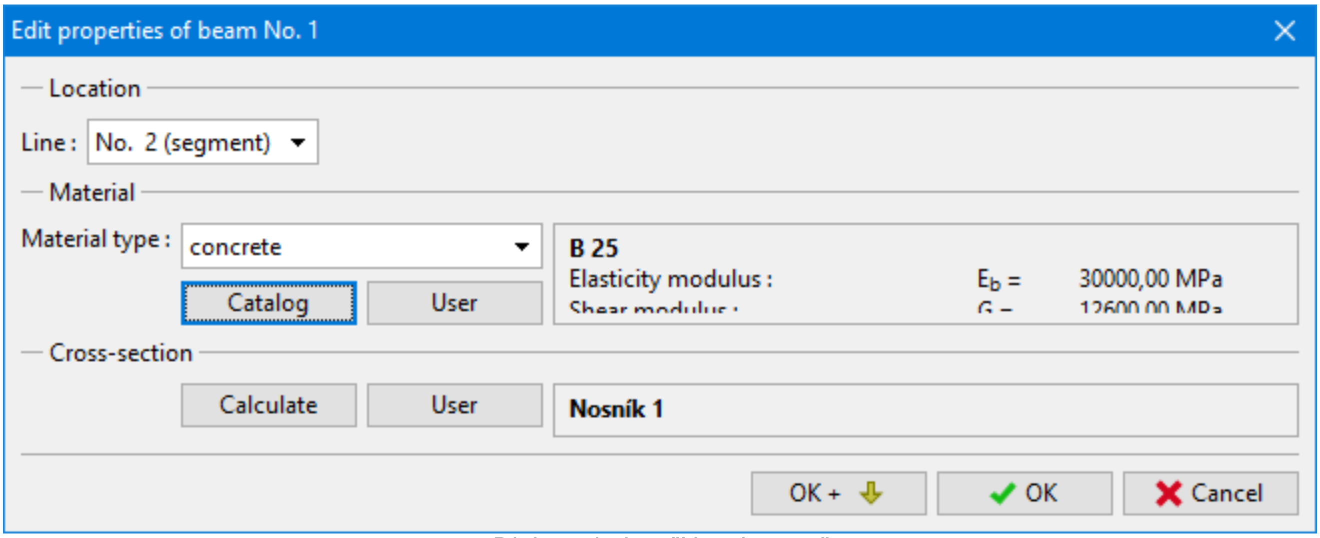
The frame "**Beams**" contains a table with the list of beams. Adding beams is performed in the "**New beams**" dialog window.

The dialog window serves to define the line number of the beam location and material and cross-section of the beam. Choosing the type of material (concrete, steel, other) then allows for assigning the material parameters either from the catalog of materials or manually using the editor of materials. The cross-section parameters (based on the type of cross-section) can be either calculated in the window "**Calculation of cross-sectional parameters**" or manually input in the "**Input of cross-sectional parameters**" window.

Added beams can also be edited on the desktop with the help of active objects.



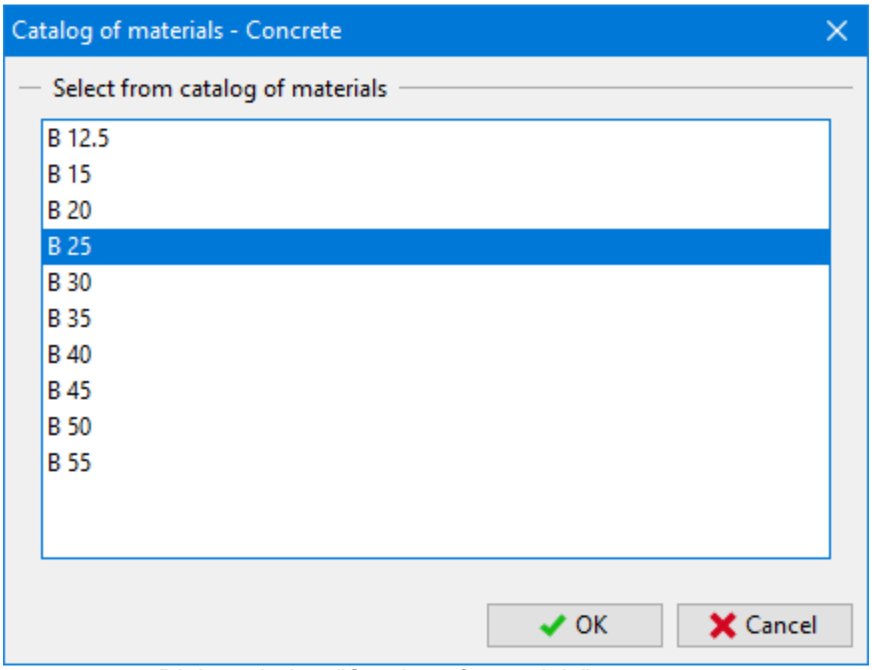
Frame "Beams"



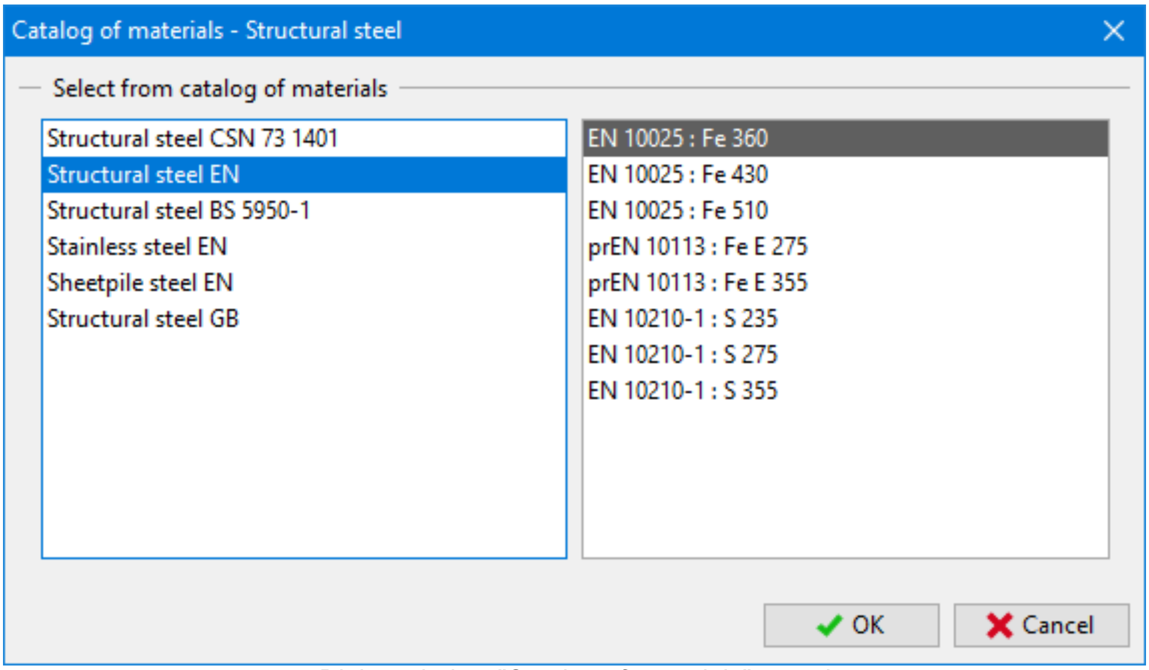
Dialog window "New beams"

Catalog of Materials

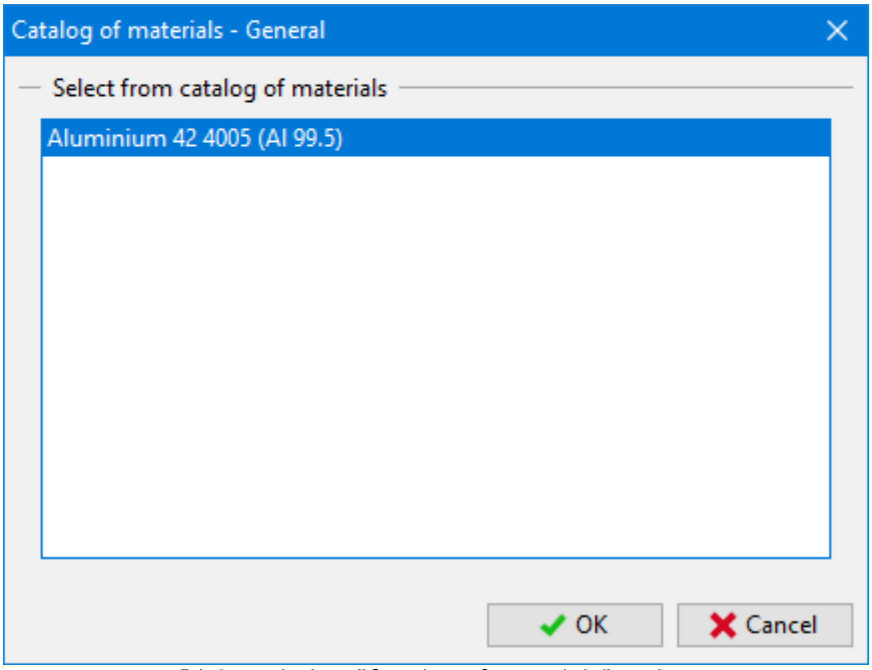
The program contains a built-in catalog of materials for concrete, steel and other materials. Only the type of material has to be specified in the dialog window. The type of cross-section is selected from the "**Calculation of cross-sectional parameters**" dialog window or "**Input of cross-sectional parameters**" dialog window.



Dialog window "Catalog of materials" - concrete



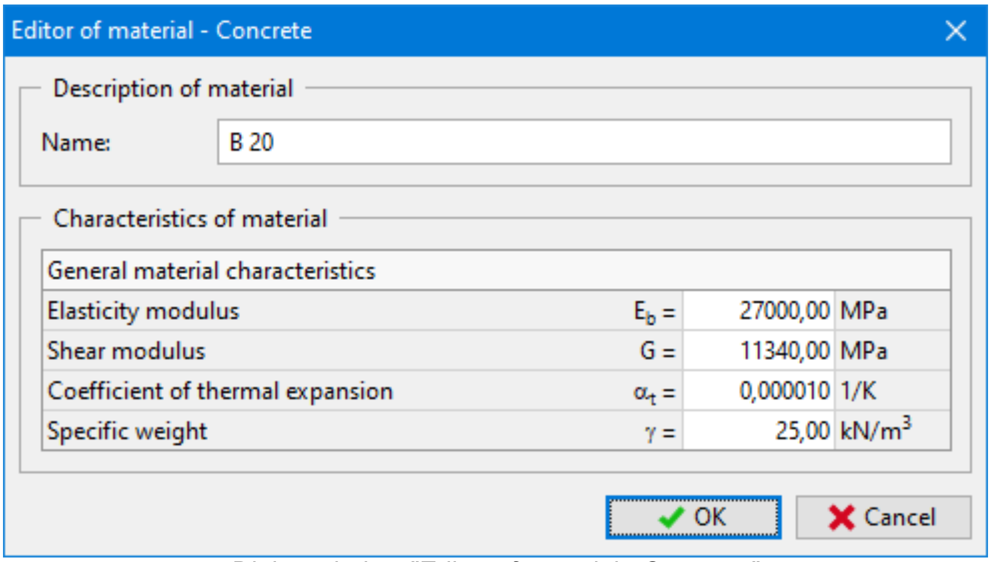
Dialog window "Catalog of materials" - steel



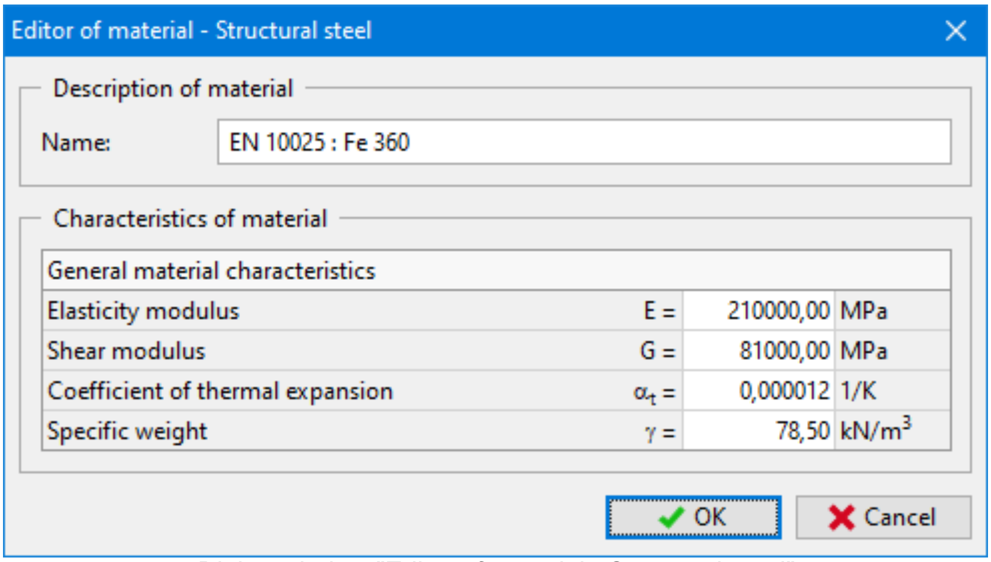
Dialog window "Catalog of materials" - other

Editor of Materials

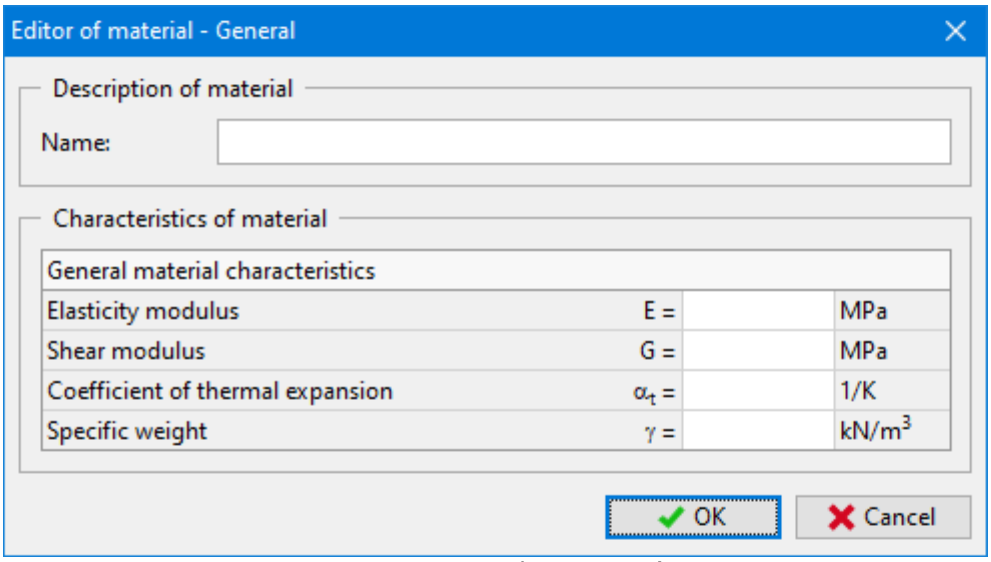
Apart from using the "Catalog of materials" the program allows the user to enter the material parameters for steel, concrete and other materials (dialog window "**Editor of material - General**") digitally. Only the type of material (material parameters) has to be specified in the dialog window. The type of cross-section is selected from the "**Calculation of cross-sectional parameters**" dialog window or "**Input of cross-sectional parameters**" dialog window.



Dialog window "Editor of material - Concrete"



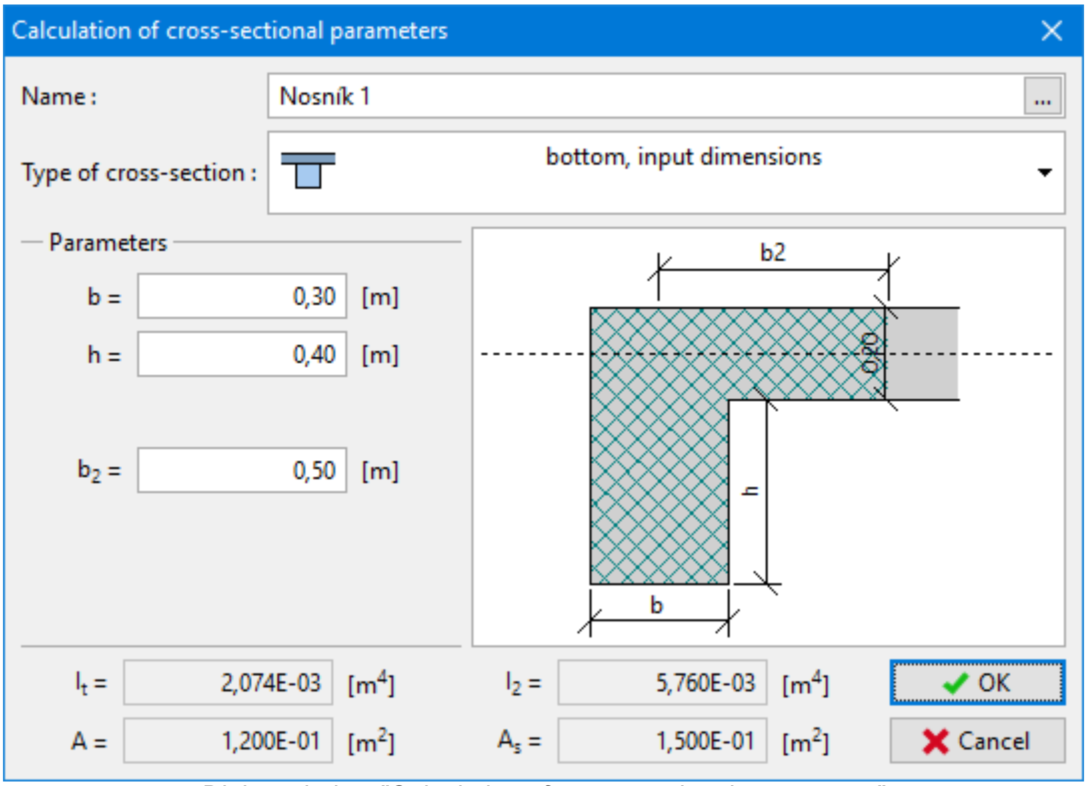
Dialog window "Editor of material - Structural steel"



Dialog window "Editor of material - General"

Types of Cross Section

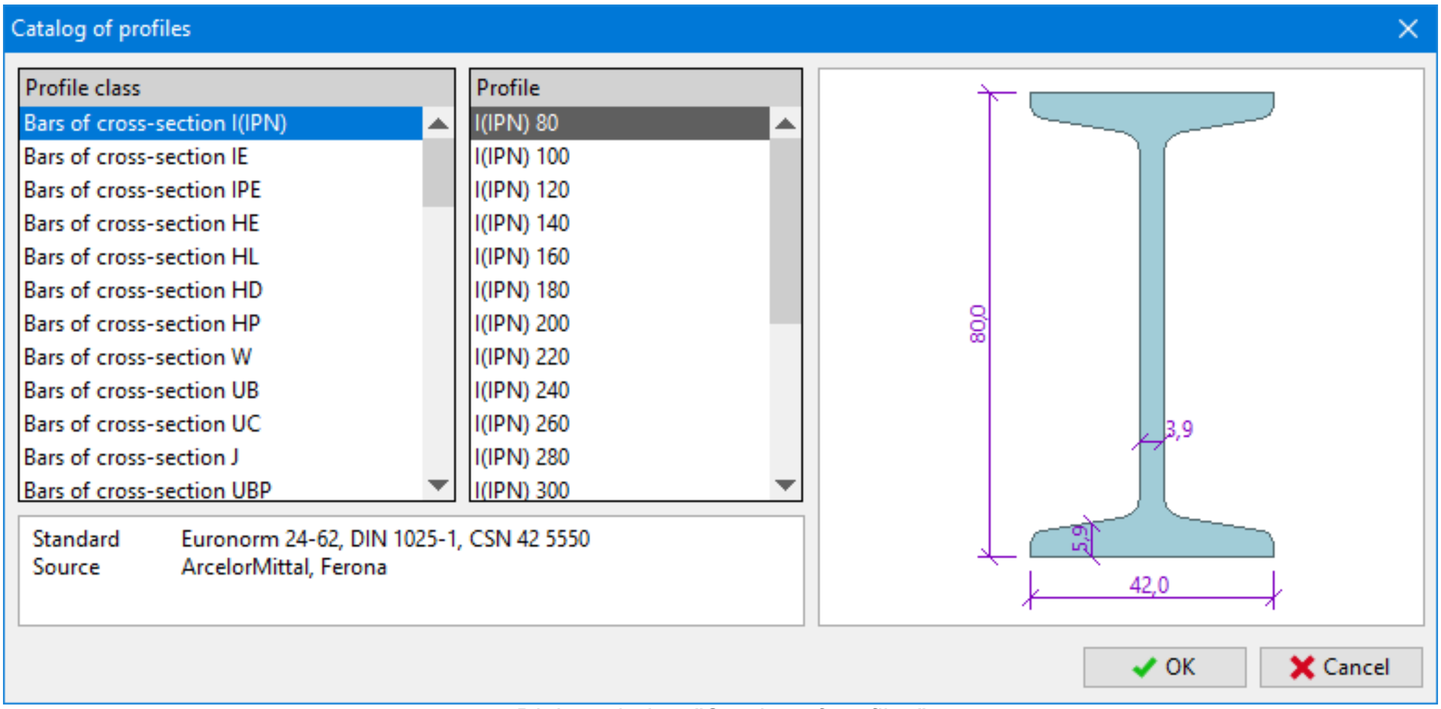
The program allows the user to either input the **cross-section parameters** in "**Calculation of cross-sectional parameters**" and "**Input of cross-sectional parameters**" dialog windows. The cross-sectional characteristics are selected from the catalog of profiles, cross-section editor dialog windows.



Dialog window "Calculation of cross-sectional parameters"

Catalog of Profiles

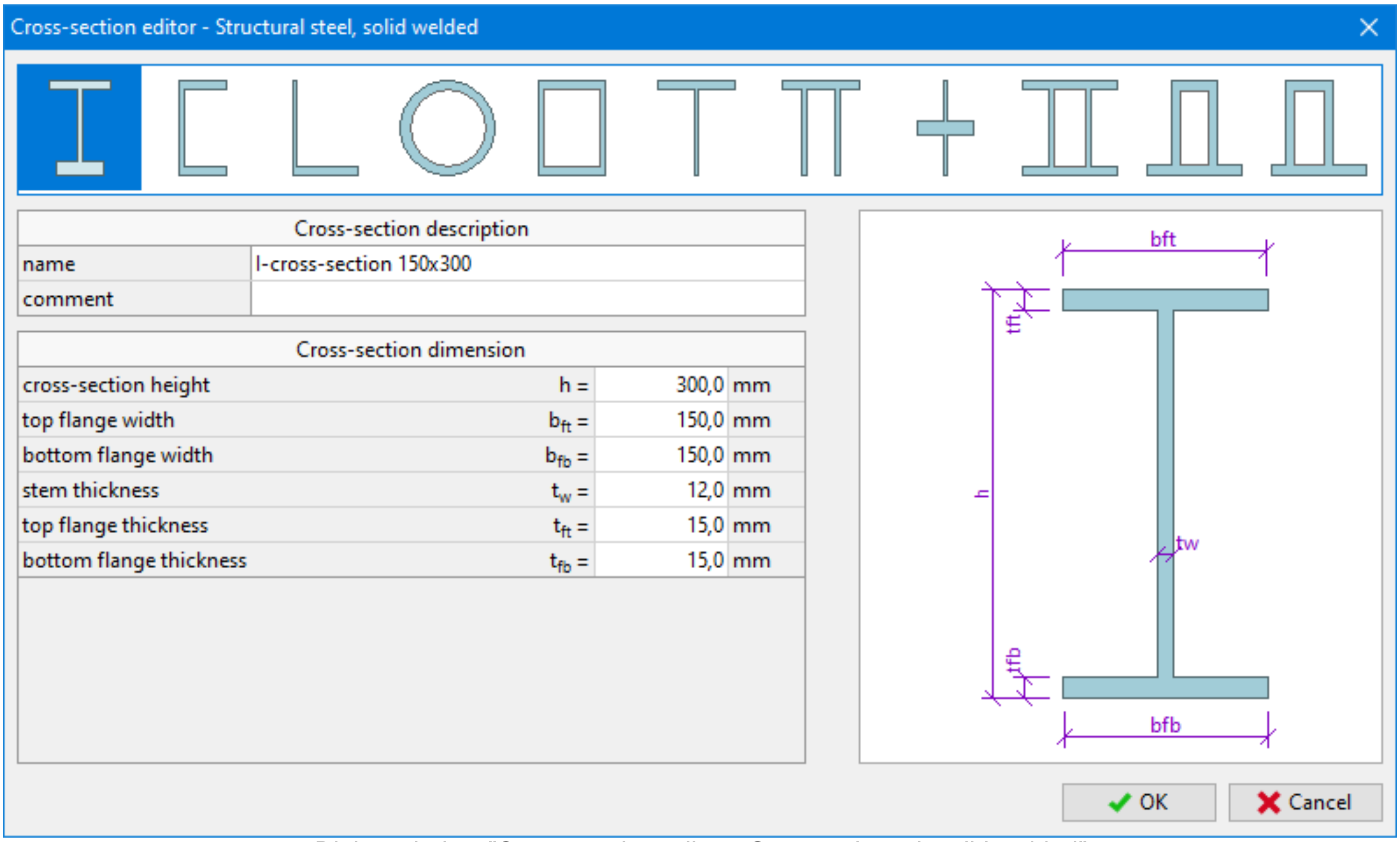
In the case of steel cross-sections the program allows for choosing a particular cross-section from the catalog of profiles. Only the type of cross-section has to be specified in the dialog window. The type of material of the cross-section is selected from the "Catalog of materials", or defined in the "Editor of materials".



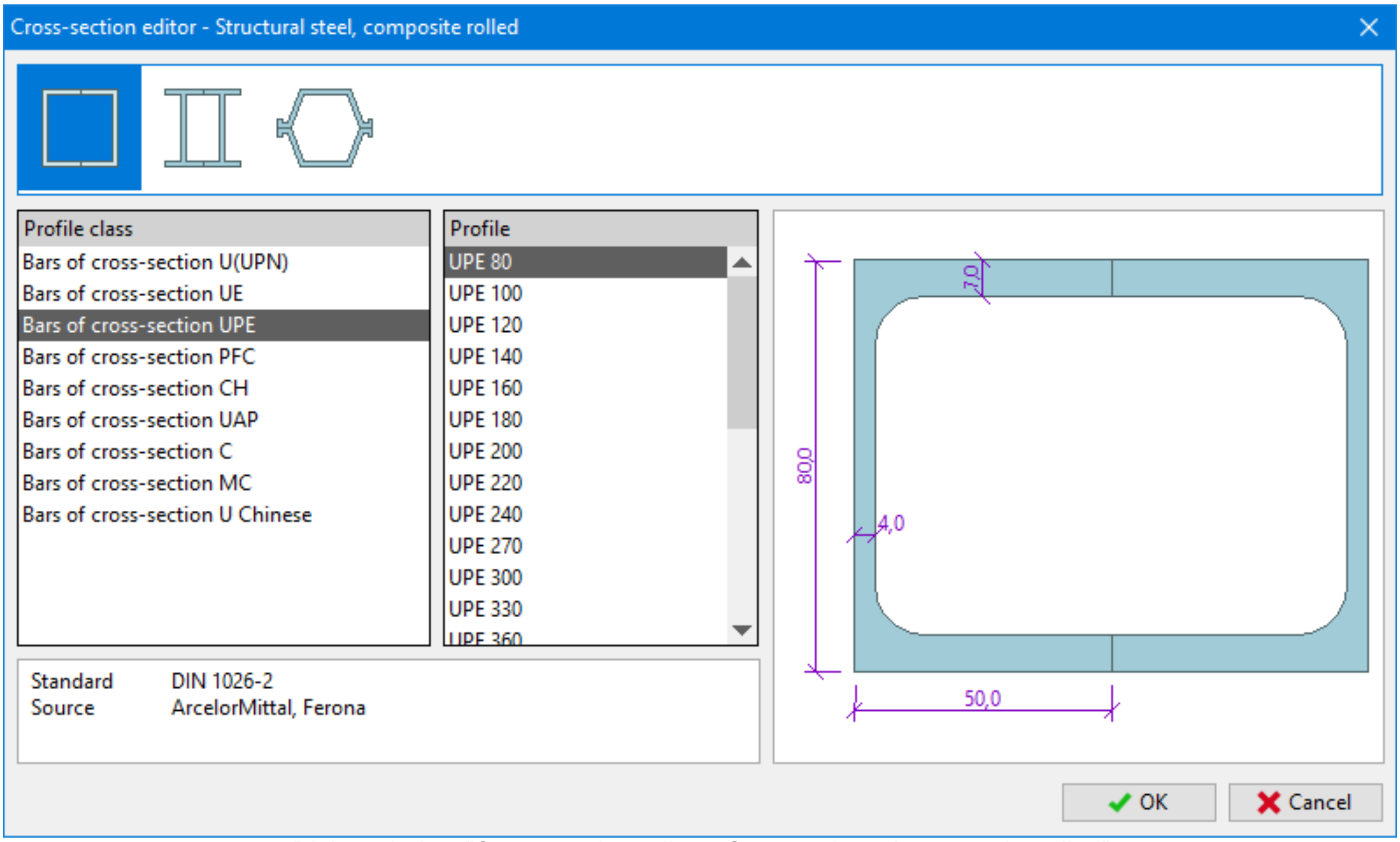
Dialog window "Catalog of profiles"

Cross Section Editor

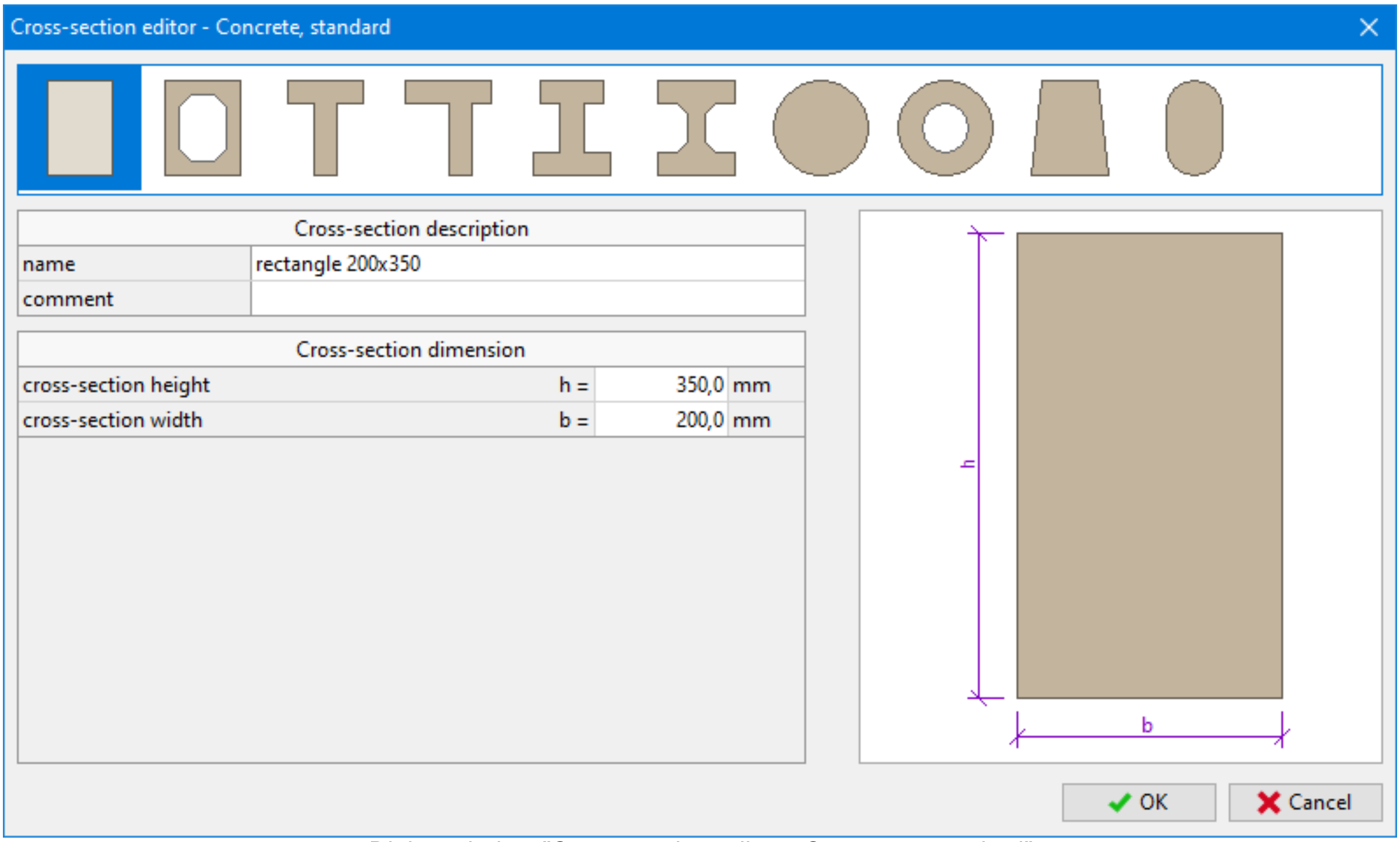
In case of steel and concrete cross-section the program allows for introducing the user-defined cross-section. Only the shape of cross-section has to be specified in the dialog window. The cross-sectional characteristics are selected from the catalog of materials, editor of materials dialog windows.



Dialog window "Cross-section editor - Structural steel, solid welded"



Dialog window "Cross-section editor - Structural steel, composite rolled"



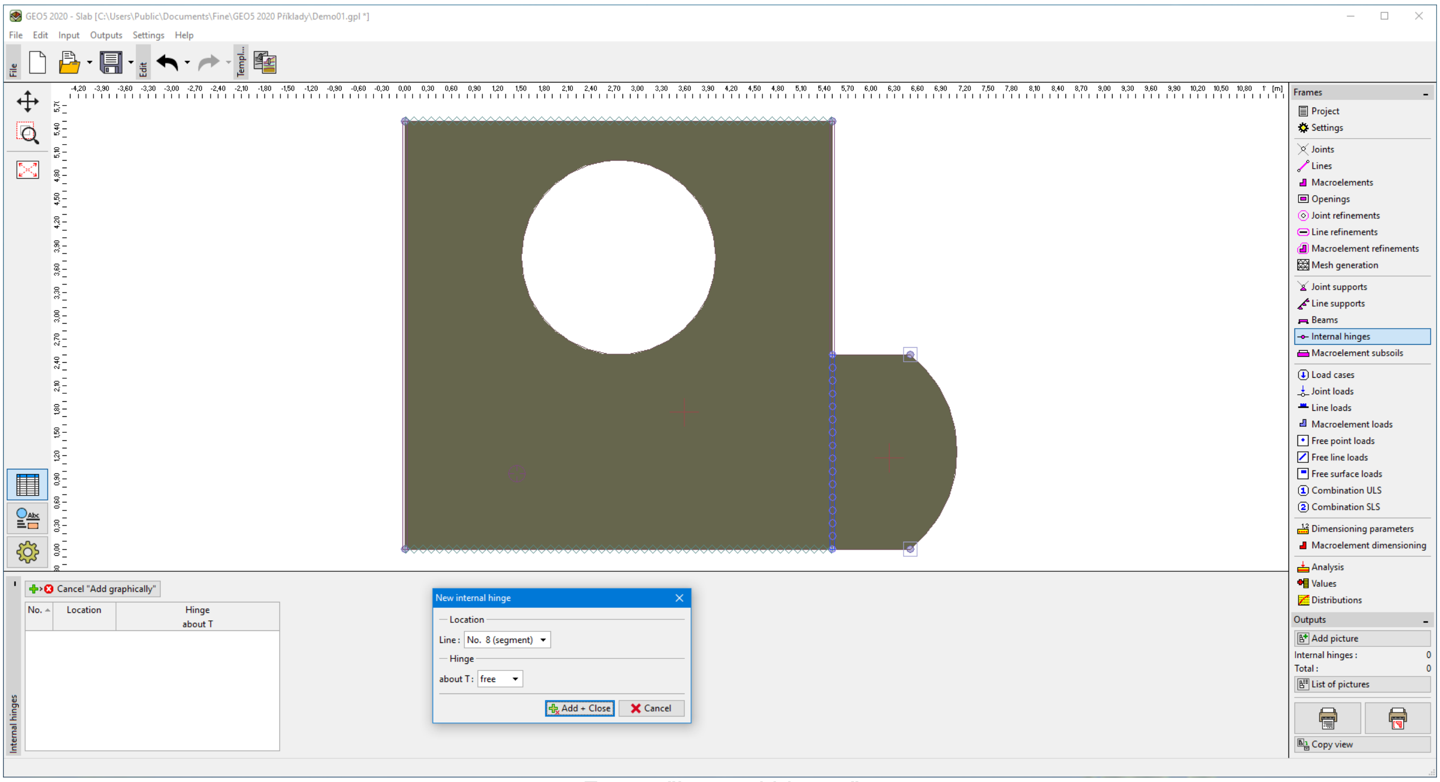
Dialog window "Cross-section editor - Concrete, standard"

Internal Hinges

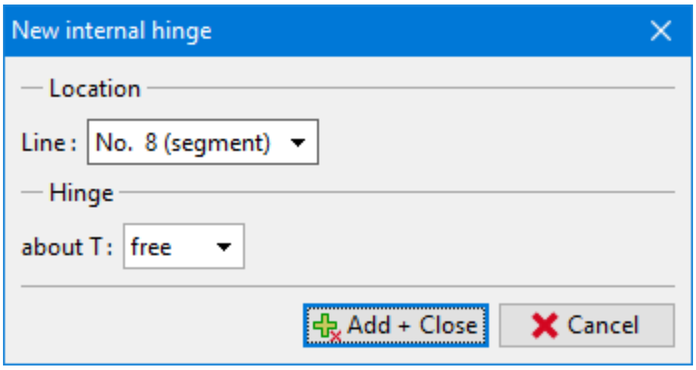
The frame "**Internal hinges**" contains a table with the list of internal hinges. Adding internal hinges is performed in the "**New internal hinges**" dialog window. The dialog window serves to define the line number of the internal hinges location and the type of the internal hinge (free, spring).

The internal hinge is a boundary condition that allows the introduction of independent rotation about the x and y axis between the two parts of the slab along a specified line while keeping the vertical deflection along this line the same. The internal hinge can be prescribed along an arbitrary line creating a boundary between two macroelements. The rotation can be either free or controled by the spring torsional stiffness $K_{\phi,T}$.

The input internal hinges can also be edited on the desktop with the help of active objects.



Frame "Internal hinges"



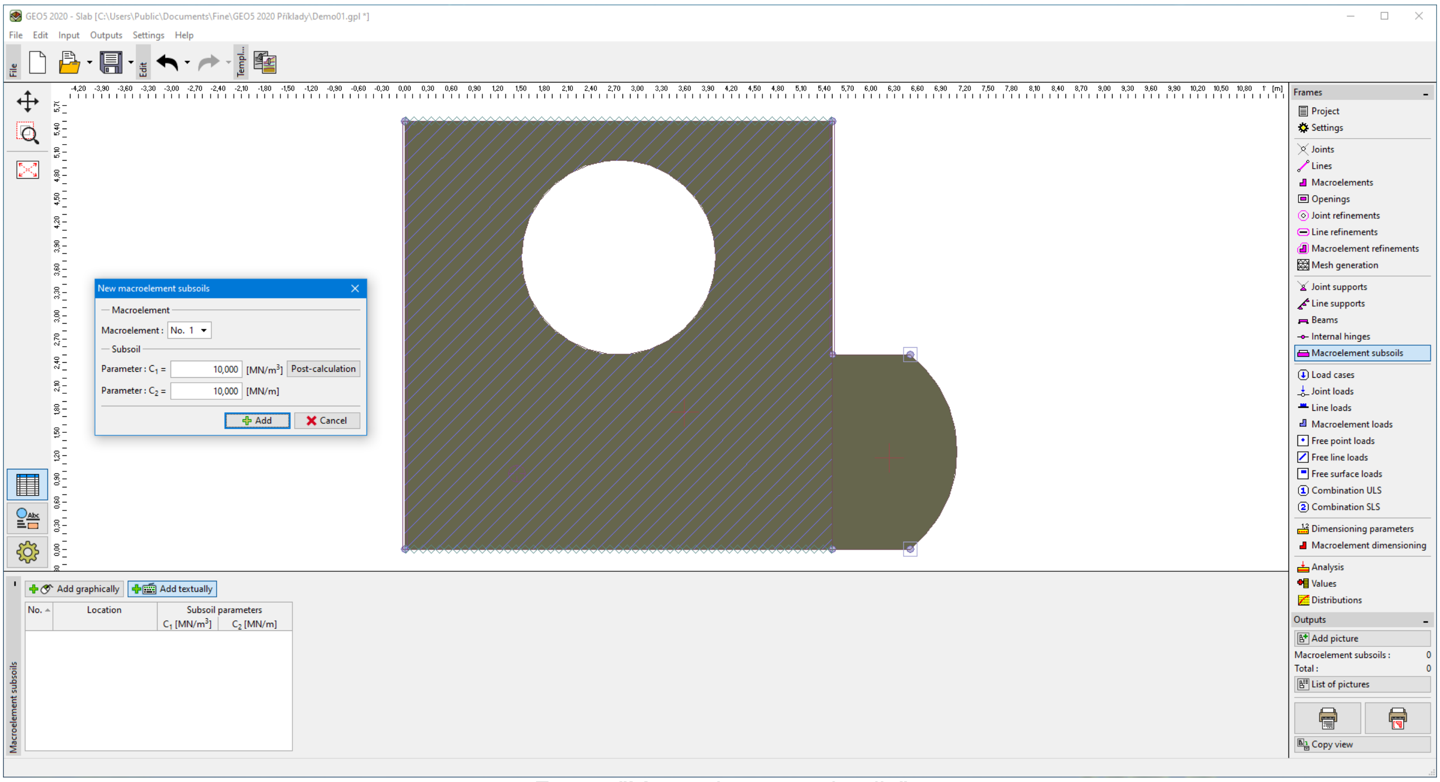
Dialog window "New internal hinge"

Macroelement Subsoils

The frame "**Macroelement subsoils**" contains a table with the list of input macroelement subsoils. Adding line supports is performed in the "**New macroelement subsoils**" dialog window.

The dialog window serves to define the macroelement number and parameters C_1 and C_2. The Winkler-Pasternak constants C_1 and C_2 can be input or calculated by the program. The latter option further requires inputting deformation parameters of soils (deformation modulus, Poisson's ratio and depth of influence zone) in the "**Compute C_1 and C_2**" dialog window. These parameters can be determined using the program "**Spread footing**" (2. limit state) and introduced into the program.

The input macroelement subsoils can also be edited on the desktop with the help of active objects.



Frame "Macroelement subsoils"

Winkler-Pasternak Parameters C1 a C2

The Winkler - Pasternak model for the solution of an elastic layer introduces the balance equation in the vertical direction as:

$$c_1 . w - c_2 . \Delta w = f_z$$

where: c_1, c_2 - constants characterizing the Winkler - Pasternak model

w - displacement in the vertical direction

f_z - vertical load acting on a layer

The elastic subsoil is introduced into the program using local stiffness matrices which are added to the stiffness matrices of individual elements resting on the subsoil. The contact stress σ is provided as an output.

Calculation of Winkler-Pasternak Constants from Deformation Parameters of Soils

The Winkler - Pasternak constants C_1 and C_2 are calculated in the program from the condition of equal compliance matrices of infinitely stiff infinite strip footing resting on the Winkler - Pasternak and elastic subsoil. . The following equations represent this condition:

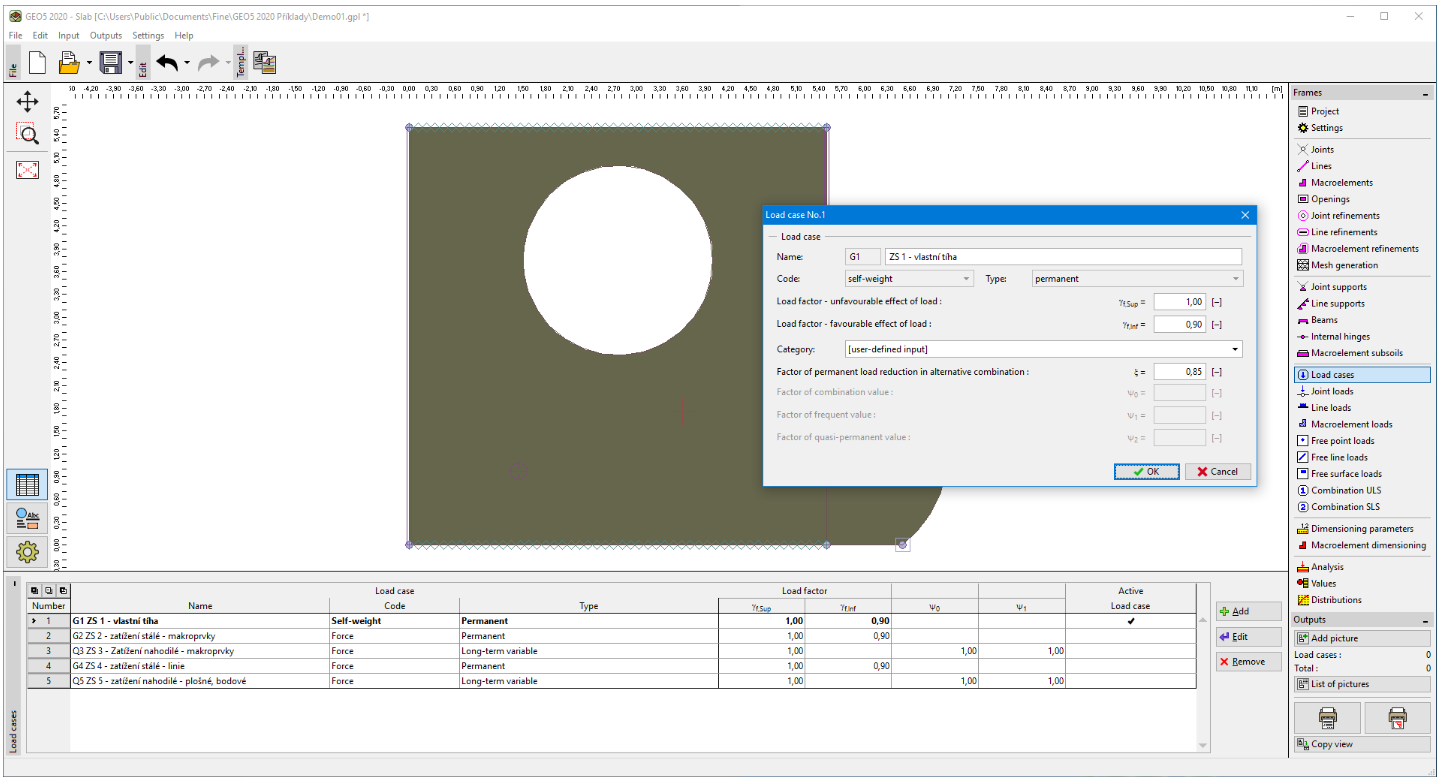
$$[\mathbf{C}] = \begin{bmatrix} \dfrac{1}{2\left[\sqrt{C_{1WP}C_{2WP}} + bC_{1WP}\right]} & 0 \\ 0 & \dfrac{1}{2\left[b^2\sqrt{C_{1WP}C_{2WP}} + bC_{2WP} + \dfrac{b^3}{3}C_{1WP}\right]} \end{bmatrix} =$$

$$= \begin{bmatrix} \sum_{n=0}^{\infty} \dfrac{1}{2\sqrt{H}\left[(2n+1)\sqrt{C_{1w}C_{2w}} + (2n+1)^2 bC_{1w}\right]} & 0 \\ 0 & \sum_{n=0}^{\infty} \dfrac{1}{2\sqrt{H}\left[(2n+1)b^2\sqrt{C_{1w}C_{2w}} + bC_{2w} + (2n+1)^2 \dfrac{b^3}{3}C_{1w}\right]} \end{bmatrix}.$$

where:
- $[C]$ - matrix of constants C_1 a C_2
- b - half-width of foundation
- c_{1w}, c_{2w} - Winkler's constants
- H depth of the deformation zone

Load Cases

The frame "**Load cases**" contains a table with the list of input load cases. Adding load cases and their parameters is performed in the "**New load case**" dialog window. Editing can be carried out with the help of the "Modify" button or by clicking the row with the required load case in the list using the left mouse button.



Frame "Load cases"

Load Case Parameters

The following parameters are defined in the "**New load case**" dialog window:

Load case identifier

The load case identifier, which is composed of the load case number and a uniliteral prefix, is displayed in front of the field for entering the name of the load case. The prefix is determined by the type of load case:

G - permanent load
Q - variable load
A - accidental load

The load case identifier is mainly used in printouts of combinations.

Load case code

The load case code determines what load can be specified for this load. The following options are available.

Self-weight - In this load case the load represents the structure self-weight and it is generated automatically by the program. Only one load case with this code can be considered in each task.

Force - An arbitrary type of force load (forces, moments) can be introduced into the load case with this code. The number of LCs is not limited.

Load type

It determines the character of load cases based on their variability in time. Selecting a particular type of load corresponds to classification according to EN 1990 standard, art. 4.1.1.

Load coefficients

It allows for specifying the load partial factor γ_f. This coefficient accounts for unfavorable deviations of values of loads from the representative ones. For permanent load it is necessary to introduce different values for favorable ($\gamma_{f,\ inf}$) and for unfavorable ($\gamma_{f,\ sup}$) load action in a combination. If the load input follows EN 1990 the default values of coefficients are taken from table A1.2(B).

Category

Classification of load cases into categories corresponds to the classification of load according to table A1.1 of EN 1990 standard. Based on this the variable load cases are assigned combination coefficients ψ_0, ψ_1 a ψ_2. The category of "**User-defined input**" allows for defining the user self-values of these coefficients. Choosing a category is possible only for load cases input, according to EN 1990 (the "Material and standards" tab serves to select the particular standard).

Combination coefficients

Basic values of coefficients to create combinations arise from EN 1990 standard and depend on the load case category. When user input is assumed, it is possible to define the user self-values of these coefficients. The following coefficients are used to create a combination:

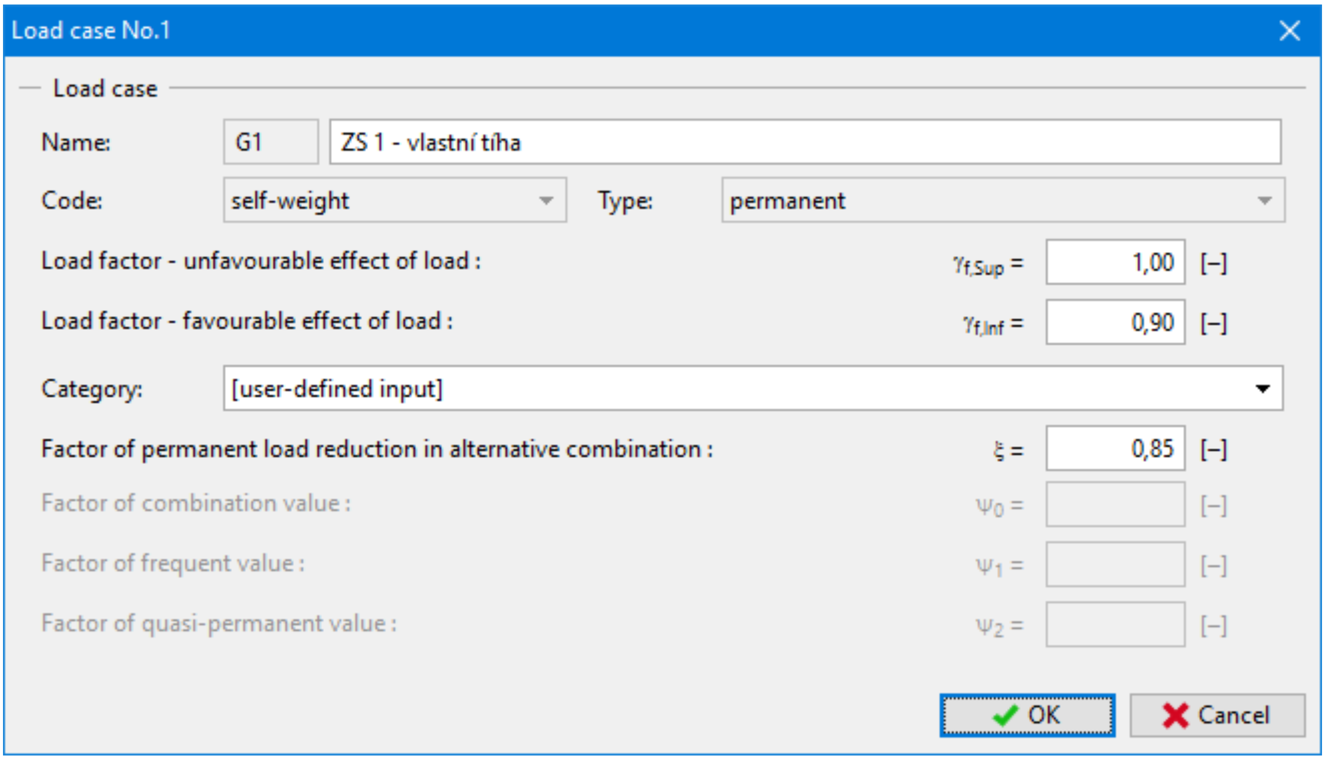
ξ - **Coefficient of reduction of permanent loads in alternative combination** - this coefficient is assigned to all permanent loads and is used when compiling alternative combinations for the bearing capacity limit state (combination to relation 6.10.b, EN 1990).

ψ_0 - **Coefficient of combination value** - coefficient for variable loads, it is used when compiling combinations for both the bearing capacity and service limit states

ψ_1 - **Coefficient of frequent value** - coefficient for variable loads, it is used when compiling accidental combinations and combinations for the service limit state

ψ_2 **Coefficient of quasi-permanent value** - coefficient for variable loads, it is used when compiling accidental combinations and combinations for the service limit state

The combination coefficients are available only for load cases input according to EN 1990 (the "Material and standards" tab serves to select the particular standard).



Dialog window "New load case"

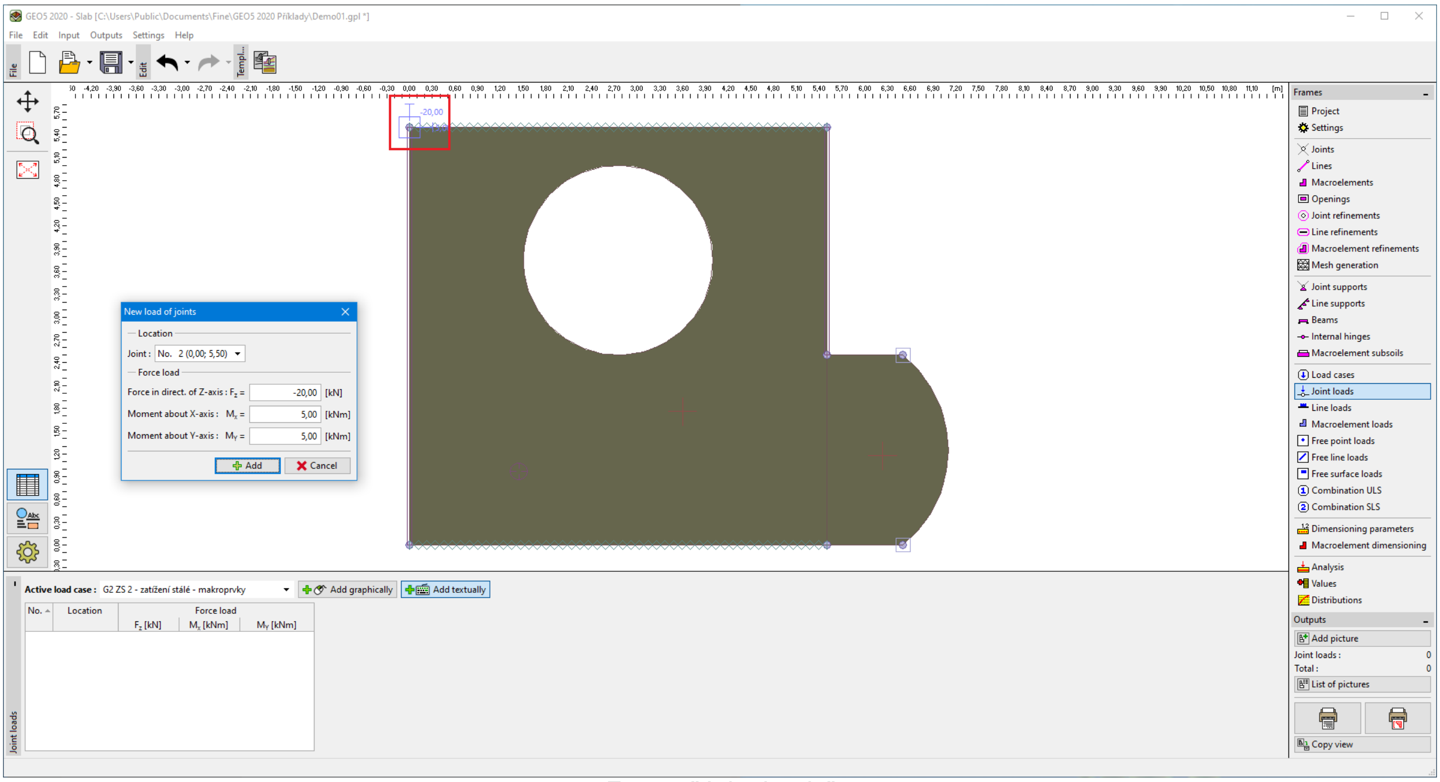
Joint Loads

The frame "**Joint loads**" contains a table with the list of input joint loads. Each joint load is assigned to a certain load case and input joint. The selection of the load is performed in the "**Active load case**" combo list. Adding (editing) joint loads is performed in the "**New load of joints**" dialog window.

The program allows for specifying either mechanical (e.g. forces) or deformational (e.g. prescribed displacements of supports) actions.

The input joint loads can also be edited on the desktop with the help of active objects.

The program employs the following coordinate systems (sign convention).



Frame "Joint loads"

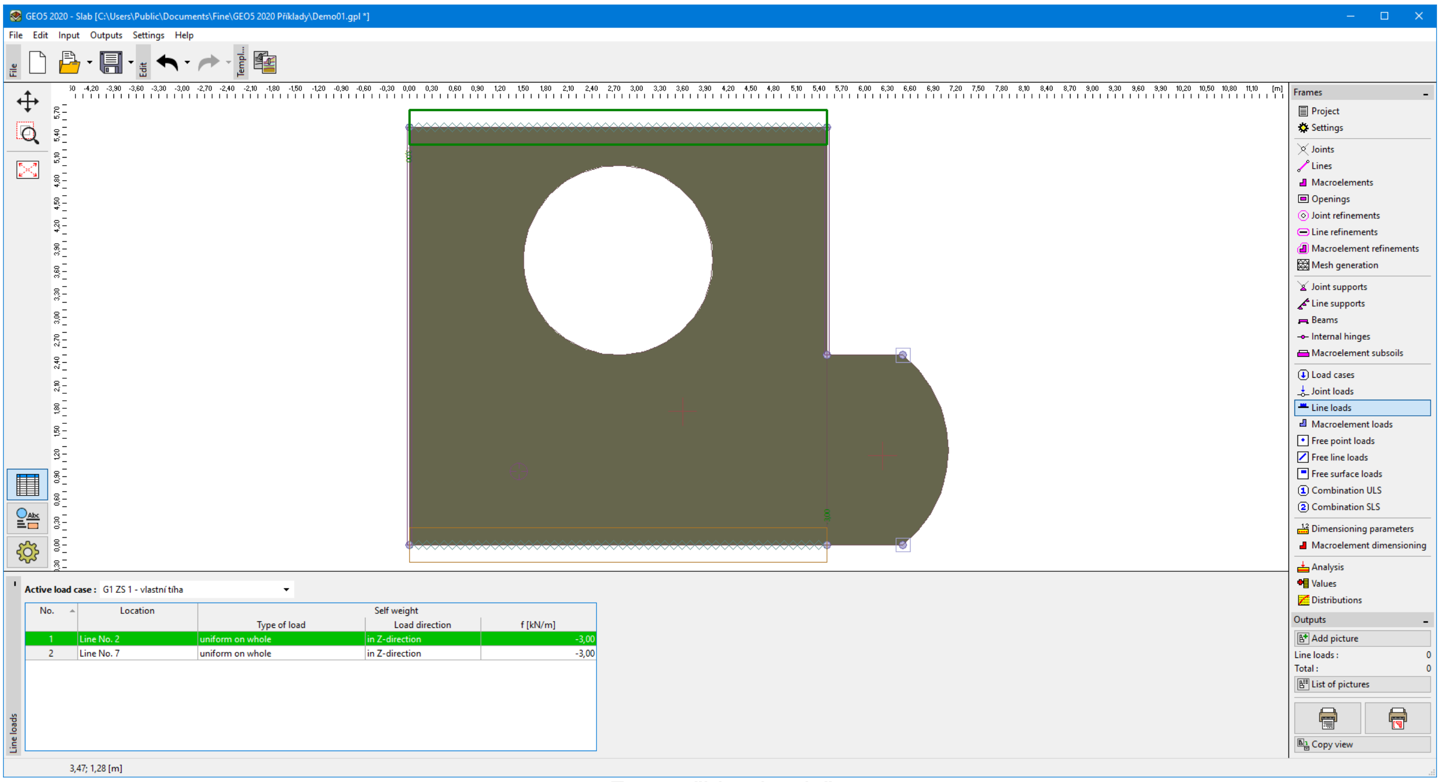
Line Loads

The frame "**Line loads**" contains a table with the list of input line loads. Each line load is assigned to a certain load case and input lines. The selection of the load is performed in the "**Active load case**" combo list. Adding line loads is performed in the "**New loads of lines**" dialog window.

The program allows for specifying either mechanical (e.g. forces), deformational (e.g. prescribed displacements of supports), or temperature actions.

The input line loads can also be edited on the desktop with the help of active objects.

The program employs the following coordinate systems (sign convention).



Frame "Line loads"

Temperature Load

The temperature load assumes a linear distribution of temperature throughout the slab thickness. Such thermal gradient causes moments in the slab given by:

$$m_t = \frac{E.h^2.\alpha.\Delta t}{12.(1-\nu)}$$

where:

E	-	elastic modulus
ν	-	Poisson's ratio
h	-	slab thickness
α	-	coefficient of thermal expansion
Δt	-	temperature difference

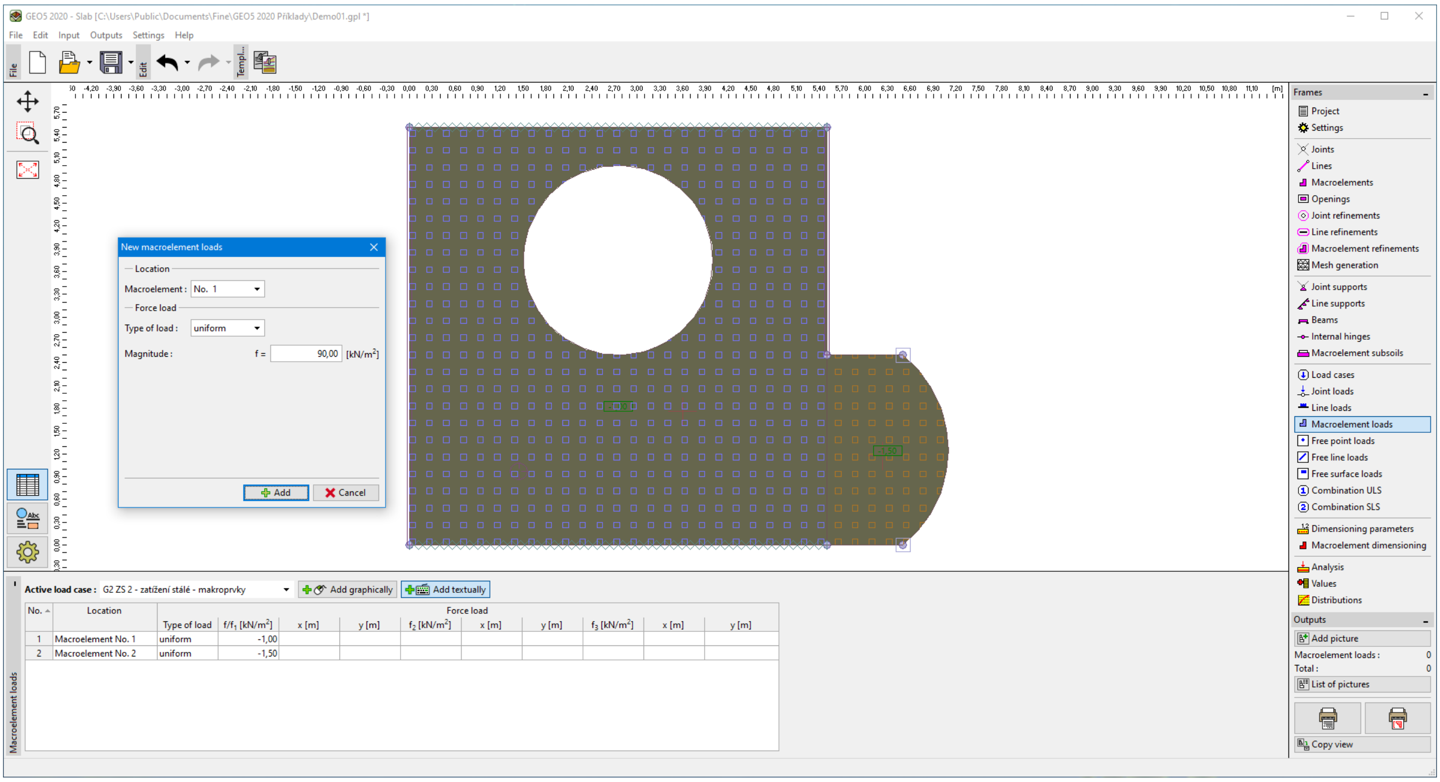
Macroelement Loads

The frame "**Macroelement loads**" contains a table with the list of input macroelement loads. Each macroelement load is assigned to a certain load case and input macroelement. The selection of the load is performed in the "**Active load case**" combo list. Adding macroelement loads is performed in the "**New macroelement loads**" dialog window.

The program allows for specifying either mechanical (e.g. forces) or temperature actions.

The input macroelement loads can also be edited on the desktop with the help of active objects.

The program employs the following coordinate systems (sign convention).



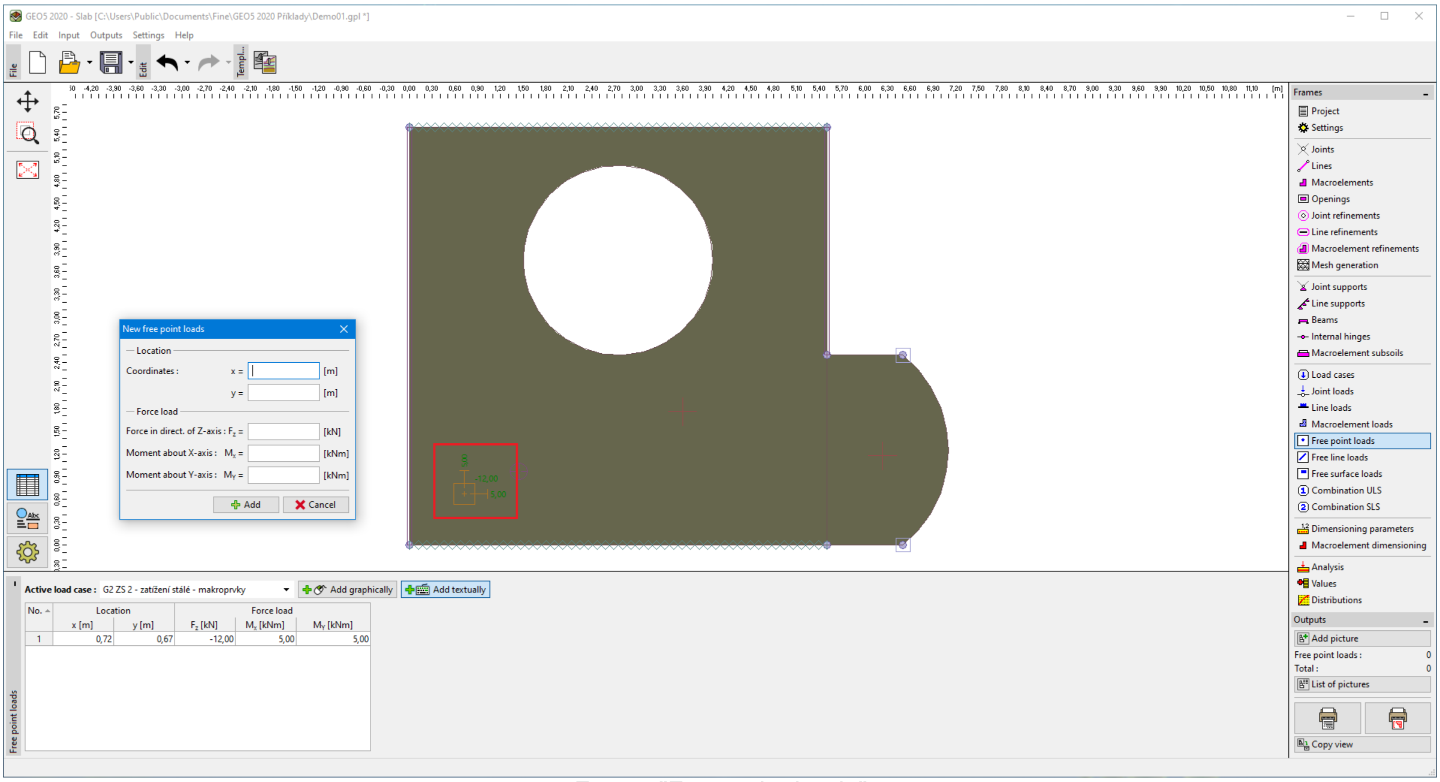
Frame "Macroelement loads"

Free Point Loads

The frame "**Free point loads**" contains a table with the list of input free point loads. Each free point load is assigned to a certain load case and can receive an arbitrary location on the slab surface. The selection of the load is performed in the "**Active load case**" combo list. Adding free point loads is performed in the "**New free point loads**" dialog window.

The input free point load can also be edited on the desktop with the help of active objects.

The program employs the following coordinate systems (sign convention).



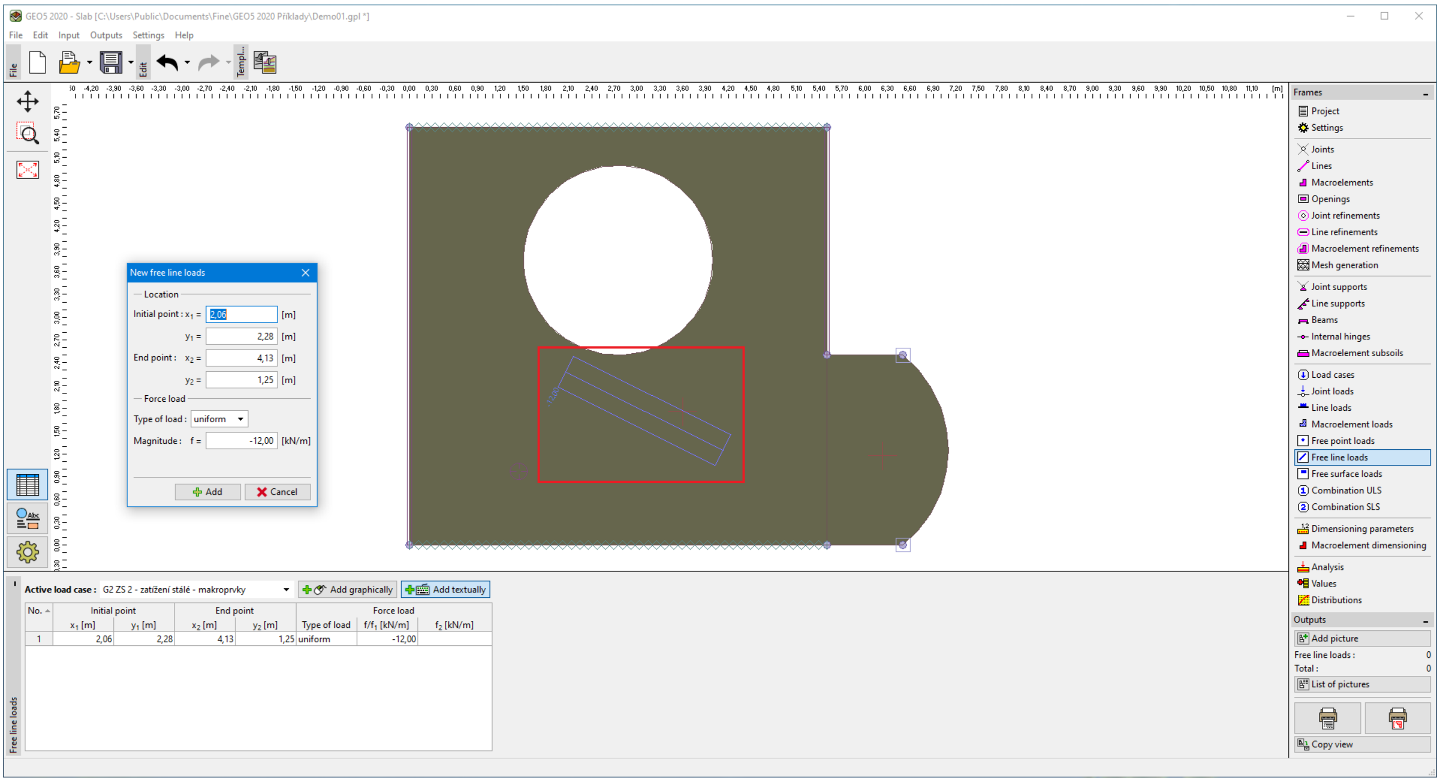
Frame "Free point loads"

Free Line Loads

The frame "**Free line loads**" contains a table with the list of input free line loads. Each free line load is assigned to a certain load case. By input of new points, it can receive an arbitrary shape and location on the slab surface. The selection of the load is performed in the "**Active load case**" combo list. Adding free line loads is performed in the "**New free line loads**" dialog window.

The input free line load can also be edited on the desktop with the help of active objects.

The program employs the following coordinate systems (sign convention).



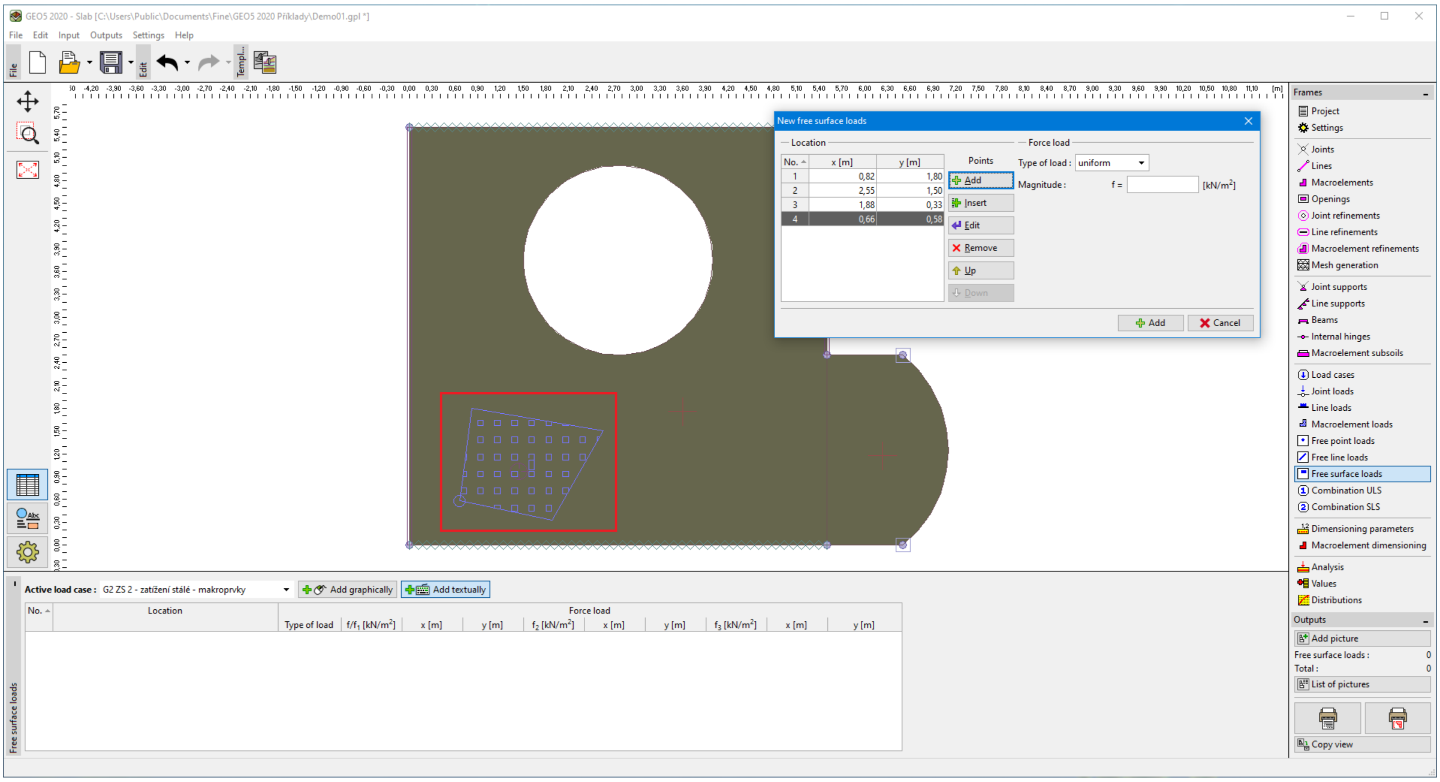
Frame "Free line loads"

Free Area Loads

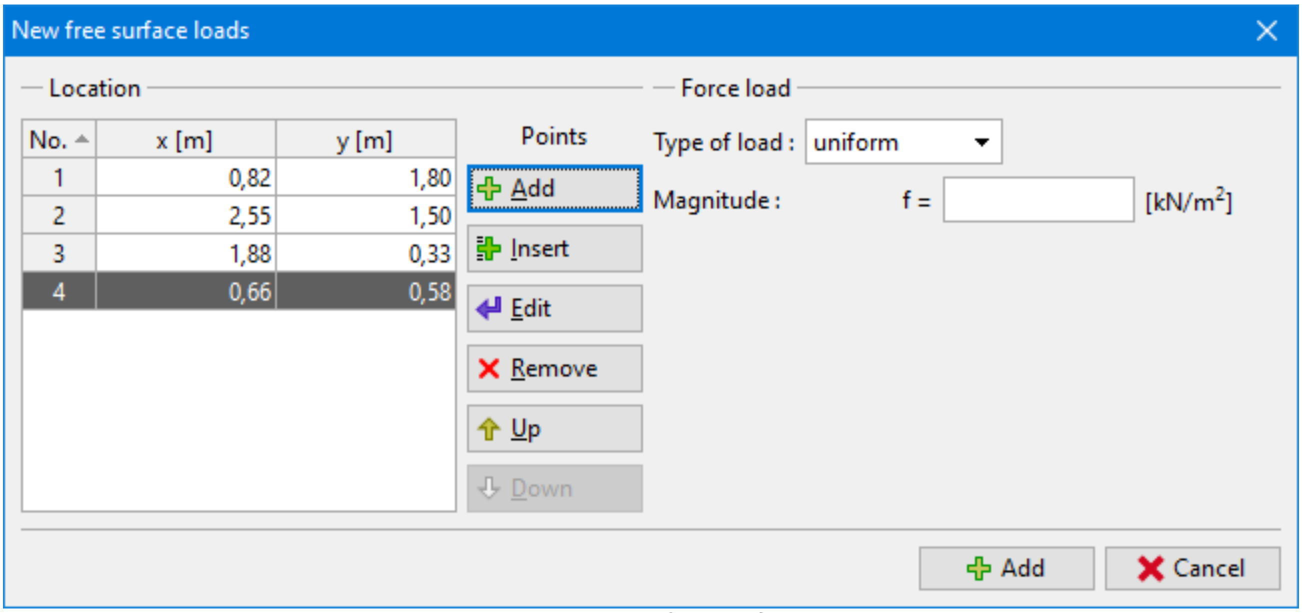
The frame "**Free area loads**" contains a table with the list of input free area loads. Each free area load is assigned to a certain load case. By input of new points, it can receive an arbitrary shape and location on the slab surface. The selection of the load is performed in the "**Active load case**" combo list. Adding free area loads is performed in the "**New free surface loads**" dialog window.

The input free area load can also be edited on the desktop with the help of active objects.

The program employs the following coordinate systems (sign convention).



Frame "Free area loads"

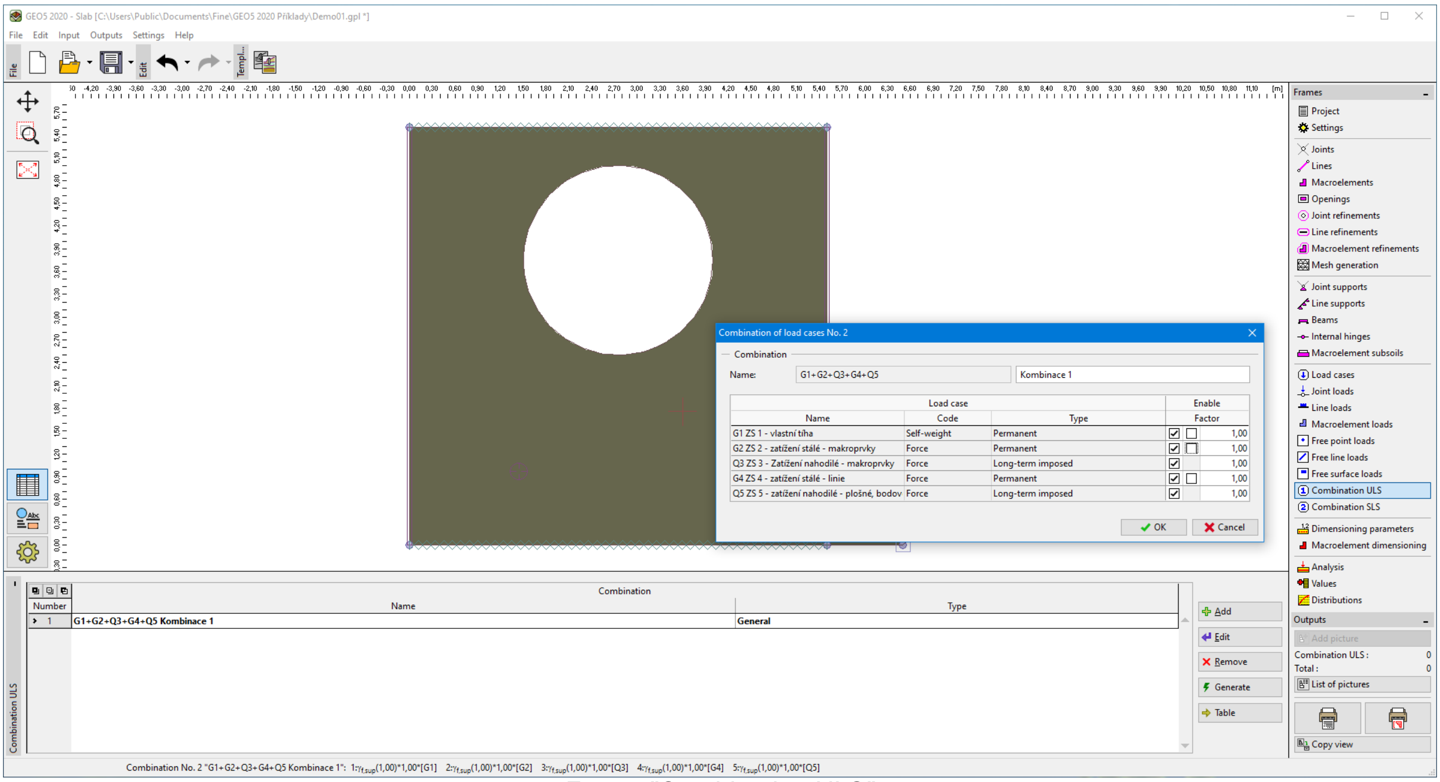


Dialog window "New free surface loads"

Combination ULS

The"**Combinations ULS**" frame contains a table with the list of input combinations of the bearing capacity limit state. Adding (editing) combinations and their parameters is performed in the "**New combination of load cases**" dialog window. Editing can be carried out with the help of the "**Modify**" button or by clicking the row with the required combination in the list using the left mouse button.

The built-in generator of combinations of load cases can be used to create individual combinations.



Frame "Combination ULS"

Parameters of ULS Combinations

The following parameters are specified in the "**New combination of load cases**" dialog window:

New combination

A brief description of combination is displayed in front of the field where the combination is defined. All considered load cases are tagged using their identifiers. The major variable loads are moved at the beginning of the list and separated from the remaining LCs by colon.

Type of combination (for combinations based on EN 1990 only)

The following combinations can be created for the bearing capacity limit state:

Basic - Basic combination based on expression 6.10 of EN 1990 standard

Alternative - Combinations based on expressions 6.10a and 1.10b of EN 1990 standard. In this case, two variants of combination are used in the analysis, one with reduced permanent LCs and the other with reduced major variable LC.

Accidental - Accidental combination based on 6.11 of EN 1990 standard.

Selection of load cases

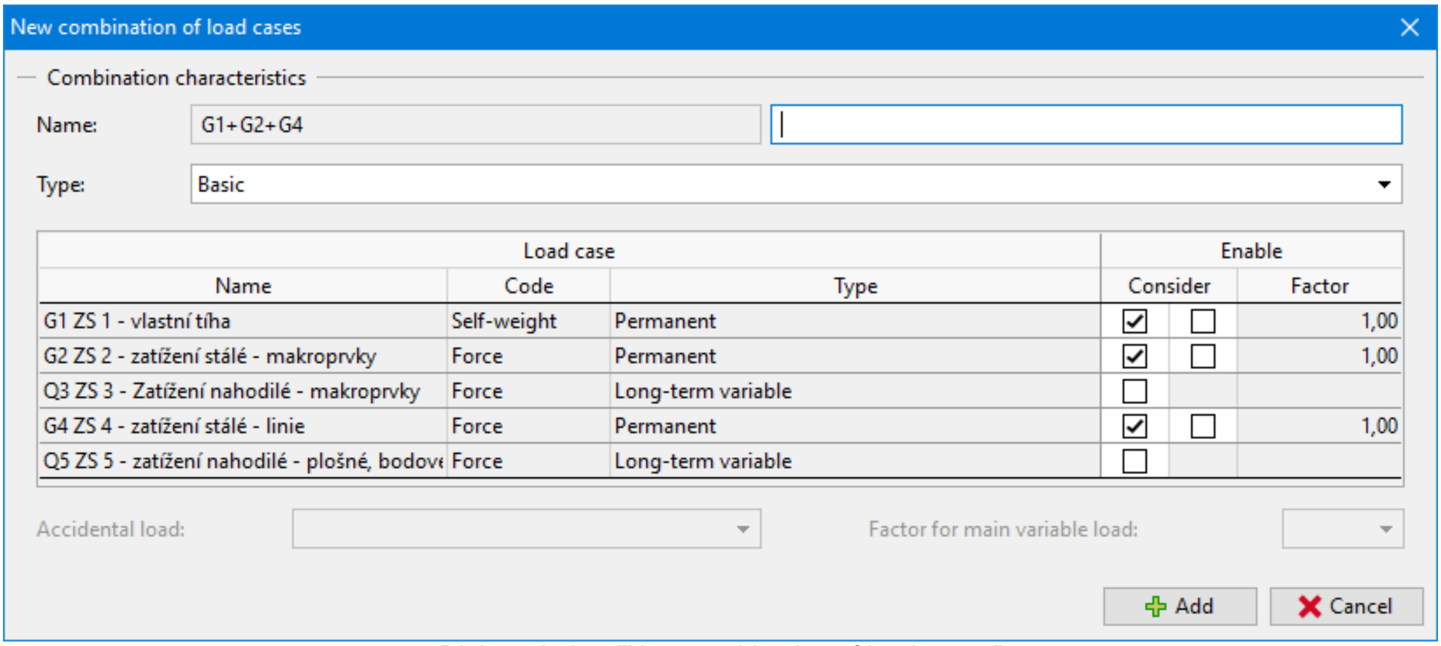
The table listing individual load cases allows for their selection to create a combination. The load case can be introduced into a combination by checking the field in the column "**Consider**" for a particular LC. Further setting in the table depends on the selection of way of inputting loads in the "Material and standards" tab.

Load according to EN 1990

A second field is available for each load case in the column "**Consider**". This field allows for assigning a favorable effect of action to permanent LCs (adopting coefficient $\gamma_{f,\, inf}$) or for specifying a variable load as the major one, respectively. The number of major variable loads in the combination is not limited. An accidental load can be introduced into combinations tagged as "**Accidental**" (only LCs tagged as "**Accidental**" are available for the selection). For accidental combinations, it is also necessary to choose whether a major variable load should be reduced by the coefficient ψ_1 or ψ_2.

General load

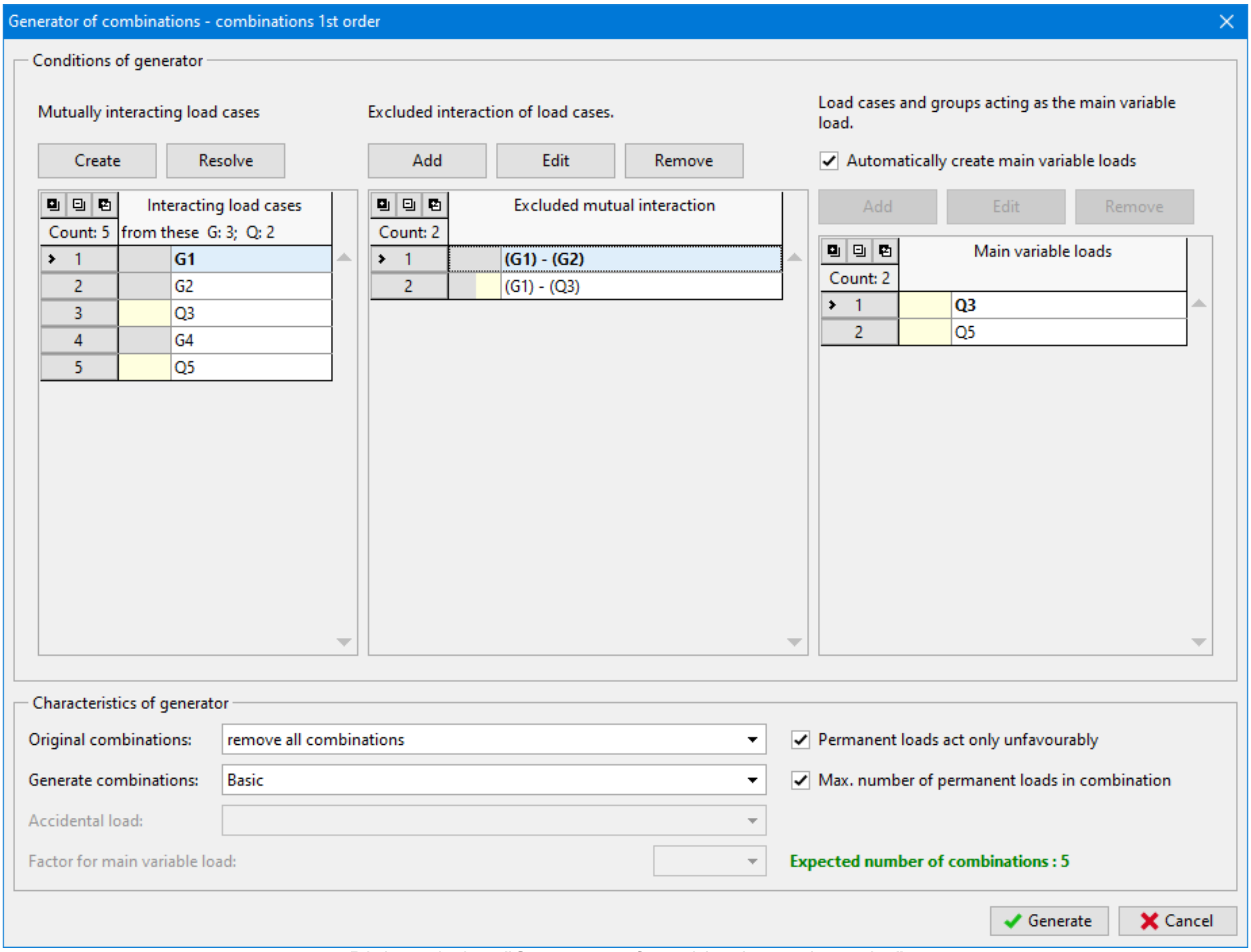
A coefficient of usability can be specified for each load case to adjust the degree of usability of the load case in the combination.



Dialog window "New combination of load cases"

Generator of ULS Combinations

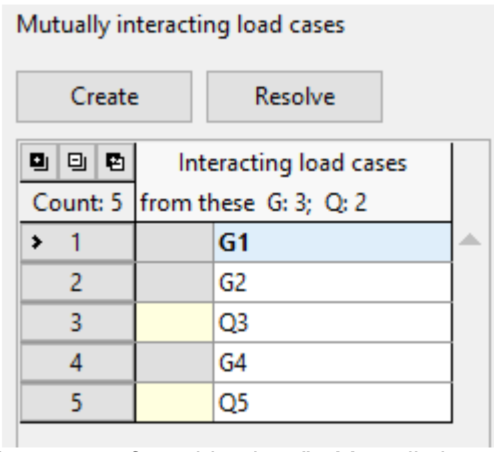
The "**Generator of combinations - 1st order**" dialog window allows for a collective compilation of combinations of load cases based on the introduced combination rules. Referring to standard EN 1990 the number of generated combinations can be relatively large and in extreme cases could considerably slow down calculations. Due to it, the information about expected number of combinations to be generated is displayed in the right bottom corner. Therefore, before launching a generation the user may check how many combinations will be generated and possibly adjust generator conditions. The top part of the window serves to define conditions for generating combinations; the bottom part contains various generator settings.



Dialog window "Generator of combinations - 1st order"

Mutually interacting load states and groups

This part makes it possible to merge those load states that should appear in combinations always together. Permanent and variable loads cannot be merged into one group. If the field **All permanent loads always in combination** is checked in the Generator parameters, the creation of groups of permanent loads has no effect on their appearance in combinations as each generated combination will always contain all permanent LCs. In such case, merging permanent LCs will only influence consideration of favorable/unfavorable effects of LCs providing the field **Permanent loads act only unfavorably** is not checked.



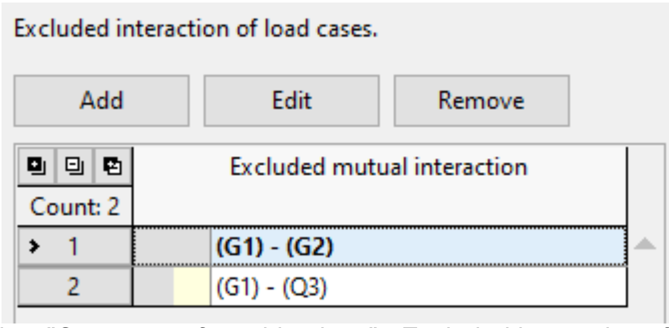
Dialog window "Generator of combinations" - Mutually interacting load states

Excluded interaction of load states

This part makes it possible to define, which LCs should not appear in a combination together. Arbitrary load cases or merged groups can be mutually excluded in dependent of the type of load case. Two options are available to define groups to be excluded:

Mutual exclusion - An arbitrary number of load cases can be introduced into one group. In such a case, the program will not generate any combination that contains at least two load cases from this group.

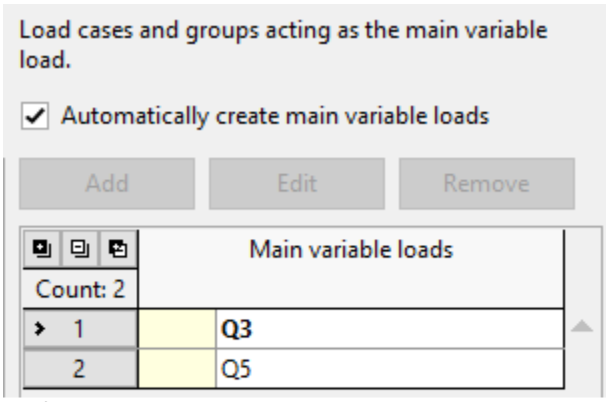
Exclusion by pairs - Providing it is necessary to create a larger number of excluding groups of two sorts, where one LC is the same (e.g. exclusion of assembly variants of permanent loads with all service load cases), it is possible to adopt this option. A load case to be excluded is first selected in the first column. The second column is then used to select an arbitrary number of LCs, which are needed to create excluding groups.



Dialog window "Generator of combinations" - Excluded interaction of load states

Load cases and groups acting as the main variable load

This part is available only when inputting loads according to EN 1990 is considered (the standard is selected in the "Material and standards" tab). When an automatic regime is assumed, then each variable load is taken as major in created combinations. If this regime is turned off, it is possible to manually adjust the list of major variable loads. For example, it is possible to remove an arbitrary load case from the list so that it will not be considered as major variable in combinations. If a new item with more load cases is add to the list then all load cases will be considered as major variable in those combinations, where they appear together.



Dialog window "Generator of combinations" - Load cases and groups acting as the main variable load

Generator parameters (parameters that can be set in the bottom part of the dialog window)

Combo list "Original combinations"

Retain original combinations - By pressing the "**Generate**" button the program will add new combinations, created according to the specified rules, to the original ones

Remove all combinations - By pressing the "**Generate**" button the program will delete all original combinations and will replace them by the new ones

Remove generated combinations - By pressing the "**Generate**" button the program will delete older combinations and will add new ones created according to the specified rules

Remove all combinations of the current type - By pressing the "**Generate**" button the program will delete all combinations of a given type and will replace them by the new ones

Remove generated combinations of the current type - By pressing the "**Generate**" button the program will delete older combinations of a given type and will add new ones created according to the specified rules

Combo list "Generate combinations"

The following types of generated combinations can be chosen for loads based on EN 1990:

Basic - Generates basic combinations for the bearing capacity limit state based on expression 6.10 of EN 1990 standard

Alternative - Generates combinations for the bearing capacity limit state based on expressions 6.10a and 1.10b of EN 1990 standard. This variant generates two times more combinations but it provides better results.

Accidental - Generates accidental combinations for the bearing capacity limit state based on 6.11 of EN 1990 standard. An accidental load case to appear in accidental combinations can be specified. It is also necessary to choose whether a major variable load will be reduced by the coefficient ψ_1 or ψ_2.

Permanent loads act only unfavorably

If this setting is not checked, the program creates all possible combinations, where introduction of all variants of favorable and unfavorable actions of permanent loads will be considered.

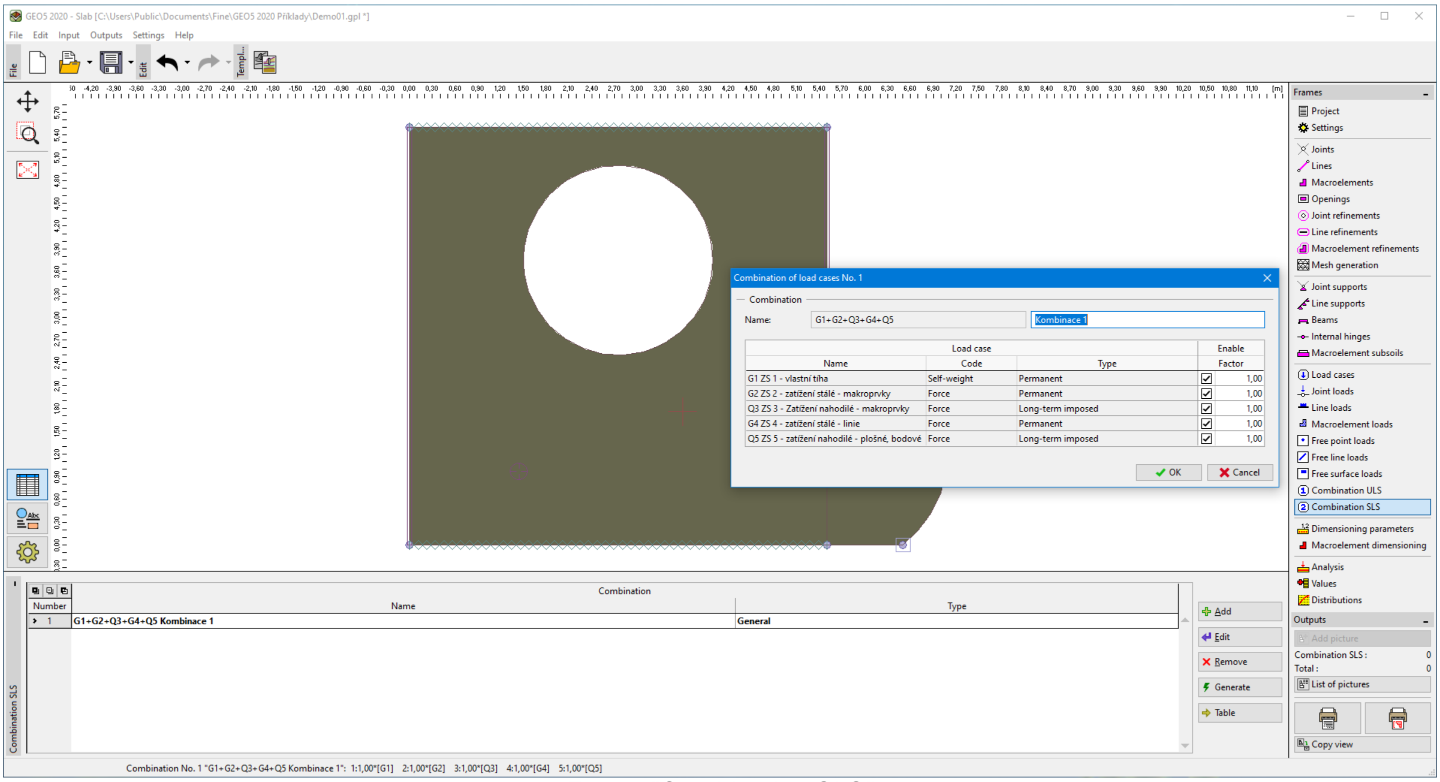
All permanent loads always in combination

If this setting is not checked, the program creates combinations in such a way that a successive introduction of all LCs into a combination will be considered.

Combination SLS

The frame "**Combinations SLS**" contains a table with the list of input combinations of the service limit state. Adding (editing) combinations and their parameters is performed in the "**New combination of load cases**" dialog window. Editing can be carried out with the help of the "Modify" button or by clicking the row with the required combination in the list using the left mouse button.

The built-in generator of combinations of load cases can be used to compile individual combinations.



Frame "Combination SLS"

Parameters of SLS Combinations

Combinations SLS serve to evaluate states that refer to the structure appearance, comfort of people or to functioning of a structure while in ordinary use. Typically, only deformations, vibrations, etc. are checked. The "**New combination of load cases**" dialog window (similarly to combinations for ULS) serves to define the following parameters:

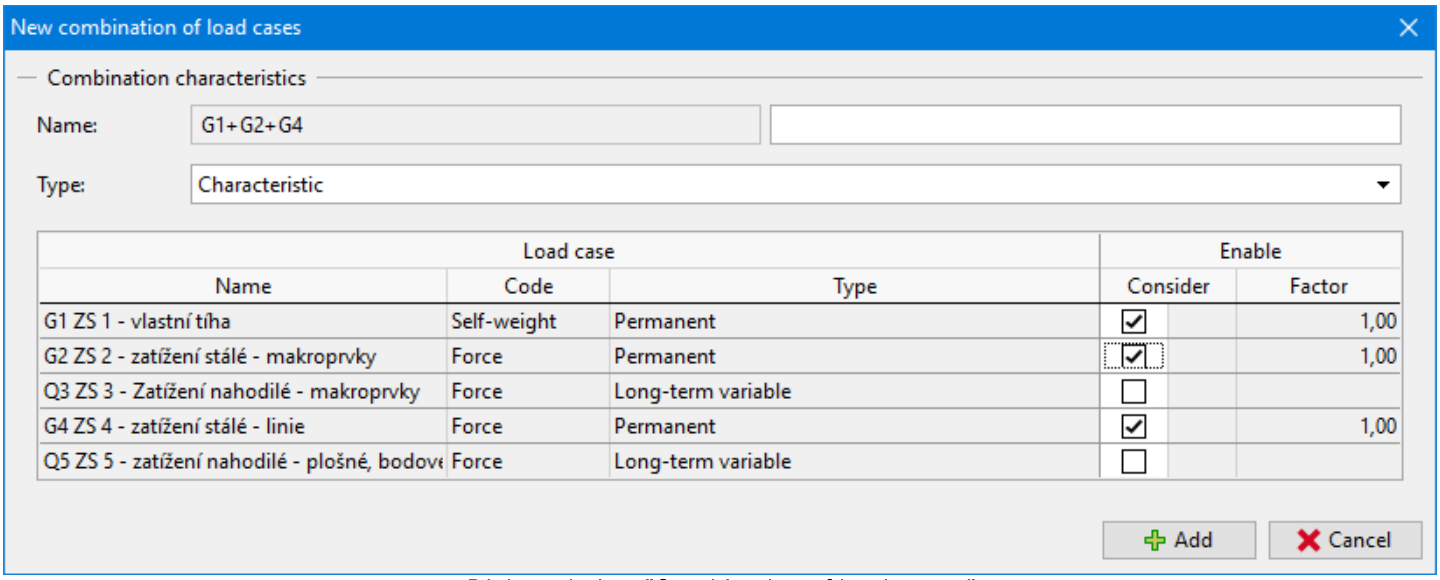
Type of combination according to EN 1990

The following combinations can be created for the service limit state:

Characteristic - combination based on expression 6.14 of EN 1990 standard
Frequent - combination based on expression 6.15 of EN 1990 standard
Quasi-permanent - combination based on expression 6.16 of EN 1990 standard

Selection of load cases

The table listing individual load cases allows for their selection to create a combination. The load case can be introduced into a combination by checking the field in the column "**Consider**" for a particular LC. A coefficient of usability can be specified for generally input combinations (select in the "Material and standards" tab) to adjust the degree of usability of the load case in the combination.



Dialog window "Combination of load cases"

Generator of SLS Combinations

The "**Generator of combinations - 1st order**" dialog window allows for a collective compilation of combinations of load cases for the service limit state. Functions of generator of combinations are explained in section devoted to the generator of combinations for the bearing capacity limit state.

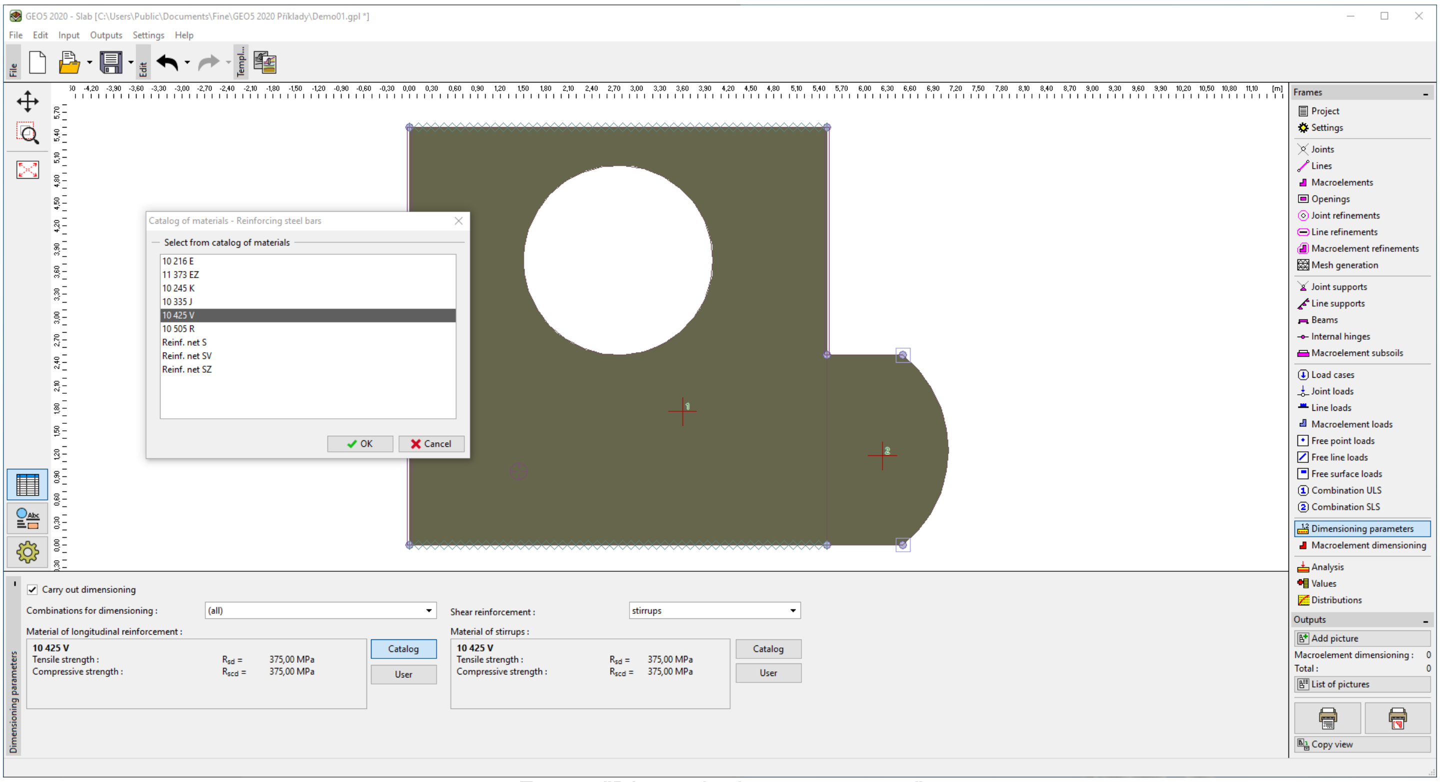
If inputting loads according to EN 1990 is set in the "Material and standards" tab, it is possible to generate the following combinations for the service limit state:

Characteristic - combination based on expression 6.14 of EN 1990 standard
Frequent - combination based on expression 6.15 of EN 1990 standard
Quasi-permanent - combination based on expression 6.16 of EN 1990 standard

Dimensioning Parameters

The frame "**Dimensioning parameters**" serves to define data for dimensioning **longitudinal** and **shear reinforcement**. The **combination number** (or all combinations) of a combination to be analyzed must be specified. The material of longitudinal reinforcements is selected either from "**Catalog of materials**", or can be introduced manually in the "**Editor of materials**". Shear reinforcement is specified in terms of **crooks**, or **stirrups** (crooks require to define their angle).

When running the dimensioning analysis the program generates values of the following quantities. The analysis is carried out according to the standard set in the "Material and standards" tab.



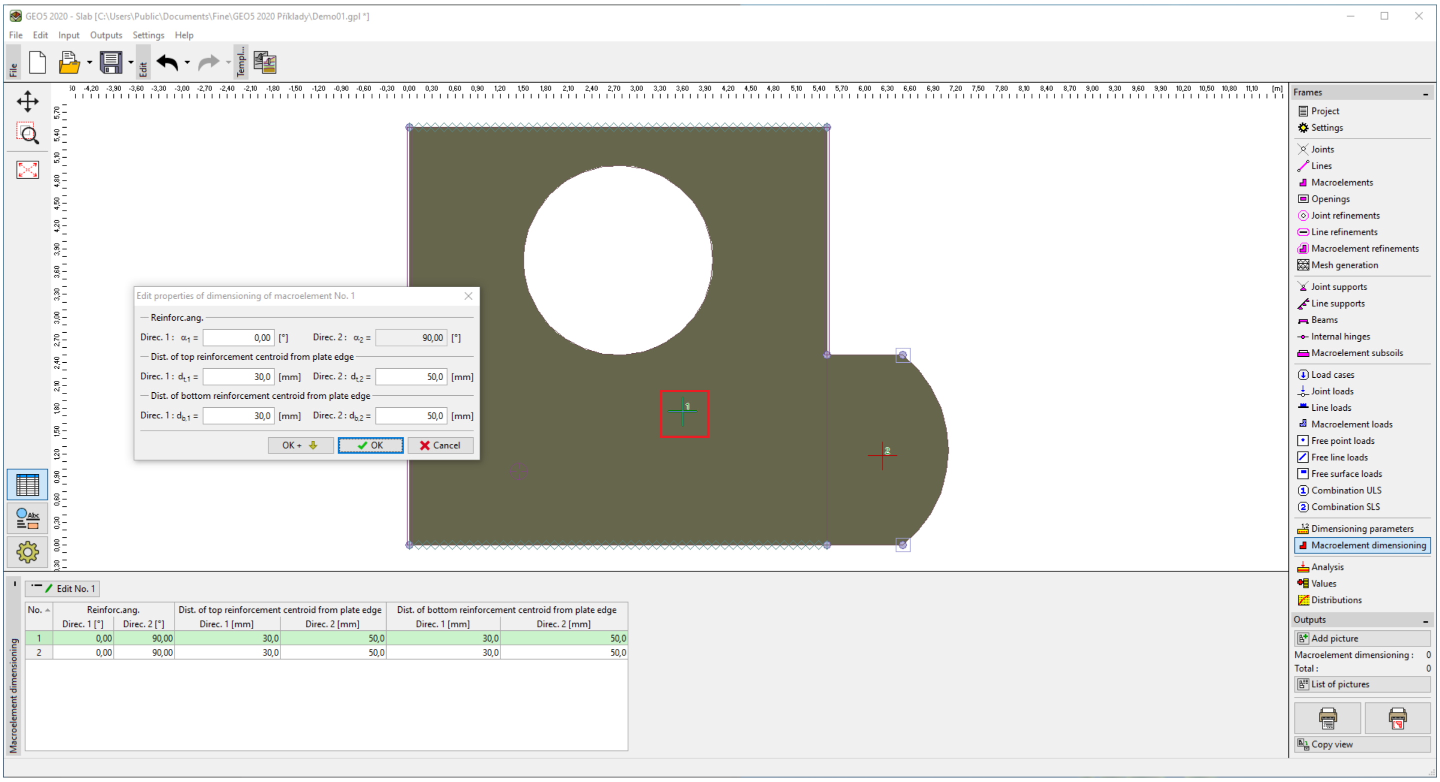
Frame "Dimensioning parameters"

Macroelement Dimensioning

The frame "**Macroelement dimensioning**" contains a table with the input macroelements. Editing reinforcement properties (**reinforcement direction** and **distance of centroid of top and bottom reinforcement from the slab edge** can be modified) is performed in the "**Modify properties of macroelement dimensioning**" dialog window.

When running the dimensioning analysis the program generates values of the following quantities. The analysis is carried out according to the standard set in the "Material and standards" tab.

The macroelement dimensioning can also be edited on the desktop with the help of active objects.



Frame "Macroelement dimensioning"

Analysis

The analysis results are displayed in the frame "**Analysis**". The "**Analysis**" is carried out using the finite element method. The dimensioning analysis is performed according to the standard set in the "Material and standards" tab.

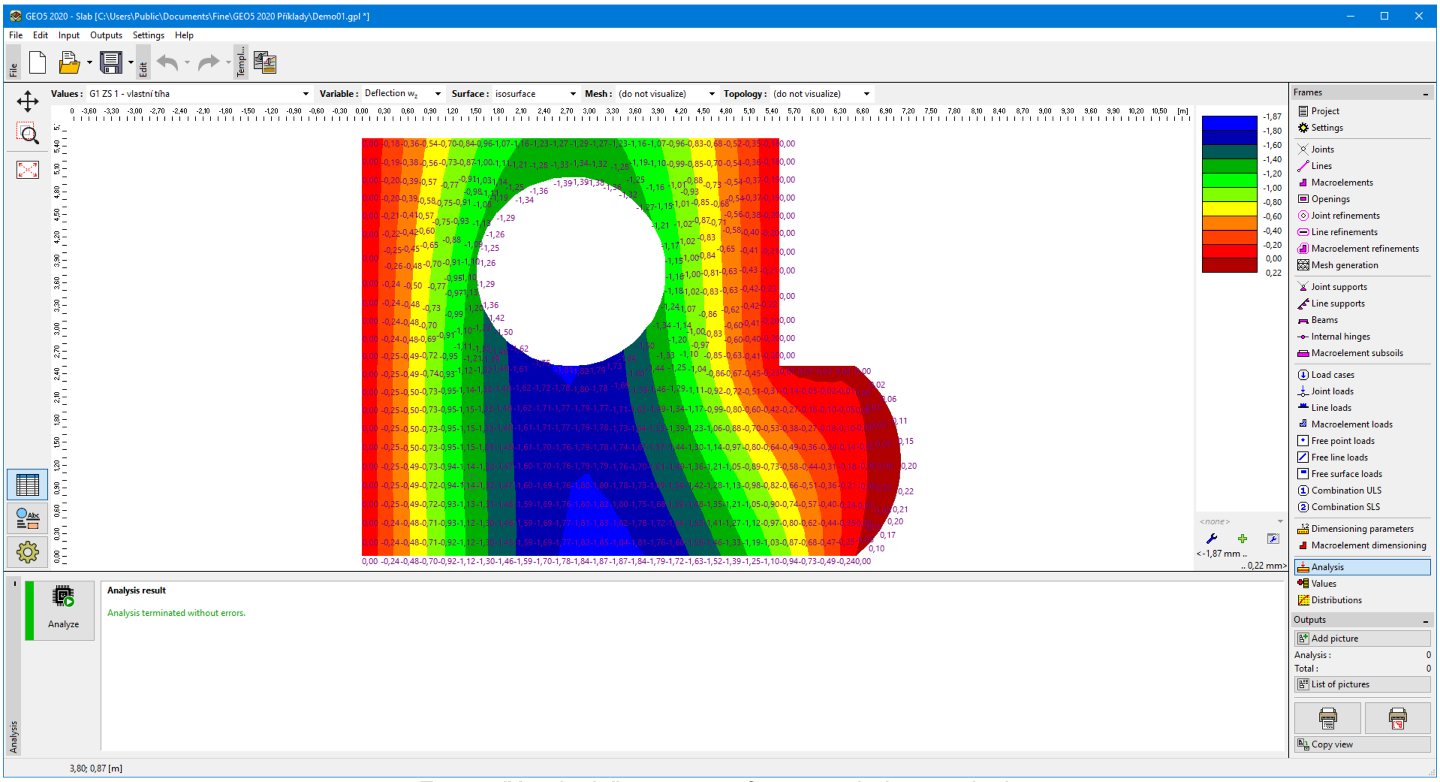
The analysis can be stopped any time by pressing the **"Interrupt"** button.

Upon completing the analysis the program immediately displays the results and information about the solution process. This information (with possible listing of errors) is shown in window in the bottom part of the frame. The principal output tool is the visualization of results on the screen. The toolbar in the top part of the screen serves to manage the graphical representation of output quantities.

The color rangeand the bottom for its setting are found in the top part of the desktop.

The program employs the following coordinate systems (sign convention).

The way the results appear on the screen can be set in the "Drawing Settings" frame.



Frame "Analysis" - screen after completing analysis

Analysis Procedure

The solution procedure is split into several steps including localization of the global stiffness matrix with the support conditions (fixed or spring supports at joint or along lines, elastic subsoil), setting up the load vector and analysis of the system of equations using the Gaussian method with the Cholesky decompositions of the global stiffness matrix. The values of primary variables w_z , φ_x a φ_y calculated at mesh nodes are then used to determine the internal forces m_x , m_y , m_{xy} , v_x and v_y together with the derived quantities m_1 , m_2 and the values of reactions developed in supports.

2D-elements

The quality of results of the slab problem derived using the finite element method is strongly influenced by the type of slab element. The present formulation exploits a deformation variant of the finite element method to derive triangular and quadrilateral elements denoted as DKMT and DKMQ (Discrete Kirchhoff-Mindlin Triangle a Quadrilateral).

Formulation of the slab element implemented in the program is based on the discrete Kirchhoff theory of bending of thin slabs, which can be considered as a special case of the Mindlin plate theory developed upon the following assumptions:

- compression of slab in the z -direction is negligible compare to the vertical displacement W_z
- normals to the mid-plane of the slab remain straight after deformation but not necessarily normal to the deformed mid-plane of the slab
- normal stress σ_z is negligible compare to stresses σ_x , σ_y

DKMT and DKMQ elements have 9 and 12 degrees of freedom, respectively - three independent displacements at each node:

W_z - elastic deflection in the direction of z - axis

φ_x - rotation about x - axis

φ_y - rotation about y - axis

The elements satisfy the following criteria:

- the stiffness matrix has correct rank (no zero energy states are generated)
- fulfill the patch test
- are suitable for the analysis of both this and thick slabs
- they show good convergence properties
- not computationally expensive

In case of well generated mesh the quadrilateral elements are preferable as show better behavior compare to triangular elements.

1D-elements

The slab can be reinforced by beams formulated on the basis of one dimensional beam element with embedded torsion and is compatible with slab elements (details can be found in literature). The primary variables are W_z, φ_x and φ_y and corresponding internal forces are M_1, M_2 and V_3 (twisting and bending moments and shear force). The beam is characterized by the moment of inertia I_t a I_2 (torsion, bending), area A and shear area A_s. These parameters can be calculated by the program based on the type of cross-section. The analysis constructs $6x6$ local stiffness matrices subsequently localized in to the global stiffness matrix of the structure.

Literature:

I. Katili, A new discrete Kirchhoff-Mindlin element based on Mindlin-Reissner plate theory and assumed shear strain fields - part I: An extended DKT element for thick-plate bending analysis, Int. J. Numer. Meth. Engng., Vol. 36, 1859-1883 (1993).

I. Katili, A new discrete Kirchhoff-Mindlin element based on Mindlin-Reissner plate theory and assumed shear strain fields - part II: An extended DKQ element for thick-plate bending analysis, Int. J. Numer. Meth. Engng., Vol. 36, 1885-1908 (1993).

Z. Bittnar, J. Sejnoha, Numericke metody mechaniky, CVUT, Praha, 1992.

Results

Visualization and interpretation of results is one of the most important parts of the program.

Based on the toolbar setting the program displays variables (elastic deflection, moments, rotation) for an arbitrary load case or LC combination, or if needed variables for dimensioning (values of necessary reinforcement areas calculated according to the standard selected in the "Material and standards" tab).

Calculation of values in user-defined points, or on lines, can be set in frames "Values" or "Distributions", respectively.

The program provides several basic types of graphical output defined in the "Slab - results visualization settings" dialog window.

- plotting structure
- surface plot of quantities
- plotting finite element mesh
- plotting grid-plotting distributions (diagrams)
- plotting values on surface
- plotting directions of moments and reactions

The toolbar "Results" in the upper part of the screen serves to selected variables to be displayed and the way they should appear on the screen. The color range is shown in the right part of the desktop. Its particular setting can be adjusted using the "Color range" toolbar.

Because properly setting outputs might be often time consuming, the program disposes of a comfortable system of selecting and storing view settings.

All outputs and selected results can be further printed out from the analysis protocol.

Toolbar "Results"

The toolbar contains the following operating elements:

Toolbar "Setting visualization of graphical outputs"

Individual elements operate as follows:

Show	**Plotting style setting**	• opens the "Slab - results visualization settings" dialog window which allows the user to be more specific in defining the plotting style
<none>	**List of plots**	• a combo list containing names of plots saved by the user
	Save plot	• saves the current plot displayed on the desktop, the dialog window serves to enter the name of the plot
	Manager of plots	• opens the "**Manager of plots**" dialog window which serves to manage (delete, change order, rename) already saved plots
Results : G1 self-weight-permanent	**Results** (loads, load cases, combination, dimensioning)	• displays the selected load cases, combination (ULS, SLS), envelopes (ULS, SLS) or dimensioning
Variable : Deflection w_z	**Type of variable**	• displays the selected variable or variable of dimensioning
Surface :	**Surface plot**	• turns on/off plotting of isolines, isosurfaces
Mesh :	**Mesh**	• turns on/off the style of plotting the FE mesh (only edges, or according to the setting in the "Slab - results visualization settings" dialog window
Topology :	**Plotting of topology**	• plotting of topology of construction

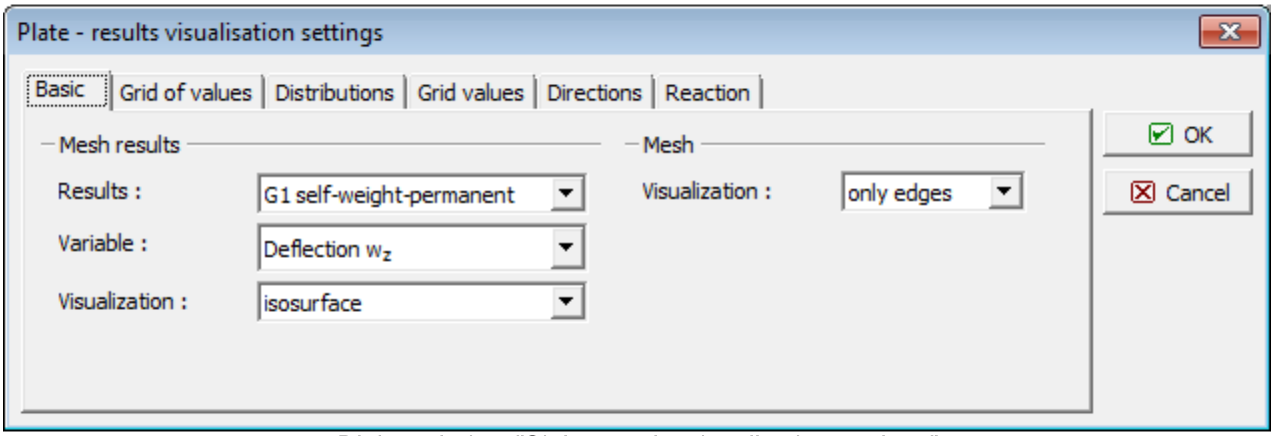
The toolbar contains **the most often used operating elements** needed to view the results on the desktop. Detailed setting of the style of plotting the results is available in the "Slab - results visualization settings" dialog window.

Similar to our other programs the results can be saved and printed. The plotting style can be adjusted in the "Drawing Settings" frame.

Results Visualization Settings

The "**Slab - results visualization settings** " dialog window serves to specify the values to be plotted and the way of their visualization. individual Settings can be later **saved** using the toolbar "Results".

The item "**Basic**" serves to set the basic parameters for plotting the results, quantities and mesh information - additional items can then be used to define visualization of other outputs.



Dialog window "Slab - results visualization settings"

List of Variables

List of quantities displayed by the program for individual load cases, combinations of load cases (ULS, SLS) or envelopes (ULS, SLS)

Variable	Unit	Description
Deflection W_z	$[mm]$	Elastic displacement in the Z-direction
Rotation φ_x	$[mrad]$	Rotation about X-axis
Rotation φ_y	$[mrad]$	Rotation about Y-axis
Moment m_x	$[kNm/m]$	Value of the moment about Y-axis
Moment m_y	$[kNm/m]$	Value of the moment about X-axis

Moment m_{xy}	$[kNm/m]$	Value of moment
Shear force V_x	$[kN/m]$	Value of the shear force in the X-direction
Shear force V_y	$[kN/m]$	Value of the shear force in the Y-direction
Moment m_1	$[kNm/m]$	Value of the principal (extreme) moment
Moment m_2	$[kNm/m]$	Value of the principal (extreme) moment
Shear force V_{max}	$[kN/m]$	Value of the shear force (extreme)
Contact stress σ	$[kN/m^2]$	Value of the contact stress

List of Variables of Dimensioning

To perform dimensioning analysis it is first necessary to choose the option "**Carry out dimensioning**" in the frame "Dimensioning parameters". Visualization of values for dimensioning can be set in the toolbar "Results". Notation of variables (particularly indexes of variables) changes according to the standards used for dimensioning of concrete and steel structures set in the "Material and standards" tab.

List of quantities displayed by the program for dimensioning

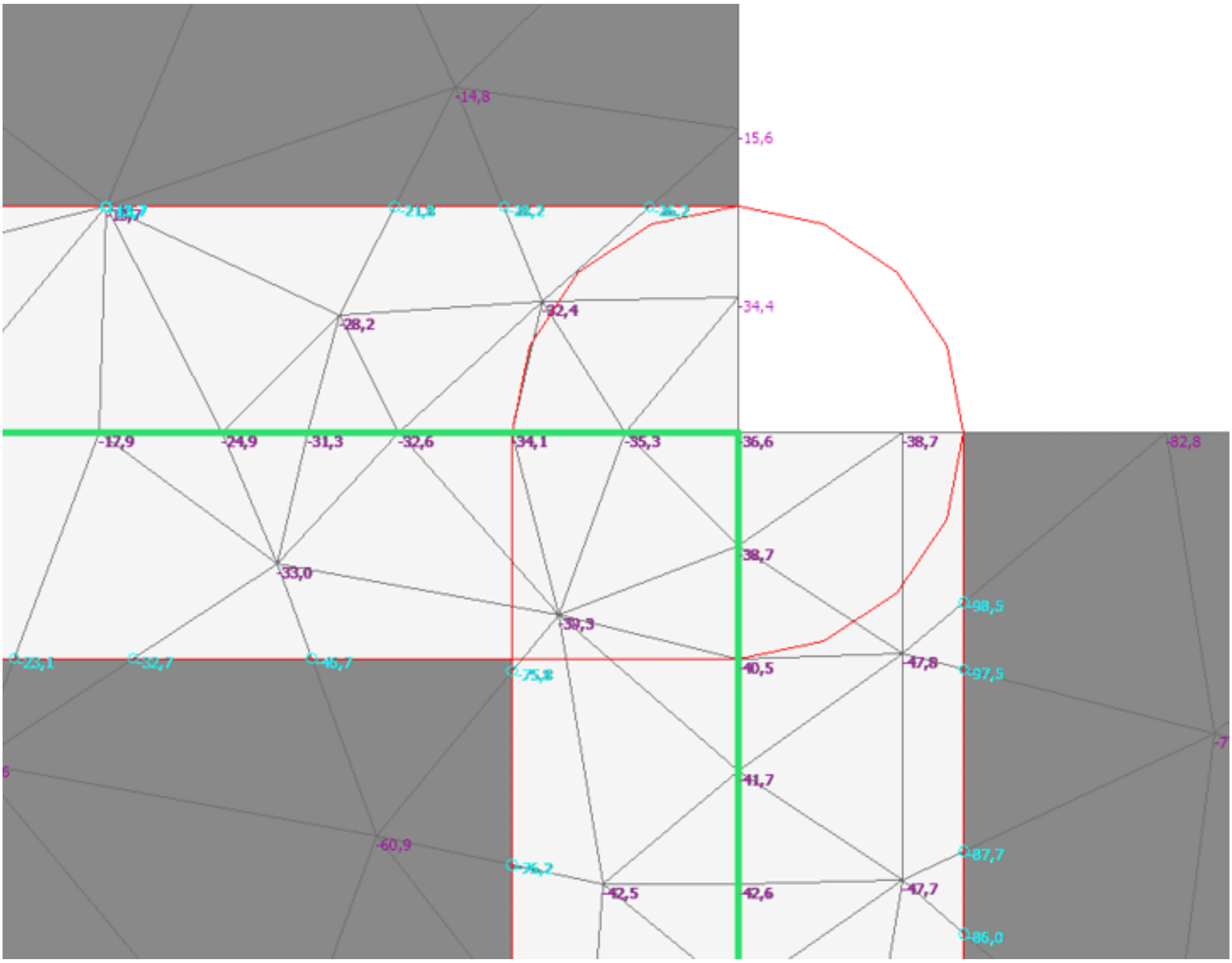
Variable	Unit	Description
Moment $M_{dim1, min}$	$[kNm/m]$	Minimal dimensioning moment in direction of reinforcement 1
Moment $M_{dim1, max}$	$[kNm/m]$	Maximal dimensioning moment in direction of reinforcement 1
Moment $M_{dim2, min}$	$[kNm/m]$	Minimal dimensioning moment in direction of reinforcement 2
Moment $M_{dim2, max}$	$[kNm/m]$	Maximal dimensioning moment in direction of reinforcement 2
Reinforcement area A_{u1}	$[mm^2/m]$	Area of upper reinforcement in direction 1
Reinforcement area A_{b1}	$[mm^2/m]$	Area of bottom reinforcement in direction 1
Reinforcement area A_{u2}	$[mm^2/m]$	Area of upper reinforcement in direction 2
Reinforcement area A_{b2}	$[mm^2/m]$	Area of bottom reinforcement in direction 2
Ratio of reinforcement μ_{h1}	$[\%]$	Reinforcement ratio of upper reinforcement in direction 1
Ratio of reinforcement μ_{d1}	$[\%]$	Reinforcement ratio of bottom reinforcement in direction 1
Ratio of reinforcement μ_{h2}	$[\%]$	Reinforcement ratio of upper reinforcement in direction 2
Ratio of reinforcement μ_{d2}	$[\%]$	Reinforcement ratio of bottom reinforcement in direction 2
Shear force V_{Ed}	$[kN/m]$	Dimensioning shear force
Reinforcement area $A_{b,nut}$	$[mm^2/m^2]$	Requested area of shear reinforcement
Shear force $V_{Rd, c}$	$[kN/m]$	Shear strength of cross-section without shear reinforcement
Shear force $V_{Rd, max}$	$[kN/m]$	Maximal allowable shear force

Reduction of Bending Moments

Dimensioning moments above supports will be reduced, if the "**Input dimensions of supports and reduce resulting bending moments**" in the "Settings" frame is selected. This will then display the "**Geometry of Support**" section in the properties of the individual supports ("Joint Supports", "Line Supports") and will reduce the dimensioning moments in the analysis.

Interpretation in the analysis

The program reduces the dimensioning moments in the areas of defined geometry of support.



- The areas of defined geometry of the individual supports that overlap or touch are joined
- The values lying at the boundary of the merged areas are calculated
- Values in the mesh nodes within the merged areas are limited by a value that is calculated from the boundary values so that the nearest boundary values have the highest influence
- The values of dimensioning moments and dimensioning shear force are limited

Limiting value

The limiting value is calculated according to the formula:

$$\upsilon_{\lim}(xy) = \sum_{i=1}^{N} \frac{k_i(xy)\upsilon_i}{\sum_{j=1}^{N} k_j(xy)}$$

where:

$$k_i(xy) = \frac{1}{\left(d\left(xy, xy_i\right)\right)^{16}}$$

where:

where:	$\upsilon_{lim}(xy)$	-	limiting value at the calculated point xy
	N	-	number of border points
	υ_i	-	value at boundary point
	$d(xy, xy_i)$	-	distance of the calculated point from the boundary point

Values

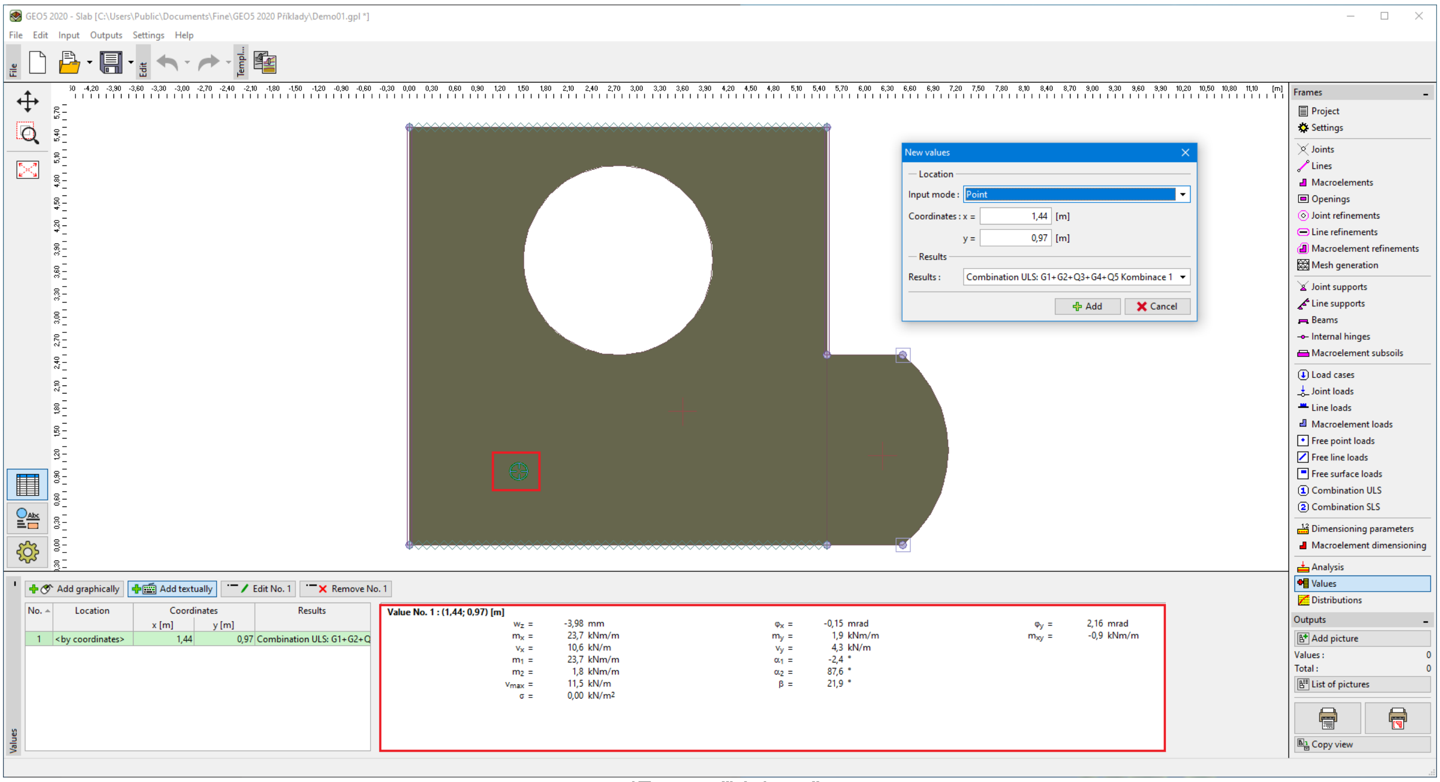
The frame "**Values**" allows for defining points (or joints) arbitrarily placed on the slab surface. For these points (joints) it is possible to display variables (deflections, moments, rotations) for an arbitrary load case or LC combination, or if needed variables for dimensioning (values of necessary reinforcement areas calculated according to the standard selected in the

"Material and standards" tab).

The frame contains a table with the list of input points (joints). Adding is performed in the "**New values**" dialog window. The window serves to specify the type of input (point, joint), coordinates and for what load case, combination, or dimensioning the resulting quantities should be displayed. The value in the supported joint corresponds to the reaction force at this support.

In the dimensioning analysis some quantities can be denoted by symbol [*]. In such a case the necessary reinforcement area and minimal degree of reinforcement is required. If the point is found on the boundary of two macroelements, the program displays two sets of values for dimensioning.

The input points (values) can also be edited on the desktop with the help of active objects.



dFrame "Values"

Distributions

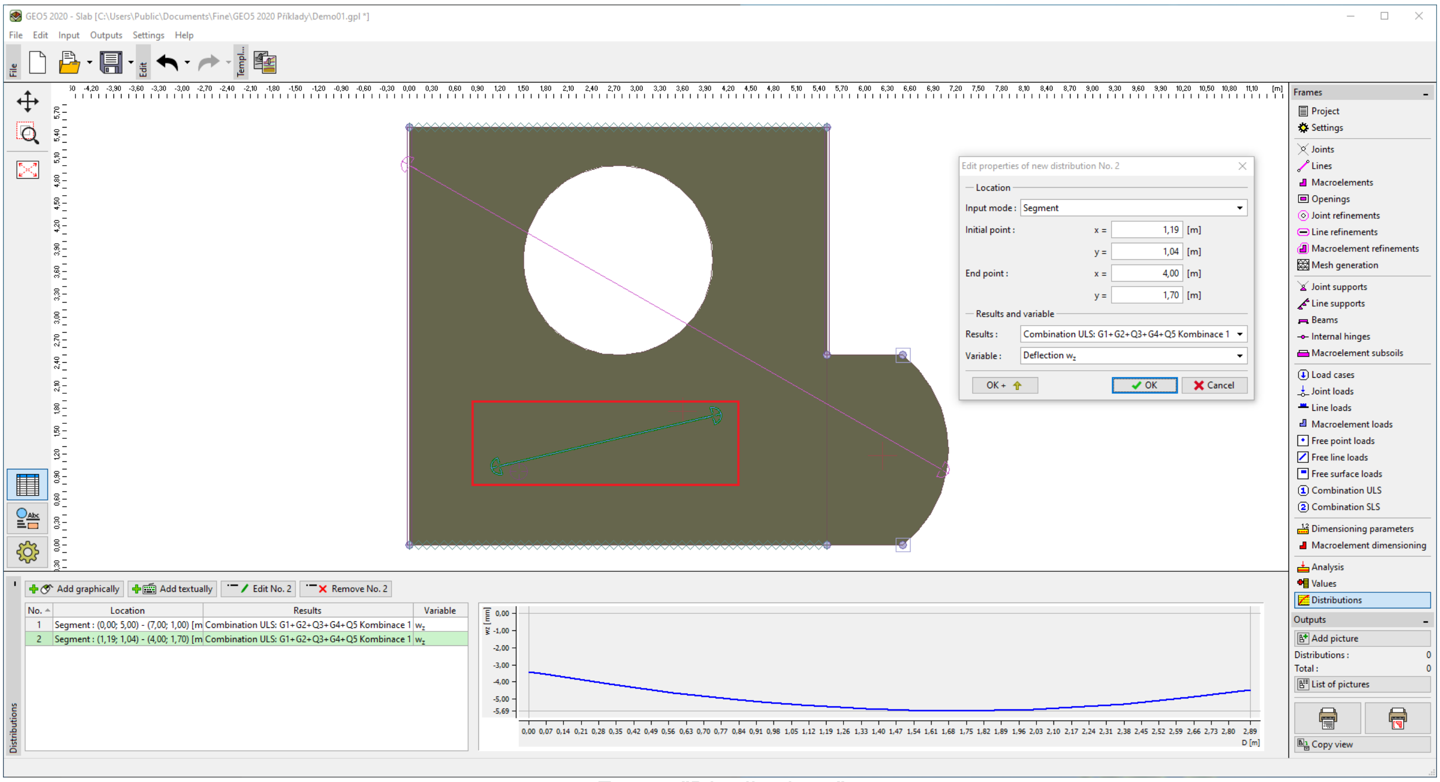
The frame "**Distributions**" serves to define general lines or lines located on the slab surface. For these segments (lines) it is possible to display variables (deflections, moments, rotations, etc.) for an arbitrary load case or LC combination, or if needed variables for dimensioning (values of necessary reinforcement areas calculated according to the standard selected in the "Material and standards" tab).

The frame contains a table with the list of input segments (lines). Adding is performed in the "**New distributions**" dialog window. The window serves to specify the type of input (segment, line), coordinates of the first and the last point, load case, combination, dimensioning and quantity.

The frame displays:

- **General distributions** — general distributions (diagrams) on an arbitrary segment (line)
- **Distributions on beams** — if a **beam** is assigned to the line it is possible to display distributions (diagrams) of other quantities (shear force V_3, bending moment M_2, twisting moment M_1)
- **Distributions on supported line** — if the line is **supported** it is possible to display distributions (diagrams) of other quantities (vertical reaction r_z, moment reaction $r_{m,\,t}$).

The input distributions can also be edited on the desktop with the help of active objects.



Frame "Distributions"

Coordinate System (Sign Convention)

Internal forces

Internal forces are defined as:

$$m_x = \int_{-h/2}^{+h/2} \sigma_x \, z dz$$

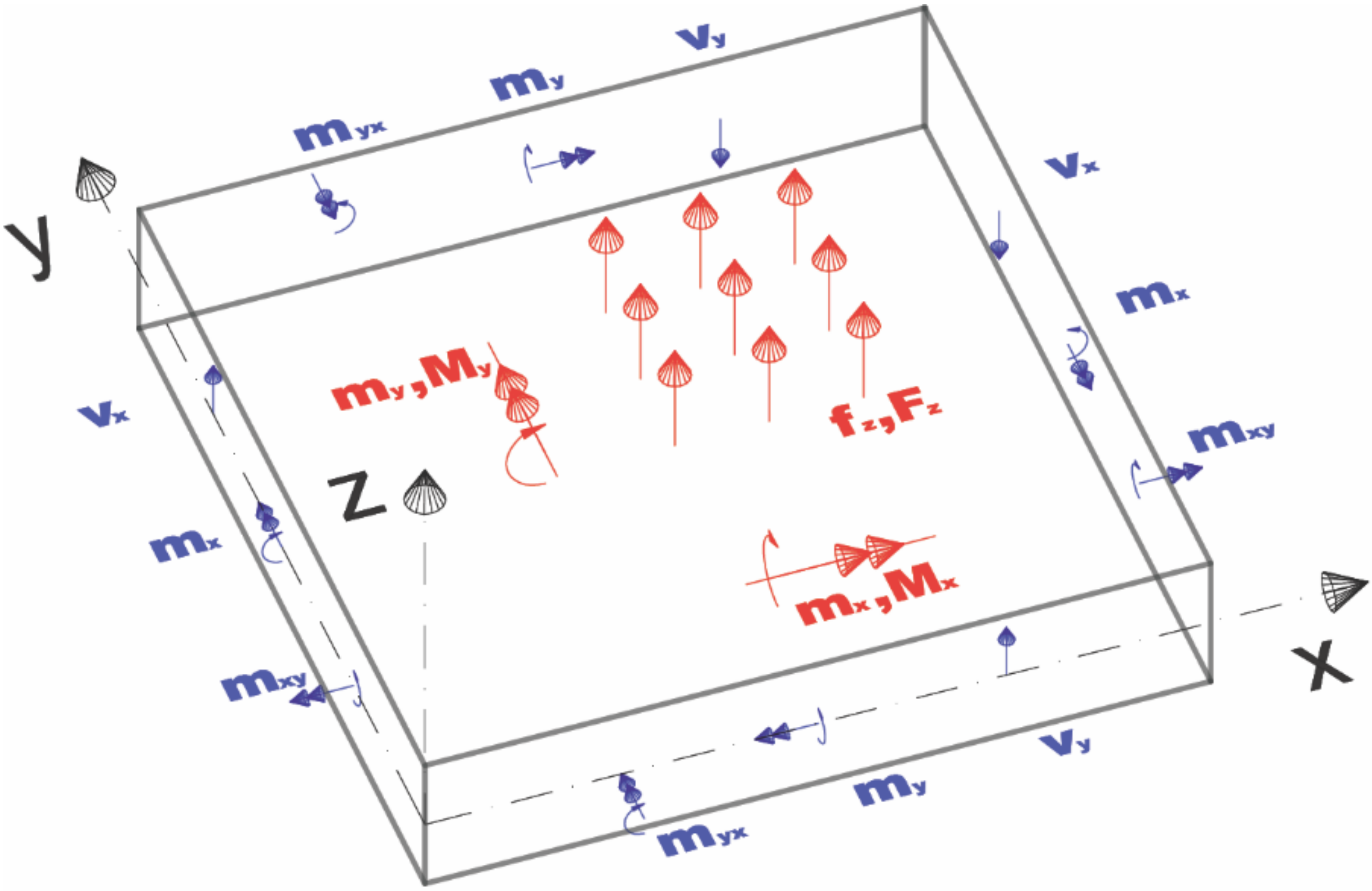
$$m_y = \int_{-h/2}^{+h/2} \sigma_y \, z dz$$

$$m_{xy} = \int_{-h/2}^{+h/2} \sigma_{xy} \, z dz$$

$$v_x = \int_{-h/2}^{+h/2} \sigma_{xz} \, dz$$

$$v_y = \int_{-h/2}^{+h/2} \sigma_{yz} \, dz$$

The positive direction of internal forces is evident from the following figure:



The principal moments and directions of principal axes are provided by:

$$m_{1,2} = \frac{1}{2}\left(m_x + m_y \pm \sqrt{\left(m_x - m_y\right)^2 + 4m_{xy}{}^2}\right)$$

$$tg\ 2\alpha_{1,2} = \frac{2m_{xy}}{m_x - m_y}$$

The meaning of individual variables is the following: internal forces can be transformed from the (x, y) coordinate system to the (x', y') coordinate system by rotating the (x, y) plane through a certain angle about the z -axis. The angle α , in particular, corresponds to a rotation angle for which the transformed $m_{x'y'}$ moment attains a zero value whereas the $m_{x'}$ and $m_{y'}$ moments attain their maximum and minimum values m_1 and m_2 , respectively.

The maximum shear force is obtained similarly:

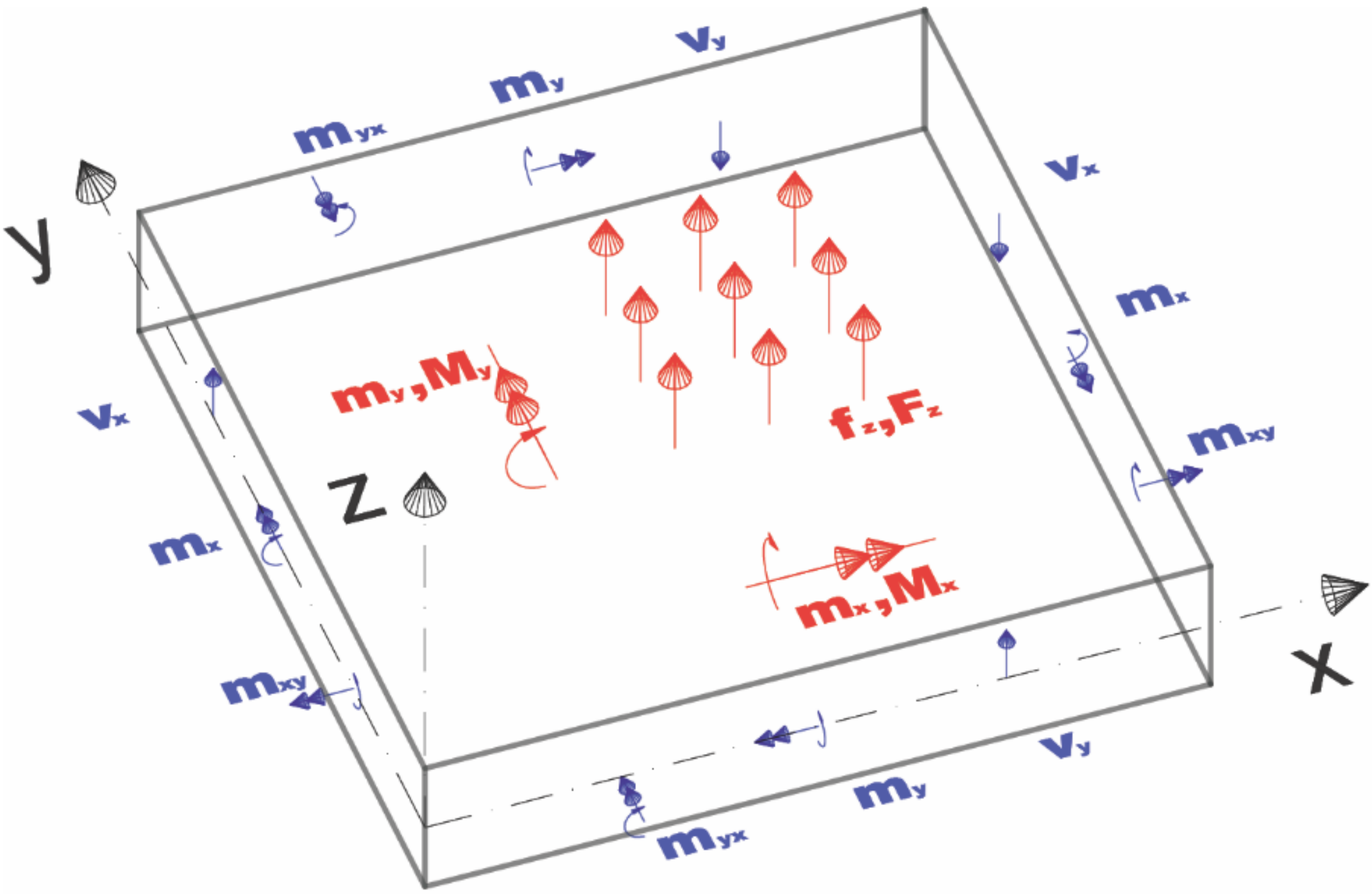
$$v_{max} = \sqrt{v_x{}^2 + v_y{}^2}$$

and the angle between v_{max} and the x -axis:

$$\beta = \text{arctg}\ \frac{v_y}{v_x}$$

Load

The sign convention of the applied force and moment load is evident from the following figure:



It is worth to point out a different sign convention applied to the load moment M (at a point or along a line) and to internal moment m . While the M_x moment rotates about the x -axis (as usual for beams), the internal moment rotating about the x -axis is denoted as m_y .

Program Beam

This program provides the analysis of foundation beams resting on elastic subsoil.

The help in the program "Beam" includes the following topics:

- The input of data into individual frames:

Project	Settings	Geometry	Subsoil	Interface	Location	Soils
Assign	Water	Supports	Load Cases	Load	Combination ULS	Combination SLS
Analysis						

- Standards and analysis methods

- Theory for analysis in the program "**Beam**":

Geostatic Stress (Uplift Pressure)	Analysis Using the Oedometric Modulus	Determination of the Influence Zone Depth

- Outputs

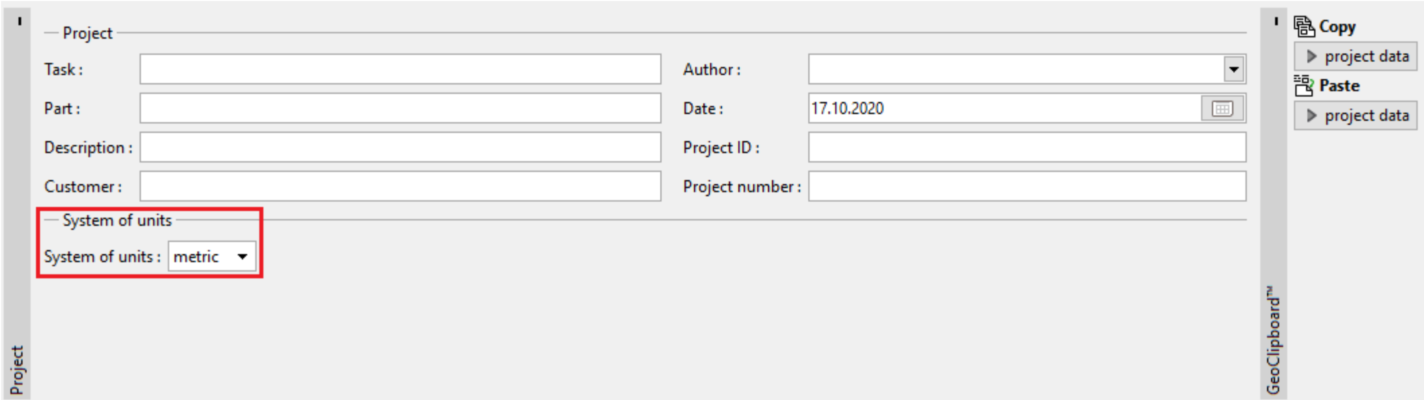
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The frame **"Project"** is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The frame "**Settings**" allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

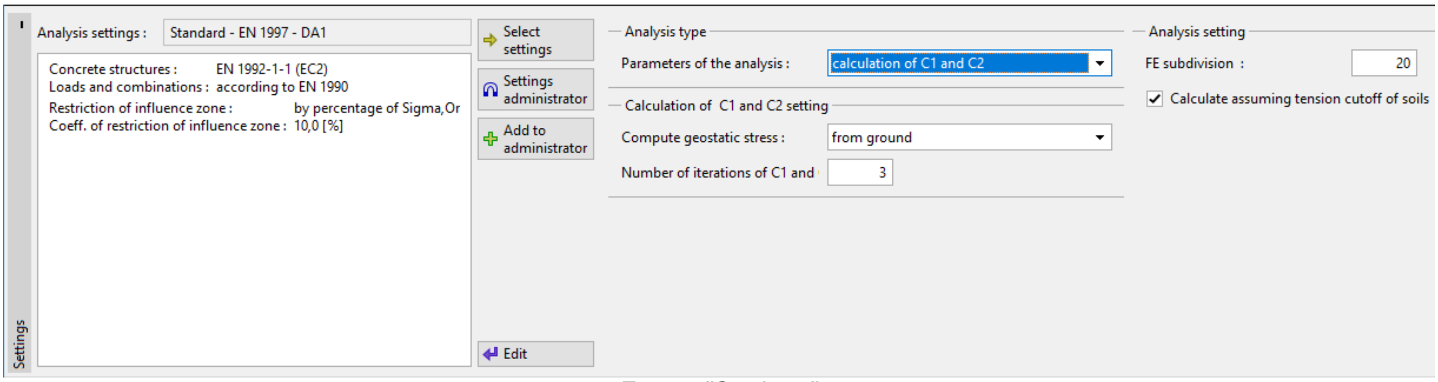
Settings of analysis parameters are performed in the "Materials and standards" tab.

This frame also allows us to specify the **subdivision of beam into finite elements** (default setting is 20 elements). Next, it is possible to define whether the soil acts in tension - we always recommend **calculating assuming tension cutoff of soils**.

A combo list allows us to select one of three ways of defining the Winkler-Pasternak **subsoil**:

- Calculation of C_1 and C_2 - the Winkler-Pasternak parameters of subsoil are calculated by the program from input parameters of the geological profile. Geostatic stress is calculated from finished ground or from footing bottom. It is possible to enter number of iterations C_1 and C_2.
- Input of C_1 and C_2 - the Winkler-Pasternak parameters of subsoil are directly specified.
- Input of E_{def}, n_y, h_z - the Winkler-Pasternak parameters of subsoil are calculated from the deformation modulus E_{def}, Poisson's ratio ν and depth of influence zone h_z.

In the first case, when parameters C_1 and C_2 are calculated, the frame "Subsoil" is not accessible. For the remaining approaches the frames "Interface", "Location", "Soils", "Assign", and "Water" are not accessible.



Frame "Settings"

Winkler-Pasternak Parameters C1 a C2

The Winkler - Pasternak model for the solution of an elastic layer introduces the balance equation in the vertical direction as:

$$c_1.w - c_2.\Delta w = f_z$$

where: c_1, c_2 - parameters of the Winkler - Pasternak model

w - deflection in the vertical direction

f_z - vertical load acting on a layer

The program allows us to calculate the parameters C_1 , C_2 from deformation parameters of soils, or from geological profile.

Calculation of Winkler-Pasternak Parameters C1 and C2 from Geological Profile

A characteristic combination of the load must be chosen when calculating the Winkler-Pasternak parameters (C_1 , C_2) from a geological profile. This combination should be considered as service and should correspond to the most frequently appearing load. Using this combination, the influence zone is calculated.

Deformation parameters (Poisson's ratio, deformation modulus) are determined for the calculated influence zone as a weighted average of deformation parameters of soils. Given these parameters, the Winkler-Pasternak constants (C_1 , C_2) are then calculated in the following way.

Calculation of Winkler-Pasternak Constants from Deformation Parameters of Soils

The Winkler - Pasternak constants C_1 and C_2 are calculated from the condition of equal compliance matrices of infinitely stiff infinite strip footing resting on the Winkler - Pasternak and elastic subsoil. This condition is represented by the following equalities:

$$[\mathbf{C}] = \begin{bmatrix} \dfrac{1}{2\left[\sqrt{C_{1WP}C_{2WP}} + bC_{1WP}\right]} & 0 \\ 0 & \dfrac{1}{2\left[b^2\sqrt{C_{1WP}C_{2WP}} + bC_{2WP} + \dfrac{b^3}{3}C_{1WP}\right]} \end{bmatrix} =$$

$$= \begin{bmatrix} \sum_{n=0}^{\infty} \dfrac{1}{2\sqrt{H}\left[(2n+1)\sqrt{C_{1w}C_{2w}} + (2n+1)^2 bC_{1w}\right]} & 0 \\ 0 & \sum_{n=0}^{\infty} \dfrac{1}{2\sqrt{H}\left[(2n+1)b^2\sqrt{C_{1w}C_{2w}} + bC_{2w} + (2n+1)^2 \dfrac{b^3}{3}C_{1w}\right]} \end{bmatrix}.$$

d

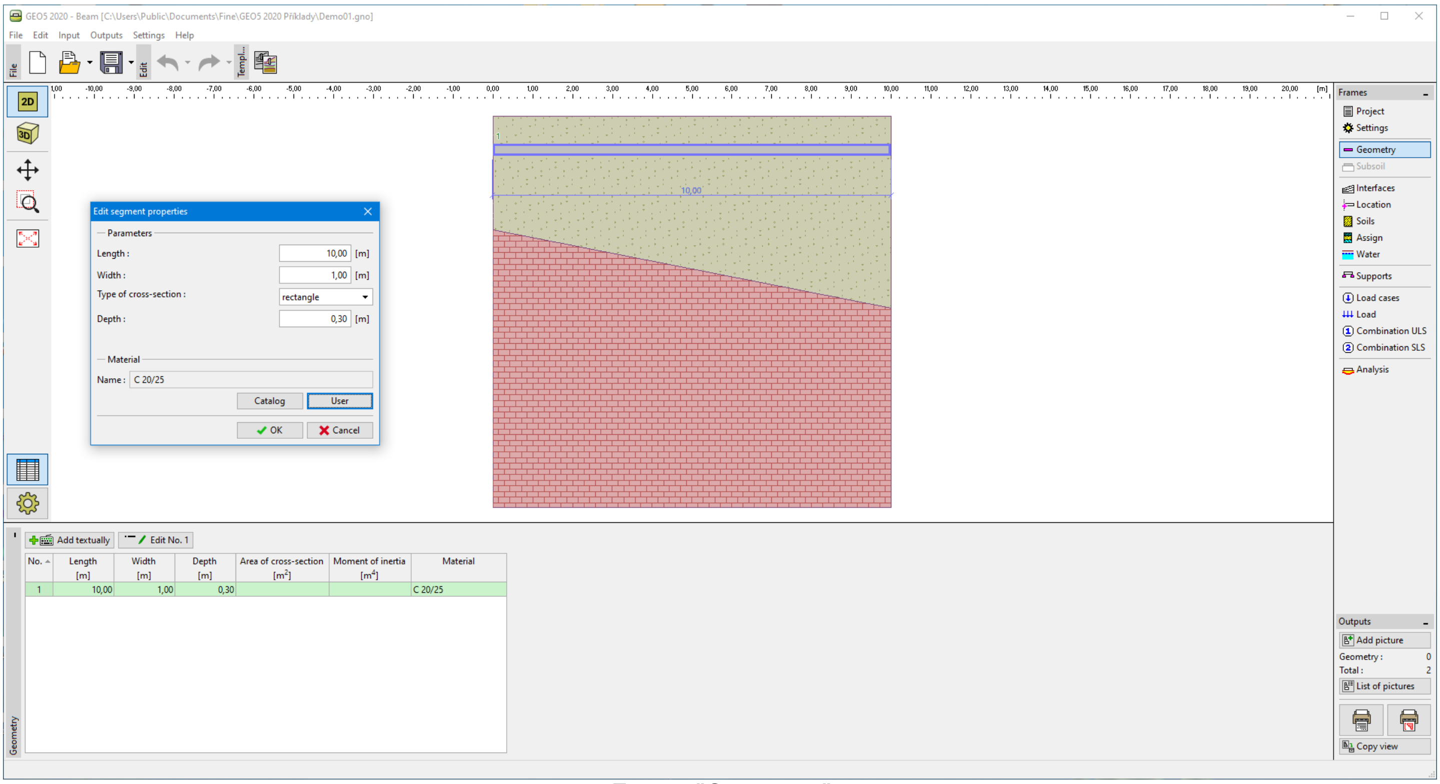
where: $[C]$ - matrix of constants C_1 a C_2

b	-	half width of foundation
c_{1w}, c_{2w}	-	Winkler's constants
H		depth of the deformation zone

Geometry

The frame "**Geometry**" contains a table with a list of input beam sections. Adding points is performed in the "**New segments**" dialog window. The window requires defining the section length, width and height (for **rectangular cross-section**). The program allows for defining a general cross-section of a beam (cross-sectional area and moment of inertia are specified).

Material of the cross-section is specified next, either using the program catalog, or by entering the material parameters (modulus of elasticity, shear modulus, self-weight).



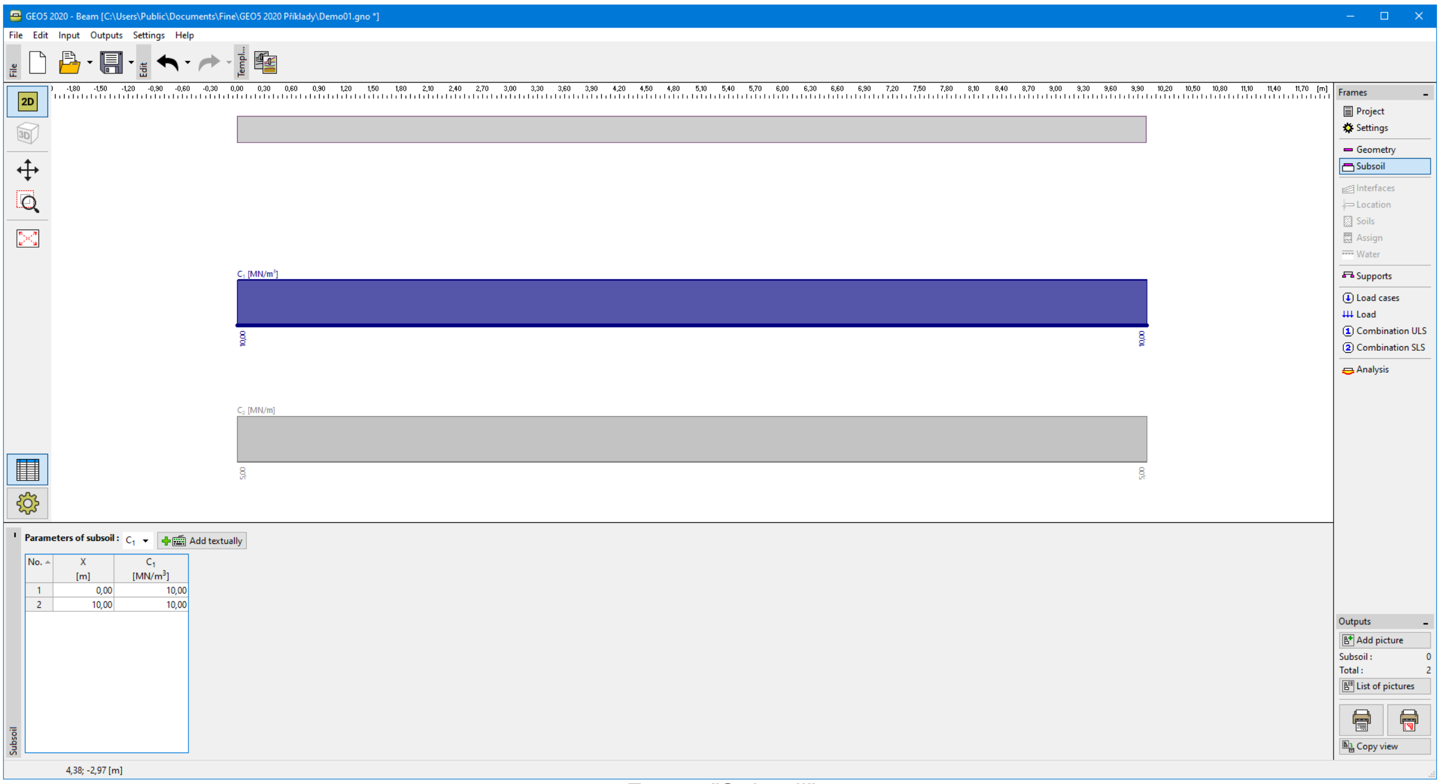
Frame "Geometry"

Subsoil

The frame "Subsoil" contains a table with the list of values of the parameters of Winkler-Pasternak subsoil C_1 and C_2 or **deformation parameters of soils** (E_{def}, n_y, h_z), respectively, depending on the setting in the frame "Settings".

Adding parameters is performed in the "**New parameters of subsoil**" dialog window.

The table shows values of the parameter that is selected from the combo list above the table. Adding (editing) points is performed in the "**New subsoil parameters**" dialog window. The window serves to specify the X -coordinate and the value of the parameter.

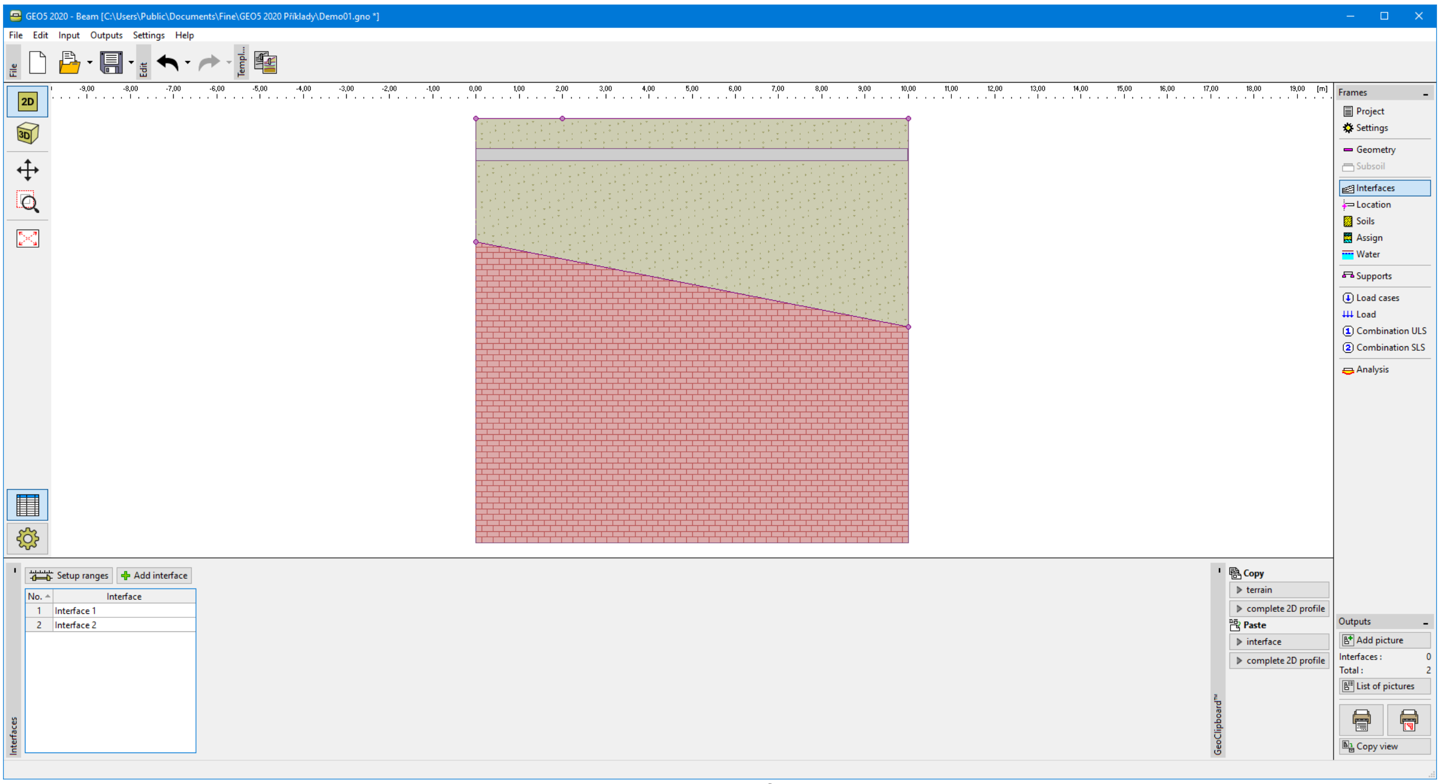


Frame "Subsoil"

Interface

The frame "**Interface**" serves to introduce individual soil interfaces into the soil body. Detailed description on how to deal with interfaces is described herein.

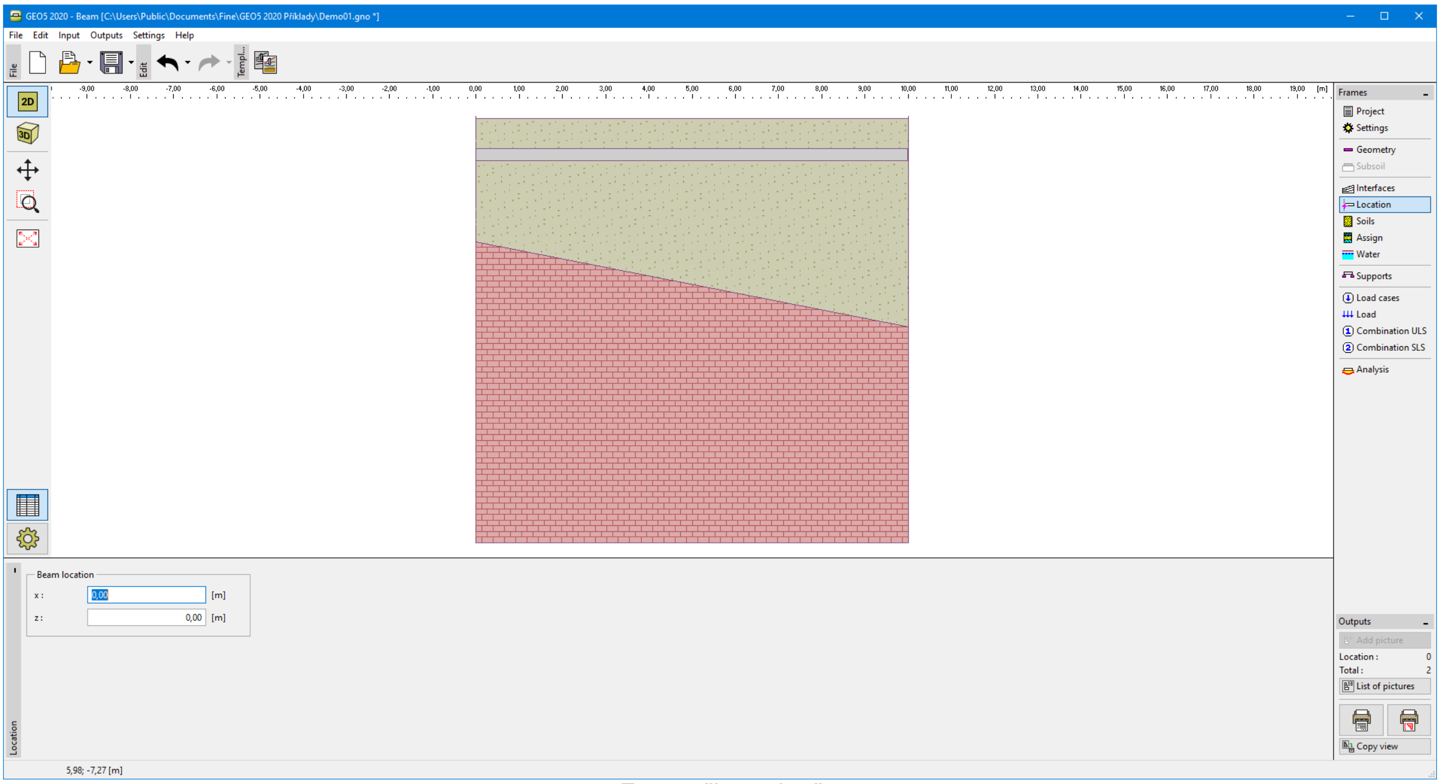
The program makes it possible to import or export interfaces in the *.DXF format. Input interfaces can be copied within all 2D GEO5 programs using "GeoClipboard".



Frame "Interface"

Location

The frame "**Location**" serves to specify the beam location. One needs to specify the beam origin - point having x and z coordinates.



Frame "Location"

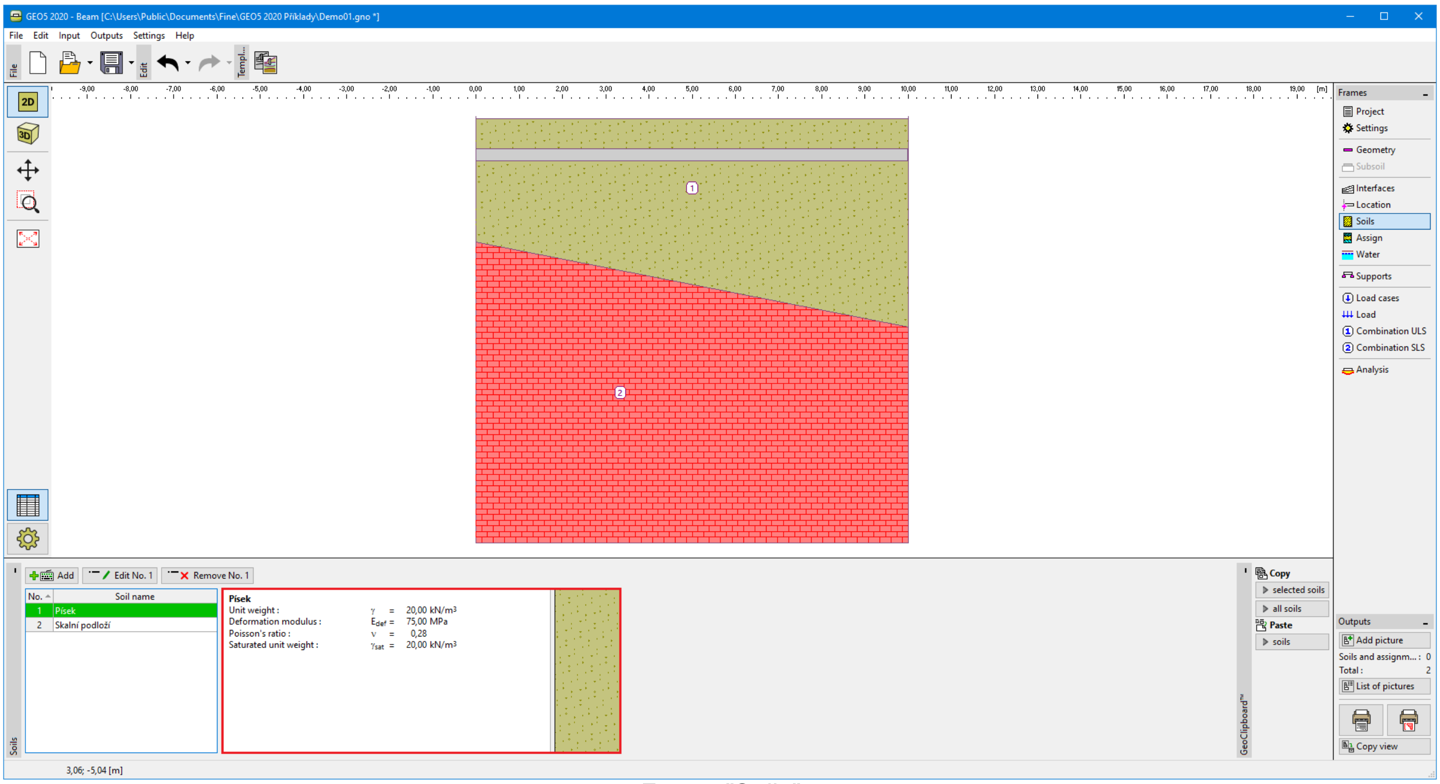
Soils

The frame "**Soils**" contains a table with the list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Settlement - oedometric modulus", "Settlement - determination of the depth of influence zone" and "Uplift pressure".

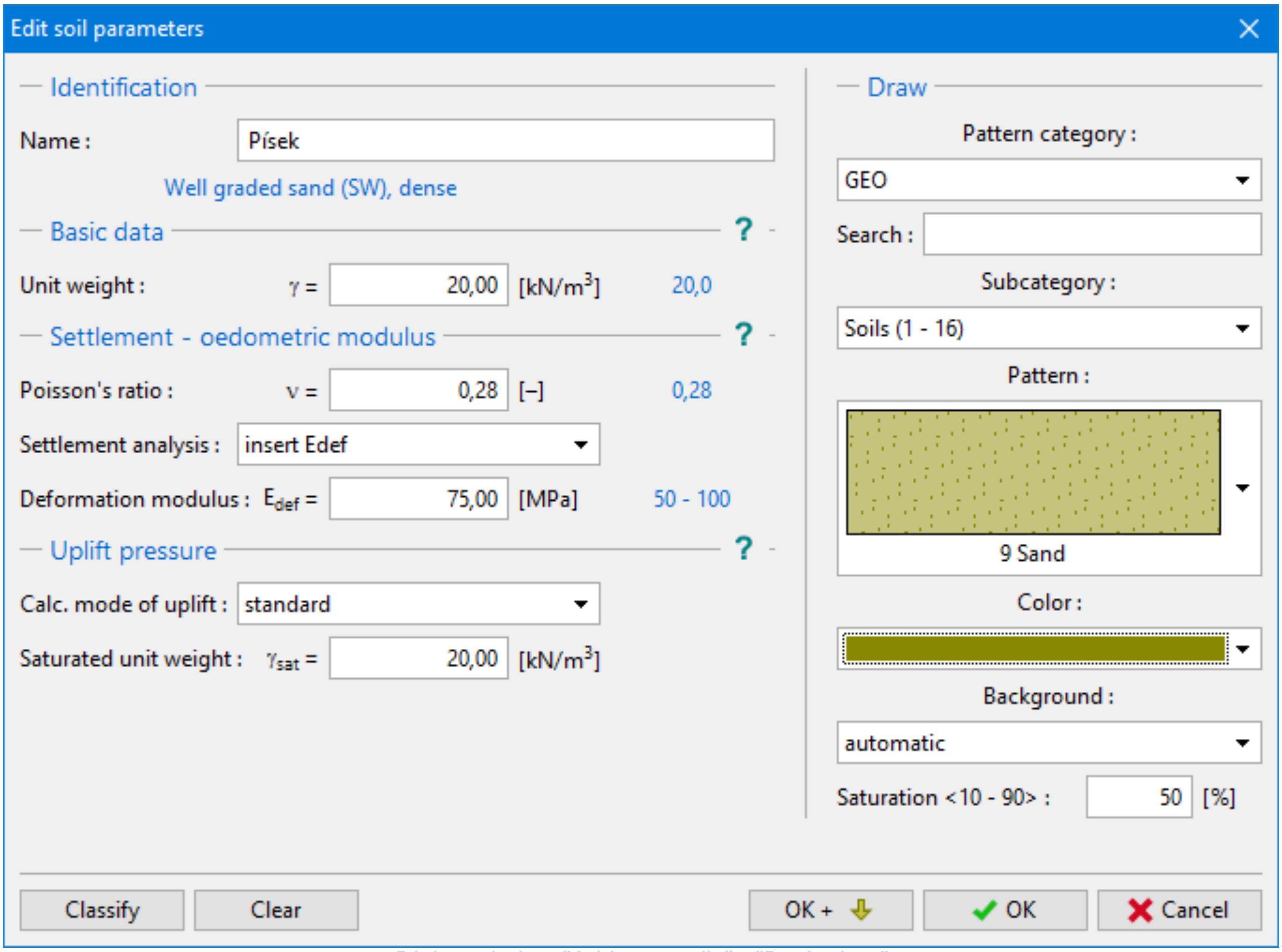
Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

Basic Data

This part of the dialog window serves to specify **the unit weight**.



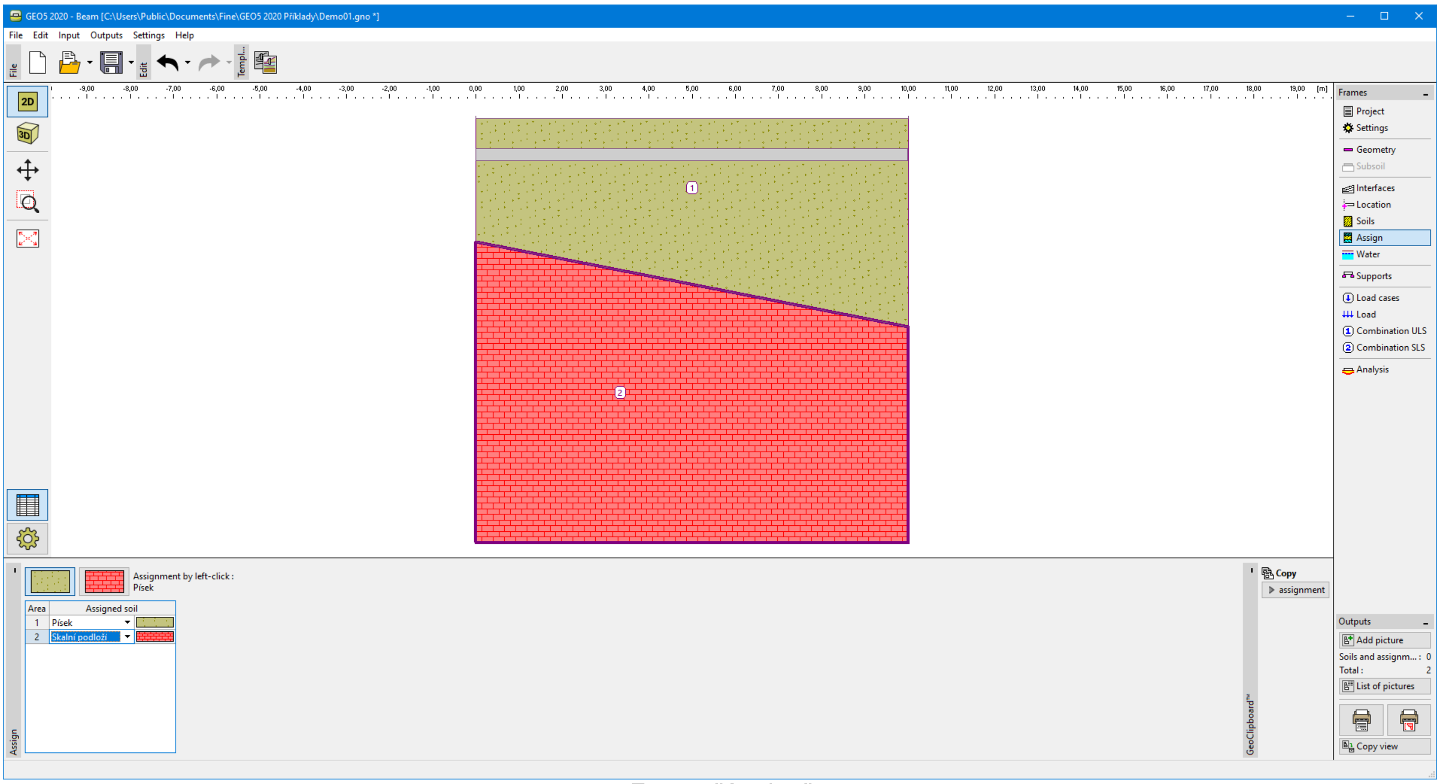
Dialog window "Add new soils" - "Basic data"

Assign

The frame "**Assign**" contains a list of layers of the profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or it is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail herein.

Assign of soils can be copied within all 2D GEO5 programs using "GeoClipboard".

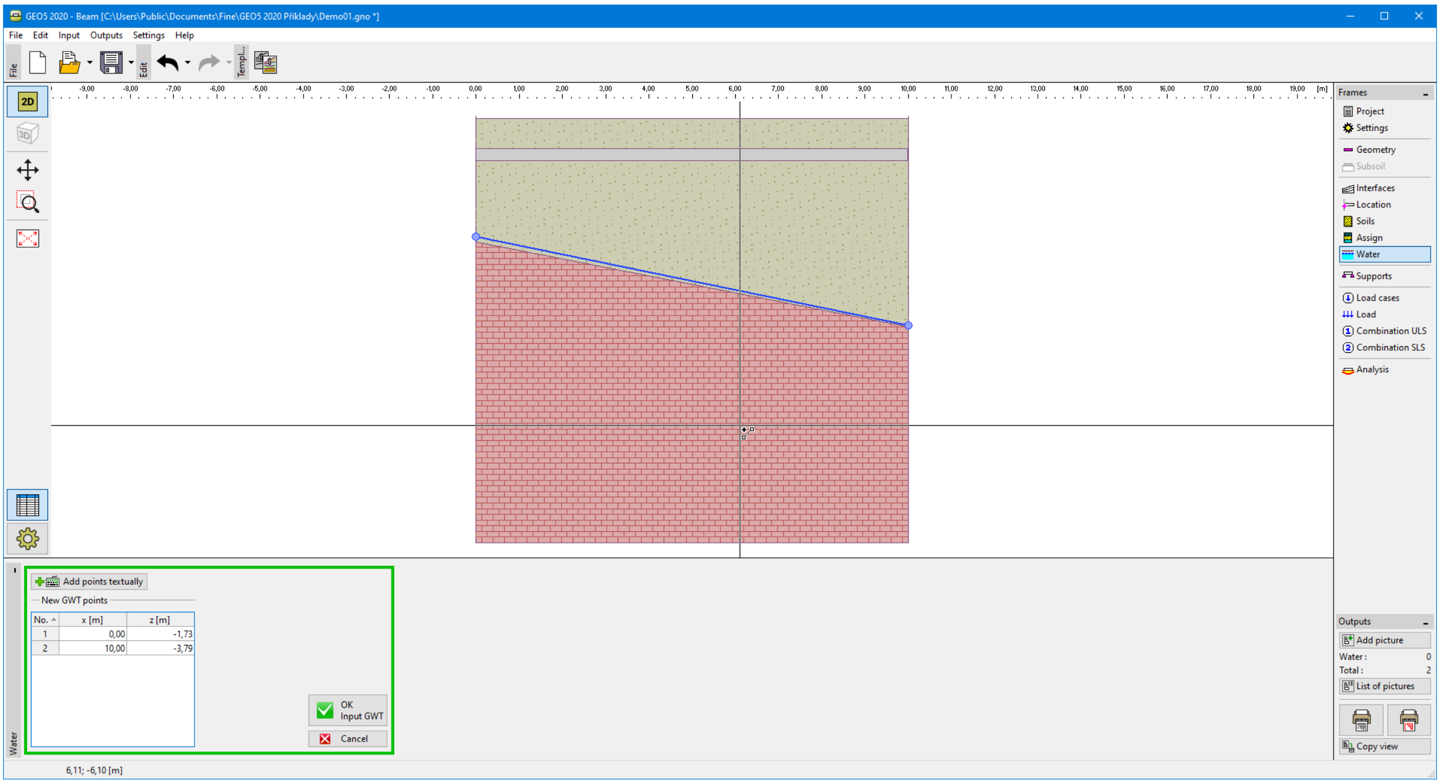


Frame "Assign"

Water

The frame "**Water**" serves to input the groundwater table. Distribution GWT is introduced the same way as interfaces of soils.

Input interfaces of water can be copied within all 2D GEO5 programs using "GeoClipboard".



Frame "Water"

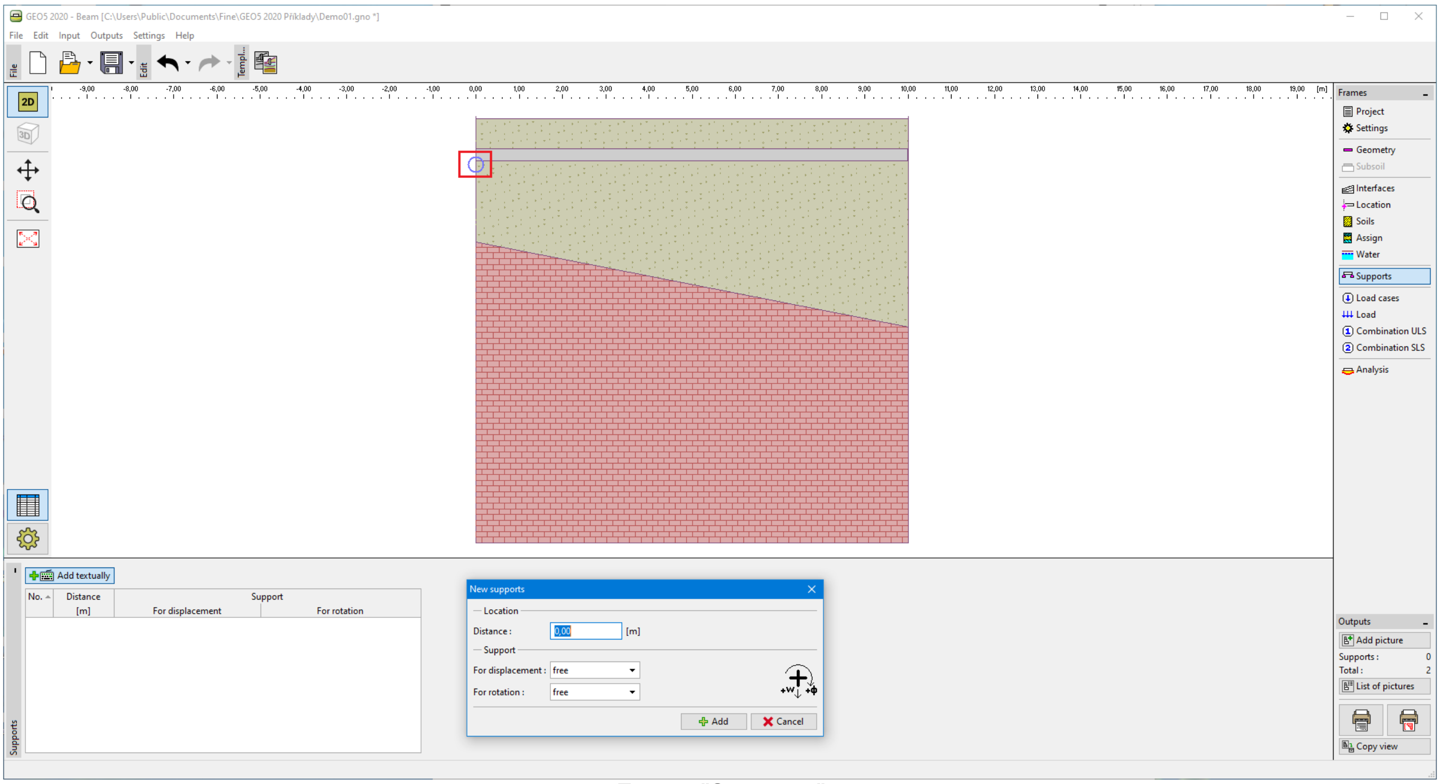
Supports

The frame "**Supports**" contains a table with the list of input supports. Adding supports is performed in the "**New supports**" dialog window. Editing can be carried out with the help of the "Modify" button or by clicking the row with the required support in the list using the left mouse button.

The type of support is determined according to a particular boundary condition specified at a given point (translation, rotation).

The following **boundary condition can be specified at a point**:

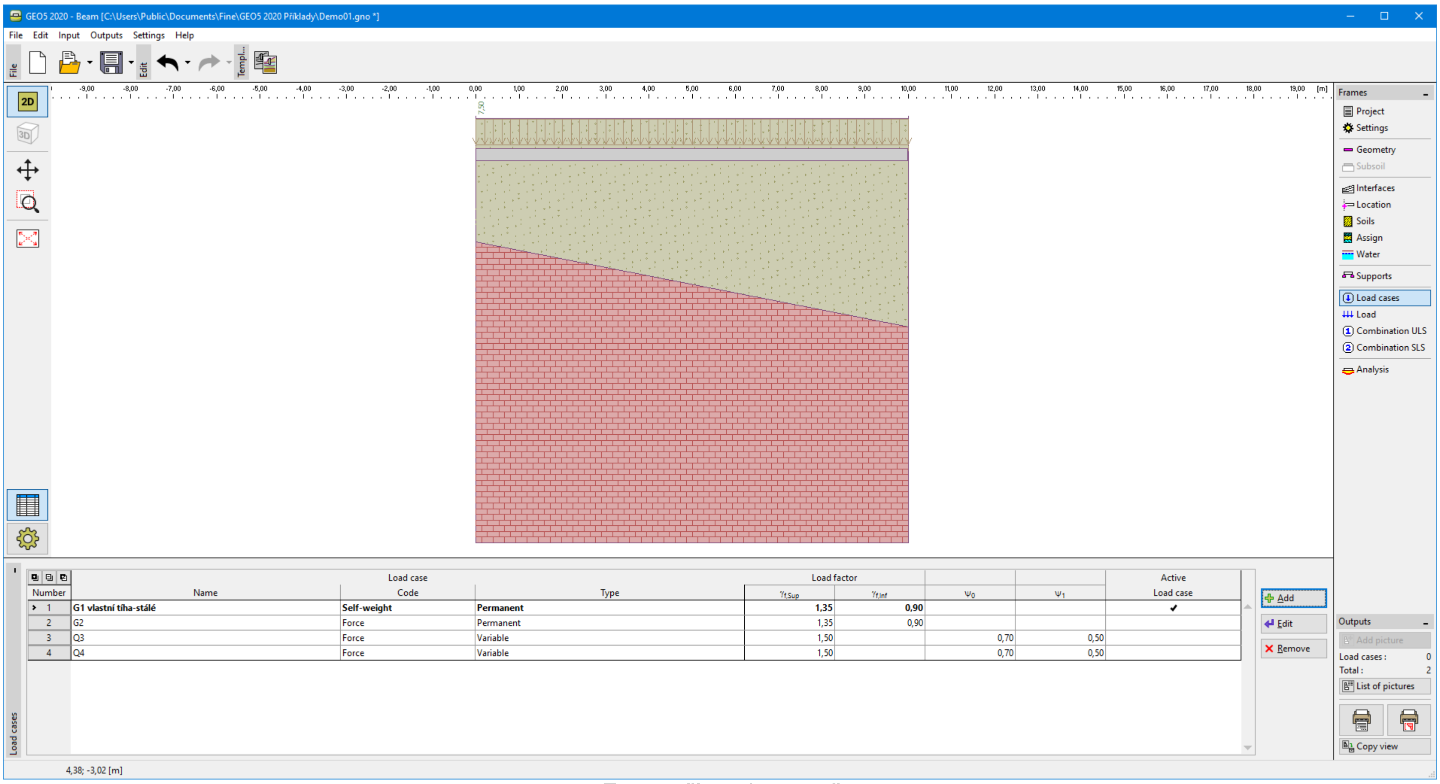
- free
- fixed
- deformation
- spring



Frame "Supports"

Load Cases

The frame "**Load cases**" contains a table with the list of input load cases. Adding (editing) load cases and their parameters is performed in the "**New load case**" dialog window. Editing can be carried out with the help of the "Modify" button or by clicking the row with the required load case in the list using the left mouse button.



Frame "Load cases"

Load Case parameters

The following parameters are defined in the "**New load case**" dialog window:

Load case identifier

The load case identifier, which is composed of the load case number and a uniliteral prefix, is displayed in front of the field for entering the name of the load case. The prefix is determined by the type of load case:

G - permanent load
Q - variable load
A - accidental load

The load case identifier is mainly used in printouts of combinations.

Load case code

The load case code determines, what load can be specified for this load. The following options are available.

Self-weight - In this load case the load represents the structure self-weight and it is generated automatically by the program. Only one load case with this code can be considered in each task.

Force - An arbitrary type of force load (forces, moments) can be introduced into the load case with this code. The number of LCs is not limited.

Load type

It determines the character of load cases based on their variability in time. Selecting a particular type of load corresponds to classification according to EN 1990 standard, art. 4.1.1.

Load coefficients

It allows for specifying the load partial factor γ_f. This coefficient accounts for unfavorable deviations of values of loads from the representative ones. For permanent load it is necessary to introduce different values for favorable ($\gamma_{f,\,inf}$) and for unfavorable ($\gamma_{f,\,sup}$) load action in a combination. If the load input follows EN 1990 the default values of coefficients are taken from table A1.2(B).

Category

Classification of load cases into categories corresponds to the classification of load according to table A1.1 of EN 1990 standard. Based on this the variable load cases are assigned combination coefficients ψ_0 , ψ_1 a ψ_2 . The category of "**User-defined input**" allows for defining the user self-values of these coefficients. Choosing a category is possible only for load cases input, according to EN 1990 (the "Materials and standards" tab serves to select the particular standard).

Combination coefficients

Basic values of coefficients to create combinations arise from EN 1990 standard and depend on the load case category. When user input is assumed, it is possible to define the user self-values of these coefficients. The following coefficients are used to create a combination:

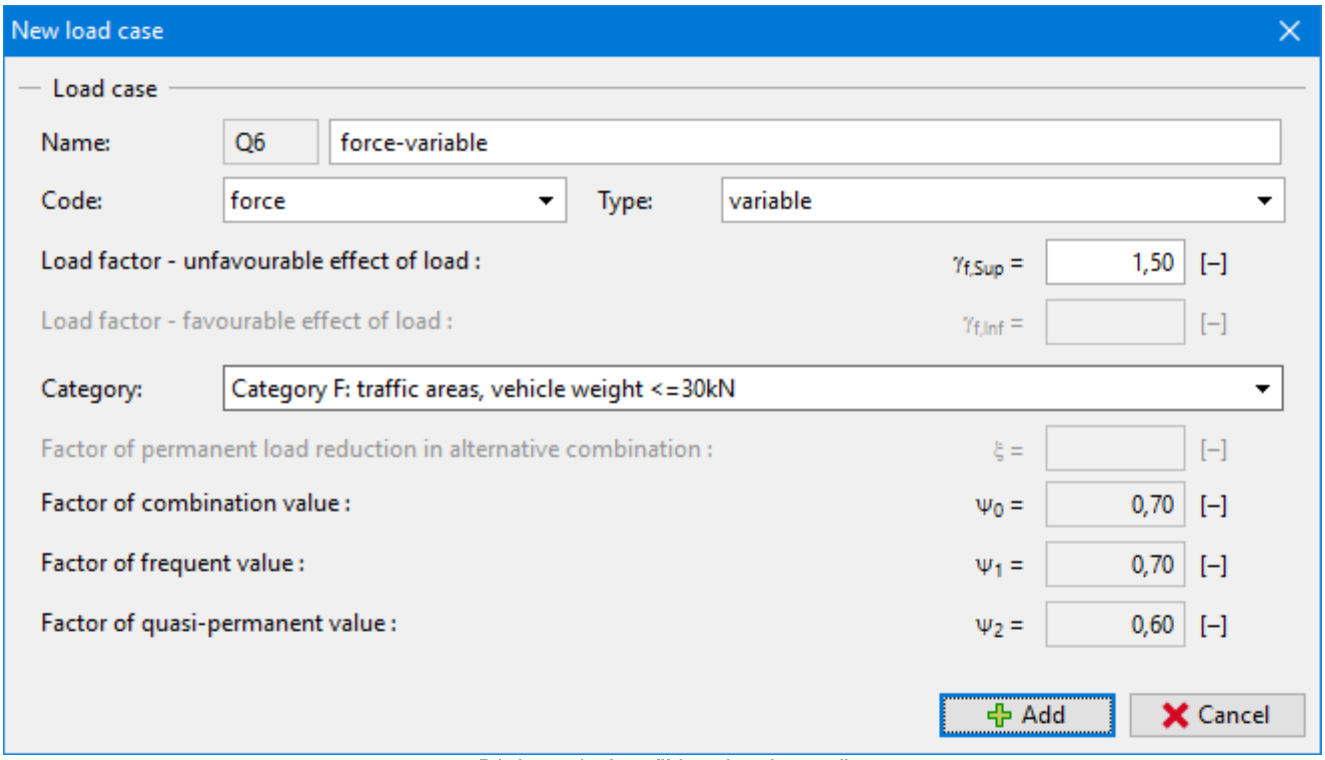
ξ - **Coefficient of reduction of permanent loads in alternative combination** - this coefficient is assigned to all permanent loads and is used when compiling alternative combinations for the bearing capacity limit state (combination to relation 6.10.b, EN 1990).

ψ_0 - **Coefficient of combination value** - coefficient for variable loads, it is used when compiling combinations for both the bearing capacity and service limit states

ψ_1 - **Coefficient of frequent value** - coefficient for variable loads, it is used when compiling accidental combinations and combinations for the service limit state

ψ_2 **Coefficient of quasi-permanent value** - coefficient for variable loads, it is used when compiling accidental combinations and combinations for the service limit state

The combination coefficients are available only for load cases input according to EN 1990 (the frame "Analysis methods" serves to select the particular standard).

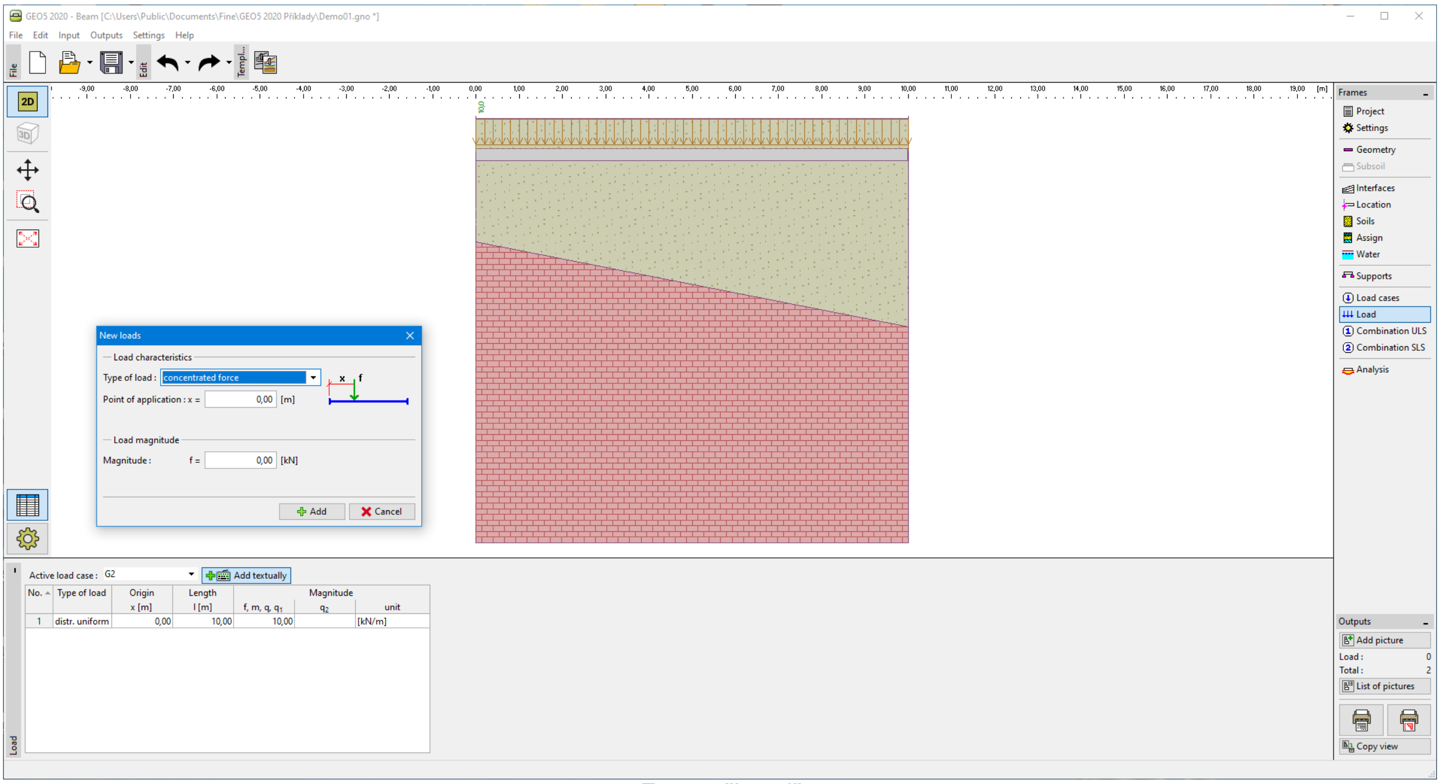


Dialog window "New load case"

Load

The frame "**Load**" contains a table with the list of input loads. Adding (editing) a load is performed in the "**New loads (Edit loads)**" dialog window. Editing can be carried out with the help of the "Modify" button or by clicking the row with the required load in the list using the left mouse button.

Each load is assigned to a load case. The load case can be selected from the "**Active load case**" combo list above the table.

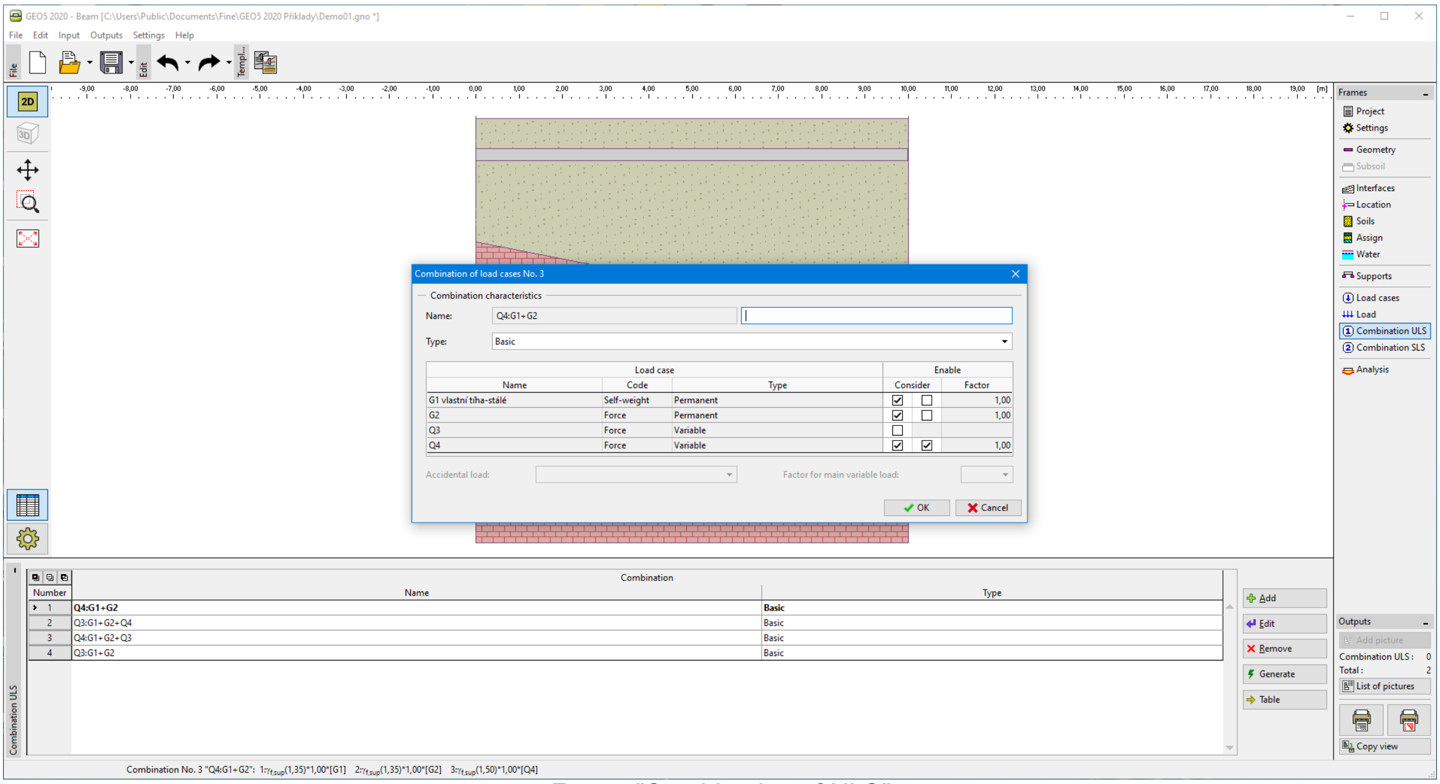


Frame "Load"

Combination ULS

The frame "**Combinations ULS**" contains a table with the list of input combinations of the bearing capacity limit state. Adding (editing) combinations and their parameters is performed in the "**New combination of load cases**" dialog window. Editing can be carried out with the help of the "Modify" button or by clicking the row with the required combination in the list using the left mouse button.

The built-in generator of combinations of load cases can be used to create individual combinations.



Frame "Combination of ULS"

Parameters of ULS Combinations

The following parameters are specified in the "**New combination of load cases**" dialog window:

New combination

A brief description of combination is displayed in front of the field where the combination is defined. All considered load cases are tagged using their identifiers. The major variable loads are moved at the beginning of the list and separated from the remaining LCs by colon.

Type of combination (for combinations based on EN 1990 only)

The following combinations can be created for the bearing capacity limit state:

Basic - Basic combination based on expression 6.10 of EN 1990 standard

Alternative - Combinations based on expressions 6.10a and 1.10b of EN 1990 standard. In this case, two variants of combination are used in the analysis, one with reduced permanent LCs and the other with reduced major variable LC.

Accidental - Accidental combination based on 6.11 of EN 1990 standard.

Selection of load cases

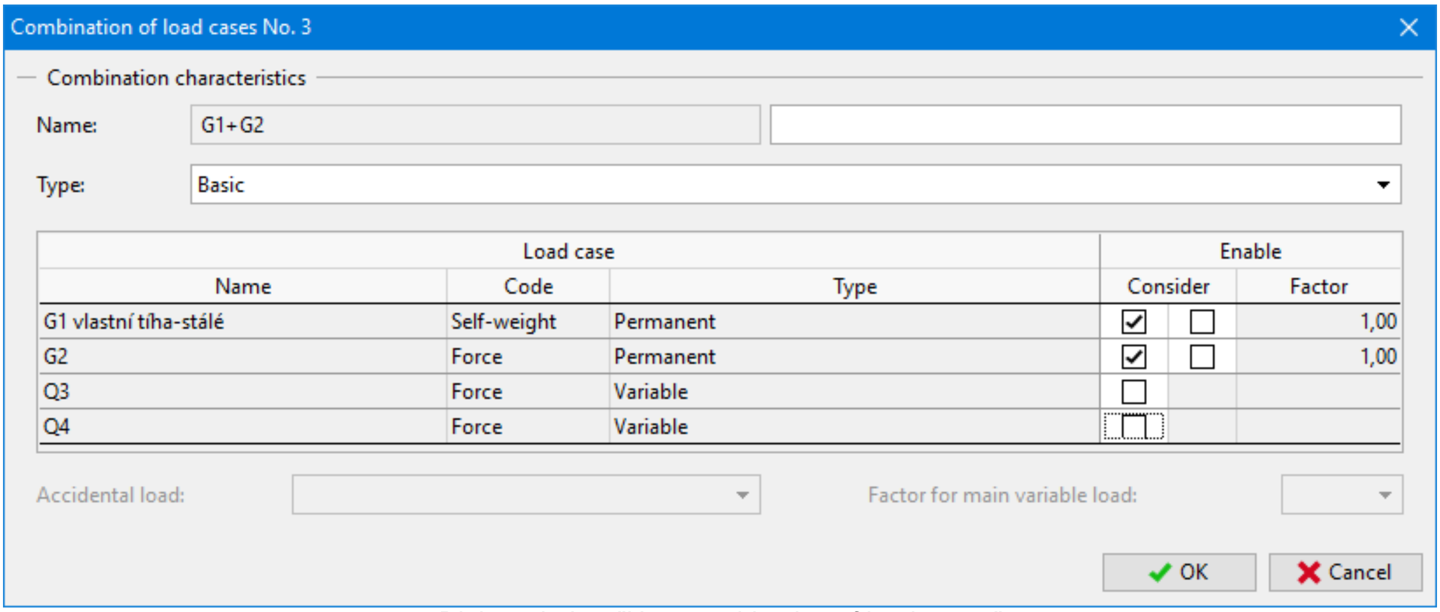
The table listing individual load cases allows for their selection to create a combination. The load case can be introduced into a combination by checking the field in the column "**Consider**" for a particular LC. Further setting in the table depends on the selection of way of inputting loads in the "Materials and standards" tab.

Load according to EN 1990:

A second field is available for each load case in the column "**Consider**". This field allows for assigning a favorable effect of action to permanent LCs (adopting coefficient $\gamma_{f,\,inf}$) or for specifying a variable load as the major one, respectively. The number of major variable loads in the combination is not limited. An accidental load can be introduced into combinations tagged as "**Accidental**" (only LCs tagged as "**Accidental**" are available for the selection). For accidental combinations it is, also necessary to choose whether a major variable load should be reduced by the coefficient ψ_1 or ψ_2.

General load

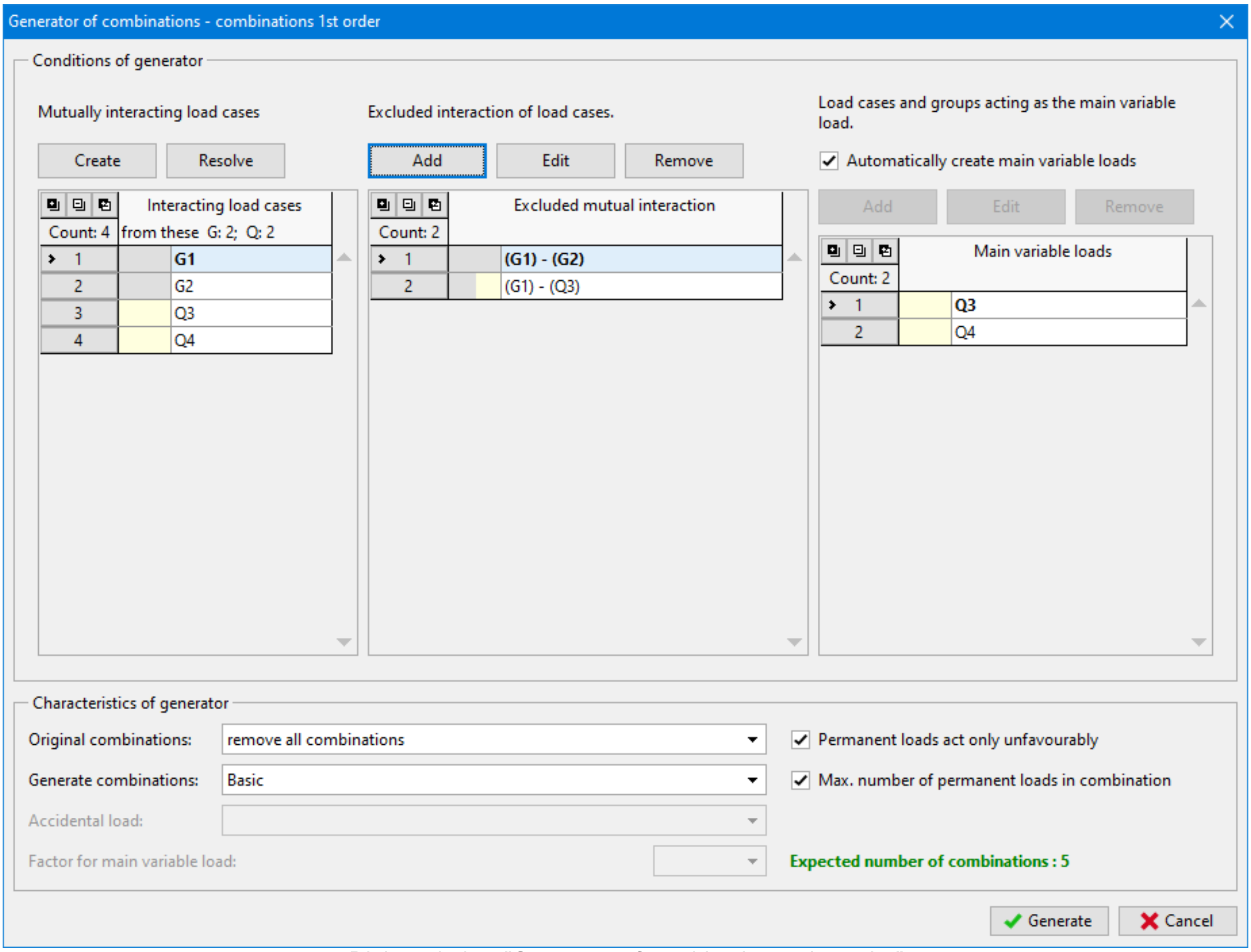
A coefficient of usability can be specified for each load case to adjust the degree of usability of the load case in the combination.



Dialog window "New combination of load cases"

Generator of Combinations

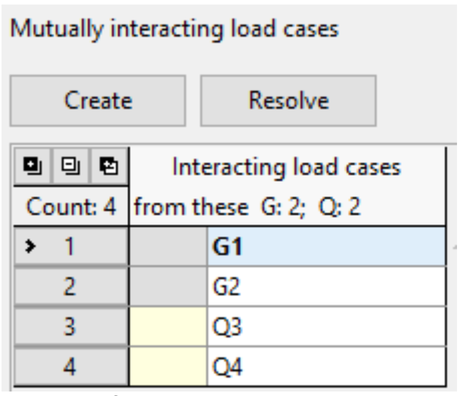
The "**Generator of combinations - 1st order**" dialog window allows for a collective compilation of combinations of load cases based on the introduced combination rules. Referring to standard EN 1990 the number of generated combinations can be relatively large and in extreme cases could considerably slow down calculations. Owing to this, information about expected number of combinations to be generated is displayed in the right bottom corner. Therefore, before launching generation the user may check, how many combinations will be generated and possibly adjust generator conditions. The top part of the window serves to define conditions for generating combinations; the bottom part contains various generator settings.



Dialog window "Generator of combinations - 1st order"

Mutually interacting load states and groups

This part makes it possible to merge those load states that should appear in combinations always together. Permanent and variable loads cannot be merged into one group. If the field "**All permanent loads always in combination**" is checked in the Generator parameters, the creation of groups of permanent loads has no effect on their appearance in combinations as each generated combination will always contain all permanent LCs. In such case, merging permanent LCs will only influence consideration of favorable/unfavorable effects of LCs providing the field "**Permanent loads act only unfavorably**" is not checked.



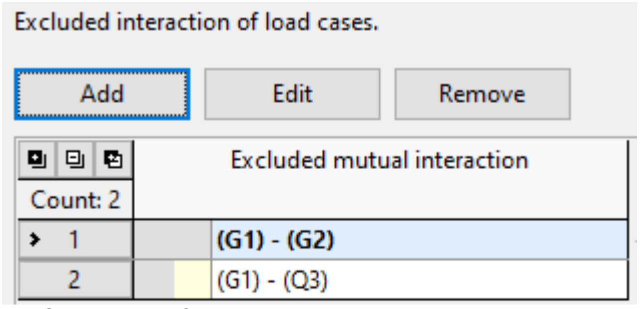
Dialog window "Generator of combinations" - Mutually interacting load states

Excluded interaction of load states

This part makes possible to define, which LCs should not appear in a combination together. Arbitrary load cases or merged groups can be mutually excluded in dependent of the type of load case. Two options are available to define groups to be excluded:

Mutual exclusion - An arbitrary number of load cases can be introduced into one group. In such a case, the program will not generate any combination that contains at least two load cases from this group.

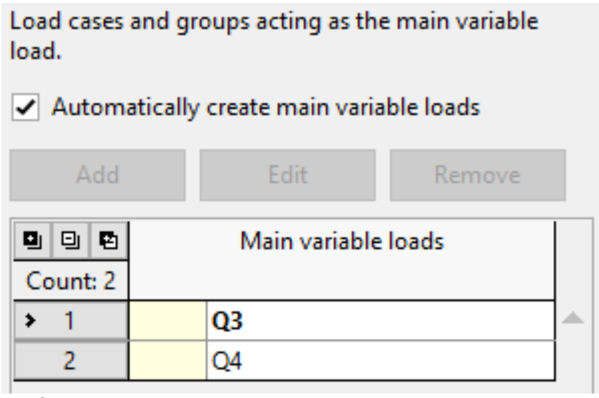
Exclusion by pairs - Providing it is necessary to create a larger number of excluding groups of two sorts, where one LC is the same (e.g. exclusion of assembly variants of permanent loads with all service load cases), it is possible to adopt this option. A load case to be excluded is first selected in the first column. The second column is then used to select an arbitrary number of LCs, which are needed to create excluding groups.



Dialog window "Generator of combinations" - Excluded interaction of load states

Load cases and groups acting as the main variable load

This part is available only when inputting loads according to EN 1990 is considered (the standard is selected in the "Materials and standards" tab). When an automatic regime is assumed, then each variable load is taken as major in created combinations. If this regime is turned off, it is possible to manually adjust the list of major variable loads. For example, it is possible to remove an arbitrary load case from the list so that it will not be considered as major variable in combinations. If a new item with more load cases is add to the list then all load cases will be considered as major variable in those combinations, where they appear together.



Dialog window "Generator of combinations" - Load cases and groups acting as the main variable load

Generator parameters (parameters that can be set in the bottom part of the dialog window).

Combo list "Original combinations"

Retain original combinations - By pressing the "**Generate**" button the program will add new combinations, created according to the specified rules, to the original ones

Remove all combinations - By pressing the "**Generate**" button the program will delete all original combinations and will replace them by the new ones

Remove generated combinations - By pressing the "**Generate**" button the program will delete older combinations and will add new ones created according to the specified rules

Remove all combinations of the current type - By pressing the "**Generate**" button the program will delete all combinations of a given type and will replace them by the new ones

Remove generated combinations of the current type - By pressing the "**Generate**" button the program will delete older combinations of a given type and will add new ones created according to the specified rules

Combo list "Generate combinations"

The following types of generated combinations can be chosen for loads based on EN 1990:

Basic - Generates basic combinations for the bearing capacity limit state based on expression 6.10 of EN 1990 standard

Alternative - Generates combinations for the bearing capacity limit state based on expressions 6.10a and 1.10b of EN 1990 standard. This variant generates two times more combinations but it provides better results.

Accidental - Generates accidental combinations for the bearing capacity limit state based on 6.11 of EN 1990 standard. An accidental load case to appear in accidental combinations can be specified. It is also necessary to choose whether a major variable load will be reduced by the coefficient ψ_1 or ψ_2.

Permanent loads act only unfavorably

If this setting is not checked, the program creates all possible combinations, where introduction of all variants of favorable and unfavorable actions of permanent loads will be considered.

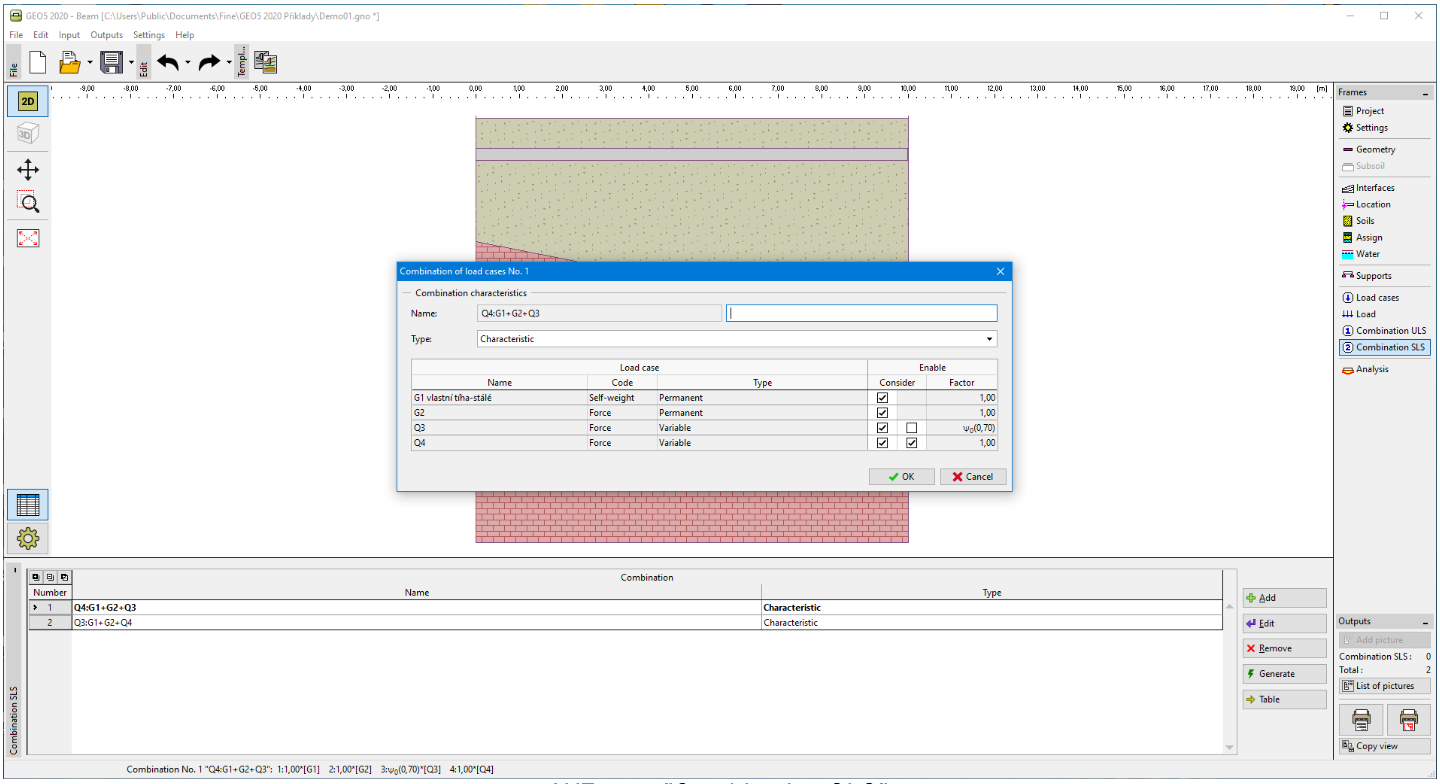
All permanent loads always in combination

If this setting is not checked, the program creates combinations in such a way that a successive introduction of all LCs into a combination will be considered.

Combination SLS

The frame "**Combinations SLS**" contains a table with the list of input combinations of the service limit state. Adding (editing) combinations and their parameters is performed in the "**New combination of load cases**" dialog window. Editing can be carried out with the help of the "Modify" button or by clicking the row with the required combination in the list using the left mouse button.

The built-in generator of combinations of load cases can be used to compile individual combinations.



WFrame "Combination SLS"

Parameters of SLS Combinations

Combinations SLS serve to evaluate states that refer to the structure appearance, comfort of people or to functioning of a structure while in ordinary use. Typically, only deformations, vibrations, etc. are checked. The "**New combination of load cases**" dialog window (similarly to combinations for ULS) serves to define the following parameters:

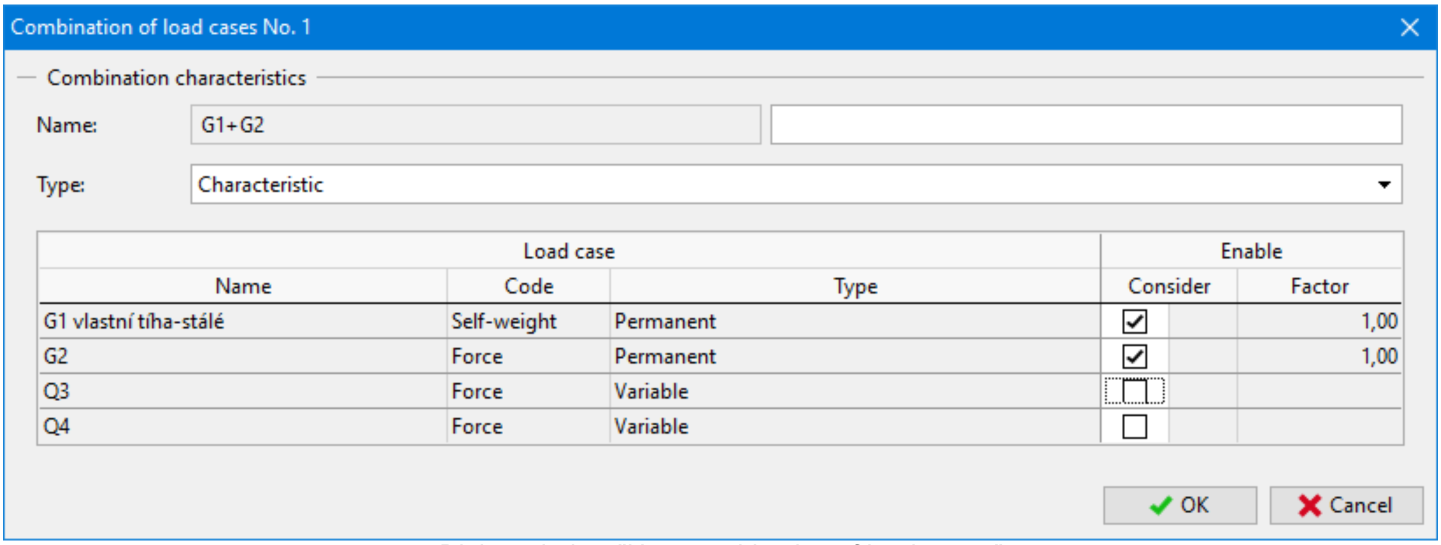
Type of combination according to EN 1990

The following combinations can be created for the service limit state:

Characteristic - combination based on expression 6.14 of EN 1990 standard
Frequent - combination based on expression 6.15 of EN 1990 standard
Quasi-permanent - combination based on expression 6.16 of EN 1990 standard

Selection of load cases

The table listing individual load cases allows for their selection to create a combination. The load case can be introduced into a combination by checking the field in the column "**Consider**" for a particular LC. A coefficient of usability can be specified for generally input combinations (select in the "Materials and standards" tab) to adjust the degree of usability of the load case in the combination.



Dialog window "New combination of load cases"

Generator of Combinations

The "**Generator of combinations - 1st order**" dialog window allows for a collective compilation of combinations of load cases for the service limit state. Functions of generator of combinations are explained in section devoted to the generator of combinations for the bearing capacity limit state.

If inputting loads according to EN 1990 is set in the "Materials and standards" tab, it is possible to generate the following combinations for the service limit state:

Characteristic - combination based on expression 6.14 of EN 1990 standard
Frequent - combination based on expression 6.15 of EN 1990 standard
Quasi-permanent - combination based on expression 6.16 of EN 1990 standard

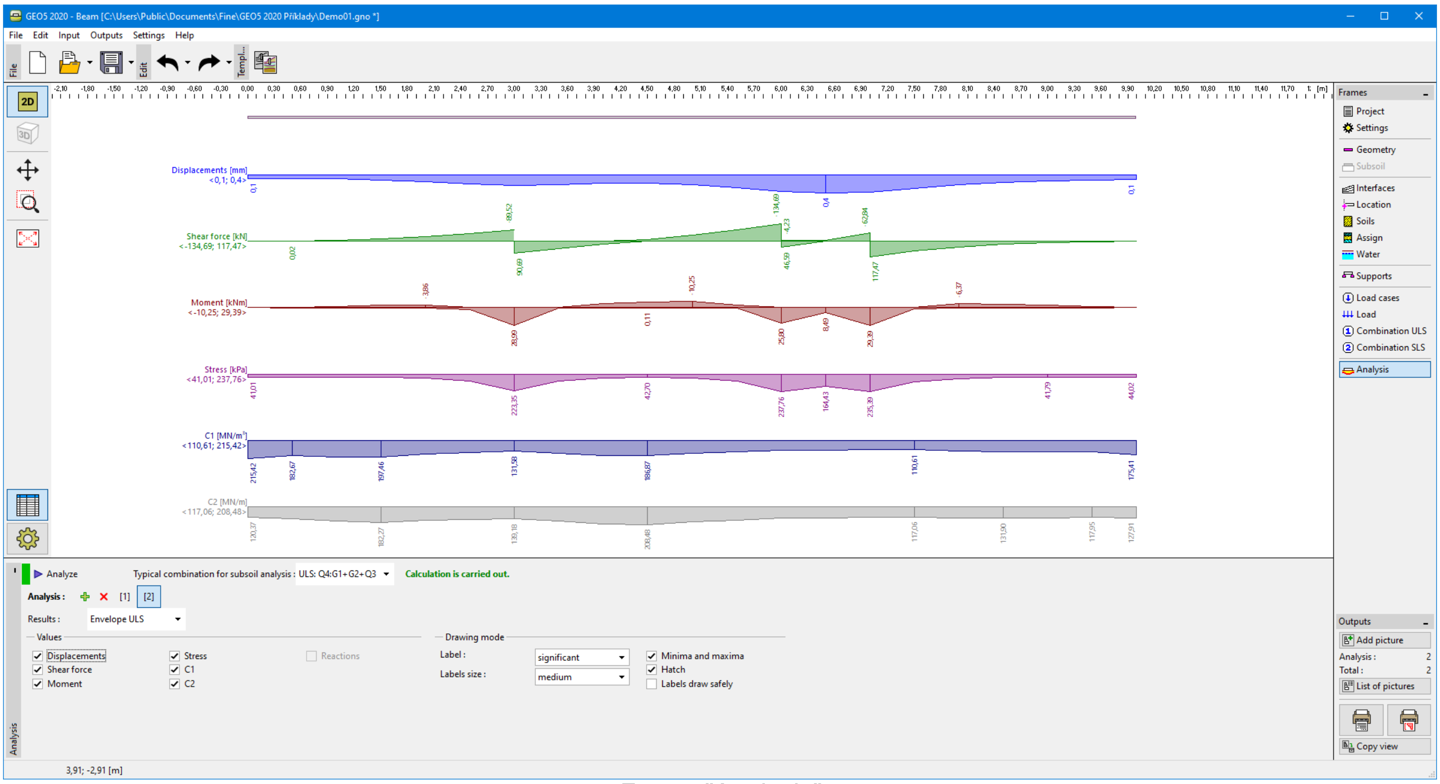
Analysis

The frame "**Analysis**" also serves to display the analysis results. The analysis is carried using the **finite element method** incorporating the Winkler-Pasternak subsoil. Several analyses, including presentation of results, can be carried out for one task. Information about the performed analysis is displayed in the top right corner of the frame. Should the analysis parameters change it is necessary to re-run the analysis by pressing the "**Analyze**" button.

If the subsoil parameters are calculated from the geological profile, it is necessary to choose "**Characteristic combination for subsoil analysis**" in the combo list.

The "**Results**" combo list serves to set combinations of load for ULS or SLS (possibly envelopes of combinations of load cases) for which the results should be displayed on the desktop.

The bottom part of the window serves to define, which variables are visualized (Displacements, Shear force, Moment…) and the way of their appearance on the desktop.



Frame "Analysis"

Program Settlement

This program is used to determine vertical settlement and time-dependent consolidation of soils under embankments, foundations, earth dams and surface loads (surcharges).

The help in the program "Settlement" includes the following topics:

- The input of data into individual frames:

  Project, Settings, Interface, Embankment, Earth Cut, Incompressible Subsoil, Soils, Assign, Surcharge, Water, Stage Settings, Analysis

- Standards and analysis methods

- Theory for analysis in the program "**Settlement**":

  Stress in Soil Body, Settlement Analysis

- Outputs

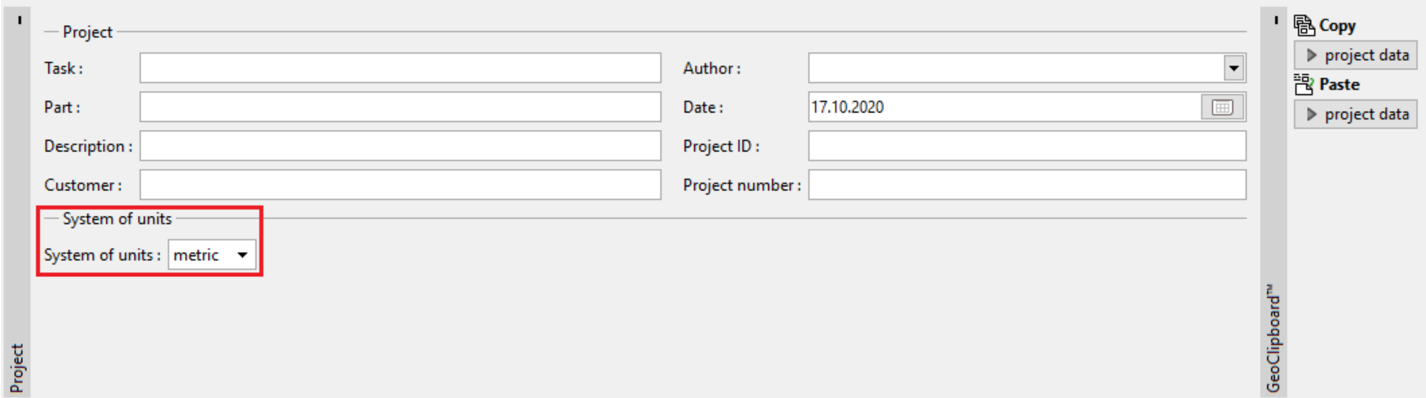
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The **"Project"** frame is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The frame "**Settings**" allows us to introduce the basic settings of the program, such as standards and theories of analysis, the way of proving the safety of a structure, and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows us to choose an already created setting from the "Settings list".

The "**Settings Administrator**" button opens the "Administrator" dialog window, which allows for viewing and modifying individual Settings. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can also be **exported and imported**.

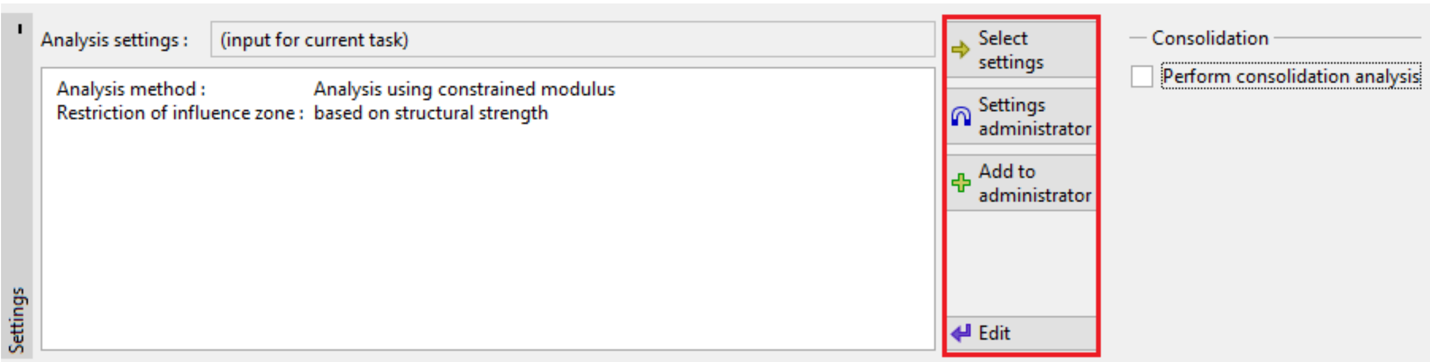
The "**Add to the administrator**" button allows us to create user-defined Settings, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

Settings of analysis parameters are performed in the "Settlement" tab.

The frame also allows us to specify whether to perform the consolidation analysis.

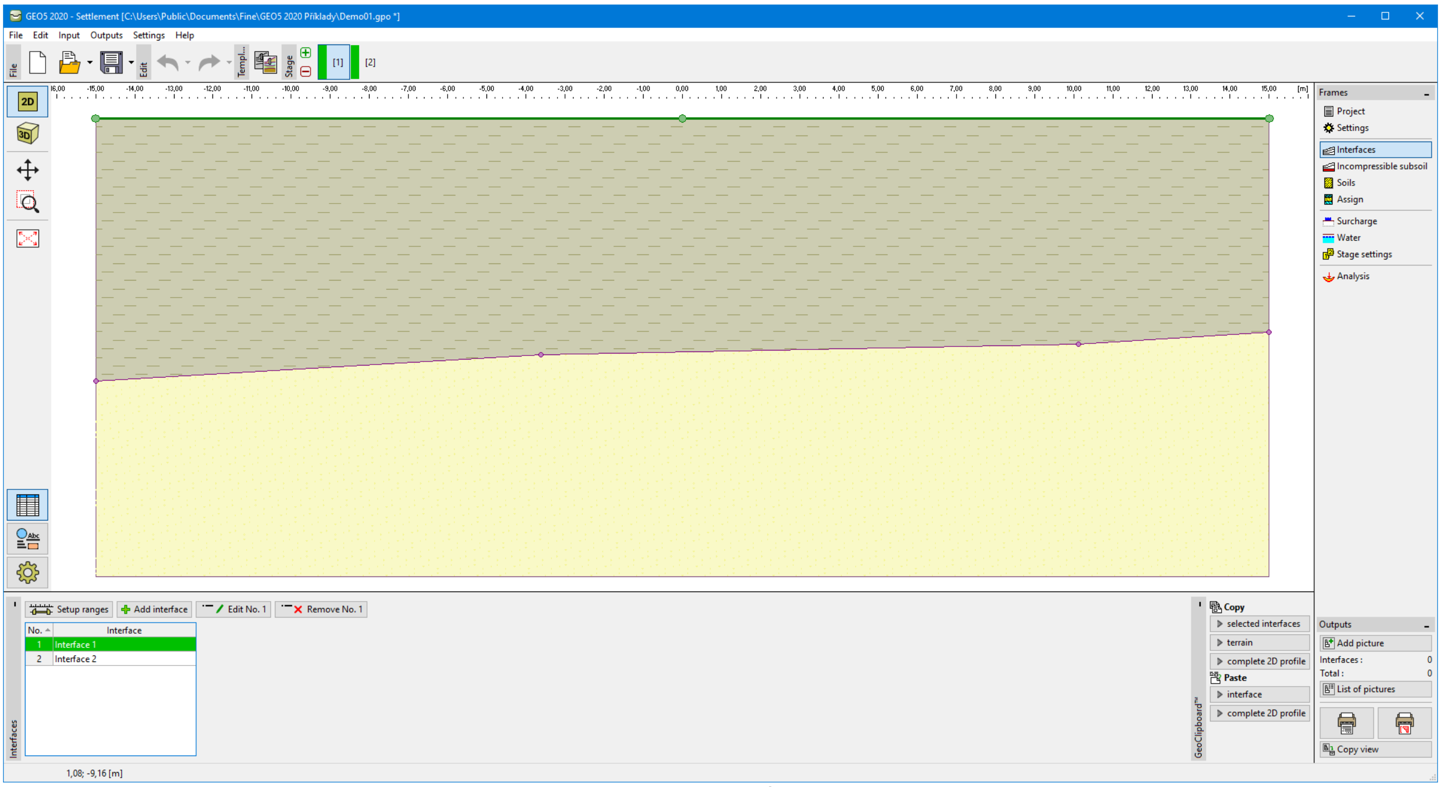


Frame "Settings"

Interface

The "**Interface**" frame serves to introduce individual soil interfaces into the soil body. Detailed description of how to deal with interfaces is described herein.

The program makes it possible to import or export interfaces in the *.DXF format. Input interfaces can be copied within all 2D GEO5 programs using "GeoClipboard".



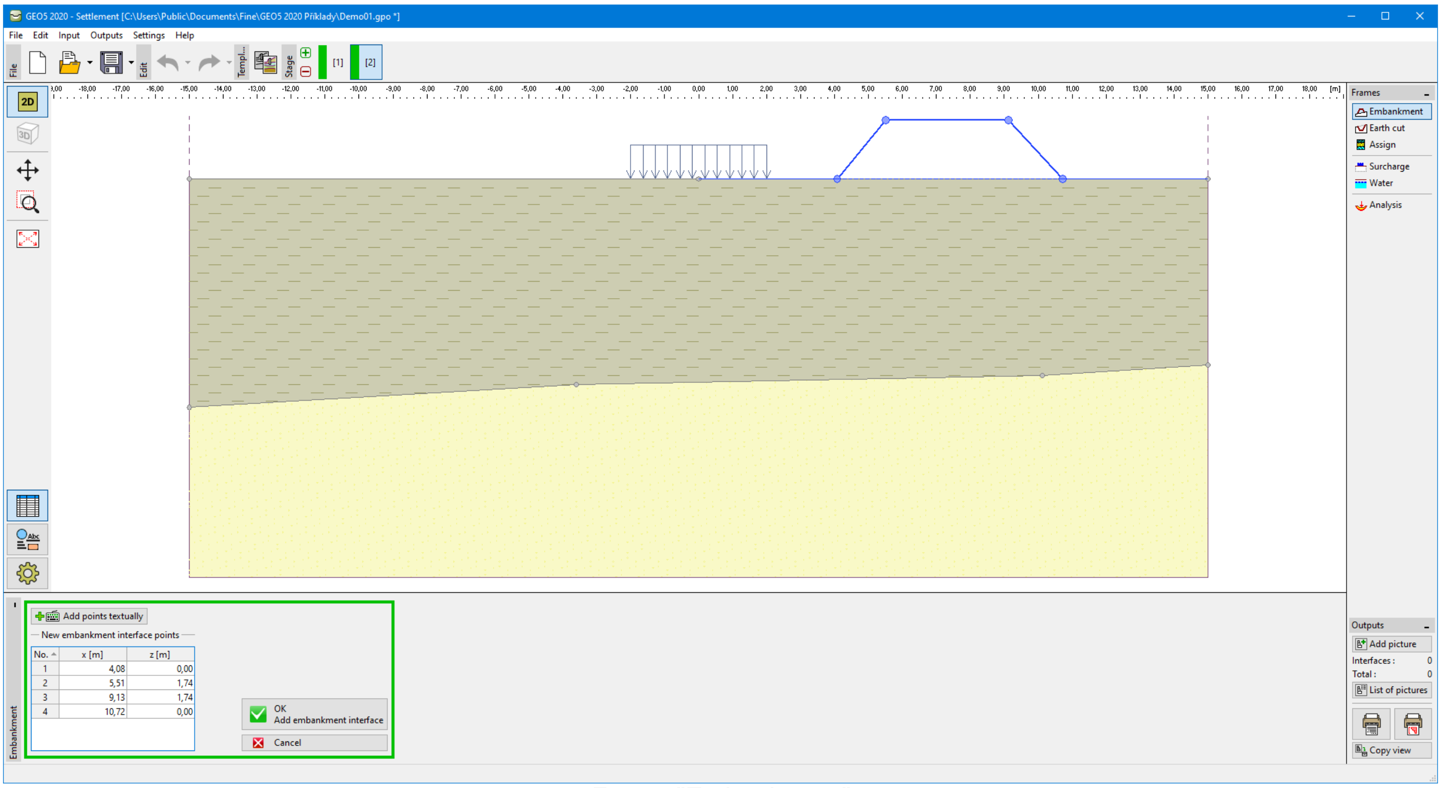
Frame "Interface"

Embankment

The "**Embankment**" frame allows for inputting interfaces to create an embankment above the current terrain. The frame contains a table with a list of interfaces forming the embankment. A table listing the points of currently selected interface of the embankment is displayed in the midsection of the frame. Inputting an embankment interface follows the same steps as used for standard interfaces.

An embankment cannot be specified in the first stage of construction. In this case, an open cut must be removed, or the embankment must be constructed in the next construction stage.

Input interfaces of an embankment can be copied within all 2D GEO5 programs using "GeoClipboard".



Frame "Embankment"

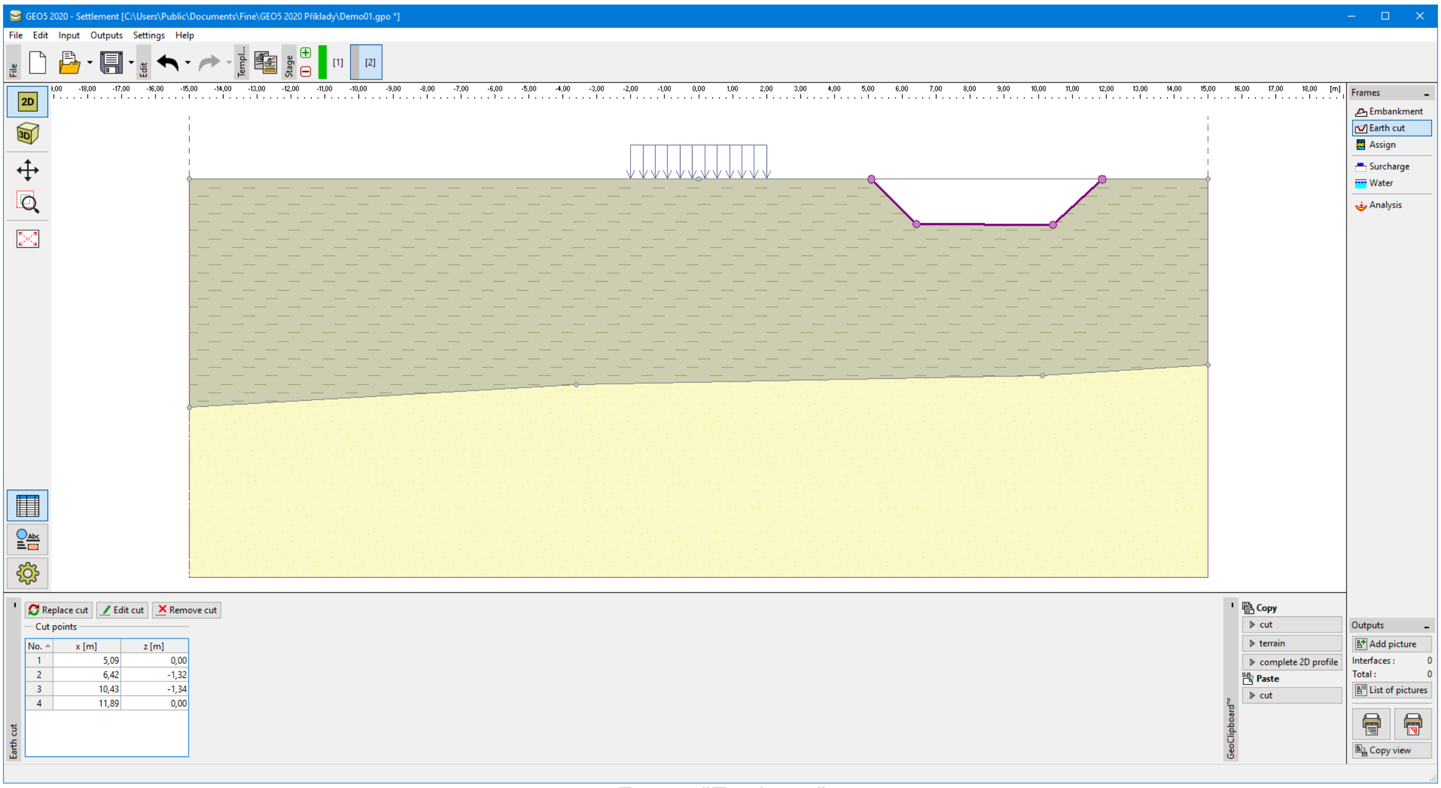
Earth Cut

The "**Earth cut**" frame serves to specify the shape of an open cut. This function allows for modifying the terrain profile within a given stage of construction. Several **earth cuts** can be introduced at the same time. In such a case, some of the lines in the cut appear partially above the terrain.

A table listing individual interface points is displayed in the left part of the frame. Inputting an earth cut interface follows the same steps as used for standard interfaces.

An earth cut cannot be built if there is an embankment already specified in a given stage. In this case, an embankment must be removed, or the earth cut must be constructed in the next construction stage.

Input interfaces of an earth cut can be copied within all 2D GEO5 programs using "GeoClipboard".



Frame "Earth cut"

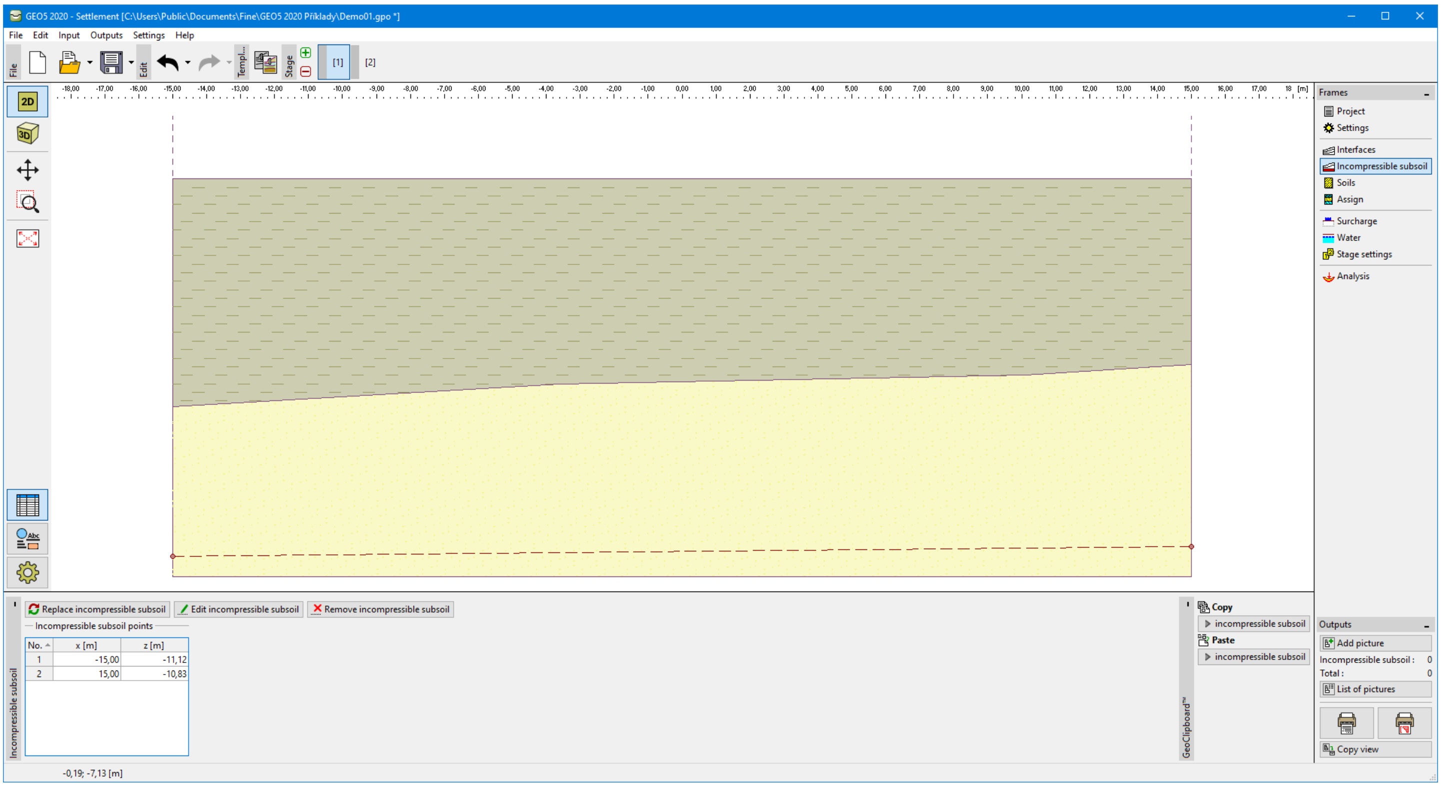
Incompressible Subsoil

The frame "**Incompressible subsoil**" serves to input a depth of **incompressible subsoil.**

Inputting the depth of incompressible subsoil is the same as when inputting standard interfaces.

Inputting an incompressible subsoil is one the options how to restrict an influence zone - if input, then both ranges and tilted sections are drawn up to a depth of incompressible subsoil. No ground deformation appears below the incompressible subsoil.

Input interfaces of an Incompressible subsoil can be copied within all 2D GEO5 programs using "GeoClipboard".



Frame "Incompressible subsoil"

Soils

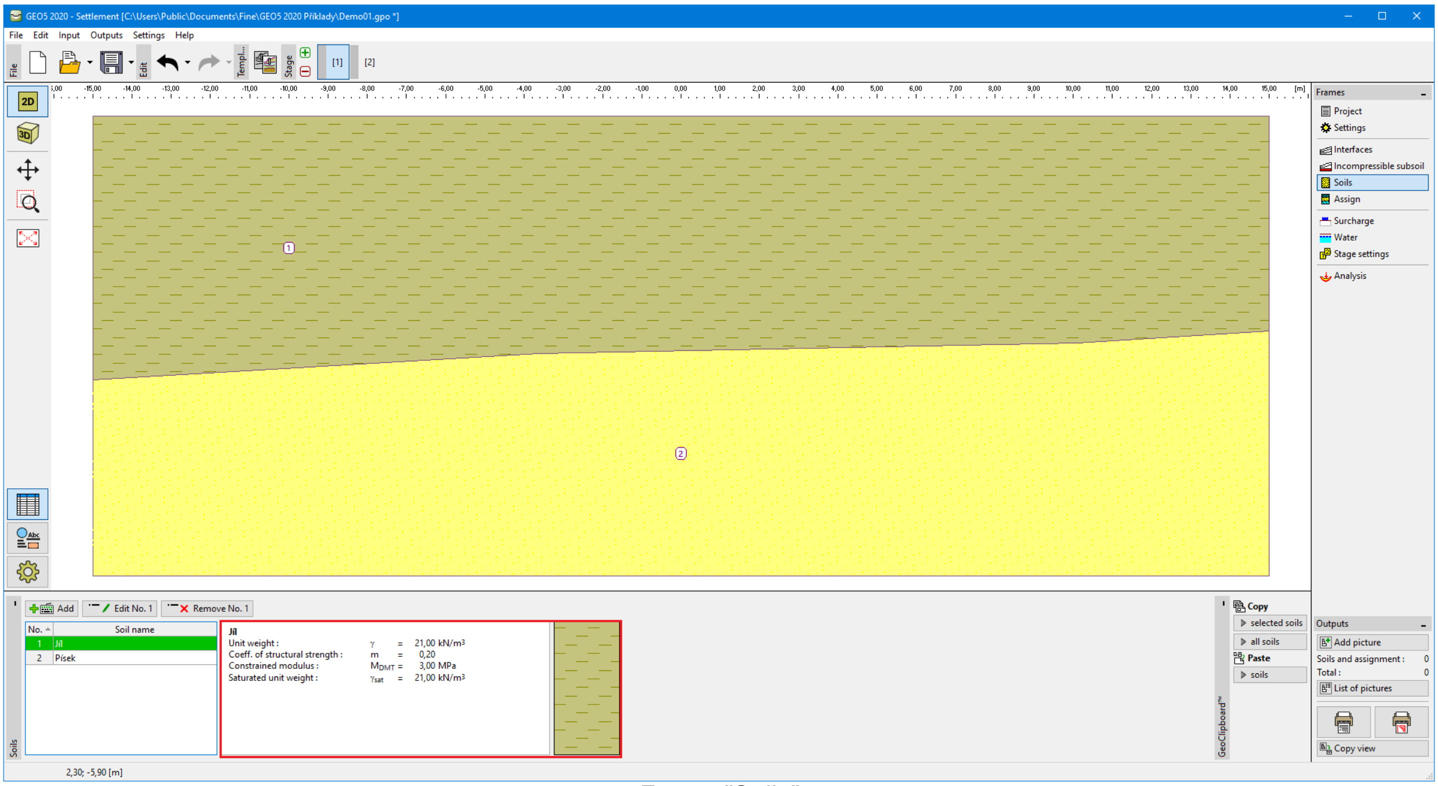
The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Uplift pressure" and "Settlement analysis". In consolidation analysis the coefficient of permeability or consolidation coefficient must be entered. The input parameters of soils are determined based on the selected theory of analysis in the "Settlement" tab.

The particular values are obtained from a geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit the built-in database of soils, which contains values of selected characteristics of soils.

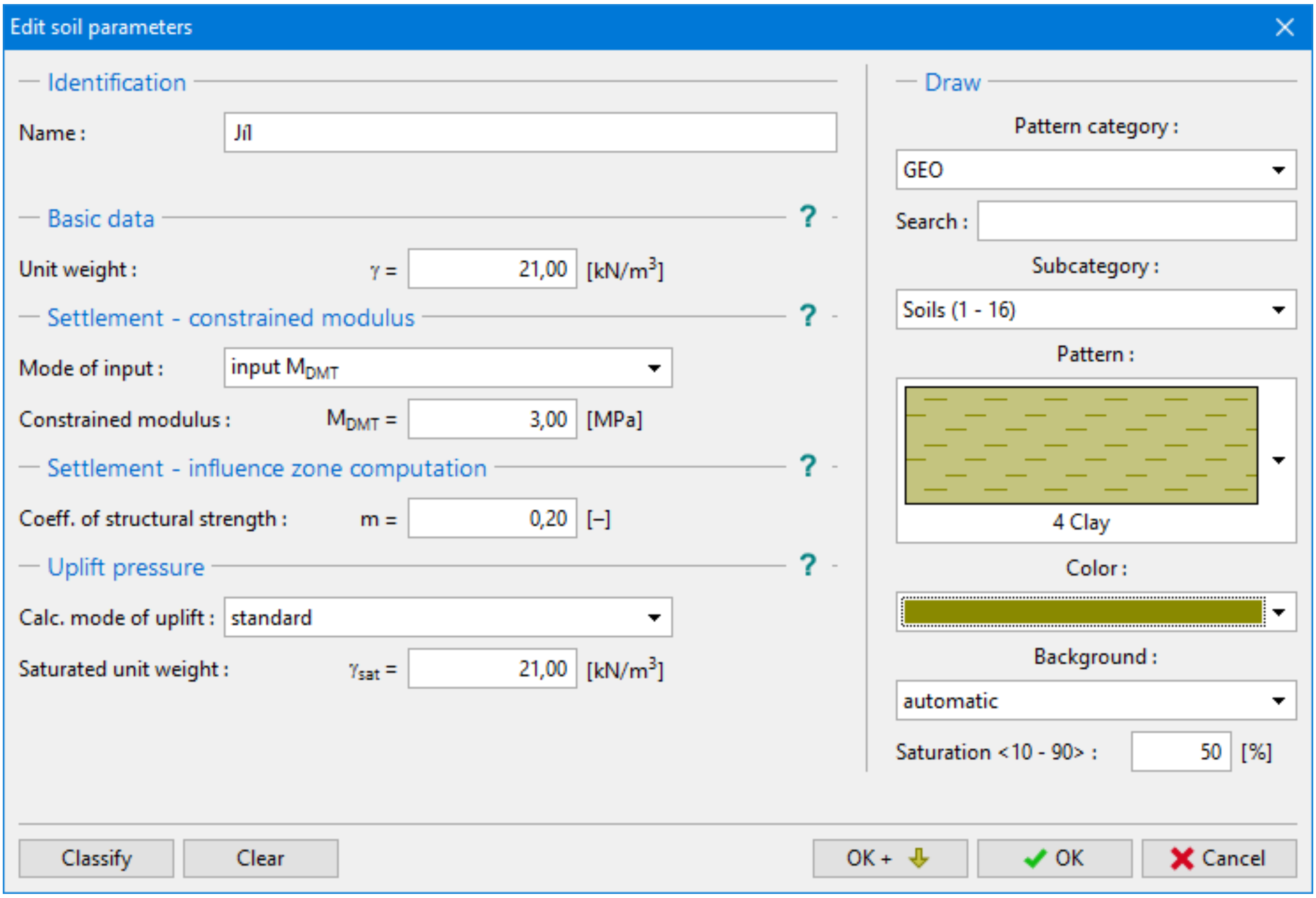
Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

Basic Data

This part of the dialog window serves to specify **the unit weight of soil**.



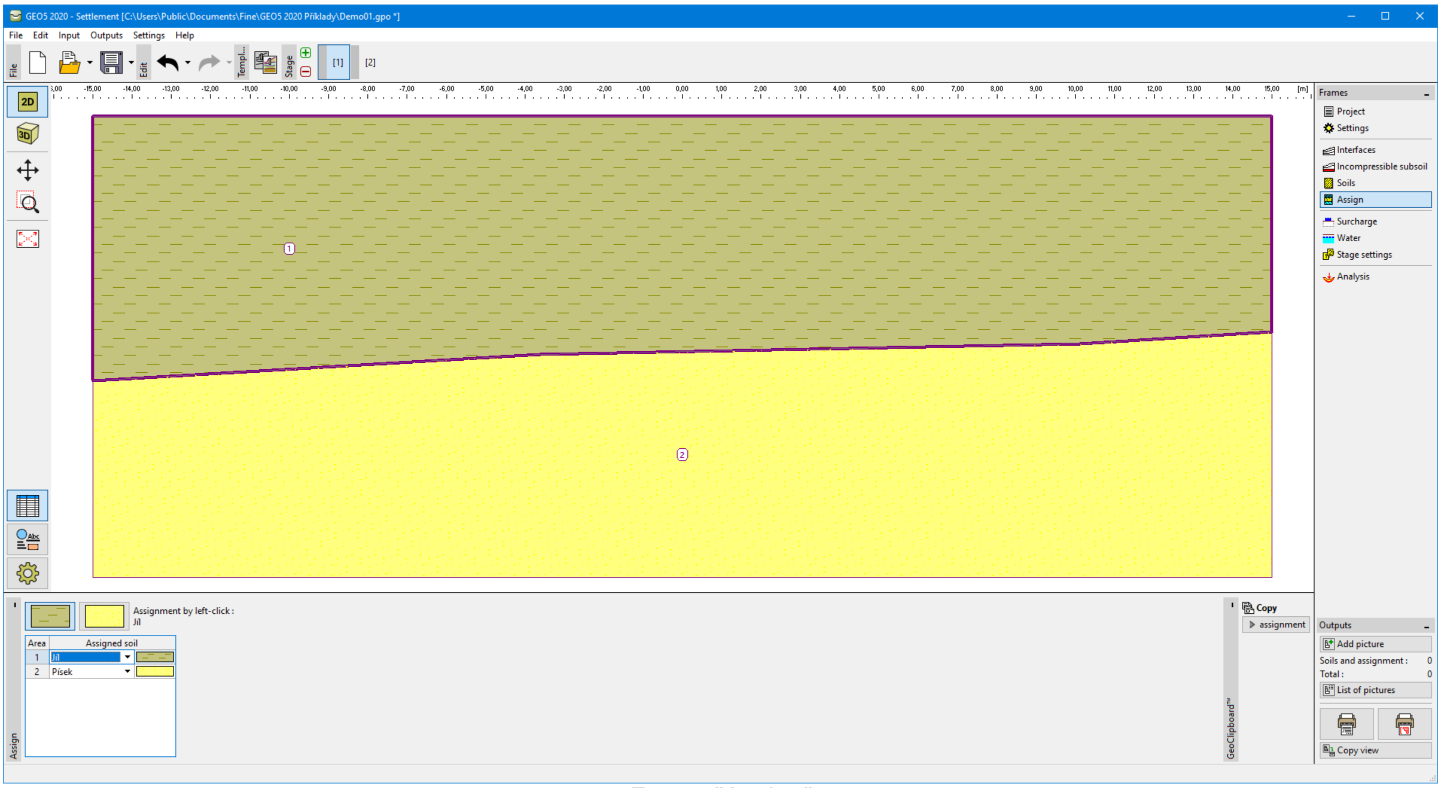
Dialog window "Add new soils" - "Basic data"

Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail herein.

Assign of soils can be copied within all 2D GEO5 programs using "GeoClipboard".



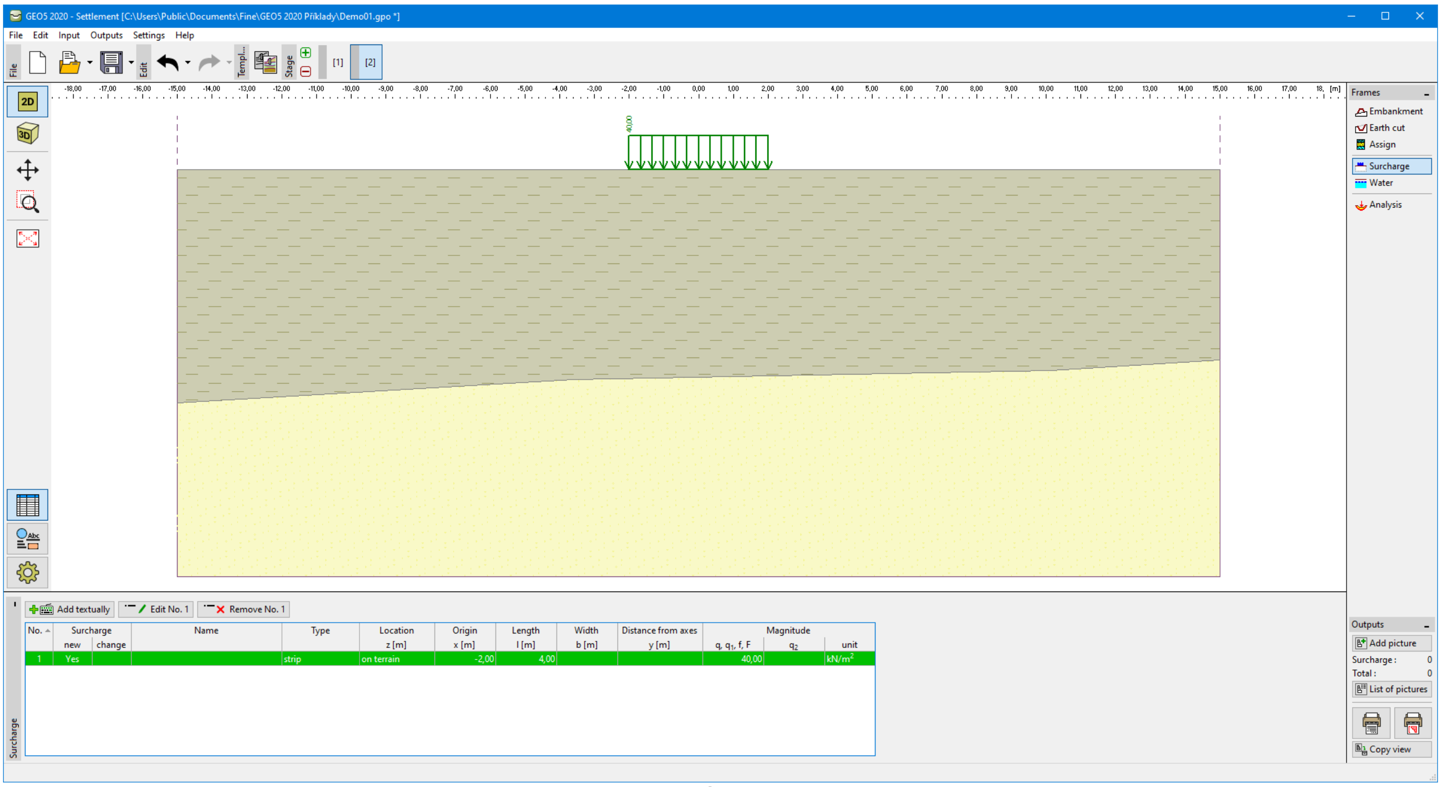
Frame "Assign"

Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active objects.

All input parameters of a surcharge can be modified in the construction stage where the surcharge was specified. Only the surcharge magnitude can be modified in all subsequent construction stages (option "**Adjust surcharge**").

Influence of surcharge on the change of stress in the soil body is described in the theoretical part of the help section.



Frame "Surcharge"

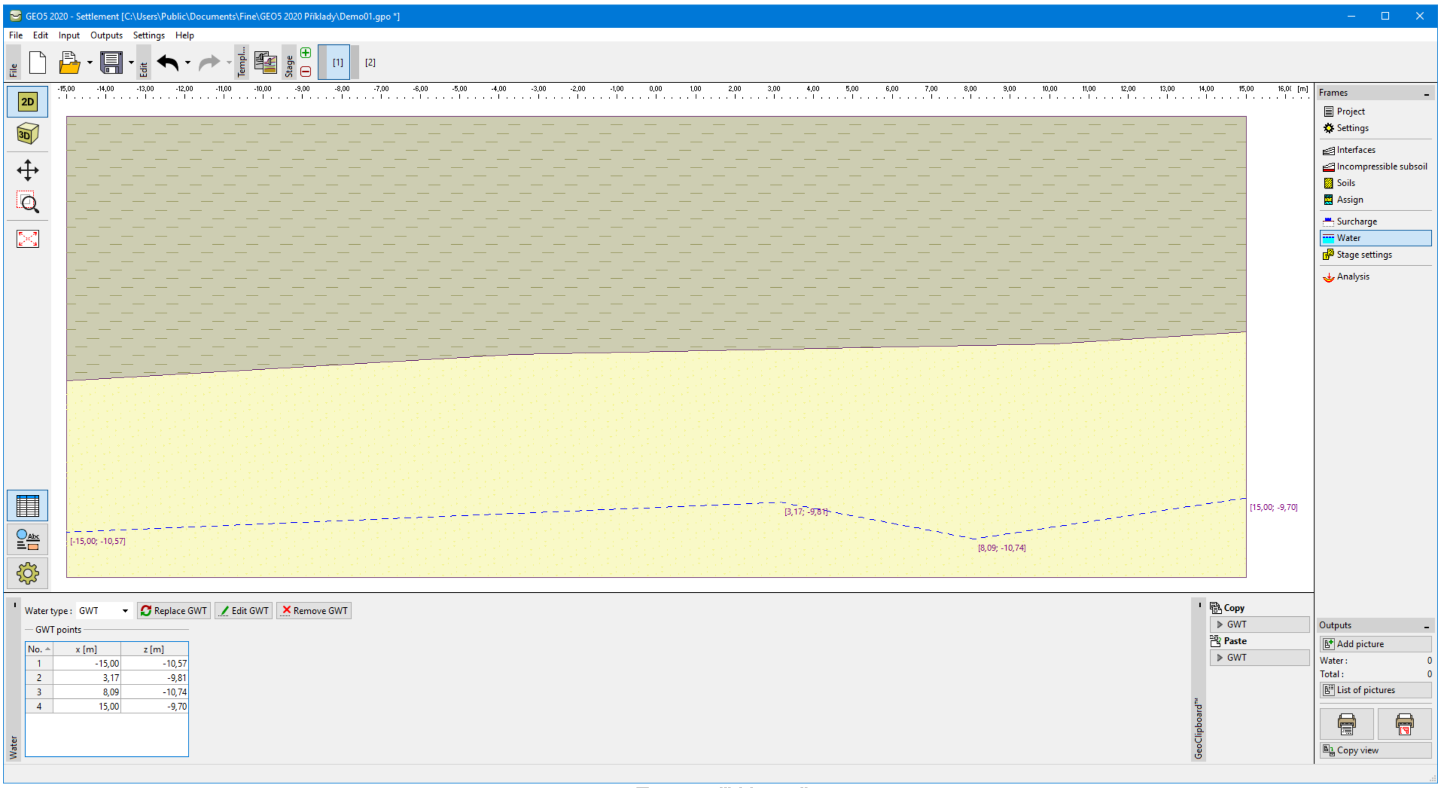
Water

The "**Water**" frame serves to set the type of groundwater table.

Inputting the ground water table or isolines, respectively, is identical with the standard input of interfaces.

If the input data in individual stages are different, the program then allows for accepting the data from the previous stage of construction by pressing the "**Accept**" button.

Input interfaces of water can be copied within all 2D GEO5 programs using "GeoClipboard".



Frame "Water"

Stage Settings

The frame "**Stage settings**" serves to input settings valid for a given construction stage.

The frame allows for specifying the position of control holes and thicknesses and locations of layers where the stress values are calculated.

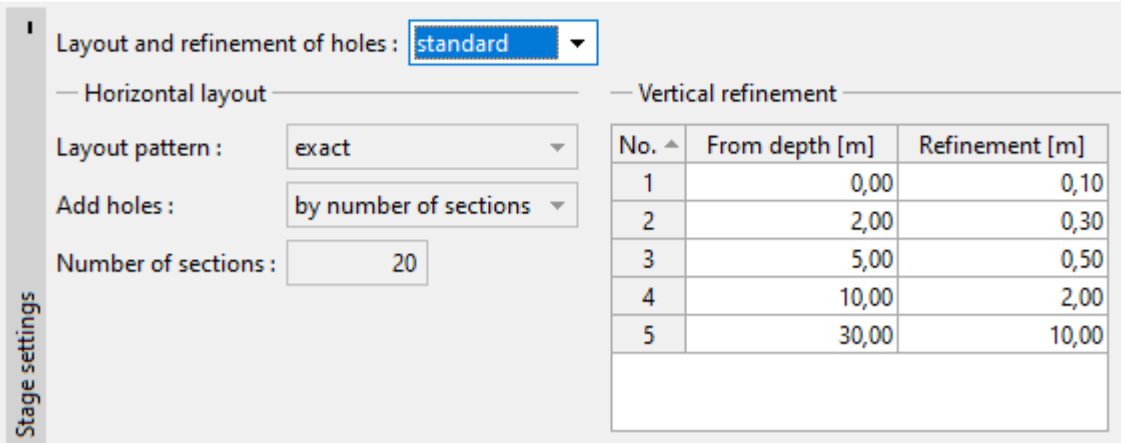
The program determines **stresses at individual control holes.** The terrain is always subdivided into twenty holes with even spacing. Additional holes are automatically generated in points specifying terrain, embankment, GWT, soil layer interfaces and end points of surcharge. The control (calculation) holes can be plotted in the frame "Analysis".

Individual holes are **divided into layers** according to the input values. The first layer always coincides with the original ground. In addition, **all points** specifying interfaces, GWT and incompressible subsoil are included. The default setting of thicknesses of layers **ensures reasonable speed and accuracy** of the analysis.

The layers are introduced up to depth of 250 m . In actual analyses, however, the depth of influence zone is restricted either by the input incompressible subsoil or by the reduction of magnitude of stress change or by the structural strength, respectively (depending on the setting in the "Settlement" tab).

The number and location of calculation holes can be adjusted when selecting the option "**User-defined**". In such a case it is possible to select both the location of holes and thicknesses and location of layers. The holes are then created according to the input - in addition, the program automatically includes all important points. When selecting the **option exact distribution**, the holes are included into all terrain points, soil layer interfaces, embankments, GWP and into end points of surcharge. When selecting the **option minimal distribution**, the holes are not included into points of interfaces of soil and embankment layers.

For standard analyses we **recommend keeping** the default setting of the analysis.



Frame "Stage settings"

Analysis

The "**Analysis**" frame displays the analysis results.

It is always required to model the structure using construction stages. **First construction stage** represents the **original state**, so the settlement is null. New surcharges or embankments are added in **further construction stages**, where the **terrain settlement** is calculated. Information about the analysis process, maximum settlement, depth of the deformation zone are listed in the window in the bottom part of the frame.

In consolidation analysis (set in the frame "Settings") this section of the frame allows us to enter consolidation parameters.

The settlement is calculated using the analysis theory, which is input in the tab "Settlement". The **depth of deformation zone** is defined either by input incompressible subsoil, method of restriction of the primary stress magnitude, or by theory of structural strength.

The results, as the main output, are displayed on the screen. To view the results, use the **horizontal bar** in the upper section of the screen and "Drawing Settings : Analysis" frame.

The frame "Drawing Settings : Analysis" allows for specifying:

- **drawing parameters**: parameters to display depression line and influence zone, to draw tilted sections, isosurfaces and isolines, etc.

- option to store individual views

The bar contains the following control items:

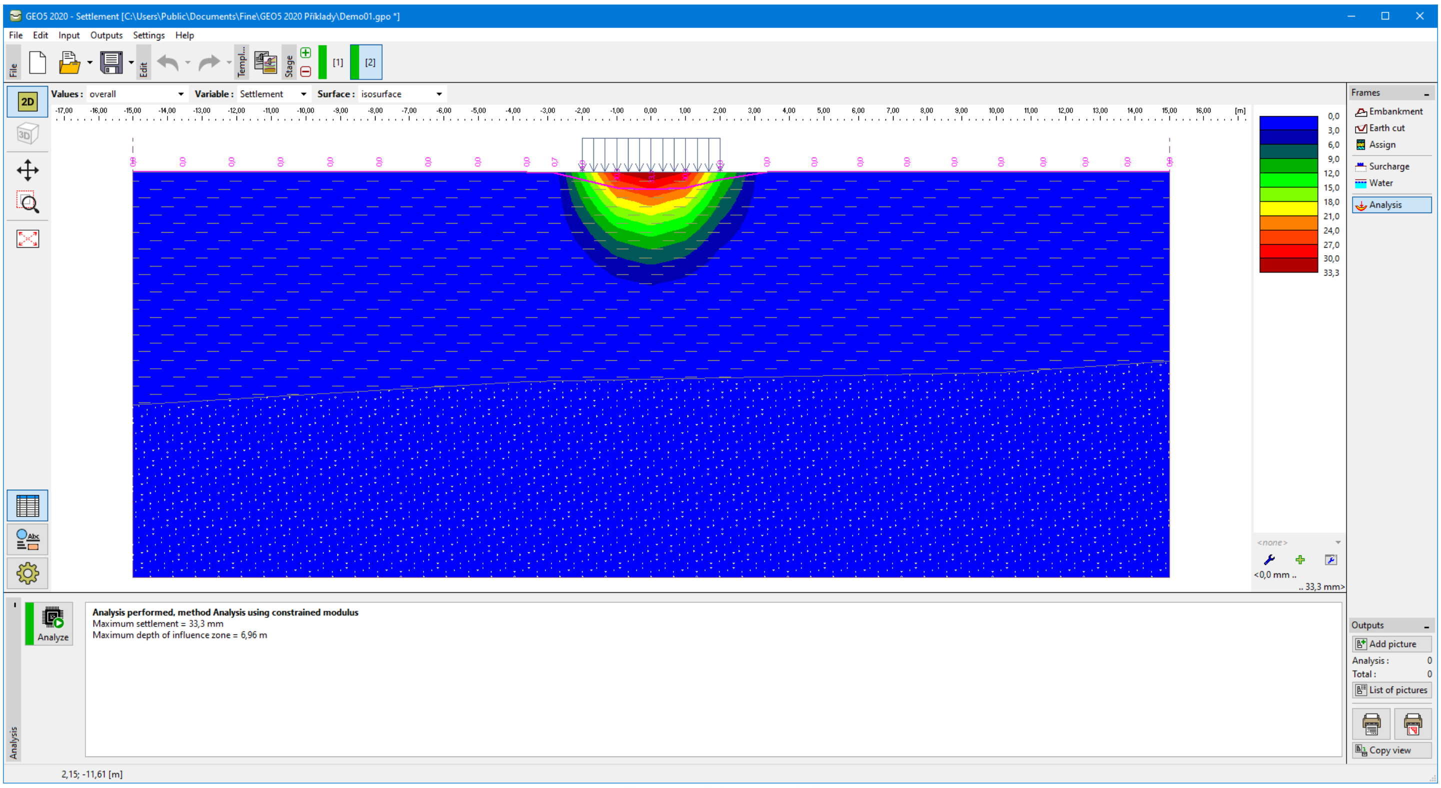
- **selecting values for visualization** - either **total** values, or their change during the **last calculation stage** or their change **in comparison with previous stages** can be plotted. The setting is available only in problems where it makes sense. It is therefore possible to display the change of stress, settlement or deformation in comparison with previous stages - however, always the current depth of influence zone is plotted.

- **selecting variables**

- SigmaZ,tot - overall vertical total stress [kPa , ksf]
- SigmaZ,eff - overall vertical effective stress [kPa , ksf]
- Pore pressure - stress due to water [kPa , ksf]
- Settlement - settlement of a point [mm , $feet$]
- Deformation - relative settlement of a layer [-]*1000*

- **plotting option** (do not plot, isosurfaces, isolines)

The **color range** is visible on the right part of the desktop. The buttons for setting the color range are located below.



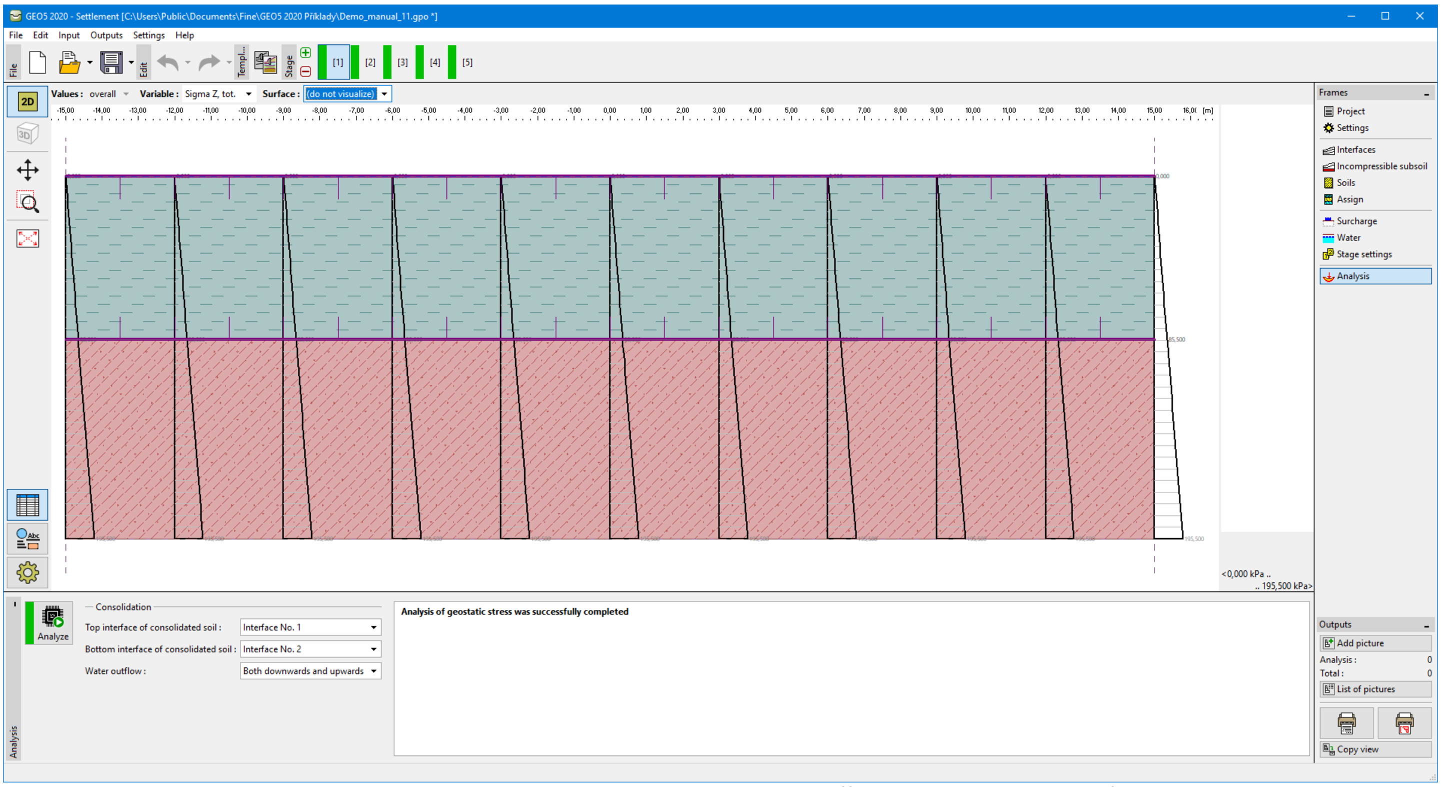
Frame "Analysis"

Consolidation Parameters

In consolidation analysis (set in the frame "Settings") the bottom window in the frame "Analysis" allows us to enter consolidation parameters.

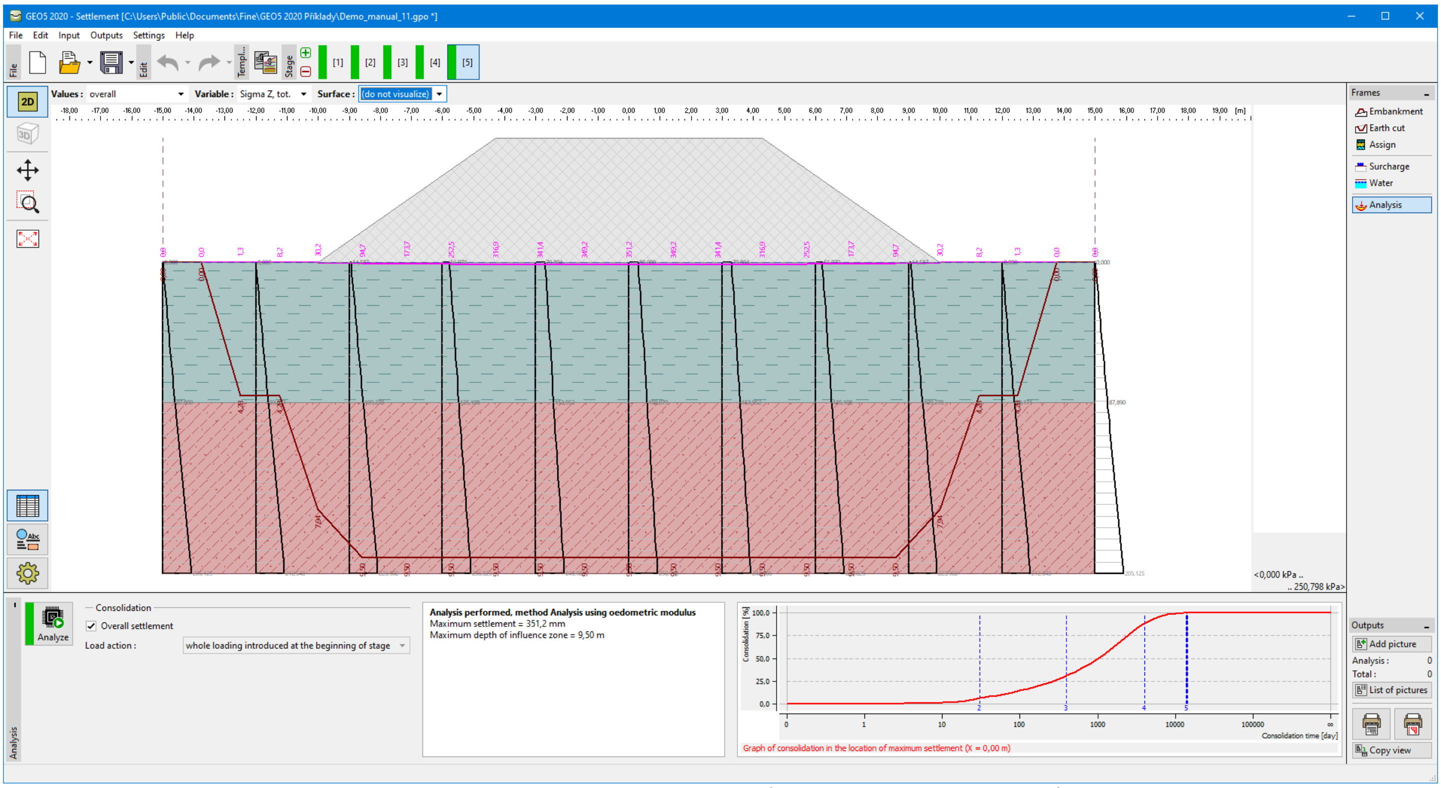
The first construction stage of calculation represents only geostatic stress at initial time of construction. Top and bottom interface of consolidated soil layer and direction of water outflow from this layer (upwards, downwards and both downwards and upwards) is entered in the first construction stage.

The program allows us to draw a time course of settlement (graph on the right of the desktop) according to relevant theory of settlement. The vertical axis shows degree of consolidation U [$\%$], on the horizontal axis is shown time of settlement t [$days$].



Frame "Analysis" - primary geostatic stress (first construction stage)

In other construction stages **time of stage duration** and load action are entered. Program allows for choosing from two options of load acting: entire load introduced at the beginning of stage or load linearly increases during stage duration. The calculation is then launched from the first construction stage to the construction stage where "**Overall settlement**" is checked (may be checked in whatever stage except first one).



Frame "Analysis" - consolidation (other construction stage)

Example: determine settlement from surcharge after 5 days, 1 month, 1 year and 5 years? Enter construction stages according to following scheme:

1. stage	Only geostatic stress
2. stage	Surcharge, time: 5 days
3. stage	No changes, time: 25 days
4. stage	No changes, time: 335 days

5. stage No changes, time: 1460 days
6. stage Check option "**Overall settlement**" and run the calculation

Program Ground Loss

This program is used for the analysis and determination of the shape of subsidence trough above excavations and to evaluate the damage to buildings situated in the affected area.

The help in the program "Ground Loss" includes the following topics:

- The input of data into individual frames:

  Project, Settings, Buildings, Profile, Soils, Assign, Geometry, Measurements, Stage Settings, Analysis, Damage

- Standards and analysis methods

- Theory for analysis in the program "**Ground Loss**":

  Stress in Soil Body, Ground Loss

- Outputs

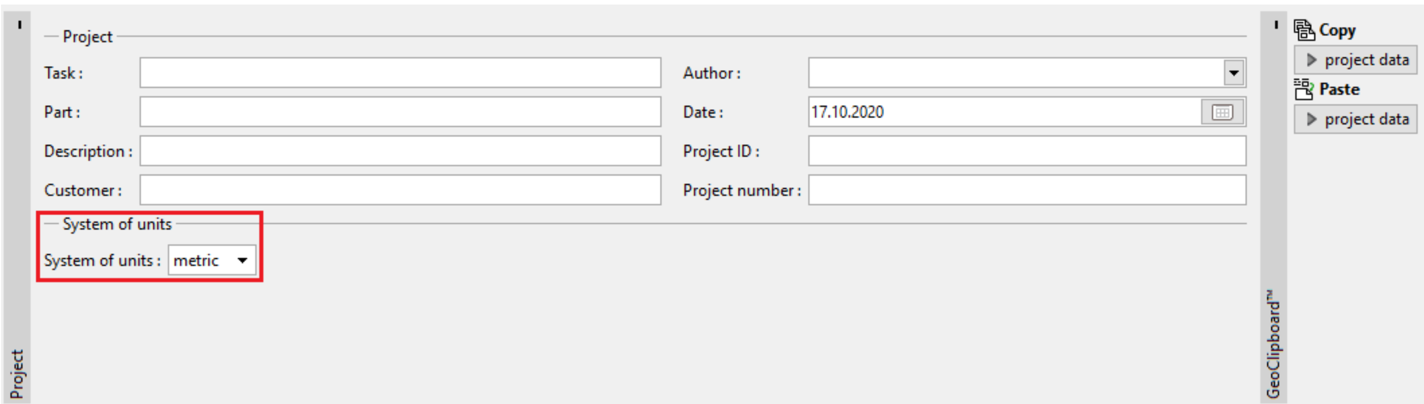
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The **"Project"** frame is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".

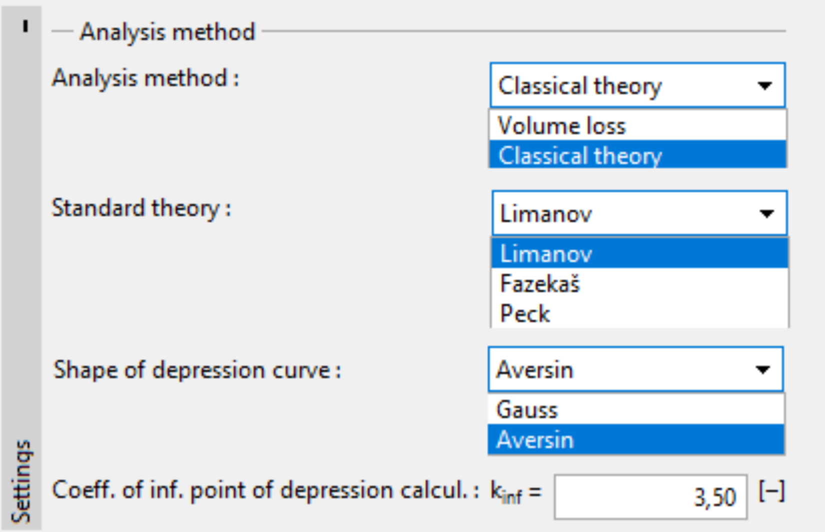


Frame "Project"

Settings

The frame "**Settings**" allows us to specify standards or methods that are used to perform the analysis.

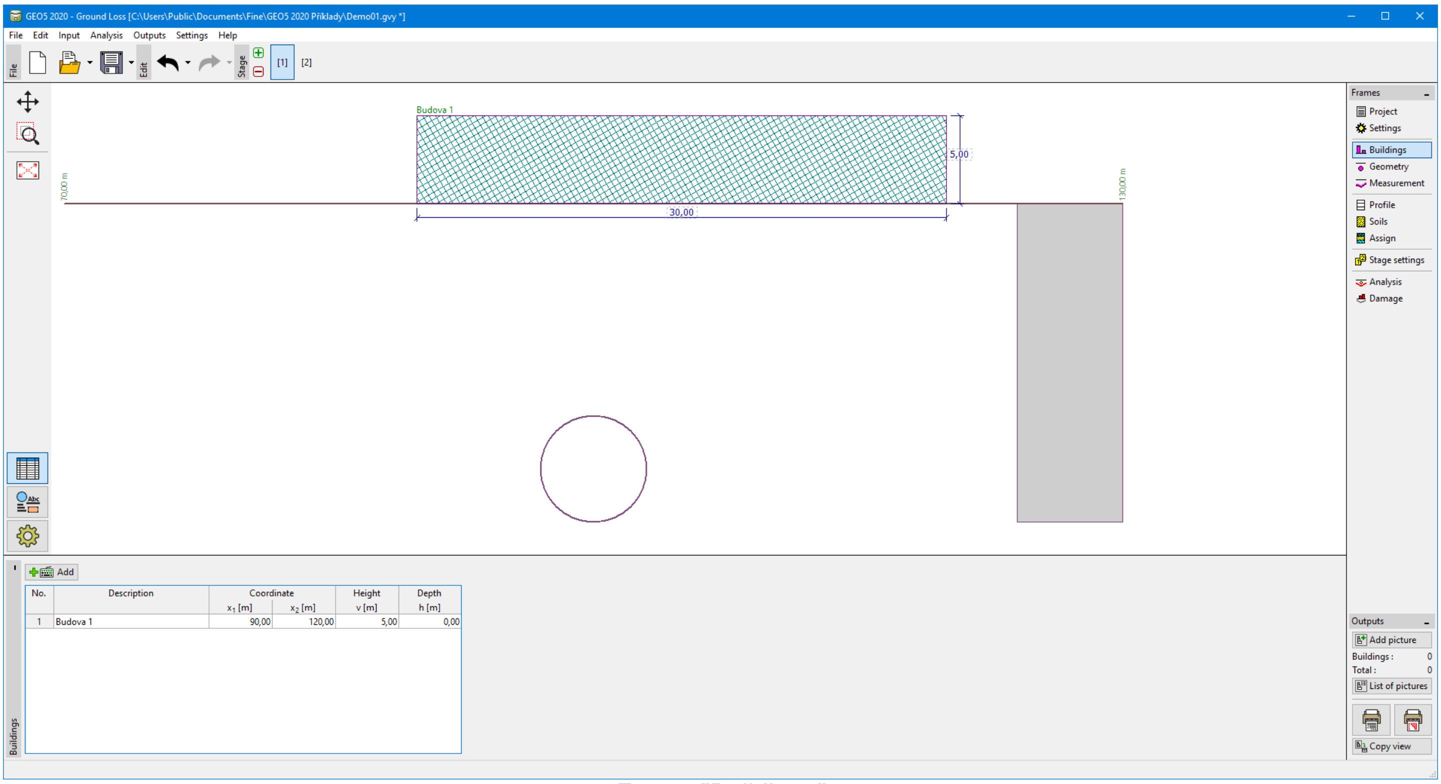
The frame "**Settings**" allows us to select the method for determining subsidence trough (Volume loss, classic theories) and its shape (Gauss, Aversin). It also allows us to input the coefficient of calculation of inflection point, (for classic theories only), which influences the shape of subsidence trough.



Frame "Analysis methods"

Buildings

The frame "**Buildings**" allows us to input objects above excavation. An arbitrary number of buildings can be specified both on a ground surface and at a given depth.



Frame "Buildings"

Profile

The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces, it is possible to edit the thicknesses of individual layers using active dimensions.

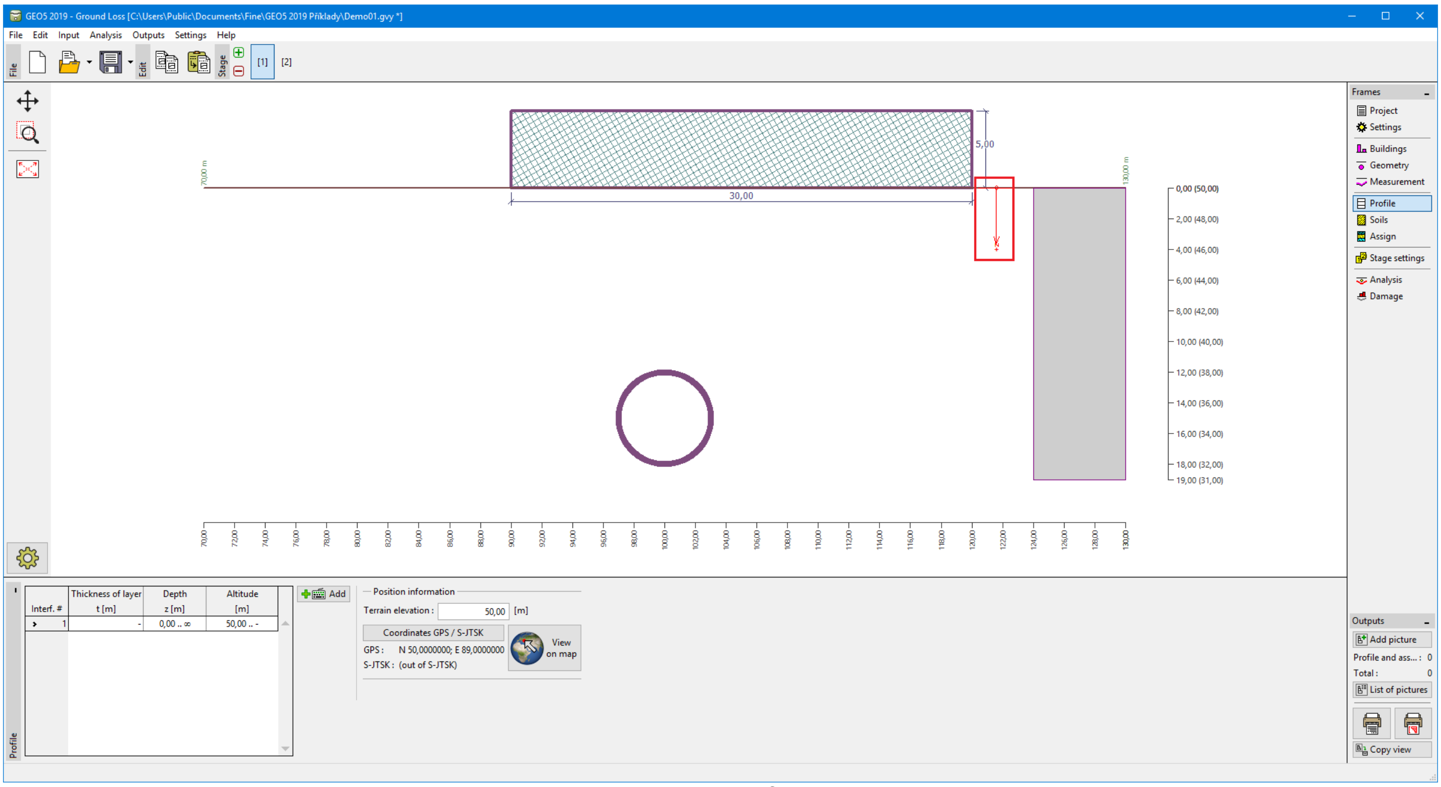
Adding a layer is performed in the "**New interface**" dialog window. The layer can be defined by z-coordinate (measured from the top of the structure) or by its thickness.

In this frame, it is also possible to enter optional information about the structure location:

- **terrain elevation** - if the terrain elevation is input, an altitude column for the layers is displayed in the table
- **GPS coordinates** - if the coordinates are input, it is possible to show a location of the structure on the Google Maps using the "**View on map**" button.

The program allows us to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window and, this way, to move the entire interface while keeping the same thicknesses of individual layers.

Data input in the frame is allowed if the **classic theory of analysis** is selected in the frame "Settings".



Frame "Profile"

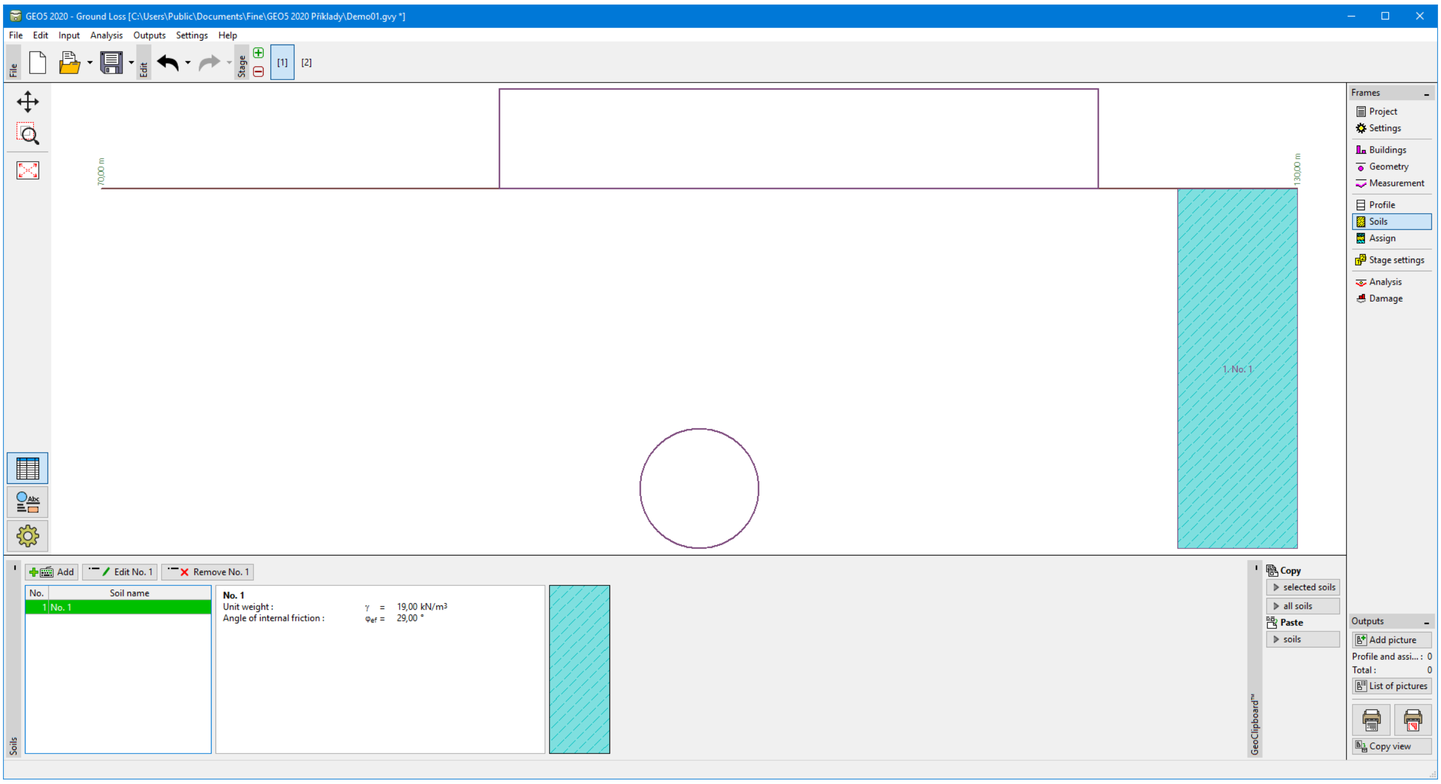
Soils

The frame **"Soils"** contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

Inputting data into the frame is allowed if the **classic theory of analysis** is selected in the frame "Settings". The particular values are obtained from a geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit the built-in database of soils, which contains values of selected characteristics of soils.

Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



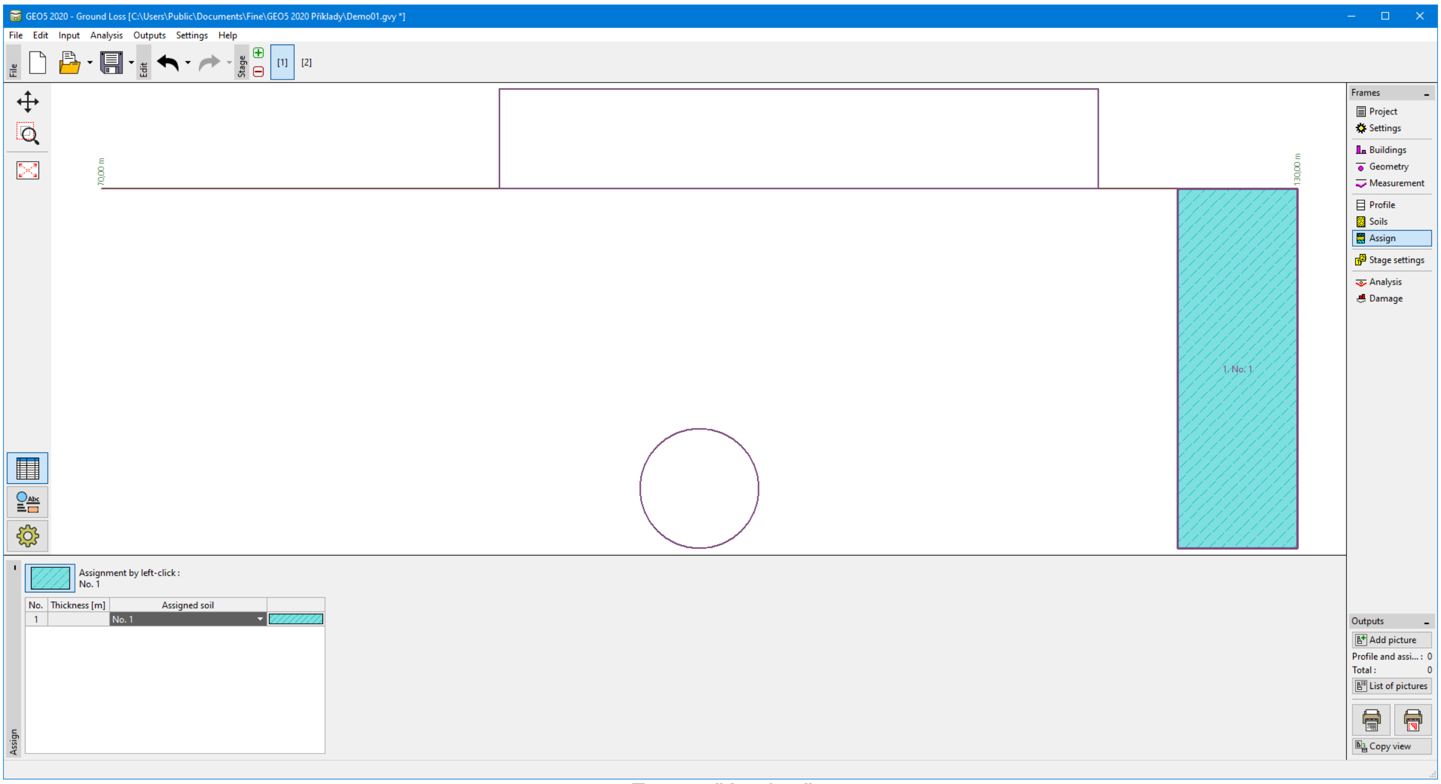
Frame "Soils"

Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail herein.

Inputting data in the frame is allowed if the **classic theory of analysis** is selected in the frame "Settings".



Frame "Assign"

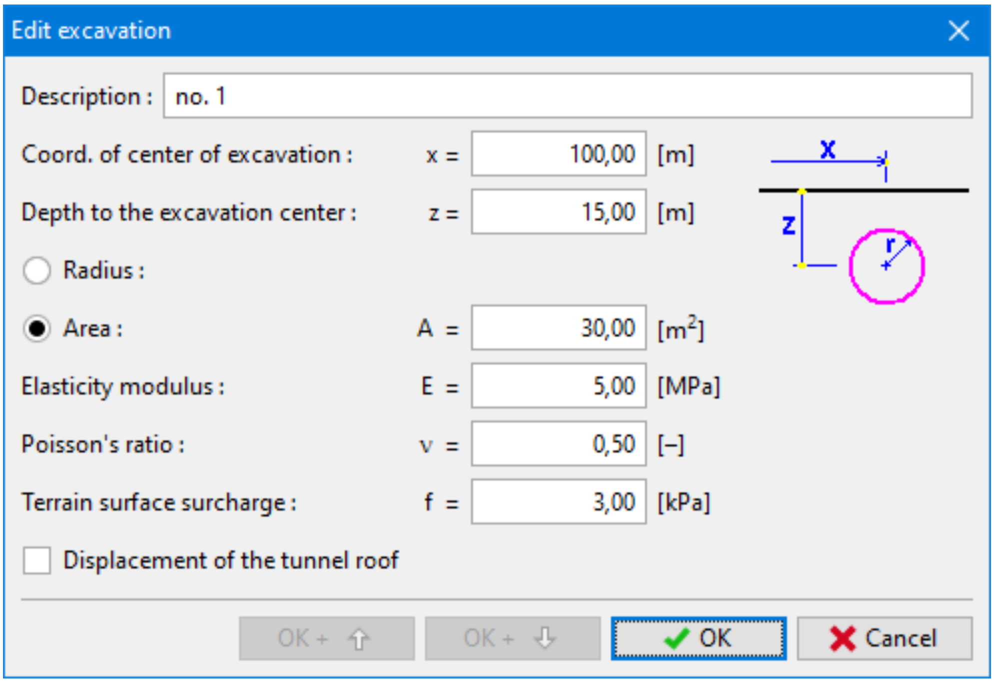
Geometry

The frame "**Geometry**" contains a table with a list of input excavations. The "**New excavation**" dialog window allows us to add excavations. The input excavations can also be modified on the desktop using the active objects.

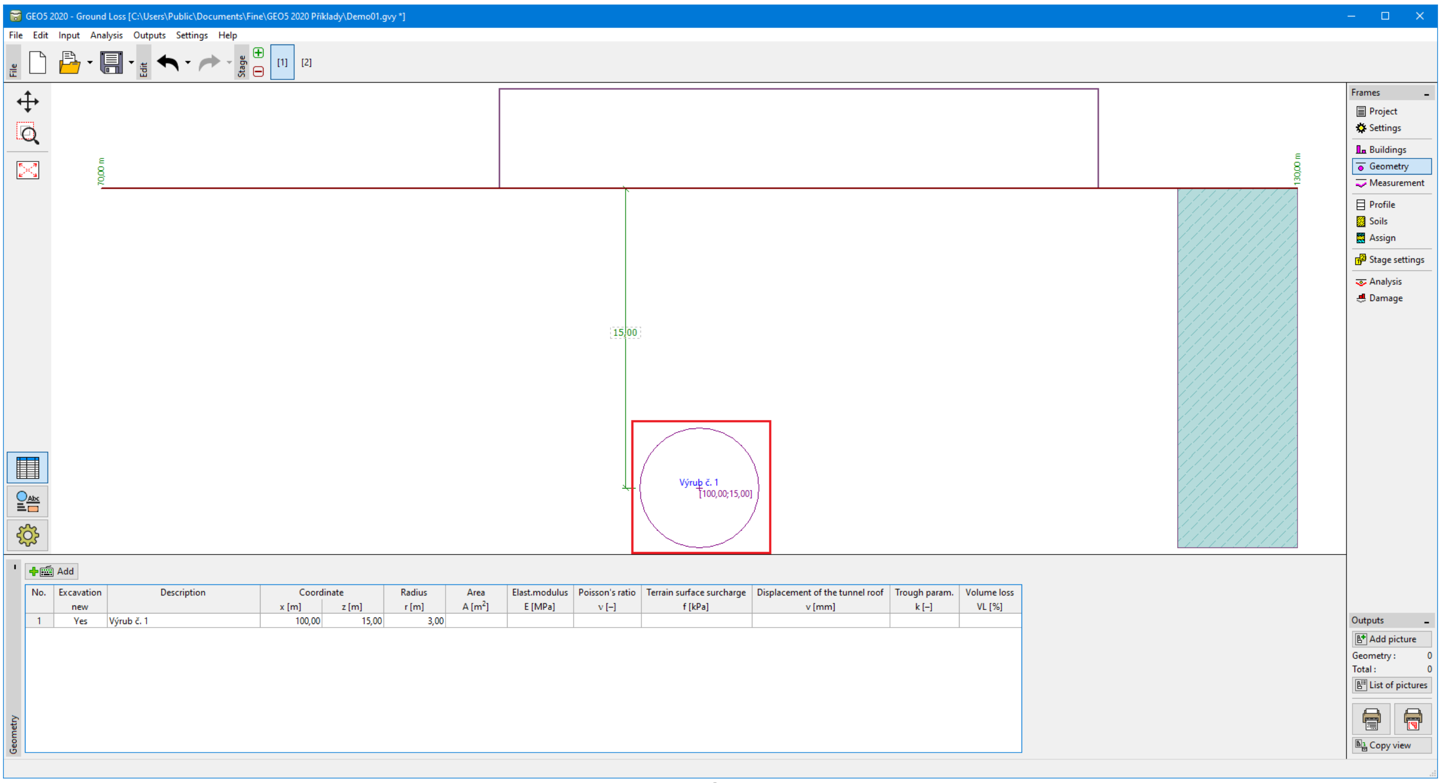
Parameters of excavation differ depending on the analysis method selected in the frame "Settings". Each excavation can be specified either by the radius or the area of excavation. Providing a sequential excavation is being input it is useful to specify the excavation area and place a fictitious center of excavation to a center of gravity of this area.

Additional input parameters are explained in more detail when describing individual analysis methods (Volume loss, classic theories).

The program allows us to export the geometry of a structure in the *.DXF format.



Dialog window "New excavation"

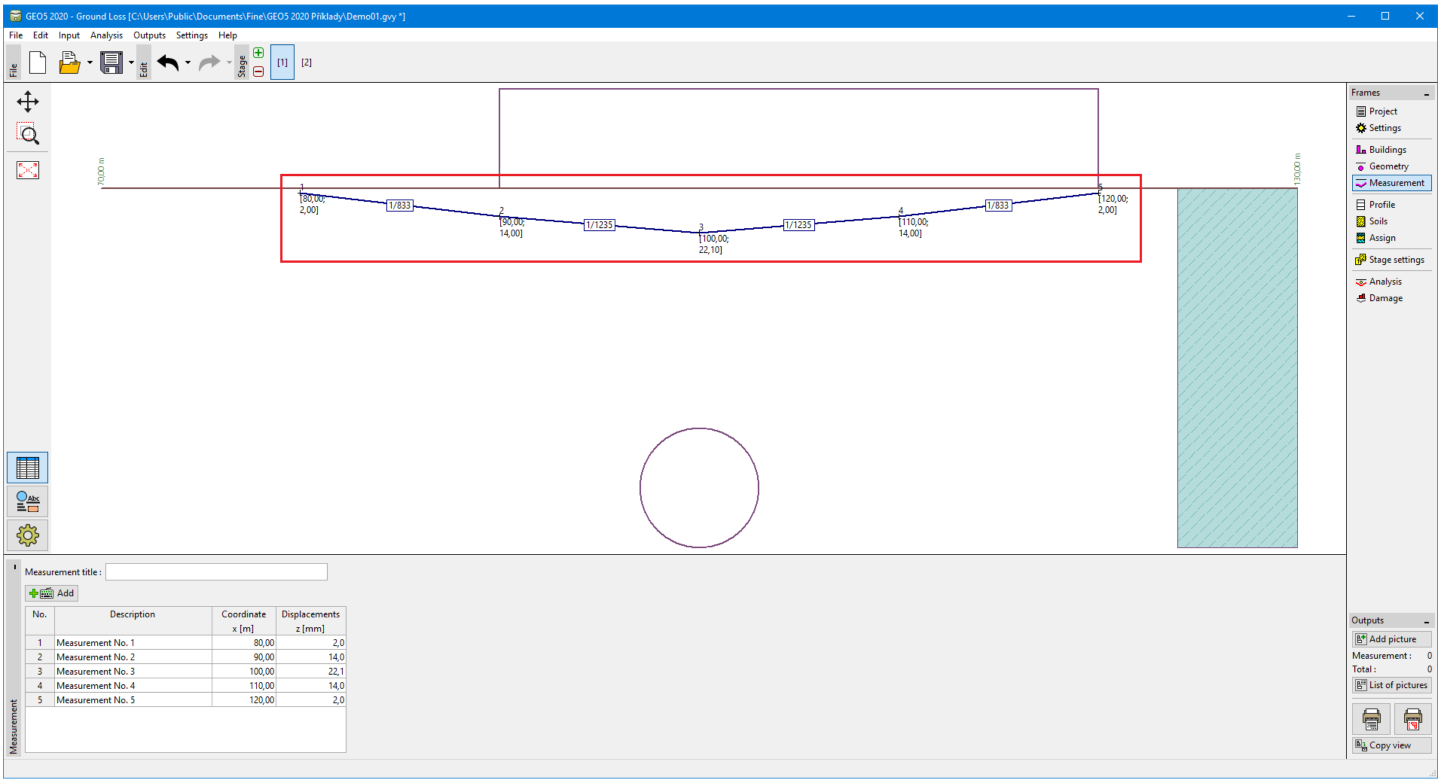


Frame "Geometry"

Measurement

The frame "**Measurement**" contains a table with a list of input measurements. The "**New measurement**" dialog window allows us to add measurements. The input measurements can also be modified on the desktop using the active objects.

Input measurements **do not influence the actual analysis** - their introduction into the program has resulted purely from designers needs. After excavating the first part of a sequential tunnel, it is useful to input the values measured in the construction site into the program and subsequently to add the excavation input parameters such that the **calculated and measured values are the same**. Practical experience shows that the values of input parameters acquired from this procedure (e.g. coefficient of volume loss) are **also valid for subsequent stages**.



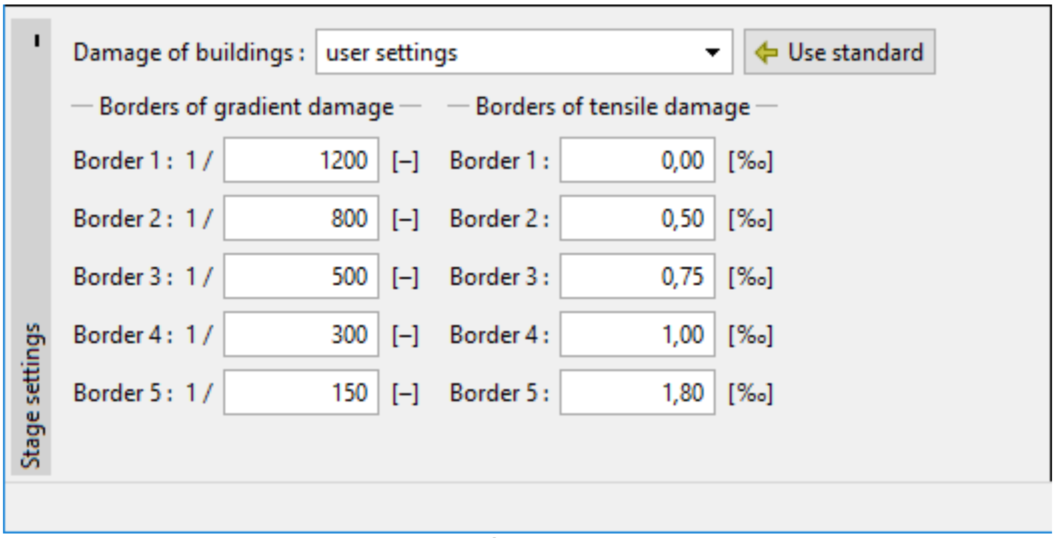
Frame "Measurement"

Stage Settings

The frame "**Stage settings**" allows us to input settings valid for a given construction stage.

The frame allows us to introduce bounds on the tensile and gradient damage. These values allow verifying the building damage in the frame "Damages". The program offers a default pre-setting (default setting for **masonry buildings**) and a user-defined setting - here, it is possible to define arbitrary criteria recommended by standards or gained from practical experience for arbitrary types of buildings.

The boundary values must be defined either in a descending or ascending order, respectively. Should you need to define fewer regions than specified in the program, it is possible to characterize certain boundaries by the same value.

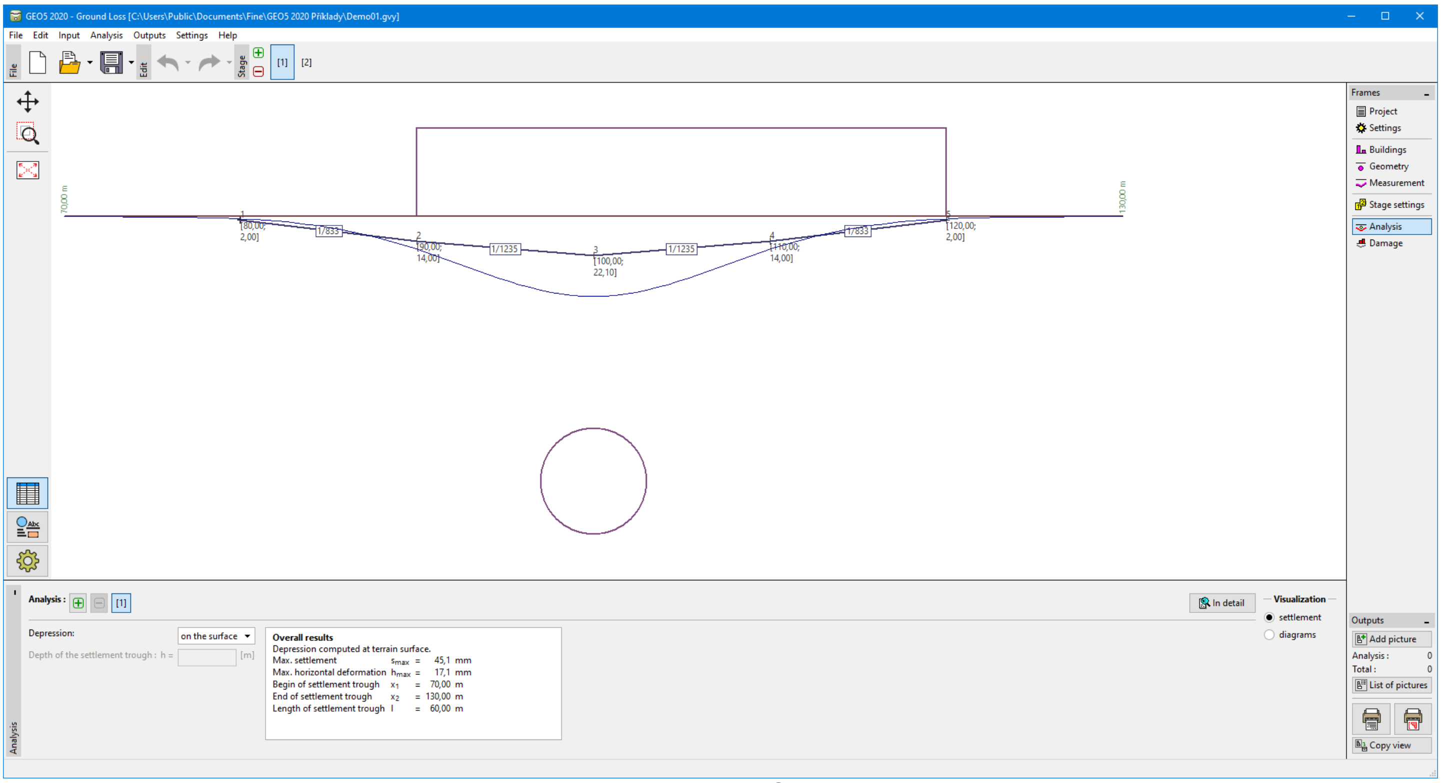


Frame "Stage settings"

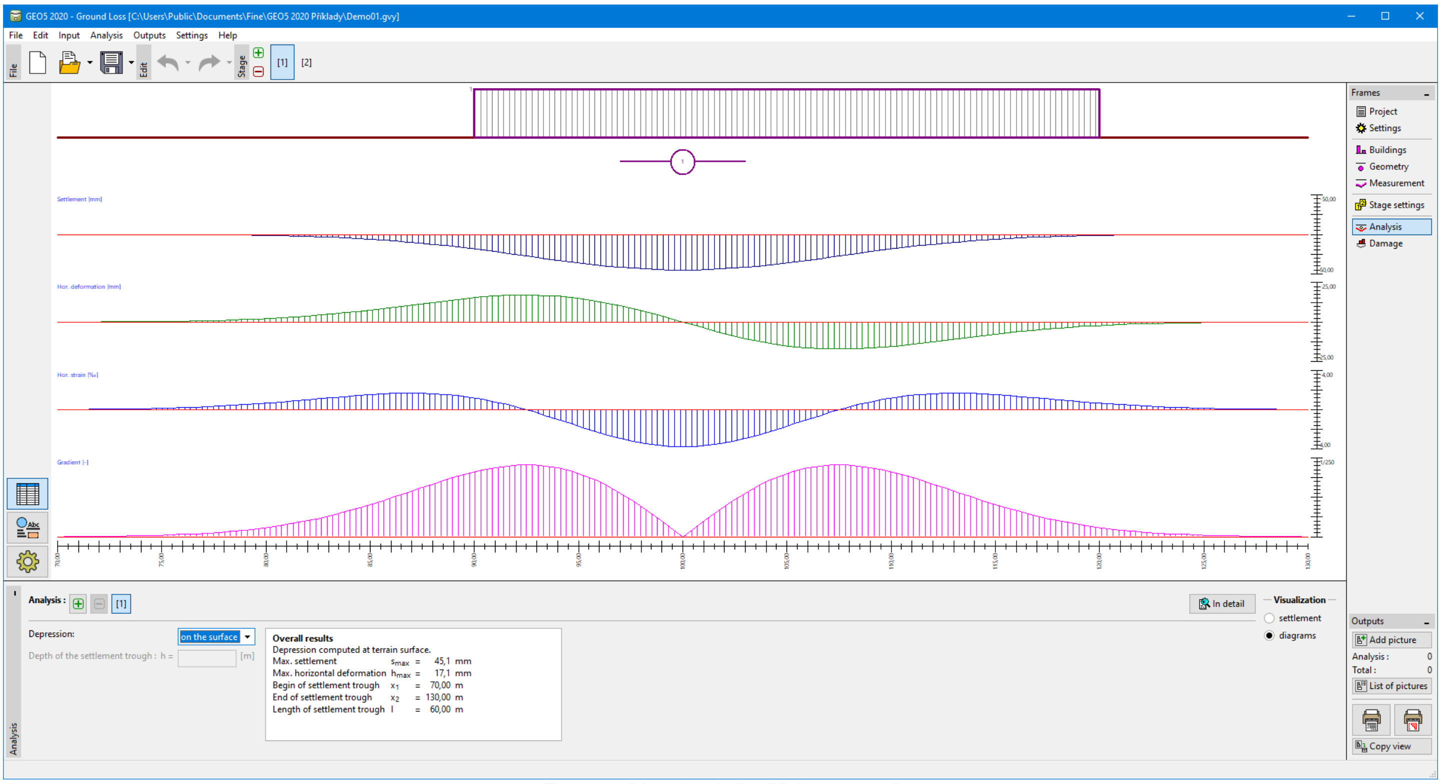
Analysis

The frame "**Analysis**" provides the results from the analysis of subsidence trough. More than one analysis at different depths below the terrain surface can be performed for a single task. The computed values are displayed on a desktop and are continuously updated whenever a change in the input data is done.

Visualization of results can be adjusted in the frame "Drawing Settings". For a quick switch between different styles of graphical presentation of results (**subsidence trough, distribution of values**), the user may use the buttons in section "**Visualization**".



Frame "Analysis" - "Settlement"



Frame "Analysis" - "Diagrams"

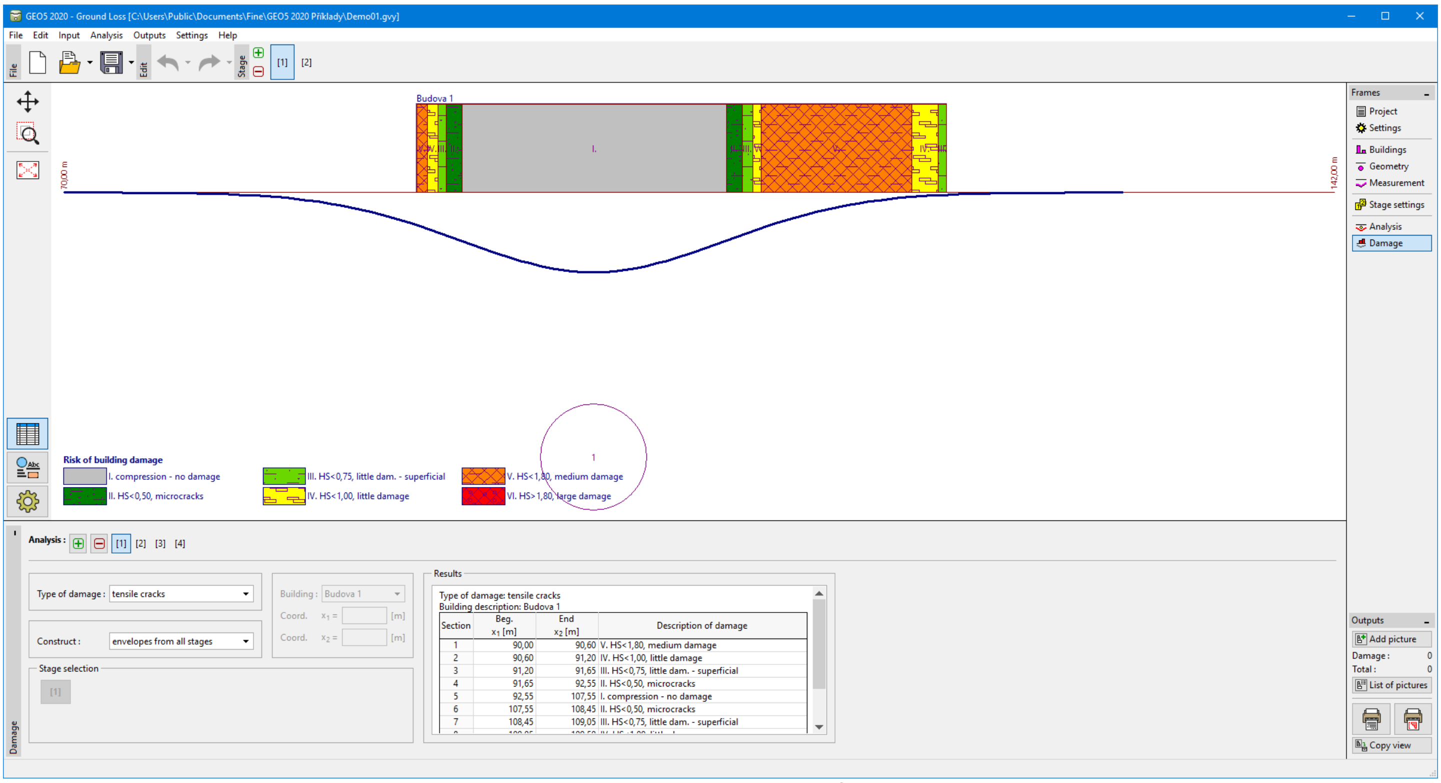
Damage

The frame **"Damages"** provides the results of the failure analysis of buildings. The program offers four types of verification:

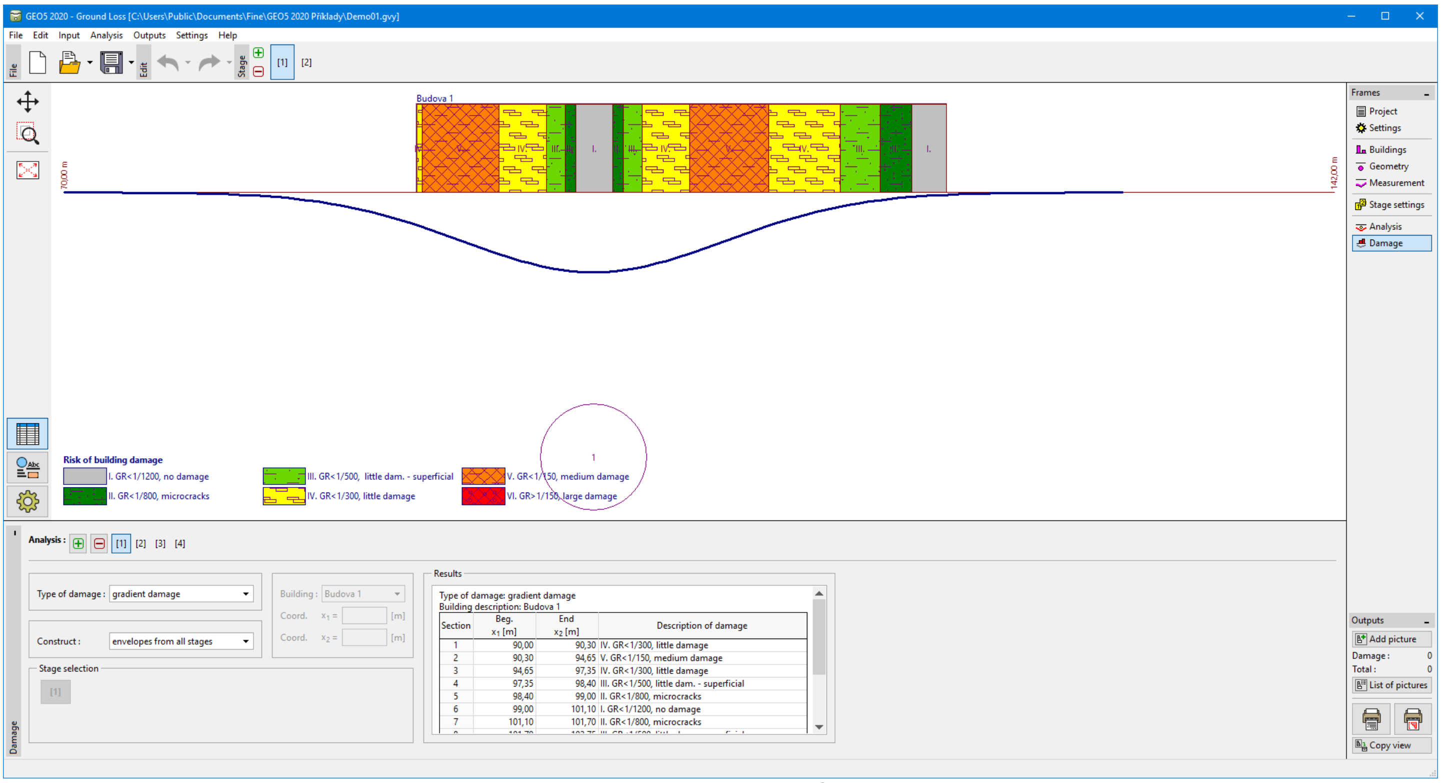
- Verification of tensile cracks
- Verification of gradient damage
- Verification of relative deflection of buildings (hogging, sagging)
- Verification of the input section of a building

The program allows us to perform an analysis for the current and all previous stages (**envelope from all stages**), or it is possible to input individual stages and evaluate their influence. Such a procedure makes it possible to find, e.g. an optimal process of excavation of sequential tunnels.

Several analyses can be carried out for a single task. Visualization of results can be adjusted in the frame "Drawing Settings".



Frame "Damages" - visualization of tensile cracks



Frame "Damages" - visualization of gradient damage

Program Stratigraphy

The Stratigraphy program (and Logs, Cross Sections and Earthworks modules) is used for:

- Creation of a digital terrain model of the construction site
- Import of all boreholes and field tests from the geological survey

- Creation of geological documentation
- Creation of geological sections
- Creation of a 3D subsoil model
- Data export to other GEO5 programs (Cross sections - "**Slope Stability**", "**Settlement**"; Output profiles - "**Spread Footing**", "**Piles**"...)

The **workflow** in the "**Stratigraphy**" program is described in **Engineering Manuals** at our website:

- EM 40 - "**Basic Work using the Stratigraphy Program**"

- EM 41 - "**Advanced Modeling in the Stratigraphy Program**"

- EM 42 - "**Creation of Field Test Documentation**"

- EM 43 - "**Interpretation of Field Tests into the Soil Profiles**"

- EM 44 - "**Creation of User-defined Template**"

- EM 46 - "**Earthworks module**"

- EM 47 - "**Export and Import Field Tests / Templates**"

The help in the "Stratigraphy" program includes the following topics:

- Data input into individual frames:

Project	Templates	Settings	Construction site	Source Data	Terrain points	Terrain Edges
Field Tests (Exploration Points)	Soils	Soil Profiles	Interfaces	Water	Geological sections	Geological model
Output Profiles	Output Sections	Output Solids				

- Earthworks - construction stages

- Outputs

- Theory for field test input in the "**Stratigraphy**" program:

  Field Testing

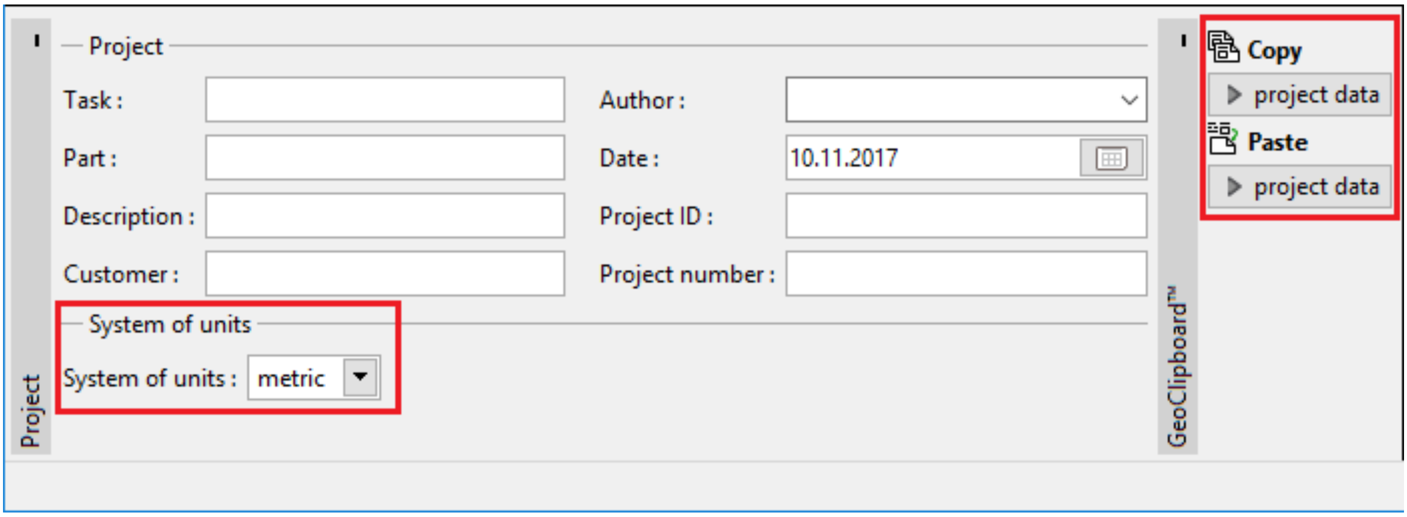
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Project

The **"Project"** frame is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs and during the creation of a geological documentation.

The frame also allows the user to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

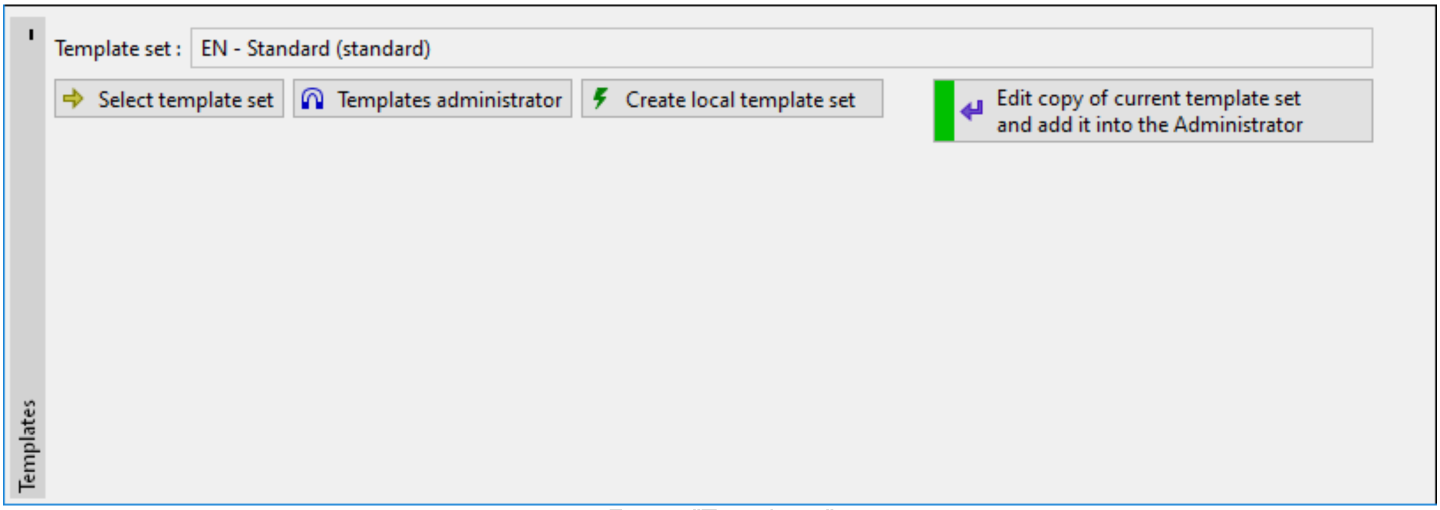
Templates

"**Templates**" frame serves to work with **geological documentation. This frame is available only for users with purchased** "**Logs**" **module**.

The template define geological documentation provided by the program. The template of each borehole contains:

- definition of input data
- definition of output protocols
- definition of mapping for export and import

The way of **working with template sets is almost the same as using settings** in the "Settings" frame in other GEO5 programs.

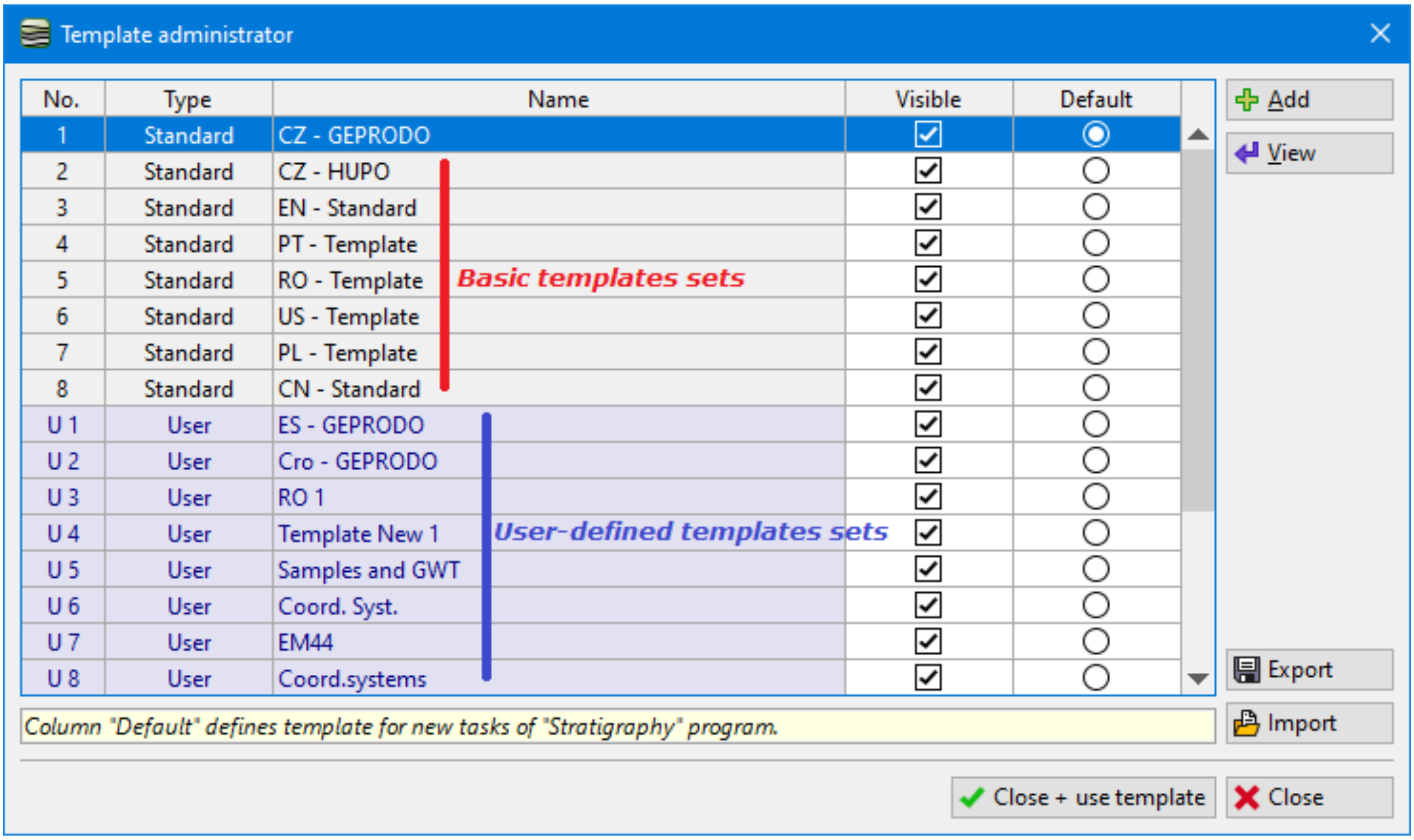


Frame "Templates"

The "**Select template set**" button allows us to choose an already created template from the "List of template sets".

The "**Templates administrator**" button opens the "Template administrator" dialog window, which allows for viewing and modifying individual template sets. It is also possible to identify the visible template sets in the "**List of template sets**". Data in the "**Templates administrator**" can also be **exported and imported**.

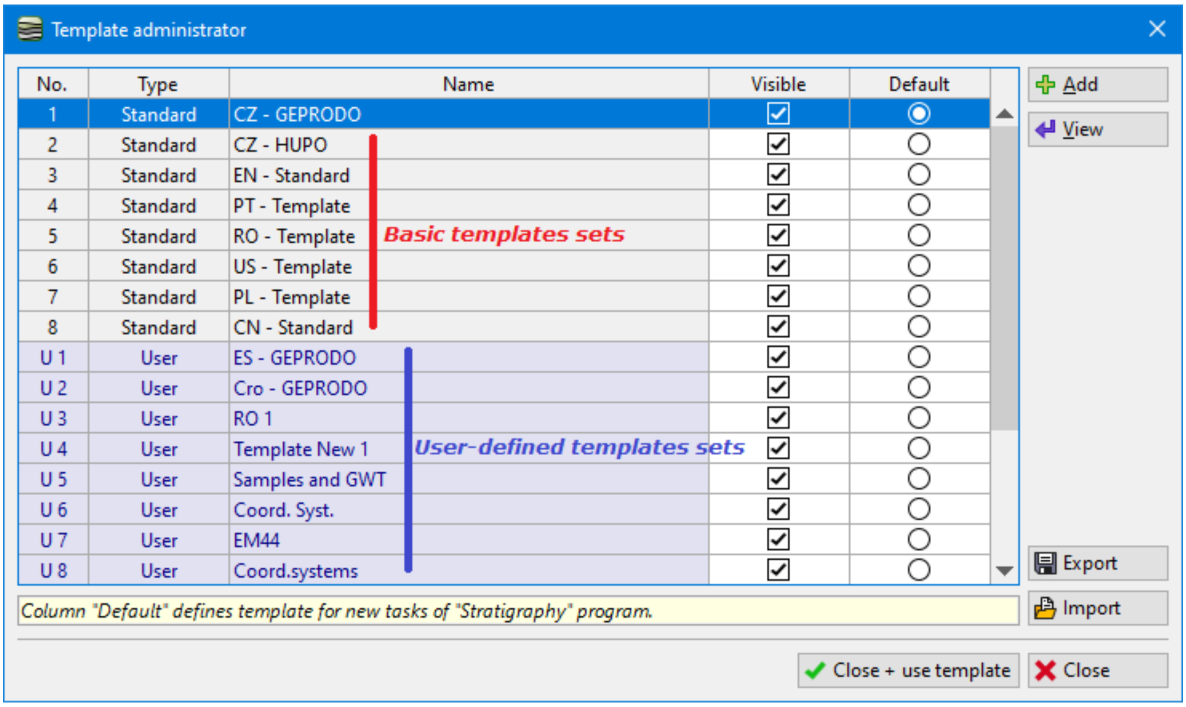
The programs not only contain the pre-defined **basic template sets**, but also allow the user to create a User-defined template sets. The "**Edit copy of current template set and add in into the administrator**" button opens a dialog window with a copy of the current template set, what can be modified. The created template set is saved into the "**Template administrator**".



The "**Create local template set**" button enables a quick visualization and editing of the current template set. Modifying any of the parameters changes the title to "**Local for current task**". The program is then working with this **local template set**. Should we consider this template set as suitable also for other tasks, we add the template set into the "**Template administrator**" by pressing the "**Add into the administrator**" button.

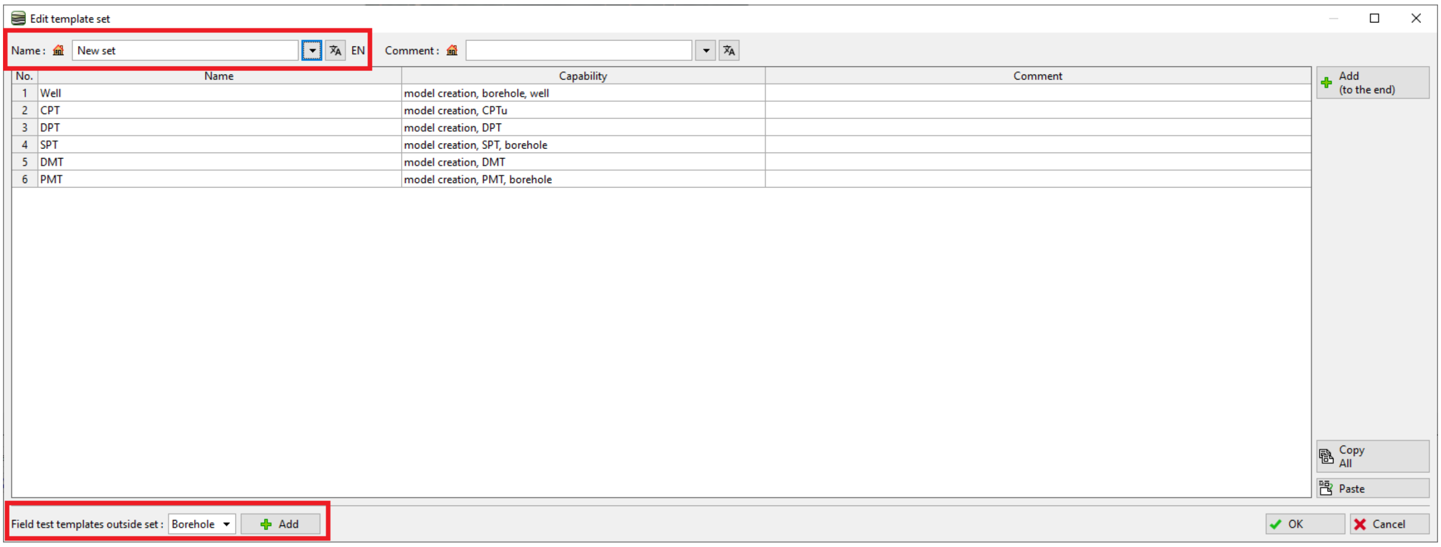
Creation of User-defined Template

The program contains the pre-defined **basic template sets** for the creation of a geological documentation for all field tests (borehole, CPT, SPT...). Because of the large number of requirements for geological protocols (logs), it is also possible to create **User-defined template sets**.



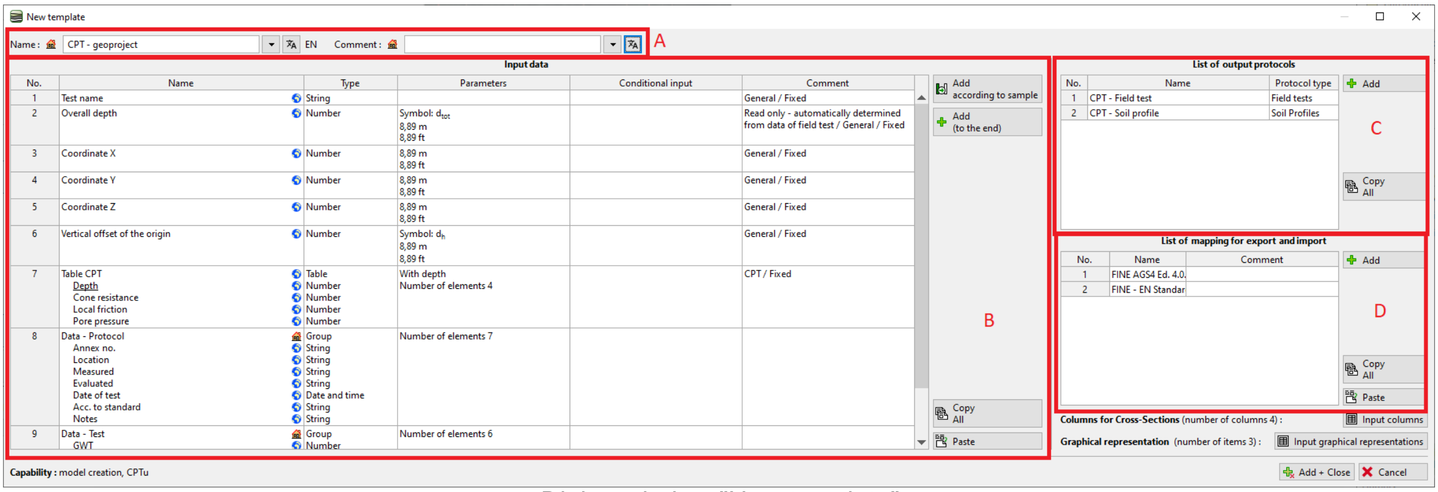
The best way to create a user-defined template is to start with a predefined template set and modify it according to the specific requirements. Using the ""**Edit copy of current template set and add in into the administrator**"" button, we define a user-defined template set in the "**Templates**" frame. All templates are displayed in the table and can be added/removed/edited.

If there is a field test in the task, which uses a template out of the current template set, this template can be added using the "Add" button in the bottom part of the window.



The dialog window for the creation/editof **a new template** contains 4 parts:

1. **Part A** - the name (and comment) of the new template is defined here (e.g. according to the country, name of company, standard...).
2. **Part B** - input data for the selected tab (field test) are defined here.
3. **Part C** - the form of output protocols is defined here for each type of field test. The editing is done in the spreadsheet, and it is similar to work e.g., in MS Excel.
4. **Part D -** mapping for export and import data



Dialog window "New template"

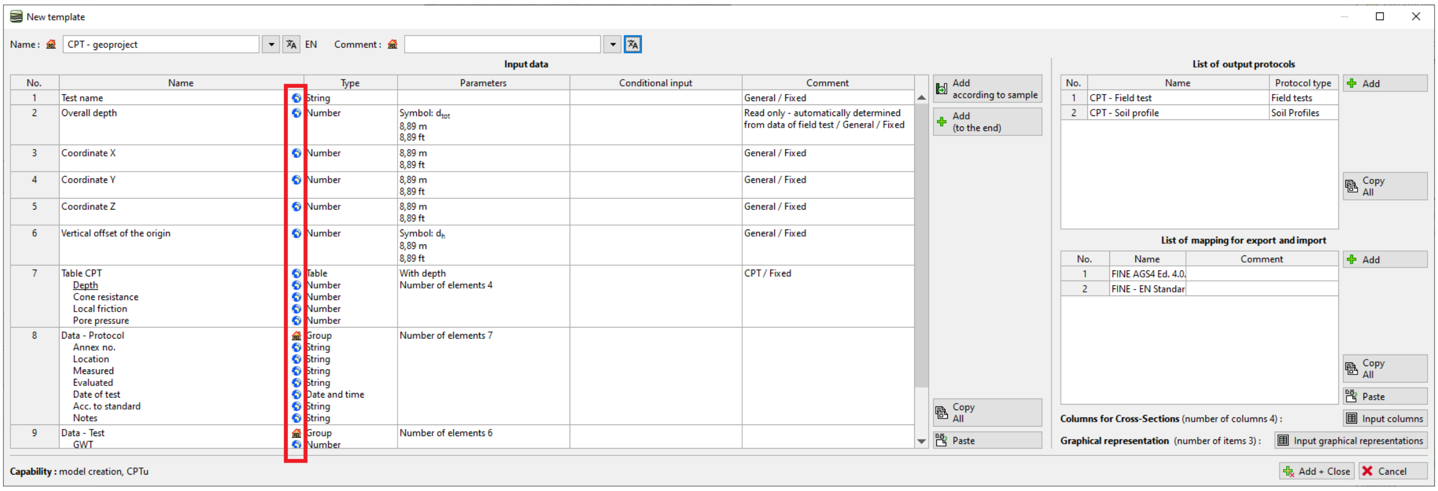
Templates are compatible with each other and preserve the specified data as much as possible. The content of any fields that are taken from the basic templates will remain unchanged when the individual templates are switched.

The creation of a user-defined Template is described in detail in the engineering manual No. 44 - Creation of User-defined Template.

Input Data

Data, which we want to enter for each field test and later print in output protocols are defined in the dialog window "**New Template**".

All data from the template is displayed on the left part of the dialog window. Data can be added, edited or removed.



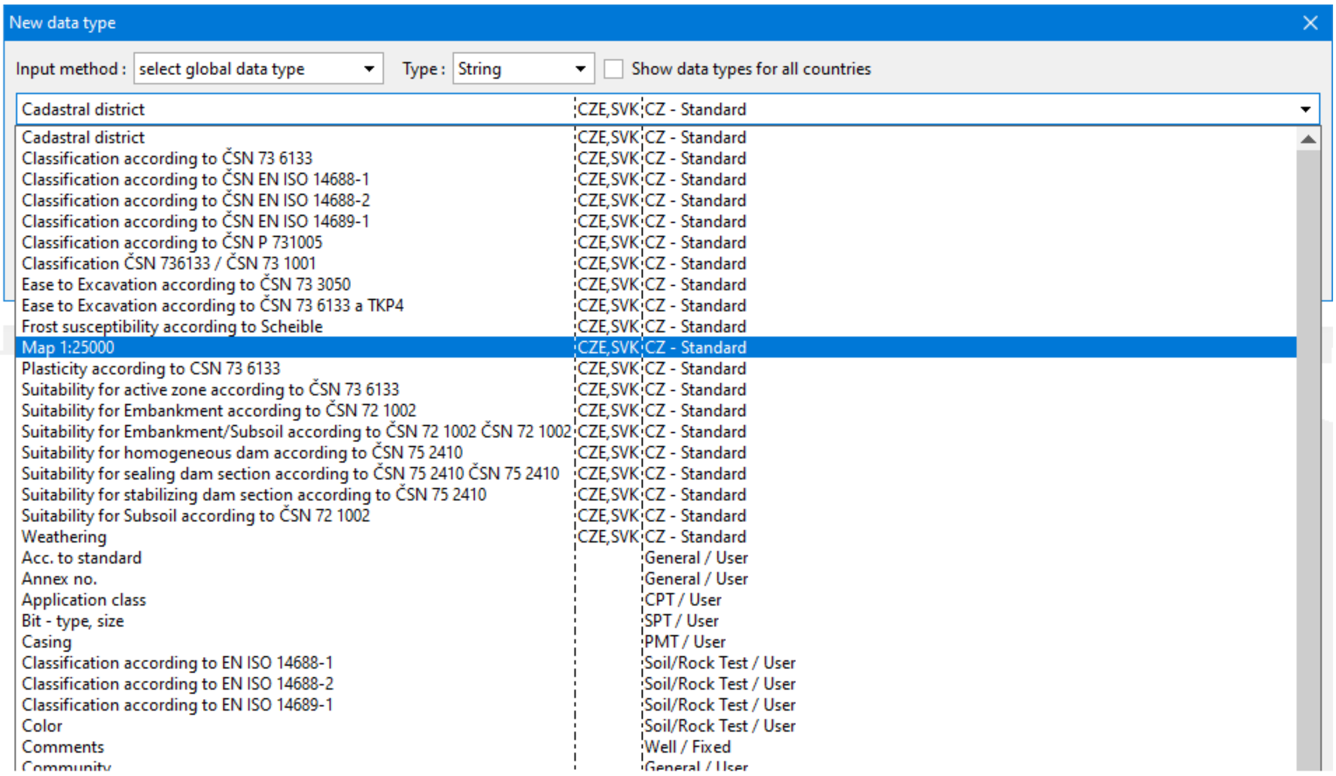
Data types in the template

Data types have assigned special symbols (house, globe). They are described in the engineering manual **No. 44 - "Creation of User-defined Template"** in detail.

We can add a new "**New data type**" (using the "**Add**" button) for the current template. First, it is necessary to select a source of data - we can use:

- **Global data type** - data types used in predefined templates as a part of "**Stratigraphy**" program
- **Existing user data type** - data types used in user-defined templates
- **New user data type** - it is possible to define a user-defined data type

It is always useful to use already defined data types. For example, if you define a new template and want to add a "**Drill machine**" data, it's always better to find data in the global library - in this case, the "**Drilling equipment**". We can edit the description when creating a template - but the relevant data (here the name of drill machine) contained in other templates or imported from different already defined formats will be **automatically loaded** into this data type.



Selection of data type

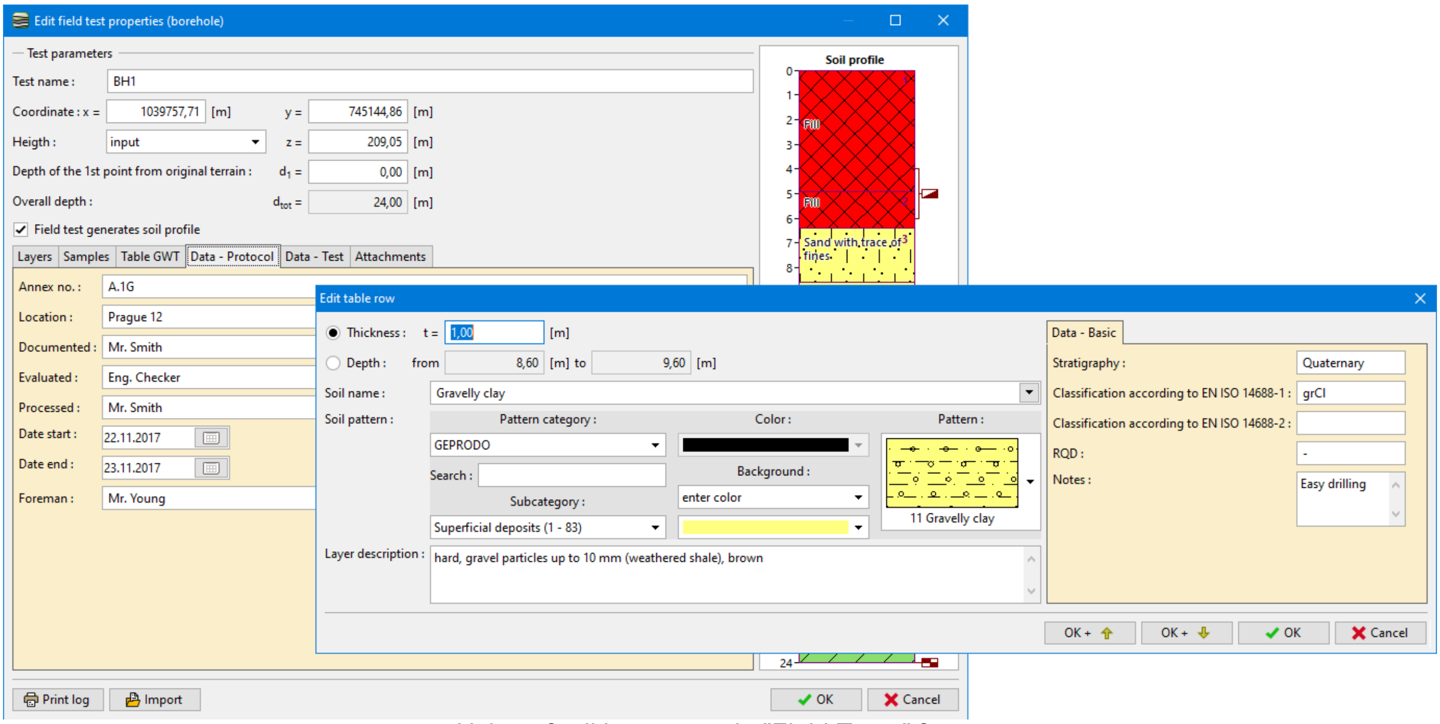
It is possible to define data of **different types**:

- **string**
- **date, time**
- **enumeration**

- **number** - it is possible to define units which are converted automatically when changing system of units
- **tables** - general or specific with compulsory depth, thickness of layer or depth interface (from-to) input
- **group** - creates a tab with data input in the group

These optional input data are entered while entering field tests in the "Field tests" frame. **Global (default) data types** can be edited (e.g. name, units or indications), but it is not possible to change a type (e.g. from text to table). If any type is missing, it is possible to add it from the predefined or user database. If the required type is not in the database, it is possible to create it. Newly created type is automatically saved into the user database.

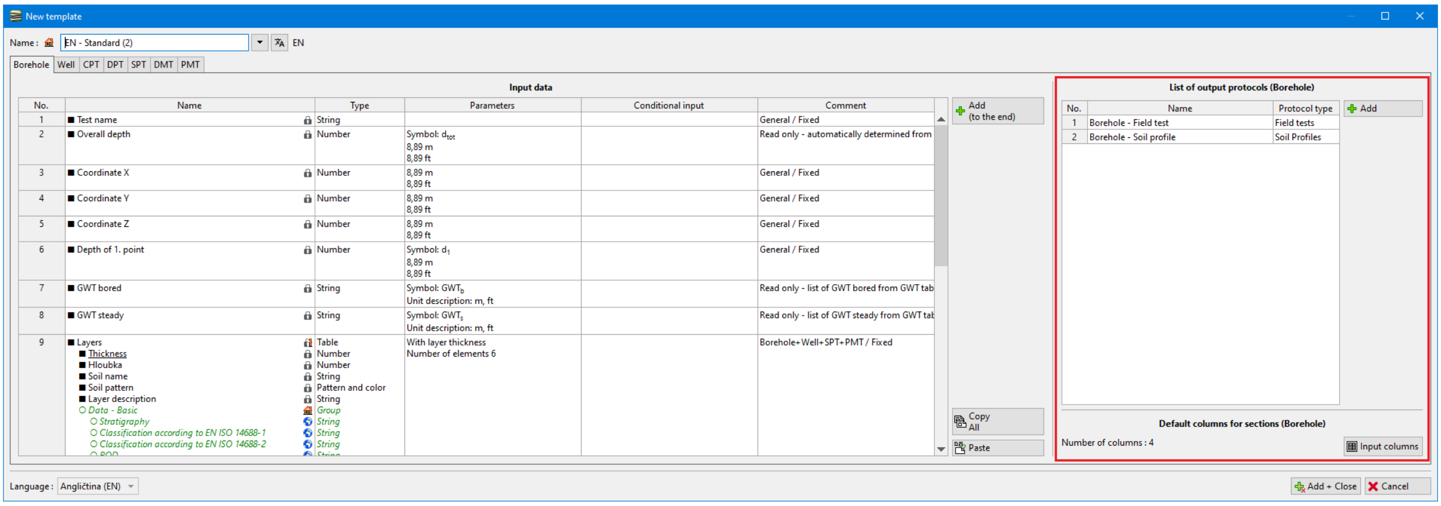
For better clarity, it is possible to sort data types into **groups** (by the creation of new data type "Group"), which determines tabs in field tests entering.



Using of editing groups in "Field Tests" frame

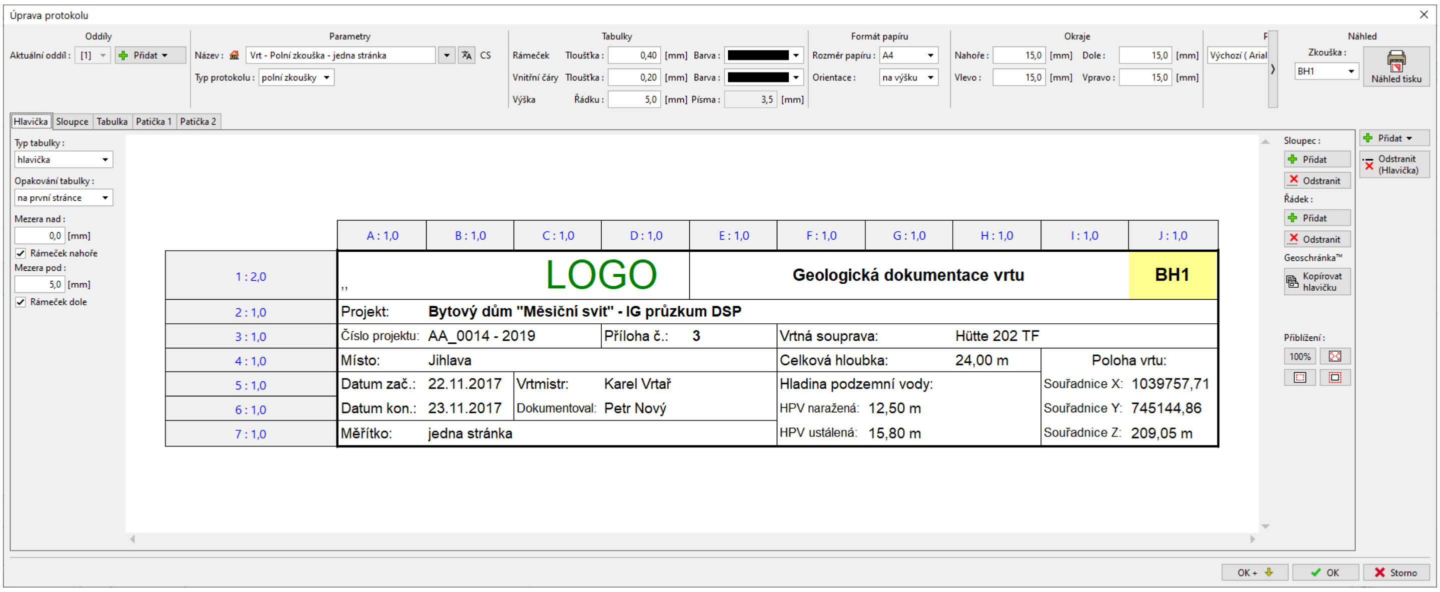
Print of Output Protocols

The design of output protocols is defined on the right side of the template dialog window. It is possible to define an arbitrary number of output protocols for each template - also for one type of field test.



In the upper part of the window, we define the general look of output - there we select if the protocol is used for "Field test" or "Soil profiles" and also we define a **scale of protocol** (one or two pages, 1:50, 1:100).

We select an edited part of the protocol by using tabs.

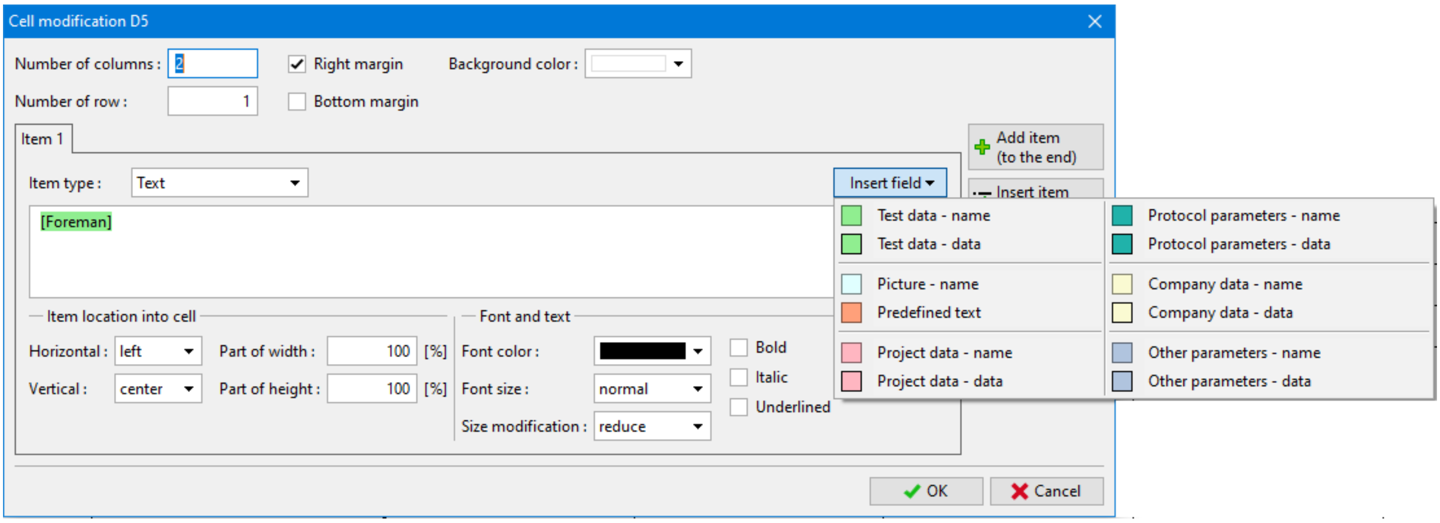


Editing of cell/field is similar to working e.g., in MS Excel - use a double click on the field. The field can contain data from various sources - as you can see in the picture.

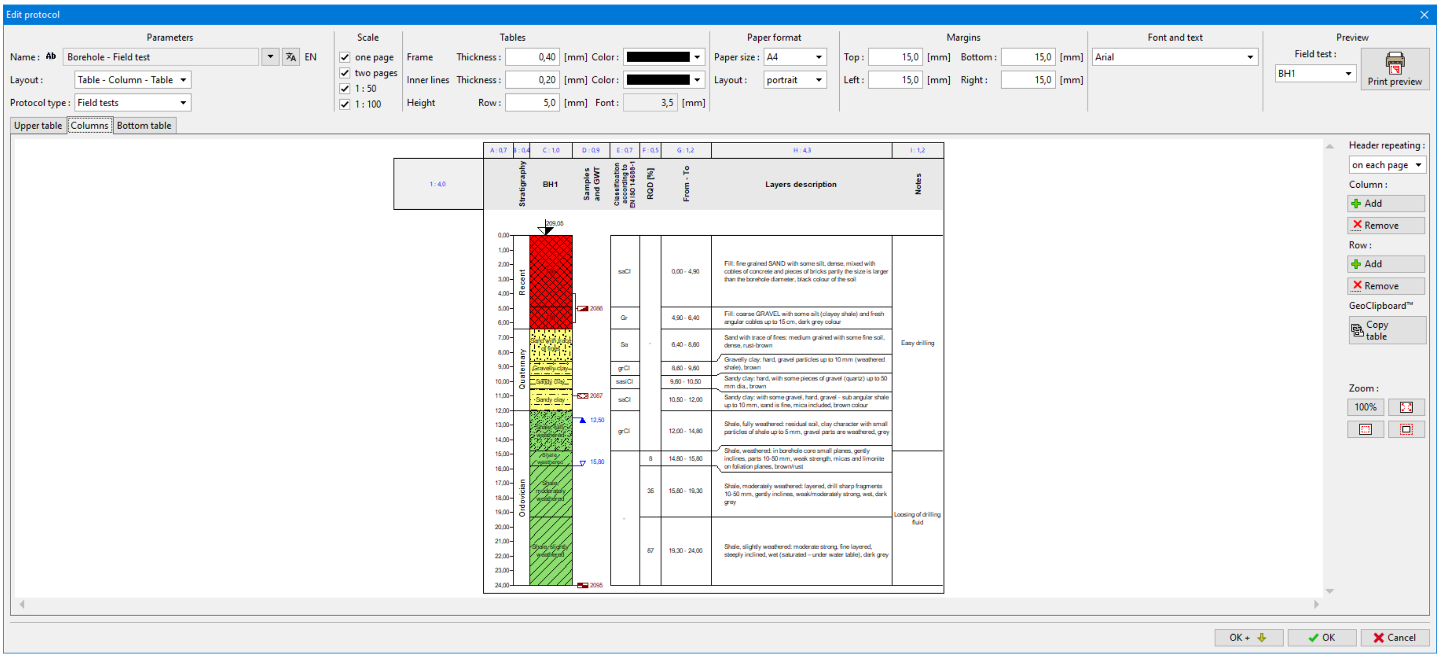
It is possible to enter **these data** to each field:

- data of field test defined in the current template
- project data - used in "Project" frame
- company data - used in "About the company" dialog window
- it is possible to select from predefined strings - item "**Predefined text**"
- it is possible to select from predefined pictures - item "**Picture - name**"
- or write an arbitrary **text into the field**.

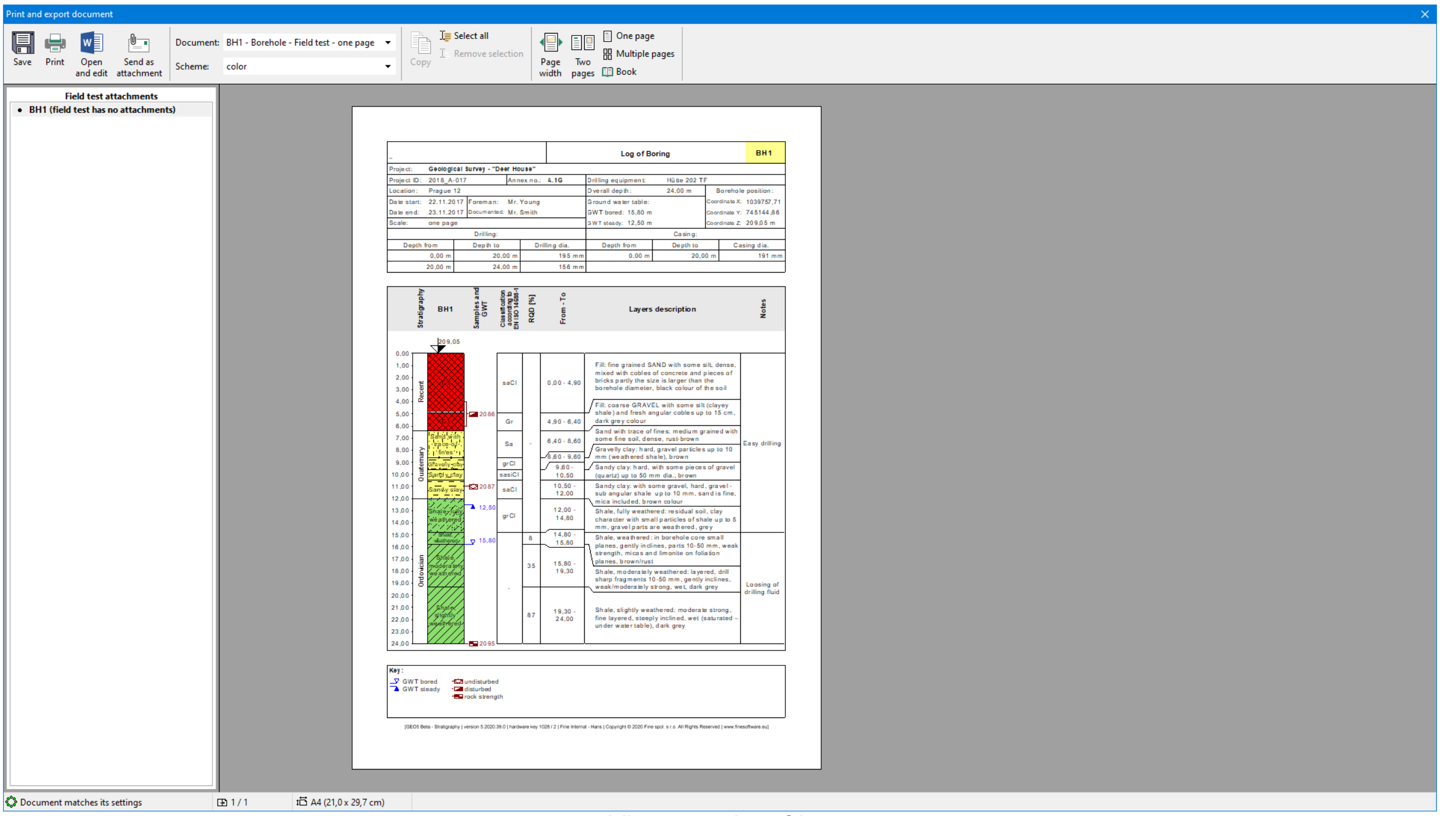
It is important to distinguish between items "**Name**" and "**Data**". "**Name**" inserts **name of datatype** (e.g. Drilling foreman), "**Data**" inserts **entered information** (e.g. Mr. Smith).



Output regarding **soil layers and measurement data** is edited in columns.



After editing protocol, we can check a final created template in the "Field Tests" or "Soil Profiles" frames - after selection of field test and pressing "**Print log**" button, the view of print is displayed.

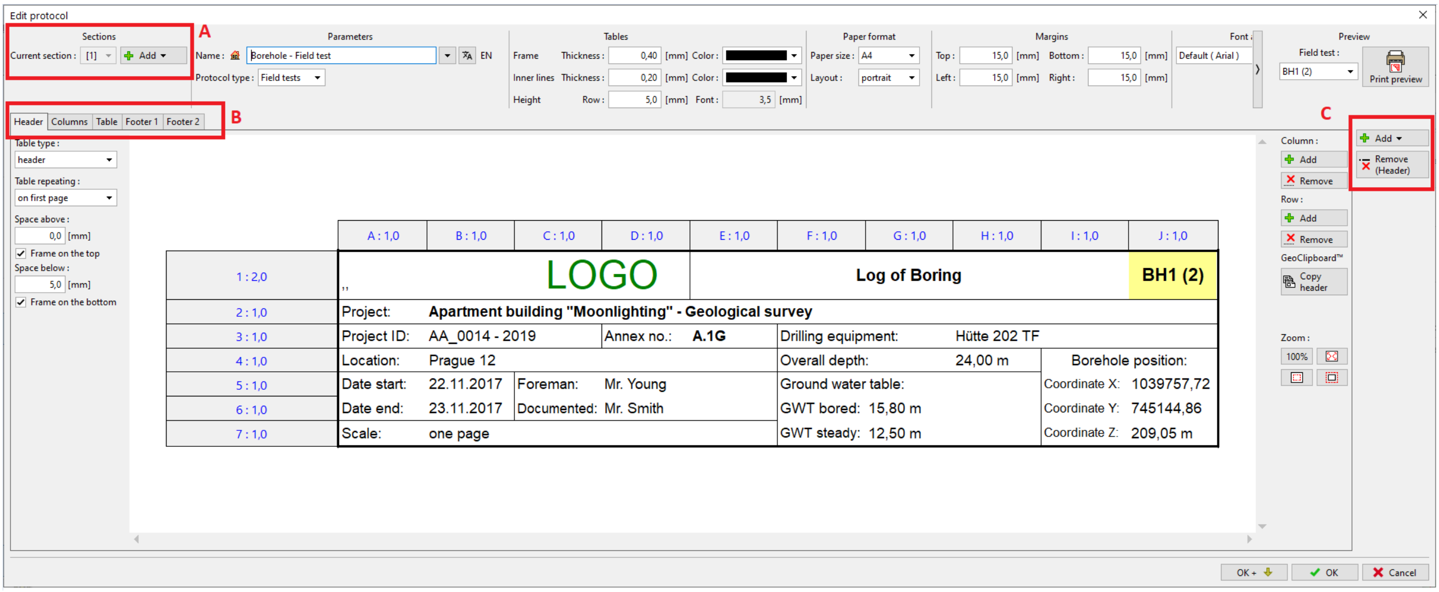


View on print of log

Layout of Output Protocol

The output protocol can contain an arbitrary number of **sections** (A).

Each section can contain **various tables and columns** (B), which are edited using the buttons in the (C) part.



Edit protocol

There are three **table types**:

- **header** - it is always placed at the beginning of the output protocol, it can be only on the first page or repeats on each page
- **footer** - it is always placed at the end of the output protocol, it can be only on the last page or repeats on each page
- **standard** - it is always placed between header and footer

The **columns** are placed between header and footer on the same level as the standard table.

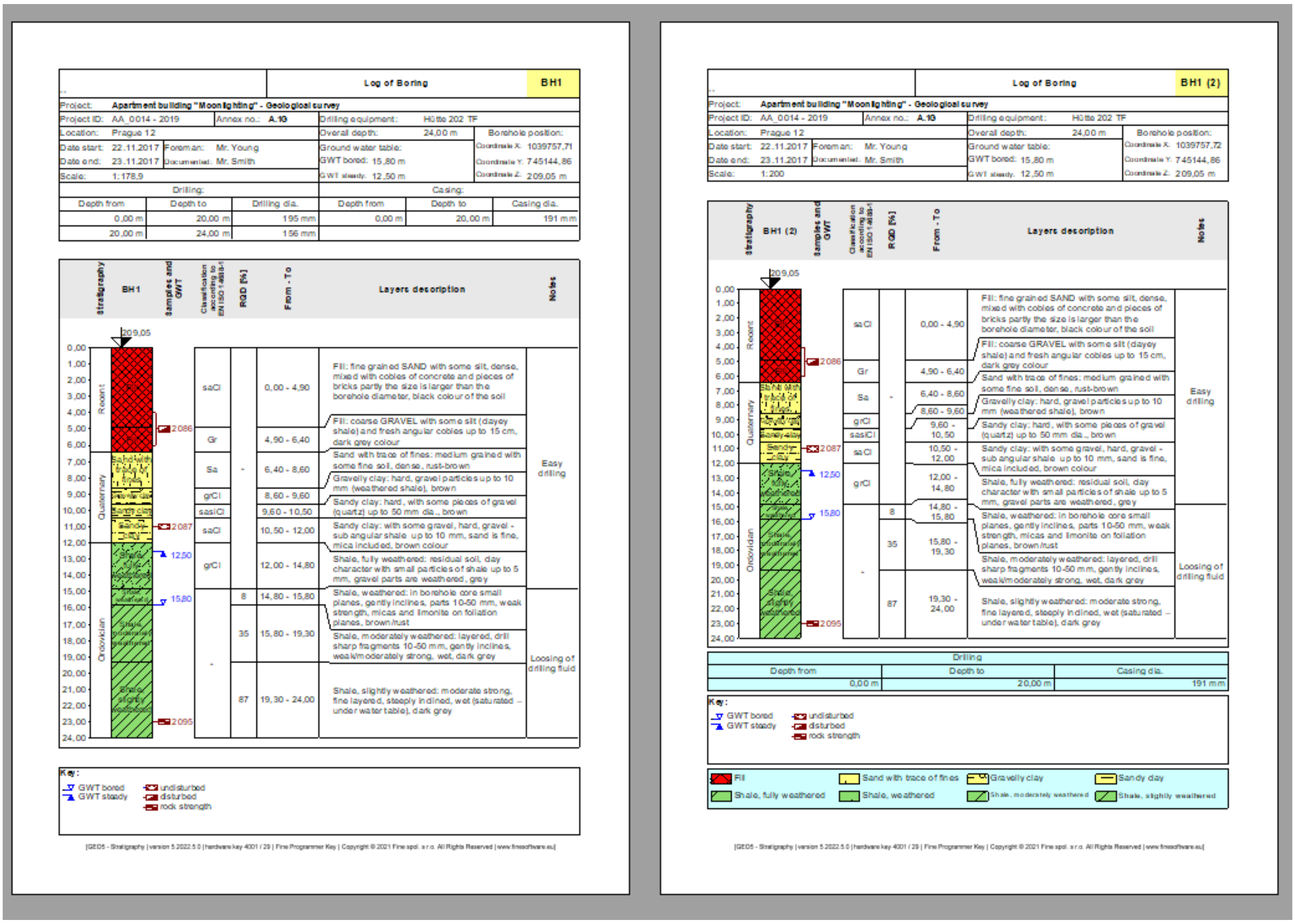
In case, that section contains **more individual parts, it is possible to change their order**. However, the basic order must be always respected: **Headers -> Standard Tables / Columns -> Footers**.

For example, if we want to use a different footer on each side of a protocol, it is necessary to create more sections:

- 1. section: header - columns - footer 1

- 2. section: header - columns - footer 2

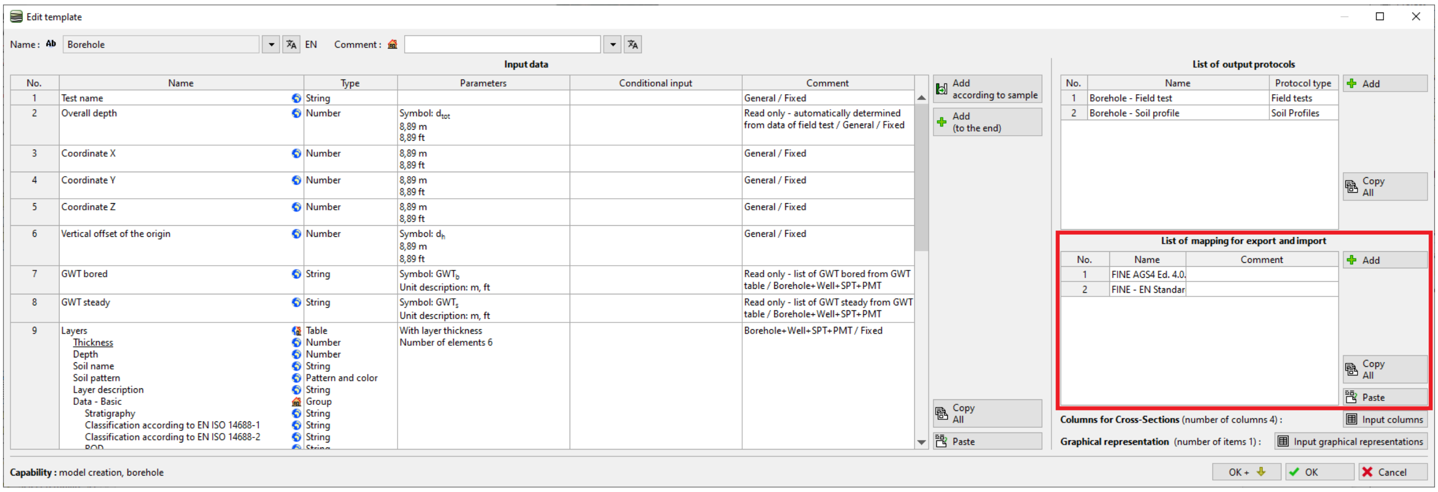
On the left side of the picture below, there is a **basic layout of the output protocol (header - columns - footer)**. On the right side of the picture, we inputted the standard table (Drilling) behind the columns and a second footer (Soils legend) at the end of the page - both new parts are colored in blue for better clarity.



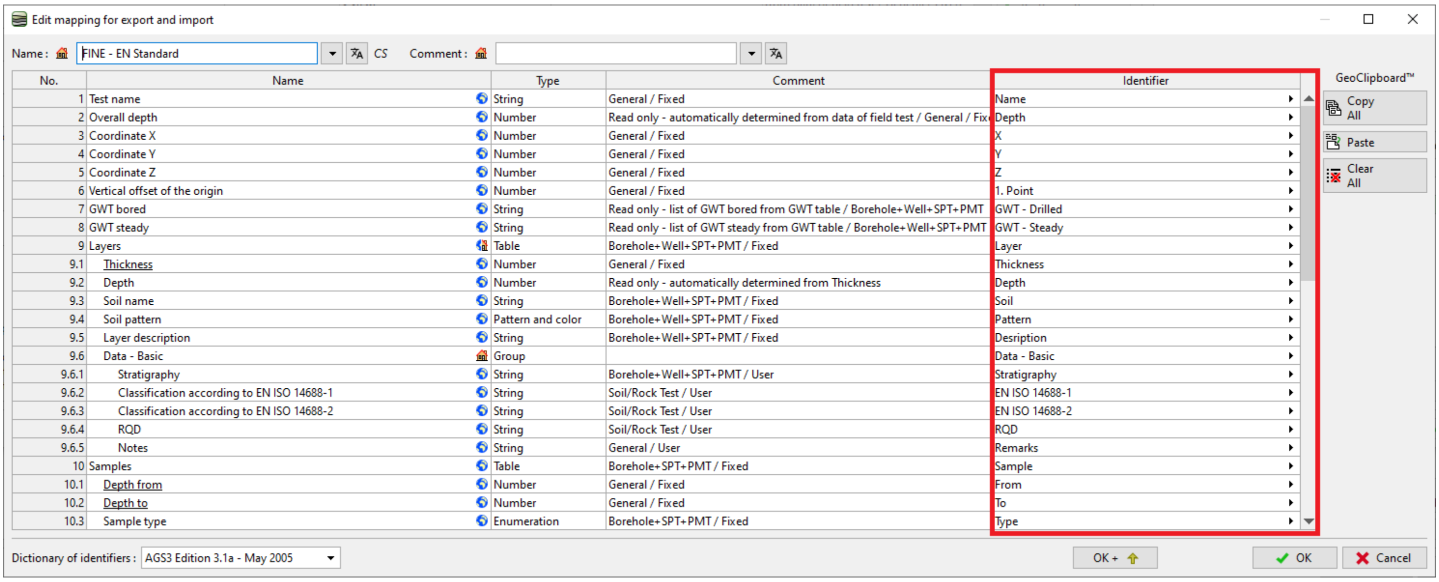
Mapping for Export and Import

The mapping determines **identifiers** which are used for **export and import** of each data.

The mapping is defined in the template. One template can have several mapping options. All templates contain the AGS mapping set as default, while some templates also contain different mapping options according to the country for which they are designed.



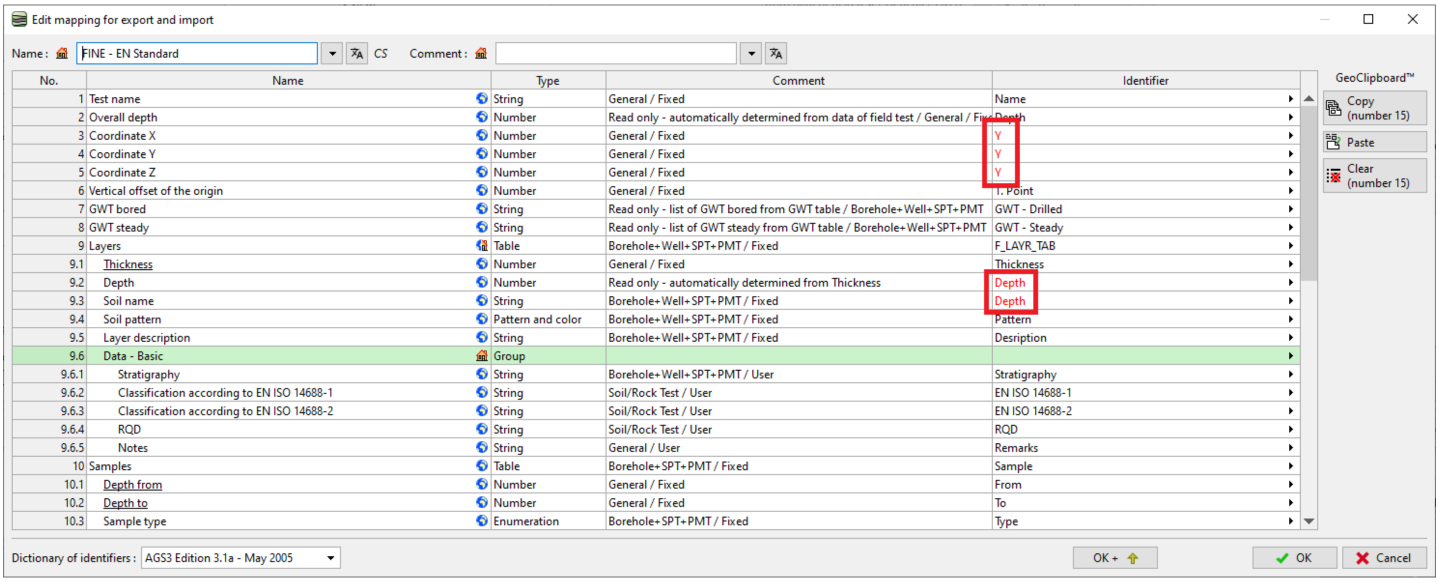
The mapping identifiers are defined and edited in the "**Edit mapping for export and import**" dialog window.


Edit mapping for export and import

Name : FINE - EN Standard CS Comment :

No.	Name	Type	Comment	Identifier
1	Test name	String	General / Fixed	Name
2	Overall depth	Number	Read only - automatically determined from data of field test / General / Fix	Depth
3	Coordinate X	Number	General / Fixed	X
4	Coordinate Y	Number	General / Fixed	Y
5	Coordinate Z	Number	General / Fixed	Z
6	Vertical offset of the origin	Number	General / Fixed	1. Point
7	GWT bored	String	Read only - list of GWT bored from GWT table / Borehole+Well+SPT+PMT	GWT - Drilled
8	GWT steady	String	Read only - list of GWT steady from GWT table / Borehole+Well+SPT+PMT	GWT - Steady
9	Layers	Table	Borehole+Well+SPT+PMT / Fixed	Layer
9.1	Thickness	Number	General / Fixed	Thickness
9.2	Depth	Number	Read only - automatically determined from Thickness	Depth
9.3	Soil name	String	Borehole+Well+SPT+PMT / Fixed	Soil
9.4	Soil pattern	Pattern and color	Borehole+Well+SPT+PMT / Fixed	Pattern
9.5	Layer description	String	Borehole+Well+SPT+PMT / Fixed	Desription
9.6	Data - Basic	Group		Data - Basic
9.6.1	Stratigraphy	String	Borehole+Well+SPT+PMT / User	Stratigraphy
9.6.2	Classification according to EN ISO 14688-1	String	Soil/Rock Test / User	EN ISO 14688-1
9.6.3	Classification according to EN ISO 14688-2	String	Soil/Rock Test / User	EN ISO 14688-2
9.6.4	RQD	String	Soil/Rock Test / User	RQD
9.6.5	Notes	String	General / User	Remarks
10	Samples	Table	Borehole+SPT+PMT / Fixed	Sample
10.1	Depth from	Number	General / Fixed	From
10.2	Depth to	Number	General / Fixed	To
10.3	Sample type	Enumeration	Borehole+SPT+PMT / Fixed	Type

It is also possible to create user-defined mappings. When creating a mapping, it is necessary to keep in mind that each identifier has to be unique and clear. The program warns us if the same identifier is already used by displaying it in red. Having multiple values with the same identifier can cause problems when importing the data.


Edit mapping for export and import

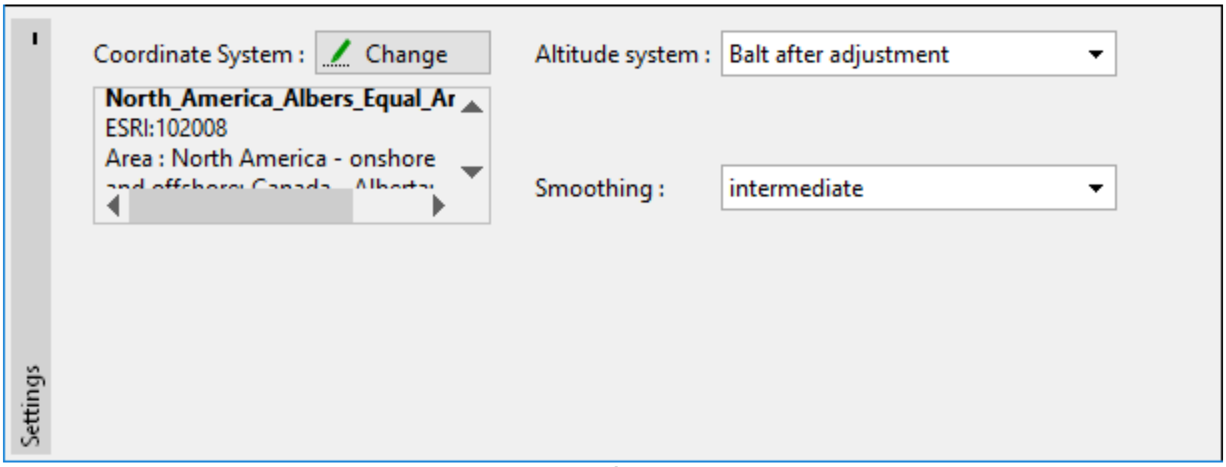
Name : FINE - EN Standard CS Comment :

No.	Name	Type	Comment	Identifier
1	Test name	String	General / Fixed	Name
2	Overall depth	Number	Read only - automatically determined from data of field test / General / Fix	Depth
3	Coordinate X	Number	General / Fixed	Y
4	Coordinate Y	Number	General / Fixed	Y
5	Coordinate Z	Number	General / Fixed	Y
6	Vertical offset of the origin	Number	General / Fixed	1. Point
7	GWT bored	String	Read only - list of GWT bored from GWT table / Borehole+Well+SPT+PMT	GWT - Drilled
8	GWT steady	String	Read only - list of GWT steady from GWT table / Borehole+Well+SPT+PMT	GWT - Steady
9	Layers	Table	Borehole+Well+SPT+PMT / Fixed	F_LAYR_TAB
9.1	Thickness	Number	General / Fixed	Thickness
9.2	Depth	Number	Read only - automatically determined from Thickness	Depth
9.3	Soil name	String	Borehole+Well+SPT+PMT / Fixed	Depth
9.4	Soil pattern	Pattern and color	Borehole+Well+SPT+PMT / Fixed	Pattern
9.5	Layer description	String	Borehole+Well+SPT+PMT / Fixed	Desription
9.6	Data - Basic	Group		
9.6.1	Stratigraphy	String	Borehole+Well+SPT+PMT / User	Stratigraphy
9.6.2	Classification according to EN ISO 14688-1	String	Soil/Rock Test / User	EN ISO 14688-1
9.6.3	Classification according to EN ISO 14688-2	String	Soil/Rock Test / User	EN ISO 14688-2
9.6.4	RQD	String	Soil/Rock Test / User	RQD
9.6.5	Notes	String	General / User	Remarks
10	Samples	Table	Borehole+SPT+PMT / Fixed	Sample
10.1	Depth from	Number	General / Fixed	From
10.2	Depth to	Number	General / Fixed	To
10.3	Sample type	Enumeration	Borehole+SPT+PMT / Fixed	Type

Settings

The basic parameters of task are entered in the frame "**Settings**":

- Coordinate system - determines the orientation of x , y axis. Entered coordinates x , y **are not recalculated** after a change of coordinate system.
- Type of terrain model and layers smoothing. Intermediate smoothing is recommended for realistic models and fast generation.
- **Altitude system of task** - is used just in the description of geological documentation. It has no influence on the model generation. Altitude coordinates z **are not recalculated** after change of altitude system.

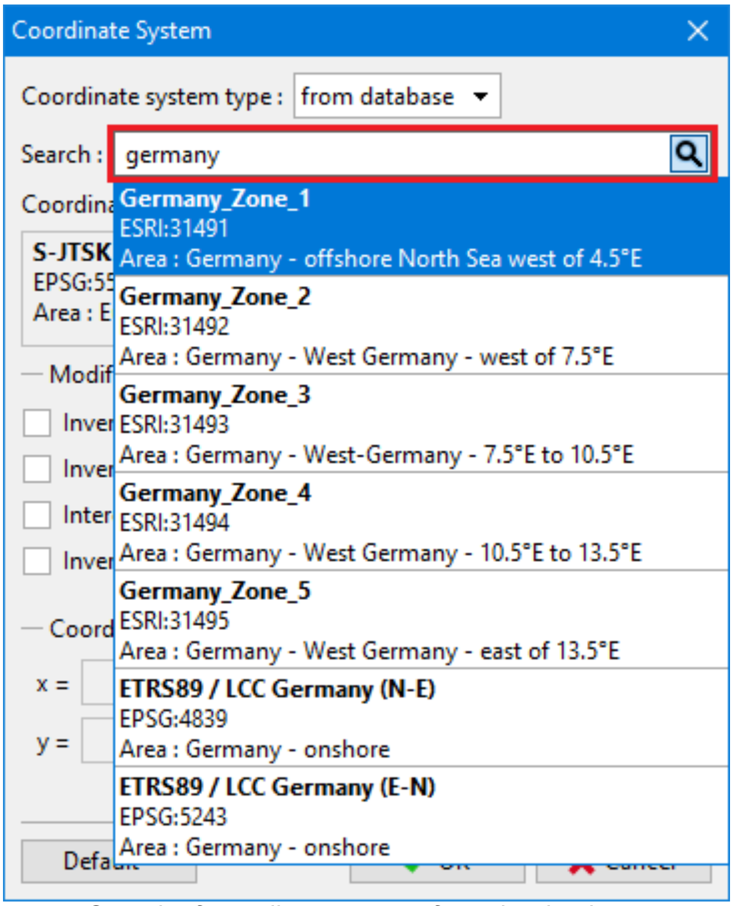


Frame "Settings"

Coordinate Systems

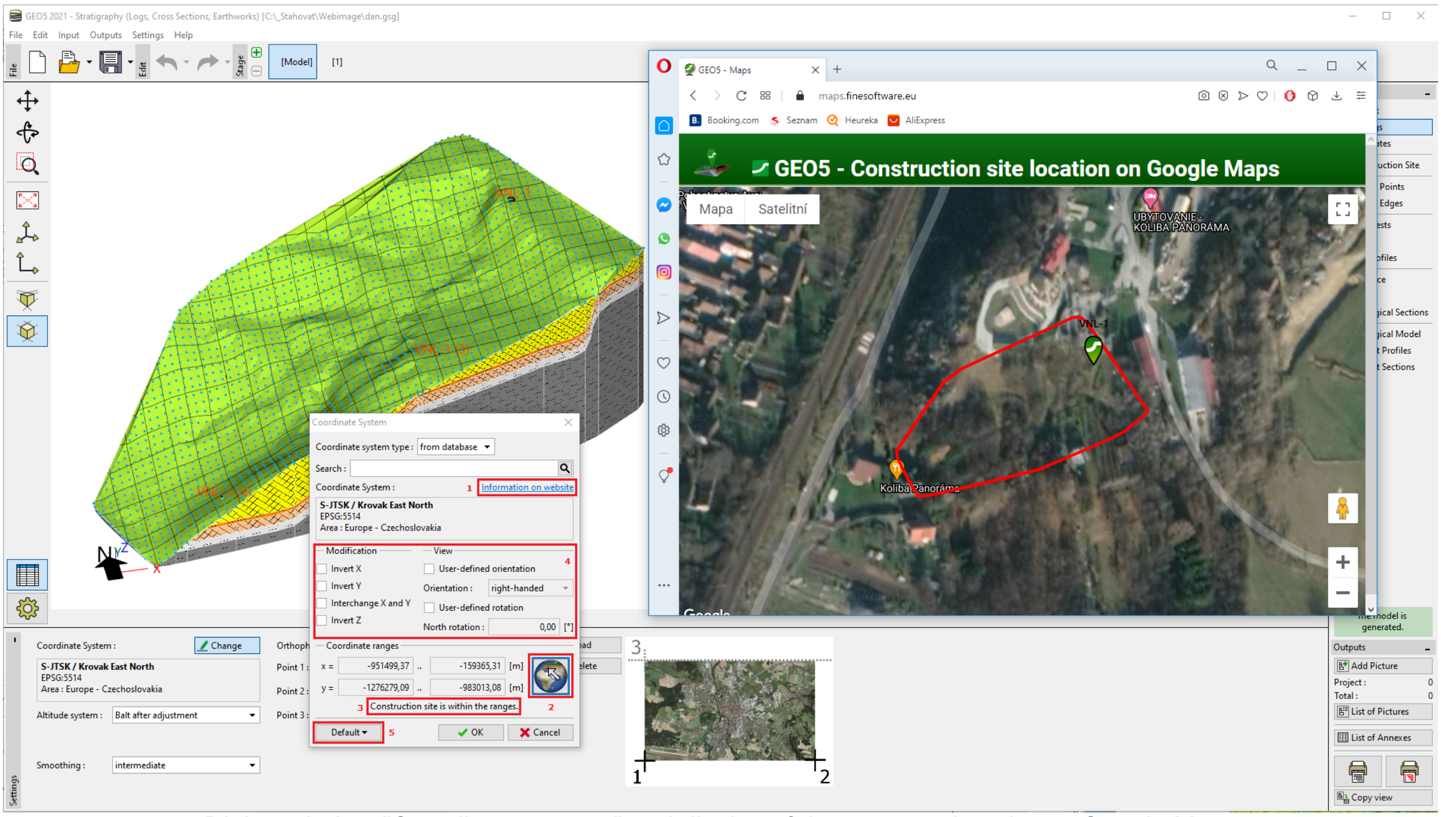
The **Stratigraphy** program allows us to input coordinates in one of **more than 6000 global coordinate systems** from the **MapTiler** database (https://epsg.io/).

The coordinate system can be found using the **index search** - after relevant text input (e.g. name of the system, area, country...) and pressing the "magnifier" button, the list of related global coordinate systems is shown.



Search of coordinate system from the database

After the selection of the **coordinate system**, all coordinates are inputted in the **range and orientation** of the selected system.



Dialog window "Coordinate system" and display of the construction site on Google Maps

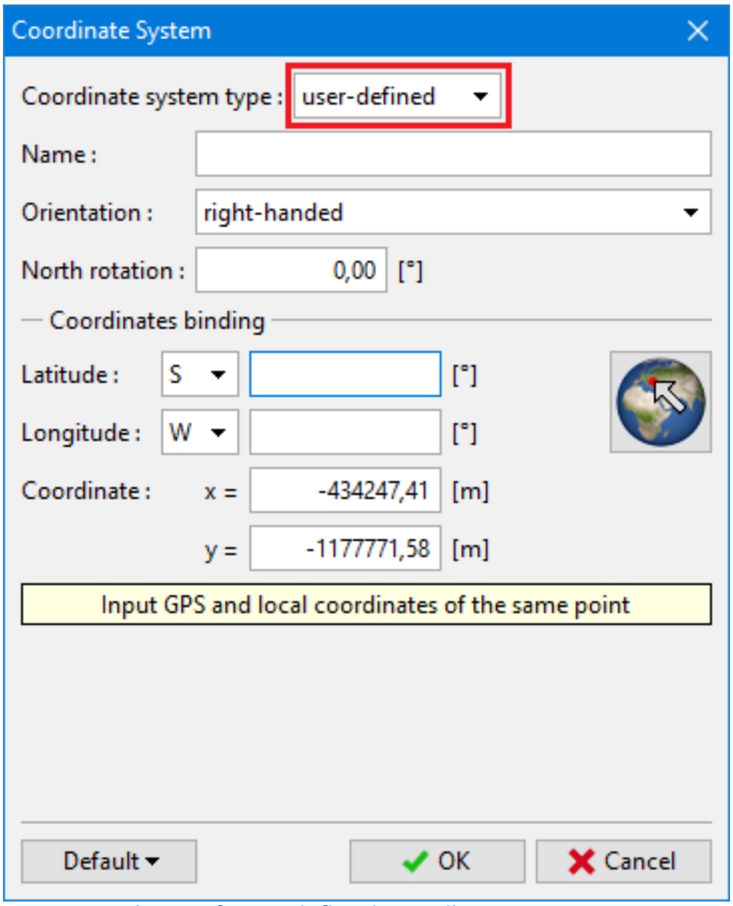
The detailed information about the selected system including the map of the covered area can be displayed using the "**Information on website**" button (**1**).

The **construction site** can be shown on **Google Maps** using the "**Globe**" button (**2**).

In case, that position on Google maps does not correspond to the real state, resp. **it is not in the range of the selected coordinate system at all** (**3**), the program allows us to perform a few changes, which can eliminate the most common mistakes in coordinate input - especially **inverting or interchanging of X, Y coordinates** (**4**).

After successful **settings** of the coordinate system, it is possible to **save it** and **load it as a default** for further tasks (**5**).

If no coordinate system from the database meets our requirements, the user-defined system can be used. In this case, **it is necessary to input** one point in both - **local system coordinates** and **GPS coordinates**.



Input of user-defined coordinate system

Construction Site

A "**Construction Site**" defines a boundary of the generated model. It can be **entered (rectangle, polygon)** or **automatically calculated (rectangle, convex hull)** from the entered terrain points and all tests. If we don´t know the exact coordinates of the construction site, they can be defined using the "Rectangle GPS" option.

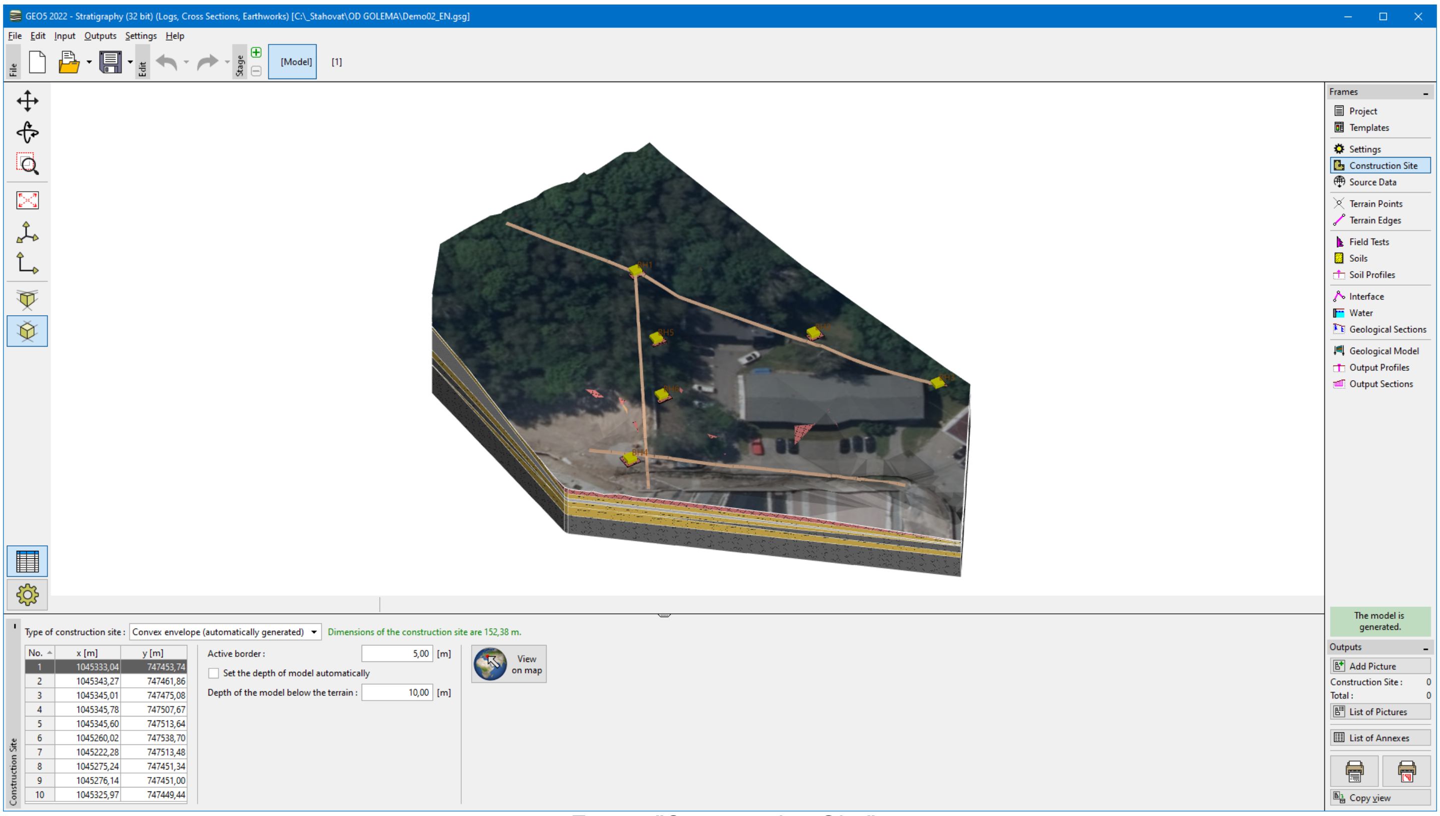
Entered points of the construction site boundary, don't have a defined z -coordinate, which is automatically calculated as a coordinate of the nearest valid point of the model (point, field test with z -coordinate).

Points and layers in the active edge are also transferred into the construction site boundary.

Terrain points and fieldtests can also be entered (imported) out of the construction site boundary. If the dimensions of the construction site are manually entered, these points and tests have a description "**Out of construction site**", and they are not displayed on the desktop. In the case of automatically calculated dimensions, the **construction site** is regenerated.

It is also possible to show the construction site on Google Maps.

If the dimensions of the construction site are **too large** (distance between points is larger than 5000 m), the program does not draw a construction site, and it is necessary to revise the model.



Frame "Construction Site"

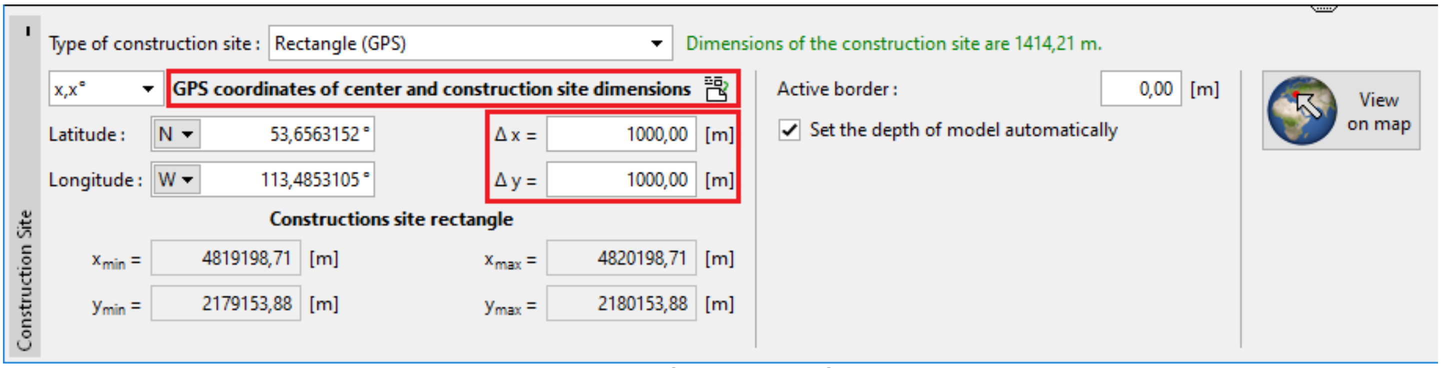
Definition of Construction Site using the GPS

If we don´t know the coordinates of the construction site, but we know the area of our interest, it is recommended to obtain coordinates as follows:

Open e.g. Google Maps in the web browser and find a middle of the required area. Using the right mouse button we display the contextual menu and copy the coordinates using the left mouse button.

Maps

In the program (in the **"Construction Site**" frame), we select a **"Rectangle (GPS)**" option. Using the button on the right from the "**GPS coordinates of center and construction site dimensions**" description, we paste the coordinates from the clipboard. Finally, we define the dimensions of the construction site (Δx , Δy).



Frame "Construction Site"

Entered GPS coordinates are transformed to the local coordinates x y using the defined coordinate system. We can **check the defined construction site** using the "**View on map**" button. In case, the GPS point is out of range of the defined **coordinate system**, the construction site will be displayed out of this area. In this case, it is necessary to define the correct coordinate system.

Defined construction site

Available map sources and points of interest (e.g. from the database of historical boreholes) can be loaded in the "Source Data" frame.

Active Edges

For a correct generation of the geological and terrain model, the **boundary conditions** (points heights in corners and edges of the construction site) are very important.

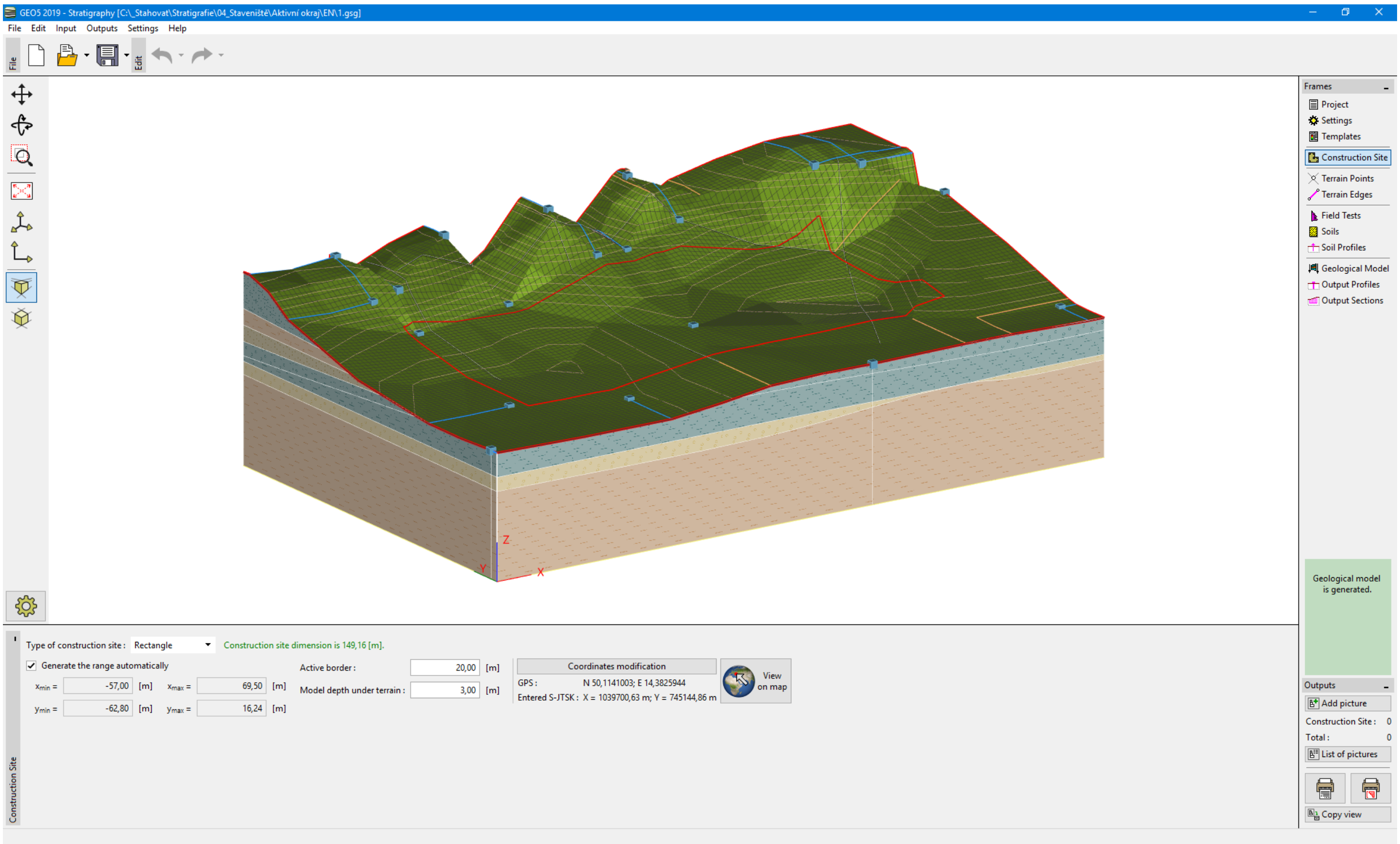
The **points and boreholes on the edges of the construction site** are automatically created when the geological model is generated. First, the z -coordinate of the closest point (or test) is assigned to the construction site points. Then, the nearest borehole is assigned to these points.

This modeling is suitable for flat terrains. In the case of slope modeling, the **model is not created correctly on the edges**, where boundary conditions are not defined. **Active edge** helps to **define a boundary condition on the edges of the construction site**.

An active edge is shown in **red color**.

When a model of the terrain is generated, all points and tests with entered z-coordinate in the active edge create an assistant point on the model boundary. **Connections between points (tests)** and assistant points are shown in **blue**.

When a geological model is generated, all boreholes and interface points in the active edge create an assistant borehole on the model boundary. **Connections between boreholes and assistant boreholes** on the edges are shown in **orange**.



Frame "Construction Site" - active edge settings

The use of active edges is also explained in the example of Construction Site Edges.

Source Data

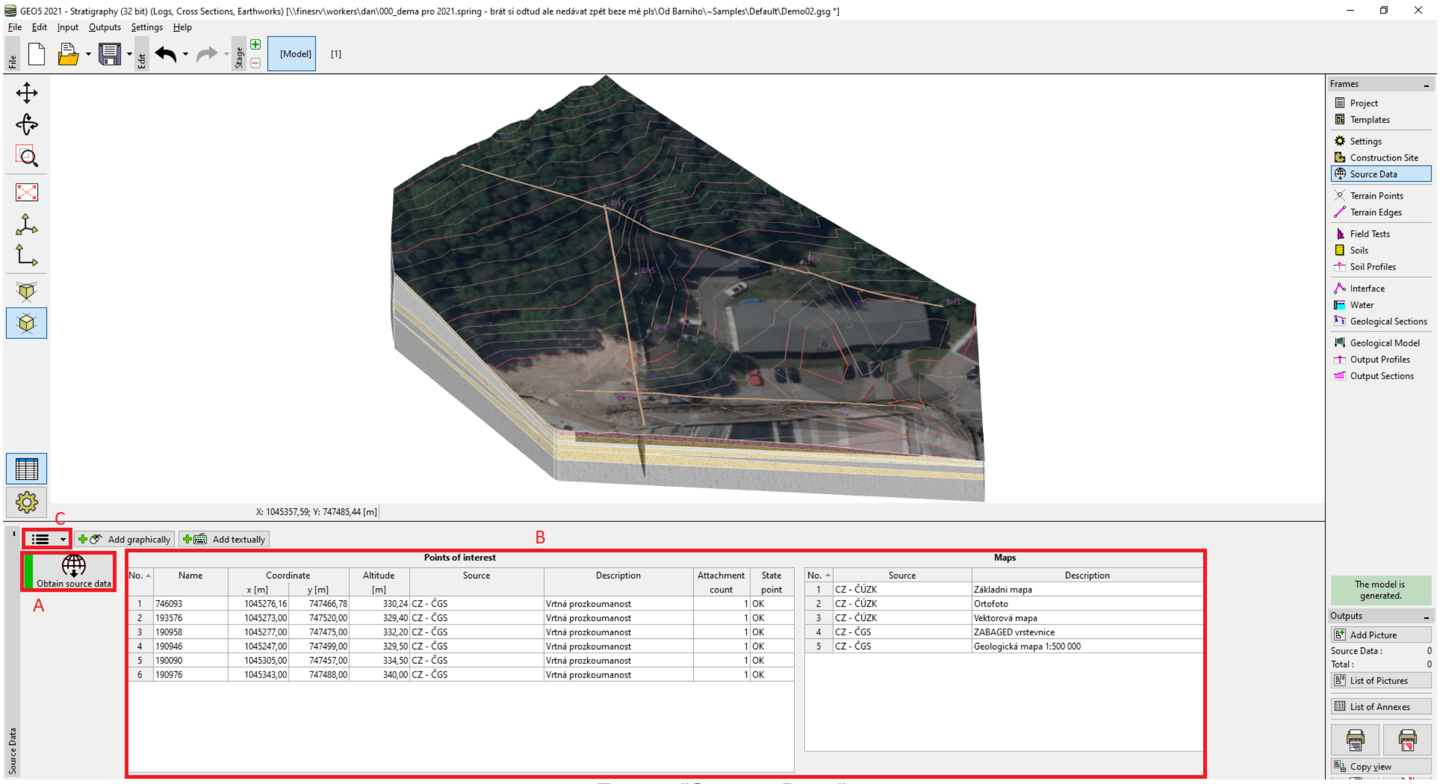
All data related to the construction site area can be loaded in this frame - e.g. maps and points of interest.

Using the "**Obtain source data**" button (**A**), the dialog window for loading of maps and points of interest is opened.

Loaded maps and points of interest are then displayed in the bottom tables (**B**).

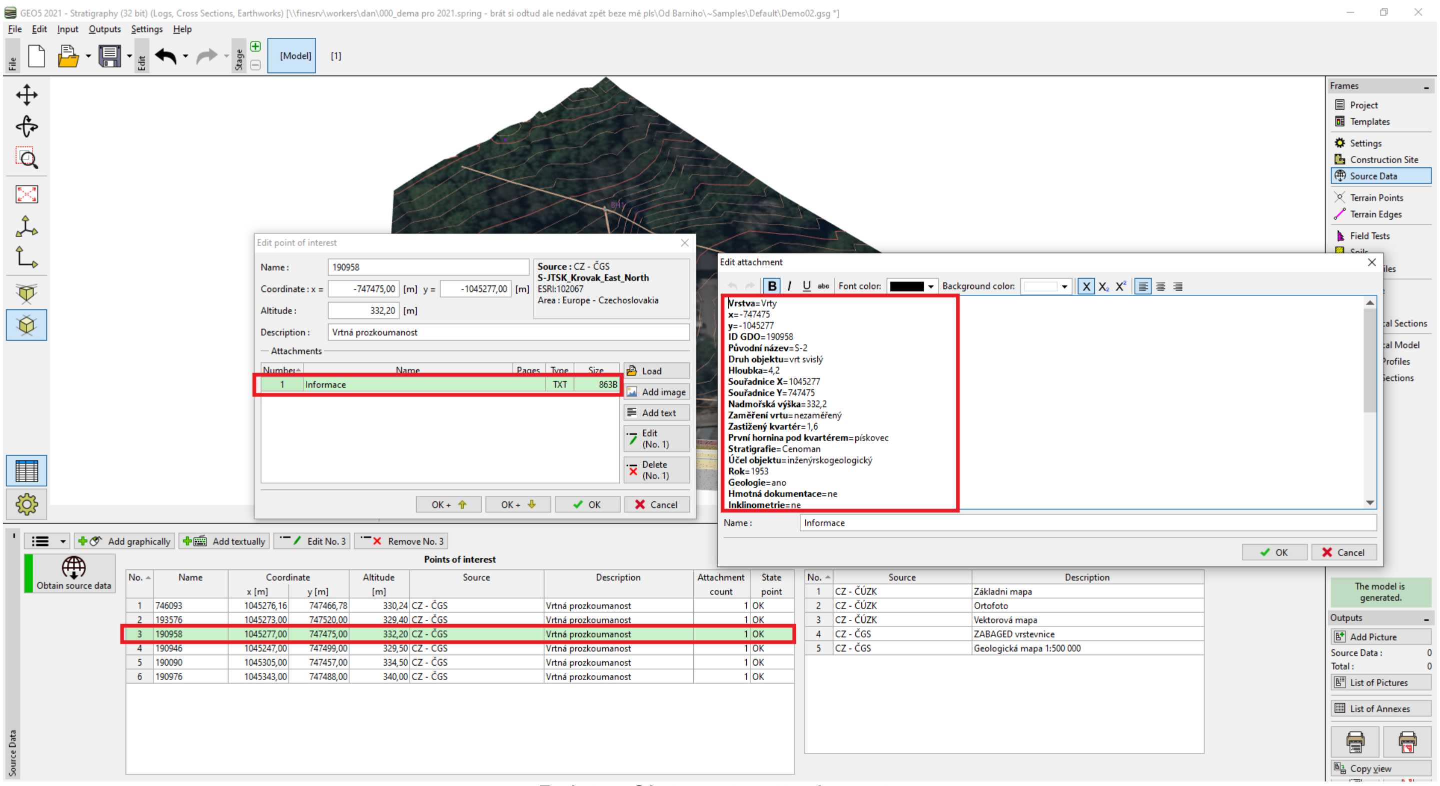
The program allows us to import also own source data (pictures, plans) and place them on the terrain.

The special data operations can be performed using the "**Options**" button (hamburger) , or right mouse click on the selected points.



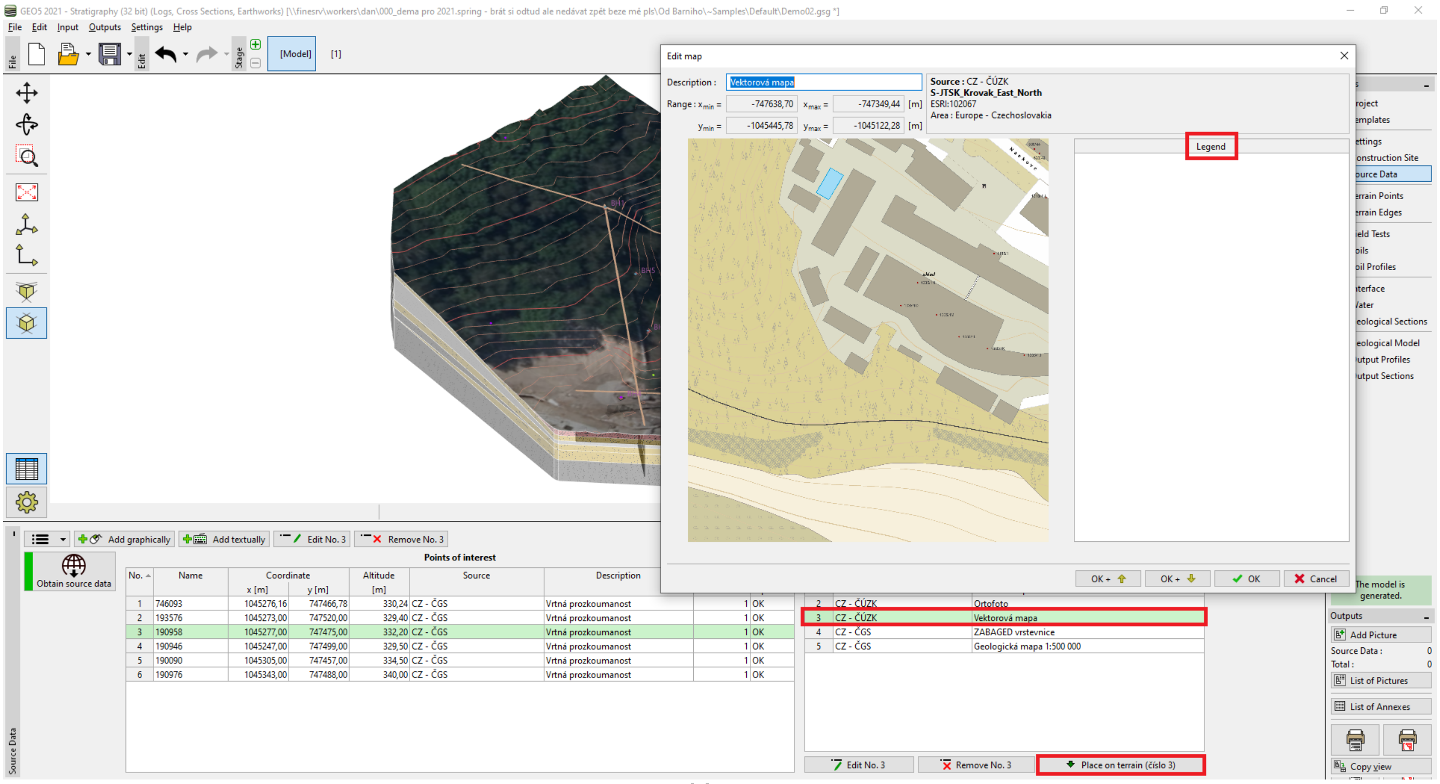
Frame "Source Data"

Points of interests can have automatically loaded attachments - e.g. the information about boreholes from the geology agency. Also, the user-defined attachments can be added to each point of interest - for this reason, the points can be defined also manually (graphically, textually).



Points of interest - attachments

The loaded maps or own data sources can be placed on the terrain as "ortophotomap" using the "**Place on terrain**" button.



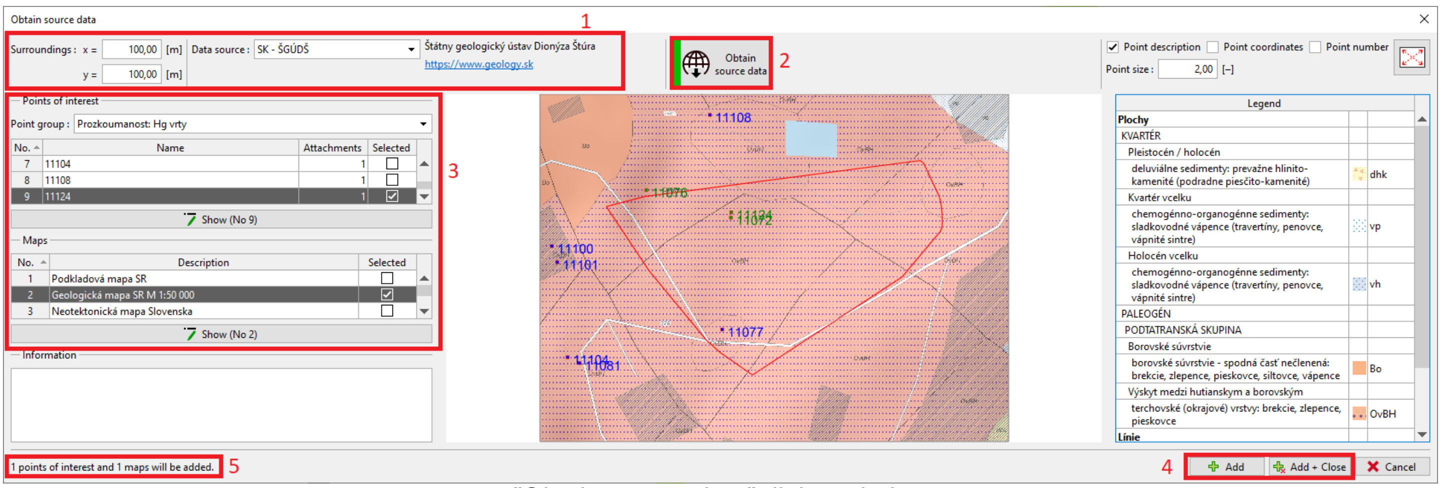
Maps

Maps and points of Interest

All data which relates to the construction site can be obtained in the "Obtain source data" dialog window. The construction site is **bordered in red** in the middle of the window. In the same part of the window, currently selected map and available points of interest are displayed. It is possible to zoom or select points using the mouse.

In the upper part of the window, the **data source and surroundings of construction site is defined** (**1**). In the current version of the program, the general sources (OpenStreetMap, Google Maps) and local sources of Czech and Slovak Republic (e.g. ČGS, ČÚZK, etc.) are implemented. We will be happy to implement other sources according to the customers feedback.

Using the "**Obtain source data**" (**2**), the available maps and points of interests will be loaded (**3**). They can be added to the model using the "**Add**" buttons (**4**) in the right bottom part of the window. The number of added points and maps is shown in the left corner (**5**).



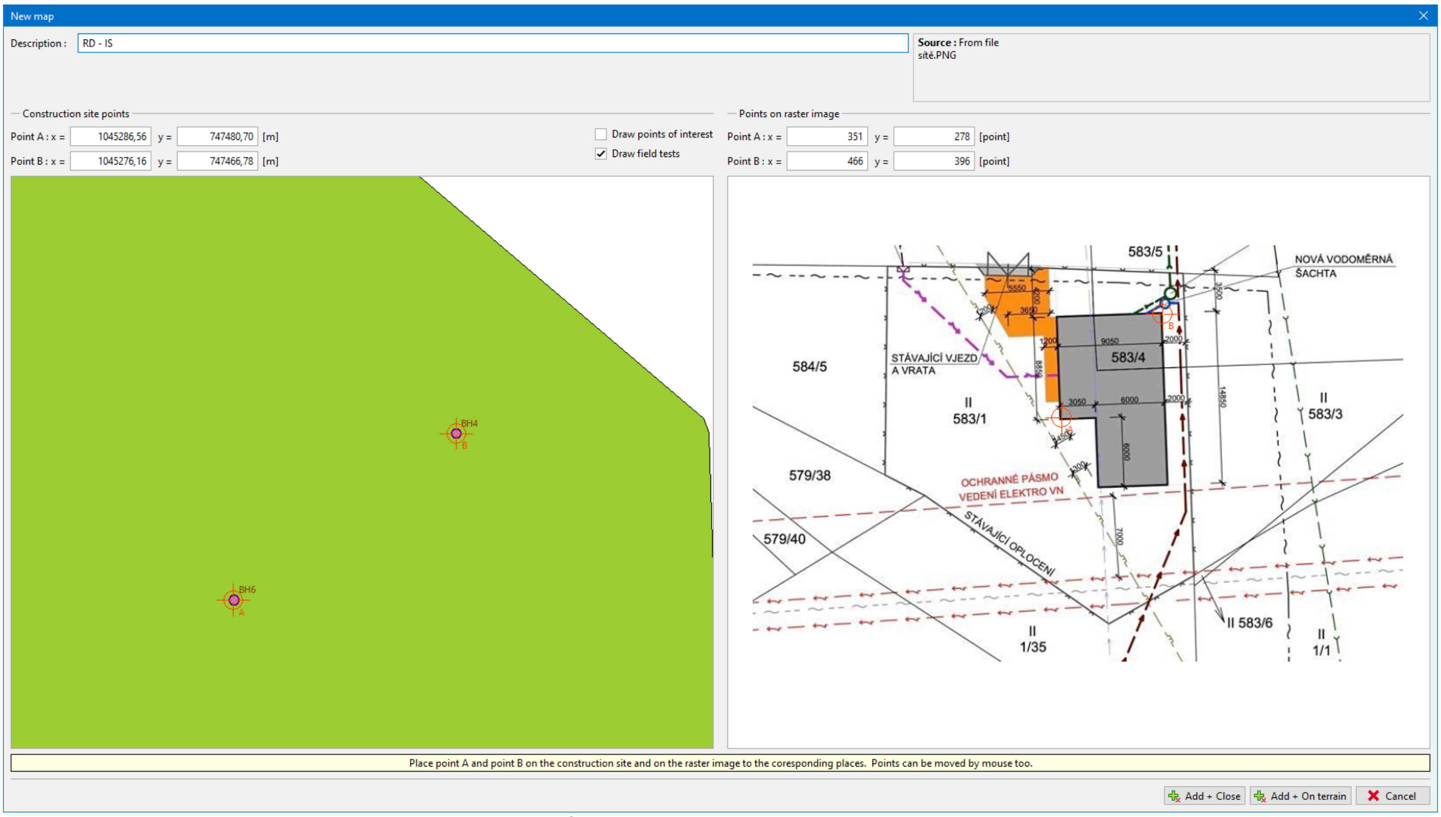
"Obtain source data" dialog window

***Note**: Legend of the map is shown in the right part of the window only in case that source data allows it (here the geological map of Slovak Republic).*

Own Source Data

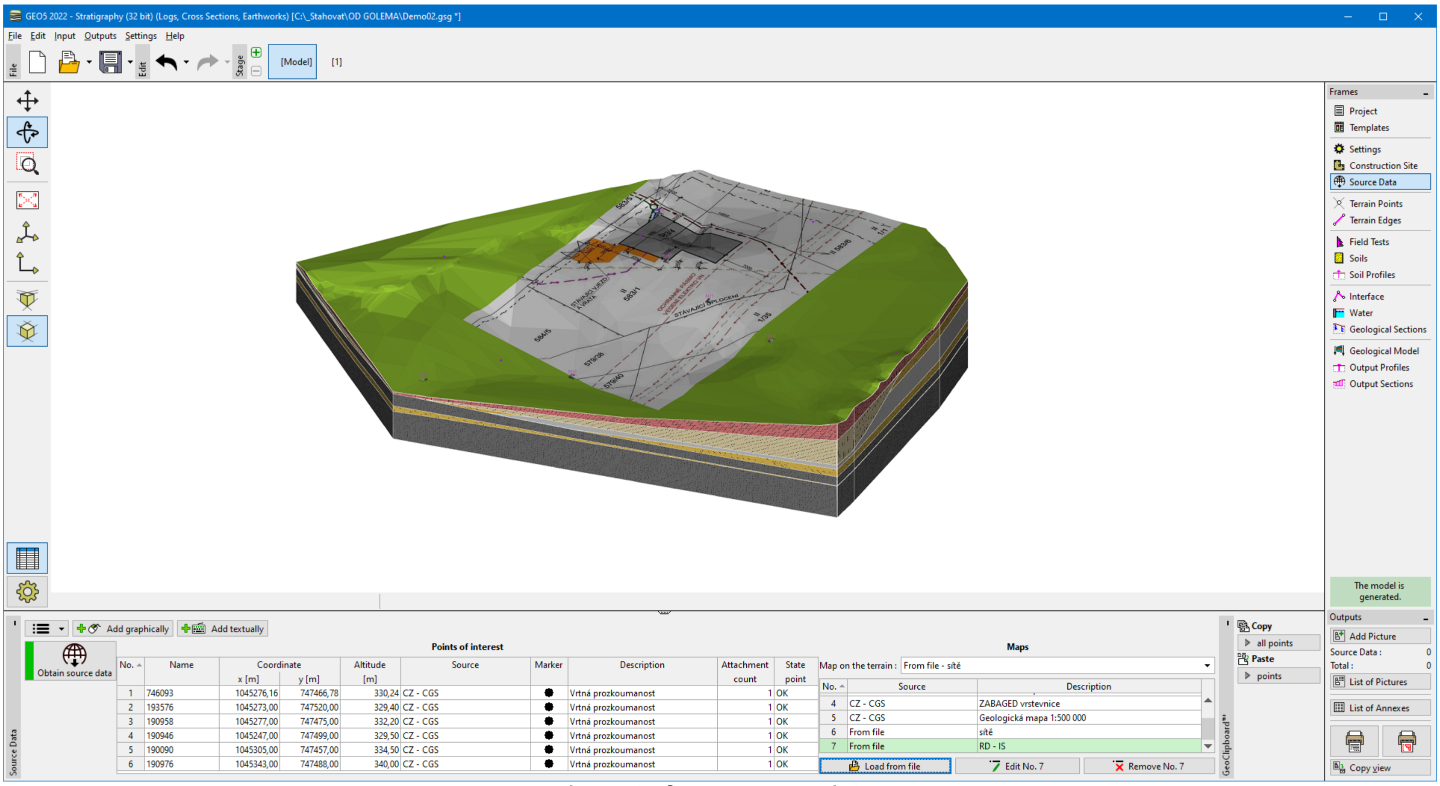
The program allows to import own source data in the ***.jpg**, ***.jpeg**, ***.jpe**, ***.png** a ***.bmp** formats. Using the "**Load from file**" button, the dialog window for import is opened. In the window, we select the file and load it.

The loaded file is shown in the right part of the "**New map**" dialog window. On the left, the construction site is displayed. It is also possible to draw all known points on the construction site - e.g. field tests or points of interest. The file is placed using the **tying two points** (A, B). In our example, we want to display the utility plan on the terrain because of the construction of the new house. We know, that boreholes BH4 and BH6 were carried out in the future corners of the planned house. So we place the points (A, B) to the position of BH4 and Bh6 boreholes on the left side and to the position of planned corners on the right side.



Placing of points to the corresponding places

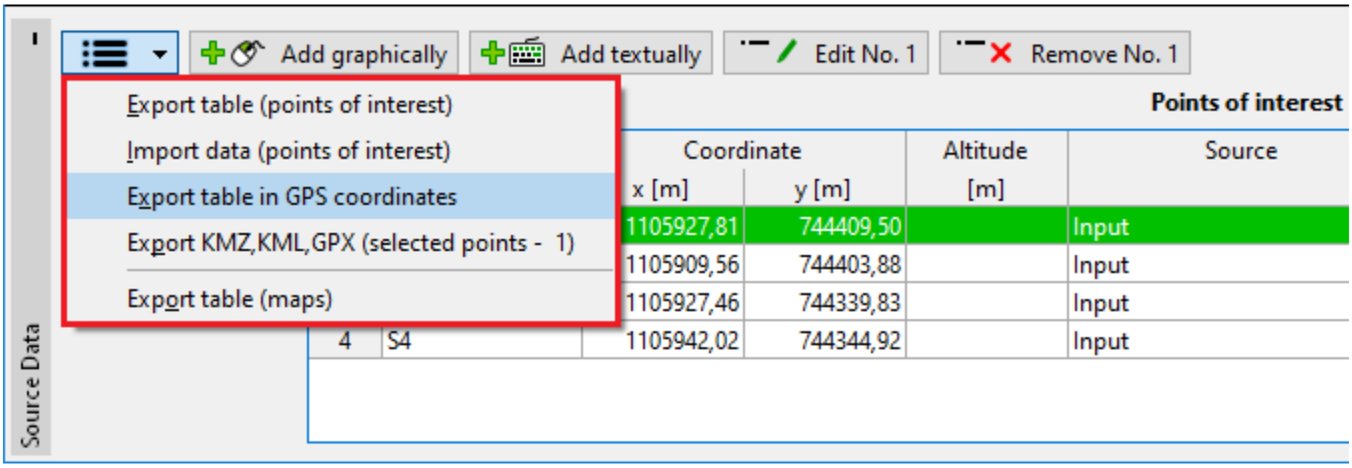
The points can be shifted using the mouse. The picture (map) is inserted using the "**Add + Close**" button into the list of sources, or it can be displayed directly in the model using the "**Add + On terrain**" button.



Import of own source data

Special Data Operations

The special data operations can be performed using the "**Options**" button (hamburger) or right-mouse click on the selected points in the table.



These tools are very useful for working with data, especially for importing and exporting data:

- **Export table** (points of interest) - it exports the table of points of interest coordinates in ***.xlsx**, ***.xls**, ***.ods**, ***.csv**, or ***.htm** format

	A	B	C	D	E	F	G	H
1	No.	Name	Type	Number of points	Coordinate z	Slope inclination	Width	Earthwork mode
2					[m]	[°]	[m]	
3	1	Embankment	polygonal	4	620,00	45,00		fill and excavate
4	2	Roadway	line	8	615,00	60,00	6,00	fill and excavate

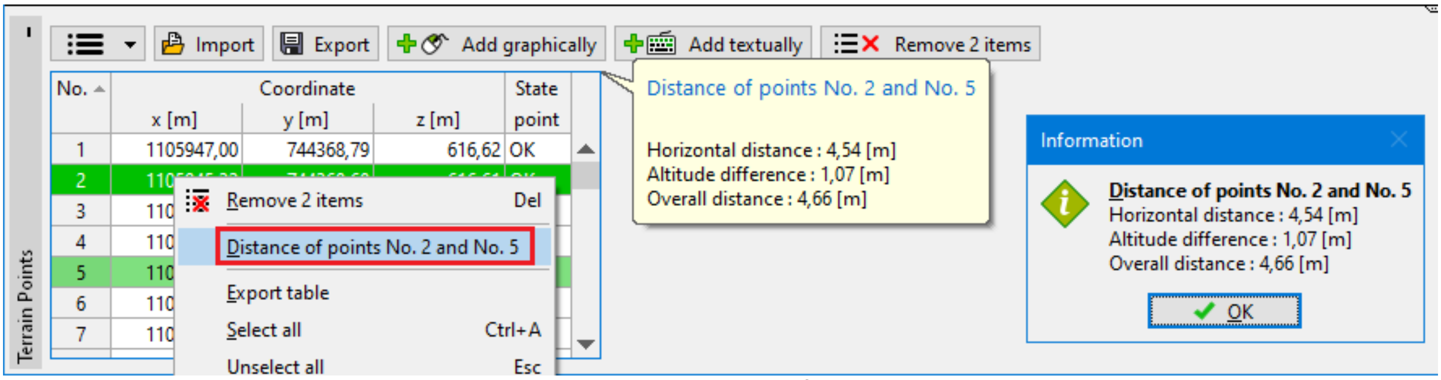
*Example of data export in *.xlsx (Excel) format*

- **Data import** (points of interest) - it allows the import of table data
- **Export table in GPS coordinates** - it exports the table of points of interest GPS coordinates in ***.xlsx**, ***.xls**, ***.ods**, ***.csv**, or ***.htm** format

	A	B	C	D	E	F	G	H
1	No.	Name	Latitude	Longitude	Altitude	Source	Marker	State
2			[°]	[°]	[m]			point
3	1	S1	49,5250397	14,5178360		Input		OK
4	2	S2	49,5252091	14,5178789		Input		OK
5	3	S3	49,5251274	14,5187889		Input		OK
6	4	S4	49,5249915	14,5187464		Input		OK

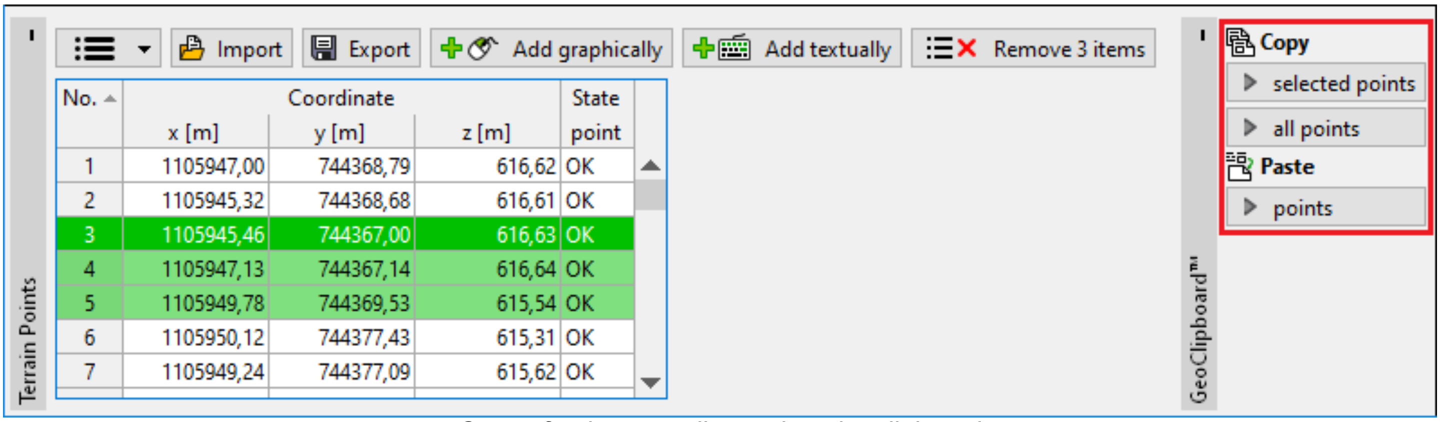
*Example of data export in GPS coordinates in *.xlsx (Excel) format*

- **Export KMZ, KML, GPX** - it exports points in ***.kmz**, ***.kml**, ***.gpx** formats. These files can be then imported into mobile devices or map applications (e.g. Google Maps, etc.).
- **Distance of points** - it measures the distance between two selected points. The points can be selected in the table, or on the desktop. After selecting two points, the bubble help with information is shown. Using the right mouse button on the selected points in the table or on the desktop, the contextual menu with the "**Distance of points**" option is shown. This option opens the "Information" dialog window where the distance is shown.



Distance measurement of two points

- **Copy of points coordinates into the GeoClipboard** - the coordinates can be copied into the clipboard. We can copy all or only the selected points. These points can be then pasted in other frames - this way, we can, for example, transfer points from **Source Data to the Terrain Points**.

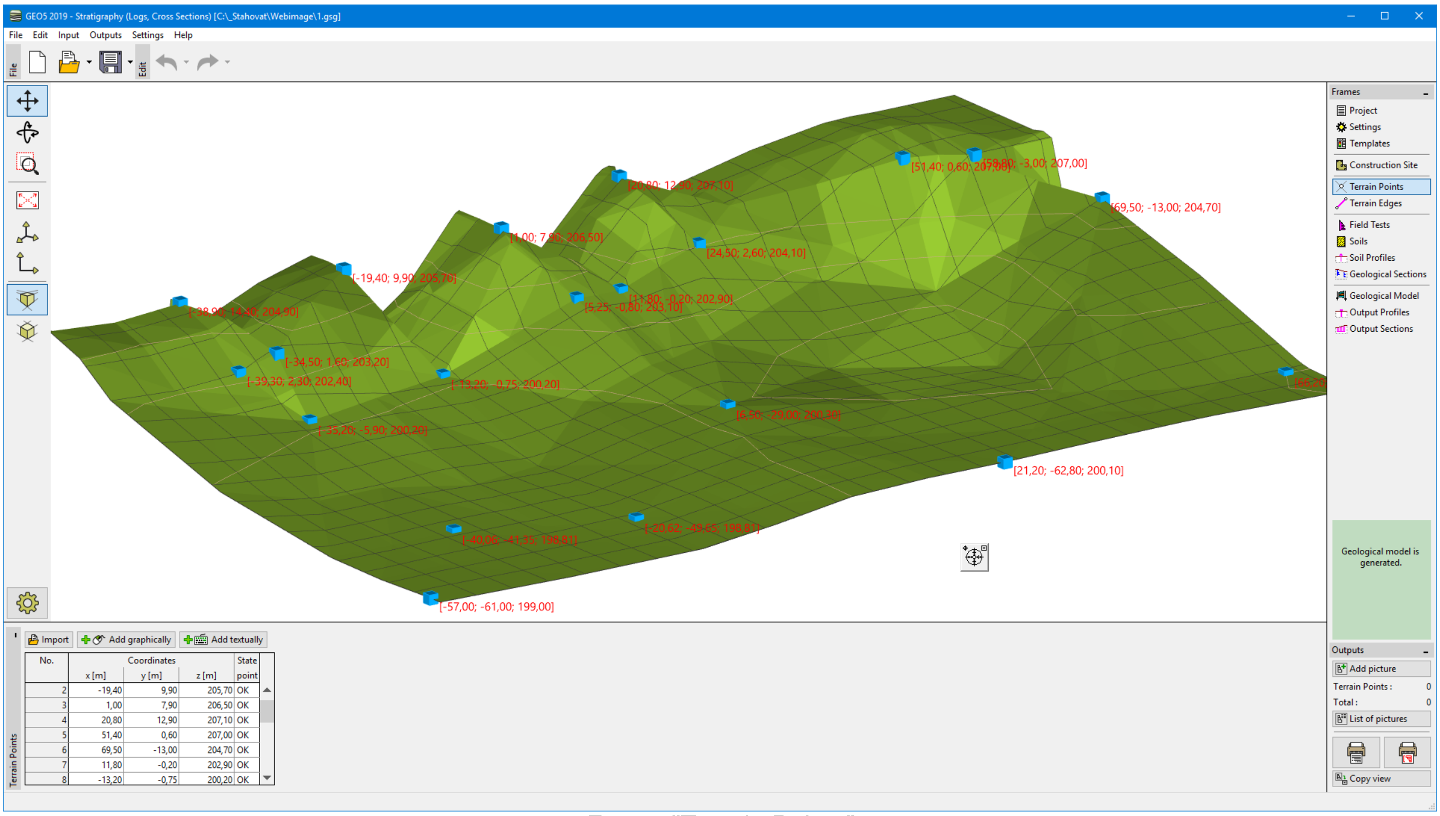


Copy of points coordinates into the clipboard

Terrain Points

The coordinates of the terrain points are entered in the frame "**Terrain Points**". The dialog window "**New terrain points**" serves to add new points.

When new points are entered into an already generated model, the heights of points Z are automatically calculated. Only one point can be added to one X, Y coordinate.



Frame "Terrain Points"

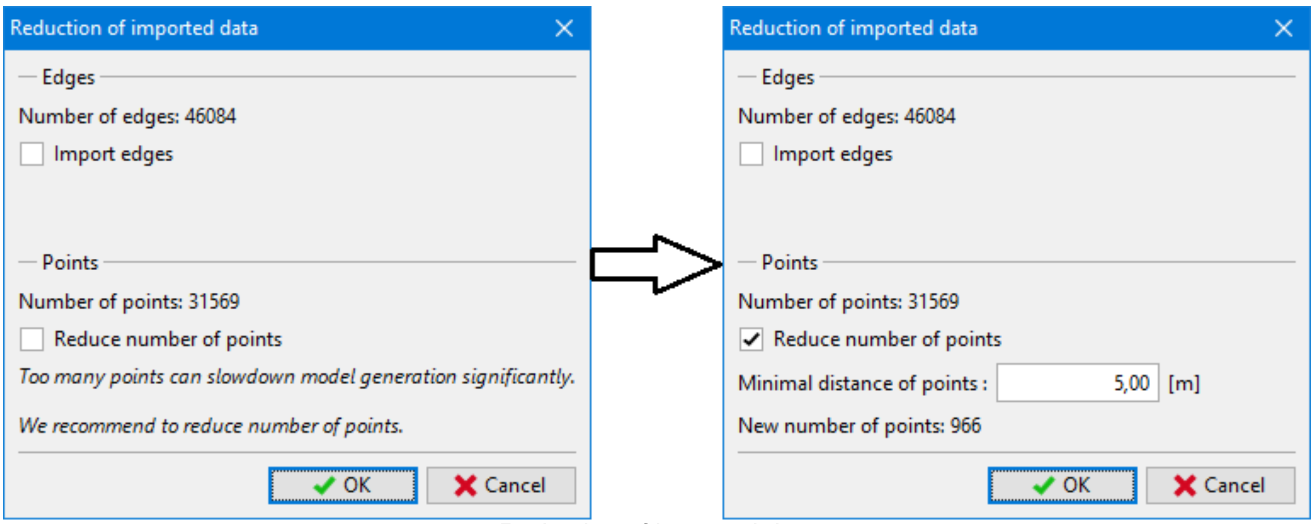
The **terrain model is always generated** after every addition or change of an arbitrary point.

The points can also be imported using the "**Import**" button in various formats.

- Text tables
- XLSX and ODS Tables
- LandXML (*.xml)
- DXF (*.dxf) - import of points is similar to using LandXML data

The program allows us to import an unlimited amount of data (points, edges), but the generation of the model with a very high number of points or edges can be time demanding and usually is unnecessary. For example, the import of edges from the contour lines is not recommended; the import of surface points only gives good results.

In the case that data contains **more than 1000 points or 100 edges**, the dialog window for the reduction of imported data is displayed. Using reduction, a new number of points or edges is shown according to the entered minimum distance between points. The program averages points which are nearer than input distance. For fluent work, it is recommended to **import maximally 10 000 points**.



Reduction of imported data

When importing edges, it is not possible to reduce the number of points at all.

Visualization of drawing on the desktop can be modified in the "Drawing Settings" frame and with the help of buttons on toolbar "Visualization".

Terrain Edges

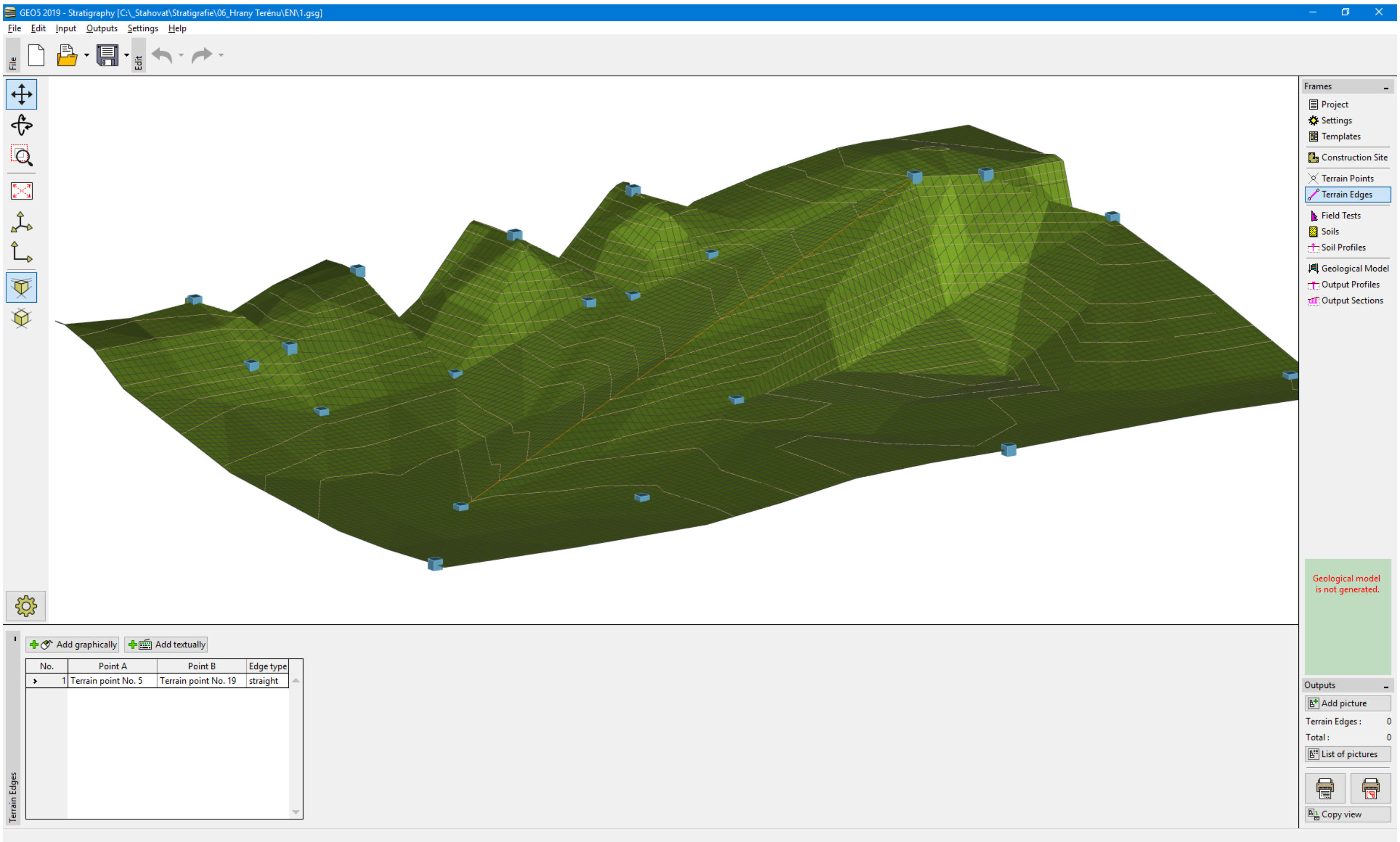
The edges connecting terrain points are entered in the "**Terrain Edges**" frame. The "**New terrain edge**" dialog window serves to add new points.

The edge is defined between terrain points or field tests.

Furthermore, the type of edge is selected:

- **straight** - points are connected by straight line
- **curved** - points are connected by curve (the edge is smoothed according to the terrain)

Just one edge can be defined between two points. The edges cannot intersect.



Frame "Terrain Edges"

The **terrain model is always generated** after any change of an edge.

Visualization of drawing on the desktop can be modified in the "Drawing Settings" frame and with the help of buttons on toolbar "Visualization".

Field Tests (Exploration Points)

The "**Field Tests**" **(Exploration Points)** frame contains a table with all input tests. The **name**, **type**, **coordinates,** and **state of the test** are visible in the table. Next to the table, the selected test is displayed.

This frame serves to input the **raw data of all field tests**, which were carried out on the construction site. It is possible to enter not only the data required to create a geological model, but also optional data for the creation of a geological documentation.

The tests should be entered in this frame, always as uncorrected in the real state (as carried out on the construction site). The modification or interpretation of tests is performed in the "Soil Profiles" frame. Each test automatically creates a soil profile (if it is not disabled using the "**Field test generates a soil profile**" button).

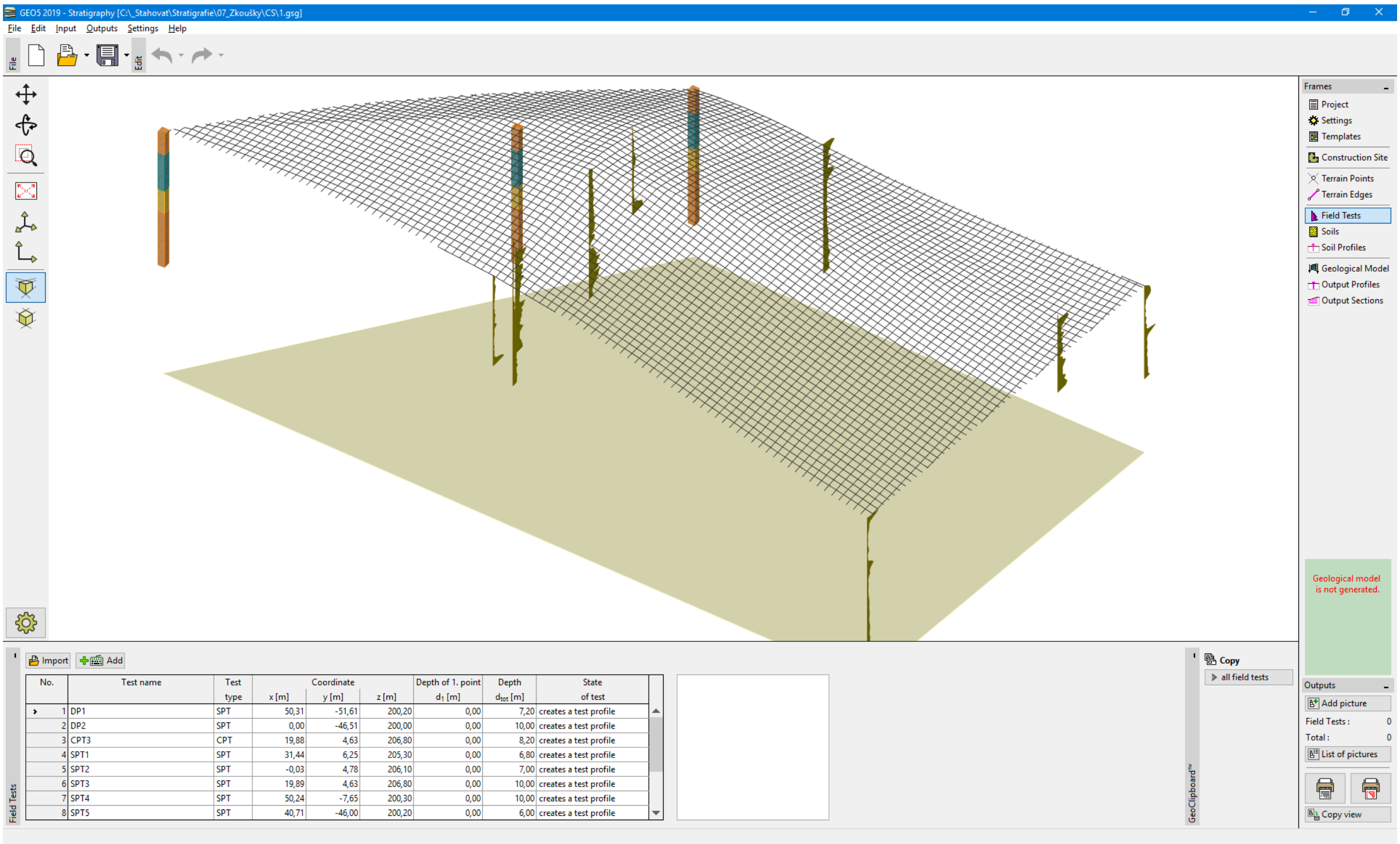
When a large number of field tests are imported, some tests can be located **out of the construction site**. In this state, the **test does not create a soil profile** and is not displayed.

The tests can be added using the "**Add**" button or imported using the "**Import**" button.

The "**Stratigraphy**" program supports these types of field tests:

- Borehole
- Well

- CPT - Cone penetration tests
- DPT - Dynamic penetration test
- SPT - Standard penetration tests
- PMT - Pressuremeter tests
- DMT - Dilatometric tests



Frame "Field Tests"

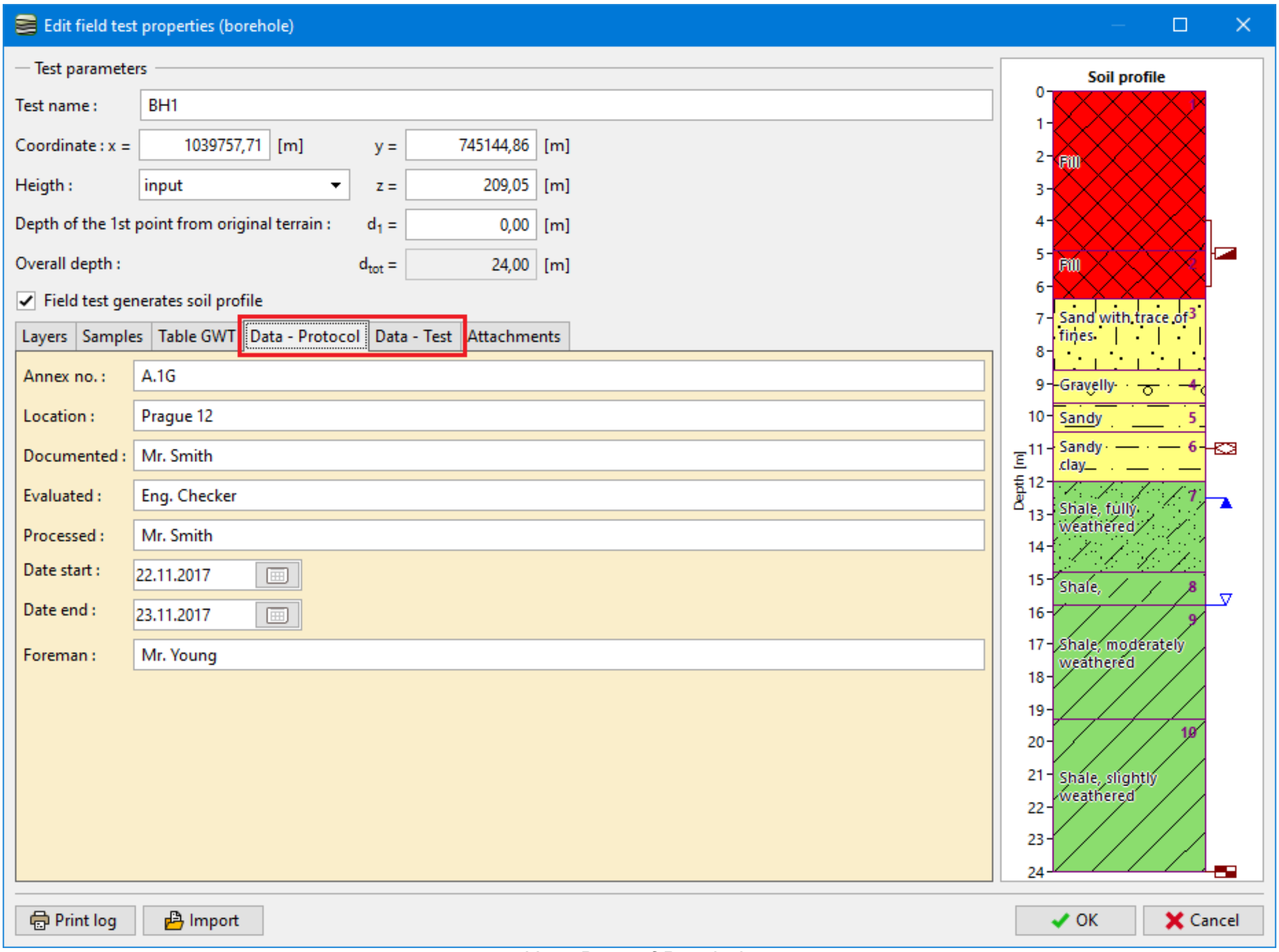
Visualization of the drawing on the desktop can be modified in any input regime based on the setting adjusted in the "Drawing Settings" frame and with the help of buttons on the toolbar "Visualization".

The "Undo" button is an important program tool. It allows us to return back to the **original state before any modification**.

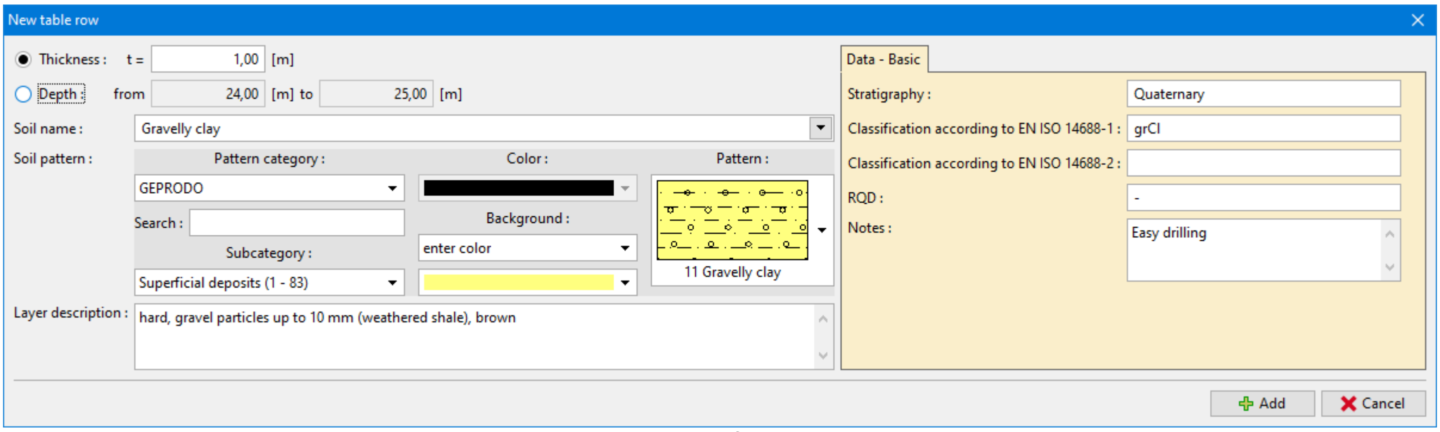
Additional Data of Field Tests

In the dialog windows for field tests input, the **basic data** for creation of a **geological model** are input**.** The users with purchased module "**Logs**" can also enter an arbitrary amount of **user-defined data**.

User-defined data are defined in the "**Templates**" frame - for each test and also for each layer. These data are important for creation of geological documentation. A larger amount of data can be sort it into the groups. Groups and their data are defined in the Edit of template dialog window



User Data of Borehole

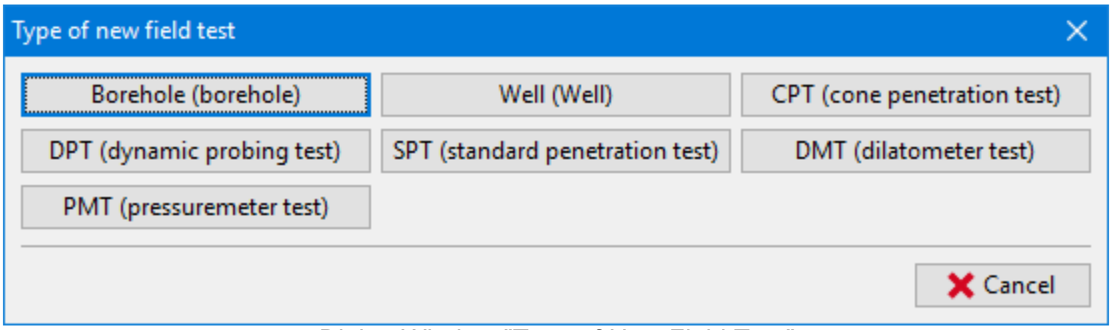


User Data of soil layer

Log style is defined in the "Templates" frame. The data are stored in the global database - it means that data are preserved when changing the template, only some of them are not displayed. Imported or entered data cannot be lost after the template switch.

Import of Tests

The "**Import**" button opens a dialog window "**Type of new field test**". In this dialog window, the type of the field test for import must be specified.



Dialog Window "Type of New Field Test"

The "**Stratigraphy**" program supports the import of these types of field tests:

Borehole

*.txt, *.xlsx, *.csv, *.ods - a general text or table format

.son,.so2 - a data format used in Czech and Slovak Republic, originally from the GeProDo software
*.accdb, *.mdb - a data of SEP3 format used in Germany

Well (Well)

*.txt, *.xlsx, *.csv, *.ods - a general text or table format

.hyd,.hyv - a data format used in Czech and Slovak Republic, originally from the GeProDo software
*.accdb, *.mdb - a data of SEP3 format used in Germany

CPT (Cone Penetration Test)

*.txt, *.xlsx, *.csv, *.ods - a general text or table format

*.spe - a data format used in Czech and Slovak Republic, originally from the GeProDo software
*.cpt - a text file standard particularly for Netherlands (used e.g., in programs Geodelft M-Serie), which serves to input elevations of individual points and values of penetration resistance (may contain more CPTs)
*.cpt - a text file Geotech AB CPT
*.cpt - a text file Gouda Geo CPT
*.cpt - a text file Hogentogler CPT
*.gef - GEF (Geotechnical Exchange Format) is a general language structure for storing and transferring of geotechnical information.
Detailed information available online:
https://publicwiki.deltares.nl/display/STREAM/GEF-CPT
*.ags a format used for transferring of geotechnical information in Great Britain.
Detailed information available online:
http://www.agsdataformat.com/datatransferv4/intro.php

DPT (Dynamic Probing Test)

*.txt, *.xlsx, *.csv, *.ods - a general text or table format

*.pen - a data format used in Czech and Slovak Republic, originally from the GeProDo software

SPT (Standard Penetration Test)

*.txt, *.xlsx, *.csv, *.ods - a general text or table format

DMT (Dilatometric Test)

*.txt, *.xlsx, *.csv, *.ods - a general text or table format

*.uni - a standardized and universal format for import of the measured data obtained from dilatometric tests

PMT (Pressumetric Test)

*.txt, *.xlsx, *.csv, *.ods - a general text or table format

A general text or table format import allows for selecting a particular **system of units** to store data of the test. When importing the program automatically converts the adopted system of units to the one used in the program.

Providing you use a certain standard of a CPT text file not supported by the program, feel free to contact us at **hotline@fine.cz** - it will be introduced into the forthcoming version.

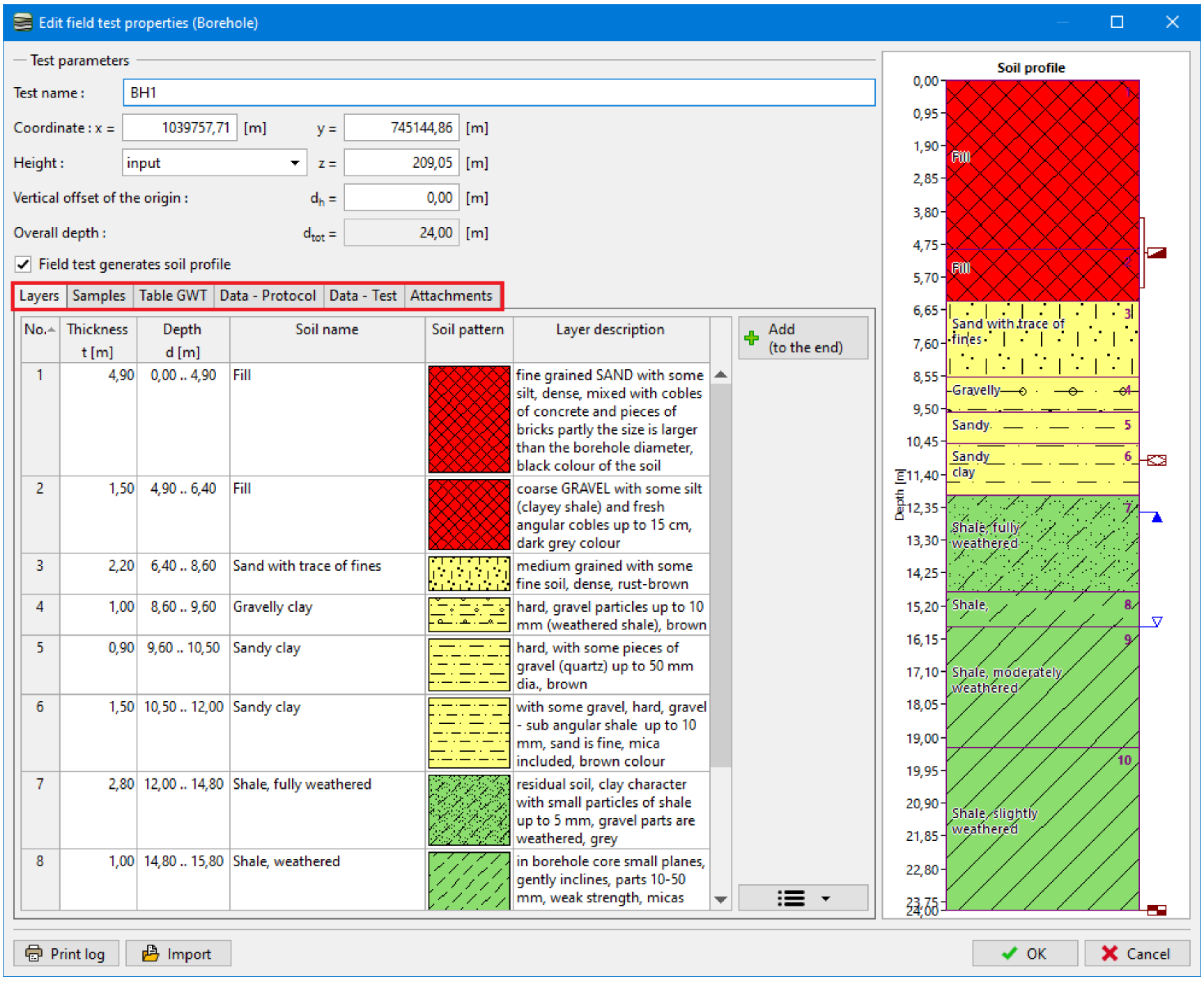
Borehole

The name and the vertical offset of the origin of the borehole is entered in the "**New field test**" dialog window. Input of coordinates is also required.

The height of the first point (z-coordinate) can be calculated automatically, if the test was carried out on the terrain level (or it can be entered manually).

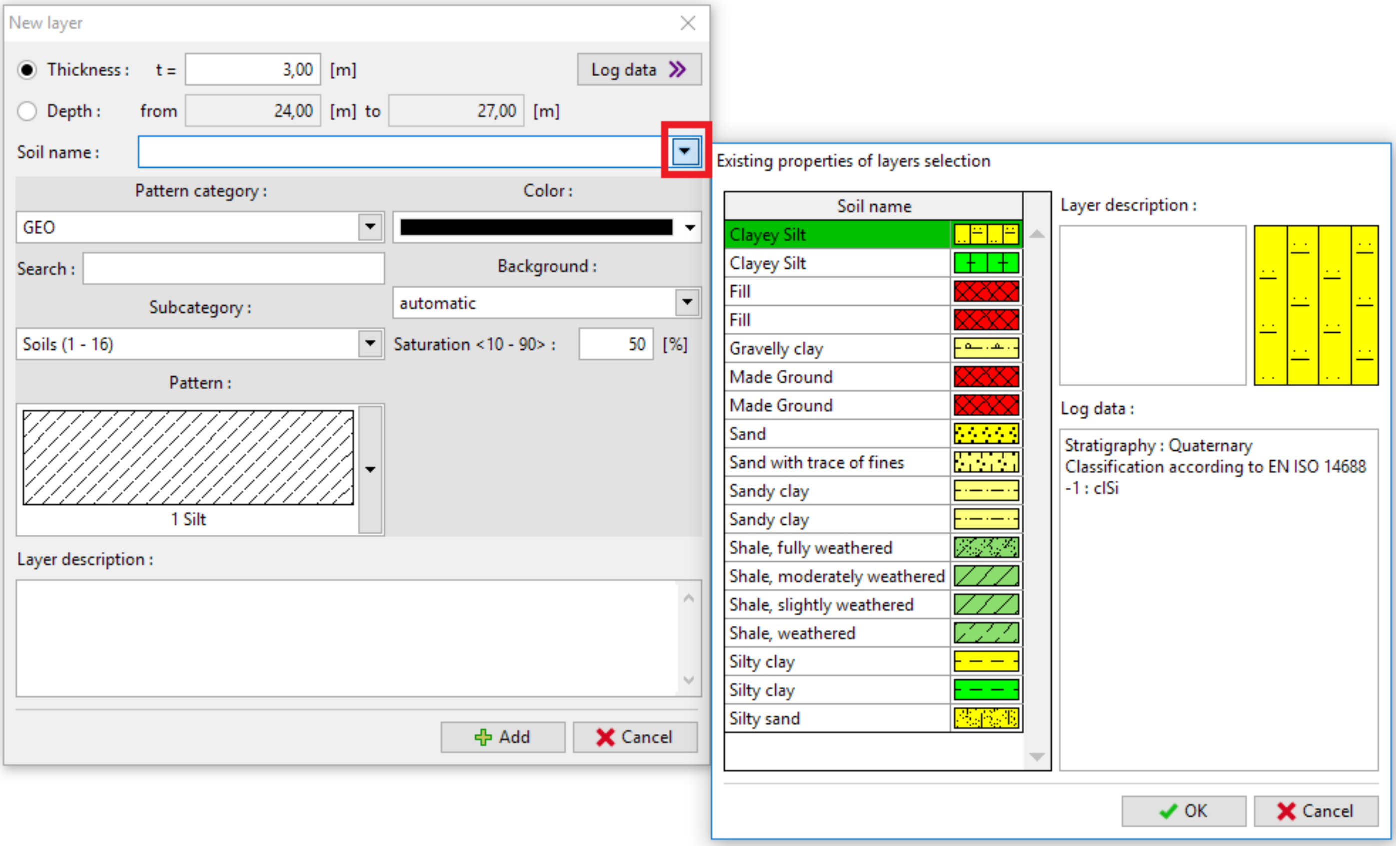
Furthermore, it is possible to enter in tab sheets:

- **Soil layers**
- **Samples**
- **Groundwater tables**
- Attachments - photos and other documents



Dialog Window "New Field Test"

The "**Add**" button opens a new dialog window "**New layers**". In this dialog window, the thickness (or depth) of the layer is entered and soil (color, pattern) is assigned to this layer. It is also possible to assign another soil from already entered soils.



Dialog Window "New Soils"

Well

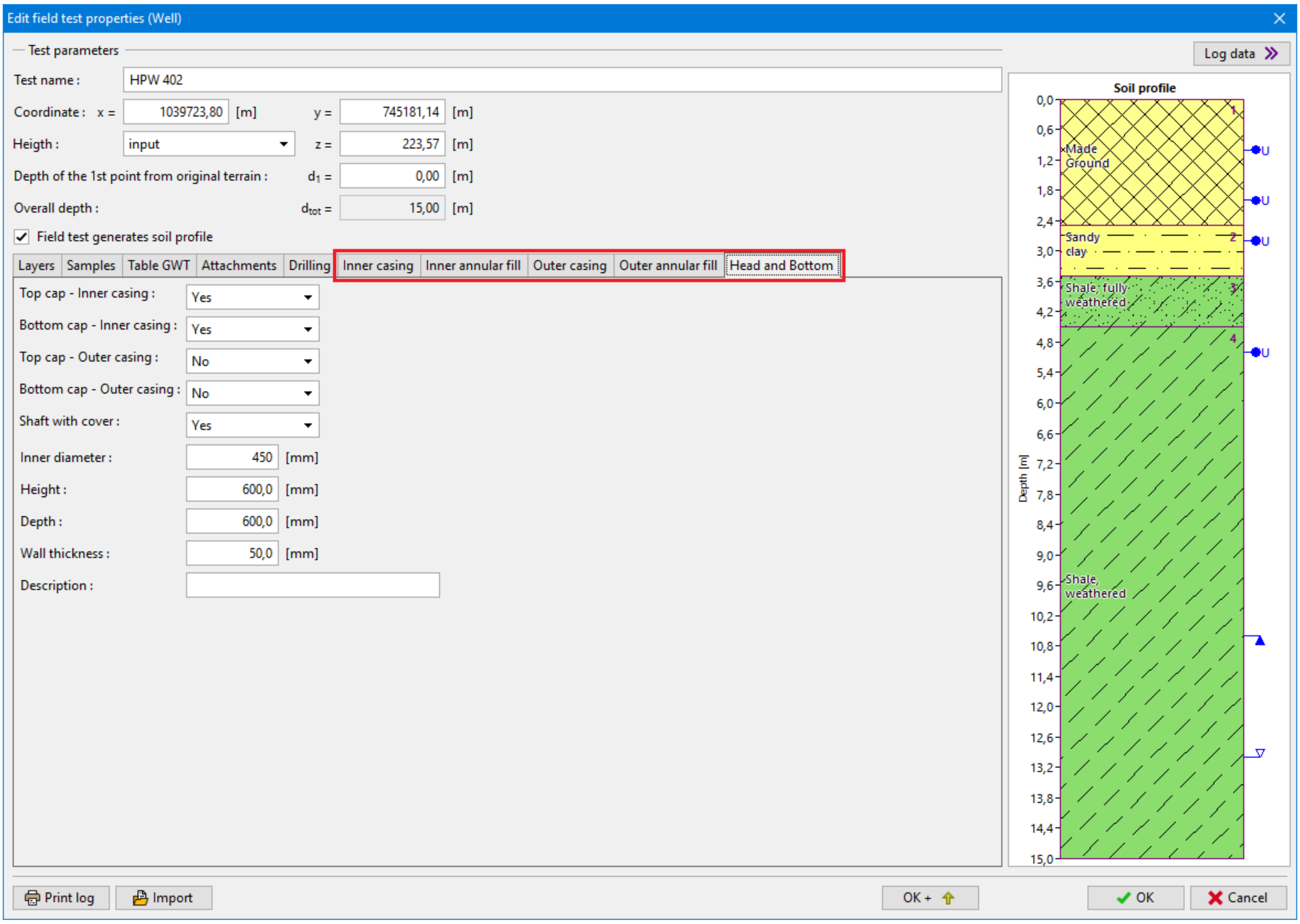
The **Well** is an extended version of the "Borehole" field test. The basic work and definition are similar to adding a "**Borehole**".

Additionally, there is a possibility to define additional parameters:

- Head and Bottom
- Casing
- Annular fill

Two separate rows of casing (inner and outer) and two separate layers of annular fill (inner and outer) can be defined in the "**Well**" field test type.

The top cap, bottom cap, and sump with cover are entered in the tab "**Head and Bottom**".

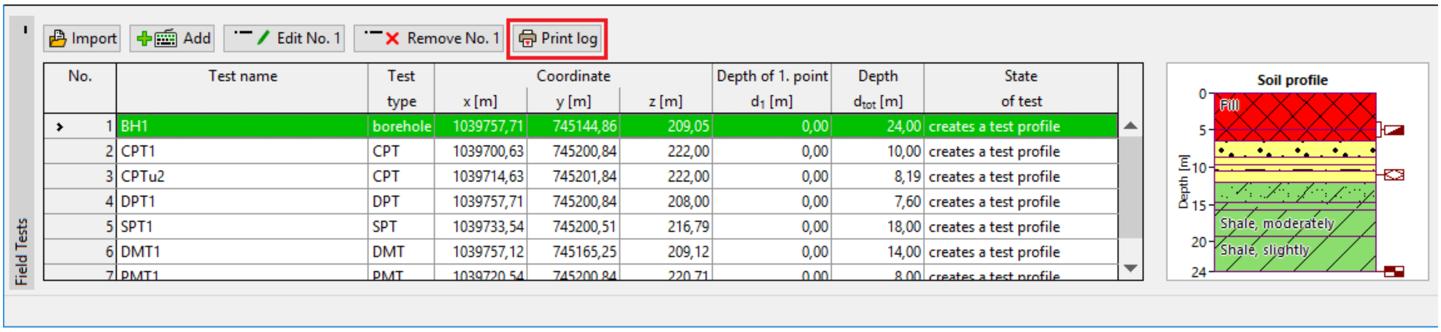


"Head and Bottom" Tab

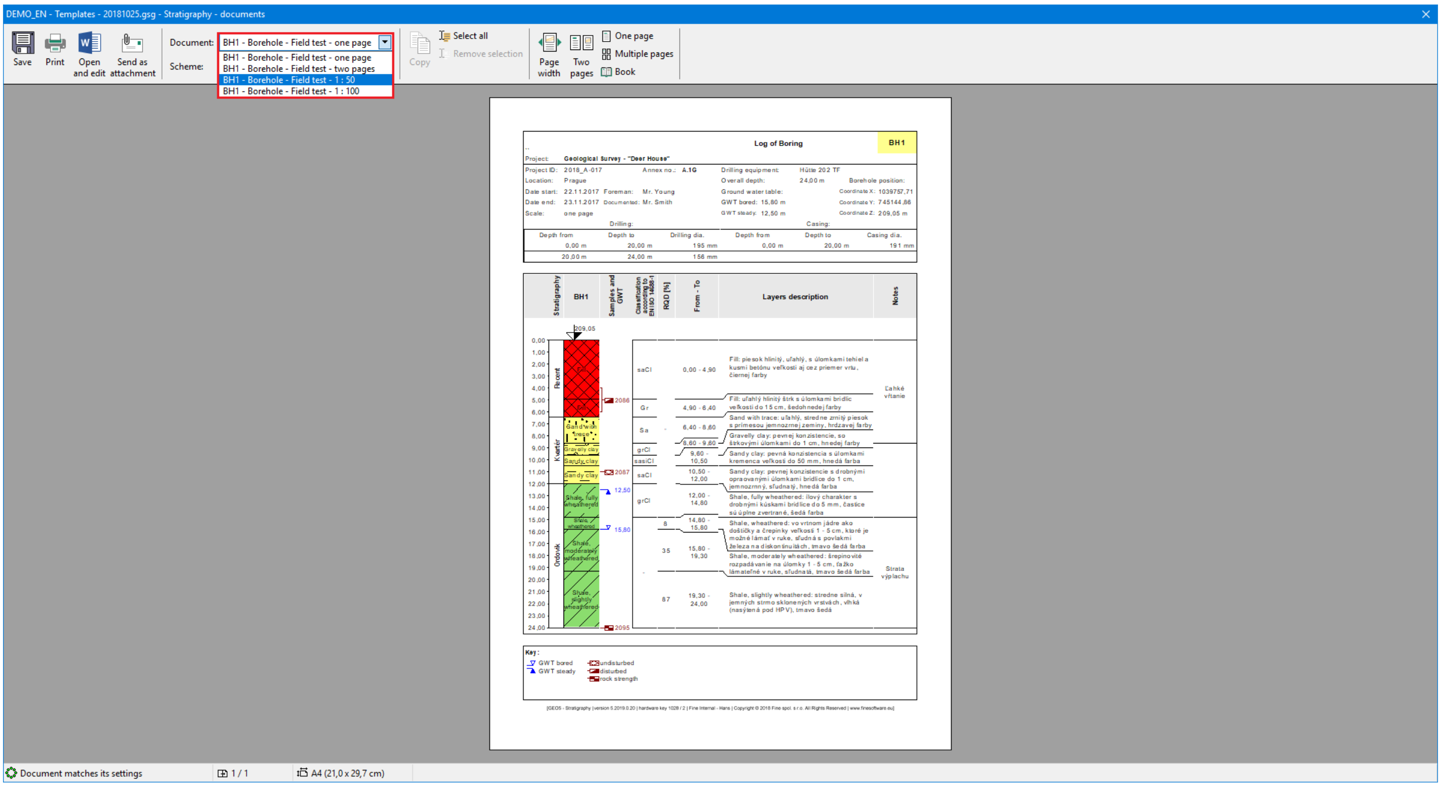
Geological Documentation

The program allows us to print a geological documentation for **field tests** and **soil profiles**.

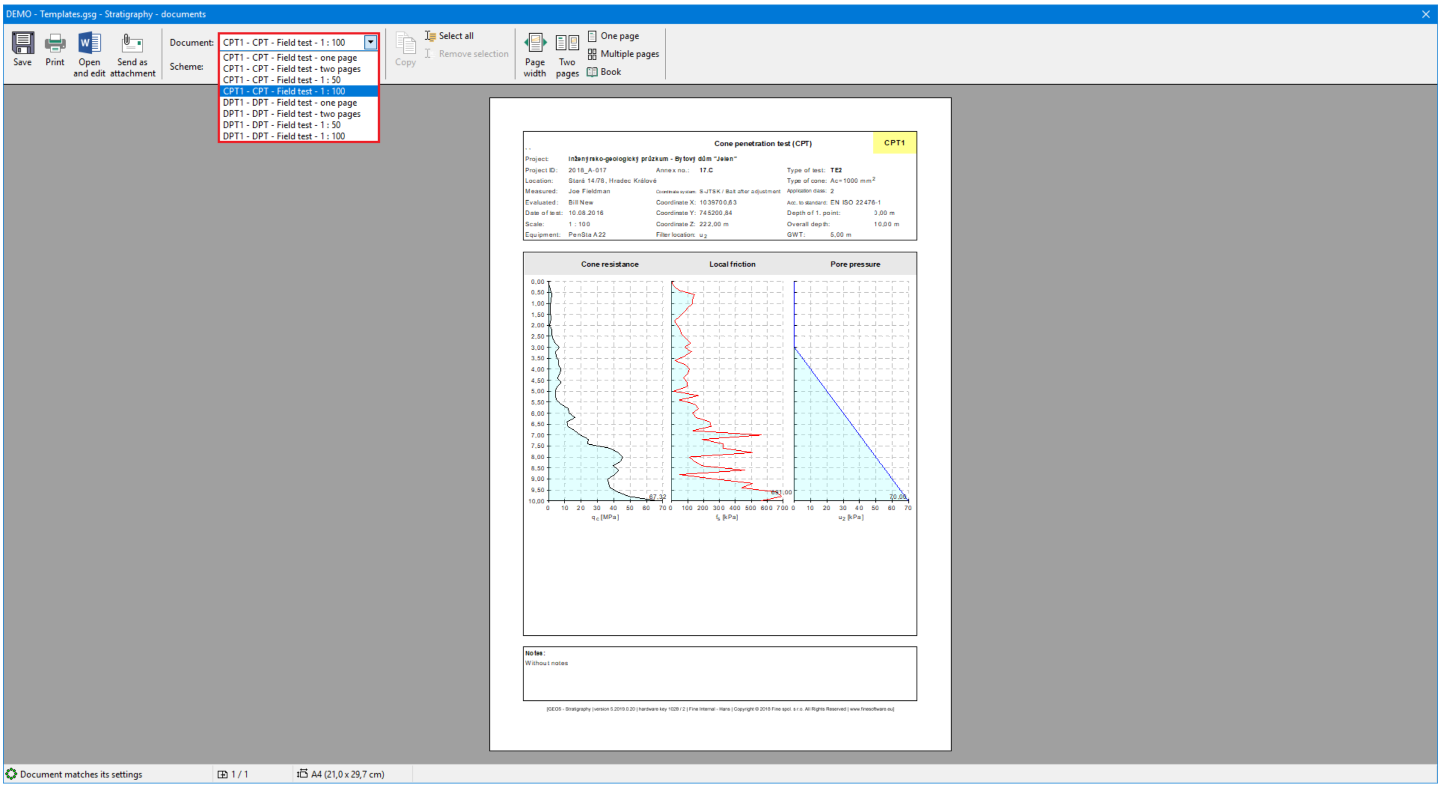
After selecting one or more tests, the "**Print log**" button is shown on the toolbar.



Using this button, the dialog window for print is opened. Here it is possible to select a template for output protocol (This function is available just for users with purchased module "**Logs**" - if this module is not installed, only one default template is available for each type of field test).



When selecting more then one test, it is possible to print more tests of identical type together - we can see three different CPTs in the picture in scale 1:100.

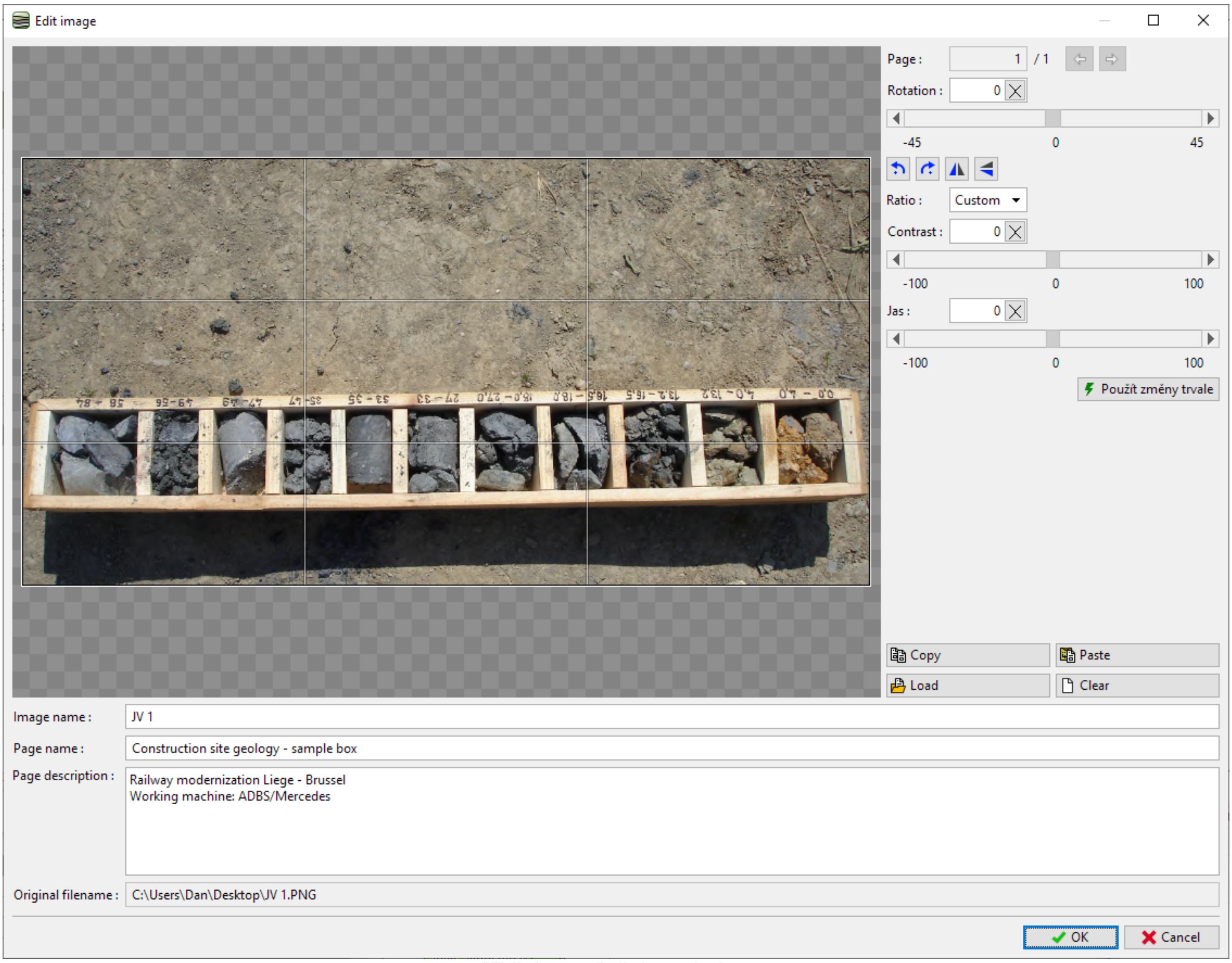


Attachments - Photos and Documents

The program allows us to load attachments in **JPG, JPEG, TIFF, BMP, PNG** formats, and **PDF** documents to the table.

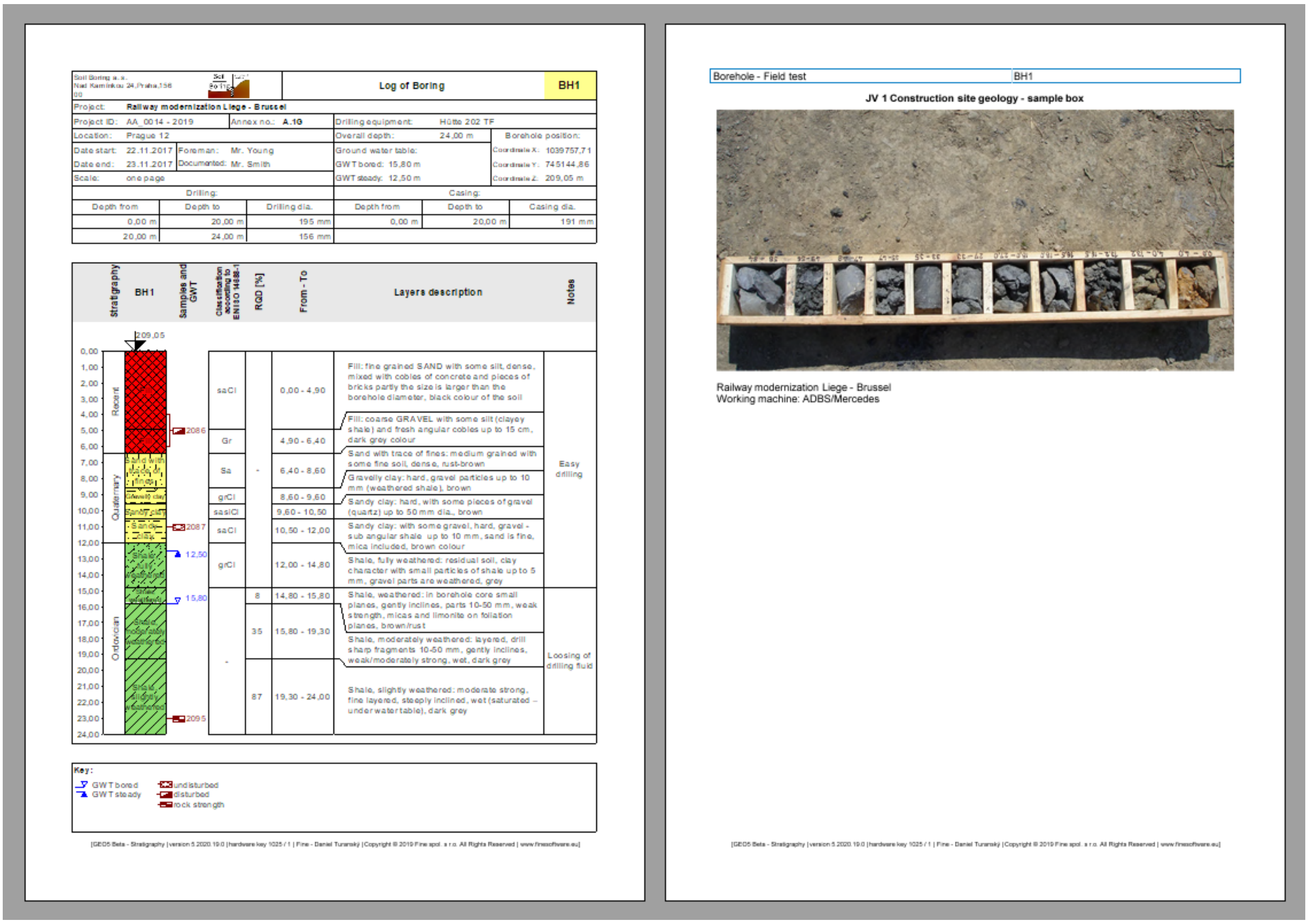
Loaded photos and documents become a **part of the GEO5 data.**

The attachments can be edited in the "**Edit image**" dialog window where it is possible to **add a name and a description of the attachment** and **edit it** - cut, rotate, change a contrast or a brightness etc.



"Edit image" dialog window

Loaded attachments are printed automatically on the next pages of the **field test log**. The image and page names are printed above the image, the page description is printed below the image. The **head,** with the name of the field test, **is generated automatically**.



Log with an attachment

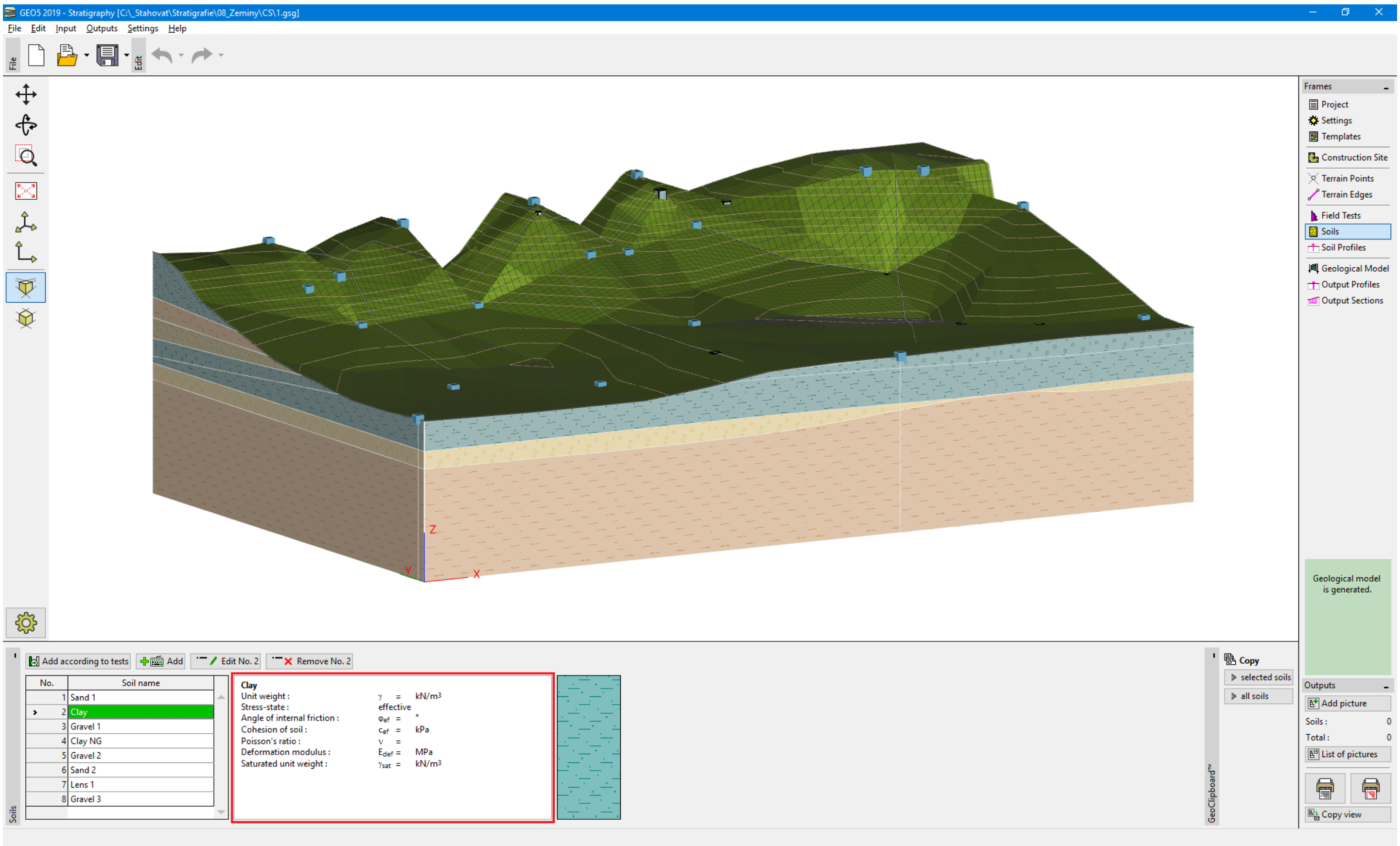
Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding a soil is performed in the "Add new soils" dialog window. It is also possible to add all soils already entered in the frame "Field Tests" using the "**Adopt from field tests**" button.

The "**Stratigraphy**" program **does not need any soil parameters** for the creation of the geological model, but these parameters can be entered for **export to other GEO5 programs**.

Data of input soils can be copied within all GEO5 programs using "GeoClipboard", or it is possible to use an export in the "Output Profiles" or "Output Sections" frames.



Frame "Soils"

Soil Profiles

The "**Soil profiles**" frame contains a table with all input soil profiles. The **name and coordinates of the original field test** and **type of the soil profile** are visible in the table. Next to the table, the selected soil profile is displayed.

Soil profiles are created automatically from the field tests. The name of the test and its coordinates cannot be entered or edited - they always correspond with the original field test.

The soil profile is an interpretation of the field test into a **geological profile**.

Soil profiles are further used for the creation of boreholes. The geological model is created from the boreholes. Each correctly entered soil profile creates a borehole (if it is not disabled using "**Profile is active for geological model generation**" button).

If one or more soil profiles are selected, the button "**Print log**" is shown on the toolbar. Using this button, we can print soil profile together with field test as a part of geological documentation.

If the soil profile **is not correctly entered**, it is displayed in red in the table with a "**not defined**" description. In this case, the borehole is not created from the test.

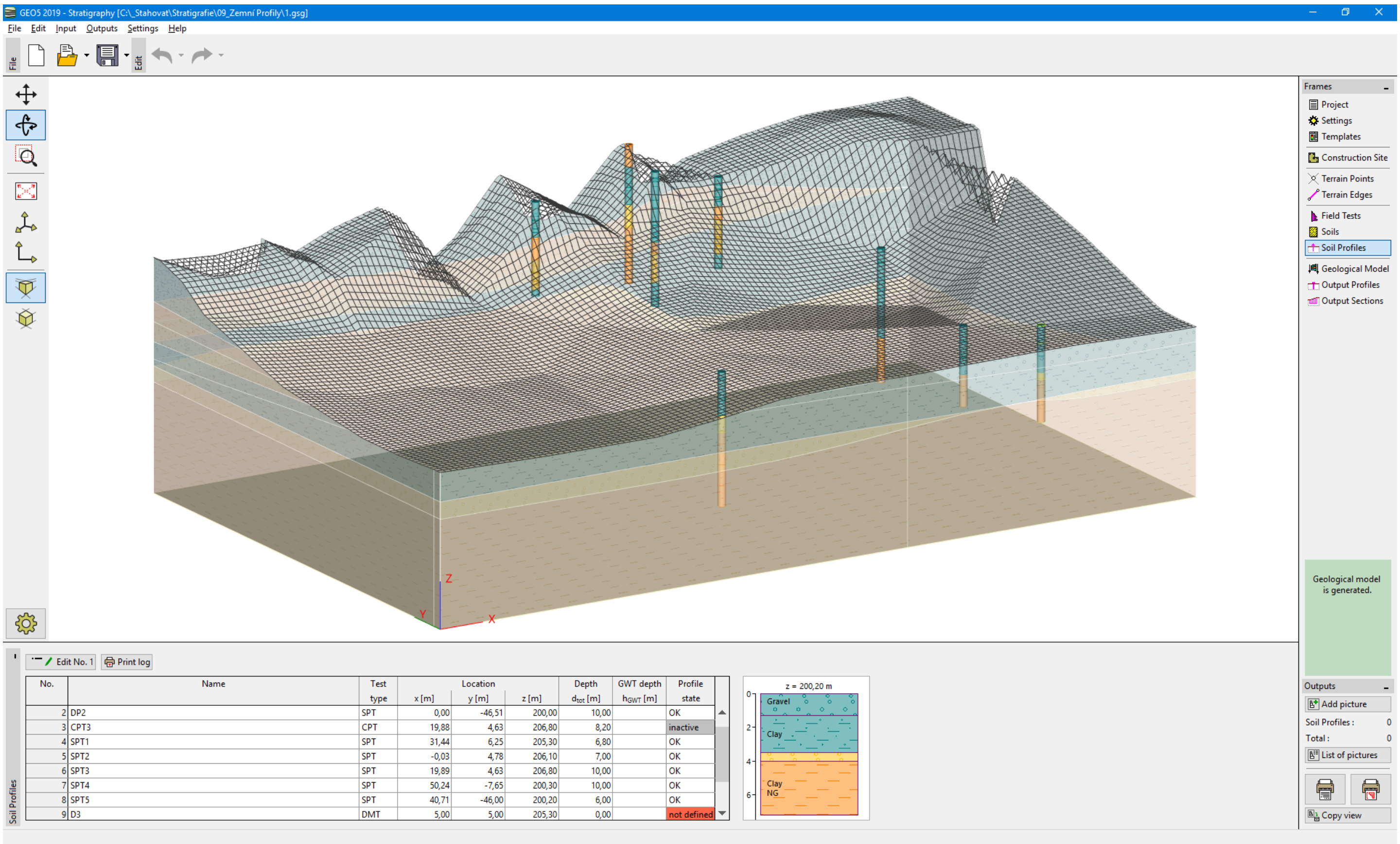
If the soil profile is located **out of the construction site**, the soil profile is not visible, and it does not create a borehole.

Creation of soil profile depends on the type of field test:

- Creation of soil profile from borehole
- Creation of soil profile from CPT
- Creation of soil profile from DPT, SPT, PMT, DMT

Interpretation of field test into a soil profile can be carried out when modeling geological sections.

During this process, the soil profiles can be directly created or edited.



Frame "Soil Profiles"

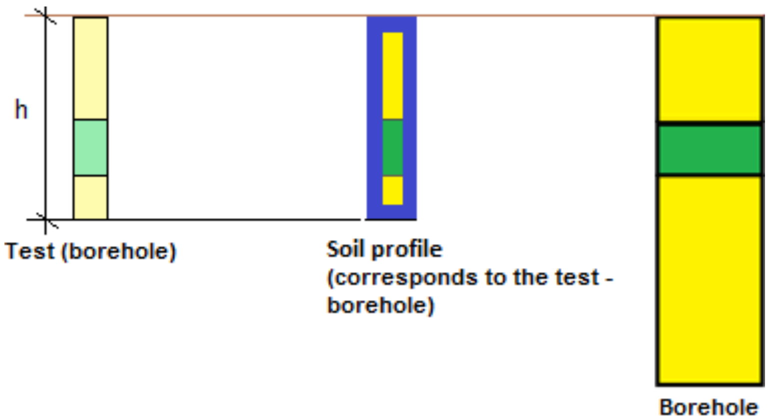
Visualization of drawing on the desktop can be modified in the "Drawing Settings" frame and with the help of buttons on the toolbar "Visualization".

The "Undo" button is an important program tool. It allows us to return back to the **original state before any modification**.

Relation between Field Test, Soil Profile and Borehole

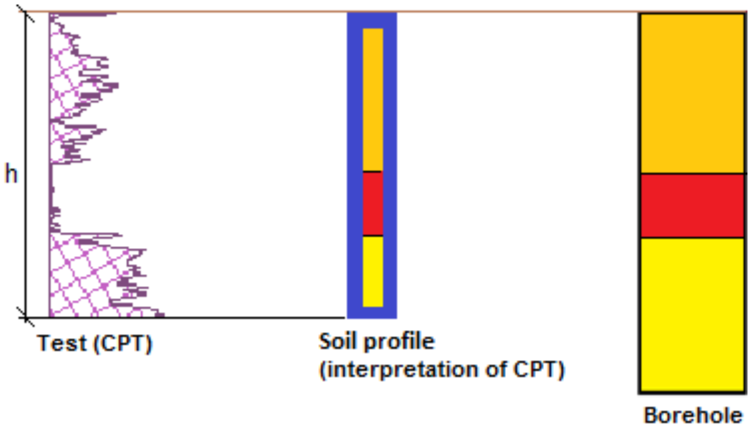
The **soil profile is created from the test** (exploration point), and the **borehole is created from the soil profile**. It is not possible to edit coordinates of the soil profile and borehole - they always correspond to the coordinates of the test.

If the type of the **test is a borehole**, the Soil profile is created automatically, however, for the creation of a 3D model, it is usually necessary to simplify it.



Relationship between the Soil Profile and Test (Borehole)

If the type of **test is not a borehole**, it is necessary to transfer this test into the soil profile (thickness of layers and assigned soils).



Relationship between Soil Profile and Test (Borehole)

Interpretation of field test into a soil profile can be carried out when modeling geological sections.

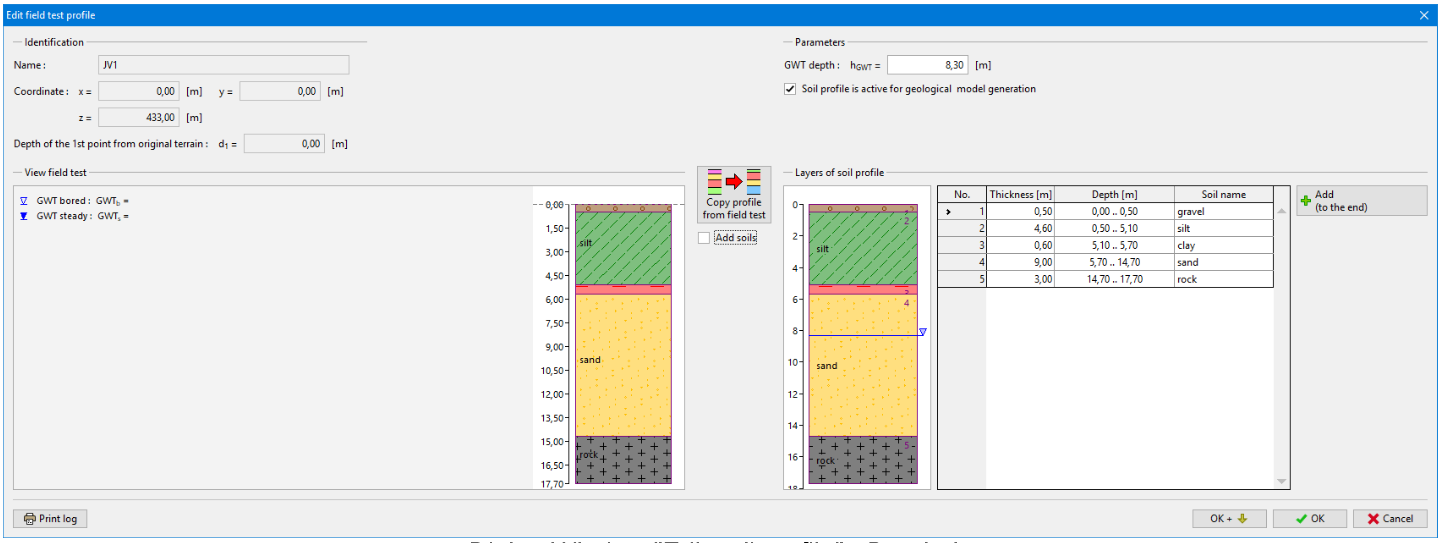
During this process, the soil profiles can be directly created or edited.

If the data of field tests are changed, the soil profile is also changed automatically. If the soil profile is changed, the borehole is also changed, and it leads to the change of the entire geological model.

Creation of Soil Profile from a Borehole

If the soil profile is created from the borehole, the "**Copy profile from field test**" button copies all layers as they were entered in the "Field Tests" frame. By "**Add soils**" checkbox, all types of soils are also copied from the field test.

The real boreholes contain a large number of soil layers; however, a geotechnical engineer works with the **simplified model**. The **geotechnical types** are used for this simplification. In the Stratigraphy program, the boreholes are entered exactly (with all soil layers), the simplification is performed when **soil profiles are created** - these profiles represent the geotechnical model. It is advantageous to create and edit soil profiles when geological sections are modeled.



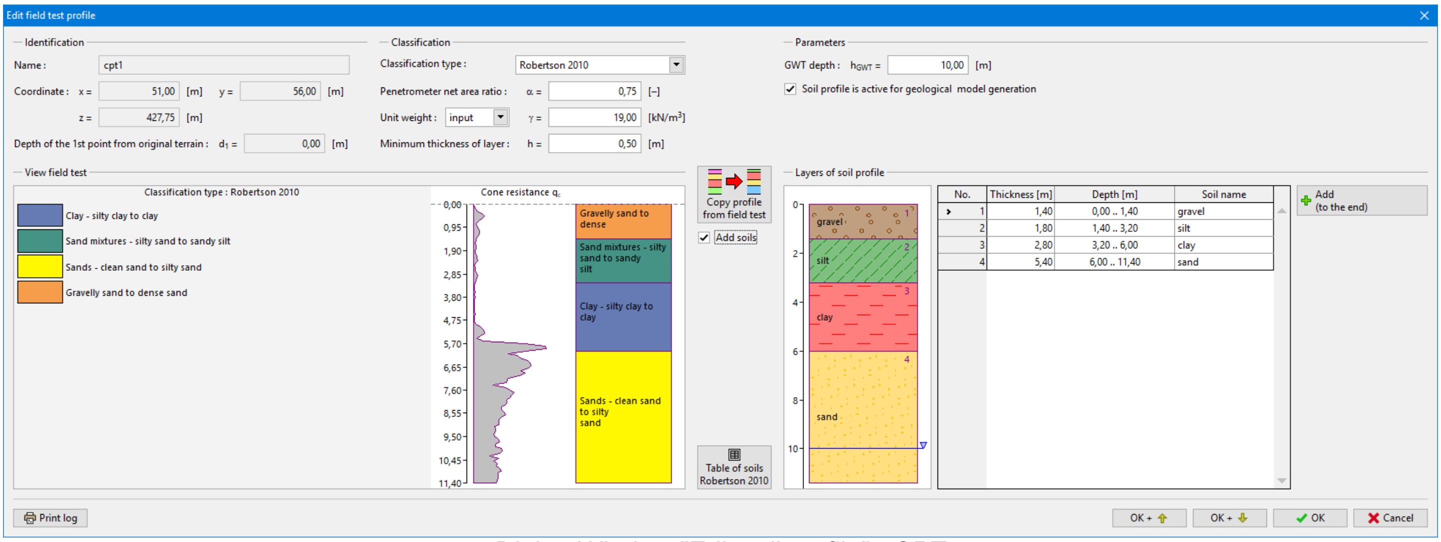
Dialog Window "Edit soil profile" - Borehole

An originally entered field test (borehole) is displayed in the left part of the dialog window. In the right part, the soil profile is displayed. The thickness of layers and assigned types of soils can be modified.

Creation of Soil Profile from a CPT

If the soil profile is created from the CPT, the program allows **two basic ways** according to **classification type**:

- **"Robertson 2010", "Robertson 1986"** - soil profile creation using the soil classification
- **Do not classify** - in this case, the soil profile is created the same way as for DPT, SPT, PMT

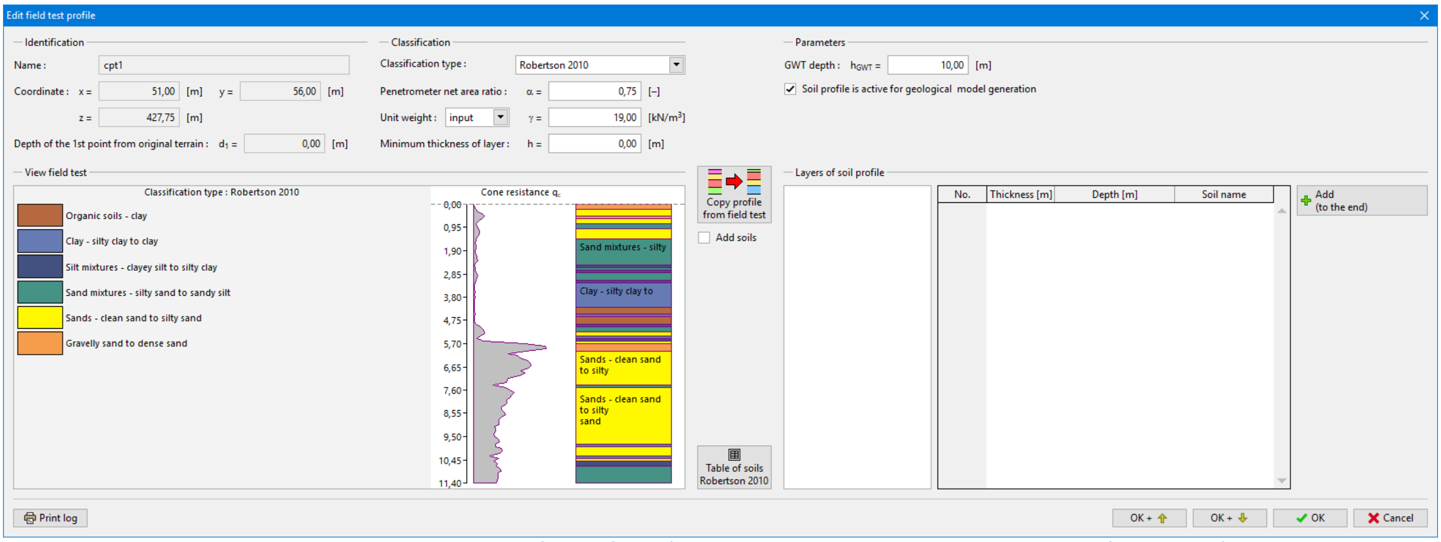


Dialog Window "Edit soil profile" - CPT

Creation of Soil Profile using Classification of Soils

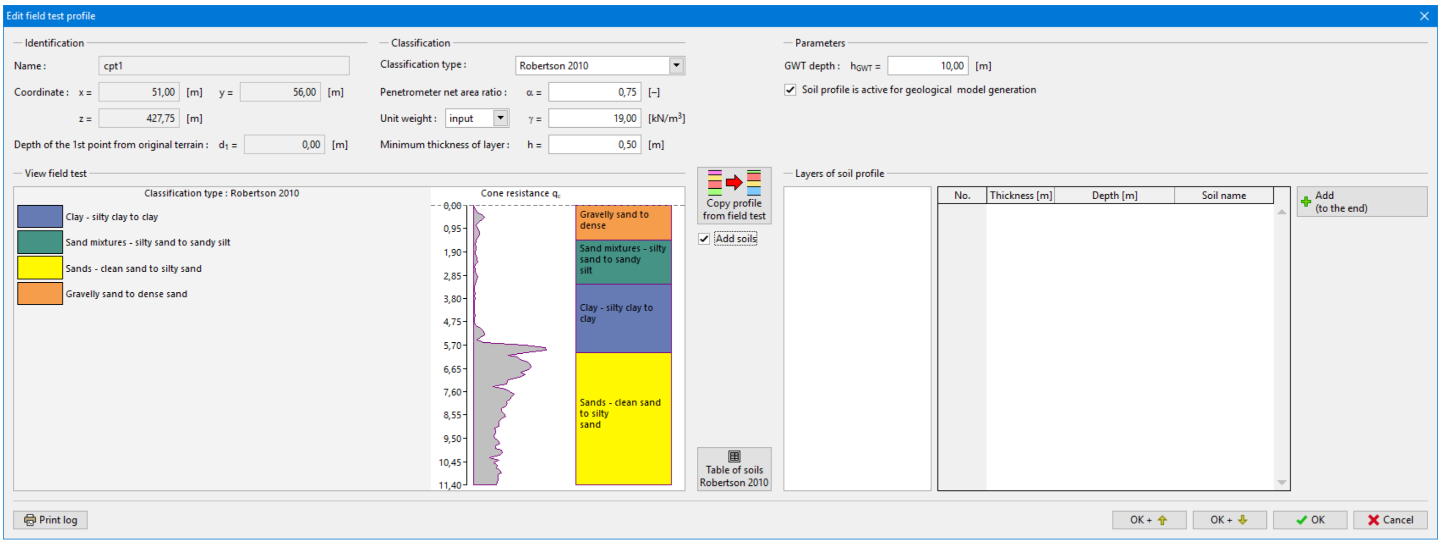
In the "**Classification**" part, the type of classification (Robertson 2010, Robertson 1986) and other parameters (Coefficient of Penetrometer α , determination of unit weight γ) must be set.

Usually, a large number of thin layers are a result of classification.



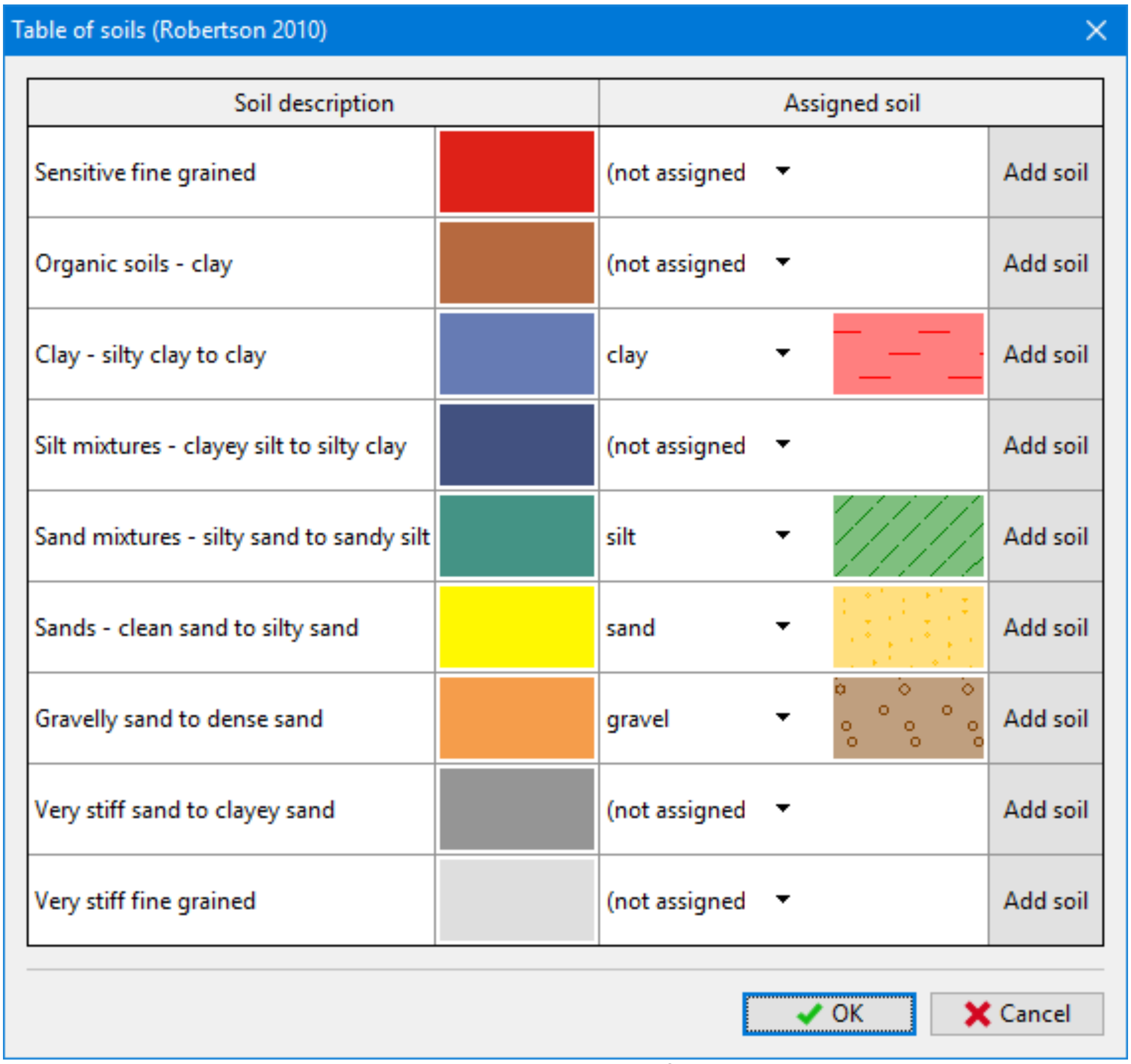
Dialog window "Edit soil profile" - CPT (without using a minimum thickness of the layer)

For this reason, it is appropriate to enter a minimum thickness of the layer h . This filter reduces the number of layers and the geological profile is clearer.



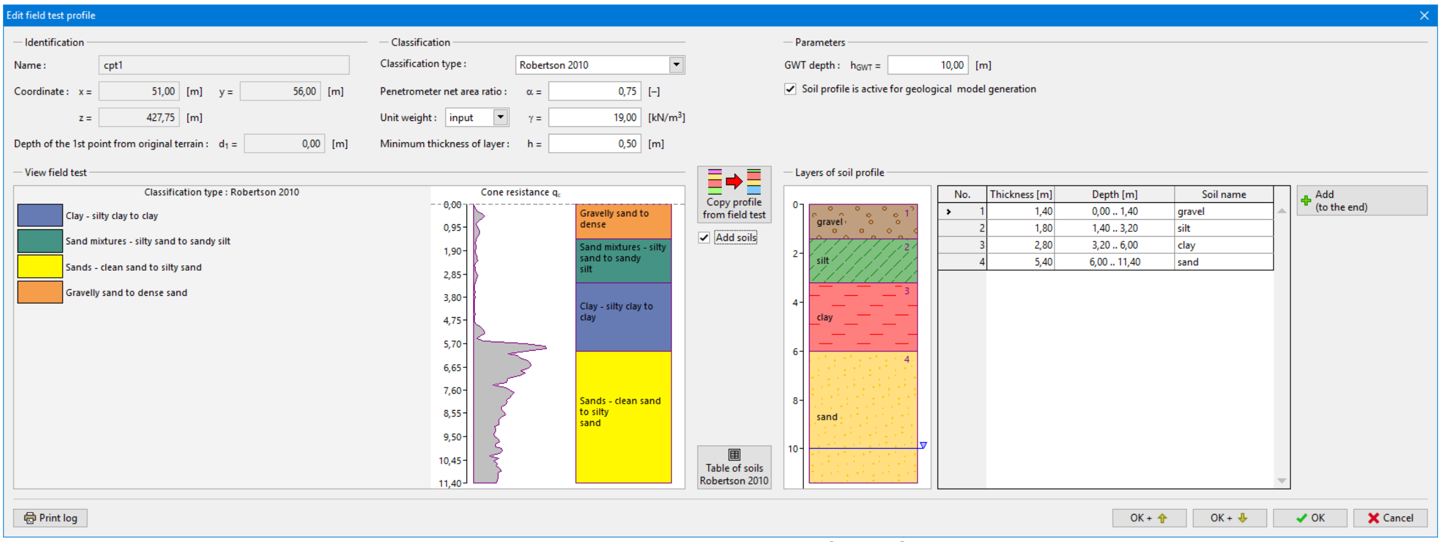
Dialog window "Edit soil profile" - CPT (with using a minimum thickness of the layer)

The "**Table of soils**" button opens a new dialog window "**Table of soils Robertson (1986** or **2010)**". It is possible to assign the entered into the descriptions of soils according to the Robertson classification. If the soil was not already entered in the "Soils" frame, it is possible to add it here using an "**Add soil**" button. It is not necessary to assign a soil into the descriptions which were not classified.



Dialog window "Table of soils"

The "**Copy profile from field test**" button creates a soil profile with the entered soils corresponding to the layers from the CPT classification.



Dialog window "Edit soil profile" - CPT

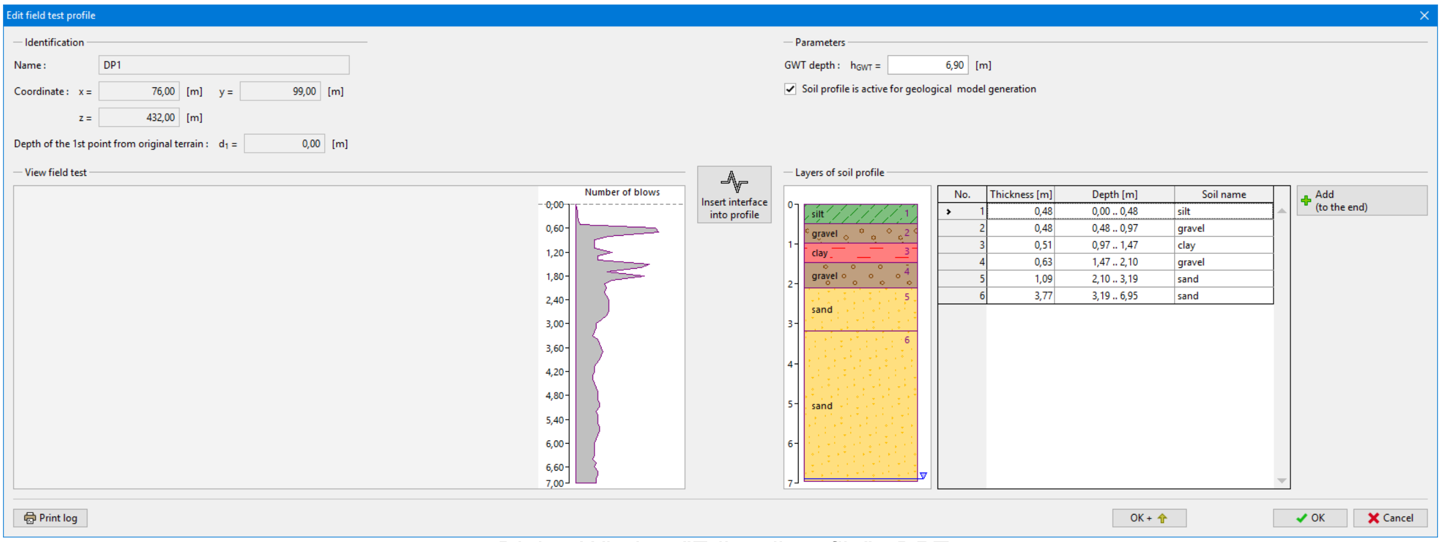
The created profile can be edited using buttons in the right part of the dialog window.

Creation of Soil Profile from an SPT, DPT or PMT

If the soil profile is created from DPT, SPT, and PMT, it is necessary to create a soil profile manually.

The measured field test (here DPT) is displayed in the left part of the dialog window. By the left mouse button, it is possible to create an interface of layers in the geological profile. These layers are displayed in the right part of the dialog window.

It is possible to enter the depth of interfaces d manually using an "**Insert interface into profile**" button. After the interface input, it is necessary to assign a soil to all layers in the table in the right part of the dialog window. If the soil was not already entered in the "Soils" frame, it is possible to add it here using the "**Add soil**" button.



Dialog Window "Edit soil profile" - DPT

Interface

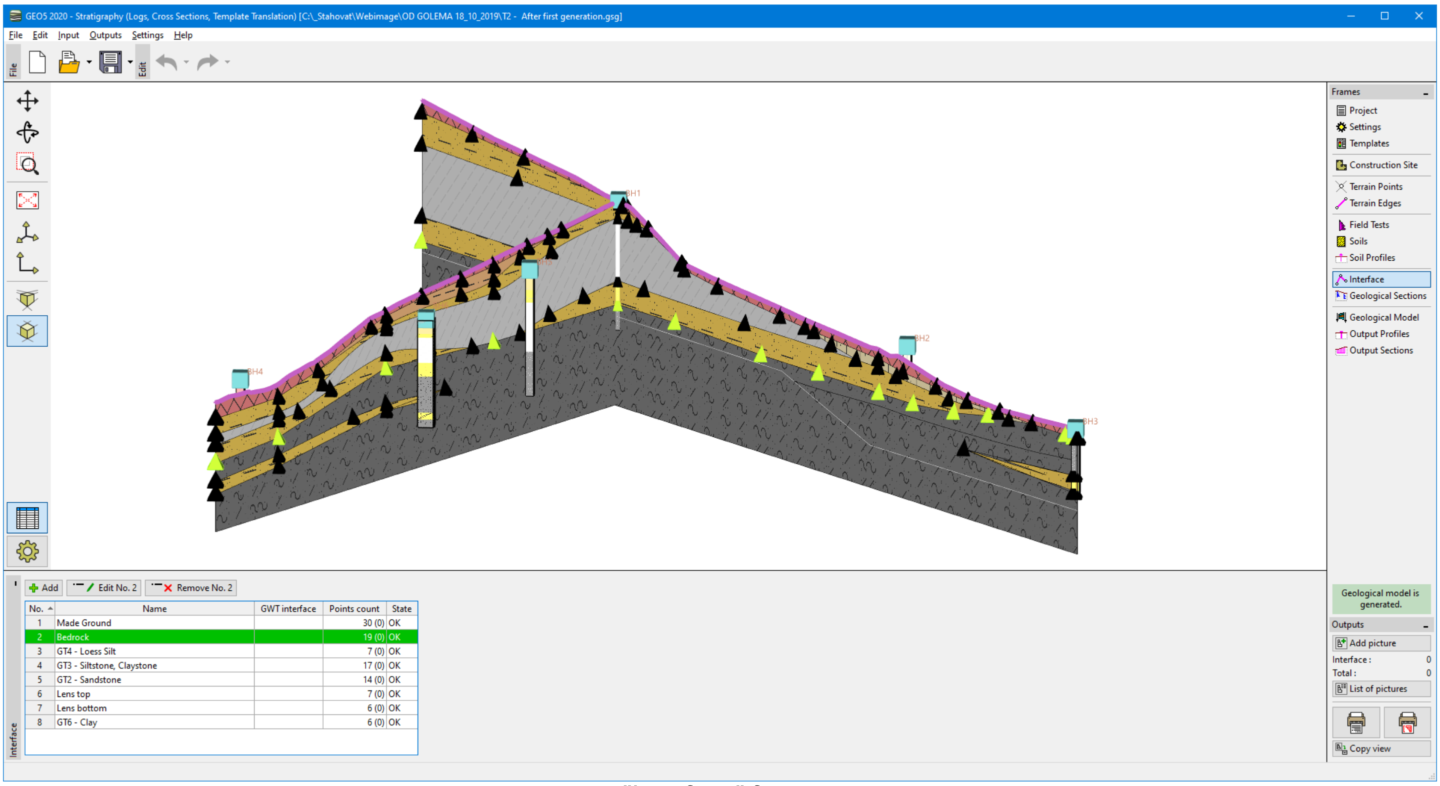
The "**Interface**" frame contains a table with a list of **defined interfaces**, which can be added or deleted. The interface is assigned to the individual layers (in the table of layers) when a geological model is created.

The **interfaces** are **created by points**. These points can be **added** to the interface in **several ways**:

- Text input of points coordinates
- Import of points (text table, XLSX and ODS table, DXF import, LandXML import)
- Automatic generation of interfaces according to the entered geological sections

The generation of interface points from entered geological sections is a base of a 3D model creation. Individual interfaces can be created or assigned directly in the dialog window for the creation of a geological section. Interface points generated from the geological section cannot be edited or deleted - they are always in full concordance with geological sections.

The "Fault" is a **special type of interface**. This type is defined and edited only **when creating a geological section** and cannot be edited here. The fault is always perpendicular to the geological section where it was created.

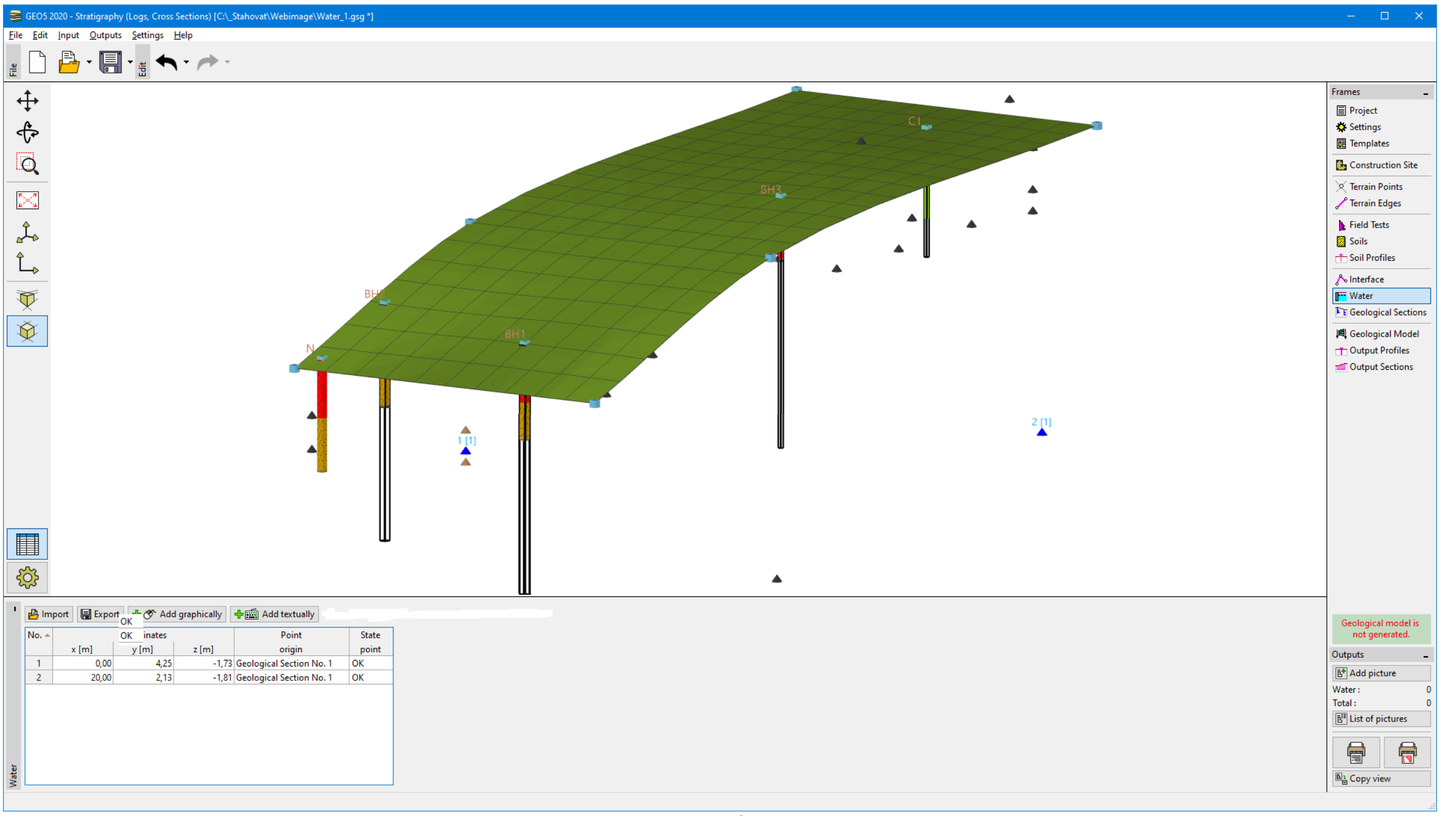


"Interface" frame

Water

The "Water" frame contains a table with a list of defined interface points.

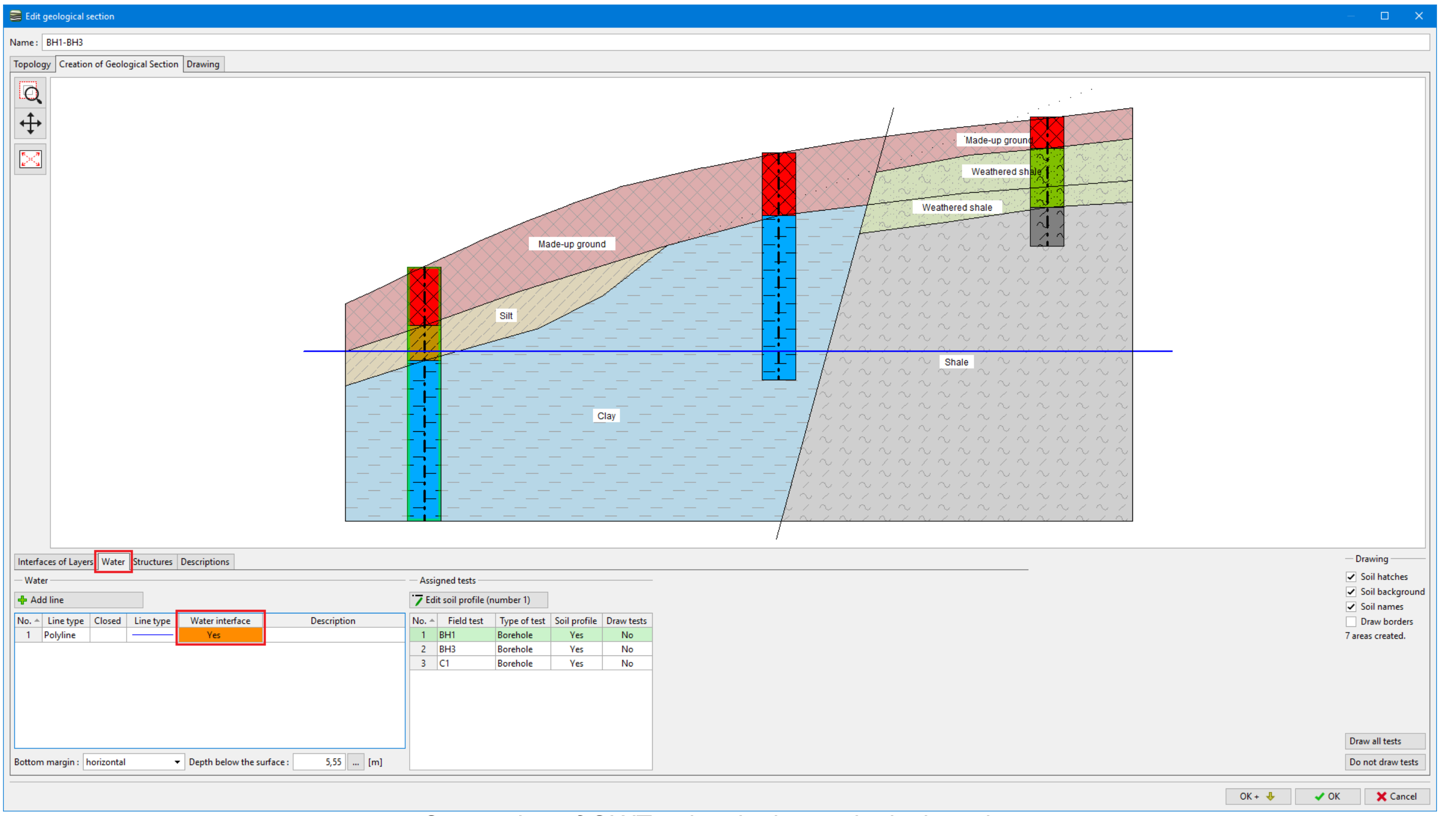
The groundwater table is generated from GWT in **active boreholes** and **GWTinterface points from this frame.**



"Water" frame

Interface points can be **defined in several ways**:

- Text input of points coordinates
- Import (text table, XLSX and ODS table, DXF, LandXML)
- Automatic generation of the interface from Geological Sections



Generation of GWT points in the geological section

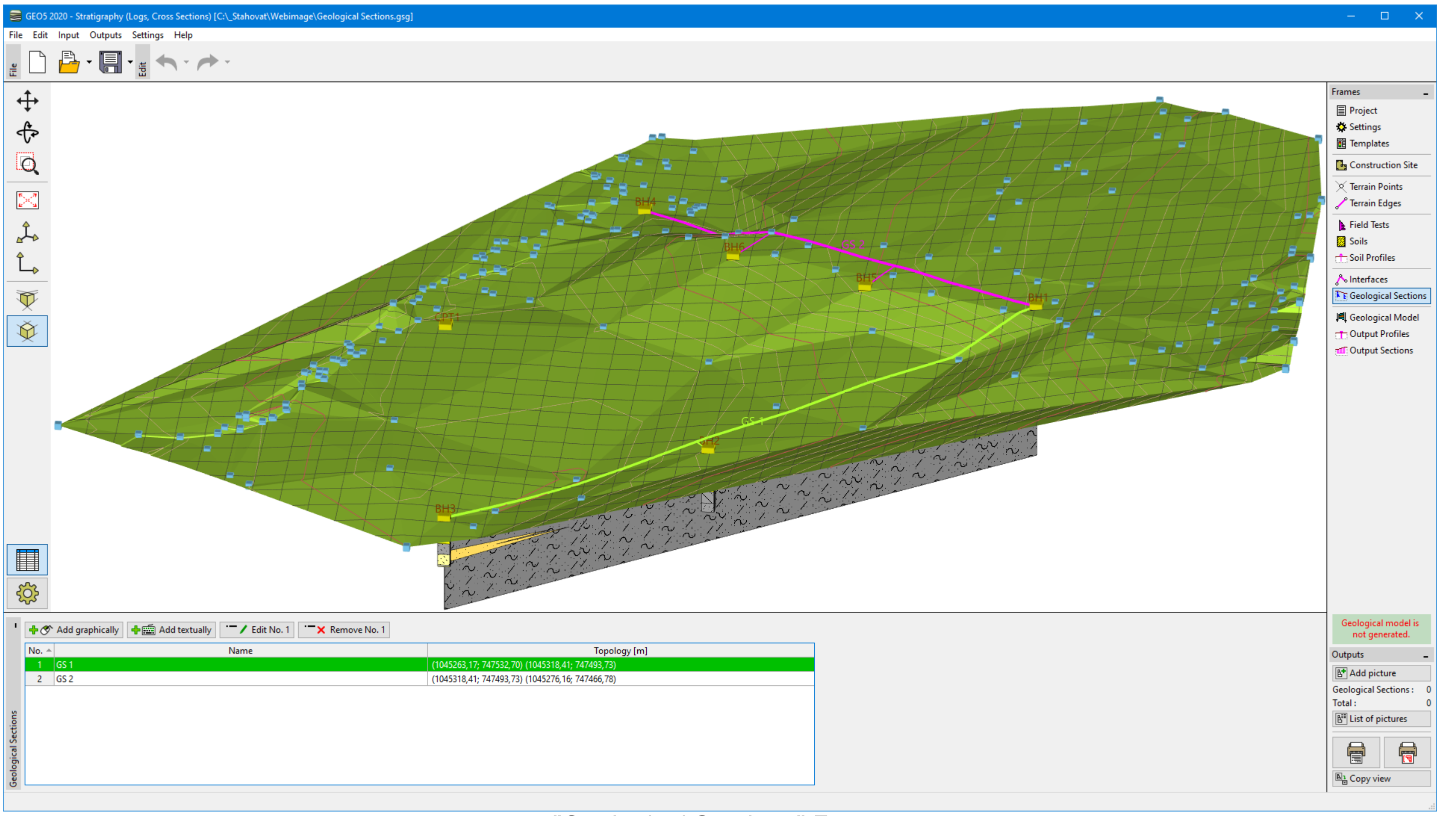
Horizontal GWT can be defined just by entering one point with the required depth.

Geological Sections

The "**Geological Sections**" frame allows us to create geological cross-sections (fences) with Field Tests and Soil Profiles displayed. Next, **the geological interfaces between soil layers**, **faults**, **the tables of ground water**, **the structures,** and other **user descriptions** are entered here. The points from defined interfaces can be copied to the "Interface" frame, where it affects the overall geological model.

We can also create geological sections from the generated geological model and use them for further **modelmodifications.**

First, it is necessary to define the required cross-section - **straight** or **polygon**. Points of geological sections can be defined within the construction site - if there is any **point out of the construction site**, the geological section is cut automatically. After input, the dialog window is opened in the "**Topology**" mode.



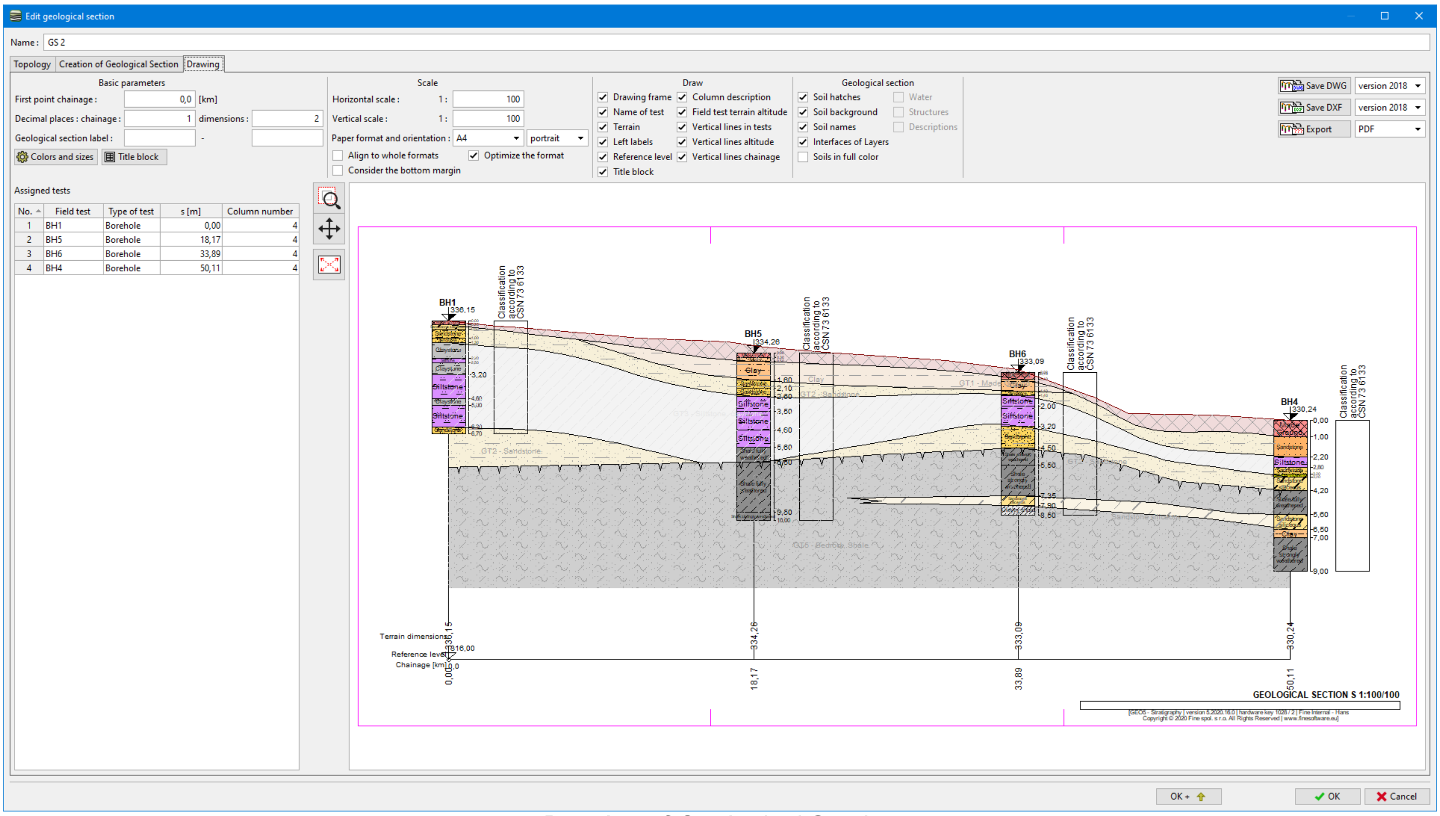
"Geological Sections" Frame

The geological section is defined and edited in a dialog window with **several tabs**.

- Topology
- Creation of Geological Section
- Drawing

The aim of geological section creation is to create and publish **geological cross-section**, define **geotechnical types of soils**, edit **soil profiles,** and generate interfaces for easy creation of a 3D model.

The **"Drawing" tab** is available only to users with purchased the **"Cross Sections"** module.



Drawing of Geological Section

Topology

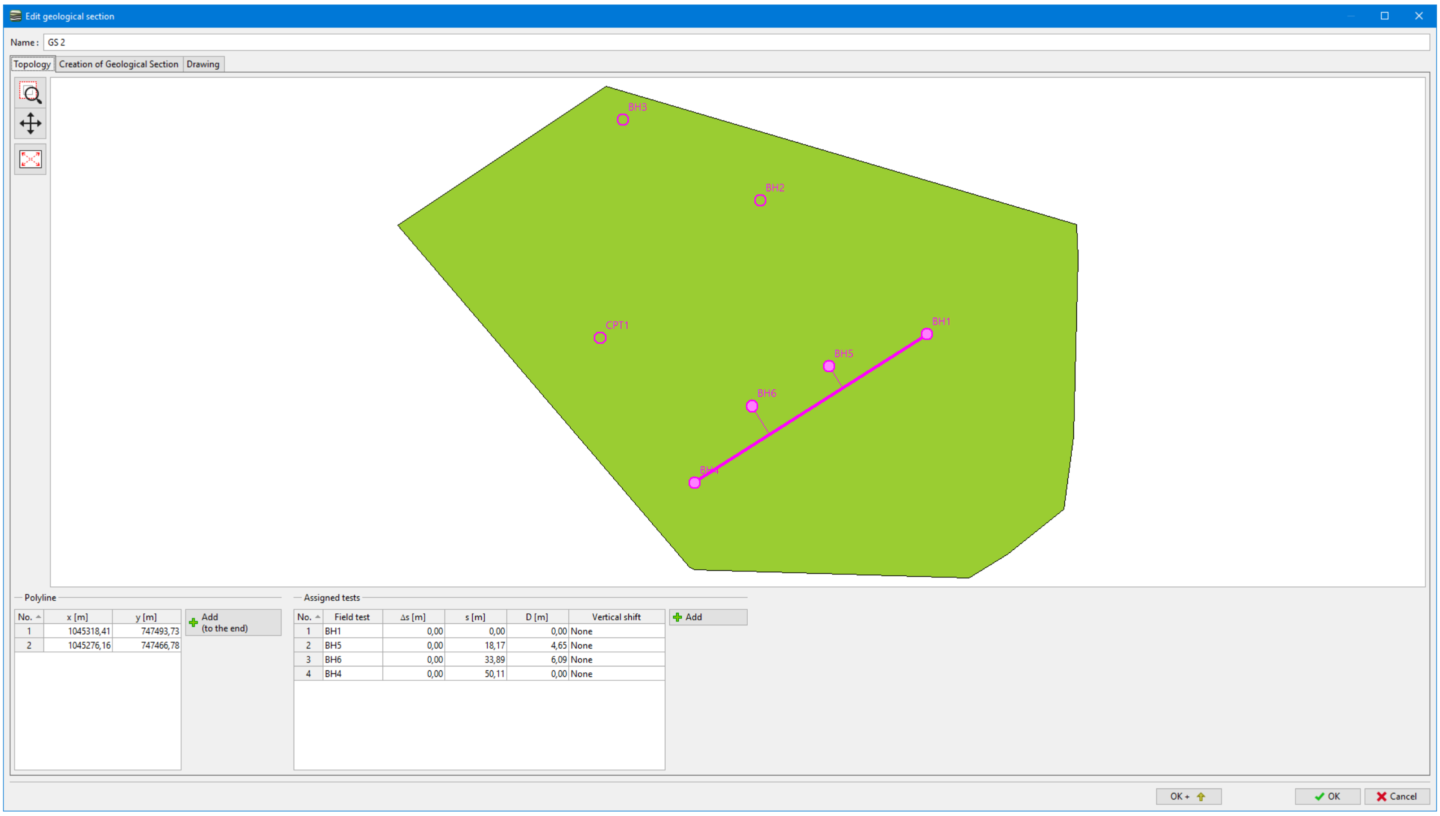
In the "**Topology**" frame, the **coordinates of section points** and **table with required field tests** are shown.

We assign all field tests we want to display in our cross-section.

The field tests are usually projected perpendicularly to the geological section, but it is possible to shift them as well. The **chainage of the field test** and its **distance from the geological section** are displayed in the "**Assigned field tests**" table.

Next, the vertical shift of the field test must be defined.

- **None** - standard option, the field test is displayed in the real altitude
- **On the terrain** - the first point of the filed test is placed on the terrain
- **Input** - user-defined vertical shift of the field test in the geological section



"Topology" Tab

Creation of Geological Section

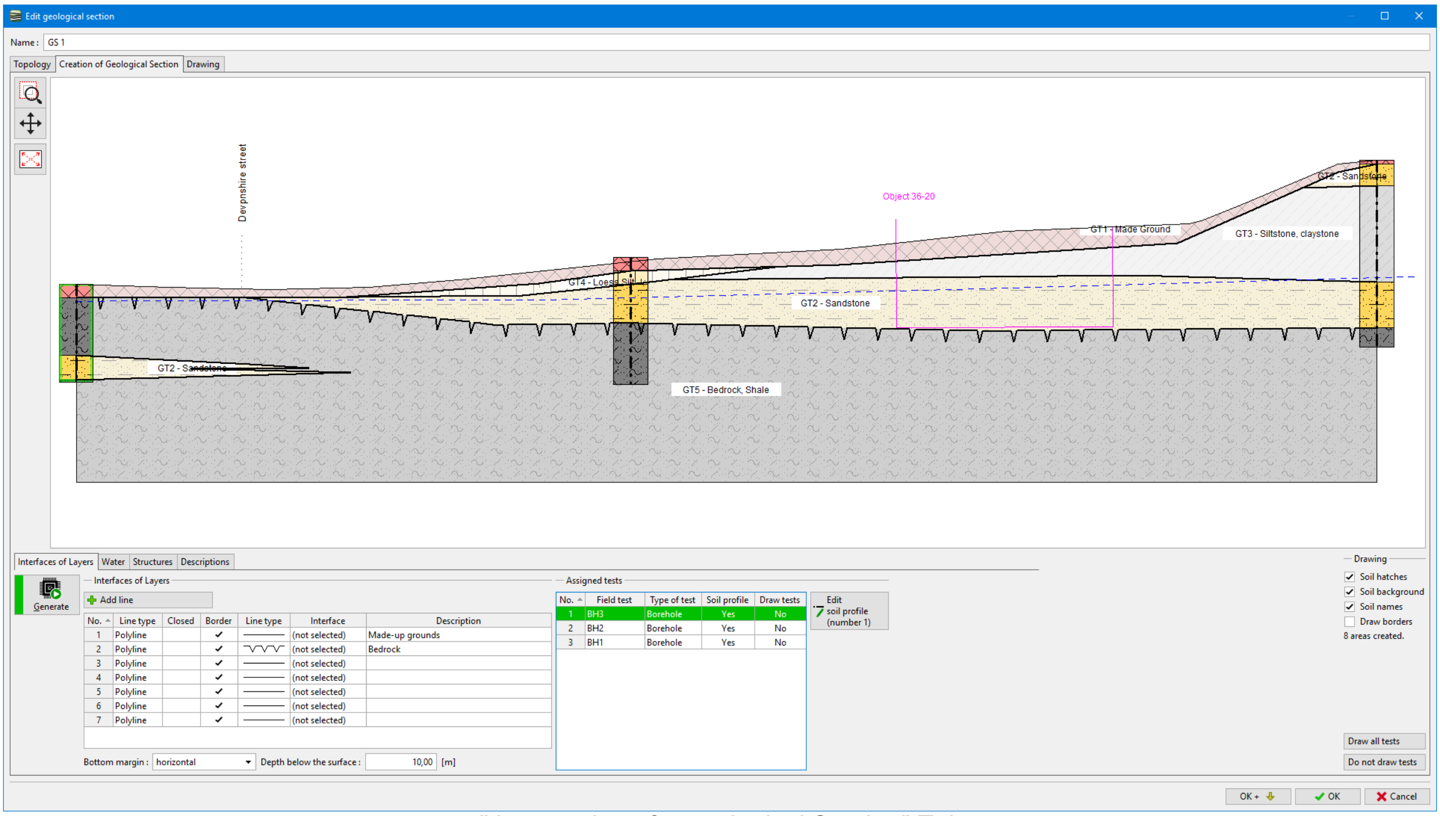
In this tab, the geological section is created.

In the subtabs, we define:

- **Interface of layers** of soils, geotechnical soil types
- Levels of **groundwater table**
- **Structures**, auxiliary lines
- **Descriptions** in the drawing

In this dialog window, it is possible to **define lines of interfaces** and this way create a geological section, and also edit soil profiles, and create points of interfaces for **3D model generation**.

The geological section can also be created from the already generated geological model.



"the creation of a geological Section" Tab

On the left side of the dialog window, there is a "**Generate**" button serving for **area generation**. The program automatically detects areas created by input lines and soil profiles. All lines marked as "**Creating border of area**" are used for areas generation. If more then one soil interferes with one area, the area is unclear, and a soil is not assigned.

On the right side of the dialog window, the **drawing of the created geological section is defined**. The possibility of **field tests drawing** is very important - the buttons "**Draw all tests**" and "**Do not draw tests**" is used when soil profiles are edited.

In the bottom part of the dialog window, the **bottom margin of the model** is defined. It can be horizontal, parallel to terrain (it is necessary to define depth below the terrain), or arbitrary continuous interface can be used.

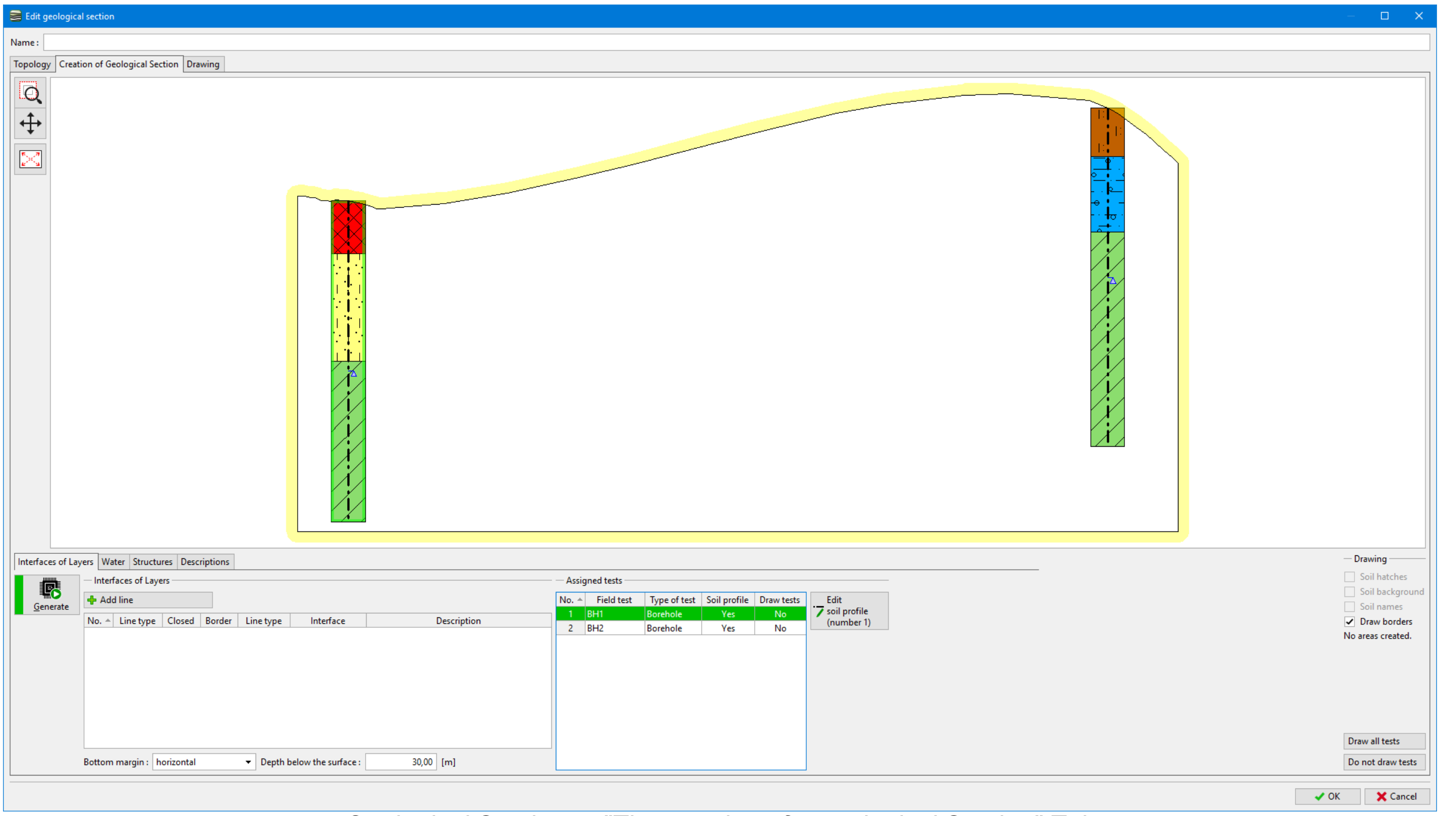
Creation of Geological Section

The geological section is created using **arbitrary lines** similar to CAD programs.

By graphical input, the arbitrary points can be defined by the left mouse button. Points of lines can be snapped to the entered soil profiles, but it is also possible to enter them on other lines or interfaces.

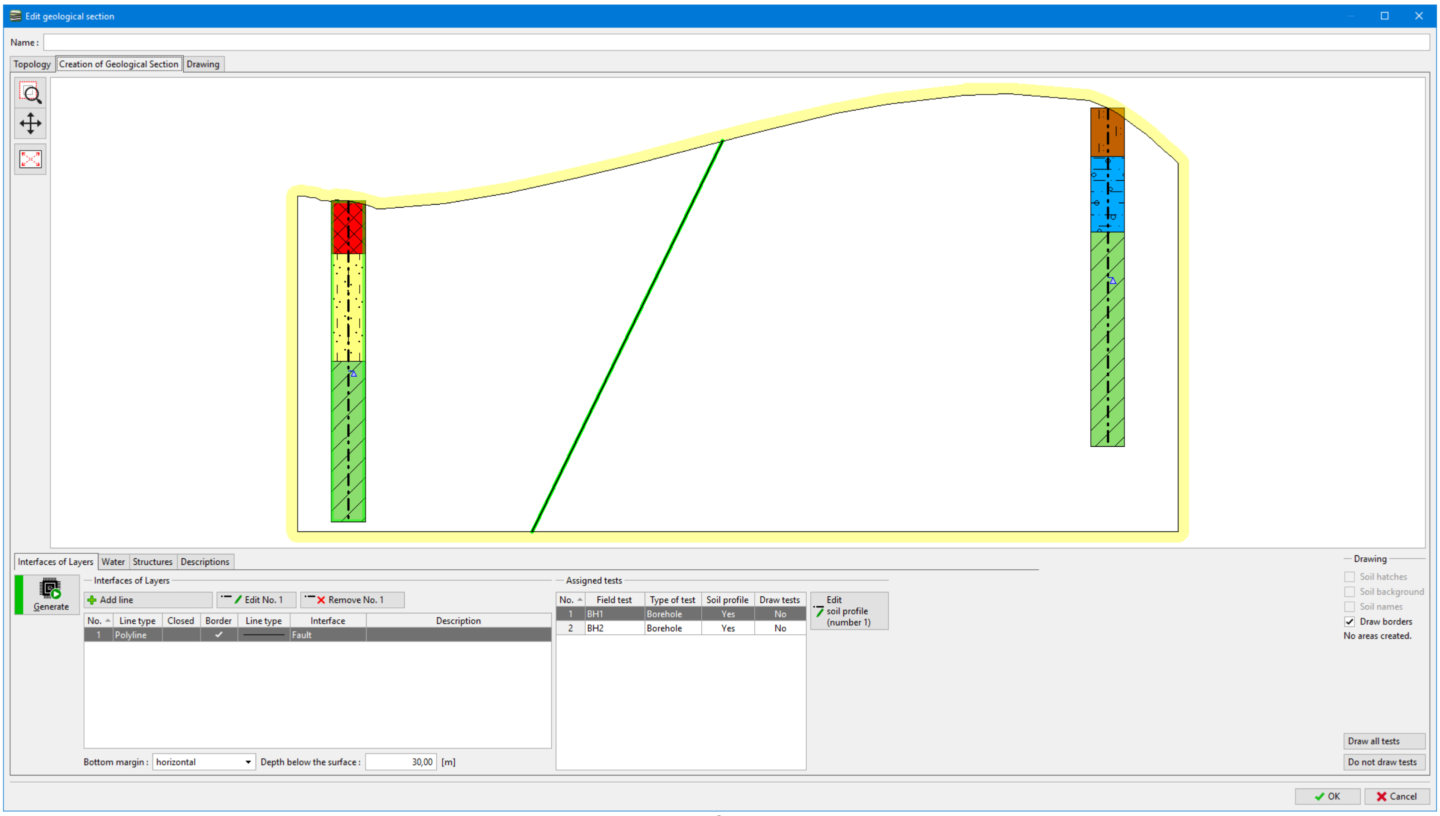
Graphical input is **terminated by double click on the same point**. It is possible to modify the geological section by shifting a point.

As an example, we show an input of geological fault between two boreholes.



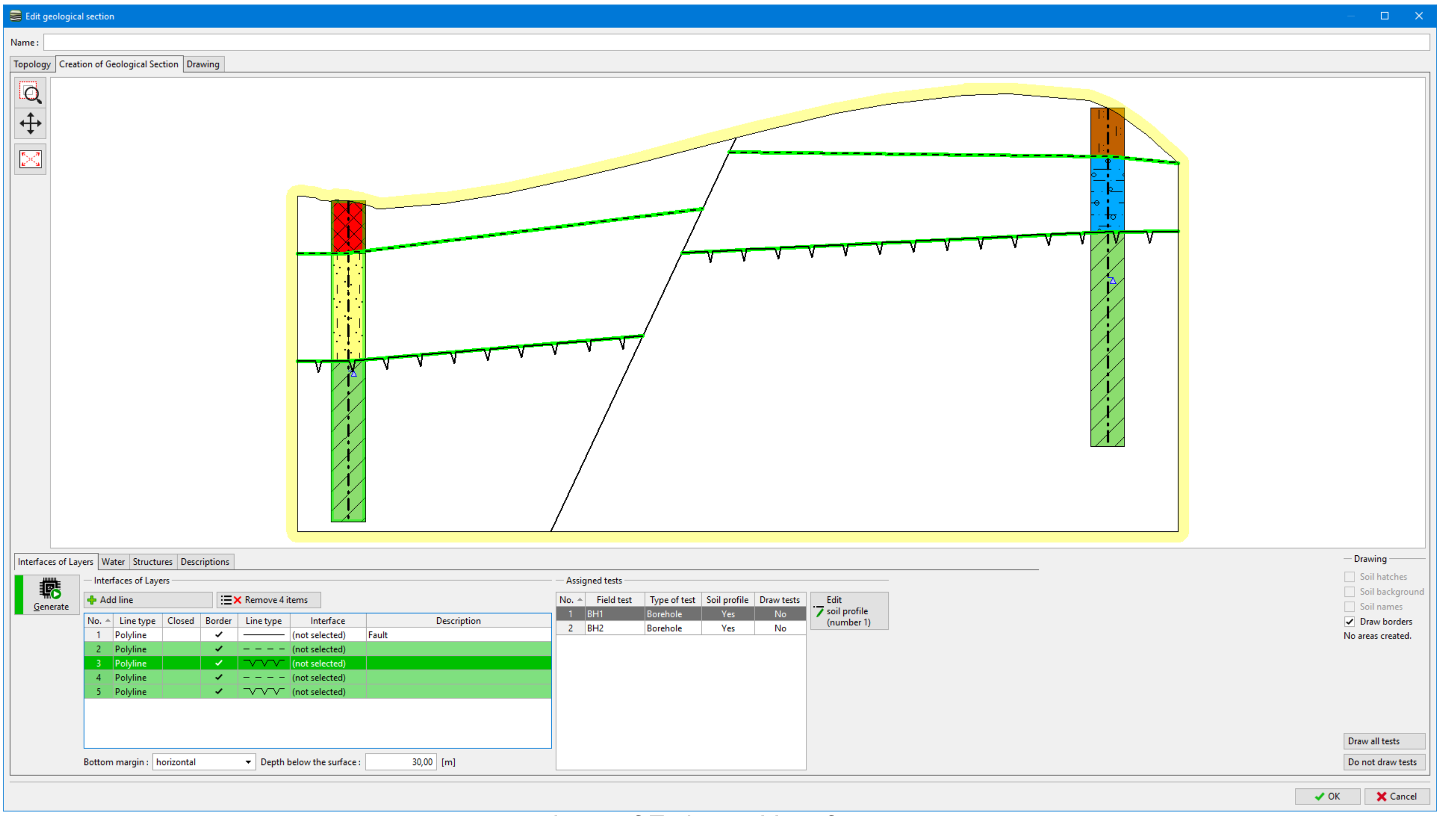
Geological Sections - "The creation of a geological Section" Tab

1. We define a fault - line going from the terrain to the rock massive.



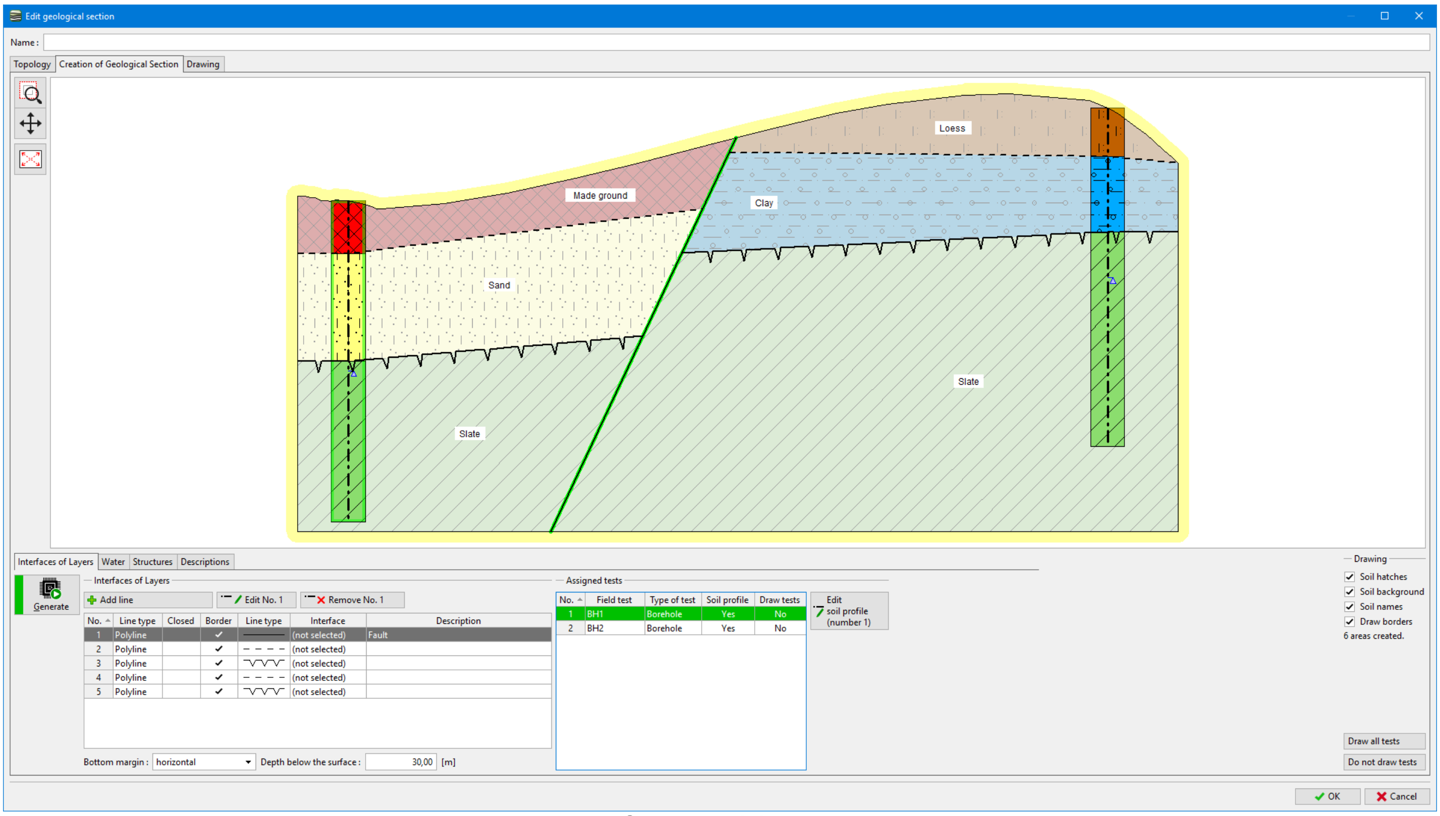
Input of a Fault

2. We define estimated interfaces, which we snap
   - on the section edge
   - on the interface in the soil profile
   - on the created fault.



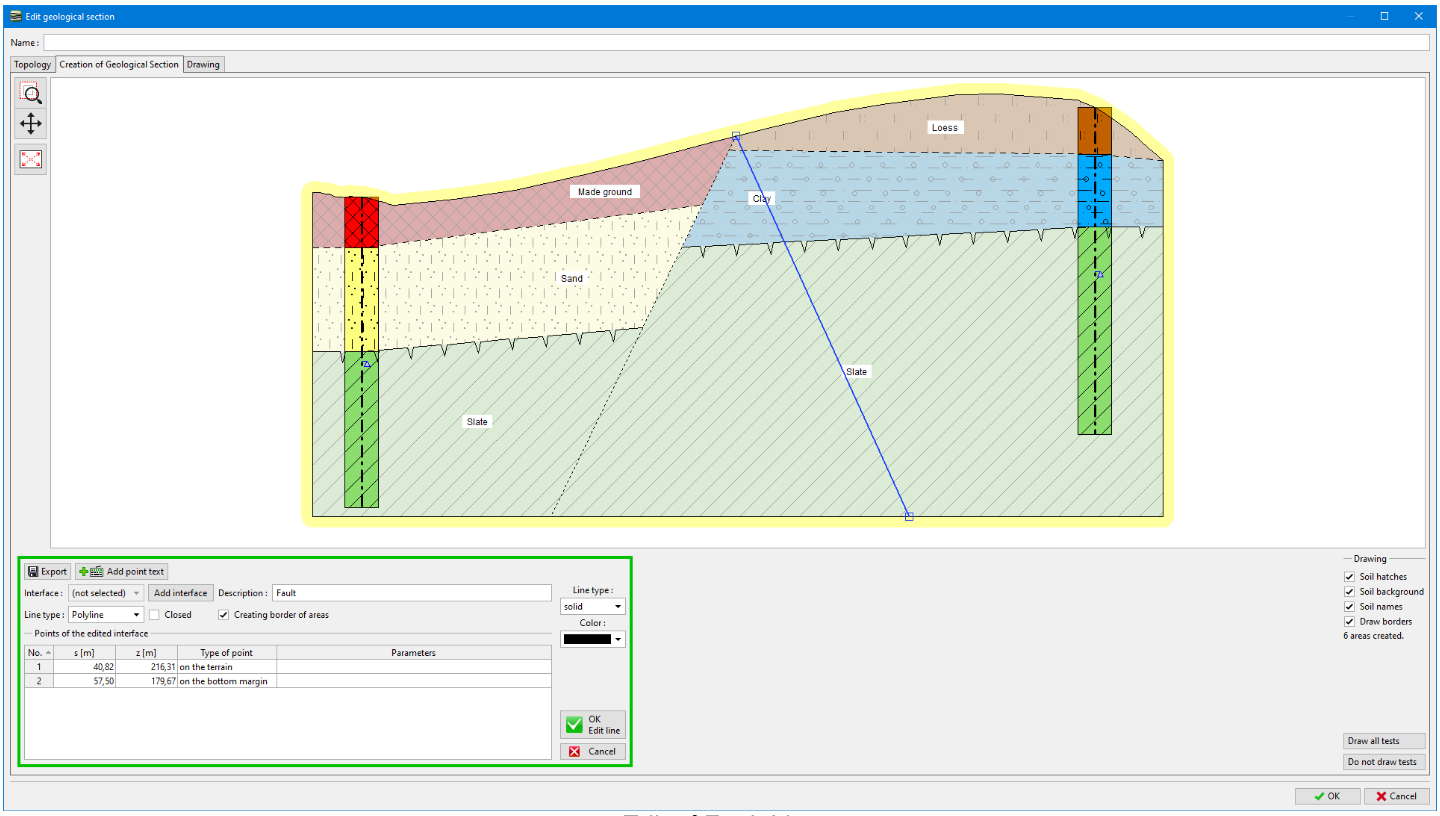
Input of Estimated Interfaces

3. We generate areas of soils.

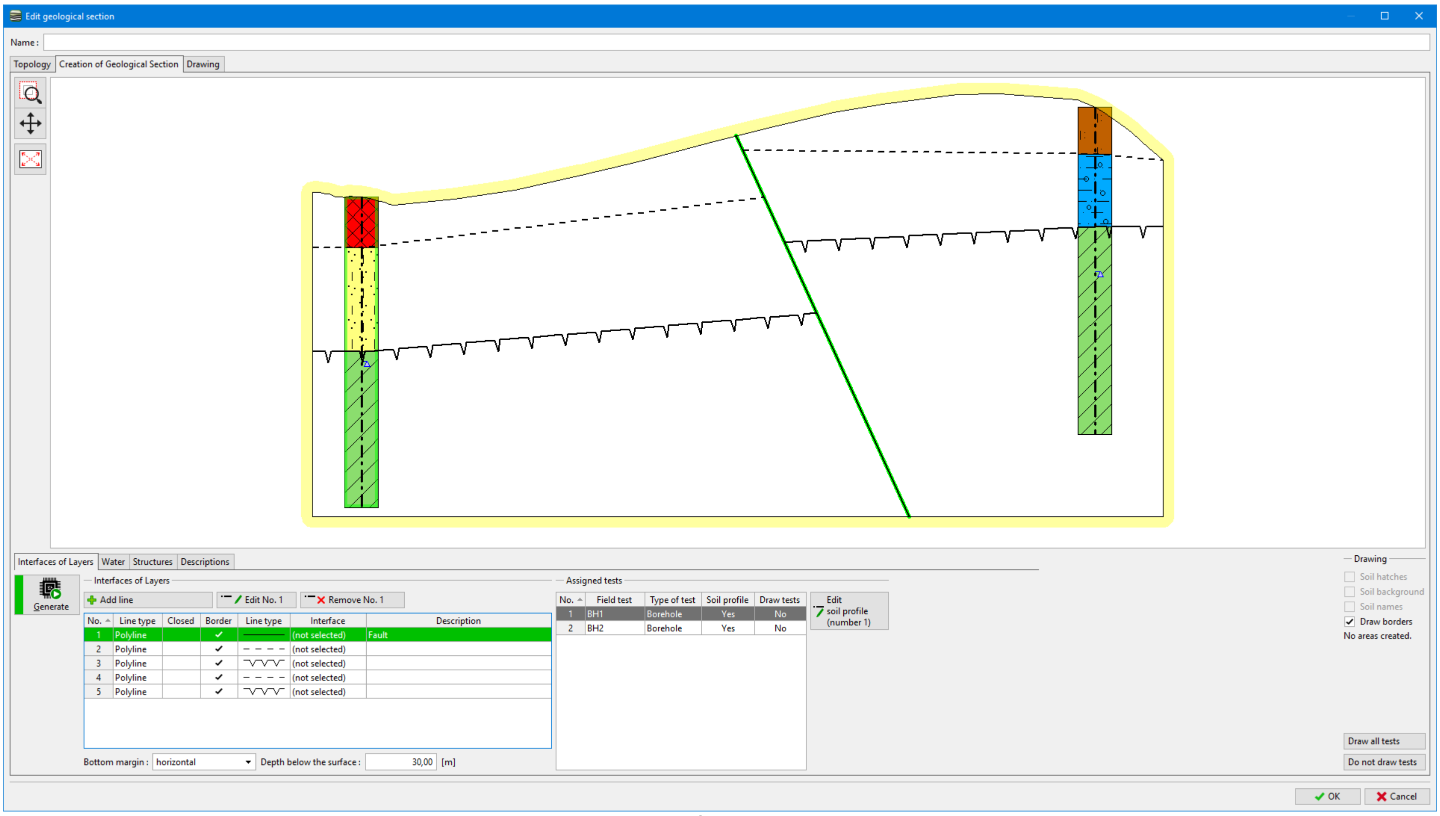


Generated Areas

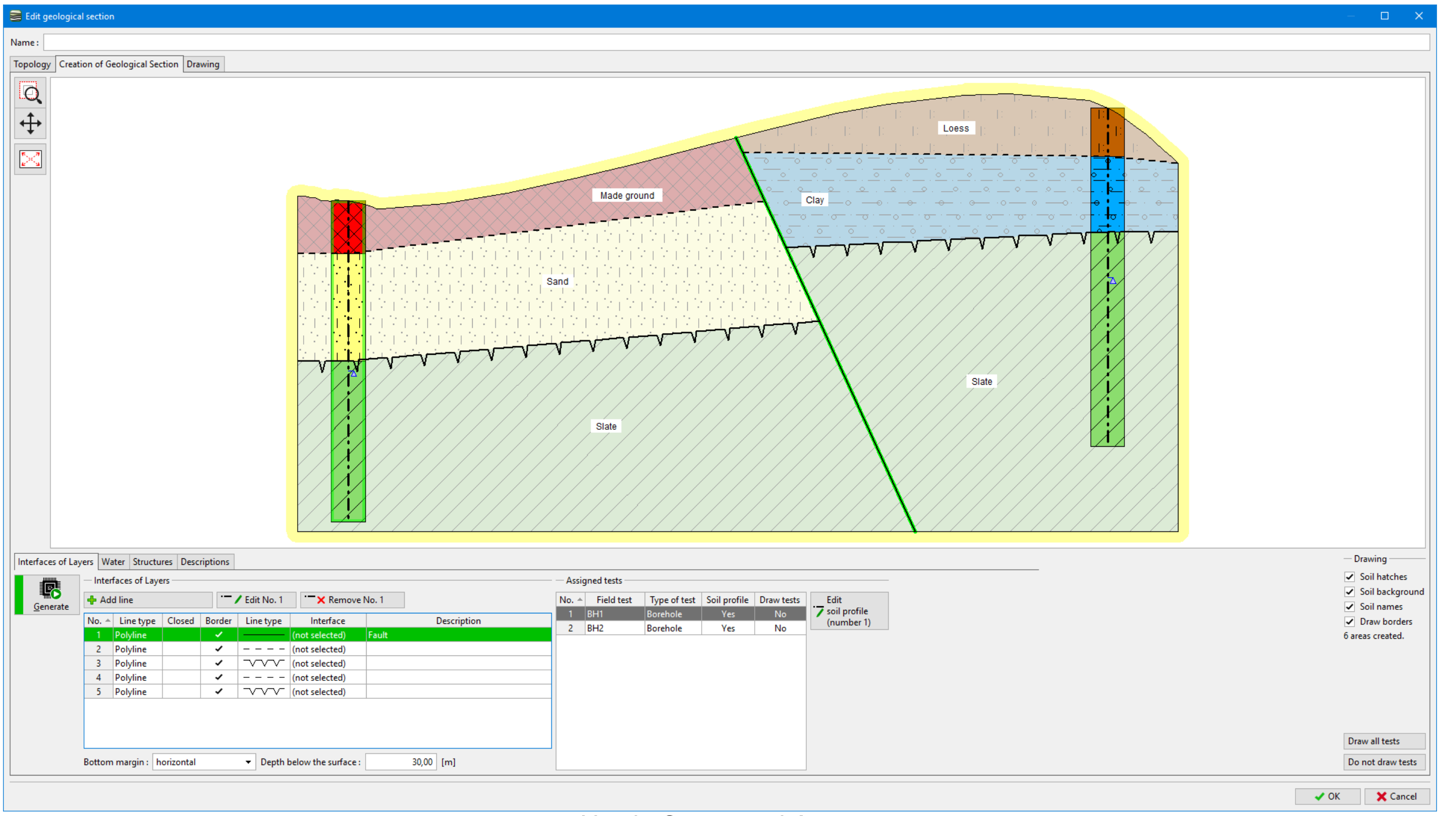
We can change the geological section by shifting the bottom point of the fault.



Edit of Fault Line



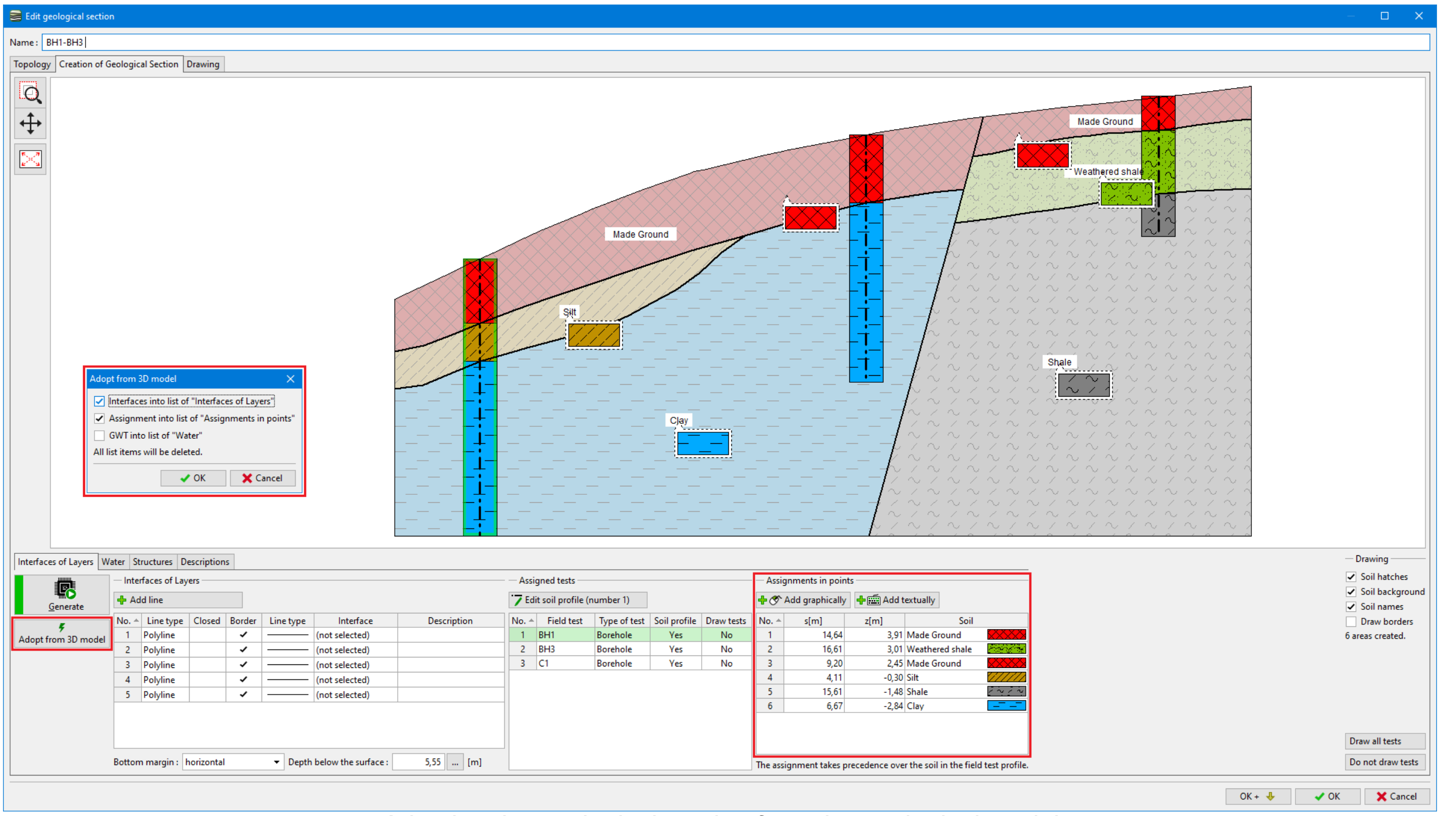
Modified Fault



Newly Generated Areas

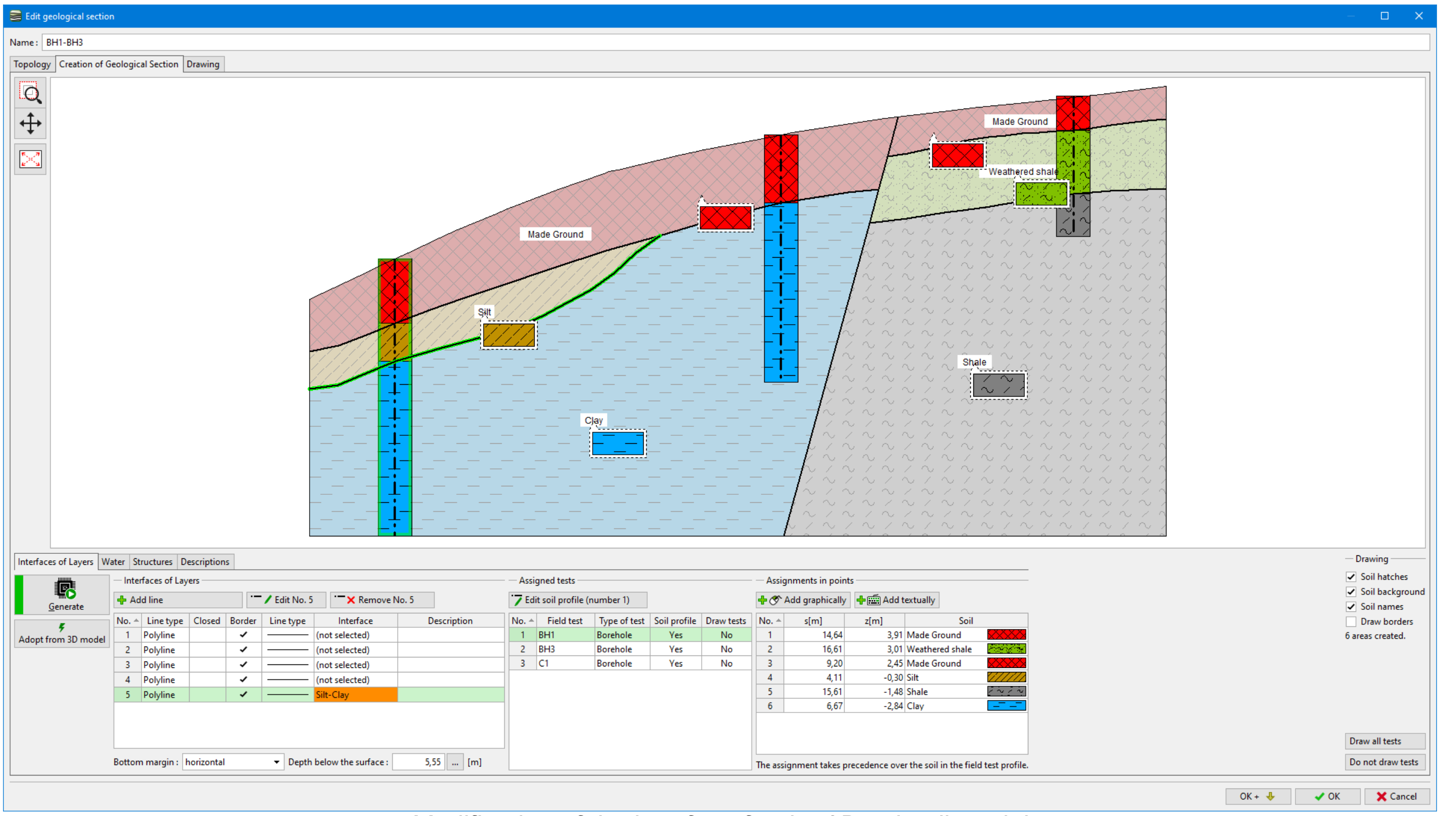
Creation of Geological Section from the Geological Model

By "Adopt from 3D model" button, it is possible to create the **Geological section from** the already generated **3D subsoil model**. The program **cuts the model**, creates lines and points of assignments. The soils in assignment points have priority against the soils in the soil profiles.



Adopting the geological section from the geological model

It is possible to work with adopted geological section normally. If we want to change the model retrospectively, we have to assign interfaces to the newly created lines. This line/interface is highlighted in orange in the table and its points are transferred to the interfaces. This way, the geological model can be changed.

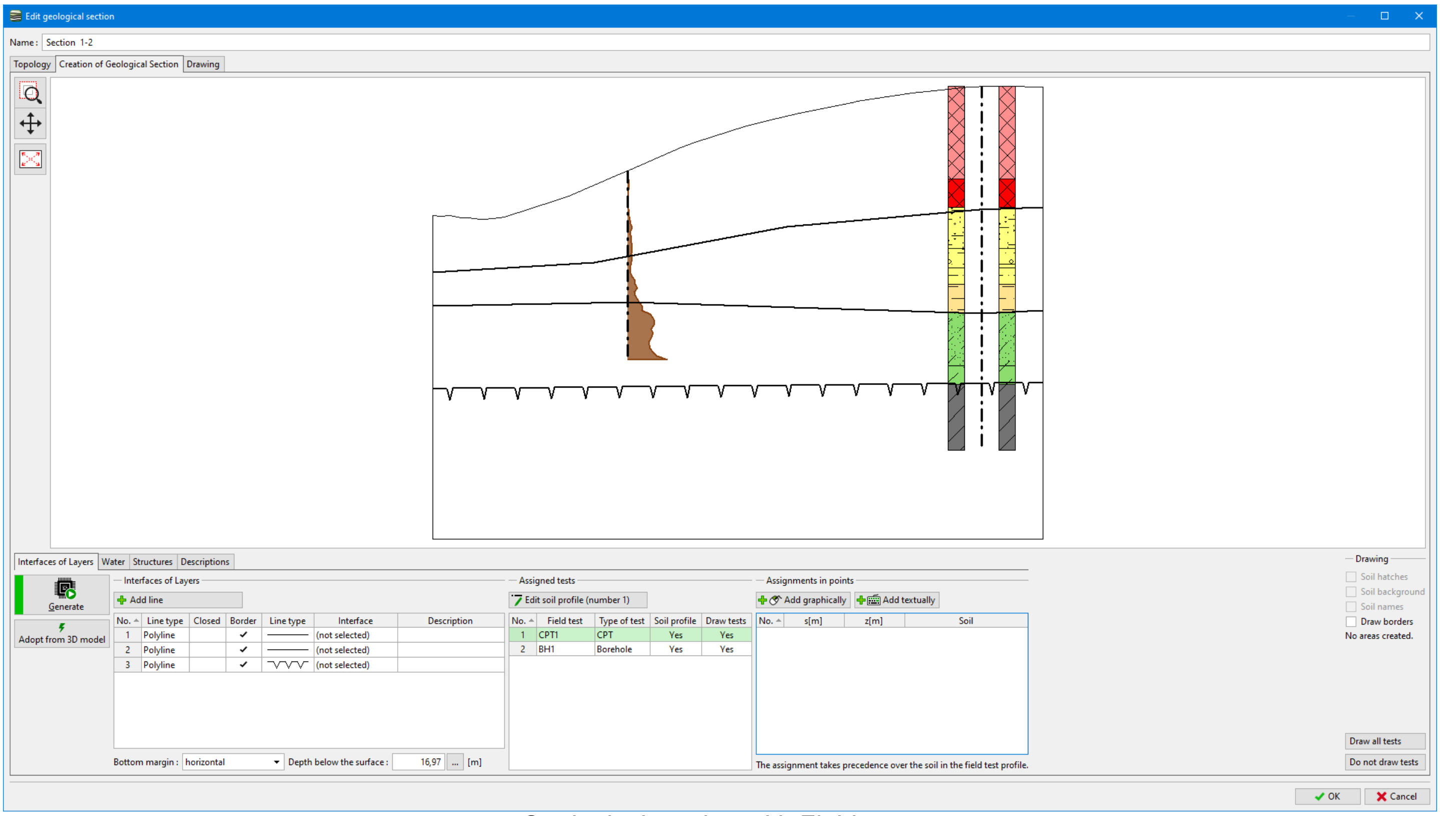


Modification of the interface for the 3D subsoil model

Edit of Soil Profiles

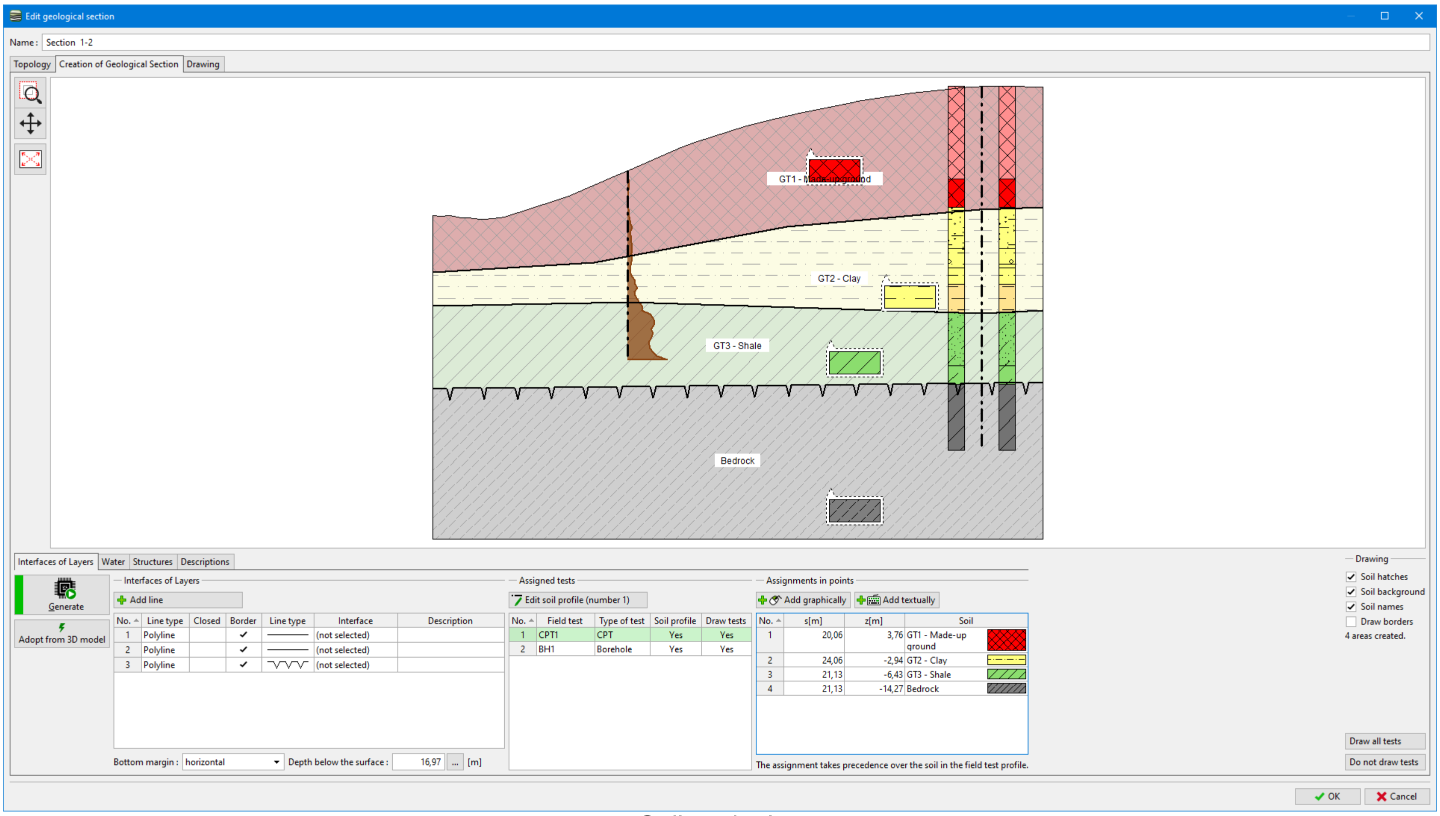
The real boreholes contain a large number of soil layers - however, the geotechnical engineer works with a **simplified model**. The **geotechnical types** are used for this simplification. The boreholes are entered exactly (with all layers), the simplification is performed when soil profiles are created - these profiles represent the geotechnical model.

In the picture below, we see borehole, CPT and our idea of the geological section.



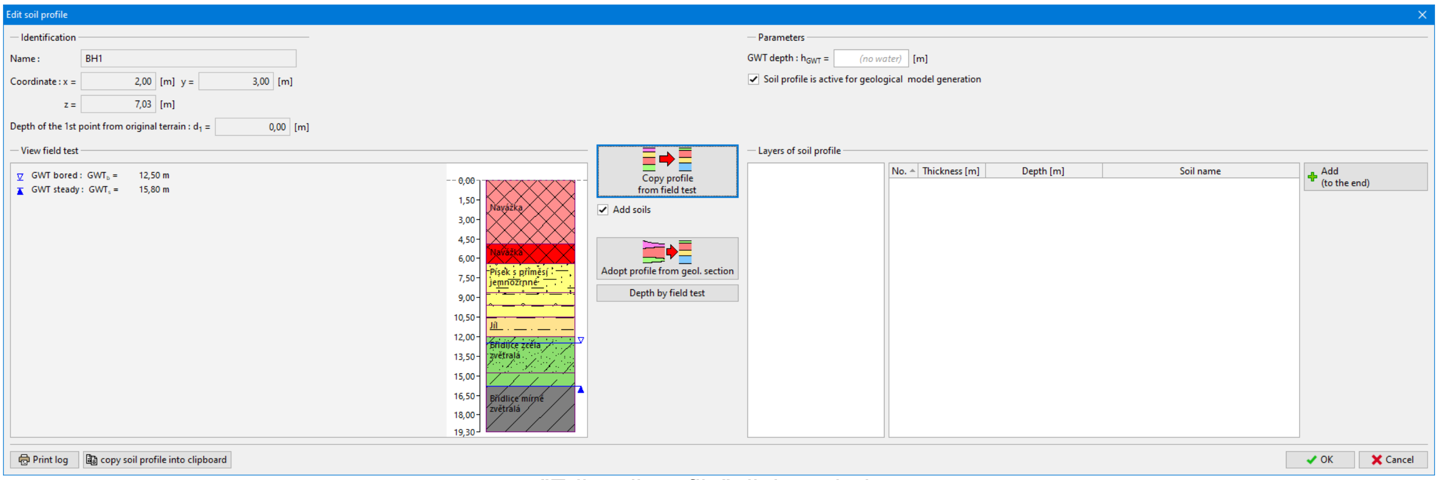
Geological section with Field tests

We assign soils/geotechnical types into the areas and generate the section.



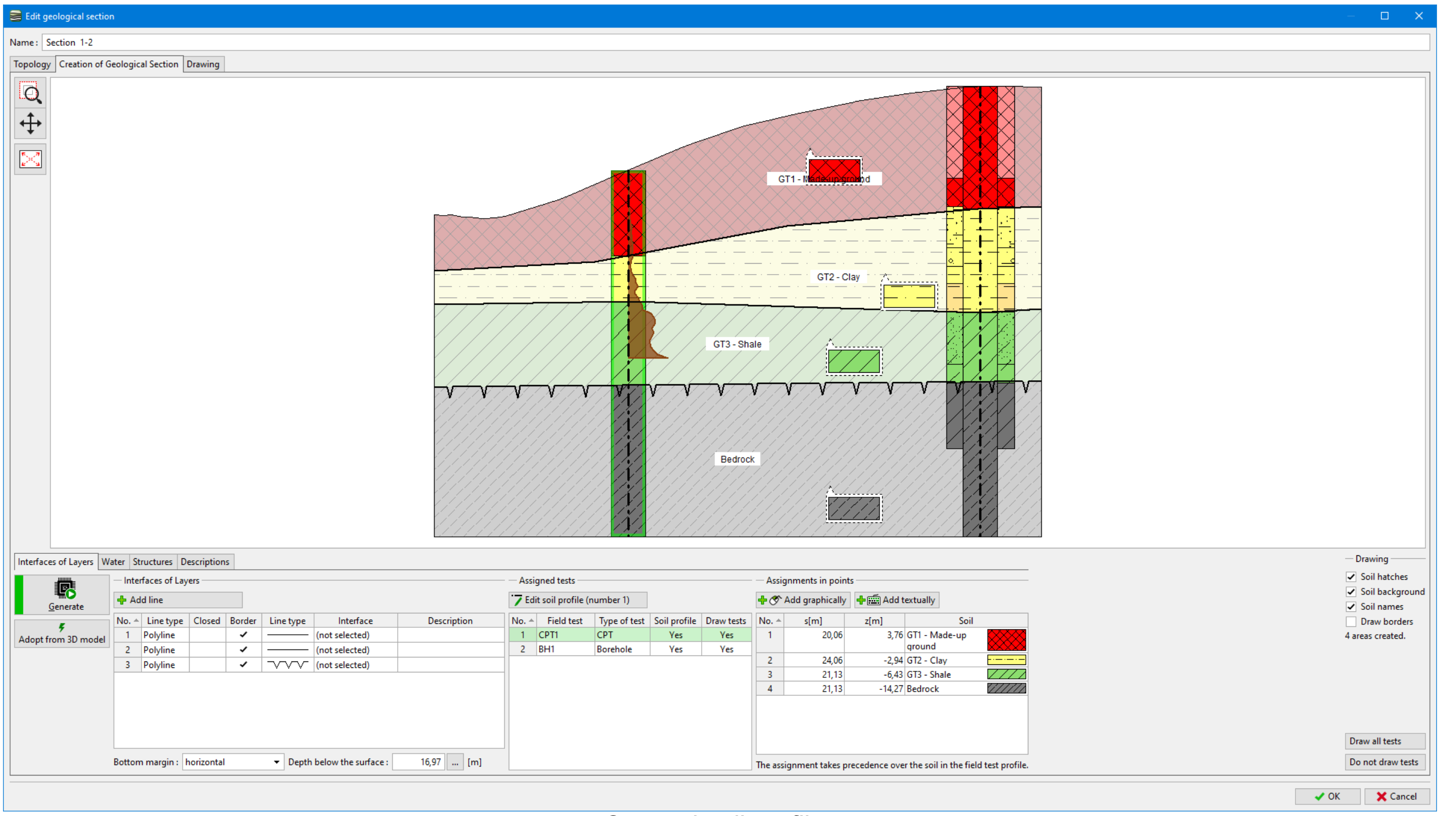
Soil assigning

We can create Soil profile from the geological section using the "Adopt profile from geological section" button.



"Edit soil profile" dialog window

Created soil profiles improve the generation of further geological sections, overall 3D subsoil model, and geological documentation of the geotechnical model.

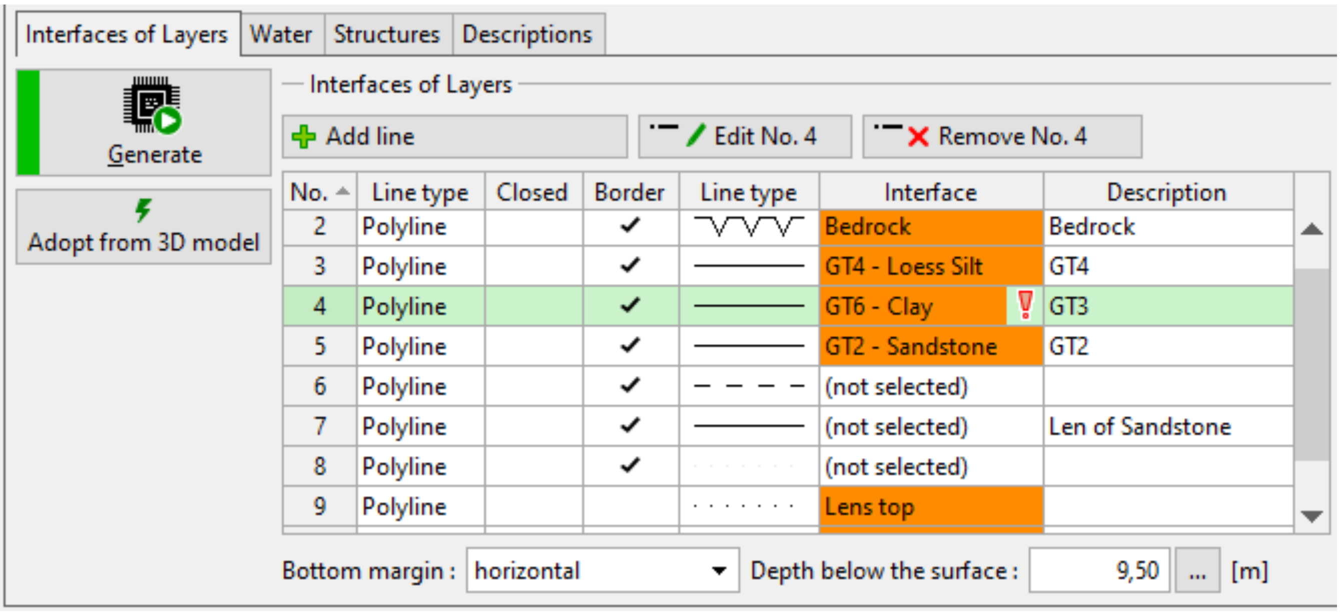


Created soil profiles

Creation of Interface points

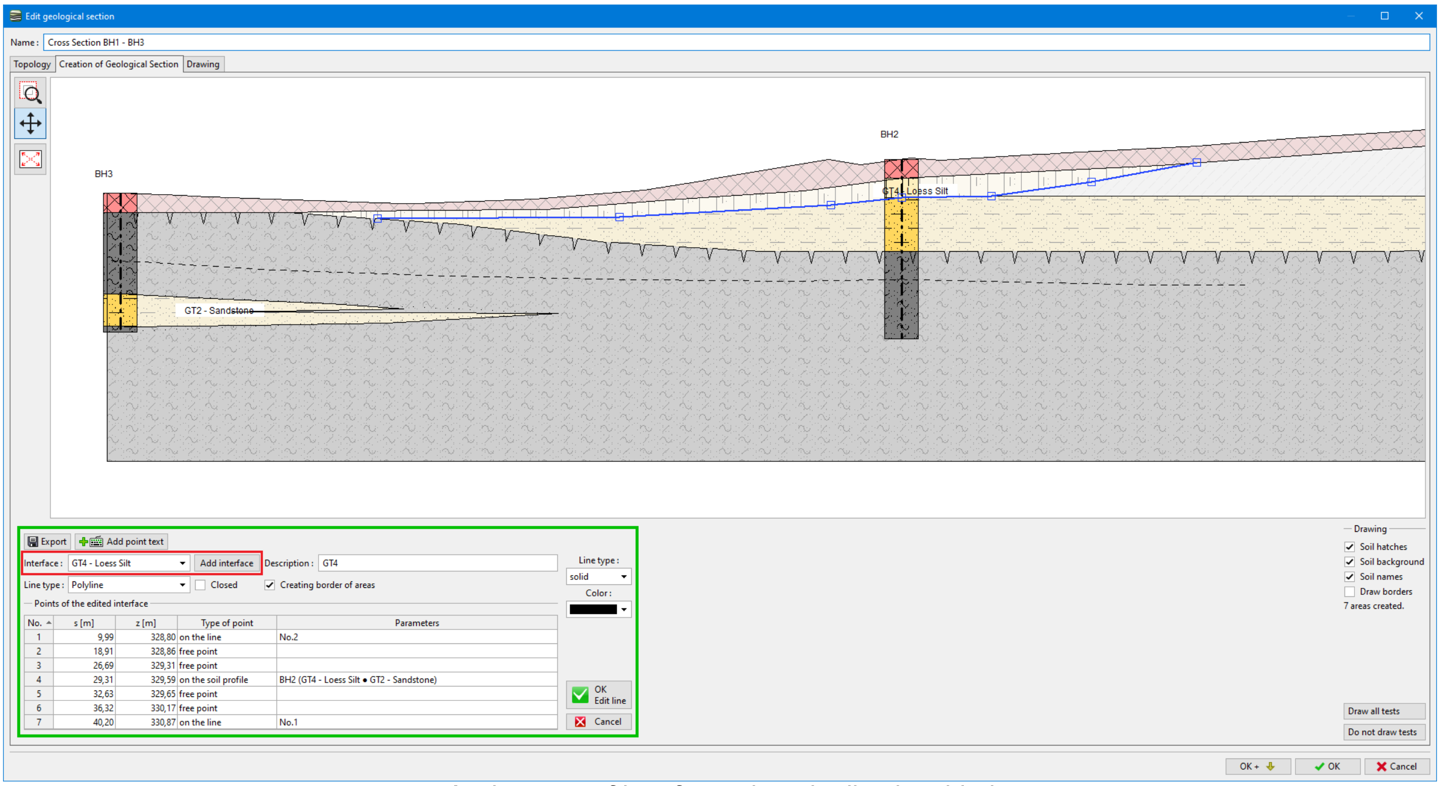
Created geological sections correspond with our ideas about the geology of the location - they are ideal input for the creation of a 3D subsoil model. It is possible to assign each line to interfaces of the future 3D model. The assignment is carried out in the **dialog window for line input** - we can **select an interface from the list** or **create a new interface** using the "**Add interface**" button. The points of lines with the assigned interface (highlighted in orange in the table) are transferred into the Interface data automatically. A special type is a modeling of soil lenses or geological faults.

Assigned interfaces are visible in the table.



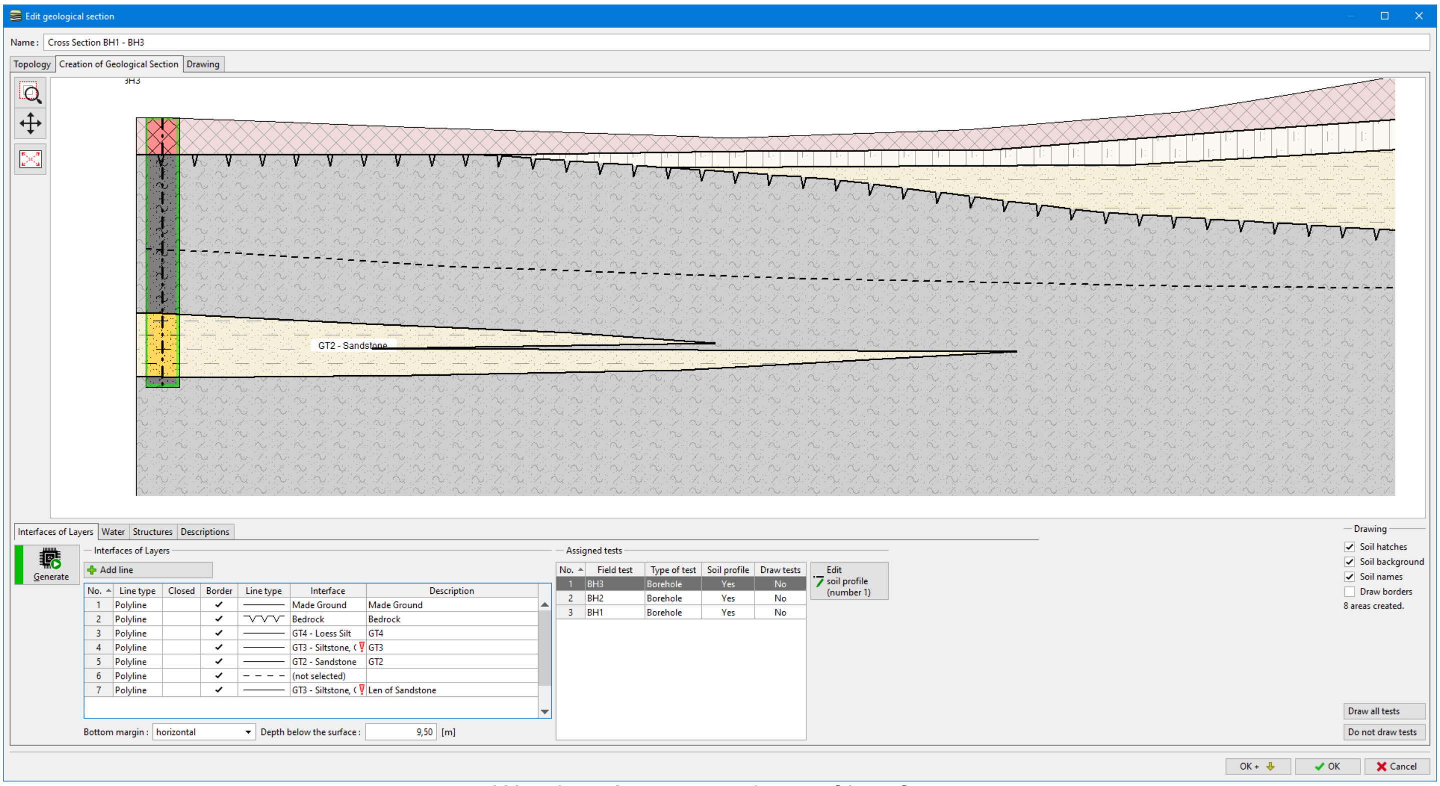
Assigned interfaces in orange

The interface can be assigned when a line is edited (or added). In the picture, we can see the assigned interface "GT4 - Loess Silt" - it is recommended to use name according to the soil (or geotechnical type) above it or combination of layers which divides (e.g. GT1-Silt/GT2-Clay).



Assignment of interface when the line is added

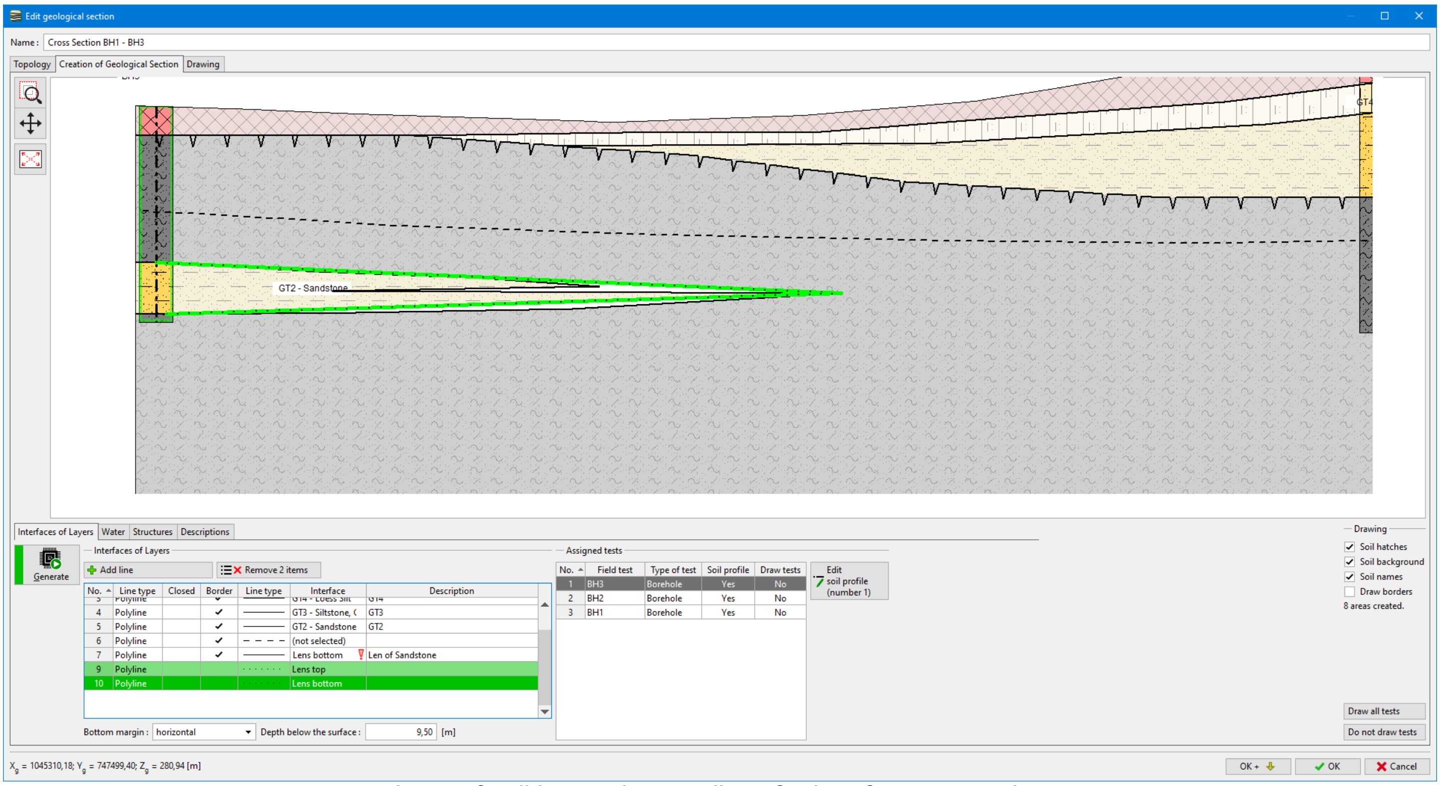
Some lines can be defined that their points lie above each other - e.g. soil lenses. If we create an interface from this line, the model could not be generated correctly. The program warns about it using a **red exclamation mark in the list of lines** (on the line of an interface).



Warning about wrong input of interface

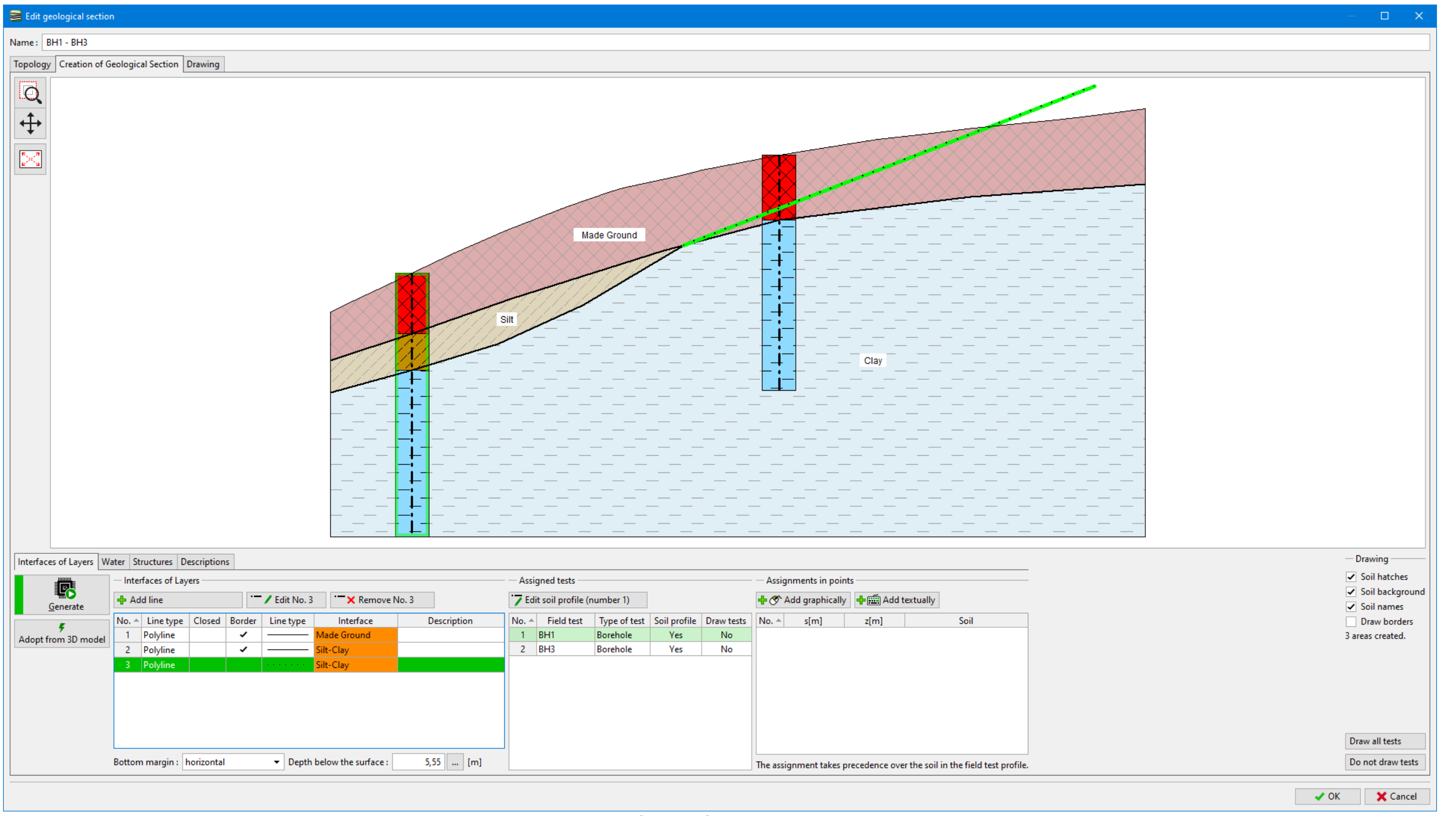
Modeling of Lenses

If we want to add points of soil lens into an interface, it is necessary **to divide it into two lines** - the upper and bottom part of lenses. The **line type** must be defined as **"auxiliary"** because this type is not displayed in the geological section drawing. Also, it is necessary to uncheck the "**Creating border of areas**" option because we do not want to change the shape of the generated areas - as shown in the following picture.



Input of soil lens using two lines for interface generation

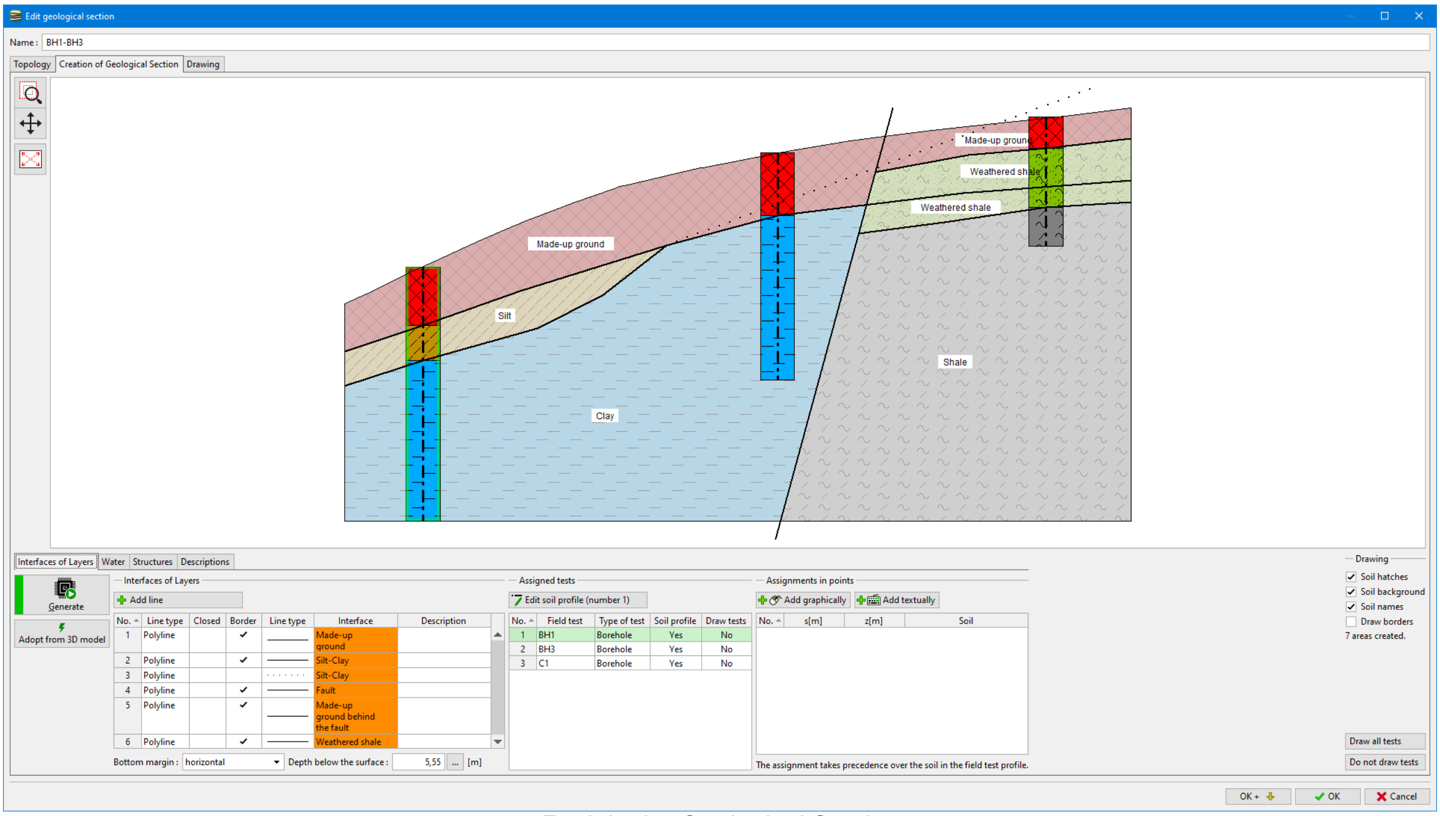
It is also recommended to model the lens interface behind the lens. This can be created as an "auxiliary" line, which is not displayed in the final drawing of the geological section. This line helps to **clearly define** the model.



Modeling of interface behind the lens

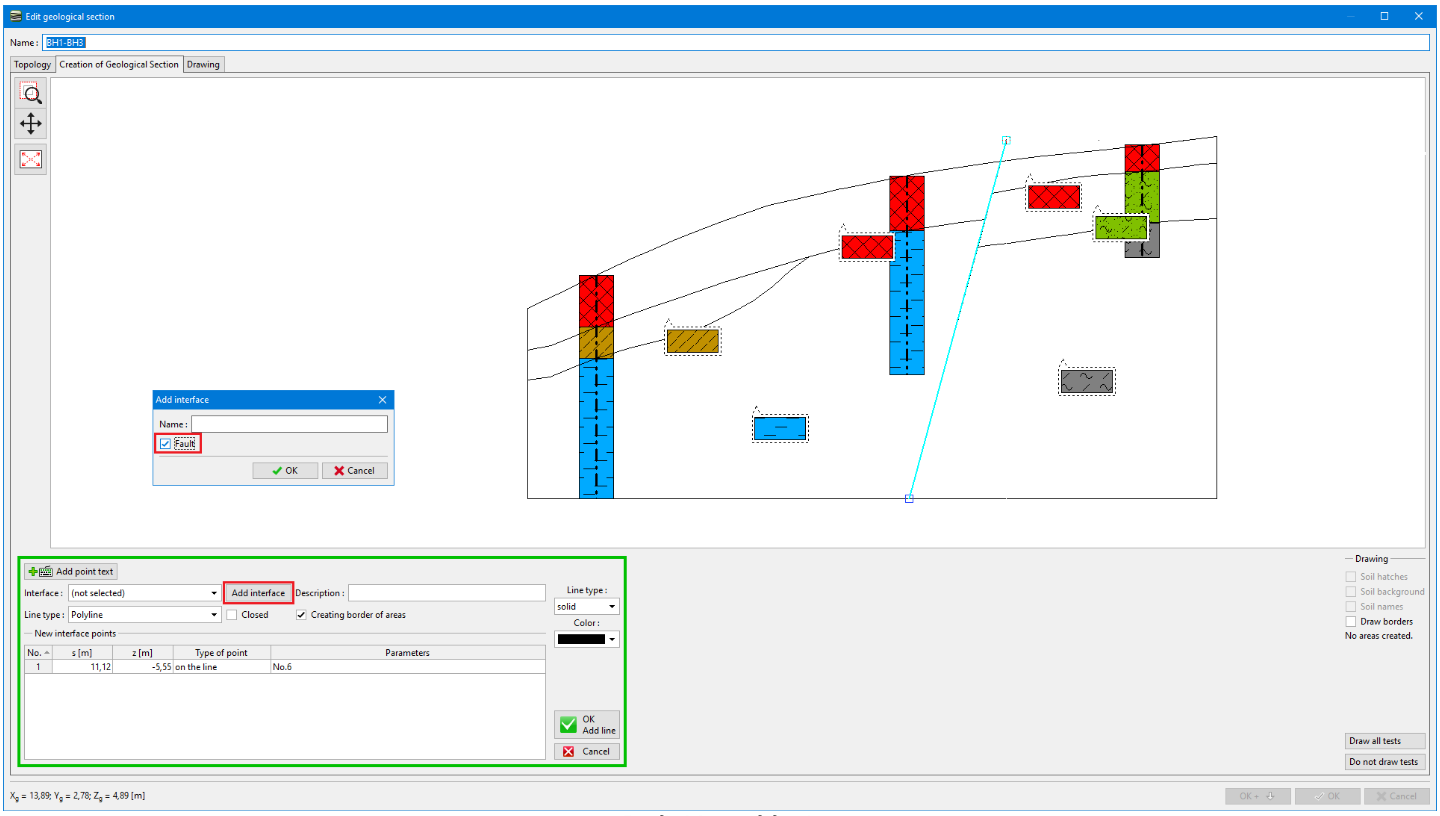
Modeling of Faults

When creating the geological section, it is possible to define a special type of interface - fault. The fault is always perpendicular to the geological section and divides the model to the individual parts. The fault interface is transferred into the list of interfaces, but can be edited only in the geological section where it was created.



Fault in the Geological Section

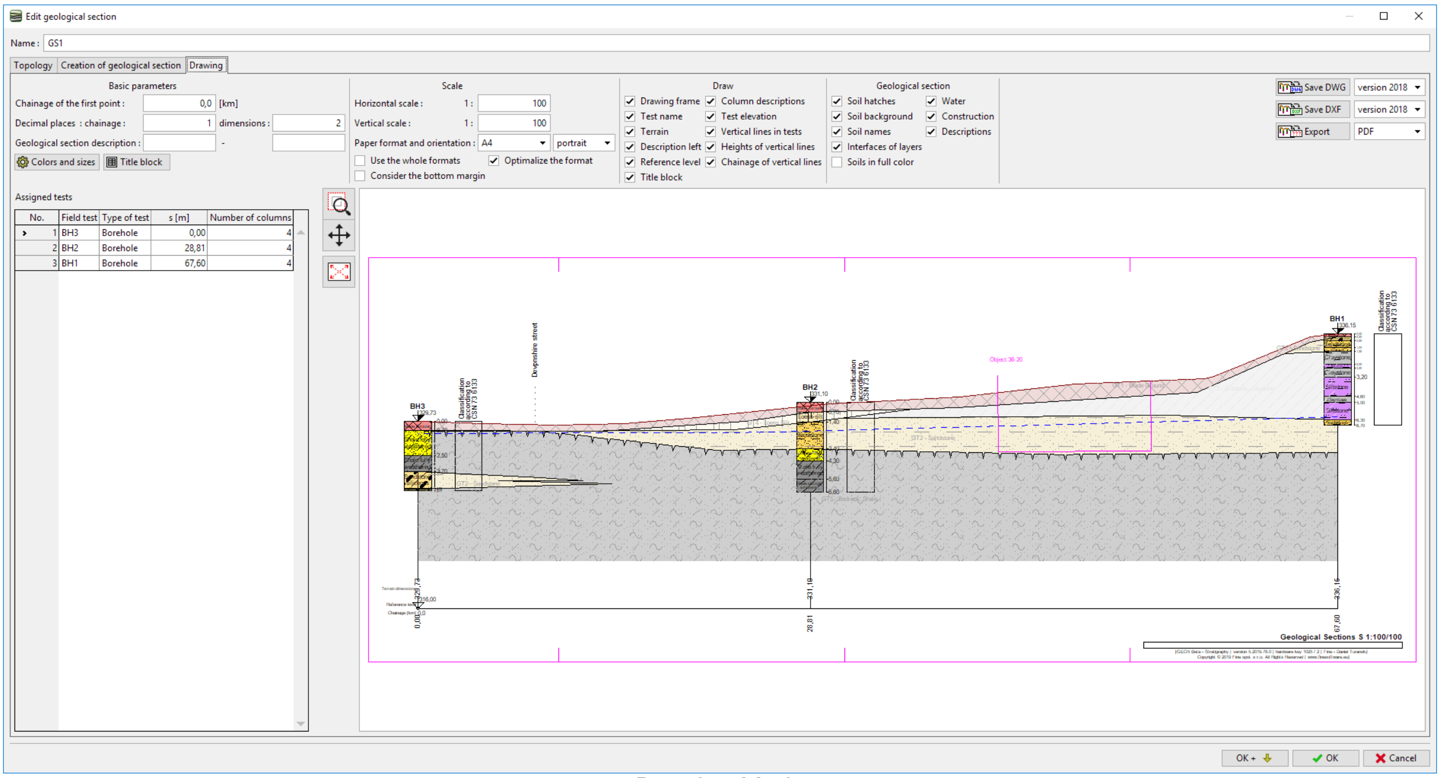
The fault is defined in the "Add interface" dialog window using "Fault" checkbox.



Definition of fault

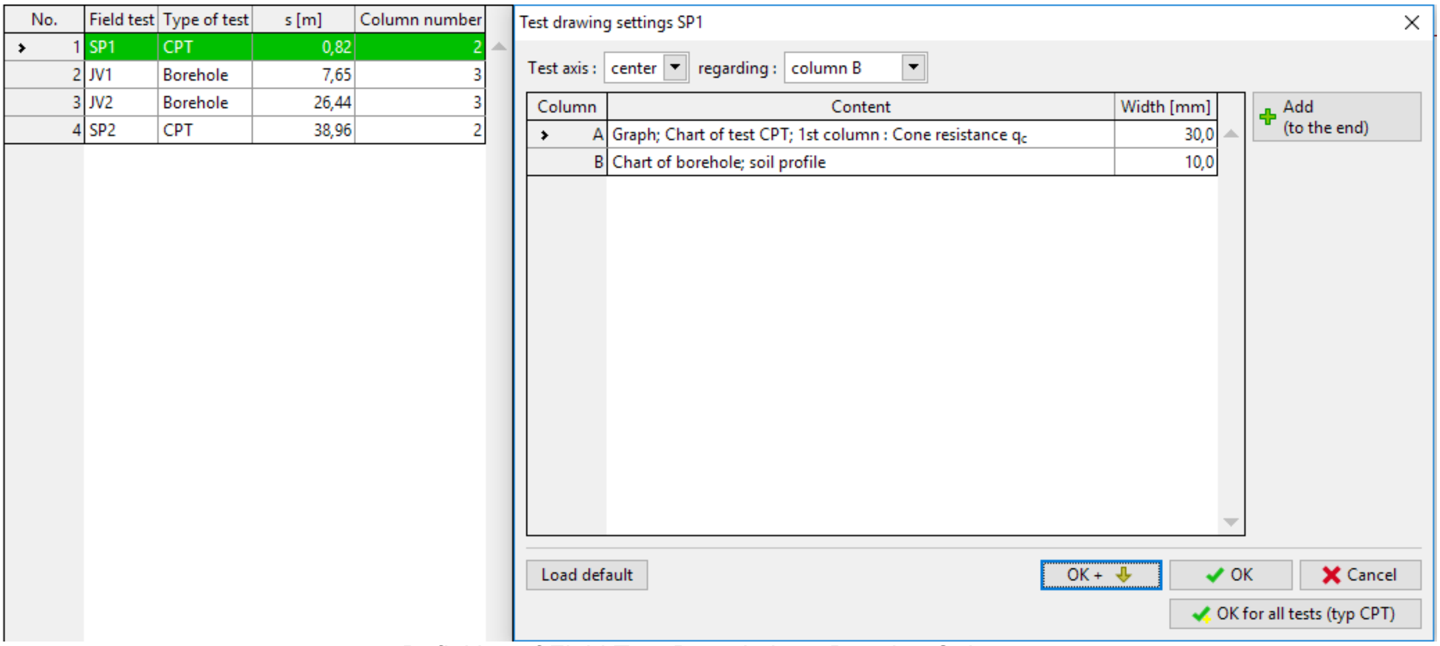
Drawing of Geological Section

In the "**Drawing**" tab, we define **what we want to plot on the drawing,** and perform a **final export of the created geological section**.



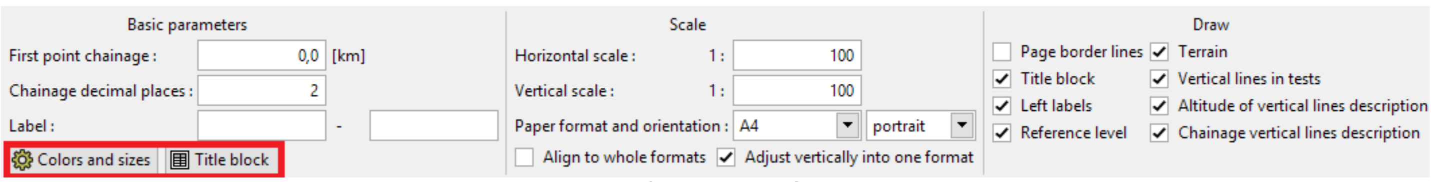
Drawing Mode

We define **drawing columns** and **location of axis** for each field test. The created settings can be used for all tests of one type using the "**OK for all tests**" button.



Definition of Field Test Description - Drawing Columns

Global drawing settings are edited in the middle part of the dialog window.



Buttons for Drawing Settings

Created drawings can be exported into **PDF, DXF, DWG, and other formats** using buttons in the right part of the dialog window. It is important to select the correct version of the format according to the CAD program, which we are using.

Geological Model

The "**Geological Model**" frame allows us to create a 3D subsoil model. The model is generated from **active boreholes** or **assigned interfaces**.

Main controls are:

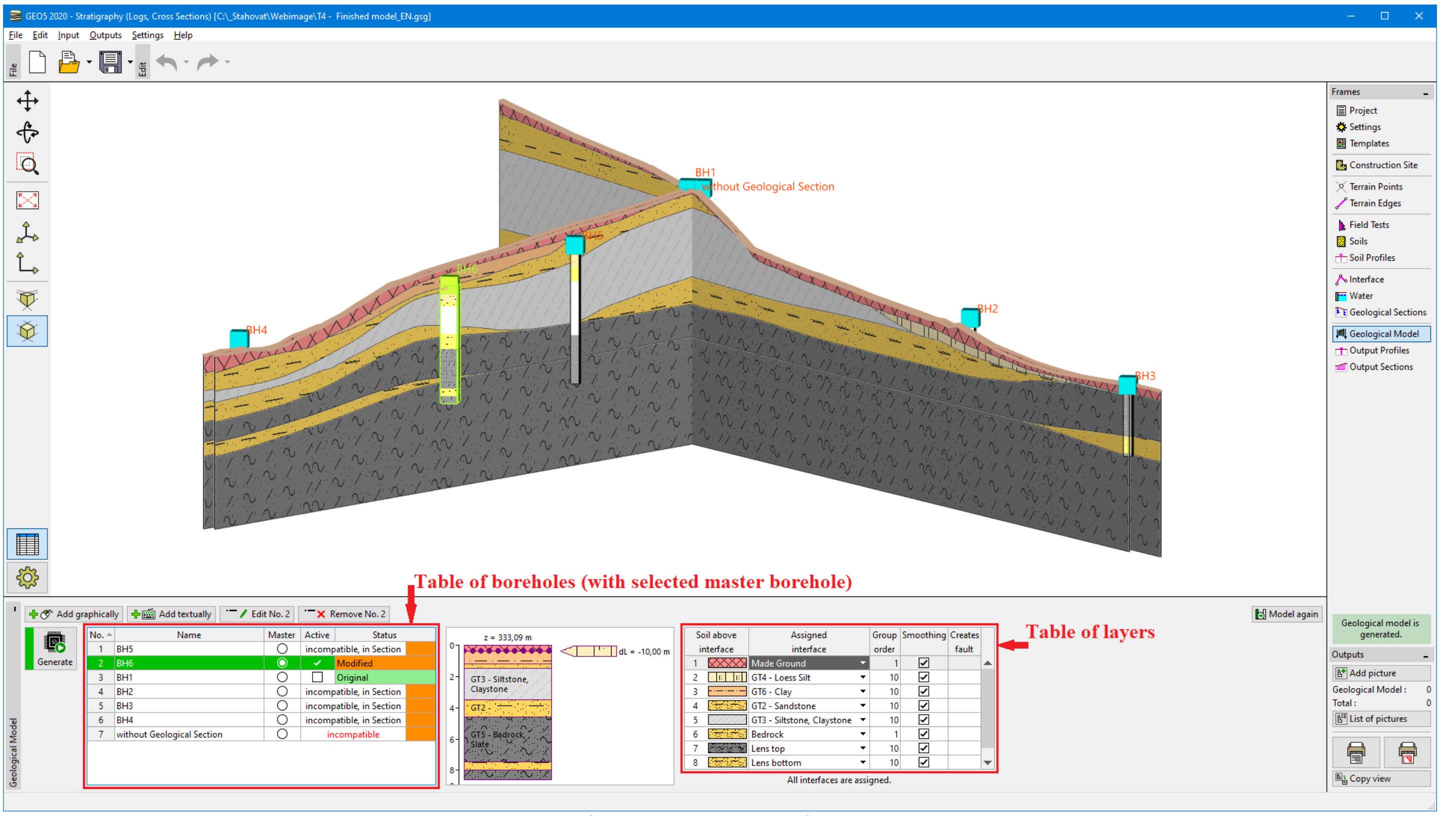
- "**Generate**" button - launches generation of a model. We recommend **a more powerful** 64-bit version for complex tasks.
- **Table of boreholes** - contains a list of boreholes, which can be added and edited
- **Visualization of currently selected borehole** displayed between the table of boreholes and table of layers (including a display of zero layers)
- **Table of layers** - contains a list of **all layers used in the mode**l, list of assigned interfaces, the order of generation, and information if the interface is smoothed or not. The content of the table corresponds to the master borehole.

Easy models can be created only from the boreholes. For models with complex geology, it is recommended to create a model from the assigned interfaces, which are generated from thegeological sections.

The workflow of modeling is shown in the engineering manuals at our websites.

- EM 40 - "Basic Work using the Stratigraphy Program"

- EM 41 - "Advanced Modeling in the Stratigraphy Program"



"Geological model" frame

"**Model again**" button - cancels the created model (In this case, all **newly created boreholeswill be deleted,** and the boreholes created from soil profiles will be **restored to the original state**).

Visualization of drawing on the desktop can be modified in any input mode based on the settings adjusted in the "Drawing Settings" frame and with the help of buttons on the "Visualization" toolbar.

Boreholes

Boreholes are (always vertical) profiles, which represent a geological profile in specific coordinates.

Boreholes can have **various** status (according to the way of creation):

- **Original borehole** - it is created automatically according to Soil profiles. Original boreholes cannot be deleted - they are always restored according to the original soil profile.
- **Changed borehole -** it is created by editing of Original borehole
- New borehole - it is created using "**Add graphically**" or "**Add textually**" buttons. It is used especially for the final modification of a 3D model.

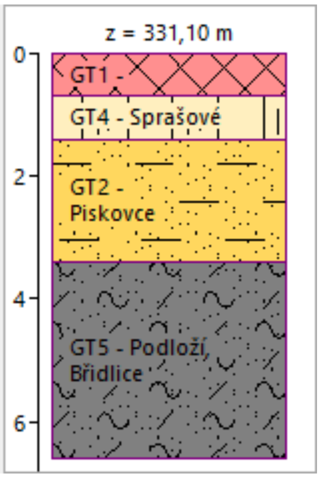
One borehole must be selected as master borehole, which determines the **number and order of soil layers** in a **geological model**. Soil layers are displayed in the table of layers.

When modeling using geological sections, it is possible to create a 3D model **with only one active master borehole**.

The boreholes **used in geological sections** are implemented in already created interfaces - they are described as "incompatible, in Section".

The boreholes which **are not used in geological sections** and are not compatible (with the master borehole) are described as "**incompatible**" in red - these boreholes should be changed to compatible boreholes and this way used in modeling.

No.	Name	Master	Active	Status
1	BH5	○	non compatible, in Section	
2	BH6	◉	✓	Modified
3	BH1	○	☐	Original
4	BH2	○	non compatible, in Section	
5	BH3	○	non compatible, in Section	
6	BH4	○	non compatible, in Section	
7	without Geological Section	○	non compatible	



When generating only from the boreholes model, it is necessary to edit boreholes to be compatible with the master borehole. Incompatible boreholes are described as "not-compatible" **in red** and are not used in model generation. /

The model is generated only from **active boreholes**. **Active boreholes** must always be **compatible with the master borehole**.

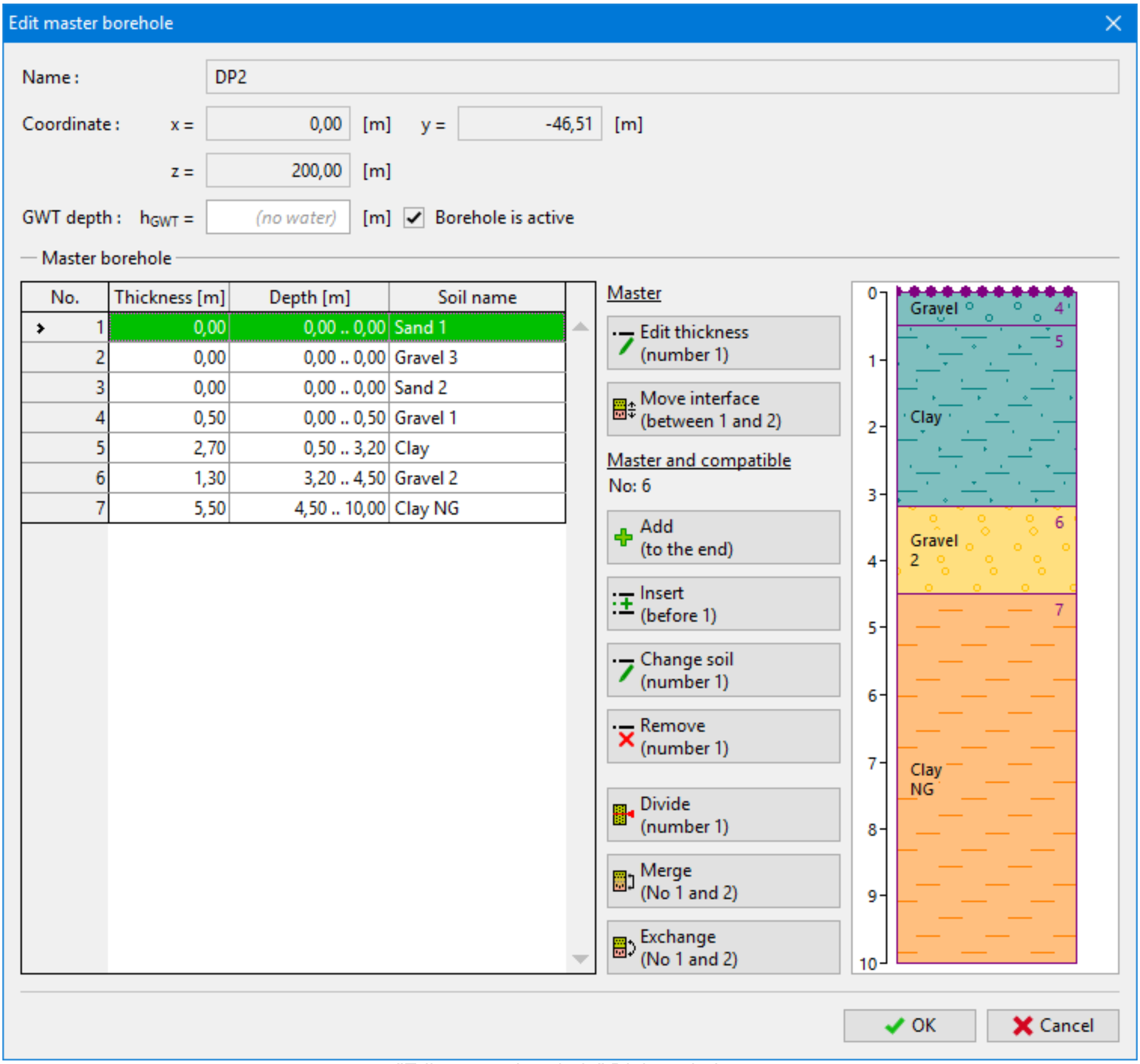
Master Borehole

The master borehole determines the **number and order of layers in the model**. **Each model** must have **one borehole** assigned to **master borehole**. The data in the Table of layers corresponds to the **selected master borehole**. When master borehole is changed, the table of layers is regenerated according to the new master borehole - again; it is necessary to assign interfaces and define an order of generation. The advantage is that each borehole can have its own table of layers defined.

The borehole with the maximum number of layers should be selected as a master borehole. The master borehole is edited during model creation to include **all layer interfaces** occurring in the model. It is necessary to add the interfaces which are not present in the master borehole. The layer interface can be defined as "not defined" - this interface is then generated according to the points from **other active boreholes** or **points from the assigned interface**.

Some modifications of the master borehole are automatically **adopted in all compatible boreholes** (adding or removing soil layer, change of soil, merging of layers...). So the compatibility of boreholes cannot be lost when editing the master borehole.

The master borehole is edited in the "**Edit master borehole**" dialog window.

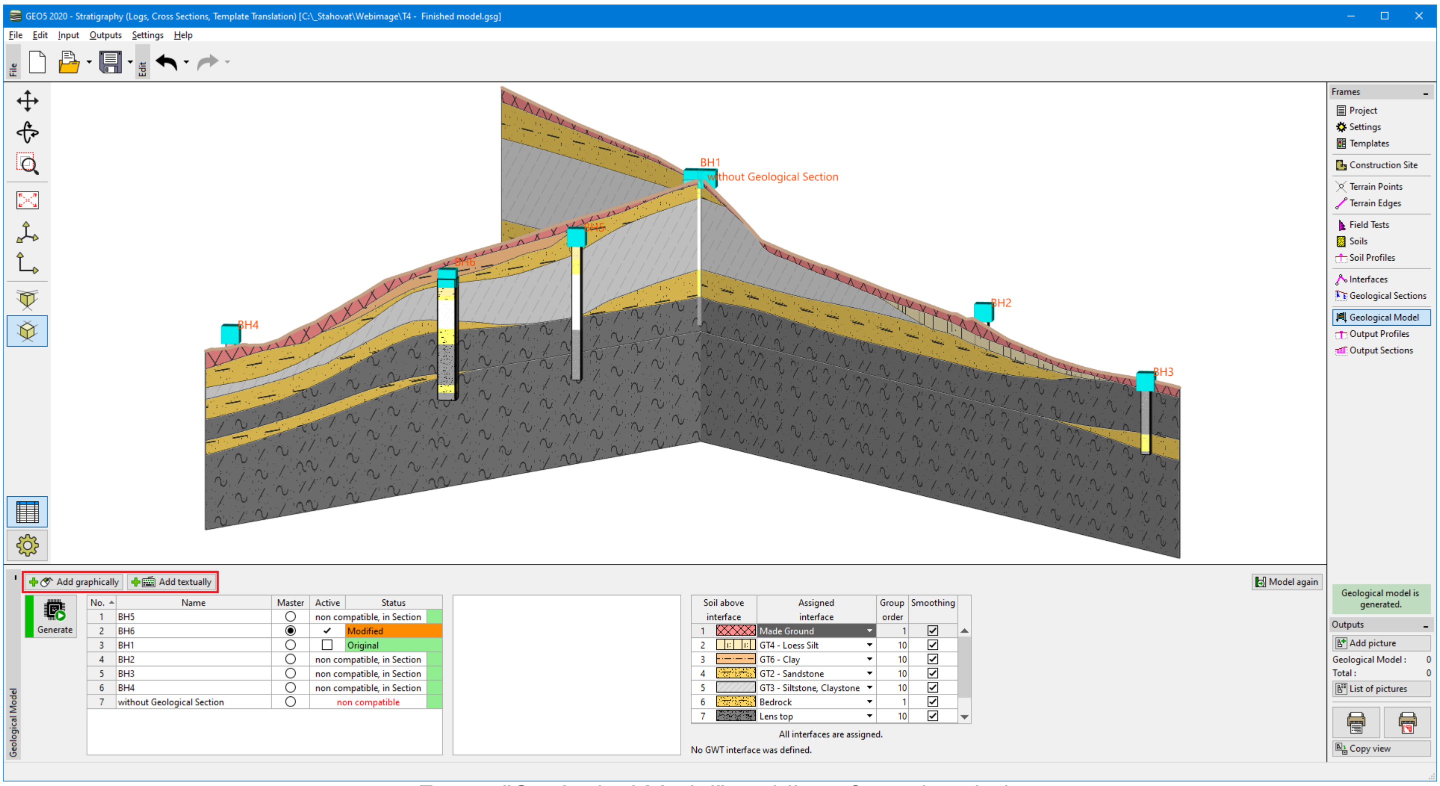


"Edit master borehole" Dialog window

Adding or removing of layers in the master borehole is automatically done also **in all compatible boreholes**. It is also possible to edit master borehole when other boreholes are edited.

Add Borehole

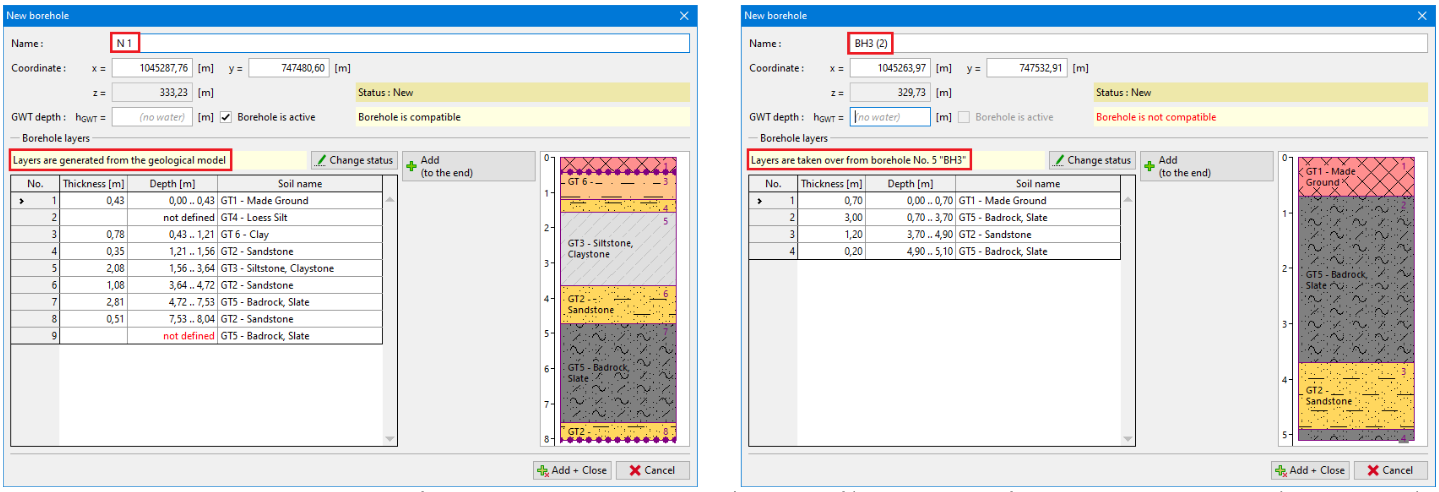
New boreholes can be added to the model using the "**Add graphically**" or "**Add textually**" buttons.



Frame "Geological Model" - adding of new borehole

If there is no borehole selected using "**Add**" button**, a new borehole is generated from the geological model** (on the left of the picture).

If there is a borehole selected, the **new borehole is a copy of the selected borehole** (on the right of the picture).

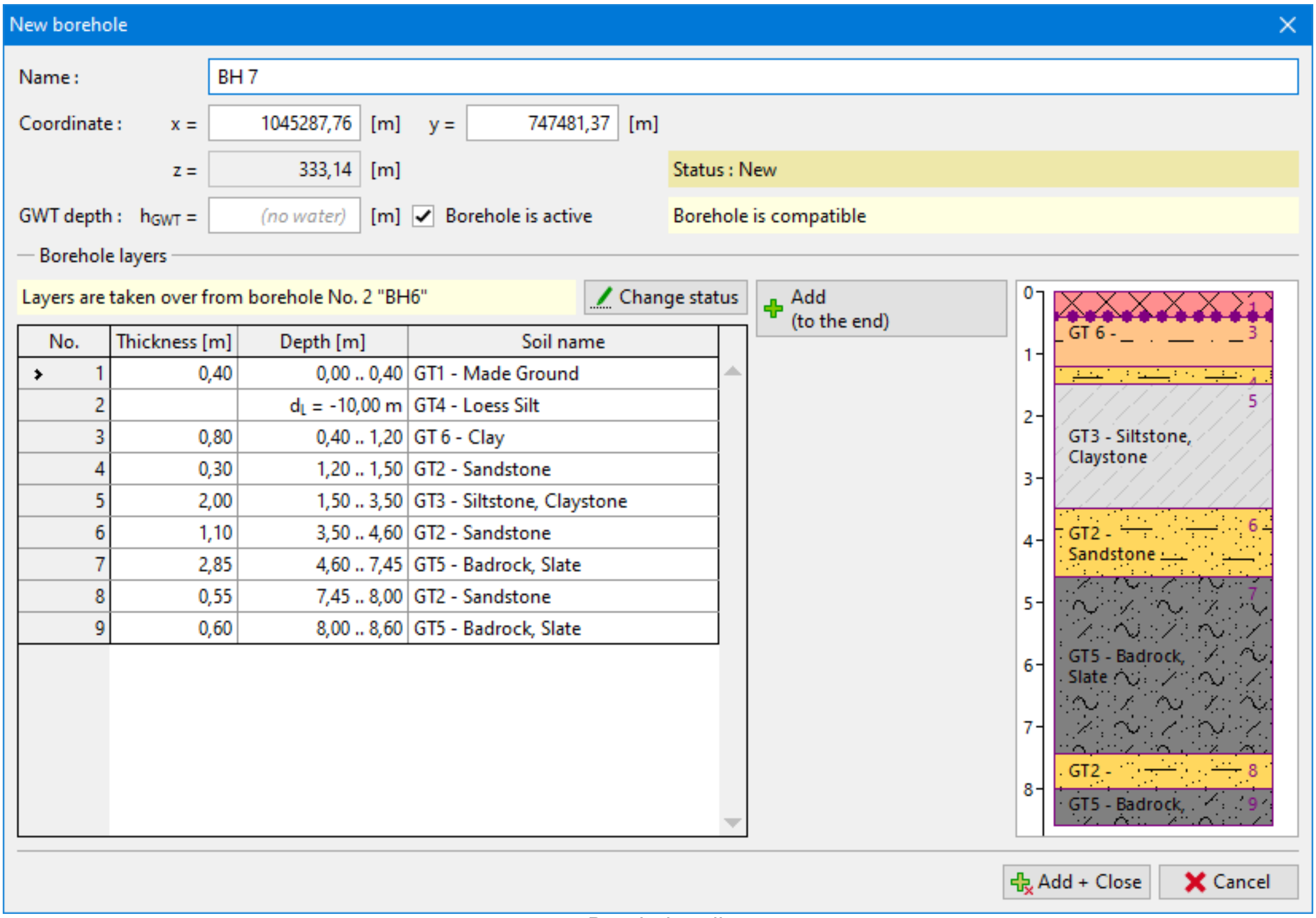


Borehole generated automatically from the geological model (on the left) and copied from another borehole (on the right)

Final geological model can be modified easily by editing of defined borehole.

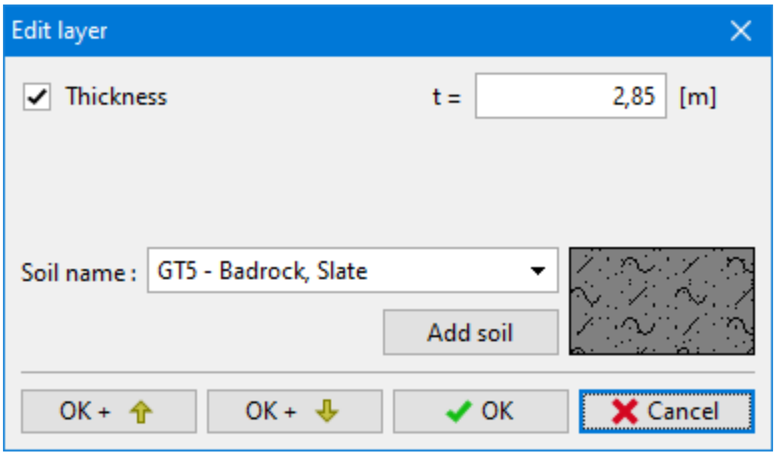
Definition of Interface in Borehole

The **thicknesses of soil layers** are entered in the boreholes - they define a **depths of interfaces between layers**. By thickness editing, the final geological model can be changed.



Borehole edit

The change of layer thickness is the most frequent way of borehole edit.



"Edit layer" dialog window

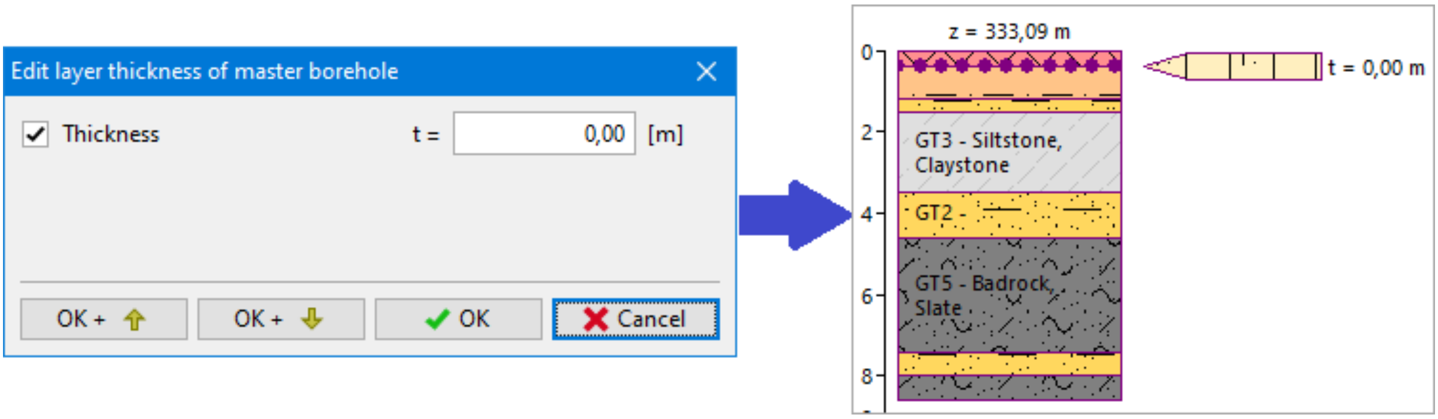
In some cases, the **layer is not within the borehole**, but it is necessary to include this layer into the model (i.e. to define an interface position in this borehole).

There are three options for defining such a layer:

- define a layer with zero thickness
- define an interface without specifying its position
- define a position of an interface anywhere in soil body or above the borehole

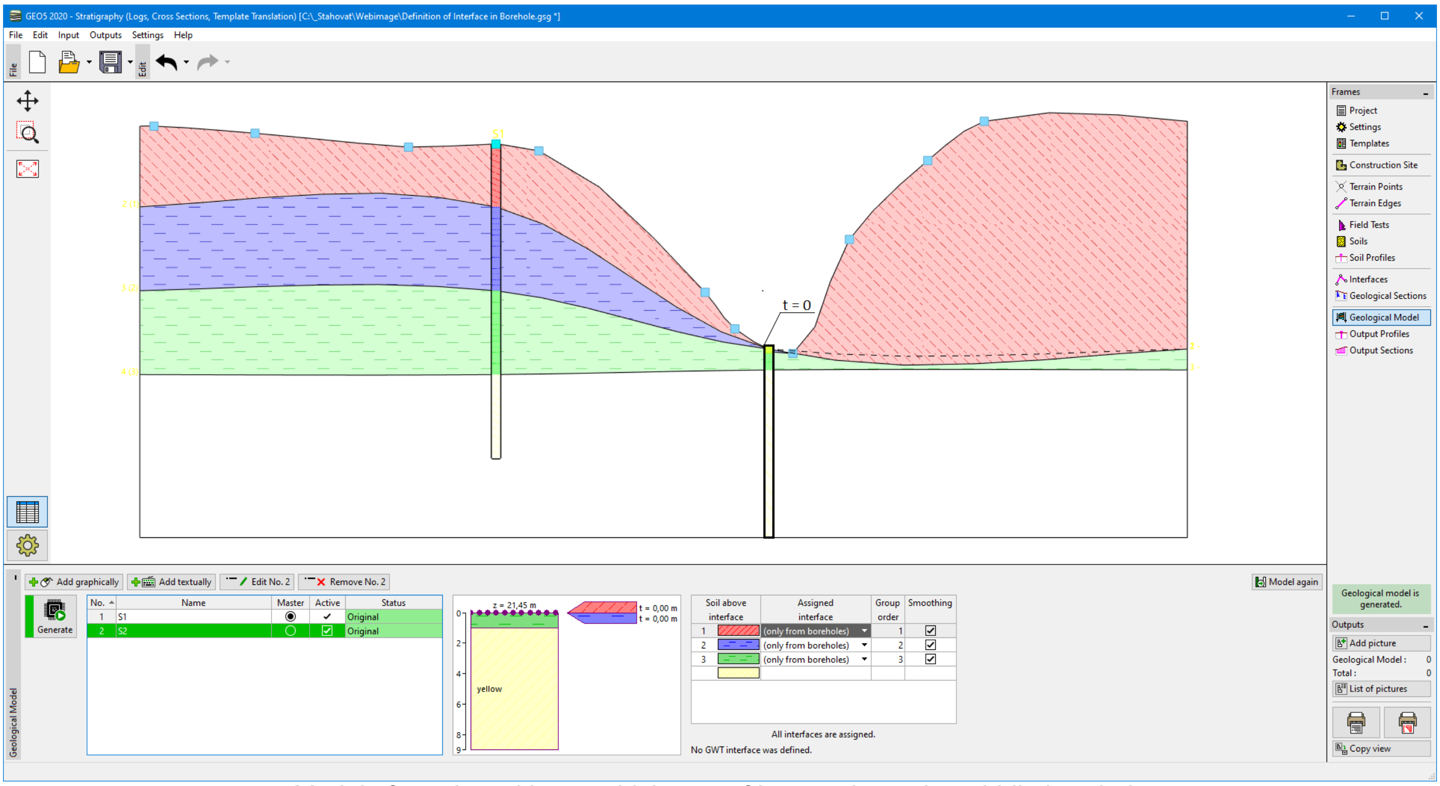
Layer with Zero Thickness

- **Edit thickness "0"** ***m*** - generation of a model is performed normally, just in the point (borehole coordinates), the **thickness layer is equal to zero**. The final model usually does not correspond to the real state.



In the picture, there is a model of erosion with layers of zero thickness above the middle borehole - the result is not realistic.

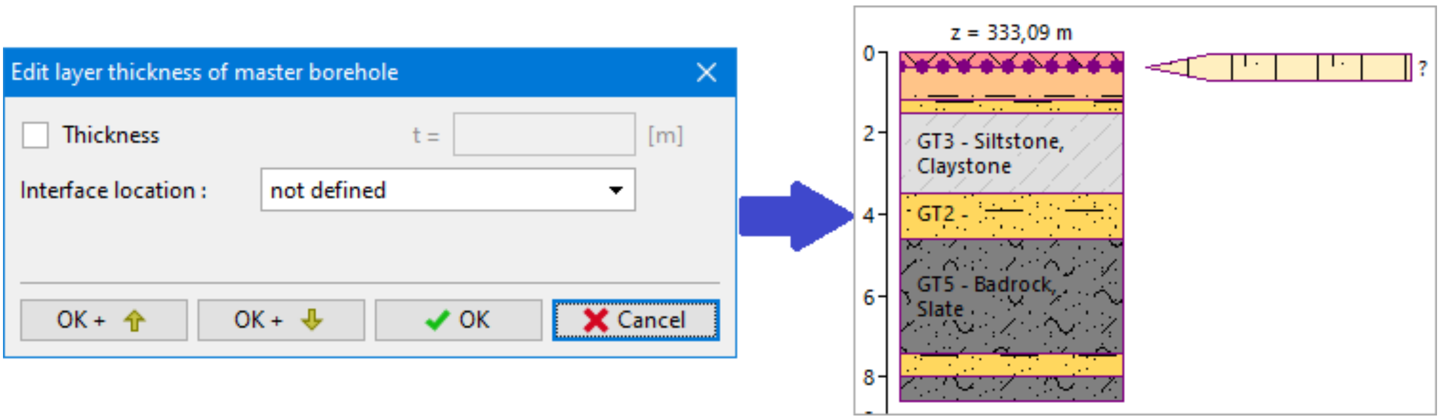
Another option is to define a depth of layer or to let the interface "not-defined".



Model of erosion with zero thickness of layers above the middle borehole

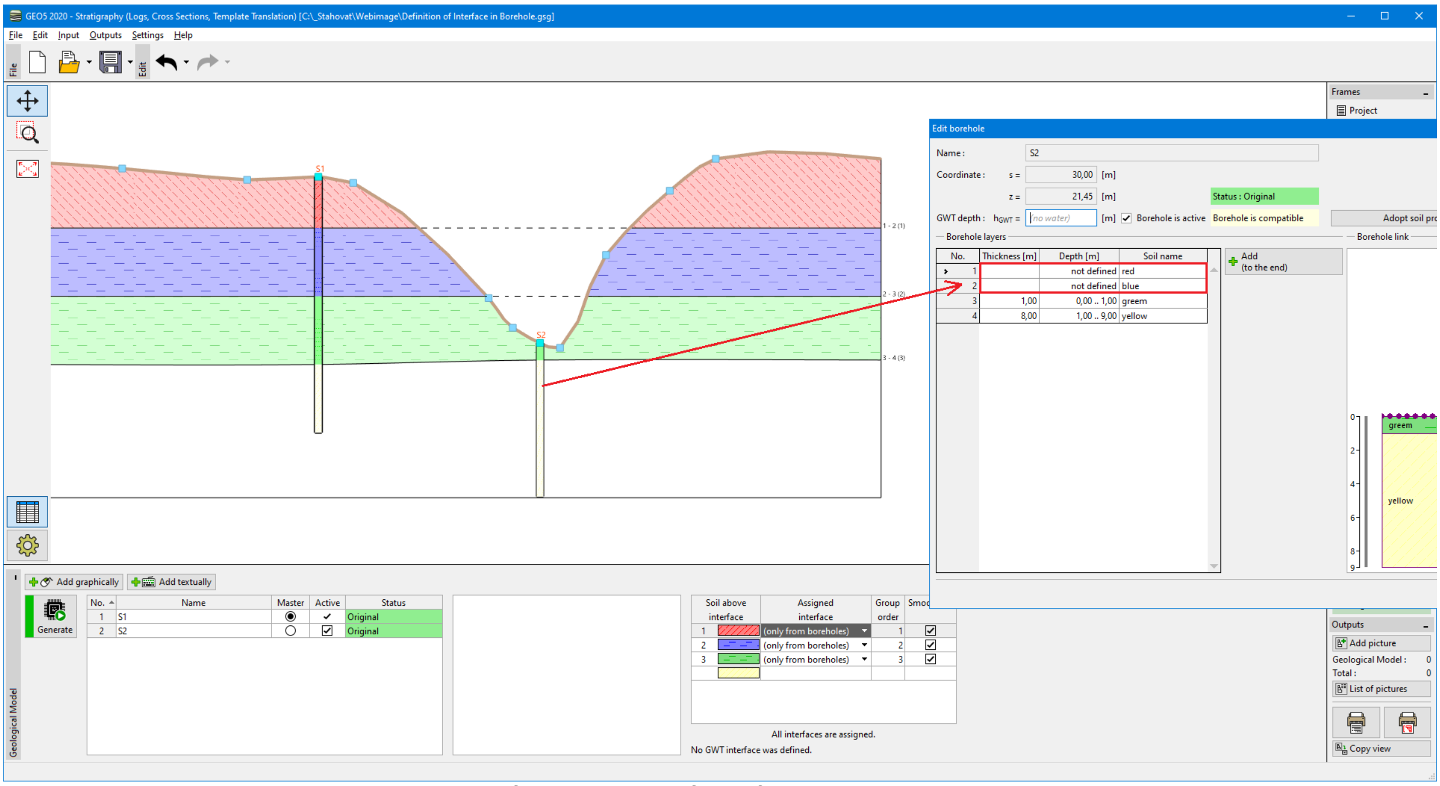
Not Defined Interface

- **"not defined" option -** we do not know the depth of the interface in the borehole. When generating a model, no points of layer interface are added in the location of the borehole. It can be used when modeling soil lenses or faults and also when the borehole is not deep enough, and we want to model only layers we know precisely. **Surrounding deeper boreholes** determine then the bottom layers.



The first two interfaces of the borehole in the middle of the model are "**not defined**". The **borehole is compatible** due to this interface input, but **interfaces are not affected by this borehole at all**.

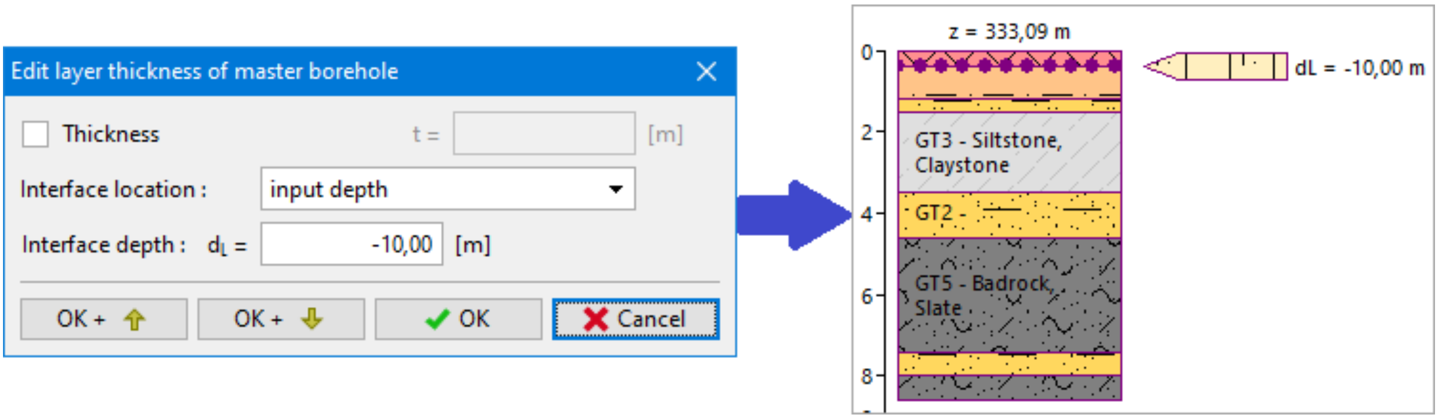
The shape of upper interfaces can be changed by the input of interface depth.



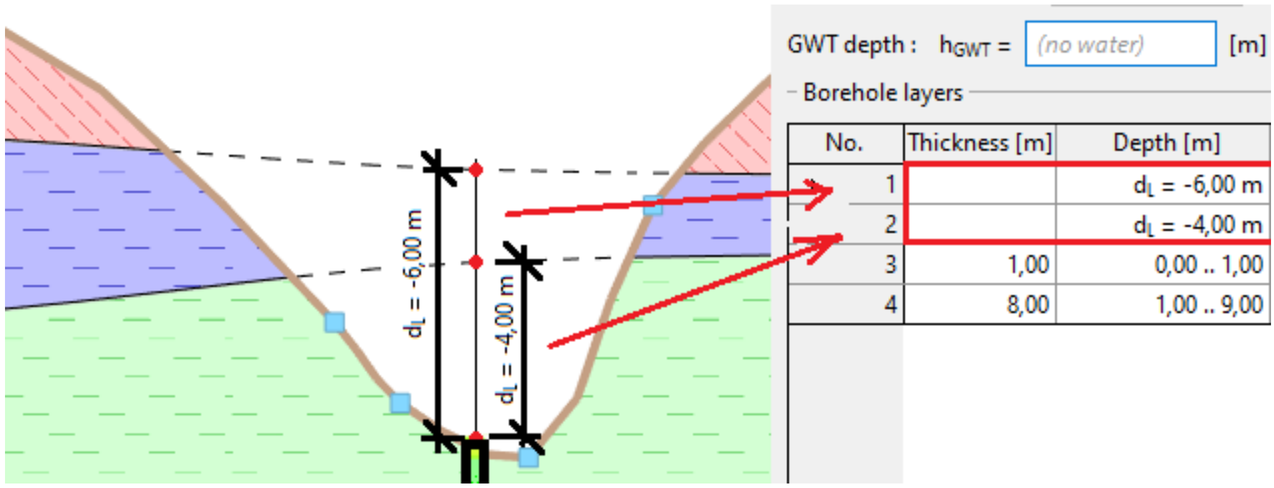
Model with not defined location of interfaces above the middle borehole

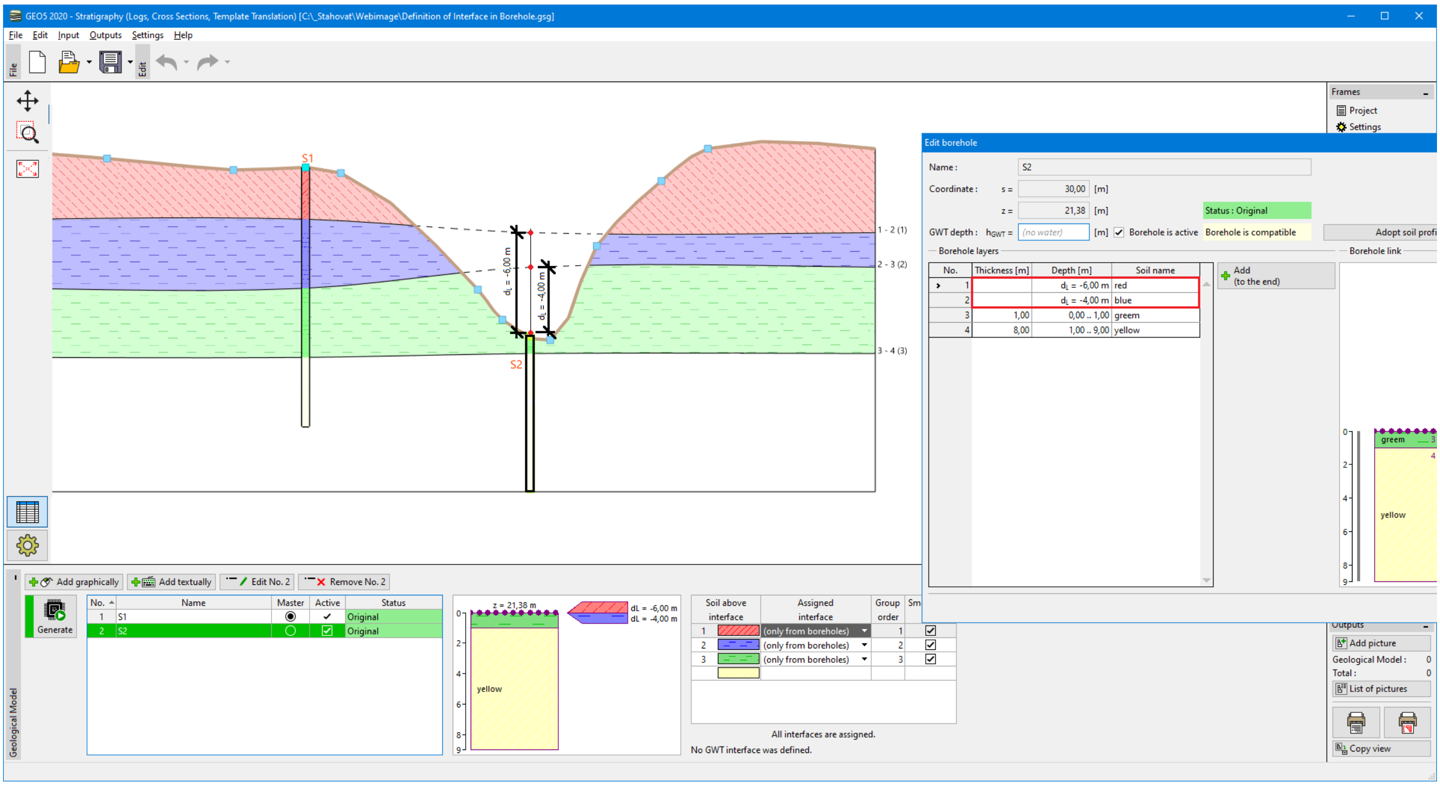
Input Depth of Interface Location

- **input depth of interface** - the depth of **fictive lower interface of the layer** is entered - the interface is generated in defined depth. The depth is measured as a **distance from the top of the borehole on the terrain, positive values downward to soil body, negative upward above the terrain.**



In the picture, we see the **edit of layers above the borehole** using input of **distance d_L** from the top of the borehole.





This way of input is used when modeling soil lenses. Using input of interface point **above the terrain** or **deeply to the soil body,** we ensure the correct generation of soil lense interface.

Table of Layers

The table of layers contains:

- **List of soils** above the interfaces
- Assigned interfaces for model generation using geological sections
- **Order** of interface generation. It is important for the final form of the geological model. Default group order is set as "10" to make further changes easy.
- Information on whether the interface is **smoothed** or not.
- Information on whether the interface is a "Fault" type.

Soil above interface	Assigned interface	Group order	Smoothing	Creates fault
1	Made-up ground	10	☑	
2	Silt-Clay	10	☑	
3	Fault	1		✓
4	Made-up ground behind the fault	10	☑	
5	Weathered shale	10	☑	

All interfaces are assigned.

Table of layers

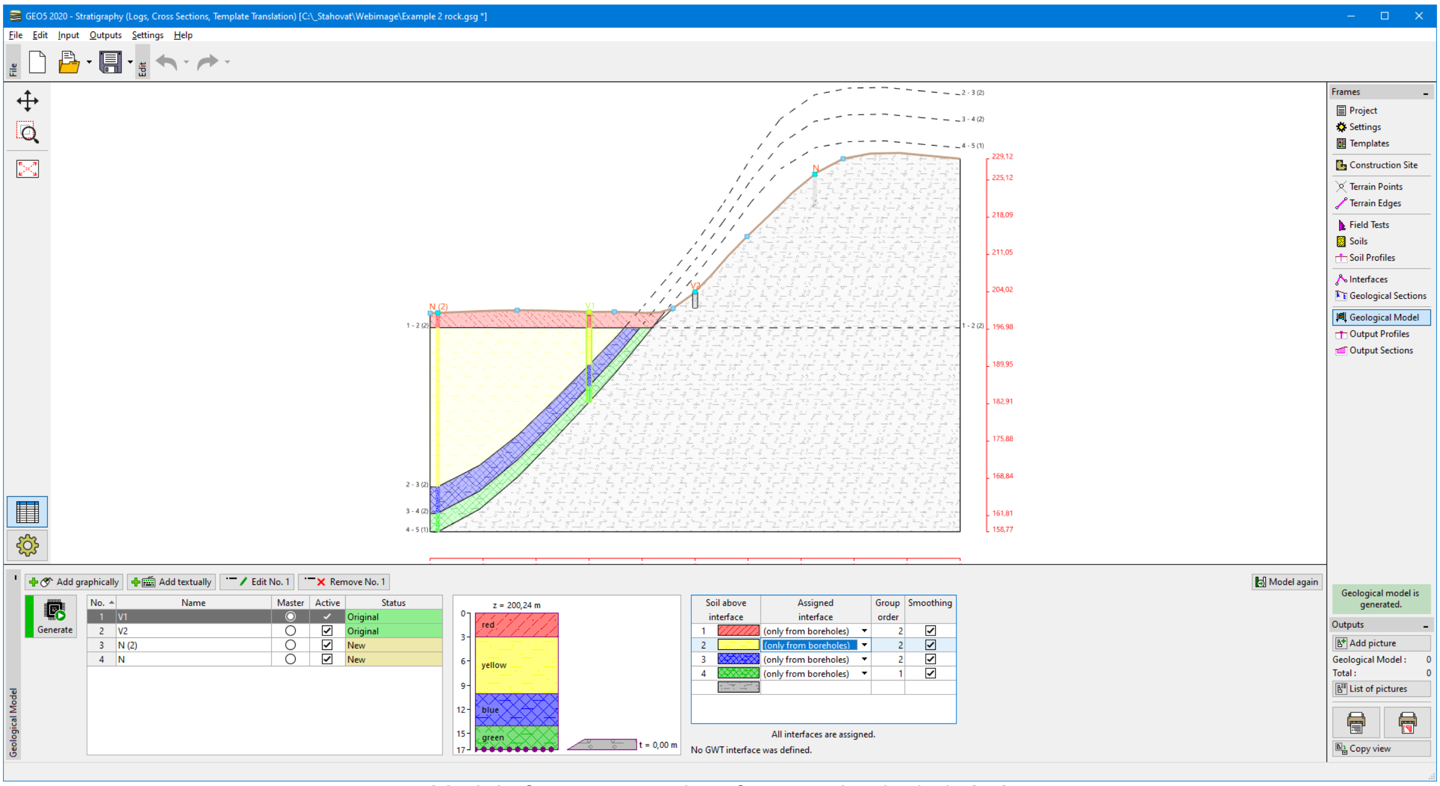
Order of Layer Generation

After terrain generation, interfaces between layers are generated:

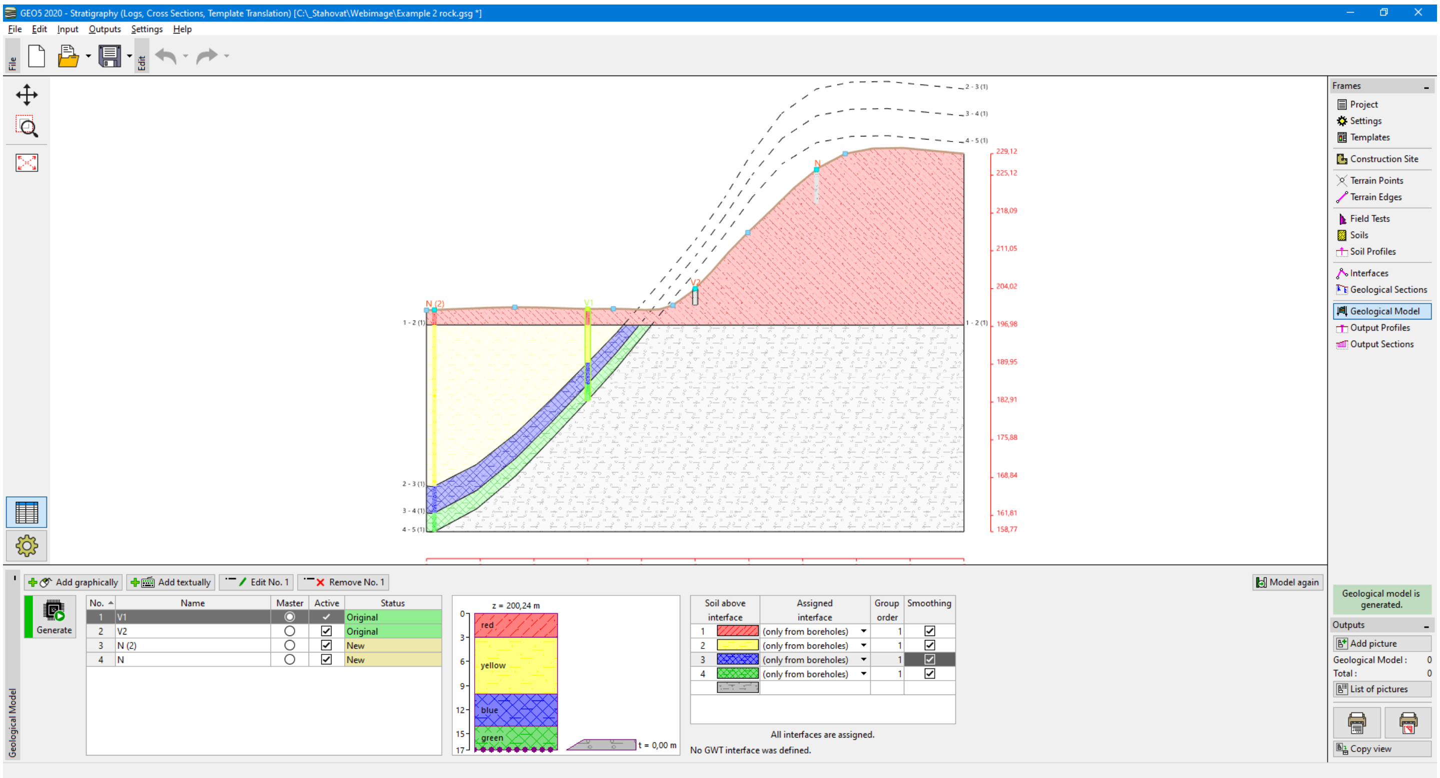
- first the interfaces with the **lowest group order number**
- for the same group order number, interfaces are generated **from the top** to the bottom

The generated interface is **"cut" by terrain** or **previously generated interfaces**.

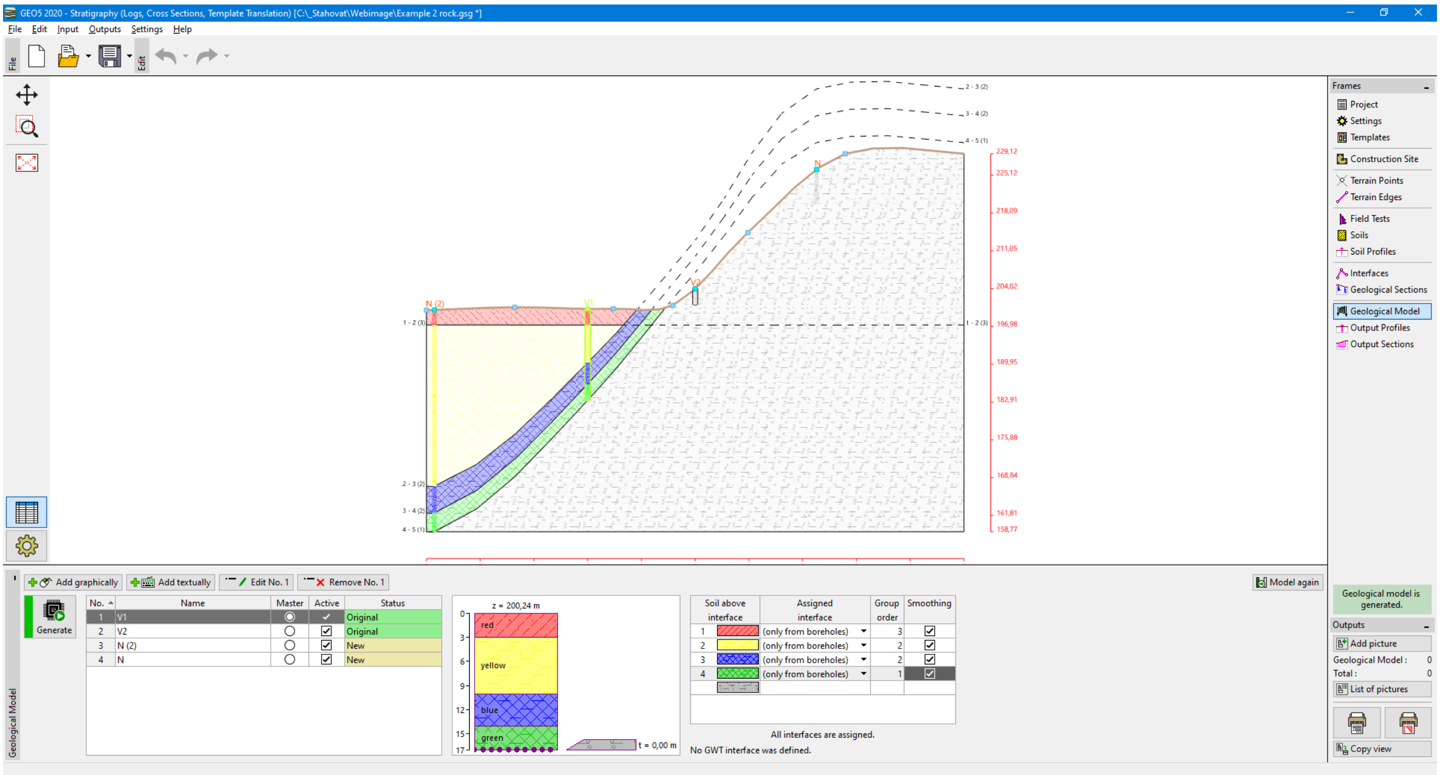
Group order of generated layers is significant when creating a model - a typical example is a rock covered by layers of sediments.



Model of outcrop - order of generation is 4, 1, 2, 3



The same model with the original order of layer generation - 1, 2, 3, 4



The same model with the changed order of layer generation 4, 2, 3, 1

This way, it is possible to model more individual models within one task - individual models are separated by faults generated as first - then the other layers are generated.

Creation of the 3D Model from Interfaces

The basic input for creating a subsoil model is the master borehole and **assigned interfaces** in the table of layers.

The **workflow for modeling** is as follows:

In the Geological Sections frame, we create **geological sections** and:

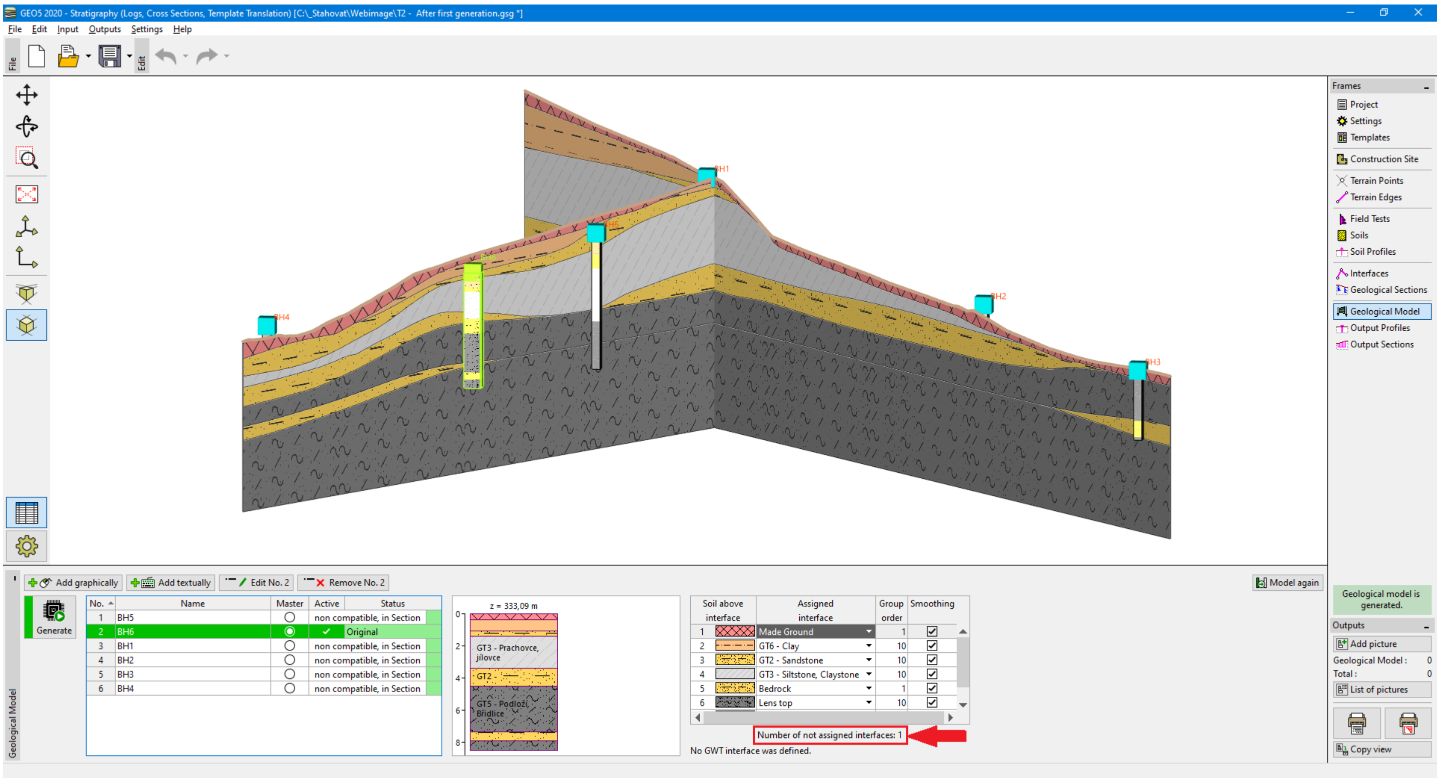
- Define lines representing interfaces between layers

- Edit soil profiles
- Create interface points, which can be edited in the "Interface" frame.

In the **Geological Model** frame, we select the master borehole. We always select a borehole **with the biggest number of interfaces** (ideally all).

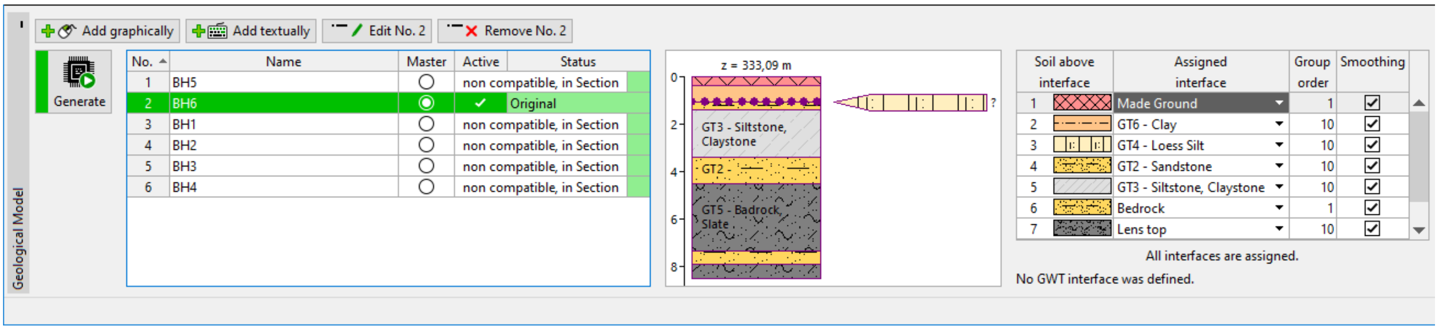
In the table of layers, we assign individual interfaces and define the order of layers generation. The interface represents a bottom border of the displayed soil. The **continuous interfaces** must be generated **as first**, then the other interfaces connected to continuous interfaces - e.g., soil lenses.

If not all defined interfaces are visible in the table of layers, it is necessary to edit master borehole and assign missing interface. The number of **not assigned interfaces to the model** is displayed below the table of layers. When all interfaces in the table are assigned, the information below the table is changed to "All interfaces are assigned".



Model with one not assigned interface before edit of the master borehole

The layers with zero thickness are displayed on the visualization of a borehole.



After each model generation, it is appropriate to compare **generated Output Sections** of model with **defined Geological Sections** to check if the 3D model was created correctly.

The final model can be easily edited by adding new boreholes and changing their layers thicknesses.

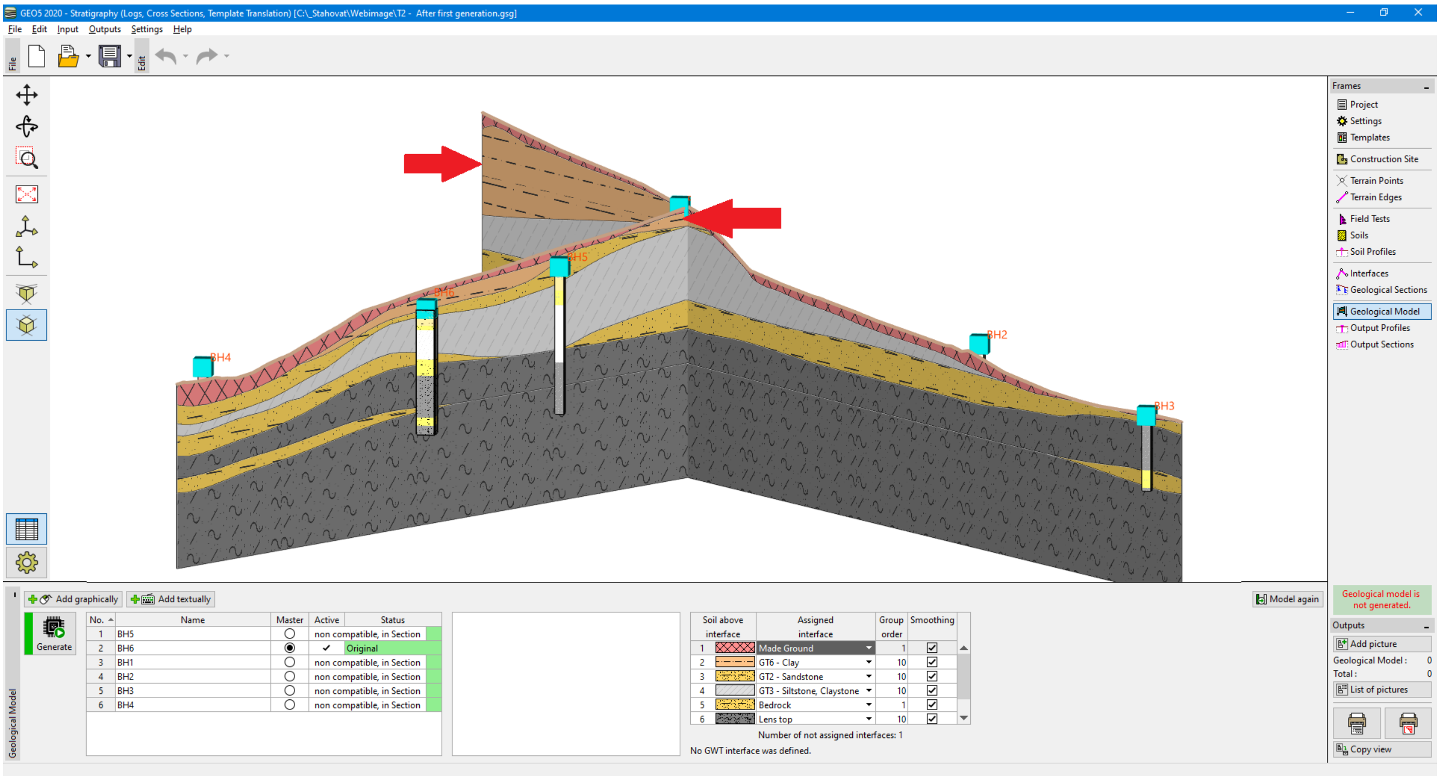
Check of the Final 3D Model

A correctly created subsoil model **meets all boundary conditions** - corresponds to defined Geological sections and Soil profiles.

Then best way how to check results is to switch over between Geological sections and Output sections and look for differences. **Output sections are automatically generated** in the locations of Geological sections. It is necessary to set correct visualization on the desktop for each frame.

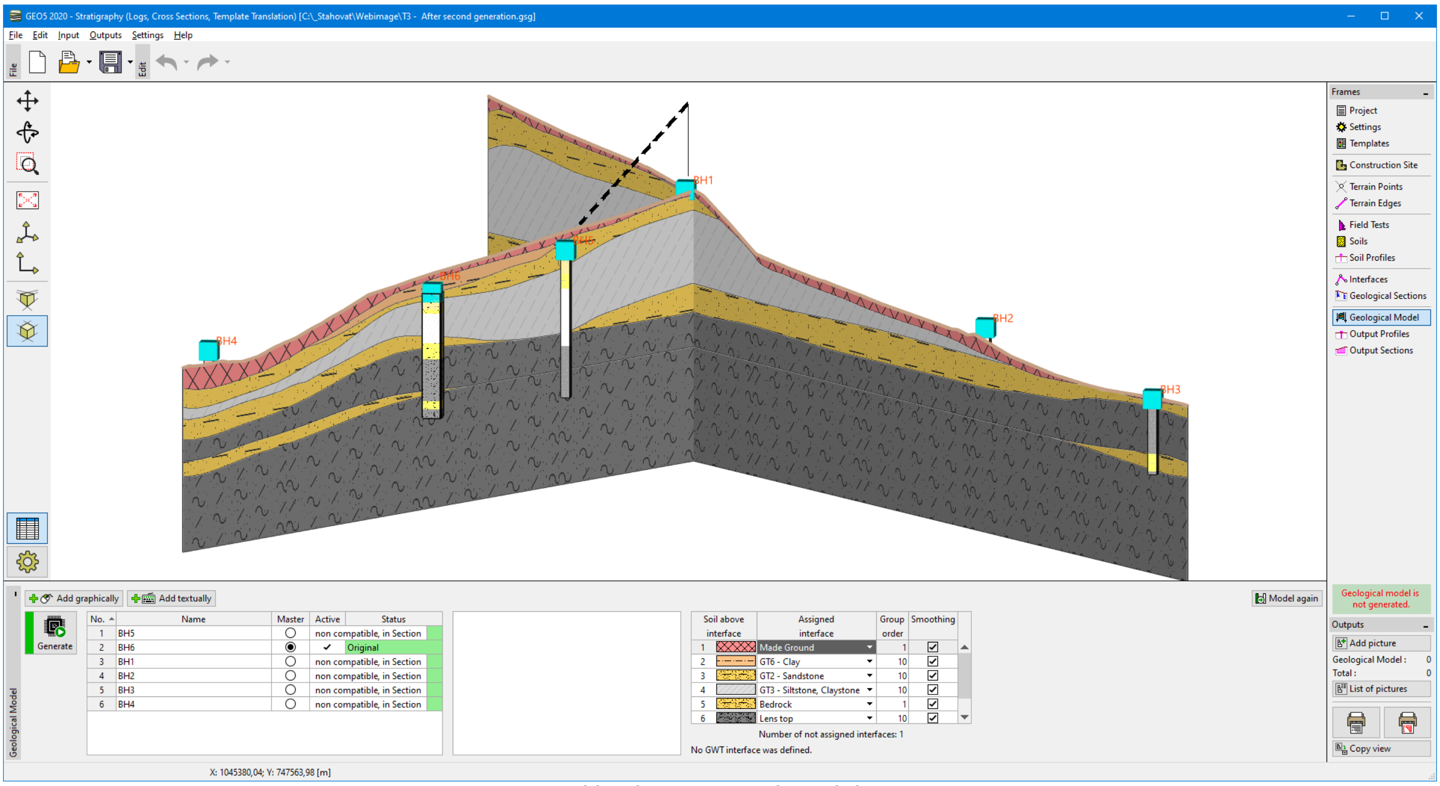
Displayed geological sections - input data for subsoil modeling

In the picture, the layer of clay is a wrongly generated. In this case, it was caused by defining an interface only on the part of the model. For this reason, we add a position of this interface above the borehole **BH1**, too.



Wrong generation of a clay layer

After the new generation, the layer of clay is created correctly.



Newly generated model

Creation of the 3D Model from Boreholes

The Geological model can be created **only from Boreholes**, but this way is more complicated than modeling from interfaces.

One borehole is always set as a master borehole. A geological model is generated from the master borehole and active compatible boreholes.

When modeling a geological model, the boreholes are subsequently edited. It is necessary to change all incompatible boreholes to a compatible boreholes performing changes of soil layers. Compatible boreholes are added to the geological model.

The new boreholes can be used for the modification of the 3D subsoil model.

The following examples show the **basic procedures** for the **creation and modification of the geological model**.

- Process of Creation of a Geological Model
- Creation of a Terrain Model
- Geological Model with Horizontal Layers
- Geological Model with Layers Following the Terrain
- Construction Site Edges - Active Edge
- Editing of Soil Layers
- Adding a New Layer into the Model

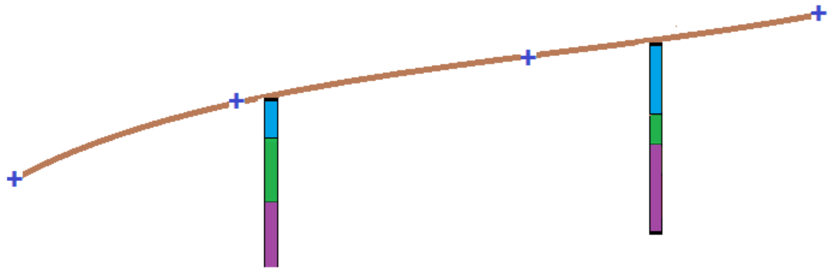
The "Undo" button is an important program tool. It allows us to return back to the **original state.**

Process of Creation of Geological Model

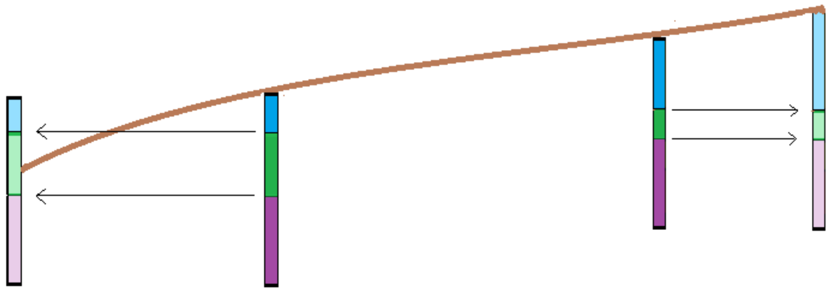
Next example

It is necessary to know a procedure of a geological model generation for a correct understanding of the creation of the geological model.

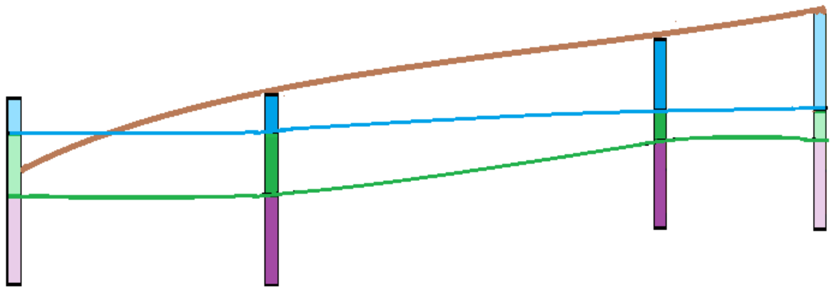
1. First, the digital terrain model is created from the terrain points and tests (exploration points) with **defined** z **-coordinate**. Then, the boreholes are placed into the model.



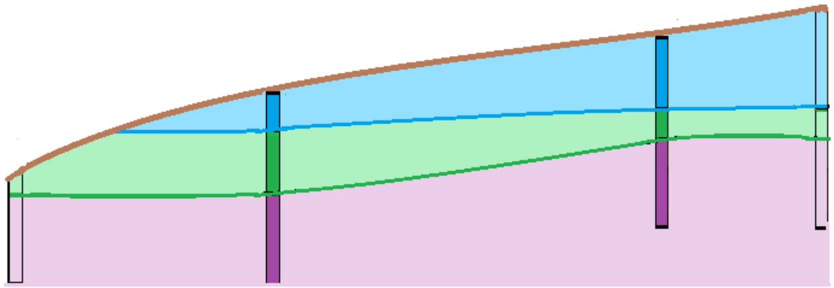
2. **Assistant boreholes** corresponding to the closest **active borehole** are automatically generated to all corner points of the "construction site". On the boundaries of the construction site, **all boreholes** from the active edge are also projected.



3. The **soil layers** and groundwater tables are generated from the boreholes and assistant boreholes (in points on construction site boundaries).



4. The **layers are verified if they do not intersect the terrain** (or other layers). The layer above the terrain is **cutted** to follow the terrain.



d

Next example

Creation of a Terrain Model

Previous example Next example

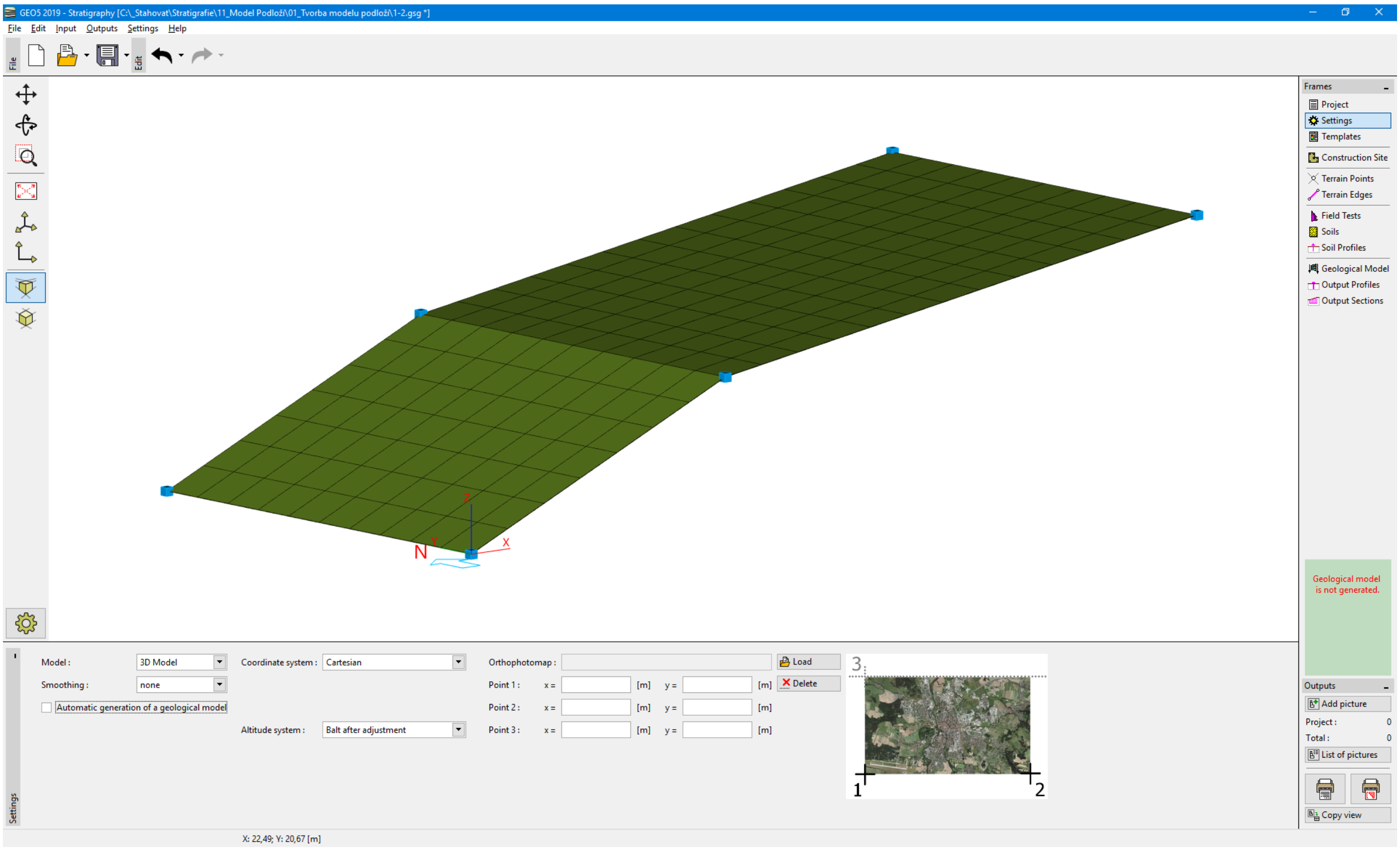
The creation of a **digital terrain model** is the first step for most tasks. The digital model of the terrain is created in the construction site dimensions from the **entered** terrain points.

The terrain points can be entered or imported. The terrain is generated automatically after each change of the entered points. The shape of the terrain is also influenced by the field tests with **defined** z **-coordinate**. When modeling slopes and broken terrains, the input of the active edge is important.

In this example, the slope modeled from six points [x ; y ; z] is shown: [0; 0; 0], [0; 10; 0], [7; 0; 3], [7; 10; 3], [20; 0; 5], [20; 10; 5].

Terrain with medium smoothing

The shape of the model can be greatly influenced by **smoothing** the surfaces between the triangles. The smoothing is entered in the frame "Settings". The above model was designed for "**intermediate**" smoothing. If the smoothing is set to "**none**", the model looks as follows.



Terrain without smoothing

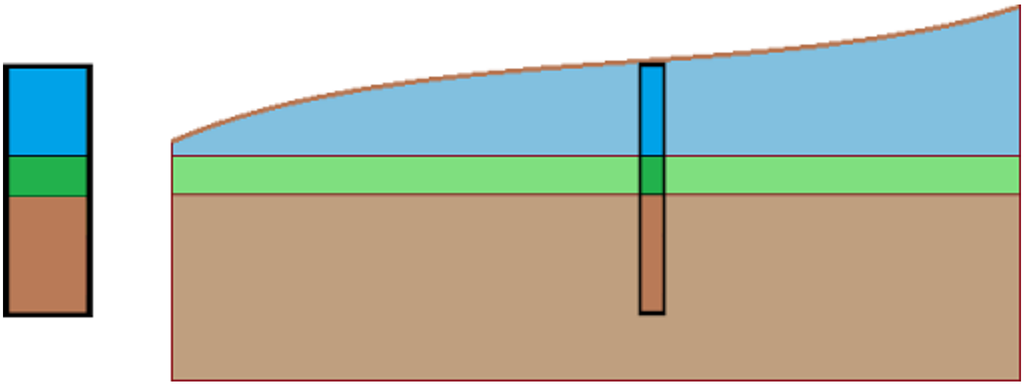
Larger smoothing allows us to create more realistic models, but the generation can be slower in case of higher number of layers. Sometimes, for a larger model, it is recommended to create the entire model without smoothing and turn it on when output documentation or cross sections are created.

Previous example Next example

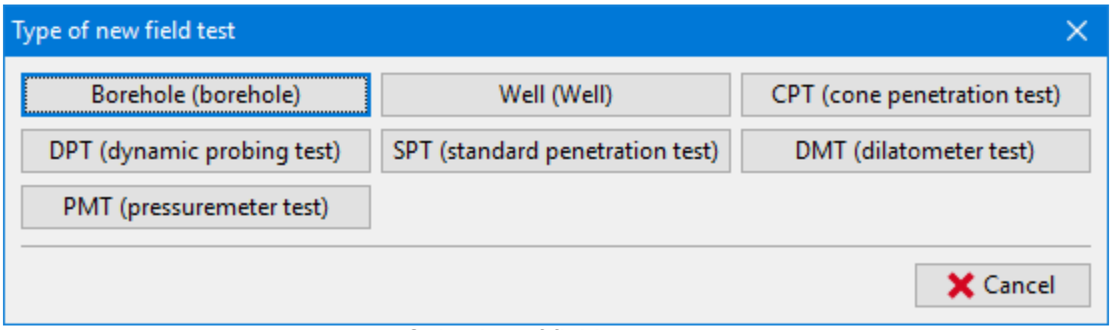
Geological Model with Horizontal Layers

Previous example Next example

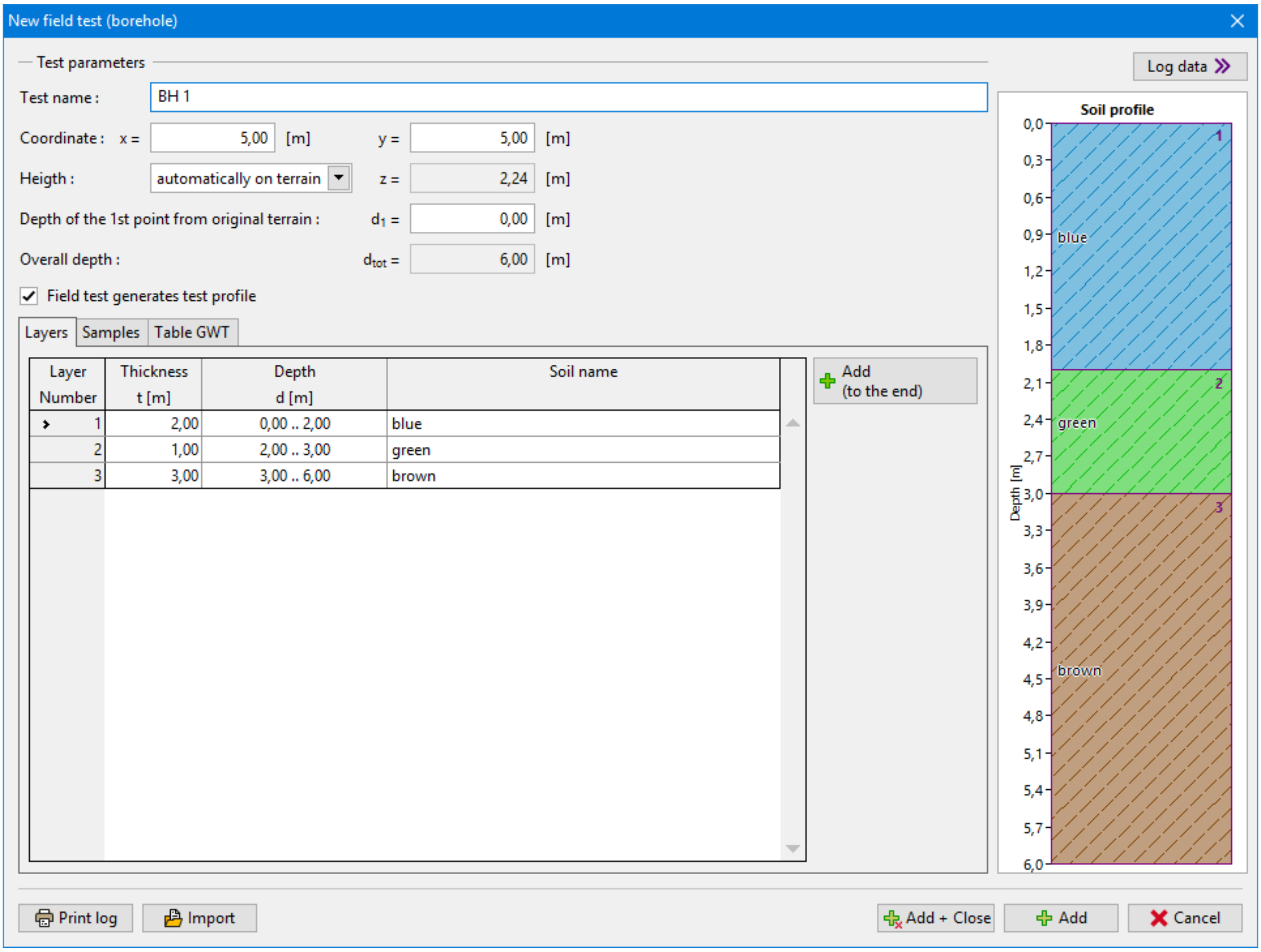
We will create a geological model with horizontal layers according to the following picture.



First, we enter a field test (type borehole) in the "Field Tests" frame - the name **BH 1**, coordinates [x ; y ; z] - [5; 5; on terrain], thickness of layers [2; 1; 3].

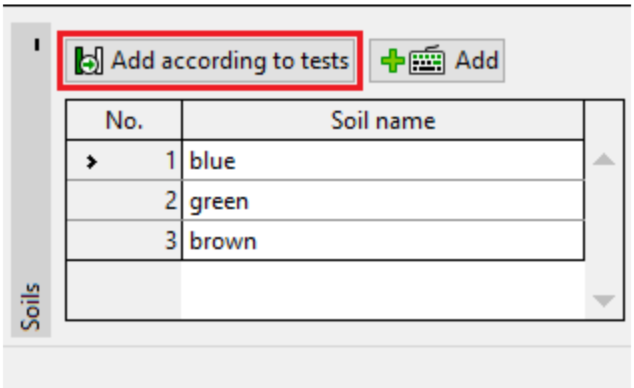


Selection of field test type



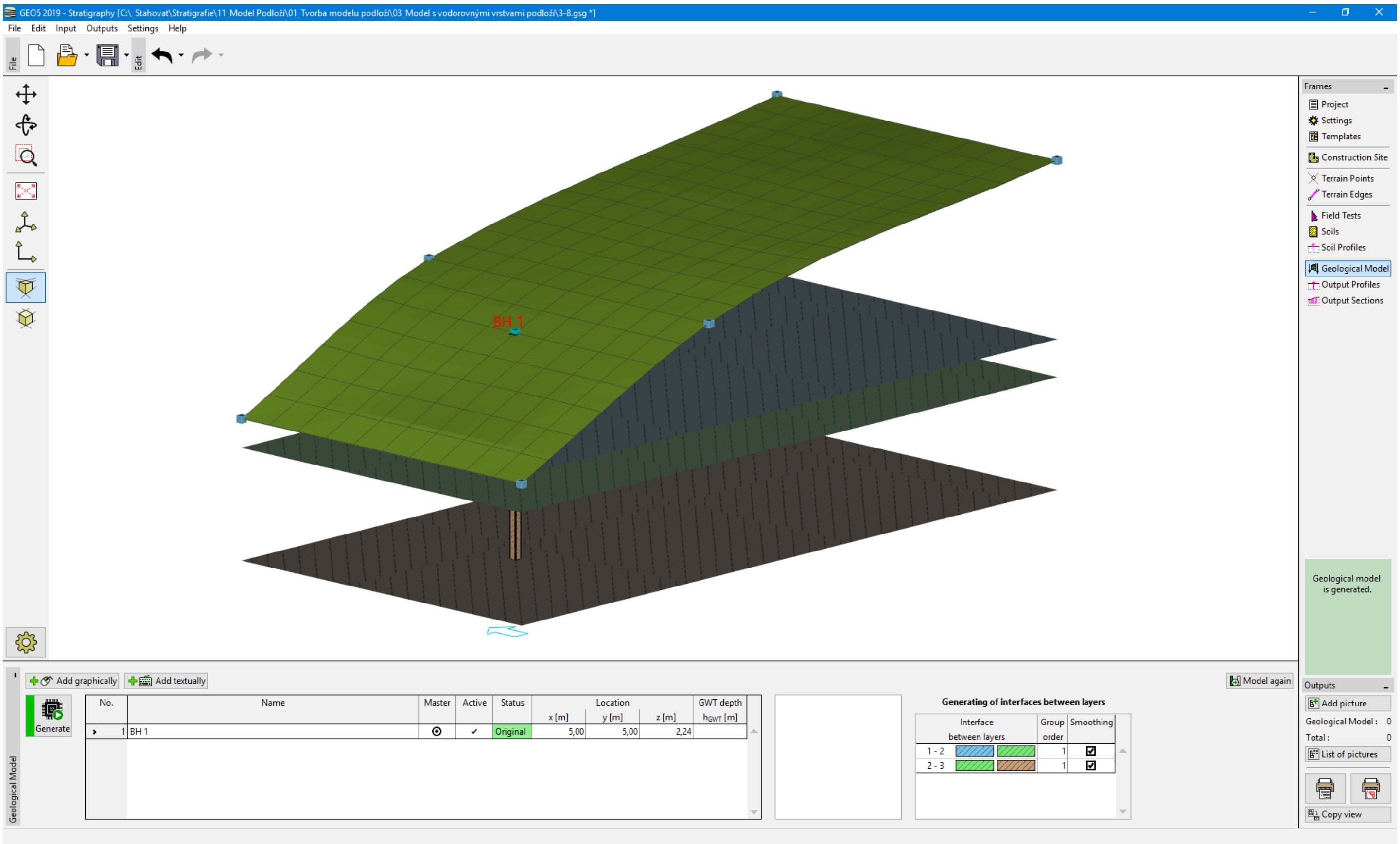
Borehole input

The **soils** [**blue**, **green**, **brown**] can be added directly in the window "**layers of borehole**" or in the "Soils" frame. The easiest way is to copy all soils from the tests using the "**Adopt from field tests**" button. The list of soils used in tests is created.



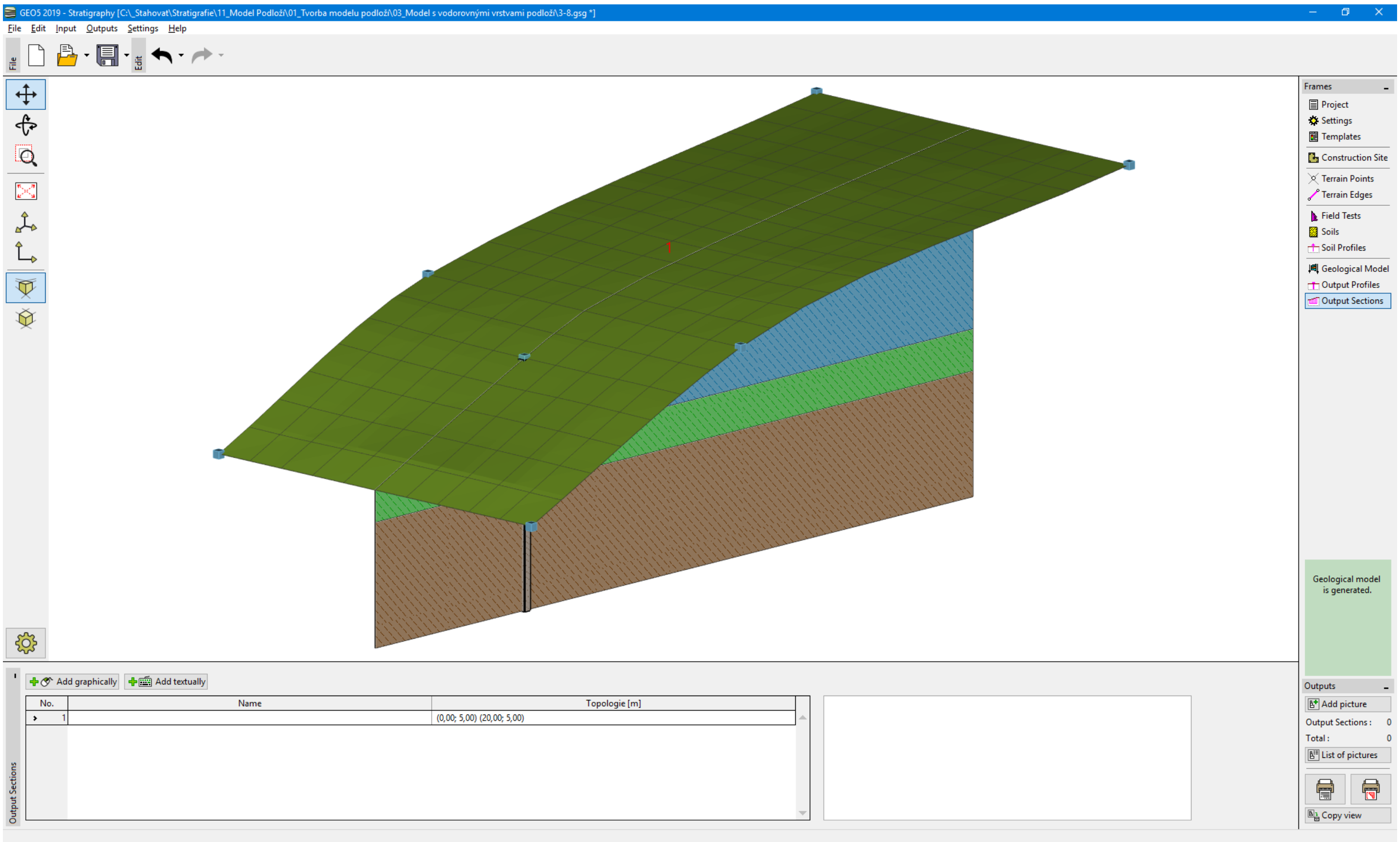
Adding soils according to tests

The soil profile and borehole are created automatically **from the test (type borehole)**. After switching to the "**Geological model**" frame, the required model is generated (if the manual generation is selected in the "Settings" frame, it is necessary to use the button "**Generate**").



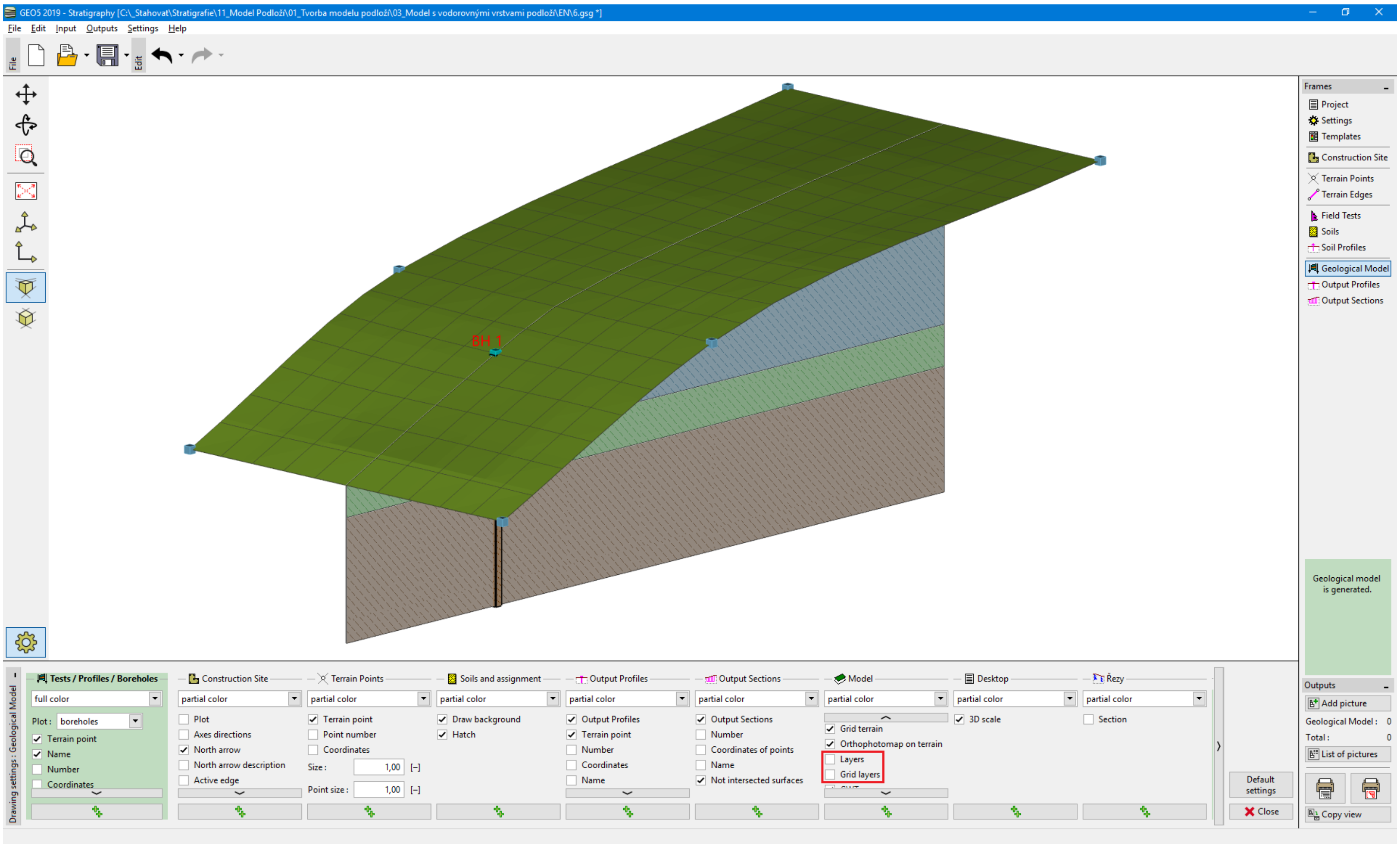
Generated model

For a clearer view, it is appropriate to enter a cross section in the frame "Output Sections" - points [x ; y] - [0; 5], [20; 5].



Visualisation of the model using Cross section

It is necessary to change the drawing settings for correct visualization of cross section in the "**Geological Model**" frame.



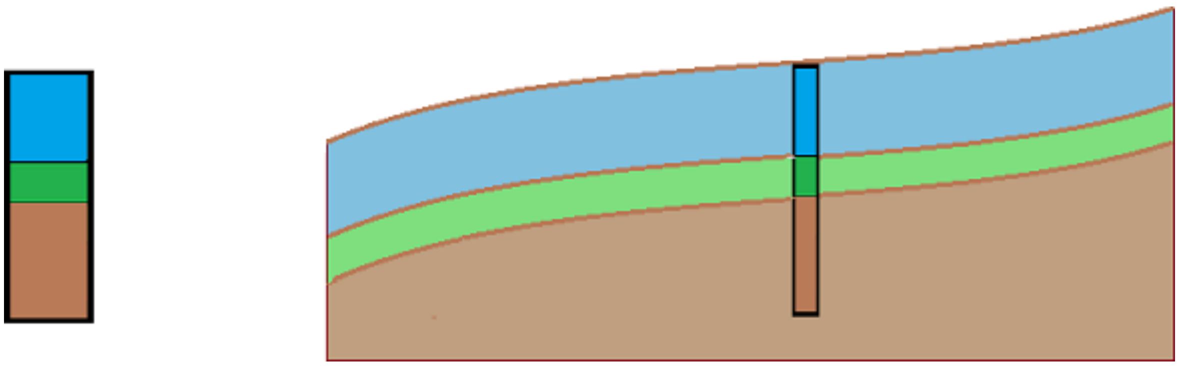
Frame "Geological model" - drawing settings

Previous example Next example

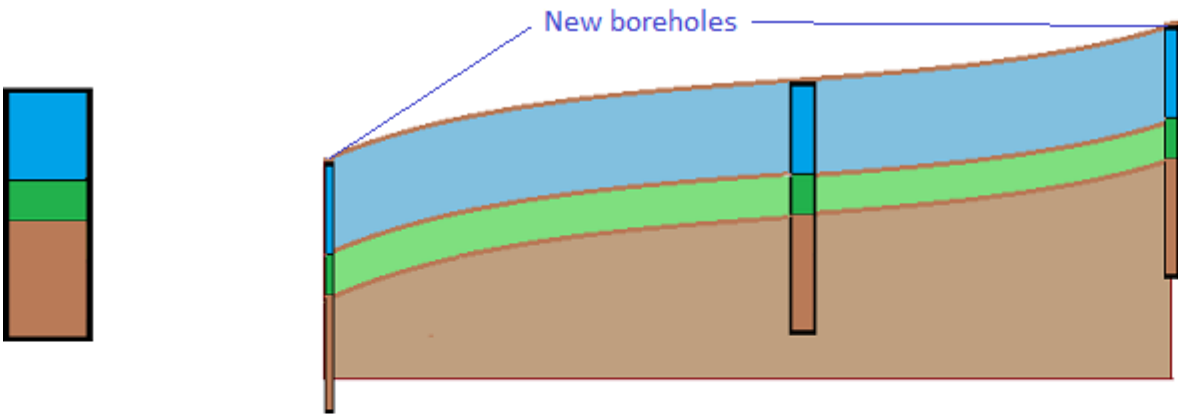
Geological Model with Layers Following the Terrain

Previous example Next example

Now we create a geological model with the layers following the terrain.



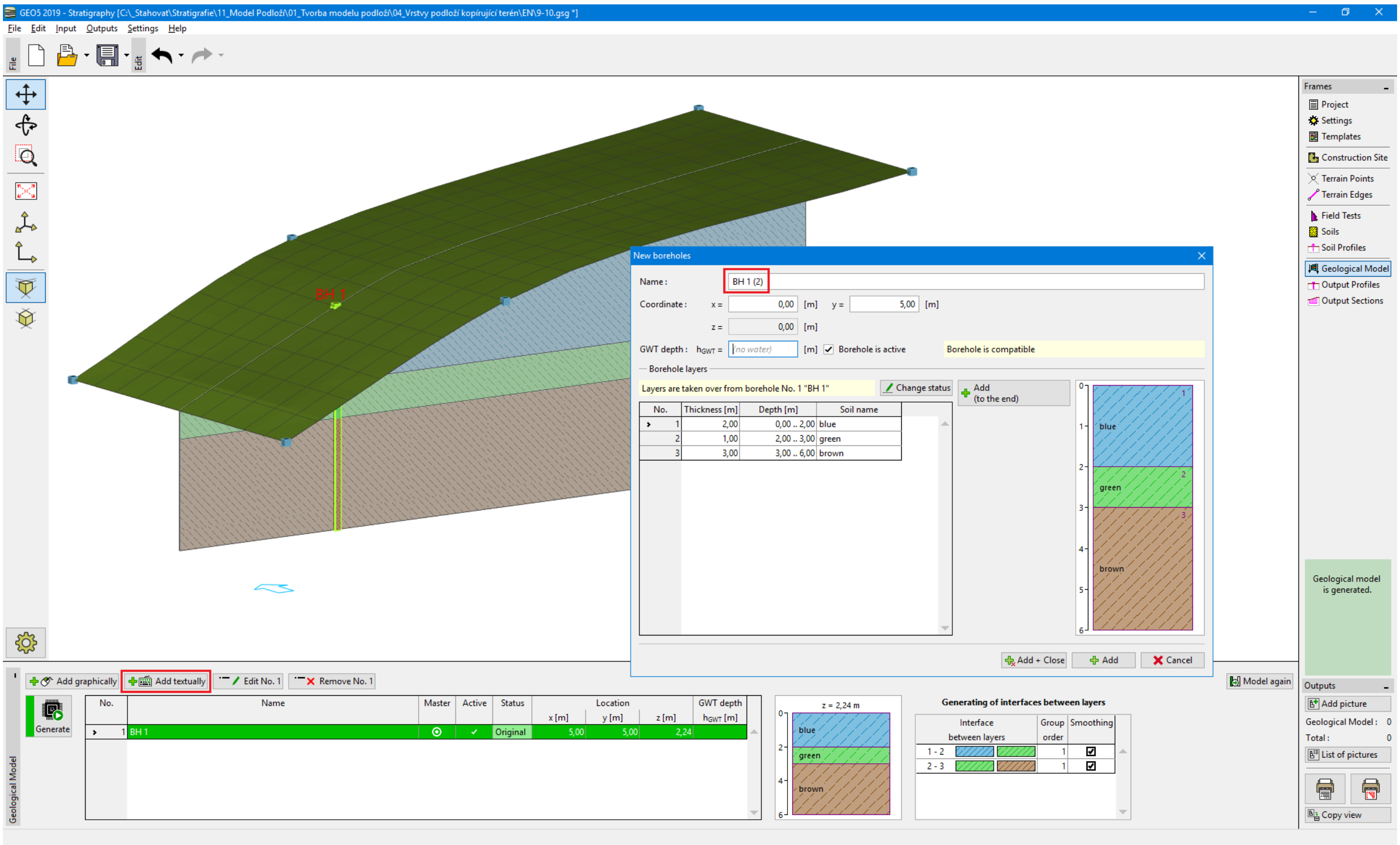
We will continue with the previous example. The procedure of modification is evident from the following picture - it is necessary to enter new boreholes on the boundaries of the construction site.



In the "**Geological model**" frame, select a borehole "**BH 1**" and define new boreholes in points [0; 5], [20; 5] (using the

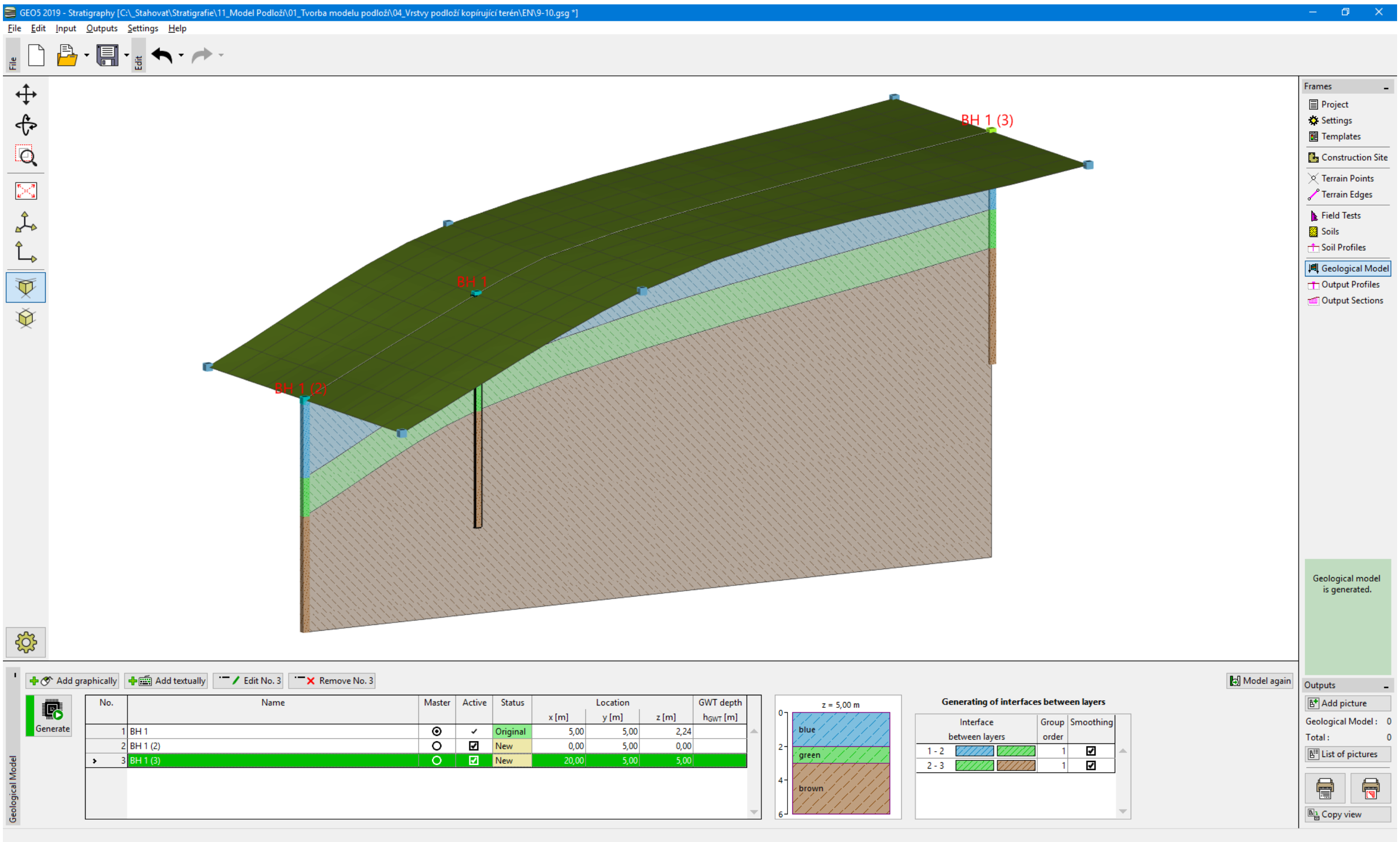
"**Add graphically**" or "**Add textually**" buttons). If the borehole were not selected before pressing "**Add graphically**" or "**Add textually**" button, new boreholes would not copy borehole "**BH 1**", but they will correspond to the already created geological model.

In the dialog window "**New boreholes**", we can see that the data of the new borehole is **copied from the borehole** "**BH 1**".



Dialog window "New borehole"

The model is created.



Final model

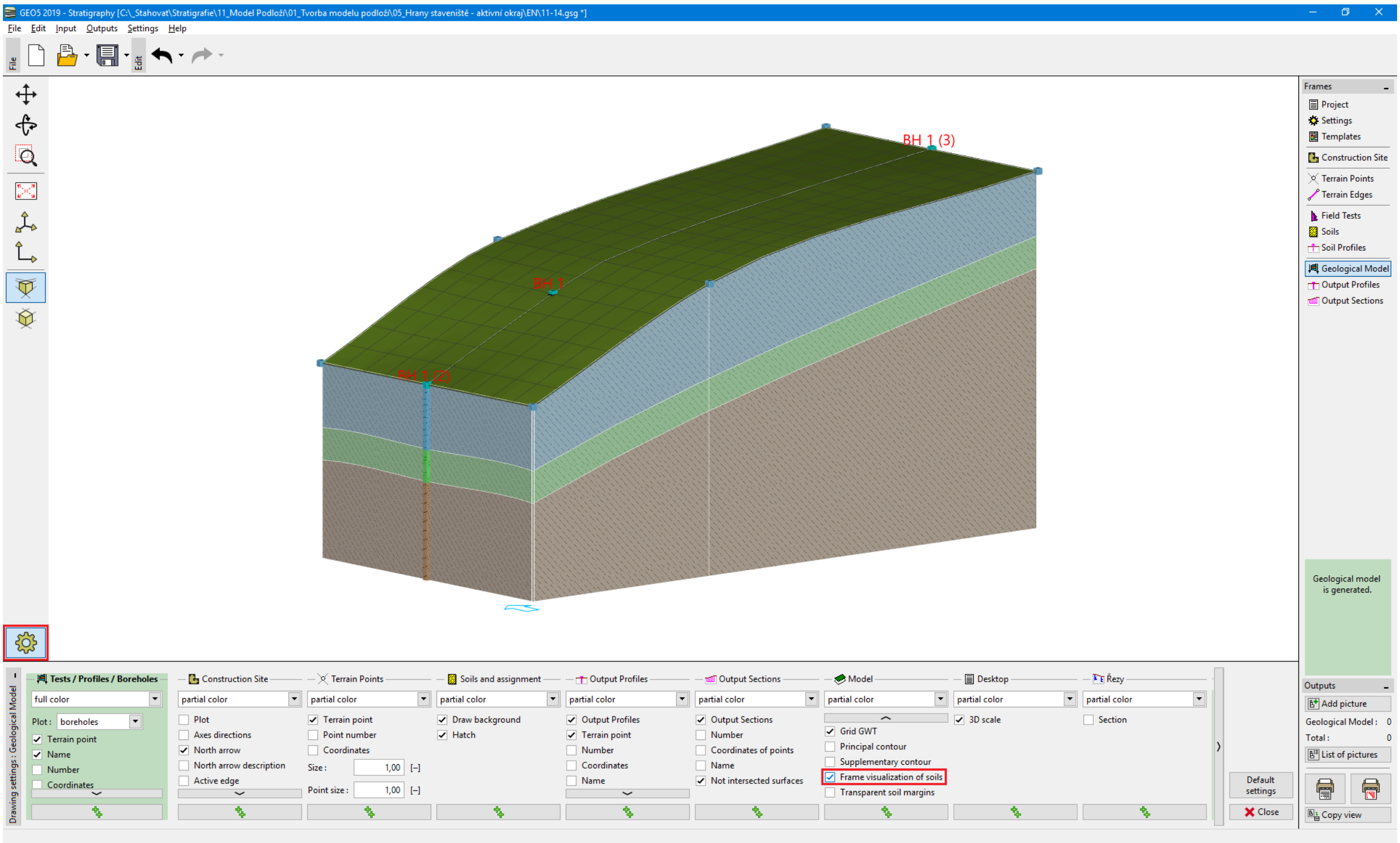
Previous example Next example

Construction Site Edges - Active Edge

Previous example Next example

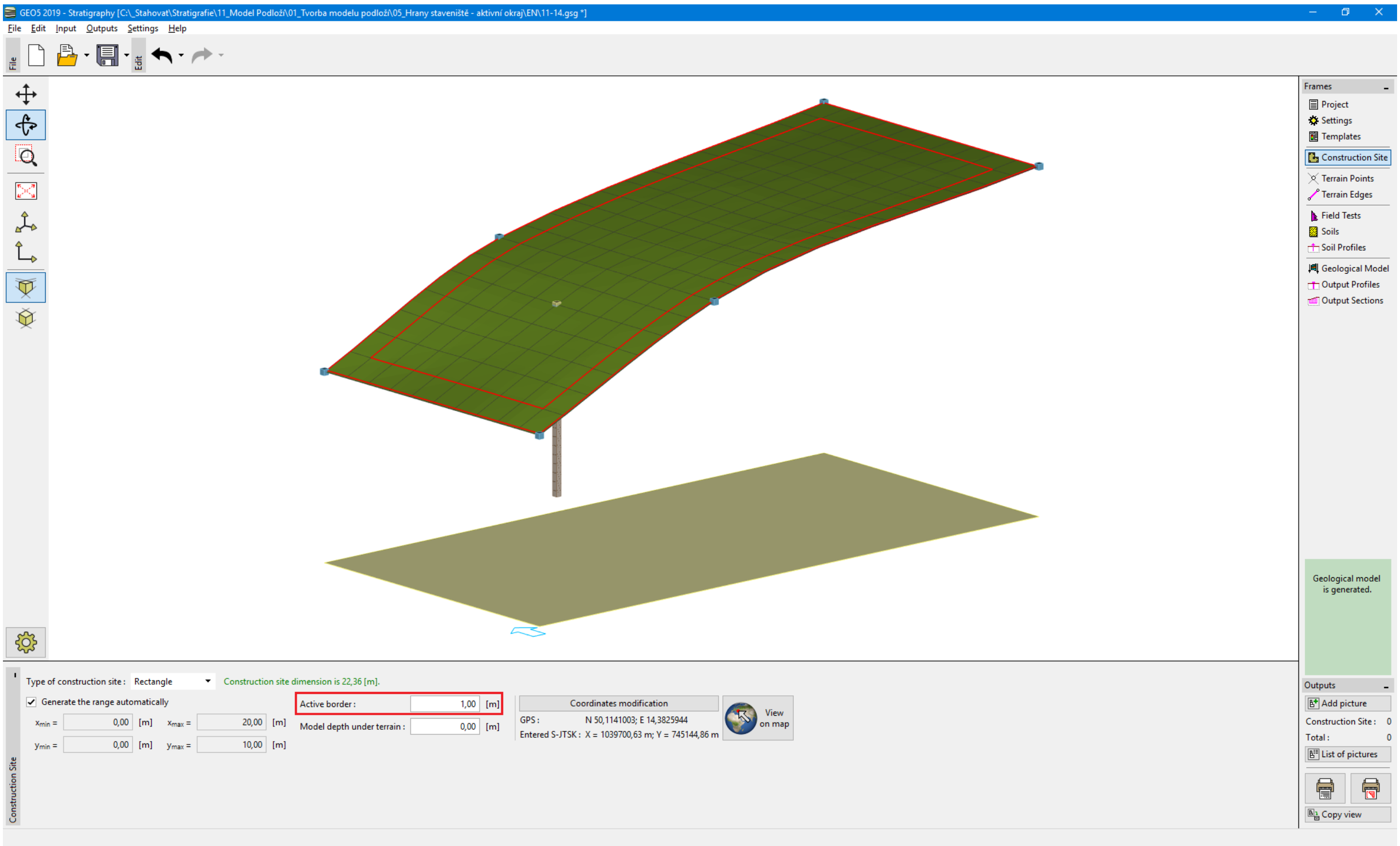
Turn on the "**Frame visualization of soils**" in the drawing settings from the previous task.

Edges of the model are created just from assistant boreholes in the model corners - layers are almost straight on the edges of the model.



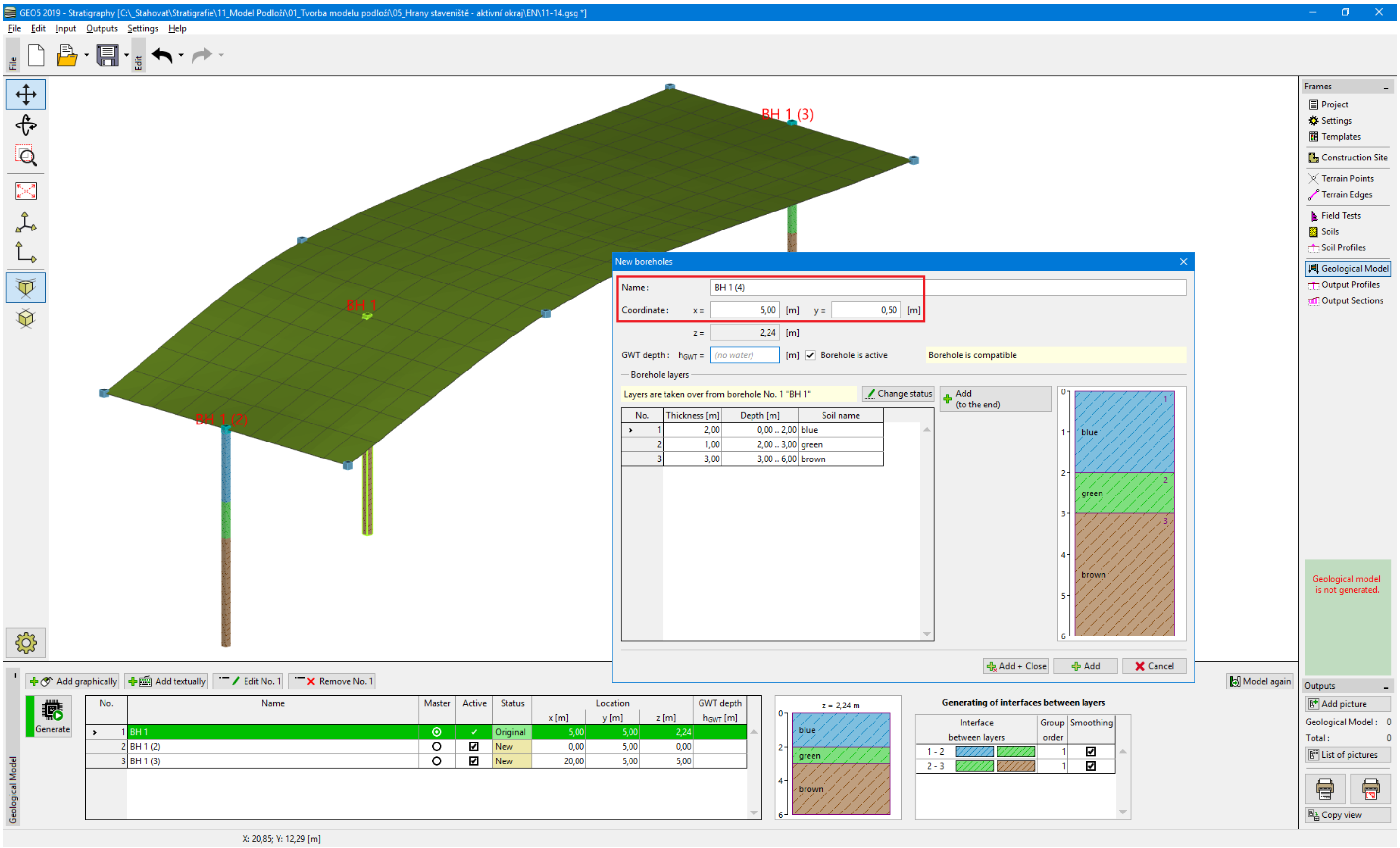
Geological Model with straight layers on edges

Switch to the "**Construction Site**" frame and enter an "Active edge" [1 m]. The active edge is red.



Active edge visualization - "Construction Site" frame

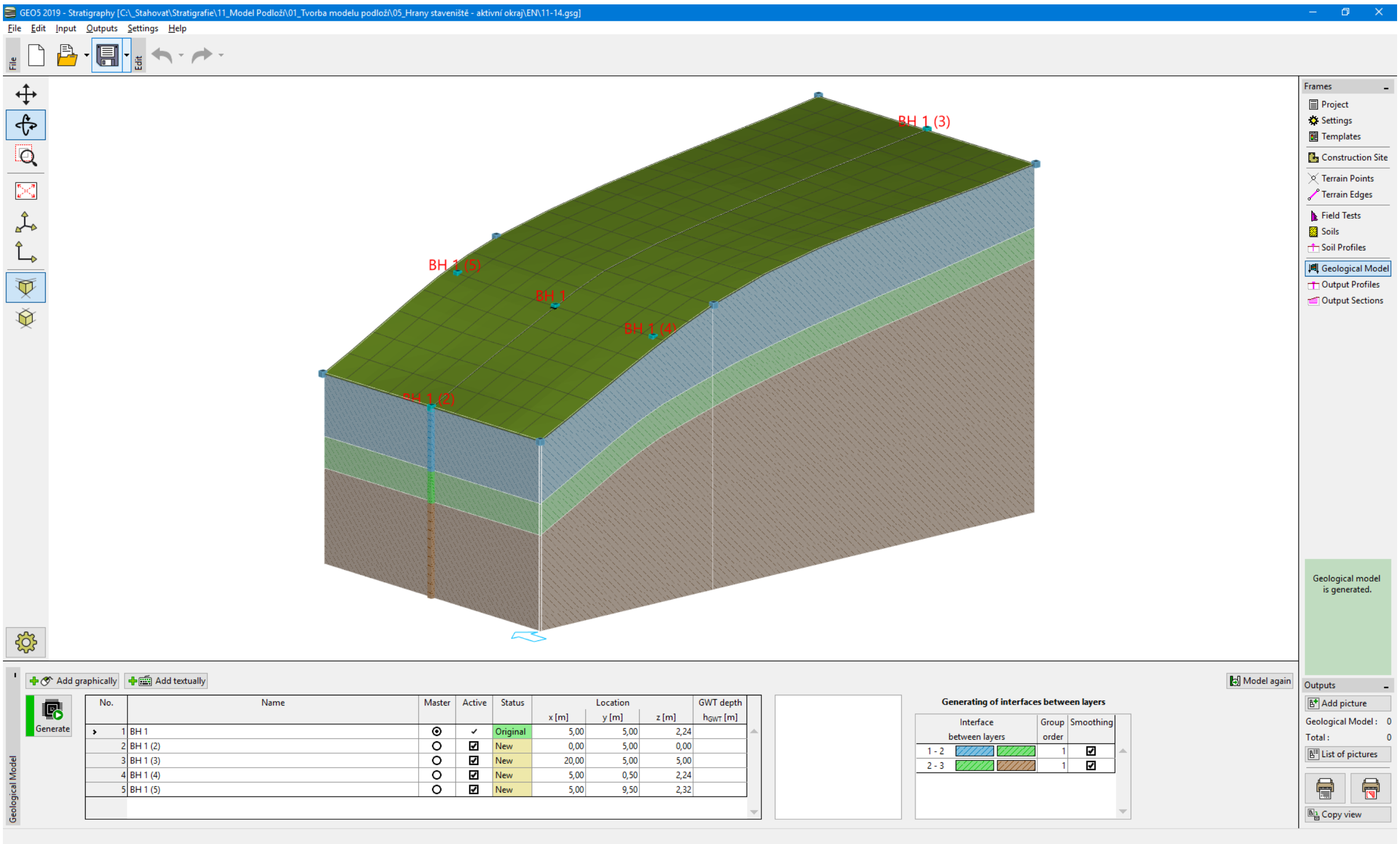
Borehole "**BH 1**" is not in the active edge, so the generated model remains the same. Switch to the "**Geological model**" frame, select borehole **BH 1**, and add two new boreholes "**BH 1(4)**" [5; 0.5] and "**BH 1(5)**" [5; 9.5] using the button "**Add textually**".



Input of new boreholes into the active zone

The edges of model have significantly changed. Now the shape of the layers is the same as in the cross section in the middle.

Both boreholes lie in the active zone - during the generation of the model, temporary assistant boreholes with the same layers are created on the borders.



Final model

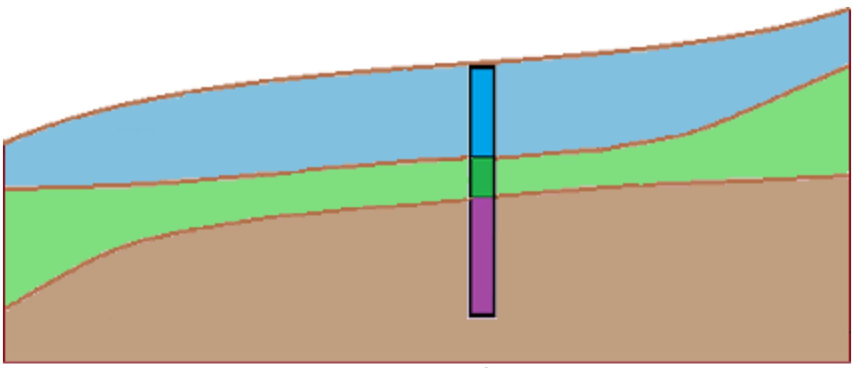
When modeling real constructions, it is reasonable to enter an active edge to reach the **closest points and boreholes** from the construction site edge.

Previous example Next example

Editing of Soil Layers

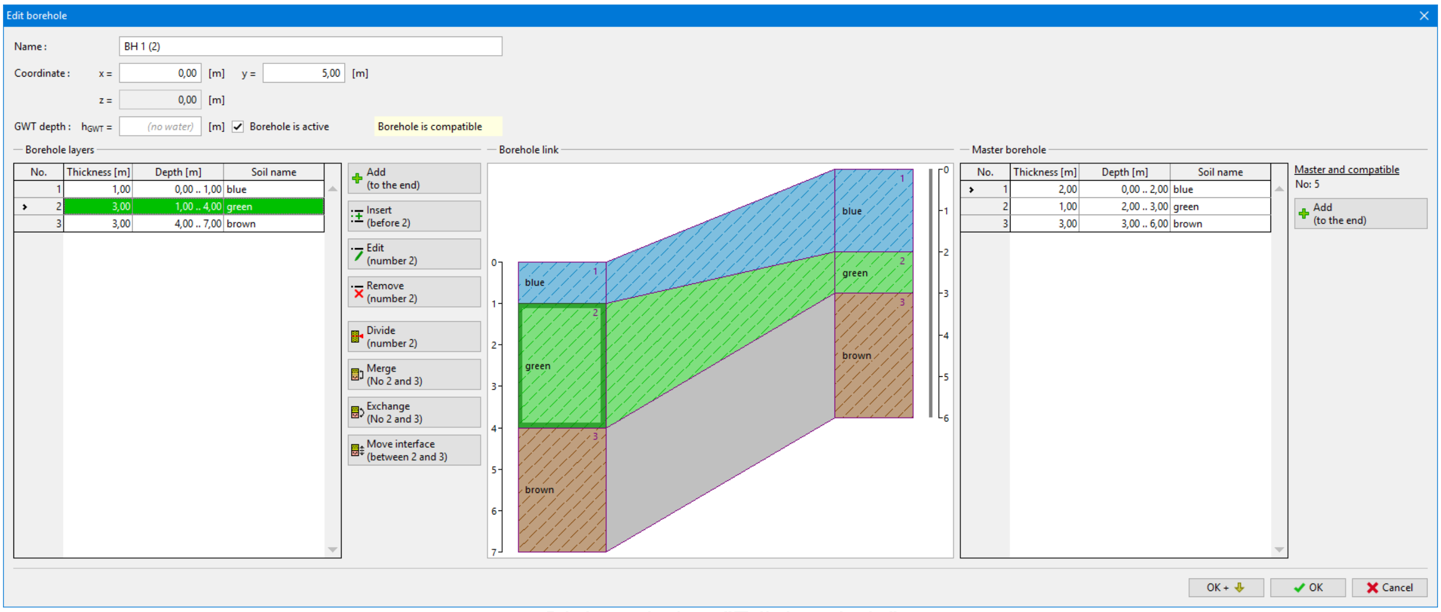
Previous exampleNext example

Now we modify the green layer to expand into the edges of model.



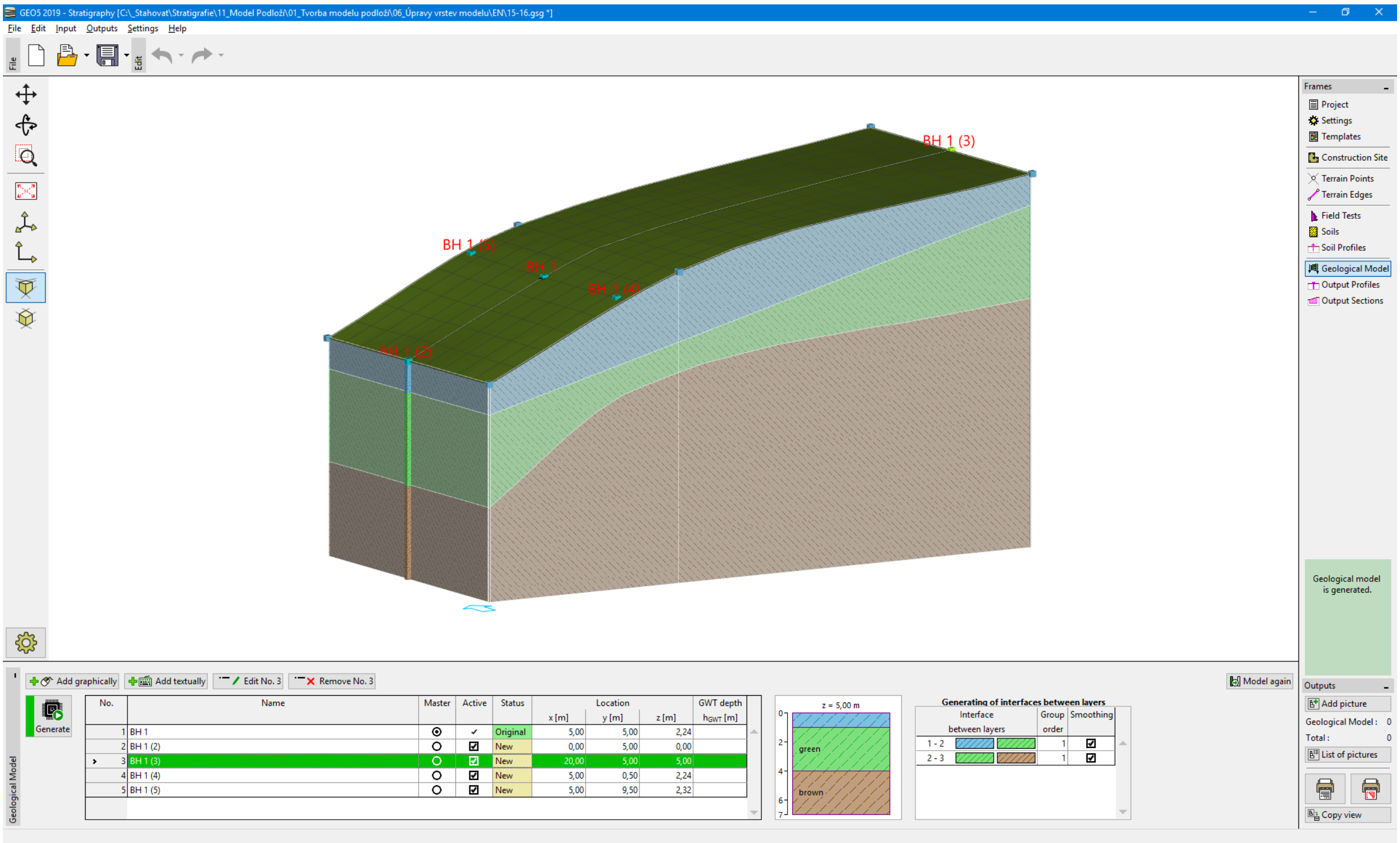
Assignment - extension of the green layer

We use the dialog window "Edit borehole". We perform the same **modifications for both new boreholes BH 1(2) and BH 1(3) - increase a thickness of the green layer from** [1 m] **to** [3 m] and **reduce the thickness of the blue layer from** [2 m] **to** [1 m].



Dialog window "Edit borehole"

The model is modified.



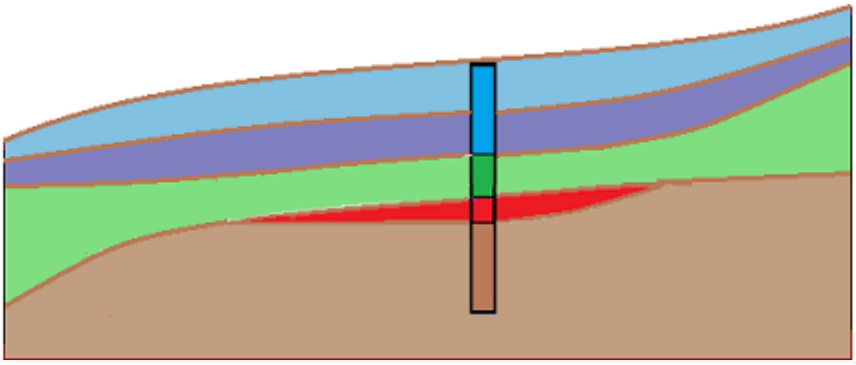
Modified model

Previous exampleNext example

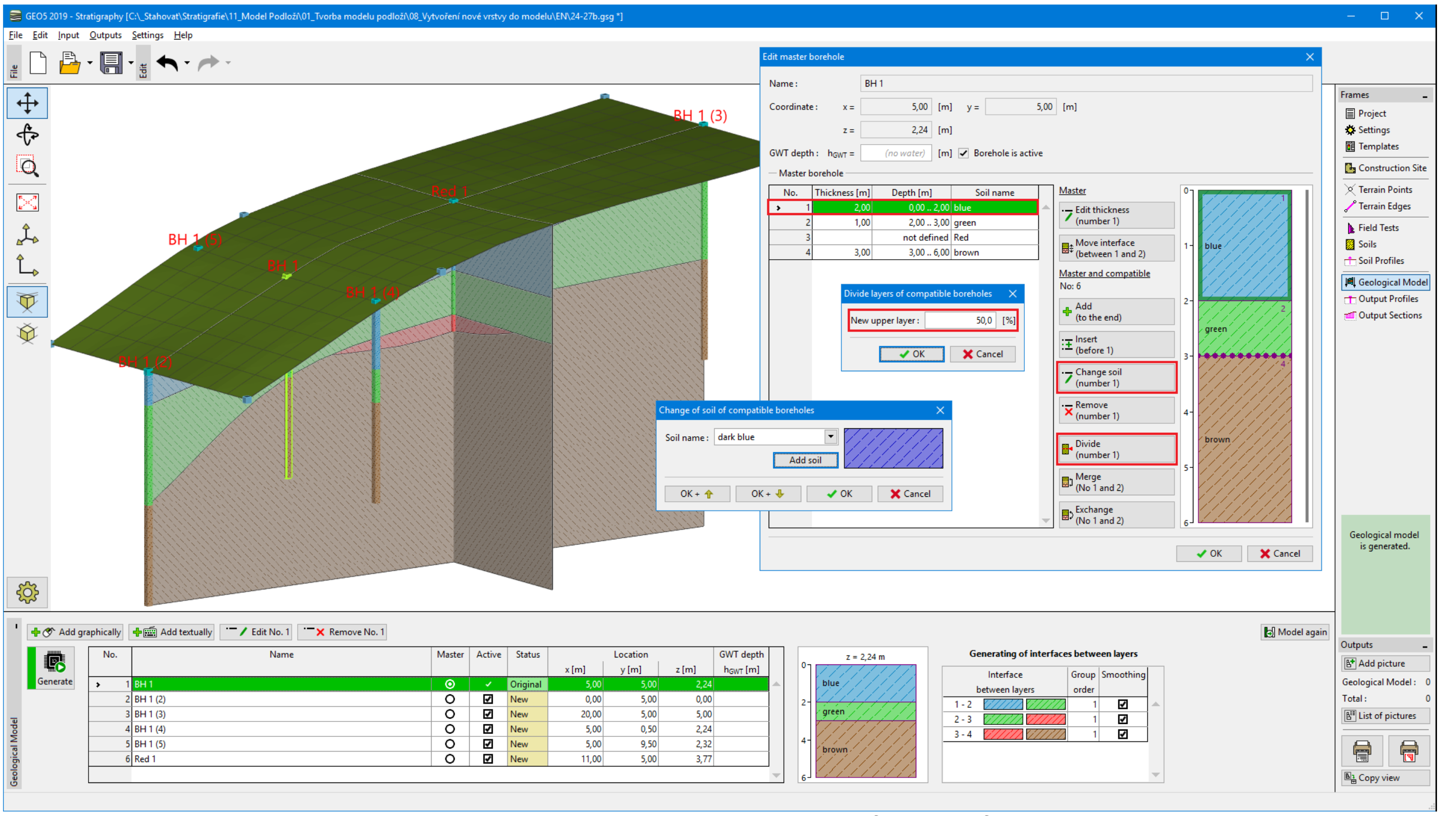
Adding a New Layer into the Model

Previous example

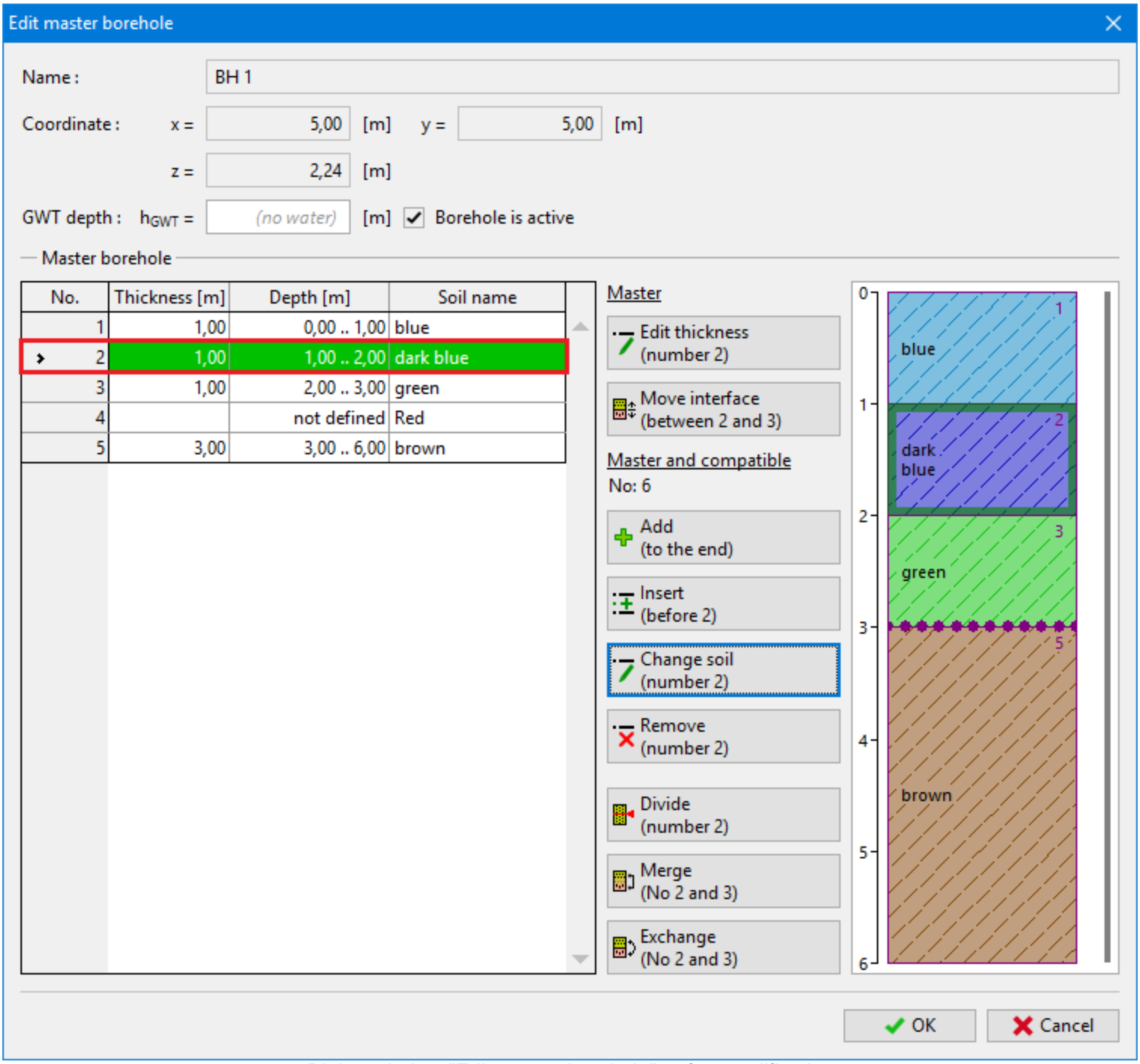
We want to divide a blue layer from the previous example into two layers - blue and dark blue. **Dividing and merging of layers** is common mainly due to changes in soil parameters in depth.



In the "**Geological model**" frame, we edit the master borehole - first, we divide the blue layer using "**Divide layer (No.1)**" button, and then, we change the new bottom layer to dark blue using "**Change soil (No.2)**"

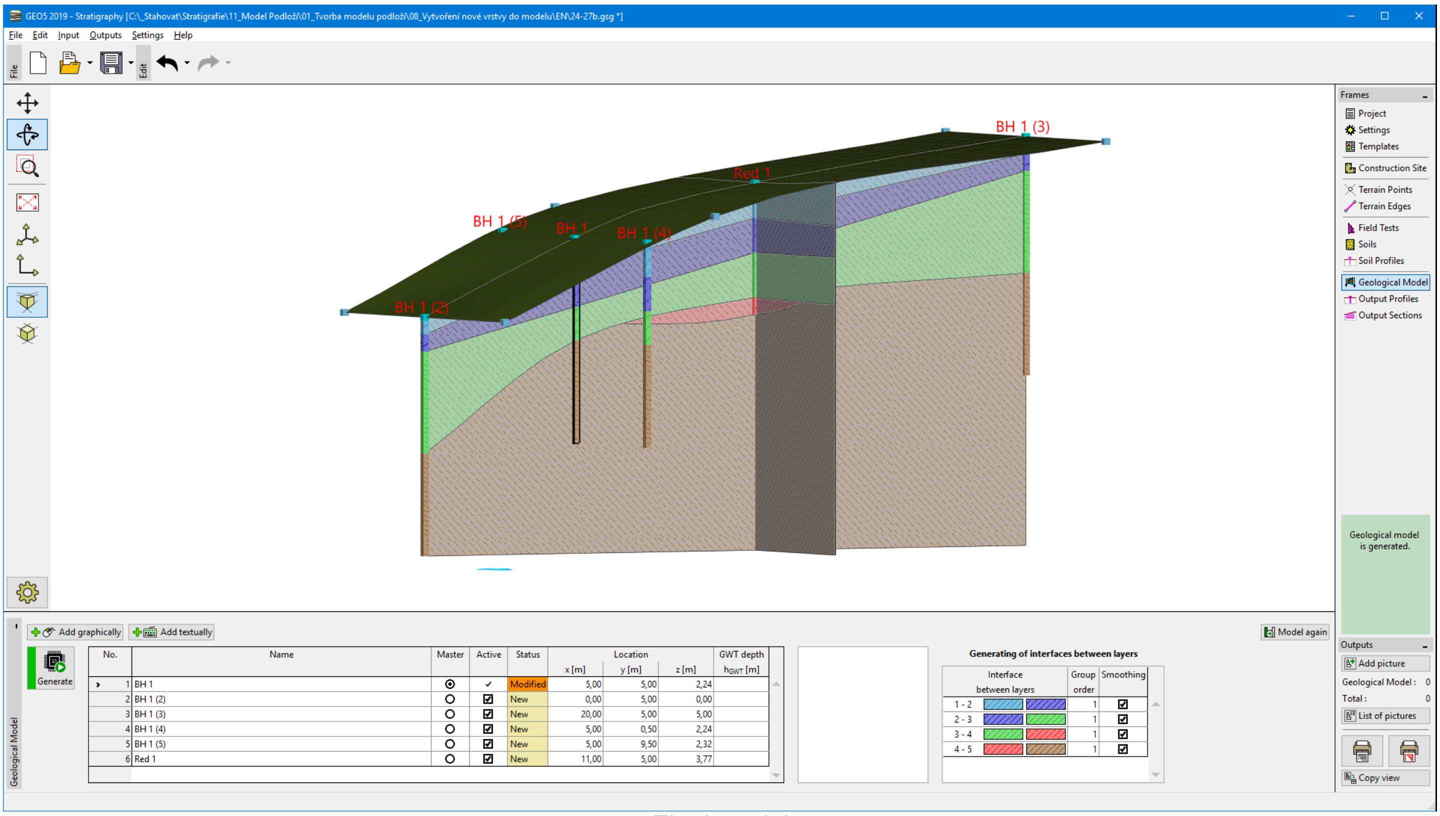


Dialog window "Edit master borehole" - before modification



Dialog window "Edit master borehole" - after modification

After model generation, the layer is changed.



Final model

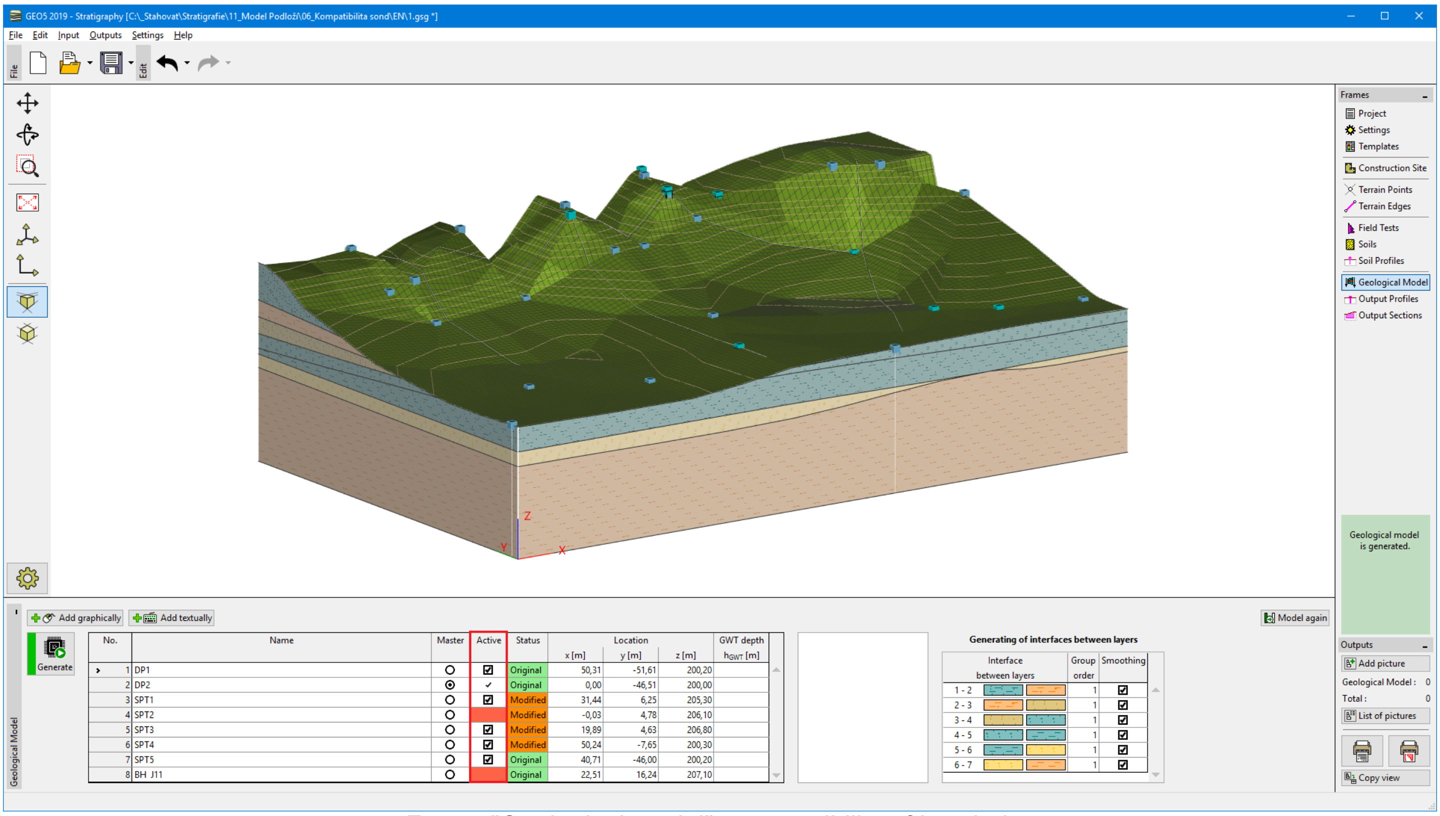
Previous example

Compatibility of Boreholes

To have the best model, we want to use all boreholes in it. For this reason, all boreholes should be compatible.

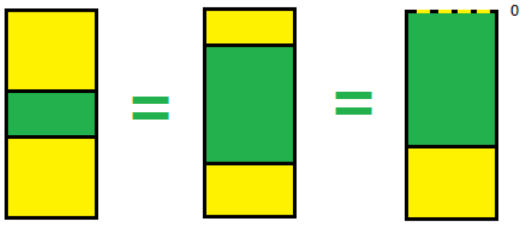
Compatible borehole has an identical count and order of layers as the master borehole. It is shown in white in the "**Active**" column. If this borehole is also active, it is used for geological model generation.

Incompatible boreholes are shown in red in the "**Active**" column and they cannot be used for the generation of the model.

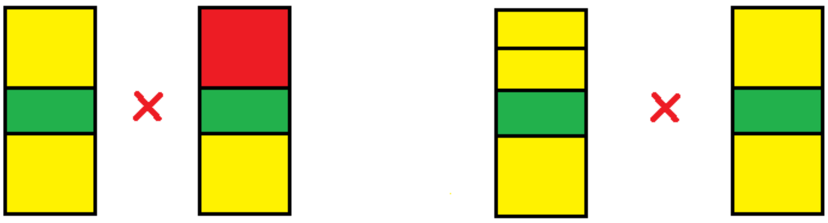


Frame "Geological model" - compatibility of boreholes

Examples of **compatible boreholes**:

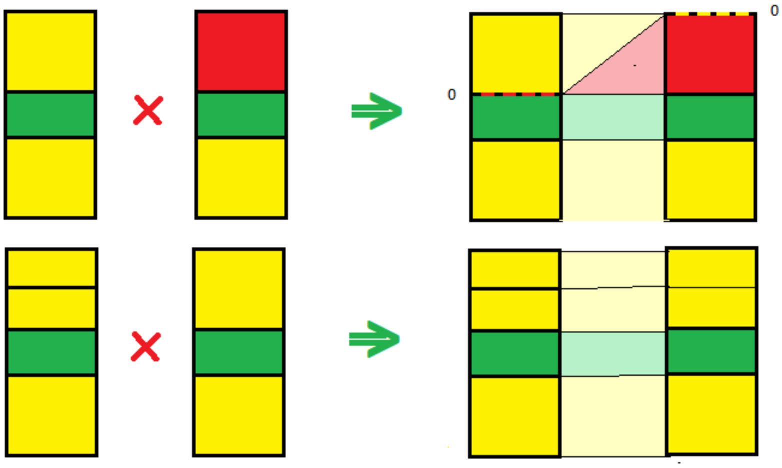


Examples of **incompatible boreholes**:



The incompatible boreholes can be changed to compatible in the "Edit borehole" dialog window - using e.g. **inserting of layers** with zero thickness or **adding (removing) soil layer**.

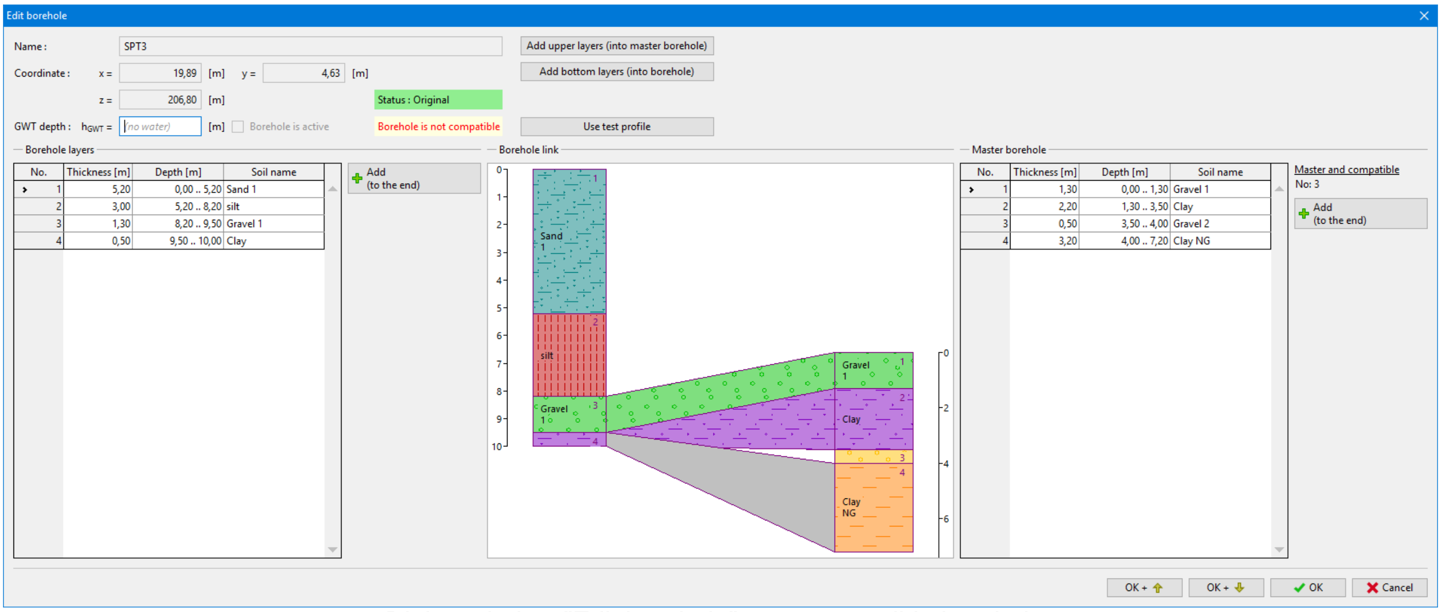
Example of modification of incompatible boreholes to compatible:



Edit Borehole

The boreholes are edited in the "**Edit borehole**" dialog window. This window allows us to **change an edited borehole** (in the left part of the dialog window) and **also the master borehole** (in the right part of the dialog window).

The tools of the dialog window are described in the example:



Dialog window "Edit borehole" - not compatible boreholes

In the left part of the dialog window, the data of **edited borehol**es are displayed. The state of a borehole (compatible/incompatible) is displayed above the borehole scheme. The aim of modification is to **change a borehole into compatible** and add it into the model.

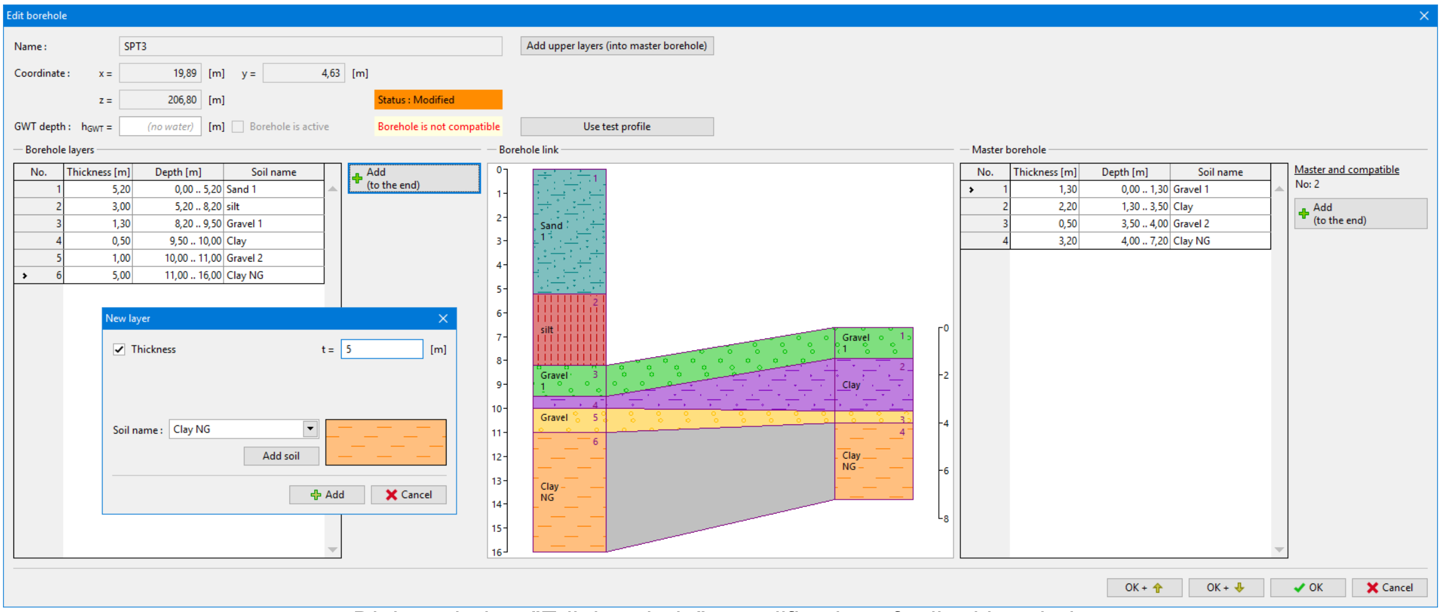
In the right part of the dialog window, the data of the **master borehol**e are displayed.

In the middle part of the dialog window, the **boreholes connection** is shown. The height of boreholes is displayed correctly, so it is obvious which layers are horizontal. If the borehole (master or changed) are created from the test, the **length of the test** is displayed in gray. In the picture, we can see that the edited borehole was created from the test, which was not deep enough to achieve yellow layers. On the other hand, the master borehole the upper layers are missing in its geological profile.

It is possible to use these buttons for borehole editing: (**"Insert"**, "**Add**", "**Edit**", "**Divide**", "**Merge**", "**Exchange**", "**Move interface**" "**Remove**") for both tables. (If no row is selected in the table, only the "**Add**" button is available. After selecting a row, other buttons are displayed).

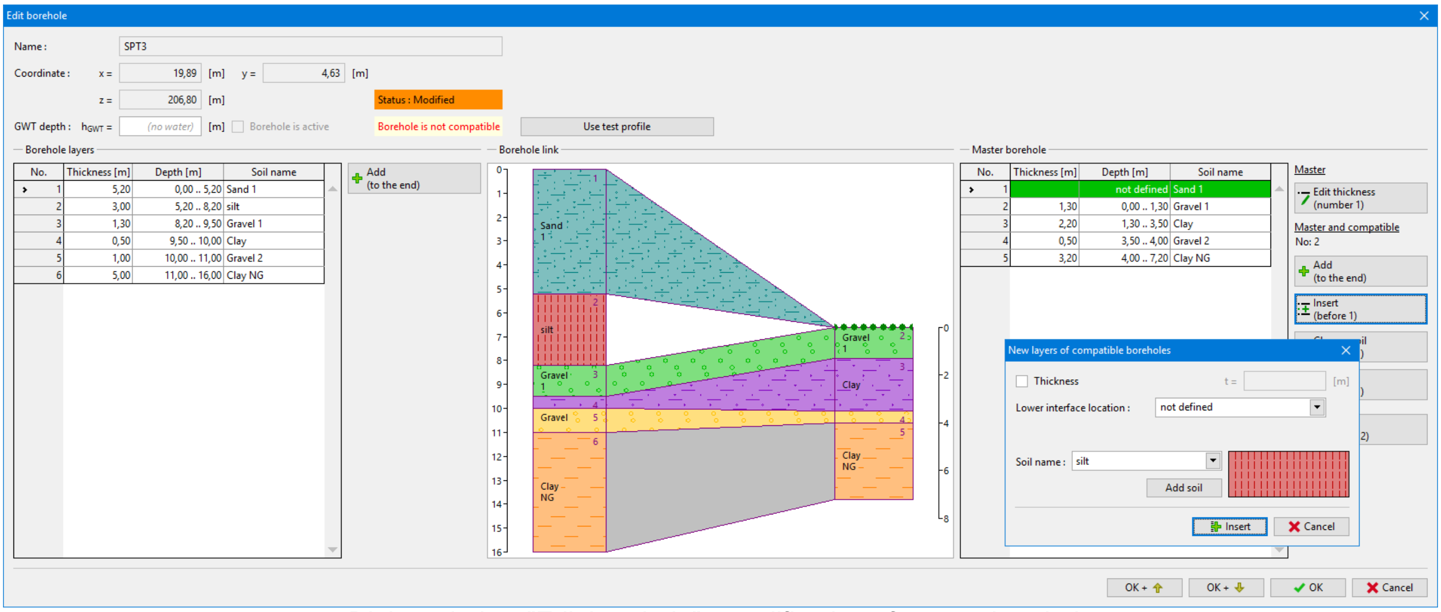
The borehole can be **changed from incompatible to compatible with these steps:**

- **Edit selected borehole on the left part of the dialog window** - Using an "**Add"** button for the layers "**Gravel 2** " (**yellow**) and "**Clay NG**" (**orange**) are added. We can enter their assumed thickness, depth of interface, or do not specify exact location.



Dialog window "Edit borehole" - modification of edited borehole

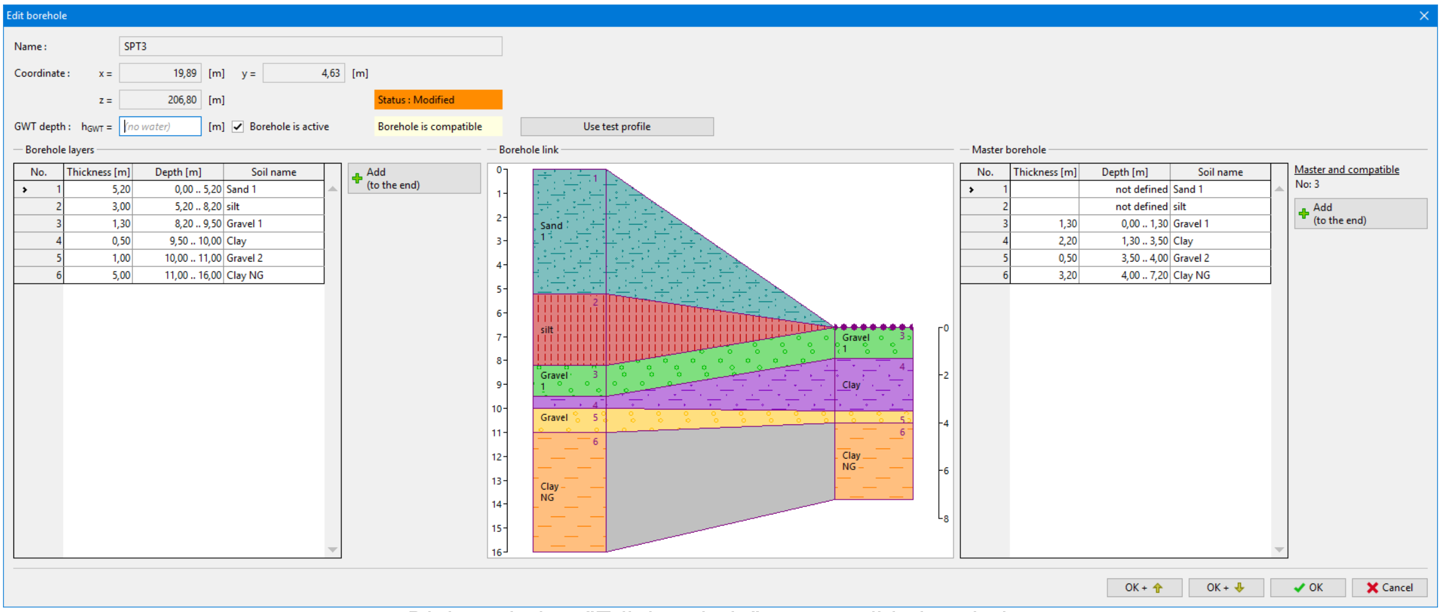
- **Edit master borehole on the right** part of the dialog window- Using an "Insert (before 1)" button (the first row in the master borehole table must be selected) for the master borehole in the right part of the dialog window, the Layers "**Silt**" (**red**) and "**Sand 1**" (**blue**). In this case, the best way is not to specify the exact location of the interface.



Dialog window "Edit borehole" - modification of master borehole

Because this way of modification is a common and little bit lengthy (for a larger number of layers), the program allows us to perform this operation in a simpler way using the "**Add upper layers**" and "**Add bottom layers**" buttons in the upper part of the dialog window.

This way, the connection between boreholes is compatible.

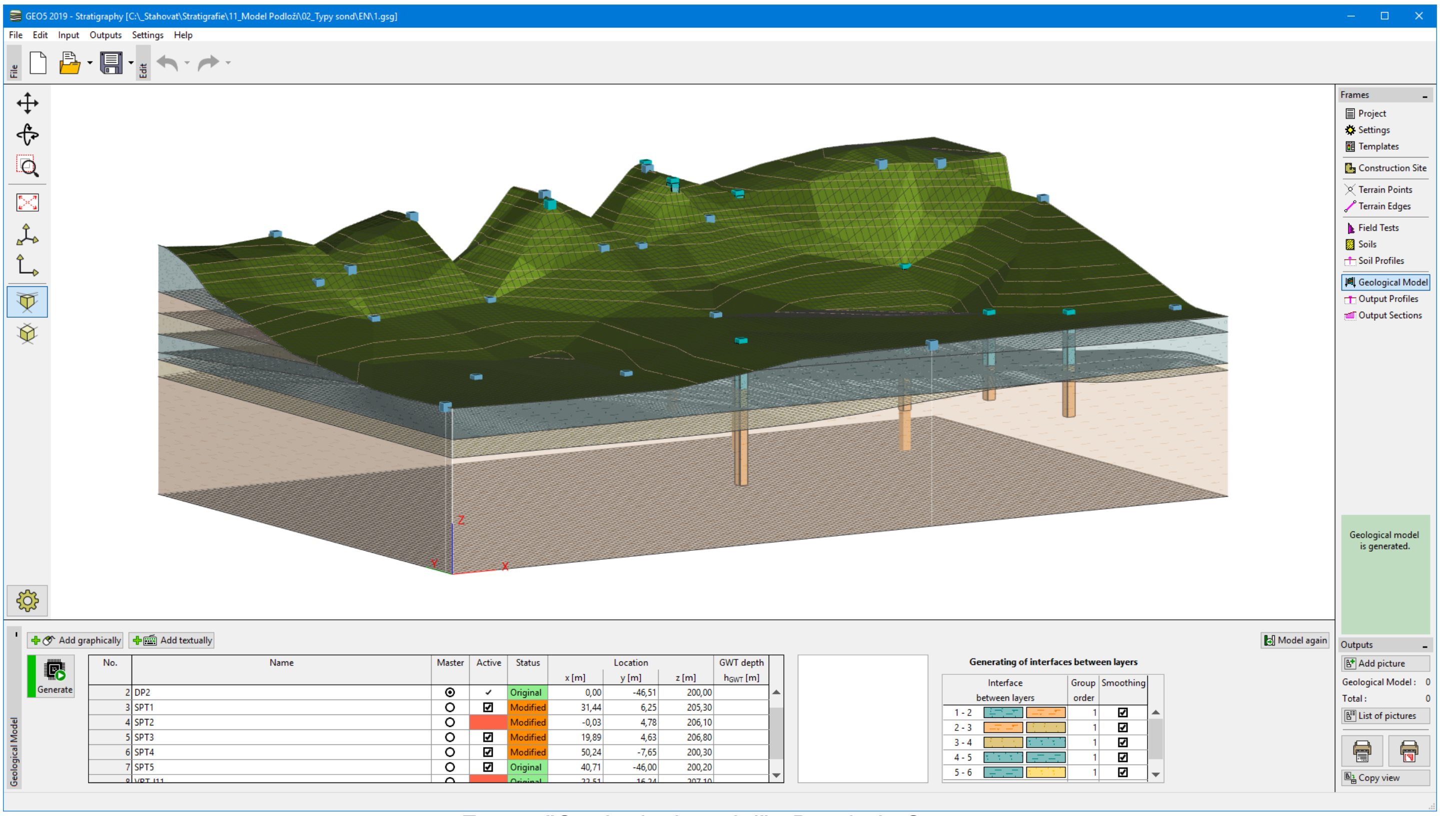


Dialog window "Edit borehole" - compatible boreholes

The borehole type is also displayed in the dialog window - if the borehole is **original** (corresponds to the soil profile) or **changed**. It is possible to return to the original borehole status using the "Copy from the soil profile" button.

Borehole Status

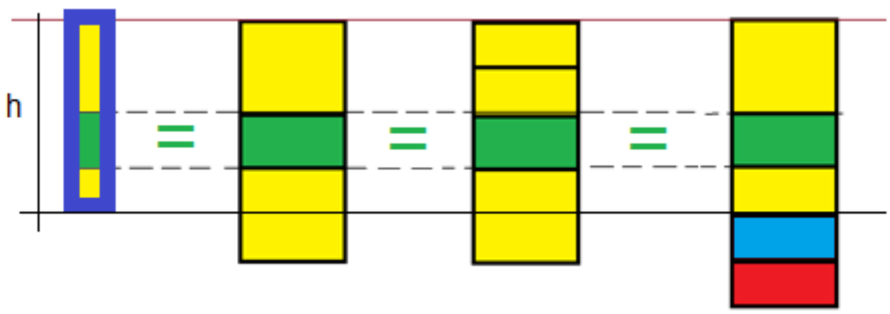
The **borehole status** determines a relationship between the original "Soil profile" and "**Borehole**". It is not necessary to know the borehole status for the generation of the geological model, but it is good to know it for a better understanding of the performed modifications of the model.



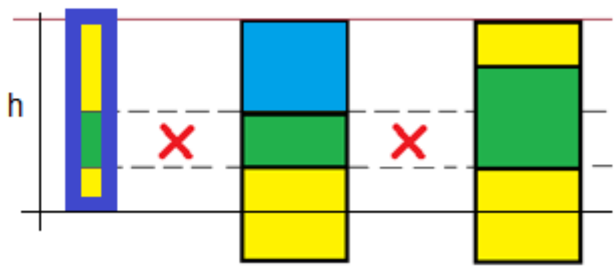
Frame "Geological model" - Borehole Status

The boreholes can have three status:

Original - the borehole is generated from the soil profile, and it corresponds to its geological profile (which means the same soil at the same depth). The layers below the overall length of the soil profile do not influence the borehole type. The borehole can have an arbitrary number of soil layers with zero thickness - the type is still original.



Changed - the borehole is created from the soil profile, but its geological profile was changed - by a change of soil or layer thickness.



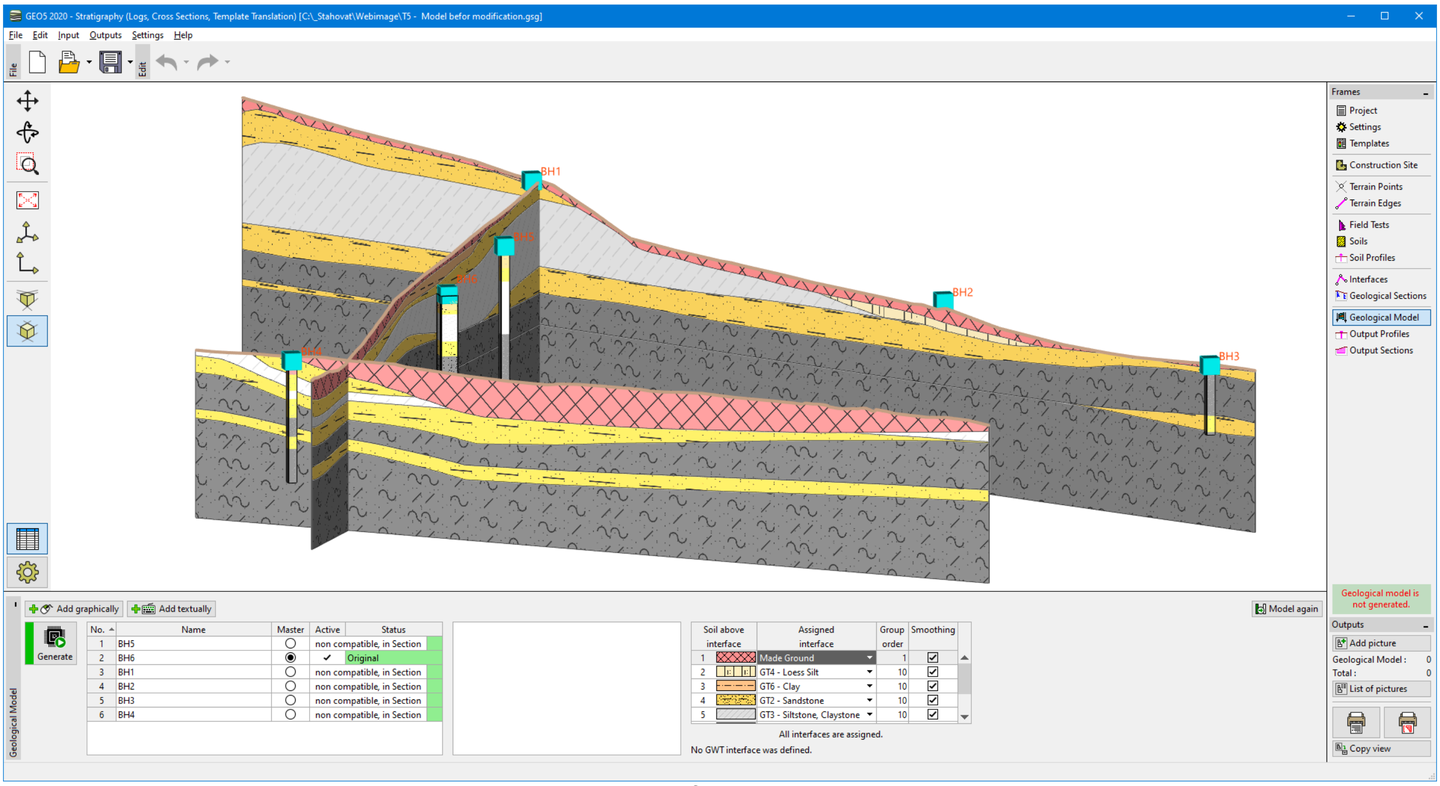
New – this borehole was created separately; it has no relationship to the tests or soil profiles. This borehole can be deleted from the model.

Modification of the Final 3D Model Using Boreholes

A correctly created model corresponds not only to defined **Geological sections** and **Soil profiles,** but it must also meet our ideas about the stratigraphy of the model.

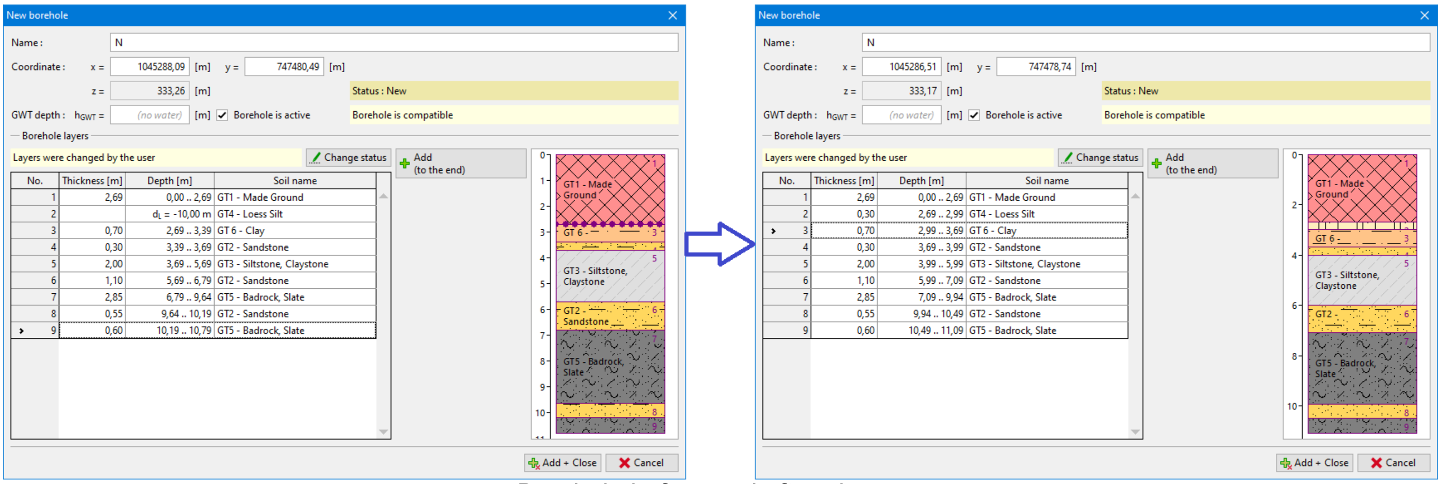
The best way of the final model check is to create a sufficient number of output sections (or visualization of **model sides**).

The final model can be easily edited by adding of new boreholes and changing their layers thicknesses. We see a too thick layer of "**Made Ground**" in the middle of the model.



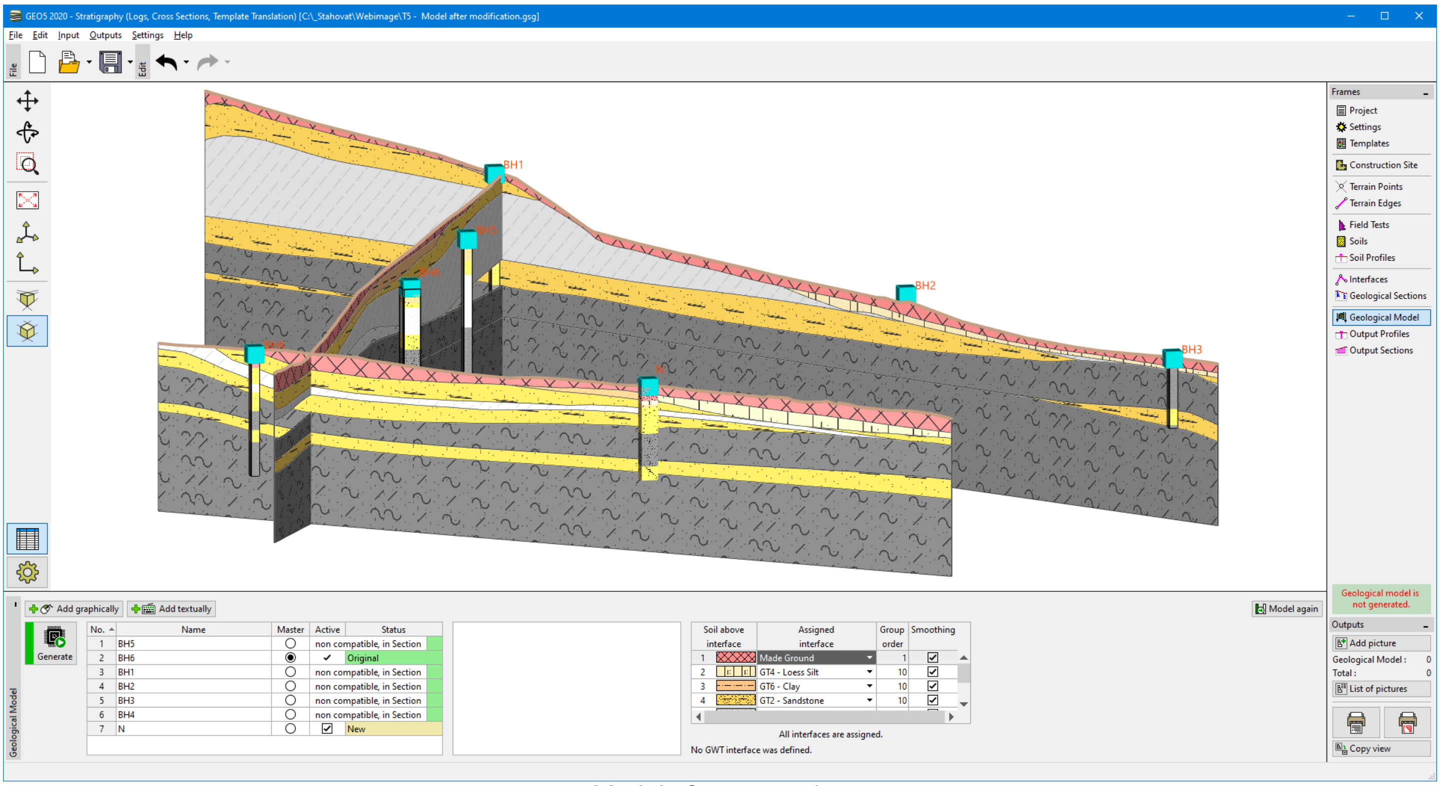
Model before correction

We add a new borehole and change the thicknesses of layers according to our ideas.



Borehole before and after changes

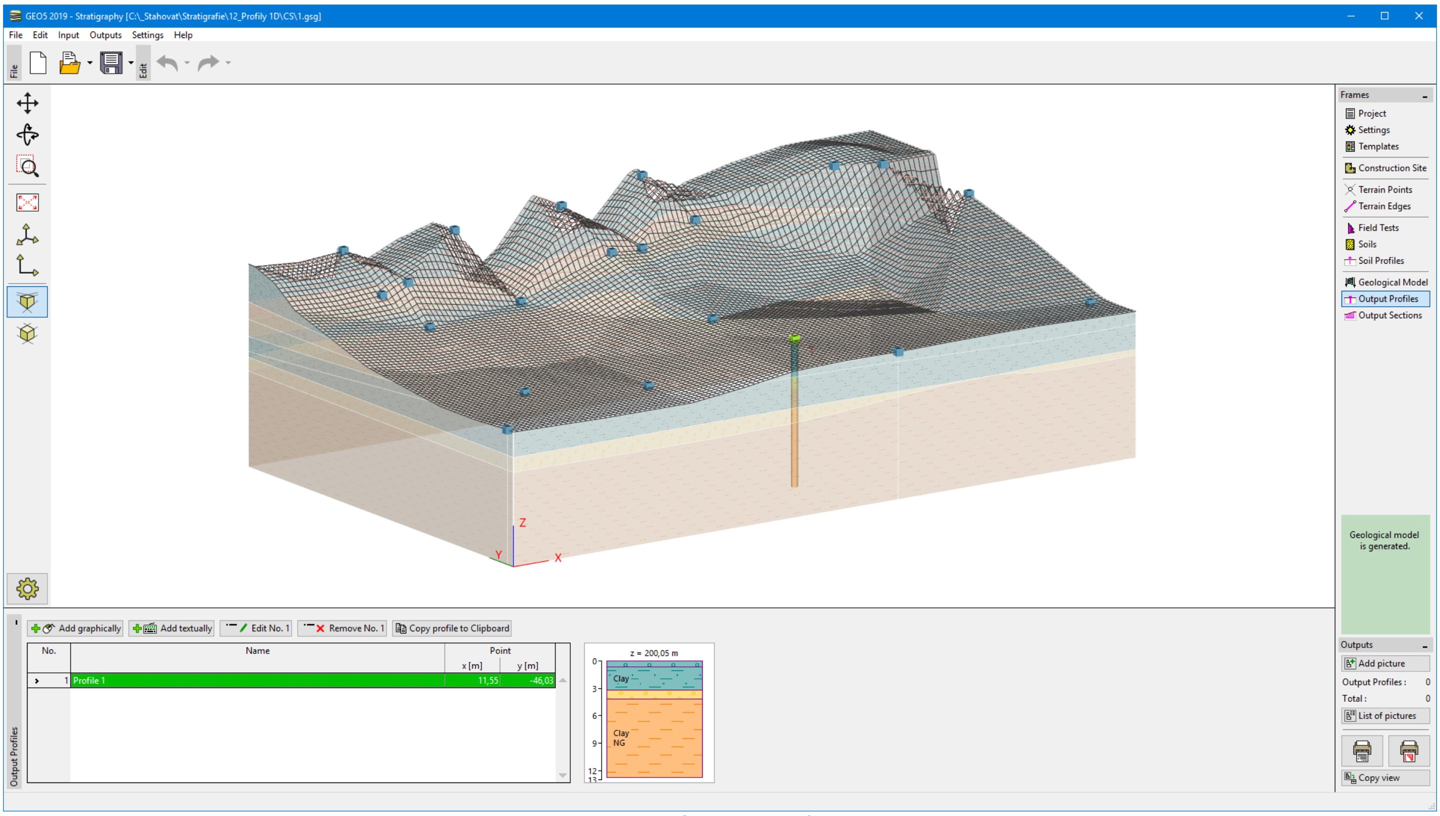
Then we generate a model and consider the accuracy of modification.



Model after correction

Output Profiles

The "**Output Profiles**" frame contains a table with all output profiles. The **name of the profile** and the **coordinates** of its location, are visible in the table. Next to the table, the selected profile is displayed.



Frame "Output Profiles"

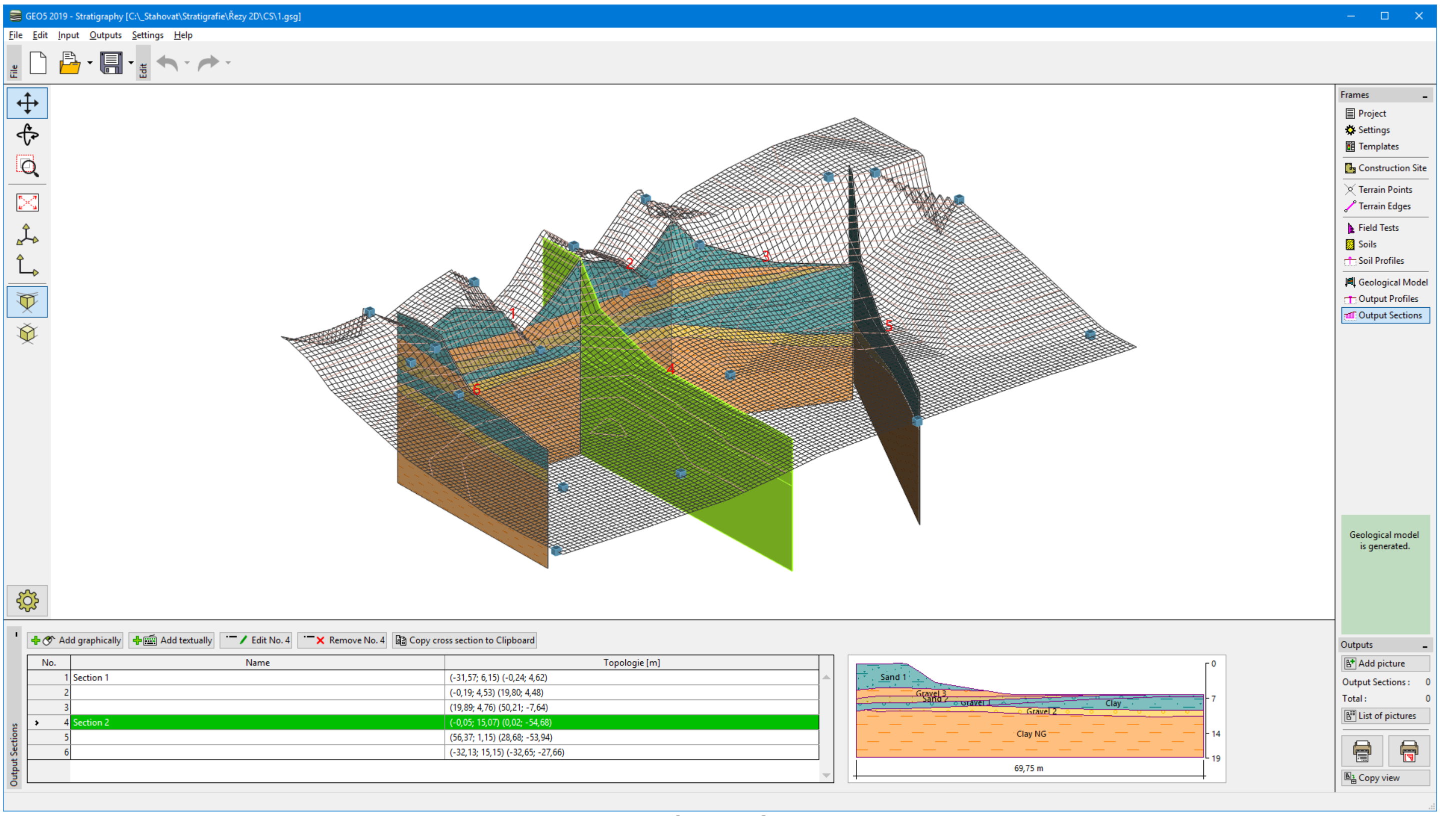
Visualization of drawing on the desktop can be modified in the "Drawing Settings" frame with the help of buttons on the toolbar "Visualization".

The "Undo" button is an important program tool. It allows us to return back to the **original state before any modification**.

Data of 1D profiles can be copied within other GEO5 programs.

Output Sections

The "**Output Sections**" frame contains a table with all output sections. The **name of the outputt section** and **coordinates** of its points are visible in the table. Next to the table, the selected output section is displayed.



Frame "Output Sections"

Visualization of drawing on the desktop can be modified in the "Drawing Settings" frame and with the help of buttons on toolbar "Visualization".

The "Undo" button is an important program tool. It allows us to return back to the **original state before any modification**.

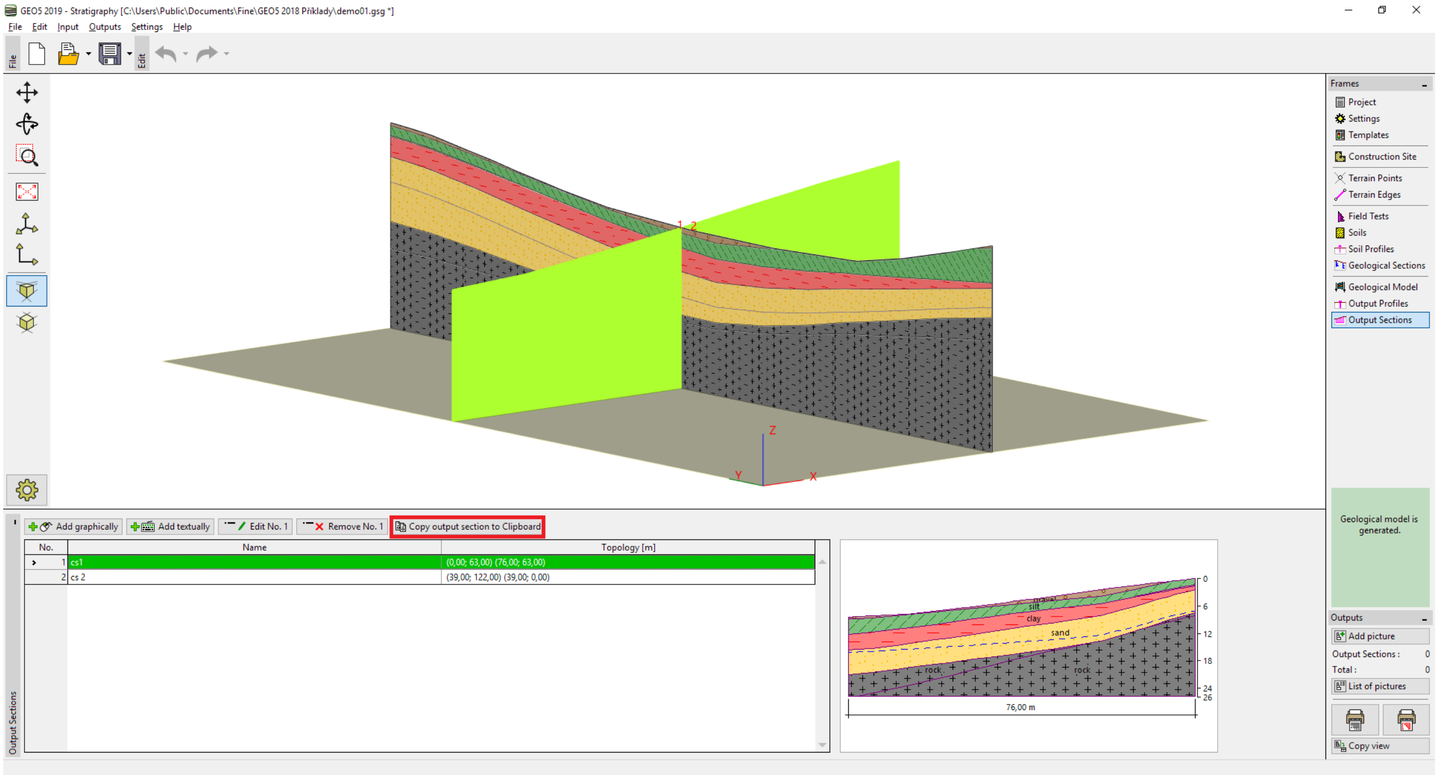
Data of output sections can be copied within other GEO5 programs.

Copying data from the Stratigraphy program to other GEO5 programs

The **Stratigraphy** program enables the user to copy and paste Output Sections and Output Profiles to other GEO5 programs.

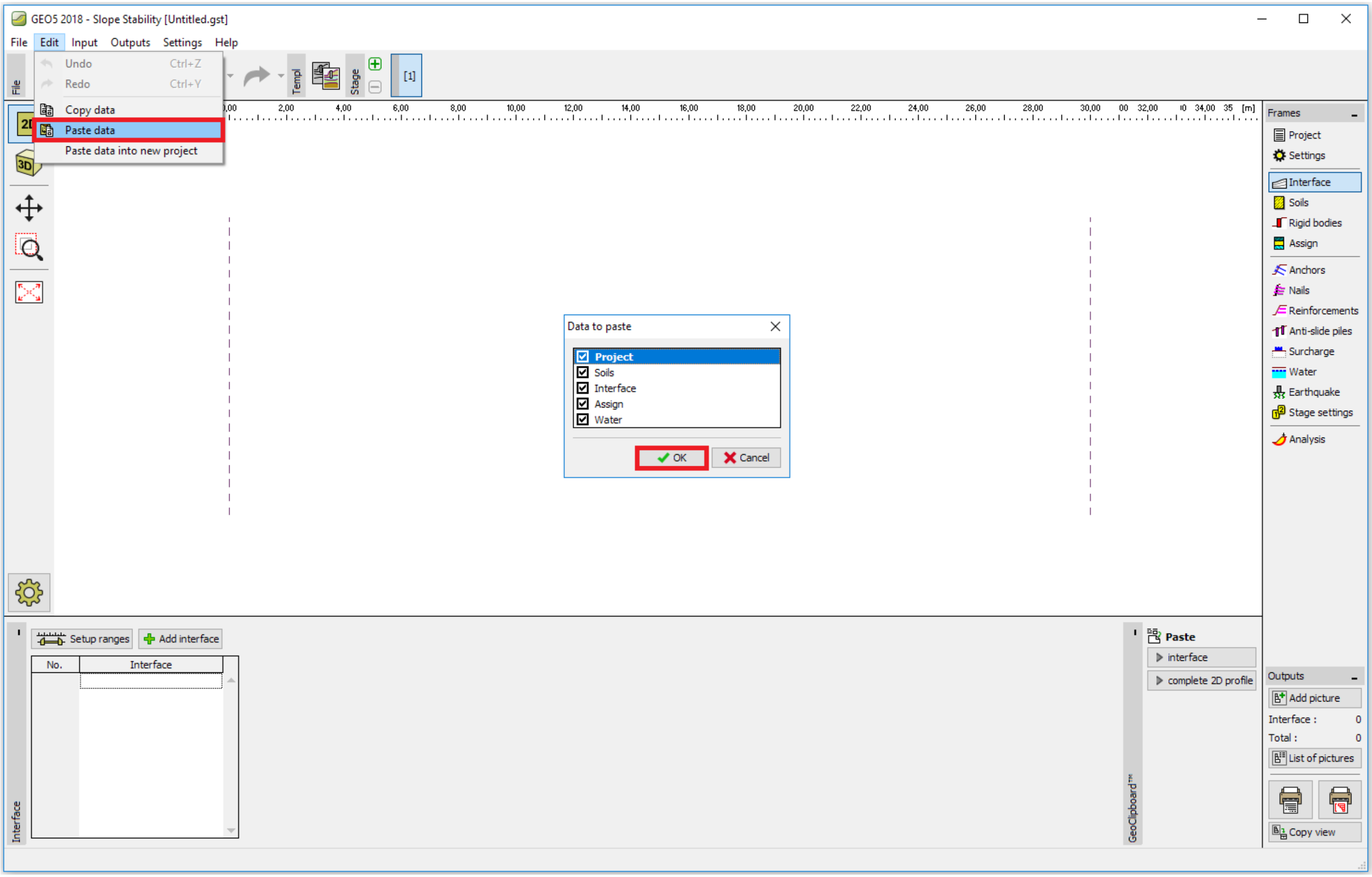
The following procedure shows how to colpy the **Output Section into the Slope Stability** program. The procedure for copying the **Output Profiles** is identical.

1. In the **Stratigraphy** program, select the output section you want to copy and use the button "**Copy output section to Clipboard**".



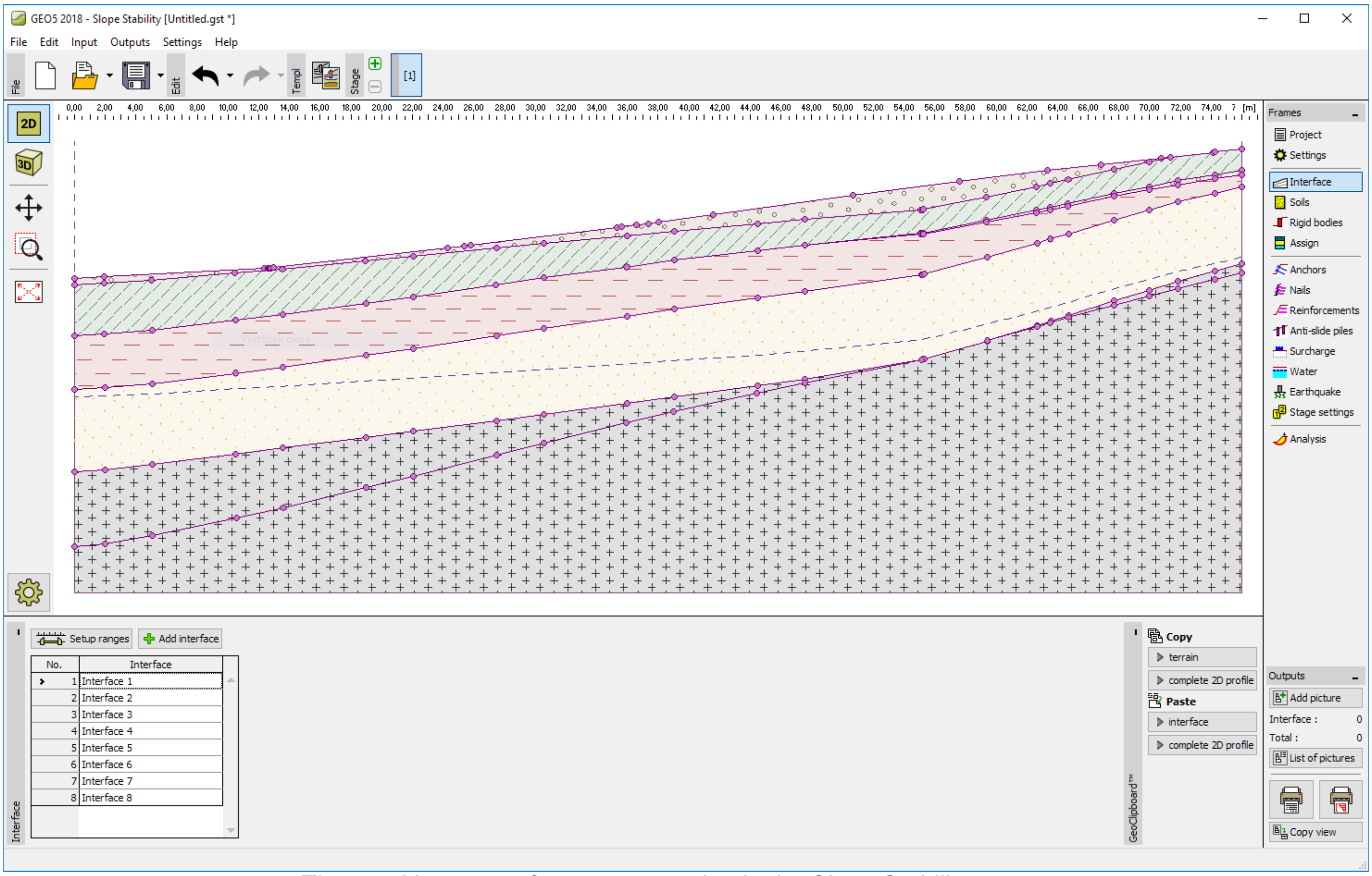
Copying an output section from the Stratigraphy program

2. Open the **Slope Stability** program and in the upper toolbar, click on "**Edit**" and then select "**Paste data**". To confirm that all data will be pasted, press "**OK**".



Pasting an output section into the Slope Stability program

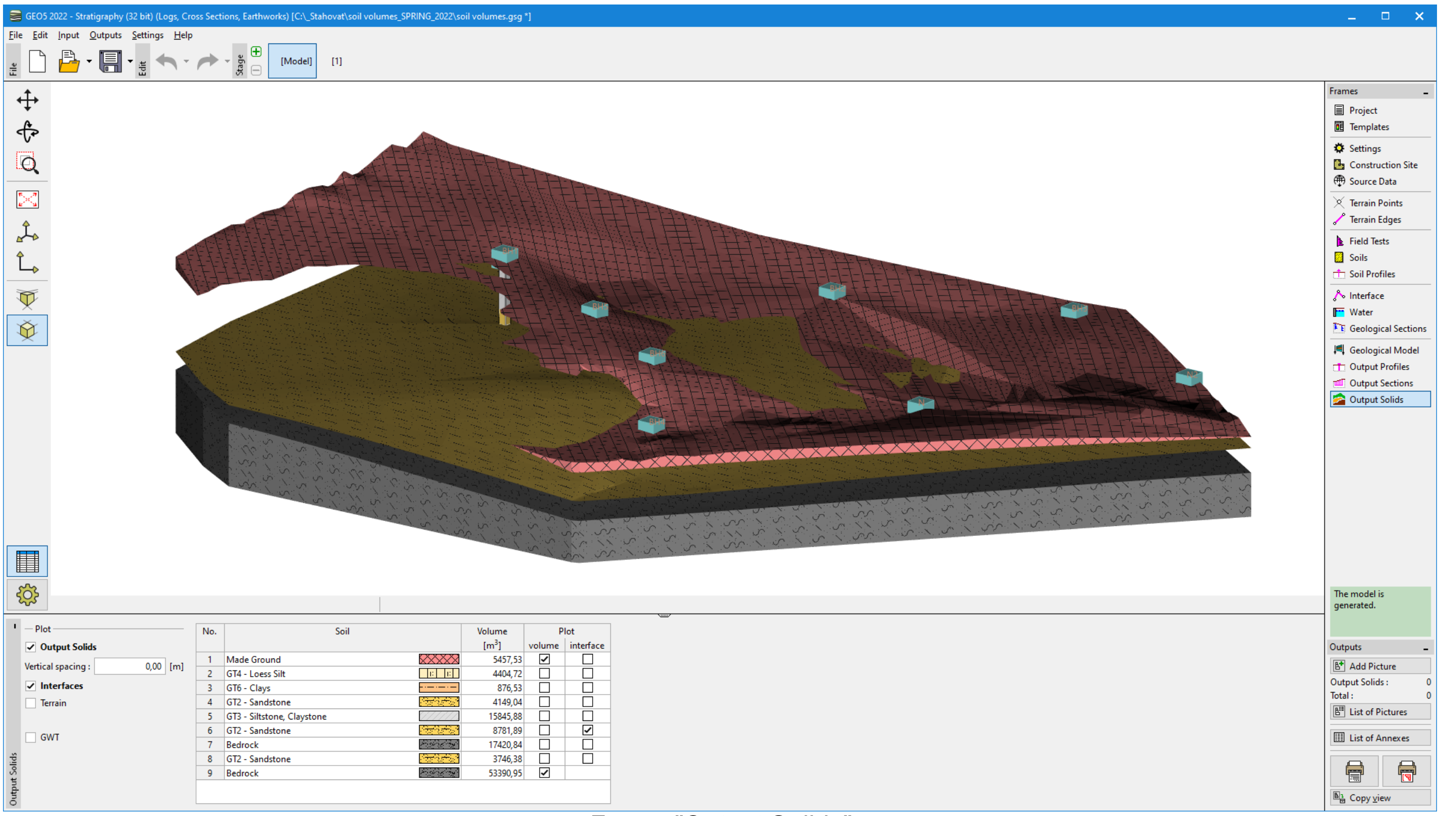
The result of this procedure is a copied cross-section, including all interfaces and soil assignments.



The resulting copy of an output section in the Slope Stability program

Output Solids

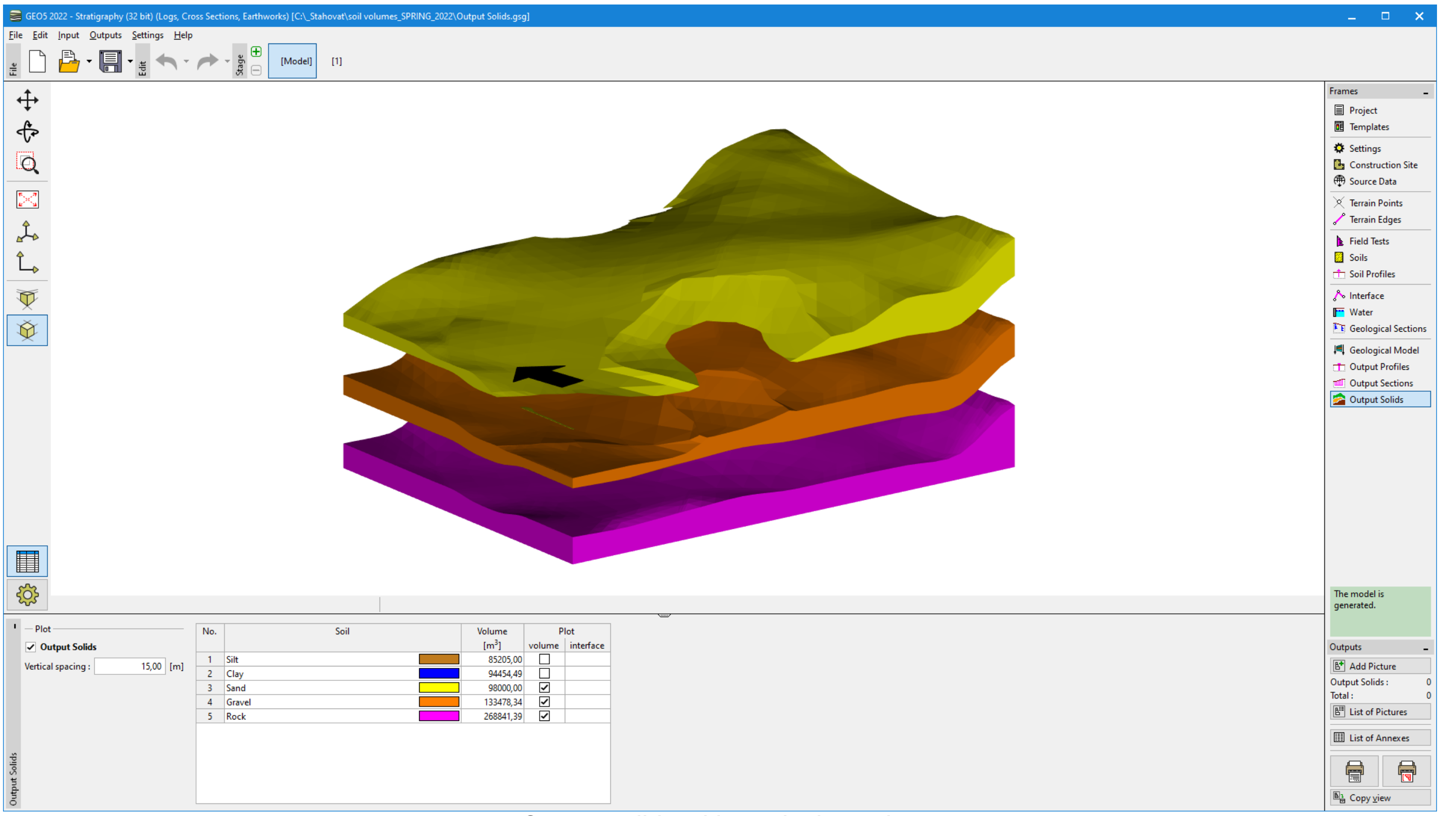
The "**Output Solids**" frame contains a table with all **output solids**. On the left side, it is possible to select the objects to display - output solids, interfaces, terrain, GWT.



Frame "Output Solids"

In the case of output solids and interfaces, it is possible to **display only some items** using the checkboxes in the table.

It is possible to define **vertical spacing** between output solids for better clarity of the layout of the layers. When defining the vertical spacing, only the output solids can be displayed.



Output solids with vertical spacing

Visualization of drawing on the desktop can be modified in the "Drawing Settings" frame and with the help of buttons on toolbar "Visualization".

The "Undo" button is an important program tool. It allows us to return back to the **original state before any modification**.

Earthworks

The "**Earthworks**" module is used for gradual **modeling of terrain changes** and **calculation of earthworks volumes**. The earthworks are modeled gradually in the individual construction stages.

- The input of data in the individual **construction stages**:

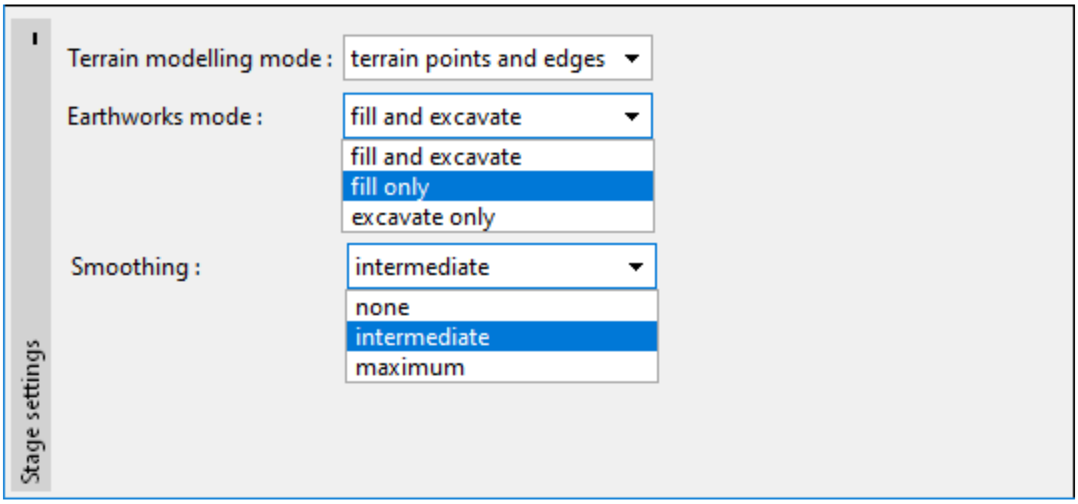
Stage Settings, External objects, Terrain Points, Terrain Edges, Earthworks, Water, Volumes, Output Profiles, Output Sections, Calculation

The **Terrain Points**, **Terrain Edges**, **Water**, **Output Profiles,** and **Output Sections** frames have the same functionality as for the creation of the geological model.

Stage Settings

In this frame, the "**Terrain modeling mode**" for the current stage is selected. Two ways can be used.

The modeling of the terrain using "terrain points and edges"



The new terrain is modelled using the terrain points and edges and it is independent of the previous construction stage or original geological model. Only the dimensions of the construction site remain the same. This approach is recommended in case we have a geodetical measurement in the given time period.

After points input, the terrain is generated automatically.

The "Earthworks mode" is an important choice. The terrain can be changed using the following modes:

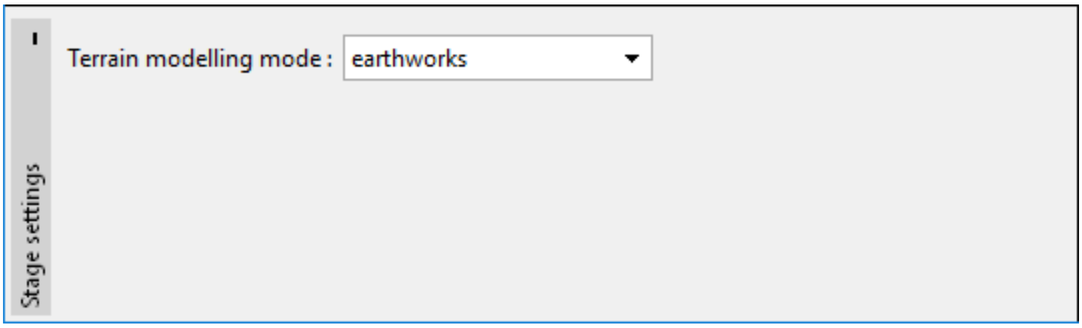
- fill and excavate against the original state - it is appropriate for landslides measurement, modeling of embankments, landfills

- fill only - in this case, the space between original and new terrain is filled with soil. It is advantageous, that the new terrain can be modeled easily by the input of only one point.

- excavate only - the original terrain is cut by new terrain. It is advantageous, that the new terrain can be modeled easily by the input of only one point.

The smoothing of the terrain in the current construction stage is also selected in this frame. If we have a detailed measurement of the whole model, it is appropriate to **smooth the terrain**. If we have only part of the terrain for cutting the model or for the embankment creation, it is appropriate to use **none smoothing**.

The modeling of the terrain using "earthworks" on the terrain of the previous construction stage.

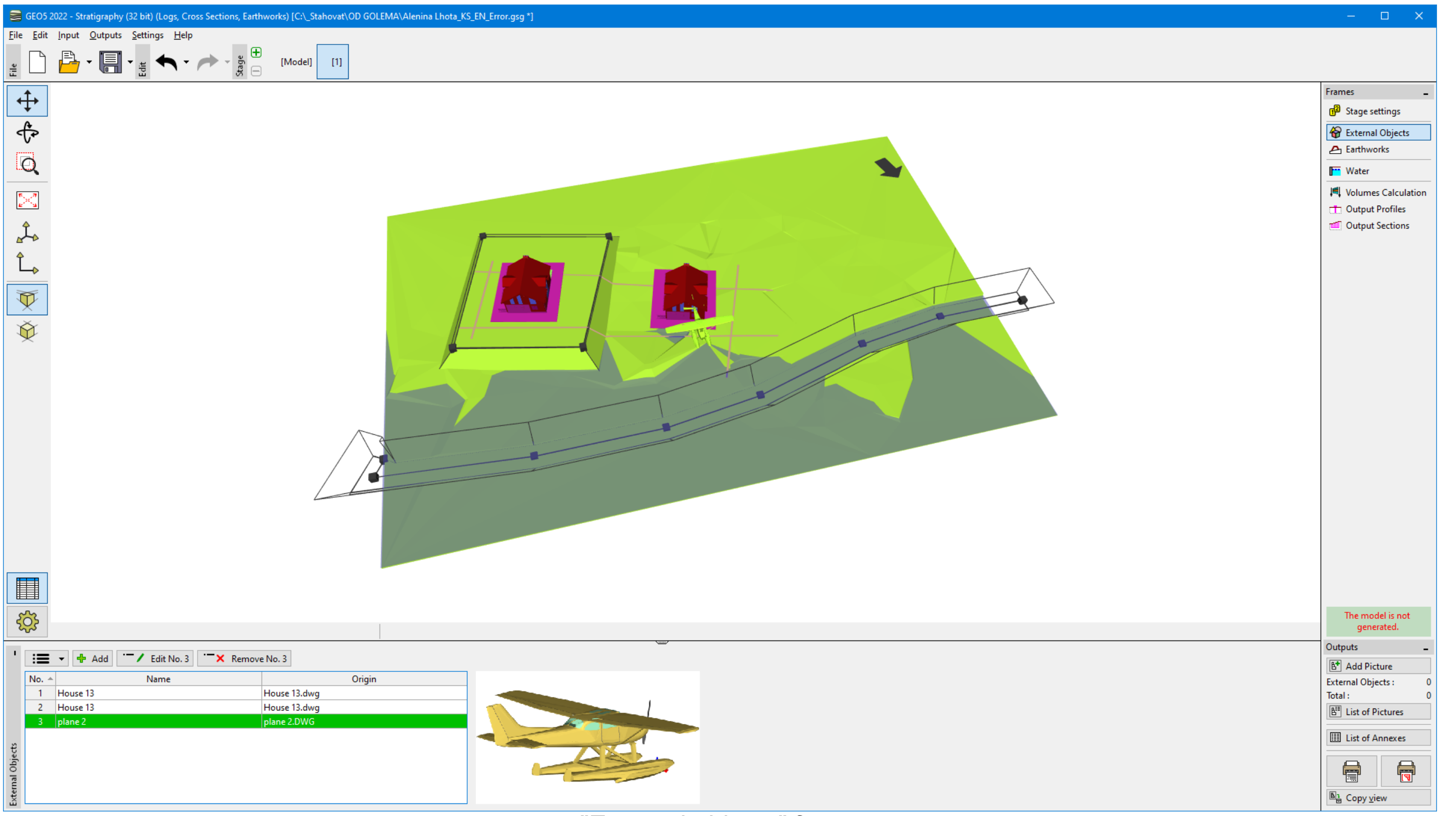


In this mode, we model the terrain changes using earthworks.

External Objects

The "**External Objects**" frame contains a table with a list of **imported** external objects. The name and origin of each object are displayed in the table - the selected object is shown next to the table.

Using the "**Add**" button and "Import from file" button we open the dialog window "**Import from file**".



"External objects" frame

Import of External Object

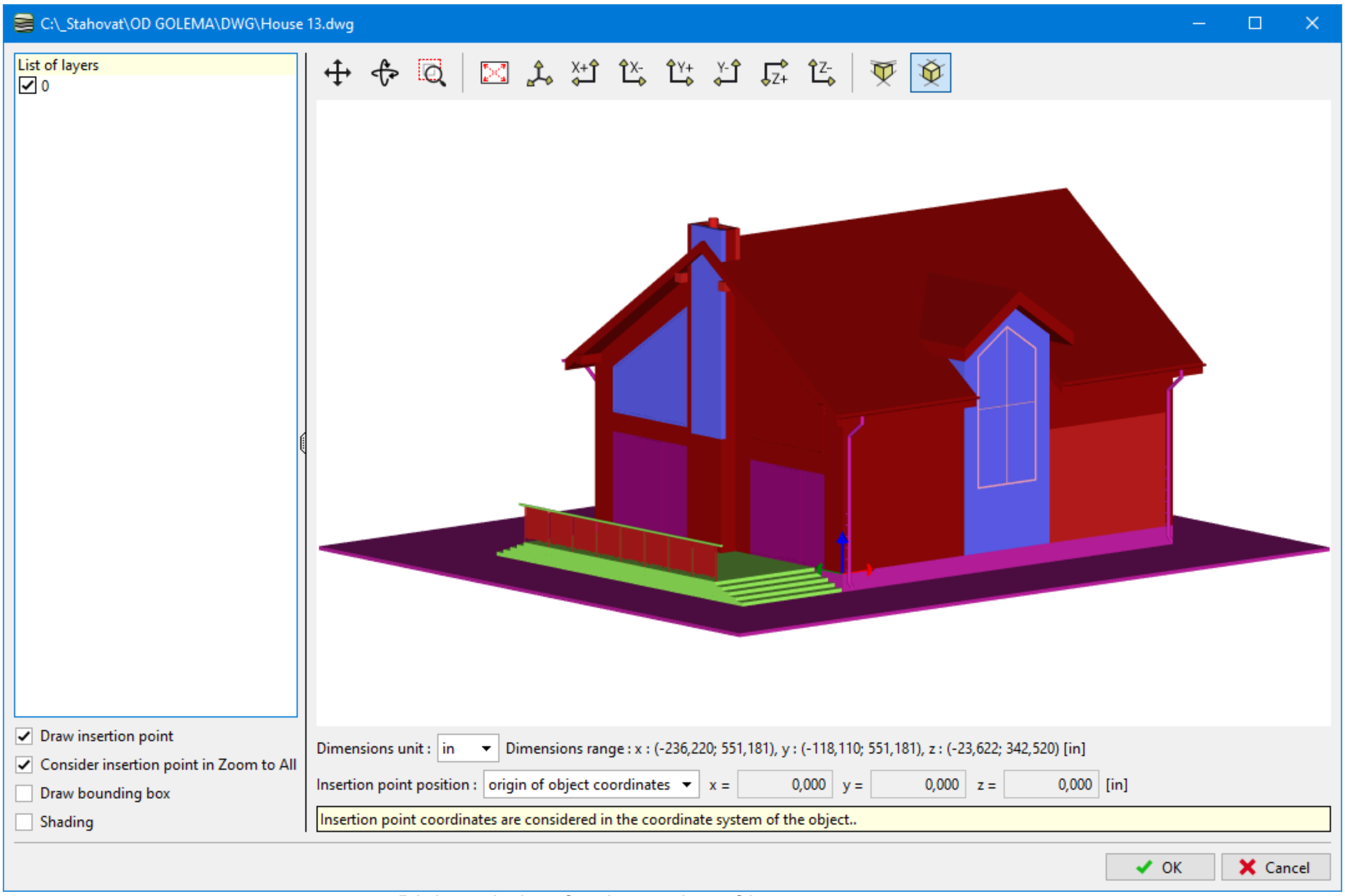
Import of object is done in the dialog window "**Import from file**". Using the button, the standard dialog window is opened - there we select the name and format of the imported file. It can be imported in the CAD formats (***.dwg**, ***.dxf**), or Fine Rendering Data (***.frd**) format.

In the dialog window, it is possible to:

- **turn on/off the layers** of the object
- **set the dimension unit** of the imported object (***mm* , *m* , *ft*** ...)
- **set the position of the insertion point**

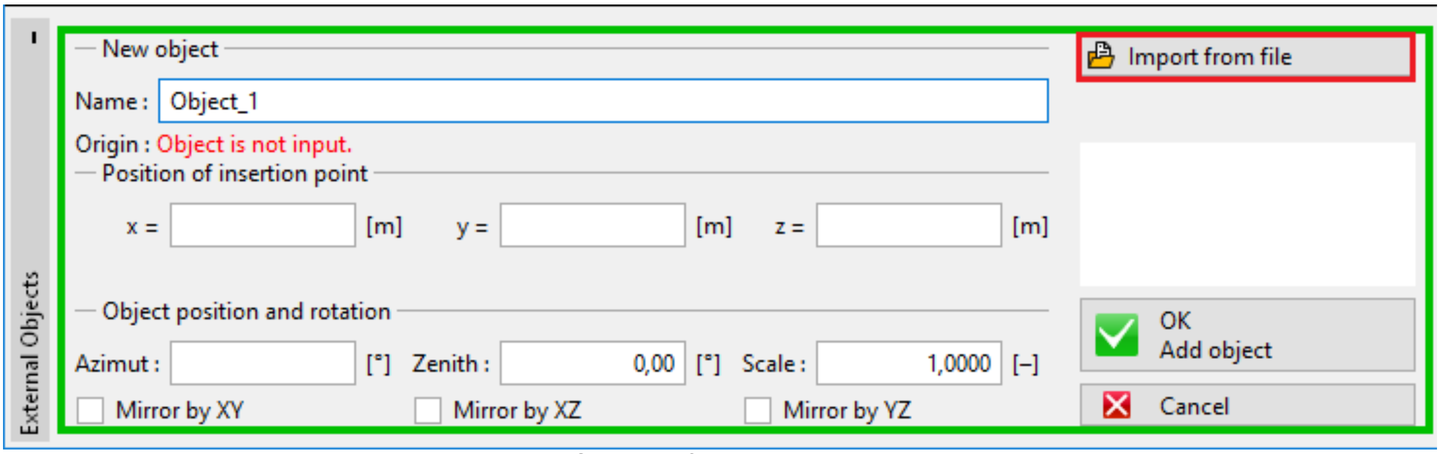
The object is inserted in two ways:

- if the coordinate system of the object **matches the coordinate system** of the task, the object is inserted in the correct coordinates,
- if the object is completely out of coordinate system ranges, the program **calculates the position of the insertion point** so that the object is approximately in the middle of the construction site.



Dialog window for the setting of import parameters

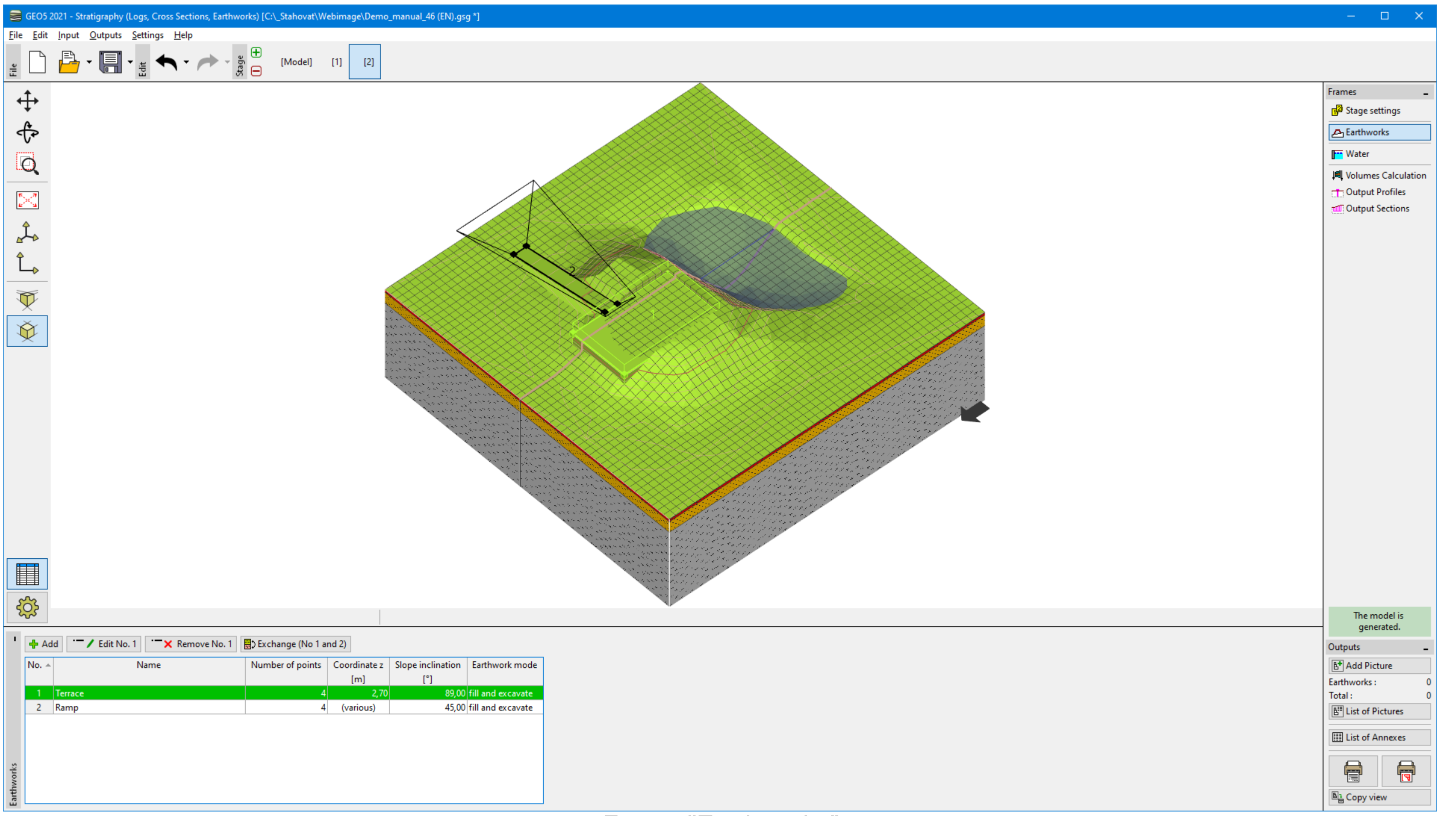
The object can be shifted by coordinates of insertion points, or it can be rotated or mirrored.



Change of object position

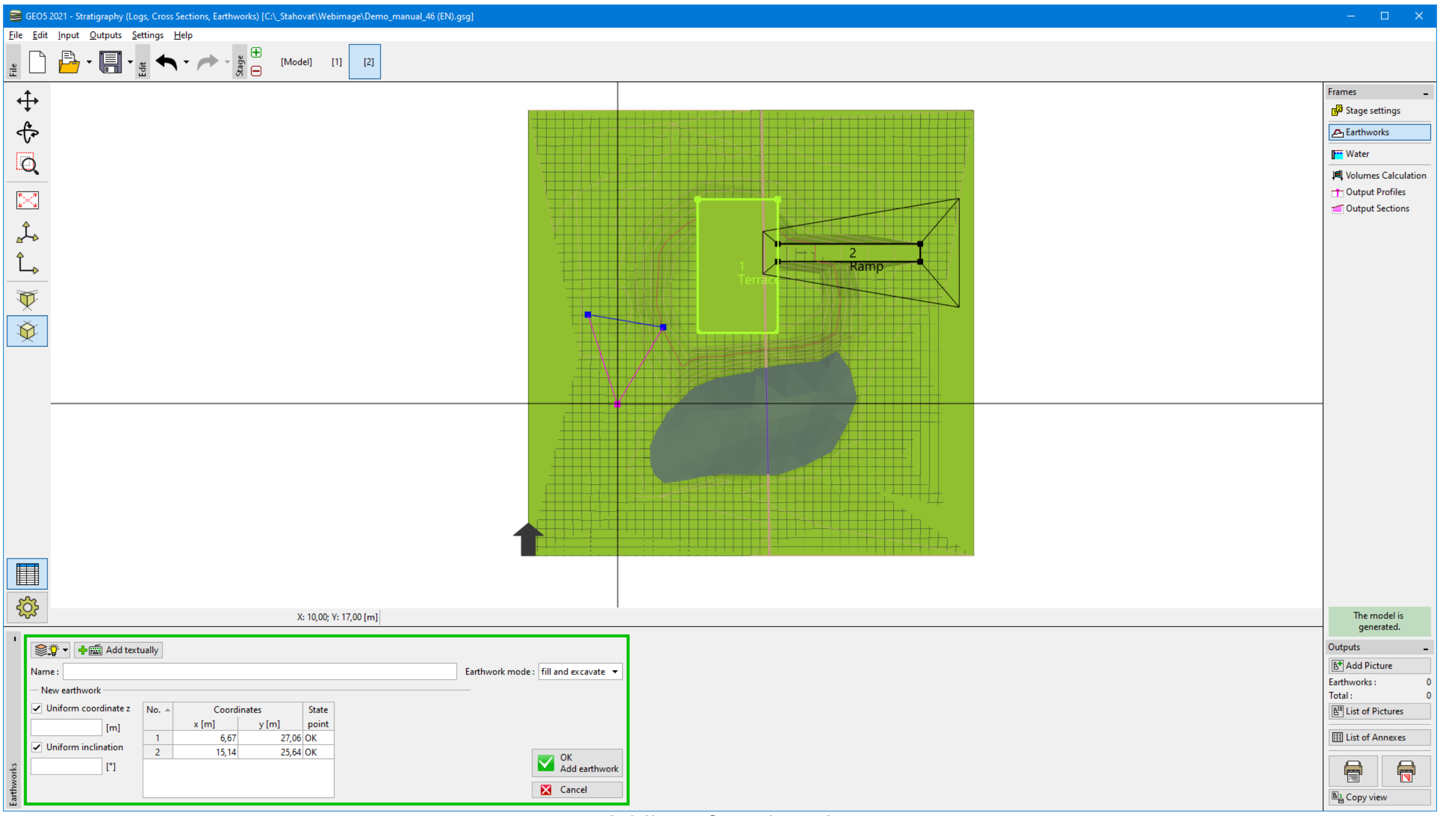
Earthworks

In this frame, it is possible to add, edit, or remove **earthworks**, which can be defined as polygonal (e.g. foundation slabs, spread footings), or line (e.g. roads, railways).



Frame "Earthworks"

The visualization of the frame is changed for the adding of **earthworks** after pressing the "**Add**" button.

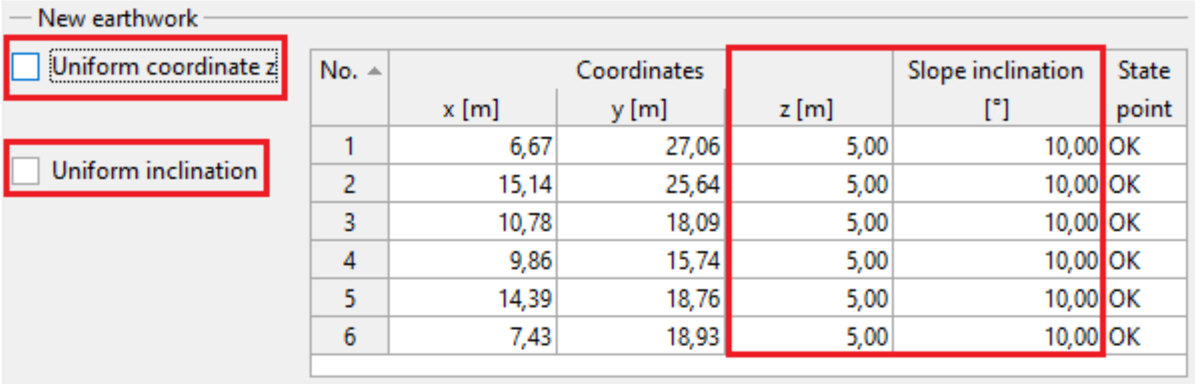


Adding of earthwork

Firstly for each earthwork, it is necessary to **define its plan view** - it can be done graphically using the mouse, or points can be inputted textually.

The "Earthworks mode" is an important choice, which determines how the terrain will be changed.

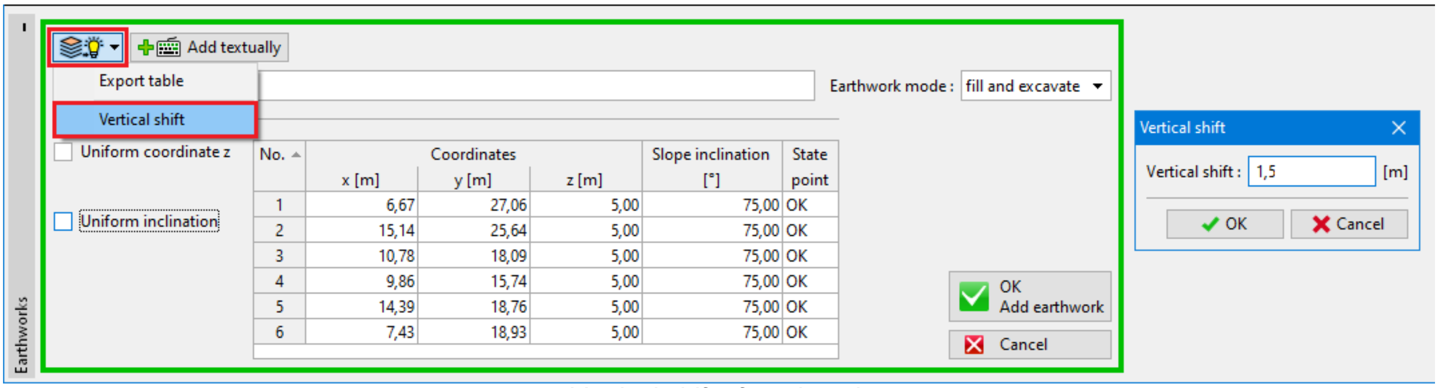
All points of earthwork can have a **uniform height (z coordinate)** and **uniform slope inclination**, which will be used for the connection of the new point and the original terrain. If we turn off the uniform input, z coordinate and slope inclination can be defined for each point separately - the table of points is extended for these data.



Non-uniform input of "z" coordinate and slope inclination

Note: If we want to use a vertical edge of earthwork, it is necessary to input 89° inclination (90° inclination is not allowed)

The special option is a **vertical shift** of the whole earthwork - it can be advantageous for transportation structures, where we want to minimalize the difference between excavated and filled soils. The calculated volumes of excavated and filled soils are shown in the "Volumes Calculation" frame.

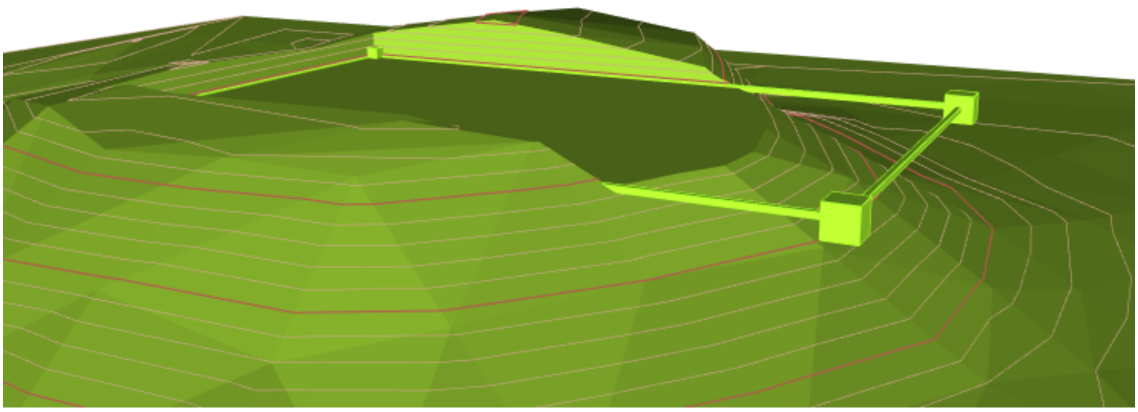


Vertical shift of earthwork

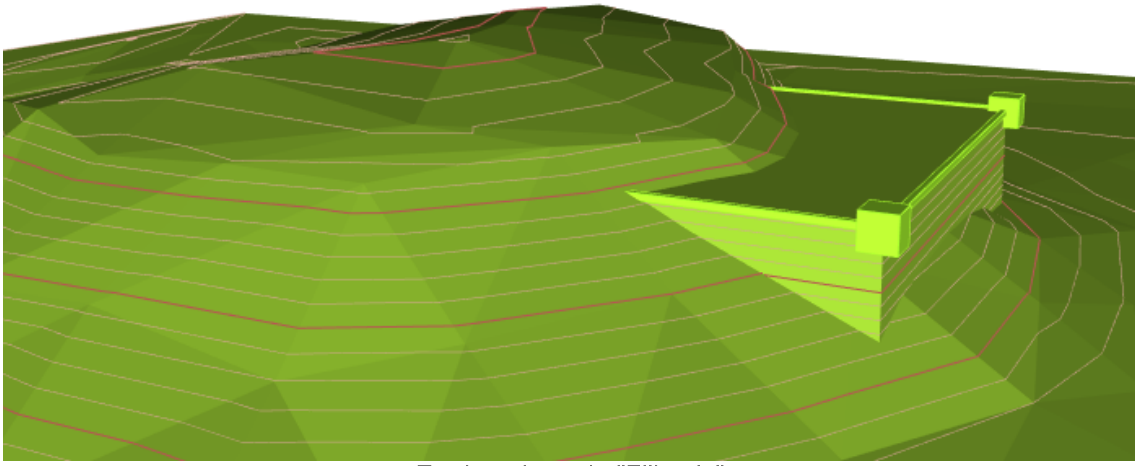
Earthwork Mode

The "**Earthwork mode**" determines how the new terrain will be changed. New terrain can be created by **filling** by new soil, **excavation of existing** soil, or by the **combination of both** modes.

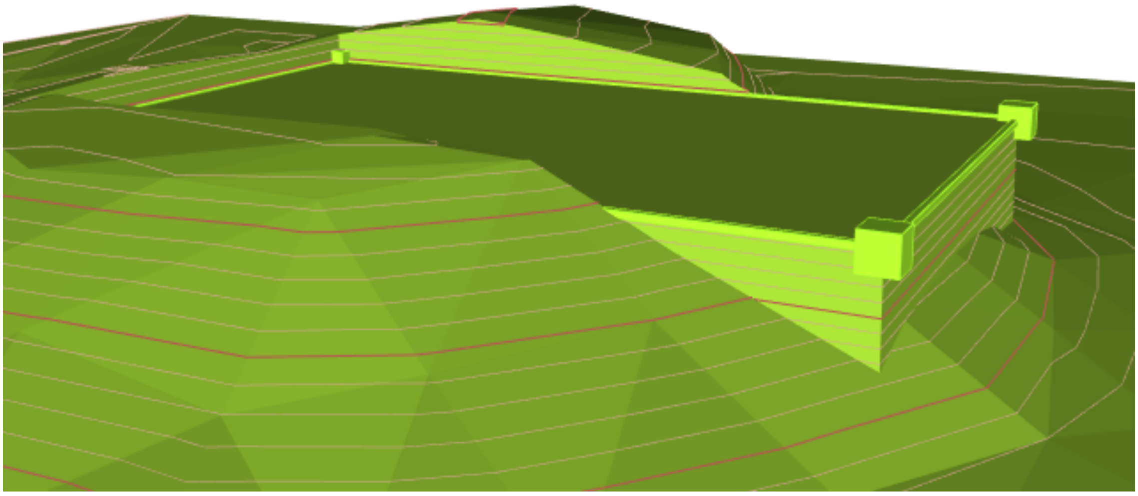
The principle is clear from the following example - all three earthworks have the same plan view.



Earthwork mode "Excavate only"



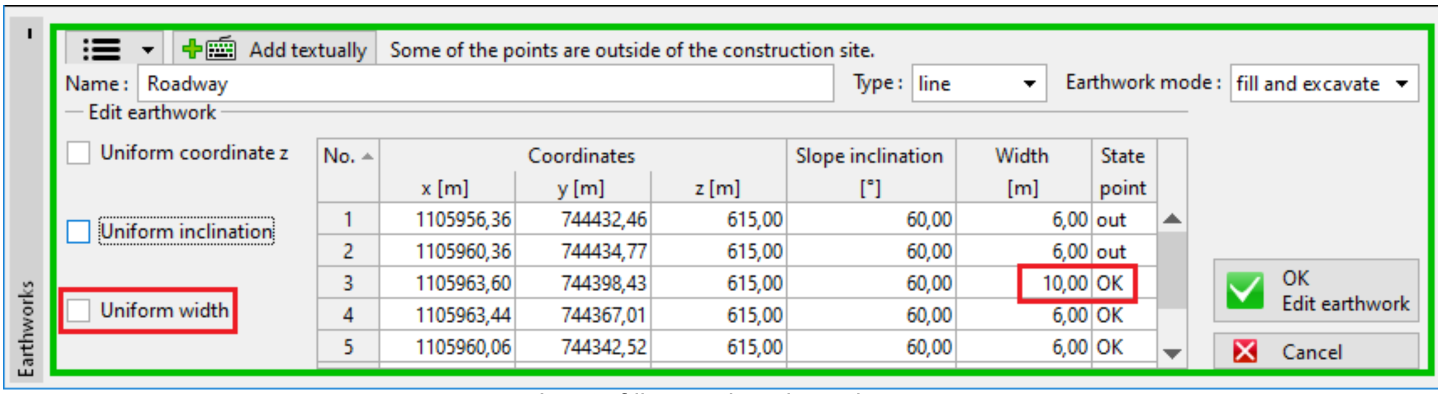
Earthwork mode "Fill only"



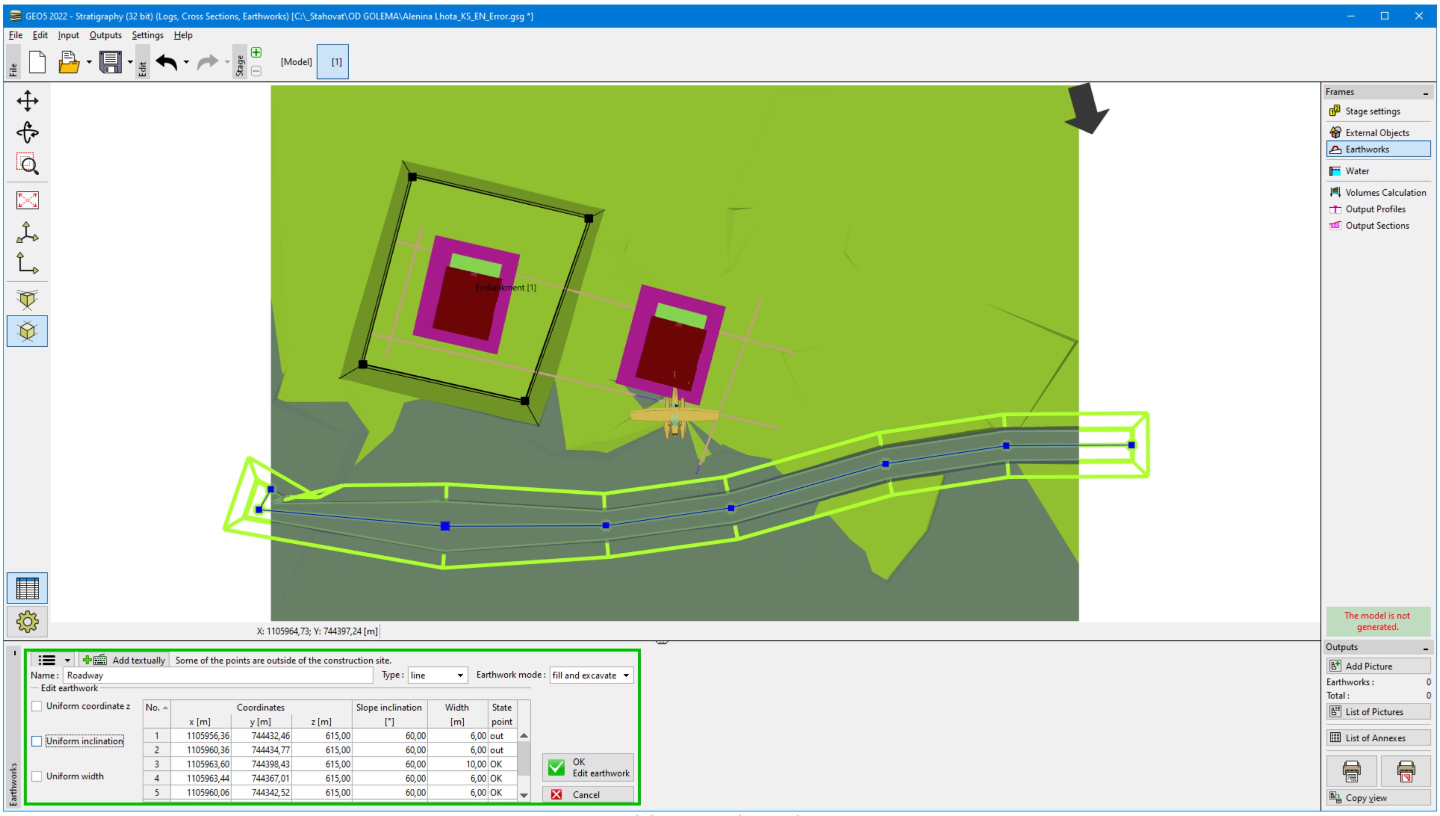
Earthwork mode "Fill and excavate"

Line Earthwork

Line earthwork is defined by its axis, which is inputted by points with (x, y) coordinates. The z coordinate, width of earthwork, and slope inclination are inputted for each section of line earthwork. In the case, that some of these parameters are the same in all points, they can be entered as uniform. For example, if we check the "**Uniform coordinate z**", all points of line earthwork have the same altitude. The same approach can be used also for earthwork width or slope inclination.



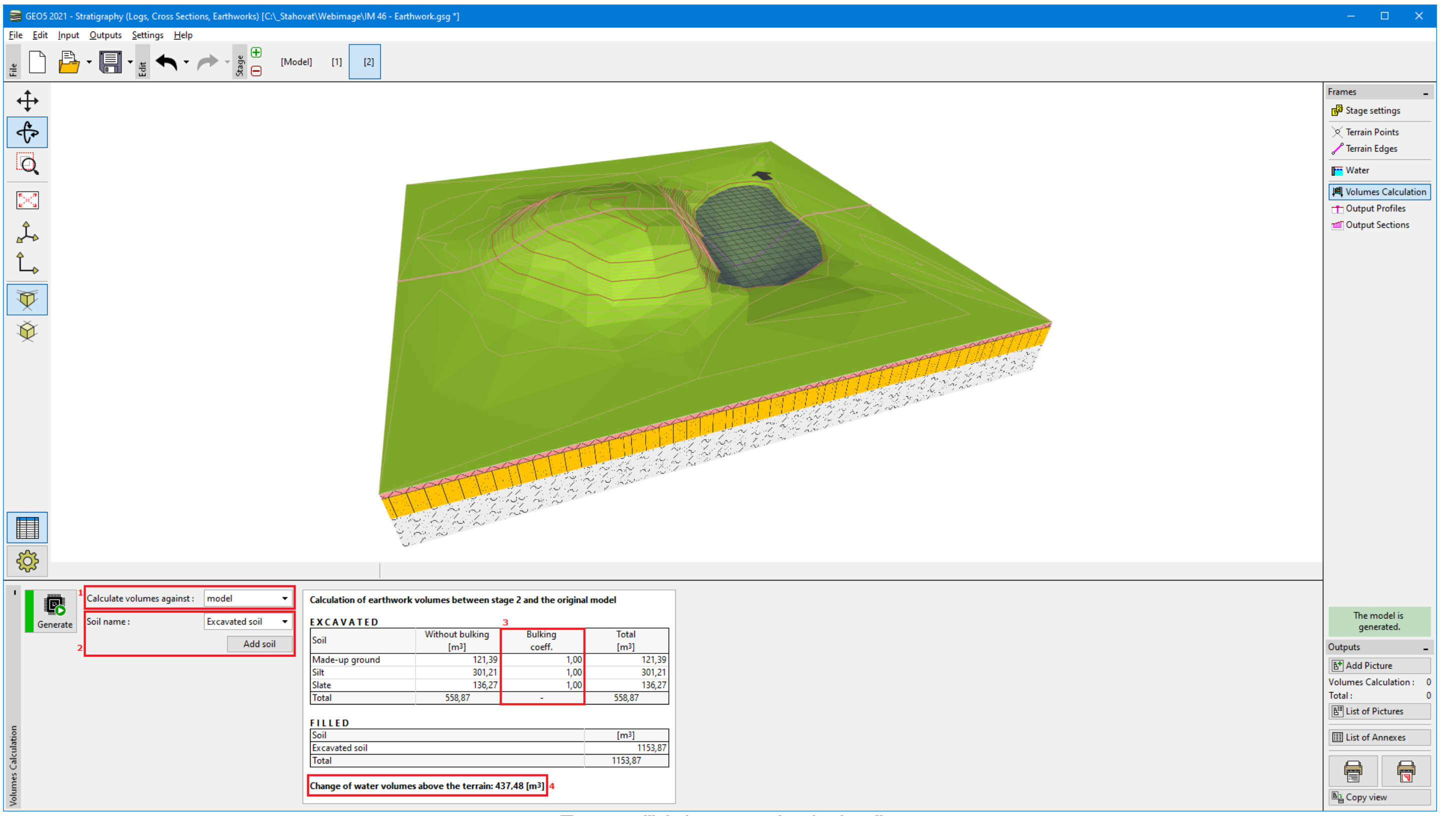
Input of line earthwork sections



Line earthwork

Volumes Calculation

In this frame, it is possible to calculate volumes of the **excavated** and **filled soils**



Frame "Volume calculation"

Volumes of soils can be calculated **against the previous construction stage** or **the original model (1).**

Further, it is possible to define **soil, which will be used for filling (2)** - it is selected from the list of already defined soils or it can be added directly in this frame using the "**Add soil**" button.

The volume of excavated soils depends on the "**bulking coefficient**" **(3)** - it is defined for each soil in the "Soils" frame.

The program can be also used for the calculation of **change of water volume above the terrain (4)**.

Program FEM

Program FEM (and modules Consolidation, Water Flow, Tunnel, Earthquake) can model and analyze a wide range of geotechnical problems including:

- terrain settlement, consolidation
- sheeting structures
- anchored support structures
- slope stability
- excavation, tunnel analysis
- calculations of tunnels, ground losses
- calculation of water flow etc.

The help in the program "FEM" includes the following topics:

- The input of data into individual frames:

Topology:	Project	Settings	Interface	Soils	Rigid Bodies	Assign
Contact Types	Lining	Free Points	Free Lines	Point Refinement	Line Refinement	Free Refinements
Mesh Generation						

Construction Stages:	Excavation	Activation	Assign	Lining	Beams	Contacts
Contacts and Beams	Point Supports	Point Flow	Line Supports	Line Flow	Anchors	Nails
Props	Reinforcements	Surcharge	Elastic Regions	Beam Loads	Water	Analysis
Monitors	Graphs	Stability	Earthquake			

- Outputs

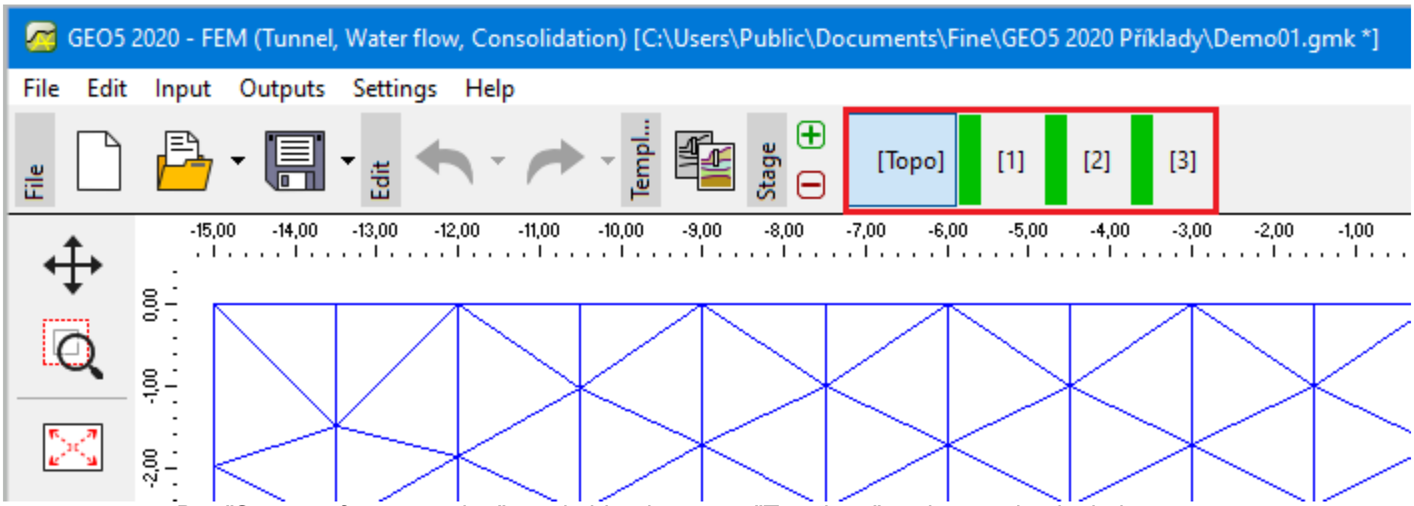
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

Topology

To input data in the FEM program slightly differs from our other programs in that it requires defining the topology of the structure prior to any calculation. This step includes introduction of interfaces between individual layers of soils, line constructions, parameters of soils and interfaces and at last generation of the finite element mesh. To avoid unexpected errors when creating a computational model the user should first become familiar with available coordinate systems.

The topology input regime is selected by clicking the "**Topo**" button on the horizontal bar "Construction Stages".



Bar "Stages of construction" - switching between "Topology" regime and calculation stages

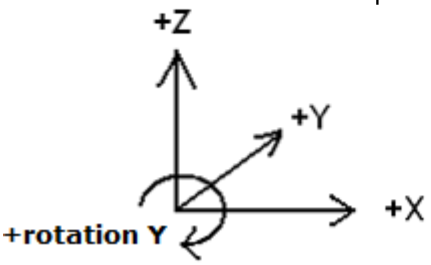
The actual analysis is performed in individual stages of construction (calculation stages), which allow the user to define activity of structures, to input beams, anchors and surcharge, to model the effect of water, etc.

Depending on the selected regime the vertical toolbar also adjusted.

Coordinate Systems

Global coordinate system

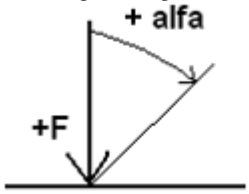
- is right-handed
- the positive X axis is directed from the left to the right
- the positive Z axis is directed from the bottom to the top
- the positive Y axis drills in to the XZ plane
- the rotation about the Y axis is positive when measured clockwise



- The GCS is used for coordinates
- in general, the positive surcharge is assumed to act in the opposite direction to the positive axis and the positive rotation follows the positive sense of the global rotation
- particular definitions of the positive direction must be carefully examined for all cases

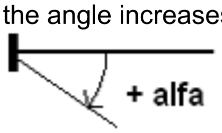
Surcharge

- is always assumed to act along the horizontal line (or at a point)
- the origin (point) and the length are the required input data
- the positive surcharge at zero angle is assumed to act in the opposite direction to the positive direction of the Z axis
- the zero angle corresponds to vertical surcharge
- the angle increases clockwise
- the angle ranges from $<-180^{\circ}$ to $180^{\circ}>$



Anchors

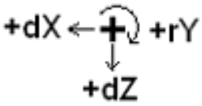
- an anchor can also be specified by the origin and an angle
- the zero angle corresponds to the direction of the X axis
- the angle increases clockwise



- the angle ranges from < -180° to 180°>

Prescribed displacements and rotation of supports

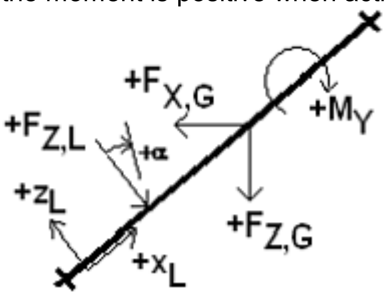
- prescribed displacements are positive in the directions of the X, Z axes and about the Y axis
- displacements are positive when developed in the opposite direction to the positive directions of the coordinate axes



- the positive rotation is measured clockwise

Load of beams

- the local coordinate system is right-handed
- the positive X_L axis of the beam is assumed in direction from the starting to the end point
- the positive Z_L axis is perpendicular and rotated counterclockwise by 90^o from the beam axis
- load can be applied three directions:
  - global Z
  - global X
  - local normal (Z)
- the positive load in the global direction acts in the opposite direction to the positive direction of the corresponding axis
- the positive load in the normal direction acts in the opposite direction to the positive direction of the local Z_L axis
- the positive load angle α is measured clockwise
- the moment is positive when acting clockwise



- definition of load along the X_L axis
  - coordinates, coordinates of the origin
  - load span
- types of load (always in above mentioned directions)
  - concentrated force
  - concentrated moment
  - distributed uniform over the entire beam
  - distributed trapezoidal over the entire beam
  - distributed uniform over a segment of the beam
  - distributed trapezoidal over a segment of the beam

Stresses and strains

- positive normal stress *Sigma* corresponds to compression, negative to tension
- positive normal strain *Epsilon* corresponds to compression, negative to tension

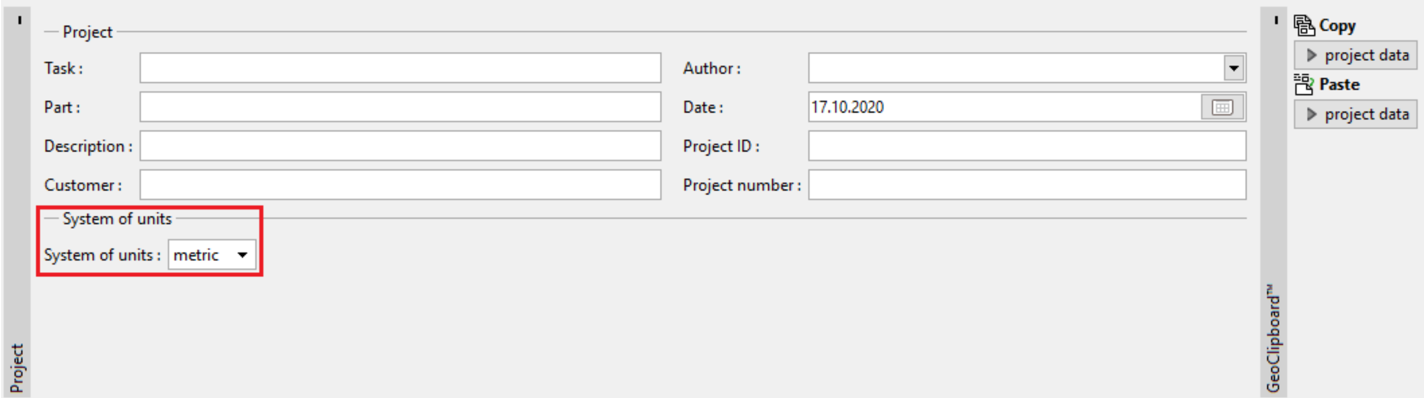
Internal forces along beams

- positive normal force corresponds to tension, negative to compression
- positive normal strain *Epsilon* corresponds to compression, negative to tension

Project

The **"Project"** frame is used to input basic project data and to specify the settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows us to switch analysis units (metric/imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

Settings

The frame "**Settings**" serves to specify standards or methods that are used to perform the analysis.

The "**Analyses**" tab allows the user to define the basic characteristics of the analysis to be carried out including the type of the problem and analysis, the method of calculating the initial stress (geostatic stress, K_o procedure) and available standards for concrete and steel structures.

The available problem (plane strain analysis, axial symmetry) **and analysis** (**stress**, slope stability, water flow, tunnels, consolidation, earthquake) **types depend on the purchased configuration of the program.**

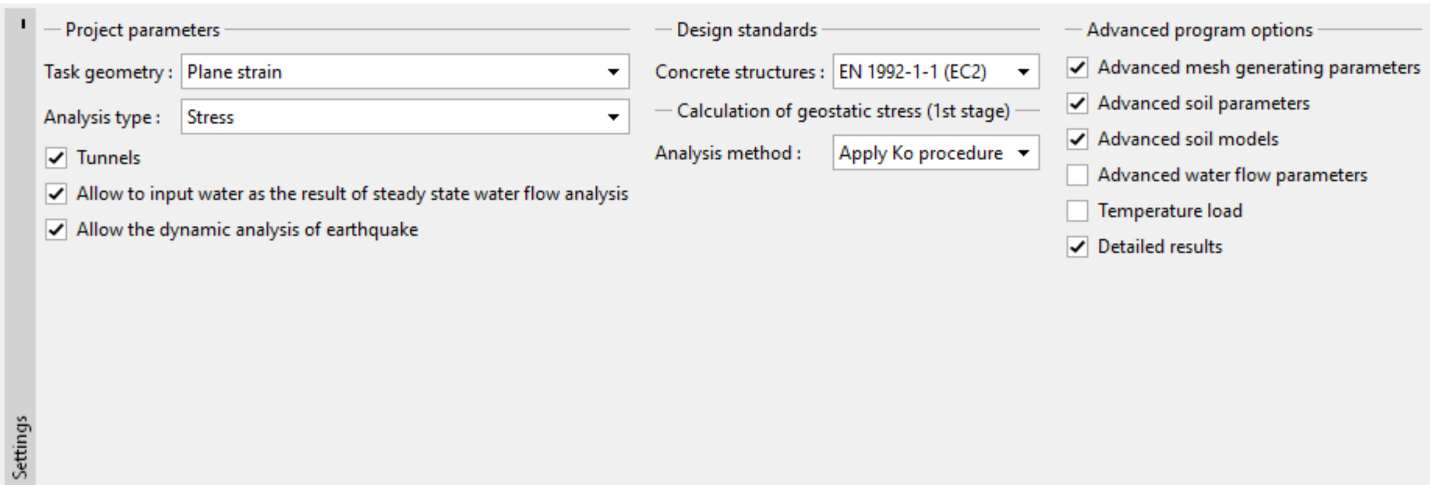
Providing all modes are available we recommend to proceed with extreme caution when selecting the type analysis - more complex types require distinctively larger number of input data and may unnecessarily complicate the use of the program.

This frame also serves to select the method for calculating the initial stress in the first stage of construction - either standard calculation of **geostatic stress** or the K_o procedure.

It is also possible to select **advanced program options**, which are usually not necessary for common analysis.

Available advanced options:

- **Advanced mesh generating parameters**
- **Advanced soil parameters**
- **Advanced soil models**
- **Advanced water flow parameters**
- **Temperature load**
- **Detailed results**



Frame "Settings"

Stability Analysis

There are two options available in the FEM program to solve the slope stability problem:

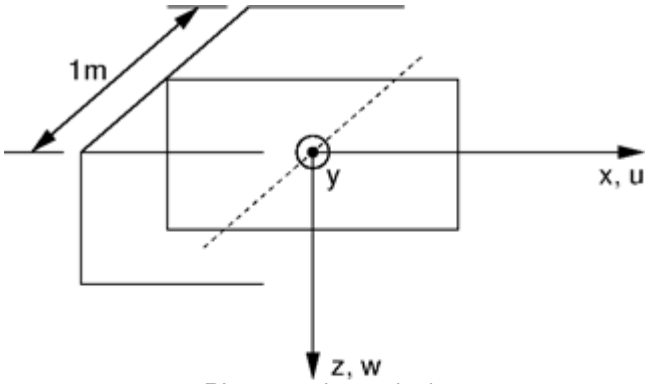
1. To set the solution type to "**Slope stability**" in the frame "Settings".

2. To run the module in the "**Slope stability**" regime in an arbitrary stage of construction of a standard analysis by pressing the "**Stability**" button - in such a case, a new secondary task (which can be saved independently) is generated. The solution then proceeds as in the step 1.

Creating model and inputting data in the "**Slope stability**" regime is performed in the same way as in the "**Stress**" regime - just the "**Analysis**" button launches the slope stability analysis of a given structure. Individual slope stability analyses in construction stages are completely independent and have no relation to **the previous stages and calculations**.

Plane Strain Analysis

This computational module is suitable for the analysis of longitudinal structures (**tunnel**, **embankment**, **dam**, etc.) characterized by a longitudinal dimensions being of the orders of magnitude larger than transverse dimensions of the analyzed domain.



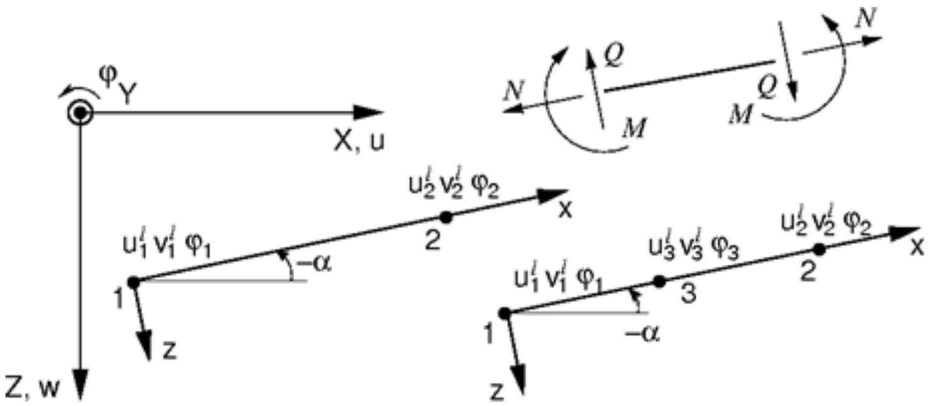
Plane strain analysis

In such a case the analysis can be carried per 1 m run of the structure, see Figure. This complies with the plane strain assumption. Components of the strain vector developed on planes normal to the longitudinal axis can then be neglected. Therefore, we assume the soil body be loaded by the components of the strain and stress vector pertinent to the transverse plane normal to the longitudinal axis and by the longitudinal normal stress that arises due to Poisson's effect. The corresponding non-zero components of the stress and strain vector are:

$$\sigma^T = \left\{\sigma_{xx}\ \sigma_{zz}\ \tau_{xz}\ \sigma_{yy}\right\}$$

$$\varepsilon^T = \left\{\varepsilon_{xx}\ \varepsilon_{zz}\ \gamma_{xz}\ \varepsilon_{yy} = 0\right\}$$

Considering beam elements the analysis corresponds to the solution of a plate strip having the cross section width equal to 1 m . Non-zero components of nodal generalized displacements are evident from the following Figure for a two-node beam element compatible with a three-node triangular plane element and a three-node beam element compatible with a six-node triangular plane element.



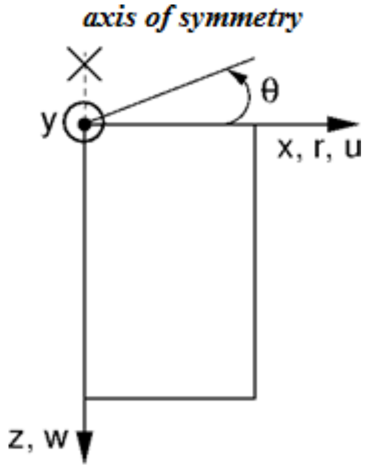
Two-node and three-node beam element

The corresponding components of internal forces, see Figure, assumed with respect to 1 m of cross sectional width are given by:

$$\sigma^T = \left\{N \equiv n_{x1},\ M \equiv m_y,\ Q \equiv q_{z1}\right\}$$

Axial Symmetry

This computational model is suitable for the analysis of structures of revolution. This assumption must be satisfied from both the construction and load point of view. A typical example is the solution of vertically loaded isolated pile, excavation of circular ditch, or pumping of water from a circular hole.



Axisymmetric analysis

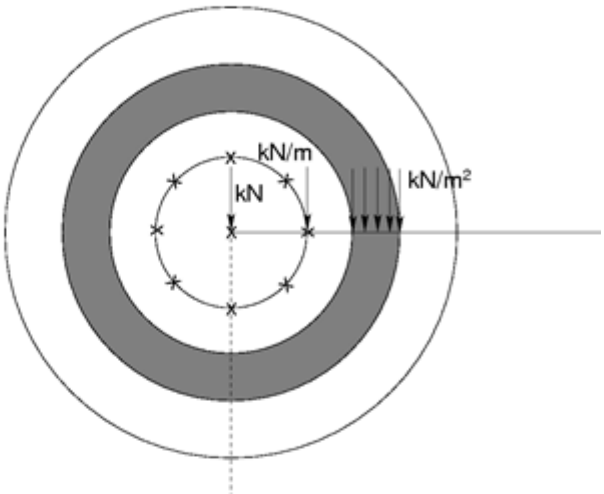
Similarly to the plane strain analysis the computational problem is three dimensional, which can, however, be transformed into a two-dimensional problem, see Figure. The analysis is then performed with respect to 1 ***m*** of arc length having diameter equal to ***x*** (***r***). The axis of symmetry always corresponds to the origin of the ***x*** (***r***) coordinate. Shear strain components in the direction of rotation can be neglected. We are then left with the stress and strain components acting on the plane of symmetry cut and the normal strain and stress component in the hoop (circumferential direction) direction. The corresponding non-zero components of the stress and strain vector are:

$$\sigma^T = \{\sigma_{xx}\ \sigma_{zz}\ \tau_{xz}\ \sigma_{\theta\theta}\}$$

$$\varepsilon^T = \left\{\varepsilon_{xx}\ \varepsilon_{zz}\ \gamma_{xz}\ \varepsilon_{\theta\theta} = \frac{u}{r}\right\}$$

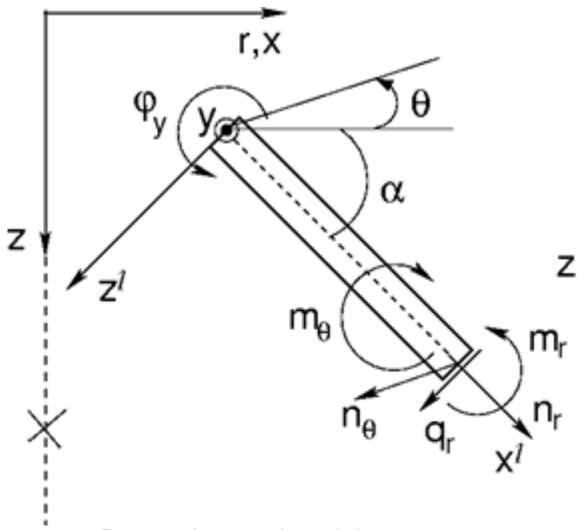
It is clear that the hoop strain, and therefore also the affected normal stresses, attains an infinite value at the axis of symmetry. Thus regarding the finite element approximation, arriving at reliable and sufficiently accurate estimates of these values requires a relatively fine mesh along the symmetry axis.

The application of line and surface load is also worth mentioning. Several examples of applying the load on terrain surface are displayed in the following Figure. Clearly, their effect increases with the distance from the axis of symmetry. Introducing such a type of load directly on the axis of symmetry has, therefore, no effect. In such a case it is necessary to choose the type of load of axis of symmetry. The program allows for the application of concentrated forces only.



Examples of load applied on terrain surface

Considering beam elements the analysis corresponds to the solution of a plane membrane of revolution including bending effects. Non-zero degrees of freedom are identical adopted for beam elements in plane strain analysis. Apart from axial (meridian) effects it is necessary to consider also membrane and bending effects in the hoop direction, see the following Figure.

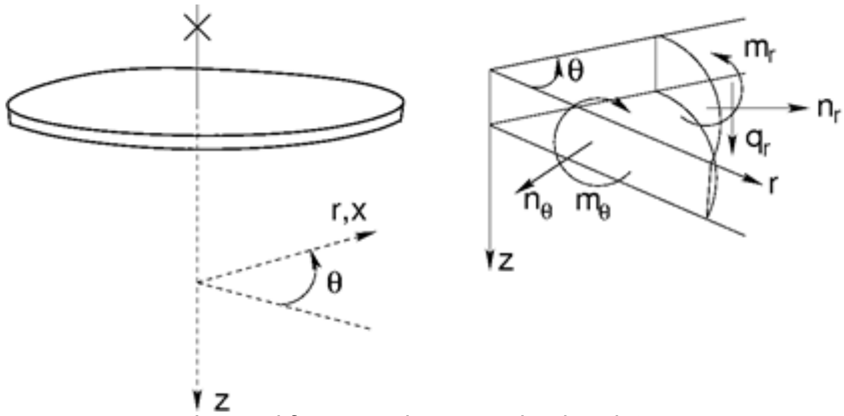


Beam element in axial symmetry

The corresponding components of internal forces, see Figure, assumed with respect to 1 ***m*** of cross sectional width are given by:

$$\sigma^T = \{n_r, m_r, q_r, n_\theta, m_\theta\}$$

In a special case of a circular plate (angle $\alpha = 0$) we may refer to radial and hoop components of internal forces, see the following Figure.



Internal forces acting on a circular plate

Internal forces are related to corresponding strains and curvatures as follows:

$$\begin{Bmatrix} n_r \\ m_r \\ q_r \\ n_\theta \\ m_\theta \end{Bmatrix} = \frac{1}{1-\nu^2} \begin{bmatrix} EA & 0 & 0 & \nu EA & 0 \\ 0 & EI_y & 0 & 0 & \nu EI_y \\ 0 & 0 & \dfrac{(1-\nu)kEA}{2} & 0 & 0 \\ \nu EI_y & 0 & 0 & EA & 0 \\ 0 & \nu EI_y & 0 & 0 & EI_y \end{bmatrix} \begin{Bmatrix} \dfrac{du^l}{dx} \\ \dfrac{d\varphi_y}{dx} \\ \varphi_y + \dfrac{dw^l}{dx} \\ \dfrac{u^l \sin\alpha - w^l \cos\alpha}{r} \\ \dfrac{\varphi_y \cos\alpha}{r} \end{Bmatrix}$$

After setting r equal to infinity we arrive at plane strain conditions. It is worth noting that in case of shear forces their magnitudes are, unlike the plane strain analysis, significantly dependent on the refinement of the finite element mesh. This holds also for vertical reactions.

Note to water flow

Recall that similarly to the reactions forces in the stress analysis the point fluxes at nodes with prescribed pore pressures are evaluated with respect to 1 ***m*** of arc length having diameter equal to ***x*** (***r***). In case of plane strain analysis the corresponding values are taken per 1 ***m*** run. The corresponding overall fluxes (inflow/outflow) can be determined from point fluxes [$m^3/day/m$] as follows:

Plane strain analysis

$$\sum Q = \sum_{i=1}^{N} Q_i \quad [m^3/ day / m]$$

Axisymmetric analysis

$$\sum Q = \sum_{i=1}^{N} 2\pi x_i Q_i \quad [m^3 / day]$$

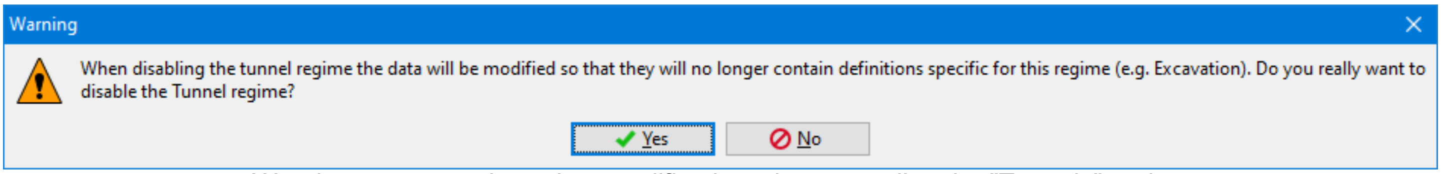
where N is the number of nodes along a given mesh line, in which the point fluxes Q_i $[m^3/day/m]$ are calculated. In case of axisymmetric analysis x_i represents the x -coordinate of a given point. Therefore, the axisymmetric analysis provides total the inflow/outflow $[m^3/day]$ through for example a cylindrical surface (vertical line) or circular surface (horizontal line).

Tunnels

The frame "Settings" in conjunction with the "**Analyses**" tab allows for selecting the option "**Tunnels**". (This module has to be **purchased** by the user - otherwise this option is not available). When selecting the "**Tunnels**" regime it is possible to define and calculate:

- **excavations** (Modeling a 3D effect at the tunnel face assuming the New Austrian method)
- **gradual degradation of beams**
- **subjecting beams to thermal load**
- **thermal load applied to selected regions** ("Advanced program options")
- **prescribing swelling stress to selected regions**
- **monitoring results**

The mode "**Tunnels**" can be switched on/off at any time. The previous results will be, however, deleted. While switching from a standard regime to the "**Tunnels**" regime is safe, proceeding in the opposite direction results into deleting all input data - a warning message, however, appears before this action is completed.



Warning message about data modification when canceling the "Tunnels" regime

Consolidation

Consolidation analysis is the optional module of **FEM** program. We can switch the consolidation analysis on in the "**Settings**" frame by switching "**Analysis type**" to "**Consolidation**".

The input has the following limitations:

- the "**Tunnels**" regime cannot be used
- only some materials models can be used: Elastic, Mohr-Coulomb, Modified Mohr-Coulomb and Drucker-Prager
- active regions can be fully specified in the 1st construction stage only, in the following construction stages only activating the regions located above the terrain of the previous construction stage is allowed. No openings or cuts are possible.
- mesh is obligatorily generated using multinode elements
- water can be entered in the 1st construction stage only and "**No water**" or "**GWT**" can only be used
- beams are always in transverse direction impermeable, only the drain along the beam on each side of the beam can be specified
- no point flow is available, the line flow on the borders have only "**permeable**" or "**impermeable**" boundary conditions

The data input and calculation is as follows:

- topology is defined the same way as in stress analysis, the finite element mesh is generated
- in the first construction stage in the "**Water**" frame the pore pressure is specified by entering groundwater table
- in the following construction stages are entered: the data for stress analysis, the flow boundary conditions, beam and contact flow parameters, analysis parameters, i.e. phase time and load action

The results of the analysis are presented in the same way as in stress analysis, in addition, the flow velocities are available.

The principle of the numerical solution is here.

Principle of Numerical Solution of Consolidation

Consolidation

In standard stress analysis the program GEO5 FEM allows for two specific approaches to the modeling of pore pressure action on a soil body. In case of undrained conditions it is assumed that all boundaries surrounding the undrained soil are impermeable, the soil is considered as volumetrically incompressible, and the applied load results in the generation of excess pore pressure within this layer. Introducing suitable boundary conditions that allow for a gradual dissipation of excess pore pressure provides passage to drained conditions. In case of drained conditions we assume that the resulting pore pressure is no longer influenced by the deformation of the soil body. Transition from drained to undrained conditions describes the theory of consolidation.

The term consolidation stands for the soil deformation in time caused by external load, which can be either constant or time dependent. This is a reological process. For the present case we limit our attention to so called primary consolidation characterized by the reduction of volume of pores and thus the change of the internal soil structure due to load accompanied by the escape of water from pores. The analysis assumes a fully saturated soil. Consolidation analysis in a partially saturated soil is not addressed by the program. The governing equation describing the flow of water (continuity equation, $\dot{a}$ represents a time derivative of a given quantity) in a fully saturated ($S = 1$, $\dot{S} = 0$) deforming soil body is provided by (recall the Richards equation for the description of transient water flow).

$$\frac{1}{\mathrm{M}}\dot{p} + \alpha\dot{\varepsilon}_{\mathrm{v}} + \nabla^{\mathrm{T}}\left(-\frac{\mathbf{K}_{\mathrm{sat}}}{\gamma_{\mathrm{w}}}\left(\nabla p - \gamma_w i_g\right)\right) = 0 \qquad (1)$$

where:

- M - Biot's modulus, assumed in the range of $M =$ (100-1000)K_{sk} (K_{sk} is unit modulus of skeleton). In general, it is a large number enforcing a volumetric incompressibility of a given fully saturated soil at very short times at the onset of consolidation. A default setting is $M = 10^6\ kPa$.
- α - Biot's parameter, typically assumed $\alpha = 1$
- p - pore pressure
- ∇p - pore pressure gradient
- $\mathbf{K}_{sat}$ - permeability matrix storing coefficients of permeability for a fully saturated soil, typical values for selected soils are stored in Table
- i_g - hydraulic gradient

The rate of change of total pressure is given by:

$$\dot{\sigma} = \mathbf{D}^{ep}\,\dot{\varepsilon} - \alpha 3\mathbf{m}\,\dot{p}_{ex} \qquad (2)$$

where:

- $\mathbf{D}^{ep}$ - current stiffness matrix
- p_{ex} - excess pore pressure
- $\mathbf{m} = \left\{\frac{1}{3}, \frac{1}{3}, 0, \frac{1}{3}\right\}^{\mathrm{T}}$ - for plane strain or axial symmetry

Note that the total pore pressure p is a sum of steady state pore pressure p_{ss} and excess pore pressure p_{ex}. It holds:

$$\dot{p}_{ss} = 0 \qquad (3)$$

The continuity equation (1) can be thus written as:

$$\frac{1}{\mathrm{M}}\dot{\mathrm{p}}_{ex} + \alpha\dot{\varepsilon}_{\mathrm{v}} + \nabla^{\mathrm{T}}\left(-\frac{\mathbf{K}_{\mathrm{sat}}}{\gamma_{\mathrm{w}}}\nabla p_{ex}\right) = 0 \qquad (4)$$

adopting the zero excess pore pressure boundary condition at the boundary with prescribed pressure as:

$$p_{ex}(t) = 0 \qquad (5)$$

and the zero flow ($q\,(t) = 0$) across the boundary with prescribed water flux density:

$$\mathbf{n}^{\mathrm{T}}\left(\frac{\mathbf{K}_{\mathrm{sat}}}{\gamma_{\mathrm{w}}}\nabla p_{ex}(t)\right) = 0 \qquad (6)$$

where: **n** - components of outward unit normal

See: **Setting hydraulic boundary conditions.**

The overall total stress is then provided by:

$$\sigma = \mathbf{D}^{el}\left(\varepsilon - \varepsilon^{pl}\right) - \alpha 3\mathbf{m}\left(p_{ss} + p_{ex}\right) \qquad (7)$$

where: $\mathbf{D}^{el}$ - elastic stiffness matrix

$\boldsymbol{\varepsilon}$ - vector of overall strains
$\boldsymbol{\varepsilon}^{pl}$ - vector of overall plastic strains

The current values of strains and excess pore pressure in equation (7) follow from the application of static equations of equilibrium and continuity equation (4) within the solution of coupled stress and water transport problem using the principal of virtual displacements. As in case of transient water flow analysis a fully implicit forward Euler method is adopted to perform time discretization of equation (4). Further details can be found in [1,2,3].

Consolidation analysis

As in case of transient water flow analysis the first calculation stage serves to set initial conditions, i.e. the distribution of geostatic stress and steady state pore pressure. The steady state pressure values also equal to the overall pressure values at the end of consolidation. The initial pore pressure values are set by specifying the groundwater table (GWT) only. It is worth to note that even if GWT table is found inside the analyzed soil body the soil below and above GWT is assumed fully saturated. This applies also to soils being introduced into the analysis in subsequent calculation stages (activating new regions). Removing or excavating soils (deactivating existing regions) is not possible with the current version of the program. The actual consolidation analysis is performed from the second stage on and requires setting the hydraulic boundary conditions, setting the time duration of a given calculation stage, setting the expected number of time steps and setting the way of introducing the load in to the analysis.

Setting hydraulic boundary conditions

The program allows for introducing two types of hydraulic boundary conditions, recall equations (5) and (6):

- Zero pore pressure condition ($p = 0$), which allows for free water outflow from the soil body, i.e. condition representing a permeable boundary. More specifically, this condition corresponds to the zero value of the excess pore pressure p_{ex}. The overall value of pore pressure along this boundary is thus equal to p_{ss}. This is a default boundary condition and is it assumed along all external domain boundaries, therefore also along the external boundaries of new regions.
- Zero flux density condition (no inflow/outflow, $q = 0$), i.e. condition representing an impermeable boundary. If needed, this condition has to be introduced manually.

The choice of a given boundary condition influences the rate of consolidation. Further details can be found in [1].

Setting the time step size - expected number of time steps in a given stage

Unlike the transient water flow analyses the initial time step size (discrete value of time increment when solving equation (4)) is in case of consolidation not directly assigned. Instead, this step is set on the bases of the specified duration of calculation stage and the input number of time steps expected for a given stage. In case of linear consolidation (all soils are assumed linearly elastic) the input number of time steps is performed. A time step reduction can take place in case of nonlinear consolidation if the lack of convergence for the current time step is encountered. This increases the number of steps to complete the stage analysis. When specifying the number of steps in relation to the stage duration one should keep in mind that at the onset of consolidation the time step should be relatively small (particularly when referring to a load stage in conjunction with nonlinear consolidation), while with gradual progress of consolidation the time step size may reach several tens of days. Further details can be found in [1].

Introducing load into the analysis

As in case of transient flow analysis the program offers two options only:

- The load is applied at one step at the beginning of the calculation stage. More specifically, a linear increase of load over the first time step is assumed. Thus if we are interested in the behavior at $t \rightarrow 0$, it is necessary to suitably choose the combination of number of time steps and duration of the first stage (e.g. 1 and 0.001) . In case of a very short time step and impermeable boundaries ($q = 0$) we simulate the response of volumetrically incompressible soil ($K \rightarrow \infty$) with a finite value of the shear modulus. The results for $t \rightarrow 0$ will then agree well with the results derived from standard stress analysis with undrained soils. Further details can be found in [1].
- The load linearly increases over the calculation stage. The load increment then depends on the current time step size. Especially in case of nonlinear consolidation one should properly specify the time span over which the load is introduced to avoid convergence difficulties.

Providing there is no load change within a given stage the above setting options are irrelevant.

Application of beam elements in consolidation

The beam permeability depends on its location and the choice of hydraulic boundary conditions. A beam found inside the soil body is in its normal direction impermeable. On domain boundaries the beam permeability in normal direction, as in case of water flow analyses, is driven by the selected boundary condition. In case of permeable boundary ($p = 0$) the beam on this boundary is fully permeable, while in case of impermeable boundary ($q = 0$) the beam on this boundary is also impermeable.

Application of contact elements in consolidation

The reason for introducing contact elements into the analysis is twofold. First, we wish to allow for a relative mutual shift between two soils, soil and rock or soil and beam element, e.g. in the analysis of interaction of soil and sheeting structure.

Second, the aim is at modeling potential drain along the beam or in general along a line to which the contact element is assigned. In every case one should realize a coupled simulation of both phenomena, i.e. stress and water flow analysis being carried out simultaneously. If not specified otherwise the program assumes the flow within a contact element being dependent on permeabilities of surrounding soils both in the longitudinal and transverse direction. In case of contact attached to the beam element the normal permeability k_n is irrelevant as the beam is assumed in this direction either fully impermeable ($k_n = 0$) or fully permeable ($k_n \rightarrow \infty$), see "**Application of beam elements in consolidation**".

General comments

Time evolution of individual variables, e.g. settlement or excess pore pressure, will be in case of linear consolidation always bounded by the solution of stress analysis when considering either undrained soils (all active soils in the analyzed domain are specified as undrained) or drained soils (standard setting, all active soils in the analyzed domain are specified as drained). The latter case coincides with the steady state analysis with total dissipation of excess pore pressure. The results of linear stress analysis with drained soils and linear consolidation derived at $t \rightarrow \infty$ must be identical. However, this does not hold for nonlinear analyses as in such cases we cannot rely on the principle of superposition. Further details can be found in [1].

Unlike water flow analyses, the solution of consolidation requires application of higher order elements (e.g. 6-node triangular or 8-node quadrilateral elements). While displacements are calculated at all nodes of a given element (quadratic approximation of displacement field), pore pressure is calculated at the corner nodes only (linear approximation of pore pressure).

Unlike one-dimensional consolidation implemented in program "**Settlement**" the two-dimensional consolidation yields at $t \rightarrow 0$ zero volumetric strain and thus also zero mean effective stress only. Individual components of the displacement field are generally nonzero.

Literature:

[1] M. Šejnoha, T. Janda, H. Pruška, M. Brouček, Metoda konečných prvků v geomechanice: Teoretické základy a inženýrské aplikace, předpokládaný rok vydání (2015)

[2] Z. Bittnar and J. Šejnoha, Numerické Metody Mechaniky II. České vysoké učení technické v Praze, 1992

[3] Z. Bittnar and J. Šejnoha, Numerical methods in structural engineering, ASCE Press, 1996

Ko Procedure

K_o procedure is the method that allows for the calculation of geostatic stress (*1 st* stage) when particular ration between vertical and **horizontal stress components** is needed. For example, when dealing with **overconsolidated soils** the actual horizontal stress can attain much higher values than found in normally consolidated soils.

When adopting **standard analysis** the initial stress is determined through the application of the finite element method. Nonlinear material models can be used to account for evolution of possible failure surfaces already in the *1st* calculation stage. In case of elastic response the ratio between vertical σ_z and horizontal σ_x stress components is provided by:

$$\sigma_x = \frac{\nu}{(1-\nu)}\sigma_z$$

where:

σ_z	-	vertical normal stress
σ_x	-	horizontal normal stress
ν	-	Poisson's ratio

This analysis may lead to evolution of plastic strains.

The K_o **procedure** generates only elastic response. The horizontal stress in the *1st* stage of construction follows from:

$$\sigma_x = K_0\,\sigma_z$$

where:

K_o	-	coefficient of horizontal stress at reast defined by the user
σ_z	-	vertical normal stress
σ_x	-	horizontal normal stress

The K_o coefficient is assumed to be a soil parameter. If the K_o parameter is not assigned, it is derived from the relation:

$$K_o = \frac{\nu}{1-\nu}$$

The resulting stresses may, however, violate the plasticity condition in the *2nd* stage of construction when nonlinear material models are used. Iteration of equilibrium is then carried out even if no changes occur in the *2nd* stage.

Water Flow

The program allows for performing either the **steady state or the transient flow analysis in a soil body**. The transient flow analysis allows for monitoring the evolution of pore pressure (pressure head) and the degree of saturation in time. Time after which the distribution of pores pressure no longer changes determines the time needed to reach the steady state conditions. This value depends both on the soil flow characteristics (coefficient of permeability, parameters of models describing the retention curve - dependence of the degree of saturation or water content on the negative pressure head or suction) and type of the analyzed problem (e.g. confined/unconfined flow). In case of steady state flow analysis, the individual stages of calculation are independent from each other. In case of transient flow analysis, the solution is performed similarly to standard stress analysis. Individual calculation stages then depend on each other. The first stage of construction stays independent and serves to set initial conditions, i.e. to assign initial pore pressures/pressure heads and degree of saturation at the onset of time dependent analysis in both a fully saturated (positive pore pressures) and a partially saturated (negative pore pressures - suction) soil. The subsequently defined stages require inputting the time duration of a given stage together with the load history (time history of hydraulic boundary conditions). The current version of the program allows us to either introduce the entire load at once at the beginning of the calculation stage or to assume that it linearly increases with time during the course of stage calculation.

In both cases (steady state/transient flow) the program describes in general the flow in an unsaturated or partially saturated medium. The flow in a fully saturated medium appears only below the groundwater table. Above the ground water table (flow in a partially saturated medium) the flow is driven by a suitable material model. To analyze problems of unconfined flow the program introduces three material models: the **Log-linear model**, the **Gardner model** and the **van Genuchten model**. When performing the transient flow analysis we recommend adopting the van Genuchten model, because this model is capable of credibly representing the retention data of soil. Since the choice of the material model influences the setting of initial conditions (initial value of degree of saturation) the program does not allow for changing material models in subsequent calculation stages. In the same spirit changing geometry in comparison to the initial stage is also not possible.

When performing the transient soil analysis it is first necessary to set in the 1st stage the initial values of the pore pressure/pressure head at time $t = 0$, particularly above the ground water table in the unsaturated or partially saturated soil (suction region). The program offers three options to introduce suction, either by performing the state analysis, or assuming an equilibrium distribution given by $p = -\gamma_w z$, where z is measured from the current location of the ground water table, or the initial values of suction can be specified directly by the user. When solving practical region we recommend not to specify values of the negative pressure head h_p smaller than -10m (p > -100 kPa), especially in case of coarse texture soils. For example, for sands the retention curve is almost flat for the values of $h_p < -1m$ and for large changes in pressure head there is almost no change in the degree of saturation. This holds also for the coefficient of relative permeability K_r, which serves to reduce the fully saturated permeability in the unsaturated or partially saturated zones. A general recommendation for setting minimal values of negative pressure head is, however, rather complicated since for fine texture soils these values may reach several hundred and for clays even thousand meters.

The next step calls for defining the boundary conditions, either at a point or along a boundary line at the beginning of a new calculation stage.

Beam and contact elements can be introduced inside the soil body. The **analysis results** are presented in the form of pore pressure and total head distributions, suction, velocities and directions of flow and information about the total inflow/outflow into or out of the soil body. In case of transient flow it is also possible to plot the distribution of the degrese of saturation inside the soil body.

Flow Analysis

Transient flow

Transient flow analysis in a partially saturated medium is driven by the solution of a general Richard's equation (equation of continuity):

$$n\,\dot{S} + div\left(-K_r \mathbf{K}_{sat} \nabla h\right) = 0$$

where:

- n - material porosity
- $\dot{S}$ - rate of change of degree of saturation
- K_r - coefficient of relative permeability
- $\mathbf{K}_{sat}$ - permeability matrix of fully saturated soil
- ∇h - gradient of total head

Time discretization of Richard's equation is based on a fully explicit Picard's iteration scheme [1]. This corresponds to a hybrid formulation ensuring conservation of mass. Owing to the solution of a generally nonlienear problem, the analysis is performed incrementally. Standard Newton-Raphson iteration scheme is used to satisfy equilibrium conditions.

Note that speed and stability of the iteration process is influenced to a large extent by the choice of the material model (the way of calculating the coefficient of relative permeability K_r, degree of saturation S and the approximation of capacity term $C = dS / dh_p$) in relation to the nonlinear properties of a given soil. A significantly nonlinear behavior is for example typical of sands where improperly prescribed initial conditions may cause numerical problems. Details can be found in [2,3].

Steady state flow

The steady state analysis assumes no change of the degree of saturation in time. The governing equation then reduces to:

$$\mathrm{div}\left(-K_r \mathbf{K}_{\mathrm{sat}} \nabla h\right) = 0$$

Unlike transient flow, the analysis is therefore time independent and the introduction of the flow boundary conditions only. However, it is still a generally nonlinear problem (e.g. unconfined flow analysis) calling for the application of the Newton-Raphson iteration method. Details can be found for example in [2,3].

Literature:

[1] M. A. Celia and E. T. Bouloutas, A general mass-conservative numerical solutionfor the unsaturated flow equation, Water Resources Research 26 (1990), no. 7, 1483-1496.

[2] M. Šejnoha, Finite element analysis in geotechnical design, to appear (2015).

[3] M. Šejnoha, T. Janda, H. Pruška, M. Brouček, Metoda konečných prvků v geomechanice: Teoretické základy a inženýrské aplikace, předpokládaný rok vydání (2015).

Earthquake

The program allows for addressing the impact of seismic actions on a geotechnical structure. The earthquake analysis can be executed in an arbitrary construction stage.

The earthquake loading is defined by an accelerogram which can be either generated by the program, imported or entered by points. The accelerogram is generated so that its **response spectrum** corresponds to the chosen elastic response spectrum defined in design codes.

Either fixed or absorbing boundary conditions can be specified **on the bottom boundary**. So-called **free-field boundary conditions** are defined **on the lateral boundaries** to correctly represent the deviation from the Free field column analysis.

The earthquake analysis **includes the solution of the eigenvalue problem** to provide the natural frequencies and corresponding mode shapes. The natural frequencies are needed in setting the parameters of **Rayleigh damping**. The results of the eigenvalue analysis include modal participation factors and modal effective mass.

To allow for running the **earthquake analysis**, check "**Allow earthquake analysis**" in the Topology regime, the frame "**Settings**".

With this option checked:

- The "Soil properties" dialogue window introduces new **parameters pertinent to dynamic analysis** including damping;
- The "**Earthquake**" frame then appears in each construction stage to allow for entering the seismic load (accelerogram) and specific boundary conditions.

The dynamic analysis outputs the mechanical quantities - displacements, stresses, strains, internal forces, etc. - in every time step.

The earthquake analysis has the following limitations:

- it is not possible to run it in the "Axial symmetry" analysis mode;
- the critical state constitutive models do not adopt "**the dynamic modulus**";
- the analysis is not allowed when "**rigid bodies**" are placed on the lateral boundaries of the numerical model.

Further details are available in the **theoretical manual** on our website.

Dynamic analysis of Earthquake

The earthquake problem is solved by the means of dynamic analysis of a continuous body. At each point x and each time instant t the following differential equation is satisfied:

$$\nabla\sigma\left(x,t\right) - c\dot{u}\left(x,t\right) - p\ddot{u}\left(x,t\right) = 0$$

where:

c	-	coefficient of viscous damping
ρ	-	mass density
u	-	displacement
$\dot{u}$	-	velocity
$\ddot{u}$	-	acceleration í
∇	-	gradient

σ - stress

The stresses are provided by:

$$\sigma_{ij} = D_{ijkl}\left(\varepsilon_{kl} - \varepsilon_{kl}^{pl}\right)$$

where: D_{ijkl} - material stiffness tensor
ε_{kl} - strain tensor
ε_{kl}^{pl} - plastic strain tensor

The strains are equal to the symmetric part of the displacement gradient:

$$\varepsilon_{ij} = \frac{1}{2}\left(u_{i,j} + u_{j,i}\right)$$

where: $u_{i,j}$ - derivative of the i-th component of the displacement in the direction of the j-axis.

Finite element discretization of the equations of motion gives the system of ordinary differential equations in the form:

$$\mathbf{M}\ddot{r}(t) + \mathbf{C}\dot{r}(t) + \mathbf{K}r(t) = F(t)$$

where: M - mass matrix
C - damping matrix
K - stiffness matrix
$F(t)$ - vector of time-dependent loading
$r(t)$ - vector of nodal displacements

As for time integration, the user may choose between the **Newmark method** and the Hilber-Hughes-Taylor **Alpha method**.

Further details are available in the **theoretical manual** on our website.

Literature:

Z. Bittnar, P. Řeřicha, Metoda konečných prvků v dynamice konstrukcí, SNTL, 1981.

T. Hughes, The Finite Element Method: Linear Static and Dynamic Finite Element Analysis, Prentice Hall, INC., Engelwood Clifts, New Jersey 07632, 1987.

Z. Bittanr, J. Šejnoha, Numerical methods in structural engineering, ASCE Press, 1996.

Eigenvalue analysis

An eigenvalue analysis providing eigenmodes and eigenfrequencies comes as a part of the earthquake analysis in GEO5. The motivation for the eigenvalue analysis is the need for low eigenfrequencies that are required when setting the **Rayleigh damping**.

The modal analysis is based on the discretized undamped equation of motion for free oscillations which come in form:

$$\left(\mathbf{M} - \omega_\alpha^2\mathbf{K}\right)\phi_\alpha = \mathbf{0}$$

where: M - mass matrix
K - stiffness matrix
ω - i -th eigenfrequency
ϕ - i -th eigenmode

When choosing a particular eigenfrequency for Rayleigh damping it is useful to assess whether the frequency corresponds more to horizontal or vertical oscillations. The modal participation factors in x and z directions.

Further details are available in the **theoretical manual** on our website.

Literature:

Z. Bittnar, P. Řeřicha, Metoda konečných prvků v dynamice konstrukcí, SNTL, 1981.

T. Hughes, The Finite Element Method: Linear Static and Dynamic Finite Element Analysis, Prentice Hall, INC., Engelwood Clifts, New Jersey 07632, 1987

Z. Bittnar, J. Šejnoha, Numerical methods in structural engineering, ASCE Press, 1996

Interface

The frame "**Interface**" serves to input interfaces between individual soils. Detailed description of how to deal with interfaces is described herein.

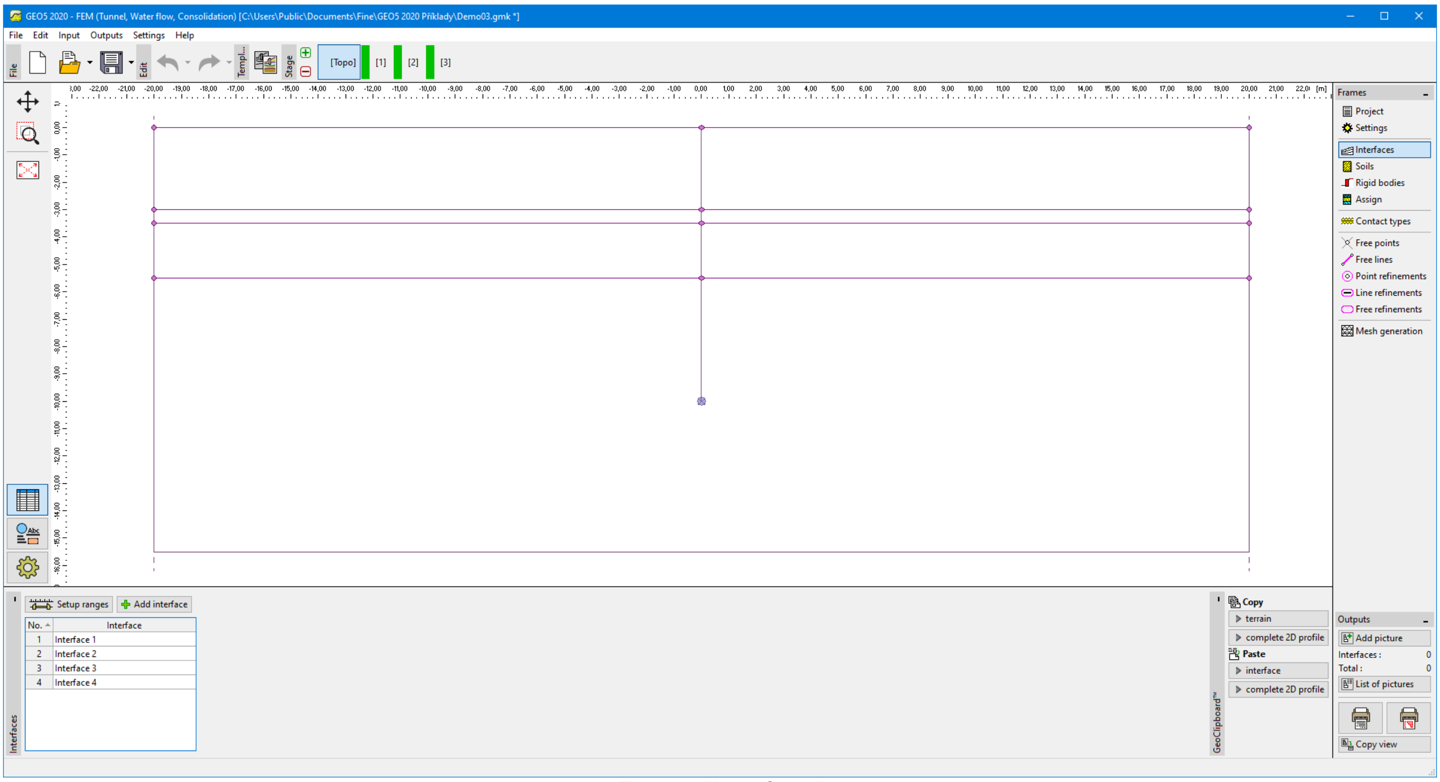
The width of the geometrical model can be usually estimated without much of a trouble (care must be take in the stability analysis to provide for sufficiently large space surrounding the critical region). The depth of a mesh however is quite important. The lowest point of a mesh can be imagined as incompressible subsoil. If there is no such layer of the soil or rock material in the geological profile it is possible to assume that at a certain depth from the ground the internal forces will vanish so that there will be no deformation. This will be the lowest point of the geometrical model.

If you are not certain about the margins of the geometrical model it is useful to proceed as follows:

- First enter larger margins, use coarser mesh and compute changes in the stress distributions within a soil body.
- In the next step modify the initial margins (regions with no apparent deformation or changes in stresses can be cut off), generate new and finer mesh and carry out a new and more accurate analysis.

Interfaces can also be imported from our other programs using clipboard.

The program makes it possible to import or export interfaces in the *.DXF format. Input interfaces can be copied within all 2D GEO5 programs using "GeoClipboard".



Frame "Interface"

Soils

The frame "**Soils**" contains a table with the list of input soils. Basic information regarding the current soil is displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

The soil input parameters depend on the selected material model, or material model in flow analysis.

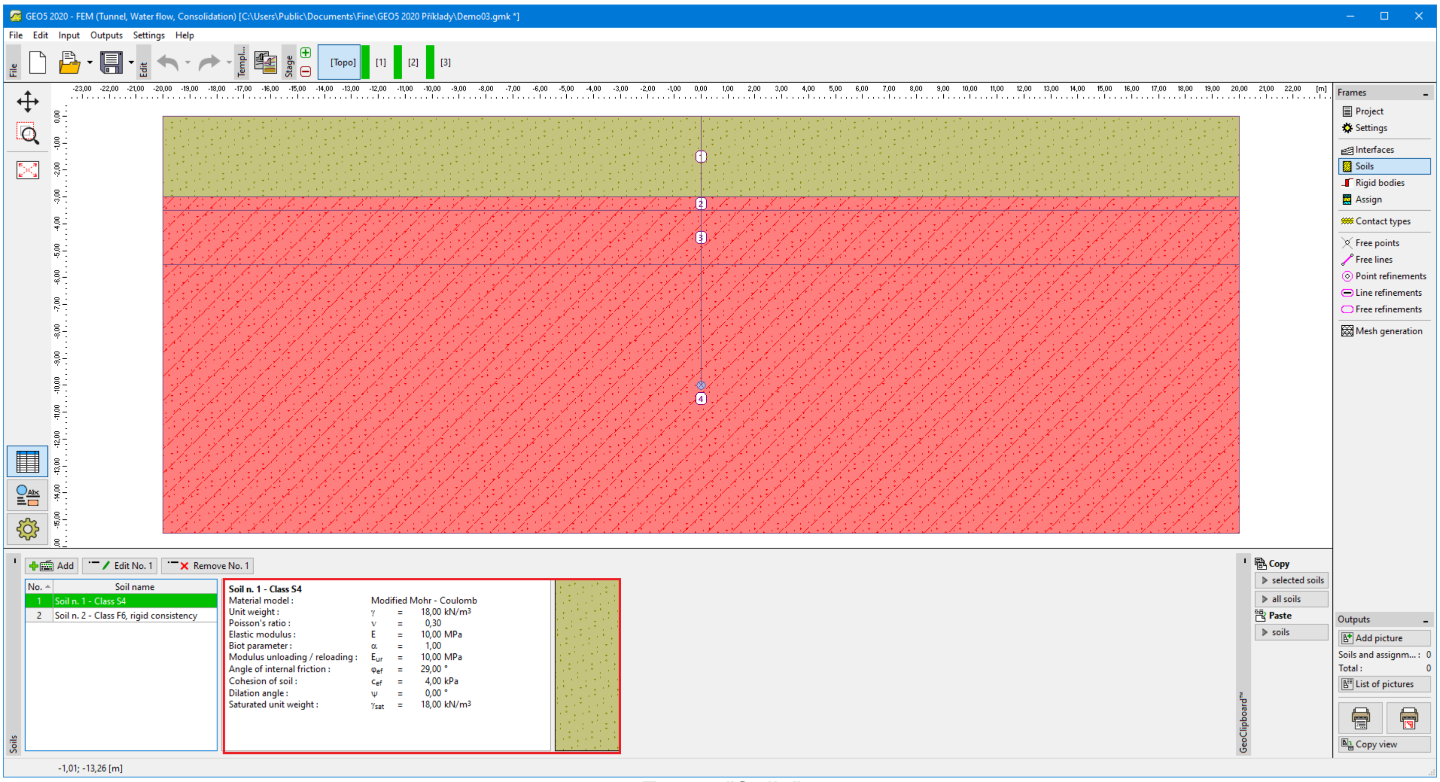
The basic material parameters are the Young modulus of elasticity E and the **Poisson's ratio** (they are needed in all models). Most nonlinear models require defining **the angle of internal friction** and **cohesion** of the soil. In both cases the analysis adopts **effective parameters** of the angle of internal friction φ_{ef} and cohesion c_{ef}.

The required list of material parameters to input also depends on the selected input mode. Assuming advanced program options (can be selected in the frame "Settings") allows us to define additional (advanced) material parameters (e.g. the Biot parameter, Effective unit modulus of water etc.). In most practical applications these material parameters are not particularly important and mostly serve to academic purposes.

Individual material models can be combined in the analysis - each soil can be assigned its own **material model**.

Adding a soil is performed in the "Add new soils" dialog window.

Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

Materials Models

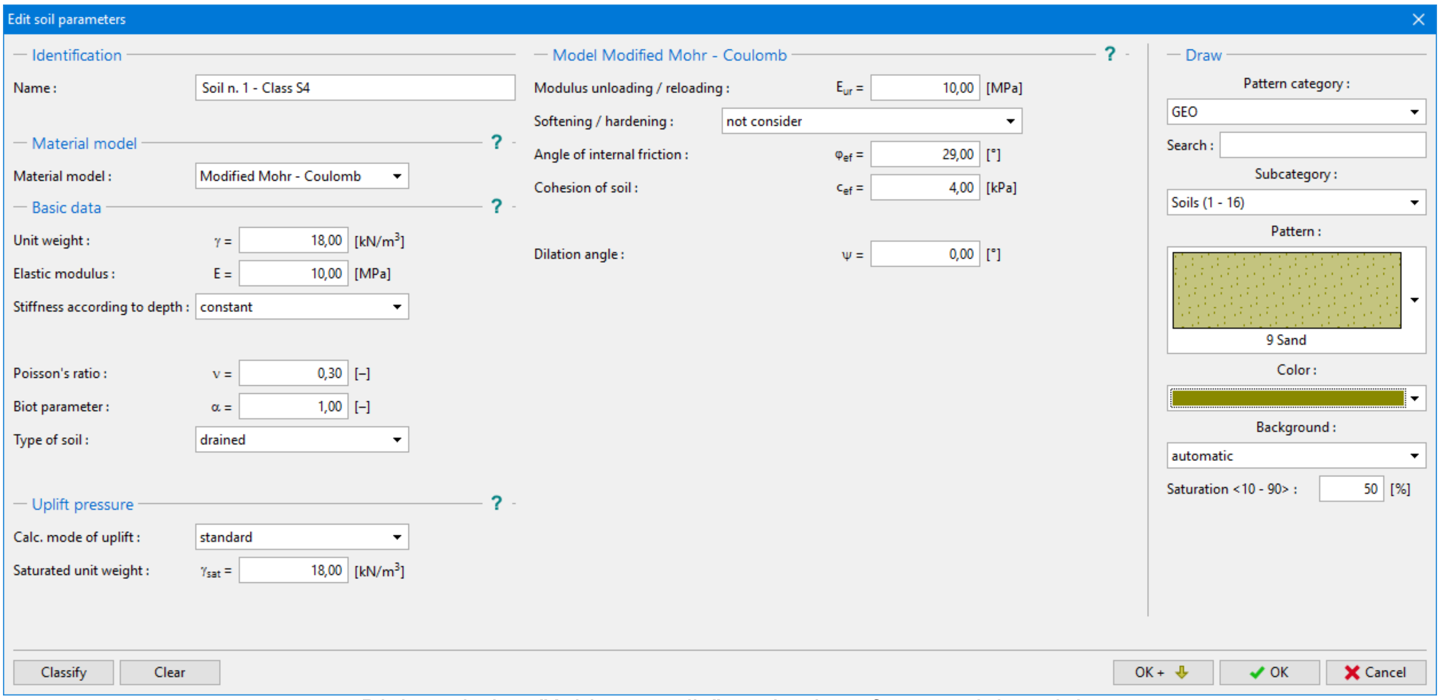
Selecting the most suitable material model together with inputting the required material parameters is one the most important but also one of the most difficult tasks when modeling structures using the finite element method.

The material model attempts to describe the soil (or rock) behavior as close to reality as possible. They can be divided into two basic groups:

- linear models
- nonlinear models

Selecting a proper material model is **essential** for the prediction of a real soil response.

Most tasks require nonlienar models (e.g., modeling of sheeting structures with linear models yields totally wrong results). In some cases, however, using linear models may prove useful and adequate and may considerably simplify the analysis.



Dialog window "Add new soils" - selection of a material model

Linear Models

Linear models give relatively fast, but not very accurate estimate of the true material response. These models can be used in cases, where only the stress or deformation states of soil mass are of interest. They provide no information about locations and possible mechanisms of failure.

They can be used to model soil behavior in regions, where only the local failure with no effect on the evolution of global failure occurs, but which may cause premature loss of convergence. Providing the main interest is in a reliable description of the soil behavior it is necessary to employ nonlinear models.

The linear models include:

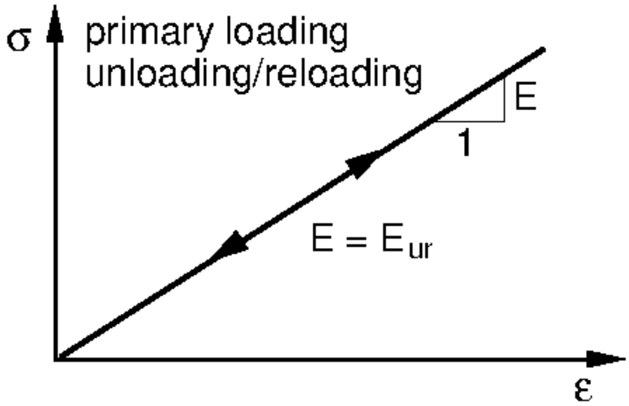
- Elastic model
- Modified elastic model

Elastic Model

The linear model is the basic material model that assumes a linear relationship between the stress and strain given by the Hooke law. The following data are required:

γ - unit weight of soil
ν - Poisson's ratio
E - modulus of elasticity

In a one dimensional problem the Hooke law describes the linear dependence of stress σ on strain ε via the Young modulus E (modulus of elasticity), see the Figure below. In this framework the linear model provides a linear variation of displacements as a function of applied loads.



Stress-strain relationship for LM

Modified Elastic Model

It is clear that for soils the linear behavior is acceptable only for relatively low magnitudes of applied loads. This becomes evident upon unloading that usually shows a rather small number of elastic deformation compare to the overall deformation. The modified linear model attempts at least to some extent to take this into account by considering different modulus for loading and unloading as plotted in the Figure.

A drop in the material stiffness along a given loading path attributed to the plastic yielding is reflected through a modulus of elasticity E, which can be imagined as a ***secant*** modulus associated with a certain stress level.

An elastic response is assumed upon unloading. To increase the clarity of model formulation the elastic modulus for the unloading branch is replaced by the unloading-reloading modulus E_{ur} that governs the response of a soil upon unloading and subsequent reloading up to the level of stress found in the material point prior to the unloading.

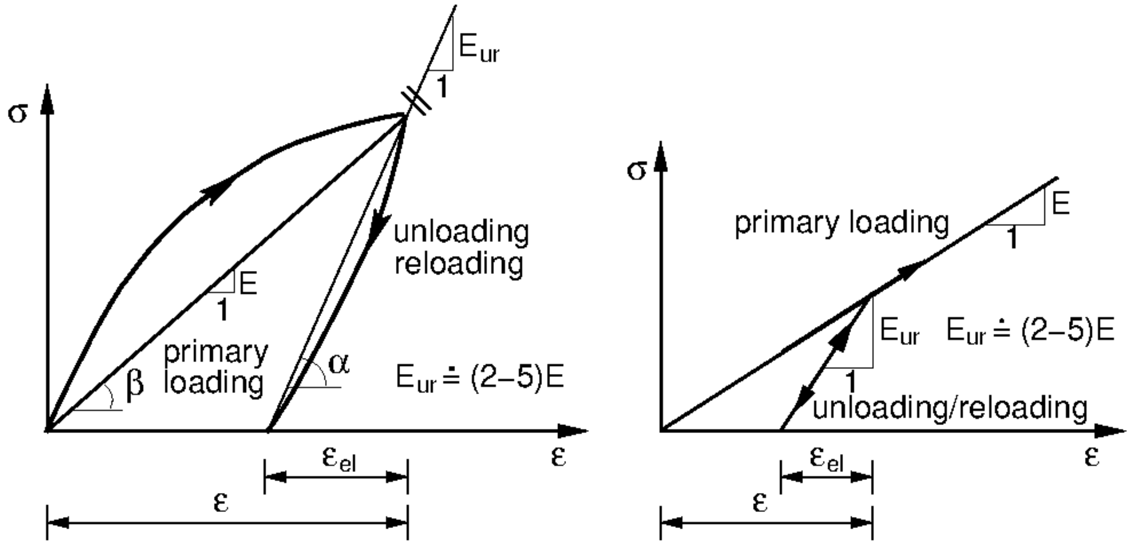
With reference to the following Figure these moduli are given by:

$$E = \tan\beta = \Delta\sigma/\Delta\varepsilon$$

$$E_{ur} = \tan\alpha = \Delta\sigma/\Delta\varepsilon^{el}$$

where: E - Modulus of elasticity (secant) [MPa]

E_{ur} - Unloading/reloading modulus [MPa]



(a) Real stress-strain diagram of soil, (b) Simplified stress-strain diagram for MLM

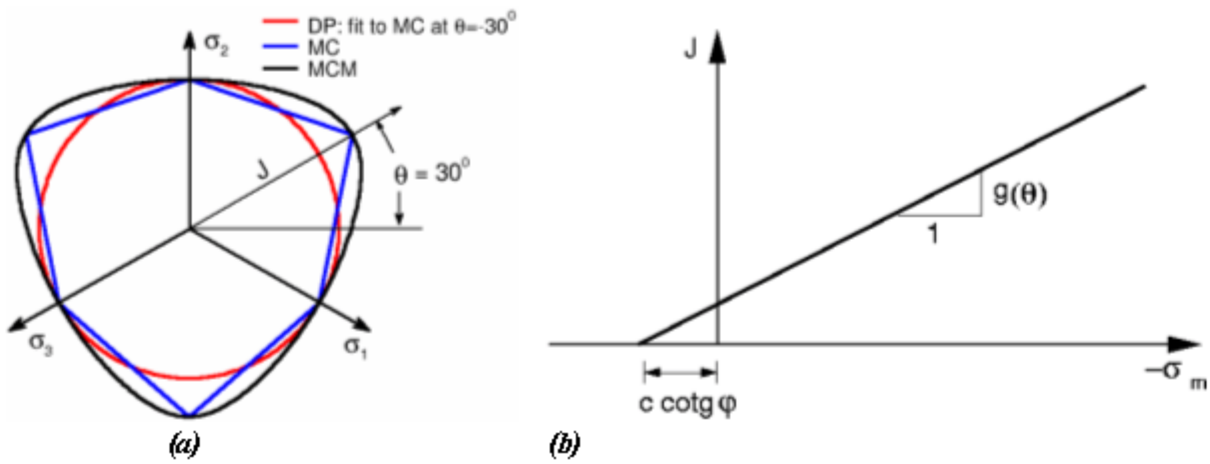
During the primary loading the response of a soil is therefore governed by the *modulus of elasticity* E while upon unloading it follows the path set by the *unloading-reloading modulus* E_{ur}. An approximate value of this modulus is ***(2-5)*modulus of elasticity*** E. In every case, both parameters should be obtained from reliable experimental measurements.

Nonlinear Models

The basic nonlinear models can be again divided into two groups.

The first class of models originates from the classical Mohr-Coulomb failure criterion. In particular, the Drucker-Prager, Mohr-Coulomb and Modified Mohr-Coulomb models fall in this category. These models can also model the hardening and softening. A common feature to these models is the evolution of unbounded elastic strains when loaded along the geostatic axis. This is evident from the figure below that shows projections of the yield surfaces into deviatoric and meridian planes, respectively. An example of the effect of the selected model is given here.

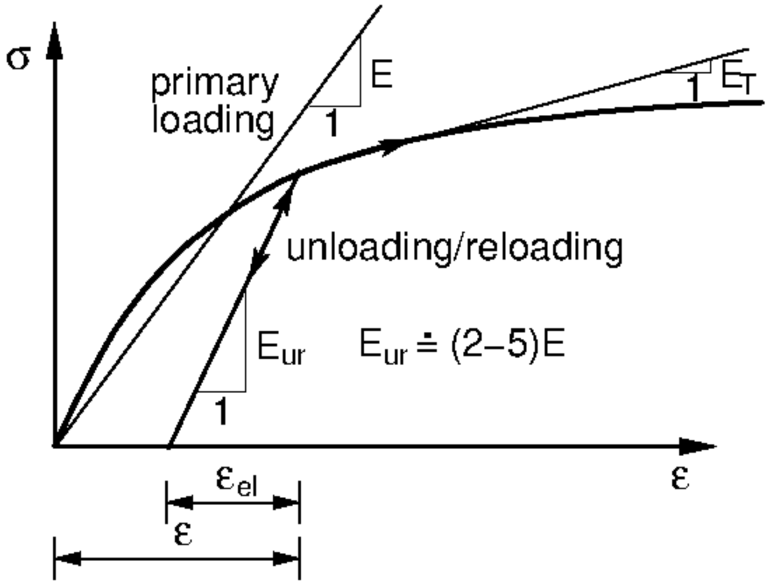
The second group of basic material models is represented by the Modified Cam-clay, Generalized Cam clay and Hypoplastic clay models employing the concept of the critical state of soils.



Projection of yield surfaces into (a) deviatoric, (b) meridian plane

Employing nonlinear models allows us to capture the typical nonlinear response of soils.

These models describe the evolution of permanent (plastic) deformation of a soil material. The onset of plastic deformation is controlled by so-called yield surface. The yield surface can be either constant (elastic-perfectly plastic material), or it can depend on the current state of stress (material with hardening/softening).



Stress-strain diagram for nonlinear models

Unlike the modified linear model the nonlinear models require specifying only the elastic modulus. A drop in the material stiffness is a result of evolution of plastic strains and corresponding redistribution of stresses. This consequently yields an instantaneous tangent material stiffness as a function of the current state of stress represented in the Figure below by an instantaneous tangent modulus E_T.

In addition to basic material parameters decribed in section "Elastic model" the nonlinear models call for the introduction of certain strength characteristics of the soil needed in the definition of a given yield surface. With reference to the first group of materials the following parameters must be specified.

φ - angle of internal friction [°]

c - cohesion of soil [kPa]

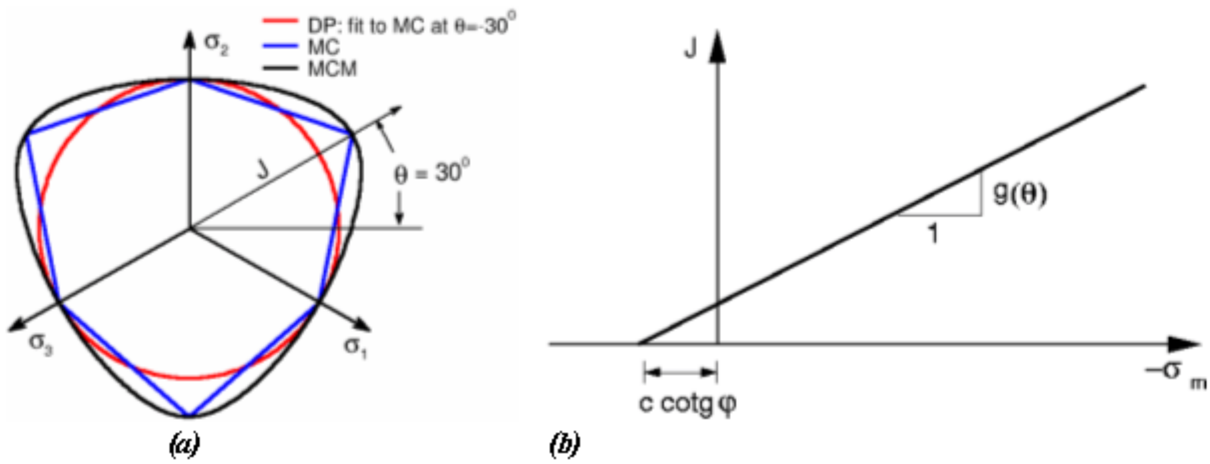
ψ - dilation angle [°]

The angle of internal friction and cohesion determine the **onset of plastic deformation**. The angle of dilation controls the evolution of plastic volumetric strain (dilation).

Mohr-Coulomb (MC)

The model requires inputting the following parameters: modulus of elasticity E, the **Poisson's ratio**, **angle of internal friction** and **cohesion**. The latter two parameters serve to define the yield condition. The formulation of constitutive equations assumes **effective parameters** of angle of internal friction φ_{eff} and cohesion c_{eff}. The angle of dilation must also be specified.

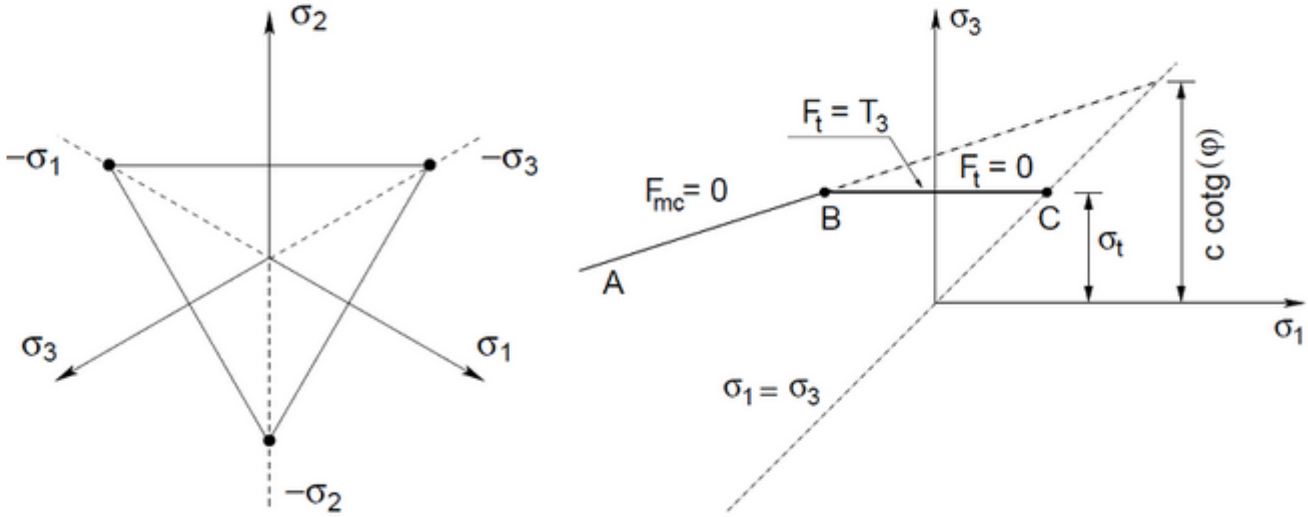
The Mohr-Coulomb yield surface can be defined in terms of three limit functions that plot as a non-uniform hexagonal cone in the principal stress space. Projections of this yield surface into deviatioric and meridian planes appear in the Figure. As evident from this Figure (part a) the MC yield function has corners, which may cause certain complications in the implementation of this model into the finite element method. The advantage on the other hand is the fact that the traditional soil mechanics and partially also the rock mechanics are based on this model.



Projection of yield surfaces into: (a) deviatoric, (b) meridian plane

Mohr-Coulomb Model with Tension Cut Off

The original formulation of the Mohr-Coulomb material model is extended by introducing the Rankine type of plasticity condition, see Figure (a), allowing for the reduction of tensile strength of soil, which in case of standard Mohr-Coulomb model is given by $c*cotg\varphi$, where c is the cohesion and φ the angle of internal friction. This value can be reduced by specifying the value of tensile strength σ_t as evident from Figure (b). If $\sigma_t > c*cotg\varphi$ the program automatically sets $\sigma_t = c*cotg\varphi$. This model can be used if the advanced program options are on.

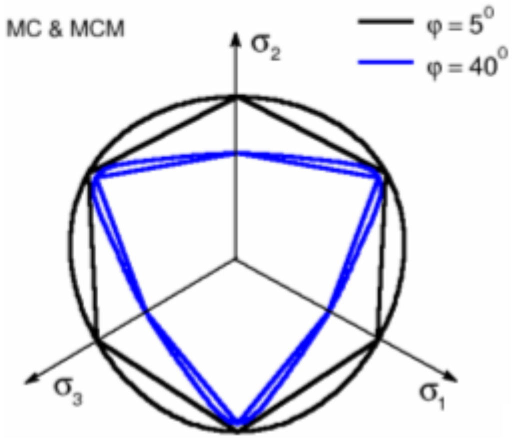


a) Projection of the Rankine yield condition in the deviatoric plane, b) Projection of the extended Mohr-Coulomb yield condition into the σ_1, σ_3 plane

Modified Mohr-Coulomb (MCM)

The model requires inputting the following parameters: modulus of elasticity E, the **Poisson's ratio**, **angle of internal friction** and **cohesion**. The latter two parameters serve to define the yield condition. The formulation of constitutive equations assumes **effective parameters** of angle of internal friction φ_{eff} and cohesion c_{eff}. The angle of dilation must also be specified.

Similarly to the DP model the Modified Mohr-Coulomb model smoothes out the corners of the MC yield surface. As suggested in the Figure the projection of the MCM yield surface into the deviatoric plane passes through all corners of the Mohr-Coulomb hexagon and as the MC yield function the MCM yield function depends on the effective mean stress σ_m and the Lode angle θ . With reference to its definition a slightly stiffer response of the material can be expected with the MCM plasticity model when compared to the MC and DP models.

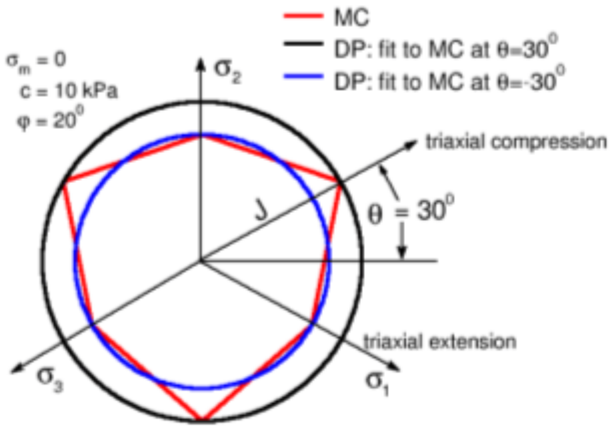


MCM and MC yield surfaces in the deviatoric plane

Drucker-Prager

The model requires inputting the following parameters: modulus of elasticity E , the **Poisson's ratio**, **angle of internal friction** and **cohesion**. The latter two parameters serve to define the yield condition. The formulation of constitutive equations assumes **effective parameters** of angle of internal friction φ_{eff} and cohesion c_{eff}. The angle of dilation must also be specified.

The Drucker-Prager model (sometimes also known as the extended von Mises model) modifies the Mohr-Coulomb yield function to avoid singularities associated with corners. Unlike the Mohr-Coulomb model the Drucker-Prager yield surface is smooth and plots as a cylindrical cone in the principal stress space. Similarly to the MC model the DP yield surface depends on the effective mean stress σ_m . The current version of the DP model implemented in FEM builds upon the assumption of triaxial extension. In other words, the yield surface projection into the deviatoric plane touches the inner corners of the Mohr-Coulomb hexagon ($\theta = -30^0$), where θ is the Lode angle.



DP and MC yield surfaces in the deviatoric plane

Softening and Hardening

Standard formulation of the Drucker-Prager and Modified Mohr-Coulomb models assumes elastic rigid-plastic behavior of the soil, when the strength parameters of soil c and φ remain constant during the analysis. The enhanced version of both models ("Advanced program options" on) allows for the evolution of these parameters as a function of the equivalent deviatoric plastic strain:

$$c = c\left(E_d^{pl}\right)$$

$$\varphi = \varphi\left(E_d^{pl}\right)$$

where: E_d^{pl} - equivalent deviatoric plastic strain is given by the following expressions:

$$E_d^{pl} = \sqrt{2 e_{ij}^{pl} e_{ij}^{pl}}$$

$$e_{ij}^{pl} = \varepsilon_{ij}^{pl} - \frac{1}{3} \varepsilon_v^{pl} \delta_{ij}$$

$$e_v^{pl} = \varepsilon_x^{pl} + \varepsilon_y^{pl} + \varepsilon_z^{pl}$$

where:

E_d^{pl} - equivalent deviatoric plastic strain

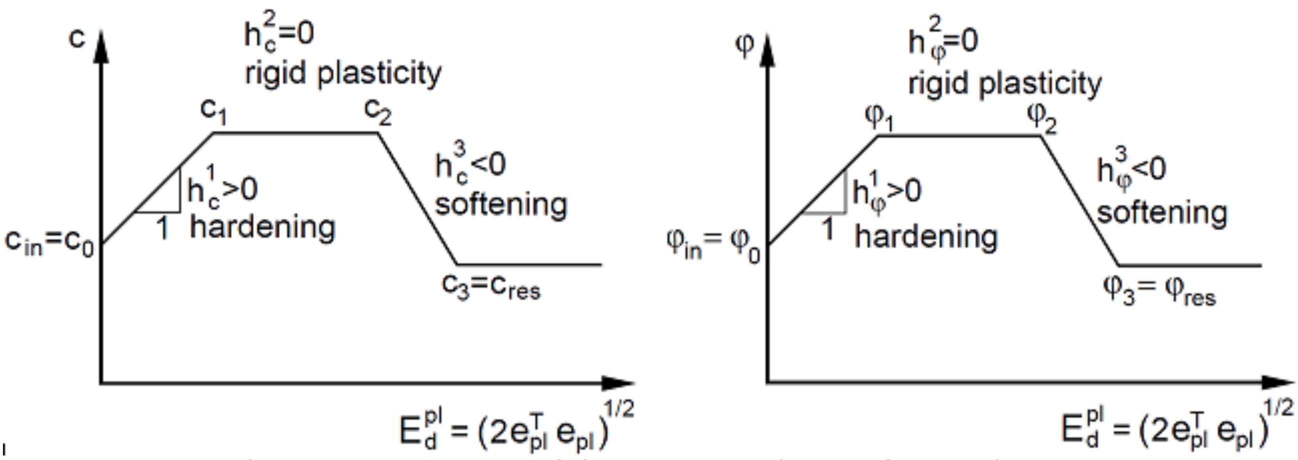
e_{ij}^{pl} - deviatoric plastic strain tensor

ε_{ij}^{pl} - plastic strain tensor

ε_v^{pl} - volumetric plastic strain

δ_{ij} - Kronecker's delta

The assumed, piecewise linear, variation of strength parameters is evident from figure.



Multi-linear hardening-softening law: dependence of a) cohesion and b) angle of internal frintion on equivalent deviatoric *plastic deformation E_d^{pl}*

Dilation angle ψ can be assumed either constant or it may evolve as a function of the angle of internal friction φ following the Rowes dilation theory:

$$\sin\psi = \frac{\sin\varphi - \sin\varphi_{cv}}{1 - \sin\varphi\sin\varphi_{cv}}$$

where φ_{cv} is the angle of internal friction at constant volume consistent with the critical state of soil (state at which the soil deforms at zero volumetric plastic strains). To prevent an infinite increase of the dilation angle (increase of tensile volumetric plastic strains) it must be bounded, e.g. in dependence on the maximum void ratio e_{max}, acceptable for a given material. The Rowes dilation theory requires the introduction of the following parameters:

φ_{cv} - angle of internal friction at constant volume [-]

e_0 - initial void ratio

e_{max} - maximum void ratio [-]

The current void ratio e can be expressed in terms of the current volumetric strain ε_v and the value of initial void ratio e_0 as:

$$\lim_{\Delta V \to 0}\frac{\Delta V}{V} = \varepsilon_v = \frac{e - e_0}{1 + e_0}$$

$$\varepsilon_v = \varepsilon_x + \varepsilon_y + \varepsilon_z$$

where: e - current void ratio

e_0 - initial void ratio

ε_v - overall volumetric strain

When the current void ratio e exceeds the maximum void ratio e_{max}, the dilation angle ψ is set to 0.

Angle of Dilation

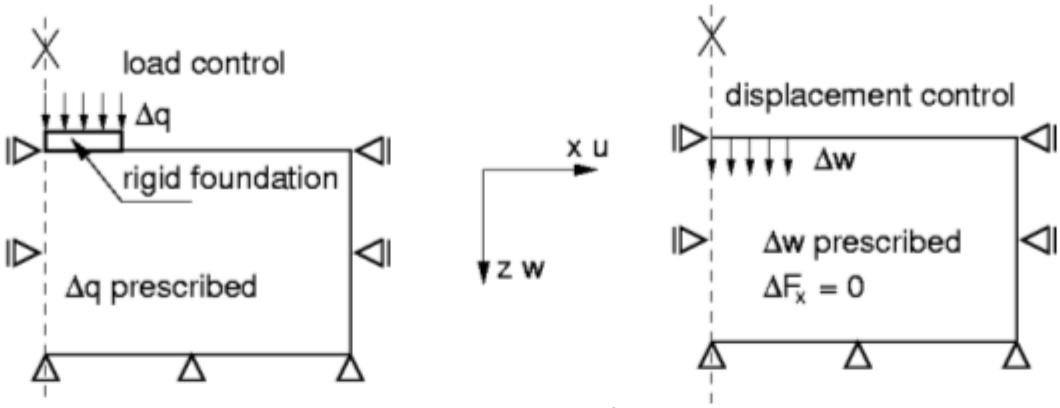
The **angle of dilation** controls an amountr of plastic volumetric strain developed during plastic shearing and is assumed constant during plastic yielding. The value of $\psi = 0$ corresponds to the volume preserving deformation while in shear.

Clays (regardless of overconsolidated layers) are characterized by a very low number of dilation ($\psi \approx 0$). As for sands, the angle of dilation depends on the angle of internal friction. For non-cohesive soils (sand, gravel) with the angle of internal friction $\varphi > 30°$ the value of dilation angle can be estimated as $\psi = \varphi - 30°$. A negative value of dilation angle is acceptable only for rather loose sands. In most cases, however, the assumption of $\psi = 0$ can be adopted.

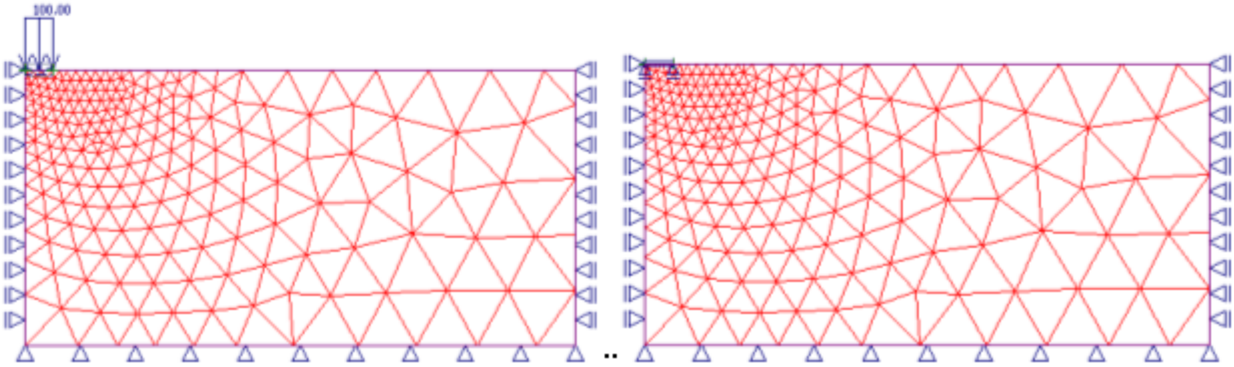
Influence of Material Model

To illustrate the effect of a particular model used to predict a structural response we present an example of a shallow foundation loaded by the distributed load q. A certain simplification of this task is the assumption of an infinitely stiff foundation loaded by the prescribed displacements.

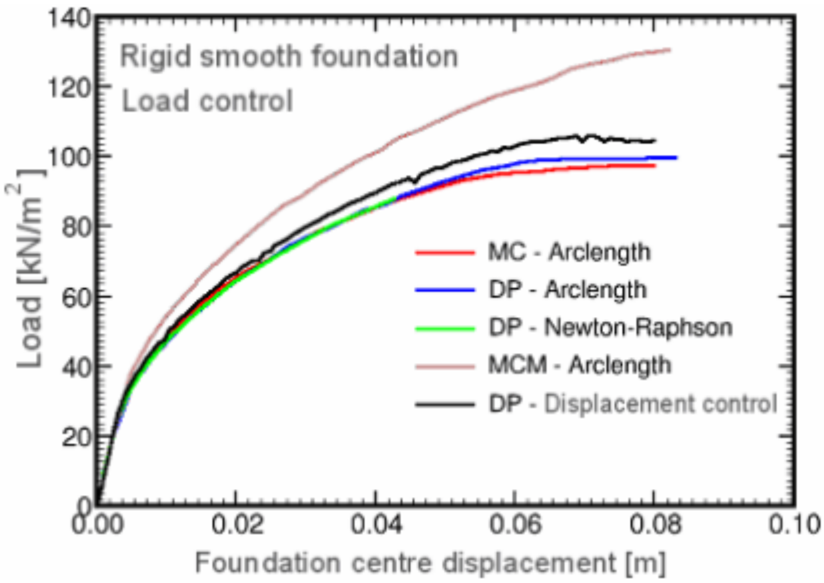
The geometrical model and finite element mesh for individual tasks appear in the Figure. The influence of soil and foundation self-weight on the resulting response is neglected. Owing to the symmetry of the model, only one half of the structure is analyzed.



Task assignment: strip foundation



Geometrical model and the finite element mesh



Analysis results

The results suggest a considerably stiffer response of the soil to the external load when using the MCM model in comparison to the DP and MC models, which in the present example show a similar behavior.

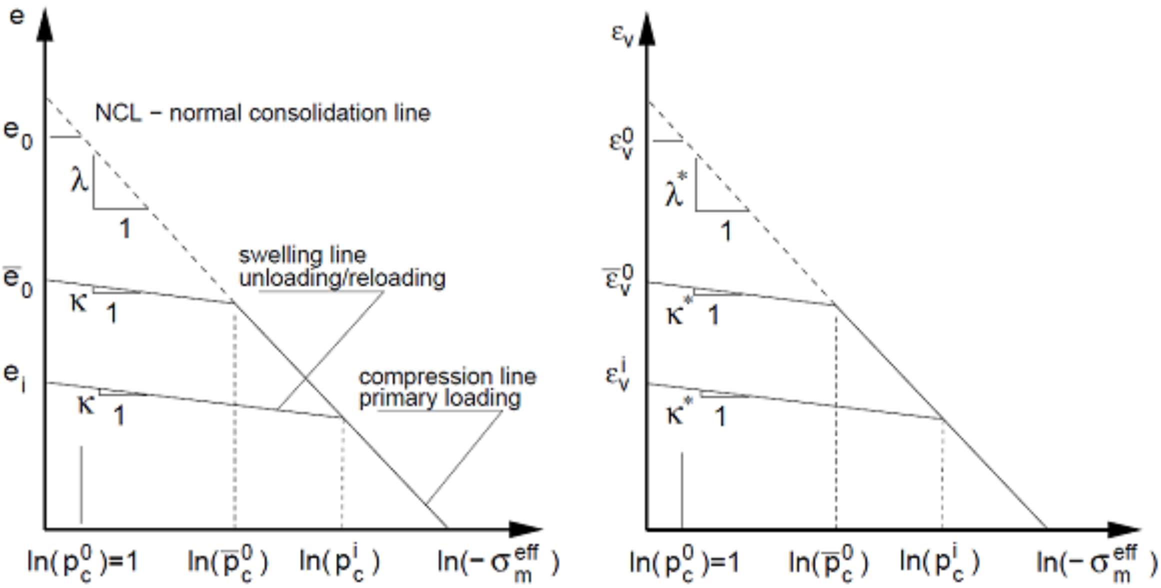
Modified Cam-Clay Model (MCC)

The MCC model was originally developed for triaxial load conditions. Experimental measurements on soft clays provided the background for the development of the constitutive model expressing the variation of void ratio e (volumetric strain ε_v) as a function of the logarithm of the effective mean stress $\sigma_m{}^{eff}$, as evident from the following Figure. Both graphs are related as follows:

$$\lambda^* = \frac{\lambda}{1+e}$$
$$\kappa^* = \frac{\kappa}{1+e}$$

κ - slope of swelling line [-]

λ - Slope of NCL (normal consolidation line) [-]
e - current void ratio [-]



Material response during isotropic consolidation (constitutive model)

The graph consists of a normal consolidation line (NCL) and a set of swelling lines. On the first load the virgin soil moves down the NCL. Next, suppose that the soil was consolidated to a certain level of stress, which is termed the preconsolidation pressure p_c, and subsequently unloaded up the current swelling line. Then, upon reloading the soil initially moves down the swelling line until reaching the stress state given by the parameter p_c, which existed prior to the unloading. At this point the soil begins to move again down the normal consolidation line (primary loading - compression line).

Parameters κ and λ can be estimated from the following expressions:

$$\lambda = \frac{C_c}{2{,}3}$$

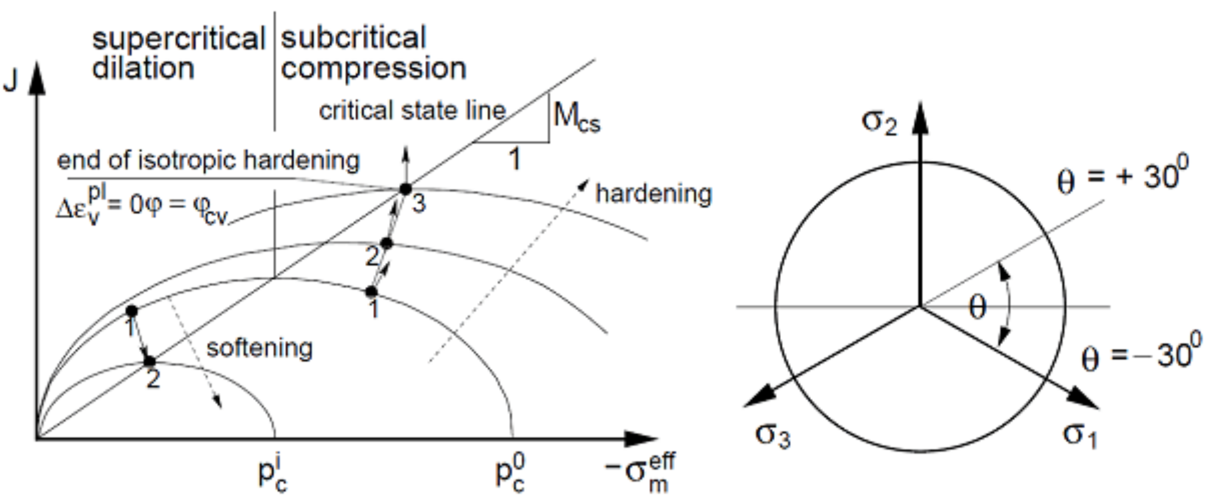
$$\kappa = 1{,}3\,\frac{1-\nu_c}{1+\nu}\,C_s$$

where: C_c - one-dimensional compression index

C_s - one-dimensional swelling index

These parameters follow from a simple oedometric test.

The yield surface is smooth without the possibility of evolution of tensile stresses. The MCC model allows, unlike the first group of models, a direct modeling of strain hardening or softening for normally consolidated or overconsolidated soils, a nonlinear dependence of the volumetric strain on the effective mean stress and limit conditions of ideal plasticity. When using the MCC model the soil loaded in shear can be plastically deformed without collapse (points 1,2 for hardening, point 2 for softening) until reaching the critical state (points 3 and 2 for hardening and softening, respectively). The soil deforms further in shear under the assumption of ideal plasticity without the change of e and σ_m^{eff}. Upon unloading, a linear response of soil is assumed.



Projection of yield function into meridial and deviatoric planes

Evolution of the yield surface (hardening/softening) is driven by the current preconsolidation pressure p_c :

$$p_c^{i+1} = p_c^i \exp\left[\frac{-\Delta\varepsilon_v^{pl}}{\lambda^* - \kappa^*}\right]$$

where:
- p_c^{i+1} - current preconsolidation pressure
- $\Delta\varepsilon_v^{pl}$ - increment of volumetric plastic strain

Apart from parameters κ and λ , the self-weight and the Poisson's ratio, the MCC model requires specifying the following three parameters:

M_{cs} - slope of the critical state line [-]

OCR - overconsolidation ratio [-]
e_0 - initial void ratio [-]

Reliable initialization of the model is described in section "Numerical implementation of MCC and GCC models".

The slope of the critical state line M_{cs} can be determined from the expression:

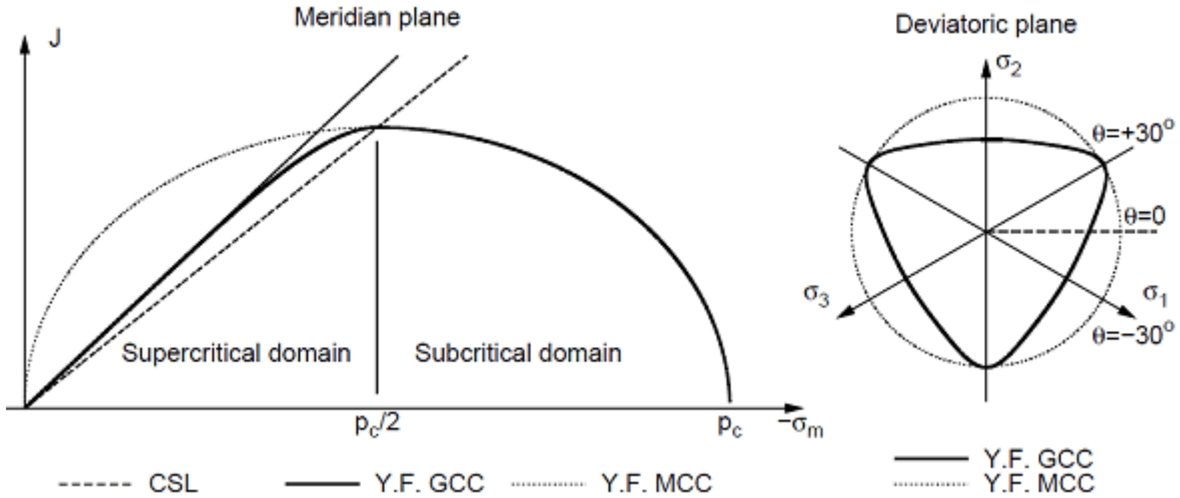
$$M_{cs}^{+30^\circ}(\varphi_{cv}) = \frac{2\sqrt{3}\sin\varphi_{cv}}{3 - \sin\varphi_{cv}}$$, for triaxial compression

$$M_{cs}^{-30^\circ}(\varphi_{cv}) = \frac{2\sqrt{3}.\sin\varphi_{cv}}{3 + \sin\varphi_{cv}}$$, for triaxial extension

where φ_{cv} is the angle of internal friction for constant volume corresponding to the critical state.

Generalized Cam-Clay Model (GCC)

This model represents a considerable improvement of the Modified Cam clay (MCC) model, particularly when modeling soils in the supercritical domain, see Figure, where the failure surface follows the classical models of Mohr-Coulomb, Drucker-Prager and Modified Mohr-Coulomb models. Unlike the Modified Cam clay model (dashed line) the GCC model plots, similarly to the MMC model, as a rounded triangle in the deviatoric plane. The MCC model plots, similarly to the Drucker-Prager model, as a circle. In the subcritical domain both models behave identically. Upon unloading, a linear response of soil is assumed. GCC model is available only when advanced program options are on. Reliable initialization of the model is described in section "Numerical implementation of MCC and GCC models".



Projection of yield surface of MCC and GCC models into meridial and deviatoric planes

The material parameters required for the Generalized Cam clay model are identical to the material data of MCC and MMC models:

κ - slope of swelling line

λ - slope of normal consolidation line (NCL)
e_0 - initial void ratio
OCR - overconsoliodation ratio
c - cohesion
φ angle of internal friction

φ_{cv} angle of internal friction at constant volume [-]

ν Poisson's ratio

Parameters κ and λ can be estimated from the following expressions:

$$\lambda = \frac{C_c}{2{,}3}$$

$$\kappa = 1{,}3\frac{1-\nu}{1+\nu}C_s$$

where: C_c - one-dimensional compression index

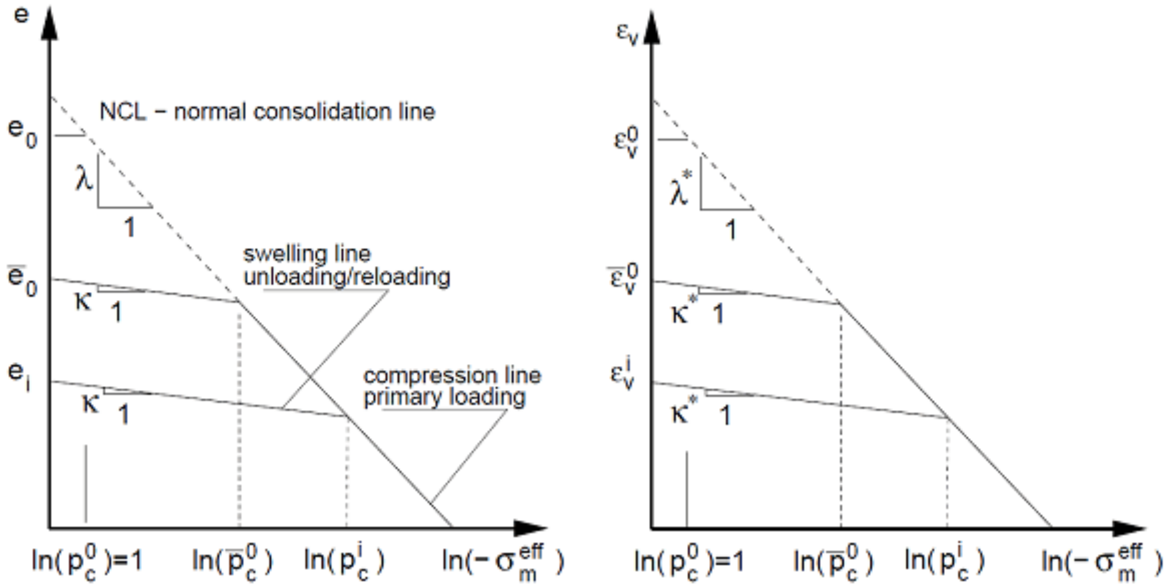
C_s - one-dimensional swelling index

These parameters follow from a simple oedometric test.

Similarly to the MCC model the formulation of the GCC model is based on the relation between the void ratio (volumetric strain) and the mean effective stress as shown in the following Figure. Both graphs are linked as follows:

$$\lambda^* = \frac{\lambda}{1+e}$$

$$\kappa^* = \frac{\kappa}{1+e}$$



Response of material during isotropic compression (constitutive law)

Evolution of the yield surface (hardening/softening) is driven by the current preconsolidation pressure p_c.

$$p_c^{i+1} = p_c^i \exp\left[\frac{-\Delta\varepsilon_v^{pl}}{\lambda^* - \kappa^*}\right]$$

where: p_c^{i+1} - current preconsolidation pressure

$\Delta\varepsilon_v^{pl}$ - increment of volumetric plastic strain

Numerical Implementation of MCC and GCC Models

An important step, ensuring a reliable application of MCC and GCC models, is the determination of the initial preconsolodation pressure p_c^{in} and the corresponding unit modulus K^{in}. These two parameters, however, are not directly specified by the user. Instead, they are derived by the program based on the assumed distribution of initial geostatic stress. Recall three basic options to derive the intial geostatic stress:

1. Using K_0 procedure

The use of K_0 procedure yields the following value of the initial mean stress:

$$\sigma_m = \frac{1}{3}\gamma\, h(1+2K_0)$$

where: K_o - coefficient of earth pressure at rest

γ - unit weight of soil

h - current depth below terrain

Assuming normal consolidation, the value of p_c^{in} is determined such that the stress derived using the K_o procedure fullfils the yield condition:

$$p_c^{in} = -\frac{J^2}{M_{cs}^2 \sigma_m} - \sigma_m$$

where: M_{cs} - slope of critical state line

J - equivalent deviatoric stress

σ_m - mean stress

Values of J and σ_m are defined by the following expressions:

$$S_{ij} = \sigma_{ij} - \sigma_m \delta_{ij} \qquad \sigma_{ij} = D_{ijkl}\ \varepsilon_{ij}$$

$$e_{ij} = \varepsilon_{ij} - \frac{1}{3}\varepsilon_v \delta_{ij} \qquad S_{ij} = 2Ge_{ij}$$

$$J = GE_d \qquad \sigma_m = K\varepsilon_v$$

$$\sigma_m = \frac{1}{3}\left(\sigma_x + \sigma_y + \sigma_z\right)$$

$$J = \sqrt{\frac{1}{2} S_{ij} S_{ij}}$$

$$e_v = \varepsilon_x + e_y + e_z$$

$$E_d = \sqrt{e_{ij} e_{ij}}$$

$$K = \frac{E}{3(1-2\nu)} = \frac{GE}{3(3G-E)} = \frac{2(1+\nu)}{3(1-2\nu)}G$$

where: E_d - equivalent deviatoric stress

e_{ij} - deviatoric strain tensor

ε_{ij} - overall strain tensor

ε_v - volumetric strain

σ_{ij} - stress tensor

s_{ij} - deviatoric stress tensor

δ_{ij} - Kronecker's delta

D_{ijkl} - elastic stiffness tensor

G - elastic shear modulus

K - elastic unit modulus

E - Young's modulus

ν - Poisson's ratio

In case of triaxial compression or extension it is possible to determine the slope of critical state line M_{cs} from the following expressions:

$$M_{cs}^{+30^\circ}(\varphi_{cv}) = \frac{2\sqrt{3}\sin\varphi_{cv}}{3-\sin\varphi_{cv}}$$

$$M_{cs}^{-30^\circ}(\varphi_{cv}) = \frac{2\sqrt{3}.\sin\varphi_{cv}}{3+\sin\varphi_{cv}}$$

In case of overconsolidated soils the initial value of p_c^{in} is modified as:

$$p_c^{in} = p_c^{in} OCR$$

The initial value of the unit modulus follows from:

$$K^{in} = -\frac{1+e}{\kappa}\sigma_m$$

where the current void ratio e is written as:

$$e = e_0 - \lambda \ln\left(p_c^{in}\right) + \kappa \ln\left(-\frac{p_c^{in}}{\sigma_m}\right)$$

For small stresses $\left|\sigma_m^{in}\right| < 1$ we get:

$$p_c^{in} = 1$$

$$K_{in} = -\frac{1+e^0}{\kappa}$$

2. Standard (elastic) analysis

Recall that the program allows for replacing the material model between stages of construction. Providing the K_o procedure cannot be used it is possible to carry out the analysis assuming elastic response of the clayey soil. The resulting stresses are used to derive the initial values of $p_c{}^{in}$ and K^{in} employing the previously defined expressions. In the next stages of construction the original elastic material model is replaced by the required MCC or GCC models.

3. Standard (plastic) analysis

This option allows the soil to be consolidated under the assumption of nonlinear behavior when generating the geostatic stress. This results in the evolution of plastic strains already in the first stage of construction. As in the K_o procedure we consider a normally consolidated soil which, during the course of deformation, moves down the normal consolidation line with the initial values of $p_c{}^{in}$ and K^{in} given by:

$$p_c^{in} = 1$$

$$K_{in} = -\frac{1+e^0}{\kappa}$$

Before the next analysis step the resulting plastic strains are set equal to zero. In some cases such an analysis may fail to converge.

Hypoplastic Clay

Hypoplastic clay is applicable for the modeling of soft fine grain soils. Similarly to all other models it belongs to the family of standard phenomenological models. As for the description of the soil response it falls into the group of critical state models (Cam clay, Generalized cam clay). This model, however, accounts for the nonlinear response of soils both in load and unloading. In comparison to other models based on the theory of plasticity, it allows for the calculation of total strains only. It thus makes no difference between elastic and plastic strains. Indication of type and location of a potential failure, in other models provided by the plot of equivalent deviatoric plastic strain, can be in case of hypoplastic clay represented by the distribution of the mobilized angle of internal friction.

When describing the soil response, the model allows for refecting a different stiffness in loading and unloding, softening or hardening in dependence on the soil compaction and the change of volume in shearing (dilation, compression). The current stiffness depends on only the load direction, but also on the current state of soil given by its porosity. Unlike Cam clay models, it strictly exlcudes tensile stresses in soil, see Figure 1a.

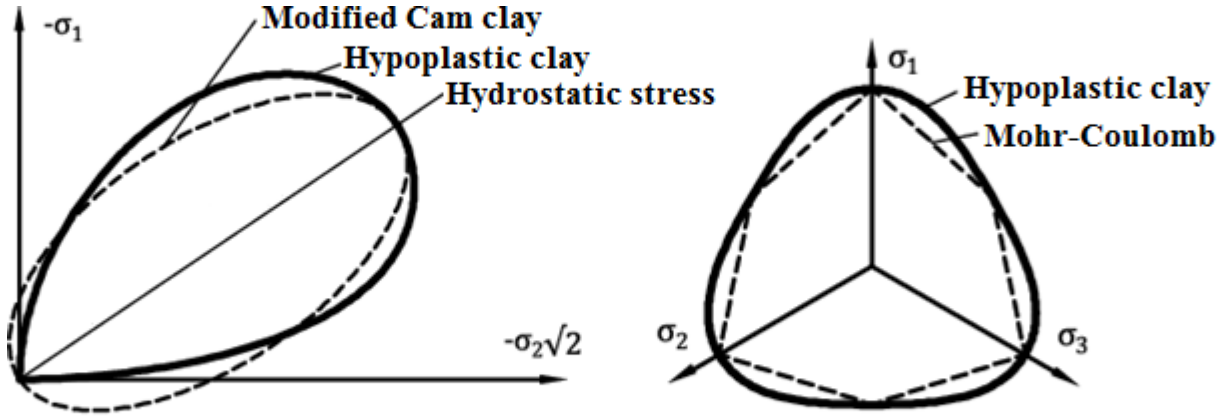


Figure 1: State boundary of hypoplastic model - (a) comparison with the yield surface of Cam clay model in the meridial plane, (b) comparison with the yield surface of Mohr-Coulomb model in the deviatoric plane

In case of hypoplastic model the standard yield surface is replaced by so called Boundary state surface. Its projection into the deviatoric plane is similar to the model, see Figure 1b. The flow rule is nonassociated resulting into a nonsymetric stiffness matrix (compare e.g. with the Mohr-Coulomb model when having different values for the angle of internal friction φ and the dilation angle ψ). Details regarding the model formulation can be found in [1].

Model parameters

The basic variant of the model requires inputting five material parameters:

- Angle of internal friction for constant volume (critical angle of internal friction) φ_{cv}
- Slope of swelling line κ^*
- Slope of normal consolidation line (NCL - normal consolidation line) λ^*
- Origin of the normal consolidation line N
- Ratio of unit and shear modulus r

Parameters κ^*, λ^* and N determine a bilinear diagram of isotropic consolidation in a log-log scale, Figure 2a. Providing the parameters of the bilinear Cam clay model (in semi-logaritmic scale, Figure 2b) are available, it is possible to input these values and the parameters of the hypoplastic model are back calculated. Paremeters of the bilinear Cam clay model are:

- Slope of swelling line κ (in semi-logaritmic scale)
- Slope of normal consolidation line λ (in semi-logaritmic scale)
- Void ratio e_{max} for normal isotropic consolidation by pressure of $1kPa$

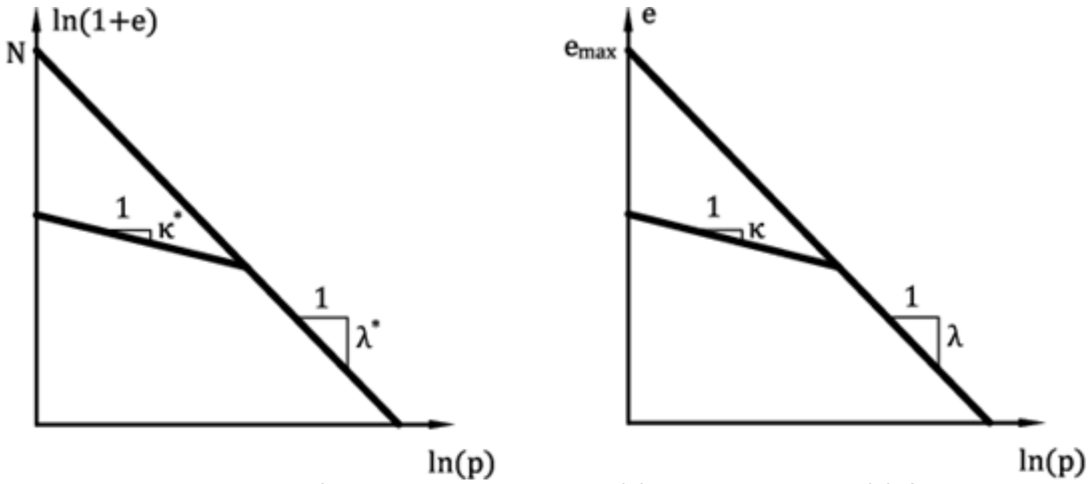


Figure 2:Bilinear diagram of isotropic consolidation - (a) Hypoplastic clay, (b) Cam clay model

Critical angle of internal frition φ_{cv}

- Identical for both original (undisturbed) and reconstituted subsequently consolidated sample
- Can be determined from standard triaxial test applying different cell pressures on a reconstituted sample
- Both drained and undrained (faster) test can be performed
- Most common values are in range of 18° - 35°

Slope of normal consolidation line λ^*

- It is determined graphically from the loading branche of oedometric or isotropic consolidation test, see Figure 3
- For stiff clays it is preferable to run the test on a reconstituted sample
- Most common values are in range of 0.04 - 0.15

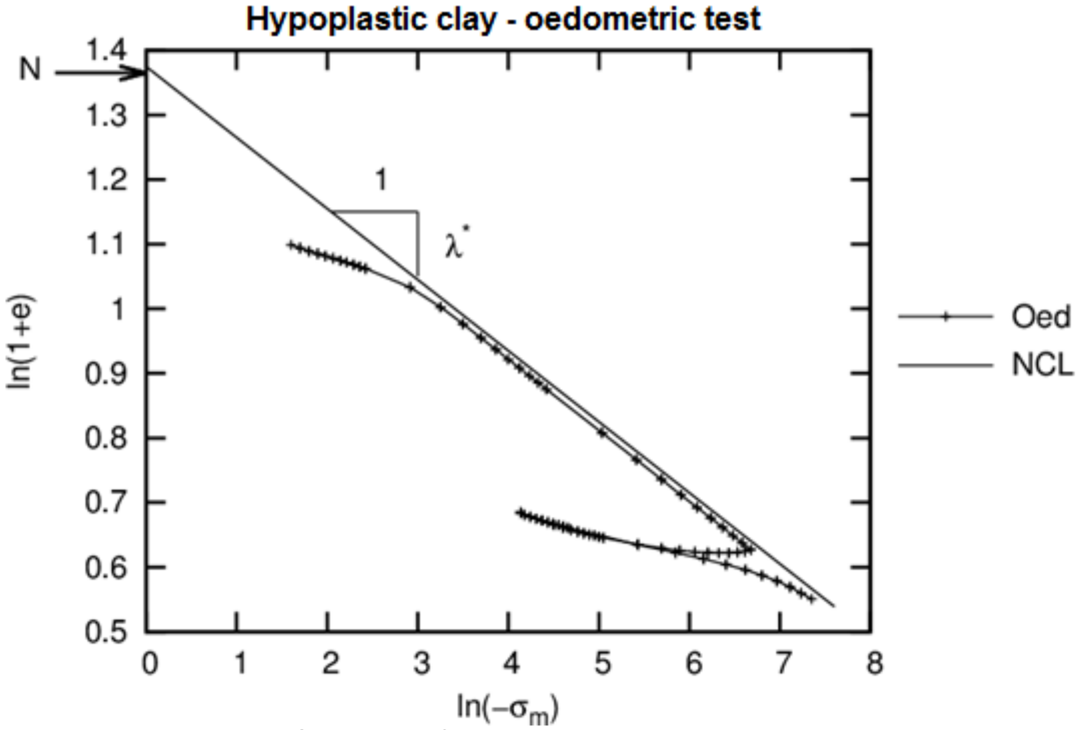


Figure. 3: Simulation of oedometric test with hypoplastic model

Slope of swelling line κ^*

- It can be determined similarly as parameter λ^* graphically or by performing a parametric study - comparing measurements and simulation along the unloading branche of oedometric or isotropic consolidation test, see Figure 3
- Most common values κ are in range of 0.01 - 0.02
- Ratio λ/κ should be large than 4.0

Origin of normal consolidation line N

- It is determined graphically from the loading branche of oedometric or isotropic consolidation test
- The test should be performed on an undisturbed sample - when surching for the intersection lambda line with the vertical axis it is possible to determine the slope lambda obtained from a reconstituted sample, see Figure 3
- Most common values are in range 0.8 - 1.6

Ratio of unit and shear modulus r

- The physical meaning of this parameter is given by the expressio $r = K_i/G_i$
- K_i corresponds to the tangent unit modulus from isotropic compression according to the normal consolidation line
- G_i corresponds to the tangent shear modulus for undrained shear test assuming the same stress state
- Parameter r can be determined by a parametric study of shear triaxial test
- Most common values are in range 0.05 - 0.7

Setting initial state of soil

In hypoplastic clay the current state of soil is associated with the current compaction represented by the void ratio. Model implementation allows for inputting the initial or current void ratio either directly or it can be back calculated using the input preconsolidation pressure OCR . In the first case, the input value e_0 corresponds to the void ratio measured on an unloaded sample extracted from a given depth. In the second case, the inputed value of e_{curr} corresponds to the void ratio of a stressed soil. In the last case, the value of OCR is specified. This parameter represents the ratio between the mean stress on NCL and the initial mean stress, see Figure 4b.

When intializing the task using the K_o procedure, the initial stress state at the beginning of the second stage is assigned the current stress state. If adopting standard analysis in the first stage (the hypoplastic clay model is introduced already in the first calculation stage) where the soil is loaded by its self-weight, the value of initial stress $p_{in} = 1\ kPa$ is assumed and it holds $e_{curr} = e_0$. Providing a different material (e.g., elastic material is considered in the first calculation stage) is replaced by the hypoplastic clay model, the initial state of stress derived in the previous stage is adopted. Recall that when using elastic material in the first calculation stage the resulting stress state corresponds to the results provided by K_o procedure for K_o (ν is the Poisson's ratio).

$$K_o = -\frac{\nu}{1-\nu}$$

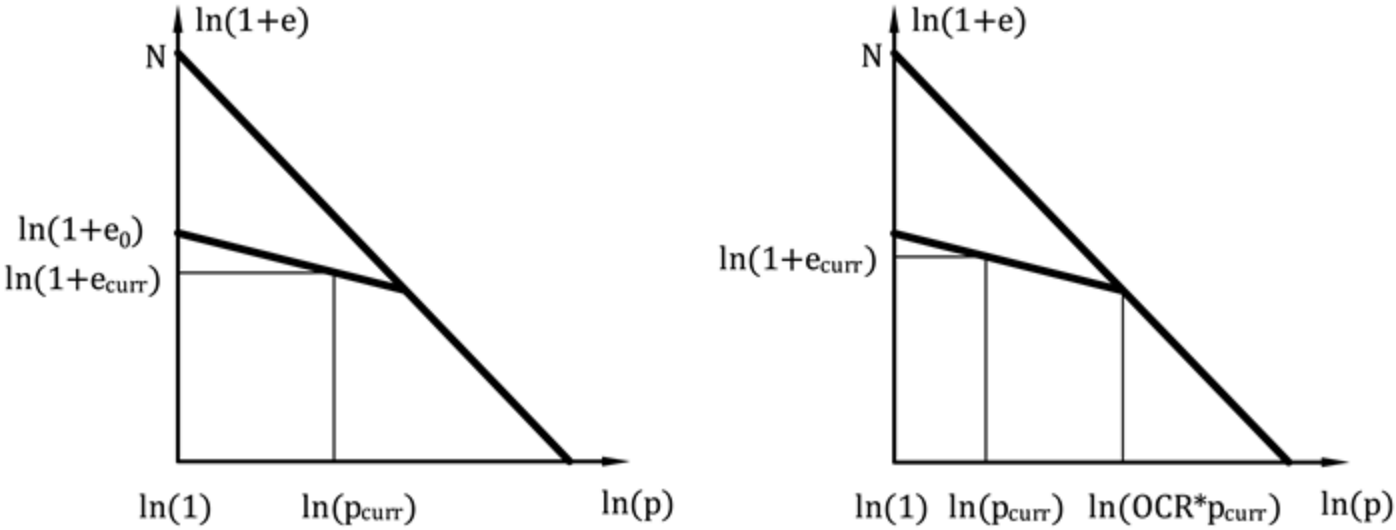


Figure 4: Iniciation of void ratio - (a) with the help of initial void ratio, (b) initiation by OCR

It is clear from Figure 5 that for normally consolidated soils the state for which $OCR = 1.0$ corresponds to an isotropic consolidation only, thus for $K_o = 1.0$. If the soil experiences a non-zero deviatoric stress state the corresponding OCR for a normally consolidated soil is greater than 1.0. An exact value of depends on both the soil parameters and stress path (the value of K_o). Figure 5 shows the dependence of the minimum for various values of K_o and different types of claye soils. Particular values are also stored in Table 1. The basic material parameters of this set of soils are listed in Table 2.

The choice of $OCR = 1.0$ for normally consolidated soils with K_o not equal to 1.0 creates a non-acceptable stress state which may result in the loss of convergence.

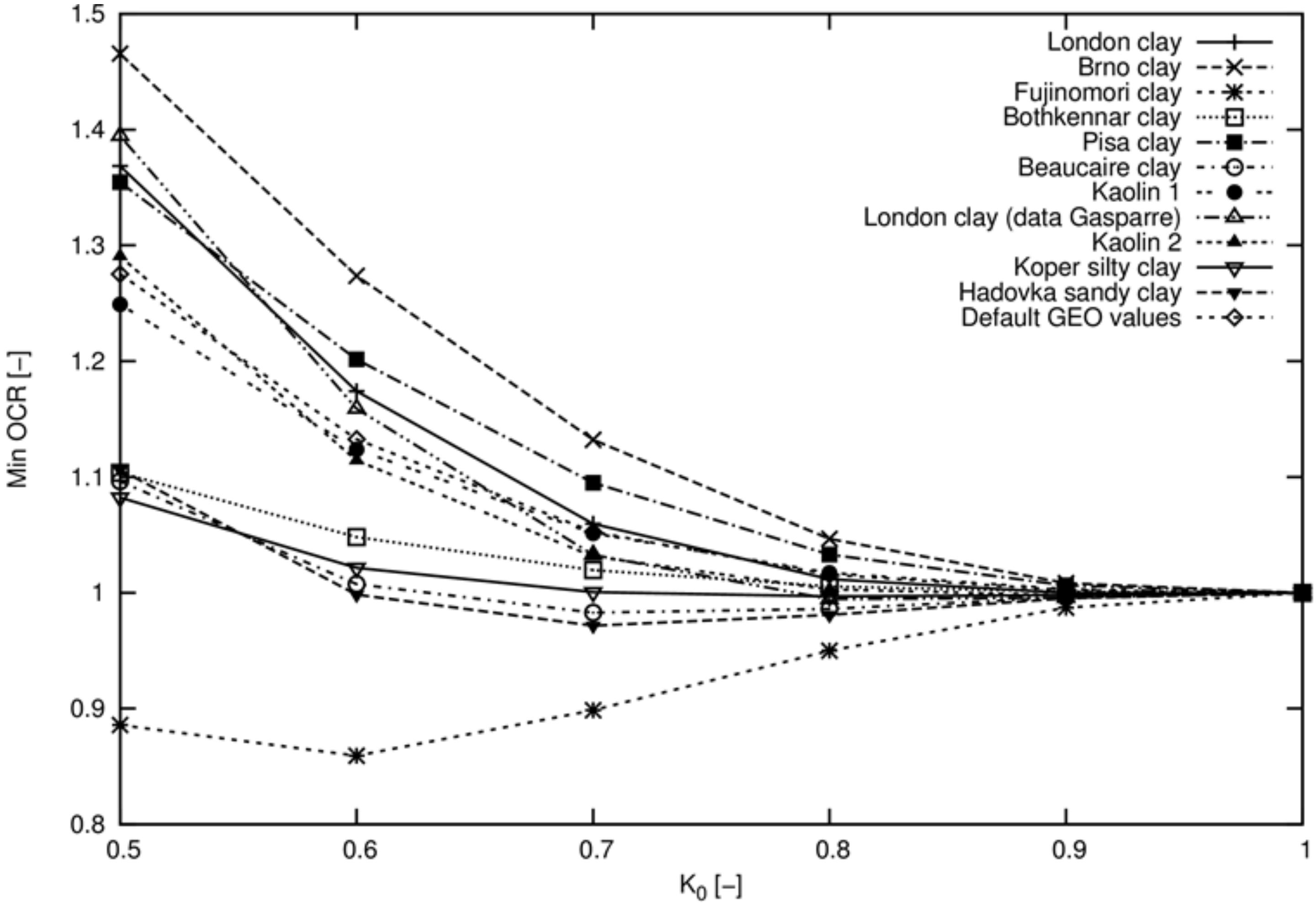


Figure 5: Dependence of OCR on the coefficient of earth pressure at rest K_o

Soil/K_o	0.5	0.6	0.7	0.8	0.9	1.0
	OCR					
London clay	1.369	1.174	1.059	1.011	1.000	1.0
London clay (data Gasparre)	1.394	1.159	1.033	0.994	0.995	1.0
Fujinomori clay	0.886	0.859	0.898	0.950	0.987	1.0
Bothkennar clay	1.104	1.048	1.019	1.001	1.001	1.0
Pisa clay	1.354	1.202	1.095	1.033	1.006	1.0
Beaucaire clay	1.096	1.008	0.983	0.986	0.996	1.0
Kaolin 1	1.249	1.123	1.051	1.017	1.003	1.0
Kaolin 2	1.291	1.114	1.031	1.001	0.998	1.0
Koper silty clay	1.081	1.021	1.001	0.997	0.998	1.0
Brno clay	1.466	1.274	1.132	1.047	1.008	1.0
Evropská (Hadovka) sandy clay	1.106	0.998	0.972	0.981	0.995	1.0
GEO FEM default values	1.275	1.132	1.052	1.016	1.002	1.0

Table 1: Oveconsolidation ratio OCR *of the selected soils as function of* K_0 *value*

Soil	φ_{cv}	λ	κ	N	r
London clay	22.6	0.11	0.016	1.375	0.4
London clay (data Gasparre)	21.9	0.1	0.02	1.26	0.5
Fujinomori clay	34.0	0.045	0.011	0.887	1.3
Bothkennar clay	35.0	0.12	0.01	1.34	0.07
Pisa clay	21.9	0.14	0.01	1.56	0.3
Beaucaire clay	33.0	0.06	0.01	0.85	0.4
Kaolin 1	27.5	0.11	0.01	1.32	0.45
Kaolin 2	27.5	0.07	0.01	0.92	0.67
Koper silty clay	33.0	0.103	0.015	1.31	0.3
Trmice clay	18.7	0.09	0.01	1.09	0.18
Brno clay	19.9	0.13	0.01	1.51	0.45
Evropská (Hadovka) sandy clay	32.4	0.0411	0.0078	0.593	0.2
GEO FEM default values	27.0	0.1	0.01	1.2	0.4

Table 2: Material parameters of the selected soils

Intergranular strain

The basic version of the model is suitable in analyses with a prevailing direction of the stress loading path. In cases with cyclic loading (loading-unloading-reloading) it is more suitable to use an advaced formulation with the concept of intergranular strain. This allows for constraing a unacceptable increase of perment deformation arising during small repeating changes in load (ratcheting). Introducing intergranular strain allows for the modeling of large stiffnes, which clays experience during small strains. This option is not part on any other models implemented in GEO FEM. The concept of intergranular strain assumes that the total soil deformation consists of a small deformation of an intergranular layer (integranular strain) and deformation caused by mutual sliding of grains. Changing the load path changes first the intergranular strain. Upon reaching the limit value of the intergranular strain, the deformation associated with the motion of grains sets on.

Adopting the concept of intergranular strain requires five additional parameters:

- Range of elastic intergranular strain R
- Parameters m_R and m_T control the small strain stiffness
- Parameters β_r and χ control the degree of stiffness degration with increasing shear strain

These parameters are calibrated after knowing already the material data of the basic hypoplastic model.

Margin of elastic intergranular deformation R

- It determines the range of maximal intergranular strain
- It can be determined by a parametric study of the degradation curve $G = G(\varepsilon_s)$, Figure 5
- Alternatively it can be considered as material independent constant $R = 10^{-4}$
- Most common values are in range $2*10^{-5}$ - $1*10^{-4}$

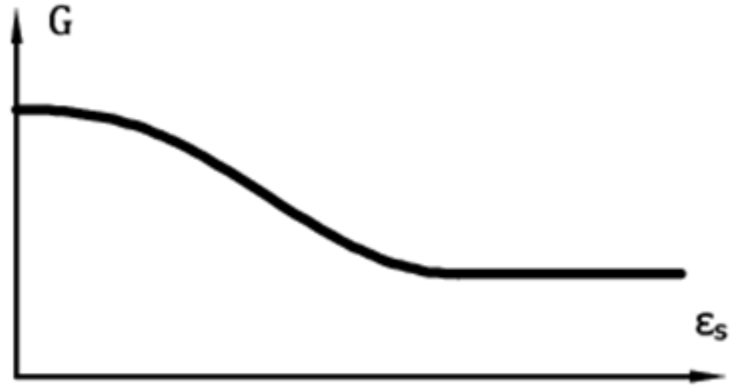


Figure 6: Curve describing the loass of stiffness of shear modulus

Parameter m_R

- It determines the magnitude of the shear modulus when changing the loading path in the meridial plane (σ_m - J) o 180°
- Linear ratio between parameter m_R and the intial shear modulus G_0 is provided by $G_0 = p*(m_r/(r*\lambda^*)$
- The initial shear modulus can be determined from the measurement of shear wave propagation [2]
- Most common values are in range 4.0 - 20.0

Parameter m_T

- It determines the magnitude of the shear modulus when changing the loading path in the meridial plane (σ_m - J) o 90°
- It holds $m_R/m_T = G_0/G_{90}$
- The ratio of initial moduli can be estimated from the ratio of these moduli for larger strains.The value of the m_R/m_T ratio is commonly in the range of 1.0 - 2.0
- Most common values of m_T are in range 2.0 - 20.0

Parameters β_r and χ

- Determine the rate of stiffness degradation with increasing shear strain
- It can be determined by a parametric study of the degradation curve $G = G(\varepsilon_s)$
- Most common values of parametr β_r are in fange 0.05 - 0.5
- Most common values of parametr χ are in range 0.5 - 6

Literature:

[1] D. Mašín, A hypoplastic constitutive model for clays, International Journal for Numerical and Analytical Methods in Geomechanics., 29:311-336, 2005.

Variable Stiffness

The soil stiffness can be specified to vary with depth. This property is defined in the dialog "**Edit soil parameters**" by choosing "**Stiffness increasing with depth**". Entering parameter k_d ensures that the finite elements will be assigned an individual value of Young's modulus depending on their depth (vertical distance from the terrain in the first construction stage) according to the equation

$$E(h) = E_0 + k_d h$$

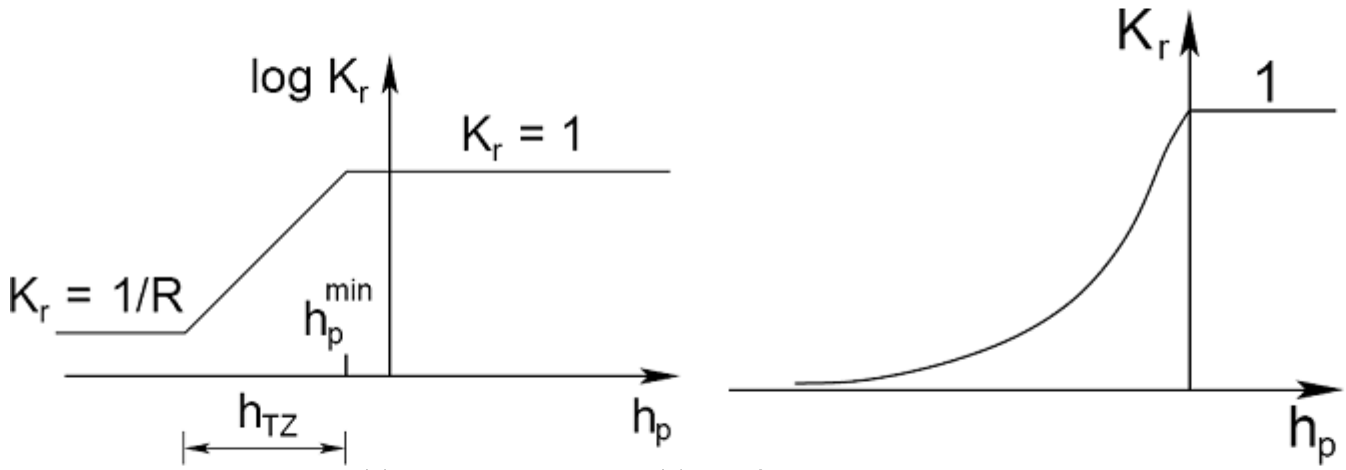
where: E_0 - Young's modulus on terrain [MPa]
h - depth [m]

Material Models in Flow Analysis

The steady state flow analysis is driven by Darcy's law specifying the relationship between the flow velocity and the gradient of hydraulic head. The current version of the program assumes constant values of coefficients of permeability independent of pore pressure.

The program also requires specifying the initial void ratio e_0 for the determination of the current porosity n and subsequently the actual velocity of water flowing through pores only $v_s = v/n$, where v is the average flow velocity through the entire seepage area. Generally, the void ratio $e_0 = 1$ corresponds to soil porosity of $n = 50\%$.

By introducing the coefficient of relative permeability K_r the program allows for tracking the transition zone between fully saturated ($S = 1$, $K_r = 1$) and unsaturated (K_r => 0) region of the soil body. As an example we may consider the problem of unconfined flow. The process of tracking the transition zone is governed by one of the three models of transition zone determining the evolution of coefficient of relative permeability K_r as a function of pore pressure head, see Figures.



(a) Log-linear model [1], (b) Van Genuchten model [2]

Log-linear model

The Log-linear transition zone model described e.g. in [1] is defined by the following parameters:

h_p^{min} - minimum value of pressure head in fully saturated region [kPa]
h_{TZ} - transition zone width [m]
R - reduction parameter, a sufficiently large number $R = 100$ až 1000 [-], default setting assumes $R = 1000$ [-]

The coefficient of relative permeability K_r is given by:

$$K_r(h_p) = 10^{\frac{\left(h_p - h_p^{min}\right)\log R}{h_{TZ}}}$$

Gardner model

This is an equivalent model depending on a single parameter α [$1/m$] only. The coefficient of relative permeability K_r is in this case given by [4]:

$$K_r(h_p) = e^{\alpha h_p}$$

Van Genuchten model

In this case, the value of coefficient of relative permeability K_r is given by:

$$K_r(h_p) = \frac{\left\{1 - (-\alpha h_p)^{n-1}\left[1 + (-\alpha h_p)^n\right]^{-m}\right\}^2}{\left[1 + (-\alpha h_p)^n\right]^{m/2}}$$

where α [$1/m$], $n > 1$, $m = 1 - 1/n$ are model parameters. Their values can be obtained from laboratory measurements of retention curves approximated by:

$$S = S_r + (S_{sat} - S_r)\,\Theta$$

$$\Theta = \left[\frac{1}{1+(-\alpha h_p)^n}\right]^m$$

where: S_{sat} - degree of saturation of fully saturated soil, default setting S_{sat} = 1
S_r - residual degree of saturation
Θ - normalized water content

Parameter Θ is in general provided by:

$$\Theta = \frac{\theta - \theta_r}{\theta_S - \theta_r}$$

where: θ_r - residual water content [m^3/m^3]
θ_S - water content of fully saturated soil [m^3/m^3]

The current degree of saturation S can be expresses as a ratio of the water content θ and porosity n as follows (it is necessary to distinguish between n representing porosity and n, which appears in the van Genuchten model, they are two different variables):

$$S_w = \frac{\theta}{n}$$

The Log-linear and Gardner models adopt a simplified version of the van Genuchten model according to [5]:

$$\Theta = K_r^b$$

where $b >$ 0 [-] is a fitting parameter allowing for a better approximation of the retention data of a given soil.

We recommend the following tables of parameters which are actually used in the program. These parameters are derived from those given in the original tables.

Optimal values of parameters of the van Genuchten model for various classifications based on USDA and FAO are presented in the following tables.

Table with regression coefficients for grain size USDA according to Van Genuchten (1991)

Soil (grain size)	K_{sat} [m/day]	**RETC**				**Rosetta**			
		e [-]	S_r [-]	α [$1/m$]	n [-]	e [-]	S_r [-]	α [$1/m$]	n [-]
Sand	7.13	0.75	0.11	14.5	2.68	0.60	0.14	3.5	3.18
Loamy sand	3.50	0.70	0.14	12.4	2.28	0.64	0.13	3.5	1.747
Sandy loam	1.06	0.70	0.16	7.5	1.89	0.63	0.10	2.7	1.448
Loam	0.25	0.75	0.18	3.6	1.56	0.66	0.15	1.1	1.474
Silt	0.06	0.85	0.07	1.6	1.37	0.96	0.10	0.7	1.677
Silt loam	0.11	0.82	0.15	2.0	1.41	0.78	0.15	0.5	1.663
Sandy clay loam	0.314	0.64	0.26	5.9	1.48	0.62	0.16	2.1	1.33
Clay loam	0.062	0.70	0.23	1.9	1.31	0.79	0.18	1.6	1.415
Silty clay loam	0.017	0.75	0.21	1.0	1.23	0.93	0.19	0.8	1.52
Sandy clay	0.029	0.61	0.26	2.7	1.23	0.63	0.30	3.3	1.207
Silty clay	0.0048	0.56	0.19	0.5	1.09	0.93	0.23	1.6	1.321
Clay	0.048	0.61	0.18	0.8	1.09	0.85	0.21	1.5	1.253

Table with regression coefficients for grain size FAO according to Van Genuchten (1998)

Soil (grain size)	K_{sat} [m/day]	e [-]	S_r [-]	α [$1/m$]	n [-]
Top soil (up to depth 1 m)					
Coarse (C)	0.600	0.68	0.062	3.83	1.3774
Medium (M)	0.121	0.78	0.023	3.14	1.1804
Medium fine (MF)	0.023	0.75	0.023	0.83	1.2539
Fine (F)	0.248	1.08	0.019	3.67	1.0120
Very fine (VF)	0.150	0.78	0.016	2.65	1.1033
Soil at depth (> 1 m)					
Coarse (C)	0.700	0.58	0.068	4.30	1.5206
Medium (M)	0.108	0.65	0.026	2.49	1.1689

Medium fine (MF)	0.040	0.70	0.024	0.82	1.2179
Fine (F)	0.085	0.93	0.021	1.98	1.0861
Very fine (VF)	0.082	1.17	0.019	1.68	1.0730

Table: FAO texture classification system

Soil	Definition
Coarse (C) Medium (M)	clay $<$ 18% and sand $>$ 65% 18% $<$ clay $<$ 35% a 15% $<$ sand nebo: clay $<$ 18% a 15% $<$ sand $<$ 65%
Medium fine (MF)	clay $<$ 35% a sand $<$ 15%
Fine (F)	35% $<$ clay $<$ 60%
Very fine (VF)	60% $<$ clay

Literature:

Details can be found in [2].

[1] D.M. Potts, L. Zdravkovič, Finite element analysis in geotechnical engineering - theory, Thomas Telford, London, 1999.

*[2] M. Th. Van Genuchten, A closed formulation for predicting the hydraulic conductivity of unsaturated soils, Journal Soil Science Society of America **44**, 239-259, 1988..*

[3] M. Šejnoha, Finite element analysis in geotechnical design, to appear (2015)

*[4] W. R. Gardner, Some steady-state solutions of the unsaturated moisture flow equation to evaporation from a water table, Soil Science **85(4)**, 228-232, 1958.*

[5] M. Šejnoha, T. Janda, H. Pruška, M. Brouček, Metoda konečných prvků v geomechanice: Teoretické základy a inženýrské aplikace, předpokládaný rok vydání (2015)

[6] USDA 1951. Soil Survey Manual. Soil Conservation Service. U.S. Department of Agriculture Handbook No. 18. US Government Printing Office. Washington DC.

*[7] Wösten, J.H.M., et. al. 1998. Using existing soil data to derive hydraulic parameters for simulation models in environmental studies and in land use planning. Final Report on the European Union Funded project. DLO Winand Staring Centre. Report 156, Wageningen, NL. **p. 106**. ISSN 0927-04537.*

Coefficient of Permeability

Ability of porous body (soils, rocks) to transport water of given properties (e.g. ground water) is denoted as seepage. The amount of water flowing through a certain area can be represented by the **coefficient of permeability**. The coefficient of permeability represents the slope of a linear dependence of water flow velocity on the gradient of total head (gradient of hydraulic head) in Dracy's law written as:

$$\mathbf{v} = n\mathbf{v}_s = -K_r \mathbf{K}_{sat} \nabla h$$

where:

$\mathbf{v}_s$	-	velocity of water flowing through pores
n		porosity
K_r		relative coefficient of permeability
$\mathbf{K}_{sat}$		permeability matrix storing coefficients of permeability of fully saturated soil k_x, k_y, which may be different along individual coordinate axes
∇h		gradient of total head

Total head at a given point of region of flow is defined as a sum of the pressure head and vertical coordinate and as such it determines the height of water in piezometer at a given point:

$$h = \frac{p}{\gamma_w} + z$$

where: γ_w - the weight of water

Example values of coefficients of permeability for various soils *(Myslivec)*

Type of soil	Coefficient of permeability k $[m/day]$	Motion of water particle by $1\ cm$ for hydraulic gradient $i = 1$ per time
Soft sand	10^2 - 10	*6 s - 10 min*
Clayey sand	10^{-1} - 10^{-2}	*100 min - 18 hrs*
Loess loam	10^{-2} - 10^{-4}	*18 hrs - 70 days*
Loam	10^{-4} - 10^{-5}	*70 days - 2 years*

Clayey soil	10^{-5} - 10^{-6}	*2 years - 20 years*
Clay	10^{-6} - 10^{-7}	*20 years - 200 years*

There are several ways for determining the coefficient of permeability k . They grouped as follows:

a) Laboratory measurements

Several types are available for the range of k 10^4 - $10^{-6} m/day$.

b) Field measurements

Dwell or sink tests, measurement of filtration velocity of flow, for the range of k 10^6 - $1 m/day$.

c) Using empirical expressions

$$k = 100\, d_{10}^2 e^2$$

Suitable for non-cohesive soils, k 10^6 - $10 m/day$, they produce only guidance values - e.g. according Terzaghi:

where: k - coefficient of permeability [cm/s]

d_{10} - diameter of effective solid particle [cm]

e - void ratio [-]

d) By calculation from time dependent consolidation process

One must know the coefficient of consolidation c_v and consolidation curve (semi-logarithmic dependence of deformation on time).This is only an indirect determination from the expression:

$$k = \frac{c_v \rho_w g a_v}{1 + e_0}$$

where: e_0 - initial void ratio

c_v - coefficient of consolidation

ρ_w - unit density of water

g - gravitational acceleration

a_v - coefficient of compressibility

Basic Data

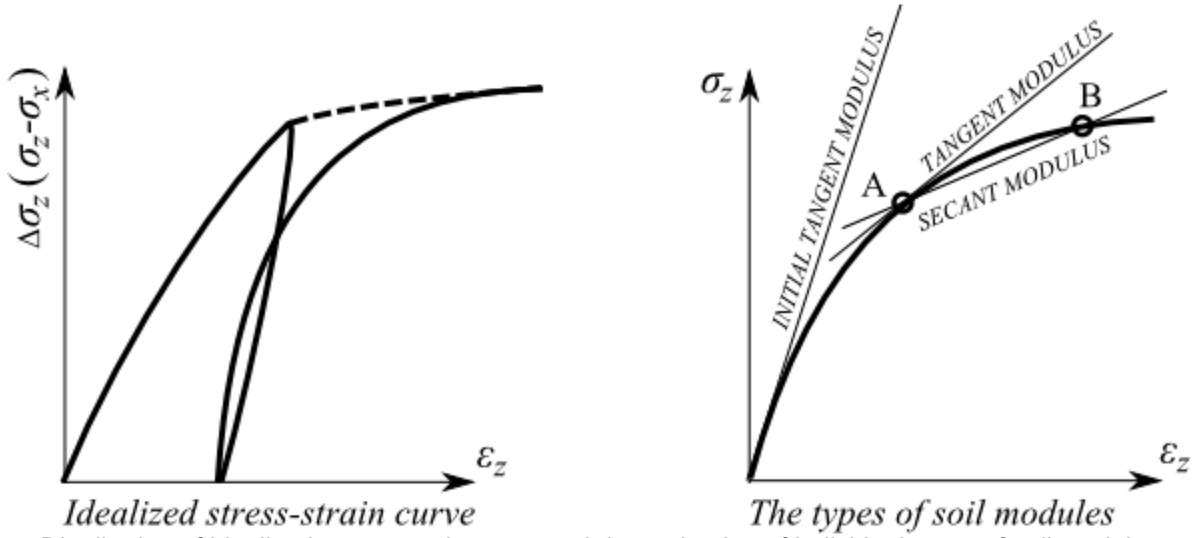
The following material parameters are required for all material models.

Unit weight γ - the unit weight of a dry soil (soil above the groundwater table, GWT) is assumed. The unit weight of a soil below the GWT is calculated from other parameters introduced in section "Uplift".

Modulus of elasticity E

The modulus of elasticity describes the material stiffness that is assumed constant over the entire load interval. In case of soils this assumption is, however, valid only for a very narrow interval of recoverable deformations. Modulus of elasticity E has no significant effect on soil behavior for nonlinear models after satisfying plasticity condition.

A straightforward answer to what definition and what value of this material parameter (initial, tangent, secant) one should use in a given material model is, unfortunately, not available. To select a given type of modulus one needs know the **soil behavior** in the analyzed geotechnical task and to assign a particular magnitude the results from a **triaxial test** for corresponding stress paths are necessary. Nevertheless, certain recommendations can be provided.



Distribution of idealized stress-strain curve and determination of individual types of soil modules

The following interpretation of Young's modulus E of elasticity is available:

- **instantaneous modulus** E_0 in case of small loads (assumption o linear dependence of strain and stress) or when instantaneous settlement is calculated
- **secant modulus** E_{50} is determined for a reference stress equal to 50% of stress at the onset of failure (used for example when analyzing spread foundations and settlement of pile foundations)
- **deformation modulus** E_{def} is determined from a stress-strain curve derived experimentally. This modulus is required when using the modified elastic model, which it assumes different behavior for load and unloading. Using this modulus when solving the problem of soil unloading (e.g., underground structures, heaving of bottom a foundation ditch) leads to larger deformations than when using the elastic modulus E_{ur} determined from unloading branch r of the stress-strain curve. Determining of deformation modulus of soil is applicable by following approximate relation:

$$E_{ur} = 3E_{def}$$

- **oedometric modulus** E_{oed} which depends on the level of stress in the soil should be used depending on the expected range of stress in the soil may experience. The relation between E_{def} and E_{oed} is provided by:

$$E_{oed} = \frac{E_{def}}{\beta}$$

$$\beta = 1 - \frac{2\,\nu^2}{1-\nu}$$

where: ν - Poisson's ratio
E_{def} - deformation (secant) modulus

- **modulus of elasticity** E_{ur} determined from the unloading branch of stress-strain curve is used when solving the problem of soil unloading (excavations) - must be defined when using the modified elastic model

The values of modules of elasticity should be determined, if possible, from a triaxial shear test. If other methods (penetration tests, pressuremeter tests etc.) are used then it becomes necessary to introduce some correlation coefficients are described in literature.

For actual modeling we recommend to perform an analysis according to the elastic material model at first and check the resulting strain field - such strains according to Hookes's law are linearly dependent on the applied load and the used elastic modulus. If the resulting strains (displacements) are already **too large** the user should **reassess the magnitude of the originally applied elastic modulus**.

Poisson's ratio ν - coefficient of transverse contraction is in case elastic homogeneous material loaded by normal stress in one direction given by:

$$\nu = \frac{\varepsilon_y}{\varepsilon_x}$$

where: ε_y - vertical strain
ε_x - horizontal strain

The Poisson's ratio is relatively easy to determine. To select its value one may take advantage of the built-in soil database. If small loads are assumed and the instantaneous modulus E_0 is used, then also the value of the Poisson's ratio ν_0 determined for the initial loading should be employed.

Geostatic Stress, Uplift Pressure

Stress analysis is based on existence of soil layers specified by the user during input. The program further inserts fictitious layers at the locations where the stress and lateral pressure (GWT, points of construction, etc.) change. The normal stress in the i^{th} layer is computed according to:

$$\sigma_i = \sum h_i \gamma_i$$

where: h_i - thickness of the i^{th} layer

γ_i - unit weight of soil

If the layer is found below the **ground water table**, the unit weight of soil below the water table is specified with the help of input parameters of the soil as follows:

- for option "**Standard**" from expression:

$$\gamma_{su} = \gamma_{sat} - \gamma_w$$

where: γ_{sat} - saturated unit weight of soil

γ_w - unit weight of water

- for option "**Compute from porosity**" from expression:

$$\gamma_{su} = (1-n)(\gamma_s - \gamma_w)$$

where: n - porosity

γ_s - specific weight of soil

γ_w - unit weight of water

$$\gamma_s = \frac{G_d}{V - V_p}$$

where: V - volume of soil

V_p - volume of voids

G_d - weight of dry soil

Unit weight of water is assumed in the program equal to $10\ kN/m^3$ or $0{,}00625\ ksi$.

Assuming inclined ground behind the structure ($\beta \neq 0$) and layered subsoil the angle β , when computingthe coefficient of earth pressure K , is reduced in the i^{th} layer using the following expression:

$$tg\beta_i = \frac{\gamma}{\gamma_i} tg\beta$$

where: γ - unit weight of the soil in the first layer under ground

γ_i - unit weight of the soil in the i^{th} layer under ground

β - slope inclination behind the structure

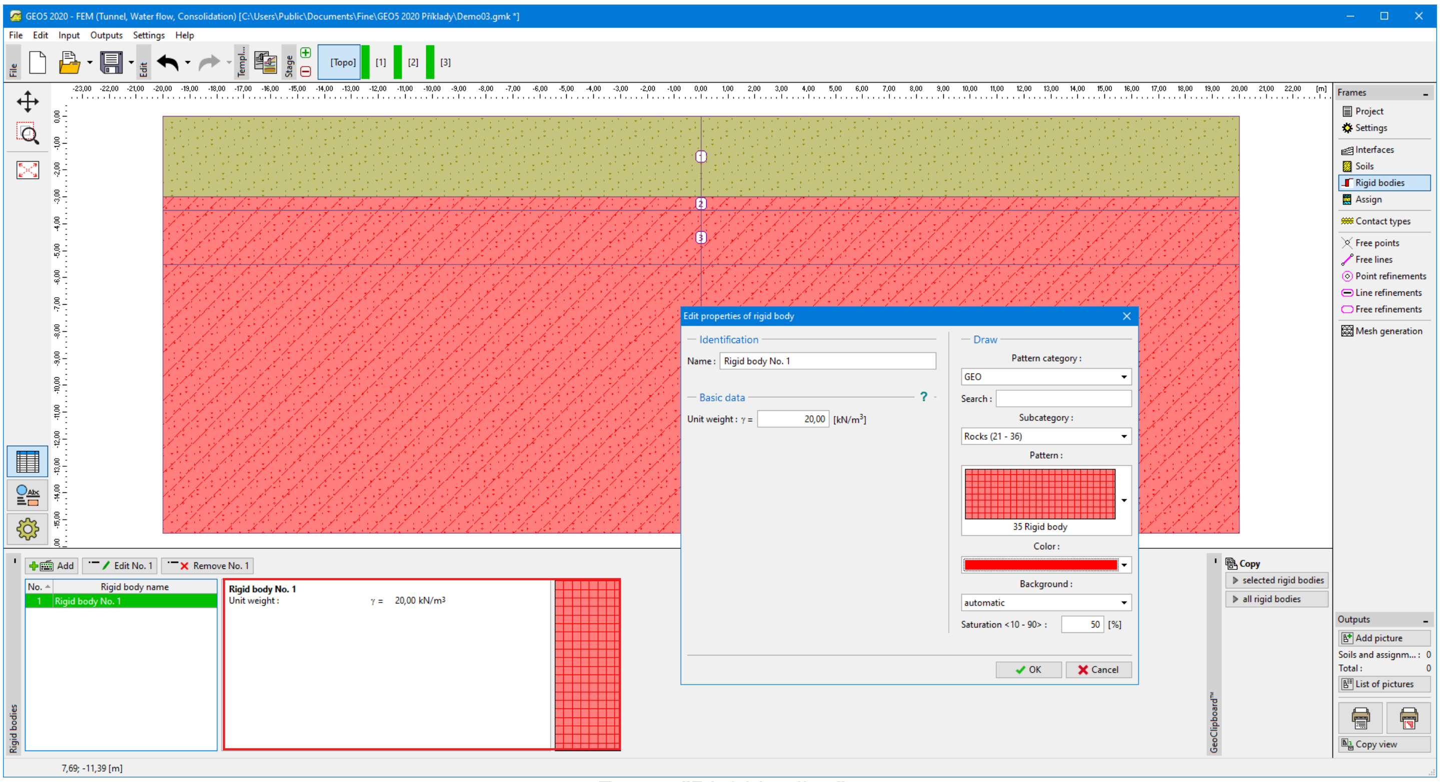
Rigid Bodies

The "**Rigid bodies**" frame contains a table with the list of input rigid bodies.

The program allows for adding rigid bodies. Here the only required input parameter is the unit weight of the rigid body. The material of the rigid body is assumed an **infinitely stiff**. These bodies serve mainly to model massive concrete structures and walls in both standard and stability analyses.

Adding rigid bodies is performed in the dialog window "**Add new rigid body**".

Input rigid bodies can be copied within all 2D GEO5 programs using "GeoClipboard".



Frame "Rigid bodies"

Assign

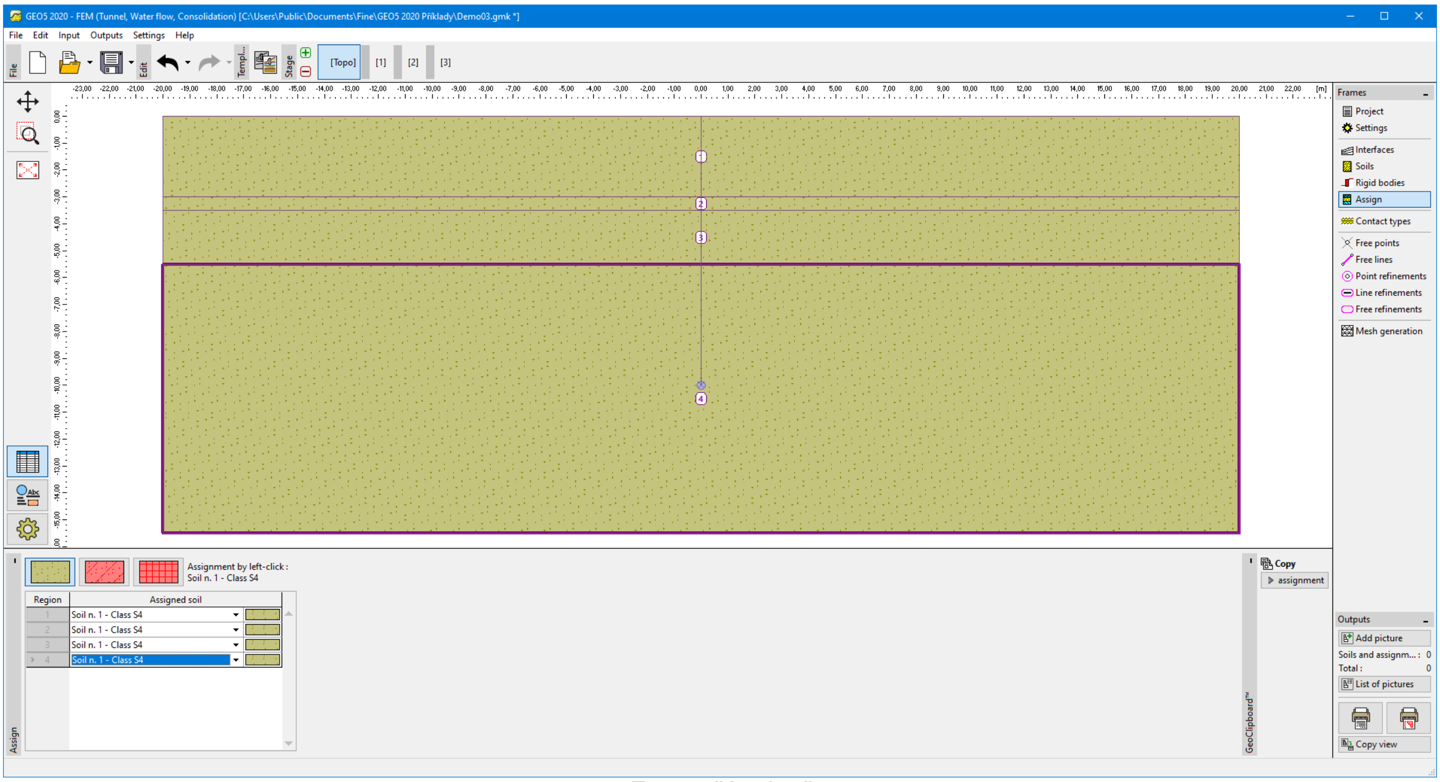
The frame "**Assign**" contains a list of layers of the profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table or is accessible from a combo list for each layer of the profile.

The procedure to assign a soil into a layer is described in detail herein.

Unlike other programs the soils, which become active in calculations stages, are assigned to regions rather than to interfaces. The regions are created automatically when creating the computational model.

When a new soil is assigned in a topology regime, it is automatically assigned to all regions in a given geological layer.

Assign of soils can be copied within all 2D GEO5 programs using "GeoClipboard".

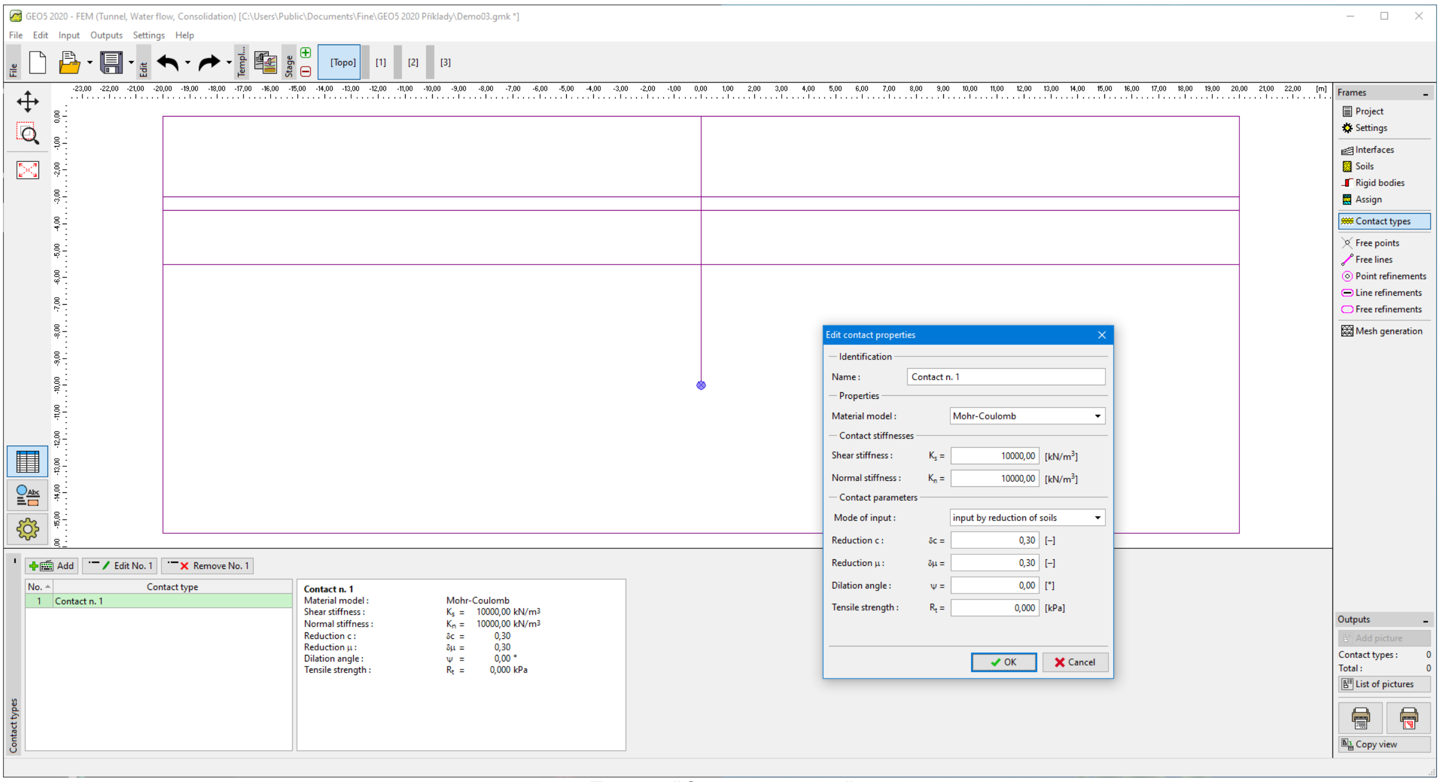


Frame "Assign"

Contact Types

The frame "**Contact types**" contains a table with the list of types of contacts. Adding contacts is performed in the "**New types of contact**" dialog window.

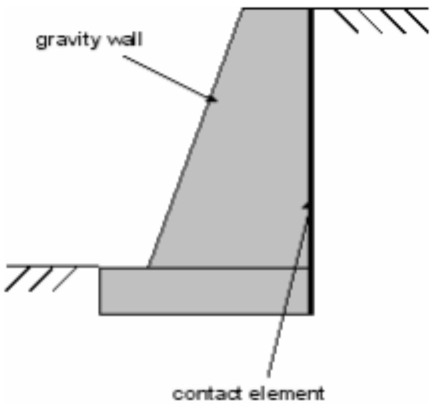
This dialog window serves to define new contact elements which can be subsequently introduced into the program using the "Beams" and "Contacts" frames. The material model of a contact element can be either linear or nonlinear.



Frame "Contact types"

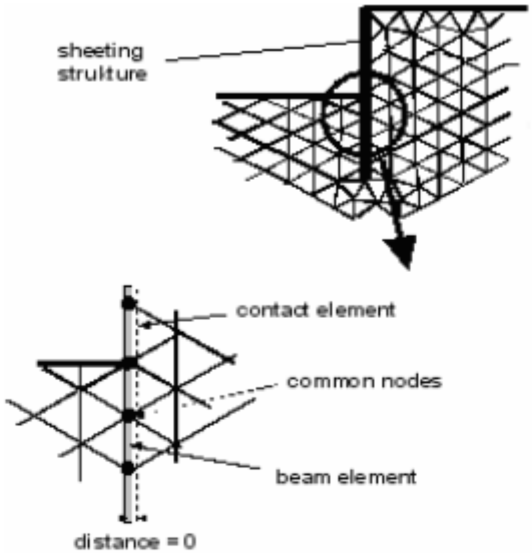
The **contact elements** are used in applications that require studying the interaction of the structure and soil. They can be further used to model joints or interfaces of two distinct materials (soil - rock interface). A typical example of using contact elements is the **modeling of sheeting structures, retaining walls, or tunnel lining**. In such applications, the contact elements are used to model a relatively thin layer of a soil or rock loaded primarily in shear.

Contacts can also be defined independently along **individual soil interfaces**.



Location of contact elements when modeling a gravity wall

The contact element is an element with a zero thickness allowing for calculating the interfacial stress as a function of a relative displacement developed along with the interface.



Construction of a sheeting wall represented by beam and contact elements

Contact Elements

Two options of the contact element material model are available. One may select either the **elastic model** with the possibility of plotting contact stresses while assuming the elastic behavior along with the interface or the **plastic model**. The plastic model is based on the classical Mohr-Coulomb model extended by including the tension cut-off.

This model is therefore well suited when modeling tensile separation. In certain applications such as sheeting structures, this model is vital for receiving meaningful predictions of the soil and structure response.

The basic model parameters are the cohesion c , coefficient of friction μ and angle of dilation ψ . The parameters c and μ can be specified also indirectly by reducing the soil strength parameters c and tan (φ) of adjacent to the contact. If the contact is assumed between two soils (rocks) then the one having smaller values of c and φ is used in the reduction step.

The contact parameters are then defined as:

$$c = \sigma_z . c_{zem}$$

$$\mu = \sigma_\mu . \tan(\varphi_{zem})$$

If no better information regarding the reduction of parameters is available one may use the following values. For steel structures in sandy soils, the reduction parameter equal to $2/3$ is reasonable while for clays the value of $1/3$ can be used. These parameters usually attain higher values when concrete structures are used. In general, the reduction parameters should be less than 1 . The dilation angle plays the same role as in the case of standard soil models. Just recall that by setting $\psi = 0$ we prior assume elastic behavior in the tension/compression. The plastic deformation is thus limited to shear.

Additional parameters of the contact material model are the elastic stiffnesses in the **normal** and **tangential** directions k_n and k_s , respectively. They can be imagined as spring stiffnesses along with a given interface. A reliable selection of the values of these parameters is not an easy task and is usually problem-dependent. To shed a light on this subject one may relate these stiffnesses to the material parameters of the soil adjacent to the contact. The following relations then apply:

$$K_n = \frac{E}{t}$$

$$K_s = \frac{G}{t}$$

where: t - assumed (fictitious) thickness of contact (interface) layer

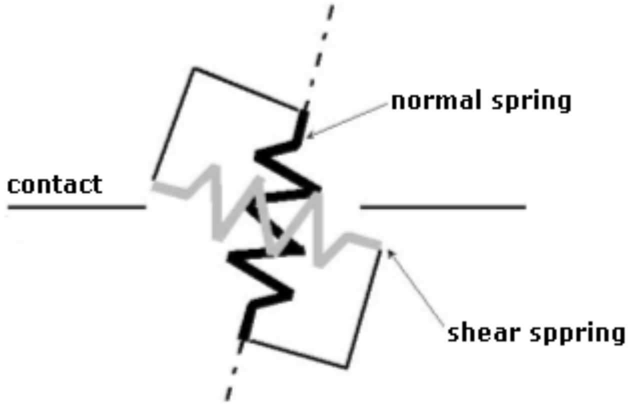
G - shear modulus of elasticity

E - Young's modulus of elasticity

In the case of distinct materials (E_1, E_2, G_1, G_2) we take the lower value of k_s and $k\ n$.

Although in case of a fully plastic behavior the selection of parameters k_s and k_n is not essential, the values assigned to these parameters are decisive for the success of the solution of a given nonlinear problem. Providing these values are two large (above $100000\ kN/m^3$) the iteration process may oscillate. On the other hand, setting the values of k_s and k_n too low (below $10000\ kN/m^3$) lead to nonrealistic deformations of structure.

The default setting in the program is $10000\ kN/m^3$.



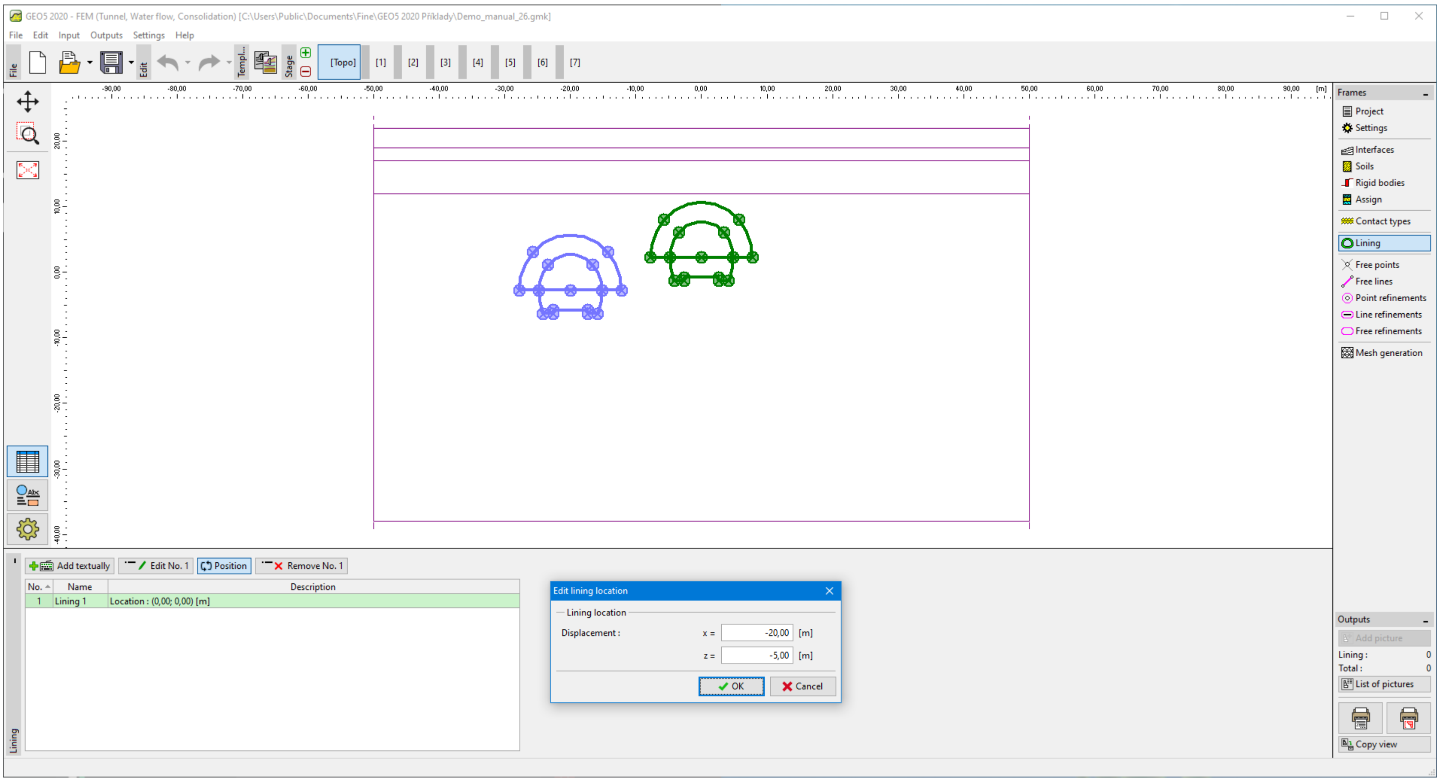
Visualization of elastic stiffnesses

Lining

The frame "**Lining**" contains a table with the list of input linings. This frame becomes accessible in the program once the "**Tunnel**" regime is activated in the frame "Settings". The "Lining - FEM" module simplifies the modeling and positioning of individual tunnel linings.

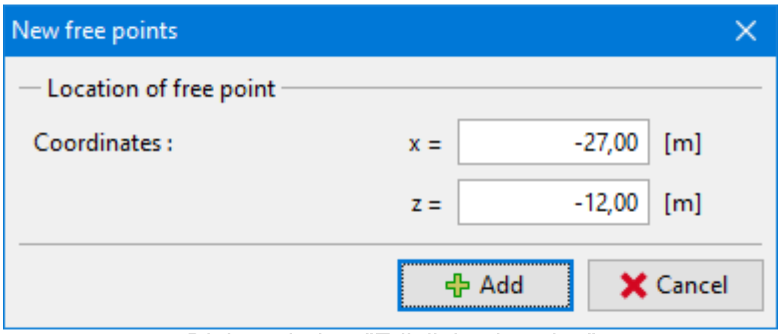
The "Lining - FEM" module is an independent program used to design linings. Free points, free lines, line refinement, anchors, beams, and beam loads created in this module are passed into the FEM program. Although behaving in a standard way, they cannot be edited in the FEM program. Editing is only possible in the "Lining - FEM" module.

Adding lining is performed in the "Lining - FEM" module. The lining can also be edited on the desktop with the help of active objects.



Frame "Lining"

Pressing the "**Position**" button opens the "**Adjust lining location**" dialog window, which allows for modifying coordinates of the lining location. To adjust lining in the FEM program is possible even without launching the "Lining - FEM" module.



Dialog window "Edit lining location"

Module Lining - FEM

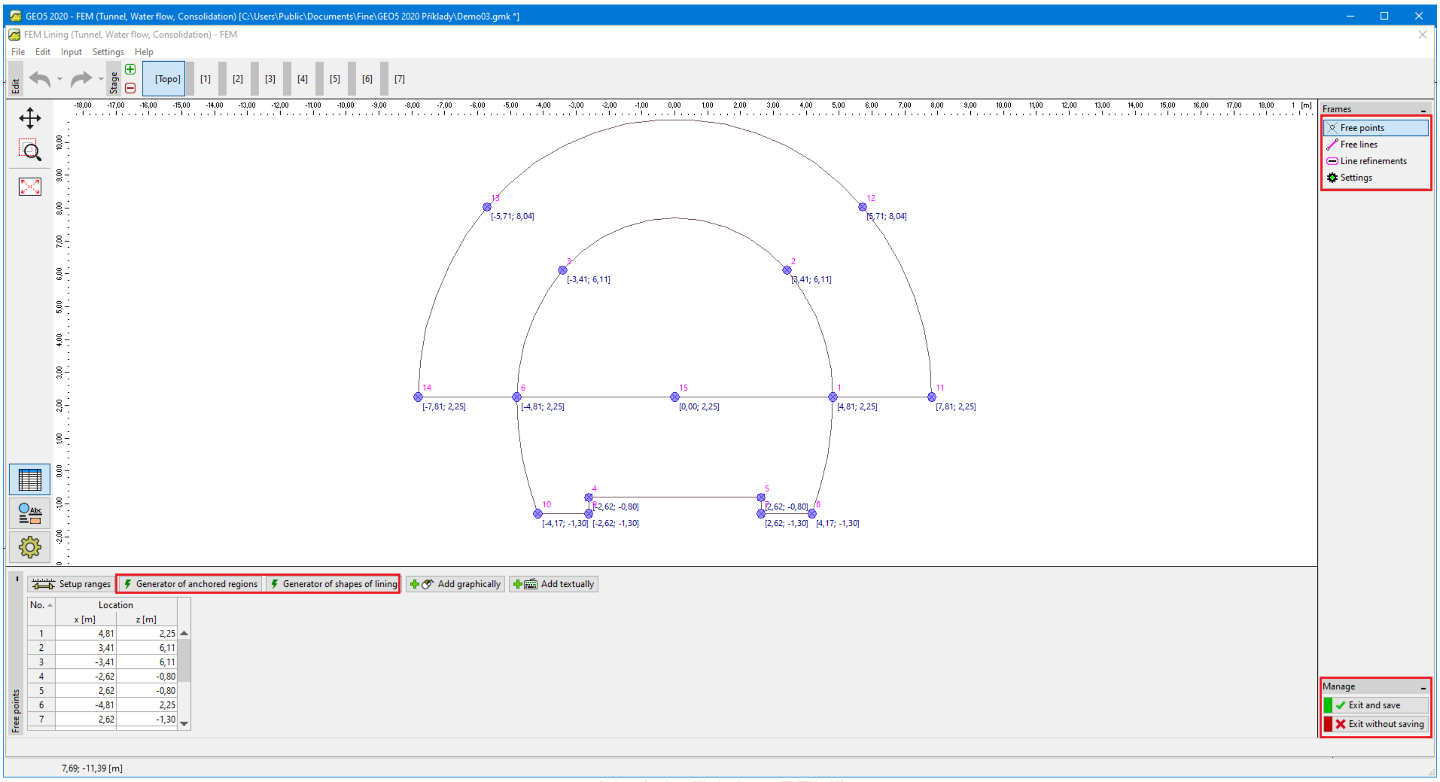
The "**Lining - FEM**" module simplifies modeling and positioning individual linings of tunnels. The module disposes of the features of the main FEM program including the "**Topology**" regime and stages of construction. In the "**Topology**" regime the module contains the "Free points", "Free lines", "Line refinement" and "Settings" frames. Frames accessible from stages of constructions are described within the stages of the construction regime of the FEM program.

The "**Exit and save**" button can be used to terminate the work in the module and to transmit data into the FEM program, whereas the "**Exit without saving**" button just terminates the work without data transmission.

The program makes it possible to import data in the *.DXF format.

The data of the lining module can be independently saved or loaded while in this dialog using standard functions "**Open**" and "**Save**". This way allows for transmitted the lining between several analyzed tasks or within a single task.

Load a lining, having less number of stages than the current state, will add the remaining stages. In the case of lining having more stages, the corresponding stages are first added to the dialog and then to the main window. The data from the lining regime cannot be loaded directly into the main window.

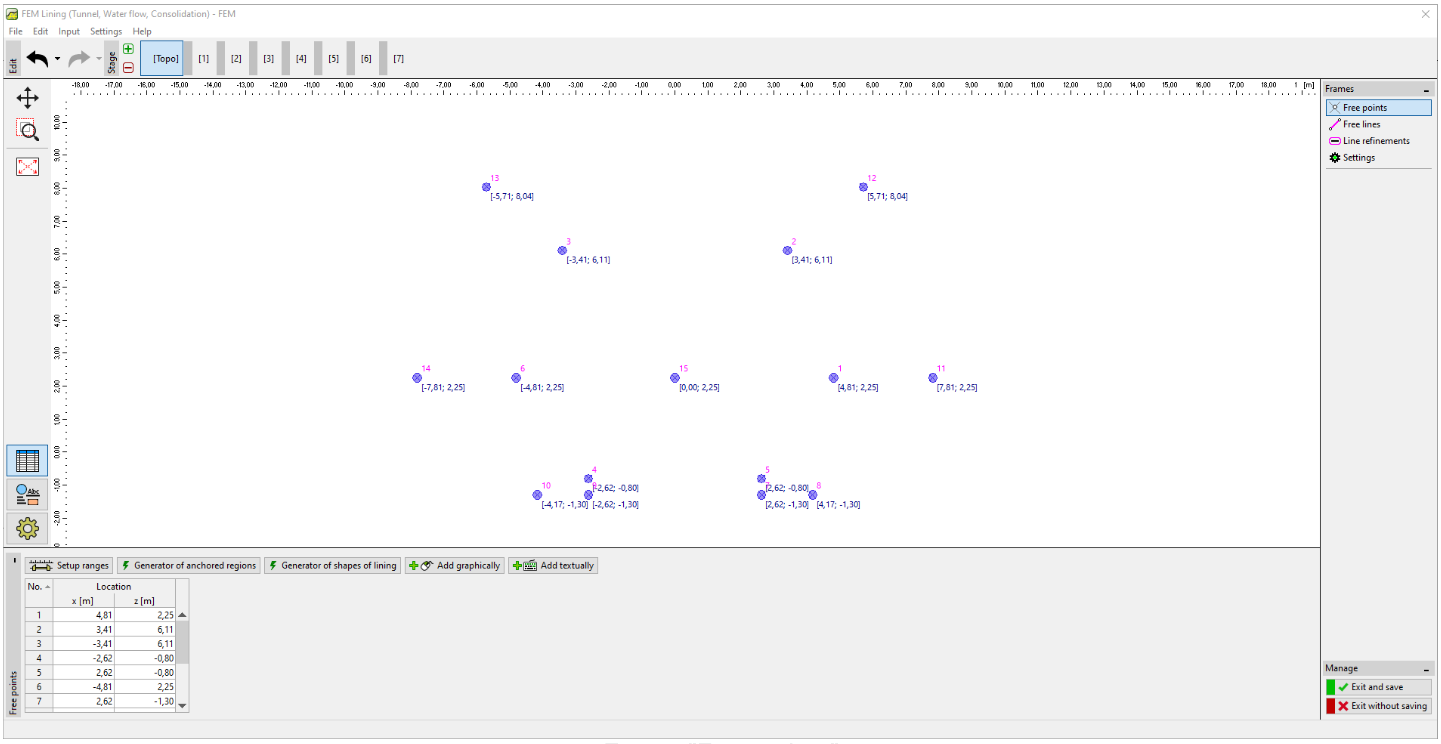


Module "Lining FEM"

Free Points

The frame "**Free points**" contains a table with the list of input free points. Working with free points follows the same guidelines as in the FEM program - frame "Free points".

The frame differs by the functions on the horizontal toolbar, which contains the "Generator of shape of lining" and "Generator of anchored regions" buttons. The function of the "**Range**" button is identical to that in the FEM program - frame "Interface".

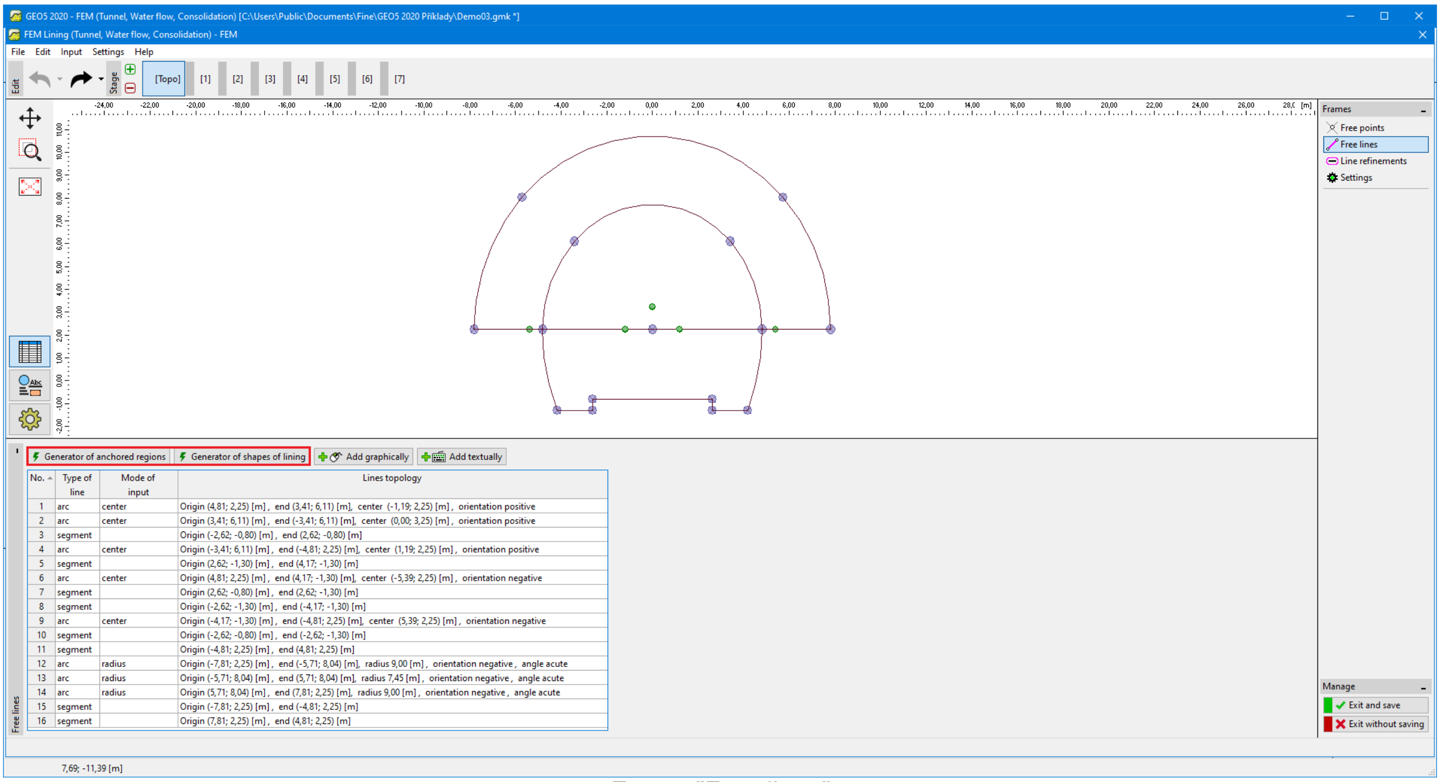


Frame "Free points"

Free Lines

The frame "**Free lines**" contains a table with the list of input free points. Working with free lines follows the same guidelines as in the FEM program - frame "Free lines".

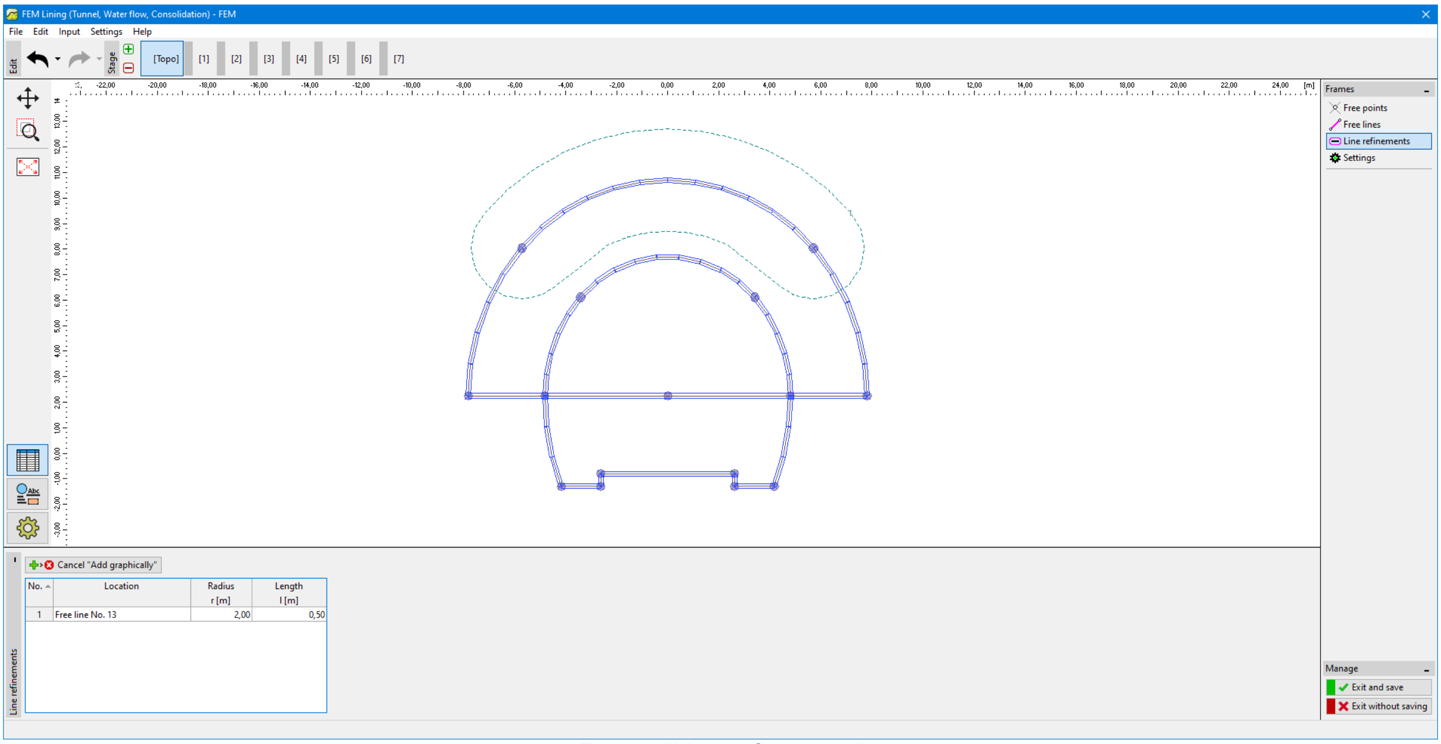
The frame differs by the functions on the horizontal toolbar, which contains the "Generator of shape of lining" and "Generator of anchored regions" buttons.



Frame "Free lines"

Line Refinement

The frame "**Line refinement**" contains a table with the list of input point refinements. Working with free lines refinement follows the same guidelines as in the FEM program - frame "Line refinement".



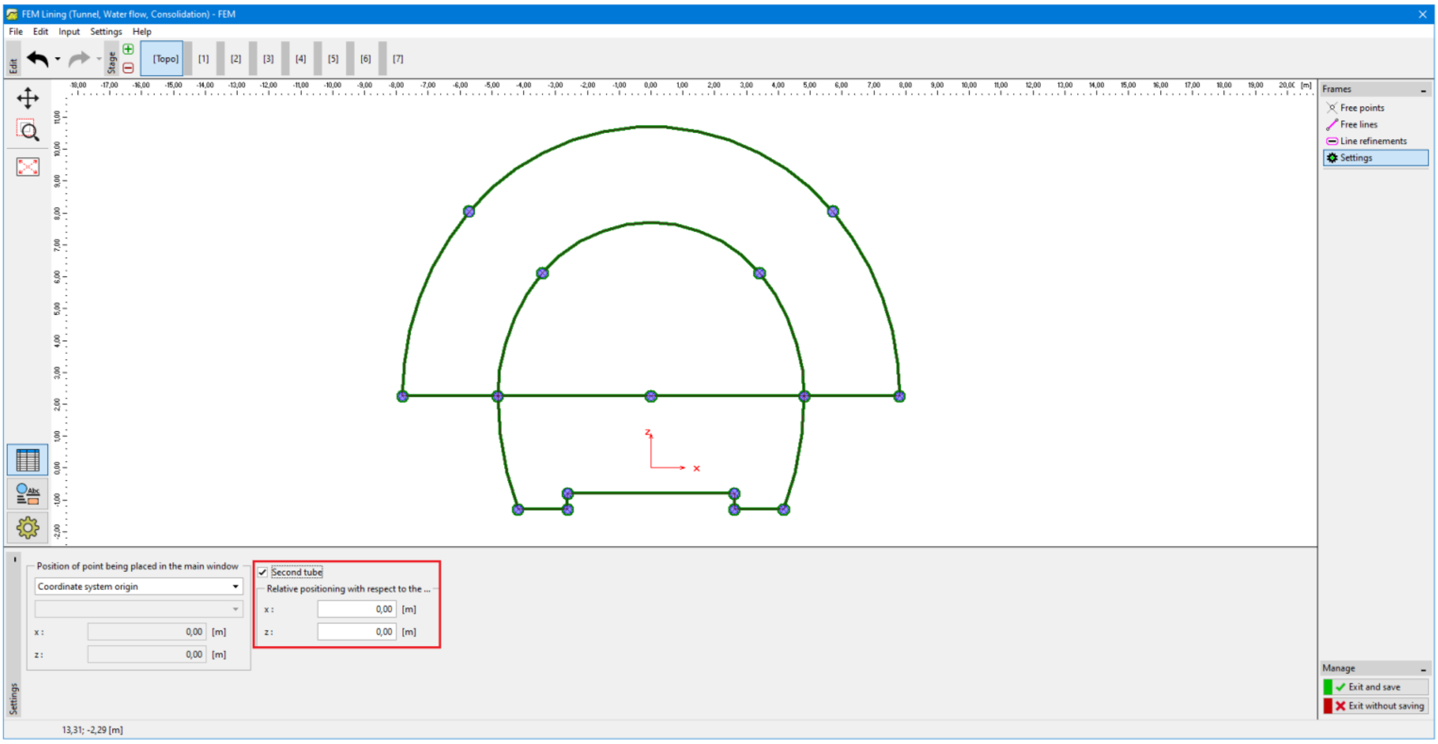
No.	Location	Radius r [m]	Length l [m]
1	Free line No. 13	2,00	0,50

Frame "Line refinement"

Settings

The frame "**Settings**" allows for redefining the location of a point to be subsequently positioned in the main window of the FEM program. The point location can be associated with the selected free point or determined by the coordinate system origin or by an arbitrary coordinate. This way allows for the exact positioning of a given point of the lining structure in the main window of the FEM program.

The use of a second tube can be activated in the right part of the frame. The second tube will appear in the frame "**Settings**" as a preview, and then after transmitting it into the FEM program. The second tube is a clone of the first one. It differs only in the positioning with respect to the originally defined structure.

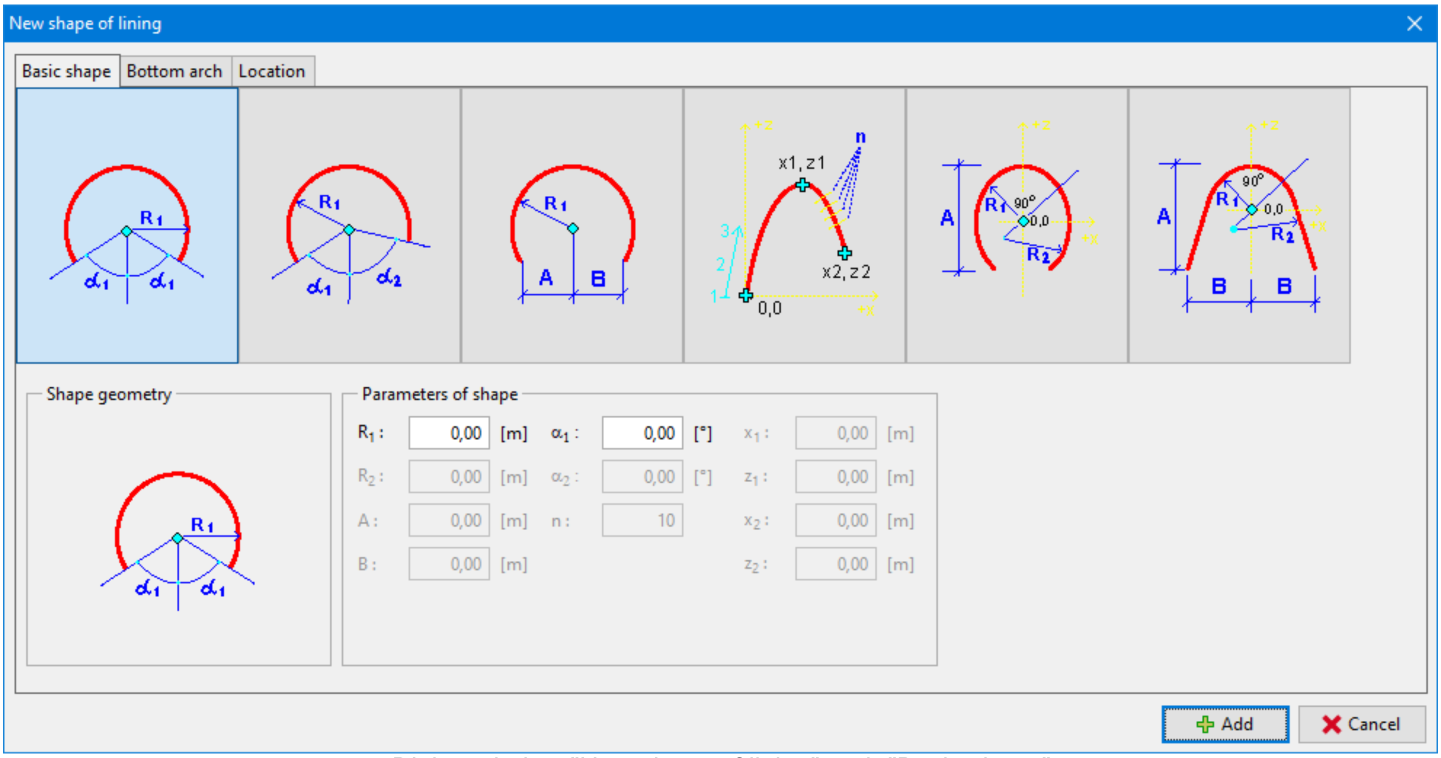


Frame "Settings"

Generator of Lining Shape

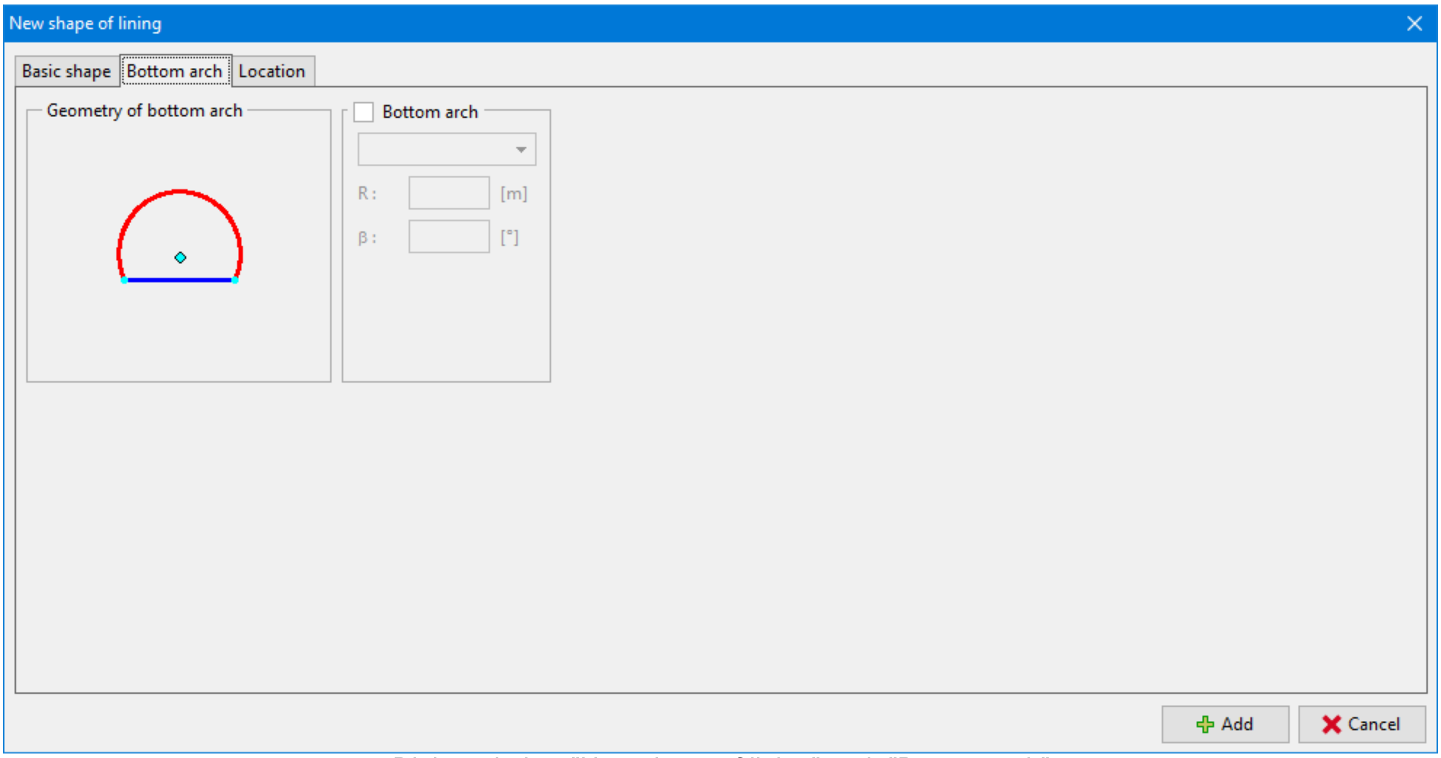
Depending on particular parameters the generator creates corresponding elements that are then operated on independently with no possibility for being parametrically modification. If the parameters of generation are acceptable, the program displays during their modification the current graphical representation of generated elements.

Six basic shapes of linings are available for generating free points and free lines in the "**New shape of lining**" dialog window. Each shape is defined by several parameters (radii, angles, height, spacing, subdivision number, control points).



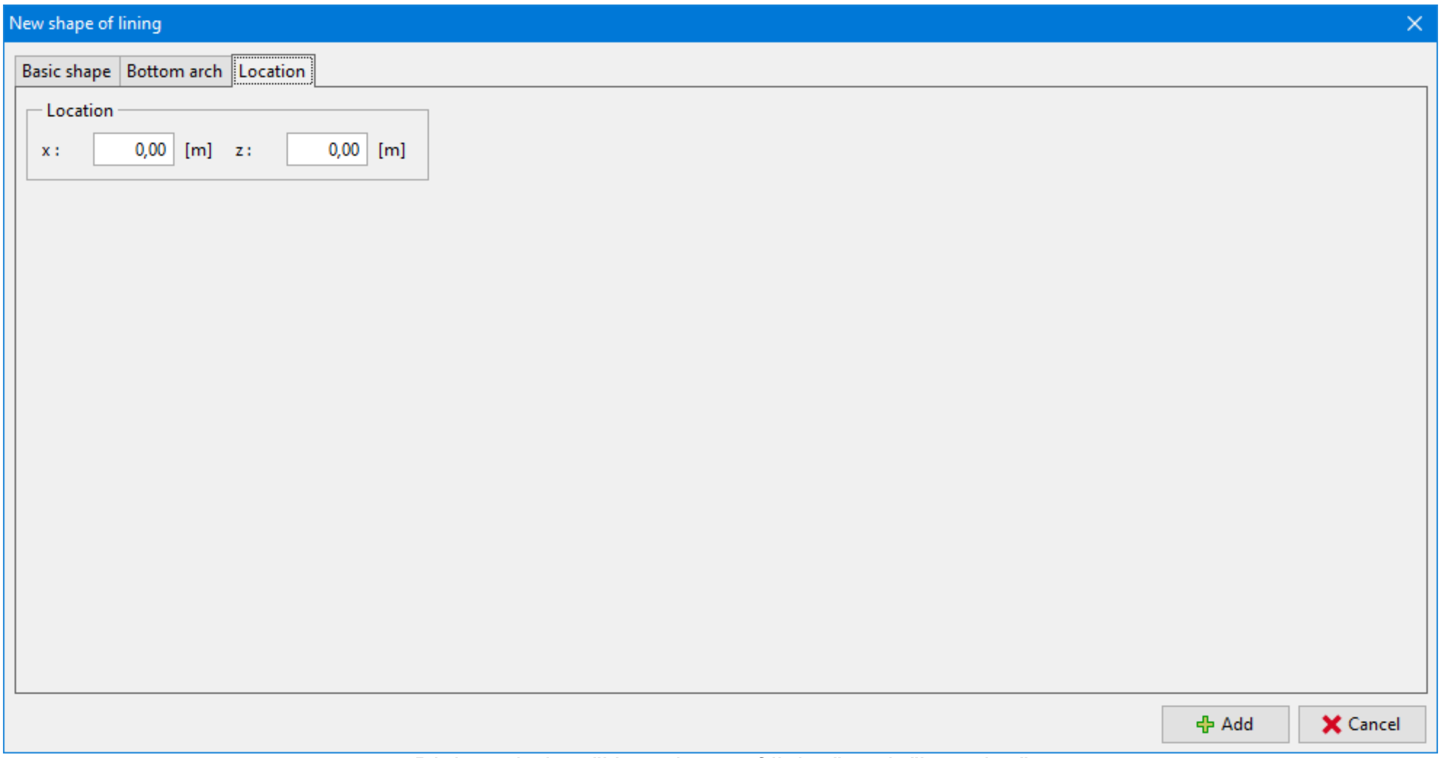
Dialog window "New shape of lining" - tab "Basic shape"

The "**Bottom arch**" tab sheep allows us to choose, whether the lining invert will be flat of arched, determined parametrically either by a radius or an angle.



Dialog window "New shape of lining" - tab "Bottom arch"

The "**Location**" tab allows, using coordinates, for changing the lining location.

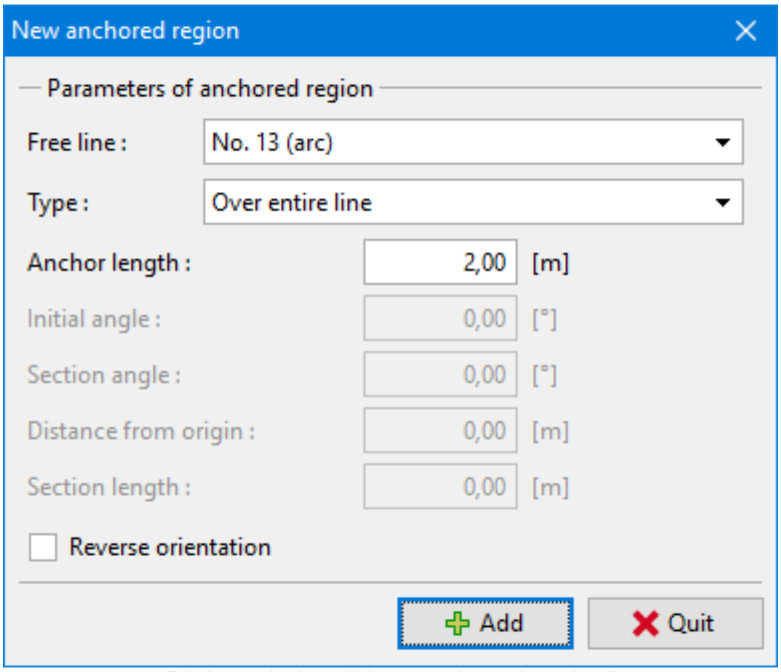


Dialog window "New shape of lining" - tab "Location"

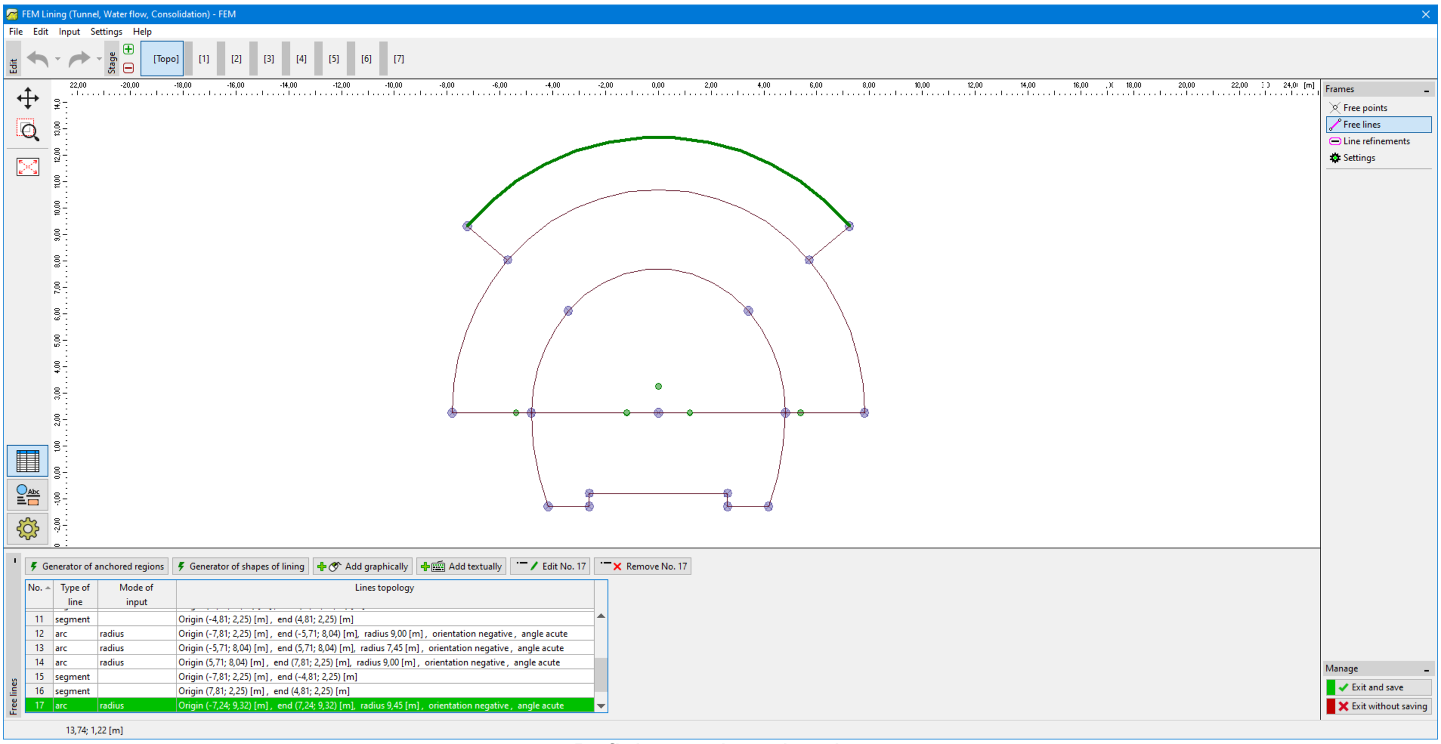
Generator of Anchored Regions

Depending on particular parameters the generator creates corresponding elements that are then operated on independently with no possibility for being parametrically modification. If the parameters of generation are acceptable, the program displays during their modification the current graphical representation of generated elements.

The "**New anchored region**" dialog window serves to generate free points and free lines based, however, on already input lines. This generates a closed region, which is then assigned in the FEM program a special soil characterizing a densely anchored region. The dialog window requires specifying a line number and parameters based on the type anchoring system (over the entire line, angle sector, origin, and length).



Dialog window "New anchored region"



Defining anchored region

Stages of Construction

Stages of construction in the "**Lining - FEM**" module and in the FEM program correspond to each other. They, however, may vary in several features.

Different behavior of stages in the "**Lining - FEM**" module:

- Possible to switch to stages of construction from the "**Topology**" regime without generating the FE mesh.
- Stages of construction added in the "**Lining - FEM**" module are, after confirming, transferred also into the FEM program.
- Stages of construction, preceding the stage from which the "**Lining - FEM**" module was launched, cannot be used.
- Stages of construction defined prior to launching the "**Lining - FEM**" module cannot be deleted.

Free Points

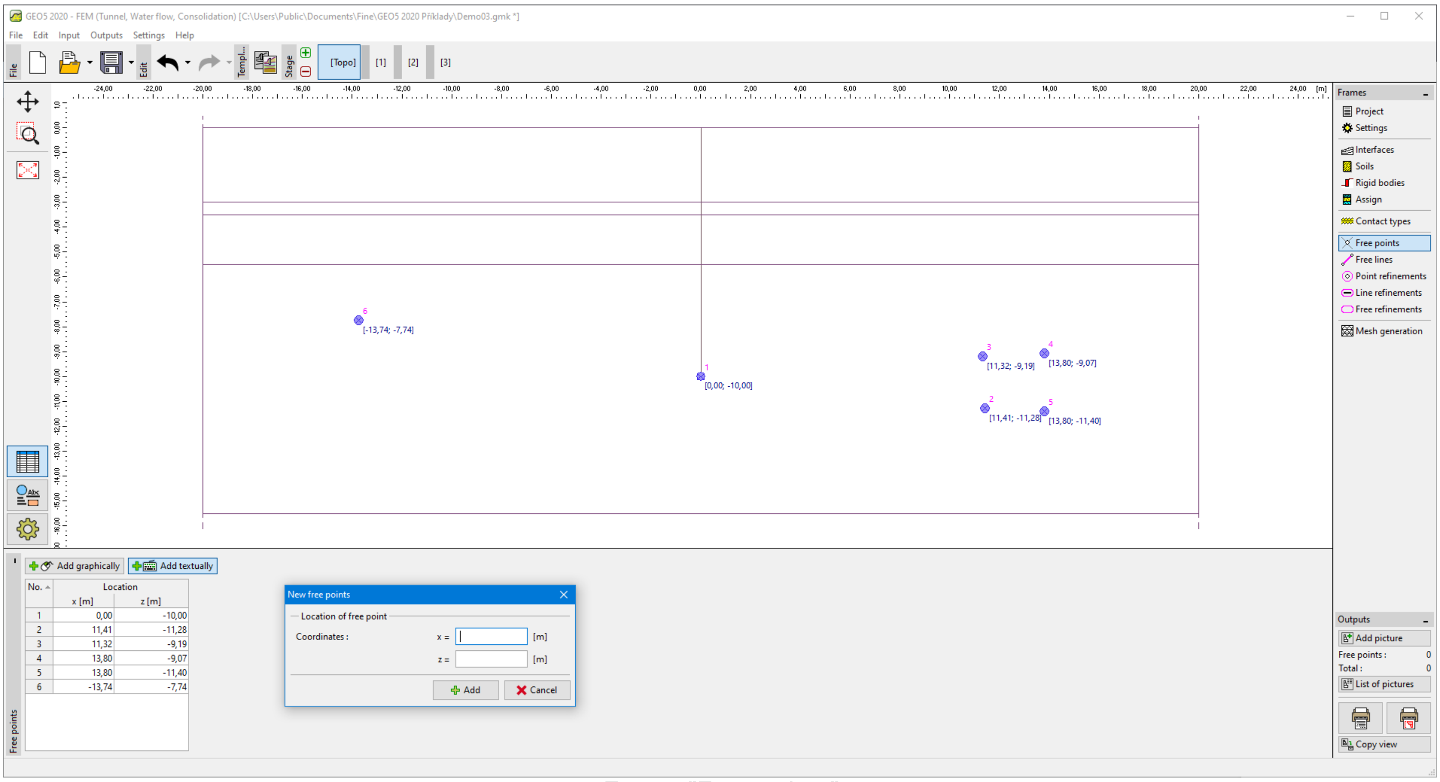
The frame "**Free points**" contains a table with the list of input free points. Adding free points is performed in the "**New free point**" dialog window.

The free points can also be edited on the desktop with the help of active objects.

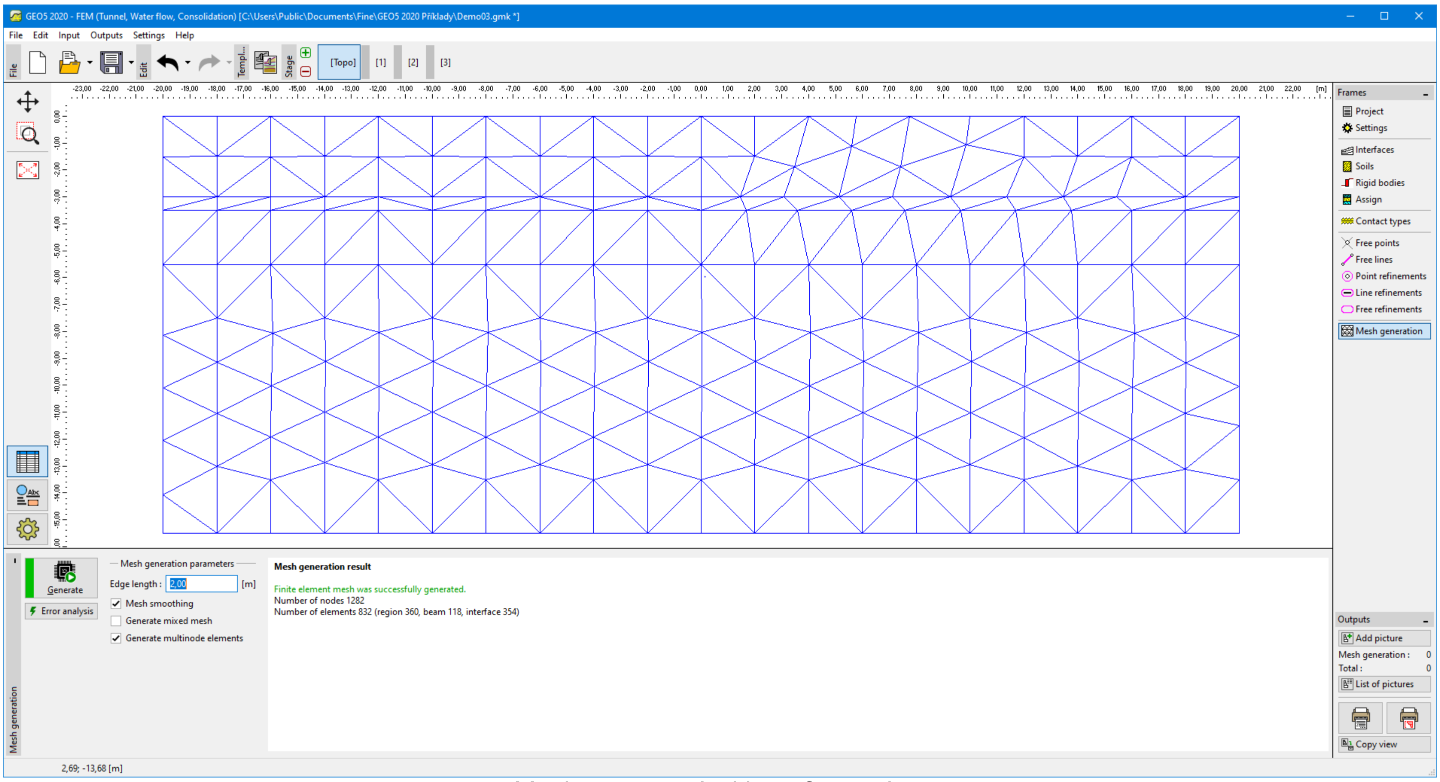
The program allows for inputting an arbitrary number of free nodes anywhere inside or outside the structure. Free nodes have several main functions:

- **nodes to define the structure** (tunnel opening, lining, sheeting, beams)
- **auxiliary points** for the mesh refinement
- points to **define a boundary condition**, to input forces, etc.

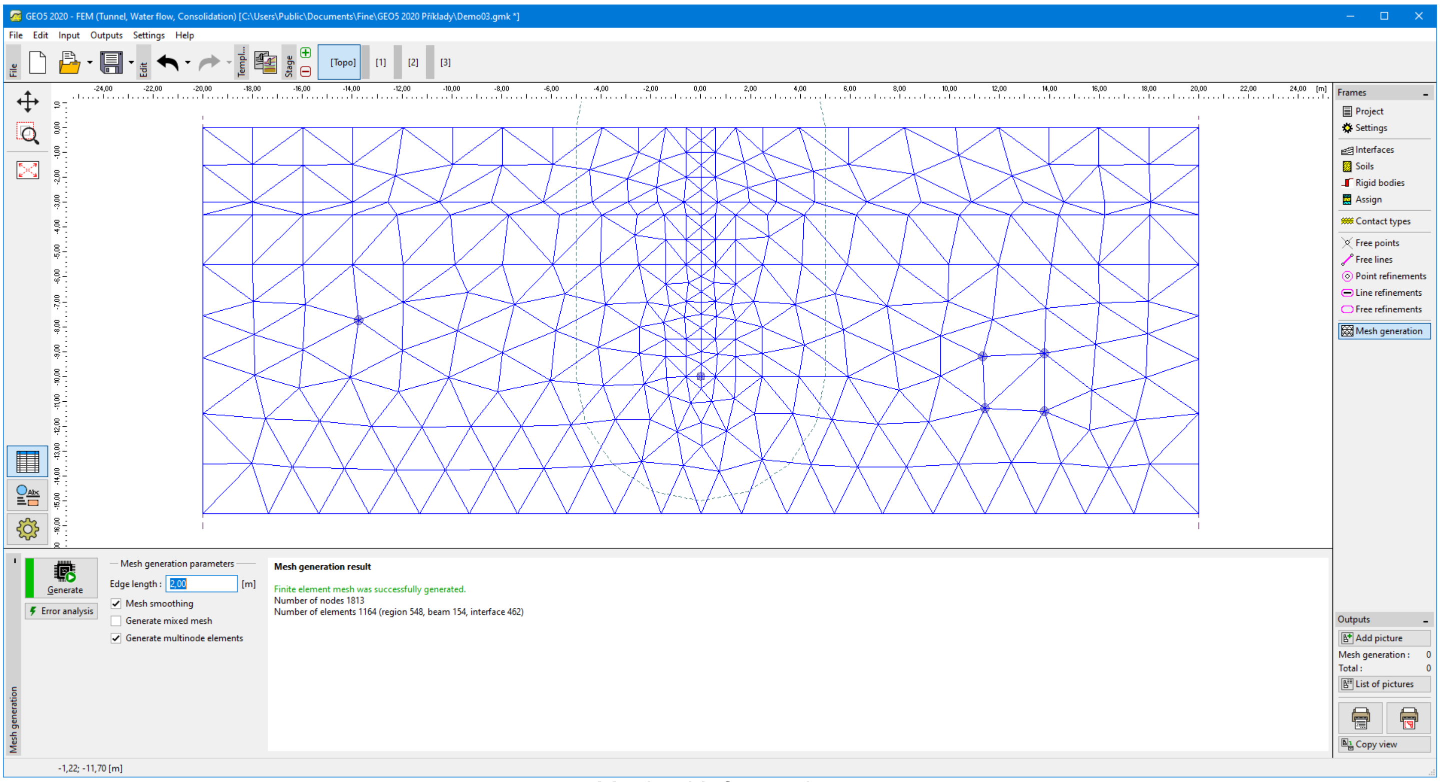
If a free node is found inside or on the boundary of a structure, it becomes **automatically a part of the finite element mesh**. This option allows an adjustment of the finite element mesh.



Frame "Free points"



Mesh generated without free points



Mesh with free points

Free Lines

The frame "**Free lines**" contains a table with the list of input free points. Adding free points is performed in the "**New free line**" dialog window.

The lines are defined **between individual points** (segments, arcs, circles) or around individual points (circles). The lines can be introduced both between free points and between points located on interfaces including the terrain surface.

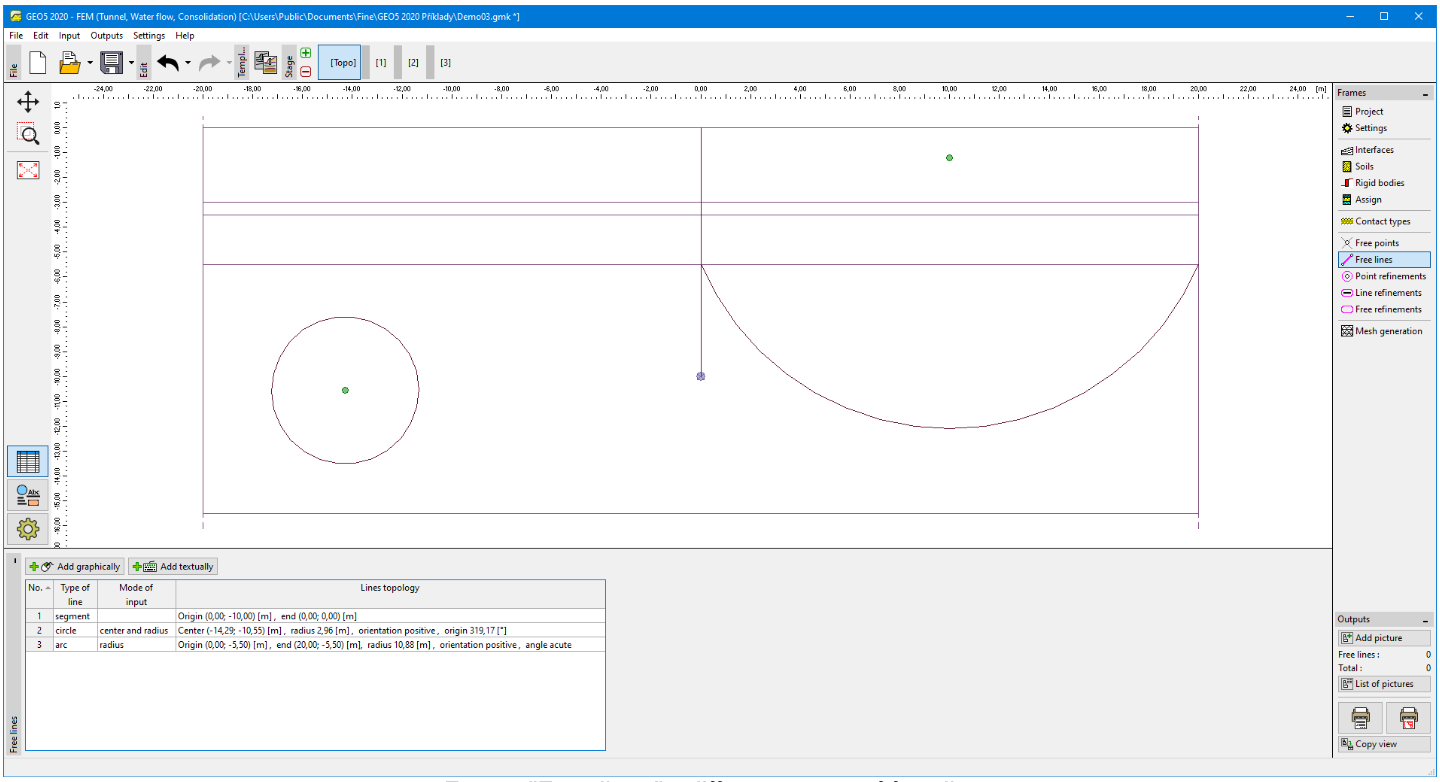
The lines may **intersect each other and may have an arbitrary number of contact points** - intersections of individual lines are determined by the program when adjusting the geometrical model. The free lines may be used to **introduce beam elements** into the model.

The type of line is selected from the combo box. The following modes are available:

- **Line type** A combo list is used to select the desired line (segment, arc, and circle).

  - **segment** Clicking individual points on the desktop with the left mouse button creates a point to point line
  - **arc** Use the combo list to choose a particular mode of defining an arc segment (third point, center, radius, height). Clicking the left mouse bottom on the desktop then selects points to define an arc. When selecting one of the following options - center, radius, or, you are further requested to select from the combo list the orientation (positive, negative).
  - **circle** Use the combo list to choose a particular mode of defining a circle (center and radius, three points). Clicking the left mouse button on the desktop then selects points to define a circle. The combo list is also used to select the orientation (positive, negative).

The free lines can also be edited on the desktop with the help of active objects.



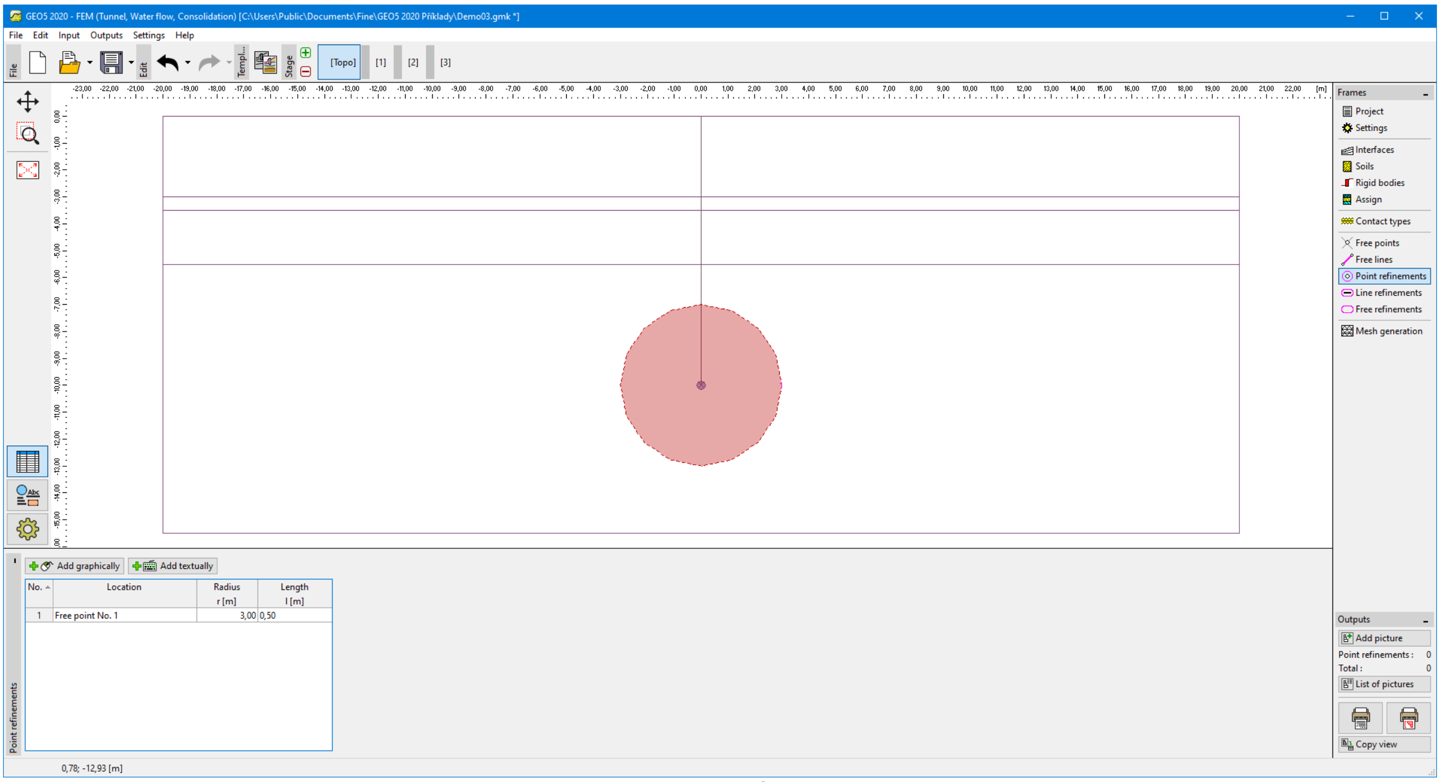
Frame "Free lines" - different types of free lines

Point Refinement

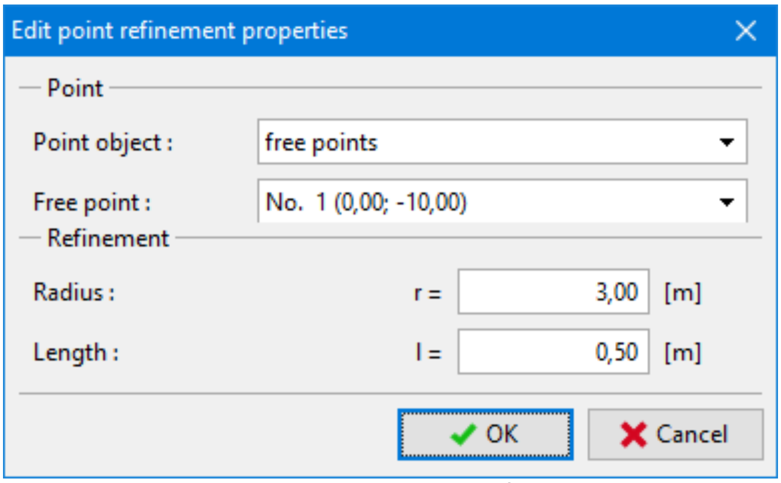
The frame **"Point refinement"** contains a table with the list of input point refinements. Adding a point refinement is performed in the **"New point refinement"** dialog window.

Refining the finite element mesh around points is an important feature, which allows us to create an appropriate finite element mesh. Both free points and points pertinent to individual interfaces including terrain can be used to refine the original finite element mesh.

The point refinement can also be edited on the desktop with the help of active objects.



Frame "Point refinement"



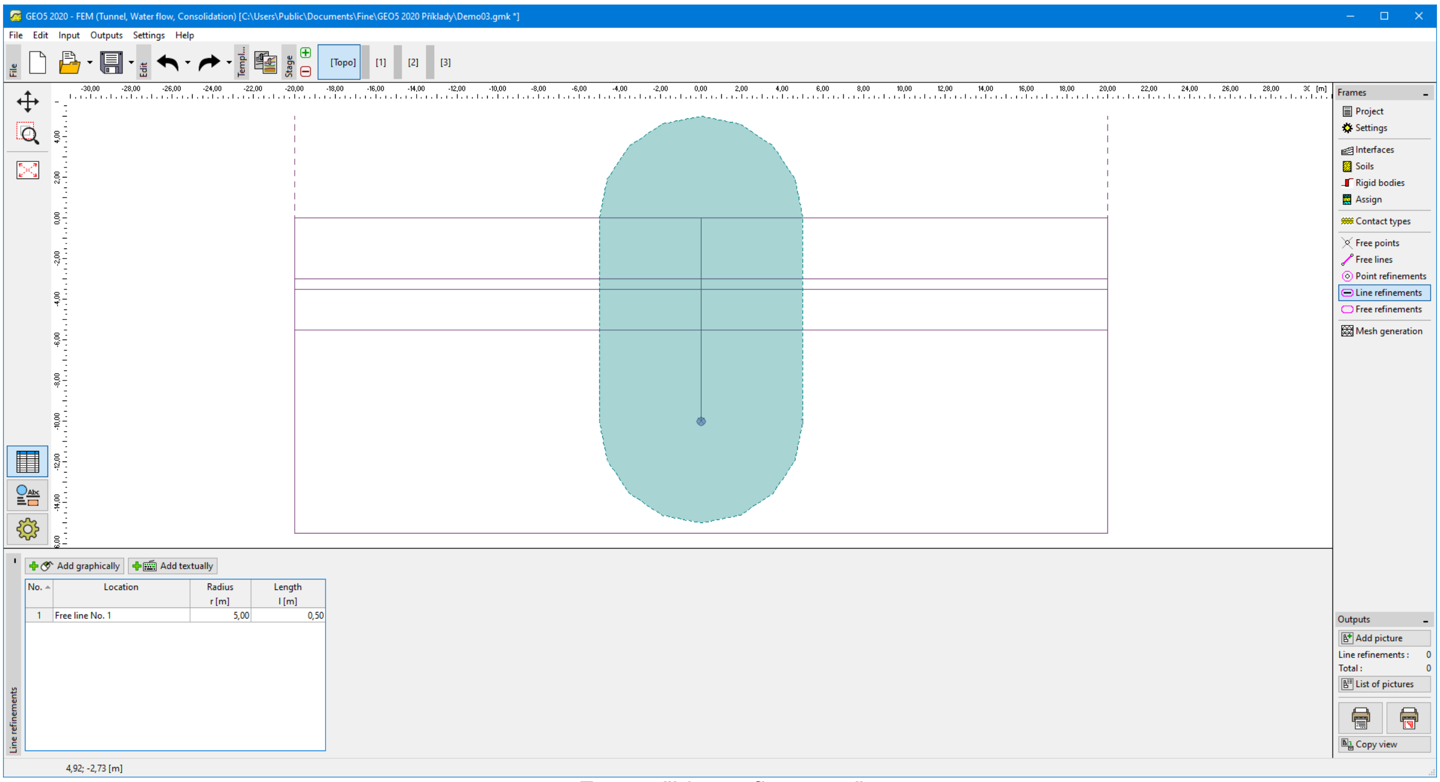
Dialog window "New point refinement"

Line Refinement

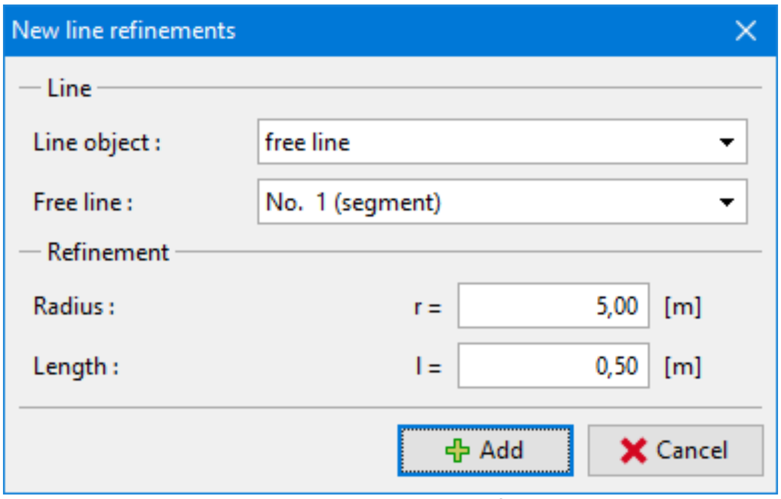
The frame "**Line refinement**" contains a table with the list of input point refinements. Adding a line refinement is performed in the "**New line refinement**" dialog window.

Refining the finite element mesh around lines is an important feature, which allows us to create an appropriate finite element mesh. Both free lines and lines pertinent to individual interfaces including terrain can be used to refine the original finite element mesh.

The line refinement can also be edited on the desktop with the help of active objects.



Frame "Line refinement"



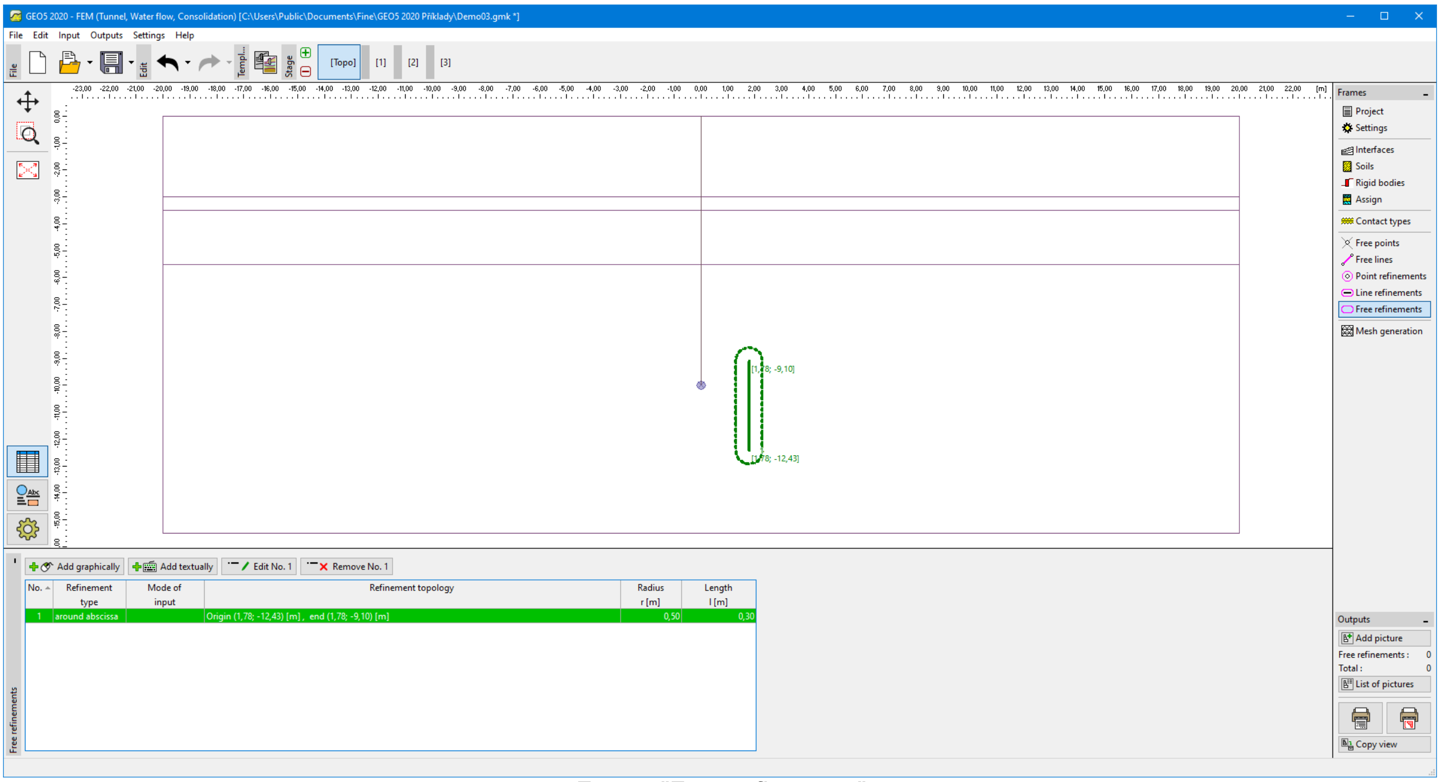
Dialog window "New line refinement"

Free Refinements

The frame "**Free refinements**" contains a table with the list of input free refinements. Adding a free refinements is performed in the "**New free refinements**" dialog window.

Refining the finite element mesh around lines is an important feature, which allows us to create an appropriate finite element mesh.

The free refinements can also be edited on the desktop with the help of active objects.



Frame "Free refinement"

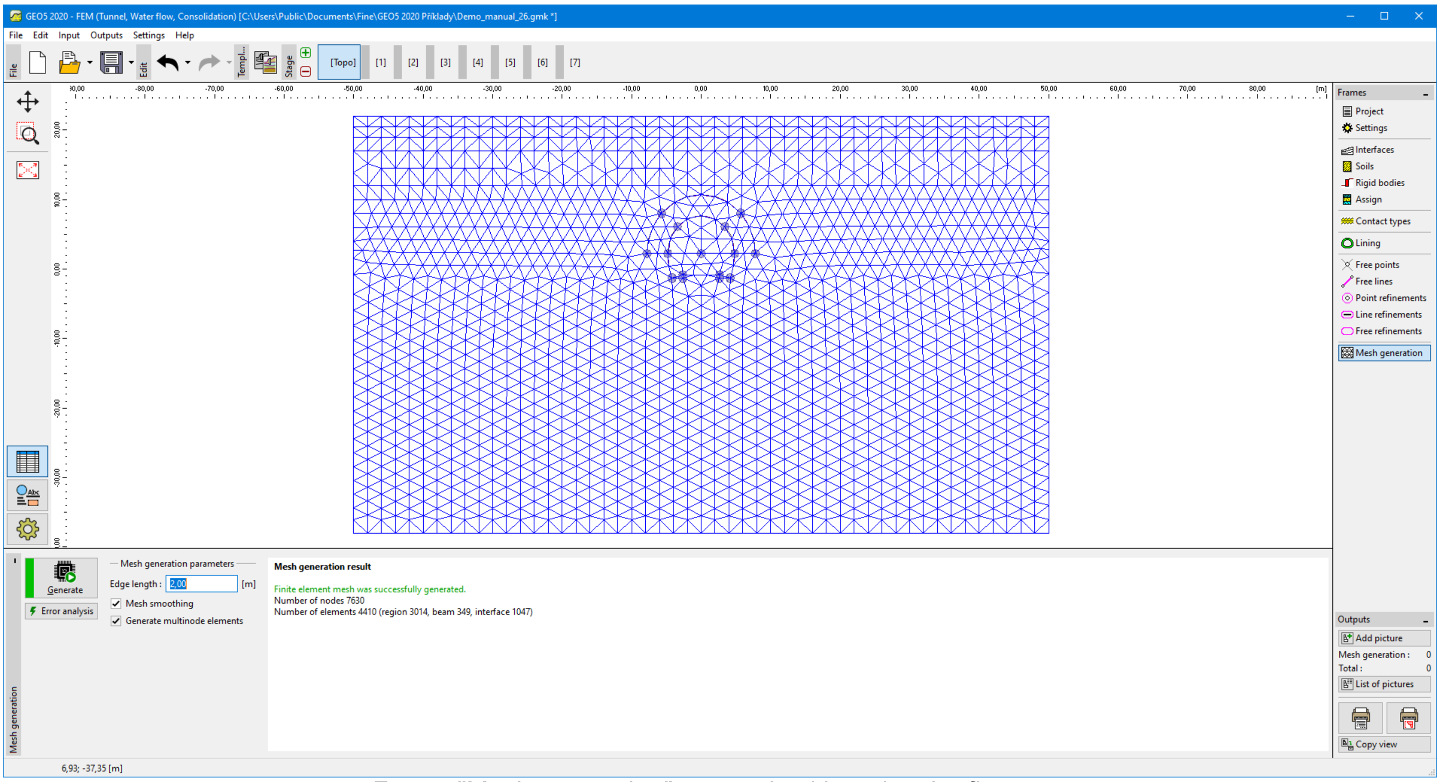
Mesh Generation

The frame "**Mesh generator**" serves to define the basic setting to generate mesh (left part) and to view information about generated mesh (right part).

A successfully generated mesh completes the topology input stage - the analysis then proceeds with the calculation stages. When generating mesh the program automatically introduces standard boundary conditions. Information about the resulting mesh including warnings for possible weak points in the mesh is displayed in the right bottom window.

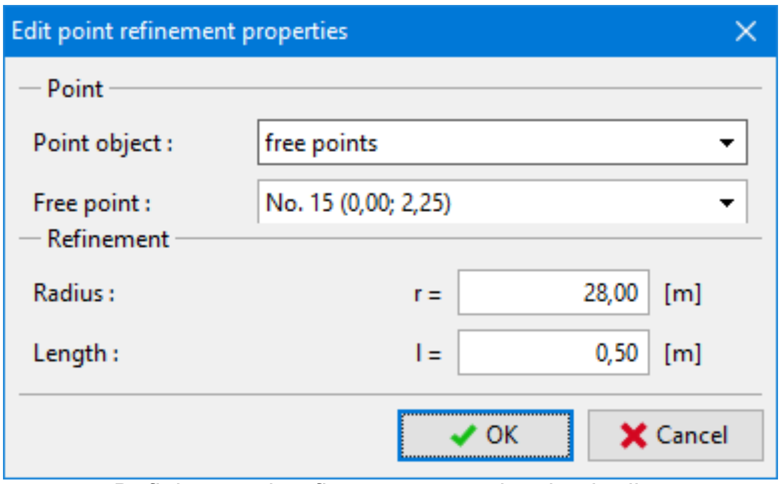
Correctly generated finite element mesh is the major step in achieving accurate and reliable results. The program FEM has an automatic mesh generator, which may substantially simplify this task. Nevertheless, **certain rules should be followed** when creating a finite element mesh:

- The basic mesh density can be specified in the "**Mesh generator**" dialog window. I is generally accepted that the finer the mesh the better the results - computation as well as post-processing, however, may slow down substantially. The goal thus becomes to find an optimum mesh density - this mainly depends on the user experiences. Meshes generated in example problems may serve as an initial hint.

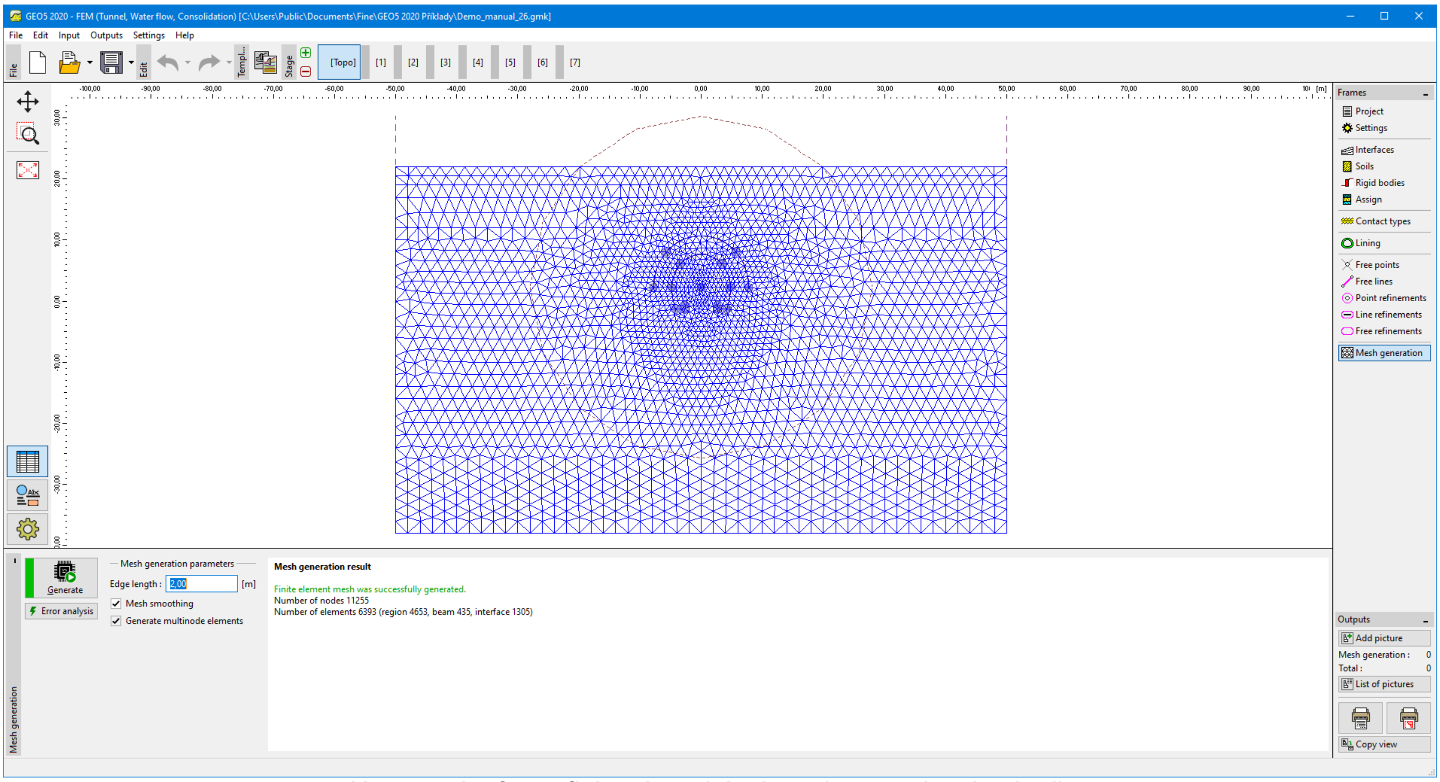


Frame "Mesh generation" - a mesh with no local refinement

- The finite element mesh should be sufficiently fine in locations in which large stress gradients are expected (point supports, corners, openings, etc.). To that end, it is possible to specify the mesh refinement in the neighborhood of these locations. The mesh refinement can be specified around individual points or lines. The spread of refinement should be at least *3-5* times the desired refinement in the center of the refinement. Also, both values (density and spread of refinement) should be reasonable in view of the prescribed mesh density that applies to the surrounding region. This assures a smooth transition between regions with different mesh densities. Singular lines should be tackled in the same way. For more complicated problems it is useful to first carry out the analysis with a rather coarse mesh and then after examining the results to refine the mesh accordingly.



Defining mesh refinement around a circular line



New mesh after refining the original mesh around a circular line

By default, program assumes ***6* -node triangular elements** with mesh smoothing. The accuracy of the results more or less corresponds to twice as fine mesh composed of 3-node triangular elements. The ***3*** -node elements are available only in the "Advanced program options" mode (checkbox "**6-node elements**") and serve merely for research and testing purposes. The stability analysis, however, can be performed with 6-node triangular elements only. In case of nonlinear analysis, these elements should be used exclusively.

The "Advanced program options" mode allows also for the generation of mixed mesh (triangular and quadrilateral elements).

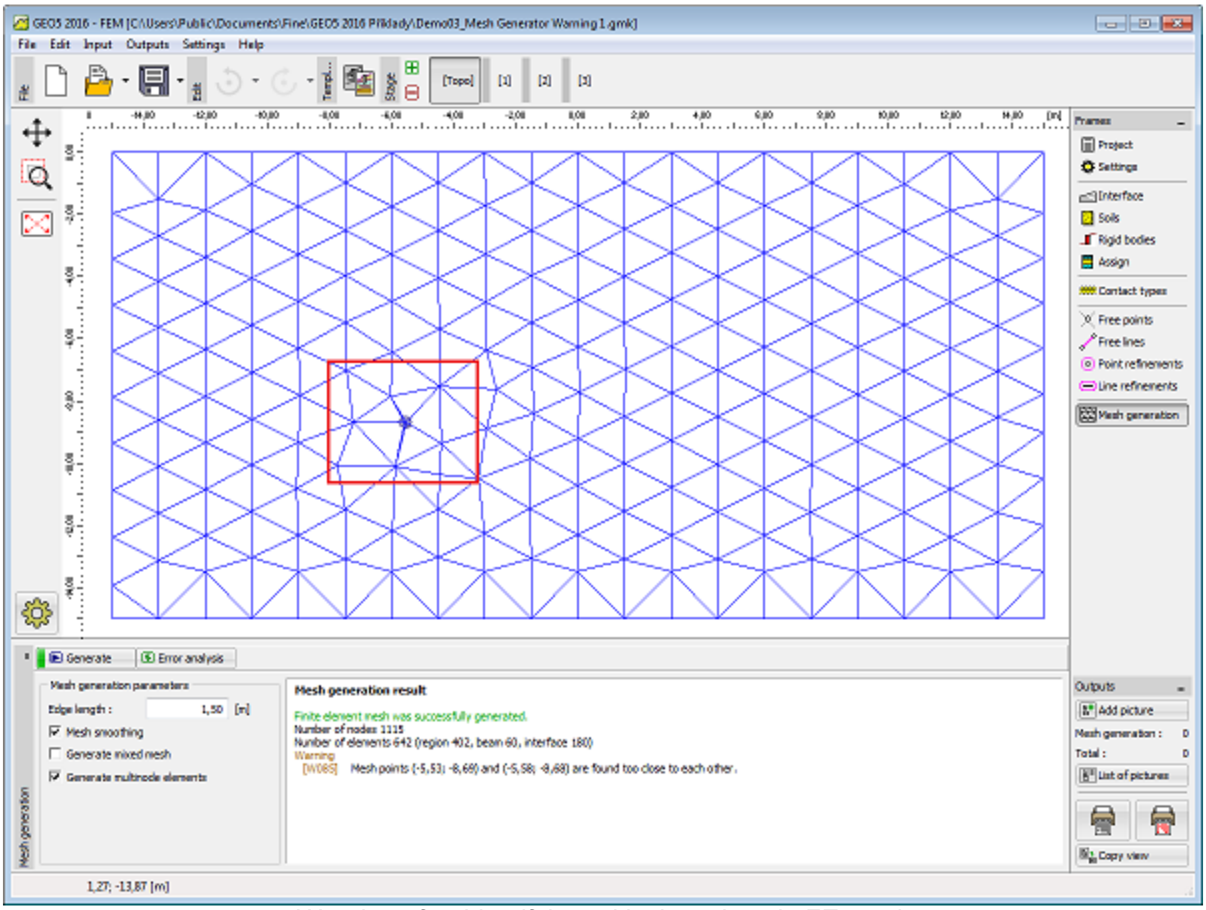
Mesh Generator Warning

In the "**Warning - structure critical locations**" dialog window the user is prompted for possible locations on the structure that may cause problems during automatic mesh generation. When positioning the cursor on individual warnings the corresponding critical region on a structure is highlighted with a red color. The following items are checked:

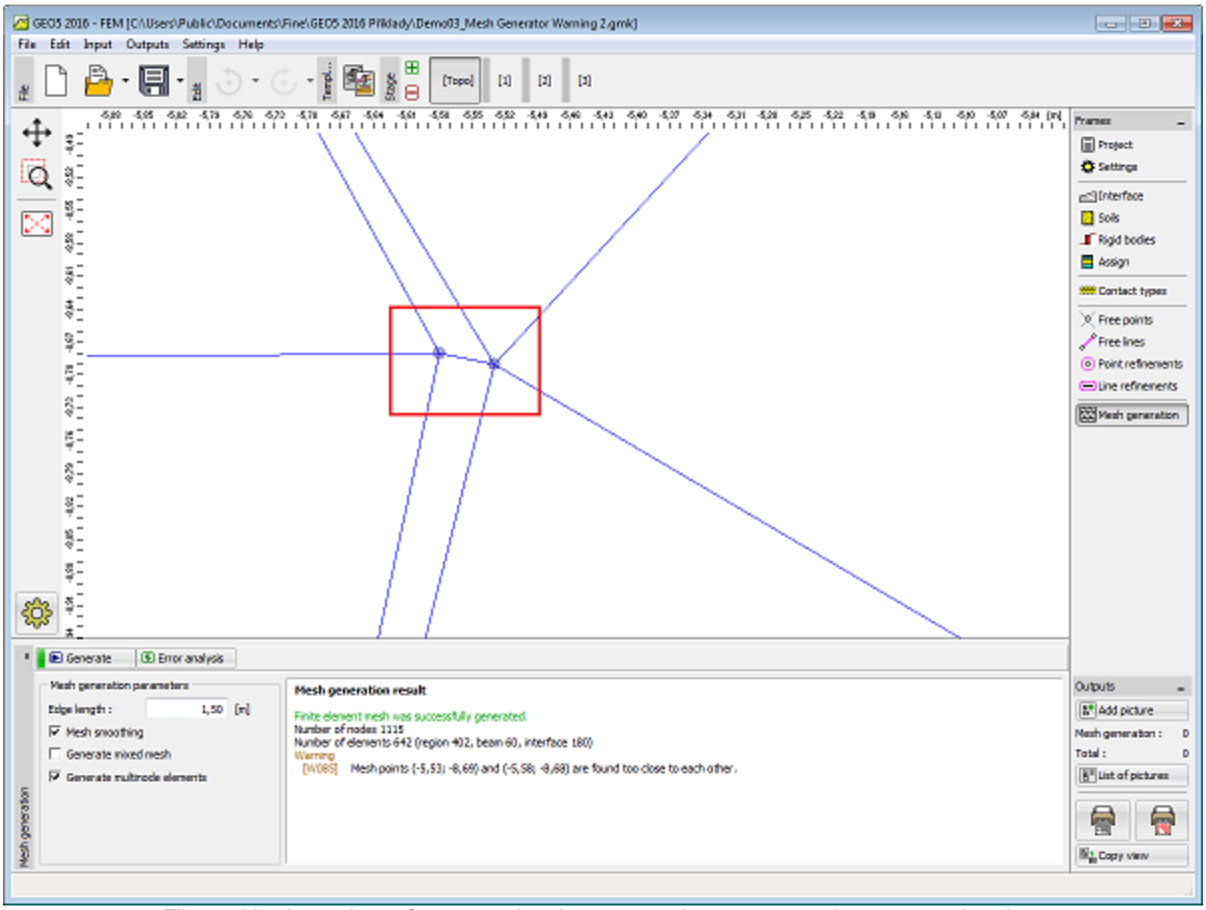
- whether the distance between two points is greater than one-tenth of the required element edge length,
- whether the distance between a point and a line is greater than one-tenth of the element edge length,
- whether the area of a region is greater than twice the element edge length,
- whether points and/or lines are found inside the structure (in the soil).

These warnings suggest locations, in which the mesh generator experience problems. The following possibilities may occur:

- the mesh is not generated => this calls for a new input of geometrical data,
- the mesh is generated => in this case it is up to the user to decide whether the mesh is reasonable - in any case, the warning can be further ignored and the analysis can be carried out.



Warning after identifying critical sections in FE mesh



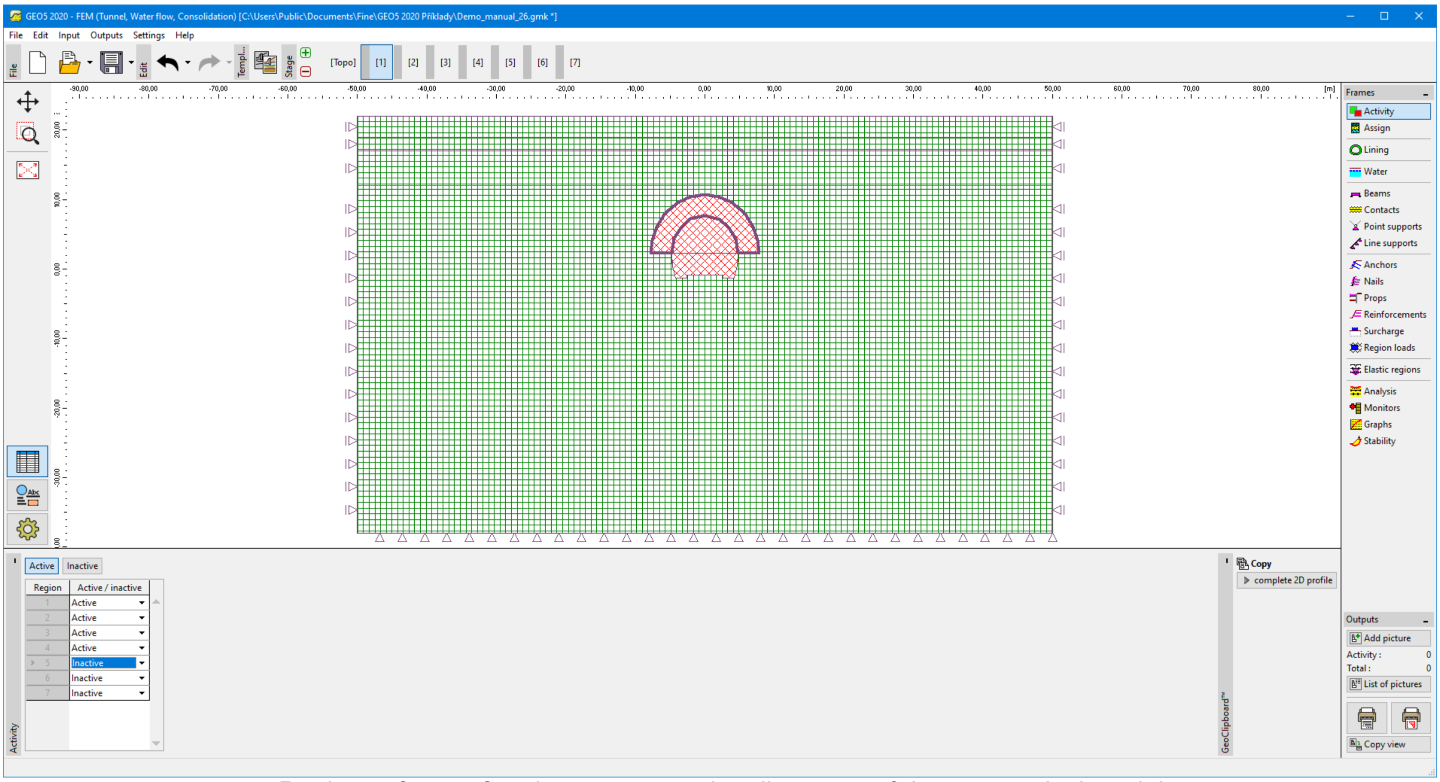
The critical section after zooming in - two points are too close to each other

Adjusting Original Geometry

The program contains a built-in **automatic corrector of the specified geometry**. This means that prior to the mesh generation the program automatically locates all points of intersection of lines, locates all closed regions, and creates a corresponding geometrical (calculation) model.

Such new regions can then be deactivated or they can be assigned a new soil. The main advantage of this system becomes evident when creating a geometrical model for tunnels (step by step excavation) or for sheeted structures. Creating even a very complicated model thus becomes rather simple and can be performed very efficiently.

Correcting the original geometrical model may cause some points in the model to be too close to each other or too small regions might be created. Warning message then appears in the right bottom window identifying such week points in the model.



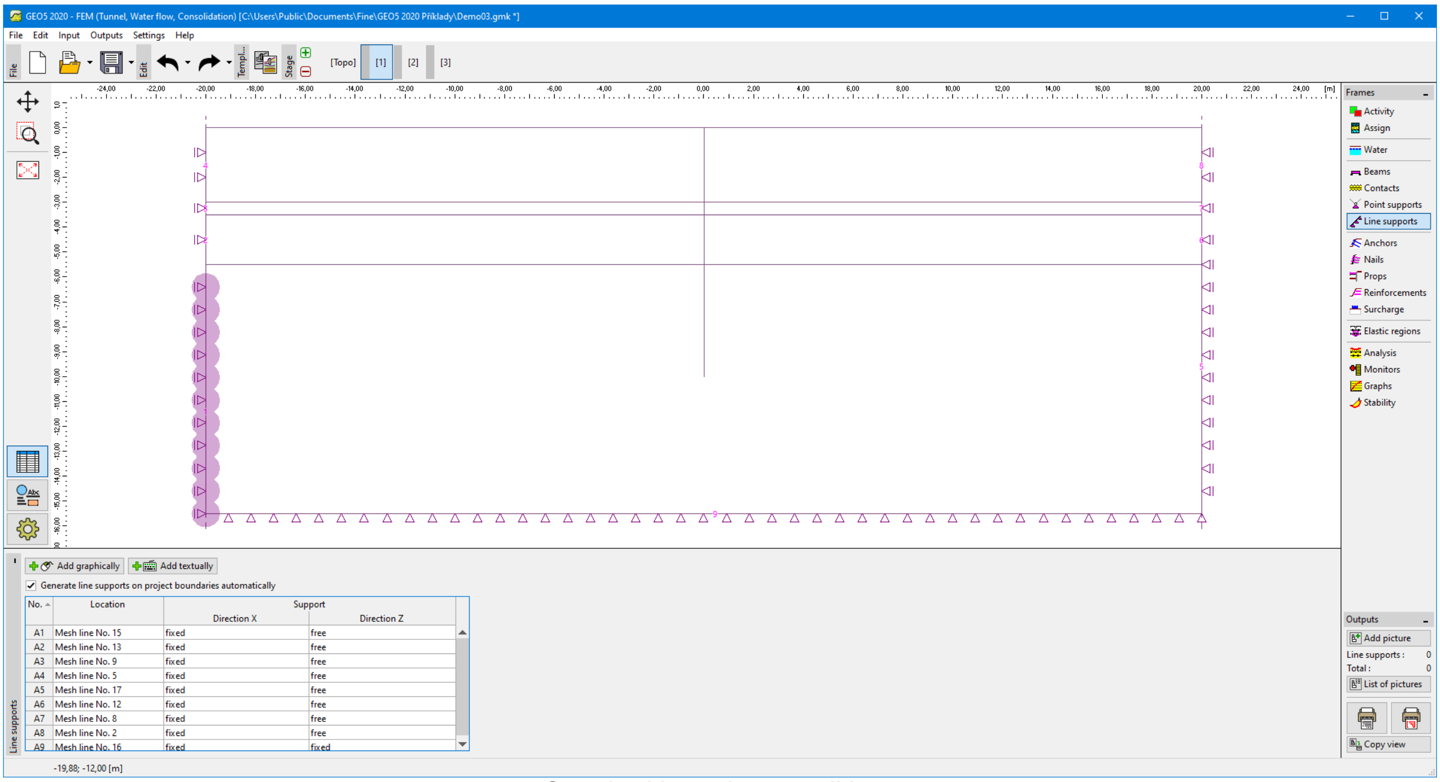
Regions after performing an automatic adjustment of the geometrical model

Standard Boundary Conditions

The program automatically generates standard boundary conditions. Therefore, **in routine problems the user does not have to enter the step of specifying supports.**

The standard boundary conditions are:

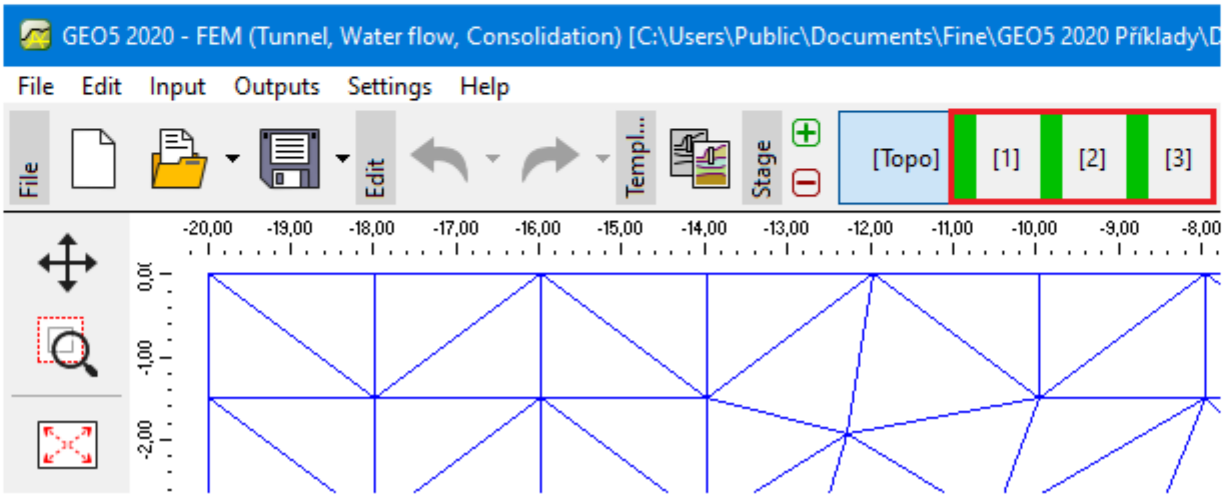
- smooth pin along the bottom edge of the geometrical model,
- sliding pin along the vertical edges of the geometrical model.



Standard boundary conditions

Construction Stages

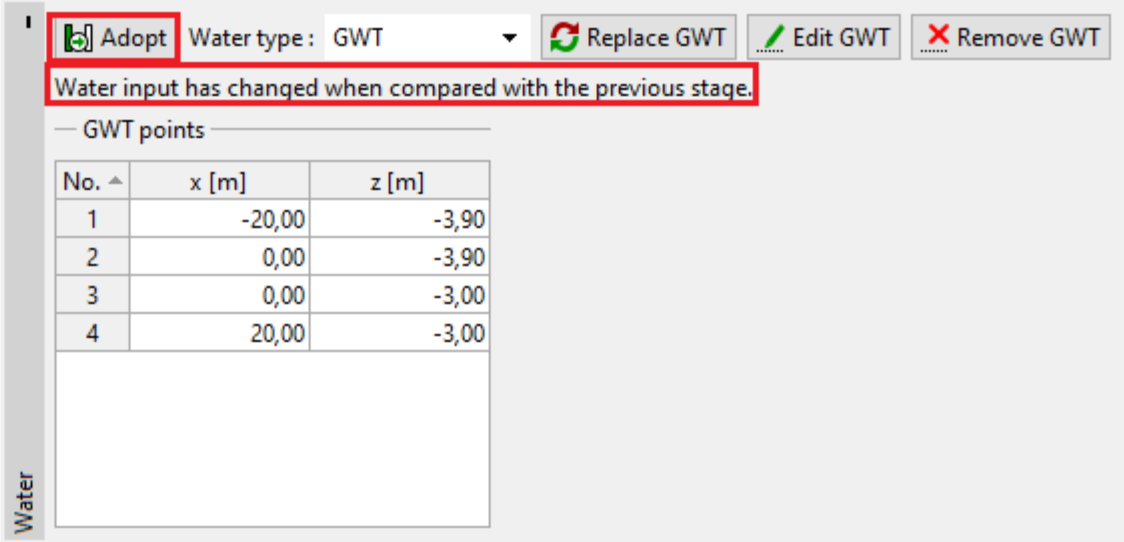
The actual analysis is performed in individual stages of construction (calculation stages) after the geometrical model and generating the finite element mesh (topology stage). One can move between calculation stages and the "**Topology**" regime using the buttons on the horizontal toolbar.



Toolbar "Construction stages" - switching between "Topology" regime and other stages of constructions

The calculation stages serve to model gradually build structures. Their correct definition and proper sequence is very important. The analysis of each stage builds (except for the stability analysis) upon the **results derived in the previous stage**. Information about individual objects and their properties is carried over from one stage to the other - when editing an existing stage or creating a new stage the program applies the principle of heredity.

Some frames ("**Water**", "**Activity**", "**Assign**") contain at the right part of the bar the "**Adopt**" button. The button becomes active once the data defined in the frame differ from those defined in the previous stage. After pressing this button the corresponding data ("**Water**", "**Activity**") is adopted from the previous stage.



Changing input data - accepting data from the previous stage of construction

The first stage of construction (**calculation of geostatic stress**) represents the initial state of the soil (rock) body before the onset of construction - displacements associated with this stage are therefore set equal to zero.

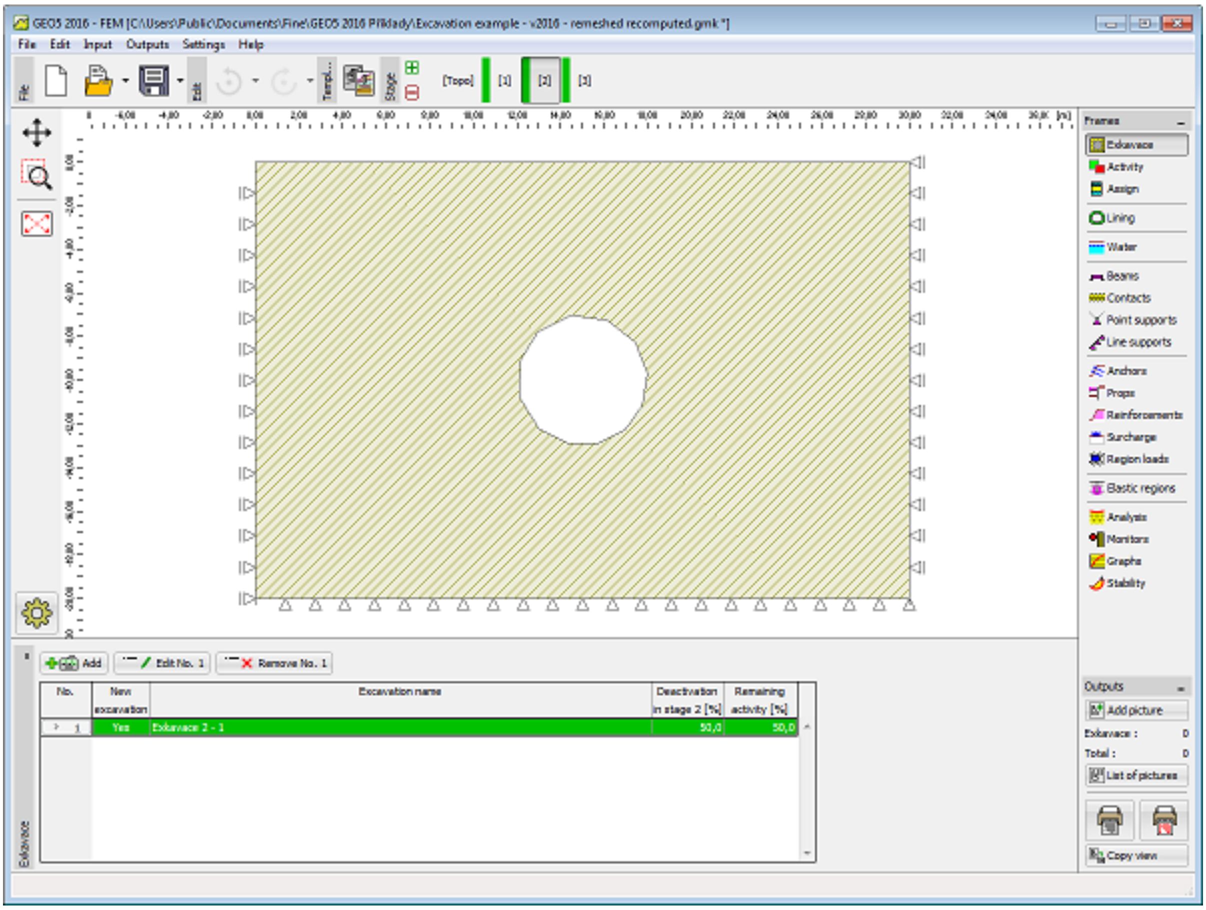
Loss of convergence may occur for a certain stage of construction. In this case (the results are not available for non-converged structure) the subsequent stages cannot be analyzed. To avoid modeling errors we recommend the user to follow the recommended way of the modeling and analysis of a structure.

Excavation

The frame "**Excavation**" contains a table with the list of excavations. Adding excavations is performed in the "**New excavations**" dialog window.

Excavation can only be defined in the second or the following construction stages.

The degree of deactivation [*%*] is specified in the dialog window "**New Excavation**".



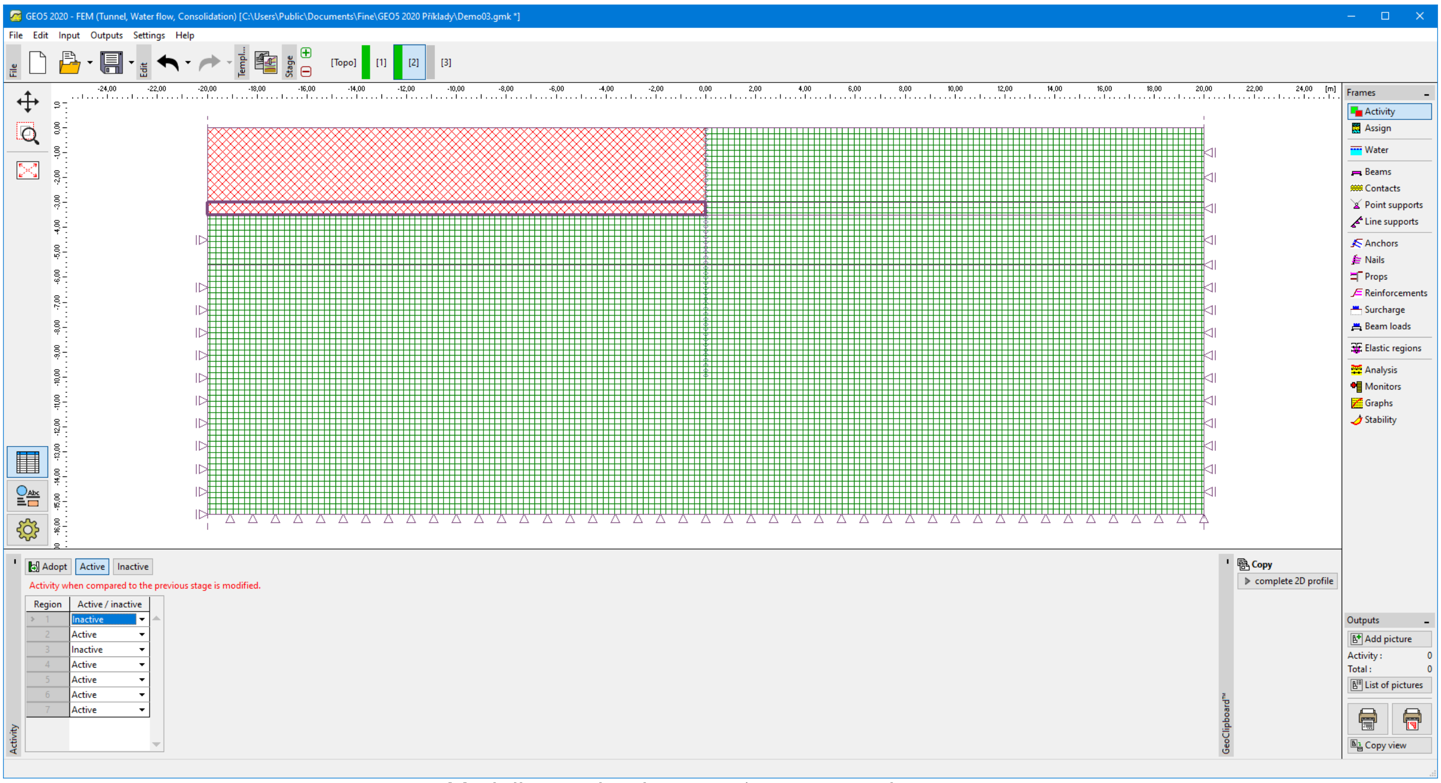
Frame "Excavation"

The excavation feature enables to partially deactivate of the forces by which the soil within the excavated profile supports the surrounding soil. This technique allows for modeling the successive excavation within a 2D (plane strain) analysis. The standard use case consists of partial deactivation of the excavated soil in one computational stage and installation of the primary lining (beam) and deactivation of the remaining supporting stress in the following stage.

In literature, this procedure is known as the Convergence Confinement Method.

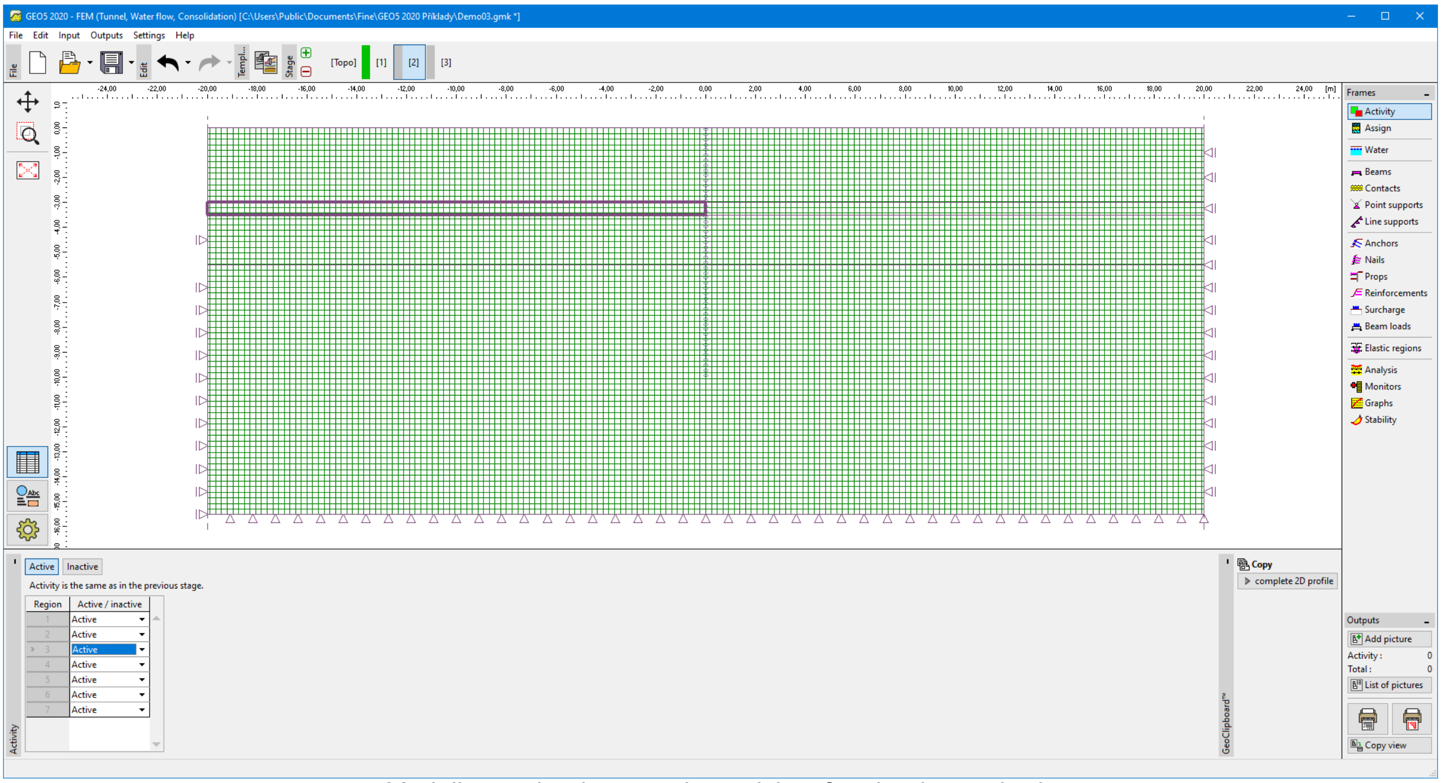
Activation

The program allows for **removing (deactivating) soils** from individual regions. As an example, we consider an embankment analysis. In such a case, it must be accounted for already in the topology regime when creating the overall geometrical model. In the *1st* calculation stage, however, it can be deactivated. A similar approach applies also to underground or open excavations (tunnels, sheeting structures). When deactivating a region below the groundwater it is necessary to correctly model the region boundary.



Modeling embankment - 1st construction stage

The embankment can be subsequently reactivated in the next construction stage.



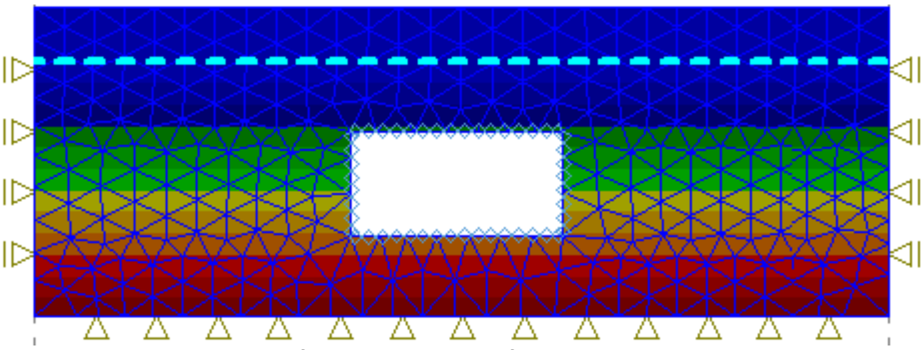
Modeling embankment - the activity of embankment body

Using GeoClipboard there's a possibility to copy the current profile as sorted interfaces and allow copying profile to another program. Copied interfaces are corrected to follow specifications to 2D profile entered from top to bottom.

Activity of Regions Below GWT

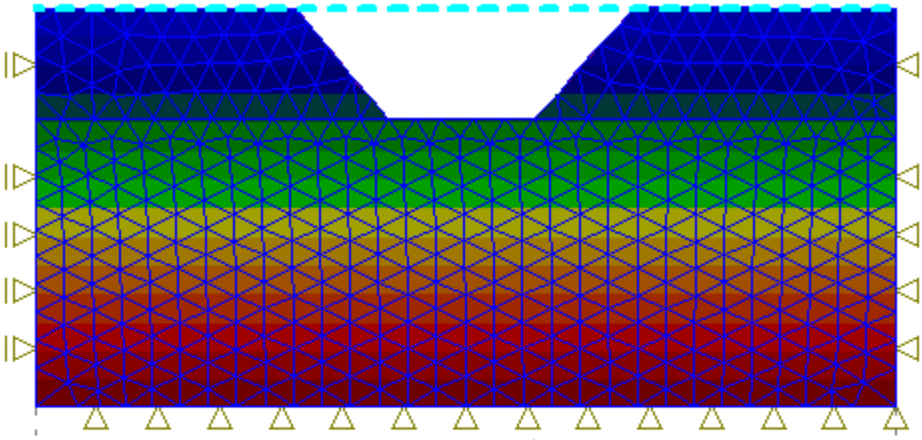
There are two cases to be considered when deactivating a region below the GWT.

1) The soil subjected to excavation **is completely enclosed by active beam elements**. The beam is then considered to be impermeable and both the soil and water are removed (removing total stresses - **the inactive region is free of water**). Owing to the impermeability of the beam elements, the pore pressure distribution remains unchanged, see the figure.



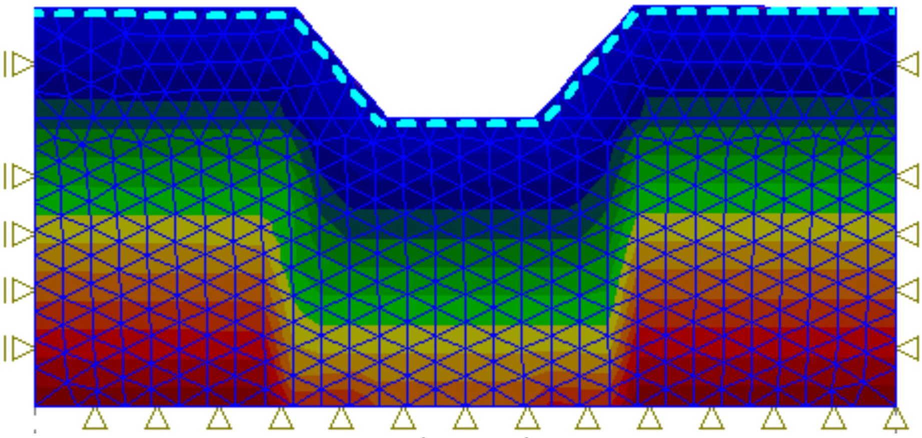
Pore pressure distribution after removing soil from region enclosed by active beams

2) The removed soil **is not enclosed by beam elements**. In such case, we assume that water in the excavated region **is still active**. This state is evident from the pore pressure distribution in the figure.



Pore pressure distribution after removing soil

Its effect can be removed by **changing the groundwater table**.



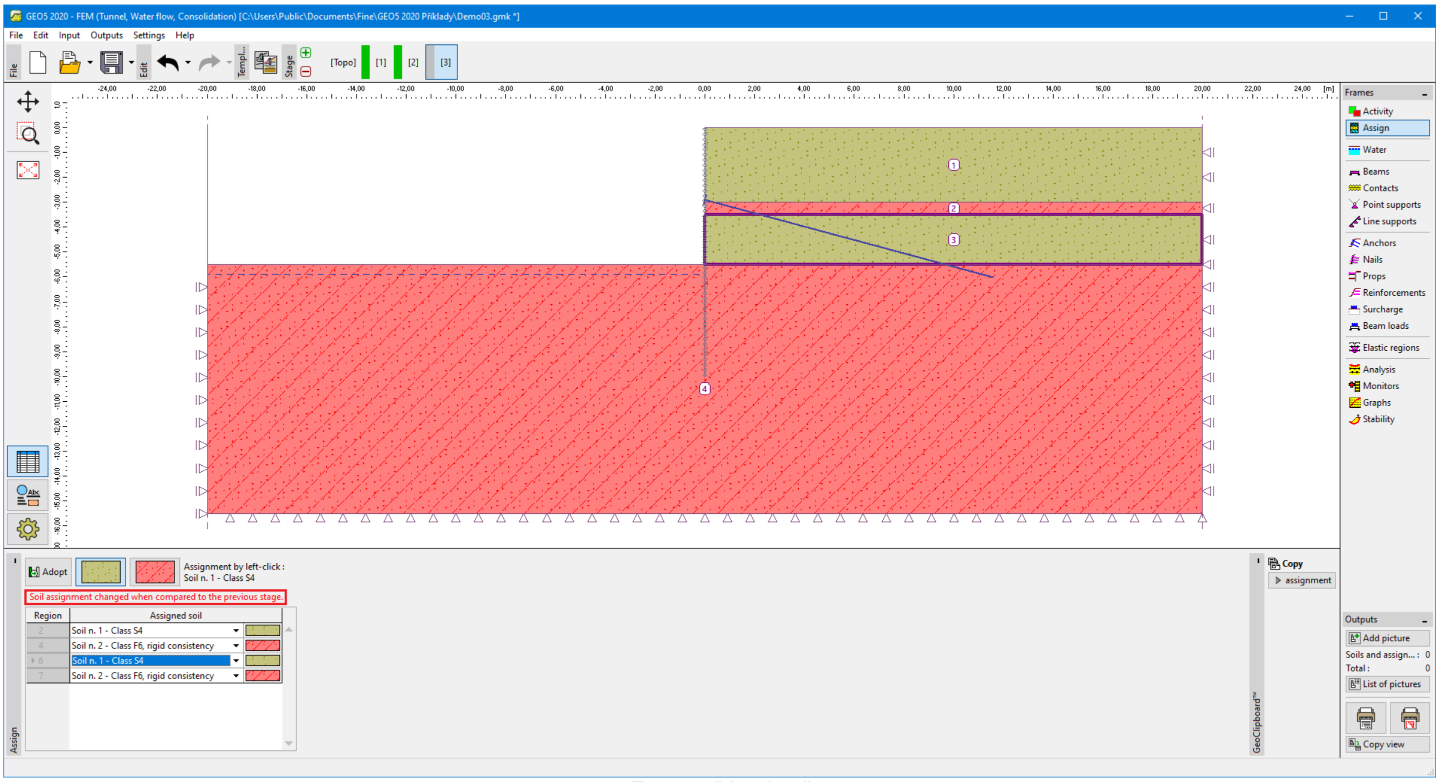
Pore pressure distribution after modifying the groundwater table

Assign

The frame "**Assign**" contains a list of layers of the profile and associated soils. Its functions are similar to the case of assigning soils in the topology regime.

In calculations stages, the active soils are assigned to regions rather than to interfaces. The regions are created automatically when creating the computational model.

Assign of soils can be copied within all 2D GEO5 programs using "GeoClipboard".

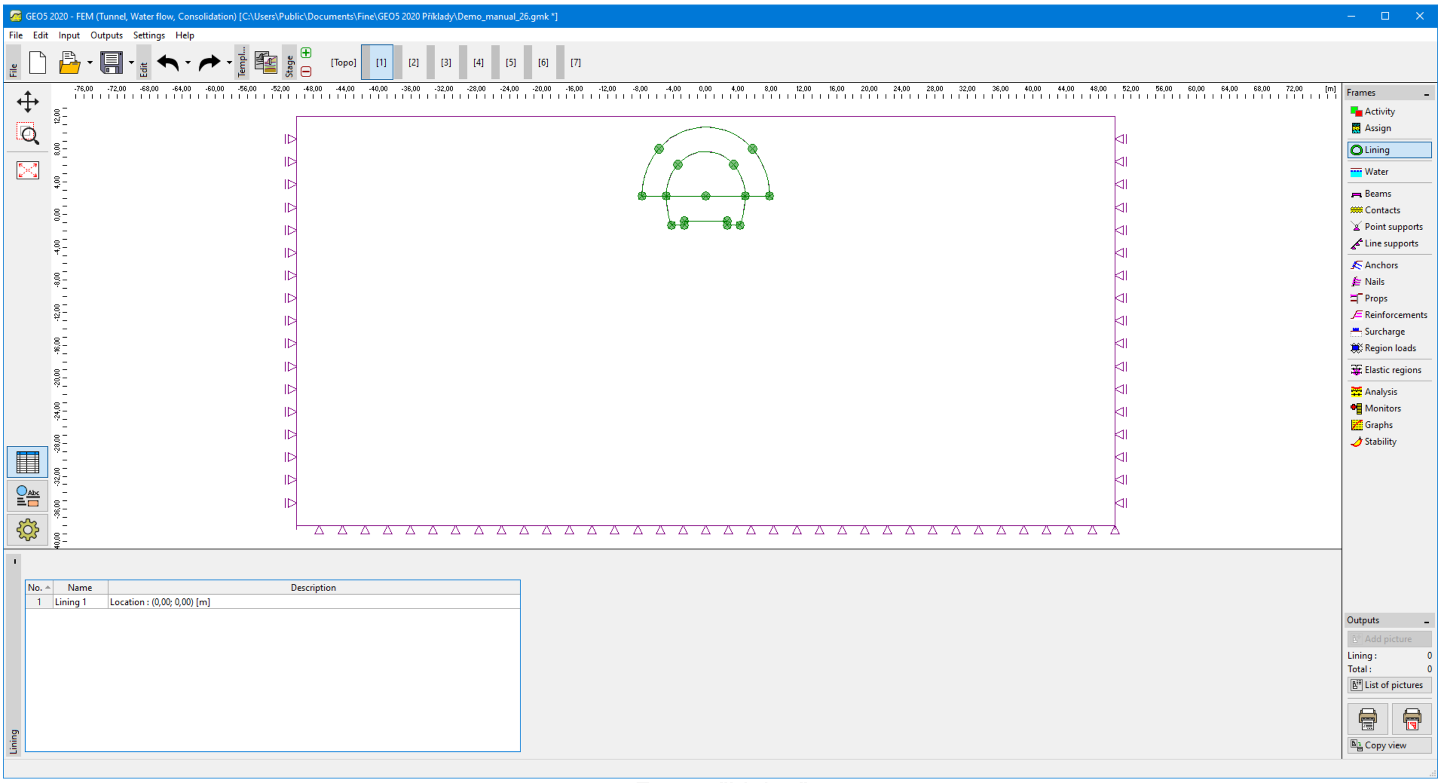


Frame "Assign"

Lining

The frame "**Lining**" contains a table with the list of input linings. This frame becomes accessible in the program once the "**Tunnel**" regime is activated in the frame "Settings". Only editing is allowed in subsequent stages of construction.

To adjust the lining the program launches the module "Lining - FEM". Its function is described in detail in the "**Topology**" regime in the "Lining" frame. In stages of construction, the "Lining - FEM" module contains the "Beams", "Anchors" and "Beam loads" frames.

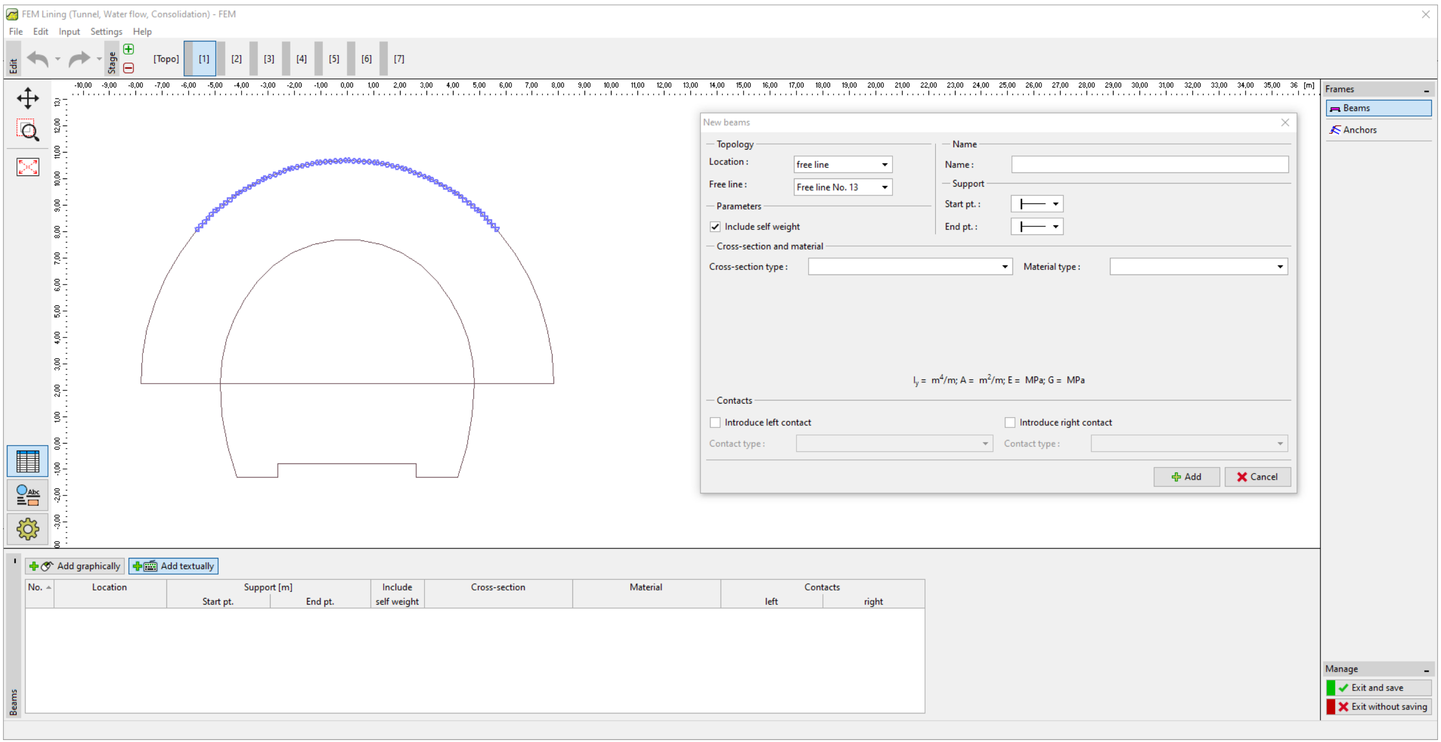


Frame "Lining"

Beams

The frame "**Beams**" contains a table with the list of beams. Actions applying to beams are identical to those used in stages of construction in the FEM program, frame "Beams".

Types of contacts to introduce contacts on beams are adopted from the FEM program.

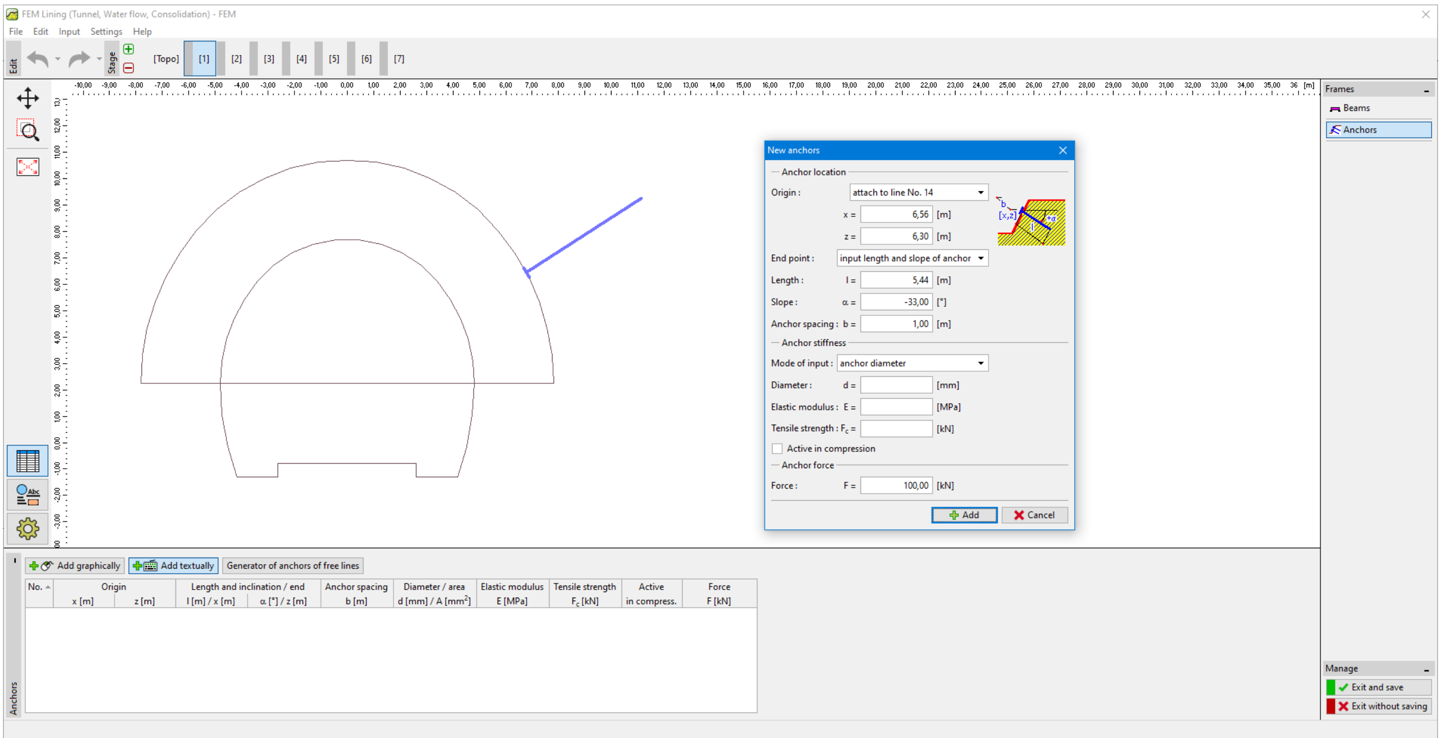


Frame "Beams"

Anchors

The frame "**Anchors**" contains a table with the list of input anchors. Actions applying to beams are identical to those used in stages of construction in the FEM program, frame "Anchors".

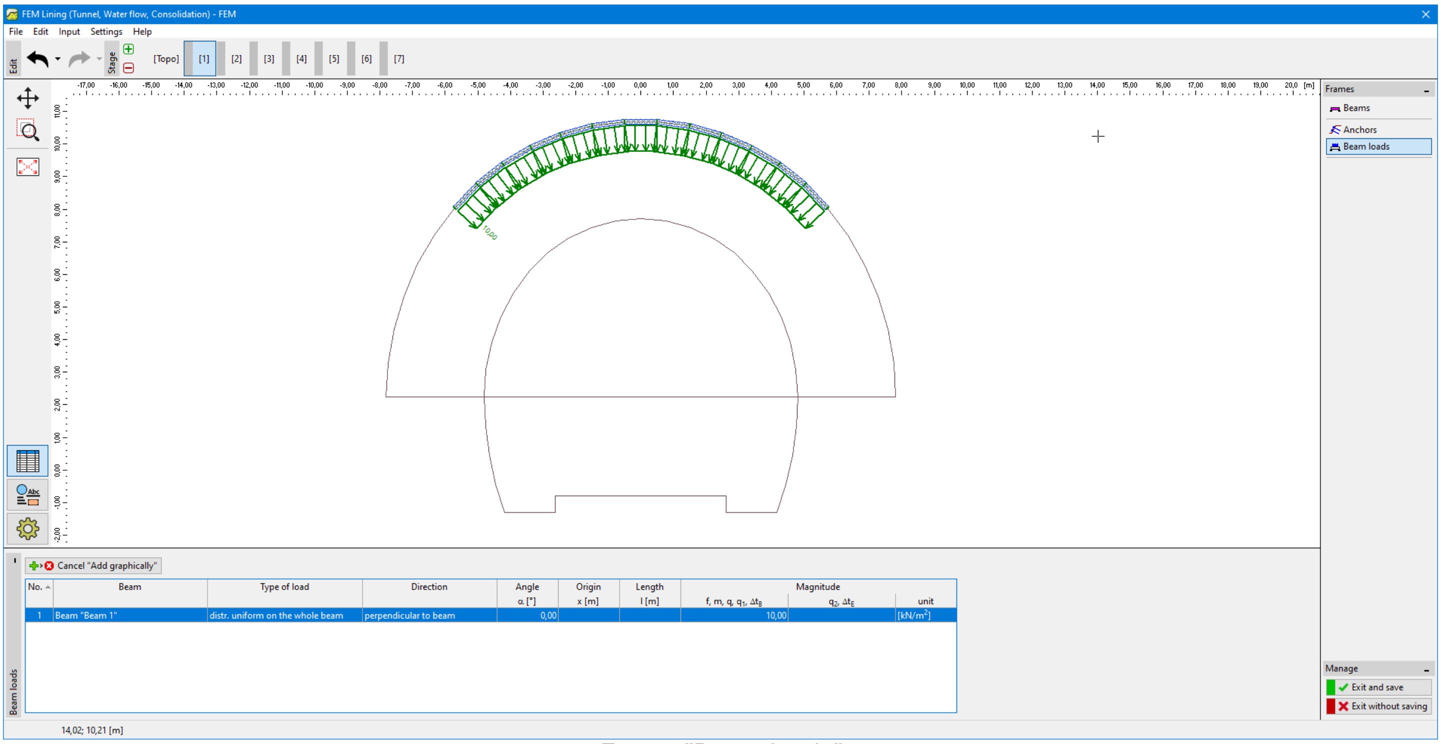
The frame differs by the function on the horizontal toolbar having the "Generator of anchors on free line" button.



Frame "Anchors"

Beam Loads

The frame "**Beam loads**" contains a table with the list of input loads. Actions applying to beam loads are identical to those used in stages of construction in the FEM program, frame "Beam loads".



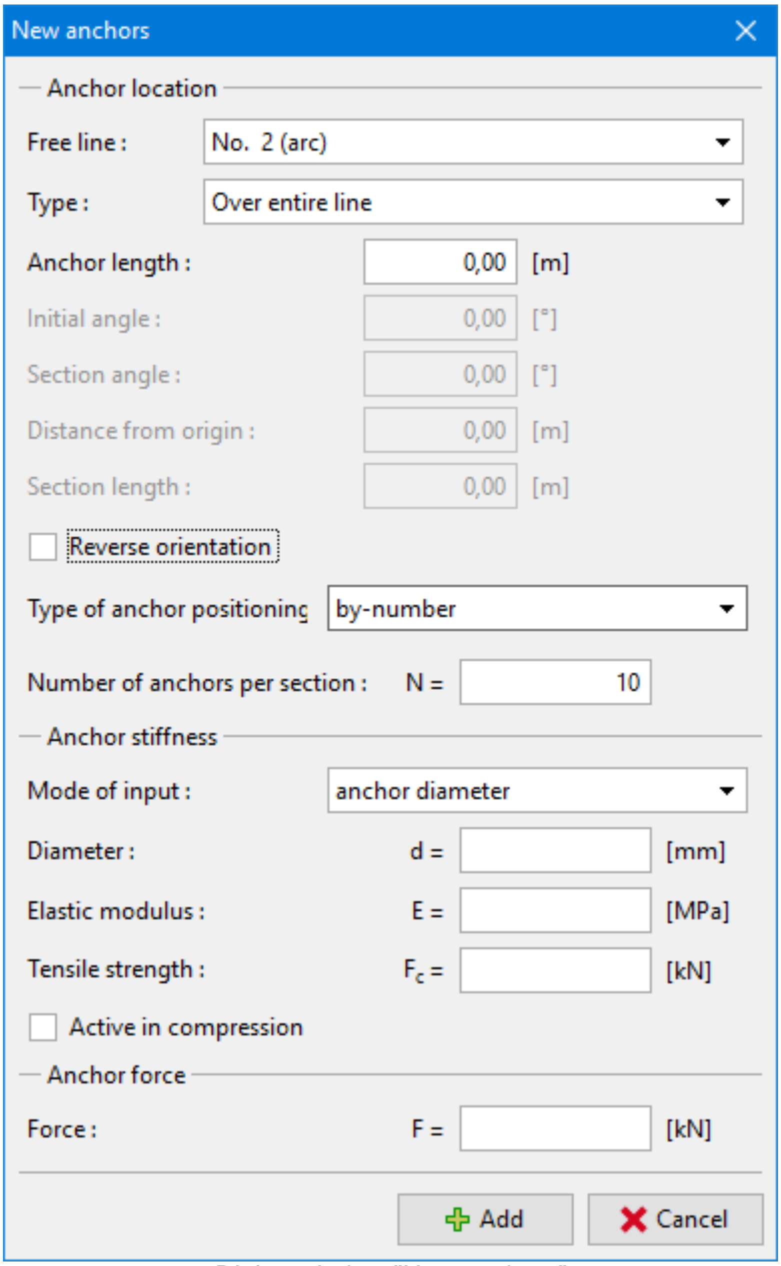
Frame "Beam loads"

Generator of Anchors on Free Line

Depending on particular parameters the generator creates corresponding elements that are then operated on independently with no possibility for being parametrically modification. If the parameters of generation are acceptable, the program displays during their modification the current graphical representation of generated elements.

The "**New anchors**" dialog window is an extension of the standard dialog window allowing for a uniform distribution of several identical anchors along a line. The spacing of anchors is generated the same way as used in the generator of anchored regions (over the entire line, over a part defined by the angle or length). There are three options to generate the number of anchors: by the number over a length, by the angle or spacing between individual anchors.

The generated anchors are attached in the FEM program to the free line defined therein.



Dialog window "New anchors"

Beams

The frame "**Beams**" contains a table with the list of beams. Adding beams is performed in the "**New Beams**" dialog window.

The beam location is selected from the combo box (mesh line, terrain segment).

Added beams can also be edited on the desktop with the help of active objects. The program employs the following coordinate systems.

The **beam elements** serve to model **beams, linings**, **sheeting walls**, etc. **Distribution internal forces** such as moment, normal and shear forces developed along a beam axis can are derived from the beam element end forces.

Beams are assigned to already defined lines (**free lines, terrain segments**) - the corresponding line then represents the **beam axis**. The program offers several basic types of cross-sections. Nevertheless, the user is free to introduce the required cross-section independently.

An important step when modeling beams is the definition of **contact elements** characterizing the interface behavior between the beam and the soil. Contact (interface) elements can be assigned to **both sides of a beam**. A correct definition of contacts is essential especially when modeling sheeting walls.

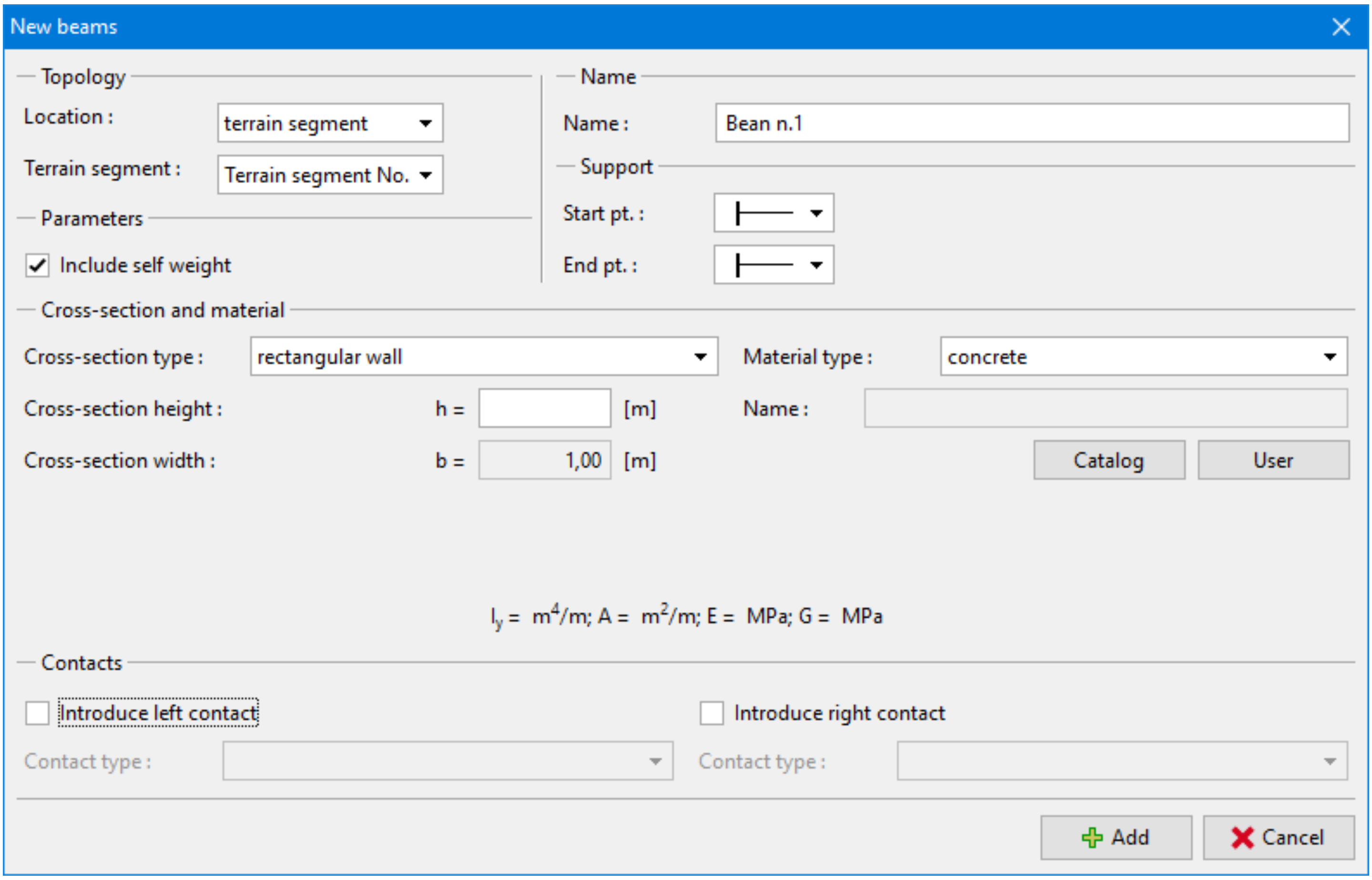
Types of endpoints connections can be specified for each beam.

In subsequent stages, the beam can be either strengthened or degraded.

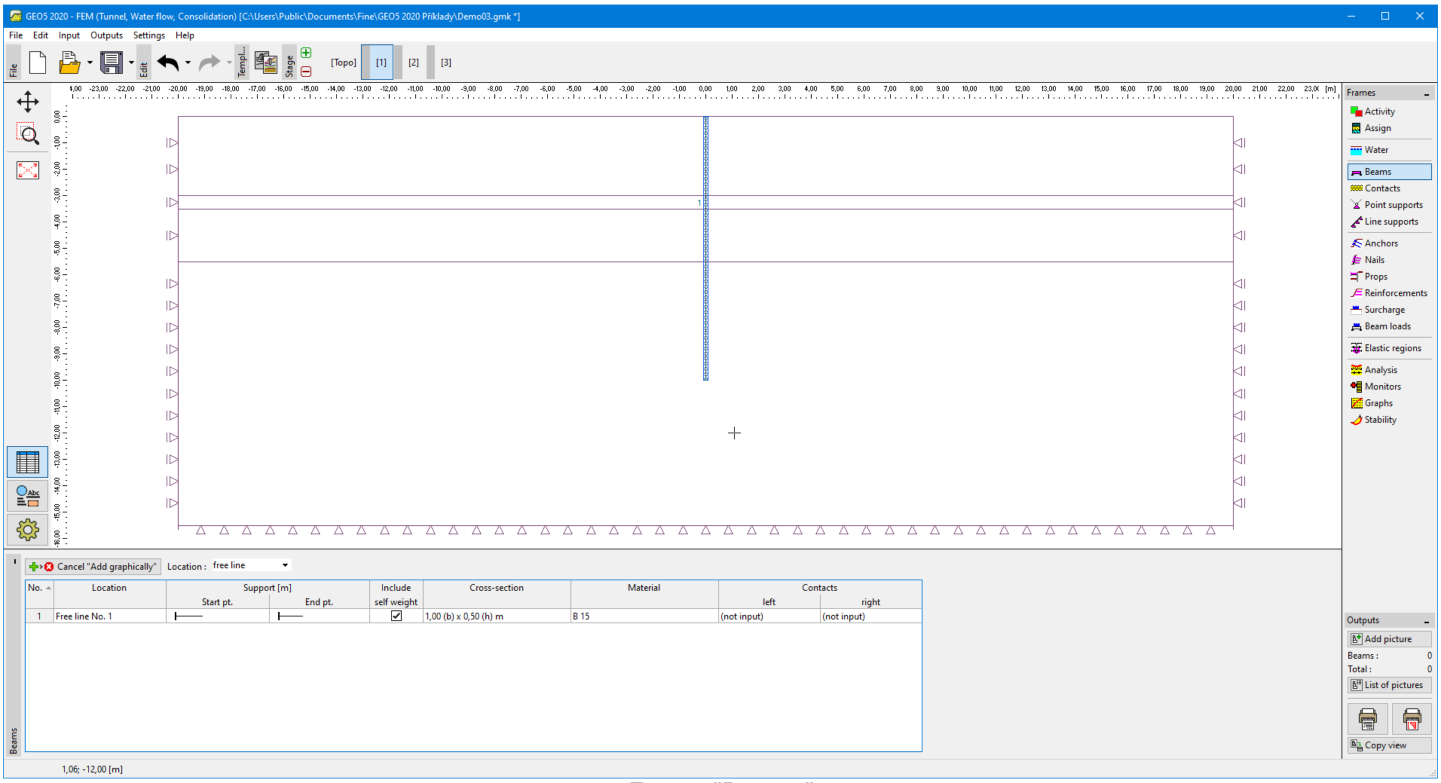
The program automatically includes the **beam self-weight** into the analysis. This feature, however, can be turned off when defining the beam.

Beams are modeled using the **beam elements** with three degrees of freedom at each node.

The beam elements are formulated on the basis of the Mindlin theory. The theory assumes that the plane cross section normal to the beam axis before deformation remains plane after deformation but not necessarily normal to the deformed beam axis. At present, the internal forces are evaluated at the element nodes and from the beam end forces.



Dialog window "New beams"



Frame "Beams"

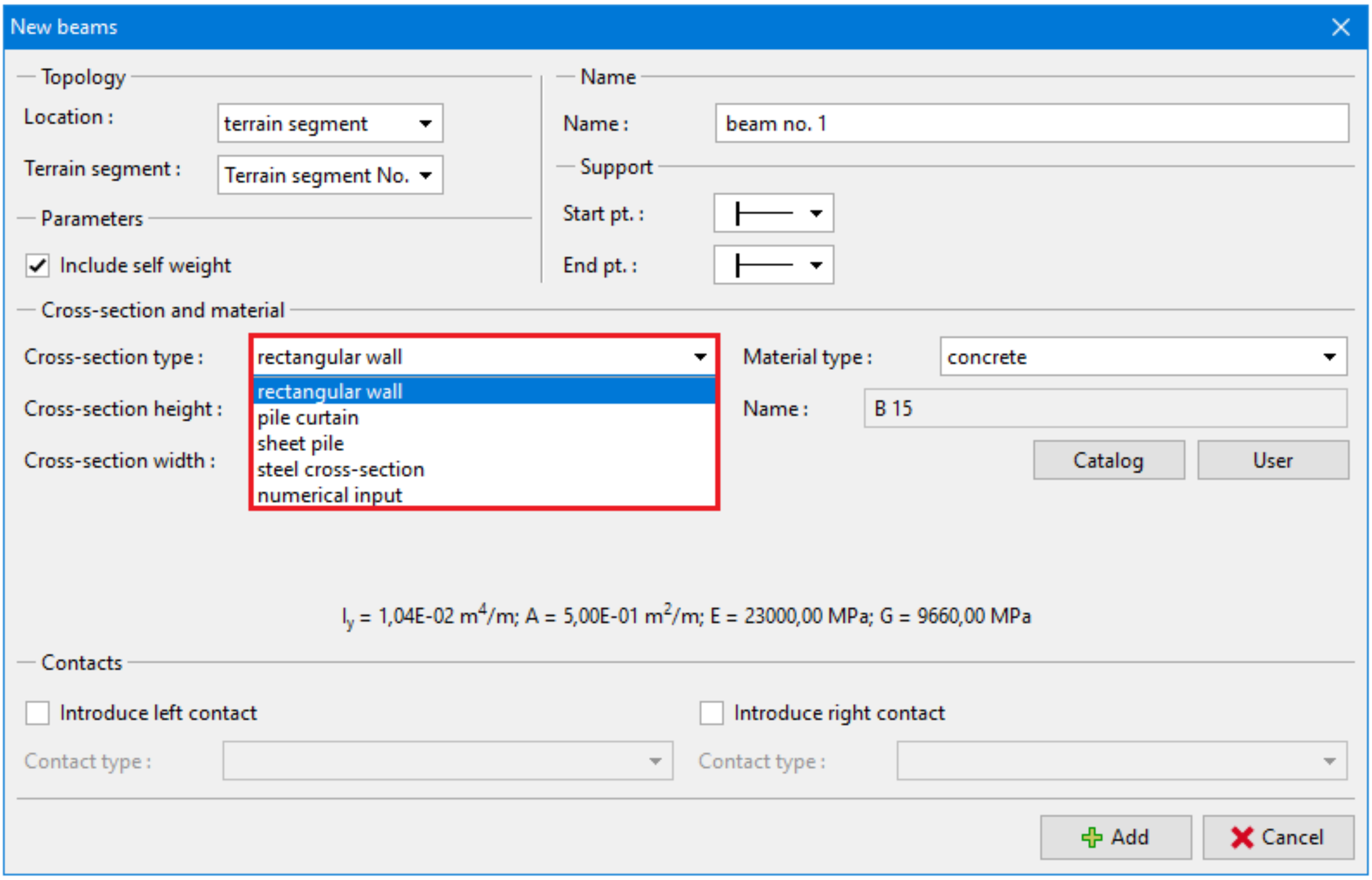
Types of Cross Section

The program allows the user to either input the **cross-section parameters digitally** or to choose from one of the predefined types of the cross-section. The type of material of the cross-section is selected from the Catalog of Materials or is introduced digitally using the Editor of Materials. The following types of beam cross-section are implemented:

- **rectangular concrete wall** - a beam wall thickness must be specified
- **pile wall** - a pile diameter and their spacing must be specified
- **steel sheet pile** - selected from the built-in database
- **steel I cross-section** - a type of cross-section from the built-in database is selected, their spacing must be specified (the type of cross-section is selected from the "Catalog of Profiles", or is defined in the "cross-section Editor", the type of material is selected from the "Catalog of Materials" or is specified digitally in the "Editor of Materials")

All input cross-sections **are automatically recalculated per** 1 m **(feet) run. The results of internal forces developed along the beams are also presented per** 1 m **(feet) run of a structure.** Thus if necessary, for piles or I cross-sections they must be adjusted depending on their spacing by the user.

Should you have your database of **sheet piles**, which is not yet built-into the program, we will be happy to implement it. You may reach us at **hotline@fine.cz**.



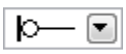
Dialog window "New beams" - the selection of the type of cross-section

Beam End-Points Connection

The program allows for three types of beam end-points connection:

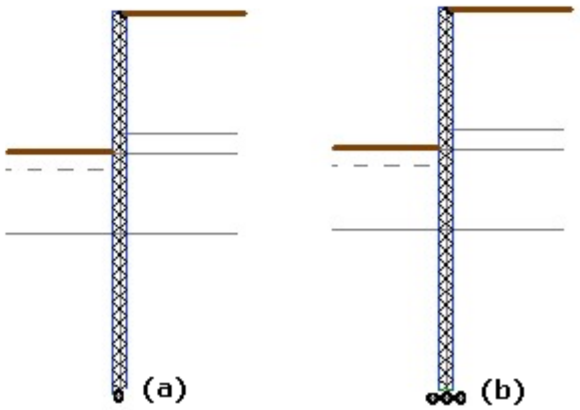
 **fixed** standard type

 **hinge** (is used to introduce an internal hinge in between beams - locations with zero bending moment)

 **foot**

The foot is a special type of a beam end-point support in the soil. It is applicable for **the beam end-point located in the soil body**. When the fixed type of connection is assumed the beam and the soil element are connected at one point (a singular connection) often causing evolution of plastic strains in the surrounding soil and loss of convergence. **The foot allows for a more realistic redistribution of contact stresses** and prevents the beam from "penetrating" into the soil, consequently stabilizing the convergence process. **By default the foot length** is assumed to be equal to the beam width -

it can be arbitrarily adjusted (for example to enlarge the pile heel).



Connection (a) without (b) with a foot

Degradation and Strengthening of Beams

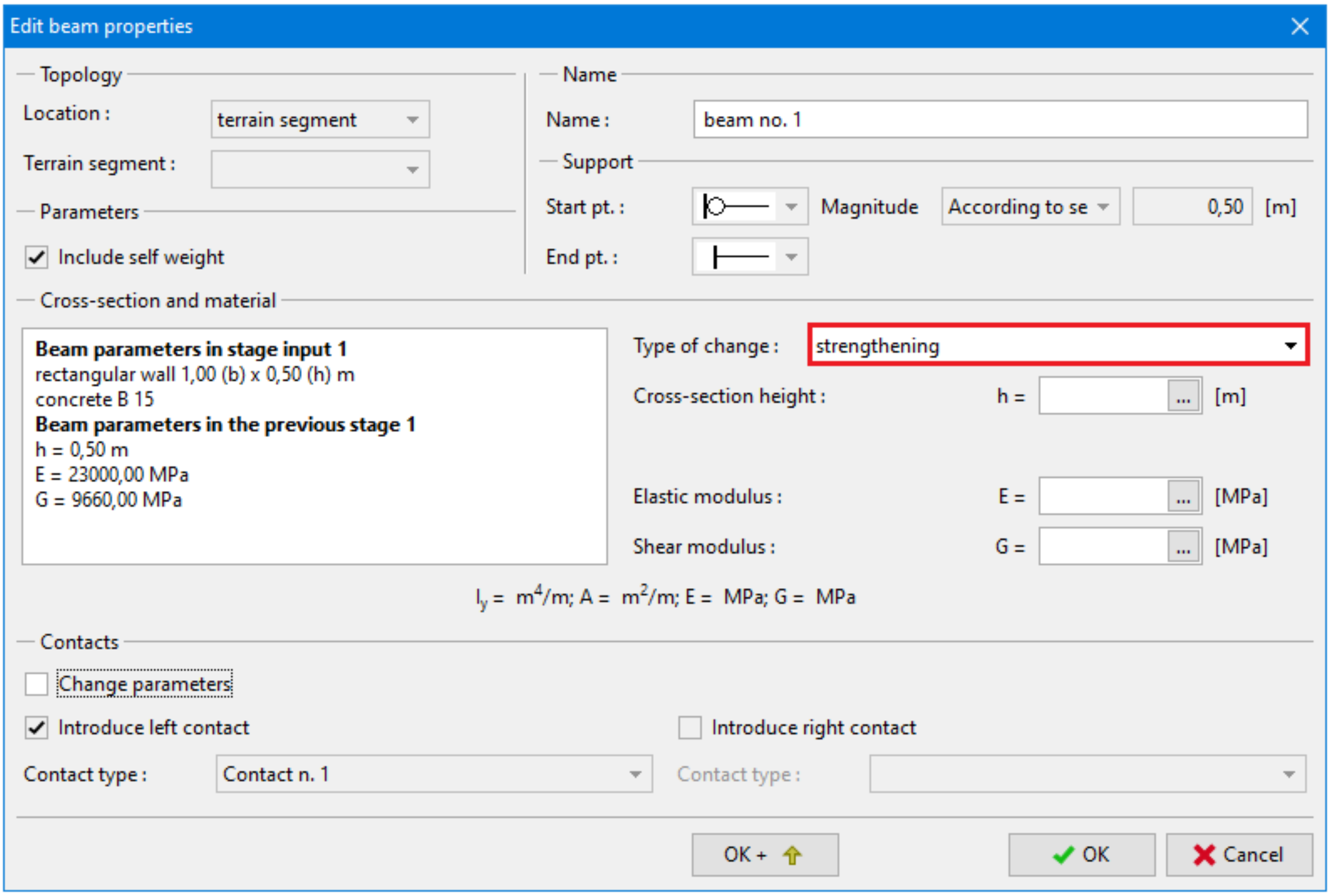
In subsequent stages, the beams cannot be edited in a standard way. Therefore, one of the following options must be selected to modify them:

- removing the selected beam from the analysis
- degrading the selected beam (applicable only in the "Tunnels" mode)
- strengthening the selected beam cross-section
- modifying the beam contact properties

The type of modification is selected from the "**Adjust beam properties**" dialog window.

A degree of **beam degradation** is specified in percentage, one hundred percent corresponds to beam removal.

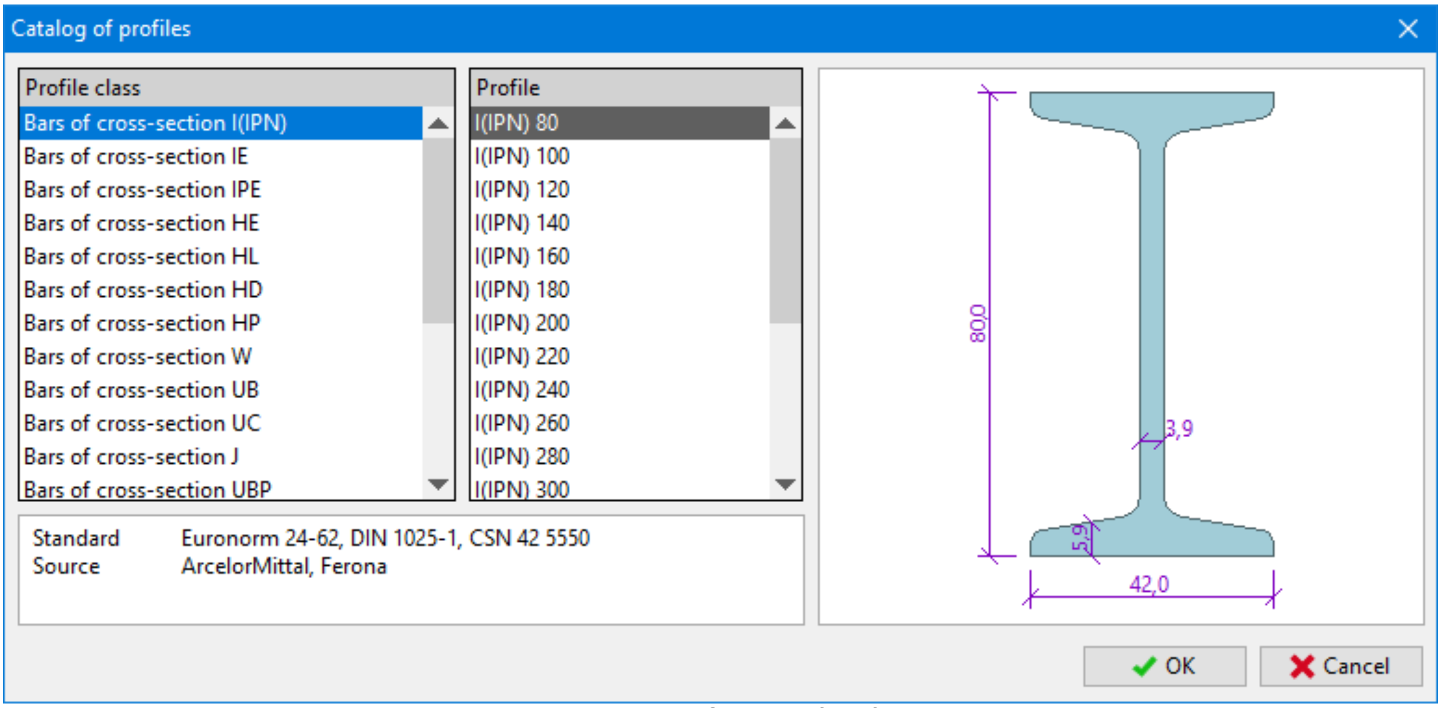
Strengthening a beam element with a rectangular cross-section can be achieved by enlarging its width (e.g., increasing the shotcrete thickness). Other cross-sections are modified by directly inputting new (larger) values of the cross-section parameters.



Dialog window "Adjust beam properties" - beam strengthening

Catalog of Profiles

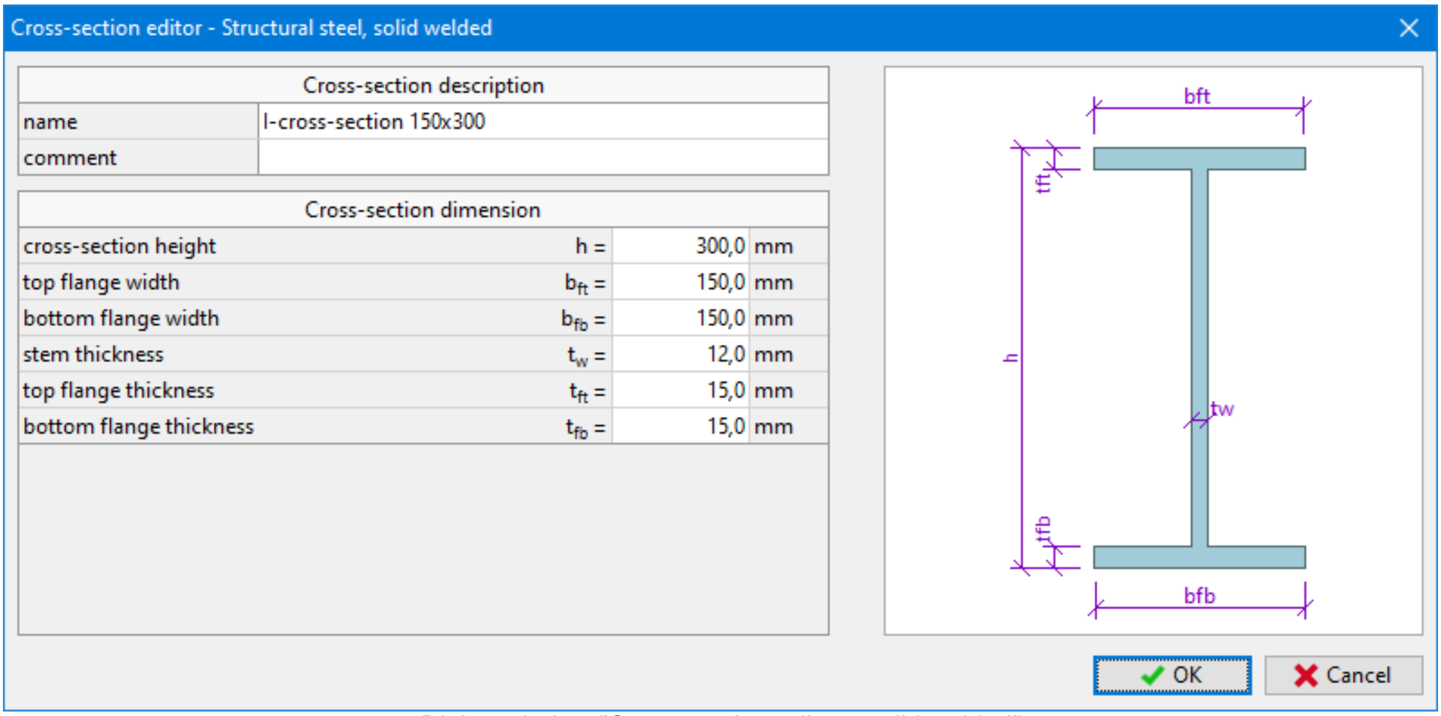
In case of a steel cross-sections, the program allows for choosing a particular cross-section from the catalog of profiles. Only the type of cross-section has to be specified in the dialog window. The type of material of the cross-section is selected similarly to other cross-sections (rectangular wall, pile wall, sheet pile…) from the "Catalog of materials", or defined in the "Editor of materials". The type of cross-section (beam) is selected in the "New beams" dialog window.



Dialog window "Catalog of profiles"

Cross Section Editor

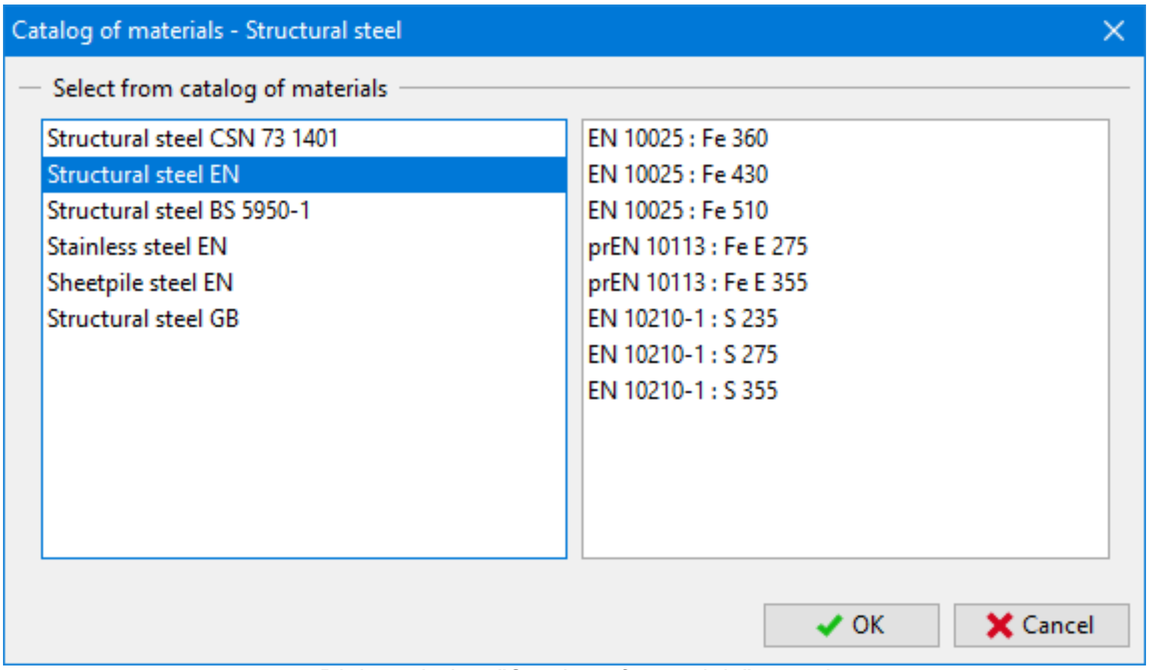
In case of steel cross section the program allows for introducing the user-defined cross section. Only the shape of cross section has to be specified in the dialog window. The type of material of the cross section is selected similarly to other cross sections (rectangular wall, pile wall, sheet pile…) from the "Catalog of materials", or defined in the "Editor of materials". The type of cross section (beam) is selected in the "New beams" dialog window.



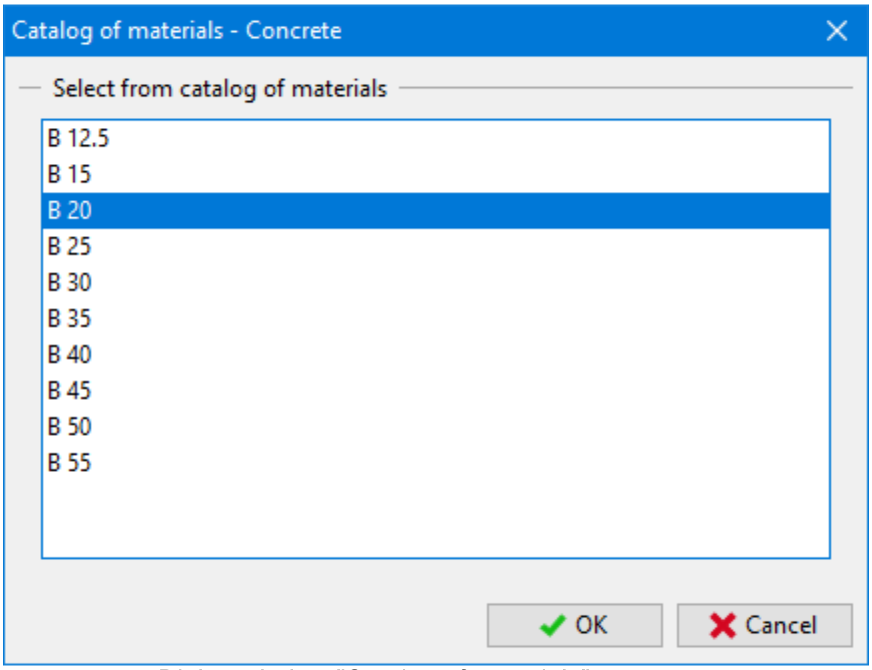
Dialog window "Cross section editor - solid welded"

Catalog of Materials

The program contains a built-in catalog of materials for concrete and steel. Only the type of material has to be specified in the dialog window. The shape of the cross-section is selected from the "Catalog of profiles", or defined in the "cross-section editor". For other types of cross-sections (rectangular wall, pile wall, sheet pile…) the type of cross-section is selected in the "New beams" dialog window.



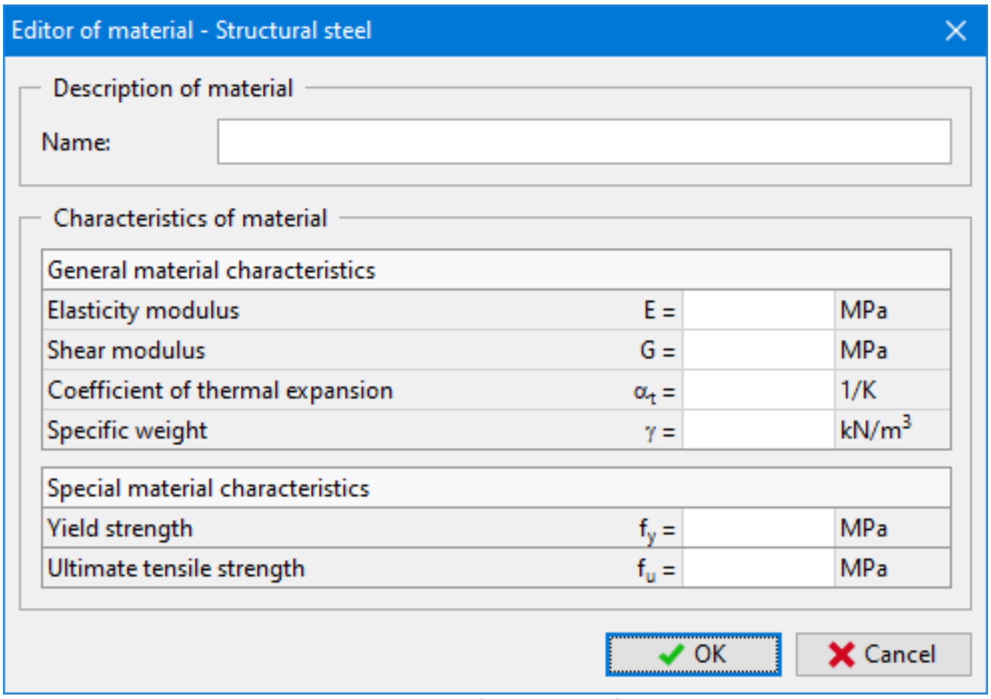
Dialog window "Catalog of materials" - steel



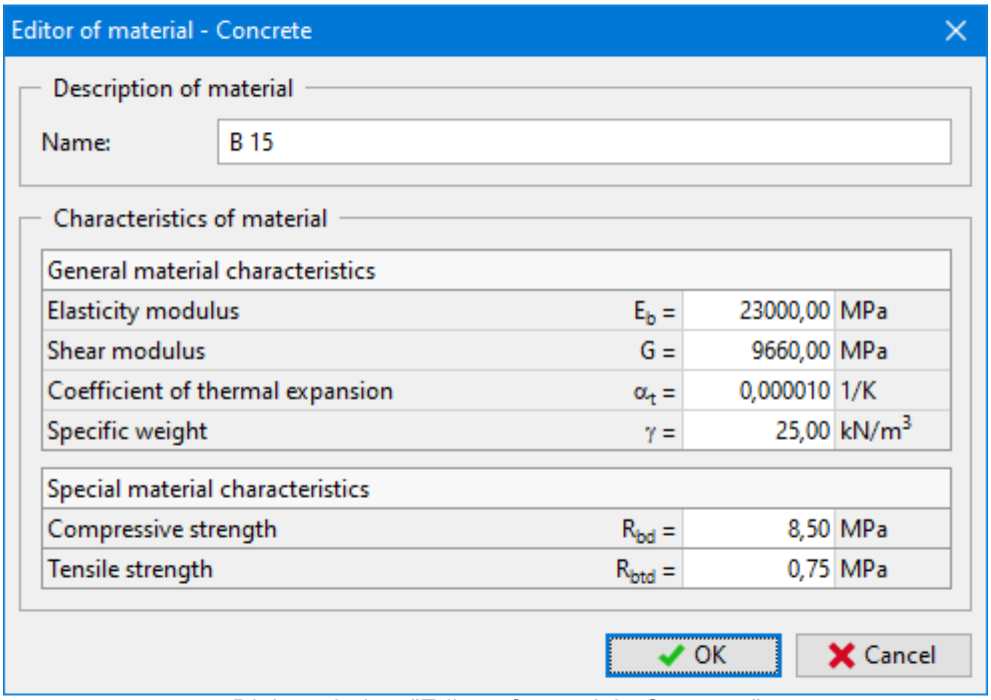
Dialog window "Catalog of materials" - concrete

Editor of Materials

Apart from using the "Catalog of materials" the program allows the user to enter the material parameters for steel and concrete digitally. Only the type of material (material parameters) has to be specified in the dialog window. The shape of the cross-section is selected from the "Catalog of profiles", or defined in the "cross-section editor". For other types of cross-sections (rectangular wall, pile wall, sheet pile…) the type of cross-section is selected in the "New beams" dialog window.



Dialog window "Editor of material - Structural steel"



Dialog window "Editor of material - Concrete"

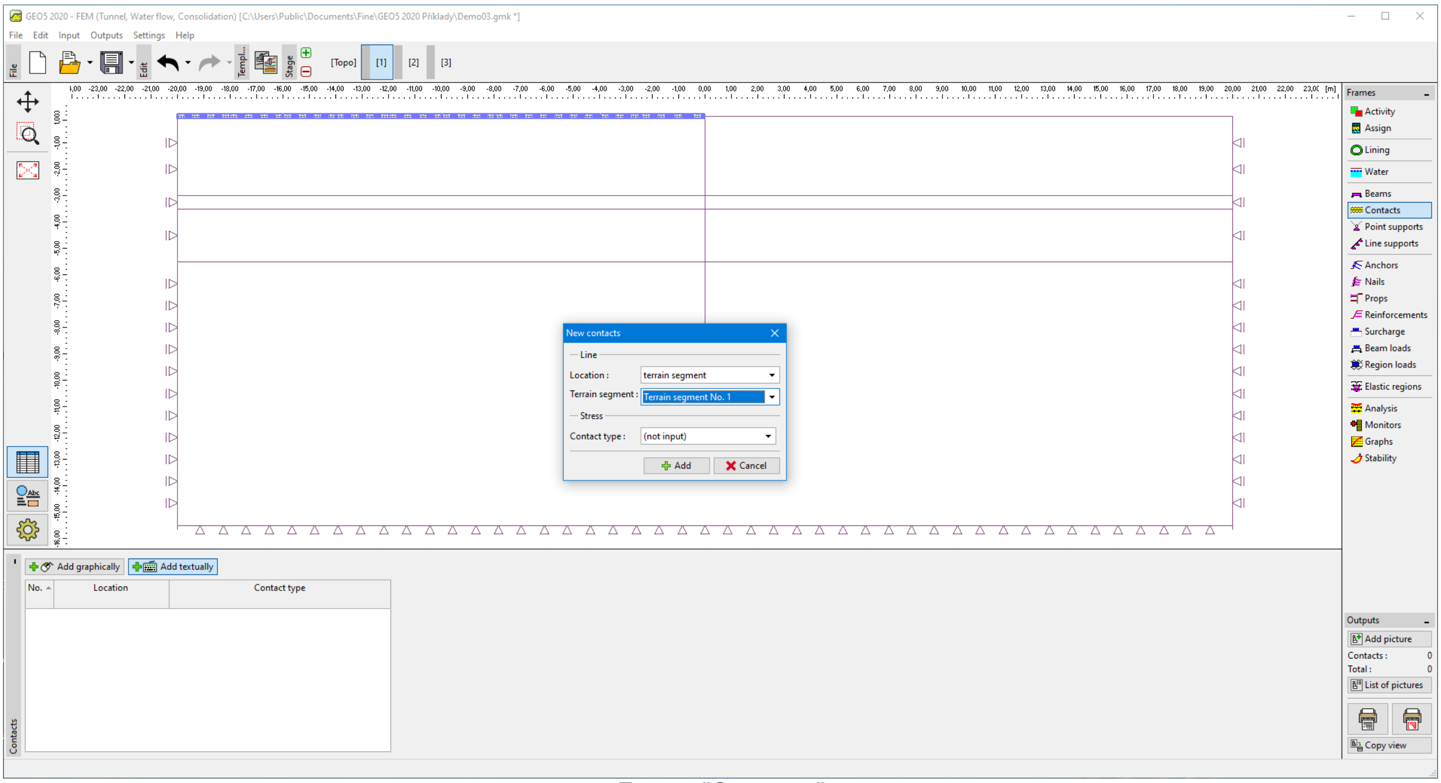
Contacts

The frame "**Contacts**" contains a table with the list of contacts. Adding contacts is performed in the "**New contacts**" dialog window.

The **contact elements** are used in applications that require a proper representation of structure-soil interaction. They can be further used to model joints or interfaces of two distinct materials (soil - rock interface). Contacts are assigned to already defined lines - free lines or mesh lines (interfaces). The contact is defined by its type.

The contact location is selected from the combo box (mesh line, terrain segment).

The input contacts can also be edited on the desktop with the help of active objects.



Frame "Contacts"

Contacts and Beams (Water Flow)

The frame "**Contacts**" ("**Beams**") contains (in mode "Water flow") a table with the list of contacts (beams). Adding contacts (beams) is performed in the "**New contacts**" ("**New beams**") dialog window.

The contact location is selected from the combo box (mesh line, terrain segment).

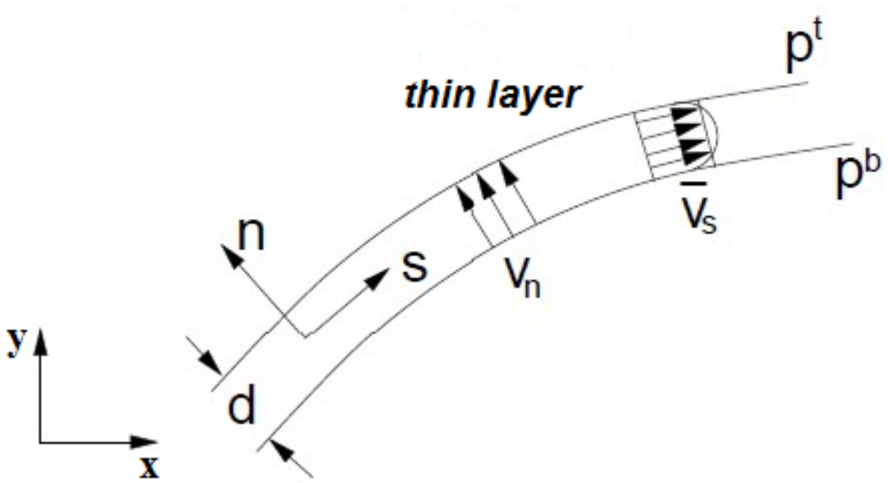
The **contact elements** are used in applications that require a proper representation of structure-soil interaction. They can be further used to model joints or interfaces of two distinct materials (soil - rock interface). Contacts are assigned to already defined lines - free lines or mesh lines (interfaces). The contact is defined by its type.

The input contacts can also be edited on the desktop with the help of active objects.

Beam or contact element can be defined as:

- **permeable**
- **impermeable**
- **partially permeable**

Contact elements allow us to model a certain barrier for flow in the soil body. Consider for example a sheeting wall represented in the stress analysis by beam elements. The sheeting wall anchored into the inside region can be considered either as fully permeable, or fully impermeable or partially permeable. Although the first two cases can also be treated using contact elements placed along the corresponding line, they are handled by the program automatically without needing these elements. The third case represents a problem of flow in a thin zone having a given thickness d , see figure:



Partially permeable contact

Corresponding fluxes in the tangent direction (s -direction) q_s and normal direction (n -direction) q_n are given by:

$$q_s = -k_s \frac{1}{2} \frac{\partial \left(h^t + h^b \right)}{\partial_s}$$

$$q_n = -k_n \frac{h^t + h^b}{d}$$

Defining contact elements, therefore requires inputting the following parameters:

k_s - permeability in tangent direction (permeability longitudinal), [m/day]

k_n - permeability in normal direction (permeability transverse), [m/day]

Point Supports

The frame "**Point supports**" contains a table with the list of point supports. Adding point supports is performed in the "**New point supports**" dialog window.

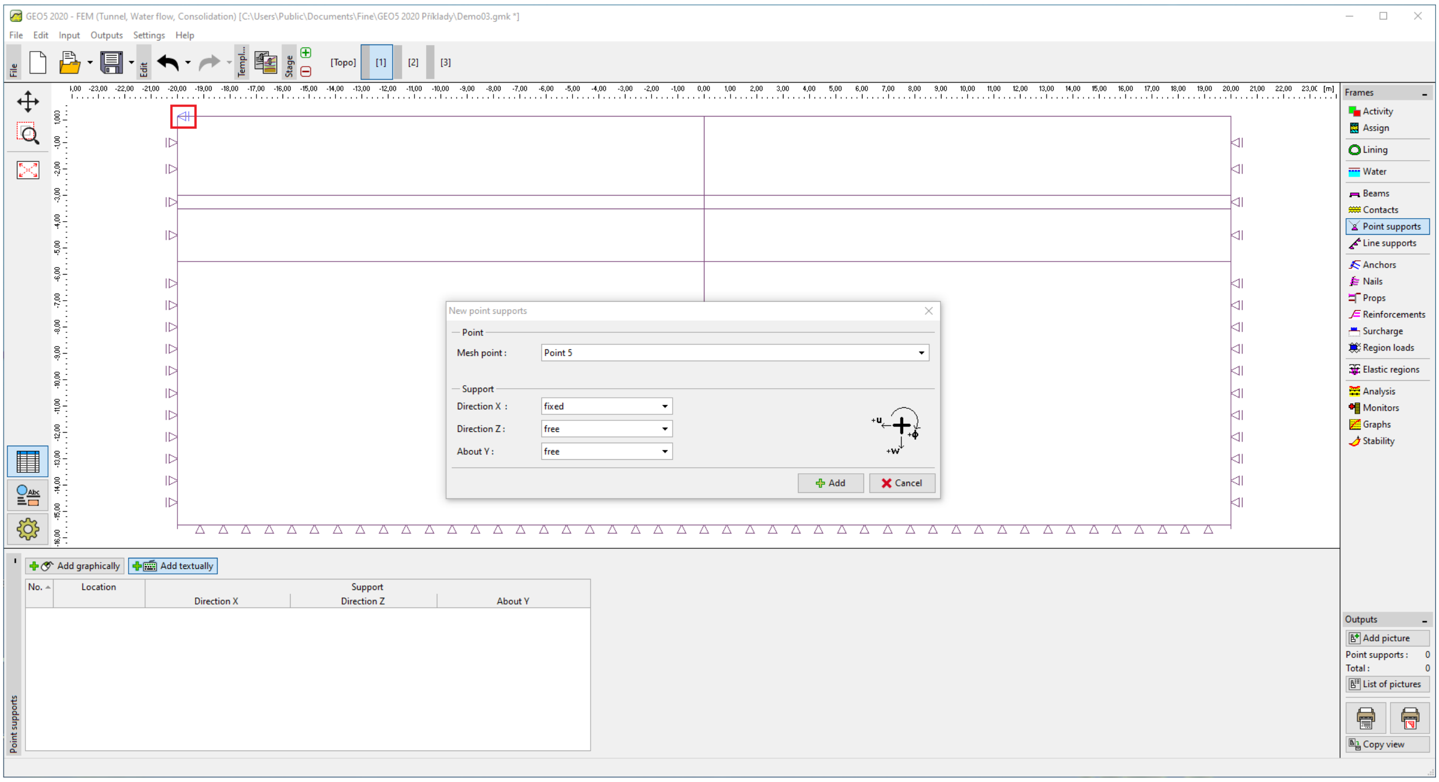
The input point supports can also be edited on the desktop with the help of active objects. The program employs the following coordinate systems.

The program contains a built-in automatic generator of standard boundary conditions. Therefore, in most problems the **boundary (support) conditions are not required to be specified**.

The following types of point supports are considered:

- **free**
- **fixed**
- **spring**
- **prescribed deformation**

Supports are defined in **the global coordinate system**.



Frame "Point supports"

Point Flow

The frame "**Point flow**" contains a table with the list of points with a flow. Adding a point flow is performed in the "**New point flow**" dialog window.

The input point flows can also be edited on the desktop with the help of active objects.

The following boundary conditions can be specified:

a) Pore pressure at a point

- Numerically - the value of pore pressure at a given point is specified [*kPa* , *ksf*]
- By specifying the location of groundwater table (total head) - coordinate of GWT is specified

b) Point inflow/outflow

Pumping/injection rate is specified [$m^3/day/m$, $ft^3/day/ft$]

Line Supports

The frame "**Line supports**" contains a table with the list of line supports. Adding line supports is performed in the "**New line supports**" dialog window.

The line support location is selected from the combo box (free line, terrain segment, mesh line).

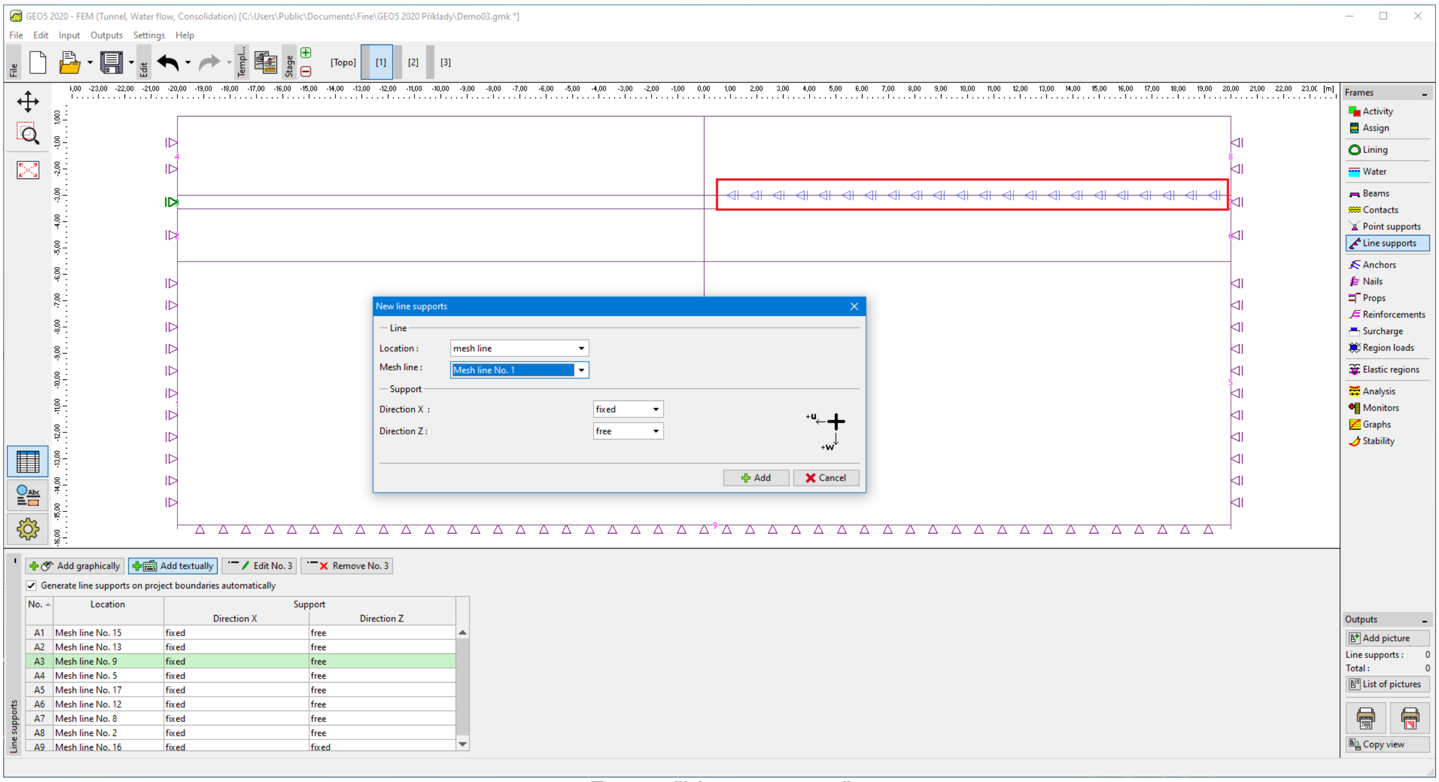
The input line supports can also be edited on the desktop with the help of active objects. The program employs the following coordinate systems.

The program contains a built-in automatic generator of standard boundary conditions. Therefore, in most problems, the **boundary conditions are not required to be specified**.

When assigning supports to a line it is first necessary to select the type of line (**free line, interface, mesh line**).

The following types of line supports are considered:

- free
- fixed
- deformation



Frame "Line supports"

Line Flow

The frame "**Line flow**" contains a table with the list of lines with a flow. Editing a line flow is performed in the "**New line flow**" dialog window.

The input line flows can also be edited on the desktop with the help of active objects.

Flow boundary conditions **must be defined on all boundary lines**. The following boundary conditions can be specified:

a) Impermeable

b) Permeable Pore pressure on a given line is equal to zero

c) Pore pressure

- distribution of pore pressure p can be specified numerically
- distribution of pore pressure can be specified by inputting the location of the groundwater table (by prescribing the total head h)

d) Inflow/outflow on a line q - it is specified in velocity units e.g. [m/day , ft/day] - the flow velocity into/out of the region is specified. The default setting corresponds to an impermeable boundary for which $q = 0$.
e) Seepage surface - this boundary condition is introduced providing the boundary cannot be uniquely divided into the part with prescribed pore pressure and the part with prescribed inflow/outflow (the exit point is not known). In such a case the analysis is performed in two steps. In the first step, the program locates the exit point. The actual flow analysis with known boundary conditions is then carried out in the second step. In some cases, both steps must be repeated several times. When enhanced input is considered the program requires entering a fictitious permeability k_v in units [m/day]. This is essentially a penalty term, a sufficiently large number in general, ensuring that along an impermeable boundary the value of total h will be equal to the y -coordinate of a given point ($q = 0$). For a part of the boundary with no flow condition, we have $k_v = 0$. Variables q and h are then related by:

$\bar{q}_n = k_v \left(h - y \right)$ if $h > 0$ ($S = 1$) inside soil body

$\bar{q}_n = 0$ if $h < 0$ ($S < 1$) inside soil body

Note: If in case of transient flow we directly define in the first calculation stage the location of the groundwater table (phreatic surface) as an initial condition, we should define in the next calculation step along the boundaries below the water level in the region of fully saturated soil a boundary with the prescribed pore pressure having a corresponding value and not the seepage surface. In the case of seepage surface, the program would immediately label this boundary as a boundary with zero pore pressure and not the originally assumed boundary with the pore pressure distribution in accordance with the expected height of GWT. In such a case, the analysis will not converge, because water, instead of flowing out of the domain at $p = 0$, will have a tendency to flow in, which not realistic.

Anchors

The frame "**Anchors**" contains a table with the list of anchors. Adding anchors is performed in the "**New anchors**" dialog window.

The input anchors can also be edited on the desktop with the help of active objects. The program employs the following coordinate systems.

The anchor head (starting point) is **automatically connected** to the terrain, an arbitrary interface, or opening (tunnel lining). The anchor head is then automatically positioned into the intersection of the anchor line determined by the input points and the selected line. The anchor can also be introduced directly by specifying coordinates of the two endpoints.

Anchors as stabilizing or reinforcing elements are represented by **elastic tensile-compressive bar element** with constant normal stiffness. The maximum allowable tensile force the element can sustain controls tensile failure of the anchor. The bar element is anchored into the soil only at its starting and endpoints. No mutual interaction between the soil and the anchor along the anchor length is considered.

Anchors are defined by their starting and end points and by their stiffness. The program automatically links the anchor element degrees of freedom to the actual degrees of freedom of the predefined finite element mesh. Therefore, the anchor can be introduced **anywhere in the structure**.

The **anchor stiffness** is specified in terms of the elastic modulus and its area. The program makes it also possible to enter the anchor diameter - the area is then determined automatically. In stability analysis problems the anchor stiffness is not considered. Its action is realized only through the prestress force introduced automatically as an external compressive force acting at the anchor head.

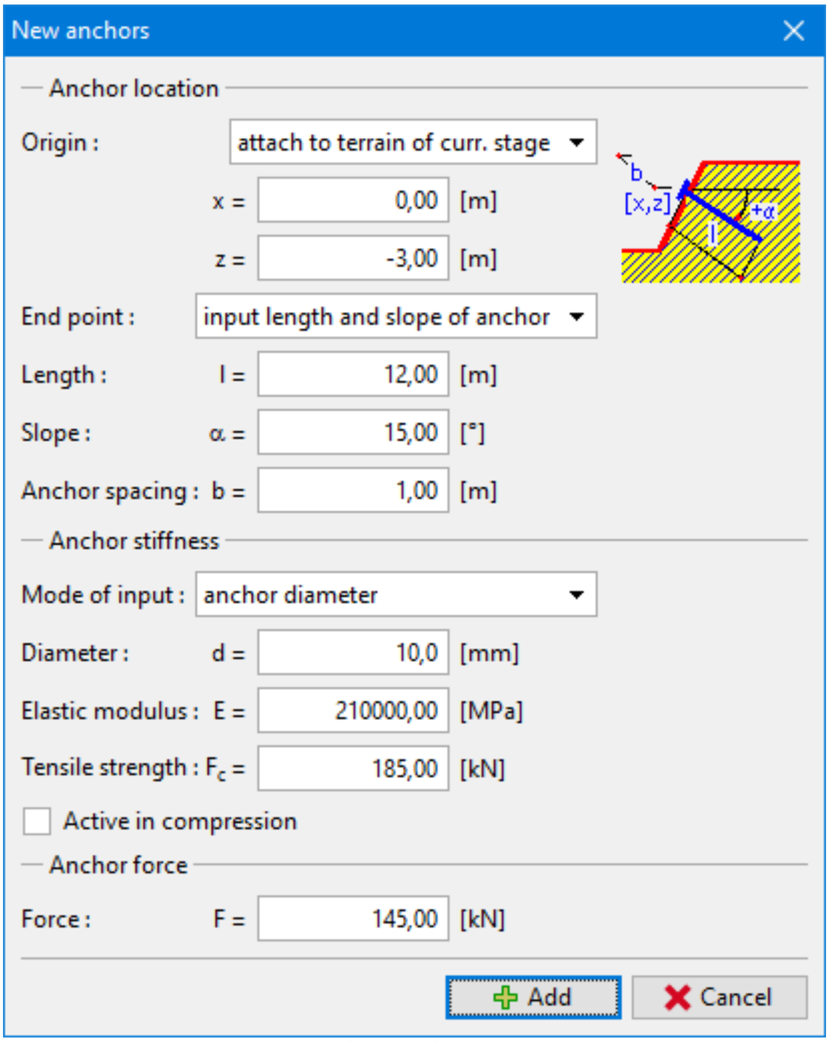
Other important parameters are the **prestress force** and the **tensile strength** (the anchor breaks when the tensile strength is exceeded). For elements with no prestress the prestress force is set equal to zero. A sufficiently large value of the anchor tensile strength may be specified to avoid anchor failure.

By default the anchor **does not support a compressive force** - anchor elements loaded in compression during a certain stage of calculation are temporarily disabled. If tension occurs in subsequent analysis run (due to change in load, geometry or material parameters of soil), the program automatically introduces these elements back into the analysis. The program makes also possible to include a compressive response of the anchor. However, for elements loaded primarily in compression we recommend defining these elements as props.

The anchor deforms during analysis. Such deformation together with the deformation of the surrounding soil may cause a **reduction of the specified prestress force** in the anchor. Providing we wish to achieve a specific prestress force in the anchor, it is necessary to either post-stress the anchor to a given value in the next calculation stage or to use a sufficiently large magnitude of the prestress force right from the beginning to compensate for a possible drop (the resulting anchor force after completion of the calculation step is displayed at the anchor head below the prescribed prestress force).

In subsequent stages the program allows only for anchor post-stressing - change of the initial prestress force, or for removing the anchor from the analysis.

Introducing pre-stressed anchors into the soil may lead to plastic deformation of the soil in the vicinity of the anchor head or root. Some modifications of the original input are then required to avoid often encountered loss of converge.



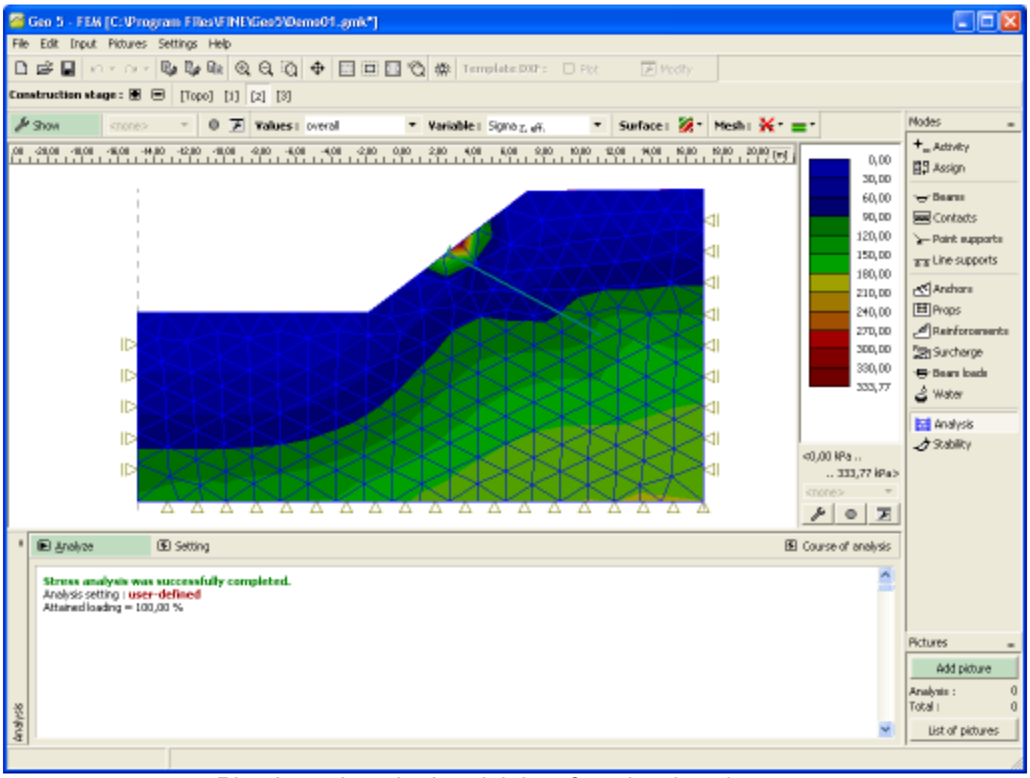
Anchor input

Anchor End Points

Introducing pre-stressed anchors into the soil may lead to **plastic deformation** of the soil in the vicinity of the anchor head or root - the analysis then often fails to converge.

In such a case we recommend the following modifications of the original input:

- to place a **beam element** under the anchor head (this results into a better transition of load into the soil),
- to place the anchor root into a **sufficiently stiff soil** (use the elastic or modified elastic material model for the soil layer around the anchor).

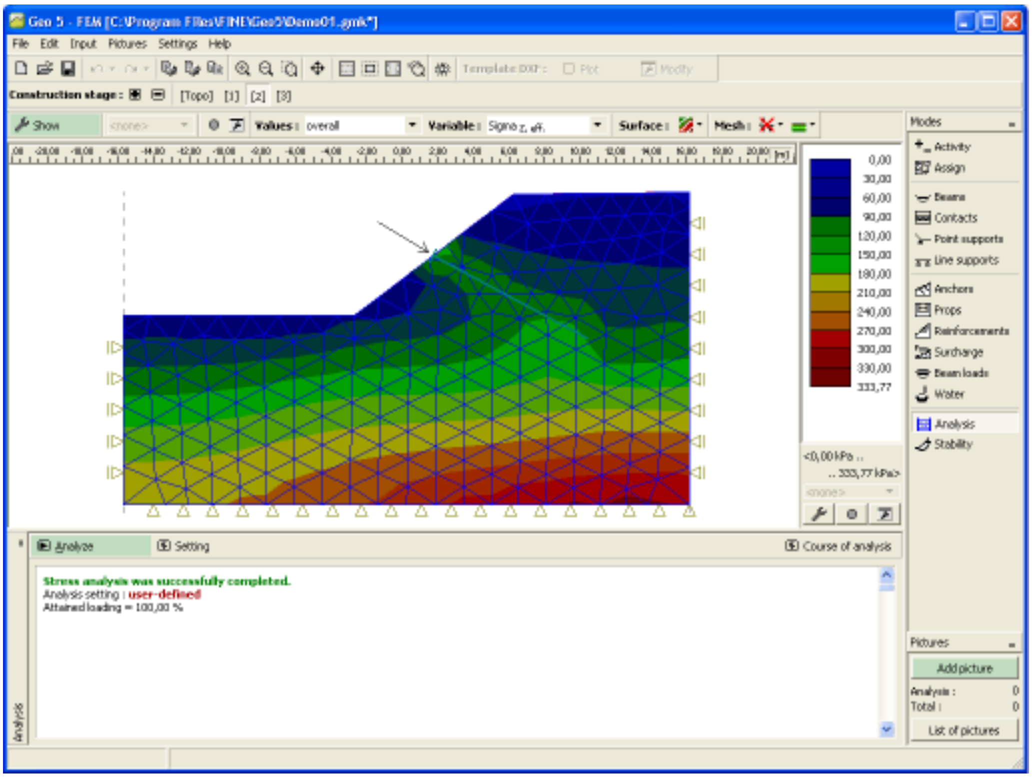


Plastic regions in the vicinity of anchor head or root

Anchors in the Stability Analysis

When performing the **stability analysis** the actual pre-stressed anchor is automatically replaced by corresponding **compressive point forces** acting at the anchor head.

The soil at the point of the applied force may, however, undergo plastic deformation. One should therefore carefully assess the resulting distribution of plastic strains. Note that the localization of equivalent plastic strain identifies the location of the potential slip surface. Therefore, if the plastic strains at the anchor head become decisive, it is necessary to introduce some modifications of the original input.



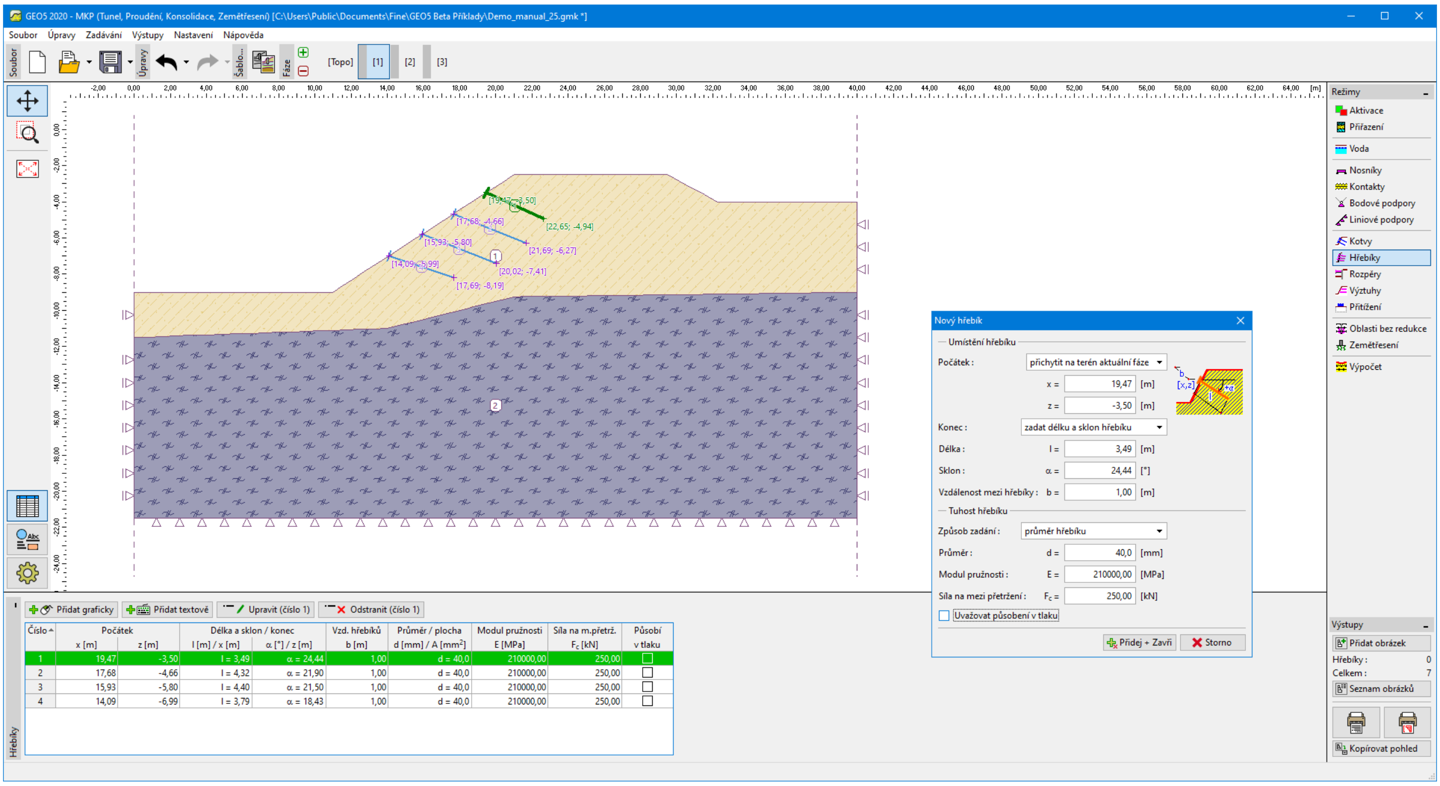
Modeling anchor in the slope stability analysis

Nails

The frame "**Nails**" contains a table with the list of nails. Adding nails is performed in the "**New nail**" dialog window. When inputting, it is possible to use the function of the grid.

The input nails can also be edited on the desktop with the help of active objects. The prograhm uses the following coordinate systems.

The nails heads are **automatically** catched on the terrain, an arbitrary interface or opening (tunnel lining). The nail can also be introduced directly by specifying coordinates of the two points too.



Frame "Nails"

Props

The frame "**Props**" contains a table with the list of anchors. Adding props is performed in the "**New props**" dialog window. When inputting, it is possible to use the function of the grid.

The input props can also be edited on the desktop with the help of active objects. The program employs the following coordinate systems.

The prop endpoints are **automatically connected** to the terrain, an arbitrary interface, or opening (tunnel lining). These points are then automatically positioned into the intersections of the prop line determined by the input points and the selected lines. The prop can also be introduced directly by specifying coordinates of the two endpoints.

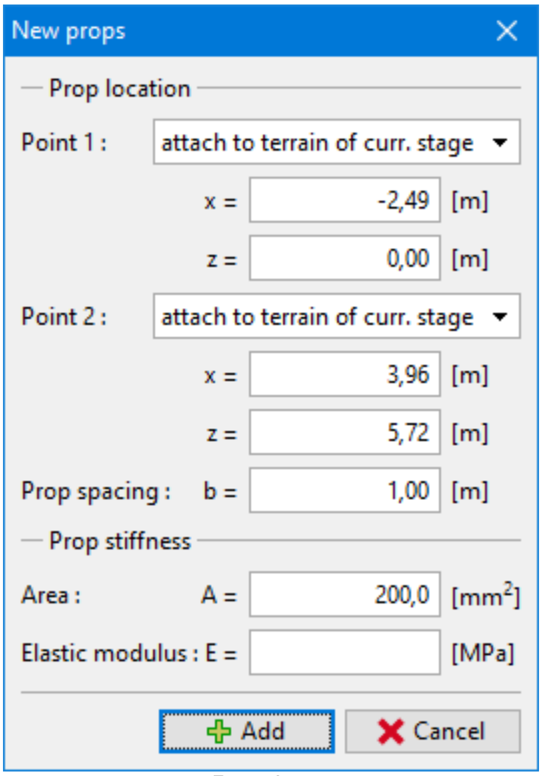
Props are represented by **elastic compressive bar elements** with constant normal stiffness. The props can sustain only a compressive load. When found in tension they are removed from the analysis.

The prop is linked to the finite element mesh in its two endpoints. No interaction is considered between the soil and the prop along its length when places into the soil.

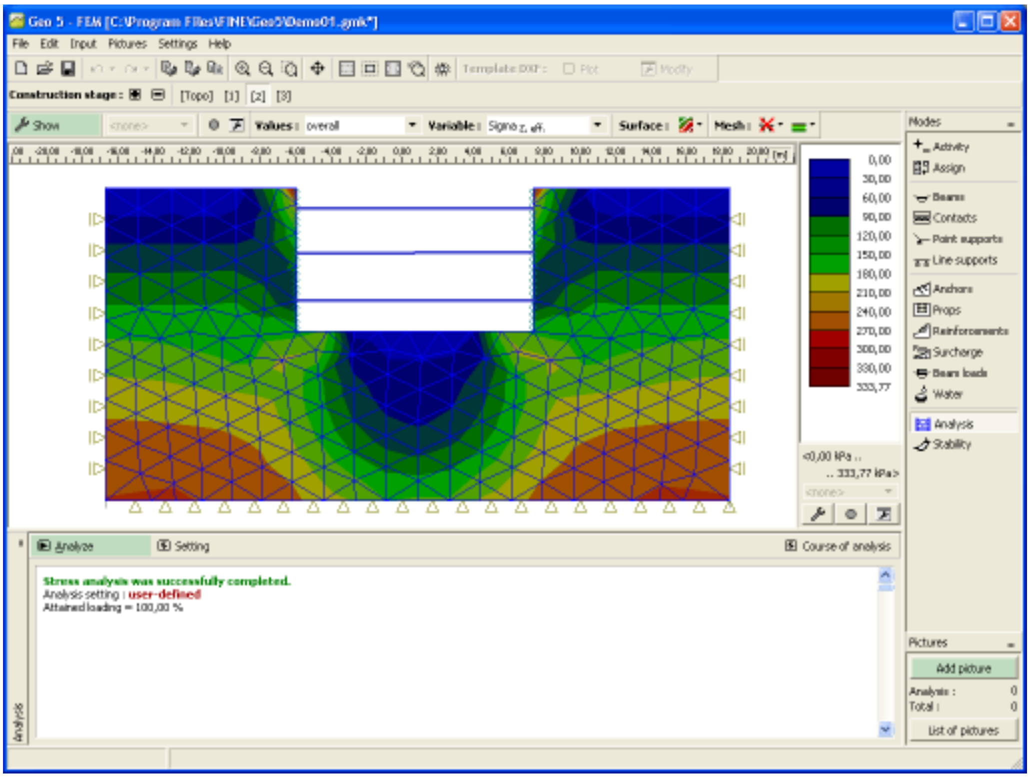
Props are defined by their starting and endpoints and by their stiffness. The program **automatically** links the **prop element degrees** of freedom to the actual degrees of freedom of the predefined finite element mesh. Therefore, the prop can be introduced anywhere in the structure.

The **prop stiffness** is specified in terms of the elastic modulus and its area. The program makes also possible to enter the prop diameter - the area is then determined automatically.

In subsequent stages the prop cannot be edited - it can be either removed or input again.



Prop input



Props - Analysis

Reinforcements

The frame "**Reinforcements**" contains a table with the list of reinforcements. Adding reinforcements is performed in the "**New reinforcements**" dialog window. When inputting, it is possible to use the function of the grid.

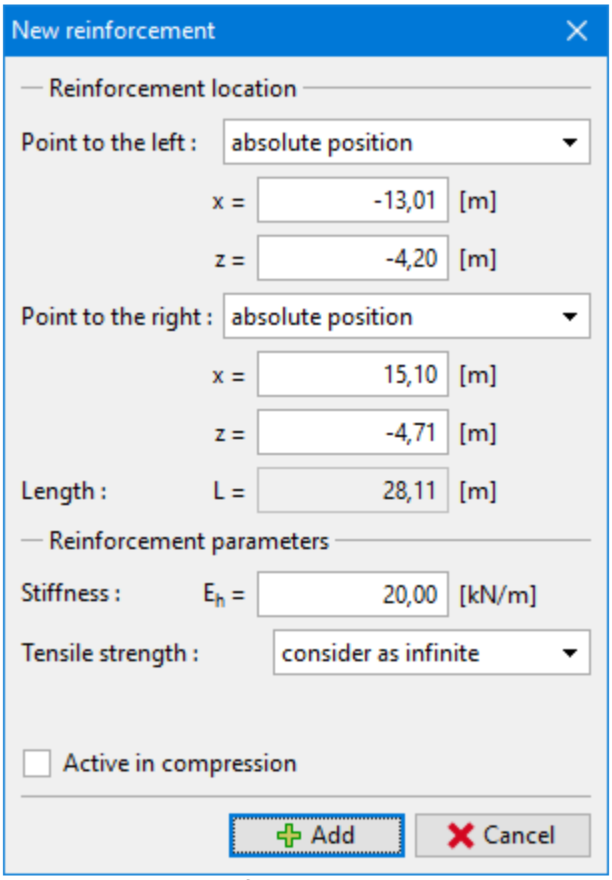
The input reinforcements can also be edited on the desktop with the help of active objects. The program employs the following coordinate systems.

The reinforcement endpoints are **automatically connected** to the terrain, an arbitrary interface, or opening (tunnel lining). These points are then automatically positioned into the intersections of the prop line determined by the input points and the selected lines. The reinforcement can also be introduced directly by specifying coordinates of the two endpoints.

Reinforcements are **tensile reinforcing elements** (geotextiles, geodrids), which are defined by their starting and endpoints and their stiffness.

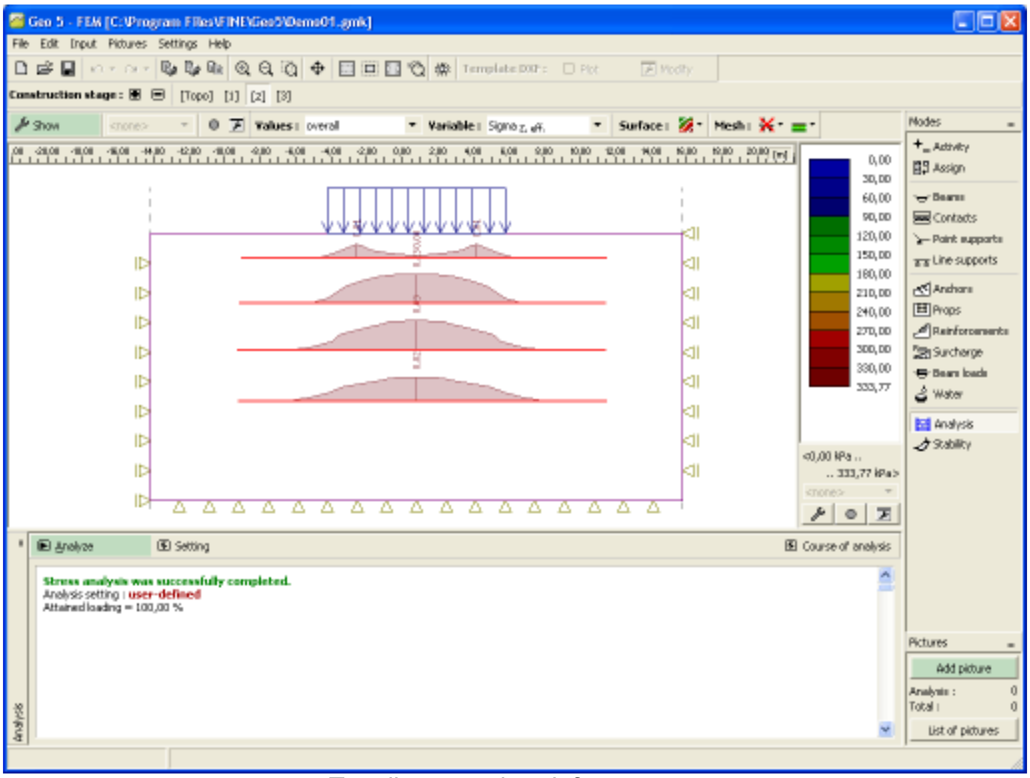
Unlike anchors or props, the reinforcement **is linked** to an underlying finite element mesh **along its entire length**. However, similar to anchors the program introduces the reinforcement endpoints into the finite element mesh automatically so the reinforcement can be specified anywhere within the mesh. Similar to anchors the reinforcement is modeled by a tensile/compressive bar element with the possibility of **transmitting only normal force**. Owing to its geometrical characteristics, the reinforcement calls for the input of the **cross-sectional stiffness taken per** 1 m (**foot**) **run of its width**. The user should contact the manufacturer for this information.

In subsequent stages the reinforcement cannot be edited - it can be only removed.



Reinforcement input

The program allows us to consider the reinforcement also in compression - by default, however, the **part of reinforcement found in compression is disabled** for the analysis. This state is simulated in the figure showing the distribution of normal tensile forces overactive parts of individual reinforcements. The compressive part of the reinforcement is **temporarily excluded** from the analysis. Similar to anchors, however, it can be automatically activated once loaded again in tension.

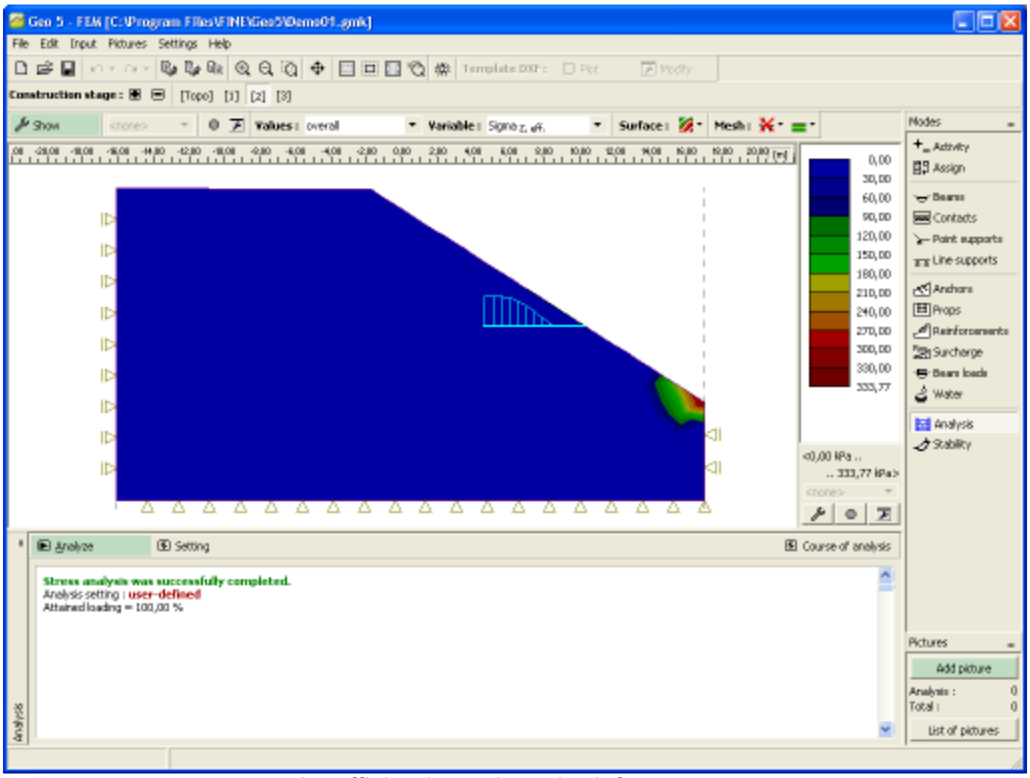


Tensile stress in reinforcements

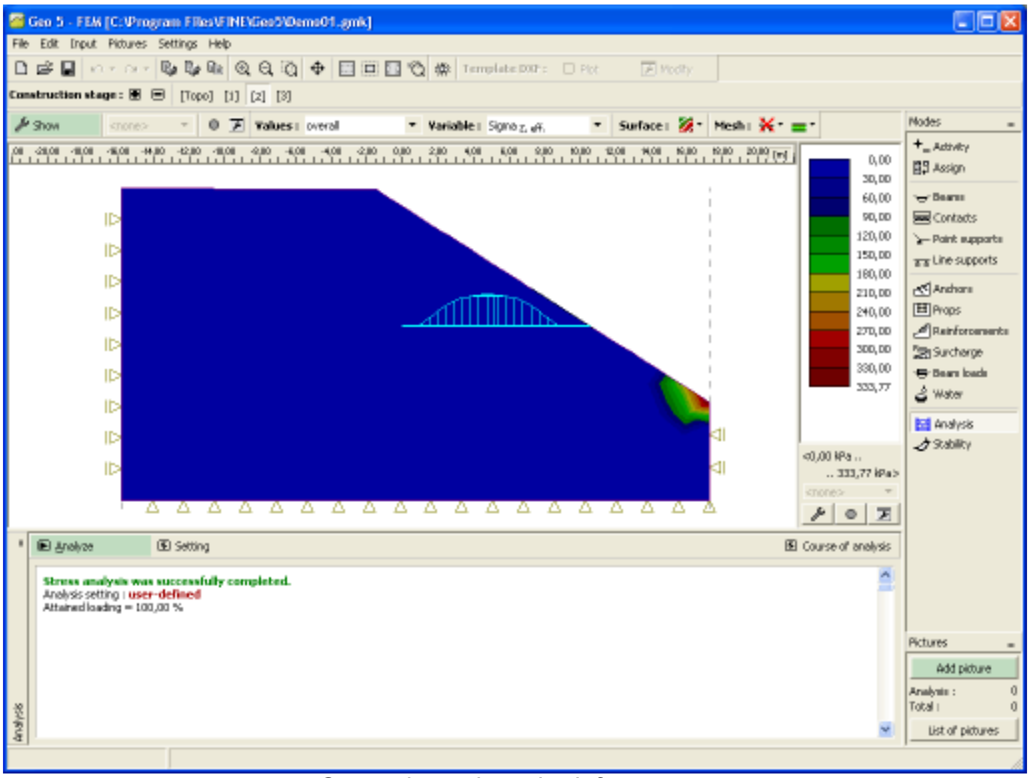
Anchoring Geo-Reinforcements

When introducing the reinforcement into the soil body it is necessary to keep in mind a **sufficient anchorage of the reinforcement** since the program **does not check the reinforcement against the shear failure**. A sudden increase of the normal force as shown in the figure suggests singularity in contact stresses and probable shear failure of the reinforcement. From that point of view, the displayed results are misleading and essentially unrealistic.

In such a case, the reinforcement should be either removed from the analysis or ensure its **sufficient anchorage** as plotted in the figure.



Insufficiently anchored reinforcement



Correctly anchored reinforcement

Axial Stiffness of Geosynthetics

Geosynthetics are tensile reinforcing elements (geotextiles, geogrids) defined by their starting and endpoints and by the axial (normal) stiffness J_z $[kN/m]$.

For **nonwoven fabrics,** the axial stiffness is usually not considered since these elements typically serve as separate layers. **Woven geotextiles** experience for small deformations very low initial stiffness - in the small strain region (up to 5%) we encounter a considerable increase of deformations under constant load.

When designing geotextiles this property must be taken into account. We thus recognize both the **long-term tension strength** in dependence on partial reduction factors (reflecting damage of elements caused by installation, creep behavior of geosynthetics, biological and chemical effects) and **initial normal stiffness** in the small strain region in the interval of 0.5% to 2%.

To determine the **minimum axial stiffness** of georeinforcements it is possible to use the following expression where for the strength corresponding to the selected strain we accept maximally 10% deviation from a linear part of tension test:

$$T_{z-x} \geq \frac{0{,}9 \cdot \varepsilon \cdot T_{max}}{\varepsilon_{max}}$$

where:

T_z	-	tensile strength at $x\%$ strain $[kN/m]$
ε	-	$x\%$ strain (relative extension) according to EN ISO 10 31 $[\%]$
T_{max}	-	maximal tensile strength according to EN ISO 10 319 $[kN/m]$
ε_{max}	-	maximal strain (relative extension) according to EN ISO 10 319 $[\%]$

Suppliers and producers of geotextiles typically provide the value of tensile strength at 2% strain. The expression then becomes:

$$T_{z-2\%} \geq \frac{1{,}8 \cdot T_{max}}{\varepsilon_{max}}$$

The **minimal** (initial) **axial stiffness** of geotextiles from a short-term experiment (load rate according to EN ISO 10 319) for x-$\%$-strain is given by:

$$J_{\varepsilon=x} \approx E \cdot A = \frac{T_{\varepsilon=x}}{\varepsilon}$$

where: ε - $x\%$-strain (relative extension) according to EN ISO 10 319 [-]

The **maximum** (theoretically attainable) **axial stiffness** of geotextiles for a short-term axial strength is determined as follows:

$$J_{\varepsilon\,max} \approx E \cdot A = \frac{T_{max}}{\varepsilon_{max}}$$

where: ε_{max} - maximum strain (relative extension) according to EN ISO 10 319 [-]

Intervals of recommended of values of axial (normal) stiffnesses of geosynthetics J_z $[kN/m]$ are listed in the following table:

Variable description	**Initial axial stiffness of geotextiles for** $\varepsilon = 2\%$	**Theoretical (maximal) axial stiffness of geotextile**
Notation (unit)	$J_{\varepsilon=x}$ $[kN/m]$	$J_{\varepsilon max}$ $[kN/m]$
Georeinforcements category	---	---
Non-woven geotextiles	-	-
Woven geotextiles	250 ÷ 500	1000
Unaxial geogrids	500 ÷ 1000	1500
Biaxial geogrids	100 ÷ 500 for $\varepsilon = 0.5\%$	2500
Triaxial geogrids	250 ÷ 500 for $\varepsilon = 0.5\%$	5000
Geomats	100 ÷ 500	1000
Drainage geocomposites	-	-
Composites	100 ÷ 500	1500
Geomeshes	-	-
Geocells	-	-

Literature:

GEOMAT ltd. (www.geomat.cz): Types of geotextiles and their function in civil engineering structures. Author: Martin Kašpar (kaspar@geomat.cz). In Czech.

HOLÝ, O., MIČA, L.: Determination of axial stiffness of geosynthetics for numerical modeling - part 1. TU Brno (paper in conference proceedings „Civil engineering structures in view of geomechanics“). In Czech.

EN ISO 10 319 (80 6125): Geotextiles - Tensile test on a wide strip. Czech standard institute, 2009. In Czech.

Surcharge

The frame "**Surcharge**" contains a table with the list of surcharges. Adding surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can also be edited on the desktop with the help of active objects. The program employs the following coordinate systems.

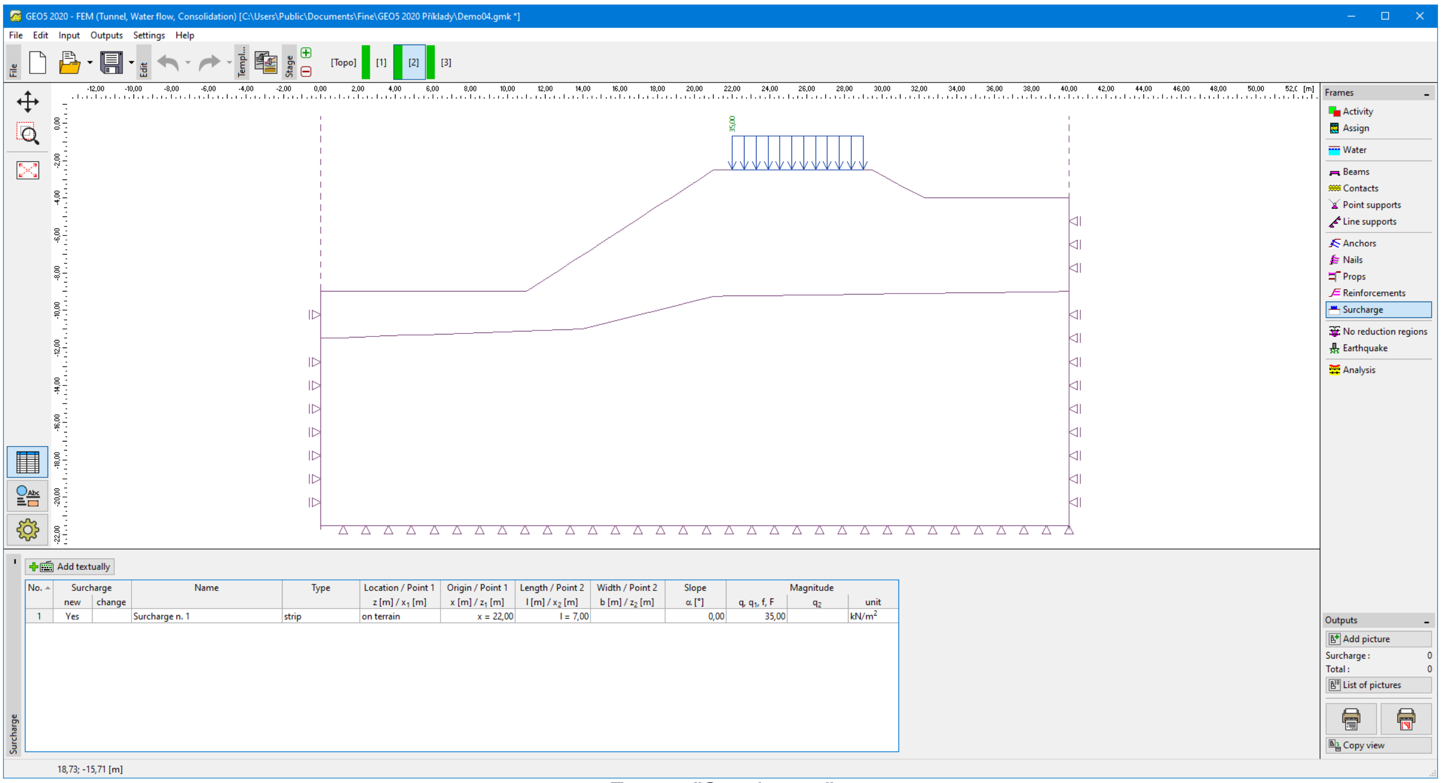
All input parameters of the surcharge can be modified in the stage of construction, in which the surcharge was introduced. In subsequent stages, it is only possible to modify its magnitude (option "**Adjust magnitude**").

This frame serves to introduce **surcharges applied only to the soil body**. The surcharge applied to a beam element is introduced in the frame - beam load.

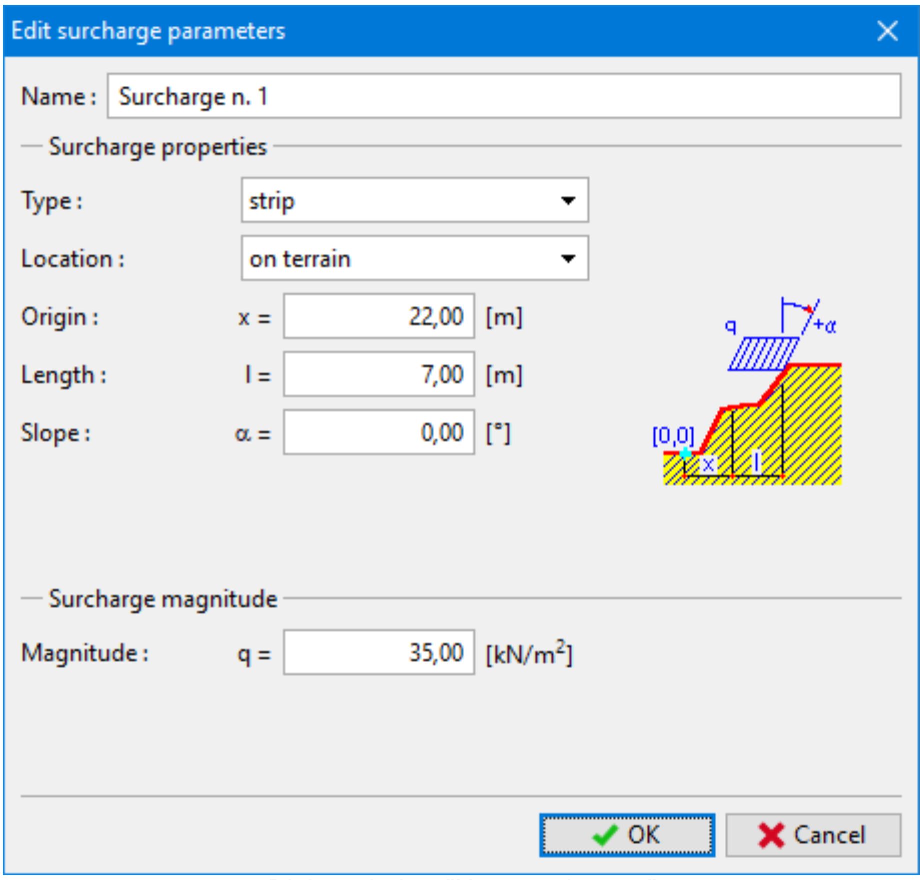
An arbitrary number of surcharges can be specified in individual stages. The surcharge may act either on the **existing interface** (including ground surface) or can be applied **anywhere in the soil body**.

In subsequent stages, we are free to either remove the input surcharge or to **modify its magnitude**.

Note that applying the surcharge directly on the ground surface may lead to **excessive plastic deformations** in the vicinity of the surcharge and the analysis may fail to converge. In such a case, one may either place a **beam element** under the applied surcharge or choose an **elastic** or modified elastic material model for the soil below the surcharge.



Frame "Surcharge"



Dialog window "New surcharges"

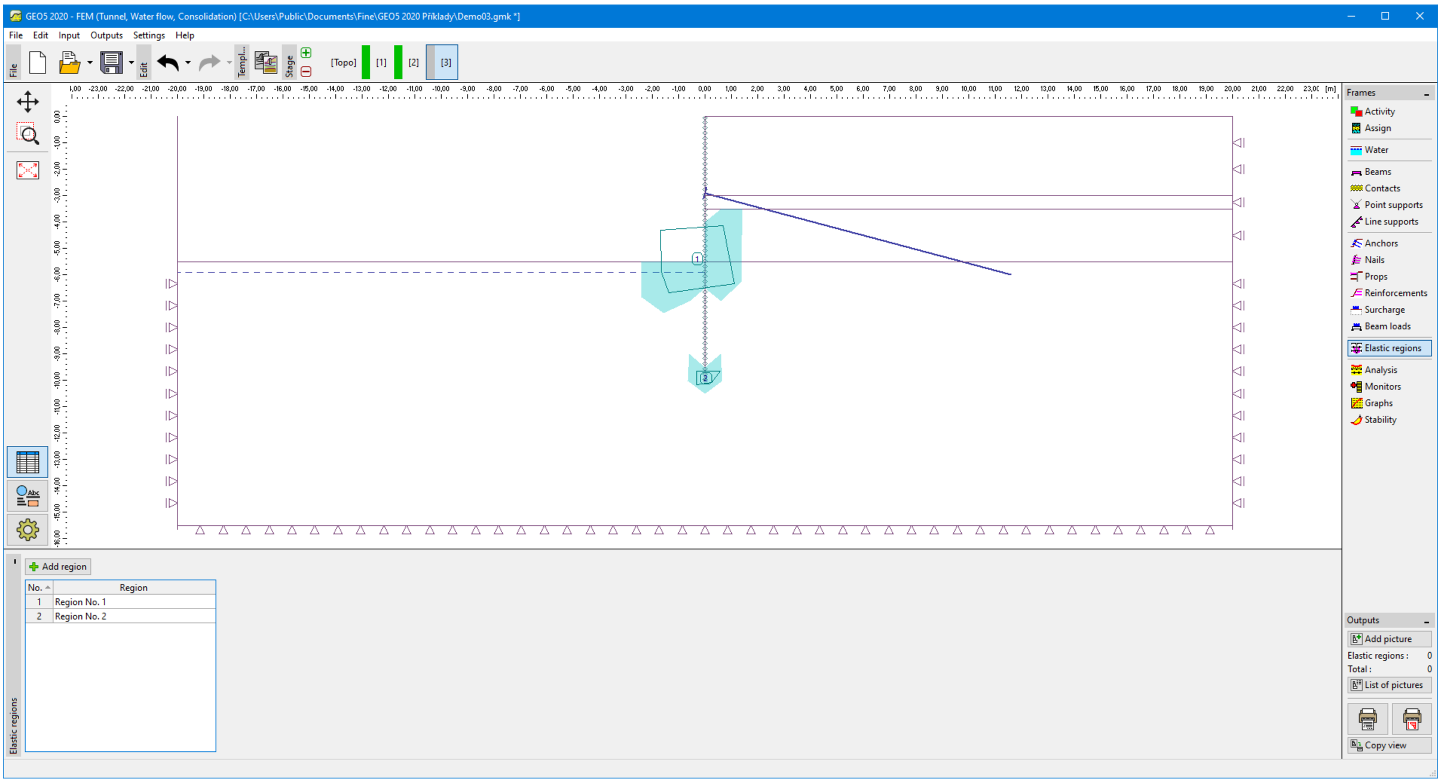
Elastic Regions

Frame "**Elastic regions**" contains a table of defined elastic regions.

Input of elastic regions is identical to the standard input of interfaces.

When loaded by stress reaching yield point the soil exhibits deformation which pertains also after unloading. This persistent deformation is termed plastic strain and can be modeled by standard models such as the Mohr-Coulomb or Drucker-Prager models or by the advanced models such as the Cam clay.

When we want to suppress the evolution of plastic strain in a chosen area and construction stage we can use the "**Elastic regions**" feature. Elastic regions are specified in teh given stage by a polygon. Each element that - at least partially - falls within this region allows for elastic strains only.



Frame "Elastic regions"

Elastic regions affect only the behavior of:

- Mohr-Coulomb model
- Modified Mohr-Coulomb model
- Drucker-Prager model

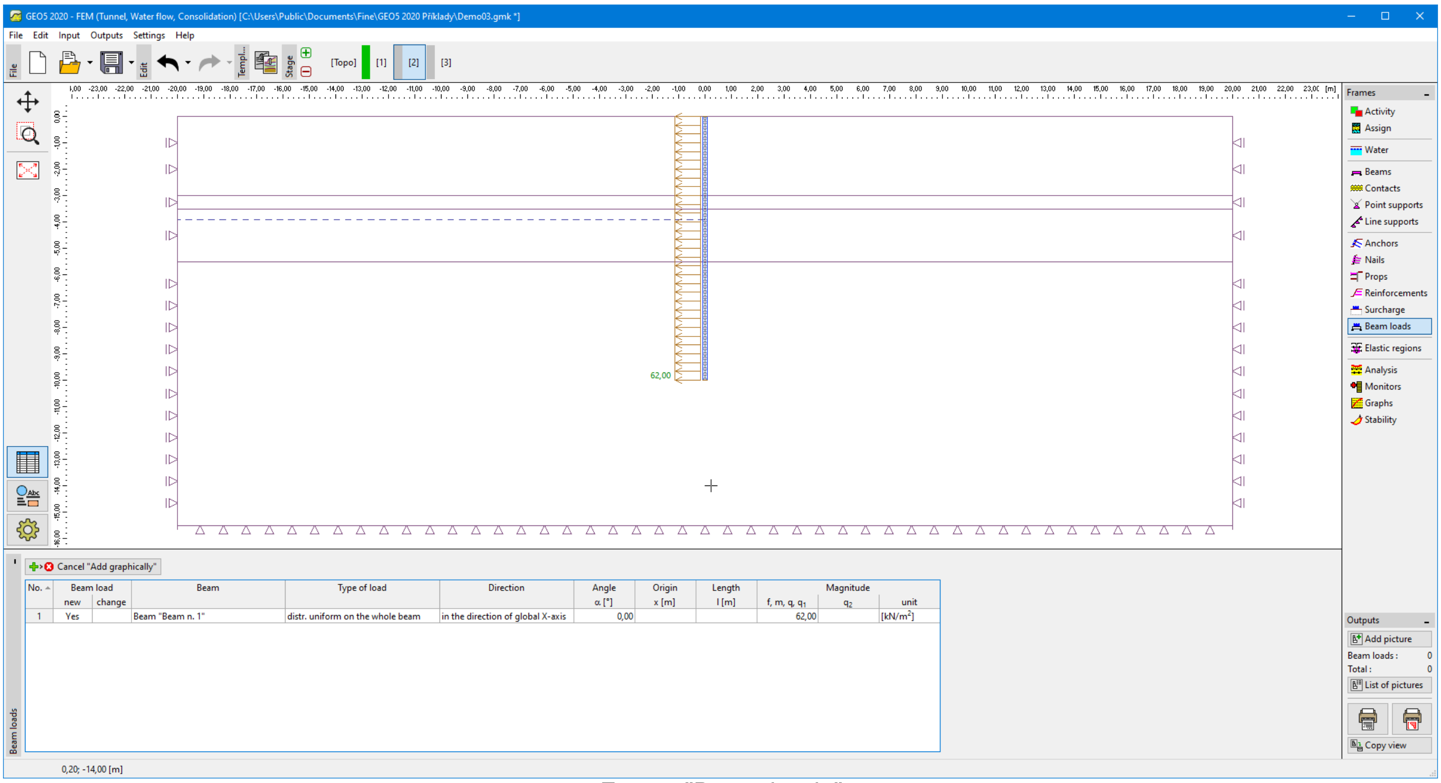
The behavior of other material models remains unaffected.

Beam Loads

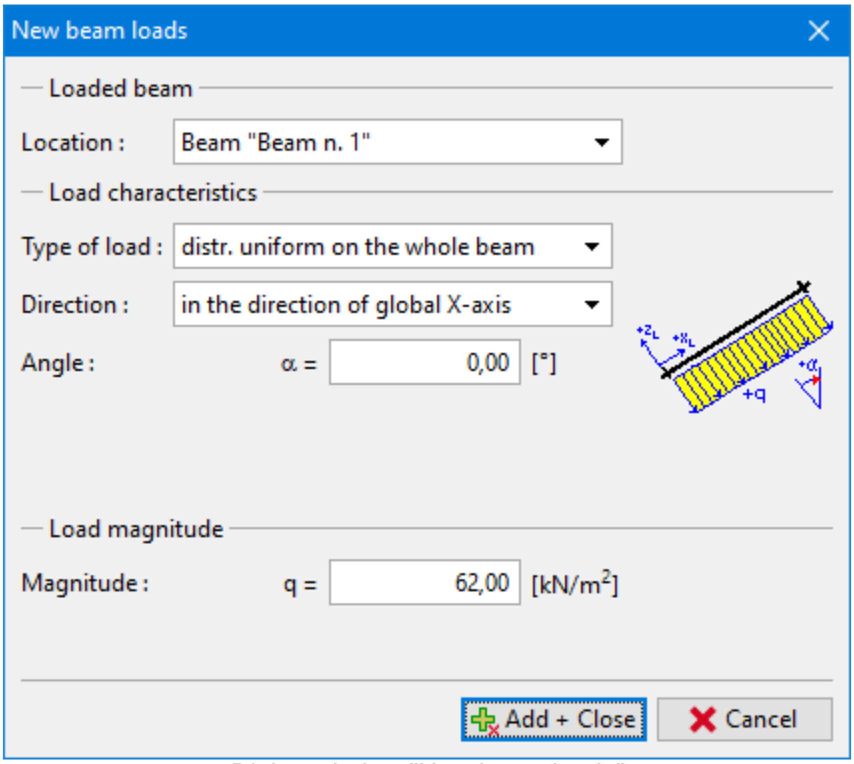
The frame "**Beam loads**" contains a table with the list of loads. Adding beam loads is performed in the "**New beam loads**" dialog window.

The input loads can also be edited on the desktop with the help of active objects. The program employs the following coordinate systems.

All input parameters of the load can be modified in the stage of construction, in which the load was introduced. In subsequent stages, it is only possible to modify its magnitude (option "**Adjust magnitude**").



Frame "Beam loads"



Dialog window "New beam loads"

Water

There are three options in the program to introduce groundwater:

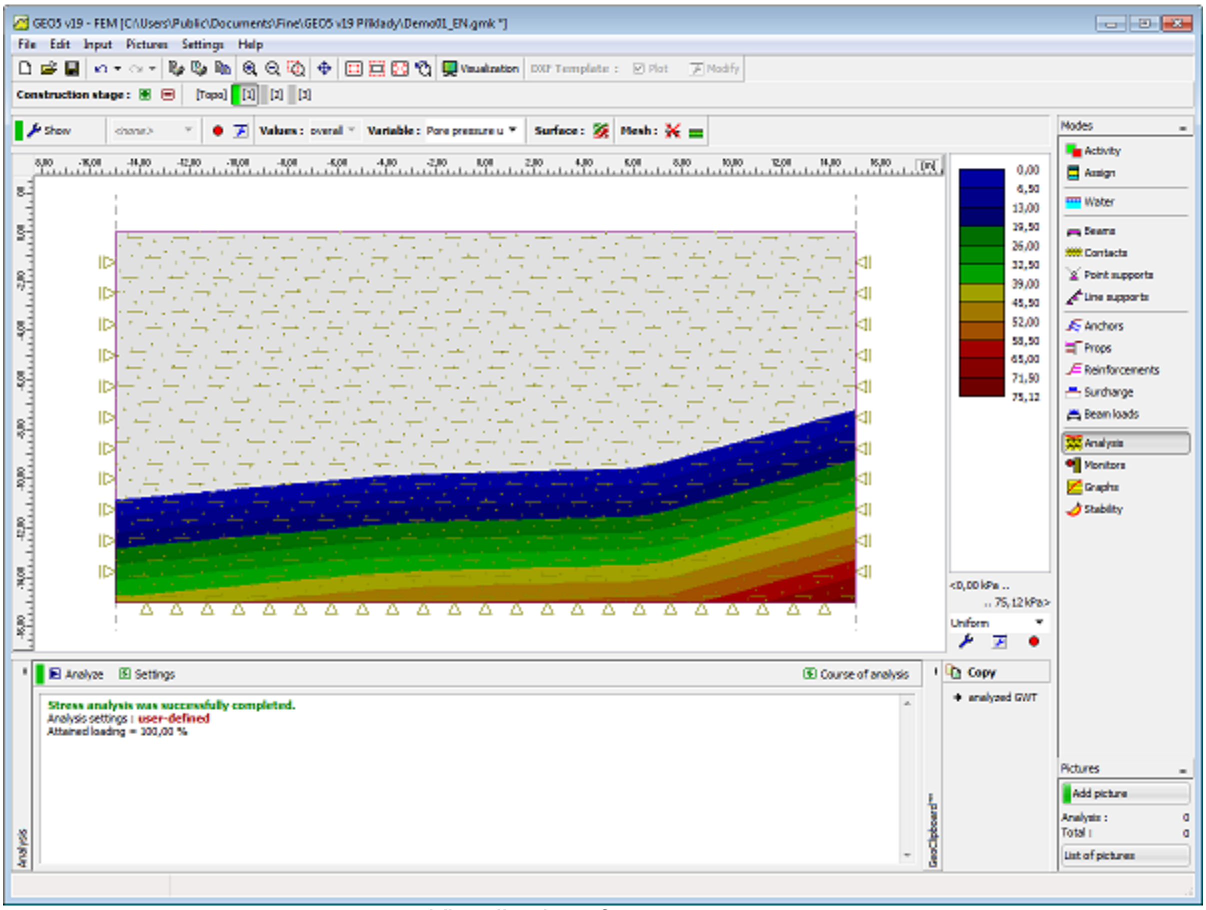
- The **groundwater table** can be specified as a continuous interface below and above the ground surface. In such a case, the program automatically adjusts the soil self-weight below the groundwater table.
- The **pore pressure** values are entered via isoline. Input is the same as interface input. The pore pressure values are inserted into the table "List of interfaces" in the left bottom part of the screen. The values between isolines follow from linear interpolation.

- **The pore pressure coefficient** r_u represents the ration between pore pressure and the geostatic stress in the soil. The values of the coefficient r_u are specified for individual isolines. The first isoline always coincides with the ground surface. The remaining isolines are introduced in the same way as interfaces between individual soil layers. The values are inserted into the table "List of interfaces" in the left bottom part of the screen. The values between isolines follow from linear interpolation.

When entering the values of pore pressure or the values coefficients r_u the **unit weight of soil is assumed in the entire body** to be equal to the unit weight γ regardless of the values of pore pressures or coefficients r_u .

The simplest way to check the input of water is to plot the distribution of **pore pressure** in the output window.

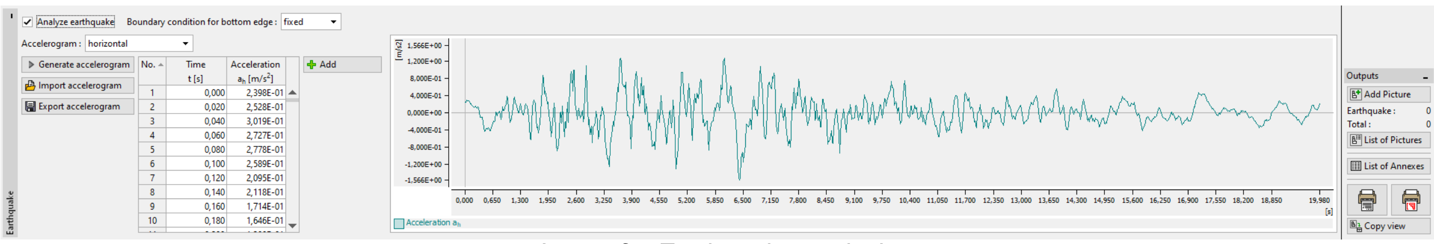
Input interfaces of water can be copied within all 2D GEO5 programs using "GeoClipboard".



Visualization of pore pressure

Earthquake

The "**Earthquake**" frame serves to input earthquake parameters for the construction stage.

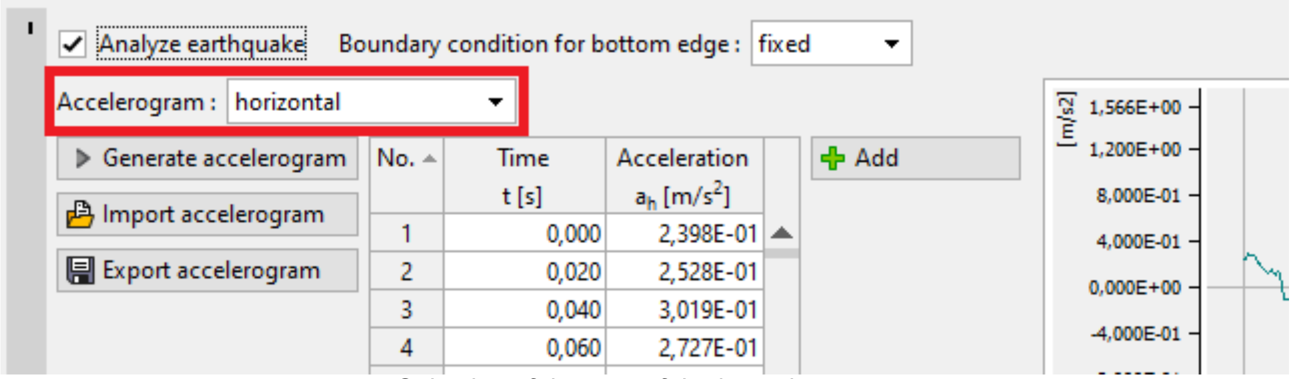


Inputs for Earthquake analysis

For the earthquake analysis it is necessary to:

- Choose the type of incoming wave: only horizontal movement (shear wave), only vertical movement (pressure wave) or a combination of both;
- Select the type of **boundary conditions**;
- Specify the **accelerogram** (the time history of acceleration) of the incoming wave.

The **accelerogram** (wave propagation) **is always introduced on the bottom boundary of the numerical model**. The user should accord a particular attention to this issue.



Selection of the type of the incoming wave

Further details are available in the **theoretical manual** on our website.

Boundary Conditions in Dynamic Analysis of Earthquake

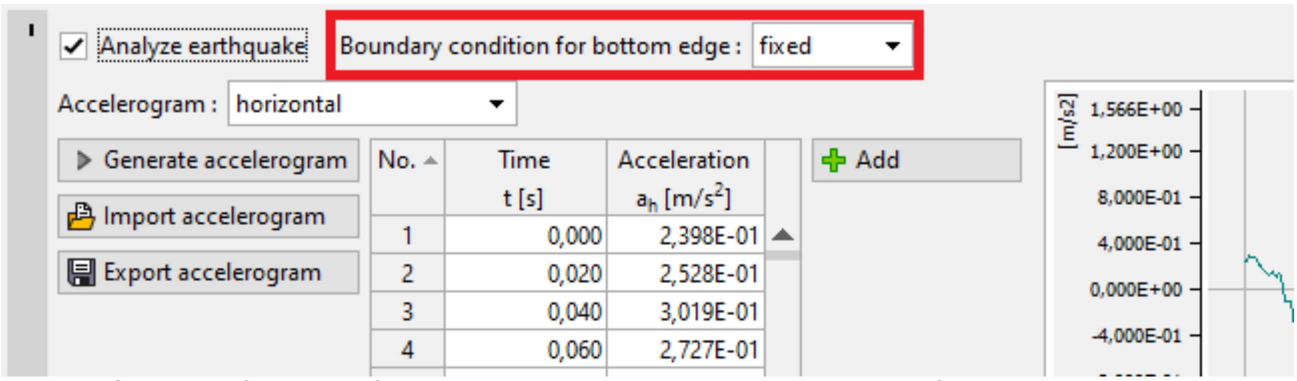
The "**Earthquake**" frame allows for setting either the **fixed** (kinematic) or **absorbing** (traction) **boundary condition on the bottom boundary of the numerical model**.

Fixed boundary condition:

Reflects the outgoing waves back into the model domain. This corresponds to the case when the model is truncated at the interface between a relatively soft soil layer and a stiff bedrock.

Absorbing boundary condition:

Transmits the outgoing wave through the model bottom boundary. Because such a wave will never return into the model, it corresponds to a dashpot, which damps this wave at this boundary. This condition is suitable whenever the model is truncated within a soil/rock layer. The material model of this layer should be elastic. The nonlinearity should be confined to top layers only.



Selection of the type of boundary conditions on the bottom boundary of the numerical model

Further details are available in the **theoretical manual** on our website.

Accelerogram

Modeling the seismic actions in the FEM program requires the knowledge of the time evolution of acceleration at the bottom boundary being represented by an **accelerogram** - a record of time instances and corresponding values of instantaneous acceleration.

Three options to specify accelerograms in the "**Earthquake**" frame are available:

1. **Generate a synthetic accelerogram** with corresponding response spectrum;
2. **Upload the recorded accelerogram** from an external data file;
3. **Create own** accelerogram

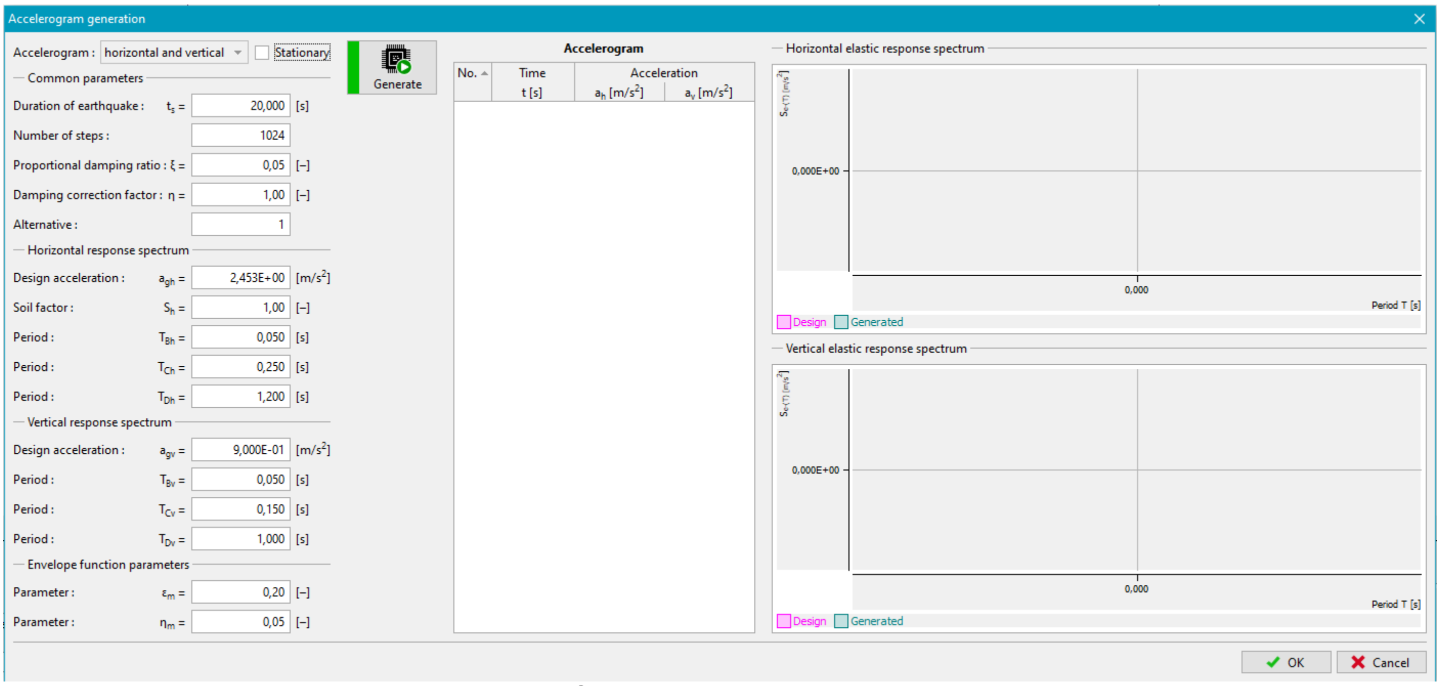
The accelerogram is interpreted as a piecewise linear distribution of acceleration. The user should ensure (verify) that the uploaded or defined accelerogram results in zero value of the velocity at the end of the record.

The generated accelerograms comply with this condition.

Generating Accelerogram

The GEO5 FEM - Earthquake module allows the user to generate an artificial accelerogram corresponding to the desired response spectrum defined in the Eurocode 8 standard.

The "**Generate accelerogram**" option in "**Earthquake**" frame prompts the "**Generate accelerogram**" dialogue window to specify parameters of the generated acceleration record.



Generating accelerogram

The generated accelerogram follows a typical shape of the acceleration record: stage of gradual increase, stage of strong motion and stage of gradual dying away. Providing the "**Stationary**" option is checked, neither the initial nor the final stage is taken into account and the amplitudes of acceleration are more or less constant in time.

Duration of earthquake t_s determines the overall span of the generated record.

Proportional damping ratio ξ specifies proportional damping. This value is used in the determination of the elastic response spectrum of the generated accelerogram.

Damping correction factor appears in the parametric description of response spectrum provided by the EN 1998-1 standard. Its value equals 1 for damping ratio $\xi =$ 0.05.

For different values of ξ it holds:

$$\eta = max\left(\sqrt{\frac{0.1}{\xi + 0.05}}; 0.55\right)$$

When adopting generated accelerograms, EN 1998-1 requires application of at least three different records. Changing the parameter "**Alternative"** allows us to receive a different record with a similar (standard) response spectrum.

In addition, it is necessary to specify the parameters pertinent to either **horizontal or vertical elastic response spectrum**.

Once specifying all parameters and then choosing "**Generate**" the program generates the acceleration record. Upon accepting by choosing "**OK**" the acceleration record is displayed in the "**Earthquake**" frame.

Literature:

EN 1998-1: Design of structures for earthquake resistance - Part 1: General rules, seismic actions and rules for buildings, (2006).

Type of Subsoil - EN 1998-1

The type of subsoil reflects the influence of local conditions on seismic loading. According to the EN 1998-1 standard we distinguish the following types: A - E, S1 and S2 with properties specified in table below.

Type of subsoil from seismic load point of view according to EN 1998-1:

Ground type	Description of stratigraphic profile	Parameters		
		$V_{s,30}$	N_{SPT} [blows/30 cm]	c_u [kPa]
A	Rock or other rock-like geological formation, including at most 5 m of weaker material at the surface.	> 800	-	-

B	Deposits of very dense sand, gravel, or very stiff clay, at least several tens of meters in thickness, characterised by a gradual increase of mechanical properties with depth.	360 - 800	> 50	> 250
C	Deep deposit of dense or medium-dense sand, gravel or stiff clay with thickness from several to many hundreds of meters.	180 - 360	15 - 50	70 - 250
D	Deposits of loose-to-medium cohesionless soil (with or without some soft cohesive layers), or of predominantly soft-to-firm cohesive soil.	< 180	< 15	< 70
E	A soil profile consisting of a surface alluvium layer with v_s values of type C or D and thickness varying between about 5 m and 20 m, underlain by stiffer material with v_s > 800 m/s.			
S_1	Deposits consisting, or containing a layer at least 10 m thick, of soft clays/silts with a high plasticity index ($PI >$ 40) and high water content.	< 100 (indicative)	-	10 - 20
S_2	Deposits of liquefiable soils, of sensitive clays, or any other soil profile not included in types A - R or S_1 d.			

Literature:

EN 1998-1: Design of structures for earthquake resistance - Part 1: General rules, seismic actions and rules for buildings, (2006).

Elastic Response Spectrum - EN 1998-1

The EN 1998-1 standard considers the **elastic response spectrum** as the basic representation of the seismic load. The spectrum is defined with the help of several basic parameters a_g, T_B, T_C, T_D, S and ξ. The design acceleration a_g can be acquired from the maps of seismic regions. The proportional damping ratio ξ determines the damping of seismic waves in the soil/rock layers. Its value is typically assumed to be equal to 5%. The values of the remaining four parameters are specified according to the **type of subsoil** (types A - F) and the type of the required spectrum (1 and 2).

The values of parameters describing the horizontal elastic response spectrum of type 1 according to EN 1998-1

Ground type:	S	T_B [s]	T_C [s]	T_D [s]
A	1.00	0.15	0.40	2.00
B	1.20	0.15	0.50	2.00
C	1.15	0.20	0.60	2.00
D	1.35	0.20	0.80	2.00
E	1.40	0.15	0.50	2.00

The values of parameters describing the horizontal elastic response spectrum of type 2 according to EN 1998-1

Ground type:	S	T_B [s]	T_C [s]	T_D [s]
A	1.00	0.05	0.25	1.20
B	1.35	0.05	0.25	1.20
C	1.50	0.10	0.25	1.20
D	1.80	0.10	0.30	1.20
E	1.60	0.05	0.25	1.20

The values of parameters describing the vertical elastic response spectrum to EN 1998-1

Spectrum	a_{vg}/a_g	T_B [s]	T_C [s]	T_D [s]
Type 1	0.90	0.05	0.15	1.00
Type 2	0.45	0.05	0.15	1.00

Literature:

EN 1998-1: Design of structures for earthquake resistance - Part 1: General rules, seismic actions and rules for buildings, (2006).

Material Parameters in Earthquake Analysis

The material parameters of soils needed in earthquake analysis are specified in the "**Edit soil properties**" dialogue window in Topology regime, the frame "**Soils**".

The dynamic modulus of elasticity corresponds to the elastic modulus at small strains. Its value is typically determined on the basis of field or laboratory measurements. **Point out that the critical state constitutive models do not adopt the dynamic modulus** in the formulation of the stiffness matrix.

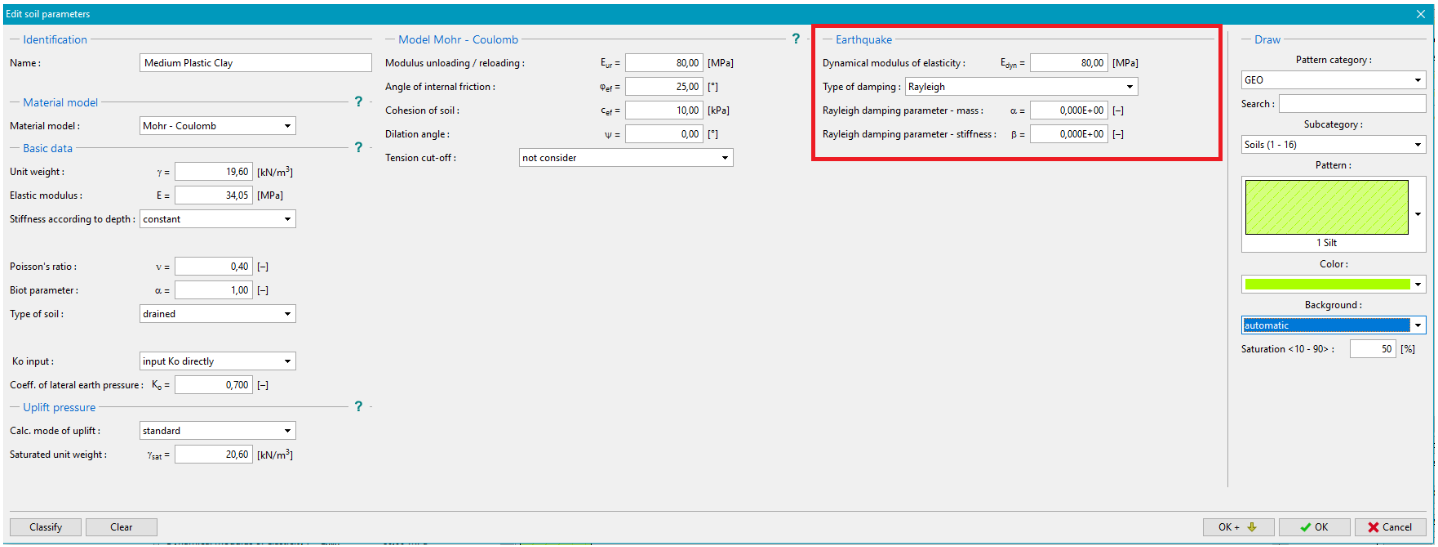
Other parameters describe the material damping. In the GEO5 FEM program the **Rayleigh damping** is assumed. The damping matrix C is then defined as a linear combination of the mass matrix M and the stiffness matrix K as:

$$\mathbf{C}^{\mathbf{M}} = \alpha\mathbf{M} + \beta\mathbf{K}$$

kde: $\boldsymbol{C}$ - damping matrix
$\boldsymbol{M}$ - mass matrix
$\boldsymbol{K}$ - stiffness matrix
$\boldsymbol{\alpha}$ - Rayleigh parameter
$\boldsymbol{\beta}$ - Rayleigh parameter

The Rayleigh parameters:

- **are determined** on the basis of the specified coefficient of proportional damping;
- **can be specified directly**.



Material parameters of soils needed for Earthquake analysis

Further details are available in the **theoretical manual** on our website.

Rayleigh Damping Parameters

When the coefficient of proportional damping ξ is specified (in frame "**Soils**") for some material the program will compute the Rayleigh damping parameters $\boldsymbol{\alpha}$ and $\boldsymbol{\beta}$ using one of the following two methods:

- The chosen eigenfrequency ω_a is assumed to be the least damped frequency associated with the damping coefficient ξ. Then the Rayleigh parameters are calculated as:

$$\alpha = \xi\omega_a$$

$$\beta = \frac{\xi}{\omega_a}$$

- The least damped frequencies appear in the interval, which is bounded by frequencies ω_a and ω_b ($\omega_b > \omega_a$) both damped by damping coefficient ξ. The Rayleigh parameters then follow from:

$$\alpha = \frac{2\xi\omega_a\omega_b}{\omega_a + \omega_b}$$

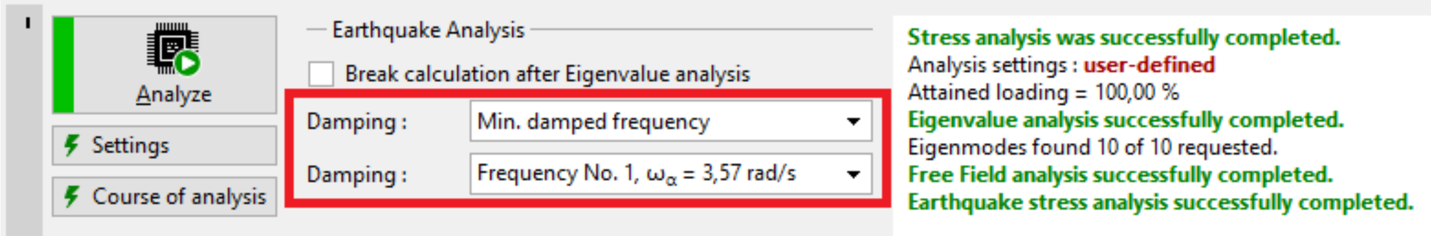
$$\beta = \frac{2\xi}{\omega_a + \omega_b}$$

The damping coefficient for an arbitrary eigenfrequency ω then reads:

$$\xi(\omega) = \frac{1}{2}\left(\frac{\alpha}{\omega} + \beta\omega\right)$$

This relationship ensures that the frequencies between ω_a and ω_b are damped less and the frequencies outside the interval are damped more than ω_a and ω_b.

Selection the approach for damping in "**Analysis**" frame.



Selection of frequencies for damping

Further details are available in the **theoretical manual** on our website.

Analysis

The analysis is performed for individual calculation stages in the frame **"Analysis"** after pressing the **"Analyze"** button. We recommend **a more powerful** 64-bit version for complex tasks.

During analysis, the program attempts to arrive at such a solution that satisfies for given load and boundary conditions of the **global equilibrium.** In most cases, this step results in an iterative process. The process of iteration and convergence of the solution is displayed on the screen.

The analysis can be stopped at any time by pressing the **"Interrupt"** button. The results are then available for the last converged load increment.

The correct results are obtained when **100% of the applied load is reached**. Due to convergence failure, the program may stop **before reaching the desired load level** - only a fraction of the total applied load is reached. In such a case it is possible to adjust standard parameters of the analysis setting.

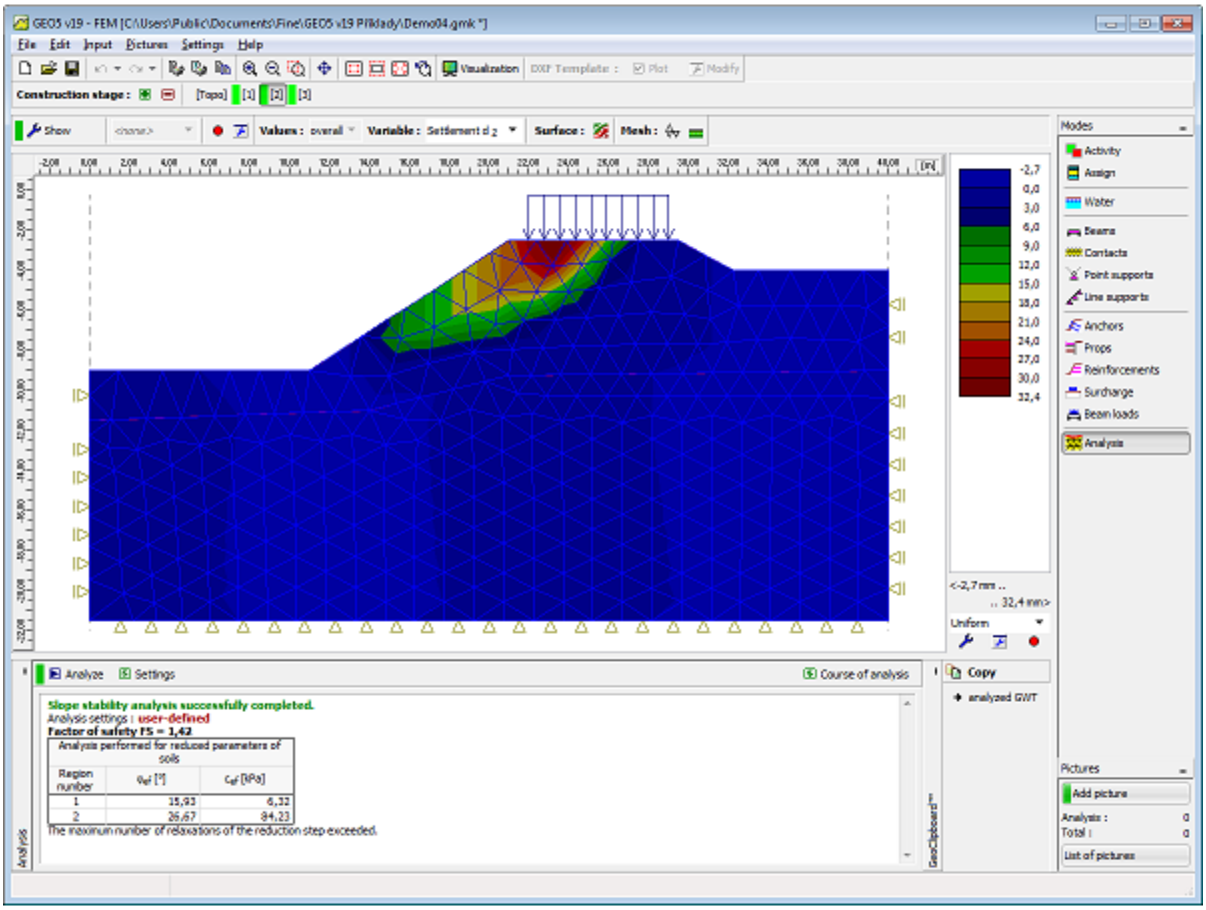
When modeling more complex engineering tasks we encourage the user to follow the recommended modeling procedure.

The transient flow analysis can be selected in the frame "Settings".

The analysis results together with information about the course of analysis appear on the screen immediately after completing the analysis.

Detailed information about the actual modeling approach is presented in section "Setting and analysis description". Visualization of results can be adjusted in the frame "Drawing Settings".

In case of considering water in analysis, in most cases, there's a possibility to copy analyzed GWT to GeoClipboard and paste it into another program.



Screen after completing the analysis

Transient Flow Analysis

The actual analysis proceeds in two and more stages ("Water flow"), where the first stage serves to set the initial conditions, i.e. the distribution of initial pore pressure, initial pressure head, degree of saturation, and relative permeability at the onset of transient flow analysis. Several options are available to set the initial pore pressure:

- With the help of a groundwater table
- Directly with the help of pore pressure interfaces
- Running the steady-state flow analysis

The first option assumes a hydrostatic (linear) distribution of pore pressure over the height. Below GWT the program generates positive pore pressures, whereas above GWT the negative pore pressures (suction) are generated. The second option allows for considering a dry soil by prescribing e.g. negative pore pressures over the entire infiltrated region. The third option requires running the steady-state analysis. Based on the assigned material model the program then determines the initial degree of saturation and relative permeability as a function of the initial pore pressure. Figure 1 shows the distribution of initial pore pressure provided by steady-state analysis for the assumed hydraulic conditions. Clearly, only the pressures below GTW are presented. The initial state in an unsaturated or partially saturated region can be partially judged by plotting, for example the distribution of initial degree of saturation as seen in Figure 2. When selecting the "**No water**" option the initial pore pressure values are set equal to zero.

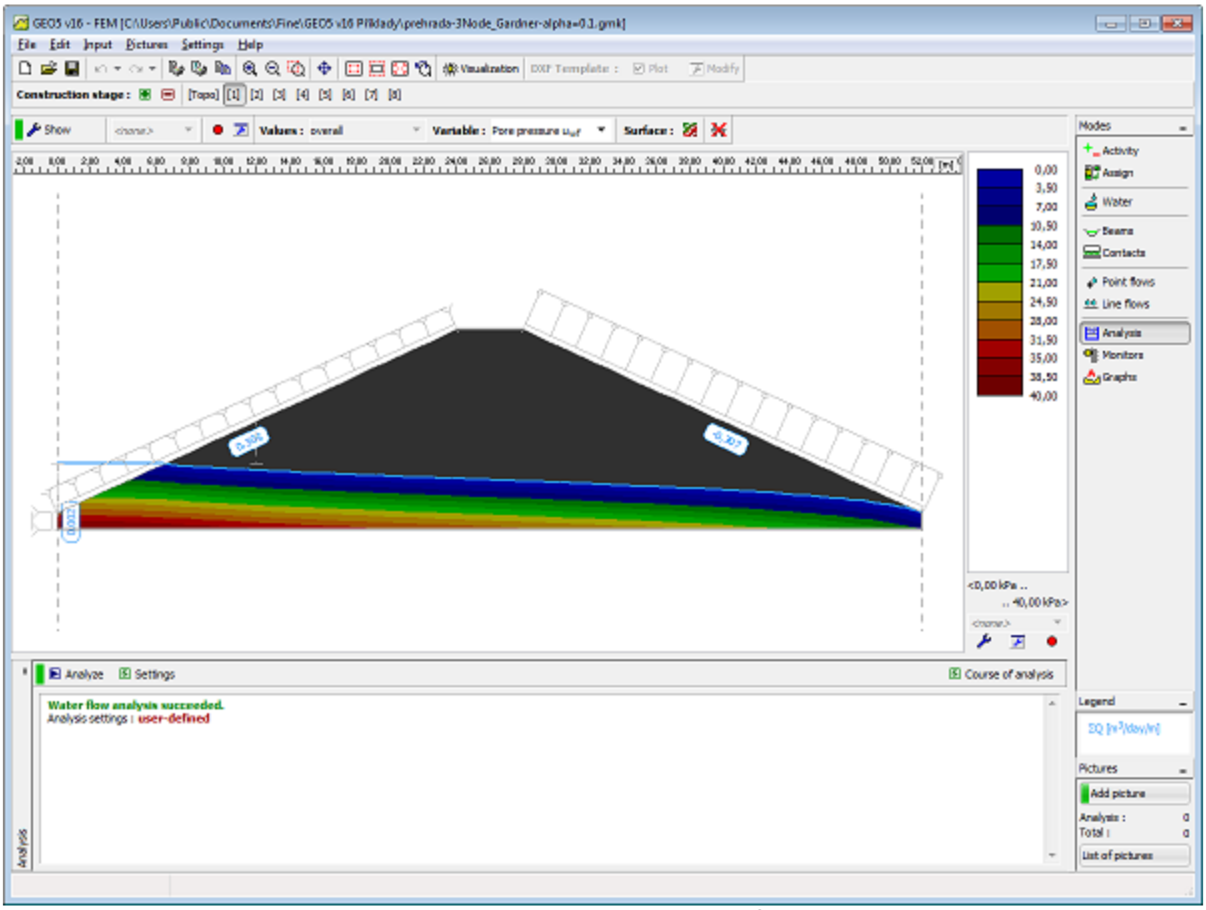


Figure 1 - First calculation stage: Distribution of initial pore pressure

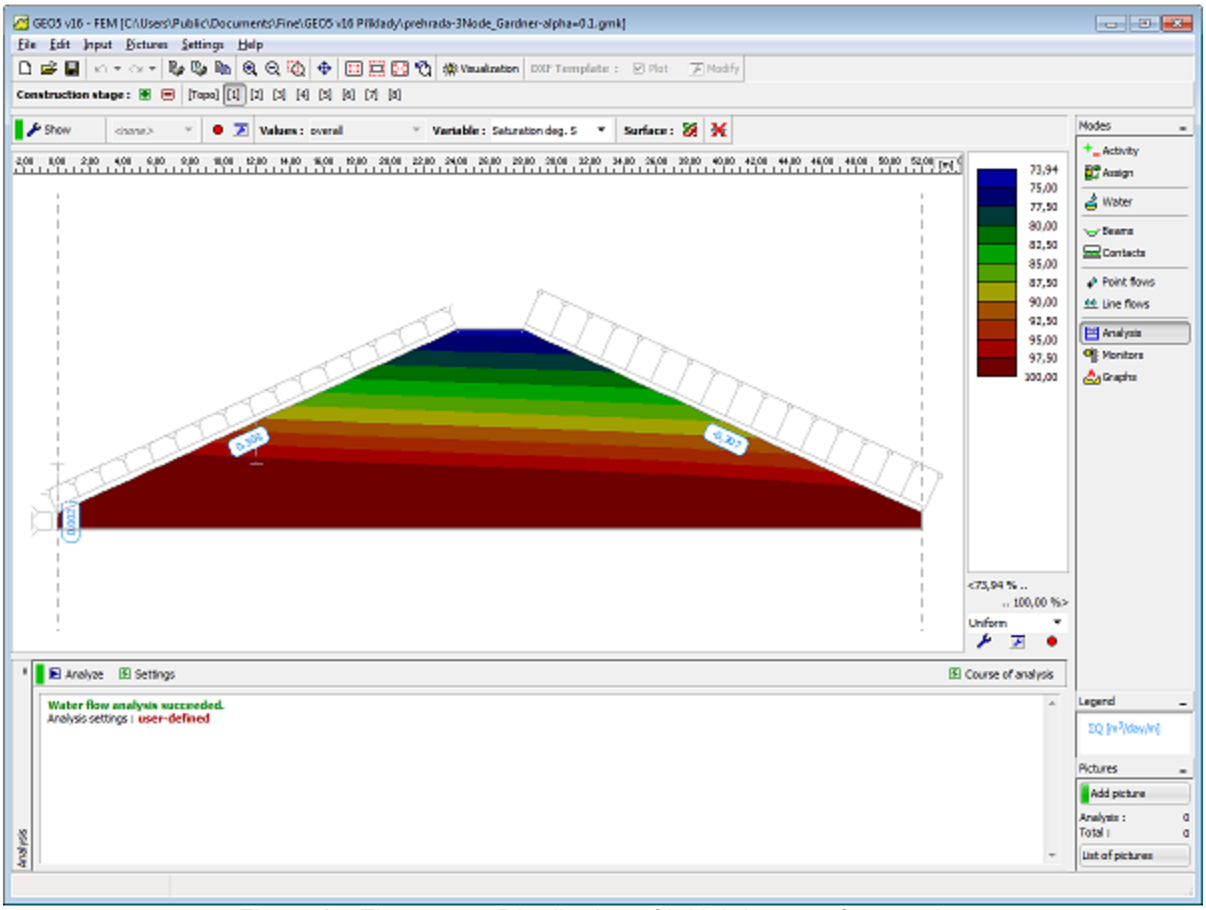


Figure 2 - First stage: Distribution of initial degree of saturation

The transient flow analysis is performed from the second stage on where the next stage follows the preceding one. Each stage requires setting the analysis time, time-dependent variation of boundary (hydraulic) conditions, and the time step length. The current version of the program allows us to either introduce the entire load at once at the beginning of the calculation stage or to assume that it linearly increases with time during the course of stage calculation ("Water flow"). In the first case, the initial time step is set to 1/10 of the assigned time step. Next, the calculation continues with the assigned time step. It is reasonable to adjust the time step during the course of analysis. A shorter time step is recommended at the beginning of the analysis. With longer times, when the solution approaches the steady-state conditions, the time step can be increased considerably (e.g. from 1/10 of the day, up to several days). Figures 3 and 4 display an intermediate state and a steady-state solution, respectively, corresponding to a sudden increase of GWT in the second calculation stage. Figures 5 and 6 show similar states associated with a subsequent rapid drawdown simulated by resetting the original level of GWT at the seventh calculation stage.

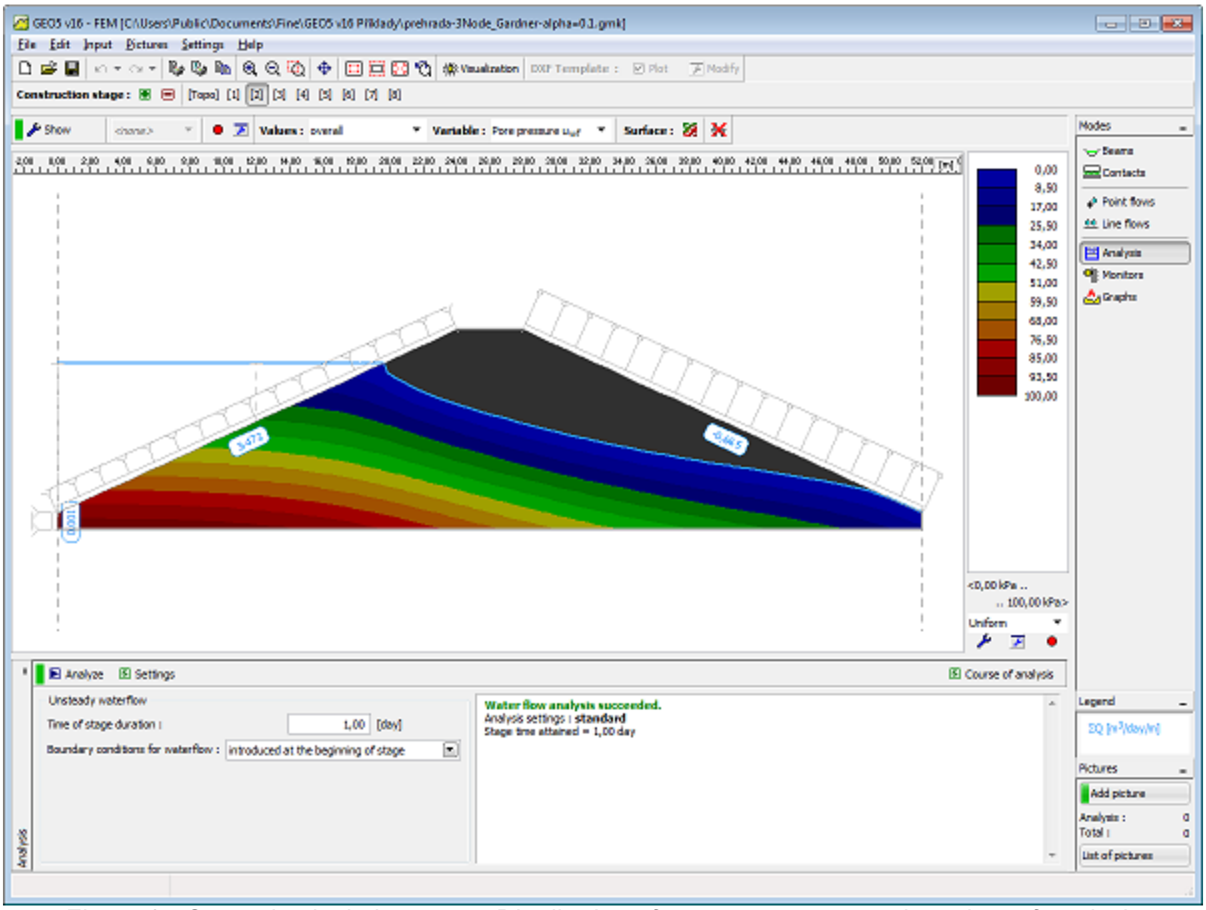


Figure 3 - Second calculation stage: Distribution of pore pressure at a given time of analysis

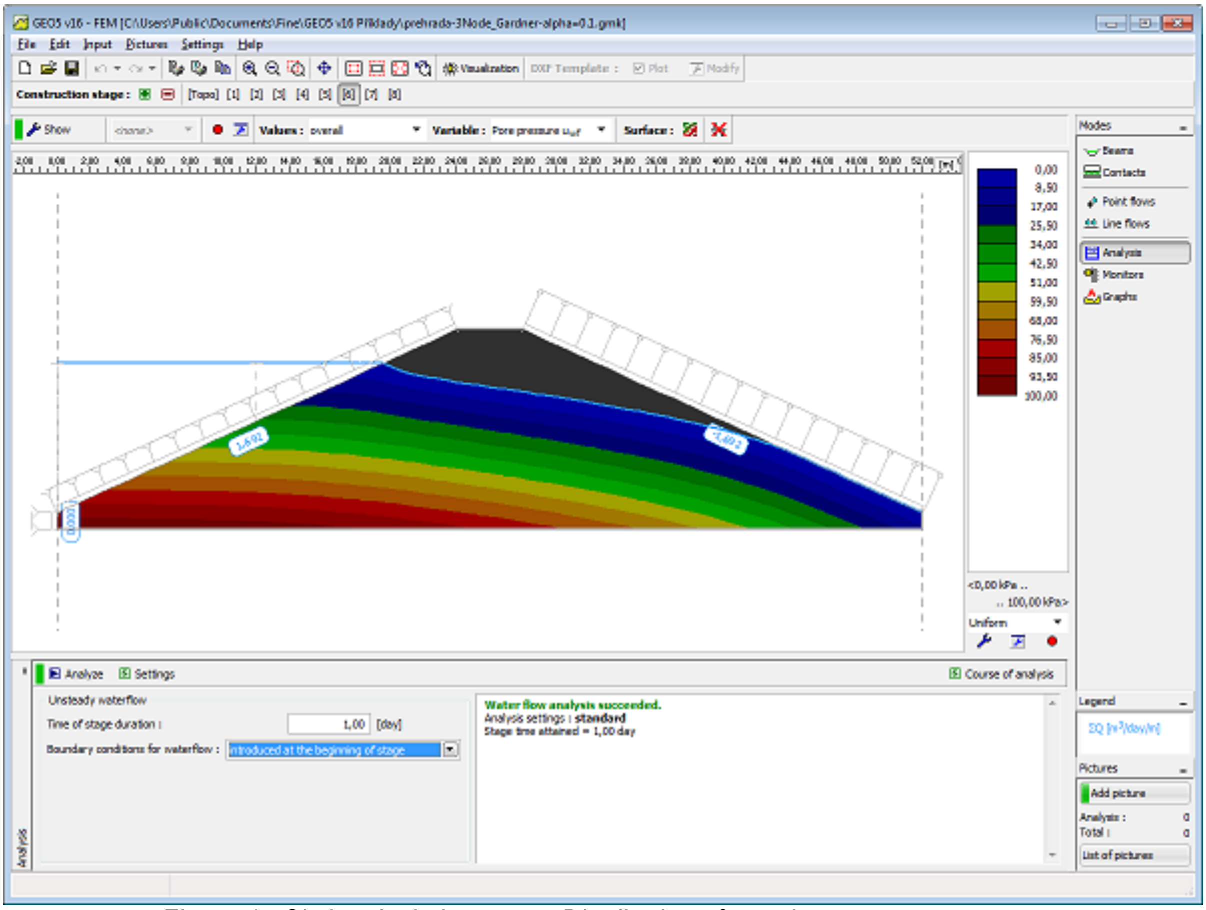


Figure 4 - Sixth calculation stage: Distribution of steady-state pore pressure

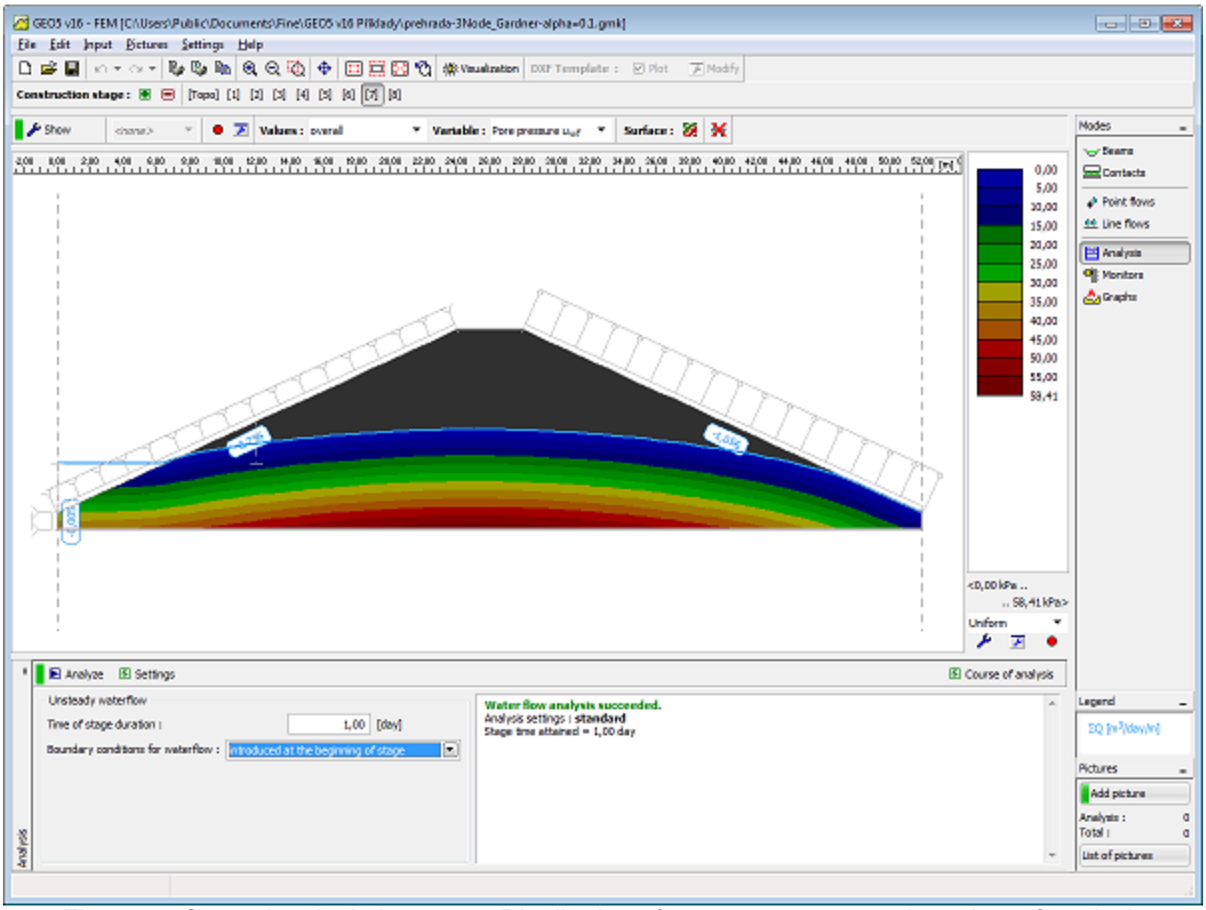


Figure 5 - Seventh calculation stage: Distribution of pore pressure at a given time of analysis

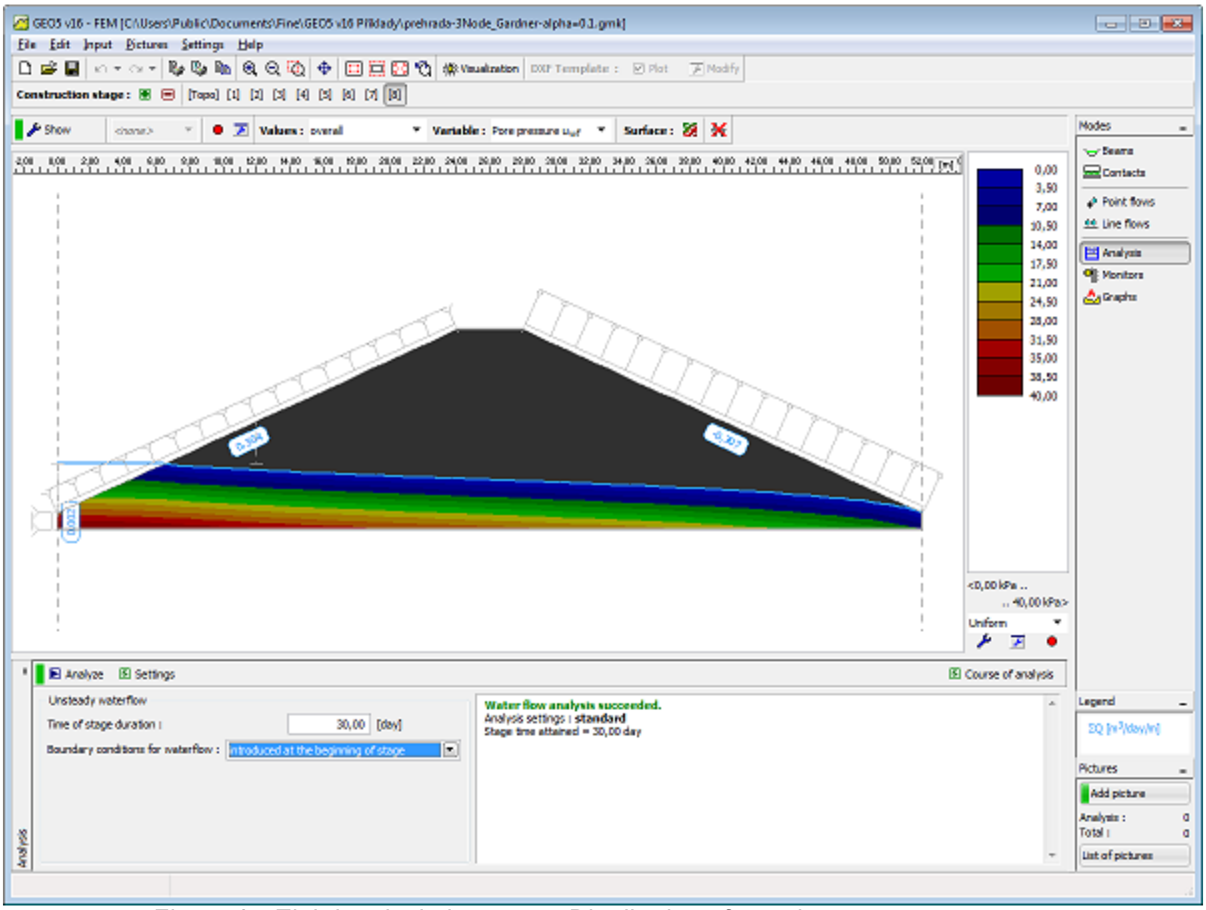


Figure 6 - Eighth calculation stage: Distribution of steady-state pore pressure

Recommended Modeling Procedure

Solving geotechnical problems using the finite element method is a relatively complex task. But yet, most users attempt to analyze the entire complex structure right from the beginning - to find the cause of possible loss of convergence may then become rather difficult. We, therefore, recommend the following approach:

1) Define the entire topology of the structure

2) Assume elastic response of soils and contact elements (use linear models)

3) Generate coarse mesh

4) Define all calculation stages

5) Perform analysis of all calculation stages (it is sufficient to launch the analysis of the last stage of construction - analyses of all previous stages are carried out automatically).

6) Asses the course of analysis

If the analysis fails, the computational model is not correctly defined - e.g. beams have too many internal hinges resulting in a kinematically undetermined structure, props are not properly connected to the structure, etc. The program contains a number of built-in checking procedures to warn the user of possible drawbacks in the model definition. Some of the errors, however, cannot be disclosed prior to running the program.

If all stages were successfully analyzed, we recommend the user to check the resulting displacements and this way also the objectivity of the used soil parameters and structural stiffness. Note that using nonlinear models always results in larger displacements in comparison to the pure elastic response - should the elastic displacements be already excessively large, we must first adjust the computational model before adopting any of the available plasticity models.

If the analysis succeeded and the displacements are reasonable, we may proceed as follows:

7) Replace linear models with suitable plastic models (Mohr-Coulomb, Drucker-Prager)

8) Perform analysis and evaluate the results according to step 6

9) Add **nonlinear contact elements**

10) Perform analysis and evaluate the results according to step 6

11) Refine and **adjust the finite element mesh** and perform the **final analysis**.

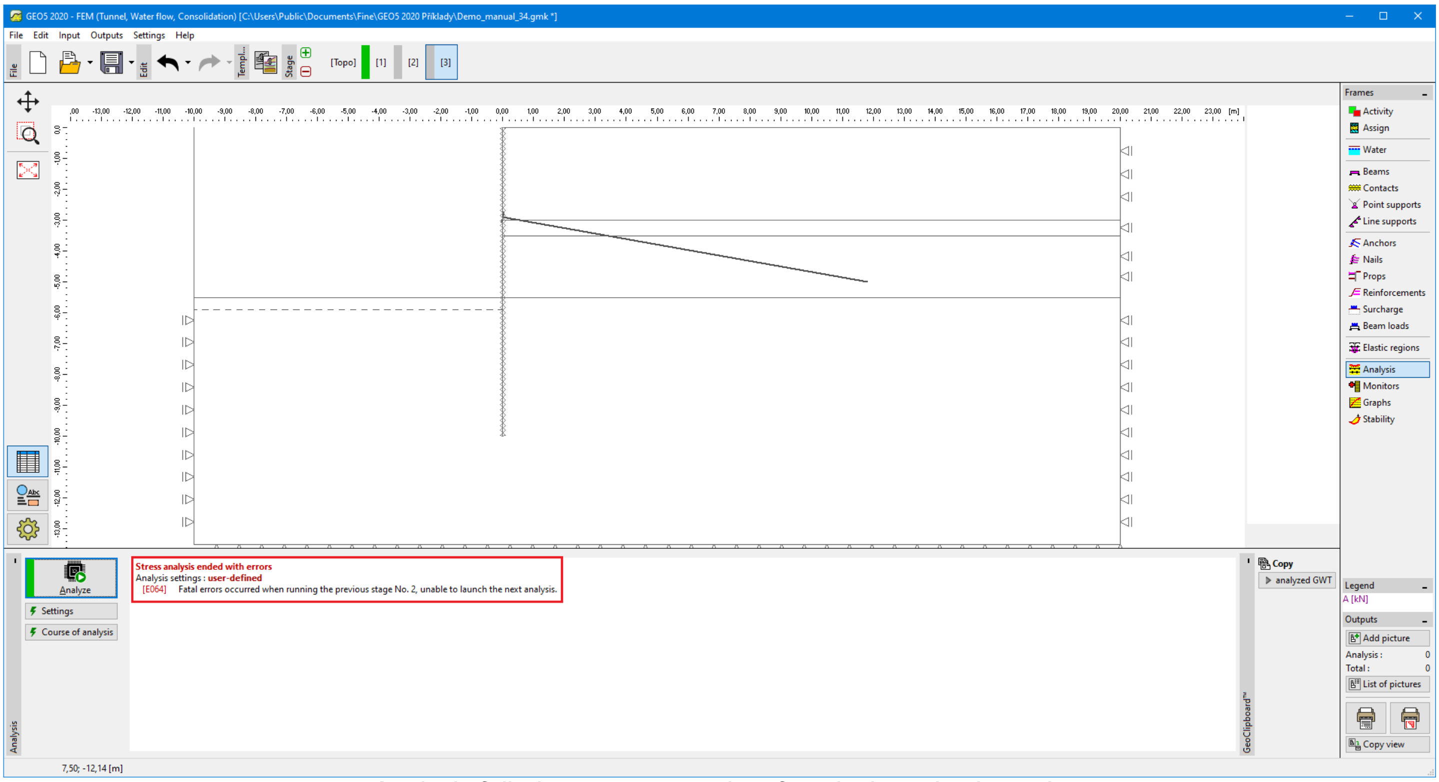
Although this approach may seem rather cumbersome and complicated, it may save a considerable amount of time when searching for the cause of failure (loss of convergence) of the analysis of complex problems.

Loss of Convergence of Nonlinear Analysis

Loss of convergence of the solution of nonlinear analysis calls for certain **modifications of the underlying computational model** - the following steps can be adopted:

- Increase the stiffness of the structure
- Decrease the applied loads
- Split the soil excavation into more steps
- Improve material parameters of existing soils
- Change the material model of soils in places of plasticity
- Add reinforcing members (beams, anchors)
- Add supports
- Change parameter settings affecting the iteration process (increase number of iterations).

Distribution of plastic strains may provide some explanation to why the analysis failed to converge. Note that the distribution of equivalent plastic strain locates the regions of probable evolution of critical failure surfaces.



Analysis failed to converge - plot of equivalent plastic strain

Settings and Analysis Description

The **default settings of parameters that drive the solution** analysis are optimized to ensure sufficient accuracy and efficiency of the analysis. Nevertheless, an experienced user may require to change the default setting or to examine the influence of parameters on the accuracy and course of the analysis. The parameter setting can be adjusted in the "**Analysis settings**" dialog window.

However, the change in standard settings, deserves a **word of caution**. Prior to making any changes, the user should be well aware of possible consequences. In particular, an improper setting may substantially slow down the computation process, which may cause divergence and eventually lead to **incorrect results**.

- Solution method
- Change of stiffness matrix
- Initial solution step
- Maximum number of iterations
- Convergence criterion
- Newton-Raphson method setting
- Arc-length method setting
- Line search method
- Plasticity

The default setting can be always recovered by pressing the "**Standard**" button.

Solution Method

The program FEM serves to analyze geotechnical problems characterized by a nonlinear response of the soil or rock body. A successful analysis of most of such problems calls for an iterative solution of a given boundary value problem. Applying the finite element method (FEM) then leads to an incremental form of the equilibrium conditions written as:

$$K_T . \Delta u = \Delta f$$

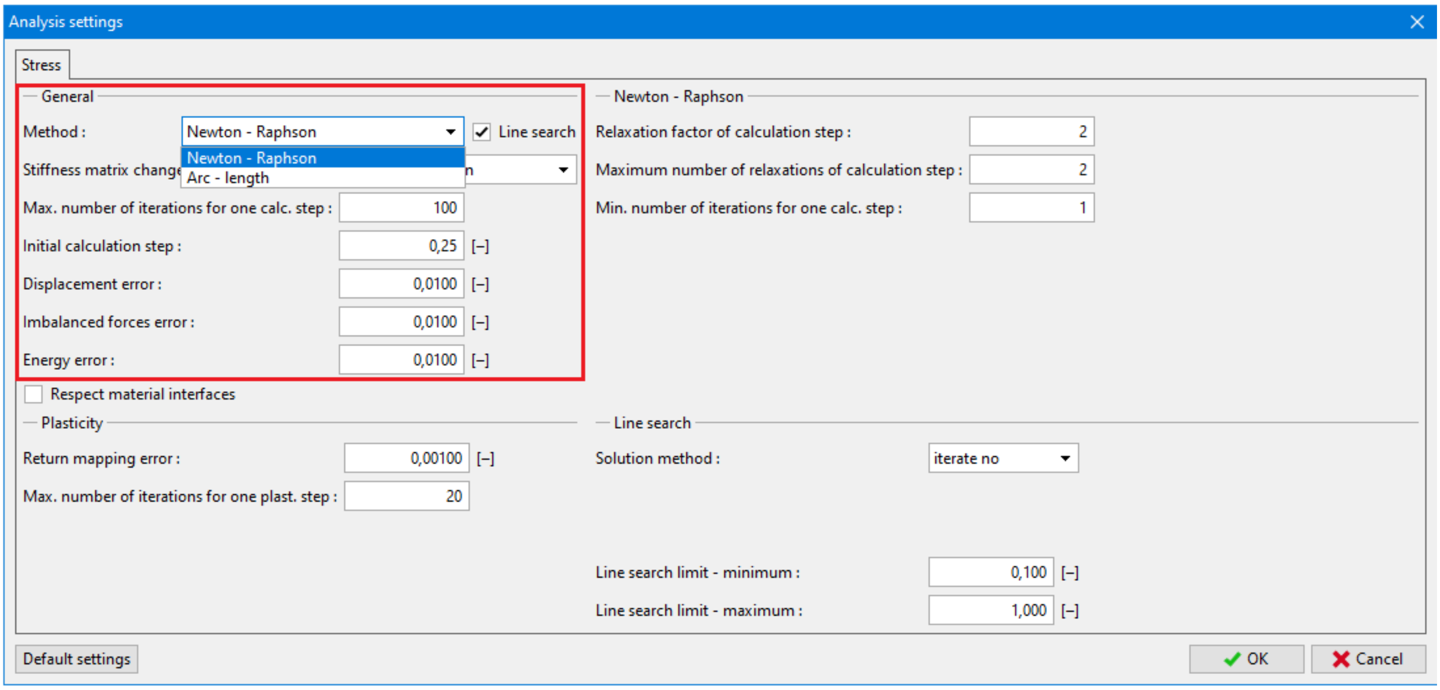
where: K_T - instantaneous stiffness matrix

Δu - vector of nodal displacement increments

Δf - vector of out-of-balance force increments

This equation can be solved only approximately using a suitable numerical method. The goal of the method is to arrive, during the process of iteration, at such a state of stress and strain that satisfies the condition $\Delta f = 0$. To that end, **the program offers** two basic methods:

1. Newton-Raphson method - NRM
2. Arc-length method - ALM



Analysis settings - setting the solution method

Change of Stiffness Matrix

The full **Newton-Raphson method** assumes that the instantaneous tangent stiffness matrix is formed at the beginning of each new iteration.

Forming a new tangent stiffness matrix only at the beginning of a new load increment leads to the so-called modified Newton-Raphson method.

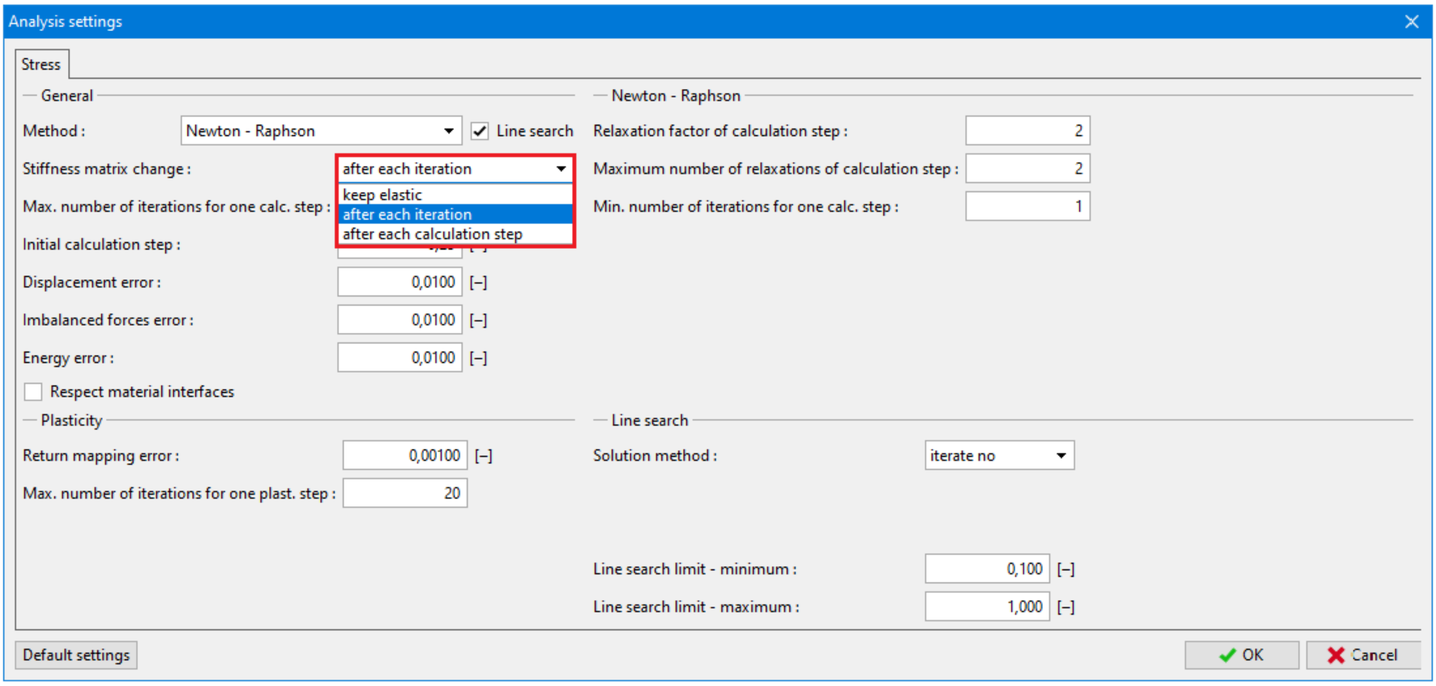
If the stiffness matrix is formed only once at the beginning of the solution analysis we obtain the so-called initial stress method.

Individual methods can be selected from the "**Analysis settings**" dialog window section "Stiffness update". The corresponding settings are:

1. **Keep elastic** - initial stress method,
2. **Each iteration** - full Newton-Raphson method,
3. **Each load step** - modified Newton-Raphson method.

The default setting assumes the full Newton-Raphson algorithm (*stiffness update after each iteration*). Note that the formulation of the stiffness matrix is consistent with the stress update algorithm. Such a formulation then ensures quadratic convergence of the full Newton-Raphson (NRM) unlike the modified NRM or the initial stress method that, in comparison with the full NRM, requires considerably more interactions to attain equilibrium.

On the other hand, it is fair to mention that the computational cost per iteration is mainly determined by the calculation and factorization of the tangent stiffness matrix. Assuming elastic response of a structure it is clearly meaningless to set up the structural stiffness matrix more than once (stiffness update - keep elastic). On the contrary, increasing the degree of nonlinearity suggests more frequent stiffness reformulations (stiffness update - Each iteration).



Newton-Raphson method - stiffness matrix update options

Initial Solution Step

The actual analysis is carried out incrementally in several load steps until the overall prescribed load is reached.

The program requires setting the **initial load step** only.

This parameter represents the **ratio between the load applied in a given load step to the overall prescribed load**. Depending on the course of iteration this parameter is adaptively adjusted.

The default setting assumes 25% of the total prescribed load. Similarly to what we have already mentioned it holds that increasing the solution complexity from the nonlinear response point of view requires a reduction of this parameter. However, in the case of an **elastic response,** this parameter can be set equal to 1, which corresponds to the solution of a given problem in one load step.

Maximum Number of Iterations

This parameter represents the **maximum number of iterations allowed** for a single load step to reach the state of equilibrium.

Exceeding this value prompts the program to automatically **reduce the current value of the assumed load step** and restart the solution from the last load level that complies with the state of equilibrium. Similar action is taken when oscillation or divergence of the program is imminent.

Convergence Criterion

For the incremental solution strategy based on one of the iterative methods to be effective, it is necessary to select suitable criteria (preset tolerances for reaching equilibrium) for the **termination of the iteration process**.

Note that loose convergence criteria may result in inaccurate results while too tight convergence tolerances may lead to an unjustified increase of computational cost spent to arrive at the results of superfluous accuracy.

In the program, the convergence is checked against the change of nodal displacement increments, the change of out-of-balanced forces, and also the change of internal energy. The last criterion gives a certain idea about how both displacements and forces approach their equilibrium values. The corresponding settings are:

1. **Displacement error tolerance** - tolerance for the change of displacement increment norm.
2. **Out-of-balanced forces tolerance** - tolerance for the change of out-of-balance force norm.
3. **Energy error tolerance** - tolerance of the change of internal energy.

The default setting is 0.01 for **all convergence tolerances**.

Setting Newton-Raphson Method

With the Newton-Raphson method the course of iteration can be driven by setting the following parameters:

1) Relaxation factor - it represents the value of reduction of the current load step for the restart providing the solution fails to converge. A new value of the assumed load step is found from the expression:

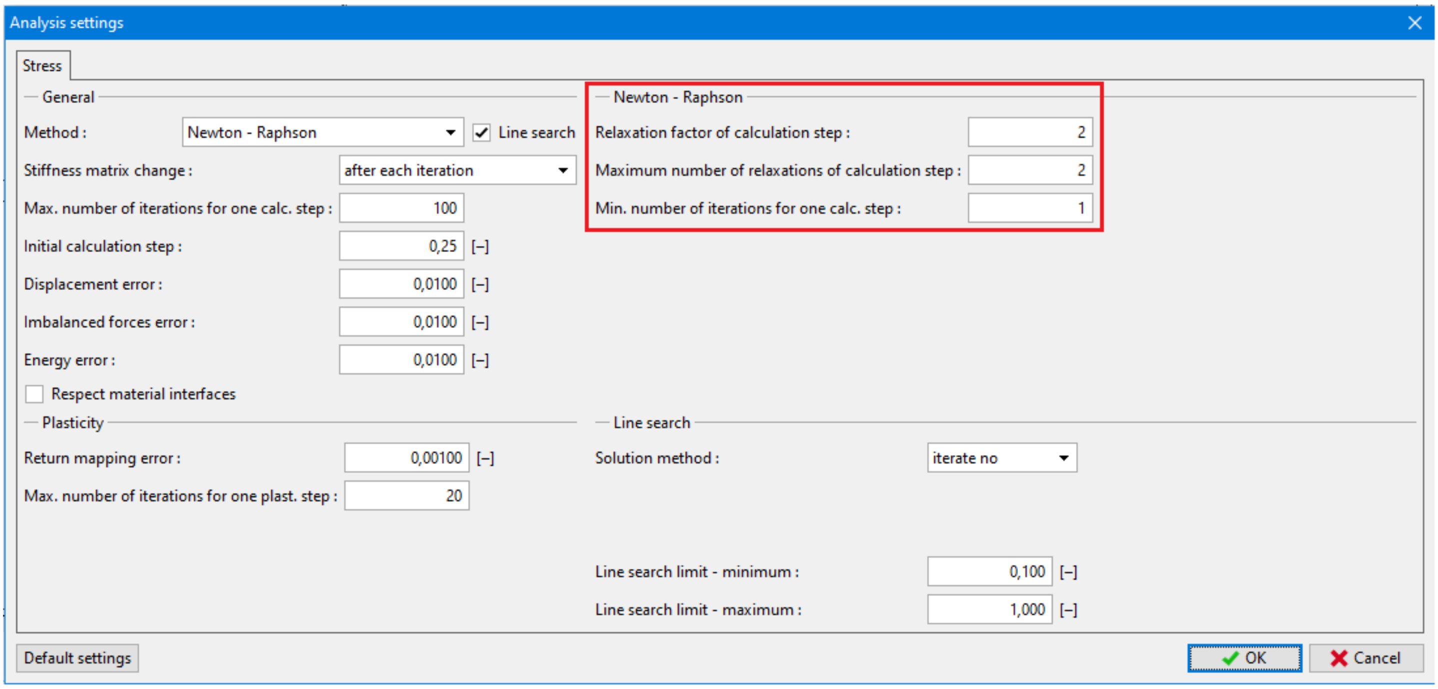
new load step = old load step / relaxation factor.

2) Max. No. of relaxations *for a single load step* - this parameter determines how many times it is possible to invoke the above action during the entire analysis. Exceeding this value prompts the program to terminate the analysis. The results are then available for the last successfully converged load level.

3) Min. No. of iterations *for a single load step* - this parameter allows for possible acceleration of the analysis. In particular, providing the number of iterations to converge in the last load step is less than the minimum one set, the load step for a new load increment is increased as follows:

new load step = old load step * relaxation factor.

The default setting of the above parameters corresponds to values displayed in the figure:



Parameters driving the iteration process

Setting Arc-Length Method

The Arc-length method (ALM) is a relatively robust method particularly suitable for the solution of problems that require the search for the collapse load of a structure. Stability analysis of earth structures (slopes, embankments) is just one particular example of such a task. Unlike the NRM where the solution is driven purely by prescribing load increments, the ALM introduces an additional parameter representing a certain constraint on the value of load increment in a given load step. The value of the load step thus depends on the course of iteration and is directly related to the selected arc length.

The basic assumption of the method is that the prescribed load varies proportionally during the calculation. This means that a particular level of the applied load can be expressed as:

$$\overline{F} = \lambda . F$$

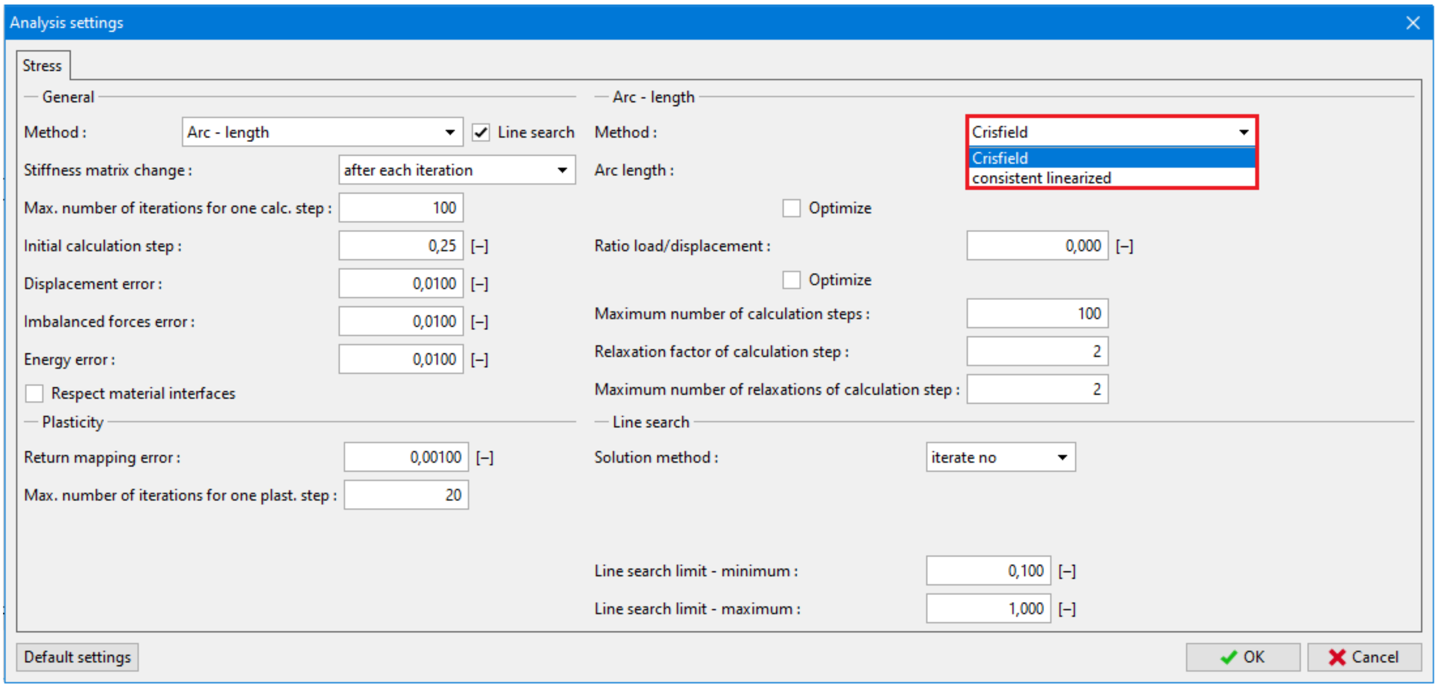
where:

where:	$\overline{F}$	-	current fraction of the total applied load
	λ	-	coefficient of proportionality
	F	-	overall prescribed load

Note that with ALM the load vector **F** represents only a certain reference load that is kept constant during the entire response calculation. The actual value of the load at the end of calculation is equal to the λ multiple of **F**; $\lambda <$ 1 represents the state when the actual bearing capacity of a structure is less than the prescribed reference load; if λ at the end of response calculation exceeds 1, the program automatically adjusts the arc length in order for the solution to converge to value $\lambda =$ 1 within a selected tolerance equal to 0.01 (1$\%$ the maximum applied load). This value cannot be changed.

The literature offers a number of ALM formulations. The program supports the method suggested by Crisfield and the consistently linearized method proposed by Ramm. The latter one is considerably simple, at least from the formulation point of view then the Crisfield method. On the other hand, it is reportedly less robust. The default setting is the Crisfield method.

Other important parameters of the method are "Setting arc length" and "Automatic arc length control".



Arc-length - setting the type of Arc-length method

Setting Arc Length

The arc length is the basic parameter affecting the response calculation. An indicator for the selection of arc length can be the course of iteration in the previous solution stage. Regardless of that the program enables the following setting:

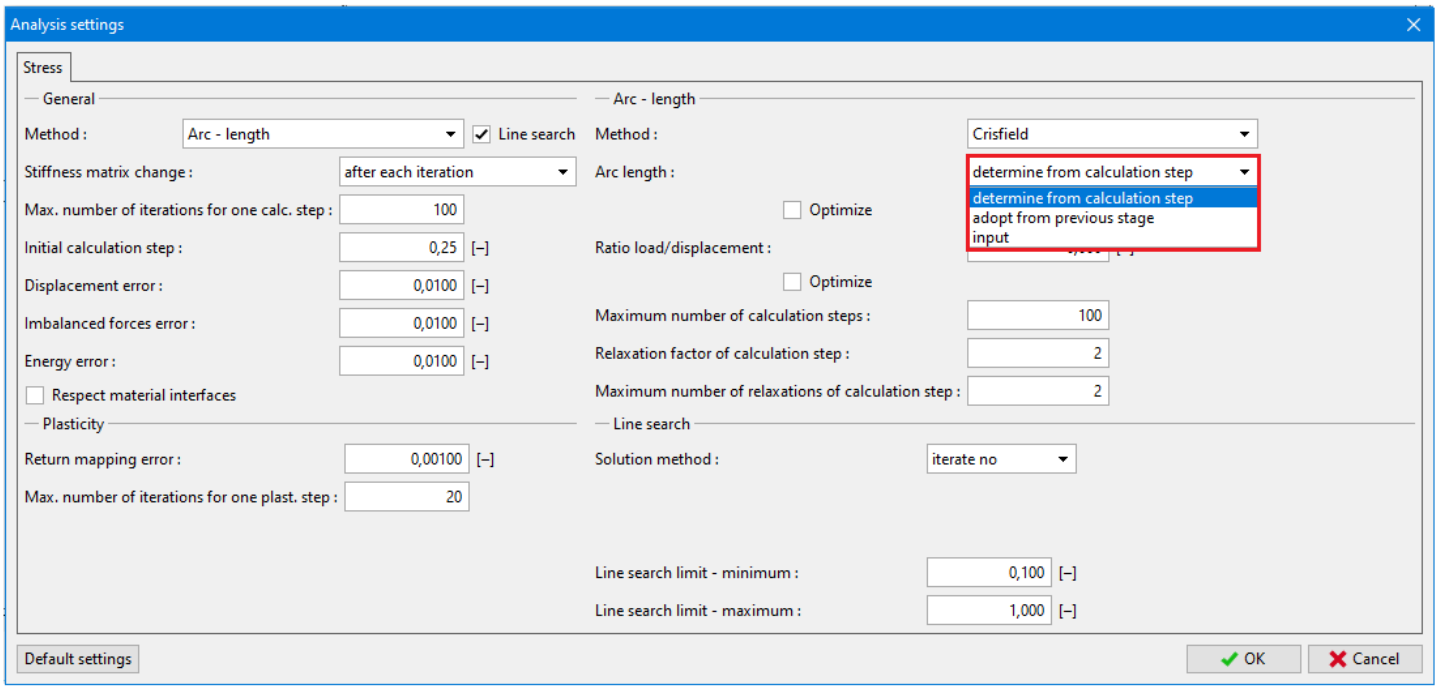
1. **Determine from load step** - the arc length is determined automatically from the initial load step.
2. **Adopt from the previous stage** - the value of arc length at the end of the previous calculation stage is used as a starting value for a new stage. This option becomes active in the second stage of construction.
3. **Input** - the value of the arc length can be directly prescribed.

Providing the structure response cannot be determined prior we recommend using the first option. Depending on the course of calculation it is possible to adjust the value of arc length and repeat the calculation. At no event, however, is it possible to ensure convergence for an arbitrary value of arc length selected. Similarly to NRM, if the convergence problems occur the program allows for the reduction of the current arc length and restarts the calculation.

The next parameter driving the iteration process is the *Maximum No. of load steps*. The program always carries on the prescribed number of load steps providing:

- parameter λ exceeds 1,
- the maximum number of relaxations of the arc length is exceeded.

Providing the analysis is terminated due to exceeding the maximum number of prescribed load steps and parameter λ is less than 1, it is necessary to increase the number of steps and restart the analysis.

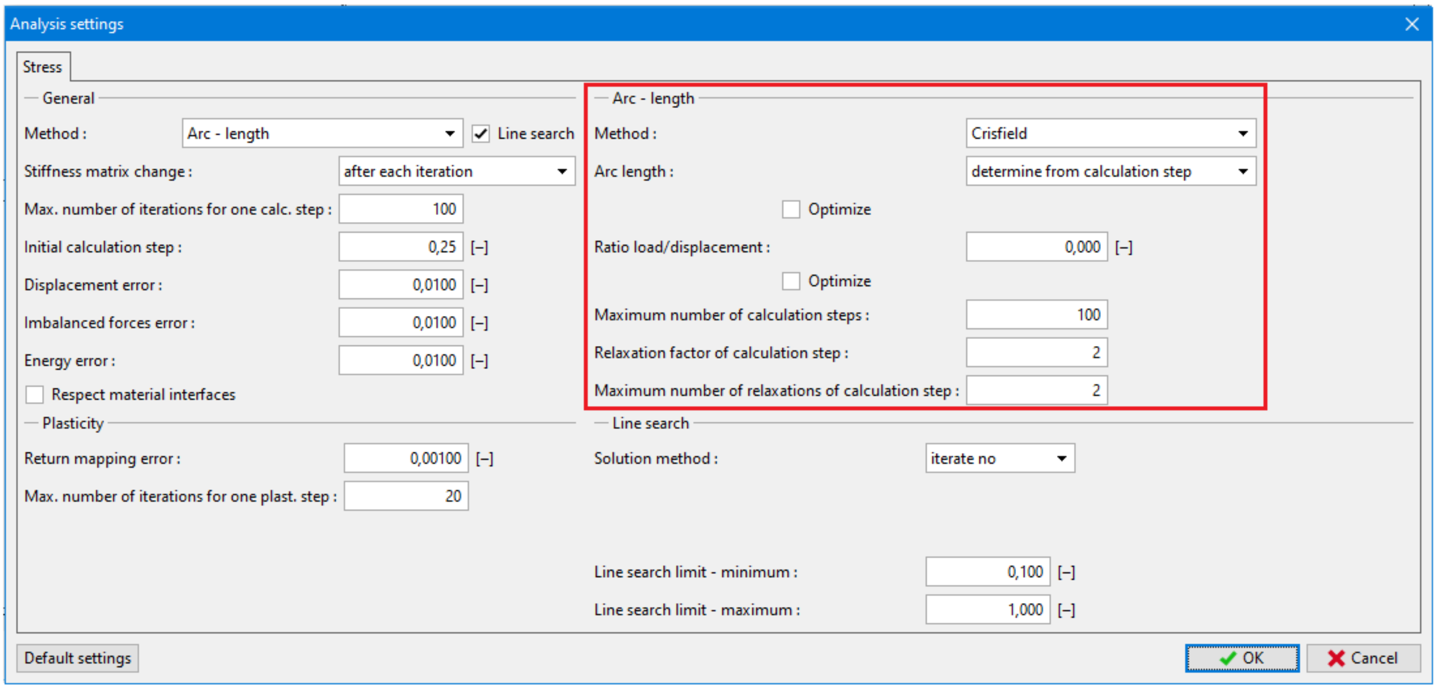


Arc-length - arc length setting

Automatic Arc Length Control

An automatic arc length control strategy constitutes a very important part of the implementation of any numerical method. The program makes it possible to adaptively adjust the current arc length for a new load step depending on the course of iteration in the previous step by activating option **Optimize**. The program will then attempt to select a value of arc length that keeps the desired number of iterations in each load step needed for convergence - option **Optim. No. of iter. in a single load step**.

The next parameter driving the process of iteration is the **Ratio load/displacement**. This parameter represents a scalar factor, which adjusts the scales of load given by parameter λ and displacement vector $\boldsymbol{u}$. providing this parameter is sufficiently large the analysis is essentially driven by load increment. Setting this parameter equal to 0 (default setting) we obtain so-called cylindrical ALM and the analysis will be driven by displacement increment. This approach is more stable and recommended by the authors. Nevertheless, the program allows for the optimization of this parameter by activating the option "**Optimize**". In such a case the current value of this parameter is set equal to the Bergan current stiffness parameter that provides a scalar measure of the degree of nonlinearity. With increasing the degree of nonlinearity this parameter is decreasing. In the vicinity of collapse load, the value of this parameter approaches zero and the solution is driven by displacement increment. This strategy thus supports the use of the cylindrical method having the **Ratio load/displacement** parameter equal to zero. As for the default setting, this option is turned off.



Arc-length - automatic arc length control

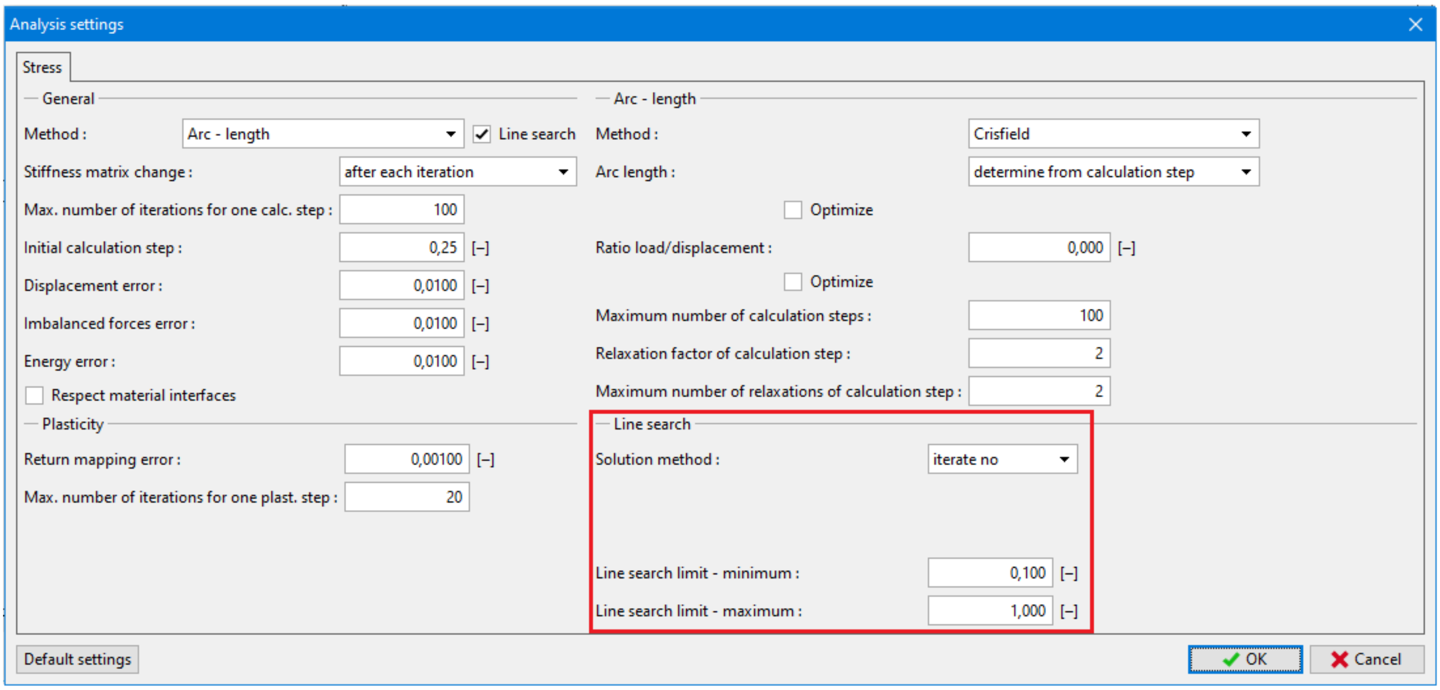
Line Search Method

The basic goal of the Line search method is to determine a scalar multiplier η that is used to scale the current displacement increment so that the equilibrium is satisfied in a given direction. The actual displacement vector at the end of the i-th iteration thus becomes:

$$u_i = u_{i-1} + \eta . \Delta u$$

Consequently, the calculation process is either accelerated, $\eta > 1$, or damped, $\eta < 1$. Obviously, with the Line search performed each iteration, the expense of the iteration increases. On the other hand, this drawback is compensated by less number of iterations needed for convergence and by the possibility of avoiding divergence or oscillation of the process of iteration. By default, the use of the Line search is enabled.

An inexperienced user is recommended to employ the default setting evident from the figure.



Line search method settings

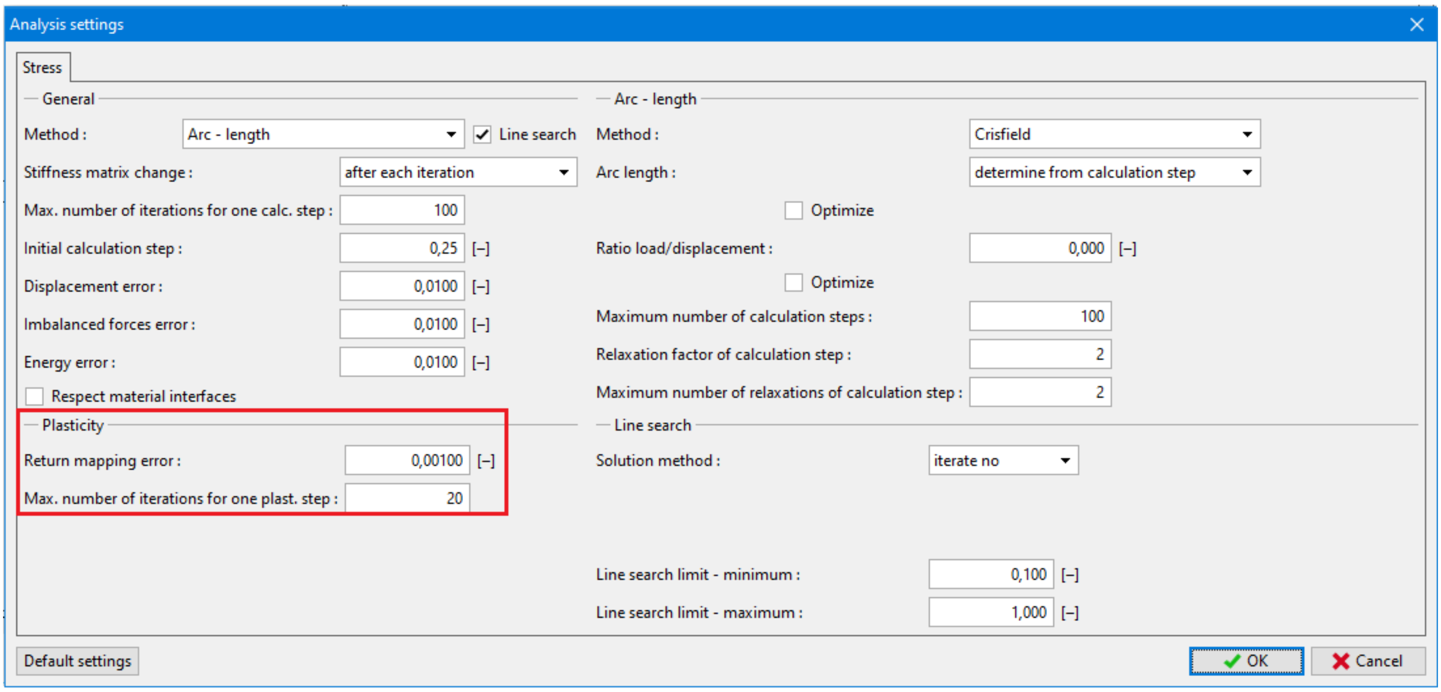
Plasticity

The **Plasticity** dialog window serves to set parameters driving the stress update procedure.

The parameter **Return to yield surface tolerance** suggests the tolerance for satisfying the selected yield condition. Assuming nonlinear hardening/softening as in case of modified cam clay model the stress return mapping requires an iteration process.

The maximum number of iterations allowed is then given by the **Max. No. of iterations for a single plastic step** parameter. When employing the rigid-plastic version of the Mohr-Coulomb, the Drucker-Prager, or the modified Mohr-Coulomb model, these parameters will not apply.

The default setting, evident in the figure, is recommended.



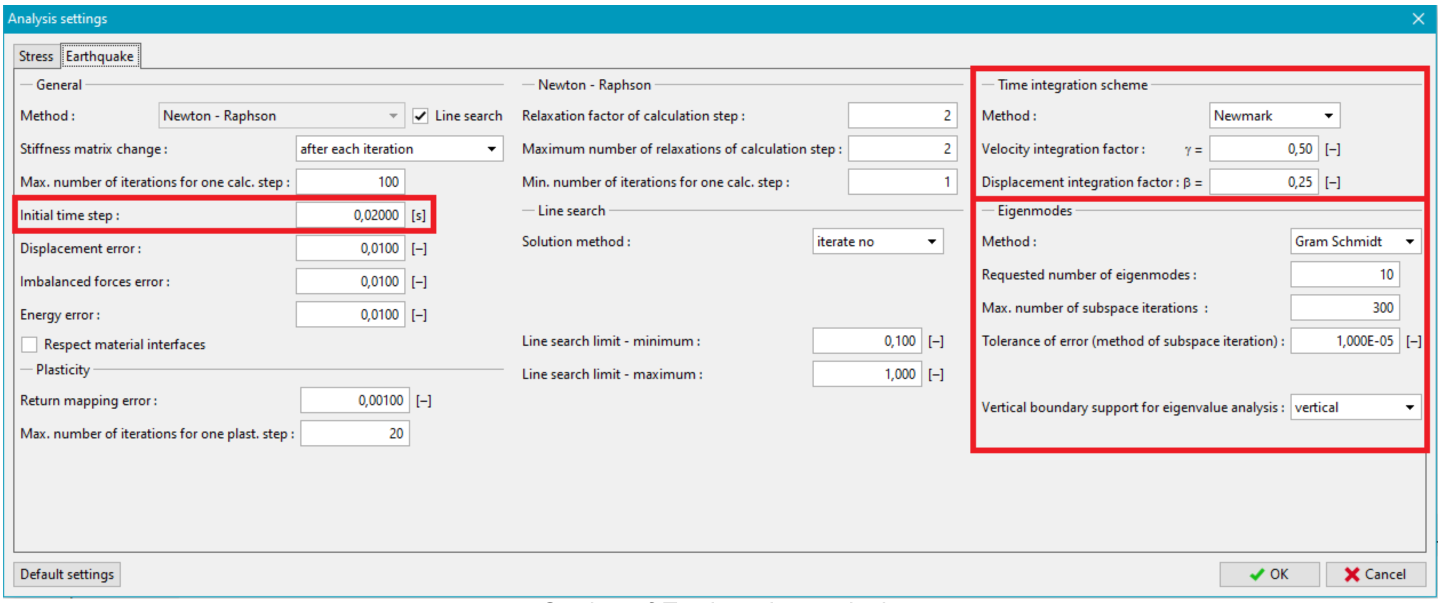
Parameters driving the stress return mapping

Setting Earthquake analysis

When performing the earthquake analysis it is possible to choose in the "**Analysis Setting**" dialogue window the type of analysis, either "**Stress analysis**" or "**Earthquake**". The stress analysis in a given stage always precedes the earthquake analysis.

Unlike in stress analysis the earthquake analysis requires setting the initial time step in s .

In addition, the parameters of the selected time integration scheme, either the **Newmark method** or the **Alpha method**, and the parameters of the Eigenvalue analysis for the **calculation of eigenfrequencies and eigenmodes** must be specified.



Setting of Earthquake analysis

The earthquake analysis always performs the following sequence of computations:

1. Stress analysis
2. Eigenvalue analysis
3. One-dimensional Free field column analysis
4. Two-dimensional earthquake analysis

The Free field column analysis serves to set the time variation of traction boundary conditions applied on the vertical boundaries of the 2D model needed in step 4. This is the only analysis which does not allow visualization of the analysis results.

Further details are available in the **theoretical manual** on our website.

Newmark method

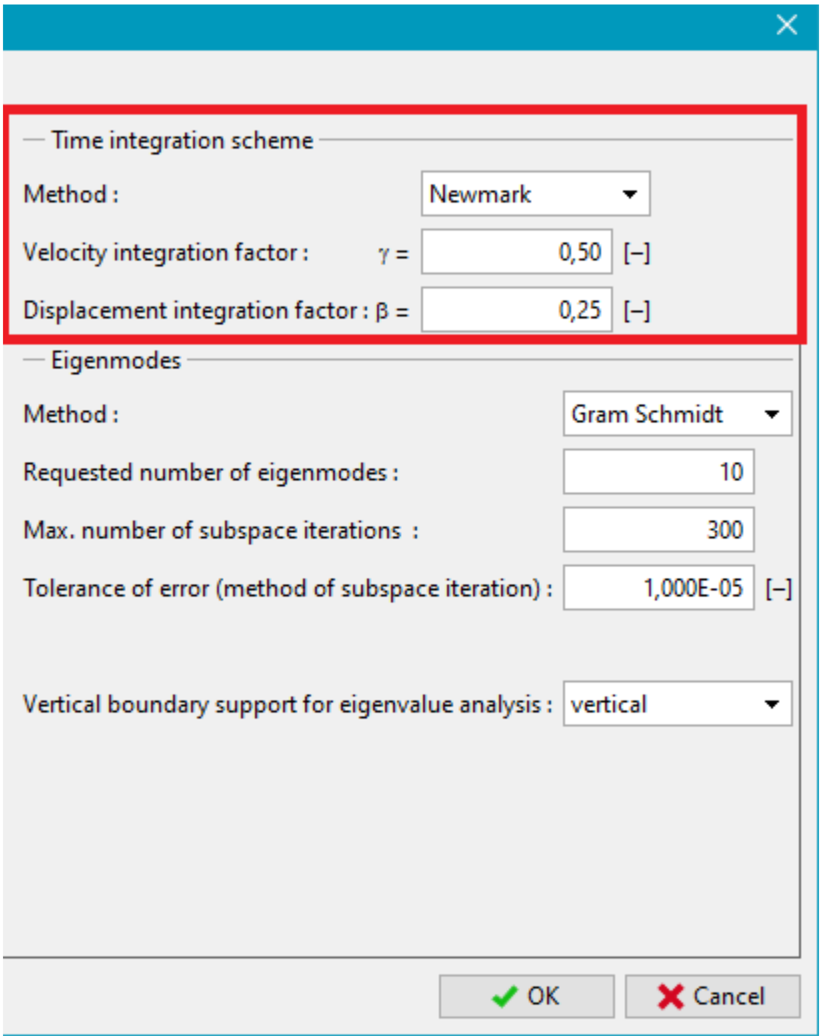
Standard Newmark method can be employed to perform time integration of basic differential equations of motion. Two basic parameters β and γ describing the evolution of displacements and velocities, respectively, must be introduced.

Setting $\gamma = 0.5$ a $\beta = 0.25$ gives the average acceleration method (trapezoidal rule). This setting recommended and complies with the following stability condition:

$$2\beta \leq \gamma \leq \frac{1}{2}$$

which ensures that the integration scheme is unconditionally stable (stability does not depend on the magnitude of the selected time step).

For $\gamma = 0.5$ the method is second order accurate (the rate of convergence with the time step reduction is 2). Also, the material damping has in such a case no effect on stability.



Setting of Newmark method

Further details are available in the **theoretical manual** on our website.

Alpha method

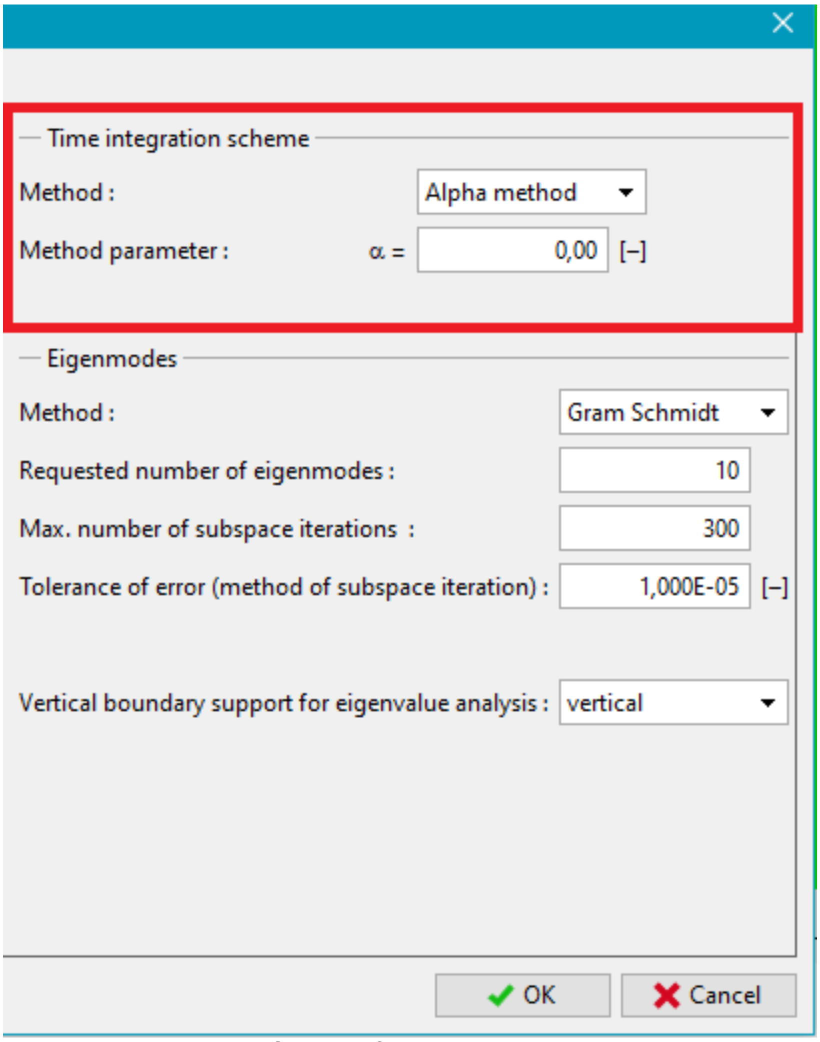
Setting $\gamma =$ 0.5 in the Newmark method yields no numerical **damping** so the participation of high frequency modes, which are just artefacts of the finite elements discretization, cannot be removed. To that end, Hilber, Hughes and Taylor developed the **Alpha method** to introduce numerical damping while retaining the unconditional stability and second order accuracy of the integration scheme and at the same time filtering out the higher modes.

The parameter α is taken from the interval:

$$\alpha \in \left[-\frac{1}{3}, 0\right]$$

And the parameters β and γ of the integration scheme are calculated directly as:

$$\beta = \frac{1-\alpha^2}{4}, \quad \gamma = \frac{1-2\alpha}{2}$$



Setting of Alpha method

Further details are available in the **theoretical manual** on our website.

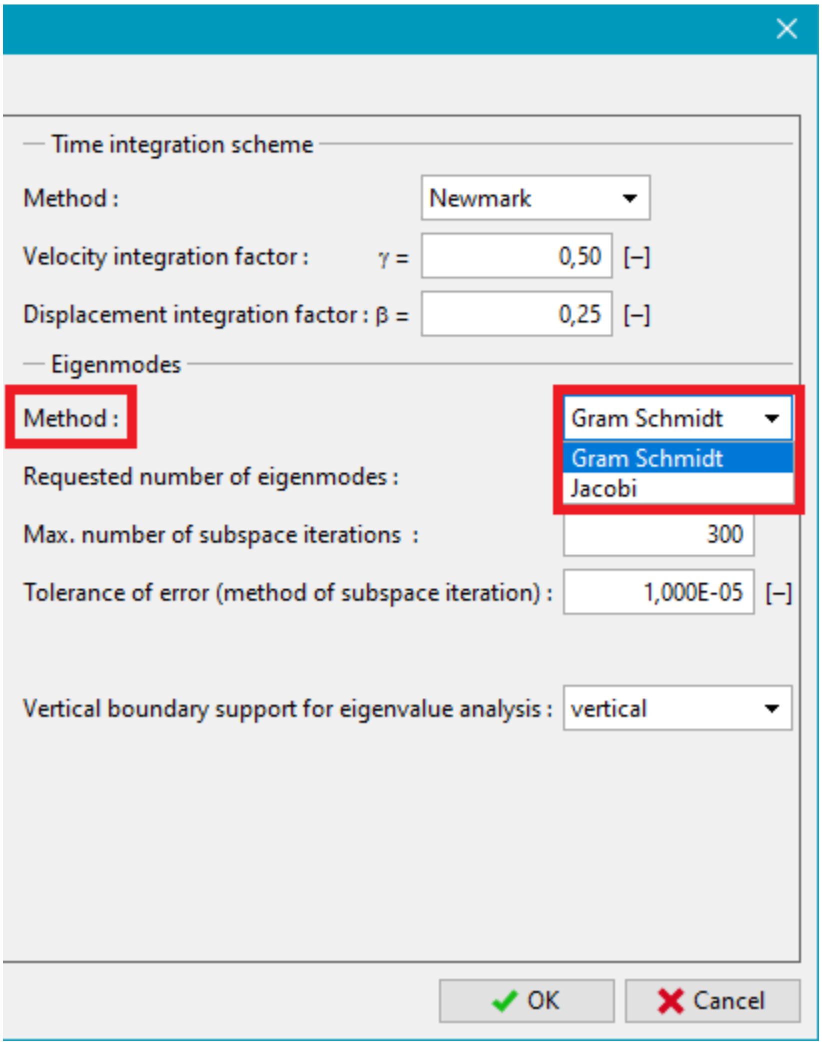
Eigenvalue analysis - calculation of eigenfrequencies and eigenmodes

Eigenvalue analysis provides the N lowest eigenfrequencies and eigenmodes of the system. This number is specified by the user. The resulting frequencies are primarily used to calculate the parameters of the "**Rayleigh damping**".

The analysis is carried out employing the inverse subspace iteration method. There are two options allowing for choosing either the **Gram Schmidt** orthogonalization method or the **Jacobi method**.

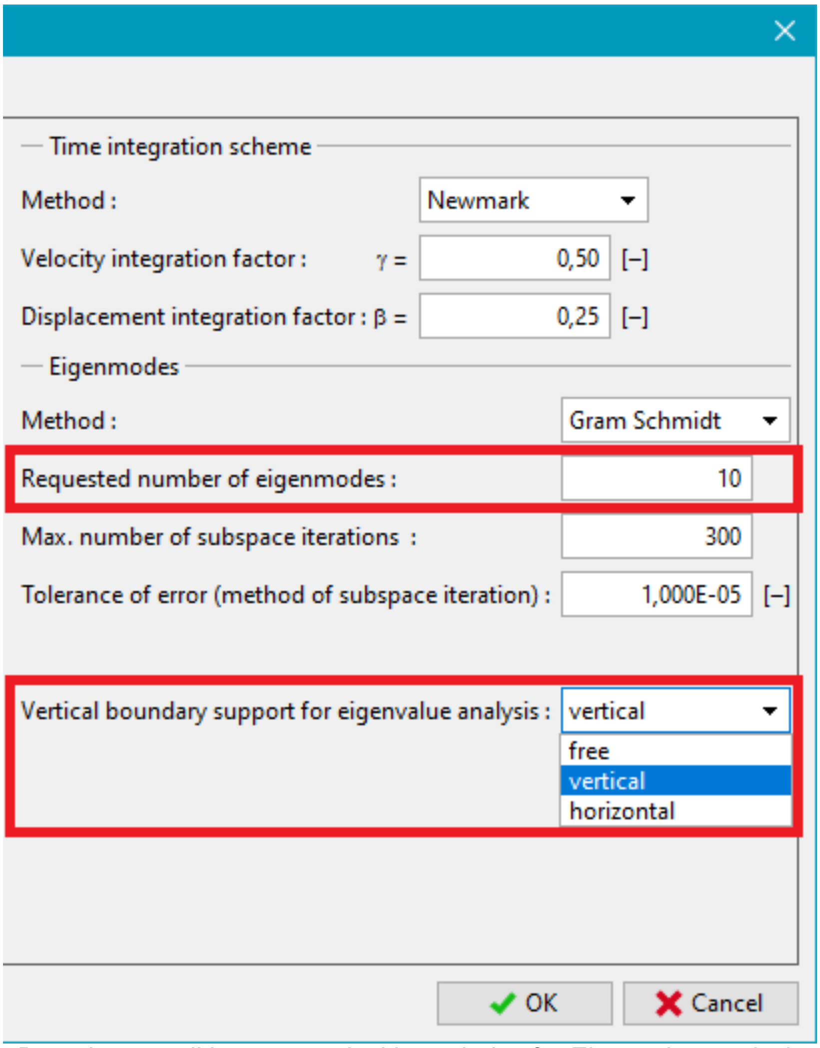
The Jacobi method typically **requires less number of iterations** and the number of found frequencies is typically higher than the number set by the user. **It may, however, happen that some frequencies are during the search omitted**. To check this, the program adopts the Sturm sequence.

The Gram Schmidt method is **computationally more demanding** but **more reliable in finding** all the desired frequencies.



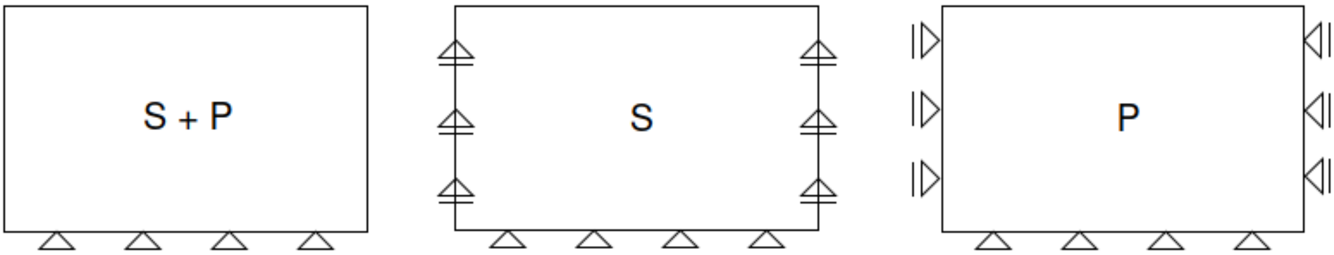
Methods for Eigenvalue analysis

When choosing eigenfrequencies for the calculation of parameters of **material damping** it is often necessary to select a suitable mode shape corresponding to the prescribed load - purely horizontal vibration mode (only the horizontal component of the acceleration vector is introduced), or purely vertical vibration mode (only the vertical component of the acceleration vector is introduced), or the combination of both modes (both components are prescribed). Eliminating some of the modes might be therefore desirable and can be achieved by selecting suitable kinematic boundary conditions on lateral boundaries of the numerical model.



Boundary conditions on vertical boundaries for Eigenvalue analysis

The program allows for the following variants of kinematic boundary conditions:



Vertical support type available for Eigenvalue analysis

An additional hint for selecting the desired modes shape for the calculation of parameters of material damping are the "Modal participation factor" $\Gamma_{\alpha,i}$ and "Modal effective mass" $m_{\alpha,i}$ representing participation of a given eigenmode in either horizontal or vertical component of the vibration.

Further details are available in the **theoretical manual** on our website.

Course of Analysis

The course of analysis can be viewed in the bottom part of the screen.

An **elastic** analysis is completed in **one computational step**. A nonlinear analysis is performed in several steps - the external load is gradually increased in several **load (calculation) steps**. The analysis is completed successfully if there is no loss of overall convergence so that 100 percent of the required load is reached.

The **default setting of parameters** that drive the solution analysis is optimized to ensure sufficient accuracy and efficiency of the analysis. Nevertheless, an experienced user may require to change the default setting or to examine the influence of parameters on the accuracy and course of the analysis. The parameters setting can be adjusted in the "Settings and analysis description" dialog window:

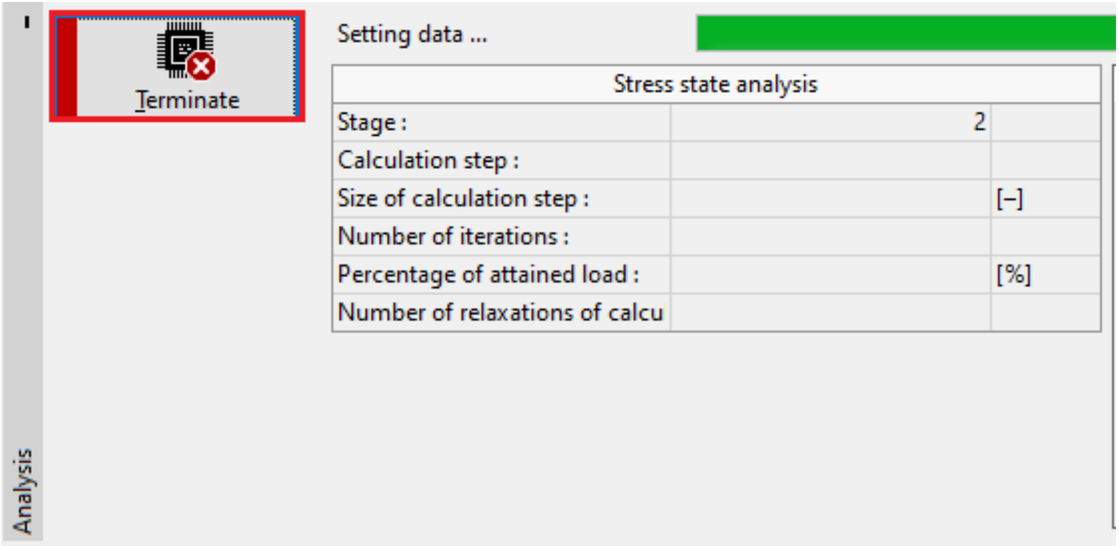
- The **Percent of the applied load** parameter gives a percentage of the overall load (excepted value) at the end of the current load step assuming successful convergence for the current load step.
- The **Step size** parameter provides the current scaling factor for the determination of load increment in the current load step.
- The **Safety factor** parameter corresponds to the expected value of the safety factor assuming successful convergence for given parameters c , φ .

The course of iteration within a given load step is characterized by the change of convergence parameters:

- η - Line search method parameter
- change of the displacement increment norm
- change of the out-of-balance force norm
- change of internal energy

If all three errors are smaller than the preset **error tolerance** (can be edited in the "**Settings**" dialog window), the analysis is for the calculation step terminated.

The "**Interrupt**" button serves to terminate the calculation process. The results are then available for the last load level that complies with the state of equilibrium.



Course of analysis

Results

Visualization (plotting) of results is one of the most important features of the program. The program allows us to select from several basic styles of graphical outputs, which are defined in the "Drawing Settings : Analysis" dialog window.

- draw **deformed mesh**
- surface plot of variables developed **inside the soil/rock body** (the total values or their increments with respect to other calculation stages can be displayed)
- **internal forces** distributed along beams, contacts
- **forces in anchors and reaction forces**
- **depression curve**
- **tilted sections** of variables
- **vectors and directions** of variables

To display results the program employs the following coordinate systems.

The toolbar "Results" in the upper part of the screen serves to selected variables to be displayed and the way they should appear on the screen. The color scheme is shown in the right part of the desktop. Its particular setting can be adjusted using the "Color scheme" toolbar.

Because properly setting outputs might be often time-consuming, the program disposes of a comfortable system of storing and managing various settings.

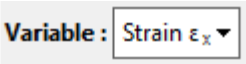
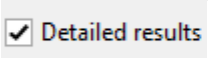
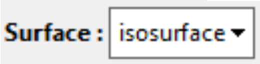
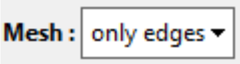
All outputs and selected results can be further printed out from the analysis protocol.

Toolbar "Results"

The control bar contains the following operating elements:

Control bar "Setting visualization of graphical outputs"

Individual elements operate as follows:

	**Values in stages of analysis**	• displays calculated values (either total or incremental values with respect to the selected stage of construction can be seen)
	**Variable type**	• displays the selected variables
	**Detailed results**	• displays the detailed results of variables
	**Surface plot**	• turns on/off plotting of isolines, isosurfaces
	**Mesh**	• turns on/off the style of plotting the FE mesh (only edges, or according to the setting in the "Drawing Settings : Analysis" dialog window
	**Displacements plot**	• selects the style of plotting deformed mesh - undeformed/deformed (deformed by the magnitude, deformed by the coefficient)

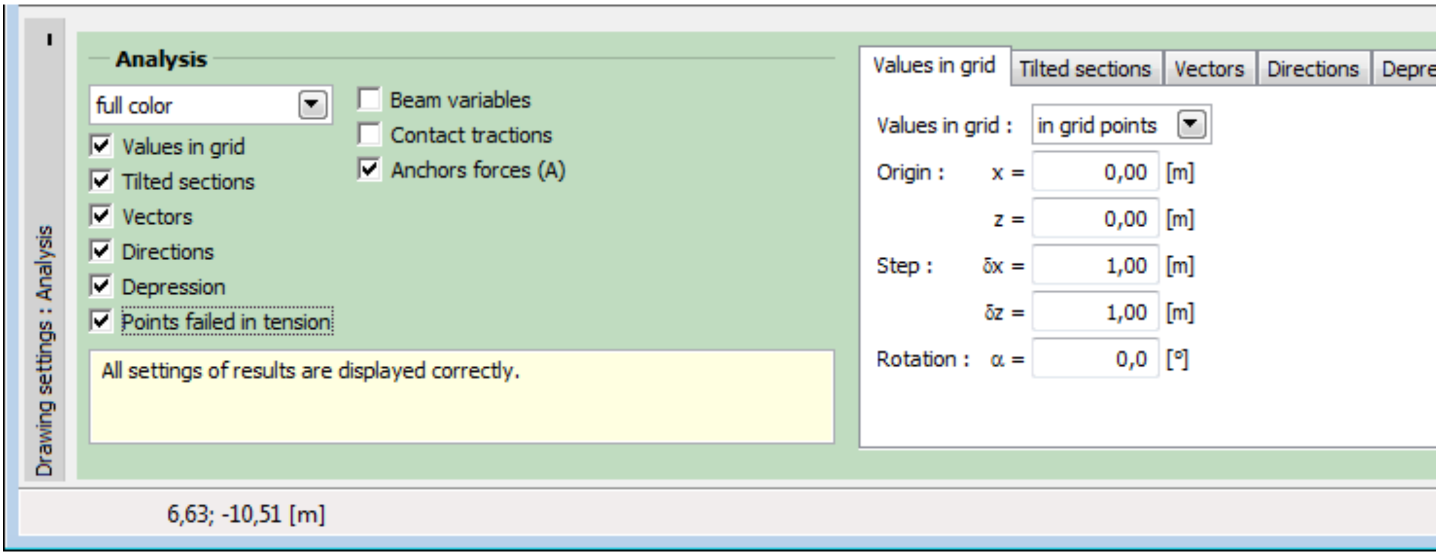
The toolbar contains **the most often used operating elements** needed to view the results on the desktop. Detailed setting of the style of plotting the results is available in the "Drawing Settings: Analysis" frame.

Similar to our other programs the results can be saved and printed. The plotting style can be adjusted in the "Drawing Settings" frame.

Drawing Settings : Analysis

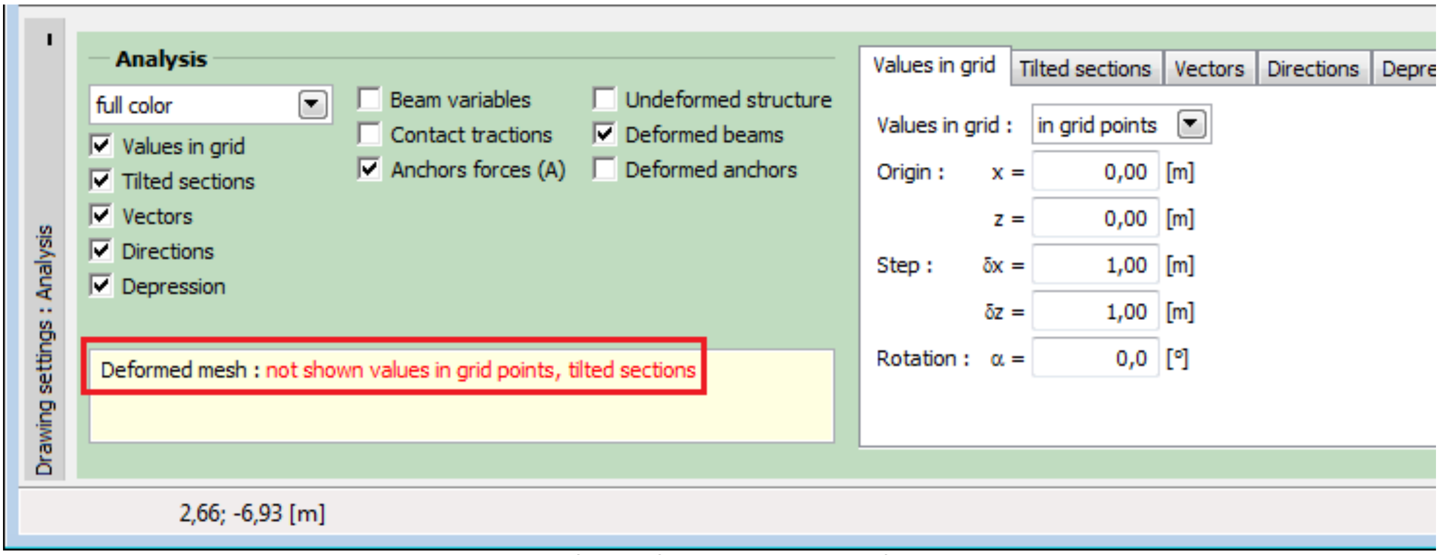
The "**Drawing settings: Analysis**" dialog window serves to select the type of variable to be displayed and the way it should appear on the screen. Individual settings can be later saved using the "Saved views" toolbar.

The tab "**Values in grid**" serves to set the basic parameters driving the visualization of surface variables and FE mesh - other tabs are used to define other types of outputs.



Frame "Drawing settings: Analysis" - tab "Values in grid"

Owing to the clarity of graphical presentation it is not possible to plot some of the results **at the same time**. It is not possible to plot a deformed mesh together with distributions of internal forces along beams - only one option must be selected. If an unacceptable combination is selected, the displays a warning message in the bottom part of the dialog window. The present example shows an unacceptable combination of *deformed mesh/values in a mesh grid* set on the toolbar "Results".



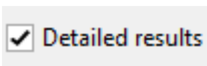
Warning for conflict in the plotting of results

List of Variables

The following variables can be displayed (values in the soil/rock body):

List of variables displayed by the program - basic variables

Notation	**Description**	**Symbol**	**Unit GEO 5**
Displacement d_Z	Displacement in the Z direction	d_z	$[mm]$
Displacement d_X	Displacement in the X direction	d_x	$[mm]$
Displacement resultant $\|d\|$	Displacement resultant	$\|d\|$	$[mm]$
Effective stress $\sigma_{x, eff}$	Effective normal stress in the X direction	$\sigma_{x, eff}$	$[kPa]$
Effective stress $\sigma_{xz\ eff}$	Effective normal stress in the Z direction	$\sigma_{z, eff}$	$[kPa]$
Effective hoop stress $\sigma_{\theta, eff}$	Effective hoop stress in the θ direction	$\sigma_{\theta, eff}$	$[kPa]$
Total stress $\sigma_{x, tot}$	Total normal stress in the X direction	$\sigma_{x, tot}$	$[kPa]$
Total stress $\sigma_{z, tot}$	Total normal stress in the Z direction	$\sigma_{z, tot}$	$[kPa]$
Total hoop stress $\sigma_{\theta, tot}$	Total hoop stress in the θ direction	$\sigma_{\theta, tot}$	$[kPa]$
Shear stress τ_{xz}	Shear stress in the XZ plane	τ_{xz}	$[kPa]$
Equivalent deviatoric stress J	Eqvivalent deviatoric stress	J	$[kPa]$
Total pore pressure u_{tot}	Total pore presure	u_{tot}	$[kPa]$
Equivalent deviatoric strain E_d	Equivalent deviatoric strain	E_d	$[\%]$
Plastic equivalent deviatoric strain $E_{d, pl}$	Plastic equivalent deviatoric strain	$E_{d, pl}$	$[\%]$
Pore presure u	Pore presure	u	$[kPa]$
Water flow velocity $v_{w, x}$	Water flow velocity in the X direction	$v_{w, x}$	$[m/day]$
Water flow velocity $v_{w, z}$	Water flow velocity in the Z direction	$v_{w, z}$	$[m/day]$
Water flow velocity resultant $\|v_w\|$	Water flow velocity resultant	$\|v_w\|$	$[m/day]$
Degree of saturation S	Degree of saturation	S	$[\%]$

List of variables displayed by the program - detailed variables (checkbox 

)

Notation	**Description**	**Symbol**	**Unit GEO 5**
Effective mean stress $\sigma_{m, eff}$	Effective mean stress	$\sigma_{m, eff}$	$[kPa]$
Total mean stress $\sigma_{m, tot}$	Total mean stress	$\sigma_{m, tot}$	$[kPa]$

Steady-state pore pressure u_{ss}	Steady-state pore pressure	u_{ss}	$[kPa]$
Excess pore pressure u_{exc}	Excess pore pressure	u_{exc}	$[kPa]$
Volumetric strain ε_v	Volumetric strain	ε_v	$[\%]$
Plastic volumetric strain $\varepsilon_{v,\,pl}$	Plastic volumetric strain	$\varepsilon_{v,\,pl}$	$[\%]$
Total head h	Total head	h	$[m]$
Total head gradient ∇xh	Total head gradient in the X direction	∇xh	[-]
Total head gradient ∇zh	Total head gradient in the Z direction	∇zh	[-]
Total head gradient resultant $\lvert\nabla\rvert$	Total head gradient resultant	$\lvert\nabla\rvert$	[-]
Seepage force $p_{w,\,x}$	Seepage force in the X direction	$p_{w,\,x}$	$[kN/m^3]$
Seepage force $p_{w,\,z}$	Seepage force in the Z direction	$p_{w,\,z}$	$[kN/m^3]$
Seepage force resultant $\lvert p_w\rvert$	Seepage force resultant	$\lvert p_w\rvert$	$[kN/m^3]$
Relative permeability K_r	Relative permeability	K_r	$[\%]$
Suction u_{succ}	Sucction	u_{succ}	$[kPa]$
Effective principal stress $\sigma_{1,\,eff}$	Effective principal stress in the 1 direction	$\sigma_{1,\,eff}$	$[kPa]$
Effective principal stress $\sigma_{2,\,eff}$	Effective principal stress in the 2 direction	$\sigma_{2,\,eff}$	$[kPa]$
Effective principal stress $\sigma_{3,\,eff}$	Effective principal stress in the 3 direction	$\sigma_{3,\,eff}$	$[kPa]$
Total principal stress $\sigma_{1,\,tot}$	Total principal stress in the 1 direction	$\sigma_{1,\,tot}$	$[kPa]$
Total principal stress $\sigma_{2,\,tot}$	Total principal stress in the 2 direction	$\sigma_{2,\,tot}$	$[kPa]$
Total principal stress $\sigma_{3,\,tot}$	Total principal stress in the 3 direction	$\sigma_{3,\,tot}$	$[kPa]$
Strain ε_x	Strain in the X direction	ε_x	$[\%]$
Strain ε_z	Strain in the Z direction	ε_z	$[\%]$
Hoop strain ε_θ	Hoop strain	ε_θ	$[\%]$
Shear strain γ_{xz}	Shear strain in the XZ plane	γ_{xz}	$[\%]$
Plastic strain $\varepsilon_{x,\,pl}$	Plastic strain in the X direction	$\varepsilon_{x,\,pl}$	$[\%]$
Plastic strain $\varepsilon_{z,\,pl}$	Plastic strain in the Z direction	$\varepsilon_{z,\,pl}$	$[\%]$
Plastic hoop strain $\varepsilon_{\theta,\,pl}$	Plastic hoop strain	$\varepsilon_{\theta,\,pl}$	$[\%]$
Plastic shear $\gamma_{xz,\,pl}$	Plastic shear in the XZ plane	$\gamma_{xz,\,pl}$	$[\%]$
Principal strain ε_1	Principal strain in the 1 direction	ε_1	$[\%]$
Principal strain ε_2	Principal strain in the 2 direction	ε_2	$[\%]$
Principal strain ε_2	Principal strain in the 3 direction	ε_2	$[\%]$
Plastic principal strain $\varepsilon_{1,\,pl}$	Plastic principal strain in the 1 direction	$\varepsilon_{1,\,pl}$	$[\%]$
Plastic principal strain $\varepsilon_{2,\,pl}$	Plastic principal strain in the 2 direction	$\varepsilon_{2,\,pl}$	$[\%]$
Plastic principal strain $\varepsilon_{3,\,pl}$	Plastic principal strain in the 3 direction	$\varepsilon_{3,\,pl}$	$[\%]$
Void ratio e	Void ratio	e	[-]
Mobilized strength R_{mob}	Mobilized strength	R_{mob}	$[\%]$

Mobilized strength

Mobilized strength variable (in percentages) shows the level of mobilized soil/rock shear strength. If $R_{mob} = 100\%$, material reached the plastic state – the shear strength is fully mobilized.

The general equation for calculation of R_{mob}:

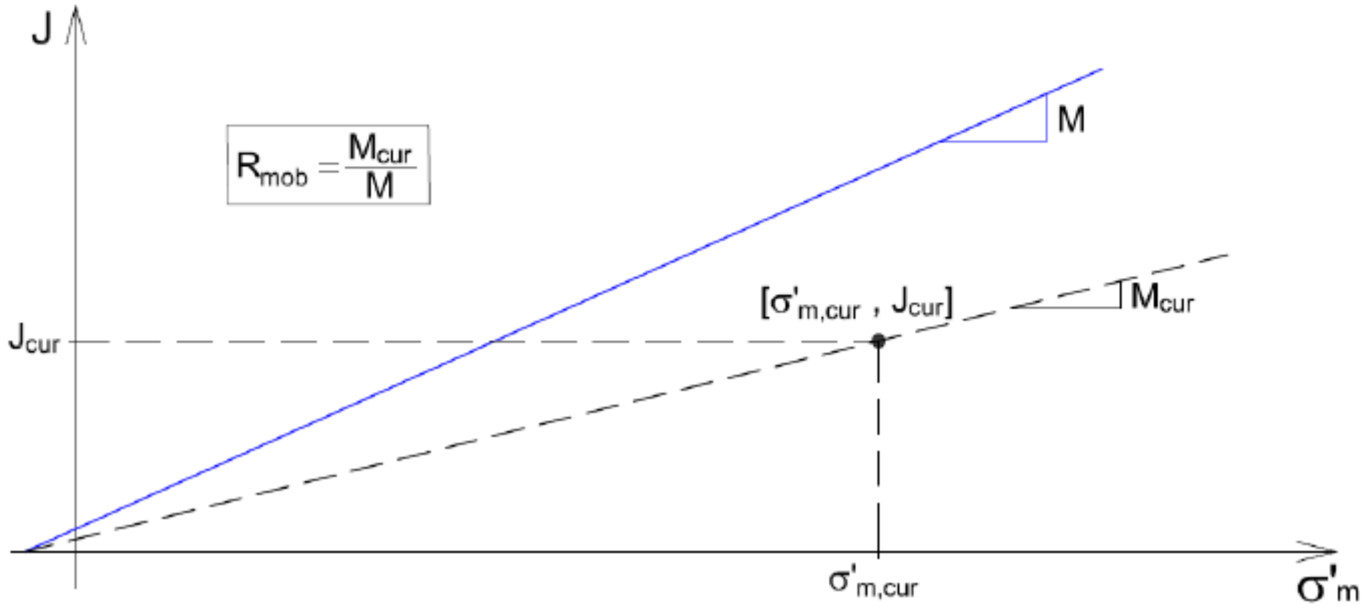
$$R_{mob} = \frac{M_{cur}}{M}\ x\ 100\ [\%]$$

where:

M_{cur} - the current mobilized shear strength [-]

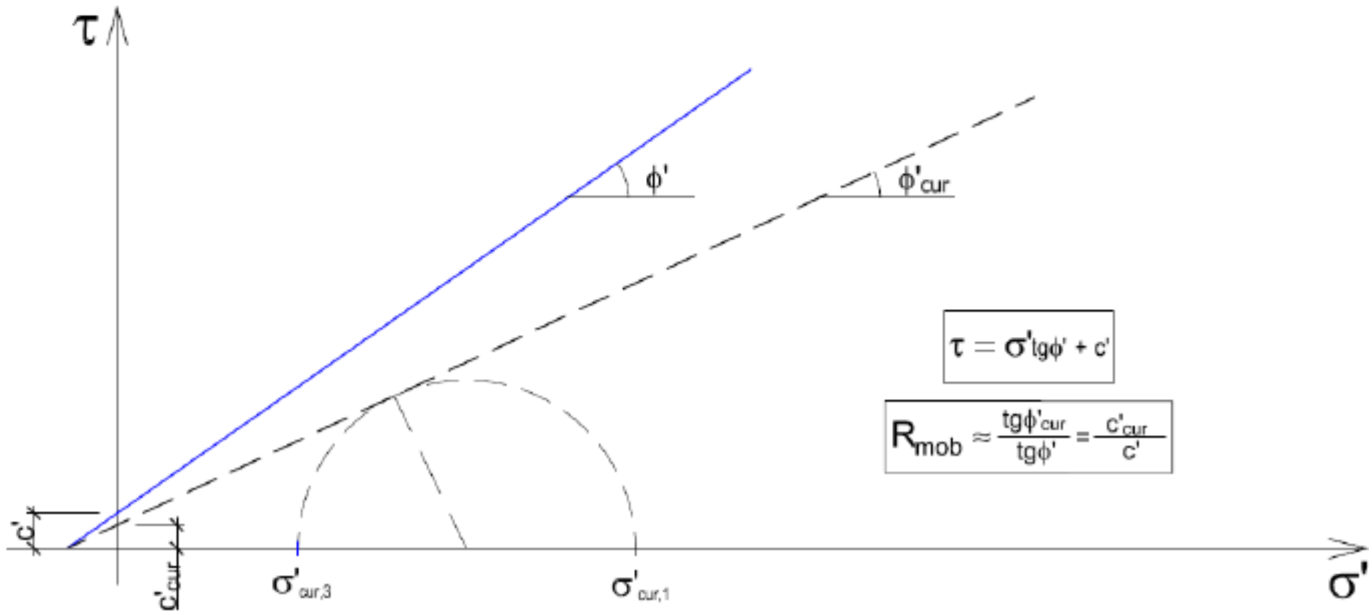
M - the shear strength (slope of plasticity/failure surface) [-]

Determination (calculation) of M_{cur} and M depends on the adopted constitutive model. Both variables (M_{cur} a M) are defined in σ'_m-J space (refer to picture below).



Mobilized shear strength in σ'_m*-J space*

Simply, for the case of 3ax compression:



Mobilized shear strength in σ'*-*τ *space*

Monitors

The frame "**Monitors**" contains a table with the list of input monitors. Adding monitors is performed in the "**New monitors**" dialog window. Either point or line-monitors can be introduced. The dialog window then serves to specify coordinates of the monitor and monitor activity.

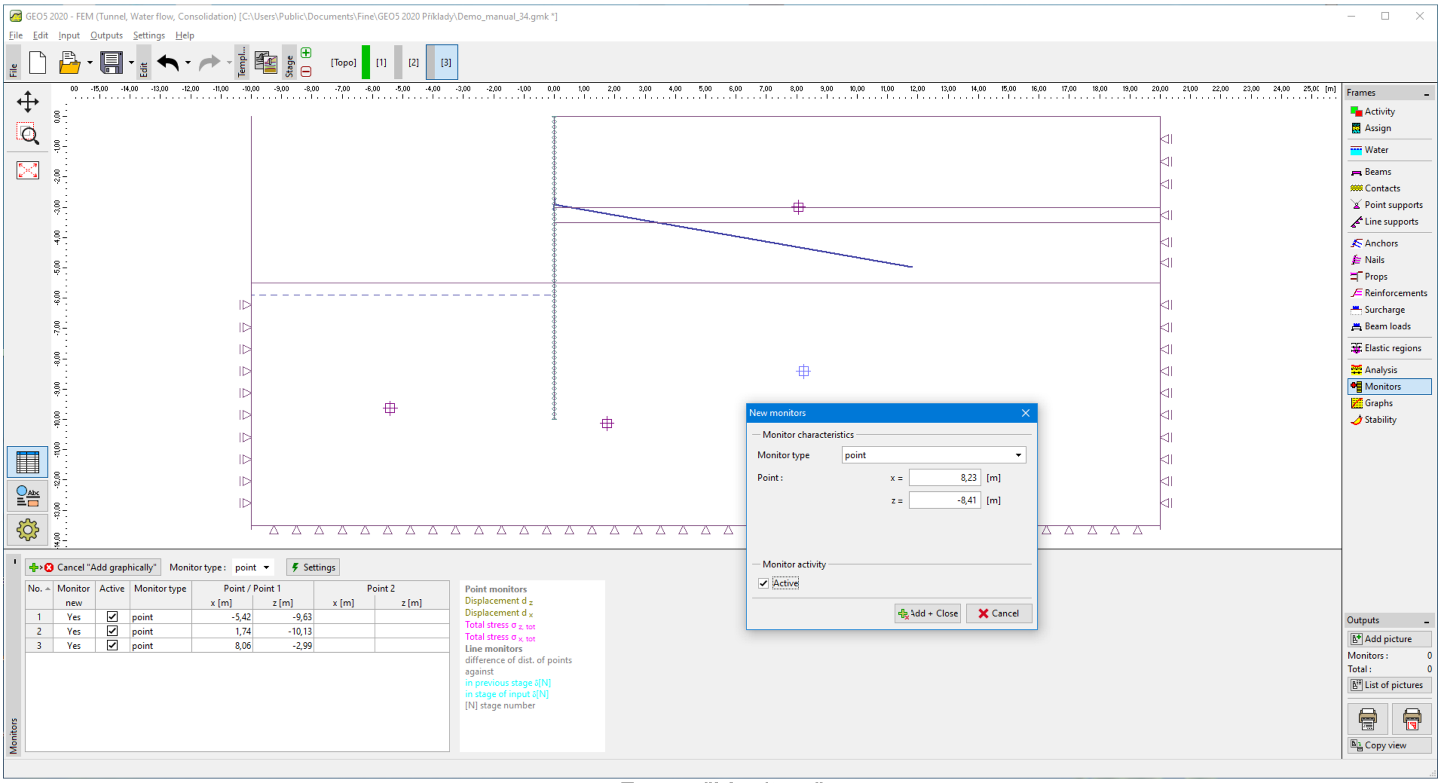
The monitors can also be edited on the desktop with the help of active objects.

The program allows for inputting an arbitrary number of points and line-monitors anywhere in the structure and also out of it. Monitors have several functions:

- displaying values of variables in a given point (point-monitor),
- displaying values of the difference of distance of two points in comparison with the previous stage $d\,[N]$ or in comparison with the input stage, where N is the stage number (line-monitor).

The point monitors store also the values of variables recorded during the analysis in individual stages. These can be written into an output protocol or used to create graphs.

The list of variables plotted for individual monitors is set in the "Monitor settings" dialog window. To open the window use the "**Settings**" button in the horizontal toolbar "**Monitors**".



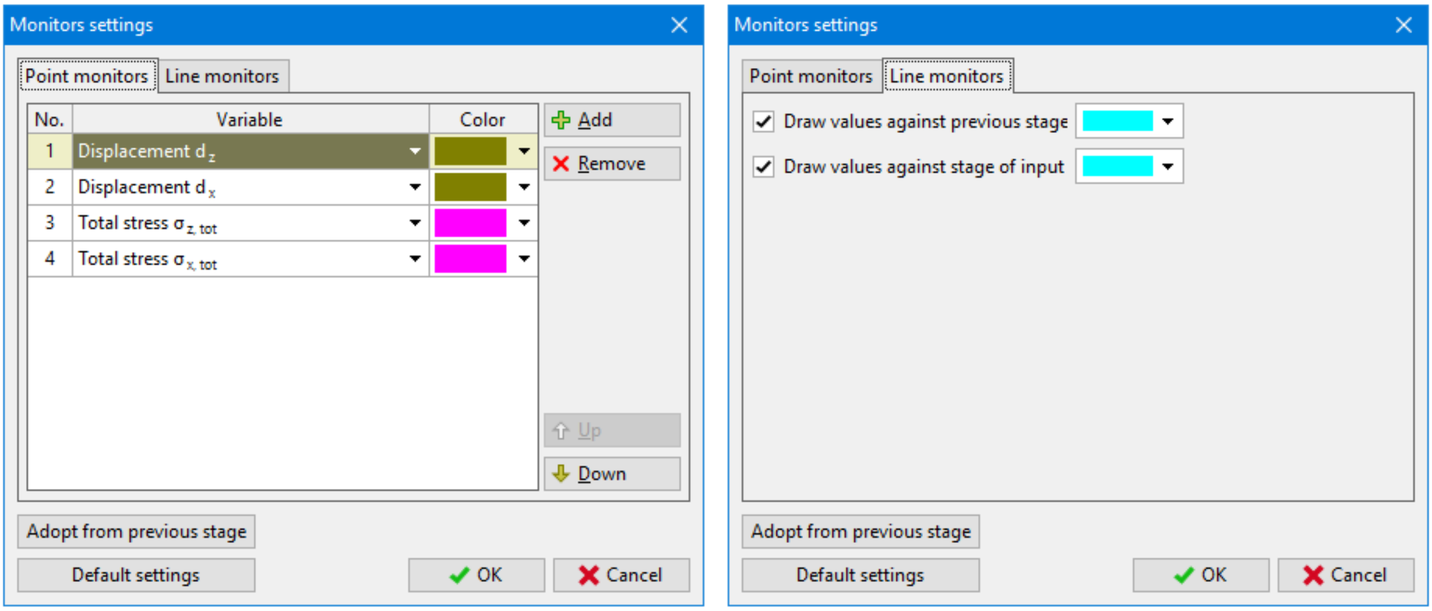
Frame "Monitors"

Monitors Settings

The "**Monitor settings**" dialog window serves to set variables whose values will be plotted for a given monitor (point-monitors). Setting for a given list of variables can be adopted from the previous stage of construction using the "**Adopt from the previous stage**" button. Four variables are displayed by default. Additional variables can be added to the list using the "**Add**" button. The variable can be removed from the list using the "**Remove**" button.

For line-monitors, the dialog window serves to activate the plot of values in comparison with the previous stage or the input stage, respectively.

For both point and line-monitors, it is possible to specify the color range of plotted values.



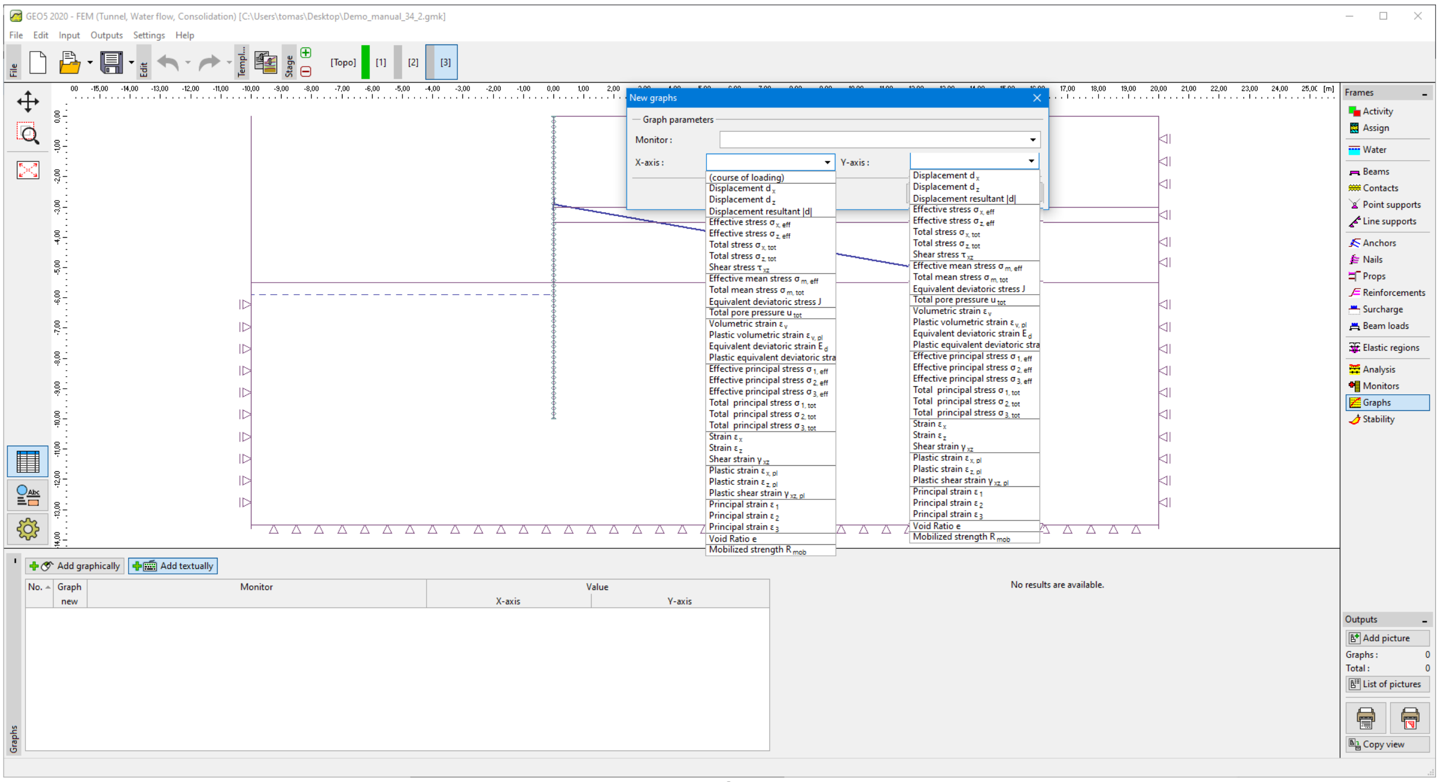
Dialog window "Monitors settings"

Graphs

The frame "**Graphs**" contains a table with the list of input graphs. Adding (editing) monitors is performed in the "**New graphs**" dialog window. The dialog window serves to enter the monitor number for which the graph will be created and the variables adopted for the X and Y-axis respectively.

The graphs can also be edited on the desktop with the help of active objects.

The program allows for inputting an arbitrary number of graphs at points of input monitors. Graphs allow for plotting a mutual dependence of individual variables stored in monitors during the course of analysis.



Frame "Graphs"

Stability

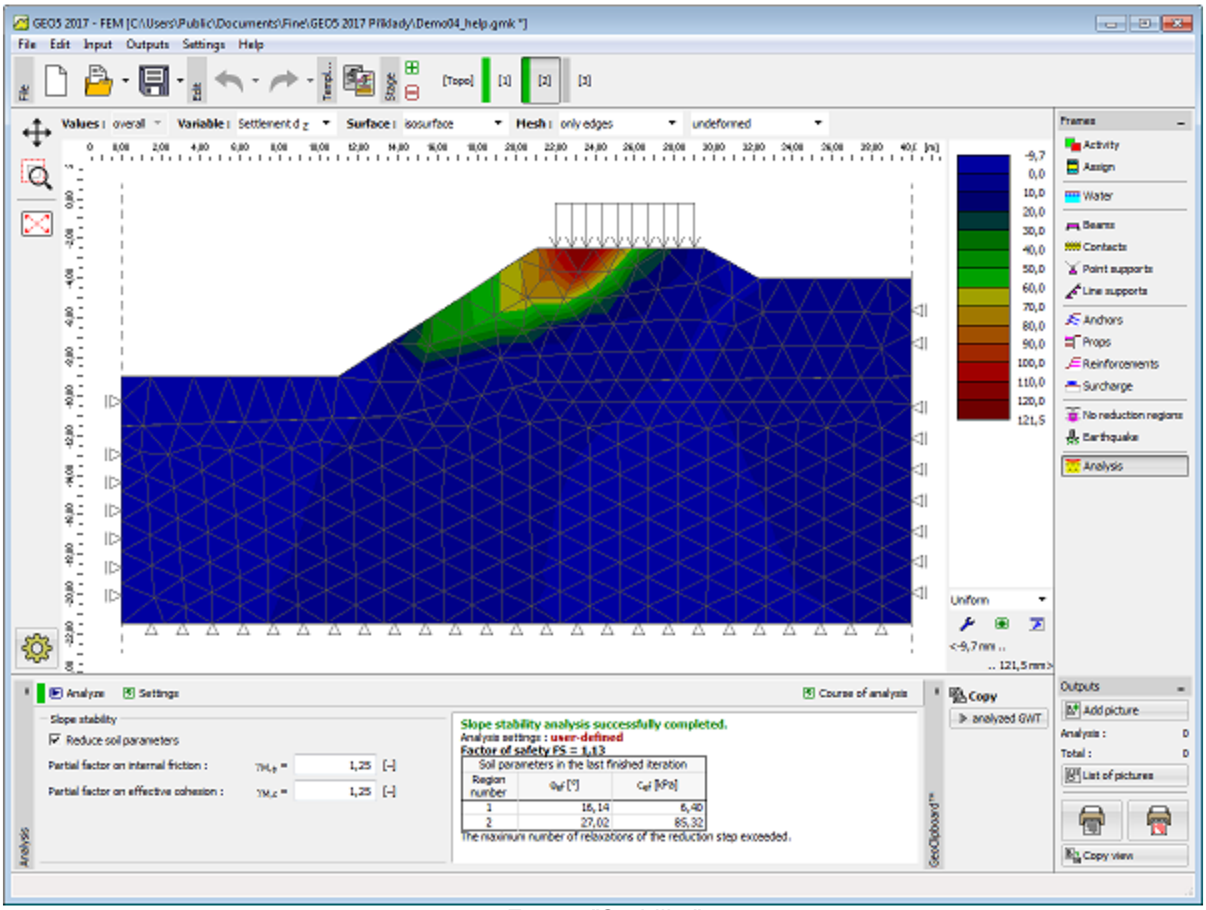
In the stability (safety factor) analysis the program **reduces the original strength parameters** - the angle of internal friction and cohesion - until failure occurs. The analysis then results in a **factor of safety** that corresponds to the classical methods of limit equilibrium.

The safety factor analysis requires using six-node elements. Since plastic slip is the main failure mechanism we also require that the Mohr-Coulomb, the modified Mohr-Coulomb, or the Drucker-Prager plasticity model be used for all soils. The default setting can be adjusted in the "**Analysis settings**" dialog window.

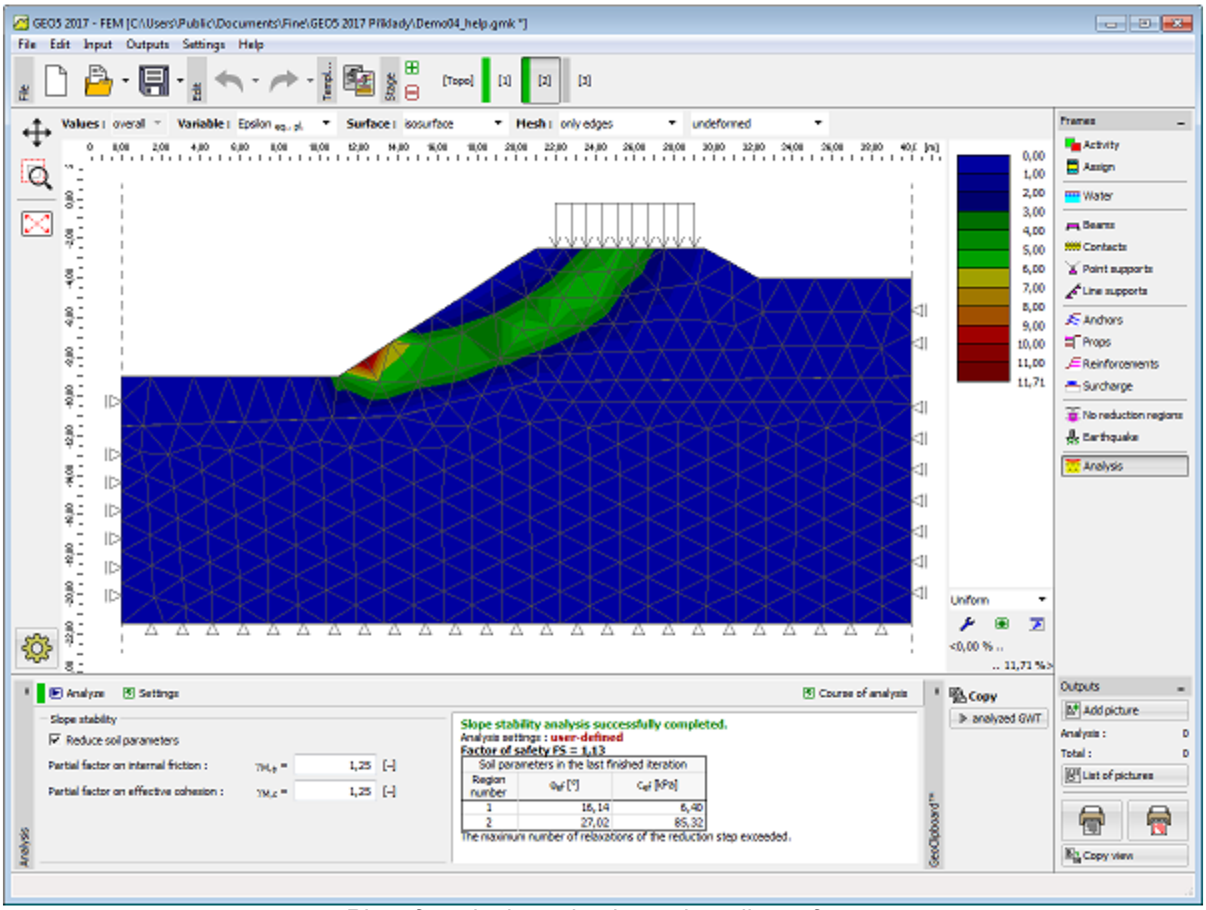
In the stability analysis mode, the only variables available for graphical representation are **displacements** (in the Z and X-directions) and equivalent total and plastic **strains**. The deformation of a soil body corresponds to the state of failure attained for the reduced soil parameters - therefore, it does not correspond to the real state of deformation of the soil body. Instead, it provides a good insight about the entire slope response of earth structure in general at the onset of failure.

A suitable way of presenting the stability analysis results are **vectors of displacements** plotted together with the **equivalent plastic strain**. The localized plastic deformation provides visible evidence about the possible location of the **critical slip surface**.

The program allows us to analyze a slope stability according to the theory of limit state or EN1997-1, design approach 3. In the frame we enter **partial factor on internal friction** $\gamma_{M,\phi}$ and **partial factor on effective cohesion** $\gamma_{M,c}$, which reduce soil parameters. If we use reduction according to EN 1997 ($\gamma_{M,\phi}$ = 1,25, $\gamma_{M,c}$ = 1,25), the slope is satisfactory for FS =1,00.



Frame "Stability"



Plot of equivalent plastic strain - slip surface

Setting Basic Parameters of Slope Stability Analysis

The safety factor analysis is based on the assumption that the total load applied to the soil/rock body is introduced in a single load step. The actual factor of safety is evaluated using the **method of reduction of strength parameters c , φ** . Regarding this, the factor of safety is defined as a scalar multiplier that reduces the original parameters c , φ to arrive at the state of failure.

Mathematically, the **factor of safety** is expressed as:

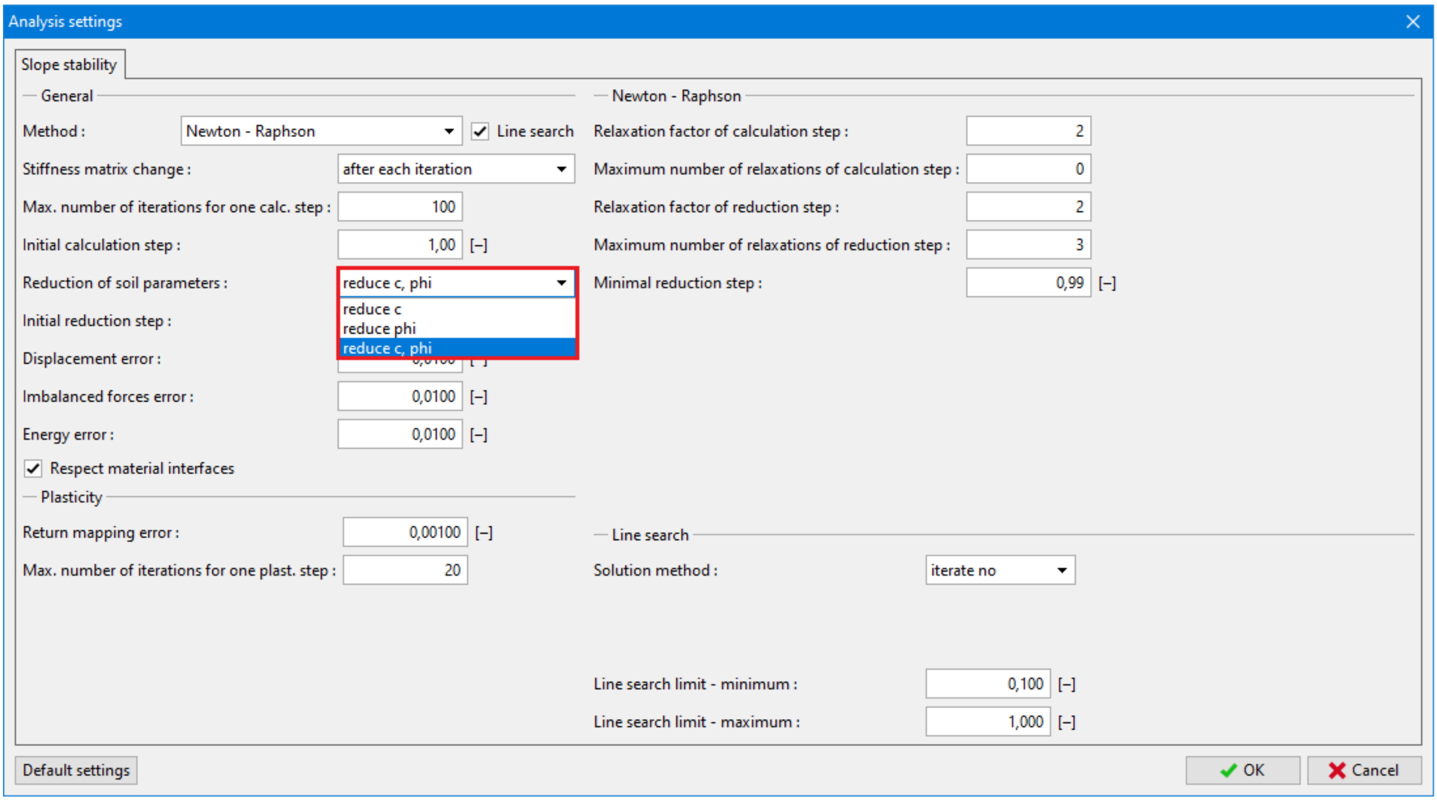
$$F = \frac{\tan \varphi^{orig}}{\tan \varphi^{failure}}$$

where: $\varphi_{original}$ - the original value of the angle of internal friction

$\varphi_{failure}$ - the value of the angle of internal friction at failure

Searching for the critical value of the factor of safety requires a systematic modification (reduction) of strength parameters c , φ leading to failure. In the framework of the NRM, the state of failure is determined as the state for which the solution fails to converge. The process of searching for critical c , φ is driven by the following parameters.

1. **Reduction** - reduction factor (scalar multiplier) to reduce parameters c , φ . During the course of analysis, this parameter is progressively updated.
2. **Min. reduction factor** - the limit value, below which the value of the reduction factor should not fall during the searching process. This parameter ensures that the computation will not continue for needless low values of the reduction factor. It is one of the parameters to terminate the searching process.
3. **Reduction of soil parameters** - this parameter allows us to define which of the parameters c , φ should be reduced. The default setting assumes that both parameters are reduced at the same time.

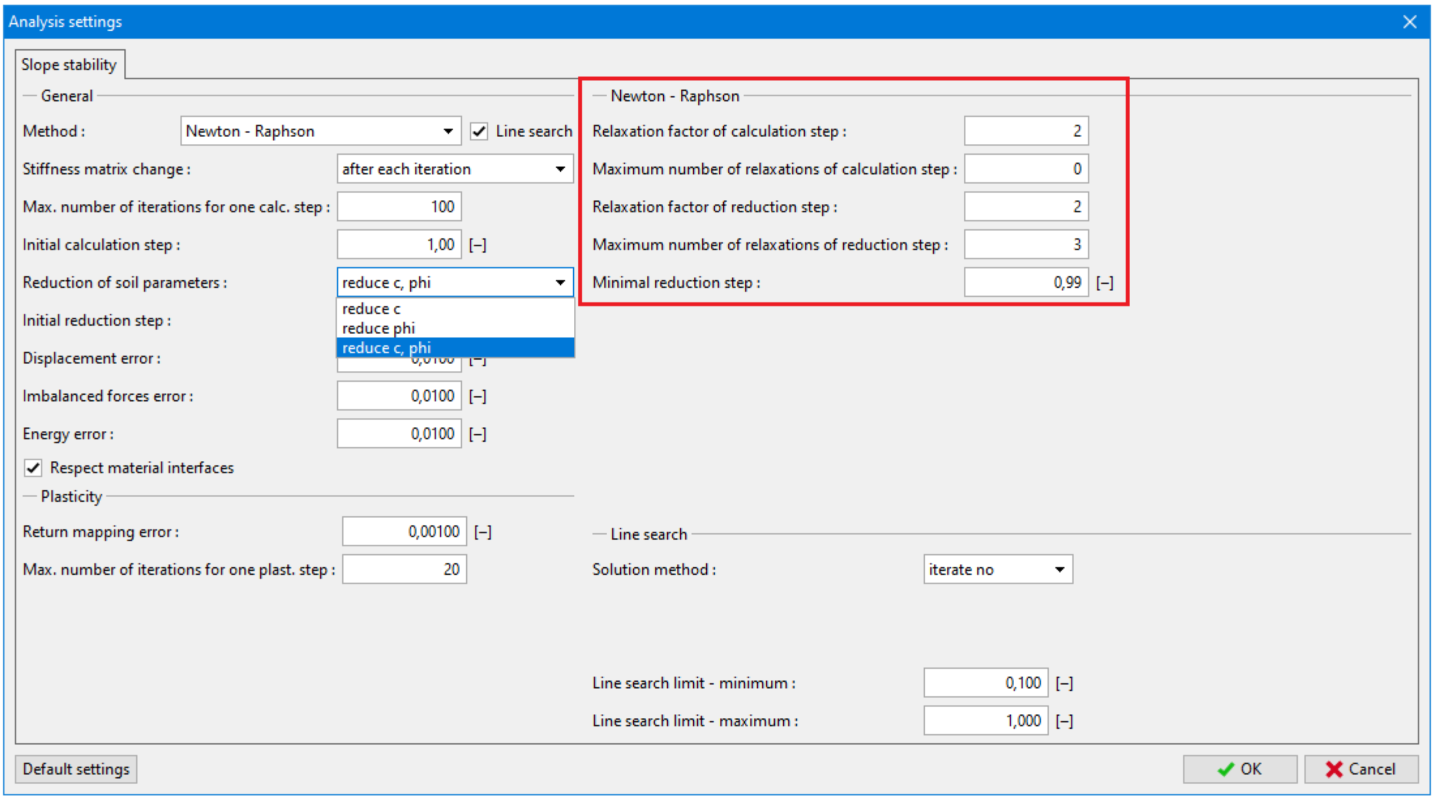


Basic parameters of slope stability analysis

Setting Driving Parameters of Relaxation of Reduction Factor

Similar to standard analysis the program adaptively adjusts the value of reduction factor. Providing the solution fails to converge for a given set of parameters c , φ , the reduction factor is relaxed and the analysis is restarted. This approach is driven by the parameters set in the "**Analysis settings**" - tab **Newton Raphson**.

The "**Relaxation factor**" serves to reduce the current value of the Reduction factor of parameters c , φ . The analysis is terminated once the value of the reduction factor drops below the minimum one or the maximum number of allowable reductions is exceeded. When selecting the NRM the program allows us to determine the parameters c , φ which bring a soil body to a stable state in cases, where the solution with the original parameters c , φ was not found. The program then precedes in the opposite way so that parameters c , φ are **systematically increased until the stable solution is found**.



Parameters driving the process of reduction of strength parameters c, φ

No Reduction Regions

Frame **"No Reduction Regions"** contains a table of defined regions where the strength parameters are not reduced during stability analysis.

The input of the regions is identical to the standard input of interfaces.

In the stability analysis, we can define regions in which the strength parameters p_{hi} a c are not reduced. The regions without reduction are input in the chosen construction stage by a polygonal boundary. The reduction of strength parameters is prevented in every finite element that is - at least partially - positioned within the region.

The regions without reduction of strength parameters are helpful in situations where the stability analysis does not lead to the global shear surface and the failure kinematics together with the value of factor of safety is unreliable.

The feature is applicable to all material models admissible in stability analysis, i.e.

- Mohr-Coulomb
- Modified Mohr-Coulomb
- Drucker-Prager

Other material models are not allowed in stability analysis.

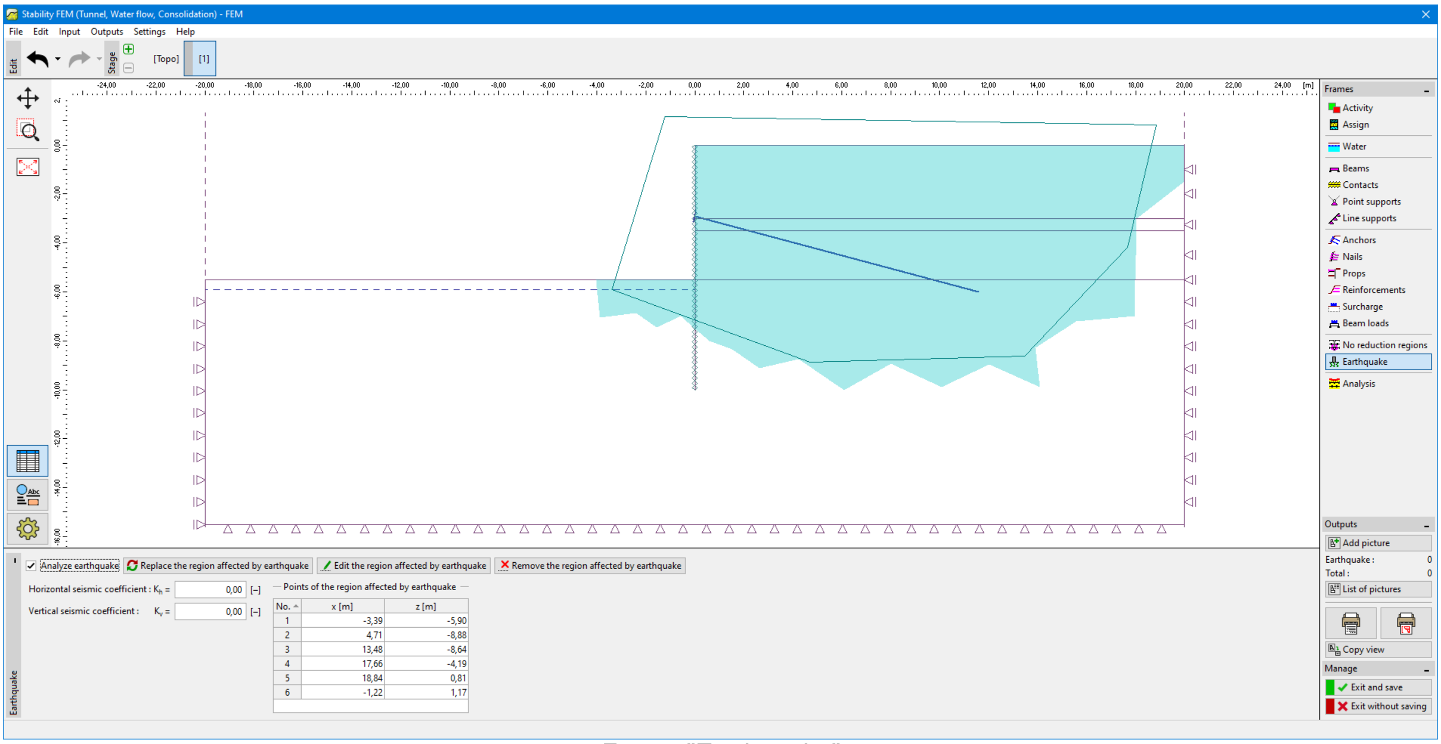
Earthquake

The "**Earthquake**" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

For the reason of successful converged analysis, it is necessary to define an area, where the **slip surface is predicted**. In this area, the horizontal and vertical acceleration acts on all finite elements (see picture). The finite elements outside this area are not affected by the earthquake. This area should not touch the edges of the task, otherwise, analysis does not have to find a solution.

The frame contains a table with a list of entered points. The input of regions is identical to the standard input of interfaces.

If not provided by measurements, the coefficients k_h and k_v can be calculated following the approach adopted from EN 1998-5.



Frame "Earthquake"

Outputs

This chapter describes the work with outputs (export document, printing pictures) in GEO5 programs using the toolbar "**Outputs**".

GEO5 programs allow creating **output document** with saved pictures from any mode of input or analysis. The pictures can be printed or exported, too.

- Print and Export Document
- Print and Export Desktop View

The work with pictures is described on pages:

- Adding Pictures
- List of Pictures

It is possible to **add user texts and images** into the output document using the "Annexes".

Page properties can be defined in each output document:

- Settings Header and Footer
- Page Properties
- Page Numbering

The information about the company can be input and printed in the output document:

- About the Company

Adding Pictures

The program allows us to store a current picture in all modes. Press the "**Add picture**" button on the "Outputs" vertical control bar. This opens the "**New picture**" dialog window and inserts the current view of the desktop into the window.

The picture is always linked to a certain input mode or analysis. (The current mode is displayed next to the picture name). When printing a document the picture is automatically added to a specific mode in the tree.

The program allows us to define the picture either for a specific stage of construction (or for the current analysis) or adjusting the setting so that the picture is added to the document in all stages of construction (or all analyses). The latter option is carried out by selecting "**all**" in the "**Stages**" combo list (or "**Analysis**" list).

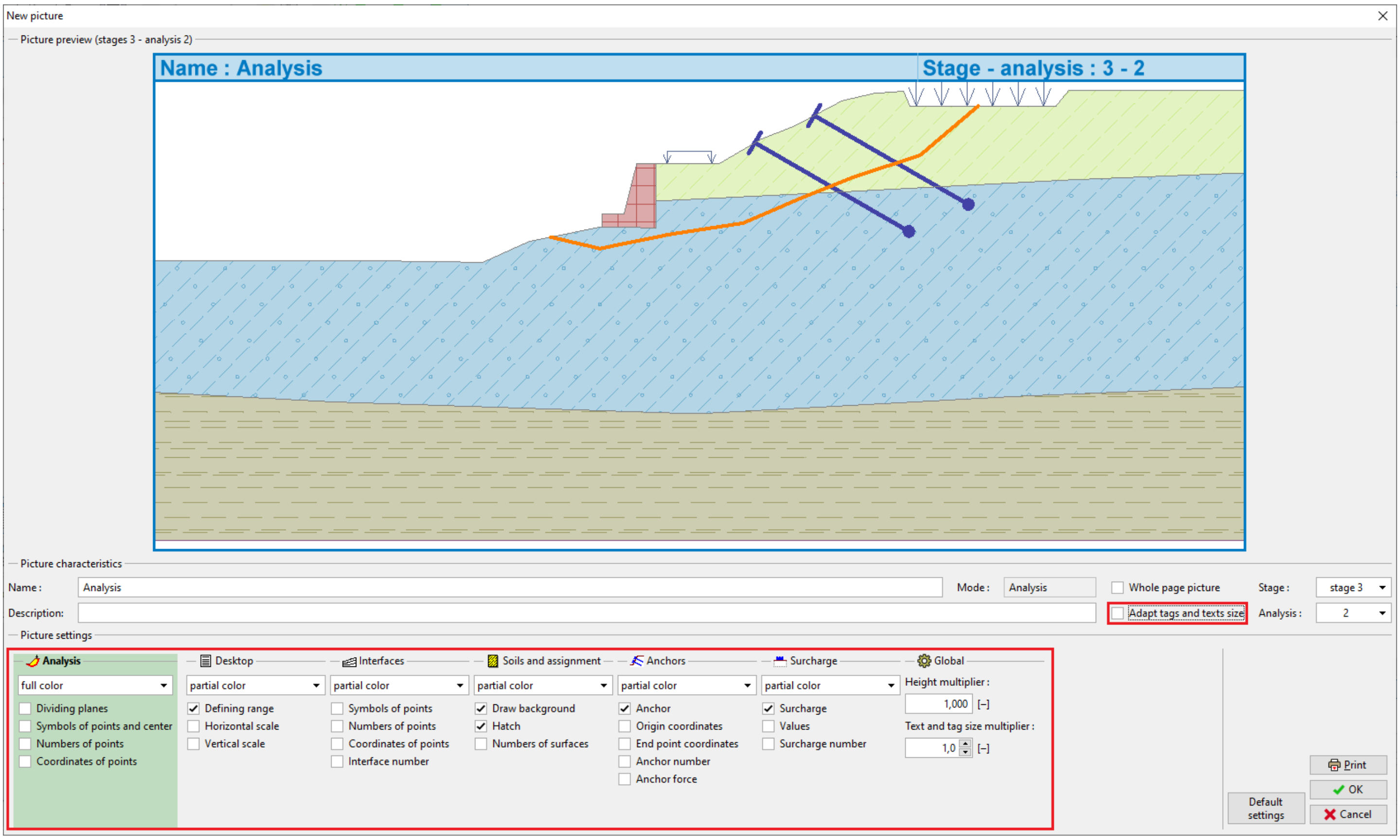
The "**Whole page picture**" checkbox allows us using the whole page picture in the document.

Warning: All input pictures are automatically regenerated whenever modifying data.

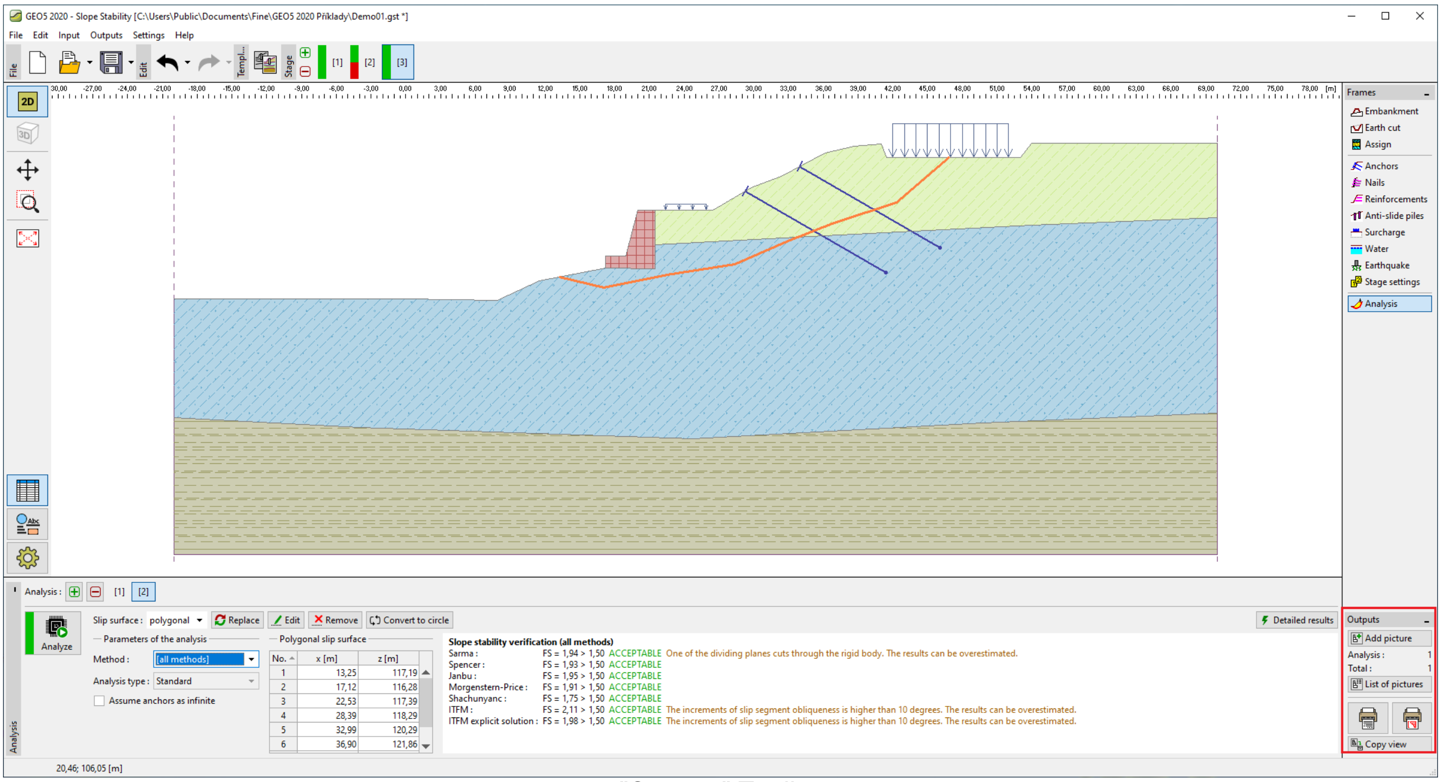
The "**Picture settings**" frame in the bottom part of the dialog window further allows adjusting the colors and style of lines (object) drawing. Settings in this part of the frame are taken from the drawing settings on the desktop. The function of the frame is the same as the desktop visualization settings and is described in "Drawing Settings".

The "**OK**" button stores the picture into the "Picture list". It can then be opened and modified at any time.

The picture can also be printed out of this window by pressing the "**Print**" button, which opens a dialog window for printing and exporting a desktop view. If the picture is active overall stages (or all analyses), then all possible combinations of pictures are printed all at once.



"New picture" Dialog window



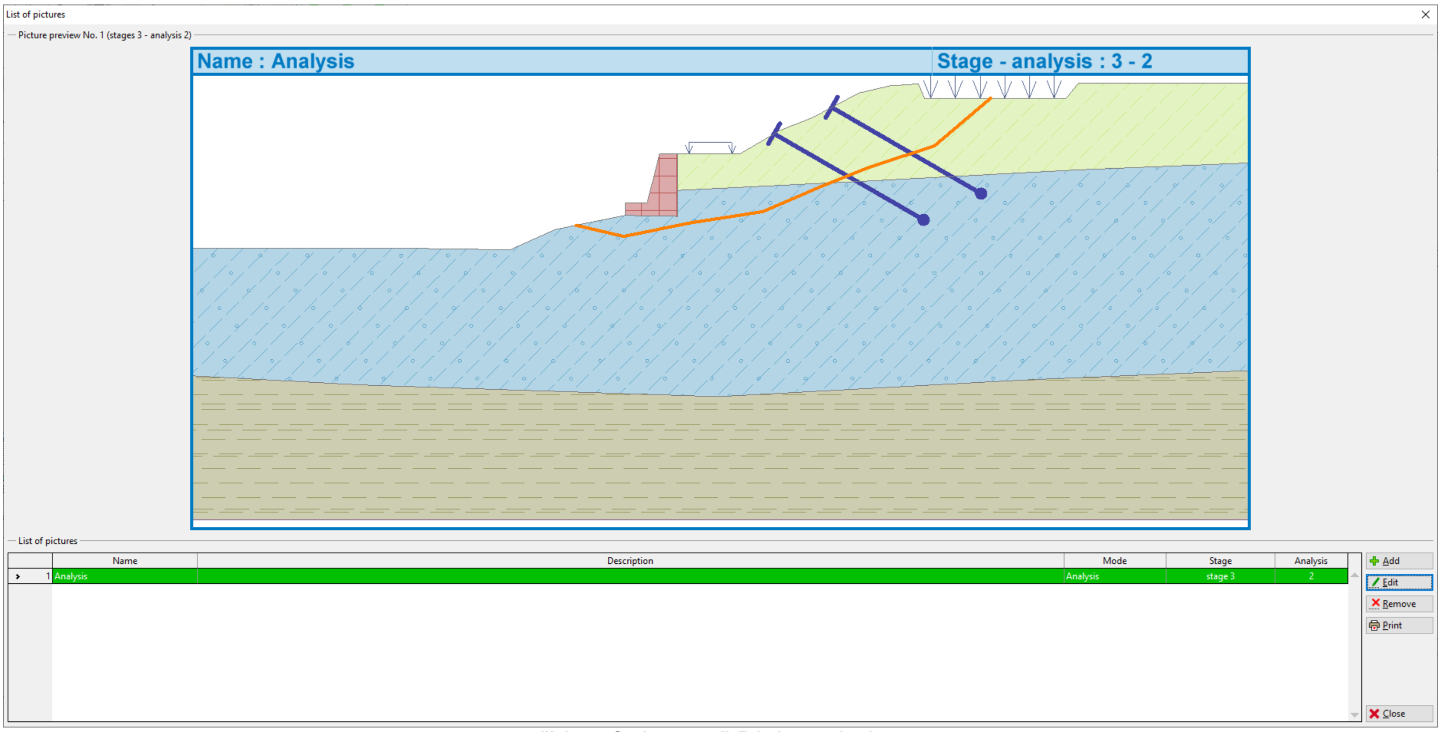
"Outputs" Toolbar

List of Pictures

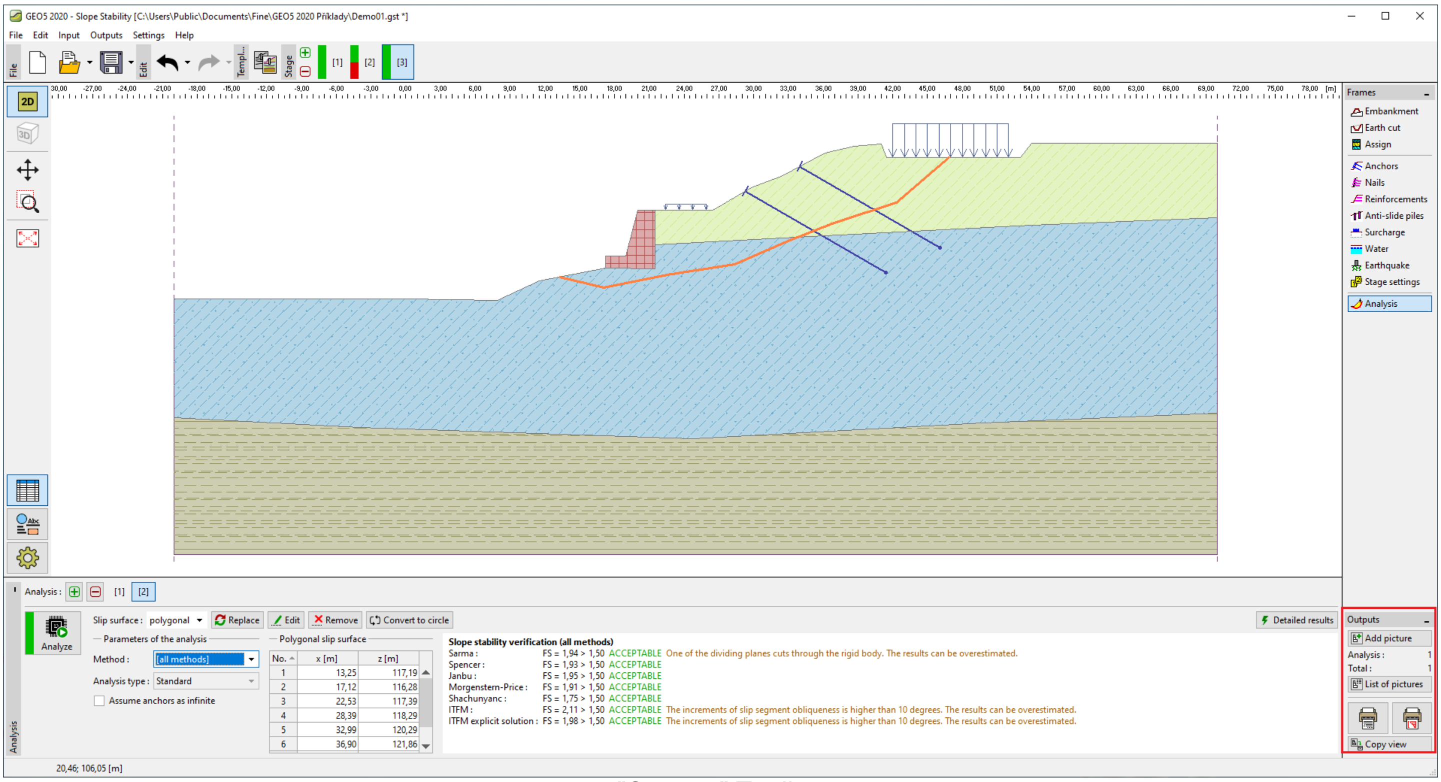
Pictures stored by the "New picture" dialog window are ordered in the table in the "**List of pictures**". The "**List of pictures**" dialog window can be opened by using the "Outputs" button on the vertical control bar. The table of the list of pictures contains the picture name and description, the mode which it was created in, and the stage of construction or the analysis number.

Individual pictures can be edited using the "**Modify**" button that opens the "**Edit picture**" dialog window (this window corresponds to the look and function of the "New picture" dialog window).

These pictures can be printed out of the window by pressing the "**Print**" button that opens a dialog window for printing and exporting a desktop view. Providing the picture is active over all stages of construction (overall analyses, respectively) then the program prints all possible combinations of the picture. If more pictures are selected then all of them will be printed out.



"List of pictures" Dialog window



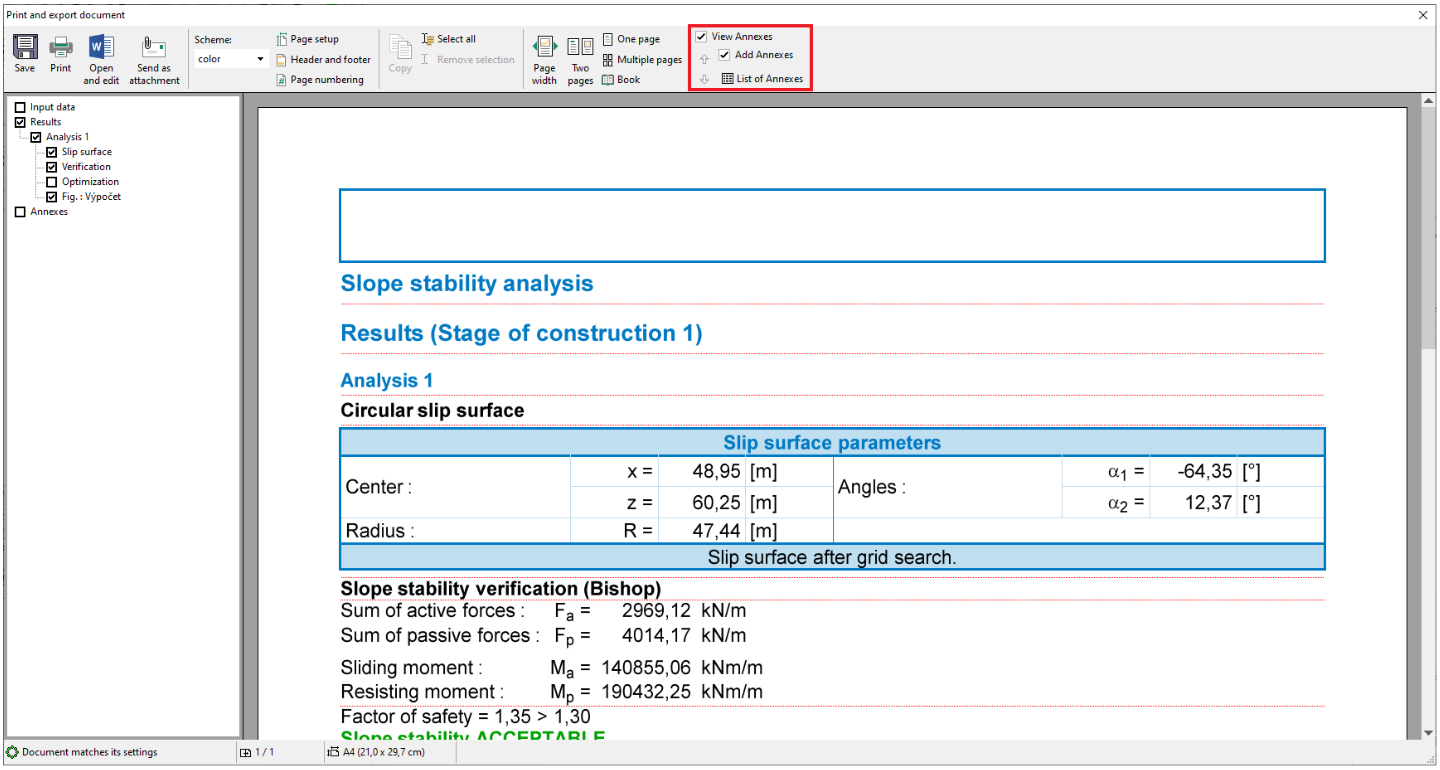
"Outputs" Toolbar

Annexes

The **output document** can contain an arbitrary amount of the **Annexes** - texts or pictures.

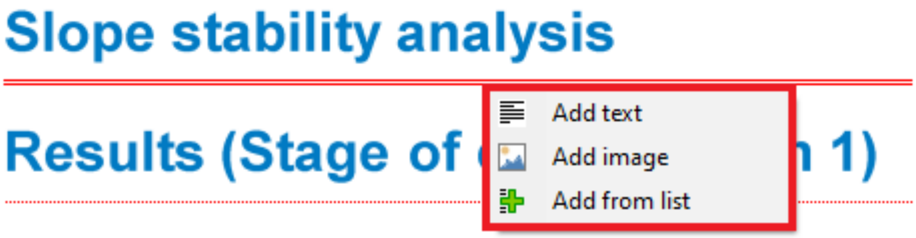
This tool is described in the **EM 45 - Using the "Annexes"**.

When the "**View annexes**" button is checked, **horizontal red lines** will appear in the output document. The annexes can be inserted by clicking on the red line.



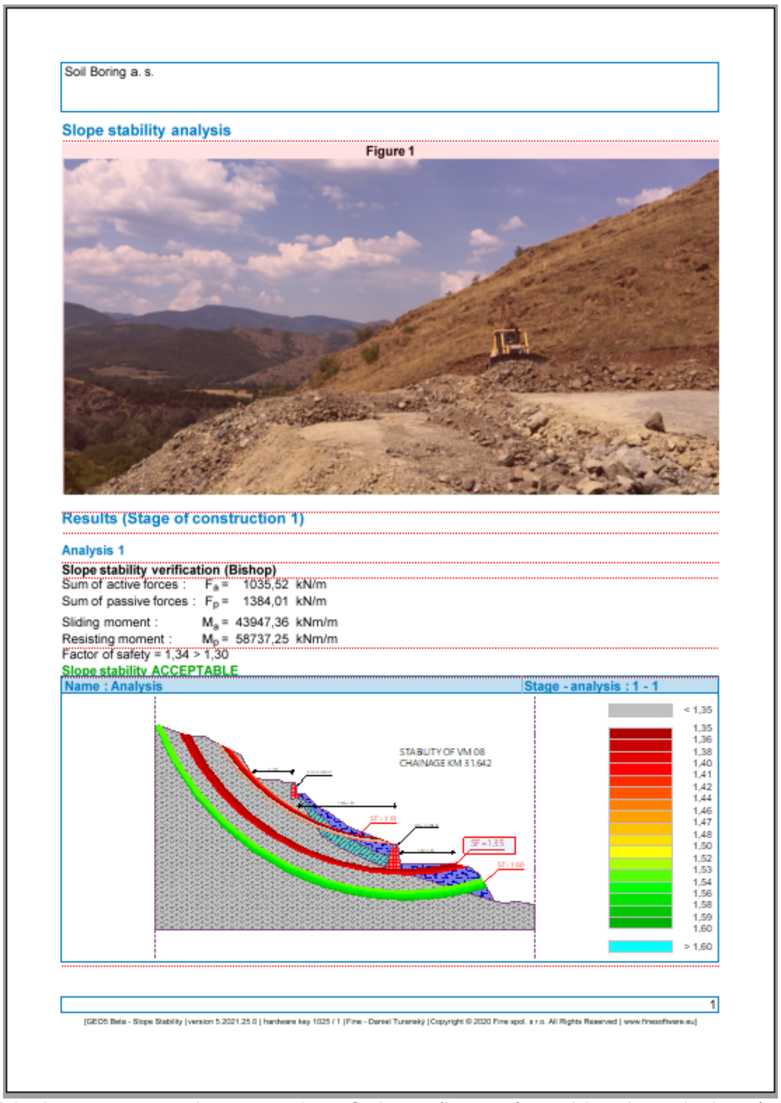
Adding annexes

By clicking on the line, the following **selection** will appear.



Selection of input

The added text or image will be inserted into the output protocol.



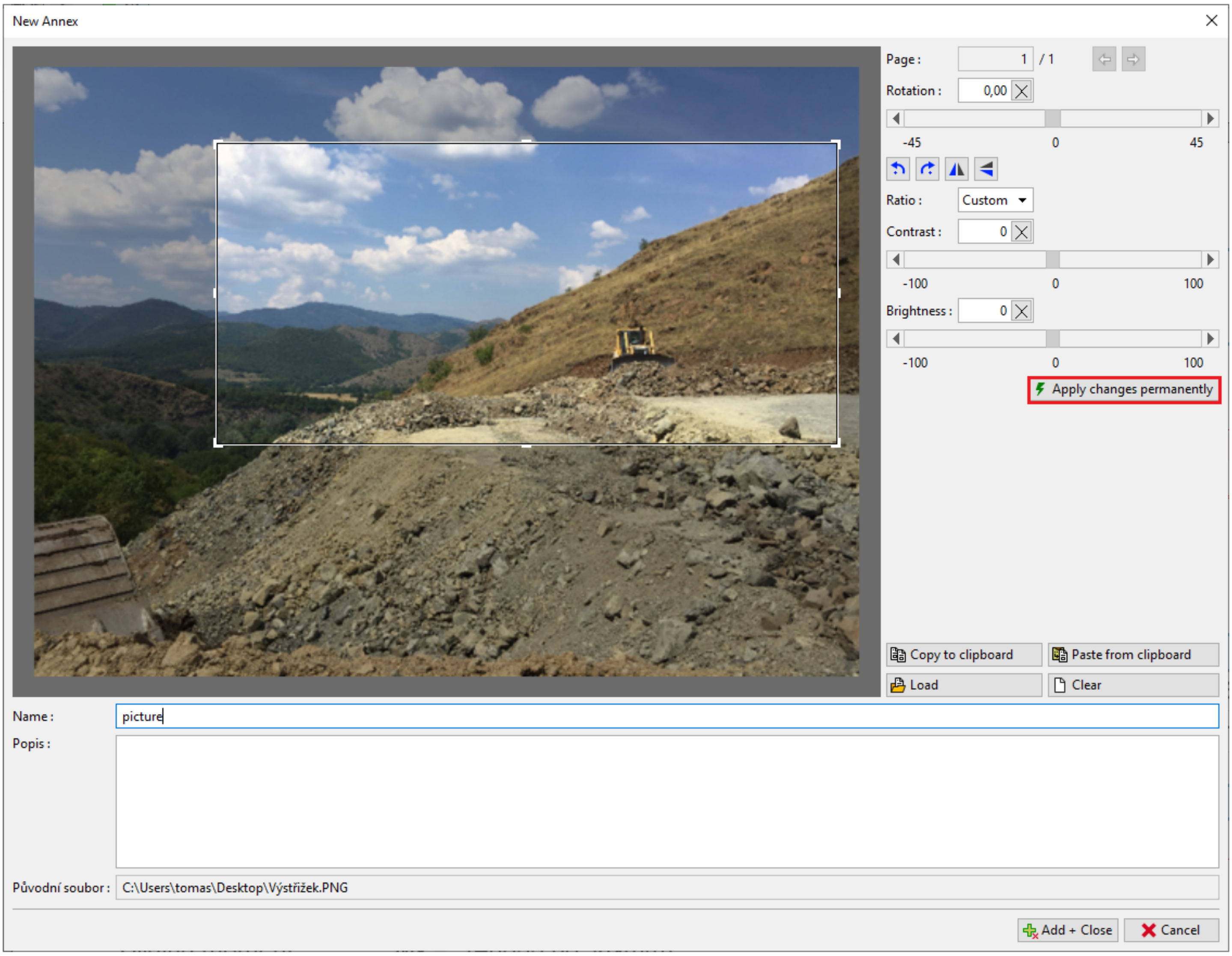
Added annexes - photography of slope (image) and its description (text)

All annexes can be viewed in the "List of Annexes".

Annexes - Pictures

The "**Add image**" function will open up a dialog window, that allows us to add pictures. We can open the picture by clicking the "**Load**" button. We can add images, in the most common formats -e.g. **JPG, JPEG, TIFF, BMP, PNG,** or **PDF**.

We can also use this frame to **edit** (crop, level the horizon, adjust the brightness and contrast) of the photo. By clicking the "**Apply changes permanently**" button, we will save it perennially and delete the original picture. The advantage of this method is, that by scaling the picture down, it will have a smaller file size, therefore the whole data file will be smaller.

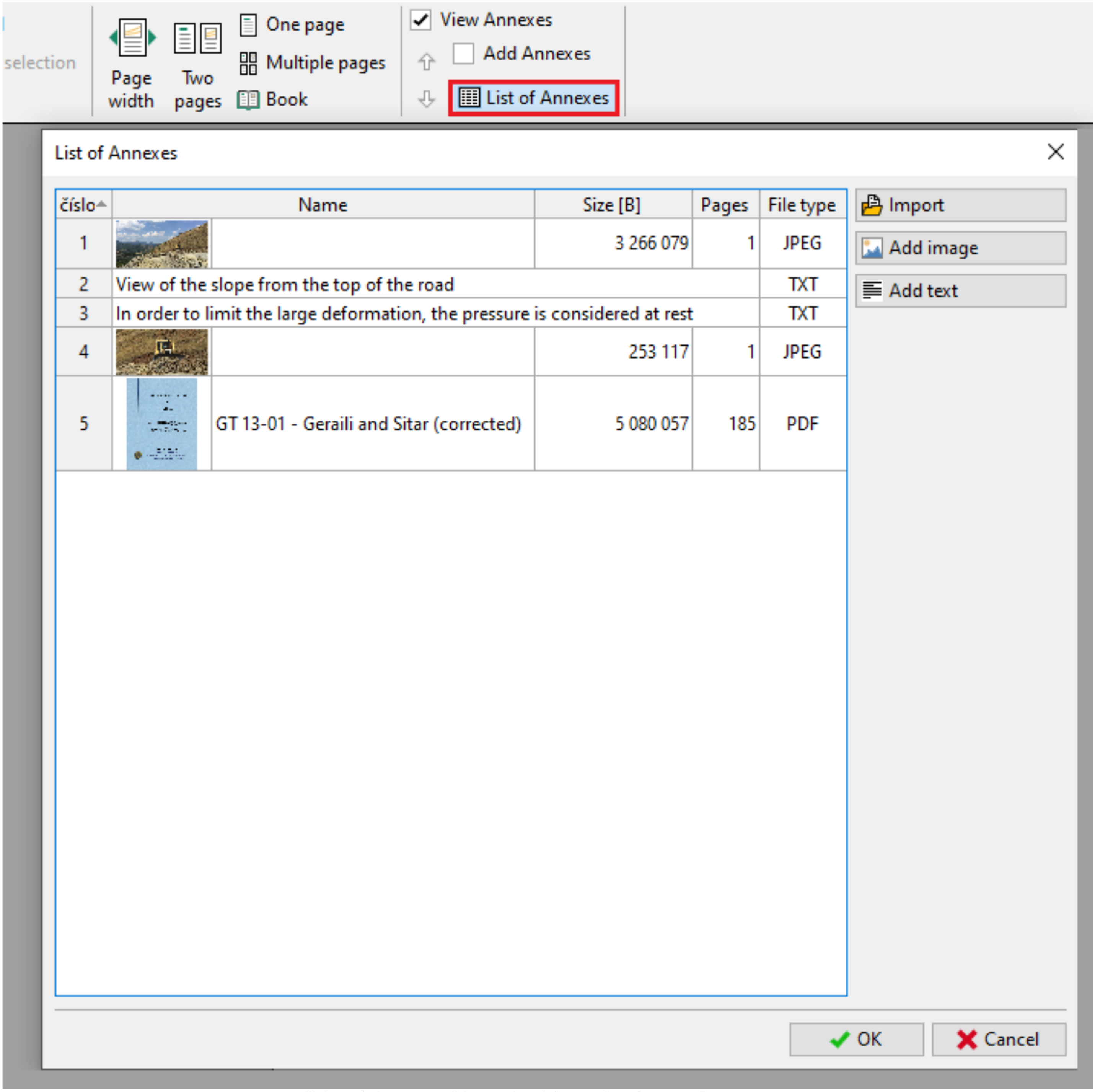


Edit and saving of the picture

All annexes are saved into the "List of Annexes".

List of Annexes

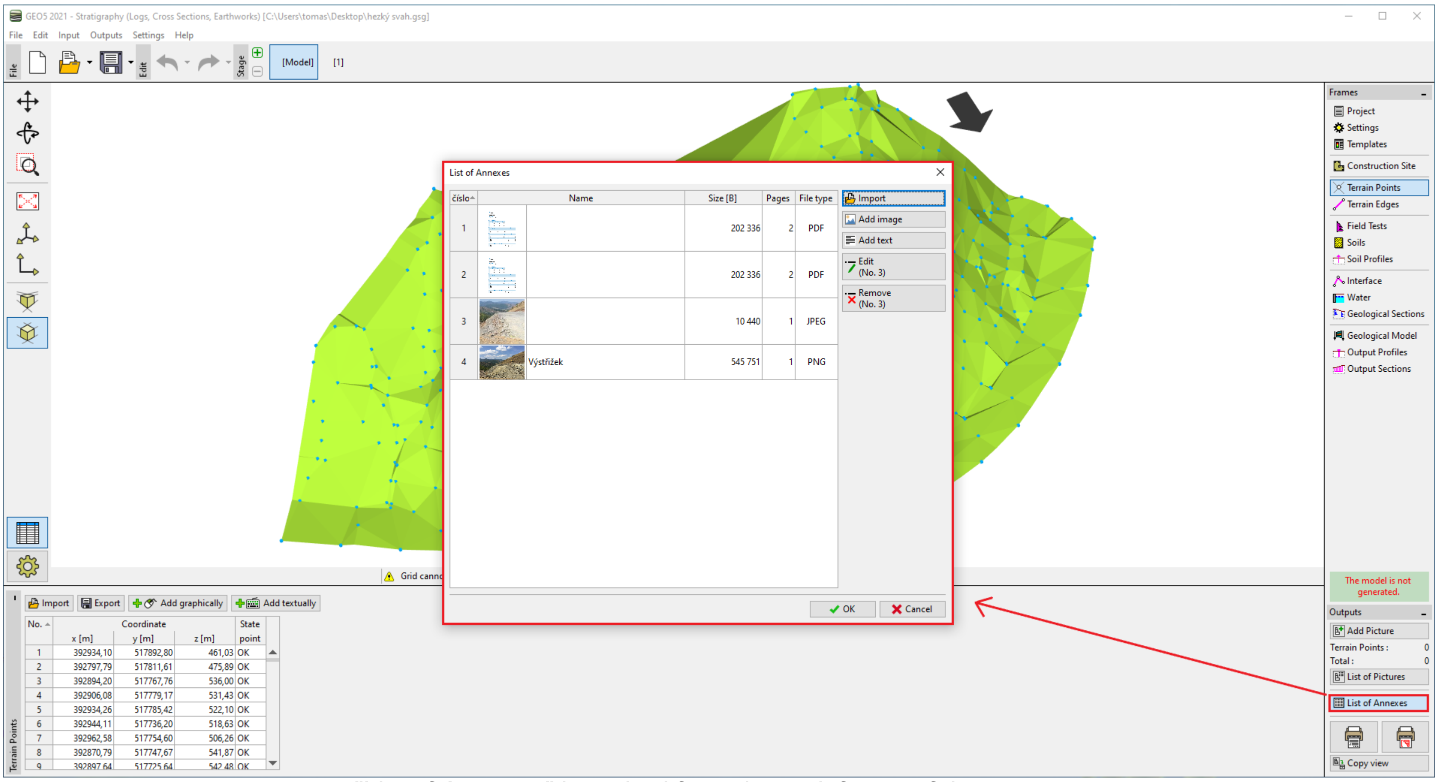
All annexes can be viewed, edited, and deleted at any time in the "**List of Annexes**".



"List of Annexes" launched from the Output Document

***Note**: An annex that was deleted from the output log remains in the data in the "**List of annexes**", therefore it is necessary to delete it from here as well.*

The "**List of Annexes**" can be used also for the **input of the external data**, that we do not want to print in the output document, but it creates a **logical attachment of the GEO5 task** - For example, in the subsoil model (Stratigraphy program), a logical annex would be a Geological survey report, borehole results and test data obtained from the geological database, laboratory results, etc. We can also input the data into the "**List of Annexes**" from the mainframe of the program.



"List of Annexes" launched from the mainframe of the program

Print and Export Document

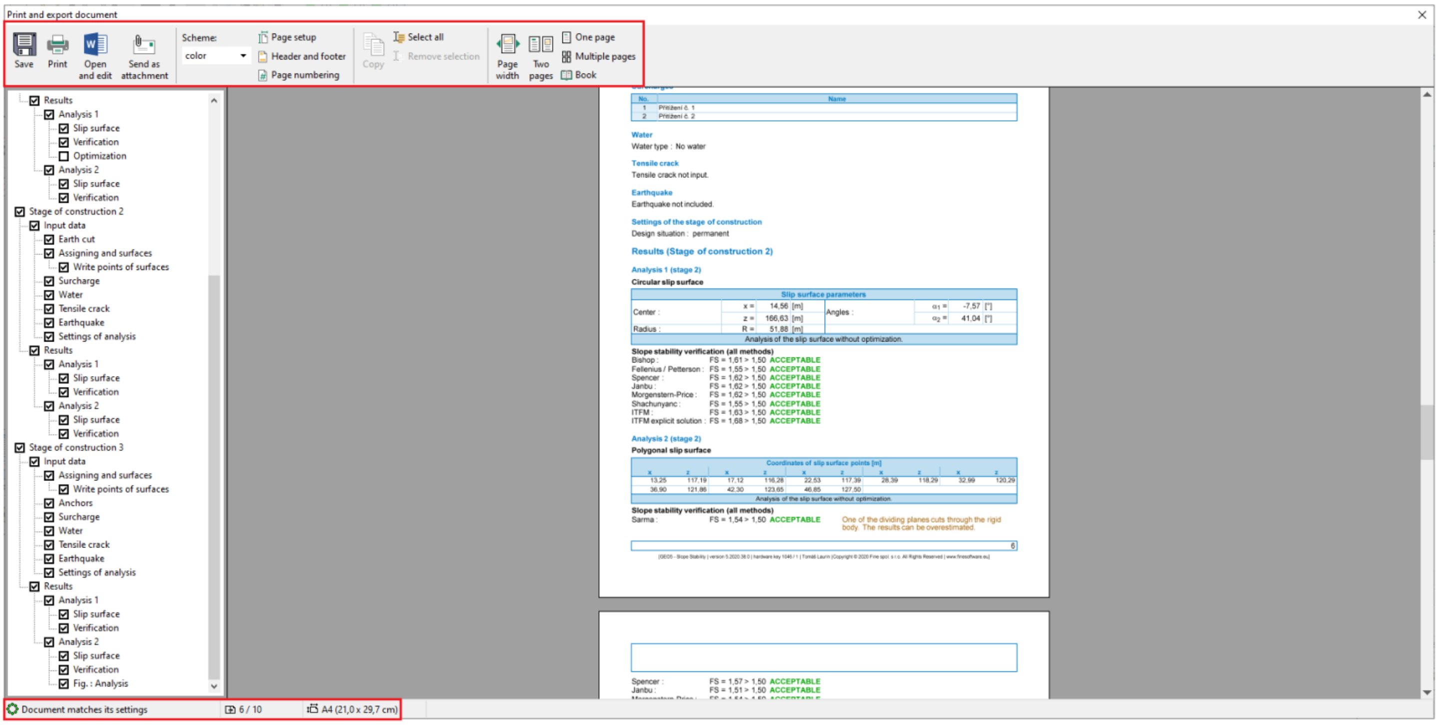
The "**Print and export document**" dialog window can be opened either from the control menu ("**Files**", "**Print document**" items), or using the "Outputs" button on the control bar. The page print preview with a generated text appears in the window.

This window generates the output document including pictures stored in the "Picture list". The **document is always up to date** - the program creates the document again based on input data (even with regenerated pictures) whenever opening this window.

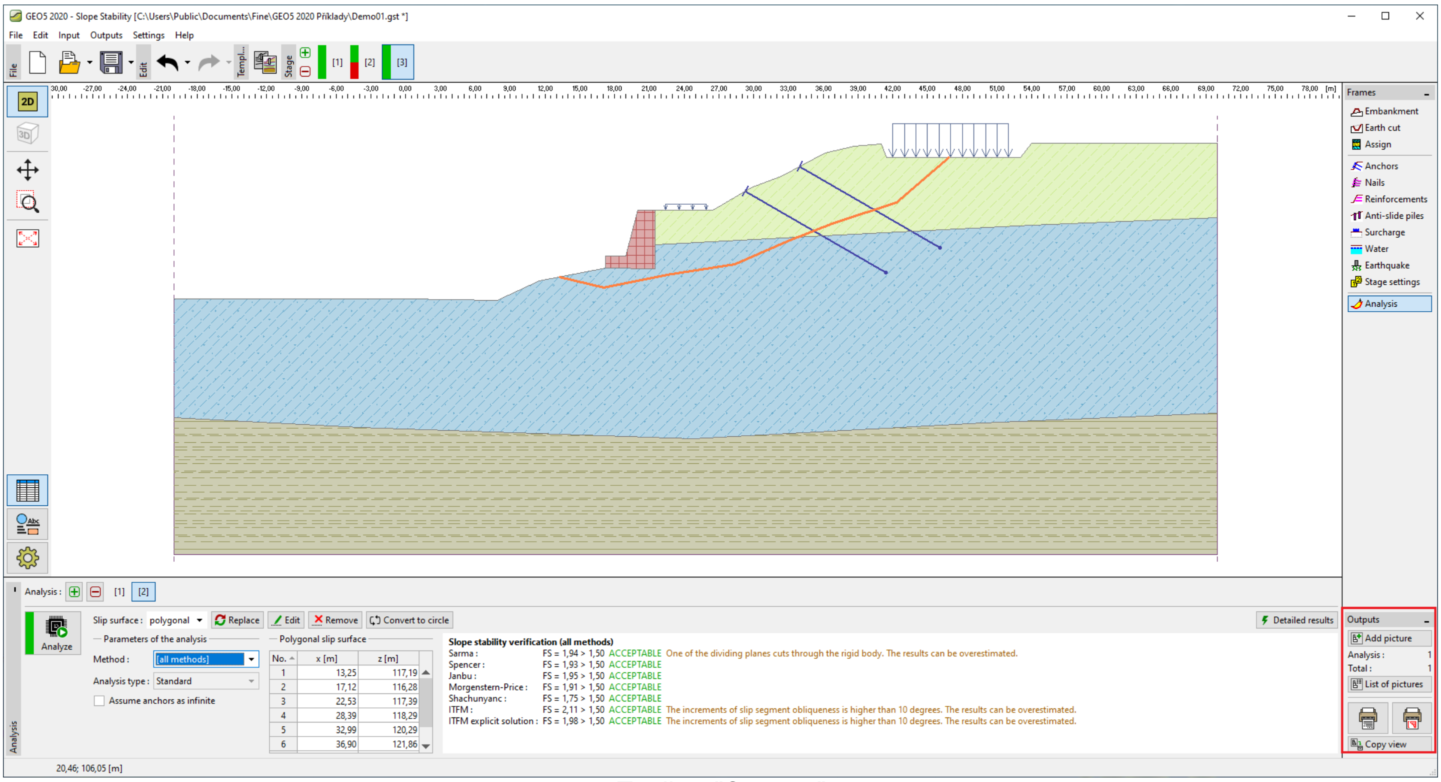
The dialog window contains its own Control bar - Print and Export with buttons used when customizing the appearance of the pages (header and footer definition, page properties, page numbering), **print,** and **export** of the document.

Only specific parts of the document including pictures can be generated by checking the corresponding "**tree**" item in the left part of the window. Selecting or deselecting an arbitrary item prompts the program to regenerate the document automatically. The mouse scroll wheel or scroll bar on the right can also be used to view the document.

The button part of the dialog window displays current information (defined page size, current document page, and the total number of pages).



Dialog window "Print and export document"



Toolbar "Outputs"

Print and Export Desktop View

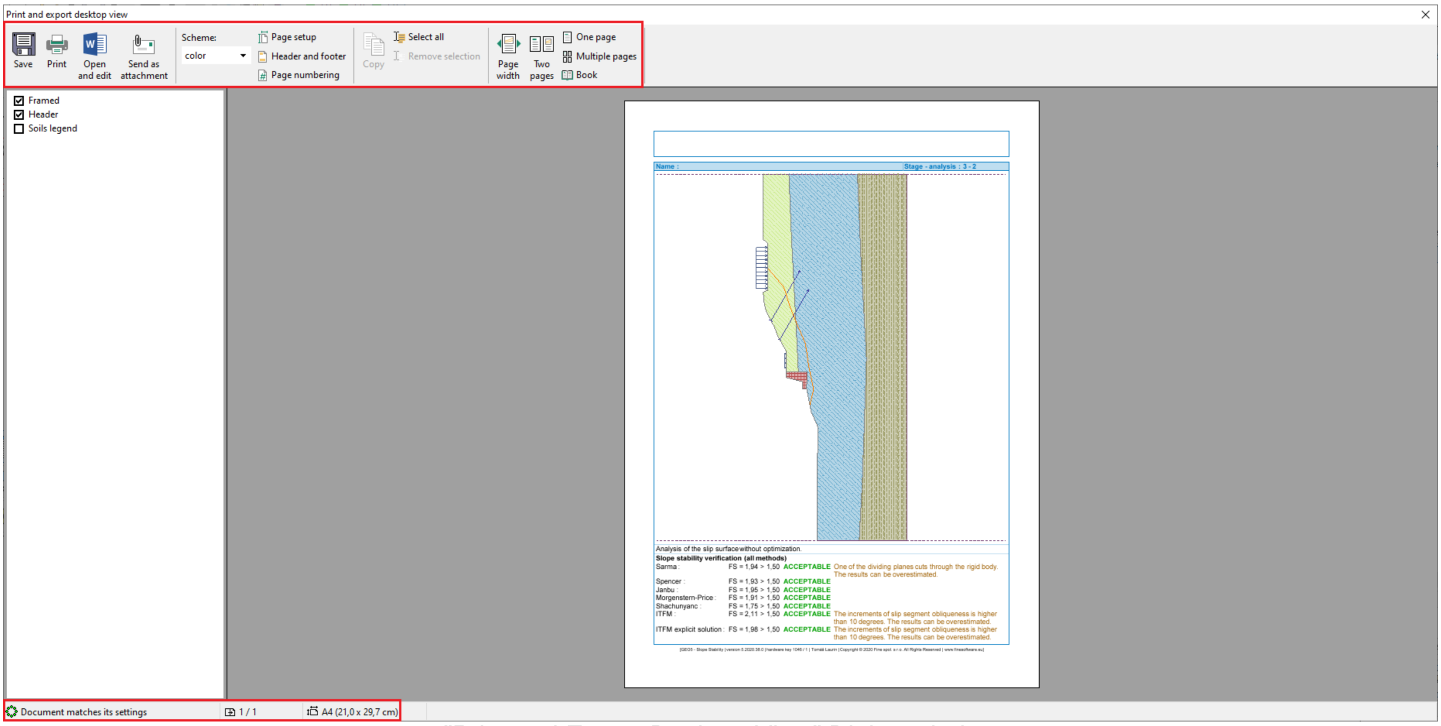
This dialog window allows us to print and export one or more pictures. Three options are available to open this window:

- Using the control menu ("**Files**", "**Print picture**" items), or the "Outputs" button on the vertical control bar.
- Using the "New picture" dialog window by pressing the "**Print**" button.
- Using the "List of pictures" dialog window by pressing the "**Print**" button.

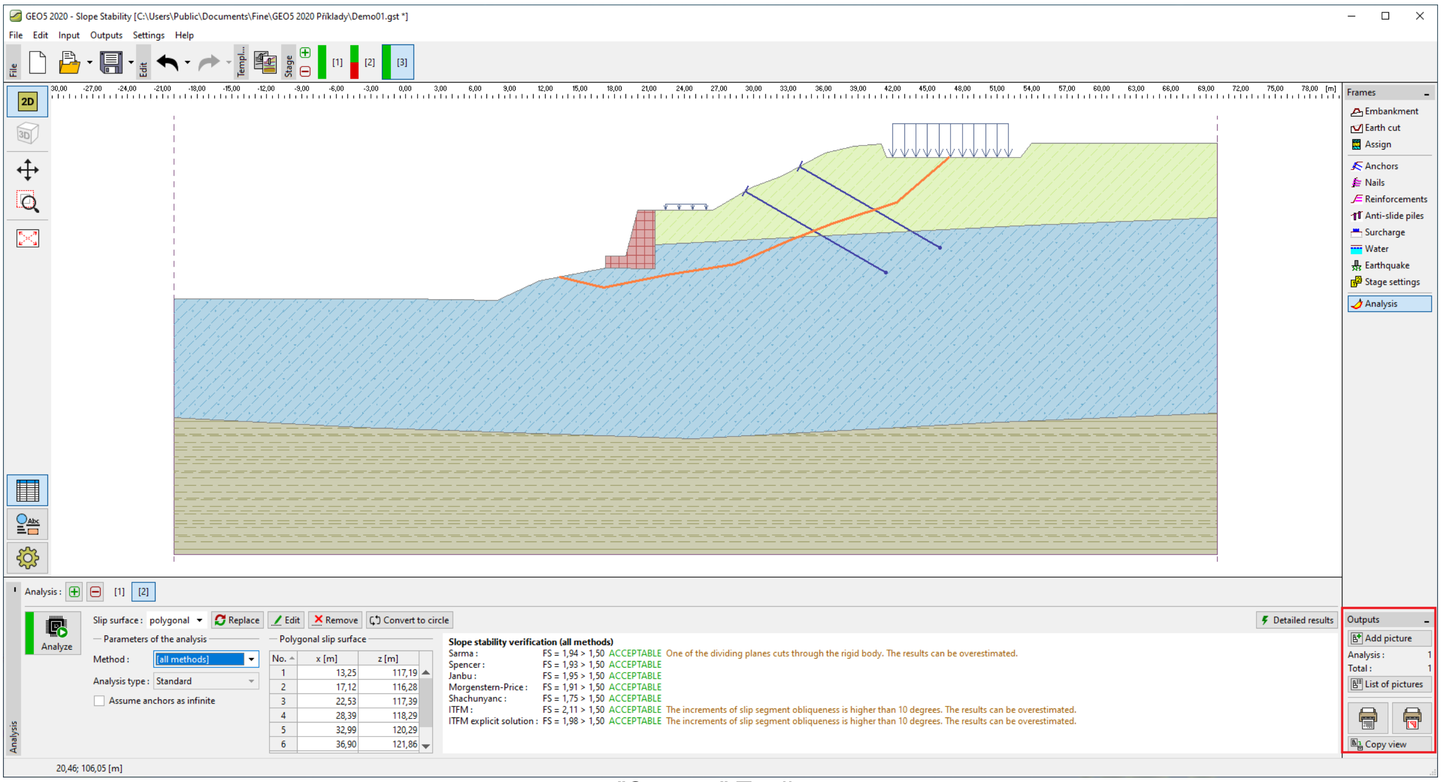
The window may contain more than one picture at the same time (when printing more construction stages or analyses) when printing more pictures from the list. Each picture is printed on a separate page. The picture preview can be adjusted by buttons on the toolbar or by the mouse scroll wheel.

The dialog window contains its own Control bar - Print and Export with buttons used when customizing the appearance of the pages (header and footer definition, page properties, page numbering), **print,** and **export** of the document.

The bottom part of the dialog window displays current information (the defined page size, a current document page, and the total number of pages).



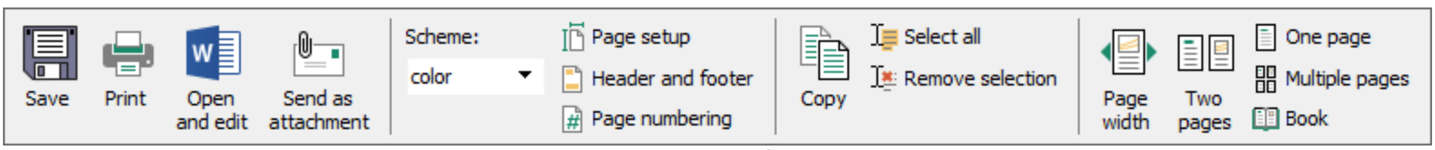
"Print and Export Desktop View" Dialog window



"Outputs" Toolbar

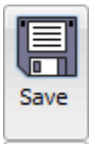
Control bar - Print and Export

The control bar of the "Print and export document" and "Print and export desktop view" dialog windows contains the following control elements:

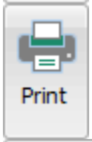


"Print and export" Control bar

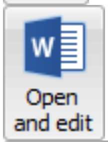
Individual control elements function as follows:



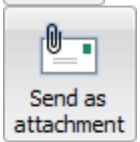
Save Opens the "**Save as**" dialog window allowing to save a file in *.PDF, *.RTF or *.TXT formats.



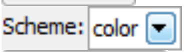
Print Opens the dialog window for "**Print**".



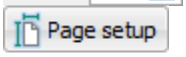
Open and Edit Opens a text editor (associated in the Windows system with *.RTF extension) that allows us to edit the page manually.



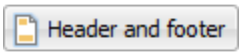
Send Opens a dialog window for a mail client and adds the picture as an attachment in *.PDF format.



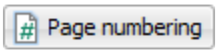
Color Style Determines the style of a picture view (color, gray scale, black & white).



Page Setup Opens the "Page setup" dialog window that allows specifying the page style (size, edges, orientation).

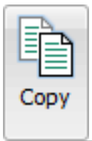


Header and Footer Opens the "Header and Footer" dialog window that allows us to input the document headers and footers.

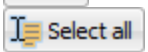


Page Numbering Opens the "Page numbering" dialog window that allows us to input the document page numbering.

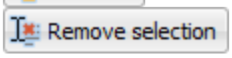
Note: The document can be scrolled using the mouse wheel, or the slider in the right part of the window.



Copy Copies the selected picture (text) to clipboard - parameters are set in the "Options" dialog window - tab "Copy to clipboard".

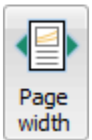


Select All Selects all elements of the document (text and pictures).

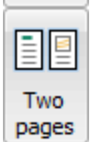


Cancel selection Cancels the entire selection (picture, text).

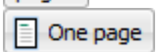
Note: It is possible to select any part of the document. This is done by moving the mouse over the desired text while keeping the left mouse button pressed.



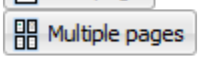
Page width Fits the page to the maximum width of the document dialog window.



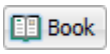
Two Pages Sets the size of the pages so that two pages are displayed in the dialog window.



One Page Modifies the page size so that the entire page in the dialog window is visible.



More Pages Sets the size of the pages so that more pages are displayed in the dialog window.



Book Displays the document in a format of a book.

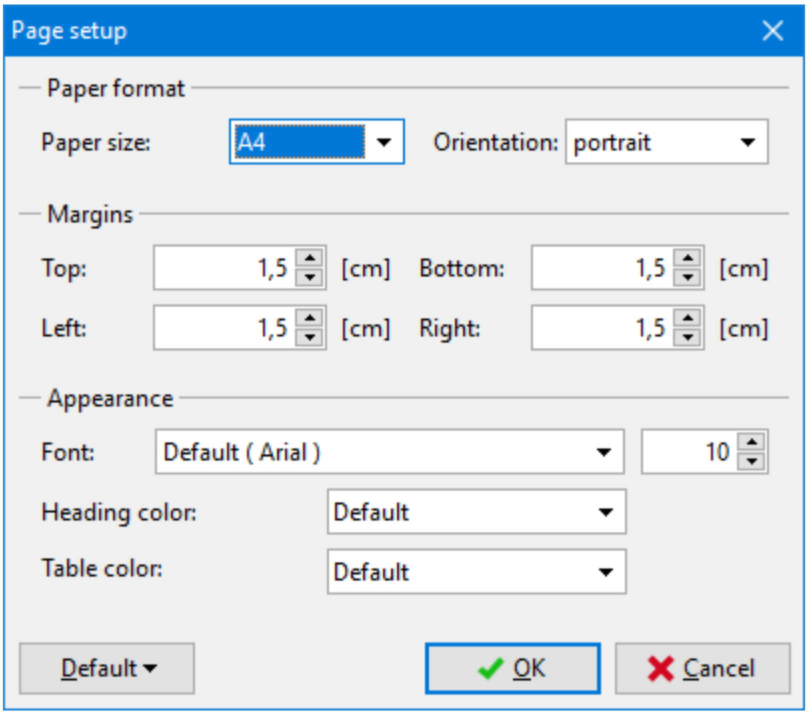
Note: **Zooming** any page of the document can be done by holding **CTRL** and rotating the mouse wheel.

Page Setup

The dialog window allows setting the page layout (paper format, print orientation and edges).

The "**Default**" button:

- Selection "**Save settings as default**" saves the content of the window as default for **any new tasks**. The default is valid for **all GEO5 programs**. Different computer users may have different defaults.
- - Selection "**Adopt default settings**" adopts the default settings to any opened task, which has the different settings.



"Page setup" Dialog window

Setting Header and Footer

The dialog window allows us to define properties of the document header and footer. The "**print header (footer)**" checkbox determines whether to print the document header (footer).

Header and footer lines may contain an arbitrary text and inserted objects implicitly defined by the program. These objects receive program information such as:

- From the "Company data" dialog window (company name, logo, address)
- From the "**Project**" frame (name and task description, author)
- From the document system data (date, time, page numbering)

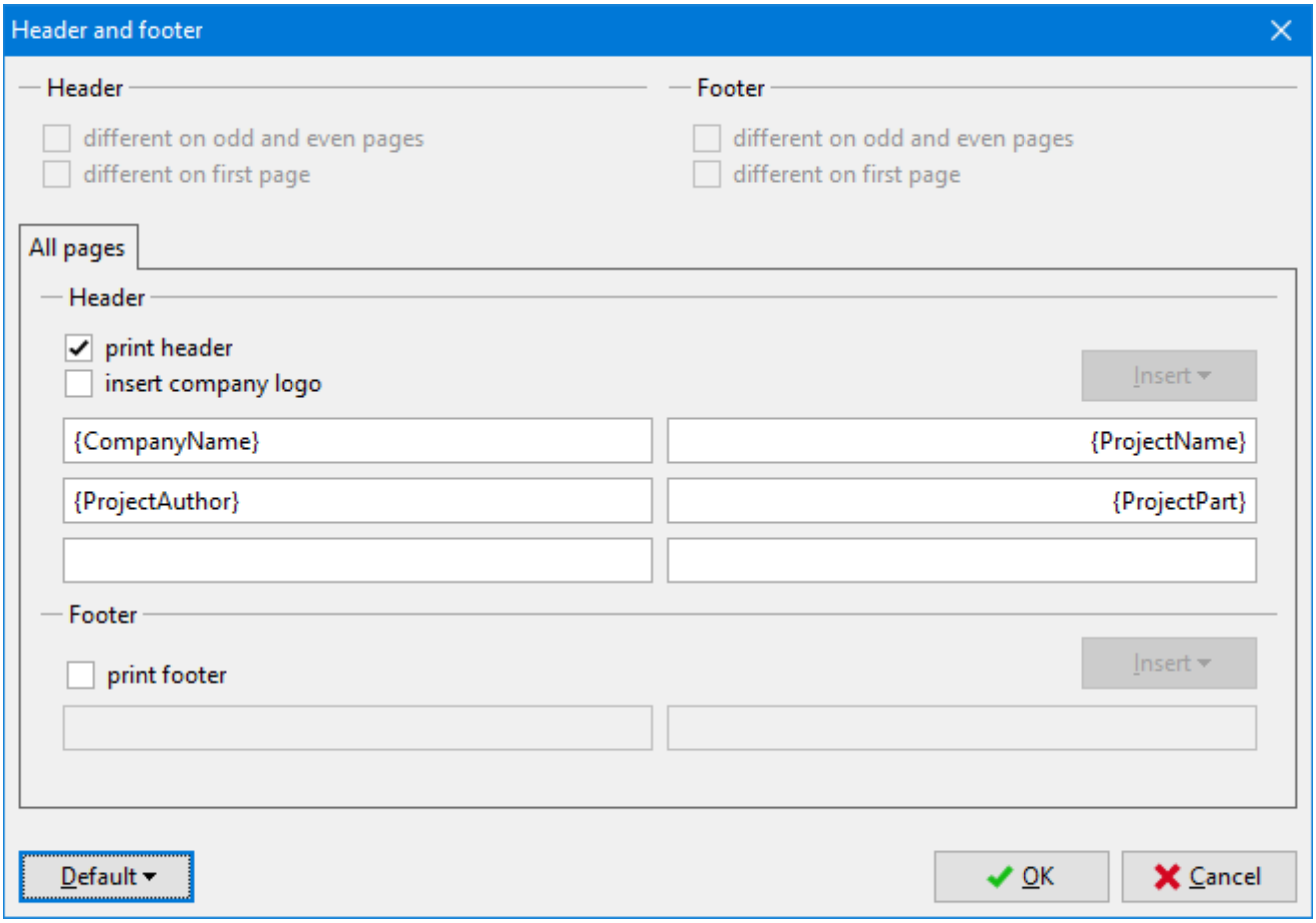
Objects can be input using the "**Insert**" button (the button opens a list of objects). The button is active only if the cursor is in one of the lines that allow us to insert a text (object). Inserted objects are written in an internal format different from the other text and placed in curly brackets.

The program allows us to define various headers for the first page or odd and even pages, respectively. Individual headers are in such a case defined in separate tabs.

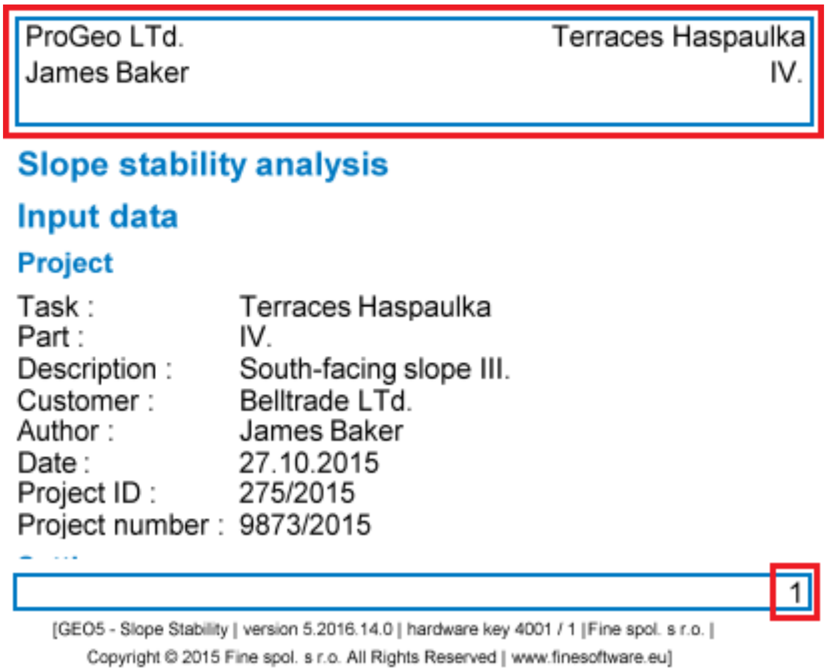
The "**Default**" button:

- Selection "**Save settings as default**" saves the content of the window as default for **any new tasks**. The default is valid for **all GEO5 programs**. Different computer users may have different defaults.
- - Selection "**Adopt default settings**" adopts the default settings to any opened task, which has the different settings.

Writing format and the resulting view are evident from the following pictures.



"Header and footer" Dialog window



View of the document header and footer

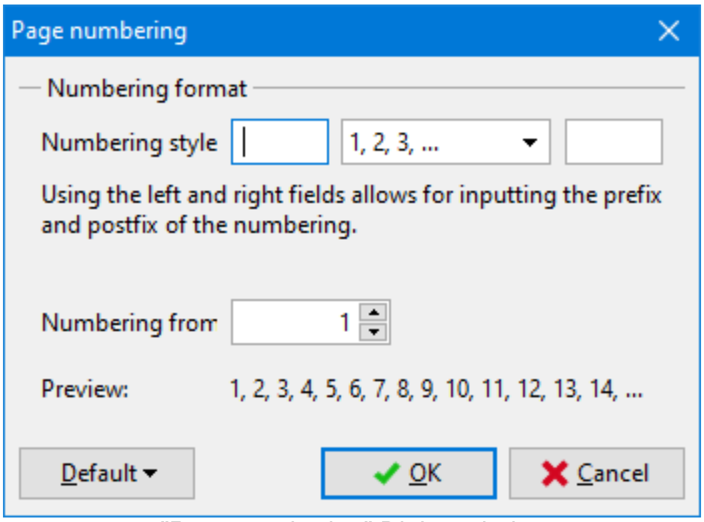
Page Numbering

This dialog window allows us to set page numbering. The combo list allows us to define the numbering style (Arabic digits, Roman digits, using symbols). A constant text can be placed both in front and behind the page number. The "**Numbering from**" option allows starting the page numbering from an arbitrary number.

The "**Default**" button:

- Selection "**Save settings as default**" saves the content of the window as default for **any new tasks**. The default is valid for **all GEO5 programs**. Different computer users may have different defaults.

- Selection "**Adopt default settings**" adopts the default settings to any opened task, which has the different settings.



"Page numbering" Dialog window

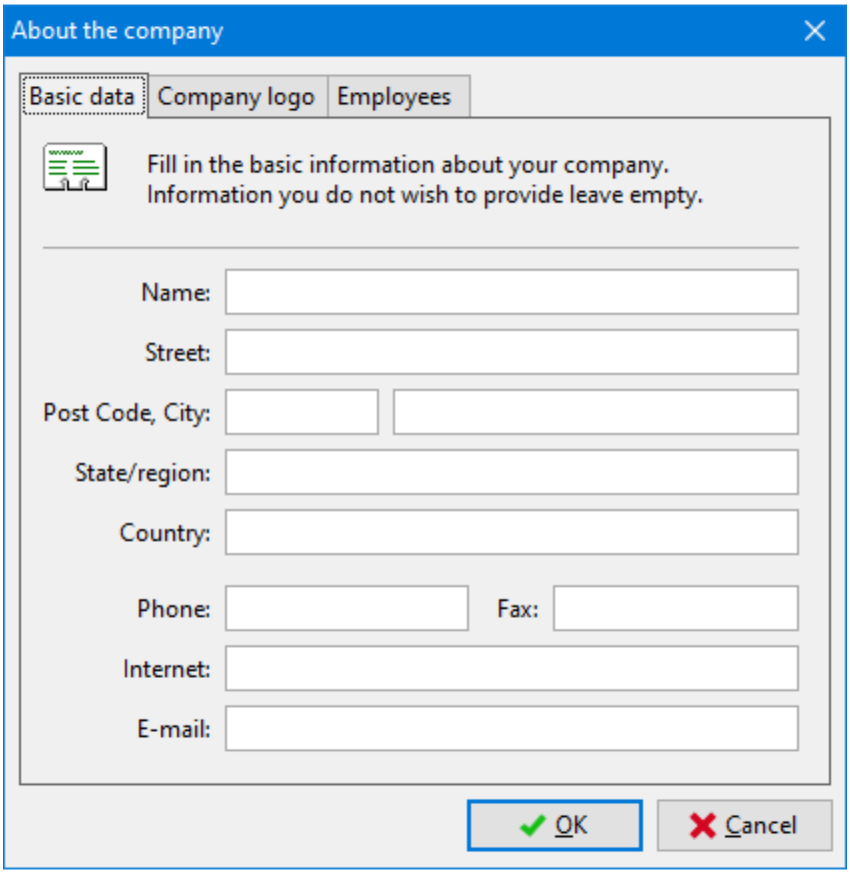
About the Company

The dialog window is launched from the managing menu ("**Settings**", "**Company**" items).

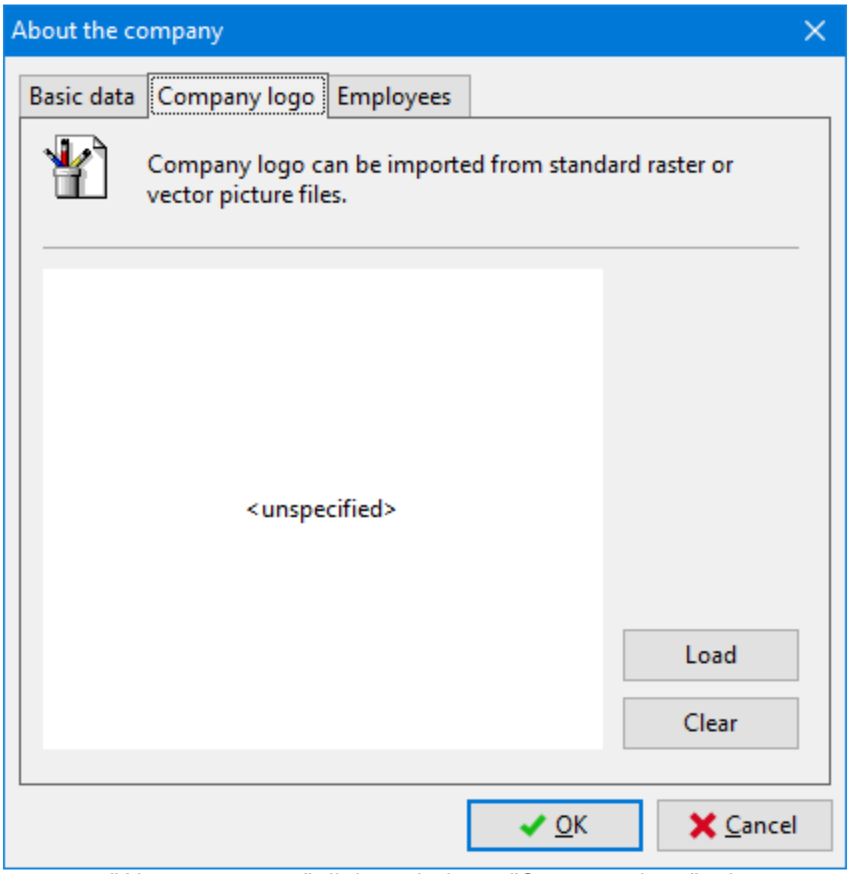
The "**Basic data**" tab allows us to specify the basic information about the company. The input data is used by the program when printing and exporting documents (pictures), in the document header and footer.

The "**Company logo**" tab allows us to load the company logo. The "**Load**" button opens a dialog window which allows us to open the picture in various formats (*.JPG, *.JPEG, *.JPE, *.BMP, *.ICO, *.EMF, *.WMF).

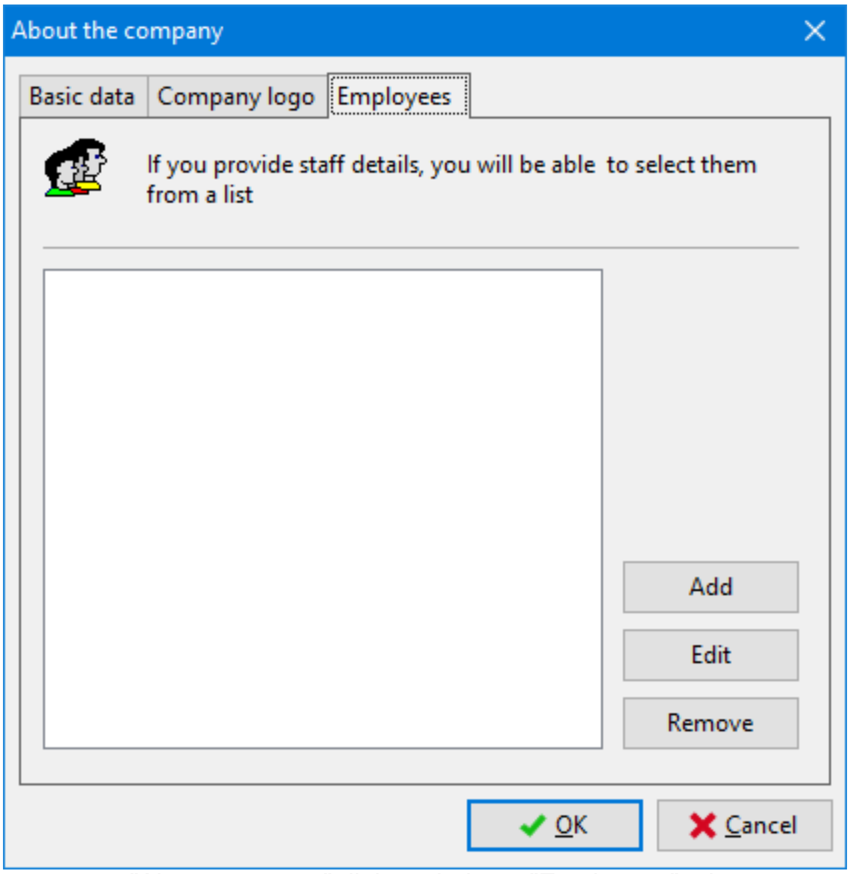
The "**Employees**" tab allows us to input a list of program users (employees). When filling the name list it is no longer necessary to fill the author's name in the "Project" frame.



"About company" dialog window - "Basic data" tab



"About company" dialog window - "Company logo" tab



"About company" dialog window - "Employees" tab

Theory

The theoretical part of the help contains all theoretical basis employed in computations within GEO5 programs.

Stress in a Soil Body

Calculation of the stress in the soil in our software is described in the following chapters:

- Geostatic stress in soil body, computation of uplift pressure
- Effective / total stress
- Stress increment due to surcharge
- Stress increment under footing

Geostatic Stress, Uplift Pressure

Stress analysis is based on the existence of soil layers specified by the user during input. The program further inserts fictitious layers at the locations where the stress and lateral pressure (GWT, points of construction, etc.) change. The normal stress in the i^{th} layer is computed according to:

$$\sigma_i = \sum h_i . \gamma_i$$

where: h_i - thickness of the i^{th} layer

γ_i - unit weight of soil

If the layer is found below the **groundwater table**, the unit weight of soil below the water table is specified with the help of input parameters of the soil as follows:

- for option "**Standard**" from the expression:

$$\gamma_{su} = \gamma_{sat} - \gamma_w$$

where: γ_{sat} - saturated unit weight of soil

γ_w - unit weight of water

- for the option "**Compute from porosity**" from the expression:

$$\gamma_{su} = (1-n).(\gamma_s - \gamma_w)$$

where: n - porosity

γ_s - specific weight of soil

γ_w - unit weight of water

$$\gamma_s = \frac{G_d}{V - V_p}$$

where: V - volume of soil

V_p - volume of voids

G_d - weight of dry soil

Unit weight of water is assumed in the program equal to 10 kN/m^3 or 0.00625 ksi .

Assuming inclined ground behind the structure ($\beta \neq 0$) and layered subsoil the angle β , when computing the coefficient of earth pressure K , is reduced in the i^{th} layer using the following expression:

$$tg\beta_i = \frac{\gamma}{\gamma_i} tg\beta$$

where: γ - unit weight of the soil in the first layer underground

γ_i - unit weight of the soil in the i^{th} layer underground

β - slope inclination behind the structure

Effective/Total Stress in a Soil

Vertical normal stress σ_z is defined as:

$$\sigma_z = \gamma_{ef}.z + \gamma_w.z$$

where: σ_z - vertical normal total stress

γ_{ef} - submerged unit weight of soil

z - depth below the ground surface

γ_w - unit weight of water

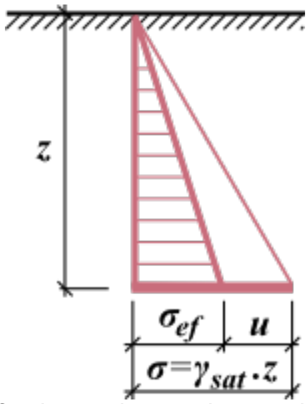
This expression in its generalized form describes so-called concept of effective stress:

$$\sigma_z = \sigma_{ef} + u$$

where: σ - total stress (overall)

σ_{ef} - effective stress (active)

u - neutral stress (pore water pressure)



Total, effective and neutral stress in the soil

Effective stress concept is valid only for normal stress σ, since the shear stress τ is not transferred by the water so that it is effective. The total stress is determined using the basic tools of theoretical mechanics, the effective stress is then determined as a difference between the total stress and neutral (pore) pressure (i.e. always by calculation, it can never be measured). Pore pressures are determined using laboratory or in-situ testing or by calculation. To decide whether to use the total or effective stresses is no simple. The following table may provide some general recommendations valid for majority of cases. We should realize that the total stress depends on the way the soil is loaded by its self-weight and external effects. As for the pore pressure we assume that for flowing pore water the pore equals to hydrodynamic pressure and to hydrostatic pressure otherwise. For partial saturated soils with higher degree of it is necessary to account for the fact that the pore pressure evolves both in water and air bubbles.

Assume conditions	**Drained layer**	**Undrained layer**
short - term	effective stress	total stress
long - term	effective stress	effective stress

In layered subsoil with different unit weight of soils in individual horizontal layers the vertical total stress is determined as a sum of weight of all layers above the investigated point and the pore pressure:

$$\sigma_z = \int_0^z \gamma\, dz + \gamma_w (z - d)$$

where: σ_z - vertical normal total stress

γ - unit weight of soil
- unit weight of soil in natural state for soils above the GWT and dry layers
- unit weight of soil below water in other cases

d - depth of the groundwater table below the ground surface

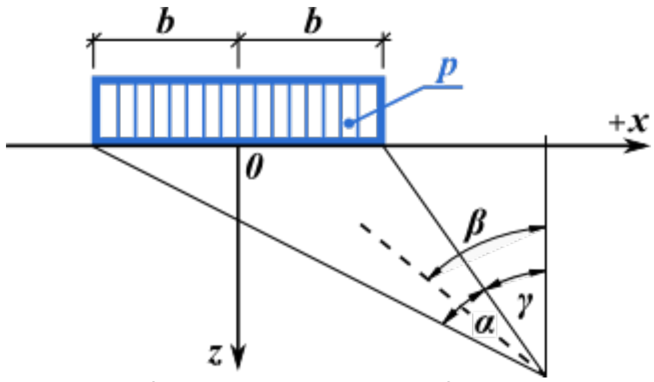
z - depth below the ground surface

γ_w - unit weight of water

Increment of Earth Pressure due to Surcharge

Earth pressure increment in a soil or rock body due to surcharge is computed using the theory of elastic subspace (Boussinesq).

Earth pressure increment in the point inside the soil or rock body due to an **infinite strip surcharge** is obtained from the following scheme:

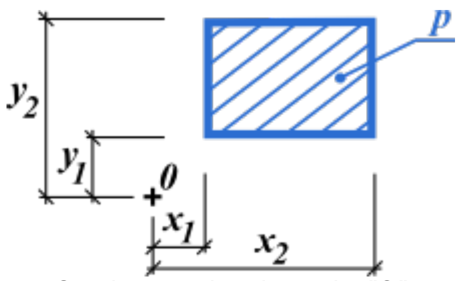


Computation of earth pressure due to infinite strip surcharge

$$\sigma_z = \frac{p}{\pi}(\alpha + \sin\alpha.\cos 2\beta)$$
$$\beta = \gamma + \frac{\alpha}{2}$$

A **trapezoidal surcharge** is automatically subdivided in the program into ten segments. Individual segments are treated as strip surcharges. The resulting earth pressure is a sum of partial surcharges from individual segments.

Stress increment due to **concentrated surcharge** is computed as follows:



Surcharge related to point "O"

$$\Delta\sigma_z = \frac{p}{2\pi}\left(\frac{x_2.z.S_2}{y_2.S_{2x}^2} + \frac{x_2.z^3}{y_2.S_{2y}^2.S_2} - \frac{x_2.z.S_3}{y_1.S_{2x}^2} + \frac{x_2.z^3}{y_1.S_{1y}^2.S_3} - \frac{x_1.z.S_4}{y_2.S_{1x}^2} + \frac{x_2.z^3}{y_2.S_{2y}^2.S_4} + \frac{x_1.z.S_2}{y_1.S_{1x}^2} - \right.$$
$$\left.\frac{x_1.z^3}{y_1.S_{2y}^2.S_1} + arctg\frac{x_2.y_2}{z.S_2} - arctg\frac{x_2.y_1}{z.S_3} - arctg\frac{x_1.y_2}{z.S_4} + arctg\frac{x_1.y_1}{z.S_1}\right)$$

where:

$$S_{2x} = \sqrt{x_2^2 + z^2} \qquad S_1 = \sqrt{x_1^2 + y_1^2 + z^2} \qquad S_3 = \sqrt{x_2^2 + y_1^2 + z^2}$$
$$S_{2y} = \sqrt{y_2^2 + z^2} \qquad S_2 = \sqrt{x_2^2 + y_2^2 + z^2} \qquad S_4 = \sqrt{x_1^2 + y_2^2 + z^2}$$

Increment of Earth Pressure under Footing

In the program "**Spread footing**", the stress distribution below foundation is determined by combining the basic load diagrams:

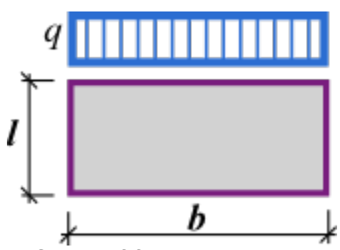


Chart of foundation load

$$\sigma_z = \frac{f}{2\pi}\left[arctg\frac{lb}{z\sqrt{l^2+b^2+z^2}} + \frac{lbz}{\sqrt{l^2+b^2+z^2}}\left(\frac{1}{l^2+z^2} + \frac{1}{b^2+z^2}\right)\right]$$

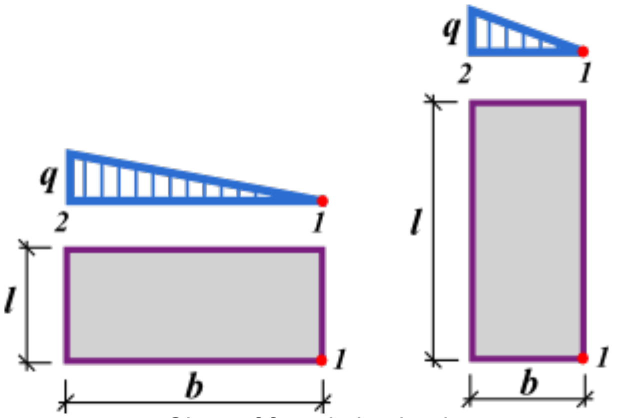


Chart of foundation load

$$\sigma_{z,1} = \frac{q}{2\pi}\left(\frac{lbz}{R\left(z^2+b^2\right)} + \frac{lz}{bR}\,\frac{R-\sqrt{l^2+z^2}}{\sqrt{l^2+z^2}}\right)$$

$$R = \sqrt{l^2+b^2+z^2}$$

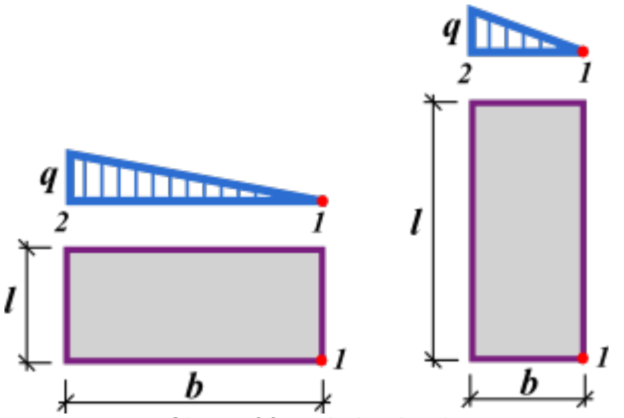


Chart of foundation load

$$\sigma_{z,2} = \frac{q}{2\pi}\left(arctg\frac{lb}{zR} + \frac{lz}{l^2+z^2}\,\frac{R-\sqrt{l^2+z^2}}{b}\right)$$

Earth Pressures

GEO5 software considers following earth pressure categories:

- active earth pressure
- passive earth pressure
- earth pressure at rest

When computing earth pressures, the program allows us to distinguish between effective and total stress state and to establish several ways of calculation of uplift pressure. In addition, it is possible to account for the following effects having on the earth pressure magnitude:

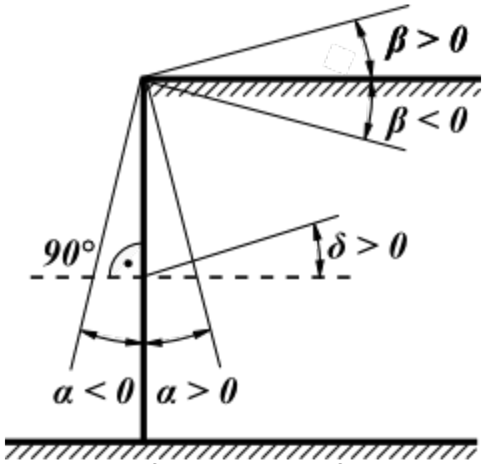
- influence of surcharge
- influence of water pressure
- influence of broken terrain
- friction between soil and back of structure
- adhesion of soil
- influence of earth wedge at cantilever jumps
- influence of earthquake

The following sign convention is used in the program, text and presented expressions.

When specifying rocks, it is also necessary to input both cohesion of rock c and the angle of internal friction of rock φ . These values can be obtained either from a geological survey, or from the table of recommended values.

Sign Convention

The following sign convention is used in the program, text and presented expressions.



Sign convention for calculation of earth pressures

- inclination of the ground surface β is positive, when the ground rises upwards from the wall
- inclination of the back of structure α is positive when the toe of the wall (at the back face) is placed in the direction of the soil body when measured from the vertical line constructed from the upper point of the structure
- friction between the soil and back of structure δ is positive, if the resultant of earth pressure (thus also earth pressure) and normal to the back of structure form an angle measured in the clockwise direction

Active Earth Pressure

Active earth pressure is the smallest limiting lateral pressure developed at the onset of shear failure by wall moving away from the soil in the direction of the acting earth pressure (minimal wall rotation necessary for the evolution of active earth pressure is about 2 $mrad$, i.e. 2 mm/m of the wall height).

The following theories and approaches are implemented for the computation of active earth pressure **assuming effective stress state**:

- The Mazindrani theory (Rankine)
- The Coulomb theory
- The Müller-Breslau theory
- The Caquot theory
- The Absi theory
- SP 22.13330.2016

For cohesive soils the tension cutoff condition is accepted, i.e. if due to cohesion the negative value of active earth pressure is developed or, according to more strict requirements, the value of "Minimum dimensioning pressure" is exceeded, the value of active earth pressure drops down to zero or set equal to the "Minimum dimensioning pressure".

The program also allows for running the analysis in total stresses.

Active Earth Pressure - The Mazindrani Theory (Rankine)

Active earth pressure is given by the following formula:

$$\sigma_a = \sigma_z . K_a = \gamma . z . K'_a . \cos\beta$$

where:

	σ_z	-	vertical geostatic stress
	K_a	-	coefficient of active earth pressure due to Rankine
	β	-	slope inclination
	γ	-	weight of soil
	z	-	assumed depth
	K'_a	-	coefficient of active earth pressure due to Mazindrani

$$K'_a = \frac{1}{\cos^2\varphi}\left[\begin{array}{l} 2.\cos^2\beta + 2.\left(\frac{c}{\gamma.z}\right).\cos\varphi.\sin\varphi - \\ \sqrt{4.\cos^2\beta.\left(\cos^2\beta - \cos^2\varphi\right) + 4.\left(\frac{c}{\gamma.z}\right)^2.\cos^2\varphi + 8.\left(\frac{c}{\gamma.z}\right).\cos^2\beta.\sin\varphi.\cos\varphi} \end{array}\right] - 1$$

where: β - slope inclination

φ - angle of internal friction of soil

c - cohesion of soil

Assuming cohesionless soils ($c = 0$) and horizontal ground surface ($\beta = 0$) yields the Rankine solution, for which the active earth pressure is provided by:

$$\sigma_a = \sigma_z . K_a$$

and the coefficient of active earth pressure becomes:

$$K_a = tg^2\left(45^\circ - \frac{\varphi}{2}\right)$$

where: φ - angle of internal friction of soil

Horizontal and vertical components of the active earth pressure become:

$$\sigma_{ax} = \sigma_a . cos(\alpha + \delta)$$
$$\sigma_{az} = \sigma_a . sin(\alpha + \delta)$$

where: σ_a - active earth pressure

δ - angle of friction between structure and soil

α - back face inclination of the structure

Literature:

Mazindrani, Z.H., and Ganjali, M.H. 1997. Lateral earth pressure problem of cohesive backfill with inclined surface. Journal of Geotechnical and Geoenvironmental Engineering, ASCE, ***123****(2): 110-112.*

Active Earth Pressure - The Coulomb Theory

Active earth pressure is given by the following formula:

$$\sigma_a = \sigma_z . K_a - 2 c_{ef} . K_{ac}$$

where: σ_z - vertical geostatic stress

c_{ef} - effective cohesion of the soil

K_a - coefficient of active earth pressure

K_{ac} - coefficient of active earth pressure due to cohesion

The coefficient of active earth pressure K_a is given by:

$$K_a = \frac{cos^2(\varphi - \alpha)}{cos^2 \alpha . cos(\alpha + \delta)\left(1 + \sqrt{\frac{sin(\varphi + \delta) sin(\varphi - \beta)}{cos(\alpha + \delta) cos(\alpha - \beta)}}\right)^2}$$

The coefficient of active earth pressure K_{ac} is given by:

for: $\alpha < \pi/4$

$$K_{ac} = \frac{K_{ahc}}{cos(\delta + \alpha)}$$

$$K_{ahc} = \frac{cos\,\varphi . cos\,\beta . cos(\delta - \alpha).(1 + tg(-\alpha) tg\beta)}{1 + sin(\varphi + \delta - \alpha - \beta)}$$

for: $\alpha \geq \pi/4$

$$K_{ac} = \sqrt{K_a}$$

where: φ - angle of internal friction of soil

δ - angle of friction between structure and soil

β - slope inclination

α - back face inclination of the structure

Horizontal and vertical components of the active earth pressure become:

$$\sigma_{ax} = \sigma_a . cos(\alpha + \delta)$$
$$\sigma_{az} = \sigma_a . sin(\alpha + \delta)$$

where:
- σ_a - active earth pressure
- δ - angle of friction between structure and soil
- α - back face inclination of the structure

Active Earth Pressure - The Müller-Breslau Theory

Active earth pressure is given by the following formula:

$$\sigma_a = \sigma_z . K_a - 2 . c_{ef} . K_{ac}$$

where:
- σ_z - vertical geostatic stress
- c_{ef} - effective cohesion of the soil
- K_a - coefficient of active earth pressure
- K_{ac} - coefficient of active earth pressure due to cohesion

The coefficient of active earth pressure K_a is given by:

$$K_a = \left(\frac{\dfrac{sin(\alpha + \pi/4 - \varphi)}{sin(\alpha + \pi/4)}}{\sqrt{sin(\alpha + \pi/4 + \delta)} + \sqrt{\dfrac{sin(\varphi + \delta) . sin(\varphi - \beta)}{sin(\alpha + \pi/4 - \beta)}}} \right)^2$$

where:
- φ - angle of internal friction of soil
- δ - angle of friction between the structure and the soil
- β - slope inclination
- α - back face inclination of the structure

The coefficient of active earth pressure K_{ac} is given by:

for: $\alpha < \pi/4$

$$K_{ac} = \frac{K_{ahc}}{cos(\delta + \alpha)}$$

$$K_{ahc} = \frac{cos\,\varphi . cos\,\beta . cos(\delta - \alpha) . (1 + tg(-\alpha) . tg\beta)}{1 + sin(\varphi + \delta - \alpha - \beta)}$$

for: $\alpha \geq \pi/4$

$$K_{ac} = \sqrt{K_a}$$

where:
- φ - angle of internal friction of soil
- δ - angle of friction between structure and soil
- β - slope inclination
- α - back face inclination of the structure

Horizontal and vertical components of the active earth pressure become:

$$\sigma_{ax} = \sigma_a . cos(\alpha + \delta)$$
$$\sigma_{az} = \sigma_a . sin(\alpha + \delta)$$

where:
- σ_a - active earth pressure
- δ - angle of friction between structure and soil
- α - back face inclination of the structure

Literature:

Müller-Breslau's Erddruck auf Stutzmauern,Stuttgart: Alfred Kroner-Verlag, 1906 (German).

Active Earth Pressure - The Caquot Theory

Active earth pressure is given by the following formula:

$$\sigma_a = \sigma_z.K_a - 2.c_{ef}.K_{ac}$$

where:
- σ_z - vertical geostatic stress
- c_{ef} - effective cohesion of the soil
- K_a - coefficient of active earth pressure
- K_{ac} - coefficient of active earth pressure due to cohesion

The following analytical solution (Boussinesque, Caquot) is implemented to compute the coefficient of active earth pressure K_a:

$$K_a = \rho.K_a^{Coulomb}$$

where:
- K_a - coefficient of active earth pressure due to Caquot
- $K_a^{Coulomb}$ - coefficient of active earth pressure due to Coulomb
- ρ - conversion coefficient - see further

$$\rho = \left[\left(1 - 0{,}9.\lambda^2 - 0{,}1.\lambda^4\right)\left(1 - 0{,}3.\lambda^3\right)\right]^{-n}$$

$$\lambda = \frac{\Delta + \beta - \Gamma}{4.\varphi - 2.\pi(\Delta + \beta - \Gamma)}$$

$$\Delta = 2.tan^{-1}\left(\frac{|cot\delta| - \sqrt{cot^2\delta - cot^2\varphi}}{1 + cosec\varphi}\right)$$

$$\Gamma = sin^{-1}\left(\frac{sin\beta}{sin\varphi}\right)$$

where:
- β - slope inclination behind the structure
- φ - angle of internal friction of soil
- δ - angle of friction between structure and soil

The coefficient of active earth pressure K_{ac} is given by:

for: $\alpha < \pi/4$

$$K_{ac} = \frac{K_{ahc}}{cos(\delta + \alpha)}$$

$$K_{ahc} = \frac{cos\varphi.cos\beta.cos(\delta - \alpha).(1 + tg(-\alpha).tg\beta)}{1 + sin(\varphi + \delta - \alpha - \beta)}$$

for: $\alpha \geq \pi/4$

$$K_{ac} = \sqrt{K_a}$$

where:
- φ - angle of internal friction of soil
- δ - angle of friction between structure and soil
- β - slope inclination behind the structure
- α - back face inclination of the structure

Horizontal and vertical components of the active earth pressure become:

$$\sigma_{ax} = \sigma_a.cos(\alpha + \delta)$$

$$\sigma_{az} = \sigma_a.sin(\alpha + \delta)$$

where:
- σ_a - active earth pressure
- δ - angle of friction between structure and soil
- α - back face inclination of the structure

Active Earth Pressure - The Absi Theory

Active earth pressure is given by the following formula:

$$\sigma_a = \sigma_z . K_a - 2.c_{ef} . K_{ac}$$

where:
- σ_z - vertical geostatic stress
- c_{ef} - effective cohesion of the soil
- K_a - coefficient of active earth pressure
- K_{ac} - coefficient of active earth pressure due to cohesion

The program takes values of the coefficient of active earth pressure K_a from a database, built upon the values published in the book: Kérisel, Absi: Active and passive earth Pressure Tables, 3rd Ed. A.A. Balkema, 1990 ISBN 90 6191886 3.

The coefficient of active earth pressure K_{ac} is given by:

for: $\alpha < \pi/4$

$$K_{ac} = \frac{K_{ahc}}{cos(\delta + \alpha)}$$

$$K_{ahc} = \frac{cos\varphi . cos\beta . cos(\delta - \alpha)(1 + tg(-\alpha).tg\beta)}{1 + sin(\varphi + \delta - \alpha - \beta)}$$

for: $\alpha \geq \pi/4$

$$K_{ac} = \sqrt{K_a}$$

where:
- φ - angle of internal friction of soil
- δ - angle of friction between structure and soil
- β - slope inclination
- α - back face inclination of the structure

Horizontal and vertical components of the active earth pressure become:

$$\sigma_{ax} = \sigma_a . cos(\alpha + \delta)$$

$$\sigma_{az} = \sigma_a . sin(\alpha + \delta)$$

where:
- σ_a - active earth pressure
- δ - angle of friction between structure and soil
- α - back face inclination of the structure

Literature:

Kérisel, Absi: Active and Passive Earth Pressure Tables, 3rd ed., Balkema, 1990 ISBN 90 6191886 3.

Active Earth Pressure - SP 22.13330.2016

Active earth pressure is given by the following formula:

$$\sigma_a = \sigma_z K_a - 2c\sqrt{K_a}$$

where:
- σ_z - vertical geostatic stress
- c - cohesion of soil
- K_a - coefficient of active earth pressure

The coefficient of active earth pressure K_a is given by:

$$K_a = \frac{cos^2(\varphi - \alpha)}{cos^2\alpha \left(1 + \sqrt{\frac{sin(\delta+\varphi)sin(\varphi-\beta)}{cos(\delta+\alpha)cos(\alpha-\beta)}}\right)^2}$$

where:
- φ - angle of internal friction of soil
- δ - angle of friction between structure and soil
- β - slope inclination
- α - back face inclination of the structure

Horizontal and vertical components of the active earth pressure become:

$$\sigma_{ax} = \sigma_a cos(\alpha + \delta)$$
$$\sigma_{az} = \sigma_a sin(\alpha + \delta)$$

where: σ_a - active earth pressure
δ - angle of friction between structure and soil
α - back face inclination of the structure

Active Earth Pressure - Total Stress

When determining the active earth pressure in cohesive fully saturated soils, in which case the consolidation is usually prevented (undrained conditions), the horizontal normal total stress σ_x receives the form:

$$\sigma_x = \sigma_z - K_{uc}.c_u$$

where: σ_x - horizontal total stress (normal)
σ_z - vertical normal total stress
K_{uc} - coefficient of earth pressure
c_u - total cohesion of soil

The coefficient of earth pressure K_{uc} is given by:

$$K_{uc} = 2.\sqrt{1 + \frac{a_u}{c_u}}$$

where: K_{uc} - coefficient of earth pressure
c_u - total cohesion of soil
a_u - total adhesion of soil to the structure

Passive Earth Pressure

Passive earth pressure is the highest limiting lateral pressure developed at the onset of shear failure by wall moving (penetrating) in the direction opposite to the direction of acting earth pressure (minimal wall rotation necessary for the evolution of passive earth pressure is about 10 $mrad$, i.e. 10 mm/m of the wall height). In most expressions used to compute the passive earth pressure the sign convention is assumed such that the usual values of δ corresponding to vertical direction of the friction resultant are negative. The program, however, assumes these values to be positive. A seldom variant with friction acting upwards is not considered in the program.

The following theories and approaches are implemented for the computation of passive earth pressure **assuming effective stress state**:

- The Rankine and Mazindrani theory
- The Coulomb theory
- The Caquot - Kérisel theory
- The Müller - Breslau theory
- The Absi theory
- The Sokolovski theory
- SP 22.13330.2016

The program also allows for running the analysis in total stresses.

Passive Earth Pressure - The Rankine and Mazindrani Theory

Passive earth pressure follows from the following formula:

$$\sigma_p = \sigma_z.K_p = \gamma.z.K'_p.cos\,\beta$$

where: σ_z - vertical geostatic stress
K_p - coefficient of passive earth pressure due to Rankine
β - slope inclination
γ - weight of soil
z - assumed depth
K'_p - coefficient of passive earth pressure due to Mazindrani

The coefficient of passive earth pressure K_p is given by:

$$K'_p = \frac{1}{\cos^2\varphi}\left[\frac{2.\cos^2\beta + 2.\left(\frac{c}{\gamma.z}\right).\cos\varphi.\sin\varphi +}{\sqrt{4.\cos^2\beta.\left(\cos^2\beta - \cos^2\varphi\right) + 4.\left(\frac{c}{\gamma.z}\right)^2.\cos^2\varphi + 8.\left(\frac{c}{\gamma.z}\right).\cos^2\beta.\sin\varphi.\cos\varphi}}\right] - 1$$

where: β - slope inclination

φ - angle of internal friction of soil

c - cohesion of soil

If there is no friction ($\delta = 0$) between the structure and cohesionless soils ($c = 0$), the ground surface is horizontal ($\beta = 0$) and the resulting slip surface is also plane with the slope:

$$\vartheta_p = 45^\circ - \frac{\varphi}{2}$$

The Mazindrani theory then reduces to the Rankine theory. The coefficient of passive earth pressure is then provided by:

$$K_p = \frac{1 + \sin\varphi}{1 - \sin\varphi} = \tan^2\left(45^\circ + \frac{\varphi}{2}\right)$$

where: φ - angle of internal friction of soil

Passive earth pressure σ_p by Rankine for cohesionless soils is given:

$$\sigma_p = \gamma.z.K_p$$

where: γ - unit weight of soil

z - assumed depth

K_p - coefficient of passive earth pressure due to Rankine

Literature:

Mazindrani, Z.H., and Ganjali, M.H. 1997. Lateral earth pressure problem of cohesive backfill with inclined surface. Journal of Geotechnical and Geoenvironmental Engineering, ASCE, ***123****(2): 110-112.*

Passive Earth Pressure - The Coulomb Theory

Passive earth pressure follows from the following formula:

$$\sigma_p = \sigma_z.K_p + 2.c\sqrt{K_p}$$

where: σ_z - effective vertical geostatic stress

K_p - coefficient of passive earth pressure due to Coulomb

c - cohesion of soil

The coefficient of passive earth pressure K_p is given by:

$$K_p = \frac{\cos^2(\varphi + \alpha)}{\cos^2\alpha.\cos(\delta - \alpha).\left(1 - \sqrt{\frac{\sin(\varphi + \delta).\sin(\varphi + \beta)}{\cos(\delta - \alpha).\cos(\beta - \alpha)}}\right)^2}$$

where: φ - angle of internal friction of soil

δ - angle of friction between structure and soil

β - slope inclination

α - back face inclination of the structure

The vertical σ_{pv} and horizontal $\sigma\, ph$ components of passive earth pressure are given by:

$$\sigma_{px} = \sigma_p.\cos(\alpha + \delta)$$

$$\sigma_{pz} = \sigma_p.\sin(\alpha + \delta)$$

where: δ - angle of friction between structure and soil

α - back face inclination of the structure

Passive Earth Pressure - The Caquot - Kérisel Theory

Passive earth pressure follows from the following formula:

$$\sigma_p = \sigma_z . K_p \psi + 2.c\sqrt{K_p \psi}$$

where:

- K_p - coefficient of passive earth pressure for $\delta = -\varphi$, see the table
- ψ - reduction coefficient ψ for $|\delta| < \varphi$, see the table
- c - cohesion of soil
- σ_z - vertical geostatic stress

The vertical σ_{pv} and horizontal σ_{ph} components of passive earth pressure are given by:

$$\sigma_{px} = \sigma_p . cos(\alpha + \delta)$$

$$\sigma_{pz} = \sigma_p . sin(\alpha + \delta)$$

where:

- δ - angle of friction between structure and soil
- α - back face inclination of the structure

Coefficient of Passive Earth Pressure Kp

Coefficient of passive earth pressure K_p for $\delta = -\varphi$											
α [°]	φ [°]	K_p when β°									
		0	5	10	15	20	25	30	35	40	45
	10	1,17	1,41	1,53							
	15	1,39	1,70	1,92	2,06						
	20	1,71	2,06	2,42	2,71	2,92					
	25	2,14	2,61	2,96	3,66	4,22	4,43				
-30	30	2,78	3,42	4,16	5,01	5,96	6,94	7,40			
	35	3,75	4,73	5,87	7,21	8,76	10,60	12,50	13,60		
	40	5,31	6,87	8,77	11,00	13,70	17,20	24,60	25,40	28,40	
	45	8,05	10,70	14,20	18,40	23,80	30,50	38.90	49,10	60,70	69,10
	10	1,36	1,58	1,70							
	15	1,68	1,97	2,20	2,38						
	20	2,13	2,52	2,92	3,22	3,51					
	25	2,78	3,34	3,99	4,60	5,29	5,57				
-20	30	3,78	4,61	5,56	6,61	7,84	9,12	9,77			
	35	5,36	6,69	8,26	10,10	12,20	14,80	17,40	19,00		
	40	8,07	10,40	12,00	16,50	20,00	25,50	36,50	37,80	42,20	
	45	13,20	17,50	22,90	29,80	38,30	48,90	62,30	78,80	97,30	111,00
	10	1,52	1,72	1,83							
	15	1,95	2,23	2,57	2,66						
	20	2,57	2,98	3,42	3,75	4,09					
	25	3,50	4,14	4,90	5,62	6,45	6,81				
-10	30	4,98	6,01	7,19	8,51	10,10	11,70	12,60			
	35	7,47	9,24	11,30	13,80	16,70	20,10	23,70	26,00		
	40	12,00	15,40	19,40	24,10	29,80	37,10	53,20	55,10	61,60	
	45	21,20	27,90	36,50	47,20	60,60	77,30	98,20	124,00	153,00	176,00
	10	1,64	1,81	1,93							
	15	2,19	2,46	2,73	2,91						
	20	3,01	3,44	3,91	4,42	4,66					
	25	4,29	5,02	5,81	6,72	7,71	8,16				
0	30	6,42	7,69	9,13	10,80	12,70	14,80	15,90			
	35	10,20	12,60	15,30	18,60	22,30	26,90	31,70	34,90		
	40	17,50	22,30	28,00	34,80	42,90	53,30	76,40	79,10	88,70	
	45	33,50	44,10	57,40	74,10	94,70	120,00	153,00	174,00	240,00	275,00
	10	1,73	1,87	1,98							
	15	2,40	2,65	2,93	3,12						

	20	3,45	3,90	4,40	4,96	5,23					
10	25	5,17	5,99	6,90	7,95	9,11	9,67				
	30	8,17	9,69	11,40	13,50	15,90	18,50	19,90			
	35	13,80	16,90	20,50	24,80	29,80	35,80	42,30	46,60		
	40	25,50	32,20	40,40	49,90	61,70	76,40	110,00	113,00	127,00	
	45	52,90	69,40	90,00	116,00	148,00	188,00	239,00	303,00	375,00	431,00
	10	1,78	1,89	2,01							
	15	2,58	2,82	3,11	3,30						
	20	3,90	4,38	4,92	5,53	5,83					
20	25	6,18	7,12	8,17	9,39	10,70	11,40				
	30	10,40	12,30	14,40	16,90	20,00	23,20	25,00			
	35	18,70	22,80	27,60	33,30	40,00	48,00	56,80	62,50		
	40	37,20	46,90	58,60	72,50	89,30	111,00	158,00	164,00	185,00	
	45	84,00	110,00	143,00	184,00	234,00	297,00	378,00	478,00	592,00	680,00
	10	1,78	1,89	2,00							
	15	2,72	2,96	3,26	3,45						
	20	4,35	4,88	5,46	6,14	6,47					
30	25	7,33	8,43	9,65	11,10	12,70	13,50				
	30	13,10	15,50	18,20	21,40	25,20	29,30	31,60			
	35	25,50	31,00	37,50	45,20	54,20	65,20	77,00	84,80		
	40	54,60	68,80	86,00	107,00	131,00	162,00	232,00	241,00	271,00	
	45	135,00	176,00	228,00	293,00	374,00	475,00	604,00	763,00	945,00	1090,00

Reduction Coefficient of Passive Earth Pressure

Reduction coefficient ψ ***for*** $|\delta| < \varphi$

φ [°]	ψ for $\lvert\delta\rvert / \varphi$					
	1.0	**0.8**	**0.6**	**0.4**	**0.2**	**0.0**
10	1.00	0.989	0.962	0.929	0.898	0.864
15	1.00	0.979	0.934	0.881	0.830	0.775
20	1.00	0.968	0.901	0.824	0.752	0.678
25	1.00	0.954	0.860	0.759	0.666	0.574
30	1.00	0.937	0.811	0.686	0.574	0.467
35	1.00	0.916	0.752	0.603	0.475	0.362
40	1.00	0.886	0.682	0.512	0.375	0.262
45	1.00	0.848	0.600	0.414	0.276	0.174

Passive Earth Pressure - The Müller - Breslau Theory

Passive earth pressure follows from the following formula:

$$\sigma_p = \sigma_z . K_p + 2.c\sqrt{K_p}$$

where: K_p - coefficient of passive earth pressure

c - cohesion of soil

σ_z - vertical normal total stress

The coefficient of passive earth pressure K_p is given by:

$$K_p = \frac{cos^2(\varphi - \alpha)}{cos^2 \alpha . cos(\delta - \alpha)\left(1 + \sqrt{\frac{sin(\varphi - \delta).sin(\varphi + \beta)}{cos(\alpha - \delta).cos(\alpha + \beta)}}\right)^2}$$

where: φ - angle of internal friction of soil

δ - angle of friction between structure and soil

β - slope inclination

α - back face inclination of the structure

The vertical σ_{pv} and horizontal σ_{ph} components of passive earth pressure are given by:

$$\sigma_{px} = \sigma_p . cos(\alpha + \delta)$$
$$\sigma_{pz} = \sigma_p . sin(\alpha + \delta)$$

where: δ - angle of friction between structure and soil

α - back face inclination of the structure

Literature:

Müller-Breslau's Erddruck auf Stutzmauern,Stuttgart: Alfred Kroner-Verlag, 1906 (German).

Passive Earth Pressure - The Absi Theory

Passive earth pressure follows from the following formula:

$$\sigma_p = \sigma_z . K_p + 2.c\sqrt{K_p}$$

where: K_p - coefficient of passive earth pressure

c - cohesion of soil

σ_z - vertical normal total stress

The program takes values of the coefficient K_p from a database, built upon the tabulated values published in the book: Kérisel, Absi: Active and passive earth Pressure Tables, 3rd Ed. A.A. Balkema, 1990 ISBN 90 6191886 3.

The vertical σ_{pv} and horizontal σ_{ph} components of passive earth pressure are given by:

$$\sigma_{px} = \sigma_p . cos(\alpha + \delta)$$
$$\sigma_{pz} = \sigma_p . sin(\alpha + \delta)$$

where: δ - angle of friction between structure and soil

α - back face inclination of the structure

Literature:

Kérisel, Absi: Active and Passive Earth Pressure Tables, 3rd ed., Balkema, 1990 ISBN 90 6191886 3.

Passive Earth Pressure - The Sokolovski Theory

Passive earth pressure follows from the following formula:

$$\sigma_p = \sigma_z . K_{pg} + c.K_{pc} + p_v . K_{pp}$$

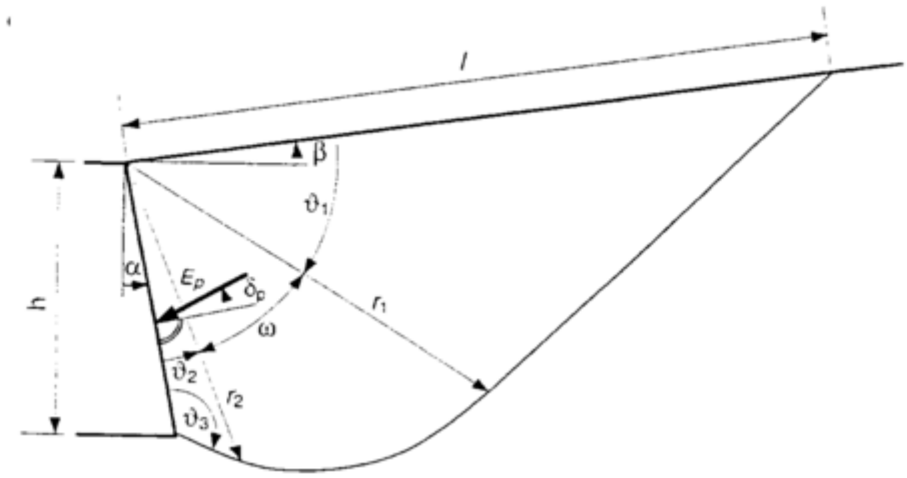
where: K_{pg} - passive earth pressure coefficient for cohesionless soils

K_{pc} - passive earth pressure coefficient due to cohesion

K_{pp} - passive earth pressure coefficient due to surcharge

σ_z - vertical normal total stress

Individual expressions for determining the magnitude of passive earth pressure and slip surface are introduced in the sequel; the meaning of individual variables is evident from Fig.:



Passive eart pressure slip surface after Sokolovski

Angles describing the slip surface:

$$\vartheta_1 = \frac{\pi}{4} - \frac{\varphi}{2} - \frac{\varepsilon_1 - \beta}{2}$$

$$\vartheta_2 = \frac{\pi}{4} + \frac{\varphi}{2} - \frac{\varepsilon_2 - \delta_p}{2}$$

$$\vartheta_3 = \frac{\pi}{4} + \frac{\varphi}{2} + \frac{\varepsilon_2 - \delta_p}{2}$$

$$\omega = \frac{\pi}{2} - \alpha + \beta - \vartheta_1 - \vartheta_2$$

$$\varepsilon_1 = \frac{\sin\beta}{\sin\varphi}$$

$$\varepsilon_2 = -\frac{\sin\delta_p}{\sin\varphi}$$

where:
- φ - angle of internal friction of soil
- δ_p - angle of friction between structure and soil
- β - slope inclination

Slip surface radius vector:

$$r_2 = \frac{h}{\cos\alpha} \cdot \frac{\sin\vartheta_3}{\sin(\vartheta_2 + \vartheta_3)}$$

$$r_1 = r_2 . e^{\omega . \tan\varphi}$$

$$I = r_1 . \frac{\cos\varphi}{\cos(\varphi + \vartheta_1)}$$

Provided that $\omega < 0$ the both straight edges of the zone r_1 and r_2 numerically overlap and resulting in the plane slip surface developed in the overlapping region. The coefficients of passive earth pressure K_{pg}, K_{pp}, K_{pc} then follow from:

$$K_{pg} = K_{pg,0} . i_{pg} . g_{pg} . t_{pg}$$

$$K_{pp} = K_{pp,0} . i_{pp} . g_{pp} . t_{pp}$$

$$K_{pc} = \cot\varphi . \left(K_{pp,0} . i_{pc} . g_{pc} . t_{pc} - \frac{1}{\cos\alpha . \cos\delta} \right)$$

where:
- φ - angle of internal friction of soil
- δ_p - angle of friction between structure and soil
- α - back face inclination of the structure

$$K_{pg,0} = K_{pp,0} = \frac{1 + \sin\varphi}{1 - \sin\varphi}$$

Auxiliary variables: i_{pg}, i_{pp}, i_{pc}, g_{pg}, g_{pp}, g_{pc}, t_{pg}, t_{pp}, t_{pc}

for:

$$i_{pg} = (1 - 0{,}53.\delta_p)^{0{,}26 + 5{,}96\varphi}, \quad i_{pp} = (1 - 1{,}33.\delta_p)^{0{,}08 + 2{,}37\varphi}, \quad i_{pc} = i_{pp}$$

$\beta \le 0$ $$g_{pg} = (1 + 0{,}73.\beta)^{2{,}89}, \quad g_{pp} = (1 + 1{,}16.\beta_p)^{1{,}57}, \quad g_{pc} = (1 + 0{,}001.\beta.\tan\varphi)^{205{,}4 + 2232\varphi}$$

$\beta > 0$ $$g_{pg} = (1 + 0{,}35.\beta)^{0{,}42 + 8{,}15\varphi}, \quad g_{pp} = (1 + 3{,}84.\beta_p)^{0{,}98\varphi}, \quad g_{pc} = e^{2.\beta.\tan\varphi}$$

$\alpha \le 0$ $$t_{pg} = (1 + 0{,}72.\alpha.\tan\varphi)^{3{,}51 + 1{,}03\varphi}$$

$\alpha > 0$ $$t_{pg} = (1 - 0{,}0012.\alpha.\tan\varphi)^{2910 - 1958\varphi}$$

$$t_{pp} = \frac{e^{-2.\alpha.\tan\varphi}}{\cos\alpha}$$

where:

$$t_{pc} = t_{pp}$$

For soils with zero value for the angle of internal friction the following expressions are employed to determine the coefficients of passive earth pressure:

$$K_{pp} = \cos\beta$$
$$K_{pc} = K_{pc,0}.i_{pc}.g_{pc}.t_{pc}$$

where:

$$K_{pc,0} = 2$$
$$i_{pc} = 1$$
$$g_{pc} = 1 + \beta$$
$$t_{pc} = \frac{1-\alpha}{\cos\alpha} \Rightarrow K_{pc} = \frac{2.(1+\beta).(1-\alpha)}{\cos\alpha}$$

Literature:

Sokolovski, V.V., 1960. Statics of Soil Media,Butterworth, London.

Passive Earth Pressure - SP 22.13330.2016

Passive earth pressure follows from the following formula:

$$\sigma_p = \sigma_z K_a + 2c\sqrt{K_p}$$

where:
- σ_z - vertical geostatic stress
- c - cohesion of soil
- K_p - coefficient of passive earth pressure

The coefficient of passive earth pressure K_p is given by:

$$K_p = \frac{\cos^2(\varphi+\alpha)}{\cos^2\alpha\left(1-\sqrt{\frac{\sin(\varphi+\delta)\sin(\varphi+\beta)}{\cos(\delta-\alpha)\cos(\beta-\alpha)}}\right)^2}$$

where:
- φ - angle of internal friction of soil
- δ - angle of friction between structure and soil
- β - slope inclination
- α - back face inclination of the structure

Horizontal and vertical components of the passive earth pressure become:

$$\sigma_{px} = \sigma_p \cos(\alpha+\delta)$$
$$\sigma_{pz} = \sigma_p \sin(\alpha+\delta)$$

where:
- σ_p - passive earth pressure
- δ - angle of friction between structure and soil
- α - back face inclination of the structure

Passive Earth Pressure - Total Stress

When determining the passive earth pressure in cohesive fully saturated soils, in which case the consolidation is usually prevented (undrained conditions), the horizontal normal total stress σ_x receives the form:

$$\sigma_x = \sigma_z - K_{uc}.c_u$$

where:
- σ_x - horizontal total stress (normal)
- σ_z - vertical normal total stress
- K_{uc} - coefficient of earth pressure

c_u - total cohesion of soil

The coefficient of earth pressure K_{uc} is given by:

$$K_{uc} = -2.\sqrt{1+\frac{a_u}{c_u}}$$

where: K_{uc} - coefficient of earth pressure

c_u - total cohesion of soil

a_u - total adhesion of soil to the structure

Earth Pressure at Rest

Earth pressure at rest rest is the horizontal pressure acting on the rigid structure. It is usually assumed in cases when it is necessary to minimize the lateral and horizontal deformation of the sheeted soil (e.g. when laterally supporting a structure in the excavation pit up to depth below the current foundation or in general when casing soil with structures sensitive to non-uniform settlement), or when structures loaded by earth pressures are due to some technological reasons extremely rigid and do not allow for deformation in the direction of load necessary to mobilize the active earth pressure.

Earth pressure at rest is given by:

$$\sigma_r = \sigma_z . K_r$$

For **cohesive soils** the Terzaghi formula for computing K_r is implemented in the program:

$$K_r = \frac{\nu}{1-\nu}$$

where: ν - Poisson's ratio

For **cohesionless soils** the Jáky expression is used:

$$K_r = 1 - sin\varphi$$

where: φ - angle of internal friction of soil

When computing the pressure at rest for cohesive soils σ_r using the Jáky formula for the determination of coefficient of earth pressure at rest K_r, it is recommended to use the alternate angle of internal friction φ_n.

The way of computing the earth pressure at rest can be therefore influenced by the selection of the type of soil (cohesive, cohesionless) when inputting its parameters. Even typically cohesionless soil (sand, gravel) must be introduced as cohesive if we wish to compute the pressure at rest with the help of the Poisson ratio and vice versa.

For **overconsolidated soils** the expression proposed by Schmertmann to compute the coefficient of earth pressure at rest K_r is used:

$$K_r = 0{,}5.(OCR)^{0{,}5}$$

where: K_r - coefficient of earth pressure at rest

OCR - overconsolidation ratio

The value of the coefficient of earth pressure at rest can be **input also directly**.

The program calculates the influence of the inclined ground surface or the back of structure and increase of pressure at rest from the surcharge.

Earth Pressure at Rest for an Inclined Ground Surface or Inclined Back of the Structure

For inclined ground surface behind the structure ($0° < \beta \leq \varphi$) the earth pressure at rest assumes the form:

$$\sigma_r = \frac{\sigma_z . K_r . sin\varphi . cos\beta}{sin\varphi - sin^2\beta}$$

where: φ - angle of internal friction of soil

β - slope inclination

σ_z - vertical geostatic stress

K_r - coefficient of earth pressure at rest

For the inclined back of wall the values of earth pressure at rest are derived from:

$$\sigma_r = \sigma_z \sqrt{sin^2 \alpha + K_r^2 .cos^2 \alpha}$$

where: α - back face inclination of the structure

σ_z - vertical geostatic stress

K_r - coefficient of earth pressure at rest

Normal and tangential components are given by:

$$\sigma = \sigma_z .\left(sin^2 \alpha + K_r .cos^2 \alpha\right)$$

$$\tau = \sigma_z .(1 - K_r).sin \alpha .cos \alpha$$

where: α - back face inclination of the structure

σ_z - vertical geostatic stress

K_r - coefficient of earth pressure at rest

The deviation angle from the normal line to the wall δ reads:

$$tg\delta = \frac{(1 - K_r).tg\alpha}{K_r + tg^2 \alpha}$$

where: α - back face inclination of the structure

K_r - coefficient of earth pressure at rest

Increased Active Pressure

The **increased active pressure** is calculated using k coefficient.

The magnitude of the pressure is calculated by the formula:

$$\sigma = k\,\sigma_r + (1 - k)\sigma_a$$

where: σ_r - pressure at rest

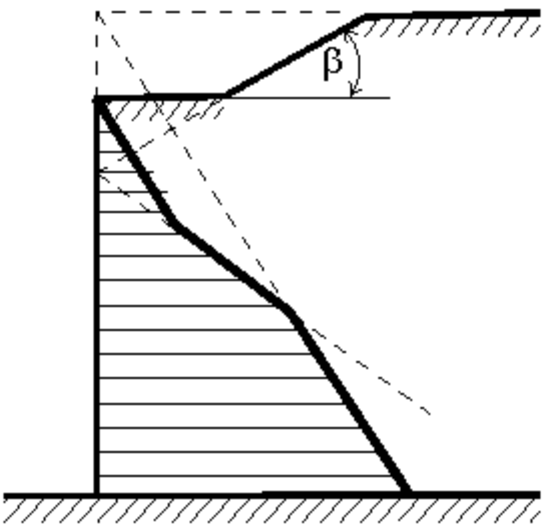
σ_a - active earth pressure

k - coefficient of increased active pressure

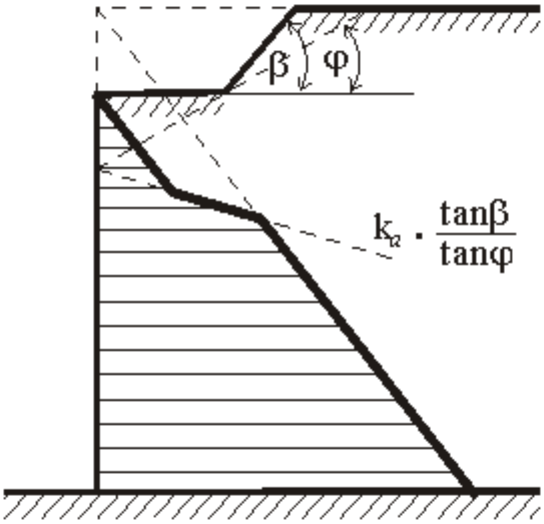
If the value of coefficient k= 1, then the acting pressure is equal to the pressure at rest. If the value of coefficient k= 0, then the acting pressure is equal to the active pressure.

Distribution of Earth Pressures in case of Broken Terrain

Figures show the procedure of earth pressure analysis in case of sloping terrain. The resulting shape of earth pressure distribution acting on the construction is obtained from the sum of triangular distributions developed by individual effects acting on the construction.



Principle of the earth pressure computation in case of broken terrain



Principle of the earth pressure computation in case of broken terrain for $\beta > \varphi$

Influence of Water

The influence of groundwater can be reflected using one of the following variants:

Without groundwater, water is not considered

Hydrostatic pressure, groundwater behind structure

Hydrostatic pressure, groundwater behind and in front of structure

Hydrodynamic pressure

Special distribution of water pressure

Without Ground Water, Water is not Considered

Without groundwater, water is not considered

In this option, the influence of ground water is not considered.

Complementary information:

If there are fine soils at and below the level of GWT, one should carefully assess an influence of full saturation in the region of capillary attraction. The capillary attraction is in the analysis reflected only by the increased degree of saturation, and therefore the value of γ_{sat} is inserted into parameters of soils.

To distinguish regions with different degree of saturation, one may insert several layers of the same soil with different unit weights. Negative pore pressures are not considered. However, for layers with different degree of saturation it is possible to use different values of shear resistance influenced by suction (difference in pore pressure of water and gas ($u_a - u_w$).

Hydrostatic Pressure, Ground Water behind the Structure

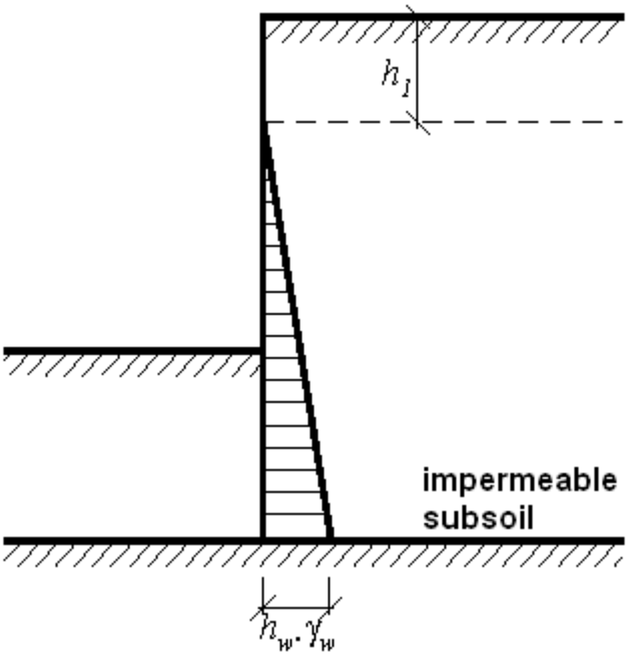
Hydrostatic pressure, groundwater behind the structure

The heel of a structure is sunk into impermeable subsoil so that the water flow below the structure is prevented. Water is found behind the back of structure only. There is no water acting on the front face. Such a case may occur when water in front of structure flow freely due to gravity or deep drainage is used. The back of structure is loaded by the hydrostatic pressure:

$$u = \gamma_w . h_w$$

where: γ_w - unit weight of water

h_w - water tables difference

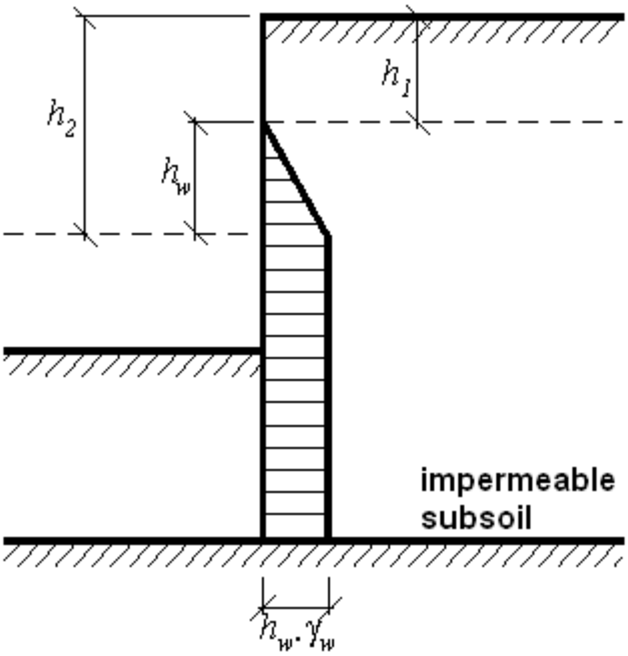


Action of hydrostatic pressure

Hydrostatic Pressure, Ground Water behind and in front of the Structure

Hydrostatic pressure, groundwater behind and in front of structure

The heel of a structure is sunk into impermeable subsoil so that the water flow below the structure is prevented. The load due water is assumed both in front of and behind the structure. The water in front of the structure is removed either with the help of gravity effects or is shallowly lowered by pumping. Both the face and back of the structure are loaded by hydrostatic pressure due to the difference in water tables (h_1 and h_2). The dimension h_w represents the difference in water tables at the back and in front of structure - see figure:

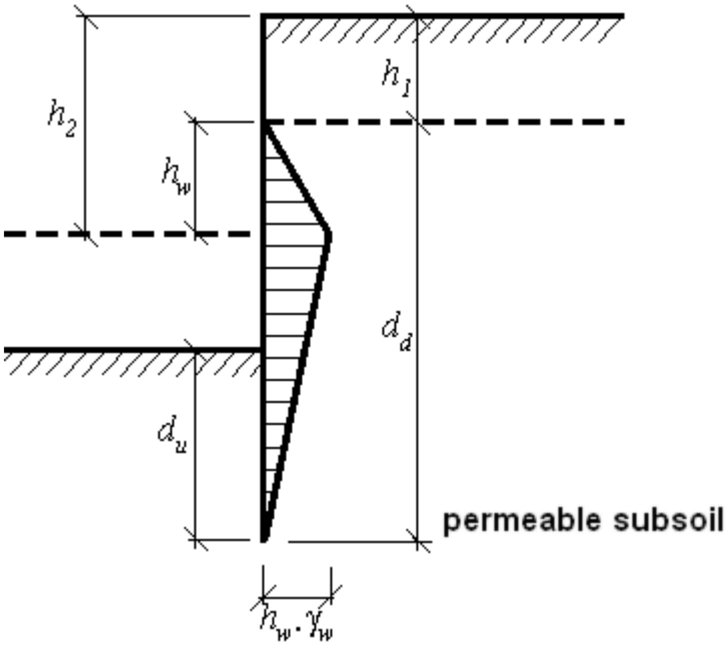


Action of hydrostatic pressure

Hydrodynamic Pressure

Hydrodynamic pressure

The heel of a structure is sunk into permeable subsoil, which allows free water flow below the structure - see figure. The unit weight of soil lifted by uplift pressure γ_{su} is modified to account for flow pressure. These modifications then depend on the direction of water flow.



Action of hydrodynamic pressure

When computing the earth pressure in the area of descending flow the program introduces the following value of the unit weight of soil:

$$\gamma = \gamma_{su} + \Delta\gamma = \gamma_{su} + i.\gamma_w$$

and in the area of ascending flow the following value:

$$\gamma = \gamma_{su} - \Delta\gamma = \gamma_{su} - i.\gamma_w$$

where: γ_{su} - unit weight of submerged soil

$\Delta\gamma$ - alteration of unit weight of soil

i - an average seepage gradient
γ_w - unit weight of water

An average hydraulic slope is given:

$$i = \frac{h_w}{d_d + d_u}$$

where:
i - an average seepage gradient
h_w - water tables difference
d_d - seepage path downwards
d_u - seepage path upwards

If the change of unit weight of soil $\Delta\gamma$ provided by:

$$\Delta\gamma = i.\gamma_w$$

where:
i - an average seepage gradient
γ_w - unit weight of water

Is greater than the unit weight of saturated soil γ_{su}, then the leaching appears in front of structure - as a consequence of water flow the soil behaves as weightless and thus cannot transmit any load. The program then prompts a warning message and further assumes the value of $\gamma = 0$.

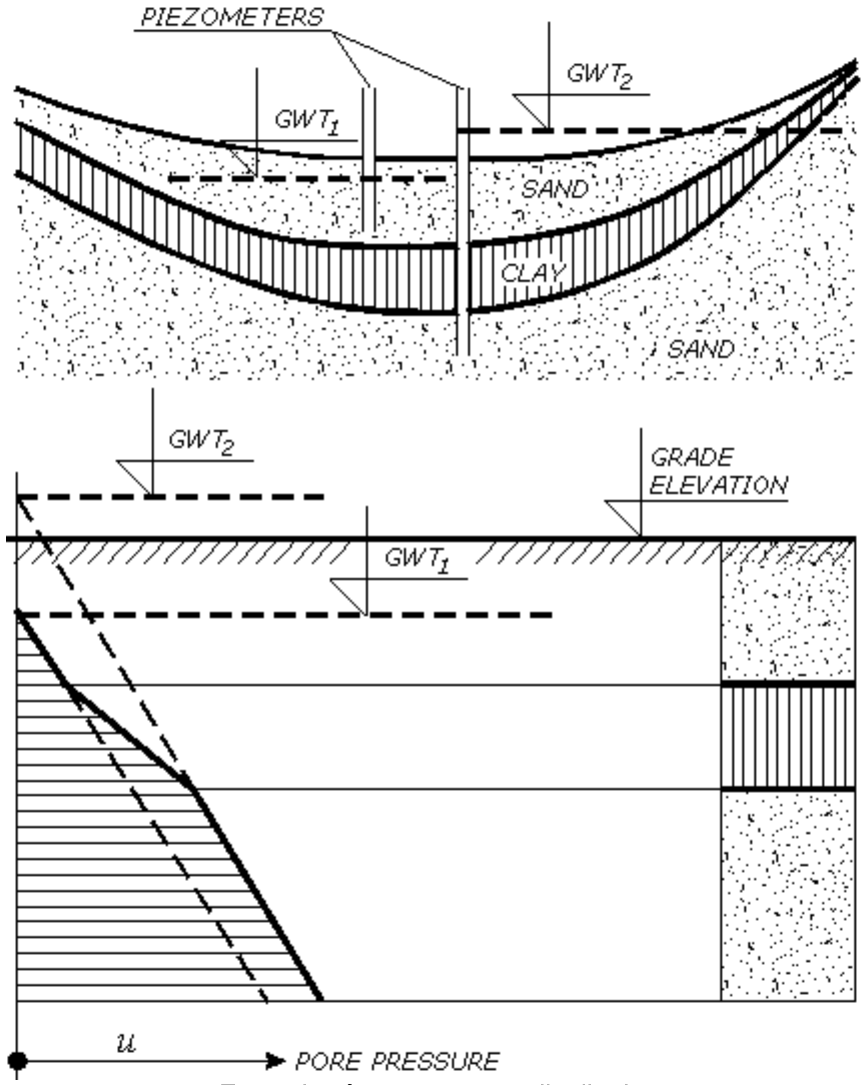
Special Distribution of Water Pressure

Special distribution of water pressure

This option allows an independent (manual) input of distribution of load due to water at the back and in front of structure using ordinates of pore pressure at different depths. The variation of pressure between individual values is linear. At the same time it is necessary to input levels of tables of full saturation of a soil at the back h_1 and in front h_2 of structure including possible decrease of unit weight δ_y in front of structure due to water flow.

Example: Two separated horizon lines of groundwater.

There are two permeable layers (sand or gravel) with one impermeable layer of clay in between, which causes separation of two hydraulic horizon lines - see figure:



Example of pore pressure distribution

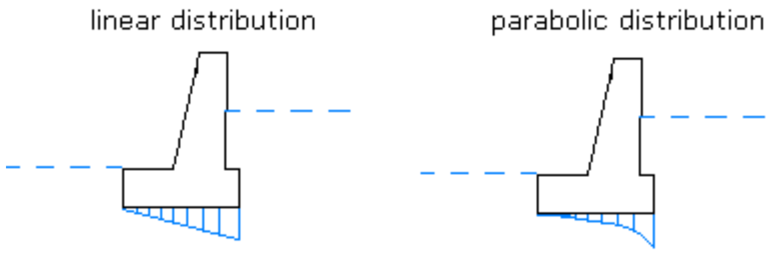
The variation of pore pressure above the clay layer is driven by free groundwater table GWT_1. The distribution of pore pressure below the clay layer results from ratio in the lower separated groundwater table GWT_2, where the groundwater is stressed. The pore pressure distribution in clay is approximately linear.

The capillary attraction is in the analysis reflected only by an increased degree of saturation, and therefore the value of γ_{sat} is inserted into parameters of soils.

To distinguish regions with different degrees of saturation, one may insert several layers of the same soil with different unit weights. Negative pore pressures are not considered. However, for layers with different degrees of saturation, it is possible to introduce values of shear resistance influenced by suction.

Uplift Pressure in Footing Bottom

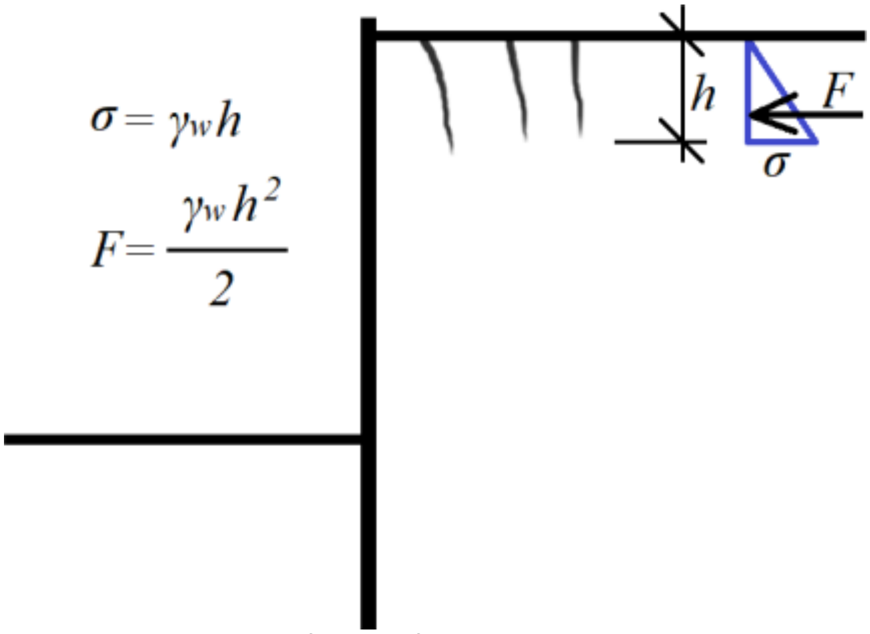
The variation of uplift pressure in the footing bottom due to difference in water tables is assumed according to expected effect linear, parabolic or is not taken into account.



Uplift pressure in footing bottom

Influence of Tensile Cracks

The program makes it possible to account for the influence of tensile surface cracks filled with water. The analysis procedure is evident from the figure. The depth of tensile cracks is the only input parameter.

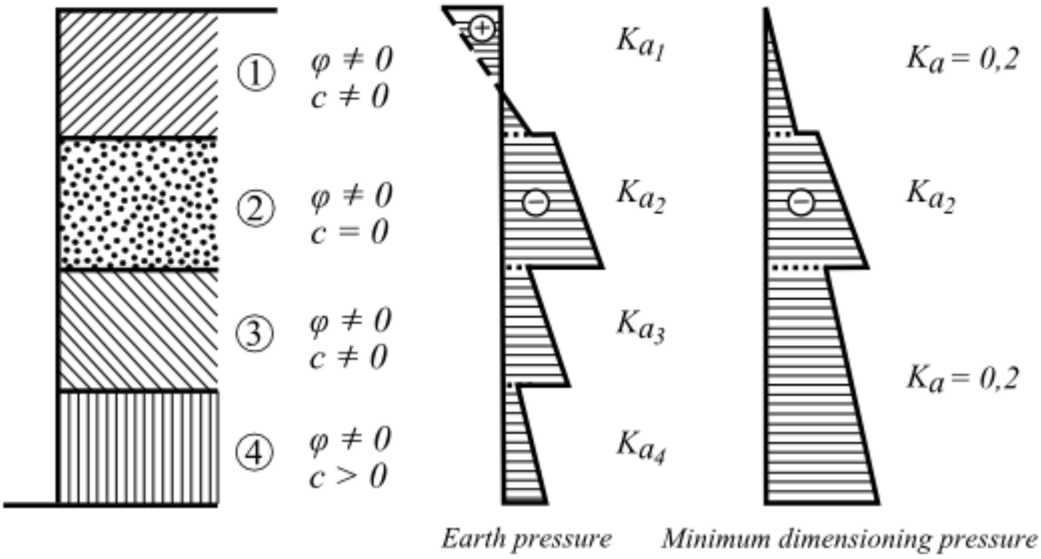


Influence of tensile cracks

Minimum Dimensioning Pressure

When determining the magnitude and distribution of earth pressures it is very difficult to qualify proportions of individual effects. This situation leads to uncertainty in the determination of earth pressure loading diagram. In reality, we have to use in the design the most adverse distribution in favor of the safety of the structure. For example, in case of braced structures in cohesive soils when using reasonable values of strength parameters of soil along the entire structure we may encounter tensile stresses in the upper part of the structure - see figure. Such tensile stresses, however, cannot be exerted on the sheeting structure (consequence of the separation of soil due to technology of construction, isolation and drainage layer). In favor of the safe design of sheeting structure particularly in subsurface regions, where tensile stresses are developed during computation of the active earth pressure, the program offers the possibility to call the option "**Minimum dimensioning pressure**" in the analysis.

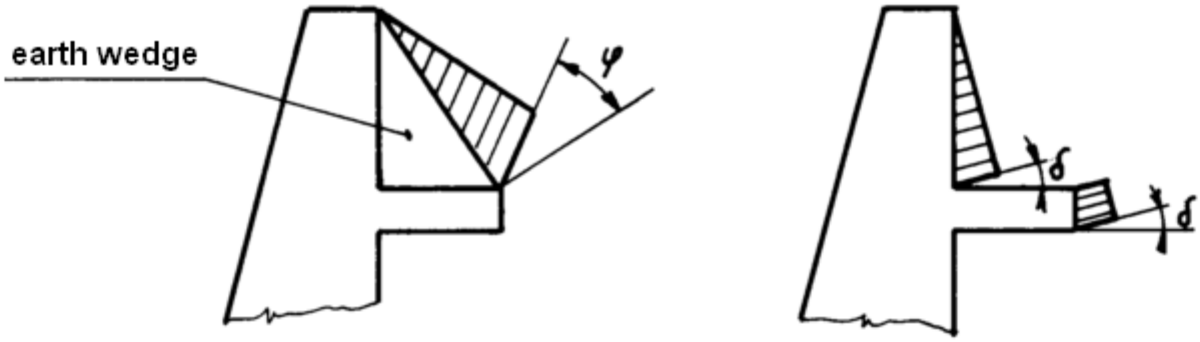
To determine the minimum dimensioning pressure the program employs for layers of cohesive soils as the minimal value of the coefficient of active earth pressure an alternate coefficient $K_a =$ 0.2. Therefore it is ensured that the value of the computed active earth pressure will not drop below *20%* of the vertical pressure ($K_a \geq$ 0.2) - see figure. Application of the minimum dimensioning pressure assumes, for example, the possibility of increasing the lateral pressure due to filling of joint behind the sheeting structure with rainwater. If the option of minimum dimensioning pressure is not selected the program simply assumes tension cutoff ($K_a \geq$ 0.0).



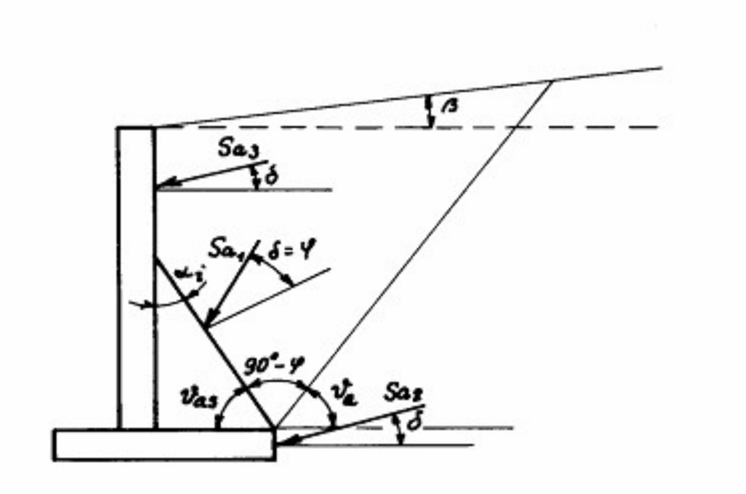
Minimum dimensioning pressure

Earth - Pressure Wedge

Providing a structure with a cantilever jump (foundation slab of cantilever wall, modification for reduction of earth pressures) is considered when computing earth pressures it is possible to compute earth pressures acting either on the real back of structure with the input angle of friction δ or on an alternate back of structure. The alternate back of structure replaces the real broken one by a slip plane passing from the upper point of the back of wall towards the outer upper point of the jump and forms an earth wedge - see figure. Standardly, a fully mobilized angle of friction $\delta = \varphi$ is assumed along this plane. In all programs, where the earth wedge can be created, the soil/soil friction angle can be reduced in the "**Stage settings**" frame. The weight of earth wedge created under this alternate back further contributes to load applied to the structure. To introduce the alternate back of structure into the analysis it is necessary to select in the program "**Earth pressures**" the option "**Consider developing of earth-pressure wedge**". In other programs the earth wedge is introduced automatically.



Calculation with and without earth-pressure wedge



Determination of earth-pressure wedge in case of active earth pressure

The slip plane of the earth-pressure wedge is inclined from the horizontal line by angle υ_a given by:

$$\upsilon_a = \varphi + \varepsilon$$

$$tan\varepsilon = \frac{cos(\varphi-\alpha).sin(\varphi-\beta).cos(\alpha+\delta)+B.cos(\varphi-\beta-\alpha-\delta)}{sin(\varphi-\alpha).sin(\varphi-\beta).cos(\alpha+\delta)+B.sin(\varphi-\beta-\alpha-\delta)+M}$$

$$M = \sqrt{(sin(\varphi-\beta).cos(\beta-\alpha)+B).(sin(\varphi+\delta).cos(\alpha+\delta)+B)}$$

$$B = \frac{2.c.cos\,\alpha.cos(\beta-\alpha).cos\,\varphi}{\gamma.h.cos(\beta-\alpha)+\dfrac{2.\sigma_z.cos\,\alpha.cos\,\beta}{\gamma.h}}$$

where:

φ - angle of internal friction of soil

β - slope inclination

δ - angle of friction between structure and soil

γ - unit weight of soil

α - back face inclination of the structure

h - height of earth wedge

The shape of the earth wedge in the layered subsoil is determined such that for individual layers of soil above the wall foundation the program computes the angle υ_a, which then serves to determine the angle υ_{as}. Next, the program determines an intersection of the line drawn under the angle υ_{as} from the upper right point of the foundation block with the next layer. The procedure continues by drawing another line starting from the previously determined intersection and inclined by the angle υ_{as}. The procedure is terminated when the line intersects the terrain or wall surface, respectively. The wedge shape is further assumed in the form of triangle (intersection with wall) or rectangle (intersection with the terrain).

Surcharge

The following types of surcharges are implemented in the program:

Active earth pressure

- Surface surcharge
- Strip surcharge
- Trapezoidal surcharge
- Concentrated surcharge
- Line surcharge
- Horizontal surcharge

Earth pressure at rest

- Surface surcharge
- Strip surcharge
- Trapezoidal surcharge
- Concentrated surcharge
- Horizontal surcharge

Passive earth pressure

- Surface surcharge

Surface Surcharge - Active Earth Pressure

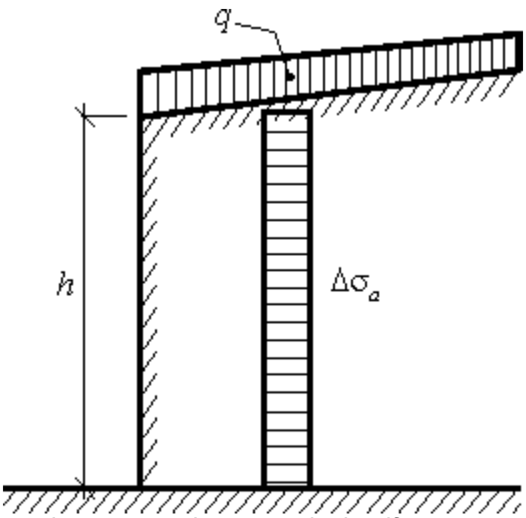
The increment of active earth pressure at rest due to surface surcharge is given by:

$$\Delta\sigma_a = p.K_a$$

where: p - vertical uniform load

K_a - coefficient of active earth pressure

The vertical uniform load p applied to the ground surface induces therefore over the entire height of the structure a constant increment of active earth pressure - see figure:



Increment of active earth pressure due to vertical uniform ground surface surcharge

Strip Surcharge - Active Earth Pressure

For vertical strip load f_a acting parallel with structure on the ground surface along an infinitely long strip the trapezoidal increment of active earth pressure applied to the structure over a given segment h_f is assumed - see figure.

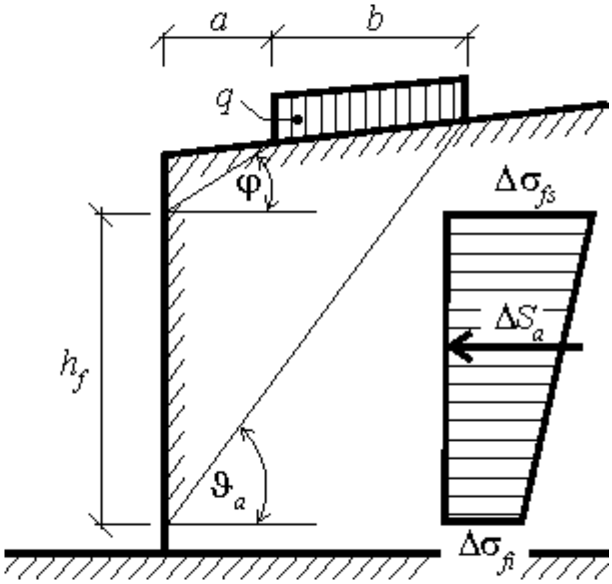


Diagram of increment of active earth pressure due to strip load f_a

This segment is determined by intersection of the structure and lines drawn from the edge points of the strip load having slopes associated with angles φ and ϑ_a. The angle ϑ_a corresponding to critical slip plane follows from:

$$\vartheta_a = \varphi + \varepsilon$$

The formula is described in more detail in section "**Active earth pressure** - line surcharge".

Variation of pressure increment is trapezoidal; the larger intensity of $\Delta\sigma_{fs}$ is applied at the upper end while the smaller intensity of $\Delta\sigma_{fi}$ at the bottom end. The two increments are given by:

$$\Delta\sigma_{fs} = \frac{f_a.b.K_{af}}{h_f}.\left(1 + \frac{a}{a+b}\right)$$

$$\Delta\sigma_{fi} = \frac{f_a.b.K_{af}}{h_f}.\left(1 - \frac{a}{a+b}\right)$$

where: f_a - magnitude of strip surcharge

b - width of the strip surcharge acting normal to the structure

h_f - section loaded by active earth pressure increment

$$K_{af} = \frac{sin(\vartheta_a - \varphi)}{cos(\vartheta_a - \varphi - \delta)}$$

where: ϑ_a - angle of critical slip plane

φ - angle of internal friction of soil

δ - angle of friction between structure and soil

The resultant of the increment of active earth pressure due to strip load f_a is provided by:

$$\Delta S_a = f_a.b.\frac{sin(\vartheta_a - \varphi)}{cos(\vartheta_a - \varphi - \delta)}$$

where: ϑ_a - angle of the critical slip plane

φ - angle of internal friction of soil

δ - angle of friction between structure and soil

f_a - magnitude of strip surcharge

b - width of the strip surcharge

For non-homogeneous soils the program proceeds as follows.

Trapezoidal Surcharge - Active Earth Pressure

The trapezoidal surcharge is subdivided in the program in ten segments. Individual segments are treated as strip loads. The resulting earth pressure is a sum of partial surcharges derived from individual segments.

Concentrated Surcharge - Active Earth Pressure

The concentrated load (resultant F due to surface or concentrated load - see figure) is transformed into a line load with a limited length. If the width of surface load b is smaller than the distance a from the back of wall (see figure) the alternate line load f having length $l+2\ (a+b)$ receives the form:

$$f = \frac{F}{l + 2.(a+b)}$$

where:

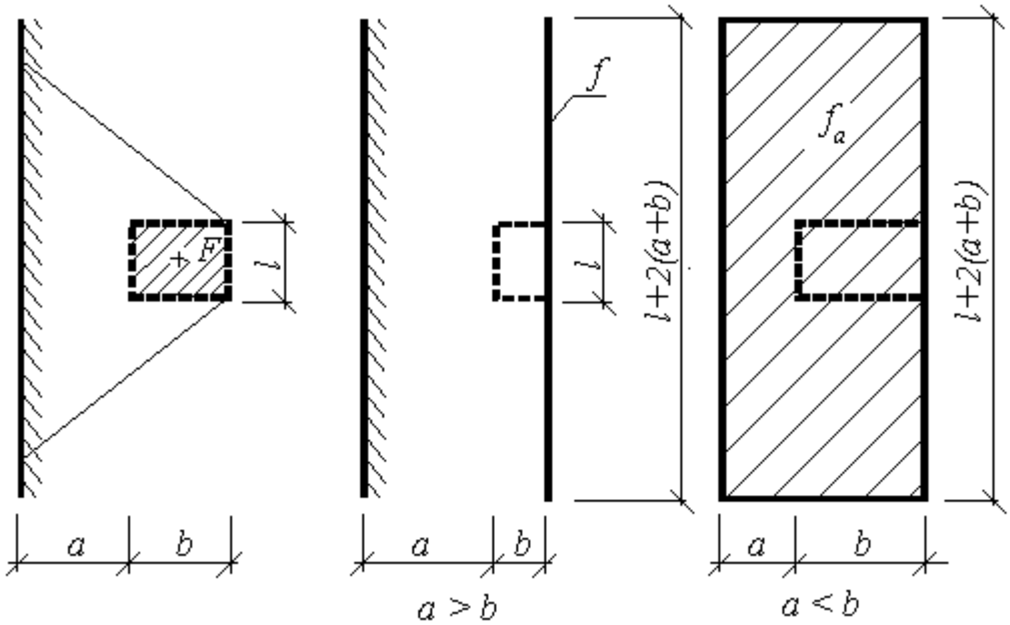
- F - resultant due to surface or concentrated load
- a - distance of load from the back of wall
- l - length of load
- b - width of surface load

If the width b of surface load is greater than the distance a from the back of wall (see figure) the alternate strip load f having length $l+2\ (a+b)$ and width $(a+b)$ reads:

$$f_a = \frac{F}{(l + 2.(a+b)).(a+b)}$$

where:

- F - resultant due to surface or concentrated load
- a - distance of load from the back of wall
- l - length of load
- b - width of surface load



Alternate load for calculation of increment of active earth pressure

For non-homogeneous soils the program proceeds as follows.

Line Surcharge - Active Earth Pressure

Vertical infinitely long line load f acting on the ground surface parallel with structure leads to a triangular increment of active earth pressure applied to the structure over a given segment h_f - see figure:

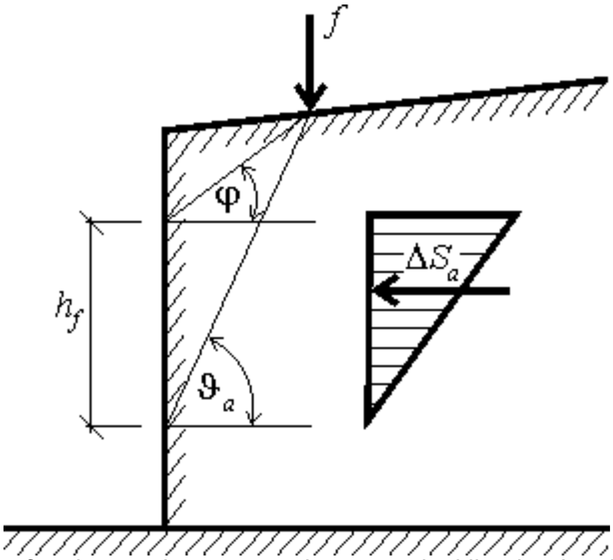


Diagram of increment of active earth pressure due to vertical line load acting on ground surface

Action of the line surcharge is determined such that two lines are drawn from the point of application following angles φ and ϑ_a (corresponding to the critical slip surface), which is provided by:

$$\vartheta_a = \varphi + \varepsilon$$

where: φ - angle of internal friction of soil

ε - angle derived from the following formulas

$$tan\varepsilon = \frac{cos(\varphi-\alpha).sin(\varphi-\beta).cos(\alpha+\delta)+B.cos(\varphi-\beta-\alpha-\delta)}{sin(\varphi-\alpha).sin(\varphi-\beta).cos(\alpha+\delta)+B.sin(\varphi-\beta-\alpha-\delta)+M}$$

$$M = \sqrt{(sin(\varphi-\beta).cos(\beta-\alpha)+B)(sin(\varphi+\delta).cos(\alpha+\delta)+B)}$$

$$B = \frac{2.c.cos\,\alpha.cos(\beta-\alpha).cos\,\varphi}{\gamma.h.cos(\beta-\alpha)+\frac{2.\sigma_z.cos\,\alpha.cos\,\beta}{\gamma.h}}$$

where: β - slope inclination

φ - angle of internal friction of soil
δ - angle of friction between structure and soil
α - back face inclination of the structure
c - cohesion of soil
γ - unit weight of soil
h - assumed depth

For non-homogeneous soil and inclination of ground surface β smaller than the angle of internal friction of the soil φ the value of the angle ε is given by:

$$cot\,g = tg(\varphi-\alpha)+\frac{1}{cos(\varphi-\alpha)}\sqrt{\frac{sin(\varphi+\delta).cos(\alpha-\beta)}{sin(\varphi-\beta).cos(\alpha+\delta)}}$$

where: β - slope inclination

φ - angle of internal friction of soil
δ - angle of friction between structure and soil
α - back face inclination of the structure

The resultant of the increment of active earth pressure due to line load f is provided by:

$$\Delta S_a = f.\frac{sin(\vartheta_a-\varphi)}{cos(\vartheta_a-\varphi-\delta)}$$

where: ϑ_a - angle of the critical slip plane

φ - angle of internal friction of soil
δ - angle of friction between structure and soil

f - magnitude of line surcharge

For non-homogeneous soils the program proceeds as follows.

Increment of earth pressure due to horizontal surcharge

Infinitely long distributed load f acting horizontally the on the ground surface leads to a triangular increment of active earth pressure applied to the structure over a given segment h_f - see figure:

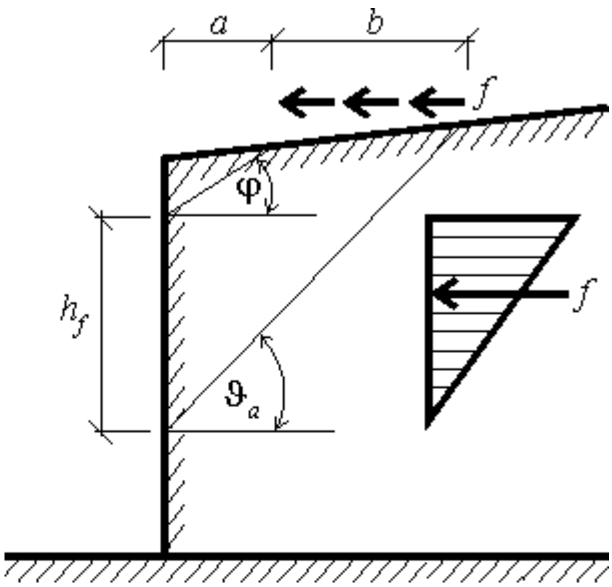


Diagram of increment of active earth pressure due to horizontal distributed load acting on ground surface

Range of the earth pressure contribution is determined by two lines drawn from the endpoints of the distributed load with angle φ and angle ϑa (corresponding to the critical slip surface), which is provided by:

$$\vartheta_a = \varphi + \varepsilon$$

where: φ - angle of internal friction of soil

ε - angle derived from the following formulas

$$tan\varepsilon = \frac{cos(\varphi-\alpha).sin(\varphi-\beta).cos(\alpha+\delta)+B.cos(\varphi-\beta-\alpha-\delta)}{sin(\varphi-\alpha).sin(\varphi-\beta).cos(\alpha+\delta)+B.sin(\varphi-\beta-\alpha-\delta)+M}$$

$$M = \sqrt{(sin(\varphi-\beta).cos(\beta-\alpha)+B)(sin(\varphi+\delta).cos(\alpha+\delta)+B)}$$

$$B = \frac{2.c.cos\,\alpha.cos(\beta-\alpha).cos\,\varphi}{\gamma.h.cos(\beta-\alpha)+\frac{2.\sigma_z.cos\,\alpha.cos\,\beta}{\gamma.h}}$$

where: β - slope inclination

φ - angle of internal friction of soil
δ - angle of friction between structure and soil
α - back face inclination of the structure
c - cohesion of soil
γ - unit weight of soil
h - assumed depth

For non-homogeneous soil and inclination of ground surface β smaller than the angle of internal friction of the soil φ the value of the angle ε is given by:

$$cot\,g = tg(\varphi-\alpha)+\frac{1}{cos(\varphi-\alpha)}\sqrt{\frac{sin(\varphi+\delta).cos(\alpha-\beta)}{sin(\varphi-\beta).cos(\alpha+\delta)}}$$

where: β - slope inclination

φ - angle of internal friction of soil
δ - angle of friction between structure and soil
α - back face inclination of the structure

The resultant of the increment of active earth pressure due to line load f is provided by:

$$\Delta S_a = f \cdot \frac{sin(\vartheta_a - \varphi)}{cos(\vartheta_a - \varphi - \delta)}$$

where: ϑ_a - angle of the critical slip plane

φ - angle of internal friction of soil

δ - angle of friction between structure and soil

f - magnitude of line surcharge

For non-homogeneous soils the program proceeds as follows.

Surcharge in Non-Homogeneous Soil

For non-homogeneous soil we proceed as follows:

- Compute the angle ϑ_a for a given soil layer.
- Determine the corresponding magnitude of force S_a and size of the corresponding pressure diagram.
- Determine the magnitude of earth pressure acting below the bottom edge of a given layer, and its ratio with respect to the overall pressure magnitude.
- The surcharge is reduced using the above ratio, then the location of this surcharge on the upper edge of the subsequent layer is determined.
- Compute again the angle ϑ_a for the next layer and repeat the previous steps until the bottom of a structure is reached or the surcharge is completely exhausted.

Surface Surcharge - Earth Pressure at Rest

An increment of uniformly distributed earth pressure at rest $\Delta\sigma_r$ caused by the vertical surface load applied on the ground surface behind the structure is computed using the following formula:

$$\Delta \sigma_r = f . K_r$$

where: f - magnitude of surface surcharge

K_r - coefficient of earth pressure at rest

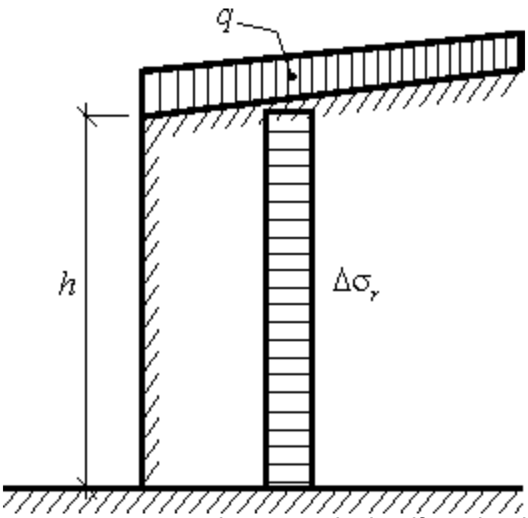


Diagram of increment of earth pressure at rest due to vertical uniform load acting on the ground surface

Strip Surcharge - Earth Pressure at Rest

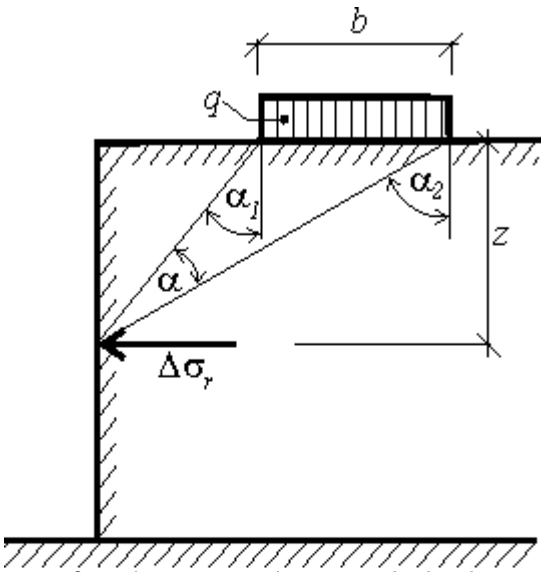
Uniform strip load f_a acting on the ground surface behind the structure parallel with vertical structure (see figure) creates an increment of earth pressure at rest $\Delta\sigma_r$ having the magnitude given by:

$$\Delta \sigma_r = \frac{f_a}{\pi}(2.\alpha - sin 2\alpha_2 + sin 2\alpha_1)$$

where: f_a - vertical strip surcharge

α , α_1 , α_2 - evident from the figure

The increase of the pressure can never be higher than pressure from surface surcharge of the same magnitude.



Increment of earth pressure due to vertical strip surcharge

Trapezoidal Surcharge - Earth Pressure at Rest

The trapezoidal surcharge is subdivided in the program in five segments. Individual segments are treated as strip loads. The resulting earth pressure is a sum of partial surcharges derived from individual segments.

Concentrated Surcharge - Earth Pressure at Rest

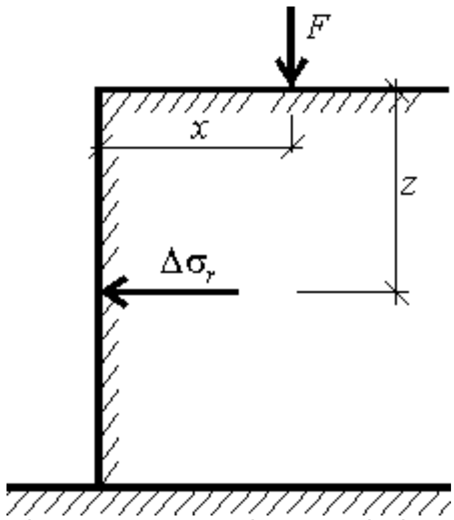
Application of concentrated force F yields an increment of earth pressure at rest $\Delta\sigma_r$ acting on the vertical structure and having the magnitude of:

$$\Delta\sigma_r = \frac{3.F}{\pi}\left(\frac{x^2.z}{r^5} + \frac{1-2.\nu}{3}\left(\frac{1}{r.(r+z)} - \frac{(2.r+z).x^2}{(r+z)^2.x^3} - \frac{z}{r^3}\right)\right)$$

$$r = \sqrt{x^2 + z^2}$$

where: F - concentrated force acting on ground surface

x, z - coordinates evident from figure



Increment of earth pressure at rest due to vertical concentrated force

Surface Surcharge - Passive Earth Pressure

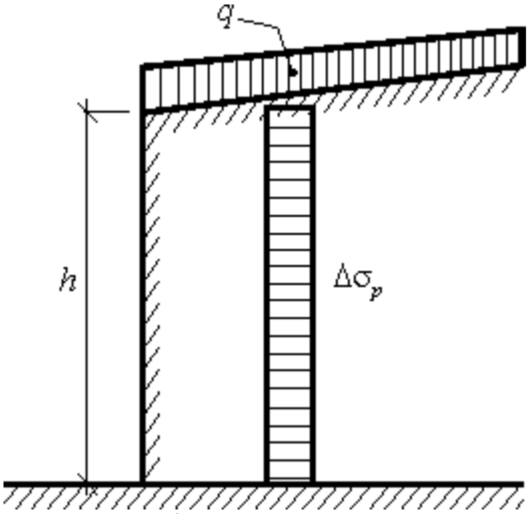
For passive earth pressure only an increment due to vertical uniform load f_a is determined using the formula:

$$\Delta\sigma_p = f_a.K_p$$

where: f_a - vertical surface surcharge

K_p - coefficient of passive earth pressure

The vertical uniform load q acting on the ground surface therefore results in a constant increment of passive pressure applied over the entire length of the wall - see the figure.



Increment of the passive earth pressure

Influence of Backfill

The programs can use one of the following variants of a backfill:

Backfill is not considered

Type 1 - backfill behind the structure in soils led from the upper edge of the foundation slab

Type 2 - backfill behind the structure in soils led from the lowest point of the foundation slab

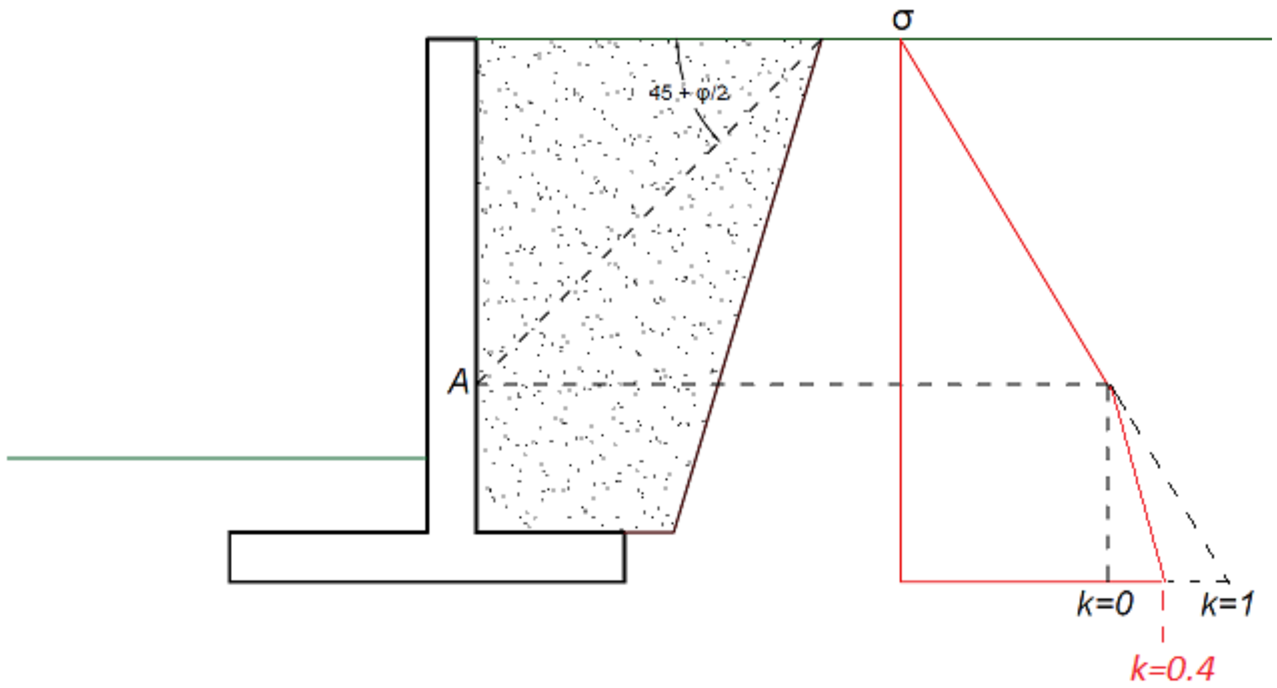
Type 3 - rock behind the wall

The soil assigned to the backfill is selected in the combo list (parameters of soils are defined in the frame "Soils"). Also, the angle of the backfill is input.

For **types 1** and **2,** the angle of the backfill α is input (max 60°). The angle of backfill has an influence only on the global stability analysis when transferring data into the "**Slope Stability**" program. The earth pressures are always calculated for the backfill soil. The original soils (behind the backfill) have no effect on the earth pressures calculation.

Note: The slope of the backfill should be such that the length of the backfill extends beyond the area of failure of the original soil. If this is not the case, the influence of the original soil on the pressure calculation will not be considered, which may lead to skewed results.

For **type 3**, the geometry and the reduction coefficient k are defined. The reduction coefficient k has an effect on earth pressures below the intersection of active pressure wedge and back face of the wall (point **A**). When $k =$ 1, the earth pressure is not reduced. When $k =$ 0, the earth pressure is constant below the intersection. When using this type of backfill, it is necessary to be sure that the soil body behind the backfill has no impact on the wall at all - for example, there is a rock massive. In this case it is not possible to transfer data into the "**Slope Stability**" program.

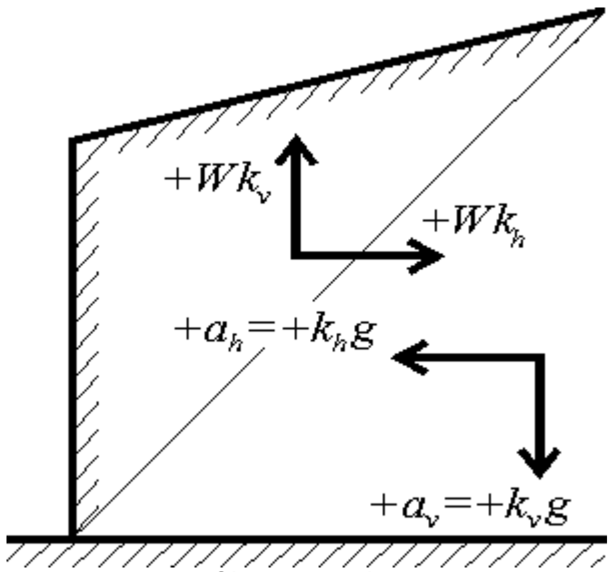


Influence of Earthquake

Earthquake increases the effect of active pressure and reduces the effect of passive pressure. The theories used in our programs (Mononobe-Okabe, Arrango, JTJ 004-89, JTS 146-2012, SL 203-97, NCMA-SRW) are derived assuming cohesionless soils without influence of water. Therefore, all input soils are assumed cohesionless when employing these theories to address the earthquake effects. Earthquake effects due to surcharge are not considered in the program - the user may introduce these effects (depending on the type of surcharge) as "**Applied forces**".

The coefficient k_h is assumed always positive and such that its effect is always unfavorable. The coefficient k_v may receive both positive and negative value. If the equivalent acceleration a_v acts downwards (from the ground surface) the inertia forces $k_v W_s$ will be exerted on the soil wedge in the opposite direction (lifting the wedge up). The values of equivalent acceleration a_v (and thus also the coefficient k_v) and inertia forces $k_v W_s$ are assumed as positive. It is clearly evident that the inertia forces act in the direction opposite to acceleration (if the acceleration is assumed upwards - $a_v = -k_v g$ then the inertia force presses the soil wedge downwards: $-k_v W_s$. The direction with most unfavorable effects on a structure is assumed when examining the seismic effects.

For sheeting structures it is possible to neglect the effect of vertical equivalent acceleration $k_v W_s$ and input $k_v = 0$.



Sign convention

The seismic angle of inertia is determined from the coefficients k_h and k_v (i.e. angle between the resultant of inertia forces and the vertical line) using the formula:

$$\psi = tan^{-1}\left(\frac{k_h}{1-k_v}\right)$$

where: k_v - seismic coefficient of vertical acceleration

k_h - seismic coefficient of horizontal acceleration

Pressure from seismic effects

Increment of active earth pressure due to seismic effects (computed from the structure bottom) follows from:

$$\sigma_{ae,i} = \sigma_{0,i}\left(K_{ae,i} - K_{a,i}\right)$$

$$\sigma_{0,i} = \sum_0^H \gamma_i . h_i \left(1 - k_v\right)$$

where: γ_i - unit weight of soil in the i^{th} layer

$K_{ae,i}$ - coefficient of active earth pressure (static and seismic) in the i^{th} layer

$K_{a,i}$ - magnitude of earth pressure in the i^{th} layer due to Coulomb

h_i - thickness of the i^{th} layer

k_v - seismic coefficient of vertical acceleration

Reduction of passive pressure due to seismic load (computed from the structure bottom) is provided by:

$$\sigma_{pe,i} = \sigma_{0,i}\left(K_{p,i} - K_{pe,i}\right)$$

$$\sigma_{0,i} = \sum_0^H \gamma_i . h_i \left(1 - k_v\right)$$

where: γ_i - unit weight of soil in the i^{th} layer

$K_{pe,i}$ - coefficient of passive earth pressure (static and seismic) in the i^{th} layer

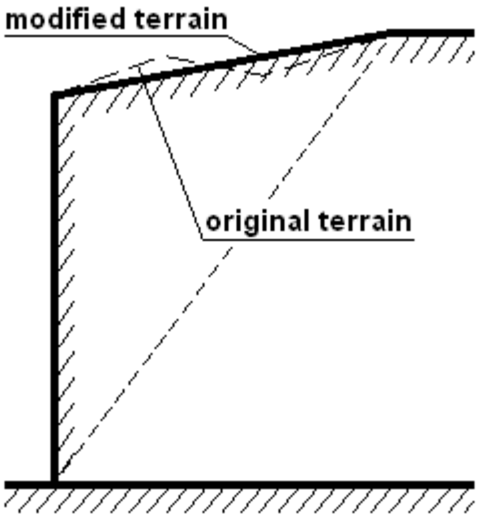
$K_{p,i}$ - magnitude of earth pressure in the i^{th} layer due to Coulomb

h_i - thickness of the i^{th} layer

k_v - seismic coefficient of vertical acceleration

Active earth pressure coefficient $K_{ae,i}$ and passive earth pressure coefficient $K_{pe,i}$ are computed using the Mononobe-Okabe theory or the Arrango theory. If there is groundwater in the soil body the program takes that into account.

The basic assumption in the program when computing earthquake is a flat ground surface behind the structure with inclination β . If that is not the case the program approximates the shape of terrain by a flat surface as evident from figure:



Terrain shape approximation

Point of application of resultant force

The resultant force is automatically positioned by the program into the center of the stress diagram. Various theories recommend, however, different locations of the resultant force - owing to that it is possible to select the point of application of the resultant force in the range of 0.33 - 0.7H (H is the structure height). Recommended (implicit) value is 0.66H. Having the resultant force the program determines the trapezoidal shape of stress keeping both the input point of application of the resultant force and its magnitude.

Mononobe-Okabe Theory

The coefficient K_{ae} for active earth pressure is given by:

$$K_{ae} = \frac{\cos^2(\varphi - \psi - \alpha)}{\cos\psi \cos^2\alpha \cos(\psi + \alpha + \delta)\left(1 + \sqrt{\frac{\sin(\varphi+\delta)\sin(\varphi-\psi-\beta)}{\cos(\delta+\psi+\alpha)\cos(-\beta+\alpha)}}\right)^2}$$

The coefficient K_{pe} for passive earth pressure is given by:

$$K_{pe} = \frac{\cos^2(\varphi - \psi + \alpha)}{\cos\psi \cos^2\alpha \cos(\psi - \alpha + \delta)\left(1 - \sqrt{\frac{\sin(\varphi+\delta)\sin(\varphi-\psi+\beta)}{\cos(\delta+\psi-\alpha)\cos(\beta-\alpha)}}\right)^2}$$

where:
- γ - unit weight of soil
- H - height of the structure
- φ - angle of internal friction of soil
- δ - angle of friction between structure and soil
- α - back face inclination of the structure
- β - slope inclination
- k_v - seismic coefficient of vertical acceleration
- k_h - seismic coefficient of horizontal acceleration
- ψ - seismic inertia angle

Deviation of seismic forces ψ must be for active earth pressure always less or equal to the difference of the angle of internal friction and the ground surface inclination (i.e. $\varphi - \beta$). If the values ψ, are greater the program assumes the value $\psi = \varphi - \beta$. In case of passive earth pressure the value of deviation of seismic forces ψ must be always less or equal to the sum of the angle of internal friction and the ground surface inclination (i.e. $\varphi + \beta$). The values of computed and modified angle ψ can be visualized in the output - in latter case the word **MODIFIED** is also displayed.

Analysis of earthquake effects (active earth pressure) - partial results

Layer No.	thick. [m]	ϕ_d [°]	ψ [°]	K_a	K_{ae}	K_{ae}-K_a	Comment.
1	0.55	29.00	75.96	0.742	3.880	3.138	
2	0.48	29.00	75.96(17.69)	0.742	3.880	3.138	MODIFIED
3	1.30	29.00	75.96(17.69)	0.742	3.880	3.138	MODIFIED

Example of the program output

Literature:

Mononobe N, Matsuo H 1929, On the determination of earth pressure during earthquakes. In Proc. Of the World Engineering Conf., Vol. 9, str. 176.

Okabe S., 1926 General theory of earth pressure. Journal of the Japanese Society of Civil Engineers, Tokyo, Japan 12 (1).

Arrango Theory

The program follows the Coulomb theory to compute the values of K_a and K_p while taking into account the dynamic values (α^*, β^*).

For active earth pressure:

$$\beta^* = \beta + \psi$$
$$\alpha^* = \alpha + \psi$$

For passive earth pressure:

$$\beta^* = \beta - \psi$$
$$\alpha^* = \alpha - \psi$$

where: β - slope inclination

α - back face inclination of the structure

ψ - seismic forces inclination

The coefficients of earth pressures K_{ae} and K_{pe} are found by multiplying the coefficients F_{ae} and F_{pe} by the values of K_a and K_p, respectively.

$$F_{ae} = \frac{cos^2(\alpha + \psi)}{cos\,\psi . cos^2\,\alpha}$$

$$F_{pe} = \frac{cos^2(\alpha - \psi)}{cos\,\psi . cos^2\,\alpha}$$

where: α - back face inclination of the structure

ψ - seismic forces inclination

If the value of the angle β^* becomes larger than φ the program assumes the value ($\beta^* = \varphi$). The values of computed and modified angle β^* can be visualized in the output - in the latter case, the word **MODIFIED** is also displayed. It is the user's responsibility to check in such case whether the obtained results are realistic.

Analysis of earthquake effects (active earth pressure) - partial results

Layer No.	thick. [m]	ϕ_d [°]	β [°]	ψ [°]	K_a	K_{ae}	K_{ae}-K_a	Comment.
1	0.30	26.50	71.43(26.50)	57.99	0.427	3.973	3.546	MODIFIED
2	0.80	26.50	71.43(26.50)	57.99	0.427	3.973	3.546	MODIFIED
3	0.40	26.50	71.43(26.50)	57.99	0.427	3.973	3.546	MODIFIED

Example of the program output

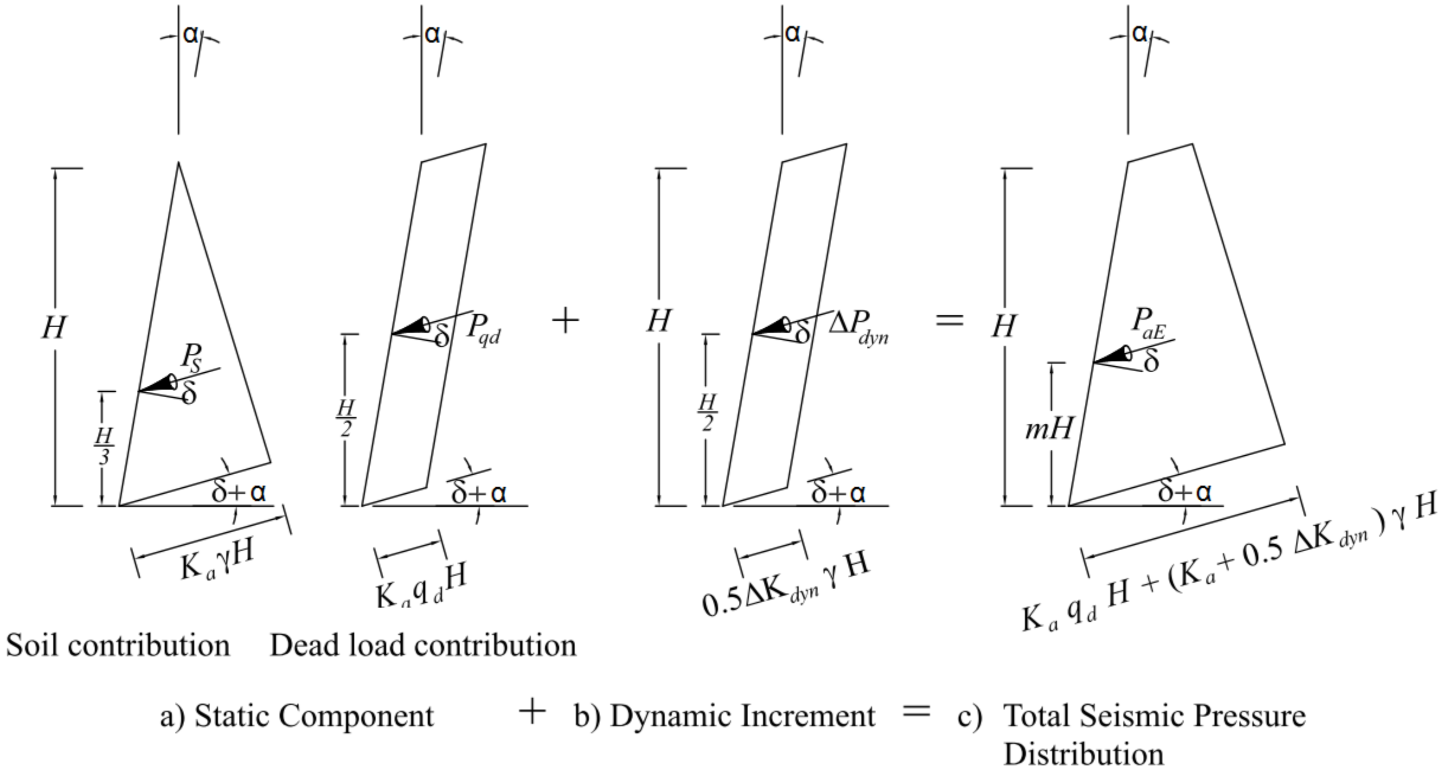
Literature:

Design of sheet pile walls, Pile Buck Inc., Vero Beach, Florida, www.pilebuck.com.

NCMA - SRW

The calculation of dynamic forces is adopted from the *NCMA Design Manual for Segmental Retaining Walls 3rd Edition*.

It is based on Mononobe-Okabe theory, but dynamic effect of active earth pressure acting on the wall is redistributed uniformly along the height of wall (reinforced soil body). The next difference in this method is reduction of length of reinforced soil by half. The horizontal seismic force is then calculated from this reduced length.

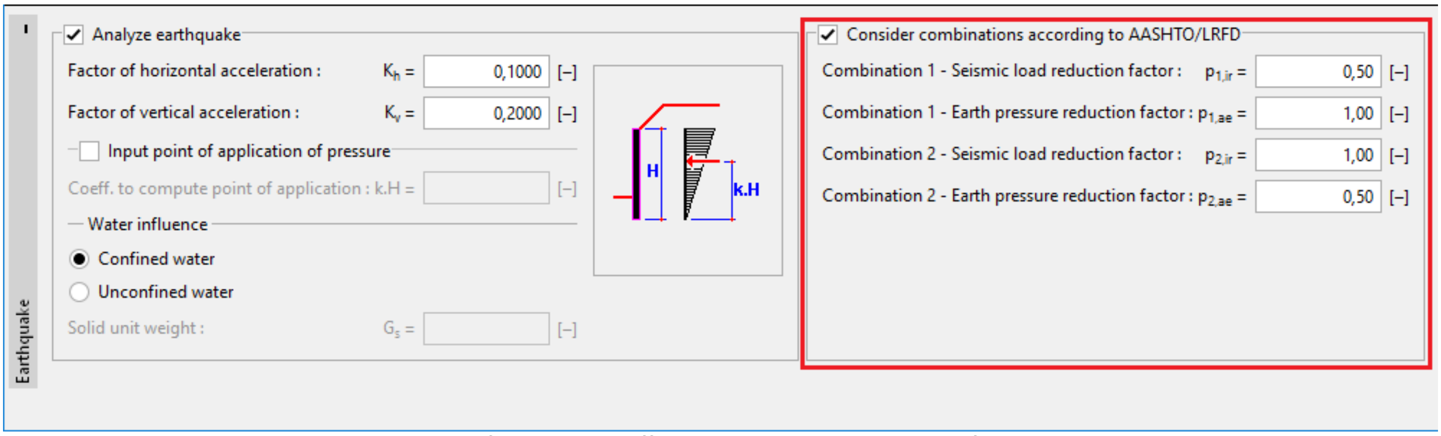


Seismic Combinations (AASHTO)

For the LRFD Verification methodology and earthquake analysis according to the Mononobe-Okabe, Arrango or NCMA -SRW, it is possible to **define coefficients for two seismic combinations in the "Earthquake" frame**.

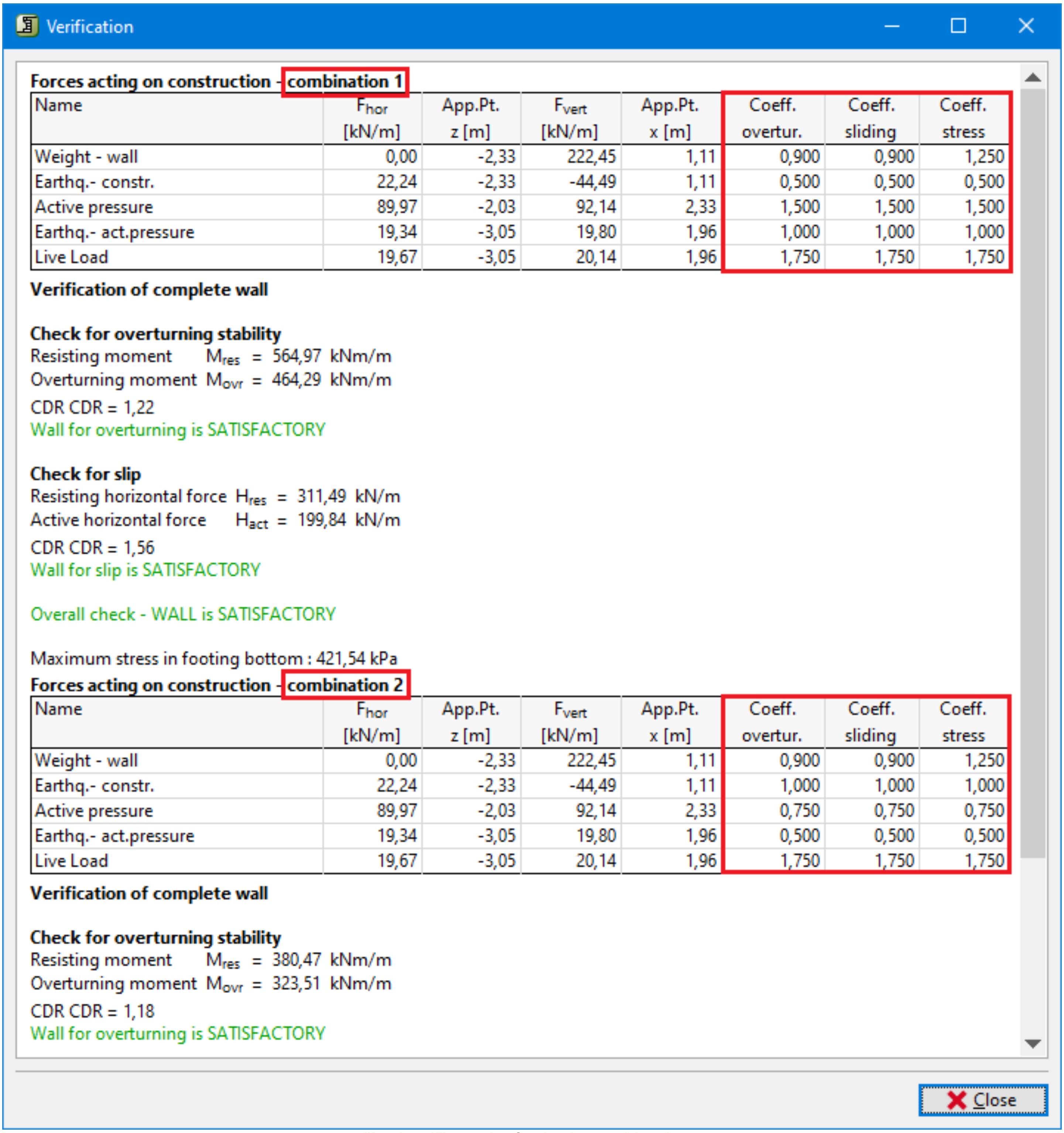
Coefficients P_{ir} reduce seismic effects of the structure.

Coefficients P_{ae} reduce earth pressures.



Input of seismic coefficients in the "Earthquake" frame

The program performs the **analysis for both combinations** and displays the worst overturning/slip results. Information about used coefficients can be found in the "In detail" results in each program.



Verification

Forces acting on construction - combination 1

Name	F_{hor} [kN/m]	App.Pt. z [m]	F_{vert} [kN/m]	App.Pt. x [m]	Coeff. overtur.	Coeff. sliding	Coeff. stress
Weight - wall	0,00	-2,33	222,45	1,11	0,900	0,900	1,250
Earthq.- constr.	22,24	-2,33	-44,49	1,11	0,500	0,500	0,500
Active pressure	89,97	-2,03	92,14	2,33	1,500	1,500	1,500
Earthq.- act.pressure	19,34	-3,05	19,80	1,96	1,000	1,000	1,000
Live Load	19,67	-3,05	20,14	1,96	1,750	1,750	1,750

Verification of complete wall

Check for overturning stability

Resisting moment M_{res} = 564,97 kNm/m

Overturning moment M_{ovr} = 464,29 kNm/m

CDR CDR = 1,22

Wall for overturning is SATISFACTORY

Check for slip

Resisting horizontal force H_{res} = 311,49 kN/m

Active horizontal force H_{act} = 199,84 kN/m

CDR CDR = 1,56

Wall for slip is SATISFACTORY

Overall check - WALL is SATISFACTORY

Maximum stress in footing bottom : 421,54 kPa

Forces acting on construction - combination 2

Name	F_{hor} [kN/m]	App.Pt. z [m]	F_{vert} [kN/m]	App.Pt. x [m]	Coeff. overtur.	Coeff. sliding	Coeff. stress
Weight - wall	0,00	-2,33	222,45	1,11	0,900	0,900	1,250
Earthq.- constr.	22,24	-2,33	-44,49	1,11	1,000	1,000	1,000
Active pressure	89,97	-2,03	92,14	2,33	0,750	0,750	0,750
Earthq.- act.pressure	19,34	-3,05	19,80	1,96	0,500	0,500	0,500
Live Load	19,67	-3,05	20,14	1,96	1,750	1,750	1,750

Verification of complete wall

Check for overturning stability

Resisting moment M_{res} = 380,47 kNm/m

Overturning moment M_{ovr} = 323,51 kNm/m

CDR CDR = 1,18

Wall for overturning is SATISFACTORY

Close

"In detail" results for both combinations

Literature:

AASHTO LRFD Bridge Design Specifications. Eighth Edition, November 2017 (chapter 11-24).

Influence of Water

When examining the influence of groundwater on the magnitudes of earth pressure the program differentiates between confined and unconfined water. Hydrodynamic pressure acting on the front face of the wall is calculated, if the wall is flooded at the front face side.

Confined water

This type is used in soils with lower permeability - app. below the value of $k = 1x10^{-3}\ cm/s$. In such soils the water flow is influenced, e.g. by actual grains (by their shape and roughness) or by resistance of fraction of adhesive water. General formulas proposed by Mononobe-Okabe or Arrango are used to analyze seismic effects. The only difference appears in substituting the value of the seismic angle ψ by ψ^* :

$$\psi^* = tan^{-1}\left(\frac{\gamma_{sat}.k_h}{\gamma_{su}.(1-k_v)}\right)$$

where: γ_{sat} - unit weight of fully saturated soil

γ_{su} - unit weight of submerged soil
k_h - seismic coefficient of horizontal acceleration
k_v - seismic coefficient of vertical acceleration

Unconfined water

This type is used in soils with higher permeability - app. above the value of $k > 1x10^{-1} cm/s$. In such soils it is assumed that water flow in pores is more or less independent of soil grains (e.g. turbulent flow in coarse grain soils). General formulas proposed by Mononobe-Okabe or Arrango are used to analyze seismic effects. The only difference appears in substituting the value of the seismic angle ψ_e by $\psi_e{}^+$:

$$\psi^+ = tan^{-1}\left(\frac{k_{he}{}^+}{1-k_v}\right)$$

$$k_{he}{}^+ = \frac{\gamma_d}{\gamma_{su}}.k_h = \frac{G_s}{G_s - 1}.k_h$$

where: γ_d - unit weight of dry soil

γ_{su} - unit weight of submerged soil
k_h - seismic coefficient of horizontal acceleration
k_v - seismic coefficient of vertical acceleration
G_S - specific gravity of soil particles

$$G_s = \frac{\rho_s}{\rho_w}$$

where: ρ_S - density of the soil solids
ρ_w - density of water

Apart from dynamic pressure the structure is also loaded by hydrodynamic pressure caused by free water manifested by dynamic pressure applied to the structures. The actual parabolic distribution is in the program approximated by the trapezoidal distribution.

The resultant of hydrodynamic pressure **behind the structure** P_{wd} is distant by y_{wd} from the heel of structure:

$$y_{wd} = 0,4.H$$

where: H - height of the structure

and its magnitude follows from:

$$P_{wd} = {}^7\!/_{12}.k_h.\gamma_w.H^2$$

where: γ_w - unit weight of water

k_h - seismic coefficient of horizontal acceleration
H - height of the structure

Hydrodynamic pressure acting on the front face of the wall

The resultant of hydrodynamic pressure on the **front face of the wall** P_{wd} is distant by y_{wd} from the heel of structure:

$$y_{wd} = 0,4.H$$

where: H - height of the structure

and its magnitude follows from:

$$P_{wd} = {}^7\!/_{12}.k_h.\gamma_w.H^2$$

where: γ_w - unit weight of water

k_h - seismic coefficient of horizontal acceleration

H - height of the structure

EN 1998-5 Seismic Effects

If the coefficients k_h and k_v are not obtained from measurements it is necessary, providing the analysis is carried out according to 1998-5 Eurocode 8: Design of structures for earthquake resistance - Part 5: Foundations, retaining structures and geotechnical aspects, to input these coefficients as follows:

$$k_h = \alpha.\frac{S}{R}$$

where:

α - ratio of the design ground acceleration on type A ground (a_g / g)

S - soil factor defined in EN 1998-1:2004, chapter 3.2.2.2

R - factor for the calculation of the horizontal seismic coefficient - see tab.

for: $\frac{a_{vg}}{a_g} > 0{,}6$ $\quad k_v = \pm 0{,}5.k_h$

for other cases: $\quad k_v = \pm 0{,}33.k_h$

Type of sheeting structure	R
Free gravity walls that can accept a displacement up to $d_r =$ 300αS (mm)	2
Free gravity walls that can accept a displacement up to $d_r =$ 200αS (mm)	1.5
Flexural reinforced concrete walls, anchored or braced walls, reinforced concrete walls founded on vertical piles, restrained basement walls and bridge abutments	1

More detailed description can be found in EN 1998-5 chapter 7.3.2.2 Seismic effects

Forces from Earth Pressure at Rest Acting on the Rigid Structure

Earth pressure at rest acts on the rigid structures (e.g. stem of the cantilever wall) in earthquake analysis. Following resultant force of this pressure is taken into account:

$$F = k_h.\gamma.H^2$$

where:

H - height of the structure

k_h - seismic coefficient of horizontal acceleration

γ - unit weight of soil

Resultant force acts in the half of the structure height.

Influence of Earthquake according to Chinese Standards

Three different Chinese standards are implemented for calculation of seismic effect for wall design which are JTJ 004-89 (**Specifications of Earthquake Resistance Design for Highway Engineering**), SL 203-97 (**Specification for seismic design of hydraulic structures**), JTS 146-2012 (**Code for seismic Design of Water Transport Engineering**). They are all based on Mononobe-Okabe theory. The main difference between Chinese standards and Mononobe-Okabe theory is that comprehensive influence factor C_z is introduced in Chinese standards which reduces seismic force by about 70 $\%$.

The advantage of choosing Chinese standards as the option for earthquake analysis is that users only need to choose the intensity of the earthquake according to which the program automatically assign values of other parameters appropriate with standards.

Influence of Earthquake according to JTJ 004-89

Only horizontal seismic force is considered according to JTJ 004-89.

Seismic force on structure

Seismic force acting on structure is provided by (Art. 3.1.5):

$$E_{ihw} = C_i C_z K_h \psi_{iw} G_{iw}$$

where:

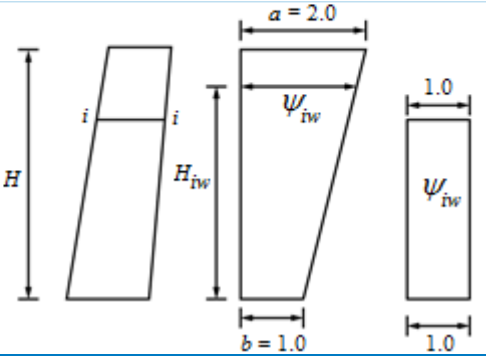
E_{ihw} - seismic force acting at the center of gravity of the wall above the i^{th} cross-section [kN/m]

K_h - coefficient of horizontal seismic acceleration

G_{iw} - weight of the structure above the ith cross-section [kN/m]

C_z	-	comprehensive influence factor, usually it's 0.25
C_i	-	importance coefficient for seismic design
ψ_{iw}	-	distribution coefficient of horizontal earthquake along the wall

Recommended value of distribution coefficient ψ_{iw} (Tbl. 3.1.5):

Wall Height $[m]$	**Security level**		**Calculation diagram for** ψ_{iw}
	Highway, A class and B class motorway	C class and D class motorway	
$H \leq$ 12	$\psi_{iw} =$ 1	$\psi_{iw} =$ 1	
$H >$ 12	$\psi_{iw} = 1 + \frac{H_{iw}}{H}$	$\psi_{iw} =$ 1	

ψ_{iw} isn't considered when $H \leq$ 12 m which means parameter a and b don't work when $H \leq$ 12 m. a is the top value of the distribution map and b is the bottom value of the distribution map.

Seismic earth pressure

When computing seismic earth pressure, Coulomb theory is used and unite weight of soil γ, internal friction angle of soil φ and angle of friction structure-soil δ is replaced by γ / cos θ, sdasdasd $\varphi - \theta$, $\delta + \theta$, where θ is a seismic angle (Art. 3.1.6).

Seismic angel θ is determined by different option of seismic fortification intensity.

Water influence

Water influence according to Chinese standard is a little different from the water influence according to Mononobe-Okabe or Arrango theory by reducing the water influence using comprehensive influence factor C_z.

Seismic bearing capacity of subsoil

Seismic bearing capacity of subsoil is provided by (Art. 2.2.1):

$$f_{aE} = \zeta_a f_a$$

where:	f_{aE}	-	seismic bearing capacity of subsoil
	ξ_a	-	adjusting coefficient for seismic bearing capacity
	f_a	-	characteristic value of bearing capacity which has been modified by the geometry of foundation

The above formula is as same as Art. 4.2.3 in GB 50011-2010 (Code for seismic design of buildings). Suggested values of ξ_a by different standards can be found here.

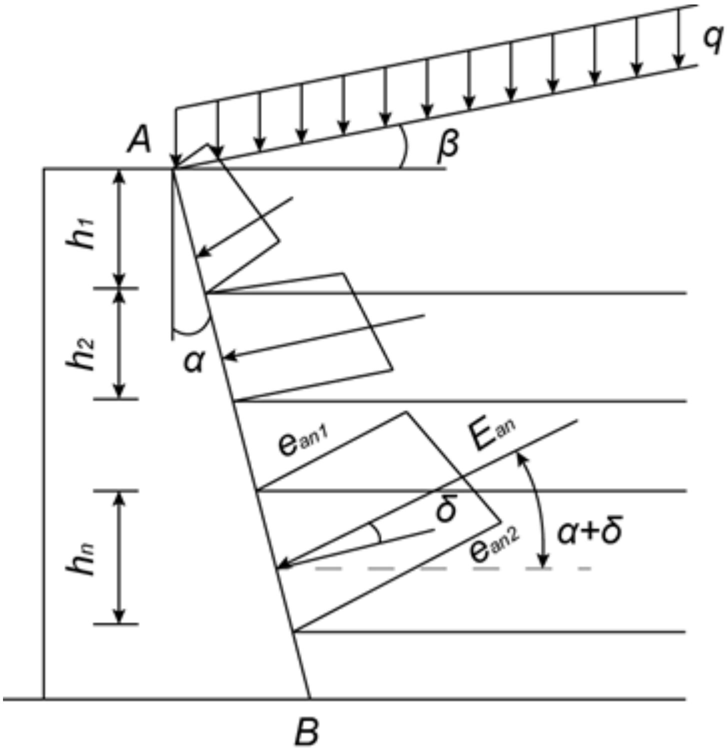
Influence of Earthquake according to JTS 146-2012

Only horizontal seismic force is considered according to JTS 146-2012.

Seismic force on structure

Calculation of seismic force acting on structure is as same as JTJ 004-89. Values of Cz can be set from 0.2 to 0.5 suggested by Art. 5.2 in JTS 146-2012.

Seismic earth pressure



Distribution of seismic active pressure (Fig. 5.3.1)

The only difference between JTS 146-2012 and JTJ 004-89 is that seismic earth pressure according to JTS 146-2012 considers the influence of cohesion.

Active seismic earth pressure is provided by (Art. 5.3.1):

$$E_{an} = \frac{1}{2}\left(e_{an1} + e_{an2}\right)\frac{h_n}{\cos\alpha}$$

where:

$$e_{an1} = \left(K_q q + \sum_{i=0}^{n-1}\gamma_i h_i\right)K_{an}\cos\alpha - 2c_n K_{acn}\cos\alpha$$

$$e_{an2} = \left(K_q q + \sum_{i=0}^{n}\gamma_i h_i\right)K_{an}\cos\alpha - 2c_n K_{acn}\cos\alpha$$

$$K_q = \frac{\cos\alpha}{\cos(\alpha-\beta)}$$

$$K_{an} = \frac{\cos^2(\varphi_n - \alpha - \theta)}{\cos\theta\cos^2\alpha\cos(\delta_n + \theta + \alpha)\left(1 + \sqrt{\dfrac{\sin(\varphi_n + \delta_n)\sin(\varphi_n - \beta - \theta)}{\cos(\delta_n + \theta + \alpha)\cos(\alpha - \beta)}}\right)^2}$$

$$K_{acn} = \frac{\cos(\alpha-\beta)\cos\varphi_n}{\cos\theta\cos\alpha\left[1 + \sin(\varphi_n + \delta_n - \beta + \alpha)\right]}$$

where:

where:	E_{an}	-	overall active pressure acting on n^{th} layer [kN/m]
	e_{an1}	-	active pressure acting on the top of the n^{th} layer [kPa]
	e_{an2}	-	active pressure acting on the bottom of the n^{th} layer [kPa]
	h_n	-	thickness of n^{th} layer [m]
	α	-	back face inclination of the structure [°]
	K_q	-	coefficient
	q	-	uniform load acting on the terrain [kPa]
	γ_i	-	unit weight of ith layer [kN/m^3], below water - buoyant unit weight is accepted
	h_i	-	thickness of the i^{th} layer [m]

K_{an}	-	coefficient of active pressure of n^{th} layer
c_n	-	standard value of cohesion of n^{th} layer [kPa]
K_{acn}	-	coefficient of seismic active pressure of n^{th} layer
β	-	slope inclination [°], and $\lvert\beta\rvert < \varphi$
φ_n	-	internal friction angle of n^{th} layer [°]
θ	-	seismic angel [°]
δ_n	-	angle of friction between structure and soil of n^{th} layer [°]

Passive seismic earth pressure is provided by (Art. 5.3.2):

$$E_{pn} = \frac{1}{2}\left(e_{pn1} + e_{pn2}\right)\frac{h_n}{\cos\alpha}$$

where:

$$e_{pn1} = \left(K_q\, q + \sum_{i=0}^{n-1} \gamma_i\, h_i\right) K_{pn}\cos\alpha + 2c_n K_{pcn}\cos\alpha$$

$$e_{pn2} = \left(K_q\, q + \sum_{i=0}^{n} \gamma_i\, h_i\right) K_{pn}\cos\alpha + 2c_n K_{pcn}\cos\alpha$$

$$K_q = \frac{\cos\alpha}{\cos(\alpha - \beta)}$$

$$K_{pn} = \frac{\cos^2(\varphi_n + \alpha - \theta)}{\cos\theta\cos^2\alpha\cos(\delta_n + \theta - \alpha)\left(1 - \sqrt{\dfrac{\sin(\varphi_n + \delta_n)\sin(\varphi_n + \beta - \theta)}{\cos(\delta_n + \theta - \alpha)\cos(\alpha - \beta)}}\right)^2}$$

$$K_{pcn} = \frac{\cos(\alpha - \beta)\cos\varphi_n}{\cos\theta\cos\alpha\left[1 - \sin(\varphi_n + \delta_n + \beta - \alpha)\right]}$$

Seismic angel θ is determined by different options of seismic fortification intensity.

Water influence

Water influence according to Chinese standard is a little different from the water influence according to Mononobe-Okabe or Arrango theory by reducing the water influence using comprehensive influence factor C_z.

Seismic bearing capacity of subsoil

Calculation of seismic bearing capacity of the subsoil is as same as JTJ 004-89.

Influence of Earthquake according to SL 203-97

Both horizontal and vertical seismic forces can be considered according to SL 203-97. In SL 203-97, seismic angle θ is derived automatically from K_h, so seismic angle θ and seismic angle blow water θ' are not visible in the frame.

Seismic force on structure

Calculation of the horizontal seismic force acting on a structure is the same as JTJ 004-89.

Vertical seismic force acting on structure is provided by (Art. 4.1.8):

$$E_{ivw} = C_0 \frac{E_{ihw}}{K_h} K_v$$

where:			
	E_{ivw}	-	vertical seismic force acting at the center of gravity of the wall above the i^{th} cross-section [kN/m]
	E_{ihw}	-	horizontal seismic force acting at the center of gravity of the wall above the ith cross-section [kN/m]
	k_h	-	coefficient of horizontal seismic acceleration
	k_v	-	coefficient of vertical seismic acceleration, usually, it's ± 2/3K_h (Art. 4.3.2)
	G_{iw}	-	weight of the structure above the i^{th} cross-section [kN/m]
	C_0	-	meeting coefficient related to the influence of horizontal seismic effect, usually, it's 0.5.

Seismic earth pressure

Calculation of seismic earth pressure is as same as JTJ 004-89. The only difference between SL 203-97 and JTJ 004-89 is that SL 203-97 has no "user-defined - input K_h , θ " as an option for seismic fortification intensity.

Water influence

Water influence according to Chinese standard is a little different from the water influence according to Mononobe-Okabe or Arrango theory by reducing the water influence using comprehensive influence factor C_z .

Seismic bearing capacity of subsoil

The calculation of the seismic bearing capacity of a subsoil is the same as JTJ 004-89.

Seismic Fortification Intensity according to Chinese Standards

There are three main options for seismic fortification intensity according to the Chinese standards. Horizontal seismic acceleration coefficient K_h and seismic angle θ are determined according to what option is selected for seismic fortification intensity.

- 7 degrees (0.1g), 7 degrees (0.15g), 8 degrees (0.2g), 8 degrees (0.3g), 9 degrees (0.4g): K_h and seismic angel s are determined according to corresponding seismic fortification intensity based on standards.
- user-defined - input K_h : K_h is input by the user and θ is determined by (Art. 4.9.1 from SL 203-97)

$$\theta = \arctan\left(\frac{C_z C_i K_h}{1 - C_z C_i K_v}\right)$$

- user-defined - input K_h , θ : K_h and θ are both input by the user.

Note: Third option is only valid for JTJ 004-89 and JTS 146-2012.

Values of K_h and θ according to corresponding seismic fortification intensity are given by the following tables:

For JTJ 004-89 (Tbl. 1.0.7 and Tbl. 3.1.6 from JTJ 004-89)

Seismic fortification intensity	7 degree		8 degree		9 degree
Coefficient of horizontal seismic acceleration K_h	0.10	0.15	0.20	0.30	0.40
Seismic angel θ	1.5°	2.3°	3.0°	4.5°	6.0°

Note: 7 degree (0.15g) and 8 degree (0.30g) are not from JTJ 004-89, because there are no value for these two situations in JTJ 004-89. They are from JTS 146-2012.

For JTS 146-2012 (Tbl. 5.3.1 from JTS 146-2012)

Seismic fortification intensity	7 degree		8 degree		9 degree
Coefficient of horizontal seismic acceleration K_h	0.10	0.15	0.20	0.30	0.40
Seismic angel θ	1.5°	2.3°	3.0°	4.5°	6.0°

For SL 203-97 (Tbl. 4.3.1 from SL 203-97)

Seismic fortification intensity	7 degree		8 degree		9 degree
Coefficient of horizontal seismic acceleration K_h	0.10	0.15	0.20	0.30	0.40

Water Influence according to Chinese Standards

Seismic water influence can be dived into two parts - influence on seismic earth pressure and dynamic water pressure. Similar to water influence according to Mononobe-Okabe or Arrango theory, water influence according to Chinese standards also has two types water influence - confined water and unconfined water. The main difference between Monobe-Okabe theory and Chinese standards is that Chinese standards reduce the water influence using comprehensive influence factor C_z .

Confined water

This type is used in soils with lower permeability - app. below the value of $k = 1*10^{-3}$ cm/s . When confined water is chosen, dynamic water pressure is not considered. The only difference from soils without water is that the seismic angle used in the calculation of seismic earth pressure is replaced by a seismic angle below water θ' .

Value of seismic angle below water θ' is determined by the following two options:

1. By seismic fortification intensity - value of θ’ is provided by the following tables:

For JTJ 004-89 (Tbl. 3.1.6 from JTJ 004-89)

Seismic fortification intensity	7 degree		8 degree		9 degree
Coefficient of horizontal seismic acceleration K_h	0.10g	0.15g	0.20g	0.30g	0.40g
Seismic angel θ	2.5°	4.5°	5.0°	9.0°	10.0°

For JTS 146-2012 (Tbl. 5.3.1 from JTS 146-2012)

Seismic fortification intensity	7 degree		8 degree		9 degree
Coefficient of horizontal seismic acceleration K_h	0.10g	0.15g	0.20g	0.30g	0.40g
Seismic angel θ'	3.0°	4.5°	6.0°	9.0°	12.0°

2. Input seismic angle - value of θ' is input by the users. When this option is chosen, the default value of θ' is provided by:

$$\theta' = \arctan\left(\frac{\gamma_{sat} C_z C_i K_h}{\gamma_{sat}\left(1 - C_o C_z C_i K_v\right)}\right)$$

If you have no idea about how to calculate the value of θ', you can use the default.

Note: For SL 203-97, there are no additional options for confined water. θ' is calculated automatically according to the above formula.

Unconfined water

This type is used in soils with higher permeability - app. above the value of $k > 1*10^{-1}$ cm/s . When unconfined water is chosen, both influence on earth pressure and dynamic water pressure is considered.

Value of seismic angle below water θ' is determined by the following three options:

1. By seismic fortification intensity - same to confined water.
2. Input seismic angle - same to confined water.
3. Input specific gravity of soil particles - the value of θ' is provided by:

$$\theta' = \arctan\left(\frac{K_{hs}^{+}}{1 - C_o C_z C_i K_v}\right)$$

where:

$$K_{hs}^{+} = \frac{\gamma_d}{\gamma_{su}} C_z C_i K_h = \frac{G_s}{G_s - 1} C_z C_i K_h$$

Note: For SL 203-97, there are no additional options for unconfined water. θ' is calculated automatically according to the above formula.

Dynamic water pressure is calculated according to the standard chosen.

For JTJ 004-89 (Art. 4.2.11 from JTJ 004-89):

$$E_w = 0.24\, C_i\, K_h\, \gamma_w d^2$$

where:
- E_w - the overall dynamic water pressure acting on the structure [kN]
- C_i - importance coefficient for seismic design
- γ_w - unit weight of water [kN/m^3]
- d - depth of water above the heel of the structure [m]

Distribution of dynamic water pressure is constant along the structure.

For JTS 146-2012 (Art. 5.4.1 from JTS 146-2012):

$$p_z = \frac{7}{8} \eta\, C_i C_z\, K_h\, \gamma_w d^{\frac{1}{2}} Z^{\frac{1}{2}}$$

where:

$$\eta = th\frac{\pi b}{4d}$$

where:
- p_z - dynamic water pressure at depth Z [kPa]
- η - reduction factor, for walls, it equals to 1.0
- Z - distance between calculation point and water table [m]

d - depth of water above the heel of the structure $[m]$

b - width of water table $[m]$

For SL 203-97 (Art. 6.1.9 from SL 203-97):

$$p_z = C_z C_i K_h \gamma_w d \, \psi(Z)$$

where: p_z - dynamic water pressure at depth Z $[kPa]$

d - depth of water above the heel of the structure $[m]$

$\psi(Z)$ - distribution coefficient of dynamic water pressure at depth Z

Value of $\psi(Z)$ is provide by the following table (Tbl. 6.1.9 from SL 203-97):

Z/d	$\psi(Z)$	Z/d	$\psi(Z)$
0.0	0.00	0.6	0.76
0.1	0.43	0.7	0.75
0.2	0.58	0.8	0.71
0.3	0.68	0.9	0.68
0.4	0.74	1.0	0.67
0.5	0.76	-	-

Importance Coefficient for Seismic Design Ci

Values of importance coefficient for seismic design C_i (Tbl. 1.0.4 from JTJ 004-89):

The importance of the motorway	Important coefficient for seismic design C_i
Important Highway and A class motorway	1.7
Highway and A class motorway or important B class motorway	1.3
B class motorway or important C class motorway	1.0
C class motorway or important D class motorway	0.6

Values of importance coefficient for seismic design Ci (Tbl. 1.0.4 from JTG TB02-01-2008):

Importance of the bridge	E1 seismic effect	E2 seismic effect
A class	1.0	1.7
B class	0.43 (0.5)	1.3 (1.7)
C class	0.34	1.0
D class	0.23	-

Adjusting Coefficient for Seismic Bearing Capacity ξa

Values of adjusting coefficient for seismic bearing capacity ξ_a (Tbl. 4.2.3 from GB 50011-2010):

Name and property of the subsoil	ξ_a
Rock, dense gravelly soil, dense gravel and coarse and medium-coarse sand, clay and silt whose $f_{ak} \geq$ 300 kPa	1.5
Medium and moderate dense gravelly soil, medium dense and moderate dense gravel and coarse and medium-coarse sand, dense and medium dense fine and silty sand, clay, silt and firm loess whose 300 $kPa \leq f_{ak} <$ 300 kPa	1.3
Moderate dense fine and silty sand, clay, silt and plastic loess whose 100 $kPa \leq f_{ak} <$ 150 kPa	1.1
Mud, muddy soil, loose sand, miscellaneous fill, newly accumulated loess, and soft loess	1.0

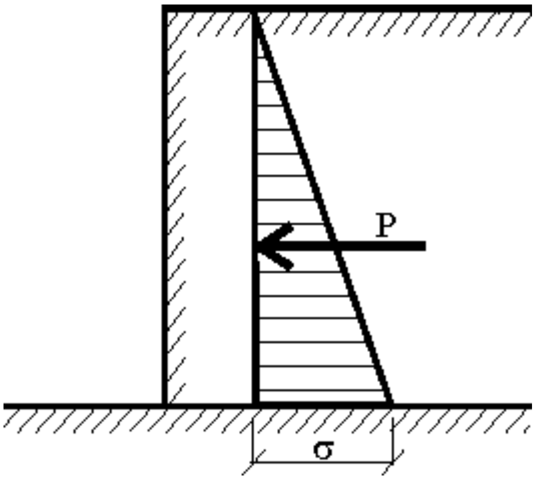
Note: In JTJ 004-89, it's table 2.2.1 which is similar to the above table.

Values of adjusting coefficient for seismic bearing capacity ξ_a (Tbl. 5.5.1 from JTS 146-2012):

Subsoil	ξ_a
Loose sand, not in liquidation status	1.0
Normal sand soil, not in liquidation status	1.3
Dense gravelly soil and bedrock	1.5

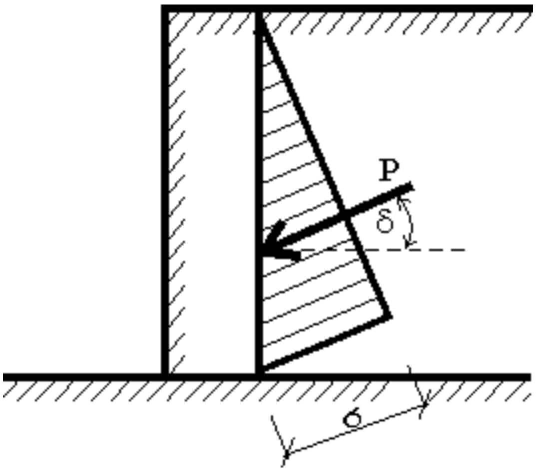
Influence of Friction between Soil and back of the Structure

The magnitude of active or passive earth pressure, respectively, depends not only on the selected solution theory but also on friction between the soil and the back of the wall and by the adhesion of soil to the structure face represented by the angle δ. If $\delta =$ 0 then the pressure σ acts in the direction normal to the back of the wall and the resultant of earth pressure P is also directed in normal to the back of the wall - see figure:



Distribution of earth pressure along the structure for $\delta = 0$

Providing the friction between the soil and the back of the wall is considered in the analysis of earth pressures, the earth pressure σ and also its resultant P are inclined from the back of the wall by the angle δ. Orientation of friction angles δ from normal to the back of the wall must be introduced in accord with the mutual movement of the structure and soil. With increasing value of δ the value of active earth pressure decreases, i.e. the resultant force of active earth pressure deviates from the normal direction - see figure:



Distribution of earth pressure along structure for $\delta \neq 0$

The magnitude δ can be usually found in the range of $\delta \leq 1/3\varphi$ to $\delta = 2/3\varphi$. The values of the orientation of the friction angle δ between the soil and the structure are stored in the table of values of δ for various interfaces and in the table of recommended values for $|\delta| / \varphi$. The value of $\delta \leq 1/3\varphi$ can be used if assuming smooth treatment of the back of sheeting structure (foil and coating against groundwater). For an untreated face, it is not reasonable to exceed the value of $\delta = 2/3\varphi$. When selecting the value of δ it is necessary to reflect also other conditions, particularly the force equation of equilibrium in the vertical direction. One should decide whether the structure is capable of transmitting the vertical surcharge due to friction on its back without excessive vertical deformation. Otherwise, it is necessary to reduce the value of δ, since only the partial mobilization of the friction on the back of the wall may occur. In case of an uncertainty it is always safer to assume smaller vale of δ.

Table of Ultimate Friction Factors for Dissimilar Materials

Values of the angle δ for different interfaces (according to the NAVFAC standards)

Interface material	**Friction factor** $tg(\delta)$	**Friction angle** δ [°]
Mass concrete on the following foundation materials:		
Clean sound rock	0.70	35
Clean gravel, gravel-sand mixtures, coarse sand	0.55 - 0.6	29 - 31
Clean fine to medium sand, silty medium to coarse sand, silty or clayey gravel	0.45 - 0.55	
Clean fine sand, silty or clayey fine to medium sand	0.35 - 0,45	19 - 24
Fine sandy silt, nonplastic silt	0.30 - 0.30	17 - 19
Very stiff and hard residual or preconsolidated clay	0.40 - 0.50	22 - 26

Medium stiff and stiff clay and silty clay	0.30 - 0.35	17 - 19
Steel sheet piles against the following soils:		
Clean gravel, gravel-sand mixtures, well-graded rock fill with spalls	0.40	22
Clean sand, silty sand-gravel mixture, single size hard rock fill	0.30	17
Silty sand, gravel or sand mixed with silt or clay	0.25	14
Fine sandy silt, nonplastic silt	0.20	11
Formed concrete or concrete sheet piling against the following soils:		
Clean gravel, gravel-sand mixture, well-graded rock fill with spalls	0.40 - 0.50	22 - 26
Clean sand, silty sand-gravel mixture, single size hard rock fill	0.30 - 0.40	17 - 22
Silty sand, gravel or sand mixed with silt or clay	0.30	17
Fine sandy silt, nonplastic silt	0.25	14
Various structural materials:		
Dressed soft rock on dressed soft rock	0.70	35
Dressed hard rock on dressed soft rock	0.65	33
Dressed hard rock on dressed hard rock	0.55	29
Masonry on wood (Gross grain)	0.50	26
Steel on steel at sheet pile interlocks	0.30	17

Adhesion of Soil

When performing the analysis in the total stress state for active or passive earth pressure it is necessary to consider the total (undrained) shear strength of soil c_u and the adhesion a of soil to the structure face. The value of adhesion a is usually considered as a fraction of the soil cohesion c. The typical values of a for a given range of the cohesion c are listed in the following table.

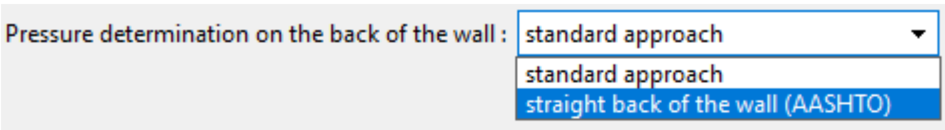
Common values of the adhesion of soil a

Soil	Cohesion c $[kPa]$	Adhesion a $[kPa]$
Soft and very soft cohesive soil	0 - 12	0 - 12
Cohesive soil with medium consistency	12 - 24	12 - 24
Stiff cohesive soil	24 - 48	24 - 36
Hard cohesive soil	48 - 96	36 - 46

Straight Back of the Wall

Standardly, the earth pressure acts on the real shape of the wall. In the case of segmental retaining walls (Gabion, Prefab Wall, Redi-Rock Wall), the earth wedges can be considered.

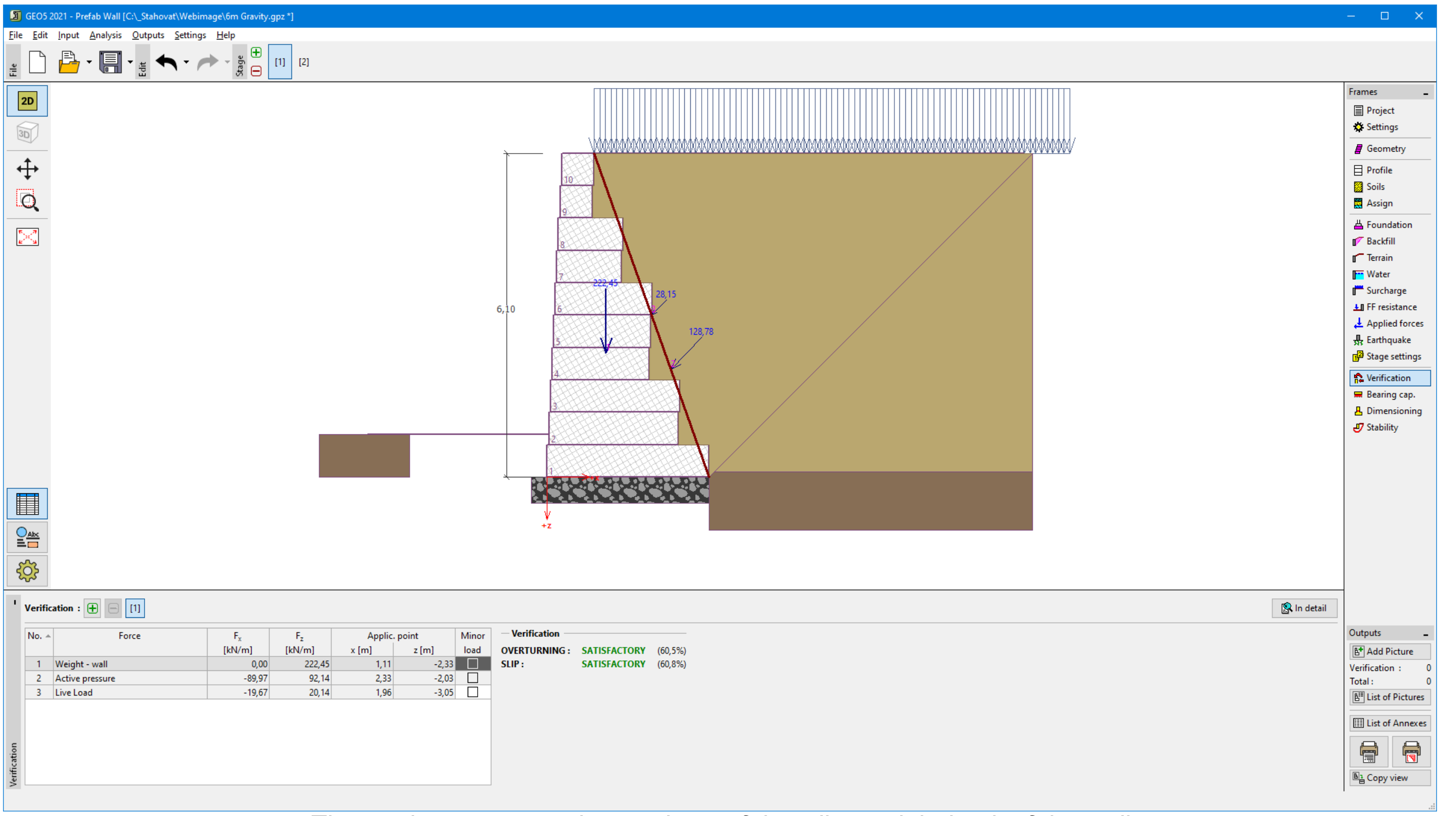
Some standards (e.g. **AASHTO**) require always the **simplified straight back of the wall for pressure analysis** in some specific cases.



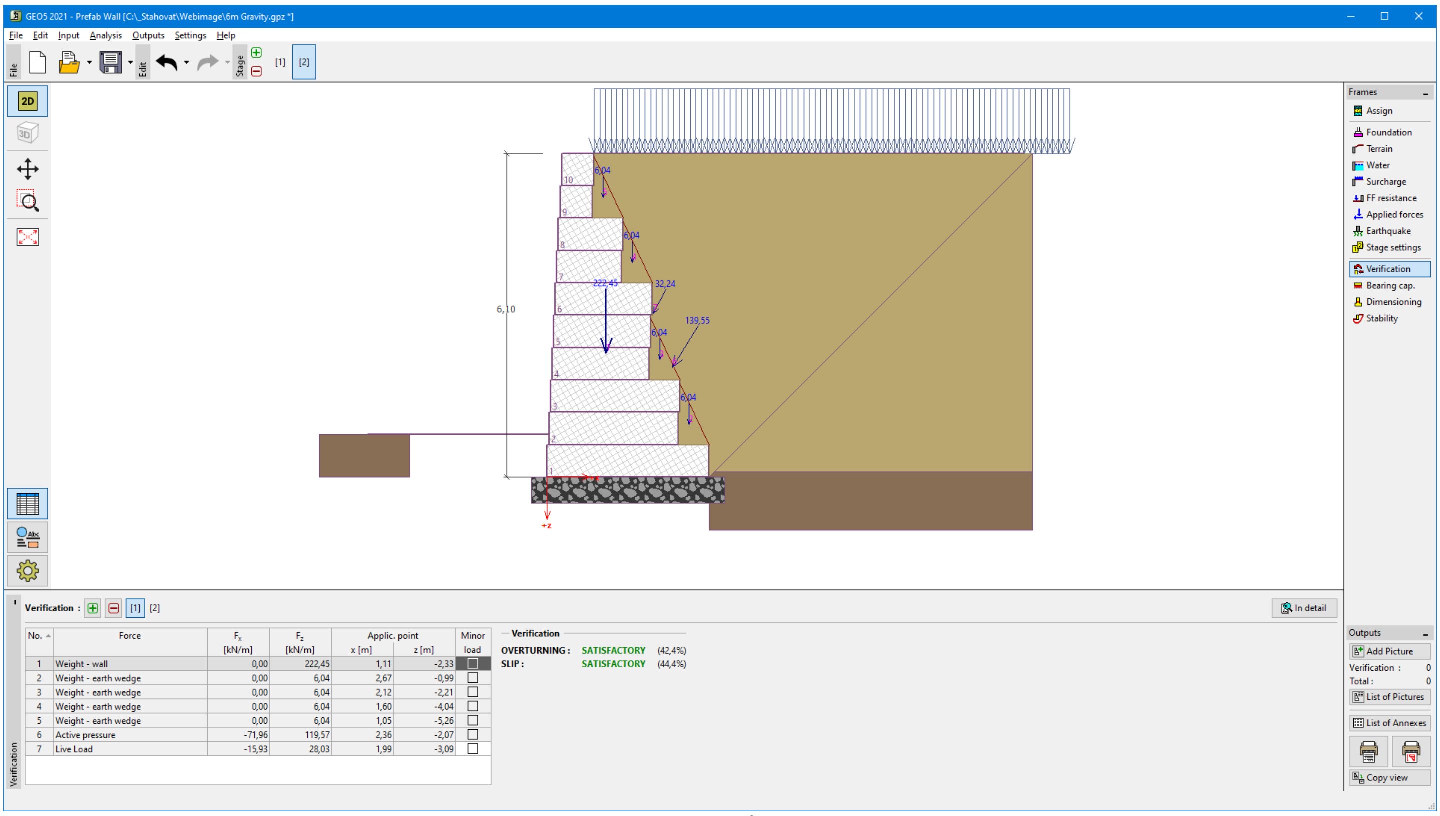
The choice of the back of the wall in the "Stage settings" frame

The back of the wall is then considered as a line between the back points of the upper and bottom block. The earth pressure then acts on this surface under friction angle δ.

The simplified back of the wall is drawn in the "**Verification**" frame.



The earth pressure acting on the prefab wall - straight back of the wall



The earth pressure acting on the prefab wall - the standard approach

Literature:

U.S. Department of Transportation, Publication No. FHWA NHI-07-071, Federal Highway Administration, June 2008.

Analysis of Walls

Verification analysis of walls can be performed using:

- the theory of the limit state (when performing the analysis according to **EN 1997** or **LRFD** the structure is verified in this particular way)

- the safety factor

In addition, the bearing capacity of foundation soil is examined for both cases.

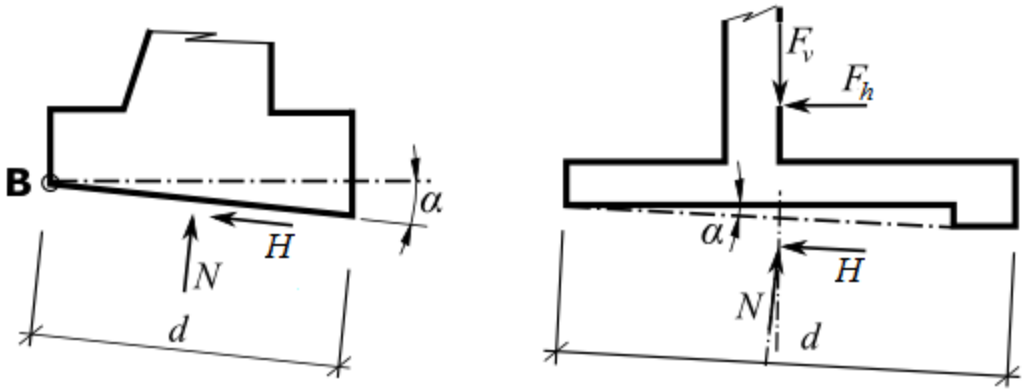
Following forces are used in the verification:

- **weight of wall** - depends on the shape and unit weight of wall (for input use the "**Material**" dialog window) - uplift pressure is introduced for walls found below the groundwater table
- **resistance on front face** - when inputting the resistance on the front face the corresponding force acts as the pressure at rest, or passive pressure or reduced passive pressure
- gravity forces of earth wedges - an arbitrary number of these forces may occur depending on the shape of the structure
- active earth pressure or pressure at rest acting on the structure - the basic load of the structure due to earth pressures - depending on the selected option in the frame "**Settings**" the pressure is computed either with or without reduction of input soil parameters
- forces due to water effects or pore pressure, respectively
- forces due to a surcharge - a single force corresponds to each input surcharge. If the magnitude of force due to surcharge is equal to zero (the surcharge has no effect on a structure) then it does not appear in the picture but only in the table listing
- **Applied forces** - forces entering the analysis are displayed
- **forces due to an earthquake** - several forces enter the analysis due to an earthquake - the increase of earth pressure acting on a structure, reduction of passive pressure on the front face of a structure, or force due to free water behind the structure
- mesh overhangs and reinforcements are displayed and included providing they appear in the analysis
- base anchorage of walls

Evaluation of Forces in the Footing Bottom

After computing forces acting on the structure the program determines the overall vertical F_v and horizontal F_h forces, computes the forces acting in the footing bottom (normal force N and shear force H):

$$N = F_v \cos\alpha + F_h \sin\alpha$$
$$H = F_v \sin\alpha + F_h \cos\alpha$$



Forces acting in the footing bottom

Verification - Limit States

The program evaluates normal and shear force in the footing bottom and then verifies the wall against overturning and sliding. For walls with a flat footing bottom and specified jump it is possible to account for the wall jump either in the form of pressure acting on the front face or by considering a wall with an inclined footing bottom.

Check for overturning stability:

$$\frac{M_{res}}{\gamma_o} > M_{ovr}$$

where: M_{ovr} - overturning moment

γ_o - reduction coefficient of overturning

M_{res} - resisting moment

Check for slip:

$$\frac{\left[\left(N\tan\varphi_d + c_d(d-2e)/\mu\right)+F_{res}\right]}{\gamma_s} > H$$

where:

N	-	normal force acting in the footing bottom
φ_d	-	design angle of internal friction of the soil
c_d	-	design cohesion of the soil
d	-	width of the wall heel
e	-	eccentricity
γ_s	-	reduction coefficient of the sliding resistance
H	-	shear force acting in the footing bottom
F_{res}	-	resisting force (from georeinforcement and mesh overlap)
μ	-	reduction coefficient of the contact base - soil

where eccentricity e :

$$e = \frac{M_{ovr} - M_{res} + \frac{Nd}{2}}{N}$$

where:

M_{ovr}	-	overturning moment
M_{res}	-	resisting moment
N	-	normal force acting in the footing bottom
d	-	width of wall heel

Horizontal components of forces are included in the shear force and overturning moment, vertical components of forces are included in the normal force and resisting moment. The resisting forces and moments also include horizontal forces from georeinforcements and overlapping meshes. The uplift pressure is considered in the overturning moment.

Verification - Safety Factor

The program evaluates normal and shear force in the footing bottom and then verifies the wall against overturning and sliding. For walls with a flat footing bottom and specified jump it is possible to account for the wall jump either in the form of pressure acting on the front face or by considering a wall with an inclined footing bottom.

Check for overturning stability:

$$\frac{M_{res}}{M_{ovr}} > SF_o$$

where:

M_{ovr}	-	overturning moment
M_{res}	-	resisting moment
SF_o	-	safety factor for overturning

Check for slip:

$$\frac{\left[\left(N\tan\varphi_d + c_d(d-2e)/\mu\right)+F_{res}\right]}{H} > SF_s$$

where:

N	-	normal force acting in the footing bottom
φ_d	-	design angle of internal friction of soil
c_d	-	design cohesion of the soil
d	-	width of wall heel
e	-	eccentricity
H	-	shear force acting in the footing bottom
F_{res}	-	resisting force (from georeinforcement and mesh overlap)
SF_s	-	safety factor for the sliding resistance
μ	-	reduction coefficient of the contact base - soil

where eccentricity e :

$$e = \frac{M_{ovr} - M_{res} + \frac{Nd}{2}}{N}$$

where:

M_{ovr}	-	overturning moment

M_{res}	-	resisting moment
N	-	normal force acting in the footing bottom
d	-	width of wall heel

Horizontal components of forces are included in the shear force and overturning moment, vertical components of forces are included in the normal force and resisting moment. The resisting forces and moments also include horizontal forces from georeinforcements and overlapping meshes. The uplift pressure is considered in the overturning moment.

Internal Sliding

This limit state evaluates the possibility of structure to slide along the reinforcement. For the selected reinforcement the program searches for a critical slip surface in the range of *45 - 90º* from the end of given reinforcement.

For each slip surfaces, the program calculates the shear and resisting forces and performs verification.

The shear forces include:

- active pressure on a fictitious wall
- forces due to surcharge behind the wall

The resisting forces include:

- resistance of the wall structure against slip (it is calculated as for the wall dimensioning)
- friction between reinforcement and the sliding block
- forces due to other reinforcements

The resisting force due to friction between reinforcement and the sliding block is given by:

$$F = N\, tg\varphi\, C_{ds}$$

where:	N	-	normal force acting on reinforcement (due to self-weight of soil and surcharge behind the fictitious wall)
	φ	-	angle of internal friction of soil surrounding the reinforcement
	C_{ds}	-	coefficient of reduction of friction on reinforcement

The actual verification is then performed based on the input specified in the "Wall analysis" tab, according to the theory of limit states and factors of safety. It has to hold:

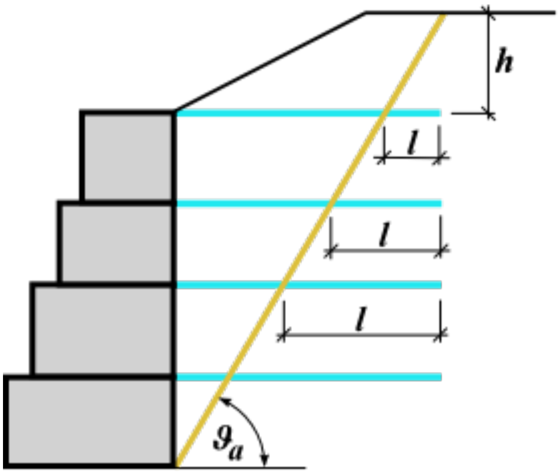
$$H_{res} > H_{act} \quad \text{resp.} \quad \frac{H_{res}}{H_{act}} > SF_{sr}$$

where:	H_{res}	-	resisting force
	H_{act}	-	active force
	SF_{sr}	-	safety factor for sliding along geo-reinforcement

Reinforcements

Reinforcements or overhangs of mesh behind the wall, respectively, may considerably increase the wall stability. The basic parameter of reinforcement is the **tensile strength R_t**. A design value of this parameter is used in all programs (except for the Redi-Rock wall program), i.e. the tensile strength of reinforcement reduced by coefficients taking into account the effect of durability, creep, environmental chemistry and installation damage. The force transmitted by reinforcement **can never exceed the assigned tensile strength R_t** (default value of *40 kN/m* is used for gabions).

The second characteristic is the **pull-out strength T_p**. This parameter determines the anchoring length, i.e. the required length of reinforcement in the soil, for which the reinforcement is fully stressed attaining the value R_t. Since the realistic values of the pull-out strength are difficult to determine, the program offers three options for their calculation, respectively for the calculation of the force F transmitted by the reinforcement.



Length of mesh step joint or reinforcement behind blocks, respectively

Calculate pull-out force

The pull-out force F is given by:

$$F = 2.\sigma.tg\varphi.C.l$$

where:

- σ - normal stress due to self-weight at the intersection of mesh and slip surface
- φ - angle of internal friction of soil
- C - coefficient of interaction (0,8 by default)
- l - length of mesh step joint behind the slip surface into the soil body

Computation of the angle υ_a is described in the chapter earth wedge.

Input reinforcement anchor length l_k

An anchoring length l_k is specified. This parameter is determined by the shear strength developed between the mesh and the soil gradually increasing from zero to its limit value (measured from the end of reinforcement fixed in soil).

$$F = \frac{l}{l_k}.R_t$$

where:

- l - length of mesh step joint behind the slip surface into the soil body
- l_k - anchoring length of reinforcement
- R_t - tensile strength

Input mesh pull-out resistance T_p

The pull-out force F is given by:

$$F = T_p.l$$

where:

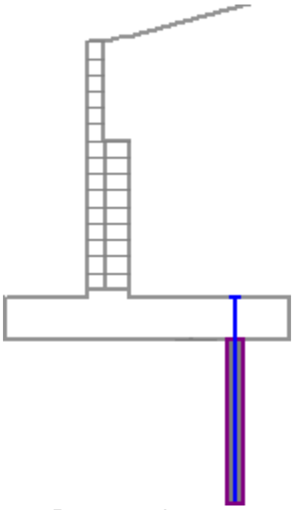
- l - length of mesh step joint behind the slip surface into the soil body
- T_p - pull-out resistance of the mesh

Base Anchorage

An anchorage of wall footing can be specified in the "**Cantilever Wall**" program. It is necessary to specify an anchor location, dimensions of a drill hole, and spacing of anchors.

Two limit states of bearing capacity are defined for an anchor:

- **bearing capacity against pulling-out $R\ e$ $[kN/m]$**
- **strength of anchor R_t $[kN]$**



Base anchorage

Bearing capacities can be either input or computed from the input values using the following expressions:

$$T_p = \frac{\pi \, d \, a}{SF_e}$$

where: T_p - pull-out resistance

d - drill hole diameter

a - ultimate bond

SF_e - safety factor against pulling-out

$$R_t = \frac{\pi \, d_s^2}{4} \frac{f_y}{SF_t}$$

where: R_t - strength of anchor

d_s - anchor diameter

f_y - yield strength of the anchor

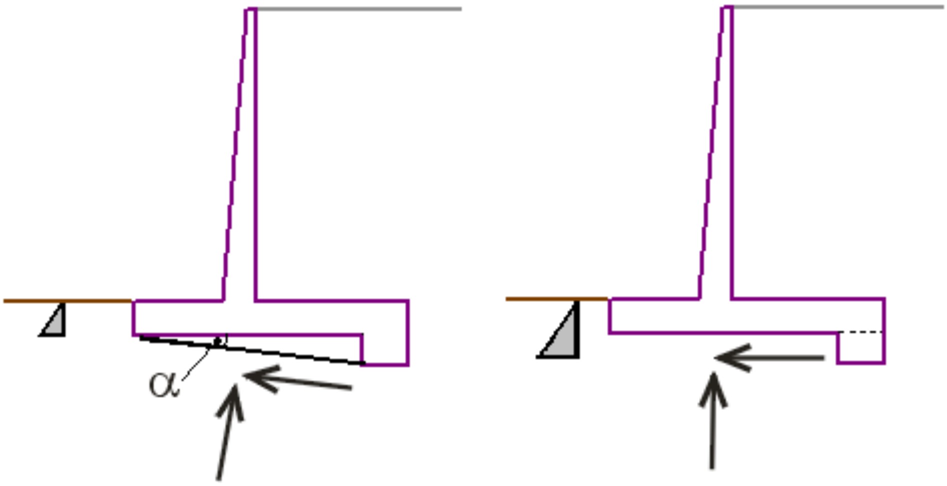
SF_t - safety factor against pulling-apart

Approximate values of bearing capacity against pulling-out

Material	**Ultimate bond** $[N/mm^2]$	**Ultimate strength for specified hole diameter** $[kN/m]$				
		65 mm	75 mm	90 mm	100 mm	150 mm
Soft shale	0.21 - 0,83	42 - 169	49 - 195	59 - 234	65 - 260	98 - 391
Sandstone	0.83 - 1,73	169 - 350	195 - 407	234 - 486	260 - 543	391 - 562
Slate, Hard Shale	0.86 - 1,38	175 - 281	202 - 325	243 - 390	270 - 433	405 - 562
Soft Limestone	1.00 - 1.52	204 - 310	235 - 358	282 - 429	314 - 477	471 - 562
Granite, Basalt	1.72 - 3.10	351 - 562	405 - 562	486 - 562	540 - 562	562 - 562
Concrete	1.38 - 2.76	281 - 562	325 - 562	390 - 562	433 - 562	562 - 562

Accounting for Wall Jump

Two options are available to account for the foundation wall jump in the analysis as shown in the figure (programs "**Cantilever Wall**" and "**Masonry Wall**").

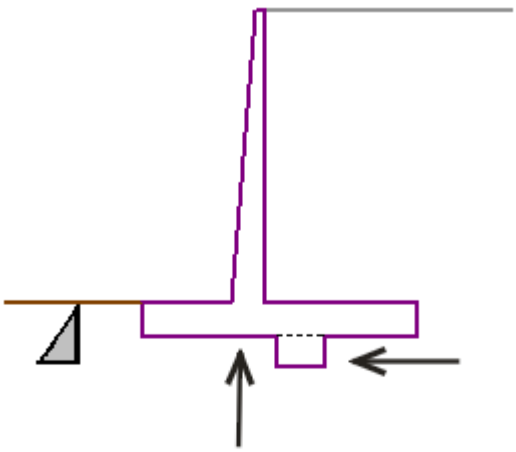


Options to account for wall jump

If the jump is assumed as an **inclined footing bottom**, then a new shape of the footing bottom is considered and the structure front face resistance is included only up to a depth of the wall front face.

If the jump influence is considered as a **front face resistance** the analysis assumes a flat footing bottom (as if there was no jump), but the structure front face resistance included up to a depth of jump. In such a case computation of the structure front face resistance must also be input - otherwise, the jump influence is neglected.

The jump introduced below the wall foundation is always considered as a structure front face resistance.

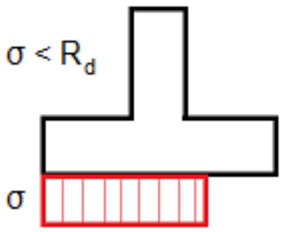


Assuming wall jump in the middle

Bearing Capacity of Foundation Soil

Verification analysis of the bearing capacity of foundation soil takes into account forces obtained from all already performed verifications of the overall stability of the structure (the theory of limit states, safety factor). To that end, the following relationships are used:

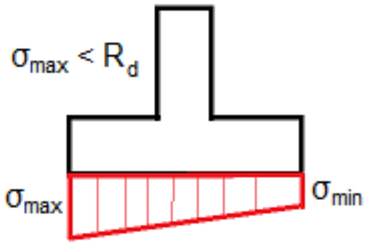
Usually, the shape of contact stress in footing bottom is considered as a constant on a reduced length of footing.



$$\sigma = \frac{N}{d - 2e} < R_d$$

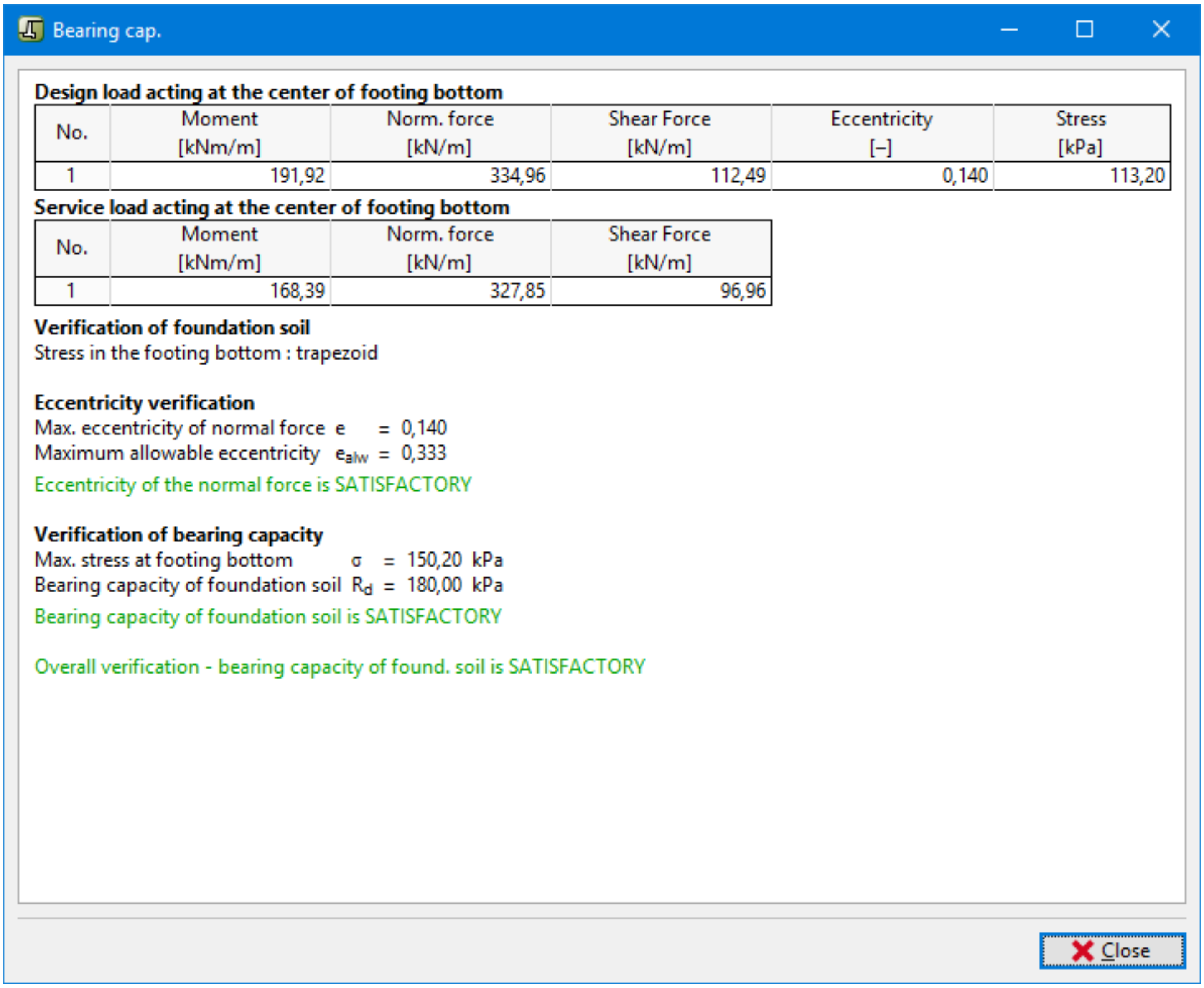
$$e \leq e_{alw}$$

Trapezoid shape of stress is required according to some standards. In this case, the verification is performed for the maximum value σ_{max}.



where:
- N - normal force acting in the footing bottom
- d - width of wall heel
- e max. eccentricity of the normal force
- R_d - bearing capacity of foundation soil
- e_{alw} - allowable eccentricity (this value is defined in the frame "**Settings**" in tab "Wall analysis")

For calculation of bearing capacity of foundation soil (in case of assuming **shallow foundation** under the wall) program allows us to calculate the **design or service load**, that acts at the center of the footing bottom. When transferring data and results in the program "**Spread Footing**" it is possible to calculate settlement and rotation of foundation correctly. For assuming a **pile foundation** in the frame "Foundation" it is possible to view internal forces in the heads of piles (for one series of piles), respectively at the center of the footing bottom (for planar pile grid).

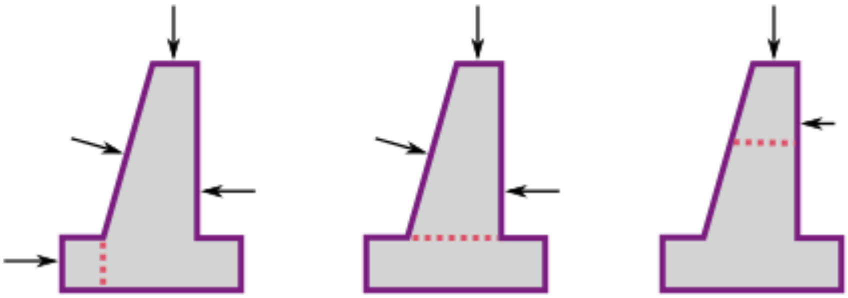


Dialog window "Bearing capacity"

Wall Dimensioning

After computing forces acting on the structure, the program determines all internal forces in the verified cross-section (normal force N, shear force Q and moment M) and then verifies the cross-section bearing capacity employing one of the settings selected in the "Wall analysis" tab.

Only the forces found above the verified joint (see figure) are assumed for dimensioning. These forces are not multiplied by any design coefficients.



Forces entering the analysis

The front jump of the wall as well as the back jump of the wall is verified against the load caused by the bending moment and shear force. Stress in the footing bottom can be assumed either **constant** (CSN) or **linear** (EC).

Assuming **linear variation of stress** in the footing bottom the distribution of stress is provided by:

$$\sigma_1 = \frac{N}{d^2} \cdot \left(4.d - 6 \cdot \left(\frac{d}{2} - e\right)\right)$$

$$\sigma_2 = \frac{N}{d^2} \cdot \left(-2.d + 6 \cdot \left(\frac{d}{2} - e\right)\right)$$

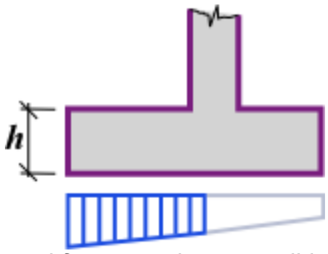
or when excluding tension:

$$\sigma = \frac{2.N}{3 \cdot \left(\frac{d}{2} - e\right)}$$

where:

- e - eccentricity of the normal force N
- d - width of the wall foundation
- N - normal force acting in the footing bottom (see verification according to limit states or factor of safety, respectively)

Bending moment and shear force are determined as reaction developed on the cantilever beam as shown in the figure:



Internal forces acting on wall jump

Verification of the back jump of the wall (top tensile reinforcement in the wall jump, respectively) is performed only in some countries and usually is not required. The programs "**Cantilever wall**" and "**Reinforced wall**" allow in version 5.5 for designing the reinforcement in the back jump of the wall. The cross-section is then assumed to be loaded by the self-weight of structure, earth wedge, surcharge, anchorage force, and the force associated with the contact pressure in the soil. Forces due the pressure are accounted for only if having a negative impact. Forces introduced by the user are not reflected at all.

The cross-section is checked against the load caused by the bending moment and shear force.

Counterfort/Buttress Wall Dimensioning

After computing forces acting on the structure the program determines all internal forces in the verified cross-section (shear force Q and moment M) and then verifies the cross-section bearing capacity employing one of the settings selected in the "Wall analysis" tab.

Wall stem - horizontal direction

Maximal moment at the center of the span between counterforts/buttresses at the front side of the wall stem:

$$M = \frac{1}{20}\sigma_{pi}l^2$$

Maximal moment at the counterfort/buttress at the backside of the wall stem:

$$M = -\frac{1}{12}\sigma_{pi}l^2$$

Maximal shear force:

$$Q = \frac{1}{2}\sigma_{pi}l$$

where: σ_{pi} - average stress at the backside of the wall stem

l - counterfort/buttress spacing

Wall stem - vertical direction

Maximal moment at the center of wall height at the front side of the wall stem:

$$M = 0.03\sigma_H Hl/4$$

Maximal moment at the bottom at the backside of the wall stem:

$$M = -0.03\sigma_H Hl$$

Maximal shear force:

$$Q = \frac{1}{2}\sigma_{pi}l$$

where: σ_H - stress at the bottom at the backside of the wall stem

H - wall height

l - counterfort/buttress spacing

Wall jump - buttress wall

Maximal moment at the center of the span between buttresses at the upper side of the wall jump:

$$M = \frac{1}{20}\sigma_{j1}l^2$$

Maximal moment at the buttress at the bottom side of the wall jump:

$$M = -\frac{1}{12}\sigma_{j1}l^2$$

Maximal shear force:

$$Q = \frac{1}{2}\sigma_{j1}l$$

where: σ_{j1} - maximal stress under wall jump

l - buttress spacing

Wall heel - counterfort wall

Maximal moment at the center of the span between counterforts at the upper side of the wall heel:

$$M = \frac{1}{20}\sigma_j l^2$$

Maximal moment at the counterfort at the bottom side of the wall heel:

$$M = -\frac{1}{12}\sigma_j l^2$$

Maximal shear force:

$$Q = \frac{1}{2}\sigma_j l$$

where: σ_j - maximal stress under wall heel

l - counterfort spacing

Counterfort / Buttress of the wall

Counterfort / Buttress is verified as rectangular cross-section. Calculation of internal forces is the same as for wall stem of the cantilever wall. Internal forces are multiplied by the loading width $l+b$ (counterfort/buttress spacing + counterfort/buttress thickness).

Internal forces at places of other checks (Wall heel - buttress wall, Wall jump - counterfort wall) are calculated in the same way, as for cantilever wall.

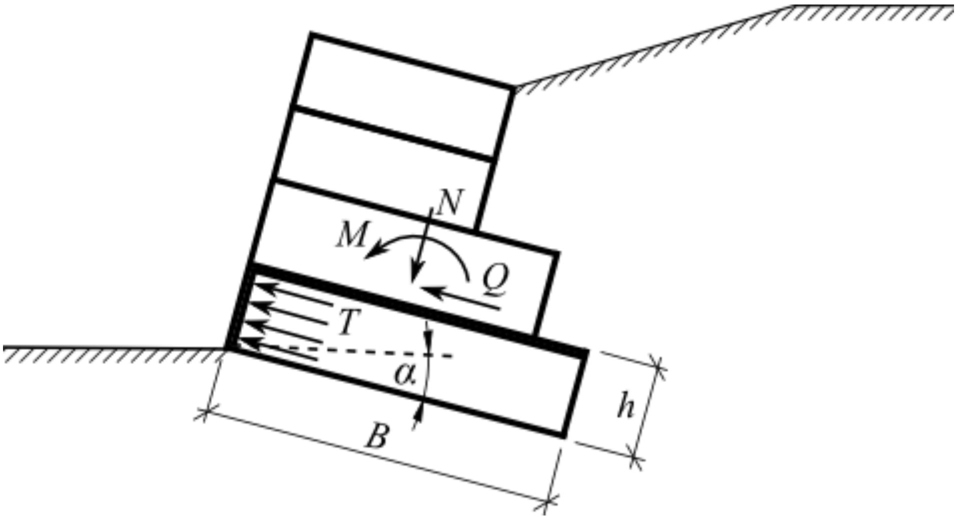
Internal Stability of a Gabion

The internal stability of gabion wall can be examined with the help of:

- the theory of limit states
- factor of safety

Verification of joints between individual blocks is performed in the "**Dimensioning**" frame. The structure above the block is loaded by active pressure and corresponding forces are determined in the same way as for the verification of the entire wall. A loose filling is used in the analysis - not hand-placed rockfill - but its effect can be simulated using a very high angle of internal friction. It can be assumed that after some time due to the action of filling aggregate the stress in meshes will dropdown. Individual sections of the gabion wall are checked for the maximum normal and shear stress. With the help of these variables, it is possible to modify the slope of the structure face by creating terraces or by increasing the slope of the face of the wall α .

Assuming the load applied to the bottom block is schematically represented as:



Load on the bottom block

Normal stress in the center of the bottom block is given by:

$$\sigma = \frac{N}{B-2e} + \frac{\gamma h \cos\alpha}{2}$$

$$e = \frac{M}{N}$$

where:

N - normal force acting on the bottom block

B - width of the upper block

e - eccentricity

M - moment acting on the bottom block

h - height of the bottom block

γ - unit weight of the bottom block material

α - gabion slope

The pressure acting on the wall of the bottom block is determined as an increased active pressure:

$$T = 0{,}5 T_r + 0{,}5 T_a$$

$$T_r = \sigma K_r$$

$$T_a = \sigma . K_a - 2 . c_d . \sqrt{K_a}$$
$$K_r = 1 - \sin \varphi_d$$
$$K_a = tg^2 \left(45 - \frac{\varphi_d}{2} \right)$$

where:

- φ_d - design angle of internal friction of the bottom block material
- c_d - design cohesion of the bottom block material
- γ - unit weight of material of the bottom block
- h - height of bottom block
- B - width of upper block
- α - gabion slope
- T - average value of pressure acting on face of the bottom block
- σ - maximal normal stress acting on the bottom block
- K_r - coefficient of earth pressure at rest
- K_a - coefficient of active earth pressure

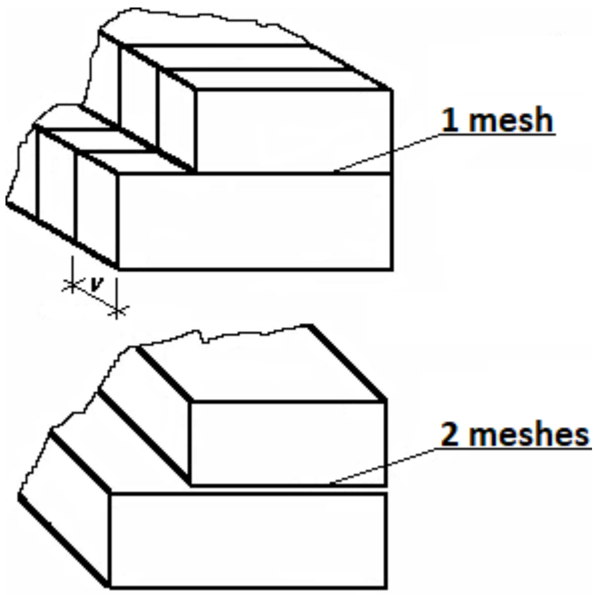
Widths of meshes of the bottom block per one running meter of the gabion wall are:

$$D_{upp} = 1$$
$$D_{total} = \frac{h}{v} + 1$$

where:

- D_{upp} - width of upper mesh between blocks loaded in tension
- D_{total} - overall width of meshes loaded in compression T
- v - spacing of the vertical meshes
- h - height of the bottom block

The program allows for analysis of gabions with both simple and double mesh placed between blocks. For double meshes, the input tensile strength of mesh (frame "Material" - the "**Edit material**" dialog window) should be twice as large as the value assumed for simple meshes.



Geometry of gabions

Internal Stability of a Gabion Wall - Safety Factor

The following cases are assumed when examining the internal stability of the gabion wall using the concept of factor of safety:

1) Check for overturning stability:

$$\frac{M_{res}}{M_{ovr}} > SF_o$$

where: M_{ovr} - overturning moment

M_{res} - resisting moment

SF_o - safety factor for overturning

2) Check for slip:

$$\frac{[(N \tan\varphi_d + c_d(d-2e)/\mu) + F_{res}]}{H} > SF_s$$

where: N - normal force

φ_d - design angle of internal friction of soil

c_d - design cohesion of the soil

d - width of block

e - eccentricity

H - shear force

F_{res} - resisting force (from georeinforcement and mesh overlap)

SF_s - safety factor for sliding resistance

μ - reduction coefficient

3) Check for bearing capacity with respect to the lateral pressure:

$$\frac{S_u}{S} > SF_n$$

$$S = \frac{Tbh}{D_{total}}$$

where: T - average value of pressure acting on face of the bottom block

S - force per one running meter of the joint

S_u - joint bearing capacity (for input, use the frame "Material")

SF_n - safety factor mesh strength (for input, use the "Wall analysis" tab - default value is 1.5)

b - width = 1 m of structure width

h height of the block

D_{total} overall width of meshes loaded in compression T

4) Check for bearing capacity of the joint between blocks:

$$\frac{N_u}{N} > SF_n$$

$$N_d = \frac{T\,b\,h}{D_{total}} + \frac{\max(0, Q - Q_{tr})}{D_{upp}}$$

$$Q_{tr} = \frac{N \tan\varphi_d + c_d B}{\gamma_f}$$

where: N_d - tensile force per one running meter of the upper joint

N_u - strength of mesh (for input, use the frame "Material")

SF_n - safety factor mesh strength (for input, use the "Wall analysis" tab - default value is 1.5)

Q_{tr} - shear force transmitted by friction and cohesion between blocks

γ_f - reduction coefficient of friction between blocks (for input, use the "Wall analysis" tab)

h - height of the block

D_{total} - overall width of meshes loaded in compression T

D_{upp} - width of upper mesh between blocks loaded in tension

Internal stability of a Gabion Wall - Limit States

Reduced parameters of the gabion material, which depend on the coefficients set in the "Wall analysis" tab, are used in the verification analysis.

1) Check for overturning stability:

$$M_{ovr} < M_{res}$$

where: M_{ovr} - overturning moment

M_{res} - resisting moment

2) Check for slip:

$$\frac{\left[\left(N \tan \varphi_d + c_d (d - 2e)/\mu\right) + F_{res}\right]}{\gamma_s} > H$$

where:
N - normal force
φ_d - design angle of internal friction of soil
c_d - design cohesion of the soil
d - width of block
e - eccentricity
γ_s - reduction coefficient of sliding resistance
H - shear force
F_{res} - resisting force (from georeinforcement and mesh overlap)
μ - reduction coefficient

3) Check for bearing capacity with respect to the lateral pressure:

$$S < S_u$$

$$S = \frac{T\, b\, h}{D_{total}}$$

where:
T - average value of the pressure acting on the face of the bottom block
S - force per one running meter of the joint
S_u - joint bearing capacity (for input, use the "Material" frame)
b - width = $1m$ of structure width
h - height of the block
D_{total} - overall width of meshes loaded in compression T

4) Check for bearing capacityof joint between blocks:

$$N_d < N_u$$

$$Q_{tr} = \frac{N \tan \varphi_d + c_d B}{\gamma_f}$$

$$N_d = \frac{T\, b\, h}{D_{total}} + \frac{\max\,(0\,, Q - Q_{tr})}{D_{upp}}$$

where:
N_d - tensile force per one running meter of the upper joint of the bottom block
N_u - strength of the mesh (for input, use the frame "Material")

Q_{tr} - shear force transmitted by friction and cohesion between the blocks
γ_t - reduction coefficient of friction between the blocks (for input, use the "Wall analysis")
h height of the block
D_{total} overall width of meshes loaded in compression T
D_{upp} width of the upper mesh between blocks loaded in tension

Calculating Abutment Forces

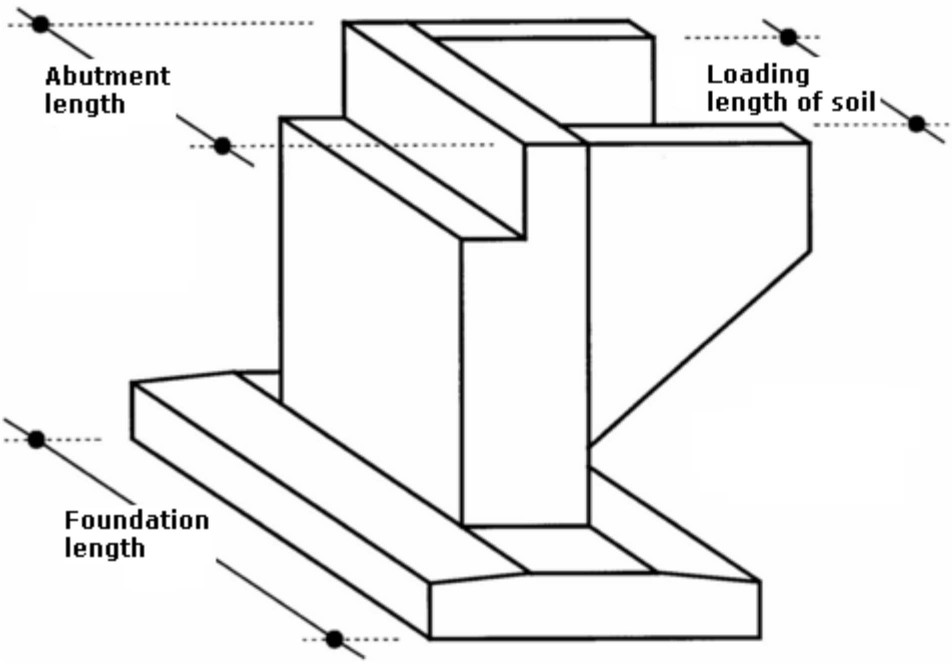
An abutment is analyzed per 1 m $(1ft)$. All forces entering the analysis are therefore adjusted in the program as follows:

- the **abutment self-weight**, assumed per 1 m $(1ft)$, is calculated from the input transverse cross-section
- **reactions inserted by the bridge and the approach slab** are input in kN (kpi) using the values for the entire abutment, these values are in the analysis divided by the **abutment length**
- **soil pressure** is determined per 1 m $(1ft)$ and then multiplied by the ratio **length of load due to soil / abutment length**,
- **weight of soil wedges** is determined per 1 m $(1ft)$ and then multiplied by the ratio **length of load due to soil / abutment length**,

- **surcharge** is determined per 1 m $(1ft)$ and then multiplied by the ratio **length of load due to soil / abutment length**,
- **input forces** and **front face resistance** are assumed per 1 m without reduction
- **wing walls** - the wing walls self-weight is computed from their geometry; before introduced in the stem design and foundation verification it is divided by the **abutment length** (it is the user's responsibility to either include or exclude the effect of wing walls in from the analysis).

Computation of individual abutment forces is described in more detail in the chapter "Wall analyses".

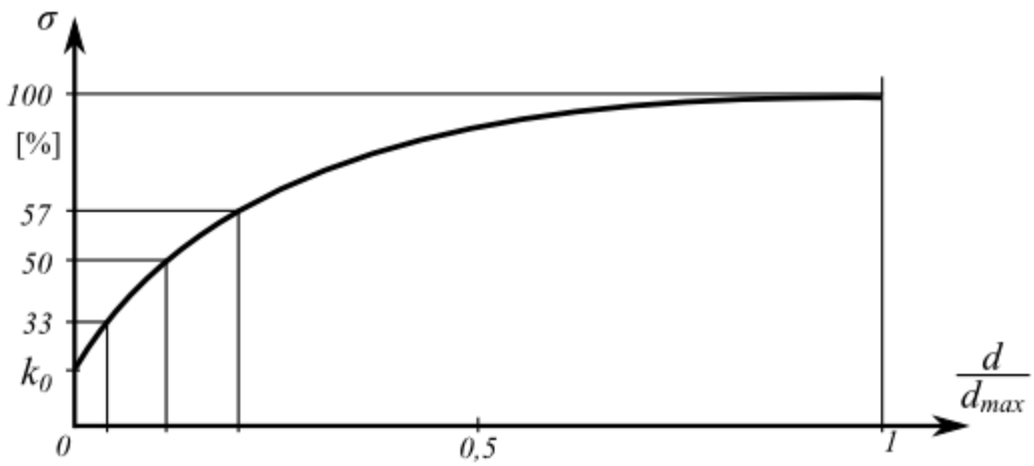
All forces acting in the foundation joint that are introduced in the verification analysis (except for the front face resistance) are multiplied by the ratio **abutment length / foundation length.**



Geometry of bridge abutment

Reduced Passive Earth Pressure

The evolution of passive earth pressure σ_p corresponds to the maximal displacement of a structure pushed into the soil. Such a displacement might not, however, occur (e.g. in case of fixed sheeting structures) and the structure is loaded by the reduced passive earth pressure σ_{ps}. The value of reduced passive earth pressure σ_{ps} can range from the value of earth pressure at rest σ_r (in case of zero deformation) up to the value of passive earth pressure σ_p. The figure shows the dependence of values of earth pressure of a cohesionless soil (soil resistance) on the actual displacement d to maximal displacement d_{max} ratio (when activating passing earth pressure σ_p).



Dependence of earth pressure values on the ratio of actual structure deformation

Nailed Slope

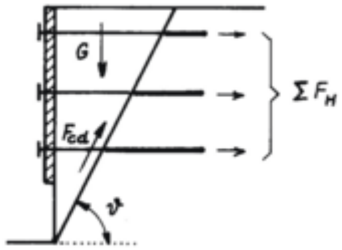
The "**Nailed Slope**" program allows the following verifications:

- Verification of structure internal stability (plane or broken slip surface, bearing capacity of nails)
- Verification of fictitious wall - the same as gravity wall verification
- Verification of structure concrete cover (dimensioning)
- Verification of mesh (dimensioning)
- Verification of overall stability using the program "**Slope Stability**"

Analysis of Internal Stability

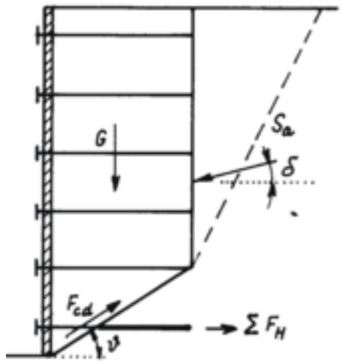
Internal stability of a structure is checked assuming two types of a slip surface:

- **Plane slip surface**



Plane slip surface

- **Broken slip surface**



Broken slip surface

In both cases, a specific slip surface is examined for a variation of angle ϑ.

When running an **optimization** analysis the calculation is carried out for all benches with a variation of the angle of slip surface ν changing *from* 1 *up to* 89 degrees with a one-degree step.

A verification analysis of internal stability can be performed using either the factor of safety or the theory of limit states depending on the setting in the "Wall analysis" tab.

The analysis checks whether a ratio of resisting and shear (driving) forces acting on a slip surface is greater than the input factor of safety. The following forces are employed:

Shear forces:

- component of gravity force parallel to slip surface
- in case of broken slip surface - component of active earth pressure acting on the vertical part of structure and parallel to slip surface (pressure is determined without reduction of input parameters)
- horizontal forces due to earthquake

Resisting forces:

- soil friction and cohesion along the slip surface
- sum of the forces transmitted by nails

Analysis of Bearing Capacity of the Nails

For each nail the following bearing capacities are either computed or input:

where: R_f - nailhead strength

R_t - nail strength against breaking

T_p - pull-out nail bearing capacity

The strength characteristics of a nail represent the basic parameters to compute the total bearing capacity of a nail.

The nail strength against breaking follows from:

$$R_t = \frac{\pi d_s^2}{4} \frac{f_y}{SF_t}$$

where:

- R_t - strength against breaking
- d_s - nail diameter
- f_y - strength of nail material
- SF_t - factor of safety against breaking

The pull-out nail bearing capacity is calculated by one of the following ways:

1. calculate from bond strength:

$$T_p = \frac{\pi d g_s}{SF_e}$$

where:

- T_p - pull-out nail bearing capacity [kN/m]
- d - hole diameter
- g_s - bond strength, given either as a parameter of the nail or as soil parameter
- SF_e - factor of safety against pull-out

2. calculate from effective stress

$$T_p = \frac{\pi d (K_a \sigma_z \tan \varphi + c)}{SF_e}$$

where:

$$K_a = \frac{1 + K_0}{2} = \frac{1 + (1 - \sin \varphi)}{2}$$

where:

- T_p - pull-out nail bearing capacity [kN/m]
- d - hole diameter
- σ_z - vertical geostatic stress
- φ - effective angle of internal friction of the soil
- c - effective cohesion of the soil
- SF_e - factor of safety against pull-out

3. calculate according to HA 68/94

$$T_p = \frac{\pi d (\sigma_n \tan \varphi + c)}{SF_e}$$

where:

- T_p - pull-out nail bearing capacity [kN/m]
- d - hole diameter
- σ_n - average radial effective stress
- φ - effective angle of internal friction of the soil
- c - effective cohesion of the soil
- SF_e - factor of safety against pull-out

Average radial effective stress σ_n is calculated by the following formula:

$$\sigma_n = \frac{(1 + K_L)\sigma_z}{2}$$

where: σ_z - vertical geostatic stress

$$K_L = \frac{1+K_a}{2}$$

where:

$$K_a = \frac{1-\sin\varphi}{1+\sin\varphi}$$

The nail head strength is evaluated by the formula:

$$R_f = \frac{\min\left(R_t; T_p l\right)\left(0,6+0,2\left(S_{\max}-1\right)\right)}{SF_f}$$

where:
- l - nail length
- S_{max} - maximum spacing of nails in a structure
- R_t - nail strength against breaking
- T_p - pull-out nail bearing capacity
- SF_f - factor of safety of a nail head strength

If the nail is not anchored to the structure cover, it is possible to set the nail head strength to zero.

Literature:

FHWA0-IF-03-017

Recommended Values of Bond Strength

The table is based on field and laboratory tests.

Bond strength: Classic drilled nails

$q_{s,k}$ $[kPa]$ 1) **according to EA Pfähle**								
Cohesionless soils			Cohesive soils			Rocks		
q_c 2) $[MN/m^2]$			c_u 3) $[kN/m^2]$			$q_{u,k}$ 4) $[MN/m^2]$		
7.5	15	25	60	150	250	0.5	5	20
55 -80	105 - 140	130 -170	30 - 40	50 -65	65 - 85	70 - 250	500 -1000	500 - 2000

Notes:

1) Determination of bond strength should be part of the geological survey

2) Cone resistance, CPT test (according to DIN EN ISO 22476-1)

3) Total cohesion of the soil

4) Uniaxial compression strength

Literature:

EA-Pfähle, ISBN: 978-3-433-03005-9

Values in the table based on soil and rock classification.

Estimated bond strength of soil nails in soil and rock *(source: Elias a Juran, 1991)*

Material	**Construction method**	**Soil / rock type**	**Ultimate bond strength** q_s $[kPa]$
Rock	**Rotary drilled**	Marl / limestone	300 - 400
		Phyllite	100 - 300
		Chalk	500 - 600
		Soft dolomite	400 - 600
		Fissured dolomite	600 - 1000
		Weathered sandstone	200 - 300
		Weathered shale	100 - 150
		Weathered schist	100 - 175

		Basalt	500 - 600
		Slate / hard shale	300 - 400
Cohesionless soils	**Rotary drilled**	Sand / gravel	100 - 180
		Silty sand	100 - 150
		Silt	40 - 120
		Piedmont residual	40 - 120
		Fine colluvium	75 - 150
	Driven casing	Sand / gravel -low overburden -high overburden	 190 - 240 280 - 430
		Dense moraine	380 - 480
		Colluvium	100 - 180
	Augered	Silty sand fill	20 - 40
		Silty fine sand	55 - 90
		Silty clayey sand	60 - 140
	Jet grouted	Sand	380
		Sand gravel	700
Fine - grained soils	**Rotary drilled**	Silty clay	35 - 50
	Driven casing	Clayey silt	90 - 140
	Augered	Loess	25 - 75
		Soft clay	20 - 30
		Stiff clay	40 - 60
		Stiff clayey silt	40 - 100
		Calcareous sandy clay	90 - 140

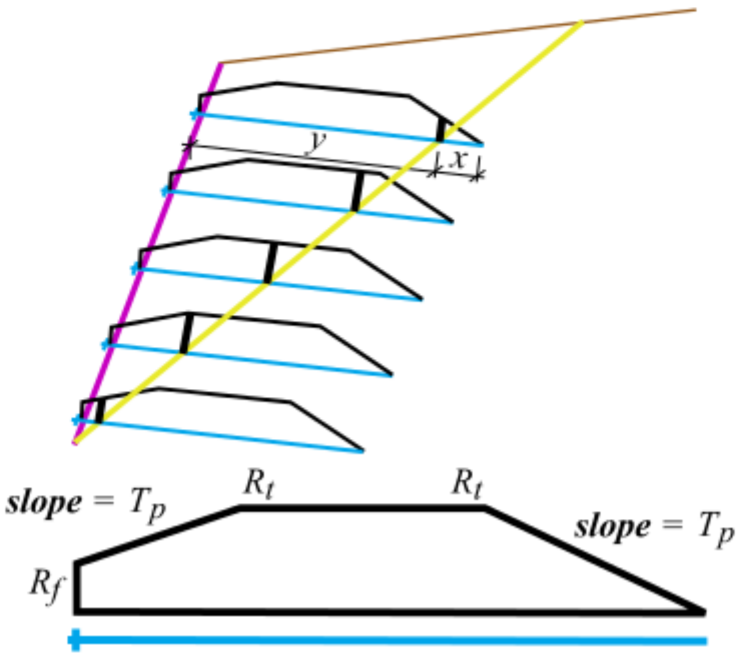
Note: Convert values in kPa to psf by multiplying by 20.9. Convert values in kPa to psi by multiplying by 0.145.

Total Bearing Capacity of a Nail

The bearing capacity of the nail is determined based on the location of its intersection with a slip surface. If a nail is found completely in front of the slip surface, then it does not enter the calculation. If a nail crosses the slip surface, then its bearing capacity is determined as:

$$F = \min\left(T_p x \; ; \; R_t \; ; \; R_f + T_p y\right)$$

where:
- x - nail length behind slip surface in direction of soil body
- y - nail length in front of the slip surface
- R_f - nail cap bearing capacity
- R_t - nail strength against breaking
- T_p - pull-out nail bearing capacity



Distribution of tensile force along the nail

Verification - Factor of Safety

The analysis checks whether a ratio of **resisting** and **shear** (driving) forces acting on a slip surface is greater than the input factor of safety.

A factor of safety on the input slips forces is thus provided by:

$$SF = \frac{F_h \cos(\upsilon + \alpha) + F_{cd}}{(G + S_{a,vert}) \sin \upsilon + S_{a,hor} \cos \upsilon}$$

$$F_h = \sum F_{h,n}$$

$$F_{cd} = \sum \frac{d_i}{d} (G \cos \upsilon + F_h \sin(\upsilon + \alpha)) \tan \varphi_i + \sum d_i c_i$$

where:

- G - gravity force
- $S_{a,vert}$ - vertical component of the active pressure
- $S_{a,hor}$ - horizontal component of the active pressure
- d_i - length of the i^{th} section slip surface
- d - length of the slip surface
- $F_{h,n}$ - bearing capacity of n^{th} nail behind the slip surface per 1 running meter
- c_i - cohesion of the i^{th} soil layer
- φ_i - angle of internal friction of the i^{th} layer
- υ - inclination of the slip surface
- α - inclination of the nails from a horizontal direction

Verification - Theory of Limit States

The analysis checks whether the **passive** (resisting) **forces** F_p acting on a slip surface are greater than the **active** (shear) **forces** F_a:

$$F_p > F_a$$

$$F_p = F_h \cos(\upsilon + \alpha) + F_{cd}$$

$$F_a = (G + S_{a,vert}) \sin \upsilon + S_{a,hor} - \cos \upsilon$$

$$F_h = \sum F_{h,n}$$

$$F_{cd} = \sum \frac{d_i}{d} (G \cos \upsilon + F_h \sin(\upsilon + \alpha)) \tan \varphi_i + \sum d_i c_i$$

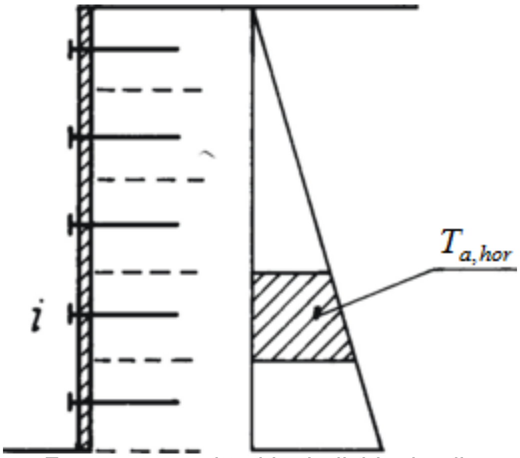
where:

- G - gravity force
- $S_{a,vert}$ - vertical component of the active pressure
- $S_{a,hor}$ - horizontal component of the active pressure
- d_i - length of the i^{th} section slip surface
- d - length of the slip surface
- $F_{h,n}$ - bearing capacity of the n^{th} nail behind slip surface per 1 running meter
- c_i - cohesion of the i^{th} soil layer
- φ_i - angle of internal friction of the i^{th} layer
- υ - inclination of the slip surface
- α - inclination of nails from a horizontal direction

Nail Force

The magnitude of **active earth pressure** is reduced using a coefficient k_n . The recommended (experimentally determined) value is $k_n =$ 0,85.

Forces transmitted by individual nails are determined such that a particular portion of the calculated earth pressure is assigned **to a given bench**. Each nail is then loaded by the corresponding portion of the active earth pressure.



Forces transmitted by individual nails

The **nail force** is provided by:

$$F_i = \frac{b\, k_n \sum T_{a,hor}}{\cos\alpha}$$

where:
- b - nail spacing
- α - nail inclination
- k_n - reduction coefficient
- $T_{a,hor}$ - active earth pressure acting on a given bench

Dimensioning of Concrete Cover

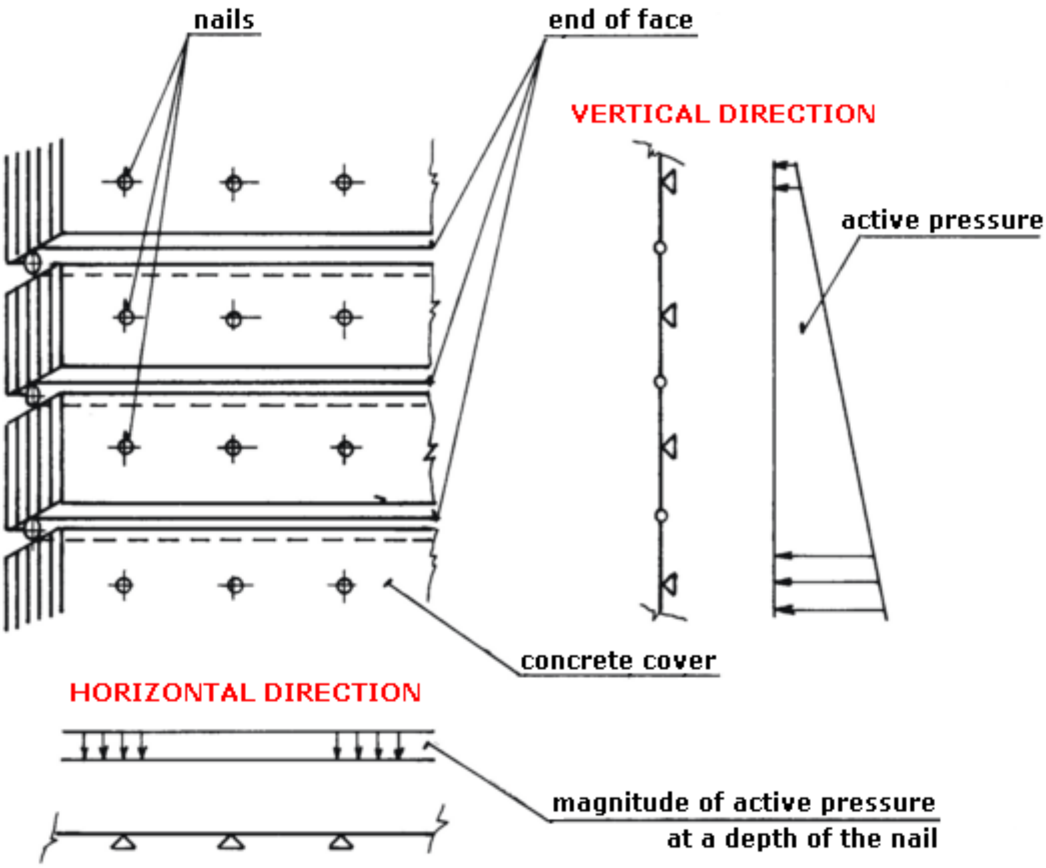
The concrete cover of a nailed slope is designed to sustain an active earth pressure. So, the structure is assumed to be subdivided into individual intermediate design strips.

In the **vertical direction,** the nail cap is modeled as support and joint between benches as an internal hinge.

In the **horizontal direction,** the program generates (by default) a structure with four supports uniformly loaded by the magnitude of active pressure up to a depth of the nail cap.

The program further allows for the verification of concrete cover reinforcement of a structure loaded by the bending moment.

Constructing scheme of the **model design** including load is evident from the figure:



Dimensioning of concrete cover

Verification of Mesh

The **mesh** design requires that the weathered layer consists of soil or heavily weathered rock.

The following assessments are carried out:

1) Check for nail shear strength

$$R_s \geq F_s$$

where: R_s - nail shear strength

F_s - shear force in the nail

2) Check for mesh puncturing resistance

$$R_p \geq F_{nail}$$

where: R_p - mesh puncturing resistance

F_{nail} - force in the nail

3) Check for mesh shear strength

$$R_s \geq S_d$$

where: R_s - mesh shear strength

S_d - shear force in the mesh

4) Check for combined strain in the nail

$$1 \geq max\left(\sqrt{\left(\frac{F_{nail}}{R_t}\right)^2 + \left(\frac{F_s}{R_s}\right)^2}, \sqrt{\left(\frac{S_d}{R_t}\right)^2 + \left(\frac{F_s}{R_s}\right)^2}\right)$$

where: F_{nail} - force in the nail

R_t - nail tension strength

F_s - shear force in the nail

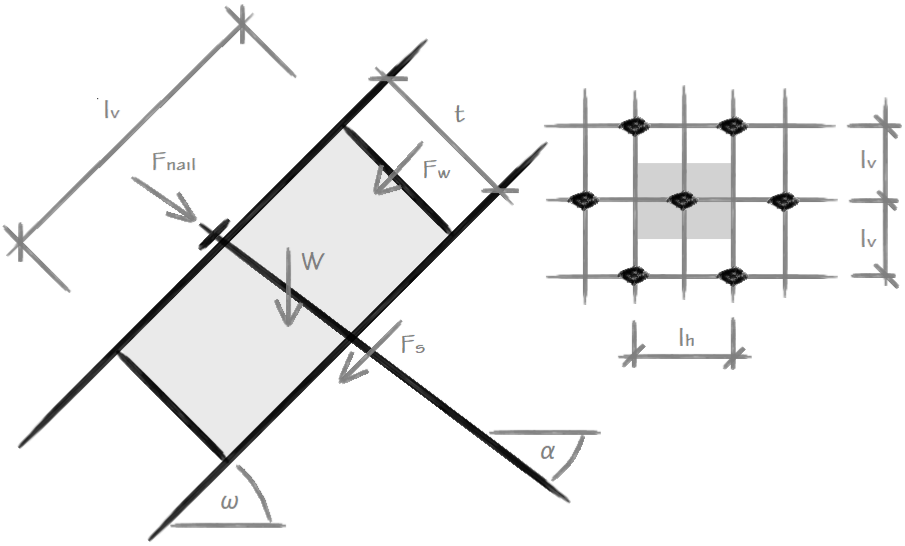
R_s - nail shear strength
S_d - shear force in the mesh

Literature:

*Krist O., Raithel M., Weingart K.: "Bemessung von Drathgeflechten zur Stabilisierung von B*öschungen". EI-Eisenbahningenieur, March 2015, 14-19.

Shear Force in the Nail

In **nail shear bearing capacity** check, the slip surface is considered at the maximum depth of the weathered layer. The shear force F_s is determined from forces acting on the soil block.



Shear force in nail

$$F_s = W.sin(\omega) - F_{nail}.cos(\omega+\alpha) + F_w - [(W.cos(\omega) + F_{nail}.sin(\omega+\alpha)).tan(\varphi) + c.A]$$

where:

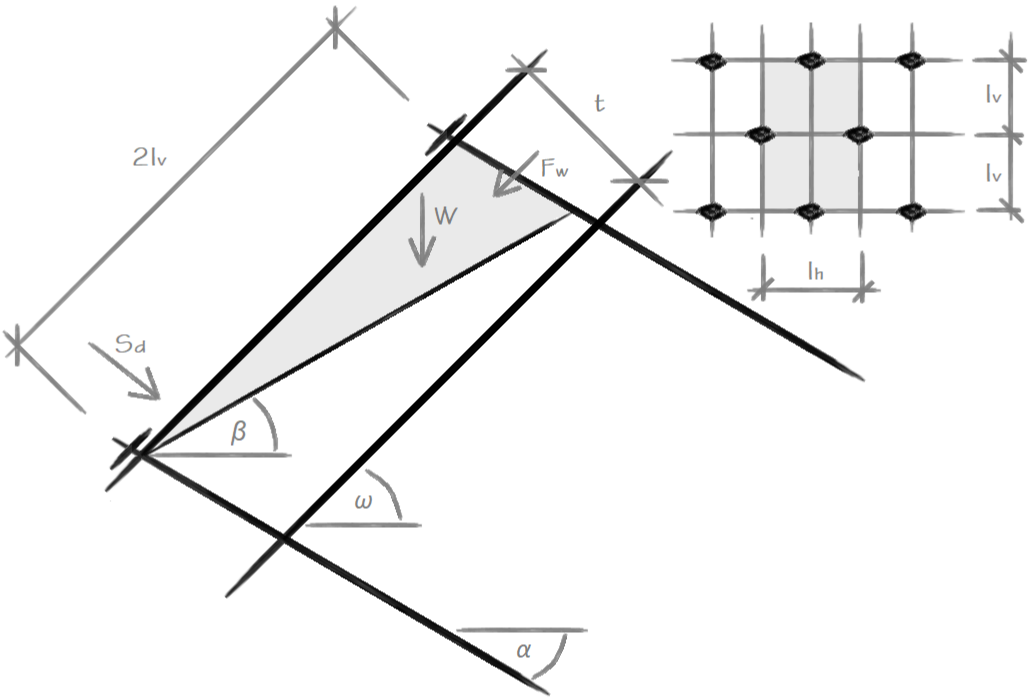
F_s - shear force in the nail
W - soil block weight
F_{nail} - force in the nail
F_w - force due to water flow
l_v - vertical spacing
l_h - horizontal spacing
t - weathered layer thickness
ω - slope inclination
α - nail inclination
φ - angle of the internal friction in a weathered layer
c - cohession in a weathered layer
A - block base area

If the water flow is considered, the program calculates with water in the entire weathered layer.

Shear Force in the Mesh

The program automatically calculates **two shapes of slip surfaces** (in the soil block formed in the area between nails) and finds the **maximal shear force**.

Plane slip surface - maximum shear force is sought over the entire weathered layer thickness.



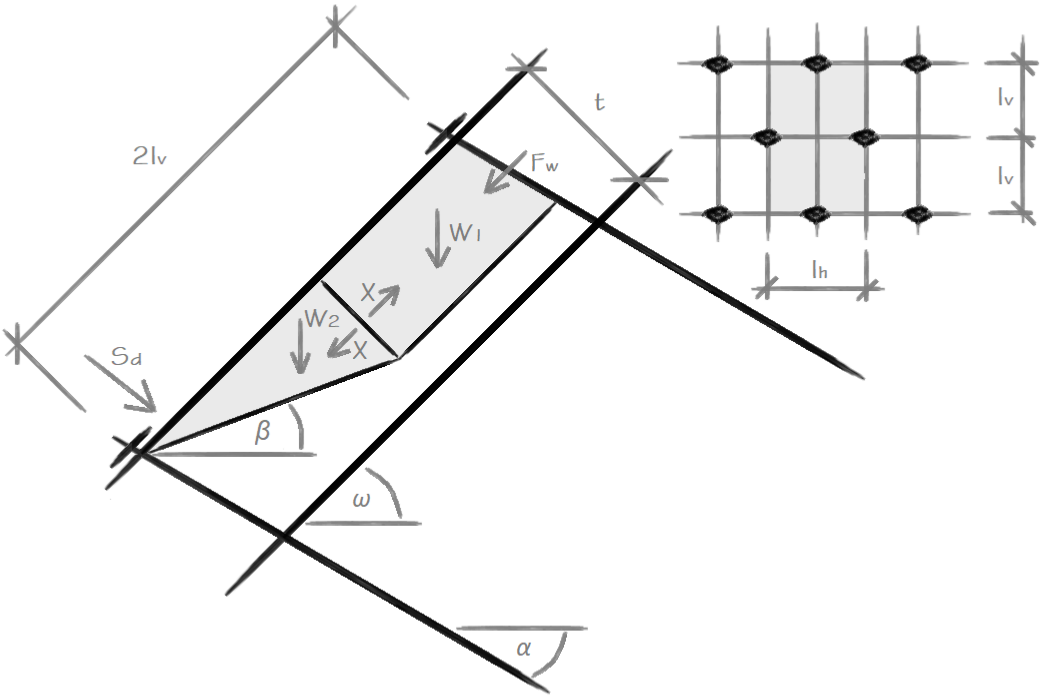
Shear force - plane slip surface

$$S_d = \frac{W.sin(\beta) + F_w.cos(\omega - \beta) - [(W.cos(\beta) + F_w.sin(\omega - \beta)).tan(\varphi) + c.A]}{cos(\alpha + \beta) + sin(\alpha + \beta).tan(\varphi)}$$

where:

S_d	-	shear force in the mesh
W	-	soil block weight
F_w	-	force due to water flow
l_v	-	vertical spacing
t	-	weathered layer thickness
β	-	inclination of the block base
ω	-	slope inclination
α	-	nail inclination
φ	-	angle of the internal friction in a weathered layer
c	-	cohession in a weathered layer
A	-	block base area

Broken slip surface - maximum shear force is sought over the entire weathered layer thickness with different inclination at the bottom block base.



Shear force - broken slip surface

When calculating a broken slip surface the force between blocks X is computed as follows.

$$X = W_1.sin(\omega) + F_{w,1} - [W_1.cos(\omega).tan(\varphi) + c.A_1]$$

where:
- X - force between blocks
- W_1 - weight of the block 1
- $F_{w,1}$ - force due to water flow in the block 1
- A_1 - block 1 base area

$$S_d = \frac{W_2.sin(\beta) + (F_{w,2} + X).cos(\omega - \beta) - [(W_2.cos(\beta) + (F_{w,2} + X).sin(\omega - \beta))).tan(\varphi) + c.A_2]}{cos(\alpha + \beta) + sin(\alpha + \beta).tan(\varphi)}$$

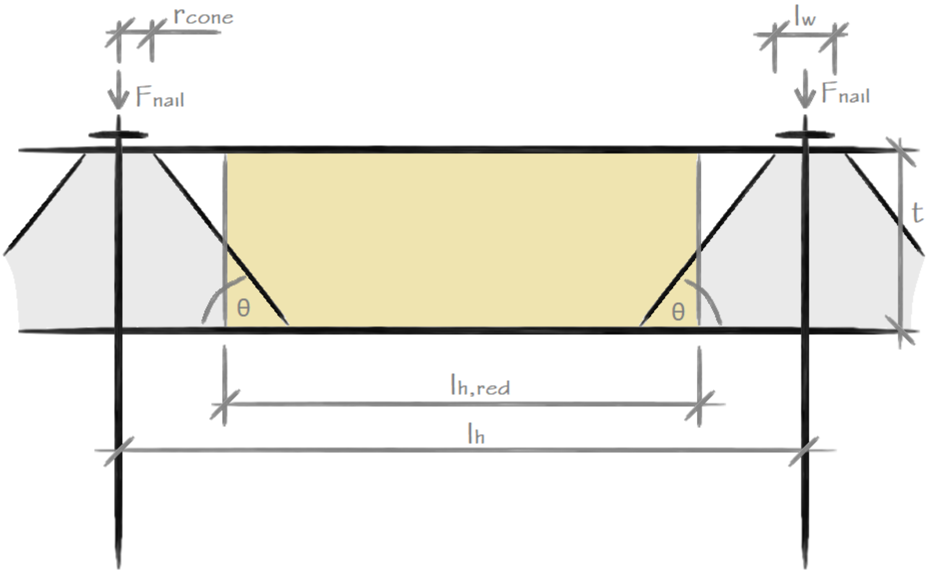
where:
- S_d - shear force in the mesh
- W_2 - weight of the block 2
- $F_{w,12}$ - force due to water flow in the block 2
- A_2 - block 2 base area

If the water flow is considered, the program calculates with water in the entire weathered layer. The pressure cone angle influences the transsmission of the nail force through the weathered layer.

Influence of Pressure Cone

The pressure cone inclination determines the transmitting of the nail force through the weathered layer. It is considered in the calculation of shear force acting on the mesh. It reduces the horizontal spacing between the nails and the width of the soil block. The reduction result is a trapezoidal shape of a soil block modified to a rectangle shape of width $l_{h,red}$.

The pressure cone angle θ is entered between 30° and 80°. The pressure cone radius in the upper part r_{cone} is considered as half of the plate length l_w.



Influence of Pressure Cone

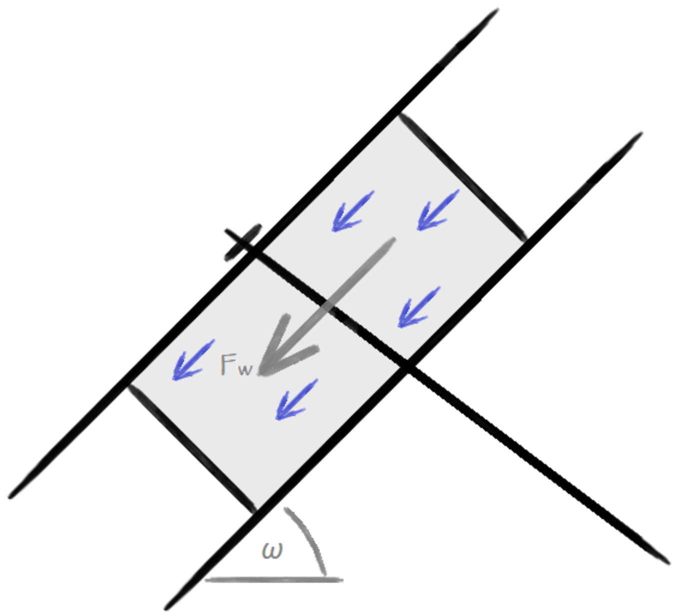
$$l_{h,red} = l_h - 2.r_{cone} - \frac{t}{tan(\theta)}$$

where:

- $l_{h,red}$ - reduced horizontal spacing between nails
- θ - pressure cone angle
- l_h - horizontal spacing between nails
- t - weathered layer thickness
- r_{cone} - pressure cone radius
- l_w - plate length

Influence of Water Flow

The force F_w considers the water flow effect. It is a part of the calculation of shear force in the nail and shear force in the mesh.



Infulence of Water Flow

$$F_w = V.\gamma_w.sin(\omega)$$

where:

- F_w - force due to water flow

V - soil block volume

γ_w - unit weight of water

ω - slope inclination

Excavation Design

GEO5 enables a complete design and analysis of the vertical sheeting structure of many types. Retaining walls can be **anchored**, **strutted, or non-anchored**.

The basic program for analysis of anchored or strutted retaining walls is "Sheeting Check". It allows calculations of real structure behavior using stages of construction, calculations of deformation and pressures acting upon the structure, verification of internal anchor stability and verification of steel, plastic, timber, or reinforced concrete cross-sections.

The "Sheeting Design" program is designated for a fast design of non-anchored structures, and basic design of anchored structures. The program calculates the required length of the structure in the soil and internal forces on the structure. The verification of a cross-section (concrete, steel, timber, plastic) can be performed.

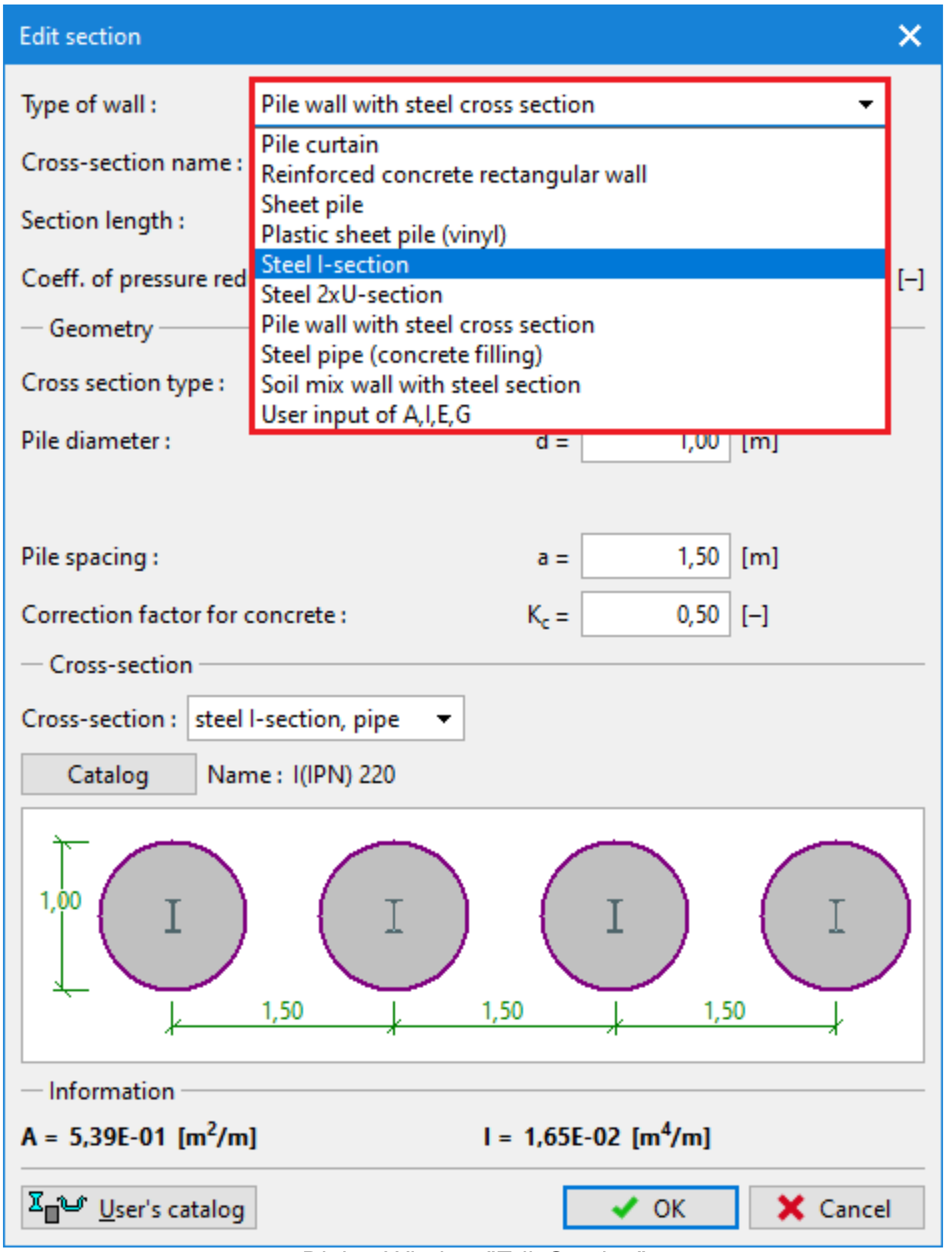
Special case of a shoring structure made in the slope to prevent a landslide is the row of anti-slide piles. The program "Anti-slide Pile" allows analysis of this structure.

Cross Sections Input

Combo list in the dialog window "**Edit section**" contains individual structural types of shoring walls:

- Pile curtain
- Reinforced concrete rectangular wall
- Sheet pile wall
- Steel I cross-section
- Plastic sheet pile (vinyl)
- Steel 2xU-section
- Pile wall with steel cross-section
- Steel pipe
- Soil mix wall with steel section
- User input of cross-sectional characteristics

Some types of cross-sections can be saved in the "User's Catalog" (button "**User's Catalog**").

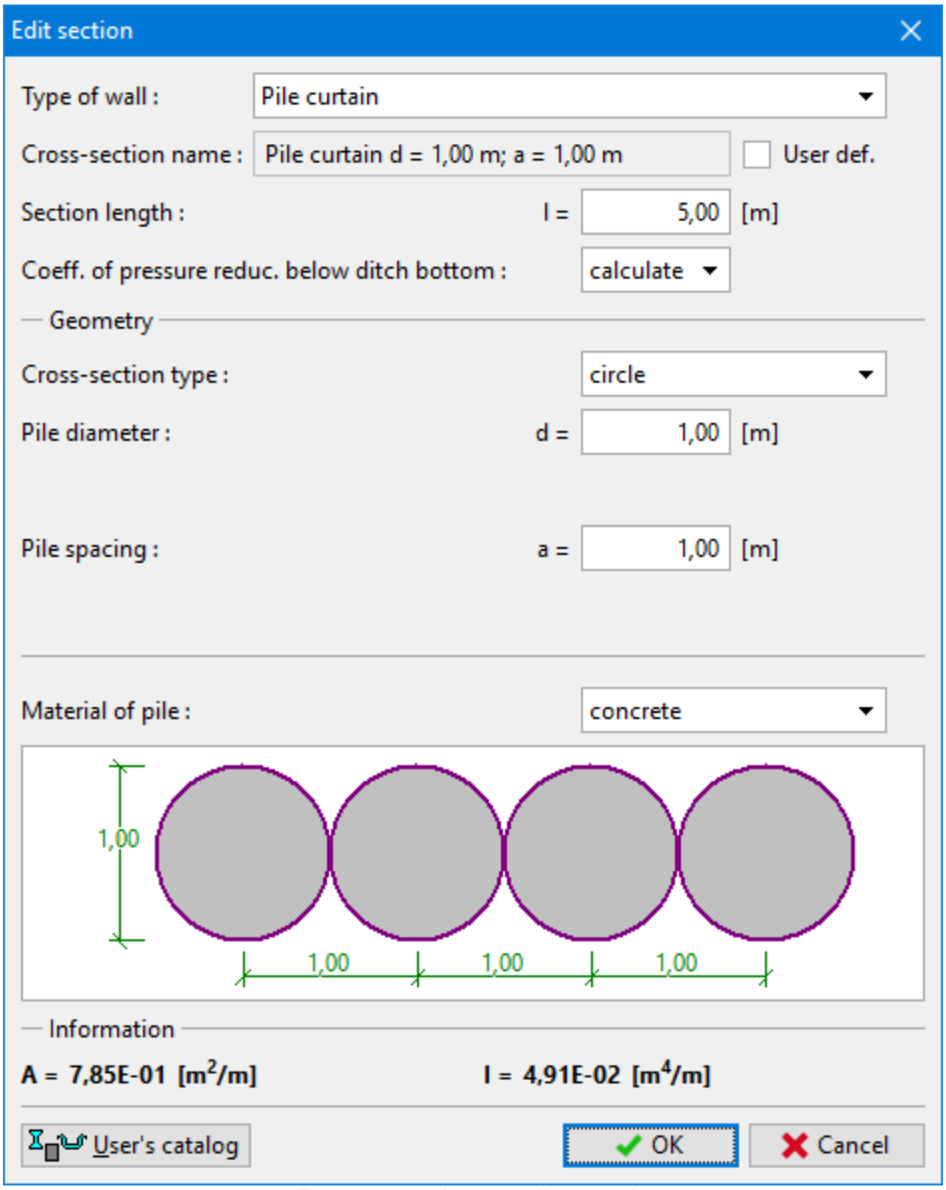


Dialog Window "Edit Section"

Pile Curtain

The **Pile curtain** requires input an of:

- **Cross-section name** (default name is generated by the program, it can be changed using "**User def.**" checkbox)
- **Section length** l
- **Coefficient of pressure reduction below ditch bottom** - this coefficient is used for reduction of pressures below ditch bottom for the calculation of braced sheeting - it can be either input or **calculated** automatically (for classical non-braced sheeting this coefficient is equal to 1.0)
- **Cross-section type** (circle, square)
- **Pile diameter** d and their **spacing** a
- **Material of pile** (concrete, wood)



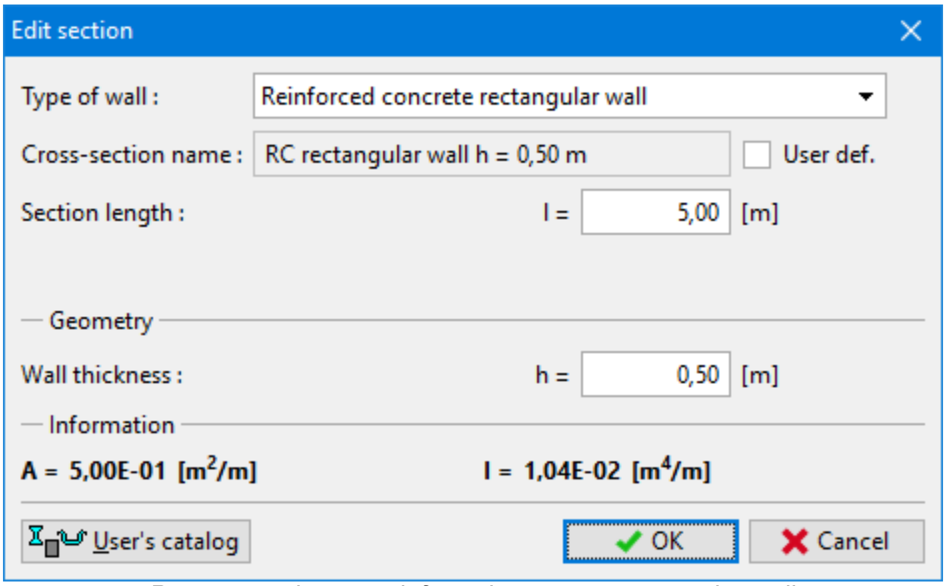
Parameters input - pile curtain

The verification of the bearing capacity of the RC section can be carried out in the "Dimensioning" frame.

Reinforced Concrete Rectangular Wall

The **Reinforced concrete rectangular wall** requires an input of:

- **Cross-section name** (default name is generated by the program, it can be changed using "**User def.**" checkbox)
- **Section length** l
- **Wall thickness** h



Parameters input - reinforced concrete rectangular wall

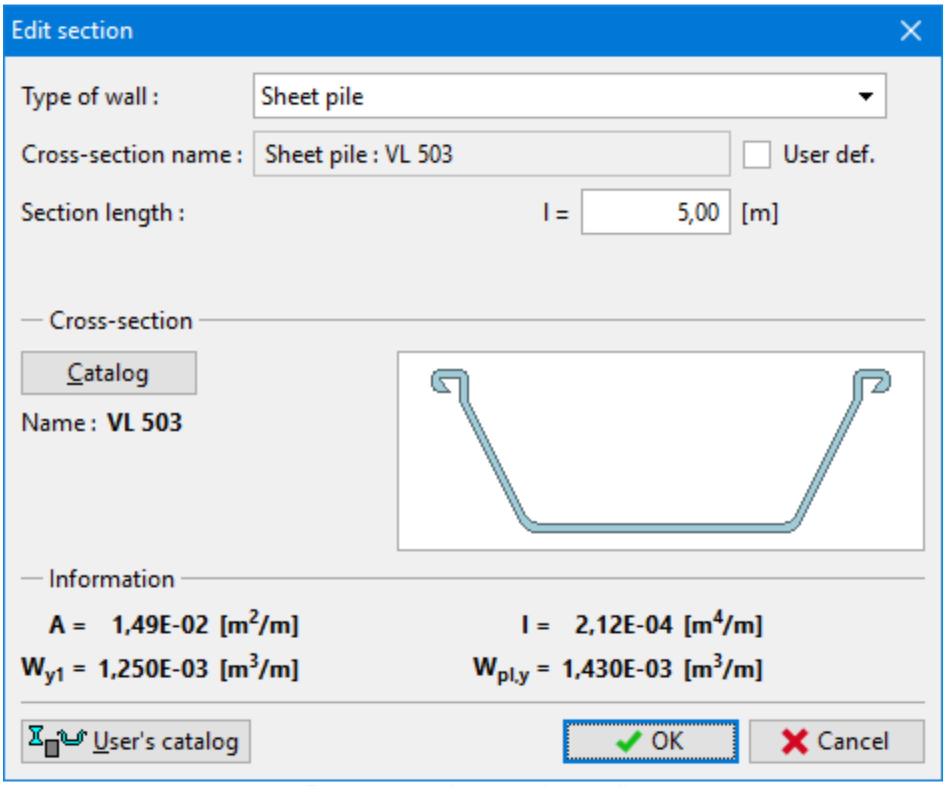
The verification of the bearing capacity of the RC section can be carried out in the "Dimensioning" frame.

Sheet Pile

The **Sheet pile** requires an input of:

- **Cross-section name** (default name is generated by the program, it can be changed using "**User def.**" checkbox)
- **Section length** l
- **Cross-section type** - it is selected in the dialog window "**Catalog of profiles**" (button "**Catalog**")

The catalog of profiles contains a lot of classes of sheet piles and casings from different manufacturers (see below). If you use different types of products, please contact us (hotline@fine.cz), we will implement it into our catalog in the program.



Parameters input - sheet pile

The verification of the bearing capacity of the steel section can be carried out in the "Dimensioning" frame.

Implemented sheet piles in our catalog

Steel sheet piles:

- *Vítkovice steel - Product catalog of sheet piles*

- *Arcelor Mittal - Steel Sheet Piling, General Catalogue 2012*
- *ThyssenKrupp Bautechnik - Sheet Piling Handbook*
- *Chinese standard GB/T 20933-2014 Hot rolled sheet pile*
- *Bethlehem Steel Sheet Piling*
- *Gerdau Steel Sheet Piling*

Steel sheet piles cold formed:

- *Mer Lion Metals - Cold Formed Steel Sheet Piles Catalogue*

Trench sheeting:

- *ThyssenKrupp Bautechnik - Kaltgewalzte Spundwandprofile – Kanaldielen*

Sheet piles O WOM/WOF:

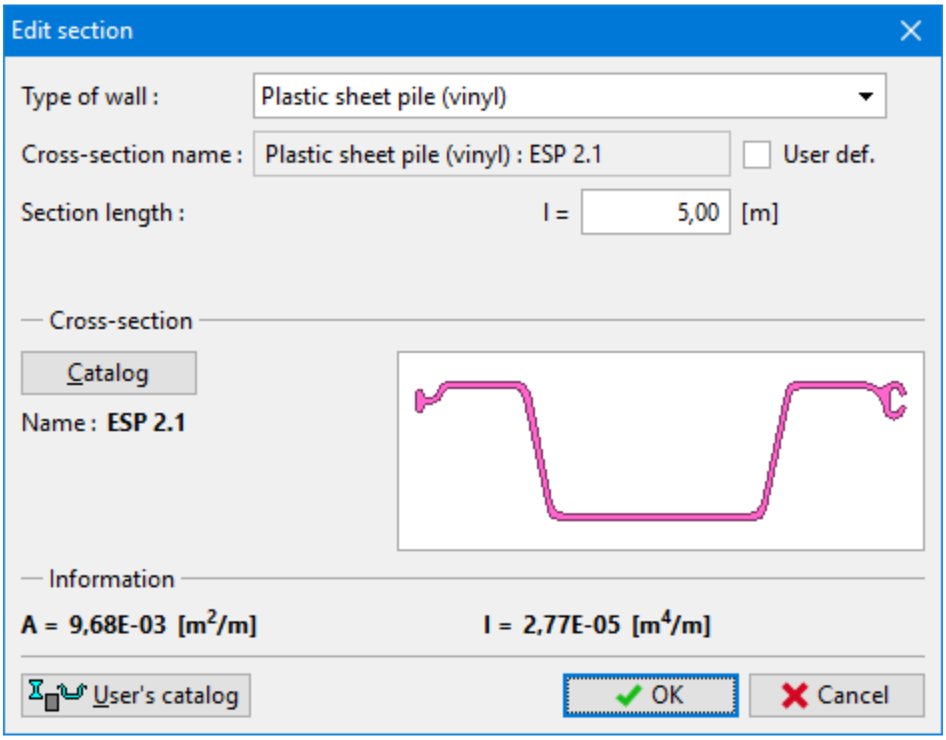
- *Mer Lion Metals - Steel Sheet Piles Catalogue*
- *PilePro Group Company - Sheet Pile Connectors*

Plastic Sheet Pile (Vinyl)

The **Plastic sheet pile** requires an input of:

- **Cross-section name** (default name is generated by the program, it can be changed using "**User def.**" checkbox)
- **Section length** l
- **Cross-section type** - it is selected in the dialog window "**Catalog of profiles**" (button "**Catalog**")

The catalog of profiles contains a lot of classes of sheet piles and casings from different manufacturers (see below). If you use different types of products, please contact us (hotline@fine.cz), we will implement it into our catalog in the program.



Parameters input - plastic sheet pile

Plastic sheet piles are very **soft for bending** and their deformations are much higher then deformation of steel or concrete structures. The standard method for determination of modulus of subgrade reaction (Schmitt) cannot be used for plastic material. If the Schmitt method is selected, the program automatically switches to the method **"Vinyl"**. In this case **modulus of subgrade reaction** is calculated from Oedometric modulus of soil - $k_h = 2/3\ E_{oed}$.

Other methods remain the same as for other types of cross-section.

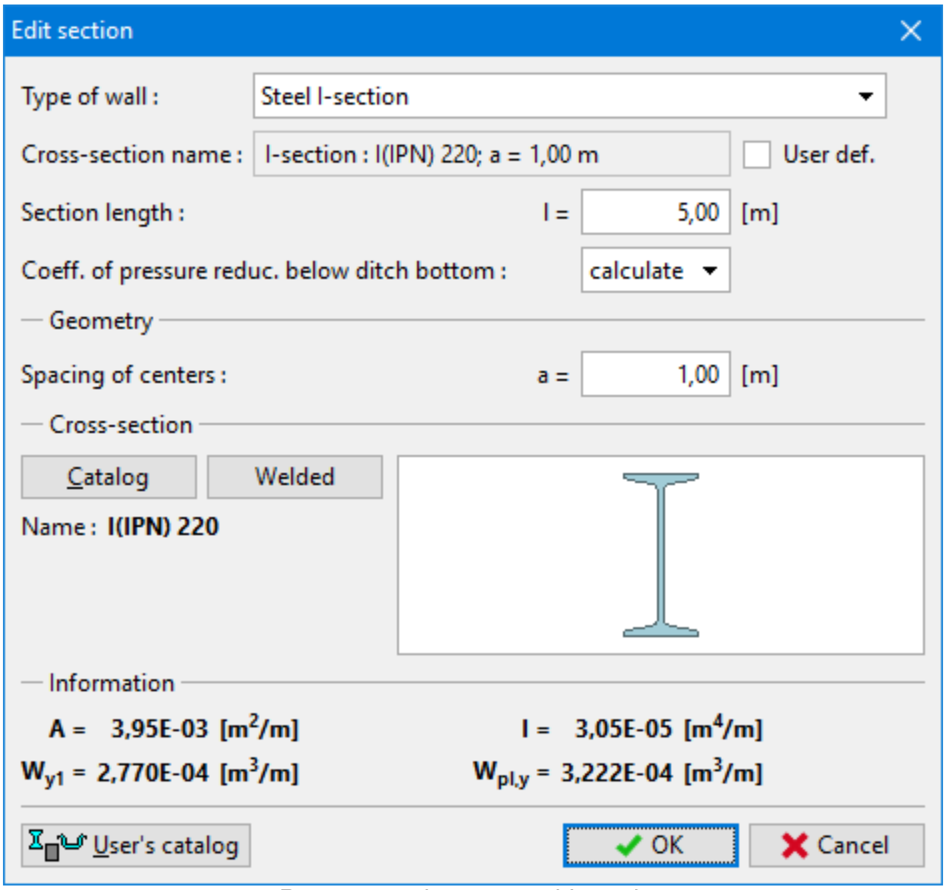
The verification of the bearing capacity of the plastic sheet pile can be carried out in the "Dimensioning" frame.

- *Katalog Everlast Synthetic Products, LLC - can be found at https://everlastseawalls.com/seawall-products/vinyl-sheet-piling*

Steel I-section

The **Steel I-section** requires an input of:

- **Cross-section name** (default name is generated by the program, it can be changed using "**User def.**" checkbox)
- **Coefficient of pressure reduction below ditch bottom** - this coefficient is used for the reduction of pressures below ditch bottom for calculation of braced sheeting - it can be either input or calculated automatically (for classical non-braced sheeting this coefficient is equal to 1.0)
- **Section length** l
- **Spacing of profiles**
- **Cross-section type** - it is selected in the dialog window "**Catalog of profiles**" (button "**Catalog**"), or in the dialog window "**Cross-section editor**" (button "**Welded**")



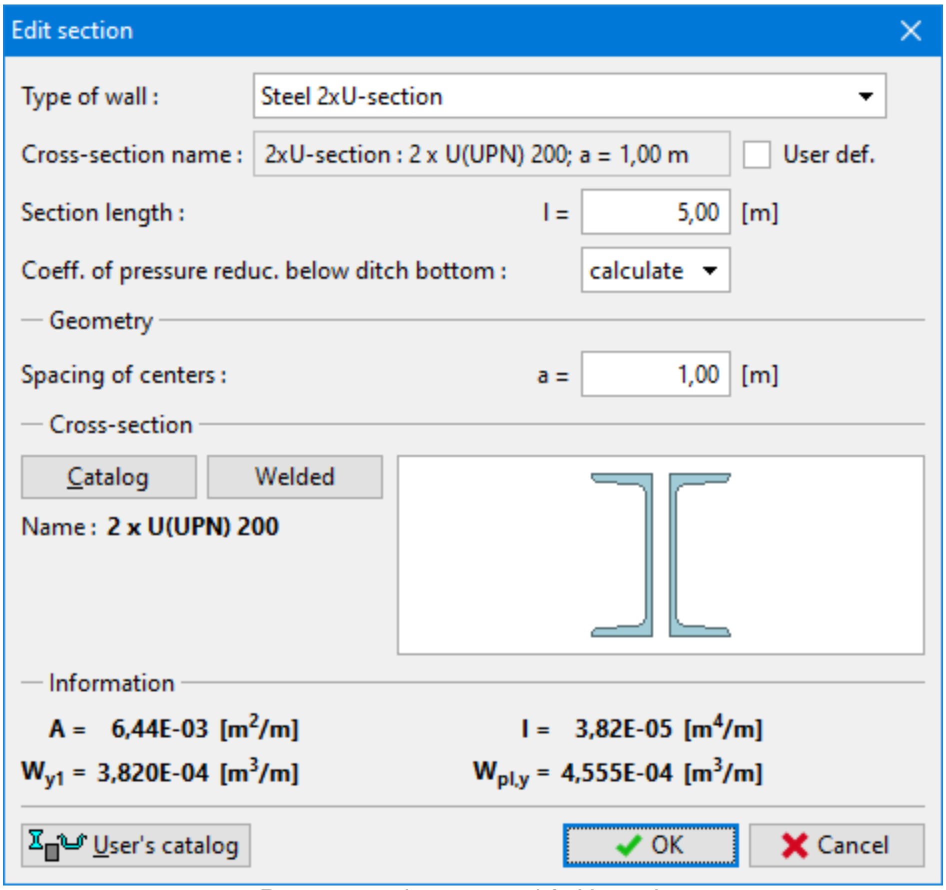
Parameters input - steel I-section

The verification of the bearing capacity of the steel section can be carried out in the "Dimensioning" frame.

Steel 2xU-section

The **Steel 2xU-section** requires input an of:

- **Cross-section name** (default name is generated by the program, it can be changed using "**User def.**" checkbox)
- **Coefficient of pressure reduction below ditch bottom** - this coefficient is used for the reduction of pressures below ditch bottom for calculation of braced sheeting - it can be either input or calculated automatically (for classical non-braced sheeting this coefficient is equal to 1.0)
- **Section length** l
- **Spacing of profiles**
- **Cross-section type** - it is selected in the dialog window "**Catalog of profiles**" (button "**Catalog**"), or in the dialog window "**Cross-section editor**" (button "**Welded**")



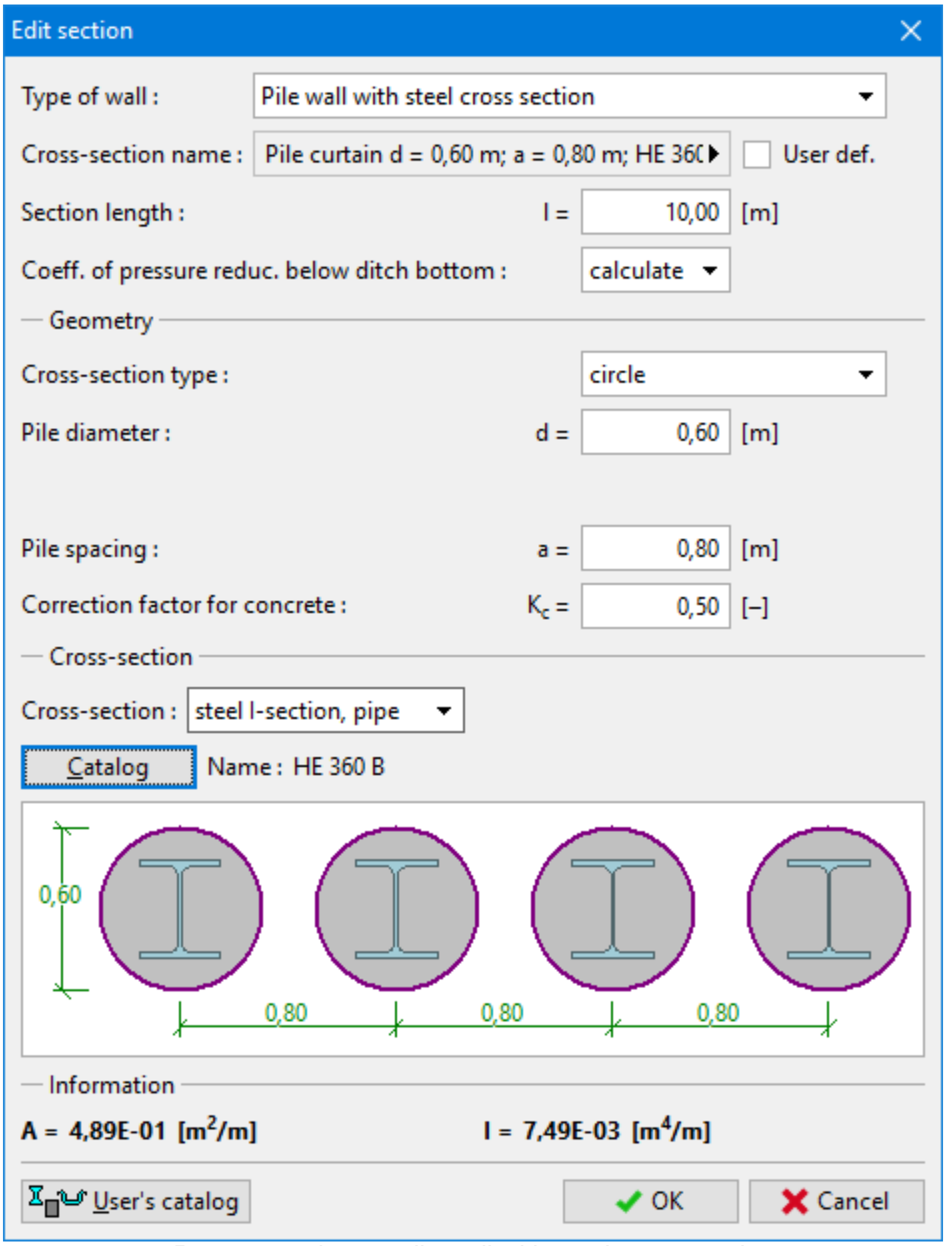
Parameters input - steel 2xU-section

The verification of the bearing capacity of the steel section can be carried out in the "Dimensioning" frame.

Pile Wall with Steel Cross Section

The **Pile wall with steel cross-section** requires an input of:

- **Cross-section name** (default name is generated by the program, it can be changed using "**User def.**" checkbox)
- **Section length** l
- **Coefficient of pressure reduction below ditch bottom** - this coefficient is used for the reduction of pressures below ditch bottom for calculation of braced sheeting - it can be either input or **calculated** automatically (for classical non-braced sheeting this coefficient is equal to 1.0)
- **Cross-section type** (circle, square)
- **Pile diameter** d and their **spacing** a
- **Steel cross-section** (I-section, pipe, 2xU-section or welded) is selected in the dialog window "**Catalog of profiles**" (button "**Catalog**")



Parameters input - pile wall with steel cross-section

For the calculation of cross-sectional characteristics, the steel section is converted to concrete.

$$I = \left(K_c I_c + I_s \frac{E_s}{E_c}\right)\frac{1}{a}$$

$$A = \left(A_c + A_s \frac{E_s}{E_c}\right)\frac{1}{a}$$

where:

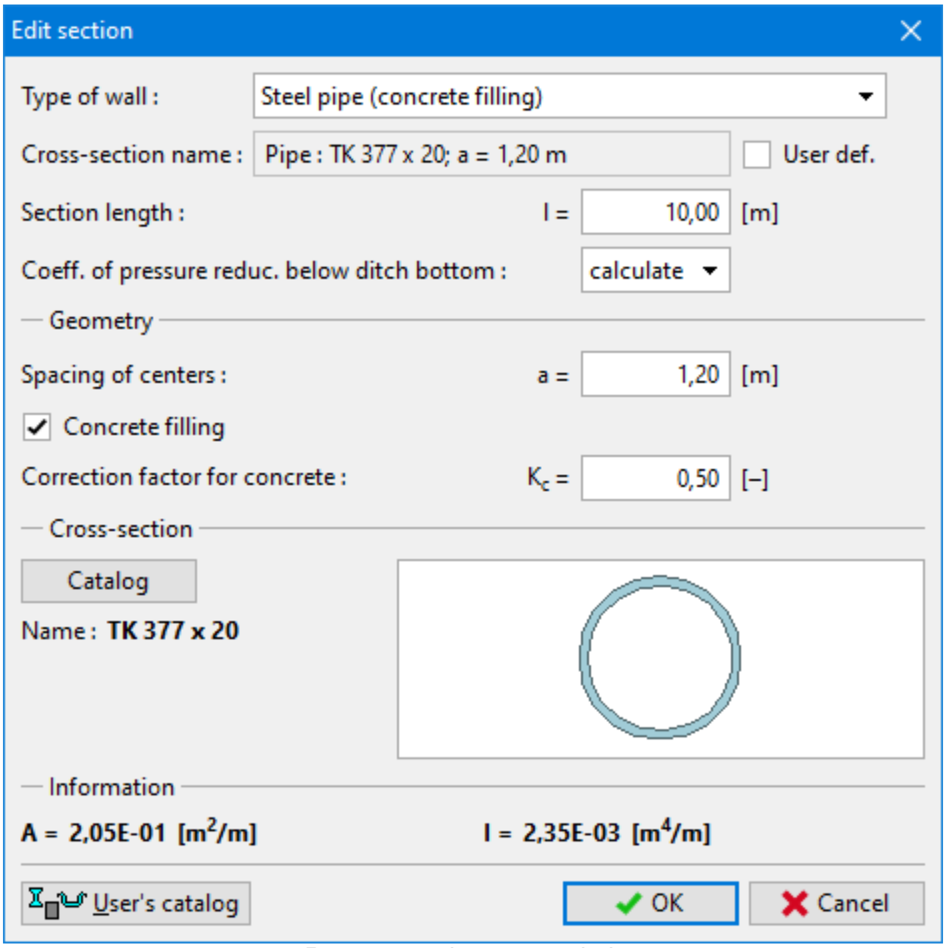
I_c	-	moment of inertia of the concrete cross-section
E_c	-	elasticity modulus of the concrete
K_c	-	correction factor for concrete
I_s	-	moment of inertia of the steel cross-section
E_s	-	elasticity modulus of steel
A_c	-	cross-sectional area of concrete
A_s	-	cross-sectional area of steel
a	-	pile spacing

The verification of the bearing capacity of the combined section can be carried out in the "Dimensioning" frame.

Steel Pipe

The **Steel pipe** requires an input of:

- **Cross-section name** (default name is generated by the program, it can be changed using "**User def.**" checkbox)
- **Coefficient of pressure reduction below ditch bottom** - this coefficient is used for the reduction of pressures below ditch bottom for calculation of braced sheeting - it can be either input or **calculated** automatically (for classical non-braced sheeting this coefficient is equal to 1.0)
- **Section length** l
- **Spacing of profiles** a
- **Correction factor for concrete** K_c (pipes filled with concrete)
- **Cross-section type** is selected in the dialog window "**Catalog of profiles**" (button "**Catalog**")



Parameters input - steel pipe

For the calculation of cross-sectional characteristics of **steel pipes filled with concrete**, the steel section is converted to concrete.

$$I = \left(K_c I_c + I_s \frac{E_s}{E_c} \right) \frac{1}{a}$$

$$A = \left(A_c + A_s \frac{E_s}{E_c} \right) \frac{1}{a}$$

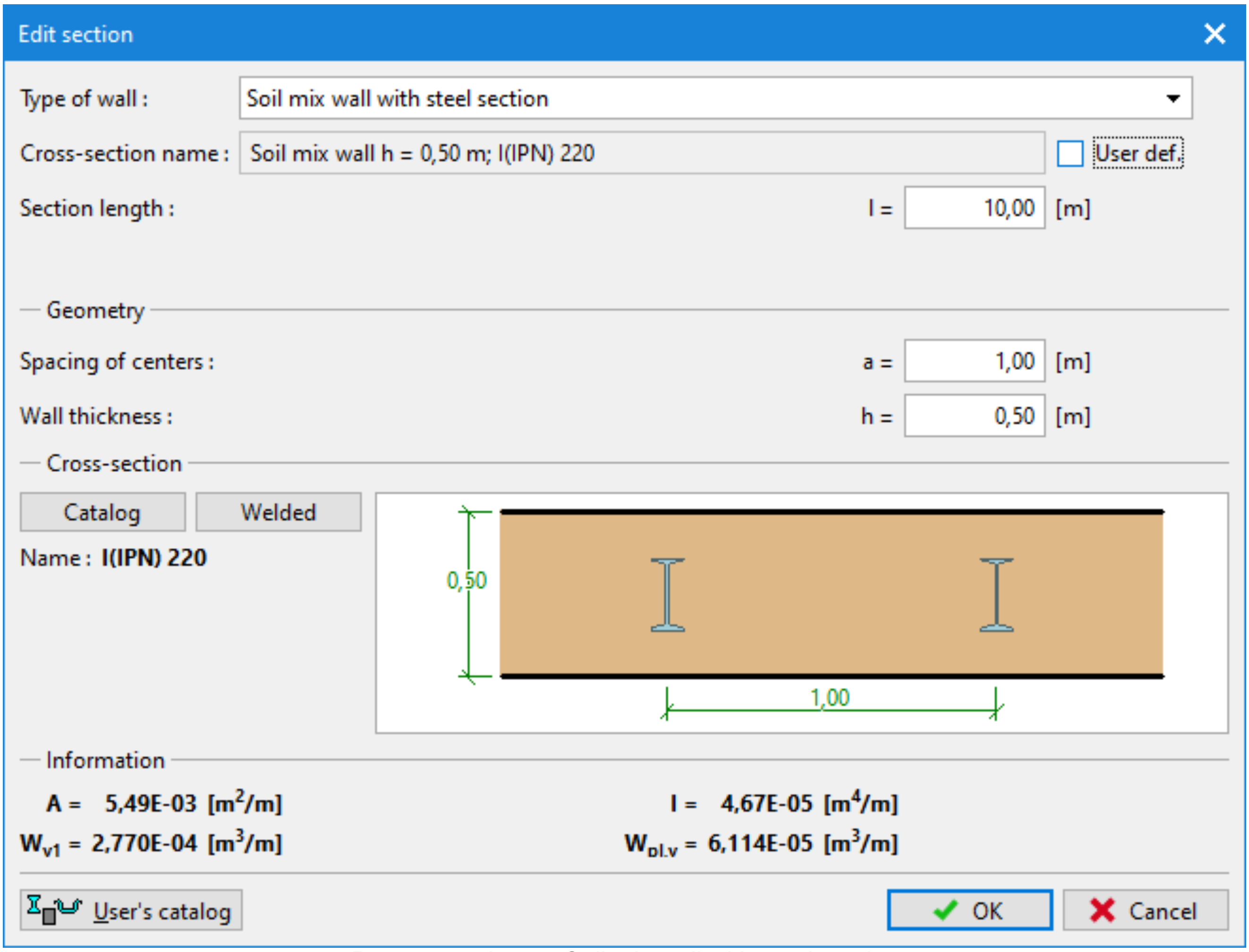
where: I_c - moment of inertia of the concrete cross-section
E_c - elasticity modulus of the concrete
K_c - correction factor for concrete
I_s - moment of inertia of the steel cross-section
E_s - elasticity modulus of steel
A_c - cross-sectional area of concrete
A_s - cross-sectional area of steel
a - pile spacing

The verification of the bearing capacity of the steel resp. combined section can be carried out in the "Dimensioning" frame.

Soil Mix Wall with Steel Section

The **Soil mix wall with steel section** requires an input of:

- **Cross-section name** (default name is generated by the program, it can be changed using "**User def.**" checkbox)
- **Section length** l
- **Spacing of profiles** a
- **Wall thickness** h
- **Cross-section type** - it is selected in the dialog window "**Catalog of profiles**" (button "**Catalog**"), or in the dialog window "**Cross-section editor**" (button "**Welded**")



Parameters input - Soil mix wall with steel section

The steel profiles are the main static elements. The verification of steel profiles is done in the "Dimensioning" frame.

The soil mix retains the soil between steel profiles. The verification of soil mix is done in the "Soil Mix" frame.

The soil mix also increases the bending stiffness of the shoring structure according to the formula:

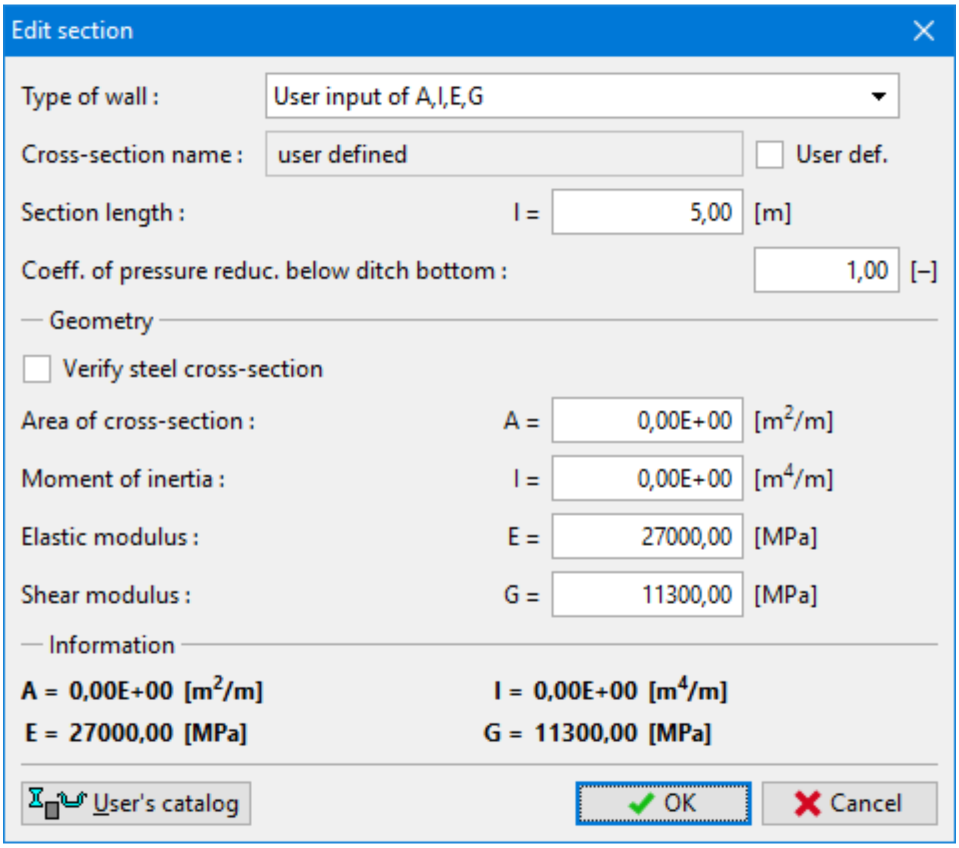
$$EI = EI_s + E_{SM} . \frac{a.\left(\frac{h}{2}\right)^3}{3}$$

where:

EI_S	-	bending stiffness of steel section
E_{SM}	-	modulus of elasticity of soil mix
a	-	spacing of profiles
h	-	wall thickness

User Input of A, I, E, G

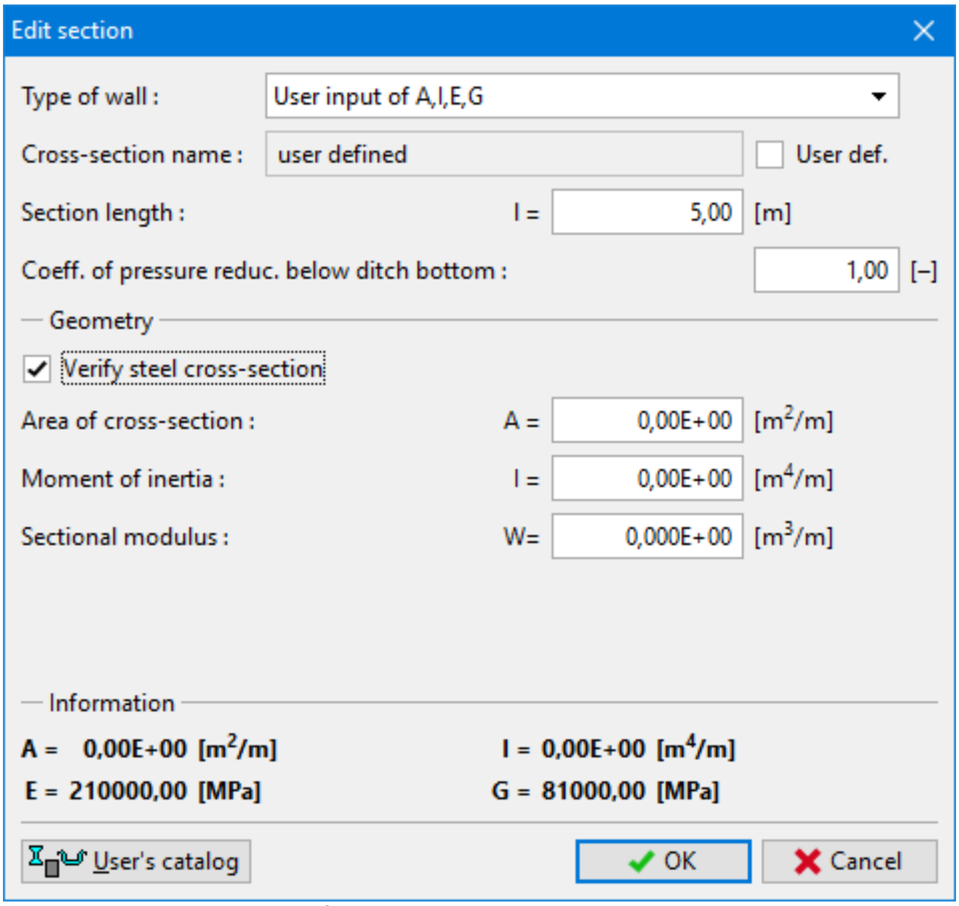
In the case of user input of geometry, it is necessary to enter values of sectional (A ,I) and material (E ,G) properties.



Input of sectional (A, I) and material properties (E, G)

In case of verification of steel cross-section (checkbox "**Verify steel cross-section**") the input of value of the sectional modulus (W) is required. Material parameters (E , G) are then transferred from the frame "**Material**".

The cross-section can be verified only for **bending and compression** in the "Dimensioning" frame.



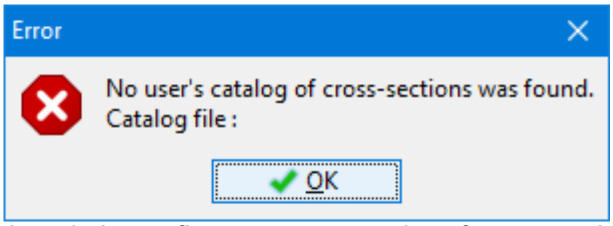
Input of sectional and material properties

User Catalog

The user catalog allows the user to define and store user-defined cross-sections and their characteristics that appear in a sheeting structure. At first use of the catalog (has not been yet created) the program prompts a warning message that no catalog was found. Then, pressing the button "**OK**" opens the "**Save as**" dialog window that allows us to enter the catalog name and saving it into a specified location by pressing the "**Save**" button (by default a folder used for saving the project data is assumed).

The program allows the user to create more than one catalog. The next catalog is created by pressing the "**New**" button - the program asks, whether the current catalog should be replaced (**the currently loaded catalog is not deleted!**) and saves the new catalog under a new name. The "**Open**" button allows loaduing an arbitrary user catalog and by pressing the "**Save as**" button for saving it under a different name.

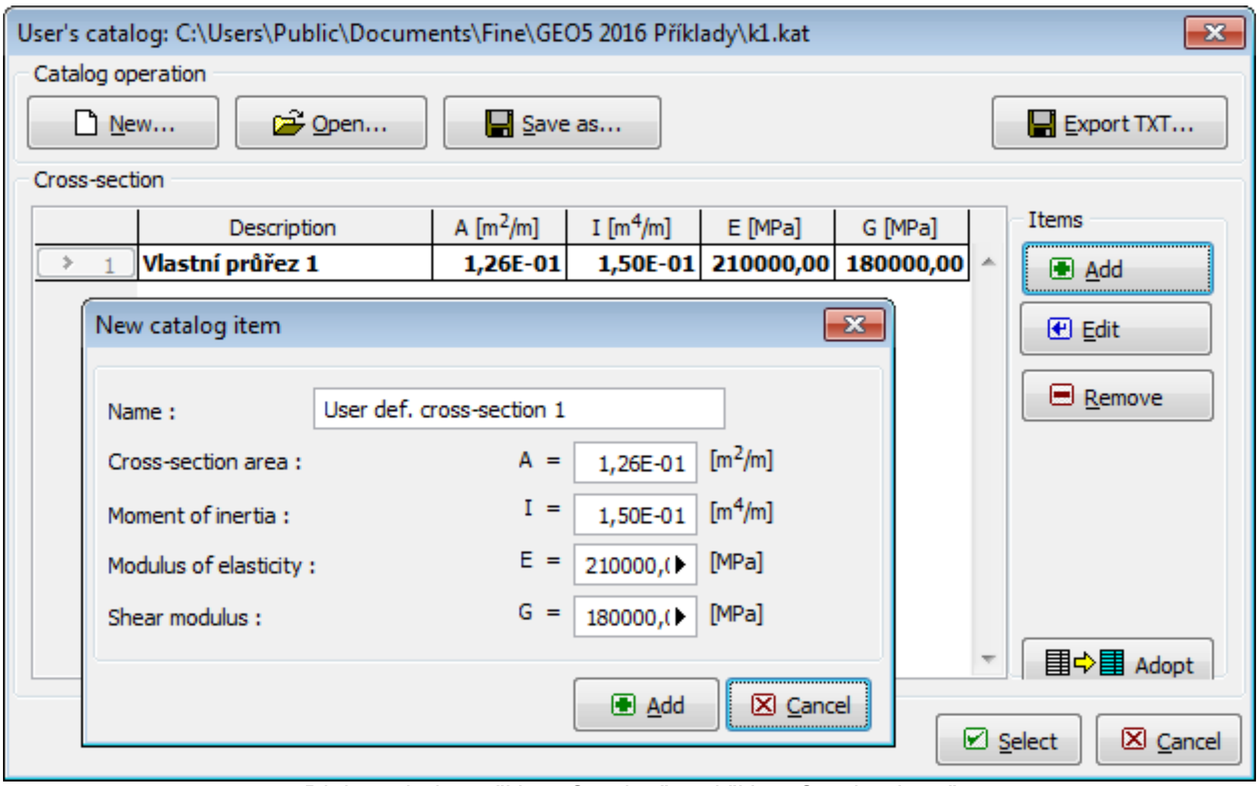
"**Export TXT**" button allows for exporting of the currently loaded user catalog to a text file.



Dialog window at first use - user catalog of cross-sections

The "**User's catalog**" dialog window contains a table listing the user-defined cross-sections. The "**Add item**" button opens the "**New catalog item**" dialog window that allows for specifying and subsequent saving of characteristics of a new cross-section into the catalog. Buttons "**Edit item**" and "**Remove item**" serve to edit individual items in the table.

The "**Accept current**" button accepts the current cross-sectional characteristics of a cross-section specified in the "New section" dialog window and opens the "**New catalog item**" dialog window that allows for modifying and saving the current cross-section.



Dialog windows "User Catalog" and "New Catalog Item"

Automatic Calculation of the Coefficient of Pressure Reduction Below Ditch Bottom

For **automatic calculation**, the coefficient of pressure reduction below ditch bottom k [-] is determined as follows:

- **circular pile curtain (a)**

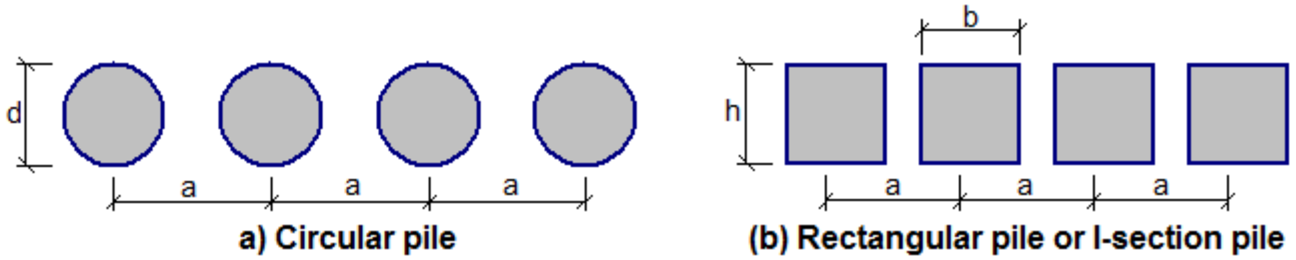
$$k = 0.9\,(1.5d + 0.5)/a \qquad (d \le 1\ m)$$
$$k = 0.9\,(d + 1)/a \qquad (d > 1\ m)$$

- **rectangular pile curtain or steel I-section (b)**

$k = (1.5b + 0.5)/a \qquad (b \leq 1\ m)$
$k = (b + 1)/a \qquad (b > 1\ m)$

Note: If coefficient $k > 1$, then $k = 1$.

where:
- d - pile diameter
- b - footprint of a rectangular pile, or flange width of steel I-section
- a - spacing of soldier beams, or spacing between piles



The coefficient of pressure reduction below ditch bottom k

Sheeting Design

Analyses in the program "**Sheeting design**" can be divided into three groups:

- Analysis of non-anchored walls (e.g. sheet pile wall)
- Analysis of anchored walls fixed at heel
- Analysis of anchored walls simply supported at heel

It is also possible to analyze soldier pile walls (braced sheeting) using this program.

Analysis of Sheet Pile Wall

A pile sheeting wall is analyzed using a standard approach that accounts for the effect of earth pressures. In general, the active earth pressure develops behind the structure while the passive earth pressure appears in front of the structure.

Based on the **theory of limit states** the program searches in an iterative way a point on the wall to satisfy the moment equation of equilibrium in the form:

$$M_{overturning} = M_{resisting}$$

Once this is accomplished, the program continues by determining the wall heel location for which the equilibrium of shear forces is fulfilled (computation of depth of fixed end). The overall length of the analyzed structure is found this way.

When applying approaches based on the **factor of safety** the program searches, in an iterative way, a point to get:

$$\frac{M_{resisting}}{M_{overturning}} = FS$$

It is obvious that the distribution of internal forces resulting from this approach is not very realistic. In some countries, however, this approach is required.

The computation can be driven either by choosing a minimal dimensioning pressure or by reduction of **passive pressure**. Assuming the actual magnitude of the passive earth pressure provides deformations of the analyzed structure, which cannot usually occur. The actual passive pressure can attain for walls free of deformation the value of pressure at rest as well as all intermediate values up to the value of passive pressure for fully deformed wall (rotation app. 10 $mRad$ - i.e. deformation 10 mm per 1 m of structure height). Therefore it is reasonable to consider reduced values of the passive earth pressure by setting the value of the "**Coefficient of reduction of passive pressure**" to less than or equal to one. The following values are recommended:

- 0.67 reduces deformations app. by one half
- 0.50 approximately corresponds to a deformation of a structure loaded by an increased active earth pressure
- 0.33 approximately corresponds to a deformation of a structure loaded by the pressure at rest, structure reaches app. 20 percent of its original deformations

Analysis of Anchored Wall Fixed in Heel

Anchored wall fixed in the heel is analyzed as a continuous beam using the deformation variant of the finite element method in order to comply with the assumption of heel fixed in the soil. The actual analysis is preceded by the determination of load due to earth pressure applied to the structure. The pressure acting on the back of a structure is assumed to be the active pressure, while the front face is loaded by the passive pressure.

The passive pressure can be reduced with the help of the **coefficient of reduction of passive pressure**. Assuming the actual magnitude of the passive earth pressure provides deformations of the analyzed structure, which cannot usually occur. The actual passive pressure can attain for walls free of deformation the value of pressure at rest as well as all intermediate values up to the value of passive pressure for a fully deformed wall (rotation app. 10 $mRad$ - i.e. deformation 10 mm per 1 m of structure height). Therefore it is reasonable to consider reduced values of the passive earth pressure setting the value of the "**Coefficient of reduction of passive pressure**" to less than or equal to one. The following values are recommended:

- 0.67 reduces deformations app. by one half
- 0.33 deformations attain approximately twenty percent of their original values

The program offers two options to **determine active pressure**:

- calculation from input soil parameters, water, surcharge, terrain including the introduction of the minimum dimensioning pressure
- inputting an arbitrary distribution of earth pressure up to the depth of the zero point (this way it is possible to introduce an arbitrary redistribution of earth pressure)

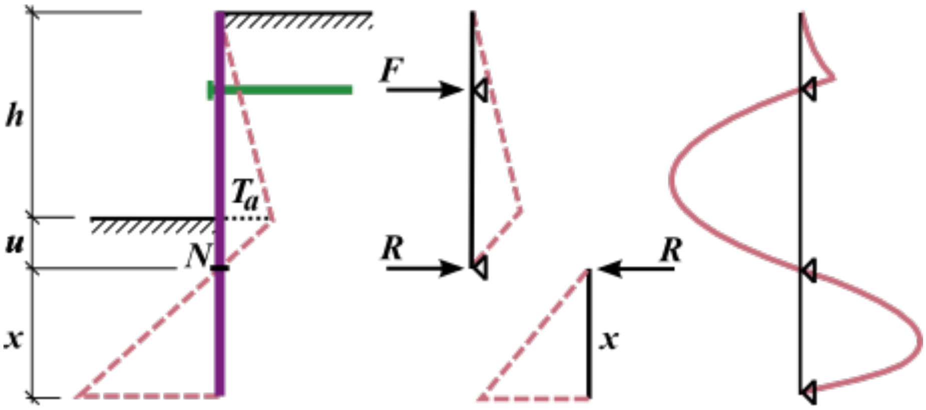
Zero-value point, i.e. the point at which the overall pressure equals zero is determined by the following expression:

$$u = \frac{\sigma_a}{\gamma . K}$$

where:

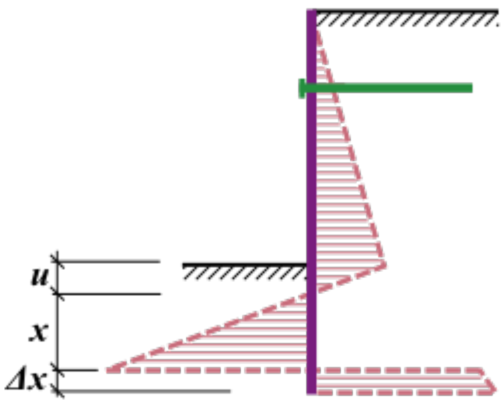
u	-	depth of the zero-value point
σ_a	-	magnitude of the active pressure behind the structure at the ditch bottom
K	-	coefficient of the overall pressure
γ	-	unit weight of the soil below the ditch bottom

The analysis of structure fixed at heel assumes that the point of zero load N (at depth u) is identical with the point of zero moment. For the actual analysis the structure is divided into two parts - an upper part (upper beam) up to the zero-value point and a lower beam:



Analysis of anchored wall fixed in heel

The upper beam is analyzed first together with the evaluation of anchor forces F and the reaction force R at the zero-value point. Then, the lower beam length x is determined such that the moment equilibrium condition with respect to the heel is satisfied (the beam is loaded by the reaction R and by the difference of earth pressures). To satisfy the shear force equilibrium the computed length of the fixed end is extended by the value Δx as shown in the figure:



Determination of the extension of the length of the wall by Δx

Analysis of Anchored Wall Simply Supported at Heel

Anchored wall simply supported at the heel is analyzed as a continuous beam using the deformation variant of the finite element method in order to comply with the assumption of a simply supported structure at the heel. The actual analysis is preceded by the determination of load due to earth pressure applied to the structure. The pressure acting on the back of a structure is assumed as active pressure, while the front face is loaded by passive pressure.

The passive pressure can be reduced with the help of the **coefficient of reduction of passive pressure**. Assuming the actual magnitude of the passive earth pressure provides deformations of the analyzed structure, which cannot usually occur. The actual passive pressure can attain for walls free of deformation the value of pressure at rest as well as all intermediate values up to the value of passive pressure for a fully deformed wall (rotation app. 10 $mRad$ - i.e. deformation 10 mm per 1 m of structure height). Therefore it is reasonable to consider reduced values of the passive earth pressure setting the value of the "**Coefficient of reduction of passive pressure**" to less than or equal to one. The following values are recommended:

- 0.67 reduces deformations app. by one half
- 0.33 deformations attain approximately twenty percent of their original values

The program offers two options to **determine active pressure**:

- calculation from input soil parameters, water, surcharge, terrain including the introduction of the minimum dimensioning pressure
- inputting an arbitrary distribution of earth pressure up to the depth of zero point (this way it is possible to introduce an arbitrary redistribution of earth pressure).

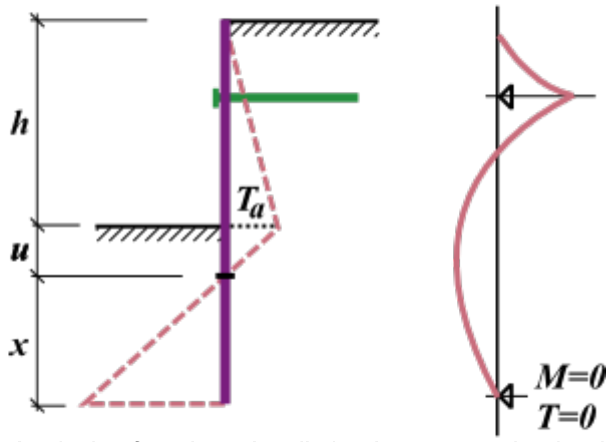
Zero-value point, i.e. the point at which the overall pressure equals zero is determined by the following expression:

$$u = \frac{\sigma_a}{\gamma . K}$$

where:

u	-	depth of the zero-value point
σ_a	-	magnitude of the active pressure behind the structure at the ditch bottom
K	-	coefficient of the overall pressure
γ	-	unit weight of the soil below the ditch bottom

For simply supported structures it is assumed that the moment and shear force are zero at the heel. The program first places the end of a structure into the zero-value point, and then it looks for the end beam location x , where the above condition is fulfilled (see Fig.). The solution procedure for a multiplied anchored walls is identical.



Analysis of anchored wall simply supported at heel

Sheeting Check

The program verifies the input structure using the method of dependent pressures or using the spring method according to JGJ 120-2012. The load applied to the structure is derived from its deformation, which allows us to realistically model its behavior and provides cost-effective designs. The analysis correctly accounts for the **construction process** such as individual stages of progressive construction of the wall (**stages of constructions**) including an gradual evolution of the deformations and a post-stressing of the anchors. The program can model any kind of braced sheeting too.

The use of the method of dependent pressures requires determination of the modulus of subsoil reaction, which is assumed either **linear** or nonlinear.

The program also allows the user to check internal stability of the anchorage system.

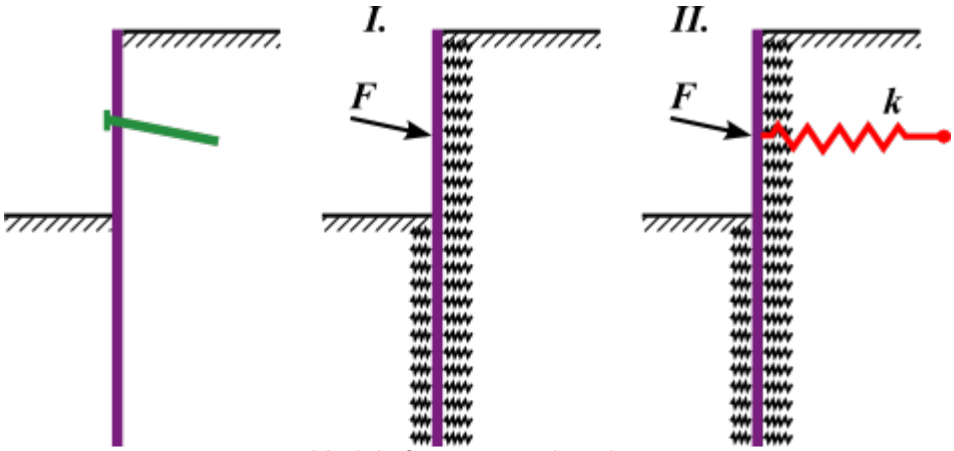
The **actual analysis** is carried out using the deformation variant of the finite element method. Displacements, internal forces, and the modulus of the subsoil reaction are evaluated at individual nodes.

The following procedure for dividing the structure into finite elements is assumed:

- First, the nodes are inserted into all topological points of a structure (starting and endpoints, points of location of anchors, points of soil removal, points of changes of cross-sectional parameters).
- Based on selected subdivision the program computes the remaining nodes such that all elements attain approximately the same size.

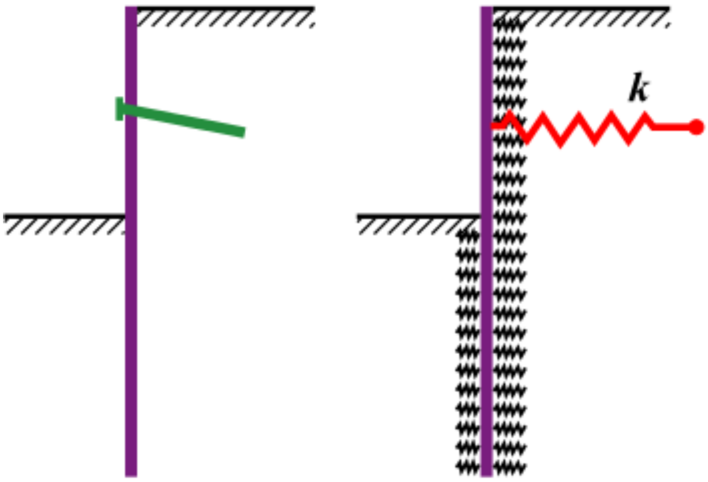
A value of the modulus of subsoil reaction is assigned to each element - it is considered as the Winkler spring of the elastic subsoil. **Supports** are placed onto already **deformed structure** - each support then represents a forced displacement applied to the structure.

In the construction stage, where are introduce**, prestressed anchors** are modeled as a force. (variant I in Fig). In other construction stages the anchors are modeled springs of stiffness k (variant II. in Fig) and force:



Model of prestressed anchor

Non-prestressed anchors are model always as spring in all construction stages**.** The force in the anchor is computed from the deformation of the structure and the anchor stiffness:



Model of non-prestressed anchor

The change of anchor force due to deformation is provided by:

$$\Delta F = \frac{k.v.\Delta w}{\cos \alpha}$$

$$k = \frac{E.A}{l}$$

where:

v	-	horizontal distance between the anchors
Δw	-	deformation increment the point of anchor application
E	-	anchor Young's modulus
A	-	anchor cross-sectional area
l	-	anchor length
k	-	anchor stiffness

α - anchor inclination

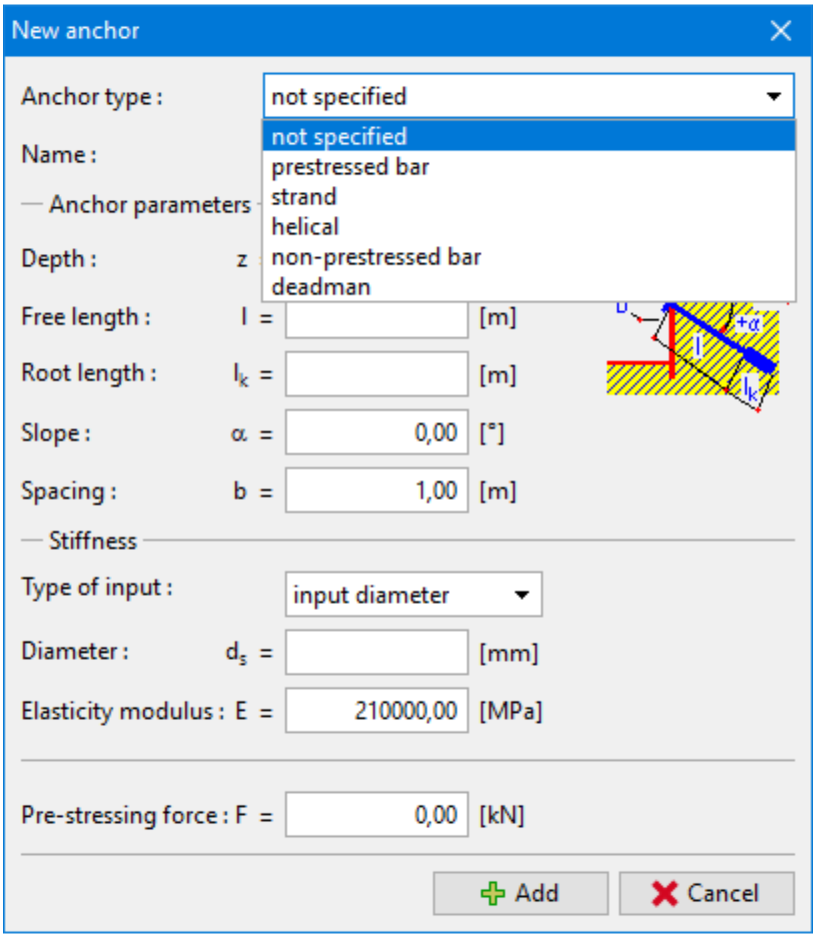
Literature:

Hurych, P.: Metoda zavislych tlaku. Sbornik konference "Automatizacia projektovania", Vysoke Tatry, 1978.

Anchor Types

The program allows us to enter prestressed and non-prestressed anchors and calculate their bearing capacity.

If the verification of anchor is not required, anchor type can be selected as "**not specified**". In this case, we just enter only information necessary for the analysis of the entire structure (geometry, stiffness).



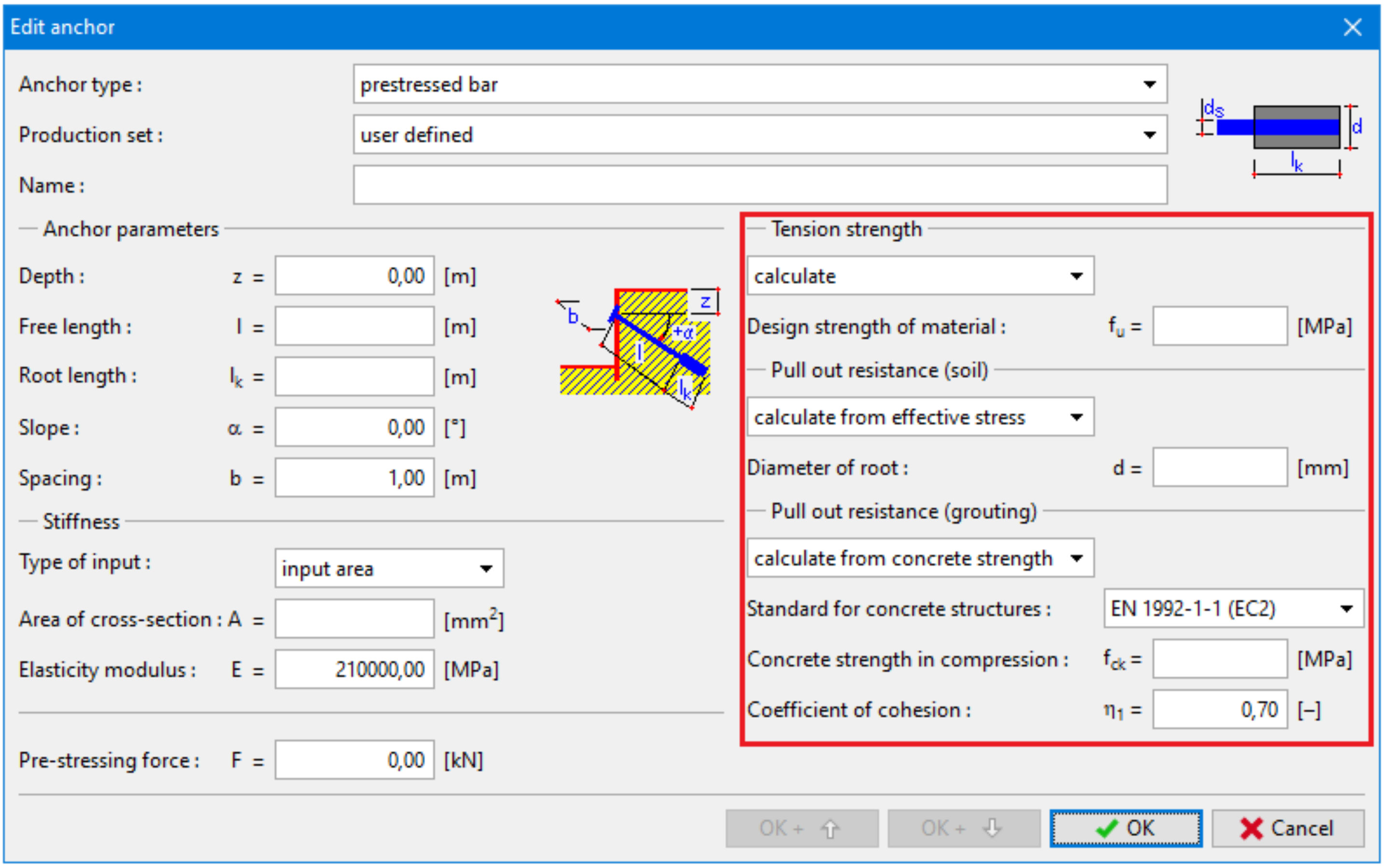
*Anchor input - **without** verification of bearing capacity*

The program allows us to enter and verify these types of anchors:

- prestressed bars
- strand
- helical
- non-prestressed bars
- deadman

The anchors can be check for three different **types of failure**.

- Strength of anchor R_t (all types)
- Pull-out resistance (soil) R_e (bars and strand)
- Pull-out resistance (grouting) R_c (prestressed bars and strand)



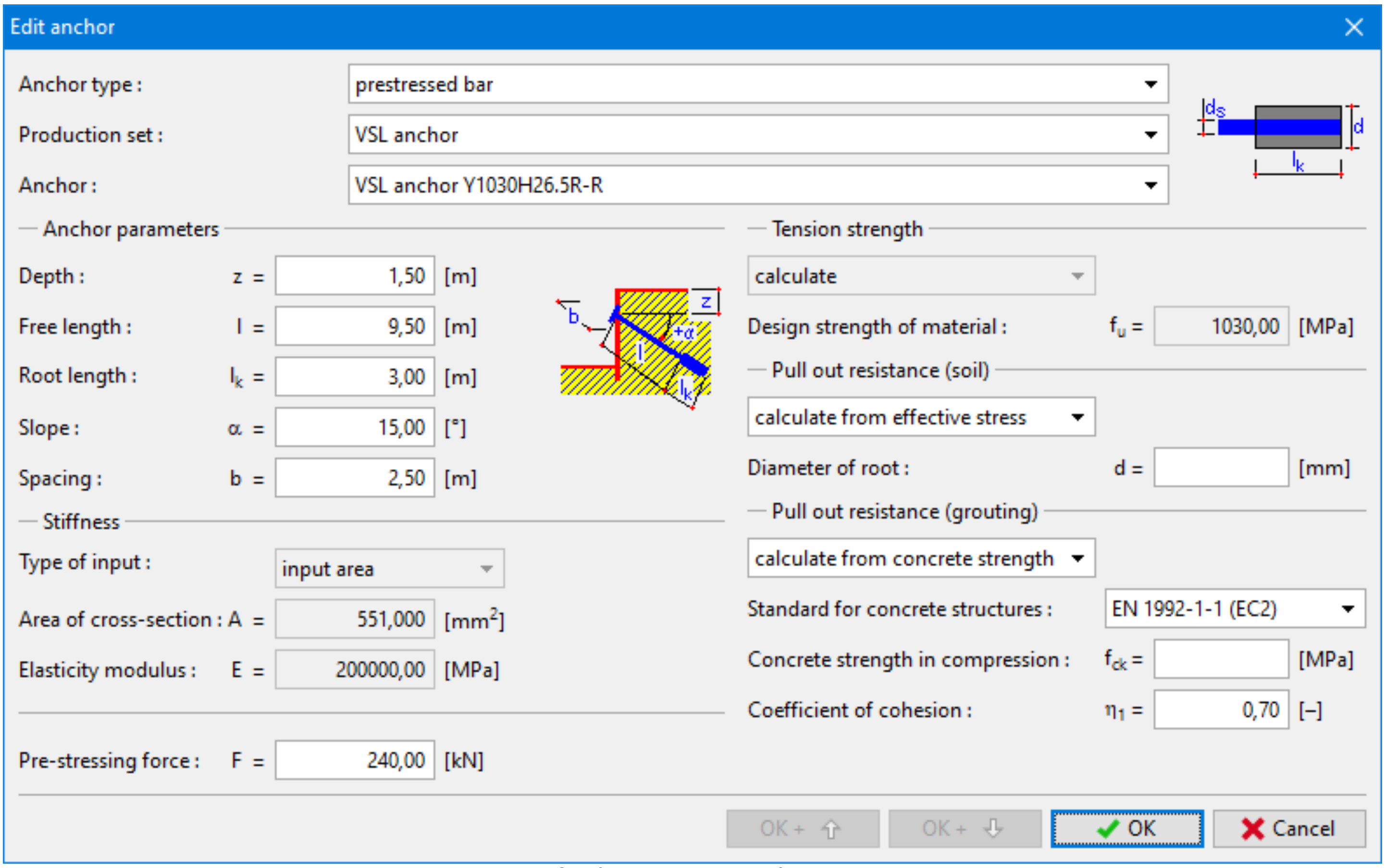
Anchor input - verification of bearing capacity

Prestressed Bar

The "**Prestressed bar**" is selected as an "**Anchor type**" in the combo list in the dialog window "**New anchor**".

The **Prestressed bar** anchor requires the input of:

- **Production set** - user-defined (the user input of parameters) or the anchor from the catalog (VSL, DYWIDAG...)
- **Location of anchor** - depth, length, root length, slope...
- **Stiffness of anchor** - anchor diameter d_s or area of anchor A , elasticity modulus E
- **Pre-stressing force**
- **Parameters for verification of anchor** (Strength of anchor, Pull-out resistance (soil), Pull-out resistance (grouting))



Anchor parameters input

Literature:

Implemented anchors in the catalog

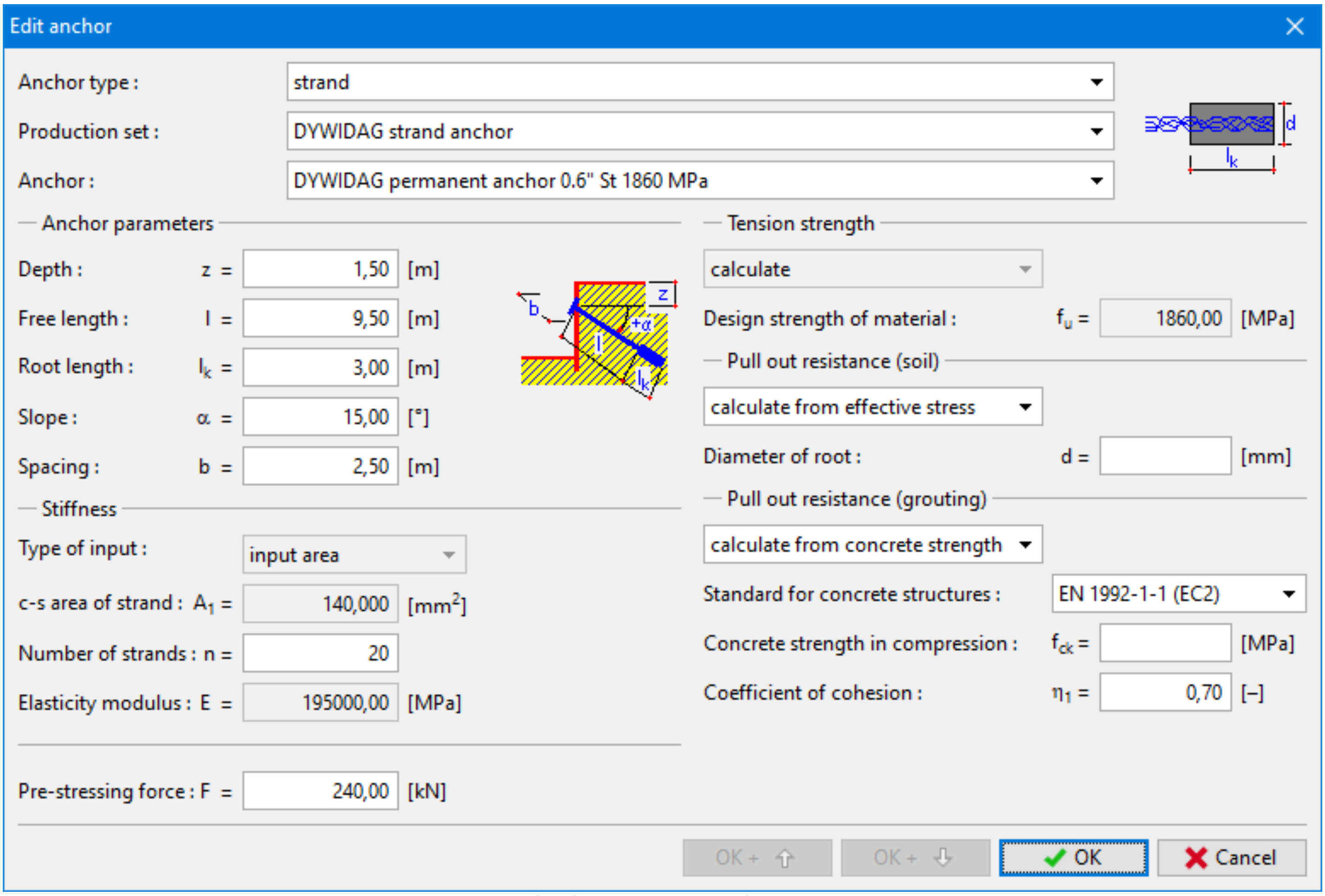
- *VSL Systems Ltd. - VSL Ground Anchor Systems - web brochure*
- *Dywidag- Systems International - Dywidag Bar Anchors, Dywidag Strand anchors - web brochures*

Strand Anchor

The "**Strand anchor**" is selected as an "**Anchor type**" in the combo list in the dialog window "**New anchor**".

The **Strand** anchor requires the input of:

- **Production set** - user-defined (the user input of parameters) or the anchor from the catalog (VSL, DYWIDAG...)
- **Location of anchor** - depth, length, root length, slope...
- **Stiffness of anchor** - diameter of strand d_1 (or area of strand A_1), Number of strands n and elasticity modulus E
- **Pre-stressing force**
- **Parameters for verification of anchor** (Strength of anchor, Pull-out resistance (soil), Pull-out resistance (grouting))



Anchor parameters input

Literature:

Implemented anchors in the catalog

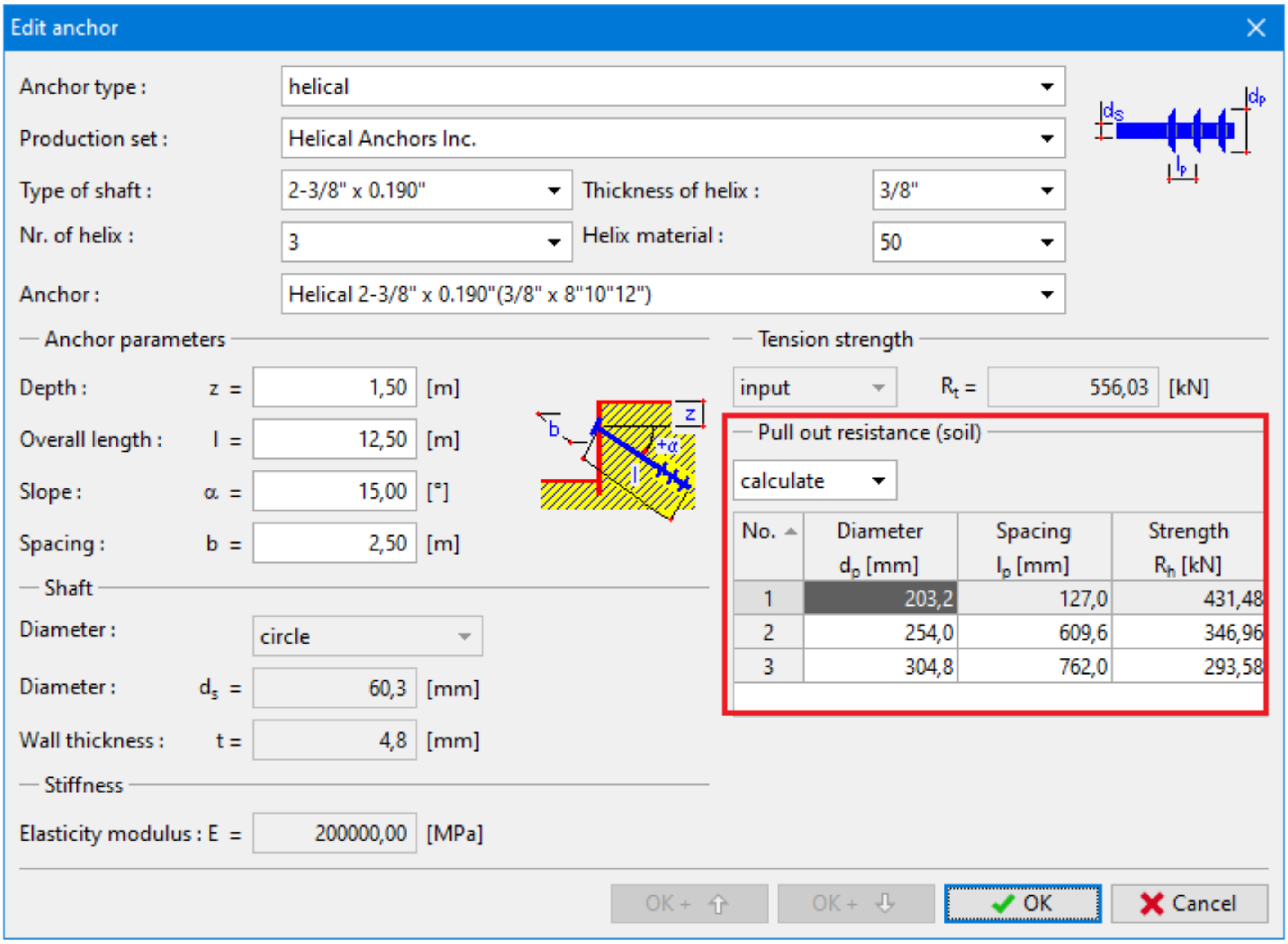
- *VSL Systems Ltd. - VSL Ground Anchor Systems - web brochure*
- *Dywidag- Systems International - Dywidag Bar Anchors, Dywidag Strand anchors - web brochures*

Helical Anchor

The "**Helical anchor**" is selected as an "**Anchor type**" in the combo list in the dialog window "**New anchor**".

The **Helical** anchor requires the input of:

- **Production set** - user-defined (the user input of parameters) or the anchor from the catalog (Helical Anchors, Mac Lean, Chance...). The anchors from the catalog are specified by the other parameters (**type of shaft, Nr. of helix, thickness, and material of helix**) - if the selected combination is **not manufactured**, the **name of the anchor is empty** and editing **cannot be finished**
- **Location of anchor** - depth, length, slope...
- **Type of shaft profile** - circle (pipe), square
- **Stiffness of anchor** - elasticity modulus E
- **Parameters for verification of anchor** (Strength of anchor, Pull-out resistance from the soil is described below)



Anchor parameters input

Calculation of pull out resistance from soil:

Option "input"

This option is required for user-defined anchors. It is necessary to enter the value of pull out resistance from soil R_e $[kN, lbf]$. The table with dimensions and locations of each helix is only for the visualization of the anchor and internal stability check.

Option "calculate"

This option is available only for anchors from the catalog. The dimensions and locations of each helix are determined by the manufacturer. The pull out resistance from the soil is calculated according to this formula:

$$R_e = \sum_{i=1}^{n} \min\left(R_{h,i}\ ;\ A_i\left(c\,N_c + \sigma_z\,N_q\right)\right)$$

where:

n	-	number of the helix
$R_{h,i}$	-	maximal bearing capacity of the helix, determined by the manufacturer
c	-	cohesion of the soil
σ_z	-	geostatical stress
N_c, N_q	-	Meyerhof´s coefficients for bearing capacity
A_i	-	the area of helix

The bearing capacity coefficients can be calculated in **standard** way:

$$N_q = e^{\pi * \tan\varphi}\ \tan(45 + \frac{\varphi}{2})^2$$

$$N_c = \left(N_q - 1\right)\cot\varphi$$

where: φ - angle of internal friction (for $\varphi = 0$ is $N_c = 2 + \pi$)

or according to **CHANCE Design Manual**:

$$N_q = 0.5(12\varphi)^{\varphi/54}$$
$$N_c = 0$$

Literature:

Chance - Technical Design Manual, Edition 4

Implemented anchors in the catalog

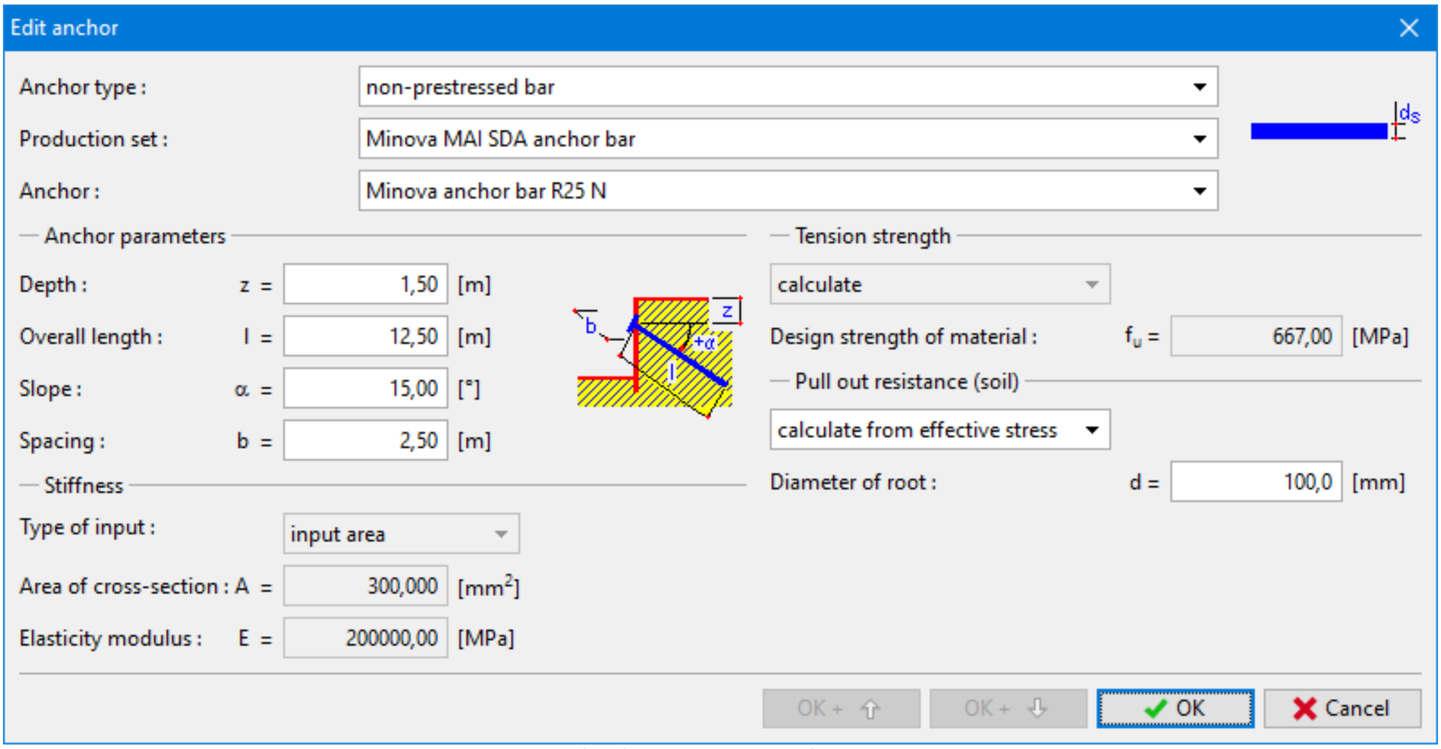
- Helical Anchors, Inc. - Product Catalog
- MacLean-Dixie - Helical Foundation Syastems - Engineering Reference Manual
- Chance - Civil Construction Product Catalog, Vol. I.

Non-Prestressed Bar

The "**Non-prestressed bar**" is selected as an "**Anchor type**" in the combo list in the dialog window "**New anchor**".

The **non-prestressed bar** anchor requires the input of:

- **Production set** - user-defined (the user input of parameters) or the anchor from the catalog (VSL, DYWIDAG...)
- **Location of anchor** - depth, length, slope...
- **Stiffness of anchor** - anchor diameter d_s or area of anchor A , elasticity modulus E
- **Parameters for verification of anchor** (Strength of anchor, Pull-out resistance (soil))



Anchor parameters input

Literature:

Implemented anchors in the catalog

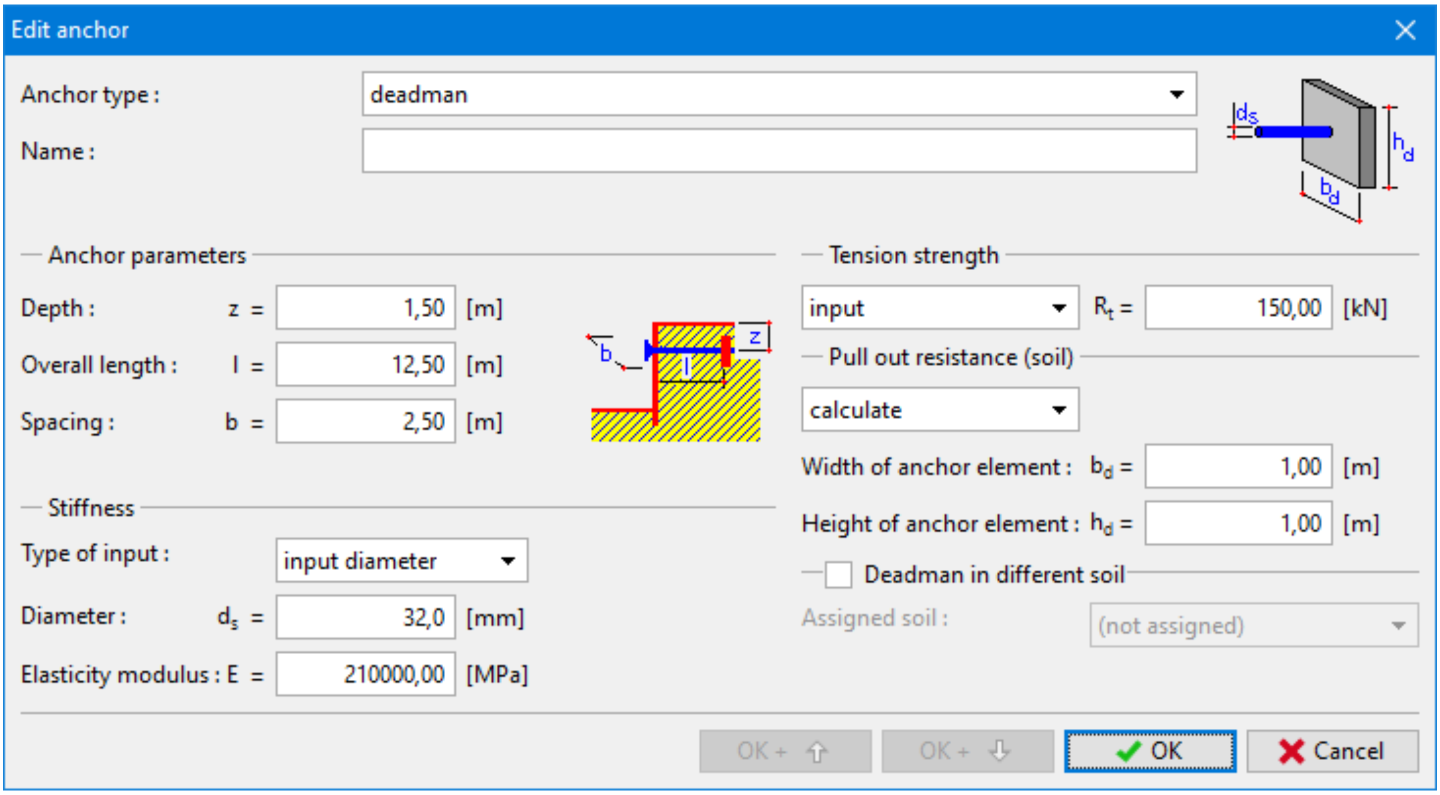
- *VSL Systems Ltd. - VSL Ground Anchor Systems - web brochure*
- *Minova Bohemia, s.r.o. - Injekční zavrtávací a kotevní tyče MAI SDA R a T, Technický list*

Deadman

The "**Deadman**" is selected as an "**Anchor type**" in the combo list in the dialog window "**New anchor**".

The **deadman** anchor requires the input of:

- **Location of anchor** - depth, length, spacing
- **Stiffness of anchor** - anchor diameter d_s or area of anchor A , elasticity modulus E
- **Parameters for verification of anchor** (Strength of anchor, Pull-out resistance from the soil is described below)



Deadman parameters input

Deadman analysis

The deadman consists of a bar with a specified length and anchor element.

The pull out resistance from the soil is equal to the passive resistance along the height of the anchoring element, which is reduced by a coefficient $\boldsymbol{k}$:

$$R_e = k.0{,}5.(2.z_B.h_D - h_D^2).K_p.\gamma$$

For $\boldsymbol{h} = 0$:

$$k = \frac{b_D}{b}$$

For $\boldsymbol{h} \geq \boldsymbol{b} - \boldsymbol{b_D}$:

$$k = \frac{3\,b_D}{b}$$

where:
- h - height of the overburden over the anchor element
- b_D - width of the anchor element
- b - spacing between the anchor elements
- h_D - height of the anchor element
- z_B - depth of the bottom edge of anchor element
- K_p - coefficient of the passive earth pressure
- γ - unit weight of the soil

The coefficient $\boldsymbol{k}$ is interpolated for the intermediate values of $\boldsymbol{h}$.

Calculation of the overburden over the anchor element:

$$h = z - \frac{h_D}{2}$$

where: z - depth of deadman

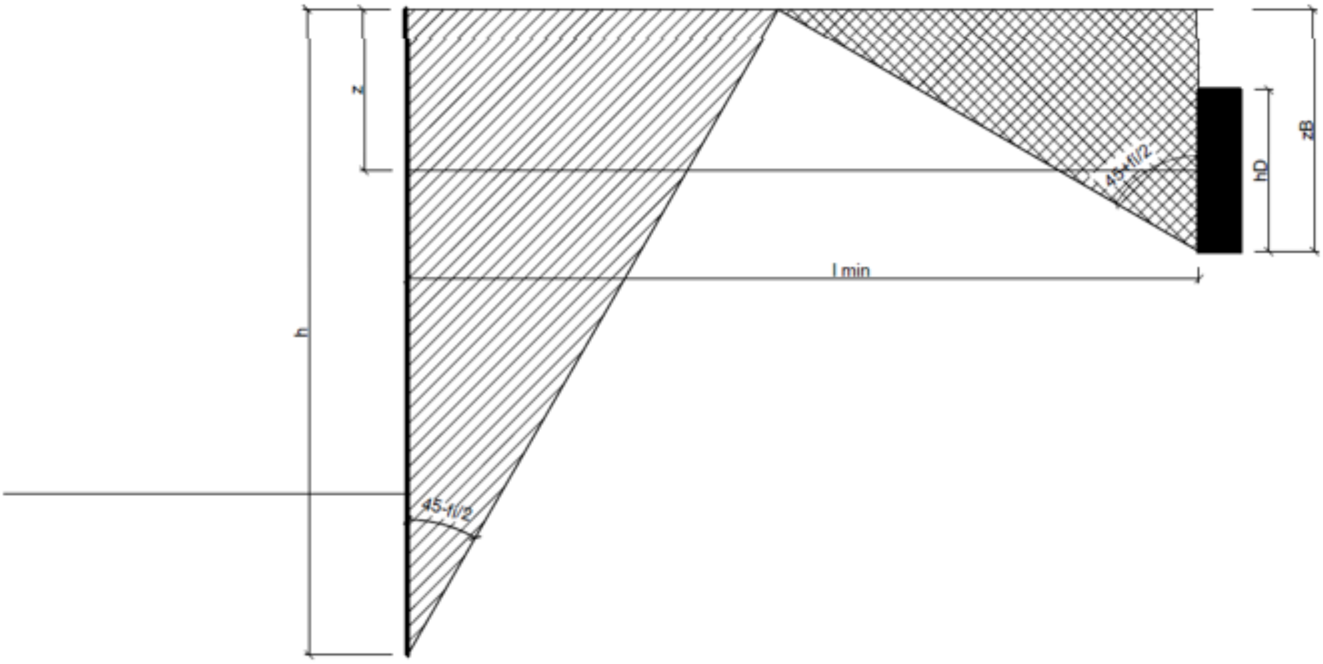
The full pull out resistance is activated if active and passive earth wedges do not intersect.

The minimum length of the bar for full pull out resistance is computed as follows:

$$l_{min} = h \tan\left(45 - \frac{\varphi}{2}\right) + z_B \tan\left(45 + \frac{\varphi}{2}\right)$$

where: φ - angle of internal friction
h - height of shoring structure

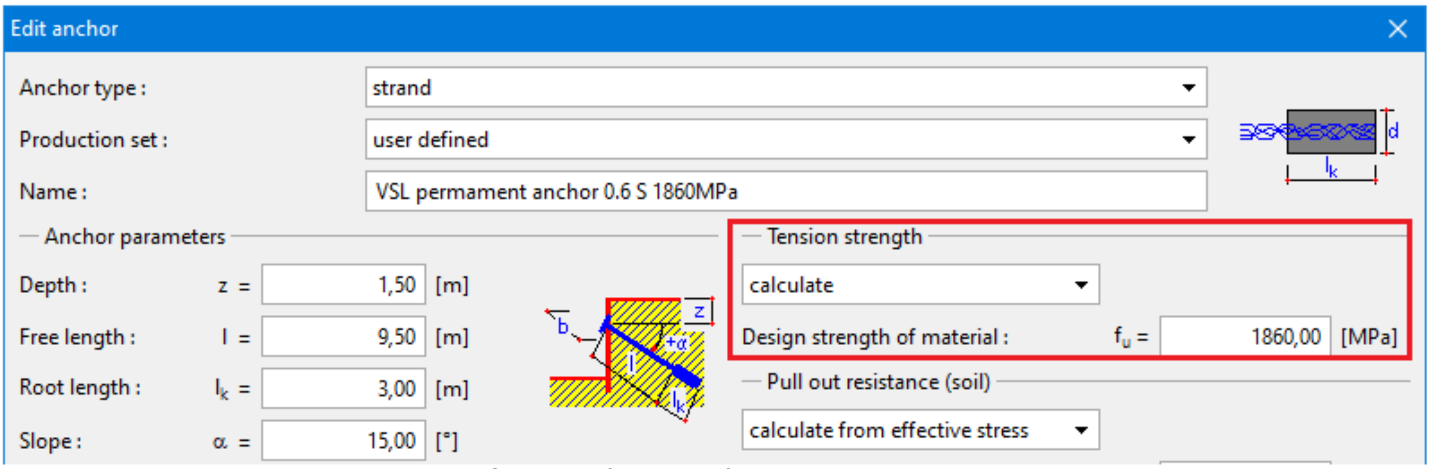
In case that the length of bar l is shorter than the minimum length of bar l_{min}, the passive pressure is considered from the intersection of active and passive earth wedges.



The principle of determining the minimum length for deadman´s bar

Anchor Strength

The method of analysis is selected in the section "**Tension strength**".



Settings of analysis for anchor strength

Option "calculate"

The calculation is the same for all types of anchors according to the formula:

$$R_t = f_u A$$

where: f_u - design strength of a material
A - area of the shaft of the anchor

Option "input"

It is necessary to enter a value of tension strength R_t $[kN, lbf]$

In the case of **strand anchors** it is necessary to enter the **area of one strand** (option "**input area**" - total area is calculated according to the formula:

$$A = A_1\, n$$

where: A_1 - area of one strand

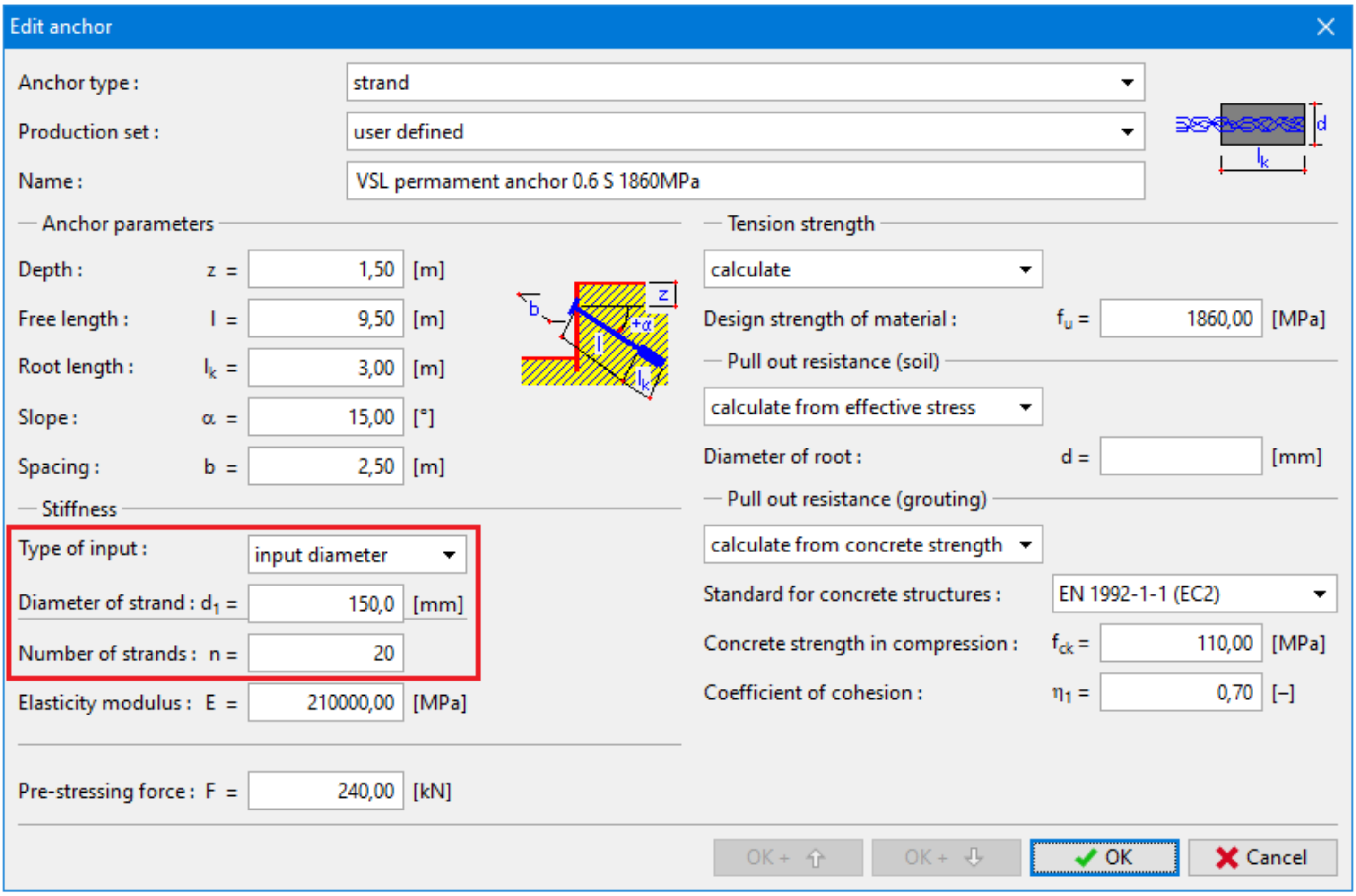
n - number of strands

In case that the diameter of strand is entered (option "**input diameter**"), the total area is calculated according to the formula:

$$A = \frac{\pi\, d_1^2}{4}\, n$$

where: d_s - diameter of one strand

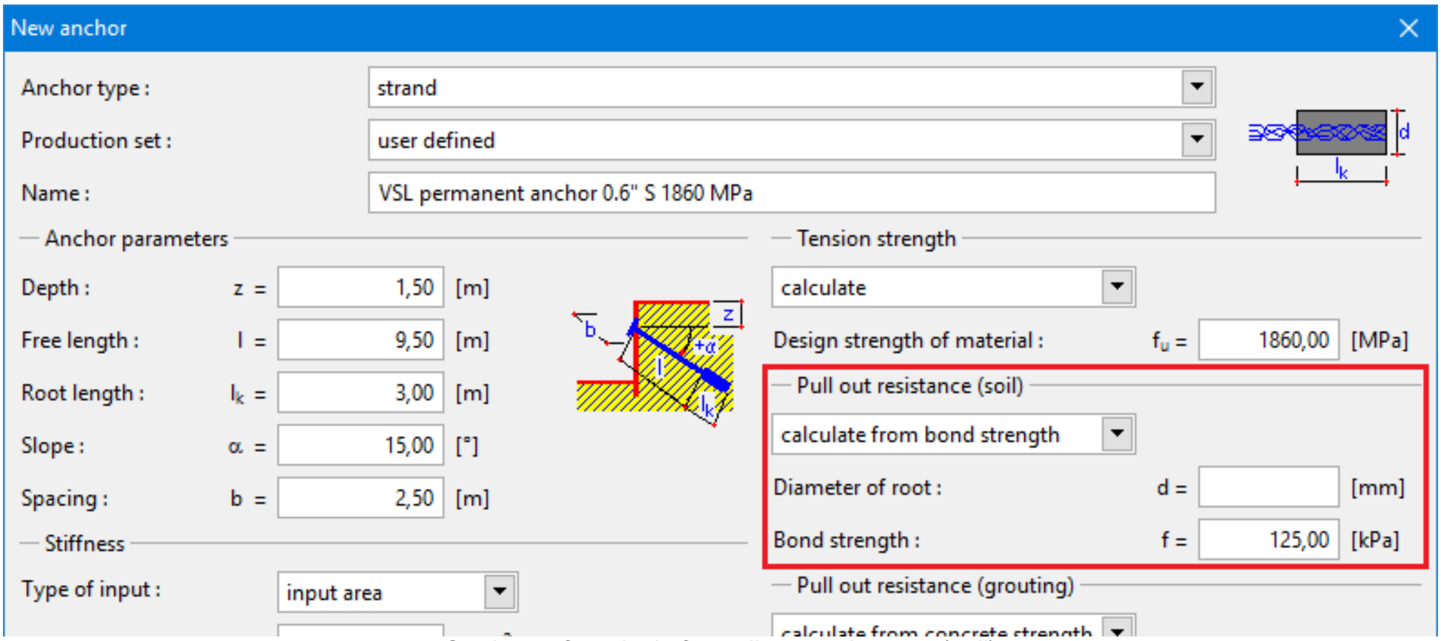
n - number of strands



Strand anchor parameters input

Pull-out Resistance (Soil)

The method of analysis is selected in the section "**Pull out resistance (soil)**".



Settings of analysis for pull out resistance (soil)

Analysis for a prestressed bar, strand, and non-prestressed bar anchors

Option "calculate from effective stress"

$$R_e = \pi\, d\, l_k\; \sigma_z \tan\varphi$$

where:
- d - diameter of the root
- l_k - root length
- σ_z - geostatical stress
- φ - angle of the internal friction of the soil

Option "calculate from bond strength"

$$R_e = \pi\, d\, l_k\, f$$

where:
- d - diameter of the root
- l_k - root length
- f - bond strength

Option "input bearing capacity per unit length"

$$R_e = R_{e,bm}\, l_k$$

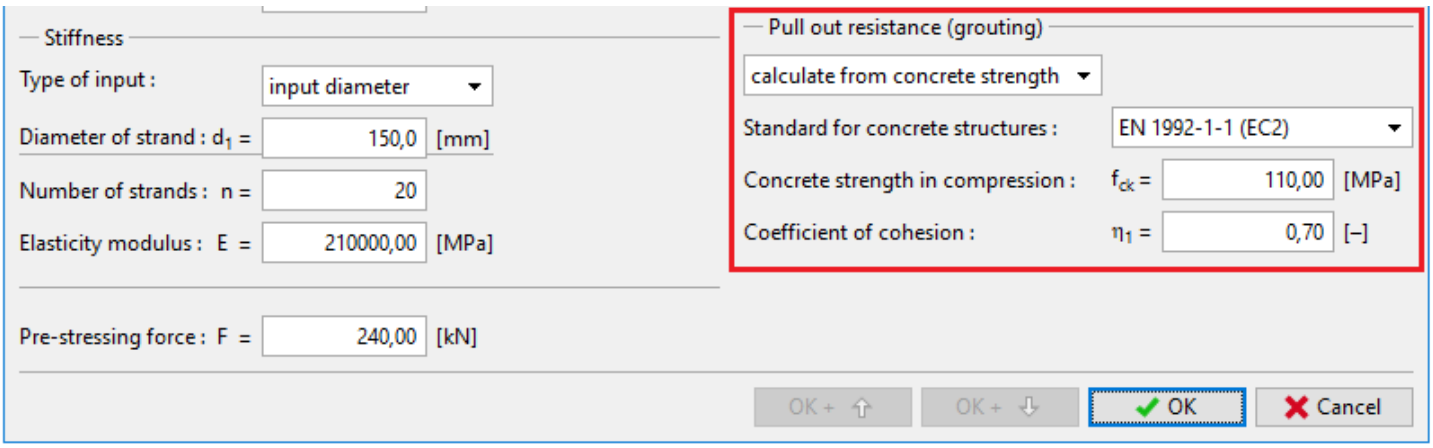
where:
- $R_{e,\,bm}$ - pull out resistance from soil per unit
- l_k - root length

Option "input"

The total value of pull out resistance (soil) is entered R_e $[kN\,,\, lbf\,]$

Pull-out Resistance (Grouting)

The method of analysis is selected in the section "**Pull out resistance (grouting)**".



Setting of analysis for pull out resistance (grouting)

This bearing capacity determines a pullout resistance from the root and is calculated equally for prestressed bar and strand anchors.

$$R_c = \pi\, d_s\, l_k\, \tau$$

Option "Calculate from shear strength"

where: d_s (d_1) - diameter of the anchor bar (strand diameter corresponding to the strand area)
l_k - root length
τ - shear strength between the anchor bar and grouting

Option "Calculate from concrete strength"

It is the same analysis as in the previous case, the shear strength τ is calculated from the parameters of concrete according to the different standards:

- EN 1992-1-1 (EC2)

$$\tau = 1{,}2\, \eta_1\, f_{ctd}$$

where: η_1 - coefficient of cohesion (1,0 for good conditions of cohesion, 0,7 in other cases)
f_{ctd} - concrete strength in tension - obtained from f_{ck} by formulas; ≤ C60/75 value

- ACI 318-11

$$\tau = 3{,}3\sqrt{f_{c'}}$$
$$\tau \leq 100\, psi (cca\ 0{,}689 kPa)$$

where: $f_{c'}$ - concrete strength in compression

- GB 50010-2010

$$\tau = \alpha\, f_t$$

where: α - coefficient of reinforcement type
1,47 – 7-strand
1,56 – 3-strand
1,79 – plain reinforcement bar
1,92 – ribbed steel wire
f_t design concrete strength in tension; ≤ C60 value

Option "input bearing capacity per unit length"

$$R_c = R_{c,bm}\, l_k$$

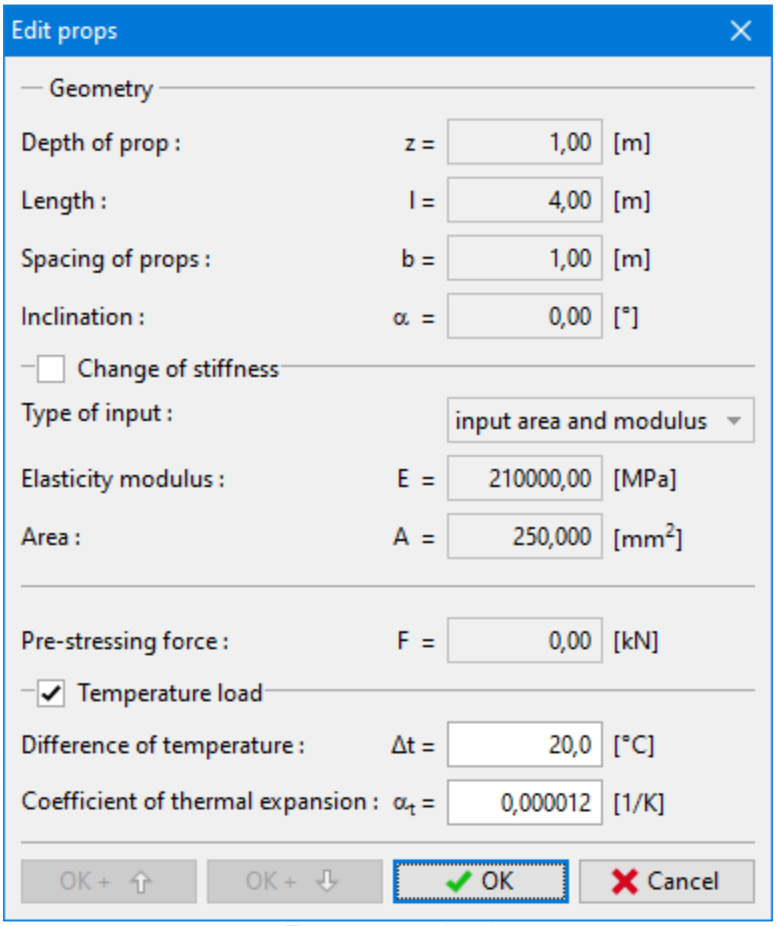
where: $R_{c,\,bm}$ - pull out resistance [kN/bm , lbf/ft] from grouting per unit length
L_K - root length

Option "input"

The value of pull out resistance (grouting) is entered R_c [kN , lbf]

Props - Temperature Load

The prop can be loaded by a **uniform temperature change**. The input requires a thermal coefficient of expansion α_t and change in temperature Δt between current and previous construction stage.



Temperature load

The magnitude of the **additional force in prop** due to change in temperature is given by:

$$\Delta F_{temp} = \alpha_t.\Delta t.E.A$$

where:
- α_t - thermal coefficient of expansion
- Δt - change in temperature
- E - elastic modulus
- A - cross-sectional area

Method of Dependent Pressures

The basic assumption of the method is that the soil or rock in the vicinity of the wall behaves as ideally elastic-plastic Winkler material. This material is determined by the modulus of subsoil reaction k_h , which characterizes the deformation in the elastic region and by additional limiting deformations. When exceeding these deformations the material behaves as ideally plastic.

The following assumptions are used:

- The pressure acting on a wall may attain an arbitrary value between active and passive pressure - but it cannot fall outside of these boundaries.
- The pressure at rest acts on an undeformed structure ($w = 0$).

The pressure acting on a deformed structure is given by:

$$\sigma = \sigma_r - k_h.w$$

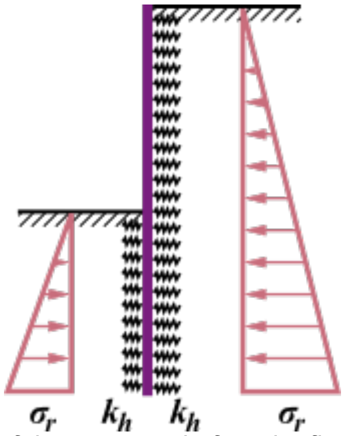
$\sigma = \sigma_a$ for: $\sigma < \sigma_a$

$\sigma = \sigma_p$ for: $\sigma > \sigma_p$

where:
- σ_r - pressure at rest
- k_h - modulus of subsoil reaction
- w - deformation of structure
- σ_a - active earth pressure
- σ_p - passive earth pressure

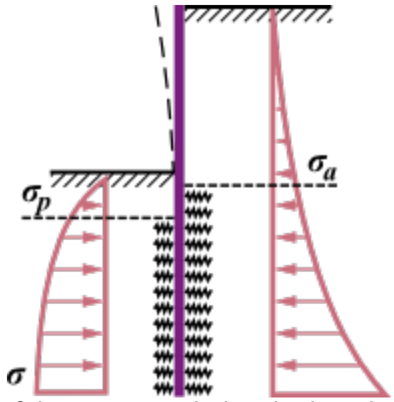
The computational procedure is as follows:

- The modulus of subsoil reaction k_h is assigned to all elements and the structure is loaded by the pressure at rest - see figure:



Scheme of the structure before the first iteration

- The analysis is carried out and the condition for allowable magnitudes of pressures acting on the wall is checked. In locations at which these conditions are violated the program assigns the value of $k_h = 0$ and the wall is loaded by active or passive pressure, respectively - see figure:



Scheme of the structure during the iteration process

The above iteration procedure continues until all required conditions are satisfied.

In analyses of subsequent stages of construction, the program accounts for plastic deformation of the wall. This is also the reason for specifying individual **stages of construction** that comply with the actual construction process.

Literature:

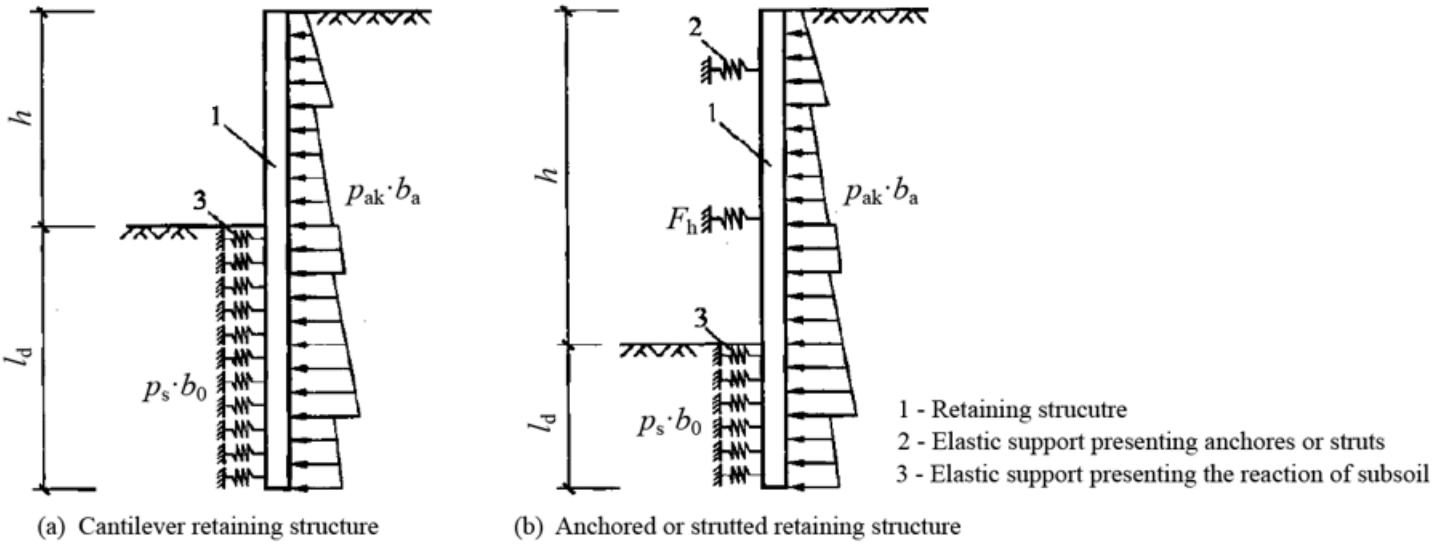
Bartak, J.: Progresivni postupy navrhovani pazenych stavebnich jam. VUT Brno, 1991.

Hurych, P.: Metoda zavislych tlaku. Sbornik konference "Automatizacia projektovania", Vysoke Tatry, 1978.

Spring Method According to JGJ 120-2012

This method is used for the analysis of sheeting structures and it is based on the Chinese standard **JGJ 120-2012** (Technical specification for retaining and protection of building foundation excavations). In principle, this theory is similar to calculation according to the method of dependent pressures, the difference is in consideration of earth pressures. The following figure shows that **behind the wall** (outside of the foundation pit) acts active earth pressure p_a or earth pressure at rest p_0 (it's defined in the "Settings" frame).

In front of the wall, there are considered springs (defined by using the modulus of subsoil reaction), which models the reaction of the soil in a horizontal direction. In case of the attainment of ultimate pressures a limiting of the size of springs is the same as for the method of dependent pressures.



Principle of spring method acording to JGJ 120-2012 for solution of sheeting structures - (a) non-anchored structure, (b) anchored or strutted structure

Literature:

JGJ 120-2012 (Technical specification for retaining and protection of building foundation excavations).

Modulus of Subsoil Reaction

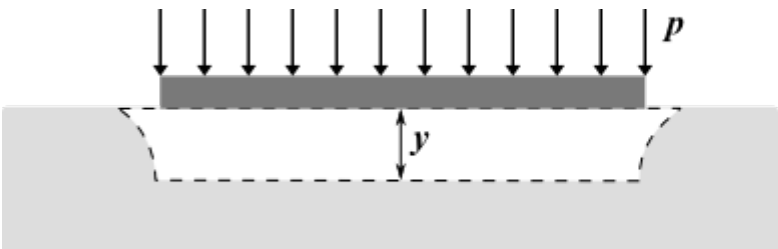
The following options are available in the program to introduce the modulus of subsoil reaction:

- **in the form of distribution** (input distribution of the modulus of subsoil reaction k_h in front and behind the structure is input)
- **as a soil parameter with a respective value** (either linear or nonlinear - curve)
- according to Schmitt
- according to Chadeisson
- according to CUR 166
- iterate by using deformation characteristics of soils
- input results of pressuremeter test (according to NF P 94-282, according to Menard)
- input results of the dilatometric test (DMT)
- according to Chinese standards ("c", "k" or "m" method)

The modulus of the horizontal reaction of a soil body generally corresponds to the spring stiffness in the Winkler model describing the relation between load applied to a rigid slab and the resulting soil deformation in the form:

$$p = k.y$$

where:
- p - load acting along the slab-soil interface
- k - stiffness of the Winkler spring
- y - translation of the slab into the subsoil



Definition of the modulus of subsoil reaction

Modulus of Subsoil Reaction According to Schmitt

This analysis of the modulus of subsoil reaction builds on the relation between oedometric modulus and bending stiffness of the structure introduced by Schmitt in Revue Francaise de Géotechnique no. 71 and 74:

$$k_h = 2{,}1\left(\frac{E_{oed}^{4/3}}{(EI)^{1/3}}\right)$$

where:

- EI - bending stiffness of the structure [MNm^2/m]
- E_{oed} - oedometric modulus [MPa]

This method is not available for the structures from plastic sheet piles.

Literature:

Schmitt, P. (1995): "Estimating the coefficient of subgrade reaction for diaphragm wall and sheet pile wall design", in French. Revue Française de Géotechnique, N. 71, 2° trimestre 1995, 3-10.

Modulus of Subsoil Reaction According to Chadeisson

Based on the measurements on sheeting structures in different soils and computation of a displacement of the structure needed to mobilize the limit value of passive pressure R. Chadeisson (1961) and A. Monnet (1994) derived expression for the determination of the modulus of subsoil reaction in the form:

$$k_h = \left[20EI\left(\frac{K_p\gamma\left(1-\frac{K_0}{K_p}\right)}{0{,}015}\right)^4\right]^{\frac{1}{5}} + A_p c' \frac{tgh\left(\frac{c'}{30}\right)}{0{,}015}$$

where:

- EI - bending stiffness of the structure [kNm^2/m]
- γ - unit weight of soil [kN/m^3]
- K_p - coefficient of passive earth pressure [-]
- K_0 - coefficient of earth pressure at rest [-]
- c' - effective cohesion of soil [kPa]
- A_p - coefficient of influence of cohesion (1 - 15) [-]

Literature:

K. J. Bakker, A. Bezuijen, W. Broere, E. A. Kwast: Geotechnical Aspects of Underground Construction in Soft Ground: Proceedings of the 5th International Symposium TC28. Amsterdam, the Netherlands, 15-17 June 2005. CRC Press, 2013, pp. 616, ISBN: 0415889138, 9780415889131.

Monnet, A.: Module de réaction, coefficient de décompression, au sujet des paramètres utilisés dans la méthode de calcul élastoplastique, Revue française de Géotechnique, 65, 1994, pp. 67 - 72.

Mitew, M.: Numerical analysis of displacements of a diaphragm wall. Warsaw University of Technology, Poland.

N. M. ILIEŞ, T. A. HULPUȘ, A. POPA: Design of Anchored Walls: The Influence of Design Approaches and Design Methods. Technical University of Cluj Napoca, Faculty of Civil Engineering, Romania, 2010.

Modulus of Subsoil Reaction According to CUR 166

The following table stores the values of the modulus of subsoil reaction derived from experimental measurements carried out in the Netherlands (described in CUR 166). The table offers secant modules, which are in the program directly transformed into secant modules of subsoil reaction - see nonlinear modulus of subsoil reaction.

	$k_{h,1}$ [kN/m^3] $p_0 < p_h < 0{,}5\ p_{pas}$	$k_{h,2}$ [kN/m^3] $0{,}5\ p_{pas} \le p_h \le 0{,}8\ p_{pas}$	$k_{h,3}$ [kN/m^3] $0{,}8\ p_{pas} \le p_h \le 1{,}0 p_{pas}$
Sand			
loose medium dense dense	12000 - 27000 20000 - 45000 40000 - 90000	6000 - 13500 10000 - 22500 20000 - 45000	3000 - 6750 5000 - 11250 10000 - 22500
Clay			
soft stiff very stiff	2000 - 4500 4000 - 9000 6000 - 13500	800 - 1800 2000 - 4500 4000 - 9000	500 - 1125 800 - 1800 2000 - 4500

Peat			
soft stiff	1000 - 2250 2000 - 4500	500 - 1125 800 - 1800	250 - 560 500 - 1125

where: p_0 - value of pressure at rest $[kN/m^2]$

p_{pas} - passive pressure $[kN/m^2]$

p_h - horizontal pressure corresponding to a given displacement of a structure $[kN/m^2]$

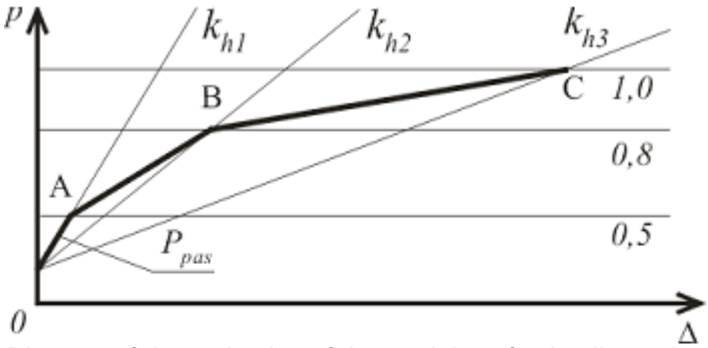


Diagram of determination of the modulus of subsoil reaction

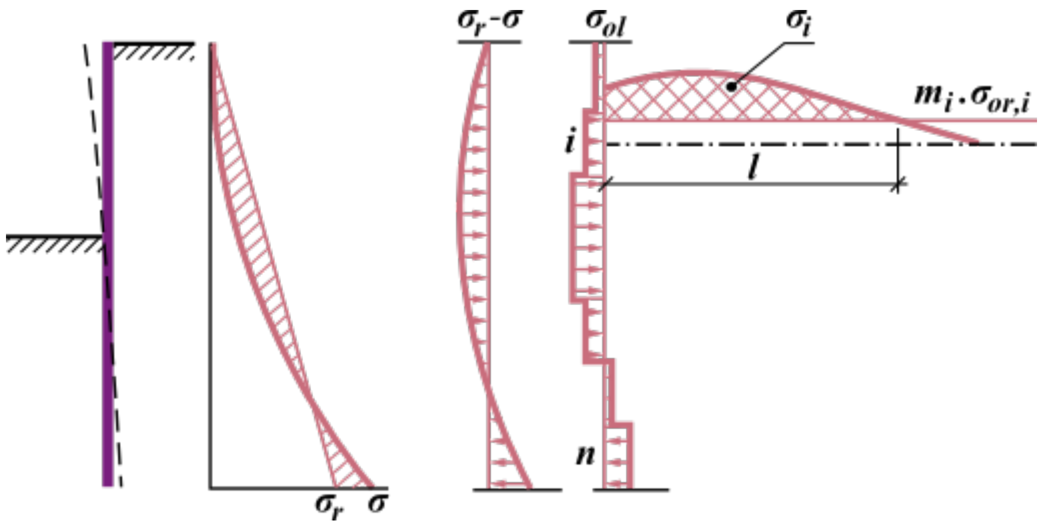
Literature:

CUR 166 Damwandconstructies, available at Civieltechnisch Centrum Uitvoering Research en Regelgeving: P.O.Box 420, 2800 AK Gouda (NL).

Modulus of Subsoil Reaction Determined from Iteration

The program allows automatic calculation of the modulus of subsoil reaction from deformational characteristics of soil from the iteration process. The procedure builds on the assumption that deformation of the elastic subspace characterized by the deformation modulus E_{def} $[MPa]$ when changing the stress state associated with the change of earth pressures is the same as the deformation of the underground wall.

The goal, therefore, is to find such values of k_h $[MN/m^3]$ so that the continuity of deformations of the wall and adjacent soil is maintained. **Plastic deformation of the structure is not considered** when performing analysis with k_h **manual iteration**. While the analysis of modulus k_h with **automatic iteration plastic deformation of the structure is considered.** Principle of manual iteration is schematically cleared by computing the modulus of subsoil reaction of the i^{th} segment of wall free of anchor, see figure:



Determination of modulus of subsoil reaction of i^{th} segment

For change of stress $\sigma_r - \sigma$ the program determines uniform load σ_{ol} $[MPa]$ of individual segments of a structure. Next, the overall change of stress passing the i^{th} segment ($\bar{\sigma}_{il}$ $[MPa*m]$) is computed. This change is caused by the additional load of the soil body due to segments 1 to n ($\sigma_{ol,1} - \sigma_{ol,n}$). The overall change of stress $\Delta\sigma_i$ is reduced by structural strength $m_i * \sigma_{or,i}$ $[MPa]$. The new value of the spring stiffness then follows from:

$$k_{n,i} = \frac{E_{def,i} \cdot \sigma_{ol,i}}{\bar{\sigma}_{il}}$$

where: E_{def} - deformation modulus of the elastic subspace [MPa]

σ_{ol} - uniform load applied to the segments of the structure [MPa]

$\bar{\sigma}_{il}$ - overall change of the stress behind i^{th} segment of the structure [$MPa*m$]

The change of stress inside the soil body is determined according to Boussinesq. Inserting the new value of k directly into the next calculation would cause an unstable iteration - therefore the value of k that is introduced into the next analysis of the wall is determined from the original value of k_p and the new value of k_n of the modulus of subsoil reaction.

$$k = k_p + 0{,}25.(k_n - k_p)$$

where: k_p - original value of modulus of the subsoil reaction [MN/m^3]

k_n - new value of modulus of the subsoil reaction [MN/m^3]

Maximum of modulus of subsoil reaction of the i^{th} layer is limited by the value:

$$k_{max,ip} = 10.E_{def,i}$$

where: $E_{def,i}$ - deformation modulus of i^{th} layer [MPa]

The **manual iterative procedure** used when computing the modulus of the subsoil reaction is as follows:

1. Determine the matrix of influence values for deriving change of stress in a depth of the soil body passing the i^{th} segment of a structure due to surcharge caused by the change of stress in other segments.
2. The first approximation of the modulus k_h in front of the wall is introduced - a triangular distribution of values at the wall heel $k_h =$ 10 MN/m^3 is assumed.
3. Perform analysis of the wall (sheeting structure).
4. Compute new values of k_h and determine new values for the next analysis.
5. The dialog window to check the iteration appears and the program waits till the next command. If the next n iterations are selected, the steps 3 and 4 are repeated n -times to arrive again at the step No. 5. The analysis is terminated in this dialog window by pressing the "**Stop**" button.

This **manual iterative process** is controlled by the user - he or she has to decide whether the results make sense. The **automatic iterative procedure** is performed without entering further iterations for calculation of modulus k_h .

Literature:

Bartak J.: Progresivni postupy navrhovani pazenych stavebnich jam, VUT Brno, 1991.

Modulus of Subsoil Reaction According to Menard

Based on the results from experimental measurements (pressuremeter) of soil response loaded by rigid slab Menard derived the following expression for modulus of subsoil reaction:

$$k_h = \frac{E_M}{\frac{\alpha.a}{2} + 0{,}133.(9.a)^{\alpha}}$$

where: E_M - pressuremeter (Menard) modulus, if necessary it can be substituted by the oedometric modulus of soil [MPa]

a - characteristic length depending on a depth of the fixed-end structure, according to Menard. The assumed depth of 2/3 of length of a fixed-end structure below the final depth of the sheeted ditch

α - rheological coefficient of soil

Approximate values of rheological coefficient of soil α

	Clay	Silt	Sand	Gravel
Overconsolidated	1	2/3	1/2	1/3
Normally consolidated	2/3	1/2	1/3	1/4
Non-consolidated	1/2	1/2	1/3	1/4

Literature:

Menard, L. (1975): "The Menard Pressuremeter: Interpretation and Application of the Pressuremeter Test Results to Foundations Design", Sols-Soils, No. 26, Paris, France.

Modulus of Subsoil Reaction According to NF P 94-282

The modulus of subsoil reaction k_h according to **NF P94-282:2009-03** depends on bending stiffness of sheeting structure

$E_{str} I_{str}$ and pressuremeter (Menard) modulus E_M. The modulus of subsoil reaction is given by the following formula:

$$k_h = 2\left(\frac{\left(\frac{E_M}{\alpha}\right)^{4/3}}{\left(\frac{E_{str} I_{str}}{B_0}\right)^{1/3}}\right) = 2\left(\frac{\left(\frac{E_M}{\alpha}\right)^{4/3}}{(E\,I)^{1/3}}\right)$$

$$E\,I = \frac{E_{str}\,I_{str}}{B_0} = \left[MNm^2 / m\right]$$

where:
- k_h - modulus of subsoil reaction $[MN/m^3]$
- E_M - pressuremeter modulus according to Menard $[MPa]$
- α - empirical coefficient depending on the type of the soil, or type of the rock [-]
- $E_{str} I_{str}$ - bending stiffness of sheeting structure $[MNm^2]$
- B_0 - characteristic (unit) length of sheeting structure [1 rm]
- E - modulus of elasticity of the material of the sheeting structure $[MPa]$
- I - moment of inertia $[m^4/m]$

Approximate values of empirical coefficient α [-] ***for various types of soils***

Type of soil	Peat	Clay		Silt		Sand		Gravel	
	α	E_M/p_{LM}	α	E_M/p_{LM}	α	E_M/p_{LM}	α	E_M/p_{LM}	α
Overconsolidated	-	> 16	1	> 14	2/3	> 12	1/2	> 10	1/3
Normally consolidated	1	9 - 16	2/3	8 -14	1/2	7 -12	1/3	6 -10	1/4
Non-consolidated	-	7 - 9	1/2	5 - 8	1-2	5 -7	1-3	-	-

Approximate values of empirical coefficient α [-] ***for various types of rocks (according to degree of violation)***

Type of rock	α [-]
Intact, strong	2/3
Slightly impaired, unweathered	1/2
Very poor, weathered	1/3
Metamorphic	2/3

Literature:

NF P94-282: March 2009, pp. 142 – 146.

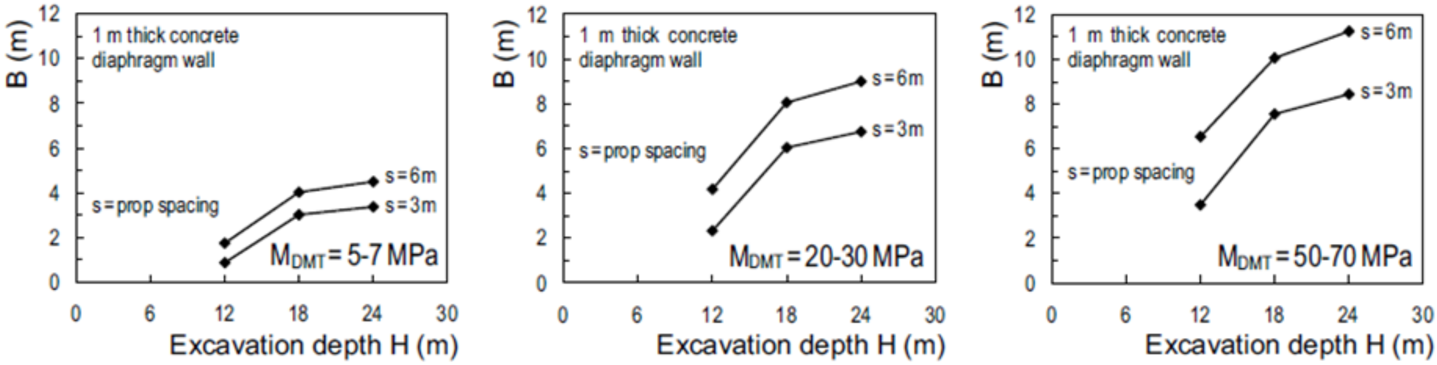
Modulus of Subsoil Reaction Specified by Dilatometric Test (DMT)

Modulus of subsoil reaction k_h $[MN/m^3]$ is determined by the following formula:

$$k_h = \frac{M_{DMT}}{B}$$

where:
- k_h - modulus of subsoil reaction $[MN/m^3]$
- M_{DMT} - constrained modulus obtained from DMT $[MPa]$
- B - characteristic length of the sheeting structure (coefficient of reduction) [1 rm]

Values of coefficient B are depending on the depth of the excavation H $[m]$, constrained modulus M_{DMT} $[MPa]$, and spacing of props.



*Graphs for determination of coefficient **B** (source: [2], Figure 7, pp. 999)*

Literature:

Monaco, P. and Marchetti, S..: Evaluation of the coefficient of subgrade reaction for design of multi-propped diaphragm walls from DMT moduli. Millpress, Rotterdam, 2004, pp. 993 – 1002, ISBN 90 5966 009 9.

Modulus of Subsoil Reaction According to Chinese standards

The calculation of modulus of subsoil reaction according to Chinese standards is based on the JGJ 120-2012 standard (Technical specification for retaining and protection of building foundation excavations) for **"m" method**.

For **"m" method**, the modulus of subsoil reaction k_h is given by the following formula:

$$k_h = m(z-h)$$

where:
- m - proportional coefficient of modulus of the subsoil reaction [kN/m^4]
- z - depth of the calculation point from the original ground [m]
- h - depth of the calculation point from the ditch bottom at the current stage of construction [m]

From previous formula it's obvious that calculation of modulus k_h is linear with the depth of point during the analysis.

Proportional coefficient m should be determined from the pile test with a horizontal load. If there are no test data, Chinese standard **JGJ 120-2012** suggest an empirical formula to estimate this coefficient:

$$m = \frac{0.2\varphi^2 - \varphi + c}{v_b}$$

where:
- c - cohesion (shear strength) of the soil [kPa]
- φ - angle of the internal friction of the soil [°]
- v_b - horizontal displacement of the sheeting structure at the ditch bottom [mm]; when the displacement is smaller than 10 mm, then $v_b =$ 10 mm

Other methods (**"c" method** and **"k" method**) are not published in JGJ 120-2012 standard, but they are based on practical experience and they are used in China profusely. Then the modulus of subsoil reaction k_h is given by the following formula:

$$k_h = a(z-h)^n$$

If exponent $n =$ 0,5, it's "**c**" **method** and then $a = c$ ($kN/m^{3.5}$).

If exponent $n =$ 0, it's "**k**" **method** and then, $a = K$ (kN/m^3).

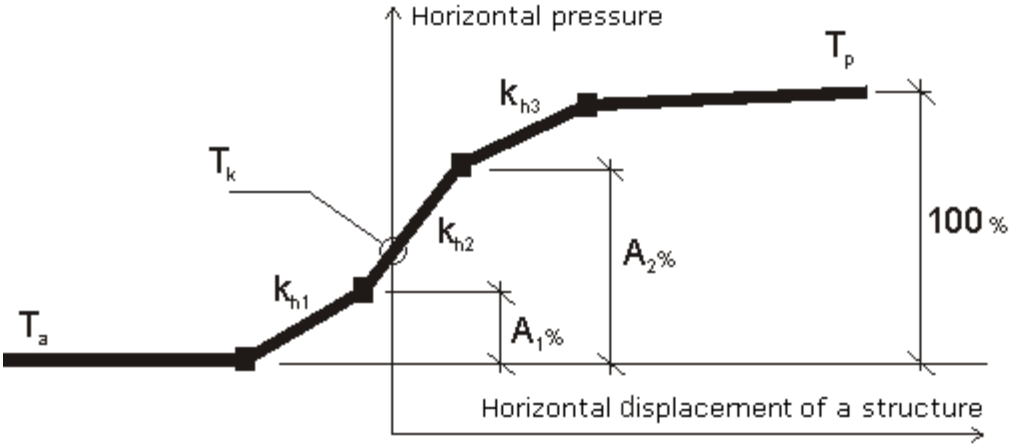
It's obvious that for "**m**" **method**, $n =$ 1.

Literature:

JGJ 120-2012 (Technical specification for retaining and protection of building foundation excavations).

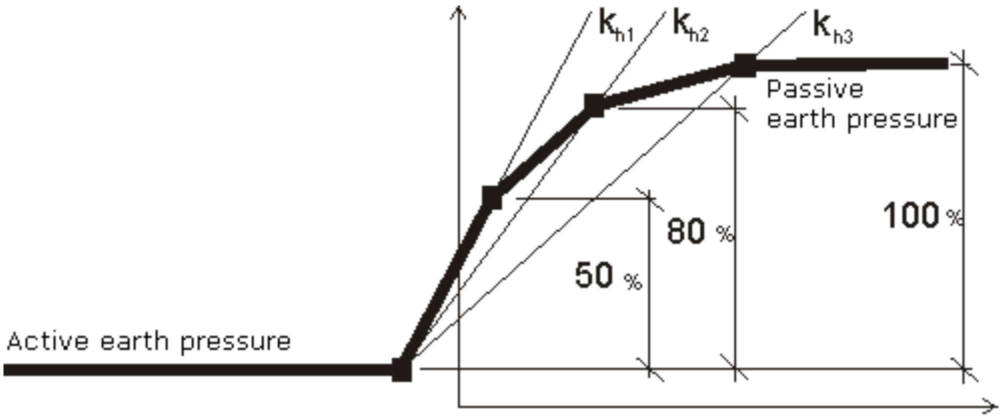
Nonlinear Modulus of Subsoil Reaction

Nonlinear model describes the dependence of the modulus of subsoil reaction k_h - i.e. change of k_h in between the threshold values corresponding to failure due to passive earth pressure T_p and active earth pressure T_a - see figure (the modulus of subsoil reaction is given by the slope of the curve; for pore pressure at rest acting on a structure it is possible to consider the value of k_{h1}). This model also accounts for spring supports and forced deflections of the structure, various boundary conditions, application of struts and anchors, etc.



Interaction model to determine k_h

The values of the modulus of the subsoil reaction can be derived subsequently from the values of secant modules of subsoil reaction (CUR 166) - see figure:



Interaction model to determine k_h *- CUR 166*

Soldier Pile Wall (Braced Sheeting)

When analyzing braced sheeting (**pile curtain**, **steel I-section** or **user input of** A , I , E , G) the following approach is adopted to determine the earth pressures:

Up to the depth of the ditch, the pressures are determined with respect to 1 rm of the structure width. Below the ditch bottom, the earth pressures are multiplied by the coefficient of reduction k (the "**Coefficient of pressure reduction below ditch bottom**"). This coefficient can be input (in the frame "Geometry" as a parameter of the section of a structure) or automatically calculated.

If "**Landfill of soil**" above the ditch (frame "Excavation") is input, then the pressures within this section are computed with respect to the entire width of the wall ($k =$ 1).

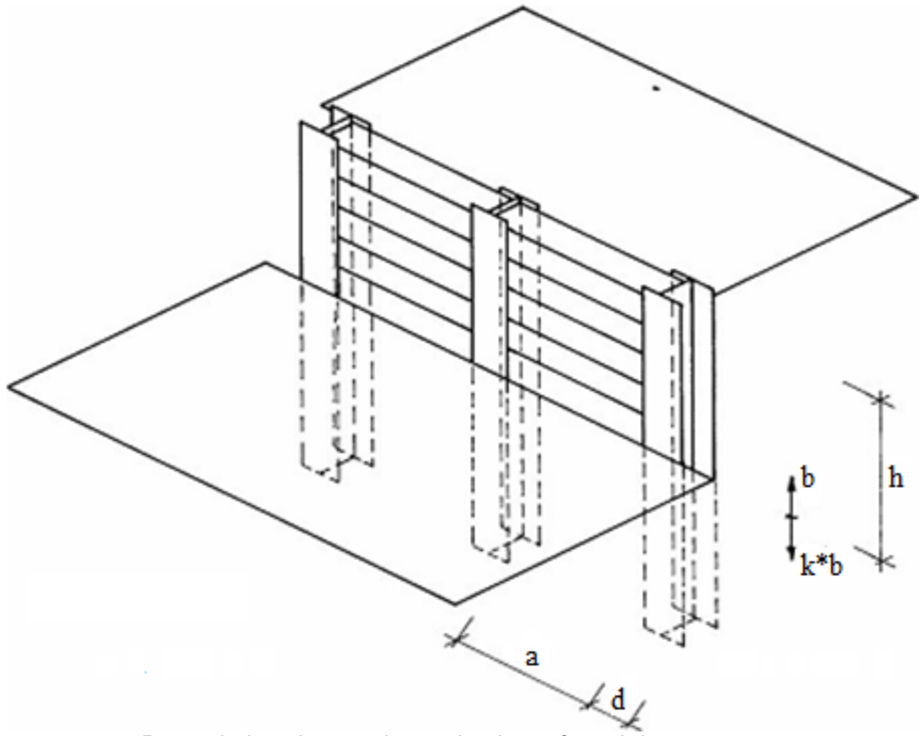
The coefficient of pressure reduction below ditch bottom k can be approximately determined (for very conservative design) according to:

$$k = \frac{d}{d + a}$$

where: a - longitudinal spacing of soldier beams, or spacing between piles

d - width of soldier beam, or diameter of the pile

The real value of the coefficient k also depends on the soil type and space effect of earth pressure. Real values of this coefficient (based on experiments) are **two to three times higher** than values calculated with the help of the previous formula.



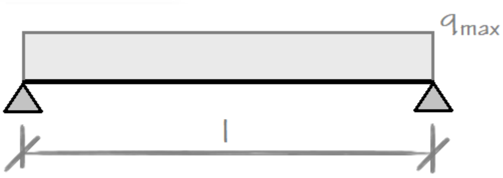
Braced sheeting - schematic view of retaining structure

Lagging

The lagging is considered as the **simply supported beam.** The **beam length** l is the same as the spacing of soldier piles.

The maximum values of internal forces (bending moment and shear force) are calculated according to the selected **type of load:**

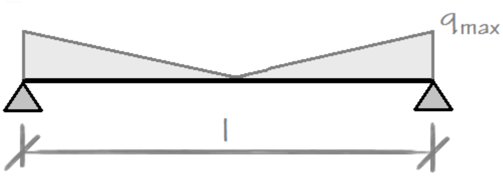
- Rectangle:



Beam static scheme - Rectangle

$$M_{max} = \frac{1}{8}.q_{max}.h.l^2$$
$$Q_{max} = \frac{1}{2}.q_{max}.h.l$$

- Triangle:



Beam static scheme - Triangle

$$M_{max} = \frac{1}{24}.q_{max}.h.l^2$$
$$Q_{max} = \frac{1}{4}.q_{max}.h.l$$

Where: q_{max} - maximum pressure (to the excavation level)
h - cross-section height
l - soldier piles spacing

Walers

The program automatically calculates the internal forces according to the **scheme** selected in the "Walers" frame.

The beam can be loaded by the **concentrated load** R or **uniform load** q .

The magnitude of the uniform load q is calculated according to:

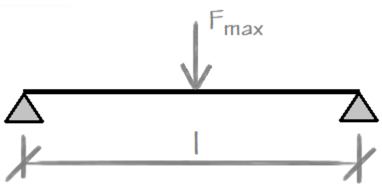
$$q = \frac{n.F_{max}}{(n-1).l + 2.l_o}$$

where: n - number of supports
F_{max} - maximum force in anchor (prop)
l - anchor or prop
l_o - overhangs length ($l/2$)

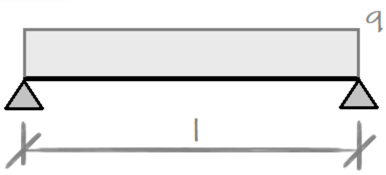
For the walers that are not defined at the same angle as the anchors or props, the load magnitude is reduced by $cos(\alpha)$.

The walers can be designed according to the following schemes:

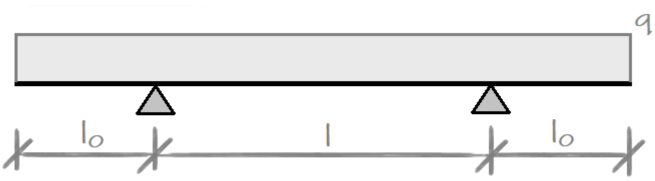
- **Simply supported beam - concentrated load:**



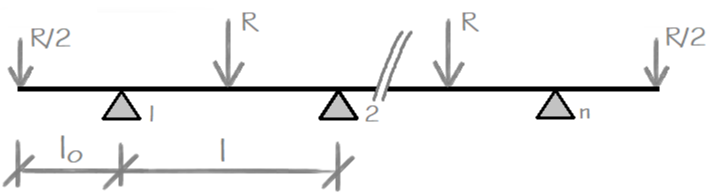
- **Simply supported beam - uniform load:**



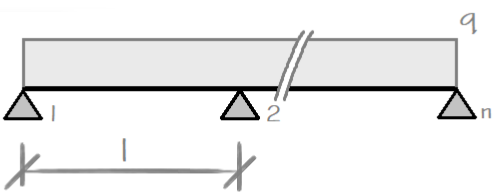
- **Simply supported beam with overhangs - uniform load:**



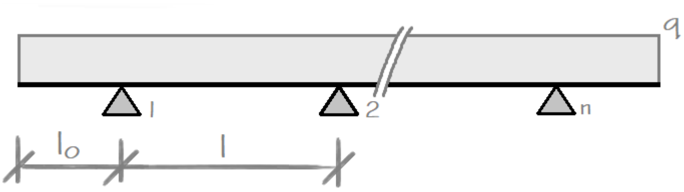
- **Continuous beam with overhangs - concentrated load:**



- **Continuous beam - uniform load:**



- **Continuous beam with overhangs - uniform load:**



In the case of **simply supported beam - concentrated load**, the anchor (prop) is considered as a concentrated load in the centre of the beam span. The beam is supported by the wall profiles.

In other cases, the anchors and props are considered as beam supports.

Note: The walers are verified only in the plane perpendicular to the wall structure resp. in the direction of the anchors.

Double-row Pile Wall

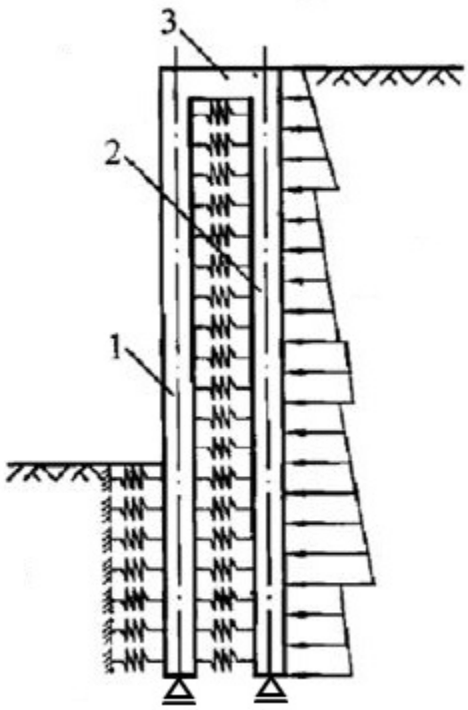
The program allows us to model the double-row pile wall in the "Geometry" frame.

The structure consists of three parts:

- front row
- back row
- connecting beam

The cross-section is selected for each part separately. The connection of the front and back row with a connecting beam is considered as rigid and the wall forms a portal frame.

The **influence of normal force** is considered for dimensioning of cross-section.



Calculation model - 1) front row, 2) back row, 3) connecting beam

The structure is loaded by three main loads:

- earth pressure behind the back row
- earth pressure in front of the front row
- earth pressure between the front and back row

The **earth pressure acting on the front and back row** is calculated in the same way as in the case of a standard wall.

The pressure between the front and back row is determined according to:

$$p_c = k_c.\Delta v + p_{c0}$$

Where:
- p_c - resultant pressure
- k_c - stiffness coefficient of soil between the front and back row
- Δv - difference of deformation of the front and back row
- $p_{c,0}$ - initial earth pressure acting between the front and back row

The soil stiffness coefficient is calculated as follows:

$$k_c = \frac{E_{oed}}{s_y - d}$$

Where: E_{oed} - oedometric modulus
s_y - distance between the front and back row
d - diameter or height of the corss-section

The initial earth pressure between the front and back row is determined by the following formula:

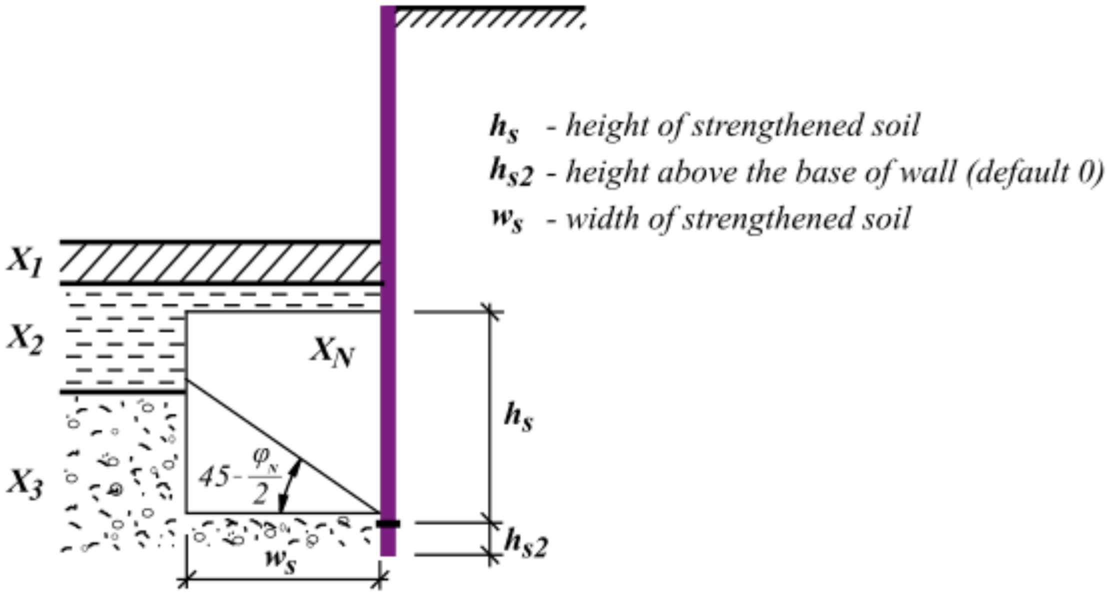
$$p_{c0} = (2\alpha - \alpha^2).p_a$$
$$\alpha = \frac{s_y - d}{h.tan(45 - \varphi_m/2)}$$

Where: α - calculation coefficient; if $\alpha > 1$ then $\alpha = 1$
p_a - active earth pressure
s_y - distance between the front and back row
d - diameter or height of the corss-section
h - final excavation depth
φ_m - average value of the angle of internal friction

Strengthening of the Soil

The program can model strengthening of the soil at the heel of sheeting structure. The strengthening of the soil is performed after installation of piles or walls which are grouted at the base of the wall. The decisive parameters are; **height of strengthened soil** h_s, or **width of strengthened soil** w_s and **parameters of strengthened soil** (φ , c).

The principle of solution is shown in the following figure.



Strengthening of the soil at the base of sheeting structure - graphic presentation of the principle of solution

The principle of calculation for strengthening of the soil at the base of the sheeting structure is described herein:

$$\mu = Min\left(\frac{w_s \tan\left(45 - \frac{\varphi_N}{2}\right)}{h}; 1\right)$$

$$X_i = X_{oi}(1-\mu) + \mu X_N$$

where: N - new layer of the strengthened soil
μ - ratio (auxiliary parameter)
X_i - arbitrary parameter at i^{th} layer of the soil
X_{oi} - original parameter at i^{th} layer of the soil

X_N - new parameter of the strengthened soil

w_s - width of the strengthened soil [m]

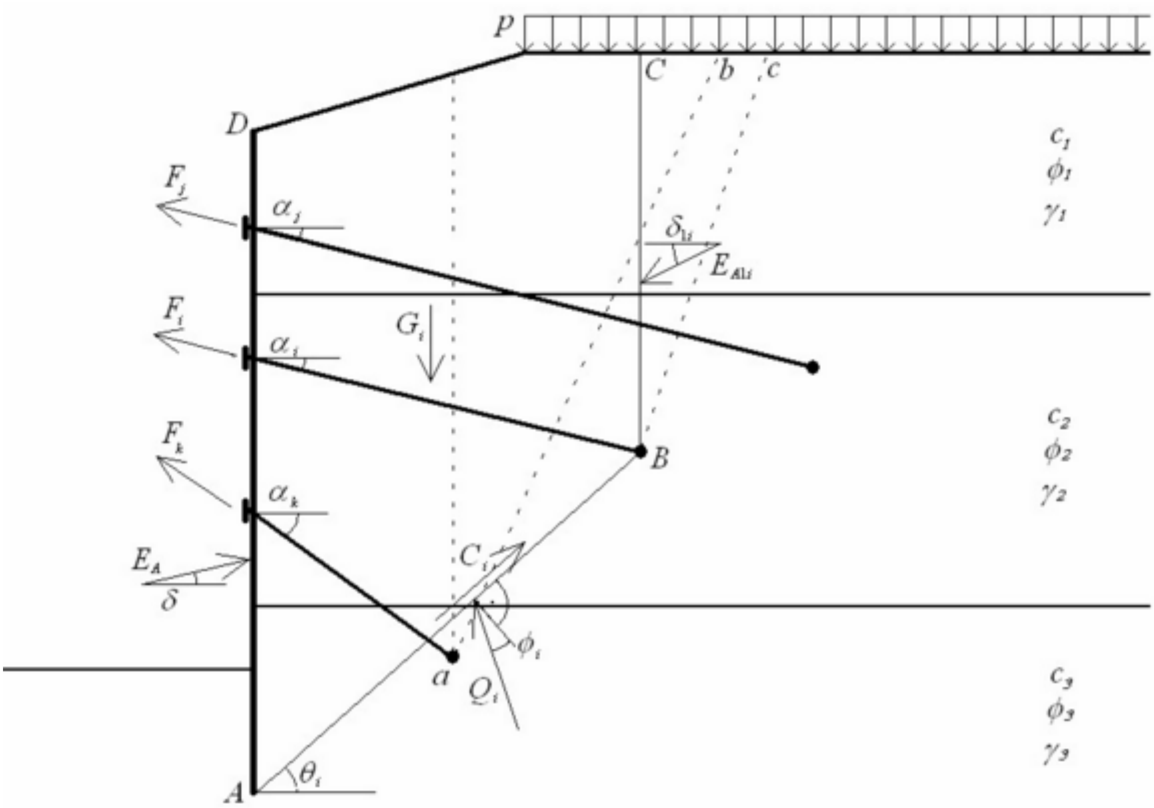
h_s - height of the strengthened soil [m]

Internal Stability of Anchors

The internal stability of an anchorage system of sheeting is determined for each layer independently. The verification analysis determines the anchor force, which equilibrates the system of forces acting on a block of soil. The block is outlined by sheeting, terrain, line connecting the heel of sheeting with anchor root, and by a vertical line passing through the center of anchor root and terrain.

The theoretical footing of sheeting construction is the point where the sum of horizontal forces under the bottom of the construction pit is equal to zero. If this point lies under the footing of the sheeting wall, the theoretical point is the footing of this wall.

The analysis is performed per one running meter of a sheeting structure. Anchor forces are therefore computed with respect to their spacing in individual layers.



Analysis of internal stability

Scheme for verification of the i^{th} layer of anchors is shown in the figure. The force equilibrium for the block $ABCD$ is being determined. The following forces enter the analysis:

E_A - resultant of active earth pressure acting on sheeting (on line AD)

E_{Ai} - resultant of active earth pressure above the root of verified anchor (on line BC)

G_i - weight of the i^{th} the soil block $ABCD$; in addition, this value incorporates the surcharge p applied on the ground surface providing the slope θ_i of slip surface AB is greater than an average value of the angle of internal friction on this surface; in case of a smaller slope of slip surface AB the ground surcharge is not considered

C_i - resultant of soil cohesion on slip surface AB

$F_j, F_k, ...$ - forces developed in other anchors, but some of them are not taken into account; only "**shorter**" anchors (comparing with the i^{th} anchor) will contribute in the equilibrium analysis of the i^{th} block; following principle is used to decide whether the given anchor (the m^{th}) is included or excluded from the equilibrium of the i^{th} block:
at first the lower anchor is selected (the m^{th} or the i^{th}); then a plane slip surface is placed from the root center of the selected lower anchor; this plane is inclined $45^o - \varphi_n/2$ from vertical line (line ab or Bc in the figure); φ_n is an average value of the angle of internal friction above the root of the lower anchor; if the i^{th} root is found above the m^{th} one and the higher located root (the i^{th}) is outside the area cut by the plane slip surface, then the m^{th} anchor is included into analysis; the same example of including the m^{th} anchor is when the i^{th} root lies under the m^{th} one and the m^{th} root is located inside the area cut by the slip surface; two opposite cases determine excluded anchors from analysis; first is the i^{th} root above the m^{th} one and the i^{th} inside the area, second is when the i^{th} root lies under the m^{th} one and the m^{th} is outside the area; from above definition resulting that "shorter" anchor F_k is included into analysis and "longer" anchor F_j is excluded from analysis (see figure)

Q_i - reaction on slip surface AB

F_i force in the analyzed anchor, the maximum allowable magnitude of this force is the result of the equilibrium analysis carried out for the i^{th} block

The solution of the equilibrium problem for a given block requires writing down vertical and horizontal force equations of equilibrium. These represent a system of two equations to be solved for the unknown subsoil reaction Q_i and the maximum allowable magnitude of the anchor force F_i.

As a result, the program provides the maximum allowable anchor forces for each row of anchors. These are then compared with those actually prescribed in anchors.

Ditch Bottom Verification

Standard verification of heave and piping is performed only in case of hydrodynamic pressure when the heel of a structure is sunk into a permeable subsoil, which allows a **free water flow below the structure**. This checks follow **EN1997-1** standard.

Verification of ditch bottom according to Chinese standards (e.g. upheavel according to Prandtl slip surface or effect of the confined aquifer) is available for all types of GWT. These checks are not standardly required and they are available only when using analysis according to Chinese standards.

Failure by Heave and Piping (EN1997-1)

Failure by heave

The stability of soil against heave due to flow of water in the subsoil (HYD) is checked according to the limit states by the equation:

$$u_{dst} \leq \frac{\sigma_{stb}}{\gamma_h}$$

where: u_{dst} - unfavourable water pressure

σ_{stb} - favourable weight of the soil

γ_h - reduction coefficient of a failure by heave

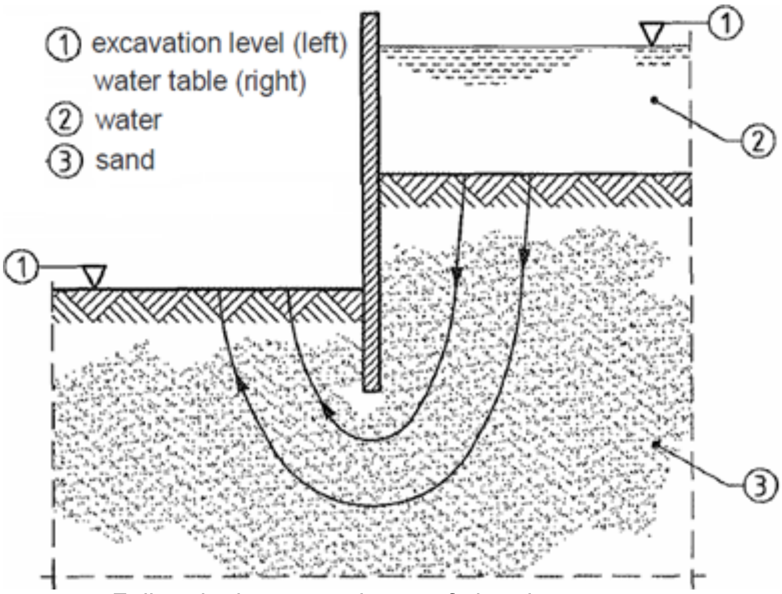
The stability of the soil against heave due to the flow of water in the subsoil (HYD) is checked according to the factor of safety by the equation:

$$\frac{\sigma_{stb}}{u_{dst}} \geq SF_h$$

where: u_{dst} - unfavourable water pressure

σ_{stb} - favourable weight of the soil

SF_h - safety factor for a failure by heave



Failure by heave - scheme of sheeting structure

Failure by piping

Failure by piping is checked according to the limit states by the equation:

$$i \leq \frac{i_c}{\gamma_p}$$

where:
- i - hydraulic gradient
- i_c - critical hydraulic gradient, where $i_c = \gamma_{su} / \gamma_w$
- γ_p - reduction coefficient of a internal erosion of soil

Failure by piping is checked according to the factor of safety by the equation:

$$\frac{i_c}{i} \geq SF_p$$

where:
- i - hydraulic gradient
- i_c - critical hydraulic gradient, where $i_c = \gamma_{su} / \gamma_w$
- SF_p - safety factor for a internal erosion of soil

Literature:

Eurocode 7: Geotechnical design - Part 1: General rules.

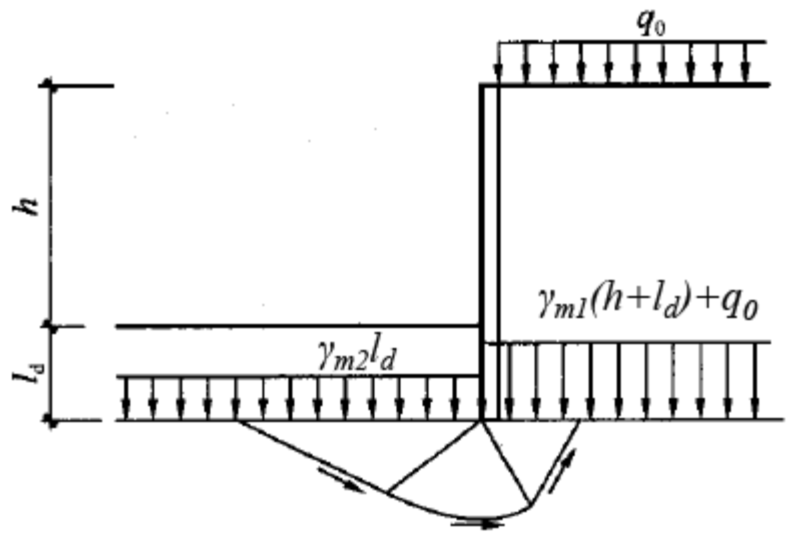
Verification of Ditch Bottom according to Chinese Standards

The program allows us to perform three independent verifications of ditch bottom according to Chinese standards:

- Upheaval check
- Heave check
- Piping of soil

Upheaval Check according to Chinese Standard

This verification follows the **Prandtl analysis of foundation soil bearing capacity**.



Upheaval check scheme - Prandtl slip surface

Verification is done using a safety factor approach and follows formulas:

$$\frac{\gamma_{m2} l_d N_q + c N_c}{\gamma_{m1}\left(h + l_d\right) + q_0} \geq SF_u$$

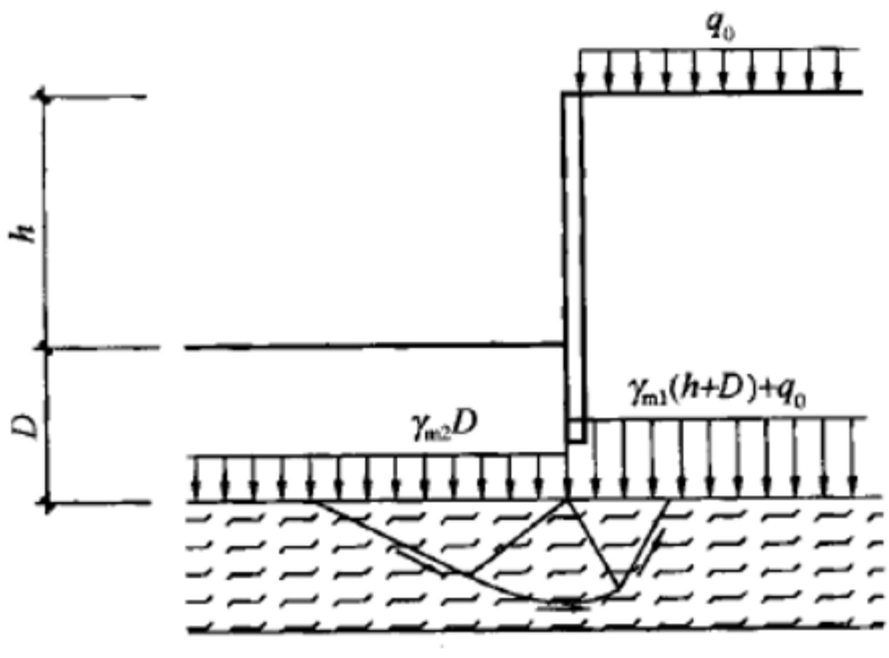
$$N_c = \frac{N_q - 1}{tan\varphi}$$

$$N_q = tan^2\left(45^o + \frac{\varphi}{2}\right) e^{\pi\, tan\varphi}$$

where:

SF_u	-	safety factor of upheaval
γ_{m1}	-	weighted average value of the soil unit weight behind the structure
γ_{m2}	-	weighted average value of the soil unit weight in front of the structure
l_d	-	embedded depth of the structure
h	-	depth of the foundation pit
q_0	-	uniform load on the terrain
N_c , N_q	-	coefficients of bearing capacity
c, φ	-	cohesion and internal friction angle of soil below the bottom of structure

Note: If there are more several layers of soil below the bottom of the structure, the software calculates the safety factor of upheaval for every layer. Depth of layer from the ditch bottom (D) is used instead of embedded depth (l_d).

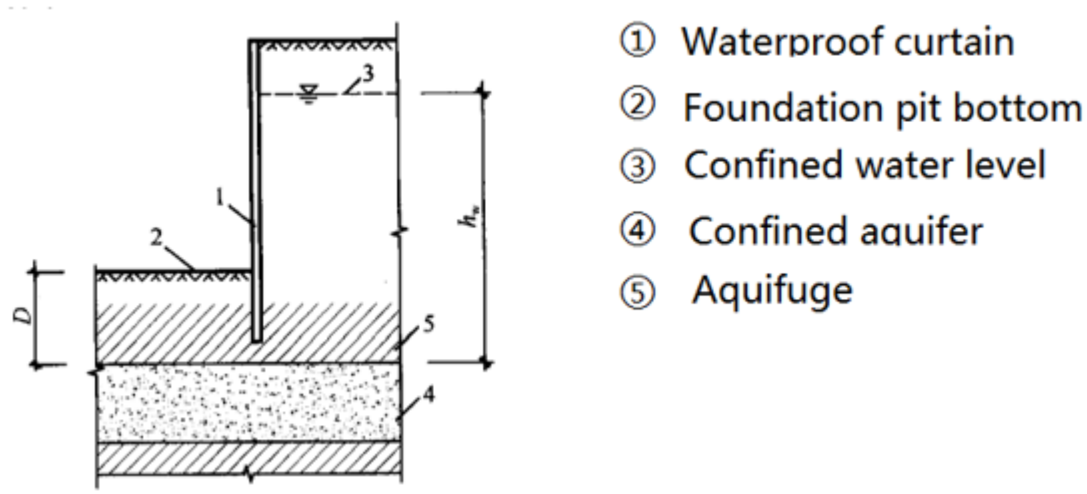


Principle of the verifciation in case of a layer in a depth D

Literature: JGJ 120-2012

Heave Check according to Chinese Standard

This verification depends on the **depth of confined aquifer** D and **height of** its **confined water level** h_w to which the water would rise if it was not restricted by aquifuge from above.



Verification is done by using a safety factor approach and follows formulas:

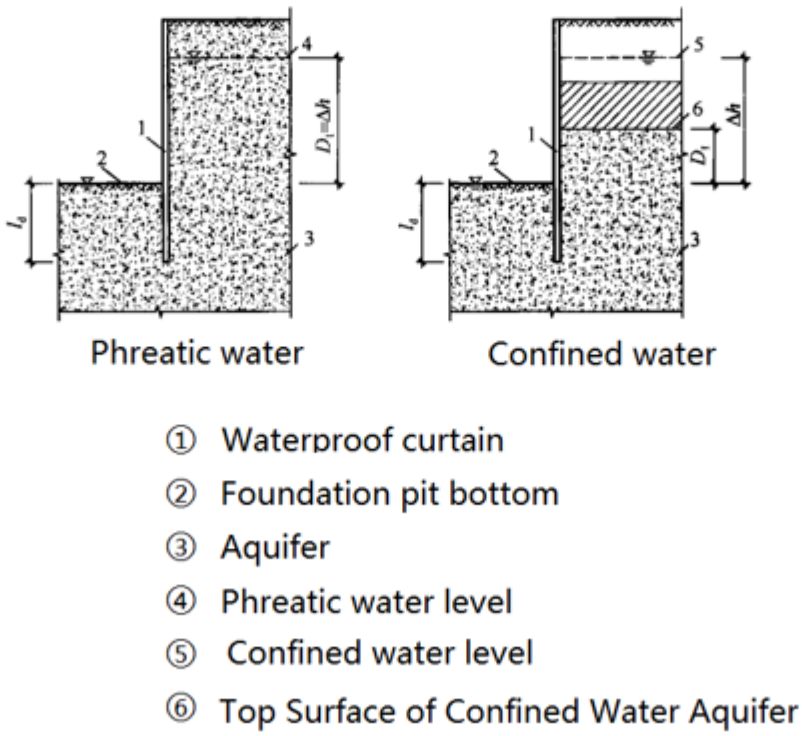
$$\frac{D\gamma}{h_w\gamma_w} \geq SF_h$$

where:

SF_h	-	safety factor of stability of the soil against heave
D	-	Distance between the top of the confined aquifer and the bottom of foundation pit
γ	-	Unit weight of the soil layer between the top of confined aquifer and the bottom of foundation pit
h_w	-	Pressure head from the top of confined aquifer
γ_w	-	Unit weight of water

Piping Check according to Chinese Standard

This verification can be used for both phreatic and confined water levels.



Verification is done using safety factor approach and follows formulas:

$$\frac{(2l_d + 0.8D_1)\gamma'}{\Delta h \gamma_w} \geq SF_p$$

where:

SF_p	-	Safety factor of stability of the soil against piping
l_d	-	Length of the waterproof curtain below the bottom of the pit
D_1	-	Depth between the bottom of foundation pit and phreatic water level or top surface of the confined water aquifer
γ'	-	Unit weight of soil under the buoyant force
Δh	-	Difference of waterhead between the inside and outside of the foundation pit
γ_w	-	Unit weight of water

Terrain Settlement behind the Shoring Structure

Settlement of terrain behind the shoring structure can be analyzed according to the following theories:

- Triangle method
- Index method
- Parabolic method
- DG/TJ08-61-2010

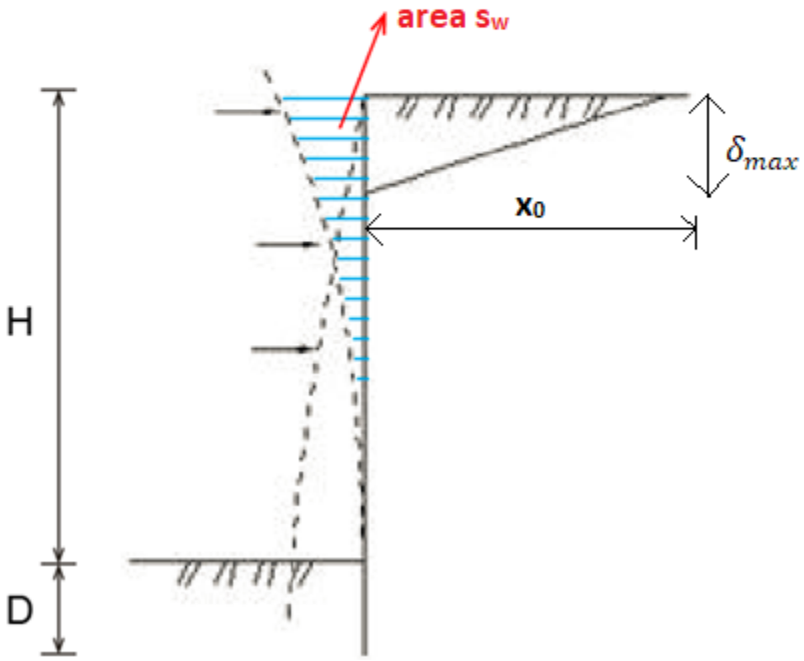
Triangle Method

Settlement of terrain behind the shoring structure is calculated according to the following formula in case of **triangle method**:

$$\delta_{max} = \frac{2 \cdot s_w}{x_0}$$

where:

δ_{max}	-	maximum settlement
s_w	-	area formed by the deformed structure and un-deformed structure
x_0	-	range of settlement



The range of terrain settlement x_0 is defined as follows:

$$x_0 = (H + D) \cdot \tan\left(\frac{\pi}{4} - \frac{\varphi}{2}\right)$$

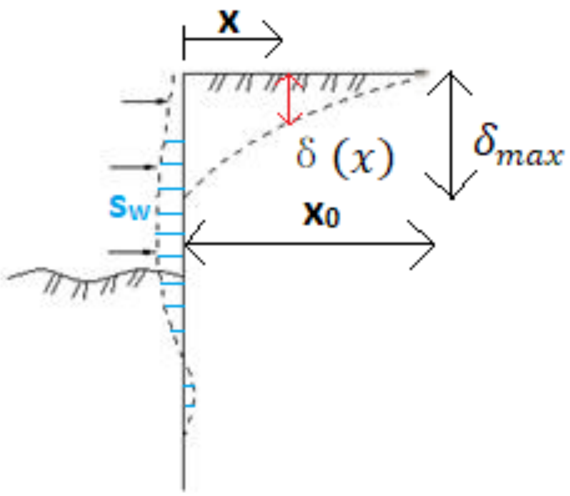
where:

H	-	depth of excavation
D	-	length of the structure below the bottom of foundation pit
φ	-	weighted average internal friction angle of all soil layers that the structure goes through

Index Method

Settlement of terrain behind the shoring structure is calculated according to the following formula in case of **index method**:

$$\delta_{max} = \frac{3 \cdot s_w}{x_0}$$



$$\delta\ (x) = e^{c(x_0 - x)} - 1$$

$$c = \ln\left(\frac{3s_w}{x_0} + 1\right)/x_0$$

where:

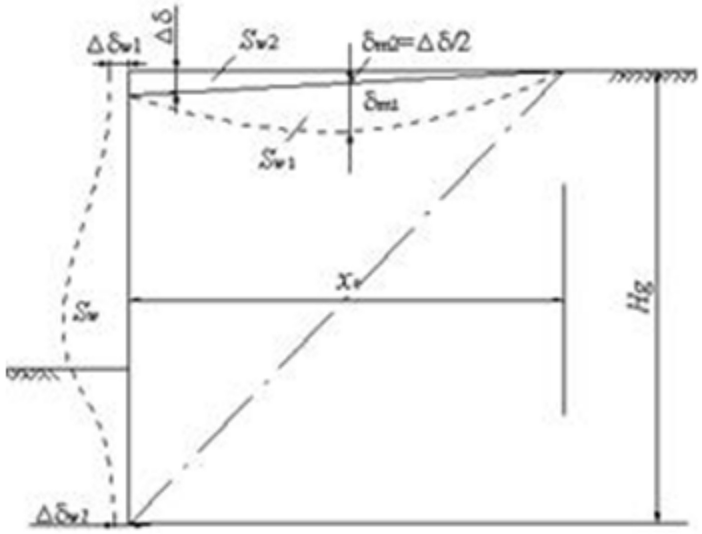
where:	δ_{max}	-	maximum settlement
	s_w	-	area formed by the deformed structure and un-deformed structure
	x_0	-	range of settlement
	$\delta\ (x)$	-	settlement at each point within the range of settlement x_0 in distance x
	x	-	distance between the calculation point and the edge of the foundation pit

Note: Calculation method for the range of settlement x_0 is as same as in case of triangle method.

Parabolic Method

Settlement of terrain behind the shoring structure is calculated according to the following formula in case of **parabolic method**:

$$\delta\ (x) = 4 \cdot \delta_{max} \cdot \frac{x \cdot (x_0 - x)}{x_0^2} + \Delta\delta \cdot \frac{x_0 - x}{x_0}$$



$$\Delta \delta = \frac{1}{2} \cdot (\delta_{w1} + \delta_{w2})$$

$$\delta_{max} = \frac{1.6 \cdot s_w}{x_0} - 0.3 \cdot \Delta\delta$$

where:

	δ_{max}	-	maximum settlement
	s_w	-	area formed by deformed structure and un-deformed structure
	x_0	-	range of settlement
	$\delta(x)$	-	settlement at each point within the range of settlement x_0 in distance x
	x	-	distance between the calculation point and the edge of the foundation pit
	δ_{w1}	-	horizontal displacement at the top of the structure
	δ_{w2}	-	horizontal displacement at the bottom of the structure

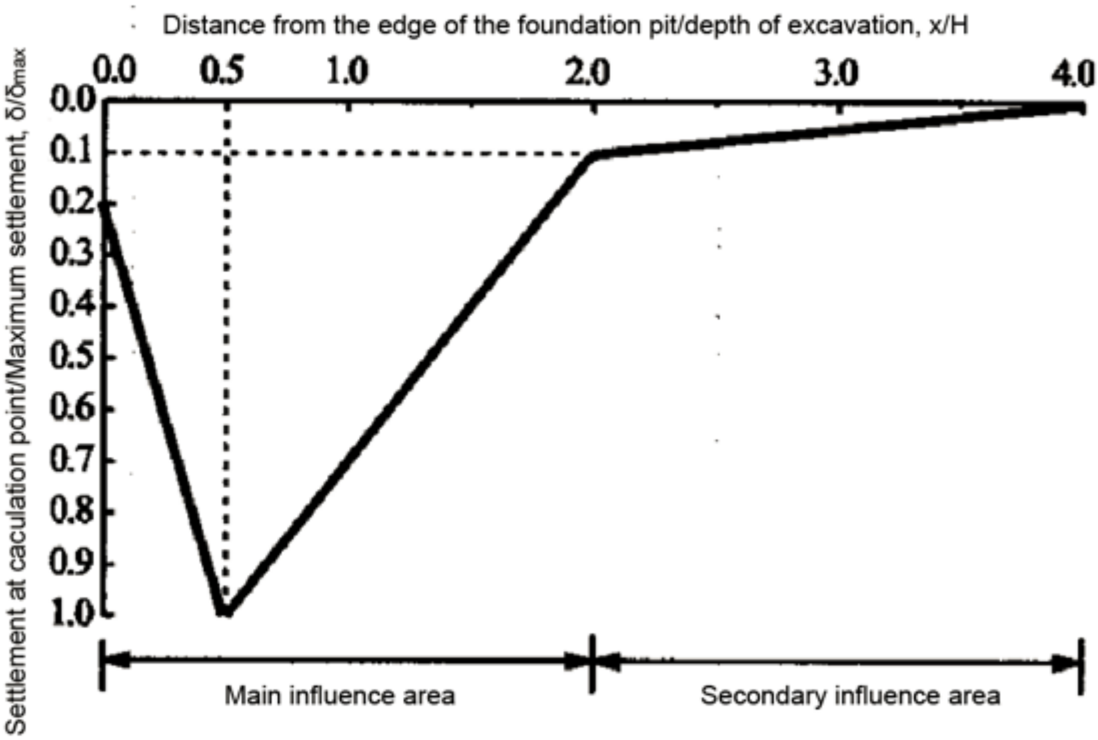
***Note**: δ_{max} is calculated from Peck theory, while $\delta(x)$ is assumed as a parabolic curve. It's obvious that the maximum value of $\delta(x)$ is not equal to δ_{max}, but the maximum settlement is still considered as δ_{max}.*

***Note**: The calculation method for the range of settlement x_0 is the same as in the case of triangle method.*

DG/TJ08-61-2010

Settlement of the terrain behind the shoring structure is calculated according to the **DG/TJ08-61-2010** (Technical code for excavation engineering in Shanghai) and it is suitable for use in soft soils.

Settlement behind the sheeting differs in two parts - **main influence area** and **secondary influenced area**.



Maximum settlement δ_{max} is calculated as follows:

$$\delta_{max} = 0.8\delta_{hm}$$

where: δ_{max} - maximum settlement

δ_{hm} - maximum horizontal displacement of the structure

The range of settlement x_0 is calculated as follows:

$$x_0 = 4H$$

where: x_0 - range of settlement

H - depth of excavation

Settlement at each point $\delta(x)$ in range of settlement x_0 is calculated as follows:

$$For\ x \leq 0.5H,\ \delta\,(x) = \frac{1.6\delta_{max}}{H}x + 0.2\delta_{max}$$

$$For\ 0.5H < x \leq 2H,\ \ \delta\,(x) = -\frac{0.6\delta_{max}}{H}x + 1.3\delta_{max}$$

$$For\ 2H < x \leq 4H,\ \delta\,(x) = -\frac{0.05\delta_{max}}{H}x + 0.2\delta_{max}$$

where: $\delta\,(x)$ - settlement at each point within the range of settlement or influenced area

x - distance between the calculation point and the edge of the foundation pit

Literature:

DG/TJ08-61-2010 (Technical code for excavation engineering in Shanghai)

Utilization of Passive Resistance

The program allows us to **compare the maximum allowable and really used passive resistance** below the ditch bottom. Even if this verification is not standardly required, it can be useful for the optimization of the embedded length of the structure in the soil.

Verification is done using safety factor approach:

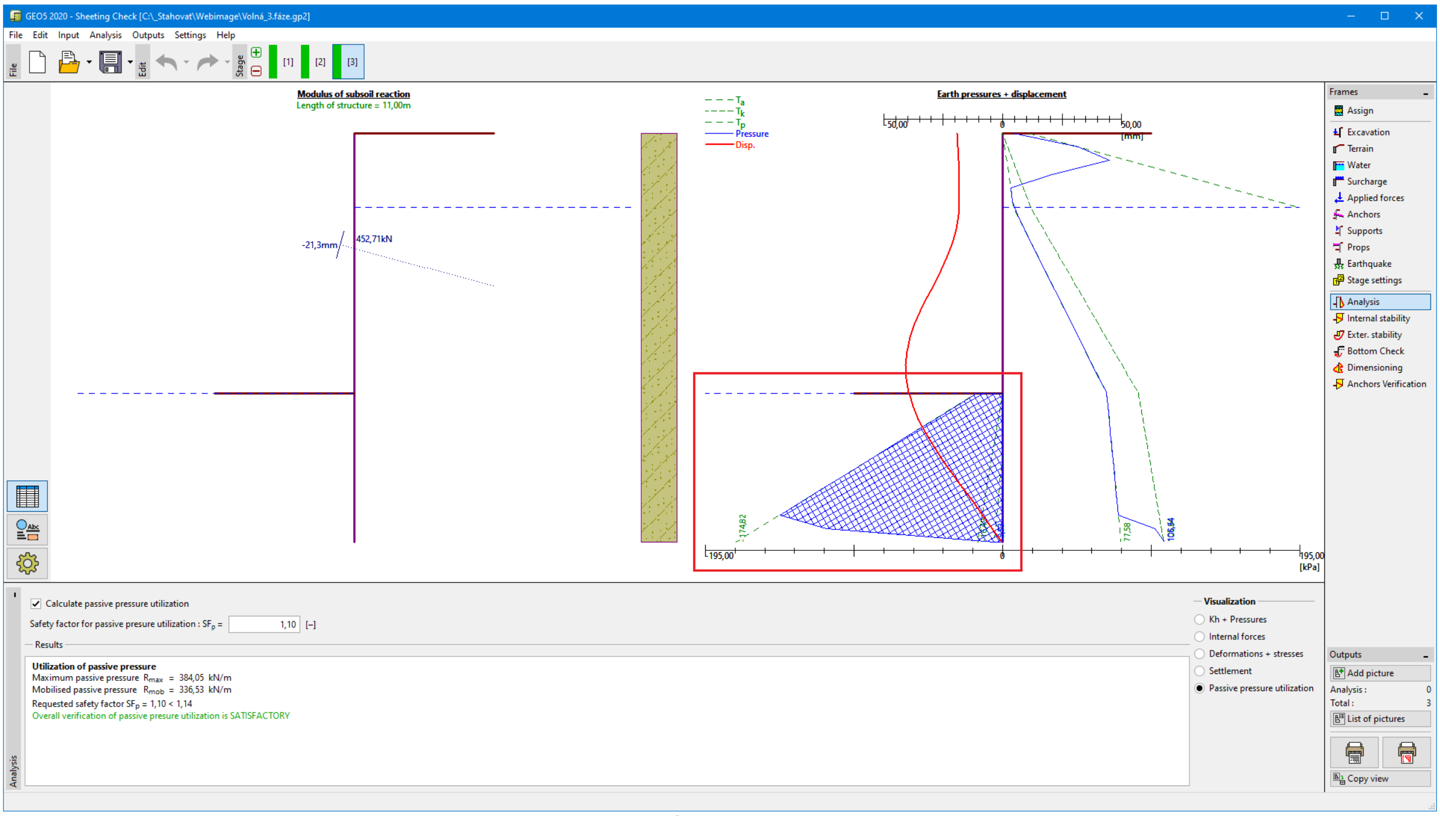
$$SF_p < \frac{R_{max}}{R_{mob}}$$

kde: SF_p - Required safety factor for the utilization of the passive resistance

R_{max} - Maximum allowable passive resistance below the ditch bottom

R_{mob} - Mobilized passive resistance below the ditch bottom

Mobilized passive resistance R_{mob} is highlighted by a pattern in the program.



Utilization of passive resistance

Vertical Bearing Capacity

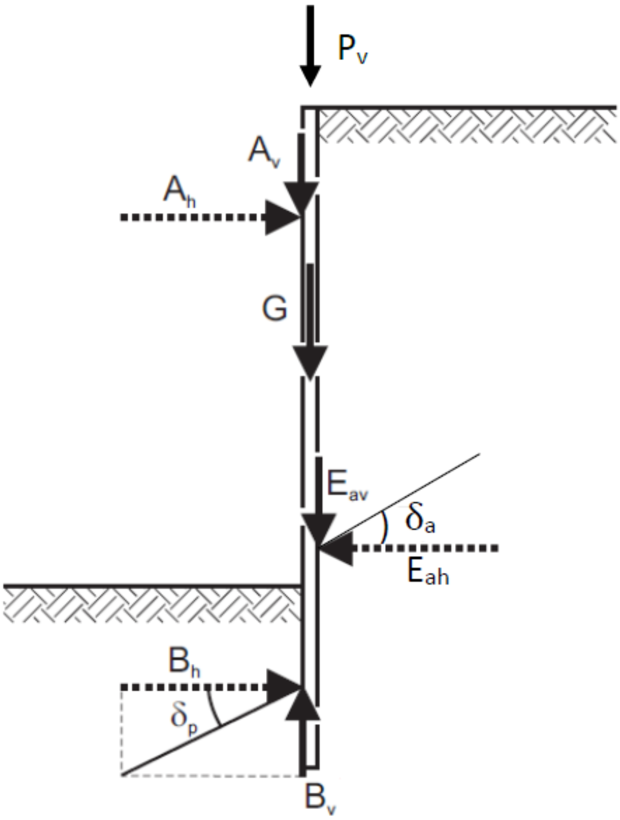
The program allows us to analyze:

- mobilization of earth resistance
- vertical bearing capacity

Both analyses can be verified according to the safety factors (ASD) or German standards EB9/EB84.

Mobilization of Earth Resistance

Mobilization of earth resistance



Forces acting on the structure

Verification according to the Safety Factors (ASD) methodology:

$$SF_f < \frac{P_v + A_v + G + E_{av}}{B_v}$$

Verification according to the EB9/EB84 methodology:

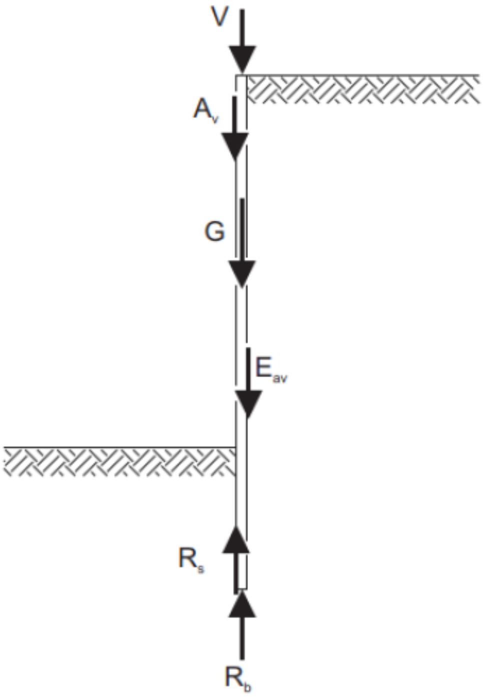
$$F_{vk} > P_{vk}$$
$$F_{vk} = P_v + A_v + G + E_{av}$$
$$P_{vk} = B_v$$

where:

SF_f	-	required safety factor for the mobilization of earth resistance
P_v	-	permanent vertical load
A_v	-	vertical forces from anchors
G	-	self-weight of wall
E_{av}	-	friction force behind the wall
B_v	-	friction force in front of the wall

Vertical Bearing Capacity

Vertical bearing capacity



Forces acting on the structure

Verification according to the Safety Factors (ASD) methodology:

$$SF_v < \frac{R_b + R_s}{V + A_v + G + E_{av}}$$

Verification according to the EB9/EB84 methodology:

$$F_{vd} < R_{vd}$$
$$F_{vd} = (P_v + A_v + G + E_{av}).\gamma_G + P_{v,t}.\gamma_Q$$
$$R_{vd} = (R_b + R_s)/k_V$$

where:

SF_v	-	required safety factor for the vertical bearing capacity
V	-	vertical load (permanent and transient)
P_v	-	permanent vertical load
$P_{v,t}$	-	transient vertical load
γ_G	-	partial factor for permanent load
γ_Q	-	partial factor for transient load
A_v	-	vertical forces from anchors
G	-	self-weight of wall
E_{av}	-	friction force behind the wall
B_v	-	friction force in front of the wall
R_b	-	base resistance
R_s	-	shaft resistance
k_V	-	reduction factor of vertical bearing capacity

Base Resistance

The vertical bearing capacity of the base is given by the formula:

$$R_b = A_b.q_0$$

where:

R_b	-	base resistance
A_b	-	base area
q_0	-	peak resistance in base

Determination of peak resistance q_0 should be part of the geological survey.

Recommended values of peak resistance q_0 [MPa] can be found in professional literature for specific soils and structure types:

- for sheet piles

	non-cohesive soil		
q_c	**7,5**	**15**	**25**
q_0	7,5	15	20

- for driven pile walls

	q_0					
	non-cohesive soil			cohesive soil		
	q_c			c_u		
s/D_{eq}	**7,5**	**15**	**25**	**100**	**150**	**200**
0,035	2,200-5,000	4,000-6,500	4,500-7,500	0,350-0,450	0,55-0,700	0,80-0,950
0,100	4,200-6,000	7,600-10,200	8,7500-11,500	0,600-0,750	0,85-0,110	0,115-0,150

- for bored pile walls

	q_0					
	non-cohesive soil			cohesive soil		
	q_c			c_u		
s/D_{eq}	**7,5**	**15**	**25**	**100**	**150**	**250**
0,02	0,550-0,800	1,050-1,400	1,750-2,300	0,350-0,450	0,600-0,750	0,950-1,200
0,03	0,700-1,050	1,350-1,800	2,250-2,950	0,450-0,550	0,700-0,900	1,200-1,450
0,10	1,600-2,300	3,000-4,000	4,000-5,300	0,800-1,000	0,1200-0,1500	1,600-2,000

where:
- q_c - cone penetration resistance, CPT [MPa]
- c_u - total cohesion of the soil [kPa]
- s - pile settlement
- D_{eq} - pile diameter

Literature:

EA-Pfähle, ISBN: 978-3-433-03005-9.

Shaft Bearing Capacity

The vertical bearing capacity of the shaft is given by the formula:

$$R_s = A_s . q_s$$

where:
- R_s - shaft resistance
- A_s - contact area
- q_s - skin friction

Note: The contact area is calculated for rectangular and pile walls, for other cross-sections, it must be defined manually [m^2/m , ft^2/ft].

Determination of skin friction q_s should be part of the geological survey.

Recommended values of peak resistance q_s [kPa] can be found in professional literature for specific soils and structure types:

- for sheet piles

	non-cohesive soil		
q_c	**7,5**	**15**	**25**
q_s	20	40	50

- for driven pile walls

	q_s					
	non-cohesive soil			cohesive soil		
	q_c			c_u		
$s\,/D_{eq}$	**7,5**	**15**	**25**	**100**	**150**	**200**
$s_{sg}*$	30-40	65-90	85-120	20-30	35-50	45-65
$s_{sg} = s_g = 0{,}1Deq$	40-60	95-125	125-160	20-35	40-60	55-80

- for bored pile walls

	non-cohesive soil		
q_c	**7,5**	**15**	**25**
q_s	55-80	105-140	130-170

where:

q_c - cone penetration resistance, CPT $[MPa]$

c_u - total cohesion of the soil $[kPa]$

$s_{sg}*$ - settlement at shaft resistance activation

$s_{sg} = s_g$ - limit settlement

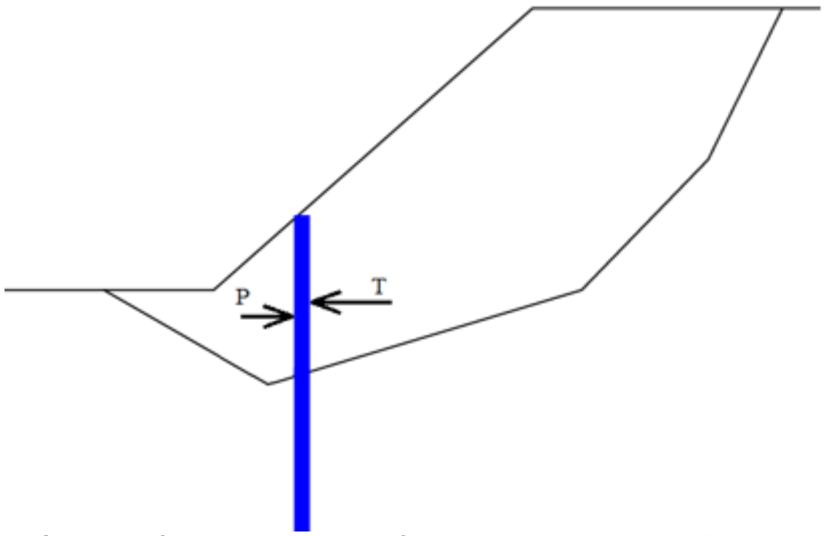
Literature:

EA-Pfähle, ISBN: 978-3-433-03005-9.

Anti-Slide Pile

Program Anti-slide pile performs analysis of anti-slide piles (calculation of internal forces + displacement, dimensioning of pile cross-section). The analysis of structure is almost the same as in the program "**Sheeting check**", the difference is the determination of pressures above the slip surface and possibility to model pile fixed into the rock.

If there is an input slope or structure found an unacceptable slip surface, then it is possible to increase the slope stability on this surface by inserting an anti-slide pile ("Anti-Slide piles" frame serves to perform this step in the "Slope stability" program). The pile has to be placed such that it crosses the slip surface and its base is found sufficiently deep below the assumed slip surface. Above the slip surface, the pile is loaded by an active force T having a tendency to shift the pile and by a passive (resisting) force P stabilizing the pile (see figure below). The difference between the passive and active forces creates a load that must be transferred by the pile to increase stability on a input slip surface up to the required value of SF_s.



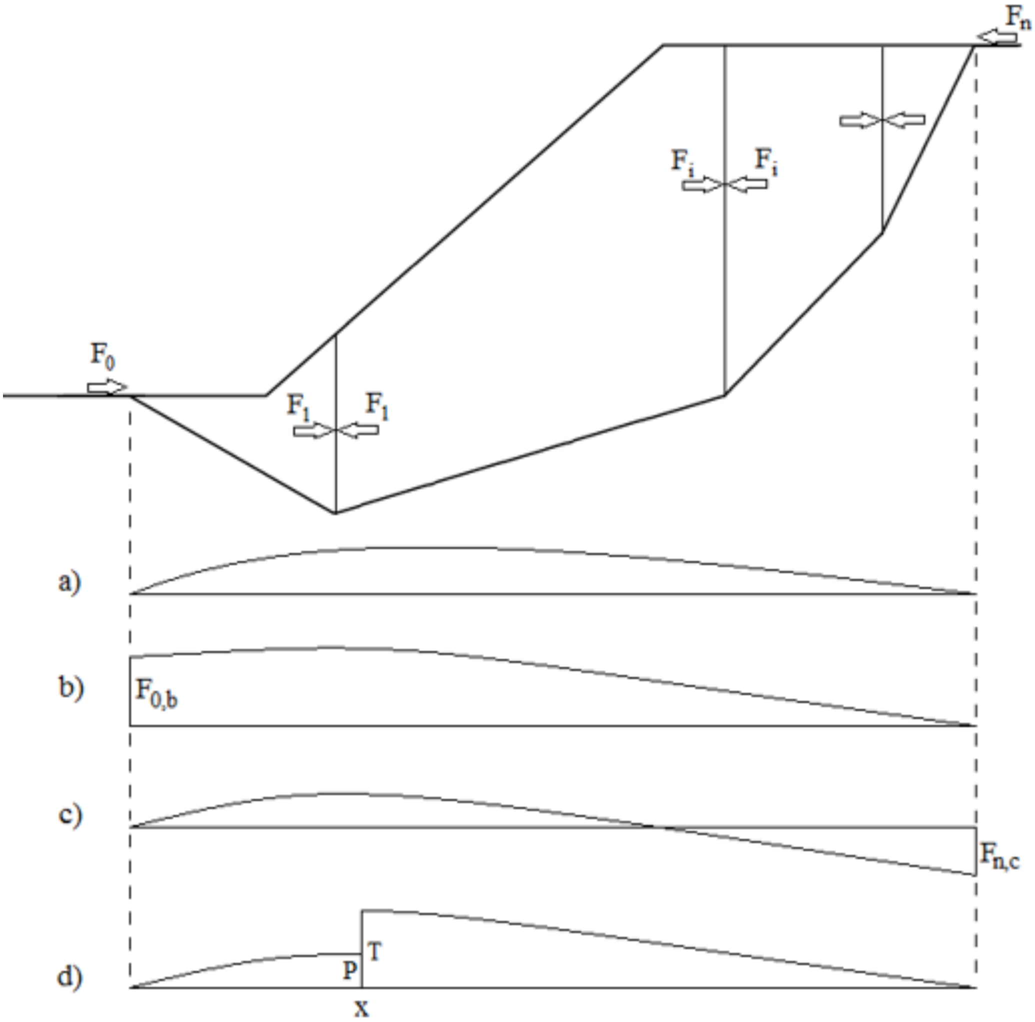
Scheme of active and passive forces acting on the anti-slide pile

If the slope stability SF without an anti-slide pile is not sufficient, the active and passive forces are in equilibrium - the pile is not loaded and there is no need to use it. For calculation of forces acting on a pile, it is thus important that the required factor of safety SF_s is greater than the one calculated for a given slip surface SF without a pile, or the required safety is larger than the one associated with an input slip surface in the absence of an anti-slide pile.

Determination of Forces Acting on an Anti-Slide Pile

Forces, acting on an anti-slide pile, are provided by the slope stability analysis. Calculation of slope stability SF is based on the equilibrium analysis of forces acting on blocks of soil above the slip surface. Vertical surfaces of individual blocks are subjected to action inter-block forces F_i and their determination is one of the steps of the slope stability analysis. If the soil blocks are exactly in the state of limit equilibrium, the inter-block forces at the beginning and at the end of the slip surface are equal to zero. The limit factor of safety SF_{lim}, for which the state of limit equilibrium is reached, is considered as a real factor of safety on an input slip surface. The distribution of inter-block forces along the sliding length is called the **pressure line**. The forces acting on an anti-slide pile are determined from the distribution of pressure lines calculated for the required factor of safety SF_s.

The following figure shows different distributions of inter-block forces F_i (pressure lines). **Graph a)** displays the distribution of forces F_i in the state of limit equilibrium, where zero values of inter-block forces are found both at the beginning and at the end of the distribution. It means that this state has been found for the value of the factor of safety SF_{lim}, which exactly expresses the degree of safety on an input slip surface. **Graph b)** displays the pressure line determined for a higher factor of safety than the one corresponding to SF_{lim}. An assumption of zero force F_n is adopted at the top of the slip surface and a non-zero force $F_{0,b}$ at the bottom of the slip surface is subsequently recorded. This means that in order to arrive at the factor of safety SF, the base of the slope would have to be loaded by the pressure force having the value of $F_{0,b}$. **Graph c)** displays the pressure line for a factor of safety SF greater than SF_{lim}. It results from the assumption of a zero force F_0 on the bottom end of the slip surface at the top and then an unbalanced force $F_{n,c}$ is encountered. Arriving at the state of equilibrium for the factor of safety SF would thus theoretically call for the action of a tensile force of this magnitude at the top end of the slip surface. **Graf d)** shows a distribution of inter-block forces for the case when an anti-slide pile is placed at point x. Part of the distribution corresponds to graph c) and serves to determine the magnitude of passive force P at point x. Above the pile the distribution according to graph b) is considered and serves to determine the active force T at point x. The difference between P and T is the force transmitted by the anti-slide pile.



Distribution of inter-block forces F_i

Figure caption (explanation):

a) distribution for the factor of safety $SF = SF_{lim}$ $(F_0 =$ 0 a $F_n =$ 0)

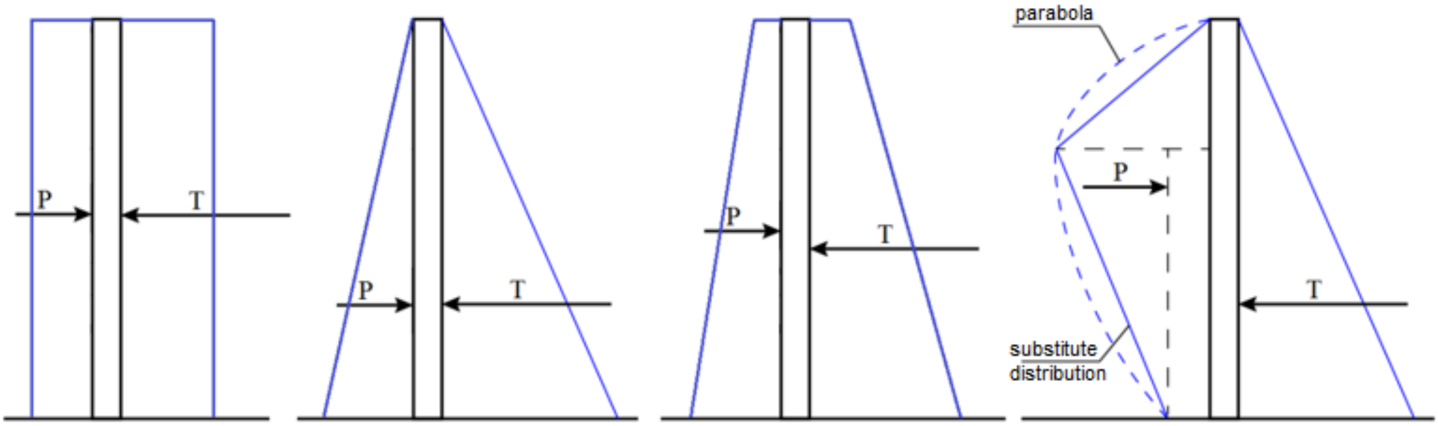
b) distribution for the factor of safety $SF > SF_{lim}$ with zero value of force at the top end

c) distribution for the factor of safety $SF > SF_{lim}$ with zero force value at the bottom end

d) distribution for the factor of safety SF with an anti-slide pile at point x

Distribution of Pressures Above the Slip Surface

The distribution of load applied to an anti-slide pile above the slip surface is determined from the magnitudes of forces P and T. **Constant**, **triangular** or **trapezoidal** distributions are considered (for "**Anti-Slide Pile**" program the distribution of active and passive forces is introduced in the "Determination of earth pressure" frame). For the passive (resisting) force P can also be considered a **parabolic** distribution, which is approximated for the simplification by combining the triangular and the trapezoidal part.



Types of distribution of load applied to an anti-slide pile above slip surface

Recommendation for active force distribution

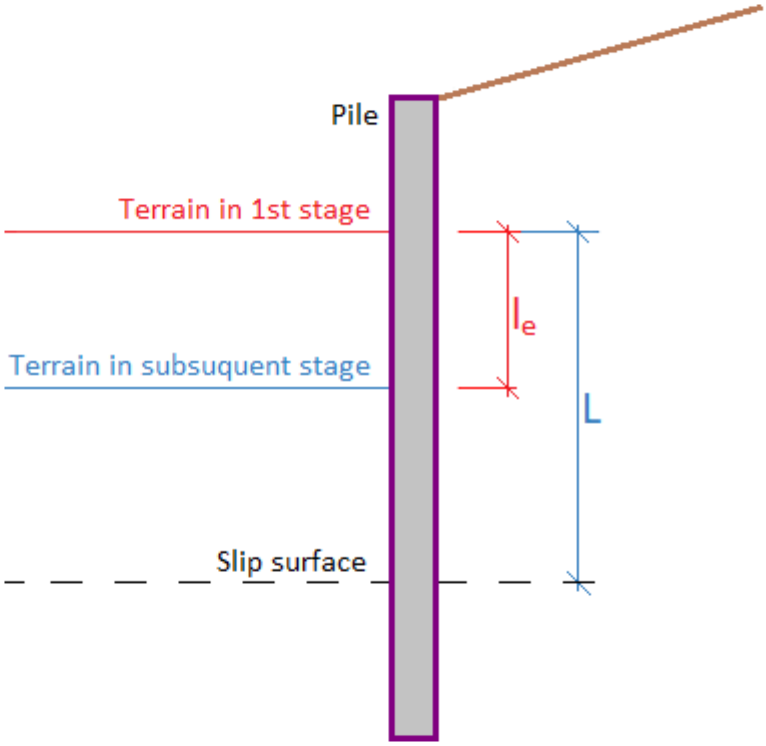
- Triangle distribution - the layers above the slip surface are made from gravel or sand
- Rectangular distribution - the layers above the slip surface are made from fine-grained soils (clay, silt)
- Trapezoidal distribution - the layers above the slip surface are made from different types of soils

Calculation of passive force in subsequent stage

If there is new excavation in front of the pile, the **passive force** can be **automatically calculated** according to the formula:

$$F = \frac{L - l_e}{L} F_0$$

Where:
- F - new passive force after excavation
- L - the depth of slip surface below the terrain in the first stage
- l_e - the distance between the terrain in the first and current stage
- F_o - original passive force before excavation



Shaft

In the frame "Analysis", program Shaft sets the load acting on the shaft.

The calculated load is then input for analysis of internal forces in the frame "Dimensioning".

Calculation of Load Acting on a Shaft

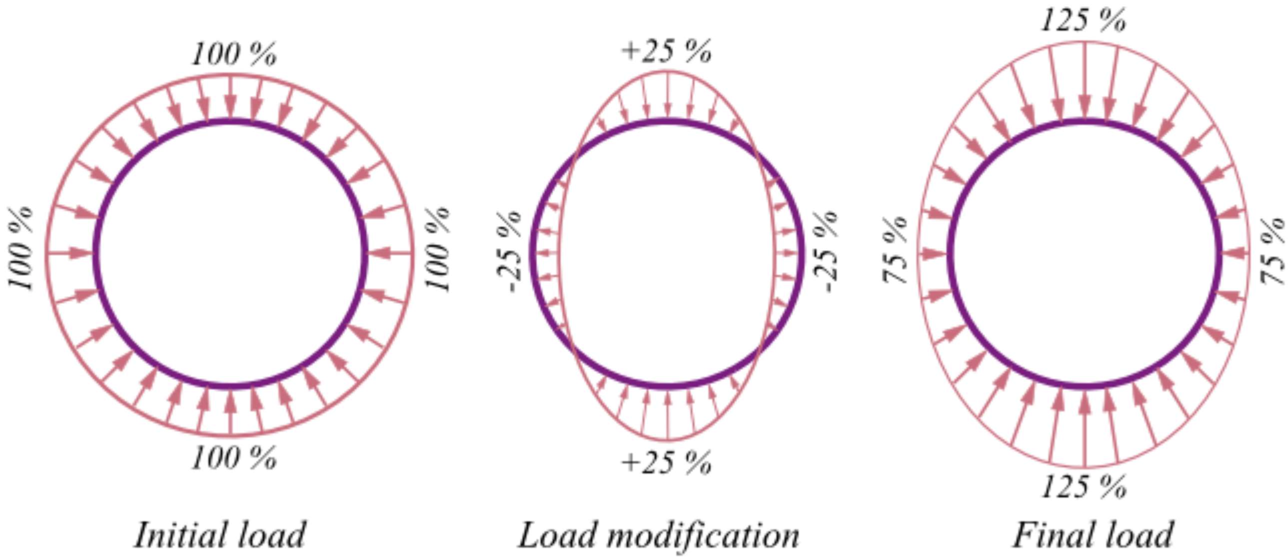
The load from earth pressure and surcharge are computed in the frame "Load analysis". The stiffness of the shaft has a major influence on earth pressure. The rigid structure does not allow deformation, so the earth pressure is much higher than on the flexible shaft.

Three types of shafts can be modeled in the program:

- Flexible - spatial active pressure is considered (earth pressure, surface surcharge and local surcharge)
- Semirigid
- Rigid - spatial pressure at rest is considered (earth pressure, surface surcharge and local surcharge)

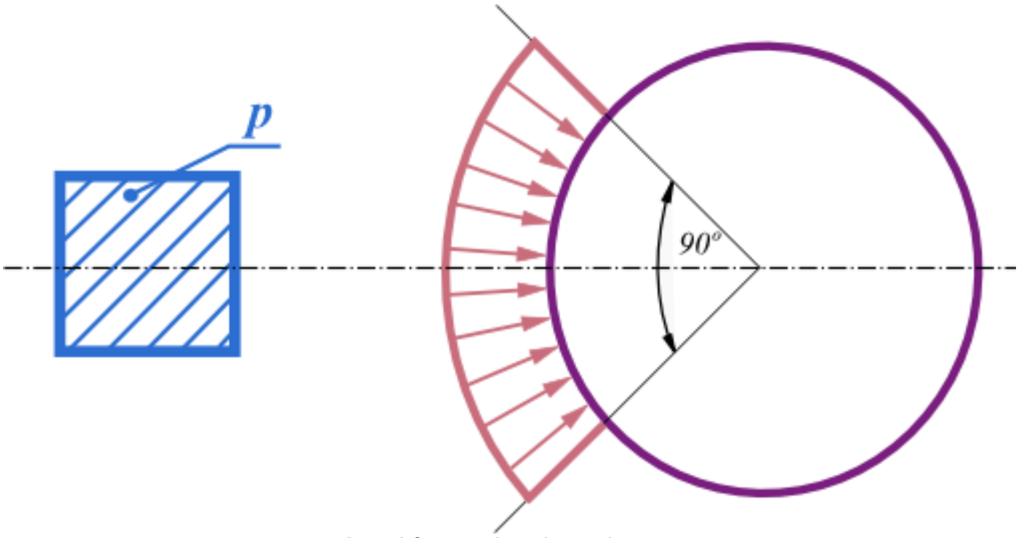
Ways of load determination

Load from earth pressure and surface surcharge acts as a uniform load on the entire diameter. This load causes structure stress only by normal force - bending moment on the shaft is theoretically equal to zero. For modeling of real behavior of the shaft, the program introduces the Reduction coefficient in compliance to the standards DIN V 4034-1 or СНиП II-94-80. The recommended value of a reduction coefficient is 25 *%* .



Modification of uniform load by reduction coefficient

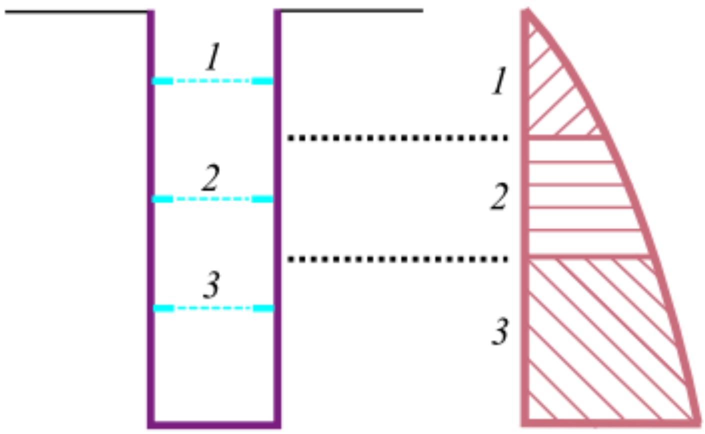
Load from a local surcharge is considered as shown in the next picture. This load is not modified by the reduction coefficient.



Load from a local surcharge

Recalculation of the load on walers

If the walers are input the program computes the load on each of them. The load depends on the axial distance between walers as shown in the picture.



Calculation of load on the waler

If no walers are input, the program computes load on unit depth (1 m , 1 ft).

If load is calculated, then program determines the distribution of internal forces on structure of the shaft.

Literature:

Berezantzev, V. G.: Earth pressure on the cylindrical retaining walls, Brussels conference on Earth pressure problems, 1958.

ČSN 73 0037: Zemní tlak na stavební konstrukce, 1990.

DIN 4085: Berechnung des Erddrucks, 1987.

Exner, K.: Hloubení jam, VŠB v Ostravě, 1986.

Cheng, Y. M.; Hu, Y. Y.: Active earth pressure on circular shaft lining obtained by simplified slip line solution with general tangential stress coefficient. Chinese Journal of Geotechnical Engineering, 27 (1), 110-115, 2005.

Link, H.; Lutgendorf, H.; Stoss, K.: Richtlinien zur Berechnung von Schachtauskleidungen in nicht standfestem Gebirge, 1976.

Sedláček, M.: Zatížení kruhových šachet prostorovým zemním tlakem. Příspěvek ke konferenci Zakládání staveb, 2014.

Snášelová, K.: Hloubení a vyztužování jam v extrémních podmínkách, ODIS VTEI pro uhelný průmysl, 1987.

Tobar, T.; Meguid, M.: Distribution of active earth pressure on vertical shafts, Geo Halifax, 2009.

Valencia, T. T.: An experimental study of the earth pressure distribution on cylindrical shafts, McGill University, Montreal, 2009.

Walz, B.; Pulsfort, M.: Raumliche Erddruck auf Schachtbauwerke in Abhangigkeit von der Wandverformung, Bergische Universitat Wuppertal, 1999.

Flexible Shaft Structure

Conventional excavation method is a typical example of using the shaft with **flexible** construction. Using this method, the rock is excavated in the first step and afterwards concrete is sprayed on to hardened surface to support the rock. There are specific technological downtime (ground excavation, shotcrete or arch timbering) with enabling stress rearrangement in the surrounding soil and the consequent value of the earth pressure acting on the shaft will be equal to the active earth pressure. This fact is well described by V.G. Berezantsev's method (1958).

Load on flexible shaft is defined using this formula:

$$p_a = K_{a\gamma}\,\gamma\,h + K_{aq}q - K_{ac}c_{ef}$$

where:
- γ - unit weight of soil
- h - depth of cross-section
- q - magnitude of a surcharge
- c_{ef} - shear strength of soil

$$K_{a\gamma} = \frac{\tan\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right)}{\eta - 1}\left(\frac{r_0}{h} - \frac{r_0}{h R_b^{\eta-1}}\right)$$

$$K_{aq} = \frac{1}{R_b^{\eta}}\tan^2\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right)$$

$$K_{ac} = \left[\frac{1-\lambda+\eta}{\eta} - \frac{\xi}{R_b^{\eta}}\tan^2\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right)\right]\cot\varphi_{ef}$$

where:
- r_0 - radius of shaft
- φ_{ef} - angle of internal friction of the soil

$\xi = 1$

$\lambda = 1$

$$\eta = \tan^2\left(\frac{\pi}{4} + \frac{\varphi_{ef}}{2}\right) - 1$$

$$R_b = 1 + \frac{h}{r_0}\tan\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right)$$

Literature:

Berezantzev, V. G.: Earth pressure on the cylindrical retaining walls, Brussels conference on Earth pressure problems, 1958.

Semirigid Shaft Structure

Sheet piles are a typical example of **semirigid** shaft support. The sheet piles are driven into the ground in the first construction stage and then the soil is excavated. The partial soil stress rearrangement in the shaft surrounding is permitted due to slightness of the construction. The resulting earth pressure value acting on the shaft construction moves between active and earth pressure.

Load on the semirigid shaft is considered an average value of load of a flexible and rigid shafts.

Rigid Shaft Structure

Shaft supported using secant pile walls is a typical example of the shaft with rigid construction. In the first construction stage, the shaft construction is built and therefore the soil is excavated. The load on the shaft due to earth pressure is equal to the at rest earth pressure because the rigid construction has minimal deformations. This construction behavior is well described by Y. M. and Y. Y. Hu theory (2005).

Load on rigid shaft is defined using this formula:

$$p_a = K_{a\gamma}\,\gamma\,h + K_{aq}\,q - K_{ac}\,c_{ef}$$

where: γ - unit weight of the soil
h - depth of the cross-section
q - magnitude of a surcharge
c_{ef} - shear strength of the soil

$$K_{a\gamma} = \frac{\tan\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right)}{(1-\sin\varphi_{ef})\,Z}\left(\left[1 + Z\tan\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right)\right]^{1-\sin\varphi_{ef}} - 1\right)$$

$$K_{aq} = \frac{1}{\left[1 + Z\tan\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right)\right]^{\sin\varphi_{ef}}}\tan^2\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right)$$

$$K_{ac} = \left[2 - \frac{1}{\left[1 + Z\tan\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right)\right]^{\sin\varphi_{ef}}}\sec^2\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right)\right]\cot\varphi_{ef}$$

where: r_0 - radius of the shaft
φ_{ef} - angle of internal friction of the soil

$\eta = \sin\varphi_{ef}$

$Z = \frac{h}{r_0}$ ratio of the depth of cut h to the radius of the shaft r_0

$$\xi = \sec^2\left(\frac{\pi}{4} + \frac{\varphi_{ef}}{2}\right)$$

$$R_b = 1 + \frac{h}{r_0}\tan\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right)$$

$$\lambda = 1 - \sin\varphi_{ef}$$

Literature:

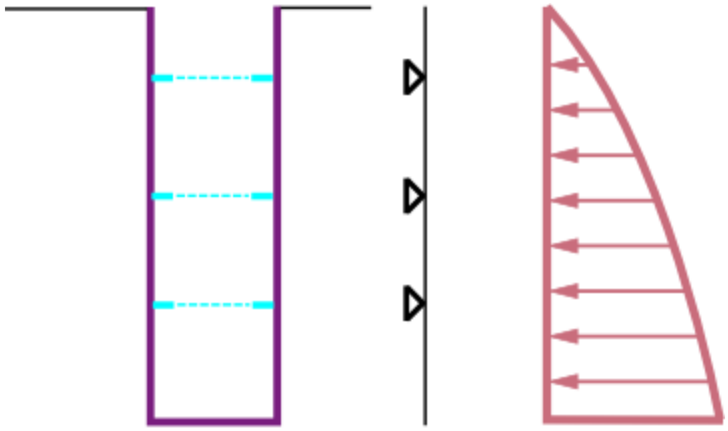
Cheng, Y. M.; Hu, Y. Y.: Active earth pressure on circular shaft lining obtained by simplified slip line solution with general tangential stress coefficient. Chinese Journal of Geotechnical Engineering, 27 (1), 110-115, 2005.

Calculation of Internal Forces on a Shaft (Dimensioning)

The program allows analysis of the internal forces acting on the structure in the horizontal and vertical direction with an intended load acting on the shaft.

Analysis of internal forces in the vertical direction

The analytical model of the structure is shown in the picture. All walers are modeled as supports. The analysis is performed for a unit width (1 ***m*** , 1 ***ft***) of the structure. Structures without walers or with one waler cannot be analysed in the vertical direction.

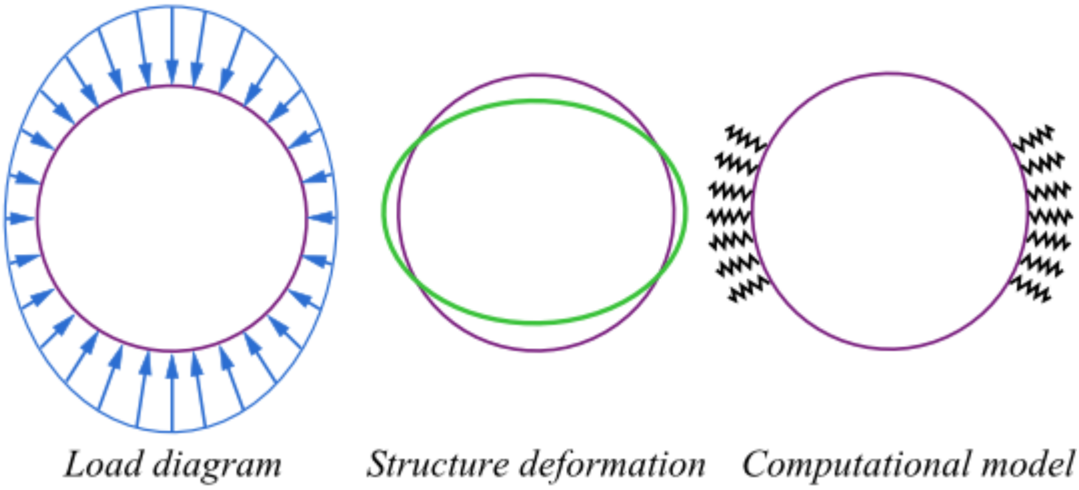


Analytical model in vertical direction

Analysis of internal forces in the horizontal direction (polygonal method)

Internal forces in the horizontal direction are computed by the polygonal method, where the circular structure is divided into 72 segments. Each segment is supported by a non linear spring, acting only in compression. The stiffness of springs is equal to the input modulus of a subsoil reaction.

The way of analysis is shown in the picture - if the structure deforms in direction to the center, the springs are removed from the analysis.



Analytical model in horizontal direction

Slope Stability

Slope stability program computes the stability of slopes and embankments with a circular or polygonal slip surfaces.

The slope stability problem is solved in a two-dimensional environment. The soil in a slope body can be found below the groundwater table, water can also exceed the slope ground, which can be either partially or completely flooded. The slope can be loaded by a surcharge of a general shape either on the ground or inside the soil body. The analysis allows us to include the effect of anchors expected to support the slope or for the introduction of horizontal reinforcing elements - reinforcements or vertical elements - anti-slide piles. An earthquake can also be accounted for in the analysis.

Two types of approaches to the stability analysis are implemented in the program - classical analysis according to the factor of safety and the analysis following the theory of limit states.

The slip surface can be modeled in two different ways. Either as a circular one, then the user may choose either from the Fellenius/Petterson, Bishop, Spencer, Janbu or Morgenstern-Price, Shahunyants, ITF method or as a polygonal one, in which case the program exploits the Sarma, Spencer, Janbu or Morgenstern-Price, Shahunyants, ITF method.

Soil Body

The soil body is formed by a **layered profile**. An arbitrary number of layers can be used. Each layer is defined by its geometry and material. The material of a layer is usually represented by a **soil** with specified properties. The geostatic stress in the soil body is determined during the analysis.

A layer can be specified also as a rigid body. Such a layer then represents bedrock or a sheeting wall. The slip surface can never pass through the rigid body.

Influence of Water

Groundwater can be assigned to the slope plane section using one of the five options:

1) Groundwater table

The groundwater table is specified as a polygon. It can be arbitrarily curved, placed totally within the soil body or introduced partially **above the ground surface**.

Presence of water influences the value of pore pressure acting within the soil and reducing its shear bearing capacity. The pore pressure is considered as the hydrostatic pressure, i.e. unit weight of the water is multiplied by the reduced height of the water table:

$$u = \gamma_w h_r$$

where: γ_w - unit weight of water

h_r - reduced height of the water table

$$h_r = h \cos^2 \alpha$$

where: h - vertical distance of point, where pore pressure is calculated and point on the water table

α - inclination of the water table

Resultant force of pore pressure at a certain section of the block is used in the calculation:

$$U = ul$$

where: u - pore pressure in the point

l - length of section

Below the groundwater table, the analysis proceeds using the unit weight of saturated soil γ_{sat} and uplift pressure; above the groundwater table the analysis assumes the input unit weight of soil γ .

The shear forces along the slip surface are provided by:

$$T = (N - U) \tan \varphi + cd$$

where: T - shear force along the slip surface segment

N - normal force along the slip surface segment

U - pore pressure resultant along the slip surface segment

φ - angle of the internal friction

c - cohesion

d - length of the slip surface segment

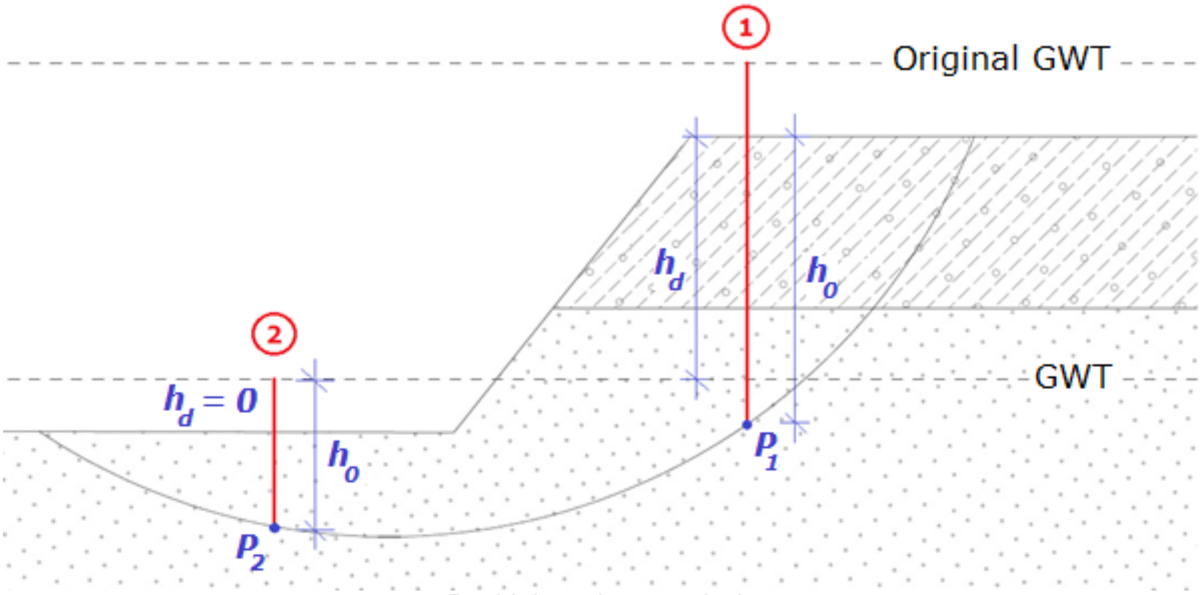
In case of a total stress (entered in the "Soil" dialog window) total parameters are used and pore pressure is considered zero.

2) Groundwater table including suction

The suction table can be introduced above the input groundwater table. A negative value of the pore pressure u is then assumed with the region separated by the two tables. Suction increases as a negative hydrostatic pressure from the groundwater table towards the suction table.

3) Rapid draw down

The original table can be introduced above the input groundwater table. The original water table simulates the state before rapid drawdown.



Rapid draw down analysis

First of all, the initial pore pressure u_0 is evaluated:

$$u_0 = \gamma_w h_0$$

where: h_0 - height from the original table to the point of evaluation P

γ_w - unit weight of water

Height h_0 is generally the distance from the point of pore pressure evaluation (**P**) to original water table - this is valid for the case when the original water table is under the terrain surface. In case of original water table above terrain there is used the height h_0 from point **P** to the level of terrain surface (***profile 1*** in the figure). Another case is the when original water table, as well as groundwater table are both above terrain - then height h_0 is the distance from groundwater table to point **P** (***profile 2*** in the figure).

The second step is to calculate the change of pore pressure in the area between the original and groundwater table:

$$\Delta u = \gamma_w h_d$$

where: h_d - height from original to groundwater table

γ_w - unit weight of water

As in previous calculation of pressure, there are three possibilities of how to get height h_d. When both water tables are under the terrain, h_d is the distance between the original and groundwater table. In case the original water table is above the terrain, then h_d is measured from groundwater table to the level of terrain (***profile 1*** in the figure). Last case is when both water tables are above the terrain - then the height h_d is zero (***profile 2*** in the figure).

Third step is the calculation of the final value of pore pressure u. Change of pore pressure Δu is multiplied by the coefficient of reduction of the initial pore pressure X, which is required for all soils (dialog window "Soils"). X coefficient of the soil in the area of point **P** is used (NOT the soil in the area between original and groundwater table). In the case of permeable soil $X =$ 1, in other case $X =$ 0. Final pore pressure is evaluated as:

$$u = u_0 - X\Delta u$$

where: u_0 - initial pore pressure

X - coefficient of reduction of the initial pore pressure

Δu - change of pore pressure

4) Coefficient of pore pressure R_u

The coefficient of pore pressure R_u represents the ratio between the pore pressure and geostatic pressure in a soil body. In the area, where R_u is positive, entered unit weight of saturated soil γ_{sat} is considered; in other case unit weight of soil γ is used.

The values of R_u are introduced with the help of isolines connecting points with the same value of R_u. Linear interpolation is assumed to obtain intermediate values. Pore pressure is established as geostatic stress reduced by coefficient R_u:

$$u = R_u \sum h_i \gamma_i$$

where: R_u - coefficient of pore pressure

h_i - height of i^{th} soil layer

γ_i - unit weight of i^{th} soil layer

5) Pore pressure values

Groundwater can be introduced directly through the pore pressure values u within the plane section of a soil body.

In the area, where u is positive, entered unit weight of saturated soil γ_{sat} is considered; in other case unit weight of soil γ is used.

The pore pressure values are introduced with the help of isolines connecting points with the same value of pore pressure. Linear interpolation is assumed to obtain intermediate values. Pore pressure values are then derived from the values of pore pressure obtained in specific points within the slope plane section.

6) Waterflow analysis

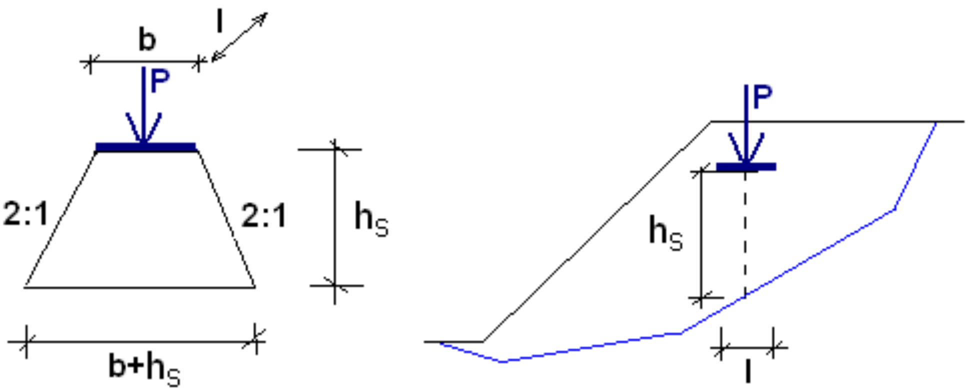
The last option serves to **analyze the pore pressures** in the "Slope Stability - Water Flow" module. This option is **available only** for the users with the "**Slope Stability - Water Flow**" **module purchased**.

Surcharge

The slope stability analysis takes into account even the surcharge caused by neighboring structures. The surcharge can be introduced either as a concentrated force or distributed load acting either on the ground surface or inside the soil body.

Since it is usually assumed that the surcharge is caused by the weight of objects found on the slope body, the vertical component of surcharge having the direction of weight (material component) is added to the weight of blocks. It means that if the earthquake effects are included this component is also multiplied by the factor of horizontal acceleration or vertical earthquake. The material surcharge component also influences the position of block centroid. The components that do not act in the direction of weight are assumed in equations of equilibrium written for a given block as weightless thus neither contributes to inertia effects of the earthquake nor position of block centroid.

The surcharge is always considered in the analysis with respect to one running meter. Providing the surcharge, essentially acting over the area $b*l$, is introduced as a concentrated force it is transformed before running the analysis into a surface load spread up to a depth of slip surface along the slope 2:1 as displayed in the figure.



Scheme of spreading the concentrated load on the slip surface

The analysis then proceeds with the resultant of surface load p having the value:

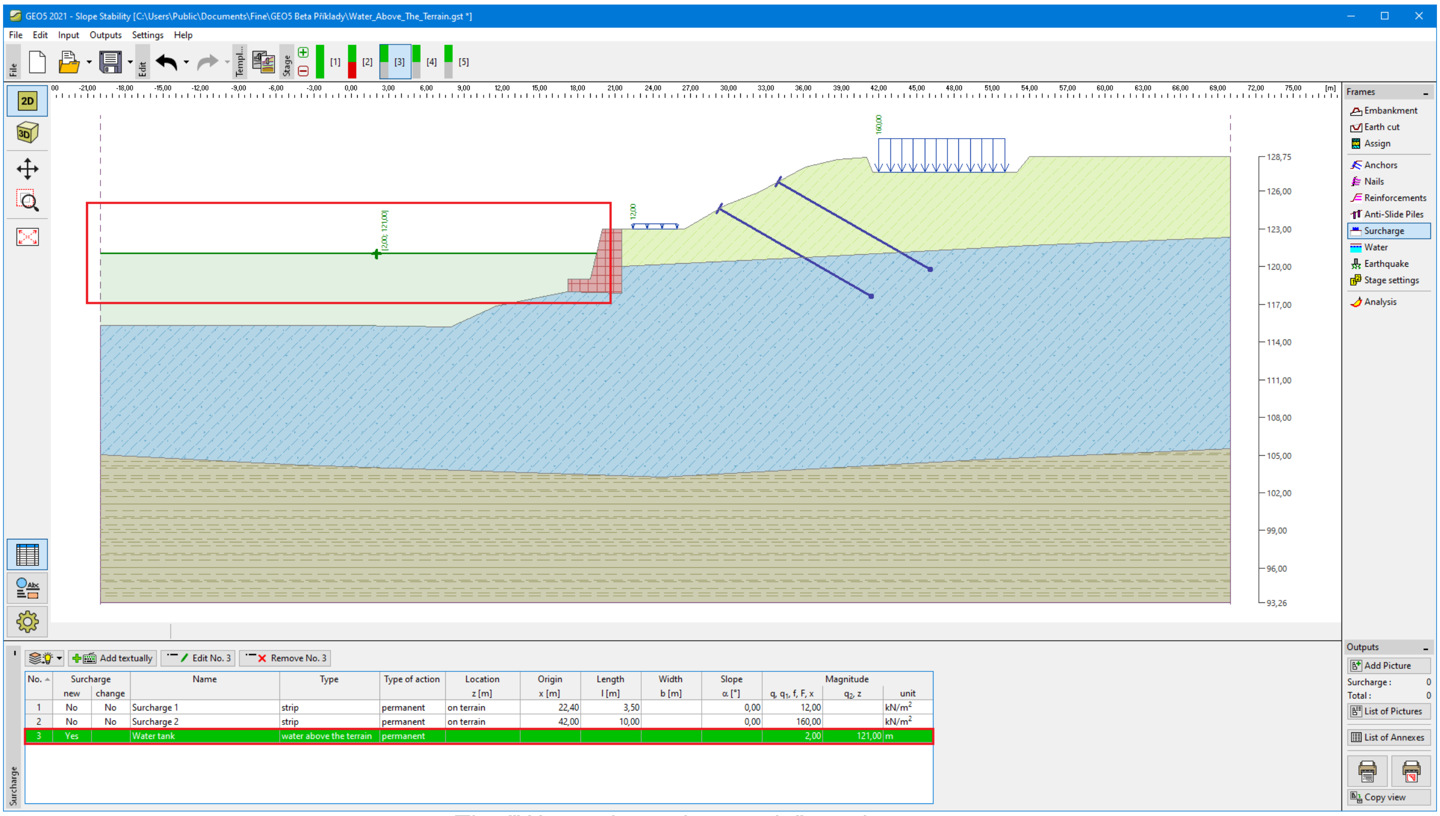
$$p = \frac{P}{(b + h_s).l}$$

It is also possible to consider a special type of surcharge - the water above the terrain.

Water above the Terrain

The **water above the terrain** is a special type of surcharge, which models the water table which does not cross into the soil body. It is typical for water reservoirs which are separated from the soil body by impermeable membranes.

In this case, the water table acts perpendicular to the terrain as a surcharge with magnitude of hydrostatic pressure.

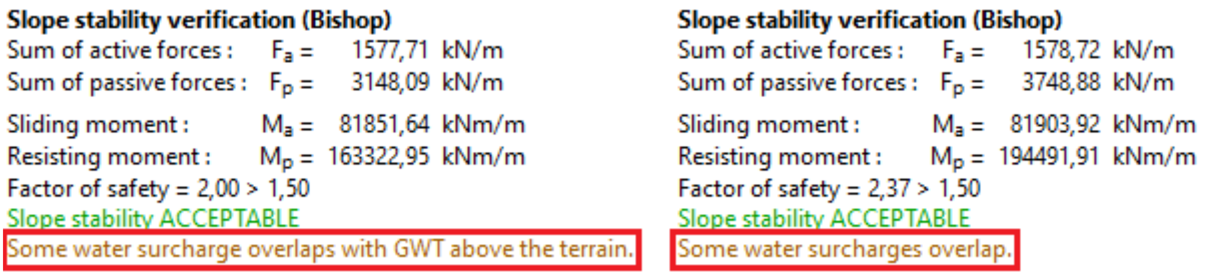


The "Water above the terrain" surcharge type

The water table is represented by one point with coordinates $[x, z]$. The program considers the water in an area bordered by the terrain or task edge and intersecting defined point.

If there is a different liquid than water in the tank, the program allows us to input the unit weight of this liquid. In this case, the program calculates the hydrostatic pressure of liquid with inputted unit weight. When using the EN1997 verification methodology, the partial factor on water γ_w is used.

It is not correct to consider the standard influence of water in the area where the water table is already defined as a surcharge. In this case, the program shows a warning in the analysis and output document. The warning is also shown if two or more "Water above the terrain" surcharges act on the same terrain parts.



Warning about water table overlap

Anchors

Anchor position is specified by head point, free length l, root length l_k and by the slope α. The anchor force is input as the pre-stressing force acting in the anchor. The head point is always located on the ground surface; the anchor force always acts in the direction of a soil body. The anchor force when computing equilibrium on a given block (slice) is added to the weightless surcharge of the slope.

Two options are available to account for anchors:

1. **Compute anchor lengths** - analysis assumes infinite lengths of anchors (anchors are always included in the analysis) and computes the required free lengths of anchors (distance between the anchor head and intersection of the anchor with the slip surface) subsequently. The anchor root is then placed behind the slip surface. This approach is used whenever we wish the anchor to be always active and thus contribute to the increase of the slope stability and we need to know its minimum distance.

2. **Analysis with specified lengths of anchors** - the analysis takes into account only those anchors that have their endpoints of roots behind the slip surface. If the slip surface intersects the free length of the anchor, a calculation is carried out with full anchor force. When the slip surface intersects the anchor root, anchor force is reduced linearly, with full force at the root begin and zero value at the end of the root. This approach is used always whenever we wish to evaluate the current state of the slope with already existing anchors, since it may happen that some of the anchors may prove to be short to intersect the critical slip surface so that they do not contribute to the increase of slope stability.

Nails

For each nail the following bearing capacities are either computed or input:

where: R_f - nailhead strength

R_t - nail strength against breaking

T_p - pull-out nail bearing capacity

Strength characteristics of a nail represent the basic parameters to compute the total bearing capacity of a nail.

The nail strength against breaking follows from:

$$R_t = \frac{\pi d_s^2}{4} f_y$$

where: R_t - strength against breaking

d_s - nail diameter

f_y - strength of nail material

The pull-out nail bearing capacity is calculated by one of the following ways:

1. calculate from skin friction:

$$T_p = \pi \, d \, g_s$$

where: T_p - pull-out nail bearing capacity [kN/m]

d - hole diameter

g_s - ultimate bond strength, given either as a parameter of the nail or as soil parameter.

2. calculate from effective stress

$$T_p = \pi \, d \left(K_a \sigma_z \tan \varphi + c \right)$$

where:

$$K_a = \frac{1 + K_0}{2} = \frac{1 + (1 - \sin \varphi)}{2}$$

where: T_p - pull-out nail bearing capacity [kN/m]

d - hole diameter

σ_z - vertical geostatic stress

φ - effective angle of internal friction of soil

c - effective cohesion of the soil

3. calculate according to HA 68/94

$$T_p = \pi \, d \left(\sigma_n \tan \varphi + c \right)$$

where: T_p - pull-out nail bearing capacity [kN/m]

d - hole diameter

σ_n - average radial effective stress

φ - effective angle of internal friction of soil

c - effective cohesion of soil

Average radial effective stress σ_n is calculated by the following formula:

$$\sigma_n = \frac{(1+K_L)\sigma_z}{2}$$

where: σ_z - vertical geostatic stress

$$K_L = \frac{1+K_a}{2}$$

where:

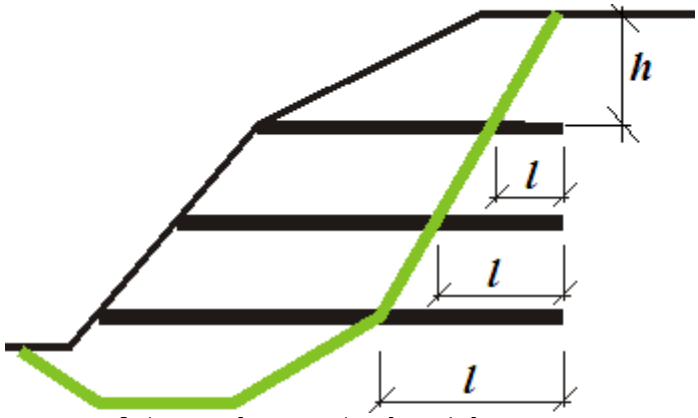
$$K_a = \frac{1-\sin\varphi}{1+\sin\varphi}$$

Nailhead strength value to be input. If the nail head is not fixed, then the strength value equals zero.

Reinforcements

Reinforcements are horizontal reinforcing elements, which are placed into the soil to increase the slope stability utilizing their tensile strength. If the reinforcement intersects the slip surface, the force developed in the reinforcement enters the force equation of equilibrium of a given block. On the contrary, the slope stability is not influenced.

The basic parameter of reinforcement is the **tensile strength** R_t. A design value of this parameter is used - i.e. the strength of reinforcement reduced by coefficients taking into account the effect of durability, creep, and installation damage. The force transmitted by reinforcement **can never exceed the assigned tensile strength** R_t.



Scheme of accounting for reinforcement

The second characteristic is the **pull-out strength** T_p. This parameter determines the anchoring length, i.e. the required length of reinforcement in the soil, for which the reinforcement is fully stressed attaining the value R_t. Since the realistic values of the pull-out strength are difficult to determine, the program offers three options for their calculation, respectively for the calculation of the force F transmitted by the reinforcement.

1) Calculate reinforcement bearing capacity

The pull-out force F is given by:

$$F = 2.\sigma.tg\varphi.C.l$$

where: σ - normal stress due to self-weight at the intersection of reinforcement and slip surface - see Fig.

φ - angle of internal friction of the soil

C - coefficient of interaction (0,8 by default)

l - length of reinforcement step joint behind the slip surface into the soil body

2) Input reinforcement anchor length l_k

An anchoring length l_k is specified. This parameter is determined by the shear strength developed between the reinforcement and the soil gradually increasing from zero to its limit value (measured from the end of reinforcement fixed in soil).

$$F = \frac{l}{l_k}.R_t$$

where: l - length of reinforcement behind the slip surface into the soil body

l_k - anchoring length of reinforcement

R_t - tensile strength

3) Input reinforcement pull-out resistance T_p

The pull-out force F is given by:

$$F = T_p . l$$

where: l - length of reinforcement behind the slip surface into the soil body

T_p - pull-out resistance of reinforcement

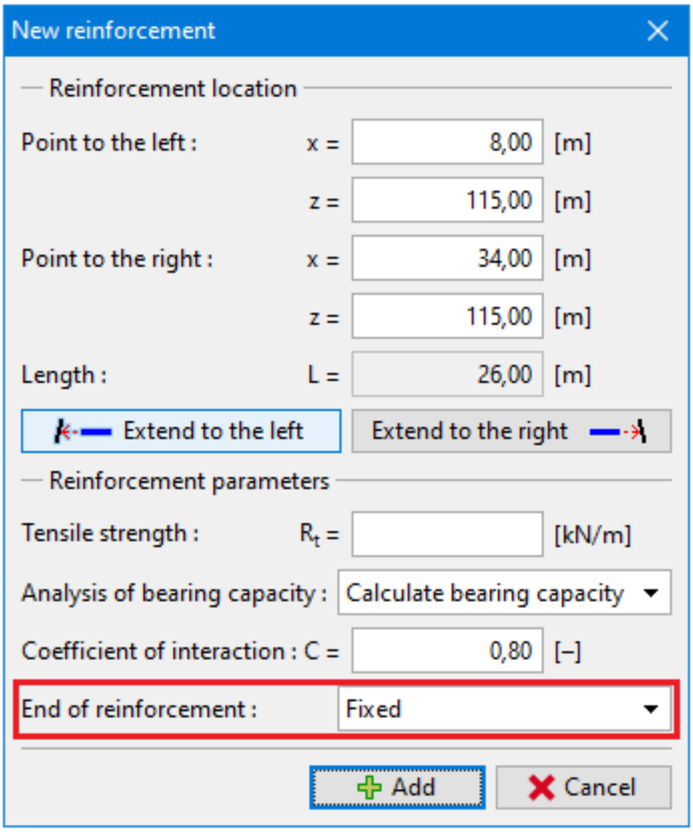
Forces in reinforcements determined on the basis of reinforcement strength may attain relatively large values. Introducing these forces in the analysis yields a higher factor of safety of a given slip surface. In the case of rigorous methods (Spencer, Janbu, Morgenstern-Price) the introduction of such forces in the reinforcements may cause the loss of convergence. This appears mainly in cases when these forces are so high that it is not possible to achieve equilibrium of forces acting on blocks while maintaining the principal assumptions of individual methods, e.g. the assumption of zero moment at the end of the slip surface. In such a case the forces in reinforcements are reduced as least as possible (to the highest acceptable values) so the method converges and attains acceptable results. The reduced values of forces are then written out as part of the stability analysis results. However, in case of no reduction, these forces are not included in the final set of results.

Reinforcement End

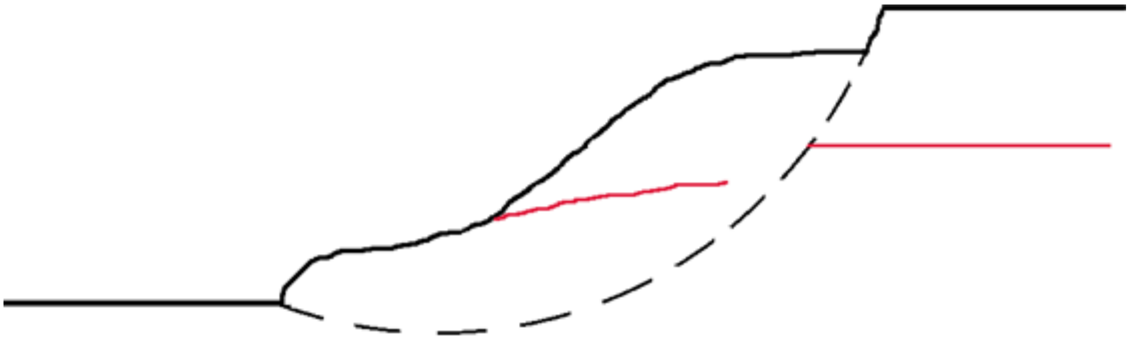
The reinforcement mounting is assumed in the program either as **fixed** or **free**.

Should the slope with reinforcement fail the one of the following reinforcement failure shown in the following figures may appear.

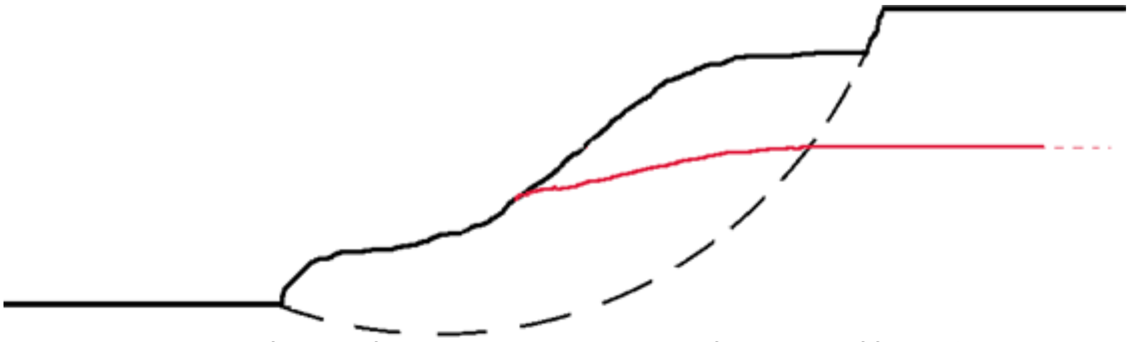
If the reinforcement at its starting point in front of the slip surface is fixed (for example fixed into the structure cladding) the 3rd type of failure is prevented - pullout of the reinforcement in front of the slip surface. The failure type 1 and 2 is always checked in the analysis, type of failure 3 is checked only for reinforcements having free endpoints that allow for such a type of failure.



"New reinforcement" dialog window - input of end of the reinforcement



Breaking of reinforcement - type of failure 1



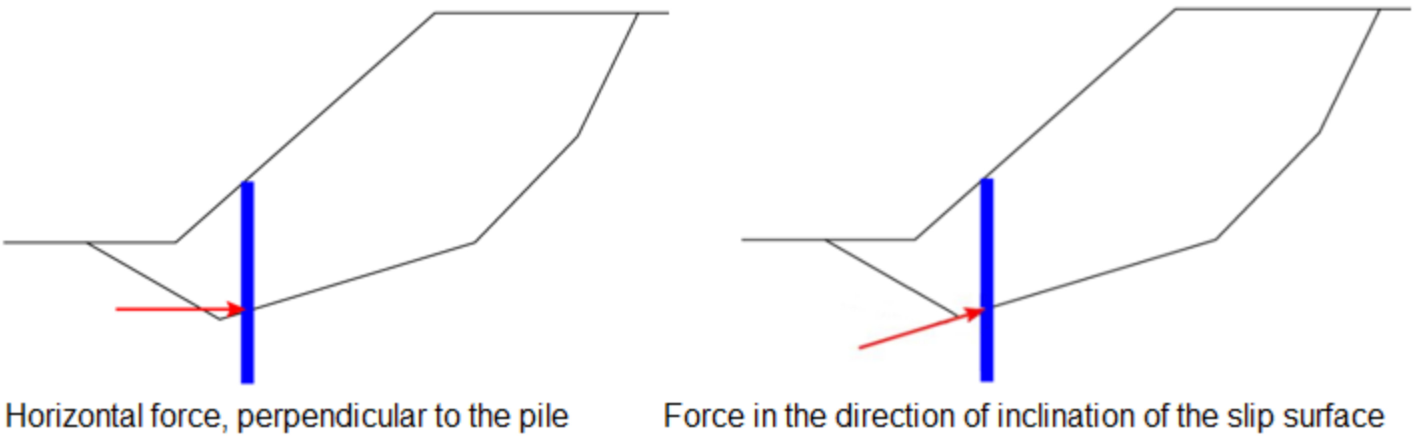
Pullout of the reinforcement behind the slip surface - type of failure 2

Pullout of the reinforcement in front of the slip surface - type of failure 3

Anti-Slide Piles

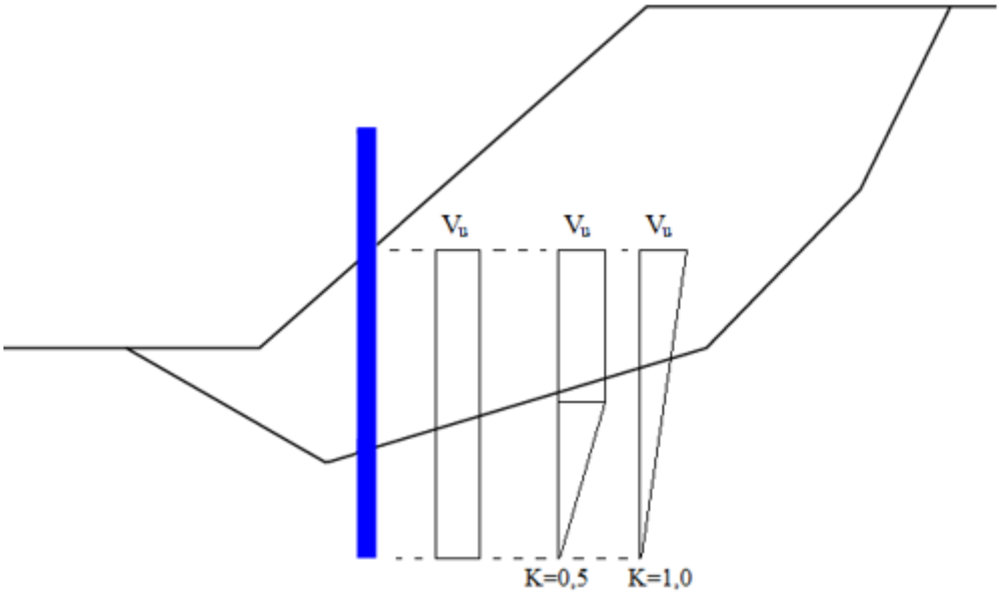
Anti-slide piles are vertical structural elements, which increase the slope stability. If the anti-slide pile intersects into the assessed slip surface, then for the calculation of the factor of safety is introducing passive (resisting) force P which corresponds to bearing capacity of pile V_u. This step is accomplished by the higher value of safety factor SF.

It is assumed that the **pile is always vertical**. Passive (resisting) force P at the intersection point with the slip surface is considered either in the horizontal direction or in a direction which corresponds to the inclination of the slip surface at that location.



Presentation of the direction of the passive (resisting) force

The value of passive (resisting) force P is always determined at 1 rm width of the slope with respect to spacing between piles. Bearing capacity of pile V_u can be specified as either with a **constant value** along the length of pile, or **increasing linearly** away from the pile base upwards.



Constant and linear distribution of bearing capacity V_u along the pile length

Linear increase of bearing capacity of the pile is described by **gradient** K, which is the ratio of the pile length, on which the ultimate bearing capacity V_u is achieved due to the length of the pile below the ground surface. If the value of gradient K approaches zero, the linear distribution of bearing capacity V_u is close to constant distribution.

The úrogram also determines active and passive forces acting on anti-slide piles above the slip surface and allows us to send data to the program Anti-slide pile, where other analyses can be performed.

Influence of an Earthquake

The program allows us to calculate the influence of earthquake according to the following standards:

- Standard analysis
- Earthquake analysis according to the Chinese standard GB 50111-2006
- Earthquake analysis according to the Chinese standard NB 35047-2015
- Earthquake analysis according to the Chinese standard GB 50330-2013
- Earthquake analysis according to the Chinese standard JTG B02-2013

The advantage of Chinese standards is to establish the intensity of the earthquake, according to which the program automatically assigned values of the coefficient K_h appropriate standards.

Earthquake Effect - Standard Analysis

The program allows us to compute the earthquake effects with the help of two variables - factor of horizontal acceleration K_h or the coefficient of vertical earthquake K_v.

Coefficient of vertical earthquake K_v

The coefficient of vertical earthquake either decreases ($K_v > 0$) or increases ($K_v < 0$) the unit weight of soil, water in soil, and material surcharge by multiplying the respective values by $1 - K_v$. It is worth noting that the coefficient K_v may receive both positive and negative value and in case of a sufficiently large coefficient of horizontal acceleration the slope relieve ($K_v > 0$) is more unfavorable than the surcharge.

Factor of horizontal acceleration K_h

In a general case, the computation is carried out assuming a zero value of the factor K_h. This constant, however, can be exploited to simulate the effect of an earthquake by setting a non-zero value. This value represents a ratio between horizontal and gravity accelerations. Increasing the factor K_h results in a corresponding decrease of the safety factor SF.

The coefficient of horizontal acceleration introduces into the analysis an additional horizontal force acting in the center of gravity of a respective block with the magnitude $K_h * W_i$, where W_i is the block overall weight including the material component of the slope surcharge.

The following table lists the values of the factor K_h that correspond to different degrees of earthquakes based on the

M-C-S scale.

M-C-S degree	**Horizontal acceleration**			**Factor of horizontal acceleration**		
(MSK-64)	[mm/s^2]			K_h		
1	0,0	-	2,5	0,0	-	0.00025
2	2,5	-	5,0	0,00025	-	0.0005
3	5,0	-	10,0	0,0005	-	0.001
4	10,0	-	25,0	0,001	-	0.0025
5	25,0	-	50,0	0,0025	-	0.005
6	50,0	-	100,0	0,005	-	0.01
7	100,0	-	250,0	0,01	-	0.025
8	250,0	-	500,0	0,025	-	0.05
9	500,0	-	1000,0	0,05	-	0.1
10	1000,0	-	2500,0	0,1	-	0.25
11	2500,0	-	5000,0	0,25	-	0.5
12		>	5000,0		>	0.5

Earthquake Analysis According to GB 50111-2006

Earthquake effects are in the stability analysis represented by horizontal and vertical forces acting at the centers of gravity of individual soil blocks. The magnitude of these forces is related to the weight of soil blocks and is calculated using horizontal and vertical earthquake coefficients. Horizontal earthquake force is always oriented out form the slope massif. Vertical force can be directed upwards or downwards, the orientation is defined by the sign of the force.

Horizontal earthquake force E_{hs} is given by the formula:

$$E_{hs} = C_i \, C_z \, K_h \, G_s$$

and vertical earthquake force E_{vs} is determined by:

$$E_{vs} = C_0 \, C_i \, C_z \, K_v \, G_s$$

where: C_i - importance coefficient for seismic design
C_z - comprehensive influence factor
C_0 - meeting coefficient related to the influence of horizontal seismic effect
K_h - coefficient of horizontal seismic acceleration
K_v - coefficient of vertical seismic acceleration
G_s - weight of the soil block

Earthquake Analysis According to NB 35047-2015

Earthquake effects are in the stability analysis represented by horizontal and vertical forces acting at the centers of gravity of individual soil blocks. The magnitude of these forces is related to the weight of soil blocks and is calculated using horizontal and vertical earthquake coefficients. Earthquake coefficients are dependant on the position of the gravity center of each block. Therefore the coefficients have individual and different values for each one of soil blocks. Horizontal earthquake force is always oriented out form the slope massif. Vertical force can be directed upwards or downwards, the orientation is defined by the sign of the force.

Horizontal earthquake force E_{hs} is given by the formula:

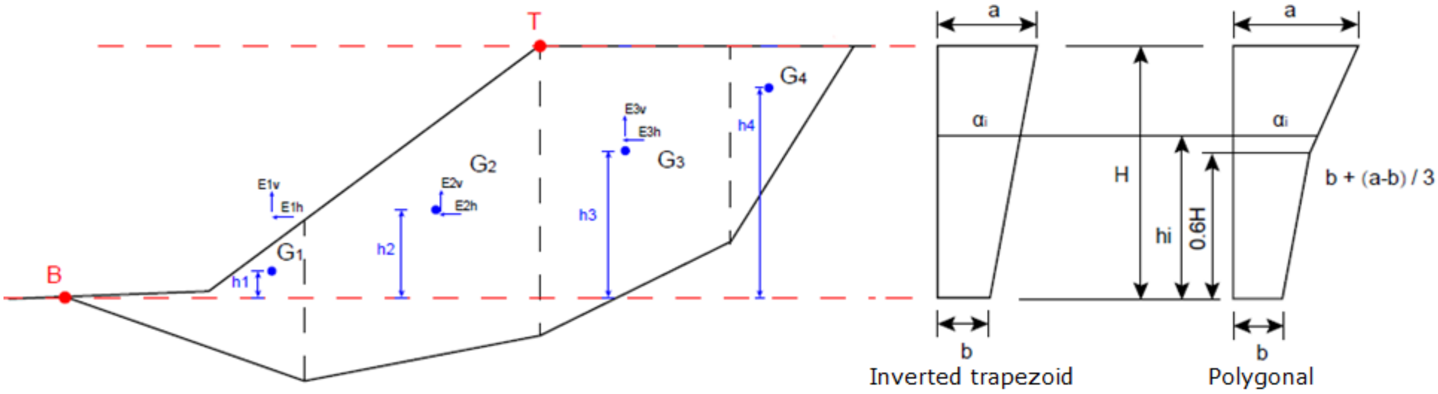
$$E_{hs} = C_i \, C_z \, \alpha_i \, K_h \, G_s$$

and vertical earthquake force E_{vs} is determined by:

$$E_{vs} = C_0 \, C_i \, C_z \, \alpha_i \, K_v \, G_s$$

where: C_i - importance coefficient for seismic design
C_z - comprehensive influence factor
C_0 - meeting coefficient related to the influence of horizontal seismic effect
K_h - coefficient of horizontal seismic acceleration
K_v - coefficient of vertical seismic acceleration
G_s - weight of the soil block
α_i - dynamic distribution coefficient of block i

There are two types of dynamic distribution used for determining of α_i value: inverted trapezoid and polygonal. The method for setting α_i value could be seen in the Figure.



Determination of dynamic distribution coefficient αi

Height H of the range of α_i is given by points **B** and **T**. The bottom point **B** is the lowest point of terrain above the slip surface and the top point **T** is the highest point of terrain above the slip surface. G_i denotes gravity center points of individual blocks and E_{ih} , E_{iv} are horizontal and vertical earthquake forces.

Earthquake Analysis According to GB 50330-2013

Earthquake effects are in the stability analysis represented by horizontal and vertical forces acting at the centers of gravity of individual soil blocks. The magnitude of these forces is related to the weight of soil blocks and is calculated using horizontal and vertical earthquake coefficients. Horizontal earthquake force is always oriented out form the slope massif. Vertical force can be directed upwards or downwards, the orientation is defined by the sign of the force.

Horizontal earthquake force E_{hs} is given by the formula:

$$E_{hs} = C_i a_w G_s$$

and vertical earthquake force E_{vs} is determined by:

$$E_{vs} = C_i a_g G_s$$

where: C_i - importance coefficient for seismic design
a_w - complex factor of horizontal acceleration
a_g - complex factor of vertical acceleration
G_s - weight of the soil block

Earthquake Analysis According to JTG B02-2013

Earthquake effects are in the stability analysis represented by horizontal and vertical forces acting at the centers of gravity of individual soil blocks. The magnitude of these forces is related to the weight of soil blocks and is calculated using horizontal and vertical earthquake coefficients. Earthquake coefficients are dependant on the position gravity of the center of each block. Therefore the coefficients have individual and different values for each one of soil blocks. Horizontal earthquake force is always oriented out form the slope massif. Vertical force can be directed upwards or downwards, the orientation is defined by the sign of the force.

Horizontal earthquake force E_{hs} is given by the formula:

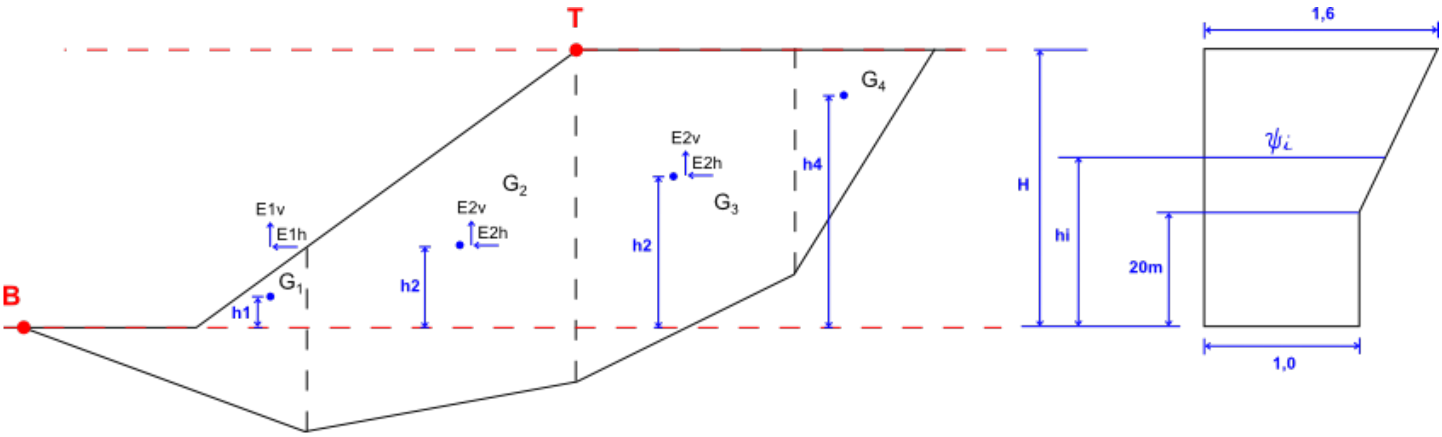
$$E_{hs} = C_i C_z \psi_i K_h G_s$$

and vertical earthquake force E_{vs} is determined by:

$$E_{vs} = C_i C_z K_v G_s$$

where: C_i - importance coefficient for seismic design
C_z - comprehensive influence factor
K_h - coefficient of horizontal seismic acceleration
K_v - coefficient of vertical seismic acceleration
G_s - weight of the soil block
ψ_i - dynamic distribution coefficient of block i

Method for setting ψ_i value could be seen in the Figure.



Determination of dynamic distribution coefficient ψ_i

Height H of the range of ψ_i is given by points **B** and **T**. The bottom point **B** is the lowest point of terrain above the slip surface and the top point **T** is the highest point of terrain above the slip surface. G_i denotes gravity center points of individual blocks and E_{ih} , E_{iv} are horizontal and vertical earthquake forces.

Verification According to EN 1997

When running the verification analysis according to EN 1997 the choice of a given "Design approach" and "Partial factors" is important. Forces and loads are reduced in all design approaches.

The value of capacity utilization V_u is calculated and then it is compared to 100% . The value of capacity utilization is given by:

$$V_u = \frac{M_a}{M_p} 100 < 100\%$$

where: M_a - sliding moment
M_p - resisting moment

In the case of design approach 2 the resisting moment M_p is determined from non-reduced soil parameters but considering the reduction of resistance on the slip surface using the coefficient γ_{Rs} .

In the case of design approach 1 and design approach 3 the program reduces for the determination of the overall resisting moment M_p the **strength parameters of soil** (angle of internal friction and cohesion).

Analysis According to the Theory of Limit States / Safety Factor

The verification parameters are input in the "Stability analysis" tab. The structure can be verified according to the factor of safety or the theory of limit states.

Verification according to the **theory of limit states**:

Soil parameters (angle of internal friction, cohesion) are in this case **reduced using the design coefficients** introduced in the "Stability analysis" tab.

The value of utilization V_u is calculated and then compared with the value of 100% . The value of utilization is given by:

$$V_u = \frac{M_a}{M_p} 100 < 100\%$$

where: M_a - sliding moment
M_p - resisting moment

The resisting moment M_p is determined considering the reduction with the help of the overall stability of construction γ_s .

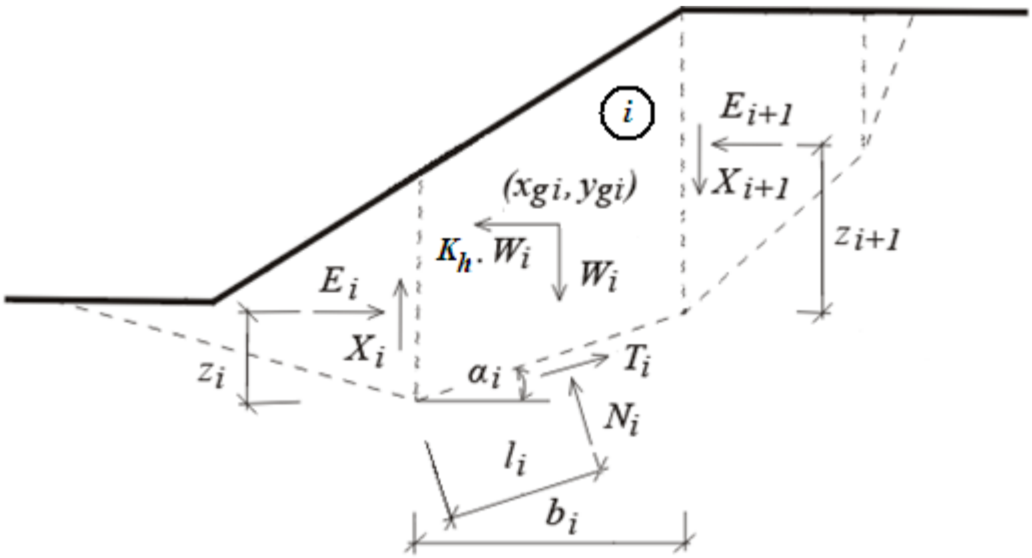
Verification according to the **factor of safety**:

$$\frac{M_p}{M_a} > SF_s$$

where: M_a - sliding moment
M_p - resisting moment
SF_s - factor of safety

Polygonal Slip Surface

The solution of the slope stability problem adopting the polygonal slip surface is based on the determination of the limit state of forces acting on the soil body above the slip surface. To introduce these forces the slip surface above is subdivided into blocks by dividing planes. Typically, these planes are assumed vertical, but this is not a required condition, e.g. the Sarma method considers generally inclined planes.



Static scheme of block

The figure shows forces acting on individual blocks of soil. If the region above the slip surface is divided into blocks, then for the evaluation of unknowns we have: n normal forces N_i acting on individual segments and corresponding n shear forces T_i; $n-1$ normal forces between blocks E_i and corresponding $n-1$ shear forces X_i ; $n-1$ values of z_i representing the points of application of forces E_i, n values of l_i representing the points of application of forces N_i and one value of the factor of safety SF . Forces X_i can be in some methods replaced by the values of inclination of forces E_i .

To following set of equations is available to solve the problem of equilibrium: n horizontal, and n vertical equations of equilibrium written for individual blocks, n moment equations of equilibrium for individual blocks and n relations between N_i and T_i forces developed on blocks according to the Mohr-Coulomb theory. In total there are $4n$ equations for $6n-2$ unknowns. This suggests that $2n-2$ unknowns must be chosen a prior. Individual methods differ from each other in the way these values are selected.

Most often points of application of individual forces acting between blocks or their inclinations are selected. Solving the problem of equilibrium it proceeds in an iterative manner, where the selected values must allow for satisfying both the equilibrium and kinematical admissibility of the obtained solution.

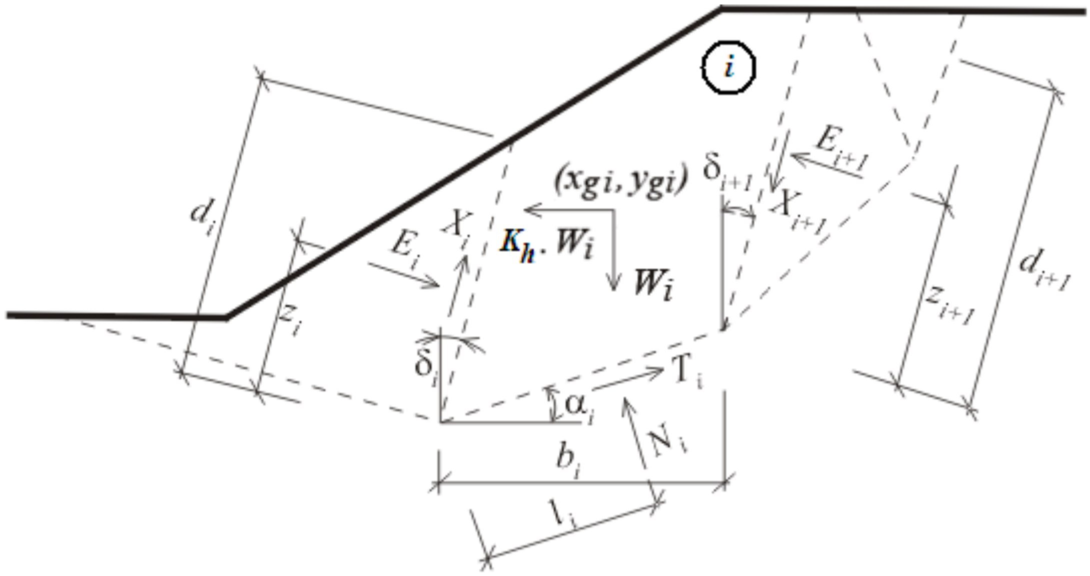
The program allows for adopting one of the following methods:

- Sarma
- Spencer
- Janbu
- Morgenstern-Price
- Shahunyants
- ITF Method

Optimization of polygonal slip surface searches the most critical surface (the lowest safety factor SF).

Sarma

The Sarma method falls within a category of general sliced methods of limit states. It is based on fulfilling the force and moment equilibrium conditions on individual blocks. The blocks are created by dividing the soil region above the potential slip surface by planes, which may have in general experience a different inclination. Forces acting on individual blocks are displayed in the following figure.



Static scheme - Sarma method

Here, E_i ,X_i represent the normal and shear forces between the blocks. N_i ,T_i are normal and shear forces on segments of a slip surface. W_i is the block weight and $K_h * W_i$ is the horizontal force that is used to achieve in the Sarma method the limit state. Generally, inclined surcharge can be introduced in each block. This surcharge is included in the analysis together with the surcharge due to water having the free water table above the terrain, and with forces in anchors. All these forces are projected along the horizontal and vertical directions, which are then summed up into components Fx_i and Fy_i .

K_h is a constant named the factor of horizontal acceleration and it is introduced into the analysis in order to satisfy the equilibrium on individual blocks. There is a relationship between K_h and the factor of slope stability SF allowing for the safety factor computation. In ordinary cases, the analysis proceeds with the value of K_h equal to zero. A non-zero value of K_h is used to simulate the horizontal surcharge, e.g. due to earthquake (see below).

Analysis process

Computation of limit equilibrium

The computation of limit equilibrium requires the solution of $6n -$ 1 unknowns, where n stands for the number of blocks dividing the soil region above the potential slip surface. These are:

- E_i - forces developed between blocks
- N_i - normal forces acting on the slip surface
- T_i - shear forces acting on the slip surface
- X_i - shear forces developed between blocks
- z_i - locations of points of applications of forces
- l_i - locations of points of applications of forces
- K_h - factor of horizontal acceleration

$5n -$ 1 equations are available for the required unknowns. In particular, we have:

a)horizontal force equations of equilibrium on blocks:

$$T_i.\cos\alpha_i - N_i.\sin\alpha_i = K_h.W_i - Fx_i + X_{i+1}.\sin\delta_i - X_i.\sin\delta_i + E_{i+1}.\cos\delta_i - E_i.\cos\delta_i$$

b)vertical force equations of equilibrium on blocks:

$$N_i.\cos\alpha_i - T_i.\sin\alpha_i = W_i - Fy_i + X_{i+1}.\cos\delta_{i+1} - X_i.\cos\delta_i - E_{i+1}.\sin\delta_{i+1} + E_i.\cos\delta_i$$

c) moment equations of equilibrium on blocks:

$$N_i.l_i - X_{i+1}.b_i\,\sec\alpha_i.\cos(\alpha_i + \delta_{i+1}) + E_{i+1}[z_{i+1} + b_1\,\sec\alpha_i.\sin(\alpha_i + \delta_{i+1})]$$
$$- E_i.z_i - W_i.(x_{gi} - x_i) + K_h.W_i.(y_{gi} - y_i) - Fx_i.rx_i + Fy_i.ry_i = 0$$

where rx_i and ry_i are arms of forces Fx_i and Fy_i

d) relationship between the normal and shear forces according to the Mohr-Coulomb theory:

$$T_i = (N_i - U_i).\tan\varphi_i + c_1.b_i.\sec\alpha_i$$

$$X_i = (E_i - PW_i).\tan\overline{\varphi}_i + \overline{c}_i.d_i$$

where:

	$P{*}W_i$	-	resultant force of pore pressure on dividing planes
	$\overline{\varphi}_i$	-	average value of internal friction angle on a dividing plane
	$\overline{c}_i$	-	average value of cohesion on a dividing plane

It is evident that n - 1 must be selected (estimated) a priori. A relatively small error is received when estimating the points of application of forces E_i. The problem then becomes statically determined. Solving the resulting system of equations finally provides the values of all remaining unknowns. The principal result of this analysis is the determination of the factor of horizontal acceleration K_h.

Computation of factor of slope stability SF

The factor of slope stability SF is introduced in the analysis such as to reduce the soil strength parameters c and $tg\varphi$. Equilibrium analysis is then performed for the reduced parameters to arrive at the factor of horizontal acceleration K_h pertinent to a given factor of slope stability SF. This iteration is repeated until the factor K_h reaches either zero or a specified value.

Influence of external load

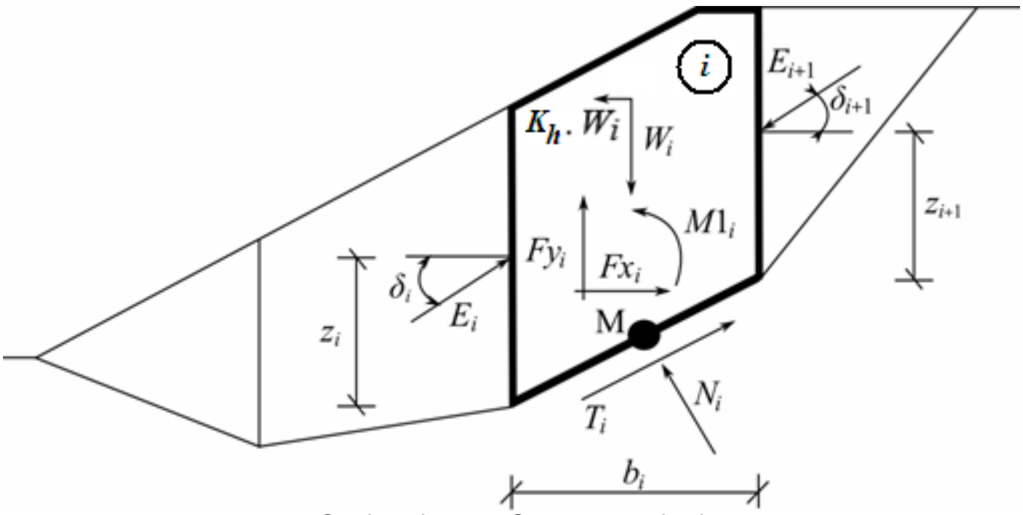
The analyzed slope can be loaded on its ground by inclined load having a generally trapezoidal shape. This load enters the analysis such that its vertical material component (if having the direction of weight) is added to the weight of a corresponding block. This results in change of both the slice weight and its center of gravity. Providing the vertical component acts against the direction of gravity it is added to force Fy_i. The horizontal component is added to force Fx_i.

Literature:

Sarma, S. K.: Stability analysis of embankments and slopes,Géotechnique 23, 423-433, 1973.

Spencer

The Spencer method is a general method of slices developed on the basis of limit equilibrium. It requires a satisfying equilibrium of forces and moments acting on individual blocks. The blocks are created by dividing the soil above the slip surface by dividing planes. Forces acting on individual blocks are displayed in the following figure.



Static scheme - Spencer method

Each block is assumed to contribute due to the following forces:

W_i	-	block weight, including material surcharge having the character of weight including the influence of the coefficient of vertical earthquake K_v
$K_h{*}W_i$	-	horizontal inertia force representing the effect of an earthquake, K_h is the factor of horizontal acceleration during the earthquake
N_i	-	normal force on the slip surface
T_i	-	shear force on the slip surface
E_i ,E_{i+1}	-	forces exerted by neighboring blocks, they are inclined from the horizontal plane by angle δ
Fx_i ,Fy_i	-	other horizontal and vertical forces acting on the block
$M1_i$	-	moment of forces Fx_i , Fy_i rotating about point M , which is the center of the i^{th} segment of the slip surface
U_i	-	pore pressure resultant on the i^{th} segment of the slip surface

The following assumptions are introduced in the Spencer method to calculate the limit equilibrium of forces and moment on individual blocks:

- dividing planes between blocks are always vertical
- the line of action of the weight of block W_i passes through the center of the i^{th} segment of slip surface represented by point **M**
- the normal force N_i is acting in the center of the i^{th} segment of slip surface, at point **M**
- inclination of forces E_i acting between blocks are constant for all blocks and equals to δ, only at slip surface endpoint is $\delta = 0$

The solution adopts the following expressions:

$$N_i = N_i' + U_i \qquad (1)$$

$$T_i = (N_i - U_i)\tan\varphi_i + c_i\frac{b_i}{\cos\alpha_i} = N_i'\tan\varphi_i + c_i\frac{b_i}{\cos\alpha_i} \qquad (2)$$

$$N_i' + U_i - W_i\cos\alpha_i + k_h W_i\sin\alpha_i + Fy_i\cos\alpha_i - Fx_i\sin\alpha_i + E_{i+1}\sin(\alpha_i - \delta_{i+1}) - E_i\sin(\alpha_i - \delta_i) = 0 \qquad (3)$$

$$N_i'\frac{\tan\varphi_i}{SF} + \frac{c_i}{SF}\frac{b_i}{\cos\alpha_i} - W_i\sin\alpha_i - k_h W_i\cos\alpha_i + Fy_i\sin\alpha_i + Fx_i\cos\alpha_i - E_{i+1}\cos(\alpha_i - \delta_{i+1}) + E_i\cos(\alpha_i - \delta_i \qquad (4)$$

$$E_{i+1}\cos\delta_{i+1}\left(z_{i+1} - \frac{b_i}{2}\tan\alpha_i\right) - E_{i+1}\sin\delta_{i+1}\frac{b_i}{2} - E_i\cos\delta_i\left(z_i - \frac{b_i}{2}\tan\alpha_i\right) - E_i\sin\delta_i\frac{b_i}{2} + M1_i - k_h W_i(y_M - y_{gi} \qquad (5)$$

where: φ_i - angle of internal friction of soil on the slip surface segment

c_i - soil cohesion on the slip surface segment

α_i - inclination of the slip surface segment

Equation (1) represents the relationship between the effective and total value of the normal force acting on the slip surface. Equation (2) corresponds to the Mohr-Coulomb condition representing the relation between the normal and shear forces on a given segment of the slip surface. Equation (3) represents the force equation of equilibrium in the direction normal to the i^{th} segment of the slip surface, whereas Equation (4) represents equilibrium along the i^{th} segment of the slip surface. SF is the factor of safety, which is used to reduce the soil parameters. Equation (5) corresponds to the moment equation of equilibrium about point **M**, where y_{gi} is the vertical coordinate of the point of application of the weight of block and y_M is the vertical coordinate of point **M**. Modifying equations (3) and (4) provides the following recursive formula:

$$E_{i+1} = \frac{\left[(W_i - Fy_i)\cos\alpha_i - (K_h W_i - Fx_i)\sin\alpha_i - U_i + E_i\sin(\alpha_i - \delta_i)\right]\frac{\tan\varphi_i}{SF} + \frac{c_i}{SF}\frac{b_i}{\cos\alpha_i} - (W_i - Fy_i)\sin\alpha_i - (K_h W_i - Fx_i)\cos\alpha_i + E_i\cos(\alpha_i - \delta_i)}{\sin(\alpha_i - \delta_{i+1})\frac{\tan\varphi_i}{SF} + \cos(\alpha_i - \delta_{i+1})}$$

This formula allows us to calculate all forces E_i acting between blocks for given values of δ_i and SF. This solution assumes that at the slip surface origin the value of E is known and equal to $E_1 = 0$.

The additional recursive formula follows from the moment equation of equilibrium (5) as:

$$z_{i+1} = \frac{\frac{b_i}{2}\left[E_{i+1}(\sin\delta_{i+1} - \cos\delta_{i+1}\tan\alpha_i) + E_i(\sin\delta_i - \cos\delta_i\tan\alpha_i)\right] + E_i z_i\cos\delta_i - M1_i + K_h W_i(y_M - y_{gi})}{E_{i+1}\cos\delta_{i+1}}$$

This formula allows us calculating for a given value of δ all arms z of forces acting between blocks, knowing the value on the left at the slip surface origin, where $z_1 = 0$.

The factor of safety SF is determined by employing the following iteration process:

1. The initial value of δ is set to zero $\delta = 0$.
2. The factor of safety SF for a given value of δ follows from equation (6), while assuming the value of $E_{n+1} = 0$ at the end of the slip surface.
3. The value of δ is provided by equation (7) using the values of E determined in the previous step with the requirement of having the moment on the last block equal to zero. Equation (7) does not provide the value of z_{n+1} as it is equal to zero. For this value, the moment equation of equilibrium (5) must be satisfied.
4. Steps 2 and 3 are then repeated until the value of δ does not change.

For the process of iteration to be stable it is necessary to avoid unstable solutions. Such instabilities occur at points where division by zero in expressions (6) and (7) takes place. In equation (7), division by zero is encountered for $\delta = \pi/2$ or $\delta = -\pi/2$. Therefore, the value of angle δ must be found in the interval $(-\pi/2\ ;\ \pi/2)$.

Division by zero in expression (6) appears when:

$$SF = \tan\varphi_i \tan(\delta_{i+1} - \alpha_i)$$

Another check preventing numerical instability is verification of the parameter m_α - following condition must be satisfied:

$$m_\alpha = \cos\alpha_i + \frac{\sin\alpha_i \tan\varphi_i}{SF} > 0{,}2$$

Therefore before iteration run it is required to find the highest of critical values SF_{min} satisfying above mentioned conditions. Values below this critical value SF_{min} are within the area of unstable solution, therefore iteration begins by setting SF to a value "just" above SF_{min} and all result values of SF from iteration runs are higher than SF_{min}.

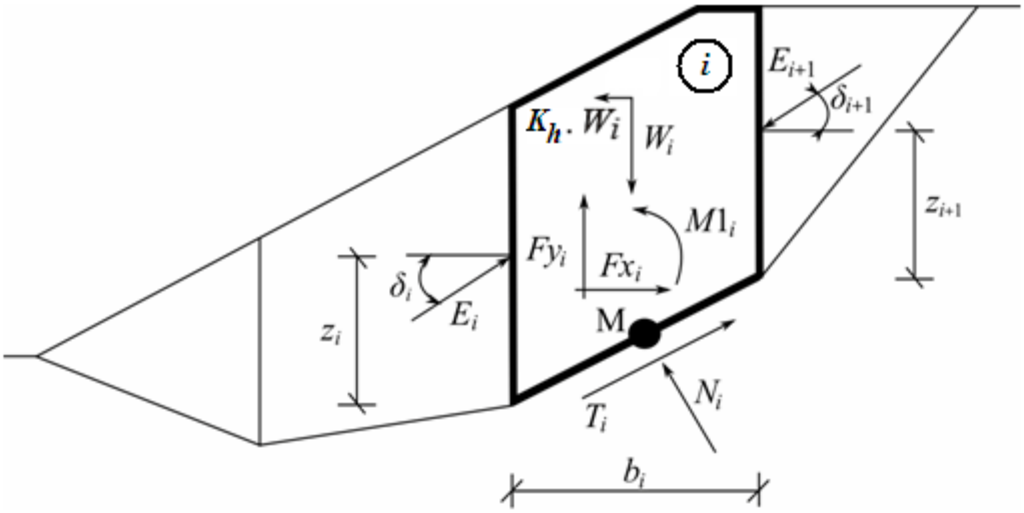
Generally rigorous methods converge worse than the simpler methods (Bishop, Fellenius). Examples with convergence problems include too steep sections of slip surface, complex geometry, a significant jump in surcharge etc. If no result is obtained, we recommend a slight change of input data, e.g. less steep slip surface, input more points into the slip surface etc. or using some of the simpler methods.

Literature:

Spencer, E. 1967. A method of analysis of the stability of embankments assuming parallel interslice forces. Géotechnique, 17(1): 11-26.

Janbu

Janbu is a general method of slices developed on the basis of limit equilibrium. It requires satisfying equilibrium of forces and moments acting on individual blocks (only moment equilibrium at last uppermost block is not satisfied). The blocks are created by dividing the soil above the slip surface by dividing planes. Forces acting on individual blocks are displayed in the following figure:



Static scheme - Janbu method

Each block is assumed to contribute due to the following forces:

W_i	-	block weight, including material surcharge having the character of weight including the influence of the coefficient of vertical earthquake K_v
$K_h{*}W_i$	-	horizontal inertia force representing the effect of the earthquake, K_h is the factor of horizontal acceleration during earthquake
N_i	-	normal force on the slip surface
T_i	-	shear force on the slip surface
E_i ,E_{i+1}	-	forces exerted by neighboring blocks, they are inclined from the horizontal plane by an angle δ_i resp. δ_{i+1} and lie at the height z_i resp. z_{i+1} above slip surface
Fx_i ,Fy_i	-	other horizontal and vertical forces acting on the block
$M1_i$	-	moment from forces Fx_i , Fy_i rotating about point **M**, which is the center of the i^{th} segment of the slip surface
U_i	-	pore pressure resultant on the i^{th} segment of the slip surface

The following assumptions are introduced in the Janbu method to calculate the limit equilibrium of forces and moment on individual blocks:

- dividing planes between blocks are always vertical

- the line of action of the weight of block W_i passes through the center of the i^{th} segment of slip surface represented by point **M**
- the normal force N_i is acting in the center of the i^{th} segment of slip surface, at point **M**
- position z_i of forces E_i acting between blocks is assumed, at slip surface endpoints is $z = 0$

The choice of position z_i can have significant influence on the convergency of the method. If we make a bad assumption of position z_i for a given slope, it can become impossible to satisfy the equilibrium conditions (algorithm does not converge). Heights z_i above the slip surface are set approximately to one third of the height of interface between the blocks. In case of unsatisfying equilibrium conditions algorithm changes the height to a different position, e.g. slightly higher within the passive zone, near the toe, and lower within the active zone, near the crest of the slope.

The solution adopts the following expressions:

$$N_i = N_i' + U_i \tag{1}$$

$$T_i = (N_i - U_i)\tan\varphi_i + c_i\frac{b_i}{\cos\alpha_i} = N_i'\tan\varphi_i + c_i\frac{b_i}{\cos\alpha_i} \tag{2}$$

$$N_i' + U_i - W_i.\cos\alpha_i + K_h.W_i.\sin\alpha_i + Fy_i.\cos\alpha_i - Fx_i.\sin\alpha_i + E_{i+1}.\sin(\alpha_i - \delta_{i+1}) - E_i.\sin(\alpha_i - \delta_i) = 0 \tag{3}$$

$$N_i'.\frac{\tan\varphi_i}{FS} + \frac{c_i}{FS}.\frac{b_i}{\cos\alpha_i} - W_i.\sin\alpha_i - K_h.W_i.\cos\alpha_i + Fy_i.\sin\alpha_i + Fx_i\cos\alpha_i - E_{i+1}.\cos(\alpha_i - \delta_{i+1}) + E_i.\cos(\alpha_i - \delta_i) = 0 \tag{4}$$

$$E_{i+1}.\cos\delta_{i+1}\left(z_{i+1} - \frac{b_i}{2}\tan\alpha_i\right) - E_{i+1}.\sin\delta_{i+1}.\frac{b_i}{2} - E_i.\cos\delta_i\left(z_i - \frac{b_i}{2}\tan\alpha_i\right) - E_i.\sin\delta_i.\frac{b_i}{2} + M1_i - K_h.W_i\left(y_M - y_{gi}\right) = 0 \tag{5}$$

where:
- φ_i - angle of internal friction of soil on the slip surface segment
- c_i - soil cohesion on the slip surface segment
- α_i - inclination of the slip surface segment

Equation (1) represents the relationship between the effective and total value of the normal force acting on the slip surface. Equation (2) corresponds to the Mohr-Coulomb condition representing the relation between the normal and shear forces on a given segment of the slip surface. Equation (3) represents the force equation of equilibrium in the direction normal to the i^{th} segment of the slip surface, whereas Equation (4) represents equilibrium along the i^{th} segment of the slip surface. SF is the factor of safety, which is used to reduce the soil parameters. Equation (5) corresponds to the moment equation of equilibrium about point **M**, where y_{gi} is the vertical coordinate of the point of application of the weight of block, and y_M is the vertical coordinate of point **M**.

Modifying equations (3) and (4) provides the following recursive formula (6):

$$E_{i+1} = \frac{\left[(W_i - Fy_i).\cos\alpha_i - (K_h.W_i - Fx_i).\sin\alpha_i - U_i + E_i.\sin(\alpha_i - \delta_i)\right].\frac{\tan\varphi_i}{FS} + \frac{c_i}{FS}.\frac{b_i}{\cos\alpha_i} - (W_i - Fy_i).\sin\alpha_i - (K_h.W_i - Fx_i).\cos\alpha_i + E_i.\cos(\alpha_i - \delta_i)}{\sin(\alpha_i - \delta_{i+1}).\frac{\tan\varphi_i}{FS} + \cos(\alpha_i - \delta_{i+1})} \tag{6}$$

This formula allows calculating all forces E_i acting between blocks for given values of δ_i and SF. This solution assumes that at the slip surface origin the value of E is known and equal to $E_1=0$.

Formula for calculating angles δ_i (7) follows from the moment equation of equilibrium (5) as:

$$\delta_{i+1} = \arctan\left(\frac{2.z_{i+1}}{b_i} + \tan\alpha_i\right) - \arcsin\frac{E_i\left(\cos\delta_i\left(z_i - \frac{b_i.\tan\alpha_i}{2}\right) + \sin\delta_i.\frac{b_i}{2}\right) - M1_i}{E_{i+1}\sqrt{\left(z_{i+1} + \frac{b_i.\tan\alpha_i}{2}\right)^2 + \left(\frac{b_i}{2}\right)^2}} \qquad (7)$$

This formula allows us calculating for a given value of δ all arms z_i of forces acting between blocks, knowing the value on the left at the slip surface origin, where $z_1 = 0$.

The factor of safety FS is determined by employing the following iteration process:

1. The initial value of angles is set to zero $\delta_i = 0$ and positions z_i to approximately one-third of interface height.
2. The factor of safety FS for a given value of δ_i follows from equation (6) while assuming the value of $E_{n+1} = 0$ at the end of the slip surface.
3. The value of δ_i is provided by equation (7) using the values of E_i determined in the previous step.
4. Steps 2 and 3 are then repeated until the value of FS does not change.

It is necessary to avoid unstable solutions for a successful iteration process. Such instabilities occur at points where division by zero in expression (6) takes place, i.e.:

$$FS = \tan\varphi_i.\tan\left(\delta_{i+1} - \alpha_i\right)$$

Another check preventing numerical instability is verification of the parameter m_α - following condition must be satisfied:

$$m_\alpha = \cos\alpha_i + \frac{\sin\alpha_i.\tan\varphi_i}{FS} > 0{,}2$$

Therefore before the iteration runs, it is required to find the highest of critical values SF_{min} satisfying above mentioned conditions. Values below this critical value SF_{min} are in the area of unstable solution, therefore iteration begins by setting SF to a value "just" above SF_{min} and all result values of SF from iteration runs are higher than SF_{min}.

Generally, rigorous methods converge worse than the simpler methods (Bishop, Fellenius). Examples with convergence problems include too steep sections of slip surface, complex geometry, a significant jump in surcharge etc. If no result is obtained, we recommend a slight change of input data, e.g. less steep slip surface, input more points into the slip surface etc. or using some of the simpler methods.

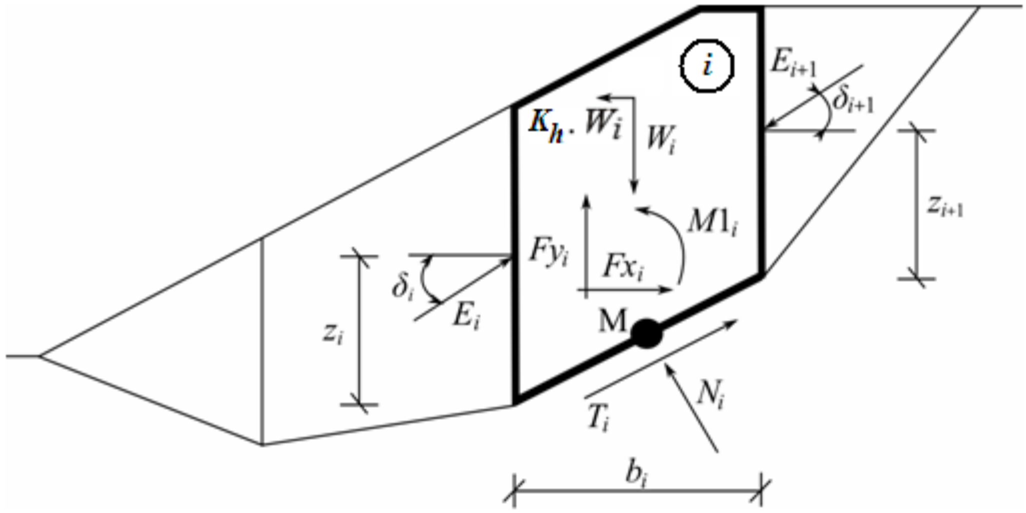
Literature:

Janbu, N. 1954. Application of Composite Slip Surface for Stability Analysis. European Conference on Stability Analysis, Stockholm, Sweden.

Janbu, N. 1973. Slope Stability Computations. Embankment Dam Engineering - Casagrande Volume, R.C. Hirschfeld and S.J. Poulos, eds., John Wiley and Sons, New York, pp 47-86.

Morgenstern-Price

Morgenstern-Price is a general method of slices developed on the basis of limit equilibrium. It requires a satisfying equilibrium of forces and moments acting on individual blocks. The blocks are created by dividing the soil above the slip surface by dividing planes. Forces acting on individual blocks are displayed in the following figure:

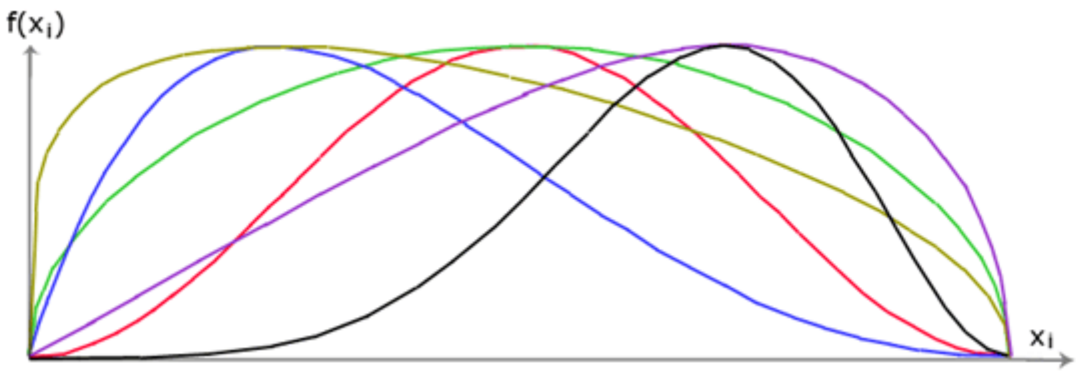


Static scheme - Morgenstern-Price method

Each block is assumed to contribute due to the same forces as in Spencer method. The following assumptions are introduced in the Morgenstern-Price method to calculate the limit equilibrium of forces and moment on individual blocks:

- dividing planes between blocks are always vertical
- the line of action of the weight of block W_i passes through the center of the i^{th} segment of slip surface represented by point **M**
- the normal force N_i is acting in the center of the i^{th} segment of slip surface, at point **M**
- inclination of forces E_i acting between blocks is different on each block (δ_i) at slip surface endpoints is $\delta = 0$

The only difference between Spencer and Morgenstern-Price method is shown in the above list of assumptions. Choice of inclination angles δ_i of forces E_i acting between the blocks is realized with the help of Half-sine function - one of the functions in the following figure is automatically chosen. This choice of the shape of function has a minor influence on final results, but a suitable choice can improve the convergency of the method. The functional value of Half-sine function $f(x_i)$ at boundary point x_i multiplied by parameter λ results in the value of inclination angle δ_i.



Half-sine function

The solution adopts the expressions (1) - (5), shown in Spencer method, i.e.:

$$N_i = N_i' + U_i \tag{1}$$

$$T_i = (N_i - U_i)\tan\varphi_i + c_i \frac{b_i}{\cos\alpha_i} = N_i' \tan\varphi_i + c_i \frac{b_i}{\cos\alpha_i} \tag{2}$$

$$N_i' + U_i - W_i.\cos\alpha_i + K_h.W_i.\sin\alpha_i + Fy_i.\cos\alpha_i - Fx_i.\sin\alpha_i + E_{i+1}.\sin(\alpha_i - \delta_{i+1}) - E_i.\sin(\alpha_i - \delta_i) = 0 \tag{3}$$

$$N_i'.\frac{\tan\varphi_i}{FS} + \frac{c_i}{FS}.\frac{b_i}{\cos\alpha_i} - W_i.\sin\alpha_i - K_h.W_i.\cos\alpha_i + Fy_i.\sin\alpha_i + Fx_i\cos\alpha_i - E_{i+1}.\cos(\alpha_i - \delta_{i+1}) + E_i.\cos(\alpha_i - \delta_i) = 0 \tag{4}$$

$$E_{i+1}.\cos\delta_{i+1}\left(z_{i+1}-\frac{b_i}{2}\tan\alpha_i\right)-E_{i+1}.\sin\delta_{i+1}.\frac{b_i}{2}-$$
$$E_i.\cos\delta_i\left(z_i-\frac{b_i}{2}\tan\alpha_i\right)-E_i.\sin\delta_i.\frac{b_i}{2}+$$
$$M1_i-K_h.W_i\left(y_M-y_{gi}\right)=0 \tag{5}$$

where: φ_i - angle of internal friction of soil on the slip surface segment

c_i - soil cohesion on the slip surface segment

α_i - inclination of the slip surface segment

- (1) relationship between the effective and total value of the normal force acting on the slip surface
- (2) Mohr-Coulomb condition representing the relation between the normal and shear forces on a given segment of the slip surface (N_i a T_i)
- (3) force equation of equilibrium in the direction normal to the i^{th} segment of the slip surface
- (4) force equation of equilibrium along the i^{th} segment of the slip surface
- (5) moment equation of equilibrium about point **M**

Modifying force equations (3) and (4) provides the following recursive formula (6):

$$E_{i+1}=\frac{\left[\left(W_i-Fy_i\right).\cos\alpha_i-\left(K_h.W_i-Fx_i\right).\sin\alpha_i-U_i+E_i.\sin\left(\alpha_i-\delta_i\right)\right].\frac{\tan\varphi_i}{FS}+}{\sin\left(\alpha_i-\delta_{i+1}\right).\frac{\tan\varphi_i}{FS}+\cos\left(\alpha_i-\delta_{i+1}\right)} \tag{6}$$
$$\frac{+\frac{c_i}{FS}.\frac{b_i}{\cos\alpha_i}-\left(W_i-Fy_i\right).\sin\alpha_i-\left(K_h.W_i-Fx_i\right).\cos\alpha_i+E_i.\cos\left(\alpha_i-\delta_i\right)}{}$$

This formula allows calculating all forces E_i acting between blocks for given values of δ_i and SF. This solution assumes that at the slip surface origin the value of E is known and equal to $E_1 = 0$.

Additional recursive formula (7) follows from the moment equation of equilibrium (5) as:

$$z_{i+1}=\frac{\frac{b_i}{2}.\left[E_{i+1}\left(\sin\delta_{i+1}-\cos\delta_{i+1}.\tan\alpha_i\right)+E_i.\left(\sin\delta_i-\cos\delta_i.\tan\alpha_i\right)\right]+E_i.z_i.\cos\delta_i-M1_i+K_h.W_i.\left(y_M-\right.}{E_{i+1}.\cos\delta_{i+1}} \tag{7}$$

This formula allows us to calculate all arms z_i of forces acting between blocks for a given value of δ_i, knowing the value on the left at the slip surface origin, where $z_1 = 0$.

The factor of safety SF is determined by employing the following iteration process:

1. The initial value of angles δ_i is set according to the half-sine function ($\delta_i = \lambda * f(x_i)$).
2. The factor of safety SF for a given value of δ_i follows from equation (6) while assuming the value of $E_{n+1} = 0$ at the end of the slip surface.
3. The value of δ_i is provided by equation (7) using the values of E_i determined in the previous step with the requirement of having the moment on the last block equal to zero. Functional values $f(x_i)$ are the same all the time during the iteration, only parameter λ is iterated. Equation (7) does not provide the value of z_{n+1} as it is equal to zero. For this value, the moment equation of equilibrium (5) must be satisfied.
4. Steps 2 and 3 are then repeated until the value of δ_i (resp. parameter λ) does not change.

It is necessary to avoid unstable solutions for successful iteration process. Such instabilities occur at points where division by zero in expressions (6) and (7) takes place. In equation (7), division by zero is encountered for $\delta_i = \pi/2$ or $\delta_i = -\pi/2$. Therefore, the value of angle δ_i must be found in the interval ($-\pi/2$; $\pi/2$).

Division by zero in expression (6) appears when:

$$FS = \tan\varphi_i . \tan\left(\delta_{i+1} - \alpha_i\right)$$

Another check preventing numerical instability is verification of the parameter m_α - following condition must be satisfied:

$$m_\alpha = \cos\alpha_i + \frac{\sin\alpha_i . \tan\varphi_i}{FS} > 0,2$$

Therefore before theiteration runs, it is required to find the highest of critical values SF_{min} satisfying above-mentioned conditions. Values below this critical value SF_{min} are in an area of unstable solution, therefore iteration begins by setting SF to a value "just" above SF_{min} and all result values of SF from iteration runs are higher than SF_{min}.

Generally, rigorous methods converge worse than the simpler methods (Bishop, Fellenius). Examples with convergence problems include too steep sections of slip surface, complex geometry, a significant jump in surcharge etc. If no result is obtained, we recommend a slight change of input data, e.g. less steep slip surface, input more points into the slip surface etc. or using some of the simpler methods.

Literature:

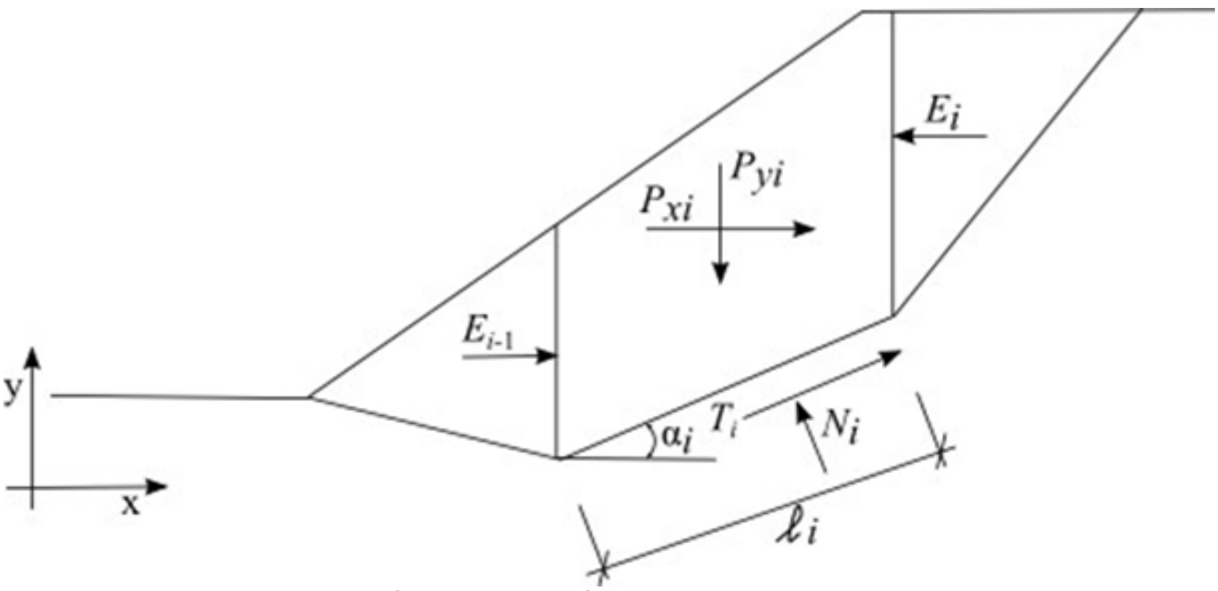
Morgenstern, N.R., and Price, V.E. 1965. The analysis of the stability of general slip surfaces. Géotechnique, 15(1): 79-93.

Morgenstern, N.R., and Price, V.E. 1967. A numerical method for solving the equations of stability of general slip surfaces. Computer Journal, 9: 388-393.

Zhu, D.Y., Lee, C.F., Qian, Q.H., and Chen, G.R. 2005. A concise algorithm for computing the factor of safety using the Morgenstern-Price method. Canadian Geotechnical Journal, 42(1): 272-278.

Shahunyants

The Shahunyants method is a general method of slices developed on the basis of limit equilibrium. It requires a satisfying equilibrium of forces and moments acting on individual blocks. The blocks are created by dividing the soil above the slip surface by dividing planes. Forces acting on individual blocks are displayed in the following figure:



Static scheme - Shahunyants method

Each block assumes action of the following forces:

where:	P_{yi}	-	resultant of vertical forces acting on a given block (block weight, block surcharge, earthquake, anchor forces, …)
	P_{xi}	-	resultant of horizontal forces on a given block (block surcharge, earthquake, anchor forces, geo-reinforcements, …)
	E_{i+1}, E_i	-	forces developed between blocks
	N_i	-	reaction below the block normal to the slip surface segment
	T_i	-	friction force on the slip surface segment
	α_i	-	inclination of the slip surface segment
	l_i	-	length of the slip surface segment
	φ_i	-	angle of internal friction of soil on the slip surface segment
	c_i	-	soil cohesion on the slip surface segment

The following assumptions are adopted in the Shahunyants method to calculate the limit state on a given block:

- dividing planes between blocks are always vertical
- slope of the forces E_i acting between blocks is zero, forces act horizontally

Solution procedure:

Forces P_{yi} and P_{xi} are first transformed with the help of expressions (1) and (2) into directions of forces T_i and N_i. For a positive angle α_i (the same way as in the schema) the force P_{Ni} acts in the direction opposite to N_i, the force P_{Qi} acts in the direction opposite to T_i.

$$P_{Ni} = P_{xi}\sin\alpha_i + P_{yi}\cos\alpha_i$$

(1)

$$P_{Qi} = P_{yi}\sin\alpha_i - P_{xi}\cos\alpha_i$$

(2)

The forces acting along the slip surface segment are related by:

$$T_i = (N_i - U_i)\tan\varphi_i + c_i l_i$$

(3)

where: U_i - pore pressure on the slip surface segment

The force equations of equilibrium are fulfilled on the block:

The equilibrium condition in the direction normal to the slip surface segment:

$$N_i = P_{Ni} + E_{i-1}\sin\alpha_i - E_i\sin\alpha_i$$

(4)

The equilibrium condition in the direction parallel to the slip surface segment:

$$T_i = P_{Qi} + E_i\cos\alpha_i - E_{i-1}\cos\alpha_i$$

(5)

Introducing Eq. (3) into Eq. (5) gives:

$$(N_i - U_i)\tan\varphi_i + c_i l_i = P_{Qi} + E_i\cos\alpha_i - E_{i-1}\cos\alpha_i$$

(6)

Next, substituting Eq. (4) into Eq. (6) gives:

$$(P_{Ni} + E_{i-1}\sin\alpha_i - E_i\sin\alpha_i - U_i)\tan\varphi_i + c_i l_i = P_{Qi} + E_i\cos\alpha_i - E_{i-1}\cos\alpha_i$$

(7)

After some algebra:

$$(P_{Ni} - U_i)\tan\varphi_i + (E_{i-1} - E_i)\sin\alpha_i\tan\varphi_i + c_i l_i = P_{Qi} + (E_i - E_{i-1})\cos\alpha_i$$

$$(P_{Ni} - U_i)\tan\varphi_i + c_i l_i - P_{Qi} = (E_i - E_{i-1})(\cos\alpha_i + \sin\alpha_i\tan\varphi_i)$$

(8)

Exploiting the following mathematical expression:

$$\cos\alpha + \sin\alpha\tan\beta = \frac{\cos\alpha\cos\beta + \sin\alpha\sin\beta}{\cos\beta} = \frac{\cos(\alpha-\beta)}{\cos\beta}$$

(9)

yields Eq. (8) in the form:

$$(P_{Ni} - U_i)\tan\varphi_i + c_i l_i - P_{Qi} = (E_i - E_{i-1})\frac{\cos(\alpha_i - \varphi_i)}{\cos\varphi_i}$$

(10)

This can be modified as:

$$(P_{Ni} - U_i)\tan\varphi_i + c_i l_i - P_{Qi} + E_{i-1}\frac{\cos(\alpha_i - \varphi_i)}{\cos\varphi_i} = E_i\frac{\cos(\alpha_i - \varphi_i)}{\cos\varphi_i}$$

(11)

to provide the recurrent expression for E_i forces acting between blocks as:

$$E_i = \frac{\left[(P_{Ni} - U_i)\tan\varphi_i + c_i l_i - P_{Qi}\right]\cos\varphi_i}{\cos(\alpha_i - \varphi_i)} + E_{i-1}$$

(12)

At this stage, the analysis enters the factor of safety K_u. The factor of safety is the value which brings the forces acting on individual blocks of soil into the state of limit states. This is achieved by multiplying active forces, i.e. forces contributing to sliding of the soil mass above the slip surface, by the factor of safety. Active forces are in Eq. (12) contained within the term P_{Qi}. This term contains, on the one hand, active forces that contribute to sliding and on the other hand forces that resist sliding. The contributing forces will be denoted as $P_{Qi,sd}$ whereas the resisting forces as $P_{Qi,ud}$. Eq. (12) then becomes:

$$E_i = \frac{\left[(P_{Ni} - U_i)\tan\varphi_i + c_i l_i - K_\mathrm{u} P_{Qi,\mathrm{sd}} - P_{Qi,\mathrm{ud}}\right]\cos\varphi_i}{\cos(\alpha_i - \varphi_i)} + E_{i-1}$$

(13)

Providing the value of P_{Qi} is positive then it contributes to sliding and will be assumed as active force $P_{Qi,sd}$. Providing the value of P_{Qi} is negative then it resists to sliding and will be assumed as force $P_{Qi,ud}$. Therefore subtracting the value $P_{Qi,ud}$, which is negative, in Eq. (13) means essentially adding the positive value, so we can formally write:

$$E_i = \frac{\left[(P_{Ni} - U_i)\tan\varphi_i + c_i l_i - K_\mathrm{u} P_{Qi,\mathrm{sd}} + \left|P_{Qi,\mathrm{ud}}\right|\right]\cos\varphi_i}{\cos(\alpha_i - \varphi_i)} + E_{i-1}$$

(14)

At the slip surface origin the value of $E_0 = 0$. The value of E_1 is then given by:

$$E_1 = \frac{\left[(P_{N1} - U_1)\tan\varphi_1 + c_1 l_1 - K_\mathrm{u} P_{Q1,\mathrm{sd}} + \left|P_{Q1,\mathrm{ud}}\right|\right]\cos\varphi_1}{\cos(\alpha_1 - \varphi_1)}$$

(15)

The value of E_2 is then given by:

$$E_2 = \frac{\left[(P_{N2} - U_2)\tan\varphi_2 + c_2 l_2 - K_\mathrm{u} P_{Q2,\mathrm{sd}} + \left|P_{Q2,\mathrm{ud}}\right|\right]\cos\varphi_2}{\cos(\alpha_2 - \varphi_2)} + \frac{\left[(P_{N1} - U_1)\tan\varphi_1 + c_1 l_1 - K_\mathrm{u} P_{Q1,\mathrm{sd}} + \left|P_{Q1,\mathrm{ud}}\right|\right]\cos\varphi_1}{\cos(\alpha_1 - \varphi_1)}$$

(16)

Similarly, we may determine the values of all forces acting between blocks. It further holds at the endpoint of the slip surface we have $E_n = 0$. Exploiting the previous expressions this can be written as:

$$E_n = \sum_{i=1}^{n}\left[(P_{Ni} - U_i)\tan\varphi_i + c_i l_i + \left|P_{Qi,\mathrm{ud}}\right|\right]\frac{\cos\varphi_i}{\cos(\alpha_i - \varphi_i)} - K_\mathrm{u}\sum_{i=1}^{n} P_{Qi,\mathrm{sd}}\frac{\cos\varphi_i}{\cos(\alpha_i - \varphi_i)} = 0$$

(17)

This equation directly provides the factor of safety K_u in the form:

$$K_\mathrm{u} = \frac{\sum_{i=1}^{n}\left[(P_{Ni} - U_i)\tan\varphi_i + c_i l_i + \left|P_{Qi,\mathrm{ud}}\right|\right]\frac{\cos\varphi_i}{\cos(\alpha_i - \varphi_i)}}{\sum_{i=1}^{n} P_{Qi,\mathrm{sd}}\frac{\cos\varphi_i}{\cos(\alpha_i - \varphi_i)}}$$

(18)

ITF Method (Imbalance Thrust Force Method)

The ITF method is a limit state method. It builds upon the equation of equilibrium of forces acting on individual blocks and does not consider the moment equation of equilibrium. The bases of the method and adopted assumptions are evident from the following figure.

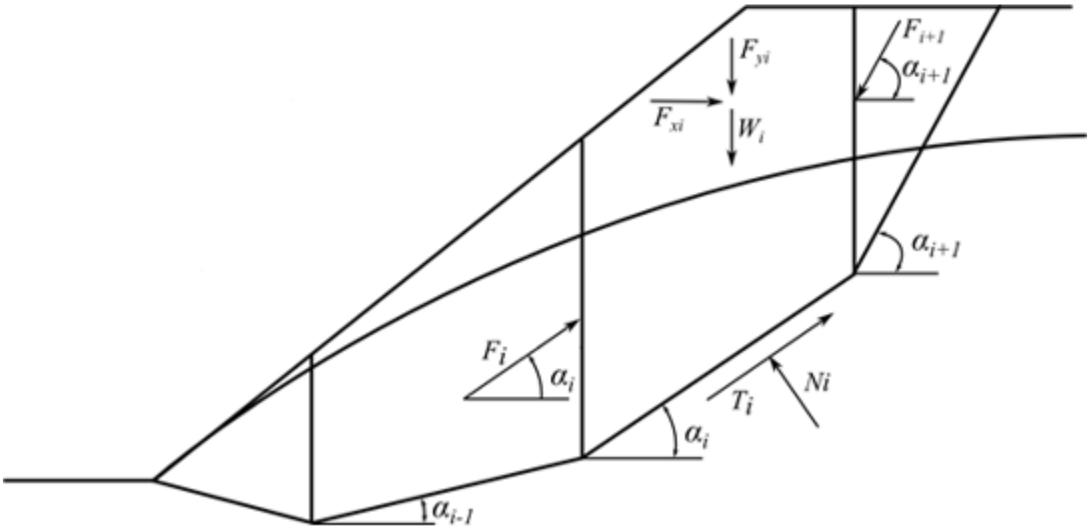


Fig. 1 Forces acting on a block - ITF Method

Consider the following assumptions concerning the forces acting on the block:

where:

W_i - weight of the i^{th} block, the weight of a part of the block below the groundwater is determined from the saturated unit weight of soil γ_{sat}

F_{yi} - represents the remaining vertical load acting on the block

F_{xi} - represents the remaining horizontal load acting on the block

F_i, F_{i+1} - forces acting between blocks along directions given by angles α_i and α_{i+1}

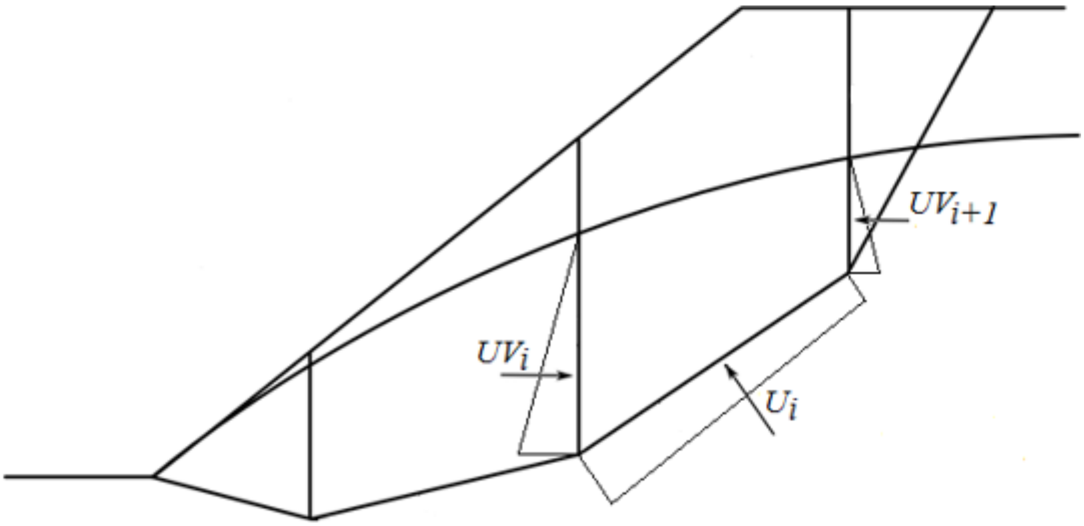


Fig. 2 Scheme of pore pressure action on the block

U_i - pore pressure resultant on slip surface segment

UV_i, UV_{i+1} - pore pressure resultants on dividing planes between blocks

The forces UV_i and UV_{i+1} are included in the horizontal forces Fx_i.

The force equation of equilibrium in the direction normal to the segment of the slip surface provides:

$$N_i = (W_i + Fy_i)\cos\alpha_i + Fx_i \sin\alpha + F_{i+1}\sin(\alpha_{i+1} - \alpha_i) - U_i \qquad (1)$$

The forces on the segment of a slip surface are related by

$$T_i = N_i \tan\varphi_i + c_i l_i \qquad (2)$$

where: φ_i - angle of internal friction of the soil

c_i - soil cohesion

l_i - length of the slip surface segment associated with the i^{th} block

The force equation of equilibrium in the direction of the i^{th} segment of the slip surface (given by angle α_i) yields the force F_i acting between blocks in the form:

$$F_i = (W_i + Fy_i)\sin\alpha_i - Fx_i\cos\alpha_i - T_1 + F_{i+1}\cos(\alpha_{i+1} - \alpha_i) \qquad (3)$$

Introducing Eqs.(1) and (2) into Eq. (3) provides:

$$\begin{aligned} F_i = {} & (W_i + Fy_i)\sin\alpha_i - Fx_i\cos\alpha_i - \\ & - \{[(W_i + Fy_i)\cos\alpha_i + Fx_i\sin\alpha_i + F_{i+1}\sin(\alpha_{i+1} - \alpha_i) - U_i]\tan\varphi_i + c_i l_i\} + \\ & + F_{i+1}\cos(\alpha_{i+1} - \alpha_i) \end{aligned}$$

and after some formal algebra, we arrive at the resulting form of the equation of equilibrium as:

$$\begin{aligned} F_i = {} & (W_i + Fy_i)\sin\alpha_i - Fx_i\cos\alpha_i - \\ & - \{[(W_i + Fy_i)\cos\alpha_i + Fx_i\sin\alpha_i - U_i]\tan\varphi_i + c_i l_i\} + \\ & + F_{i+1}[\cos(\alpha_{i+1} - \alpha_i) - \sin(\alpha_{i+1} - \alpha_i)\tan\varphi_i] \end{aligned} \qquad (4)$$

The equilibrium condition will be fulfilled by introducing the factor of safety SFS into the analysis such that the strength parameters of a given soil c and $tan\varphi$ are divided by this value. Eq. (4) then becomes

$$\begin{aligned} F_i = {} & (W_i + Fy_i)\sin\alpha_i - Fx_i\cos\alpha_i - \\ & - \{[(W_i + Fy_i)\cos\alpha_i + Fx_i\sin\alpha_i - U_i]\tan\varphi_i + c_i l_i\} / SF + \\ & + F_{i+1}[\cos(\alpha_{i+1} - \alpha_i) - \sin(\alpha_{i+1} - \alpha_i)\tan\varphi_i / SF] \end{aligned} \qquad (5)$$

Eq. (5) then gives the searched factor of safety SF through the process of iteration. This process proceeds such that the force F_n equal to $0\ kN$ is applied at the highest (end) point of the slip surface. The forces F_i acting in between blocks are determined for a given value of the factor of safety SF from Eq. (5). This step is repeated for various values of SF until we find such SF for which the force F_0 at the slope base becomes equal to $0\ kN$. No tension is assumed along the slip surface. If the equilibrium condition requires the value of normal force N_i being negative, which means that the soil is loaded in tension, then the value of this force is set equal to zero in the next iteration step and the shear force T_i acting on a given segment is determined based on the soil cohesion only.

The ITF method is quite sensitive with respect to the shape of the slip surface. In case the slip surface contains sharp segment discontinuities the resulting factor of safety is generally larger when compares to reality. It is recommended that the slope difference between adjacent segments of the slip surface be less than 10°. This is checked automatically by the program and if the slope difference is found greater the programs prompt a warning that the results might be overestimated. This is usually not the problem of a circular slip surface but should be kept in mind in case of polygonal slip surfaces.

ITF Method - explicit solution

The explicit solution of the ITF method assumes a different way of introducing the factor of safety into the analysis. The mathematical solution then does not require iterations and the resulting factor of safety is calculated directly in one step. With this approach, the resulting factor of safety is typically higher which may the solution totally devalued, particularly in cases concerning polygonal slip surfaces with large slope differences of adjacent segments.

The solution exploits Eq. (4) to which the factor of safety SF is introduced such that it multiplies the active components of forces, i.e. the components acting in the sliding direction. The equilibrium condition then becomes :

$$\begin{aligned} F_i = {} & [(W_i + Fy_i)\sin\alpha_i - Fx_i\cos\alpha_i]SF - \\ & - \{[(W_i + Fy_i)\cos\alpha_i + Fx_i\sin\alpha_i - U_i]\tan\varphi_i + c_i l_i\} + \\ & + F_{i+1}[\cos(\alpha_{i+1} - \alpha_i) - \sin(\alpha_{i+1} - \alpha_i)\tan\varphi_i] \end{aligned} \qquad (6)$$

For lucidity, we introduce the component of active forces as:

$$A_i = (W_i + Fy_i)\sin\alpha_i - Fx_i\cos\alpha_i$$

and next to the component of passive forces as:

$$P_i = [(W_i + Fy_i)\cos\alpha_i + Fx_i\sin\alpha_i - U_i]\tan\varphi_i + c_i l_i$$

and an auxiliary function:

$$\psi_i = \cos(\alpha_{i+1} - \alpha_i) - \sin(\alpha_{i+1} - \alpha_i)\tan\varphi_i$$

Eq. (6) can then be written in more compact form as:

$$F_i = A_i\,SF - P_i + F_{i+1}\,\psi_{i+1} \qquad (7)$$

The next step assumes the known force $F_n=0$ to provide expressions of forces between blocks F in the form:

$$F_{n-1} = A_{n-1}\,SF - P_{n-1}$$

$$F_{n-2} = A_{n-2}\,SF - P_{n-2} + \left(A_{n-1}\,SF - P_{n-1}\right)\psi_{n-1}$$

$$F_{n-3} = A_{n-3}\,SF - P_{n-3} + \left(A_{n-2}\,SF - P_{n-2}\right)\psi_{n-2} + \left(A_{n-1}\,SF - P_{n-1}\right)\psi_{n-1}\psi_{n-2}$$

Etc.…

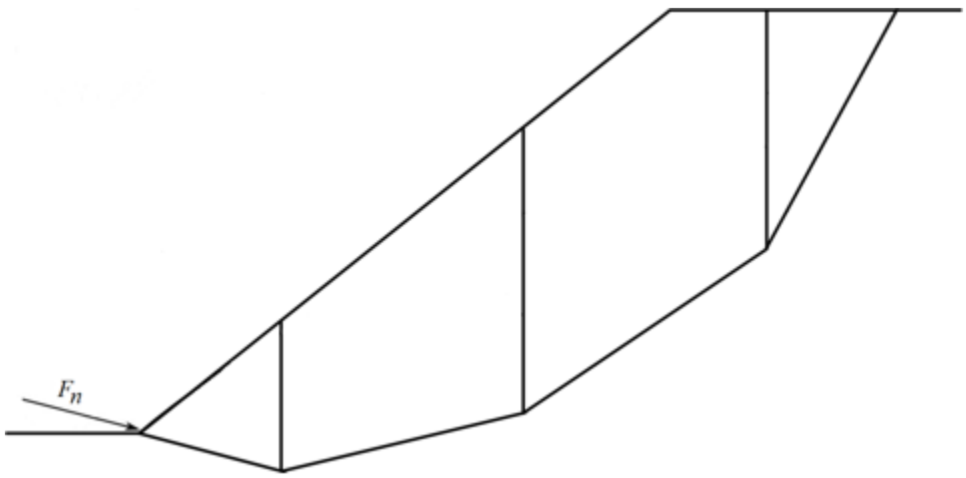
$$F_0 = \left[A_0 + \sum_{i=1}^{n-1}\left(A_i\prod_{j=1}^{i}\psi_j\right)\right]SF - P_0 - \sum_{i=1}^{n-1}\left(P_i\prod_{j=1}^{i}\psi_j\right) \qquad (8)$$

And since the force on the bottom origin of the slip surface should be equal to $0\ kN$, we get the final form of the factor of safety SF as:

$$SF = \frac{P_0 + \sum_{i=1}^{n-1}\left(P_i\prod_{j=1}^{i}\psi_j\right)}{A_0 + \sum_{i=1}^{n-1}\left(A_i\prod_{j=1}^{i}\psi_j\right)} \qquad (9)$$

Surplus sliding force

Surplus sliding force F_n is introduced as a force acting in the bottom point of the slip surface - see picture. With this force, the limit equilibrium can be reached for the desired value of design safety factor F_{sd} and its value can be useful for the design of a retaining wall. The surplus sliding force has a reasonable value if the design safety factor is higher than the actual one. When the actual safety factor is high enough (the slope stability is acceptable), surplus sliding force value is equal to zero.



Surplus sliding force F_n

The surplus sliding force can also be used as a criterion of optimization. In that case, the optimization process searches the slip surface with a maximum value of surplus sliding force. This kind of optimization process can be used, if the slope stability for actual slip surface is not acceptable, because in case of an acceptable slip surface, the surplus sliding force is not calculated (equals zero). For acceptable slip surface stability, the optimization process uses as a criterion the factor of stability FS only.

Optimization of Polygonal Slip Surface

The slip surface optimization proceeds such that the program changes subsequently locations of individual points of this surface and checks, which change of location of given point results in the maximal reduction of the factor of slope stability SF. The endpoints of the optimized slip surface are moved on the ground surface, internal points are moved in the vertical and horizontal directions. The step size is initially selected as one-tenth of the smallest distance of neighboring points along the slip surface. With every new run, the step size is reduced by one half. Location of points of slip surface is optimized subsequently from the left to the right and it is completed when there was no point moved in the last run.

When optimizing the polygonal slip surface the iteration process may suffer from falling into the **local minimum** (with respect to the gradual evolution of locations of nodal points) so not always the process is terminated by locating the critical slip surface. Especially in case of a complex slope profile, it is, therefore, advantageous to introduce several locations of the initial slip surface. A combination with the approach used for circular slip surfaces is also recommended. Therefore, the critical slip surface assuming circular shape is found first and the result is then used to define the initial polygonal slip surface.

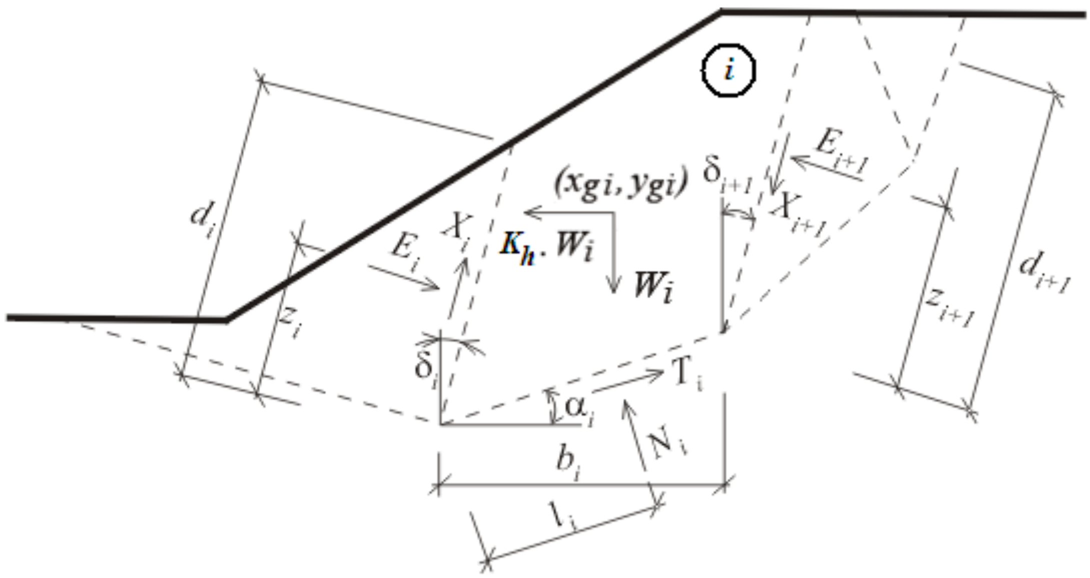
The optimization process can be restricted by various constraints. This becomes advantageous especially if we wish the searched slip to pass through a certain region or to bypass this region. The restriction on the optimization process can be performed in two different ways:

1. Optimization restrictions are specified as a set of segments in a soil body. The optimized slip surface is then forced to bypass these segments during optimization.
2. Another way of restricting the optimization process is to fix the location of selected points along the optimized slip surface or allow for moving these points only in one of two directions, either vertically or horizontally.

For ITF method, the surplus sliding force can be used as the criterion of optimization. In that case, the optimization process searches the slip surface with a maximum value of surplus sliding force F_n instead of the minimum stability factor FS. This optimization criterion can be applied when the actual slip surface is not acceptable. For an acceptable slip surface, the optimization process uses the stability factor FS only.

Changing the Inclination of Dividing Planes

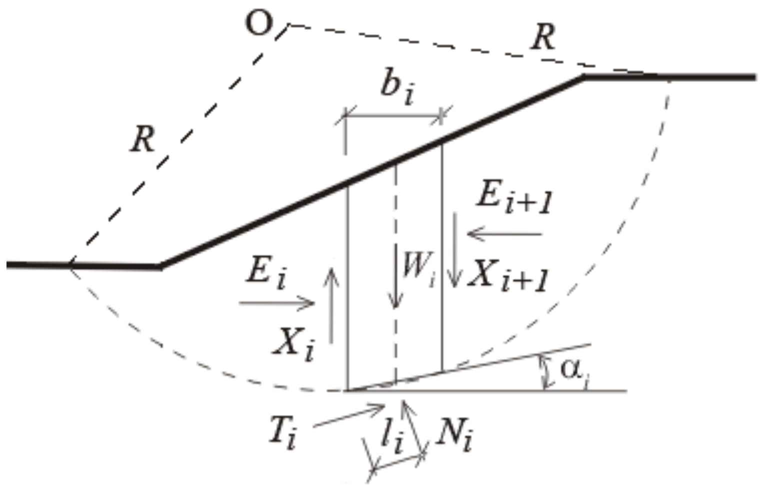
Changing the inclination of dividing planes is applied for Sarma's method only. It is evident from the figure that the planes dividing individual blocks do not have to be vertical and not even mutually parallel. In the first stage of analysis when the optimization procedure moves points along the slip surface it assumes vertical alignment of dividing planes. To arrive at an even smaller value of the slope stability it is possible to change the mutual alignment of dividing planes. This process is again performed in several runs with a limited value of rotation step and this step is again reduced in the course of optimization. This stage of optimization is terminated once the rotation step drops below the value of 1^o and no change of rotation occurred during the last optimization run.



Static scheme - Sarma method

Circular Slip Surface

All methods of limit equilibrium assume that the soil body above the slip surface is subdivided into blocks (dividing planes between blocks are always vertical). Forces acting on individual blocks are displayed in the figure.



Static scheme of slice

Here, X_i and E_i are the shear and normal forces acting between individual blocks, T_i and N_i are the shear and normal forces on individual segments of the slip surface, W_i are the weights of individual blocks.

Individual methods of slices differ in their assumptions of satisfying the force equations of equilibrium and the moment equation of equilibrium with respect to the center **O**.

The program allows for adopting one of the following methods:

- Fellenius / Petterson
- Bishop
- Spencer
- Janbu
- Morgenstern-Price
- Shahunyants
- ITF Method

Groundwater specified within the slope body (using one of the five options) influences the analysis in two different ways. First when computing the weight of a soil block and second when determining the shear forces. Note that the effective soil parameters are used to relate the normal and shear forces.

Introducing anchor forces and water above the ground surface into the analysis

Anchor forces are considered as an external load applied to the slope. They are taken with respect to one running meter [kN/m] and introduced into the moment equation of equilibrium. These forces should contribute to additional stability, if that cannot be achieved in a different way. There is no limitation to the magnitudes of anchor forces and therefore it is necessary to work with realistic values.

Influence of the water above the ground surface is considered as set forces acting perpendicular on the ground surface together with pore pressure along the slip surface, which is derived depending on the depth of the slip surface measured from the groundwater table. The forces acting on the ground surface enter the moment equation of equilibrium as forces acting on respective arms measured towards the center of the slip surface.

Optimization of circular slip surface searches the most critical surface (the lowest SF).

Fellenius / Petterson

The simplest method of slices assumes only the overall moment equation of equilibrium written with respect to the center of the slip surface. The shear and normal forces between blocks X_i and E_i are neglected. The factor of safety SF follows directly from the following expression:

$$FS = \frac{1}{\sum_i W_i . \sin \alpha_i} . \sum_i \left[c_i . l_i + \left(N_i - u_i . l_i \right) . \tan \varphi_i \right]$$

where:

u_i	-	pore pressure within the block
c_i, φ_i	-	effective values of soil parameters
W_i	-	block weight
N_i	-	normal force on the segment of the slip surface
α_i	-	inclination of the segment of the slip surface

l_i - length of the segment of the slip surface

Literature:

Petterson KE (1955) The early history of circular sliding surfaces. Geotechnique 5:275-296.

Bishop

The simplified Bishop method assumes zero X_i forces between blocks. The method is based on satisfying the moment equation of equilibrium and the vertical force equation of equilibrium.

The factor of safety SF is found through a successive iteration of the following expression:

$$FS = \frac{1}{\sum_i W_i . \sin\alpha_i} . \sum_i \frac{c_i . b_i + (W_i - u_i . b_i) . \tan\varphi_i}{\cos\alpha_i + \frac{\tan\varphi_i . \sin\alpha_i}{FS}}$$

where:

u_i - pore pressure within the block

c_i, φ_i - effective values of the soil parameters

W_i - block weight

α_i - inclination of the segment of the slip surface

b_i - horizontal width of the block

Literature:

Bishop, A.W. (1955) "The Use of the Slip Circle in the Stability Analysis of Slopes", Geotechnique, Great Britain, Vol. 5, No. 1, Mar., pp. 7-17.

Spencer

This method assumes non-zero forces between blocks. The resultants of shear and normal forces acting between blocks have constant inclinations. The Spencer method is a rigorous method in the sense that it satisfies all three equations of equilibrium - the force equations of equilibrium in the horizontal and vertical directions and the moment equation of equilibrium. The factor of safety SF is found through the iteration of inclination of forces acting between blocks and the factor of safety SF. Further details can be found in the section describing the analysis of the polygonal slip surface.

Janbu

Janbu method assumes non-zero forces between blocks. Method satisfies the force equations of equilibrium in the horizontal and vertical directions for all blocks and the moment equation of equilibrium for all but the last (uppermost) slice. The assumption of this method is the choice of position of forces acting between the blocks. The factor of safety SF is found through the iteration of forces acting between blocks and then inclinations of these forces are calculated. Further details can be found in the section describing the analysis of a polygonal slip surface.

Morgenstern-Price

This method assumes non-zero forces between blocks. The resultants of shear and normal forces acting between blocks have different inclinations at each block (shape of Half-sine function). Morgenstern-Price is a rigorous method in the sense that it satisfies all three equations of equilibrium - the force equations of equilibrium in the horizontal and vertical directions and the moment equation of equilibrium. The factor of safety SF is found through the iteration of inclination of forces acting between blocks and the factor of safety SF. Further details can be found in the section describing the analysis of a polygonal slip surface.

Shahunyants

Further details can be found in the section describing the analysis of a polygonal slip surface.

ITF Method (Imbalance Thrust Force Method)

Further details can be found in the section describing the analysis of a polygonal slip surface.

Optimization of Circular Slip Surface

The goal of the optimization process is to locate a slip surface with the smallest factor of slope stability SF. The circular slip surface is specified in terms of 3 points: two points on the ground surface and one inside the soil body. Each point on the surface has one degree of freedom while the internal point has two degrees of freedom. The slip surface is defined in terms of four independent parameters. Searching for such a set of parameters that yields the most critical results requires sensitivity analysis resulting in a matrix of changes of parameters that allows fast and reliable optimization procedure. The slip surface that gives the smallest factor of slope stability is taken as the critical one. Parameters of individual slip surfaces and results from optimization runs can be displayed in the output document.

This approach usually succeeds in finding the critical slip surface without encountering the problem of falling into a local minimum during iteration. It, therefore, appears as a suitable starting point when optimizing general slip surfaces such as the polygonal slip surface.

The optimization process can be restricted by various constraints. This becomes advantageous especially if we wish the searched slip surface to pass through a certain region or to bypass this region. The restriction on the optimization process can be performed in two different ways:

1. Optimization restrictions are specified as a set of segments in a soil body. The optimized slip surface is then forced to bypass these segments during optimization.
2. Another way of restricting the optimization process is to fix the location of lthe eft or right endpoint of the optimized slip surface.

For ITF methods, the surplus sliding force can be used as the criterion of optimization. In that case, the optimization process searches the slip surface with a maximum value of surplus sliding force F_n instead of the minimum stability factor FS. This optimization criterion can be applied when the actual slip surface is not acceptable. For an acceptable slip surface, the optimization process uses the stability factor FS only.

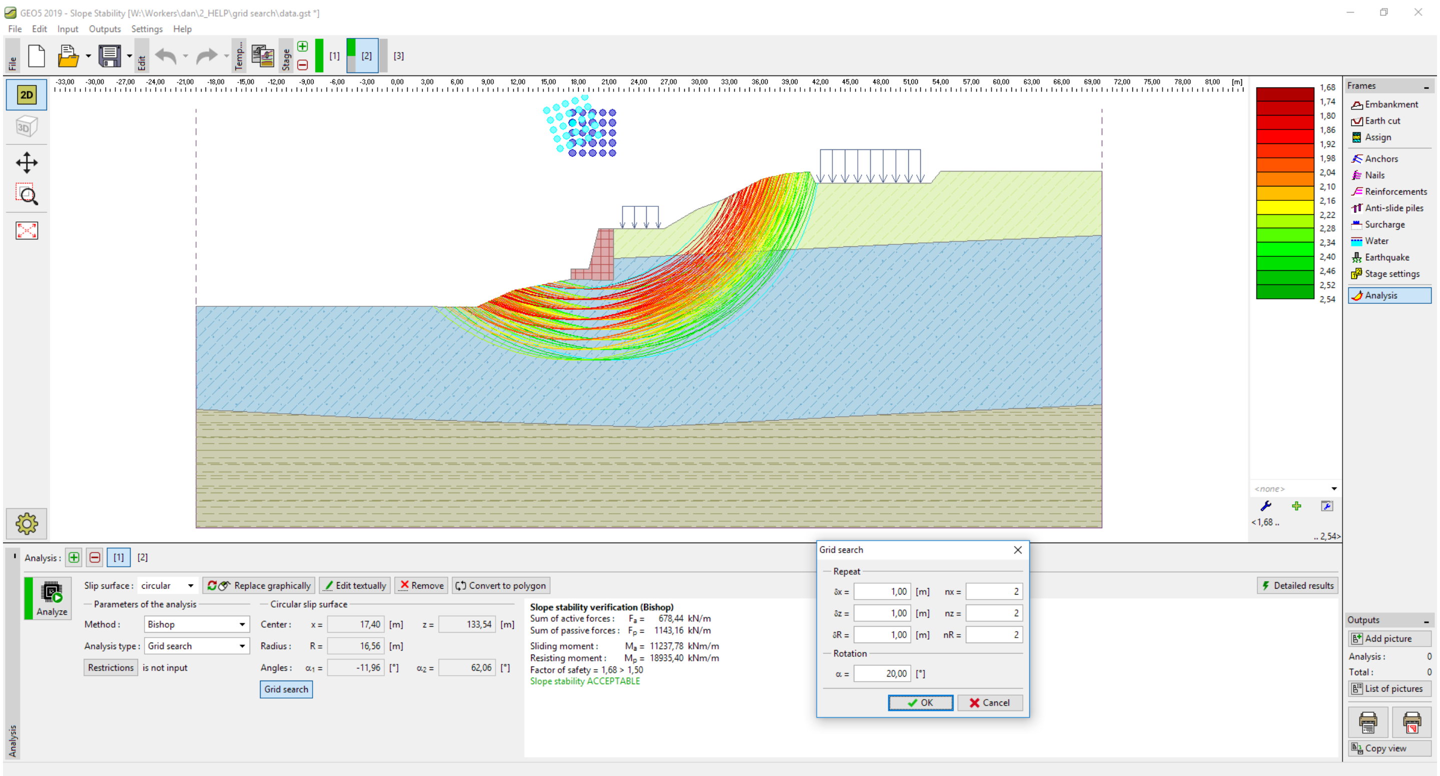
Grid Search

With circular slip surfaces, it is sometimes convenient to **display several slip surfaces at one time**. To do so, select the grid search option.

The calculation of the grid is always done based on the currently inputted slip surface, therefore, we recommend to run the optimization of circular slip surface first.

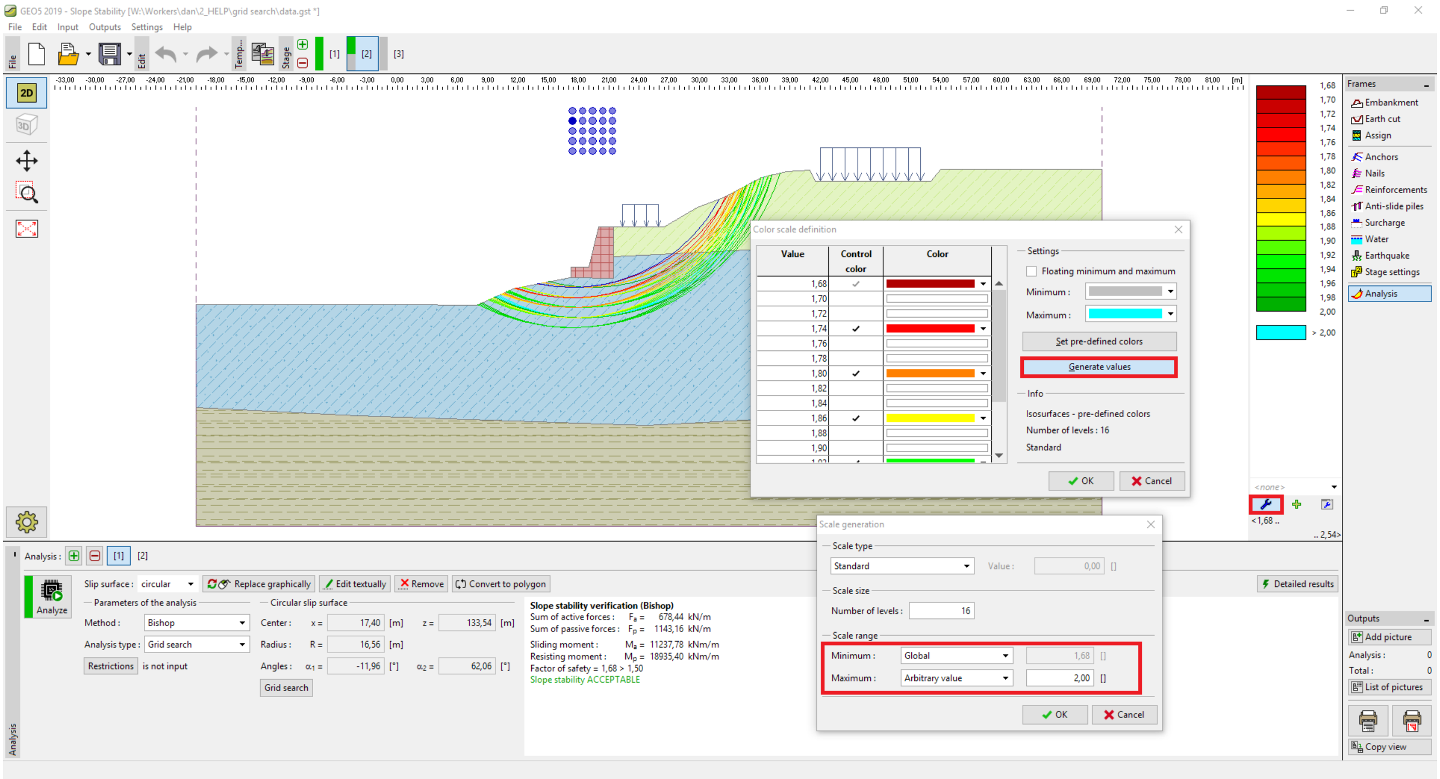
It is also possible to input slip surface restriction.

In the dialog window, input the grid parameters (center, radius, rotation). After the analysis is performed, all slip surfaces are calculated and are displayed in the selected color scale.



Slip Surface Grid Parameter Settings

For better representation, it is sometimes convenient to consider the safety factor only **until a certain level** - this may be set in the dialog window **"Color Scale Definition"** by pressing the button **"Generate Values"**. Slip surfaces outside of the given range are not displayed.



Scale Generation until a Certain Level

This method can also be used as **handmade optimization**:

- Use the input slip surface and run the calculation.
- The slip surface with the highest utilization is displayed in blue and its center is bold.
- After the calculation is performed again, the worst slip surface is considered and the grid is calculated again.
- The process can be repeated until the final center of the slip surface is in the center of the grid.

In any case, this procedure is less trustworthy than the standard slip surface optimization algorithm and we recommend to use it very carefully.

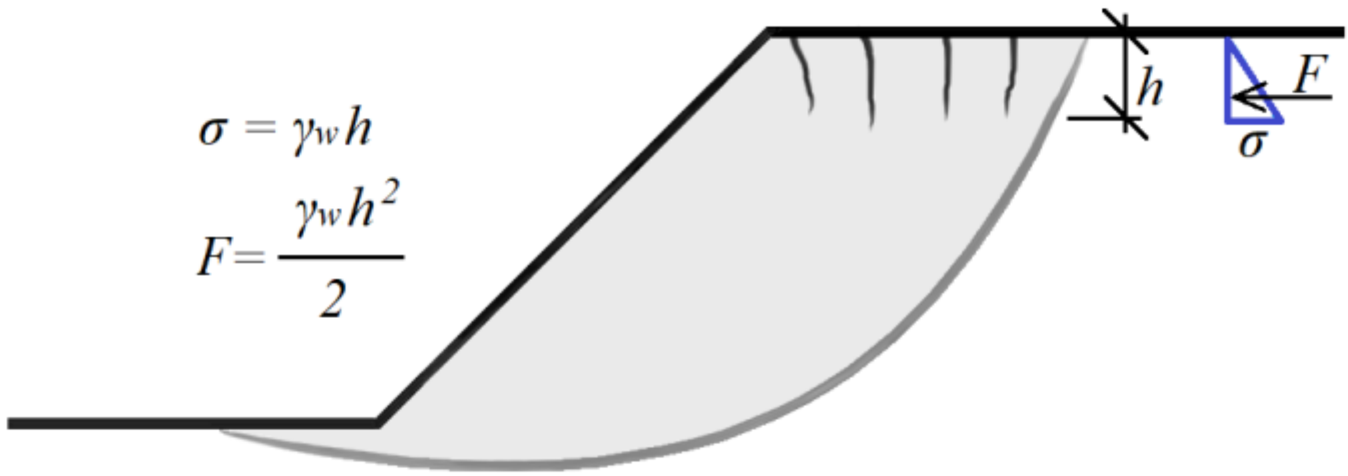
Foliation

Soils can be introduced with foliation. It means that along an angle specified in terms of a certain interval, which in turn is introduced as one of the soil parameters <***Starting Slope***; ***End Slope***> the soil experiences significantly different (usually worse) parameters (c a φ).

If the slope of a slip surface segment or the slope of the interface between blocks is assumed within the interval <***Starting Slope***; ***End Slope***>, the analysis proceeds with the modified parameters of c and φ.

Influence of Tensile Cracks

The program makes it possible to account for the influence of tensile cracks that appear on terrain surface and are filled with water h. The only input parameter is the depth of tensile cracks. The effect of cracks is incorporated when calculating normal and shear forces in sections of a slip surface containing cracks - in a section with tensile cracks the shear strength parameters are set to zero ($c = 0$, $\varphi = 0$). Next, a horizontal force F due to the presence water in a tensile crack is introduced in the analysis (see figure):

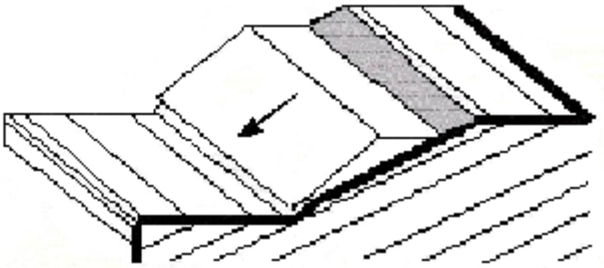


Influence of tensile cracks

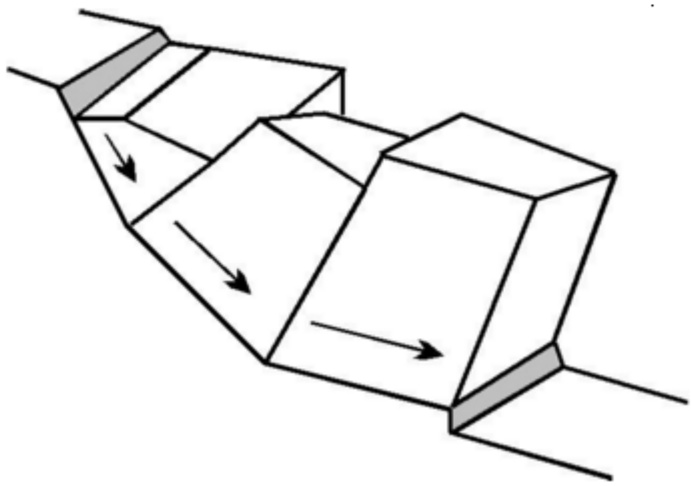
Rock Stability

Program for stability analysis of rock slope treats the following types of failure of rock faces:

- Sliding on the plane slip surface
- Translation on the polygonal slip surface
- Fall of the rock wedge



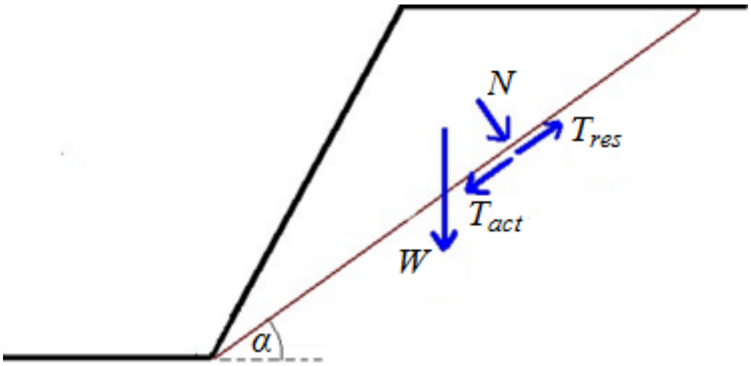
Failure of a rock face due to sliding on the plane slip surface



Translation on the polygonal slip surface

Plane Slip Surface

Failure on the plane slip surface is manifested by a rock block sliding down along this surface. The rock wedge can be specified with a tension crack. The solution procedure of stability requires determination of the **normal force** N acting on the slip surface, the **shear force** T_{act} (**active**), and the **resisting shear force** T_{res} (**passive**).



Forces on the slip surface

The shear strength parameters and the normal force N acting on the slip surface are the main input data for the determination of the resisting shear forces T_{res}. Calculation of the **active shear force** T_{act} and the **normal force** N is further influenced by the self-weight of block (depends on the geometry and unit weight of rock), anchorage, surcharge, influence of the water, and seismic effects. The active force T_{act} and the normal force N are determined as a sum of all forces entering the analysis.

The program offers several types of plane slip surfaces:

- Smooth
- Undulated
- Stepped

The resulting verification can be carried out either according to the selected verification methodology based on the input in the "Settings" frame.

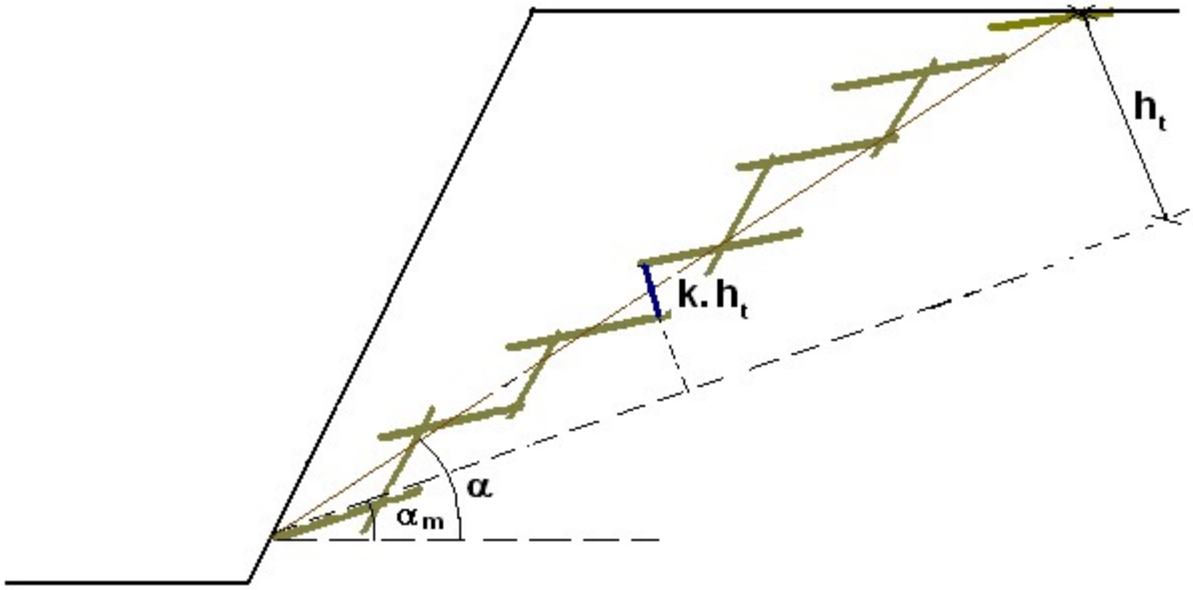
Stepped Slip Surface

If the rock body contains a system of parallel discontinuous cracks inclined to the top face of a rock and the second system is indistinctive, then it is possible to consider a formation of a stepped (jagged) slip surface in the rock body. This surface can be introduced into the program using the Calla and Nicholas theory, which increases resistance on the stepped slip surface by $\Delta\tau$.

$$\Delta\tau = \sigma_n tg\, v + T$$
$$T = \sum k.h_t.T_0$$

where:

	σ_n	-	normal stress acting in the direction normal to the slip surface
	v	-	waviness angle
	T	-	effective tensile strength of steps in the intact rock
	k	-	part of the height h_t associated with steps in the intact rock (not created by a secondary system of planes) $\sum k \in \langle 0,1 \rangle$
	h_t	-	normal height of stepped wedge resting on an inclined plane of the principal system of discontinuity planes
	T_0	-	tensile strength of intact rock



Stepped slip surface

Literature:

W.S. Dershowitz, H.H. Einstein - Characterizing rock joint geometry with joint system models Journal Rock Mechanics and Rock Engineering, Springer Wien ISSN 0723-2632 , Issue Volume 21, Number 1 / January, 1988 Pages 21-51.

Tensile Strength of Rock

Tensile strength T_e is 20 to 30x smaller than the strength of the rock in simple compression σ_c.

Strength in simple tension T_o for selected intact rocks [MPa]

Basalt	3 - 18
Gneiss	7 - 16
Granite	11 - 21
Limestone	3 - 5
Marble	7 - 12
Quartzite	4 - 23
Sandstone	5 - 11
Schist	5 - 12
Slate	2 - 17
Tuff	2 - 4

Undulated Slip Surface

If the undulated surface is considered (on scale 1 to 10 m) - it is possible to account for slip surface waviness by angle ν:

$$\nu = \alpha - min(\alpha_i)$$

where: α - slip surface gradient

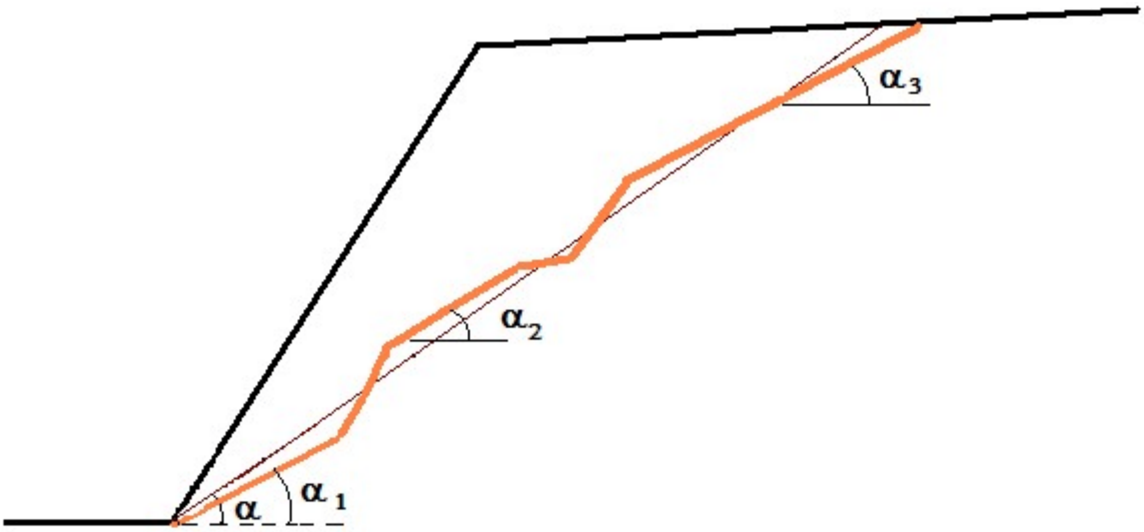
α_i - gradient of the i-th fault of the slip surface

The waviness increases the tensile strength τ on the slip surface by $\Delta\tau$:

$$\Delta\tau = \sigma_n . tg\,\nu$$

where: σ_n - normal stress acting in the direction normal to the slip surface

ν - waviness angle



Undulated slip surface

Literature:

Miller, S.M. (1988). Modeling Shear Strength at Low Normal Stresses for Enhanced Rock Slope Engineering, Proc. Of 39th Highway Geology Symp, 346-356.

Anchorage of Rock Slope

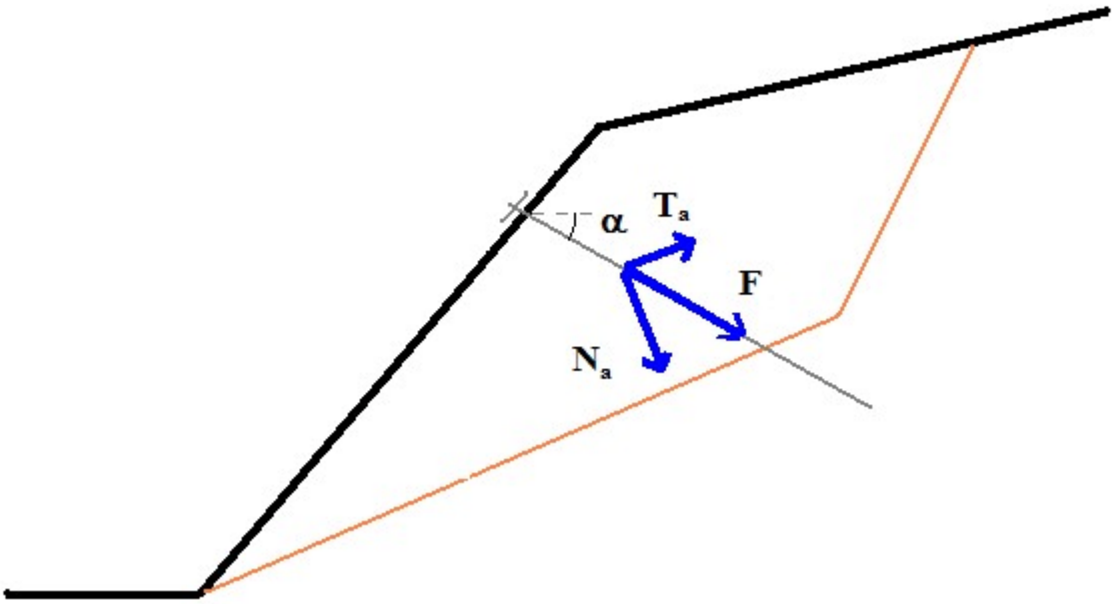
Two types of anchors can be defined when running the slope stability analysis on a plane slip surface:

Active

An active anchor is represented by a pre-stressed anchor, for which the anchor forces are activated before the sliding of a rock block takes place. The normal force increases the normal stress on a slip surface and as such also the resisting forces; the tangent component of the normal force is either added to or subtracted from the shear (active) forces.

Passive

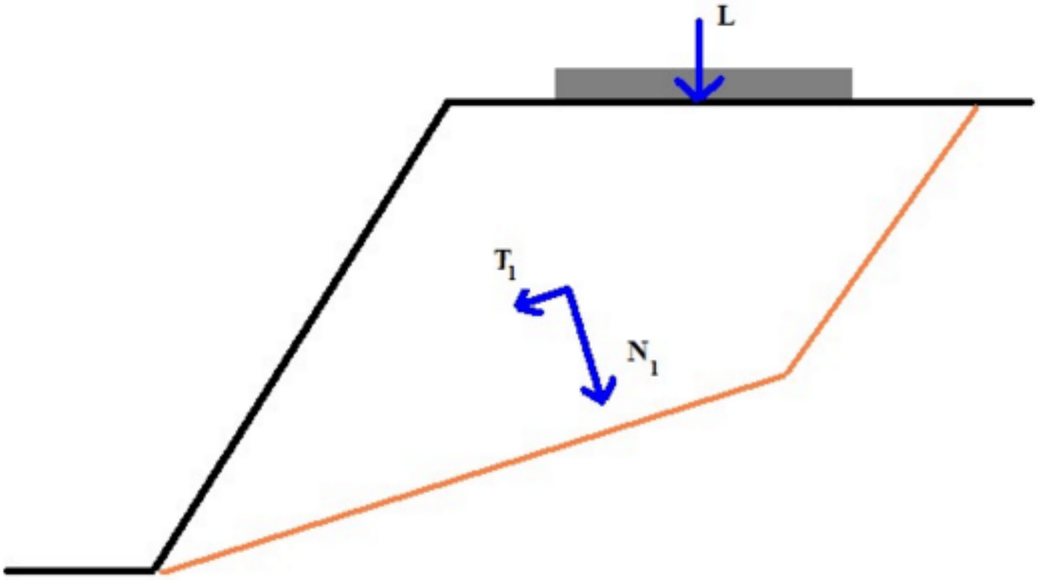
A passive anchor is activated by sliding off a rock block (i.e. not pre-stressed anchors). The normal force increases the normal stress on a slip surface and as such also the resisting forces; the tangent component of the normal force is added to the resisting forces.



Resolution of anchor force

Surcharge of Rock Slope

The surcharge resultant is determined first. The normal component of the resultant force increases the normal stress on a slip surface and as such also the resisting forces T_{res}, the tangent component is either added to or subtracted from the shear (active) forces T_{act}.



Resolution of surcharge

Influence of Water Acting on Slip Surface

The following options to account for water effects are available in the program:


Without groundwater, water is not considered


Hydrostatic pressure, GWT above toe of slope


Hydrostatic pressure, GWT on tension crack


Hydrostatic pressure, GWT on tension crack, max


Hydrostatic pressure, water acting on tension crack only

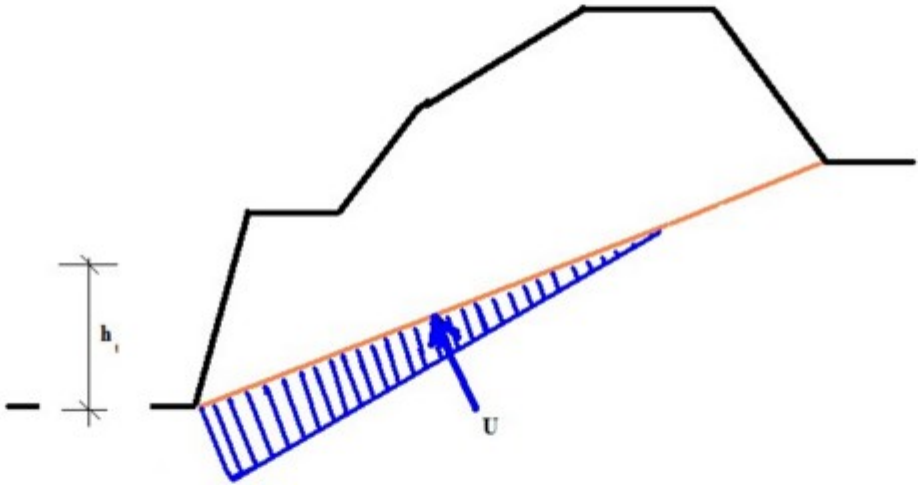

Own water force acting on slip surface only


Own water force behavior

GWT Above Slope Toe


Hydrostatic pressure, GWT above toe of slope

The slip surface is either entirely or partially below the groundwater table (water can not outflow from slip surface), the maximal water pressure is at the toe of the face.



Hydrostatic pressure on the slip surface

The value of water pressure u at the heel of the slope is given by:

$$u = \gamma_w h_t$$

where: γ_w - unit weight of water

h_t - height of GWT above the toe of the slope

The resulting compressive water force acting in the direction normal to the slip surface is given by:

$$U = \frac{1}{2}\gamma_w h_t \left(\frac{h_t}{sin\alpha} \right)$$

where: γ_w - unit weight of water

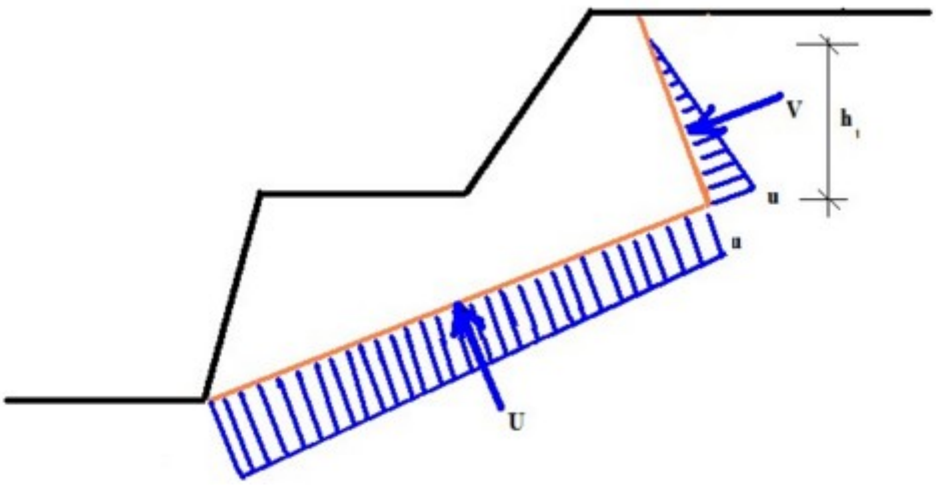
h_t - height of GWT above the toe of the slope

α - deflection of slip surface from horizontal

GWT on Tension Crack

GTW on tension crack

The slip surface is entirely below the groundwater table; the GWT either intersects the tension crack or is aligned with the terrain, the maximal value of uplift pressure is at the toe of the face.



Hydrostatic pressure on slip surface and on tension crack, max. value at the toe of the slope

The value of uplift pressure u at the intersection of slip surface and tension crack is given by:

$$u = \gamma_w h_t$$

where: γ_w - unit weight of water

h_t - height of GWT above the line of intersection of slip surface and tension crack

The resulting compressive water force V acting in the direction normal to the tension crack is given by:

$$V = \frac{1}{2}\gamma_w h_t \left(\frac{h_t}{cos\varphi}\right)$$

where: γ_w - unit weight of water

h_t - height of GWT above the line of intersection of slip surface and tension crack

φ - deflection of tension crack from vertical

The value of hydrostatic pressure u_1 at the toe of the slope is given by:

$$u_1 = \gamma_w H_w$$

where: γ_w - unit weight of water

H_w - height of GWT above toe of slope

The resulting compressive water force U acting in the direction normal to the slip surface is given:

$$U = \frac{1}{2}(u + u_1)\left(\frac{H_w - h_t}{sin\alpha}\right)$$

where: u - water pressure acting on the line of intersection of slip surface and tension crack

u_1 - water pressure at the toe of the slope

h_t - height of GWT above the line of intersection of slip surface and tension crack

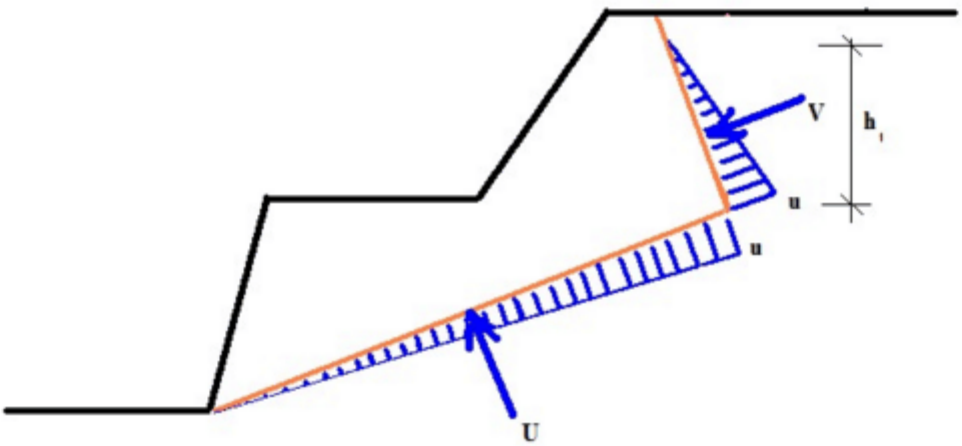
α - deflection of slip surface from horizontal

H_w - height of GWT above the toe of the slope

GWT on Tension Crack, Max. Tens. Crack

GWT on tension crack

The slip surface is entirely below the groundwater table, the GWT either intersects the tension crack or is aligned with the terrain (water can outflow at the slope heel), the maximal value of uplift pressure is at the intersection of tension crack and slip surface.



Hydrostatic pressure on slip surface and on tension crack

The value of uplift pressure u at the intersection of slip surface and tension crack is given by:

$$u = \gamma_w h_t$$

where: γ_w - unit weight of water

h_t - height of GWT above the line of intersection of slip surface and tension crack

The resulting compressive water force V acting in the direction normal to the tension crack is given by:

$$V = \frac{1}{2}\gamma_w h_t\left(\frac{h_t}{cos\varphi}\right)$$

where: γ_w - unit weight of water

h_t - height of GWT above the line of intersection of slip surface and tension crack

φ - deflection of tension crack from vertical

The resulting value of pressure u_1 at the toe of the slope is equal to zero.

The resulting compressive water force U acting in the direction normal to the tension crack is given:

$$U = \frac{1}{2}u_1\left(\frac{H_w - h_t}{sin\alpha}\right)$$

where: u - water pressure acting on the line of intersection of slip surface and tension crack

h_t - height of GWT above the line of intersection of slip surface and tension crack

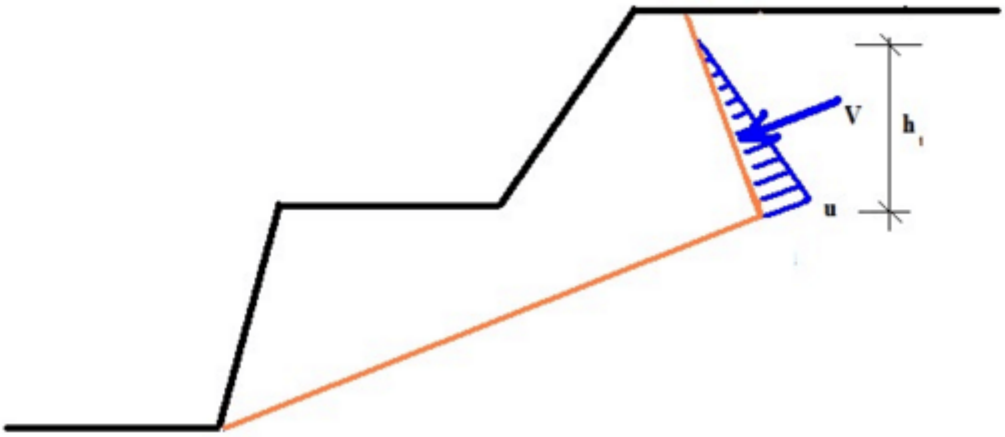
α - deflection of slip surface from horizontal

H_w - height of GWT above the toe of the slope

Water Acting Only on Tension Crack

Water acting on tension crack only

The slip surface is fully dry (seepage is not possible); the GWT either intersects the tension crack or is aligned the terrain, the maximal value of uplift pressure is at the intersection of slip surface and tension crack.



Water acting on tension crack only

The value of uplift pressure u at the intersection of slip surface and tension crack is given by:

$$u = \gamma_w h_t$$

where: γ_w - unit weight of water

h_t - height of GWT above the line of intersection of slip surface and tension crack

The resulting compressive water force V acting in the direction normal to the tension crack is given by:

$$V = \frac{1}{2}\gamma_w h_t\left(\frac{h_t}{cos\varphi}\right)$$

where: γ_w - unit weight of water

h_t - height of GWT above the line of intersection of slip surface and tension crack

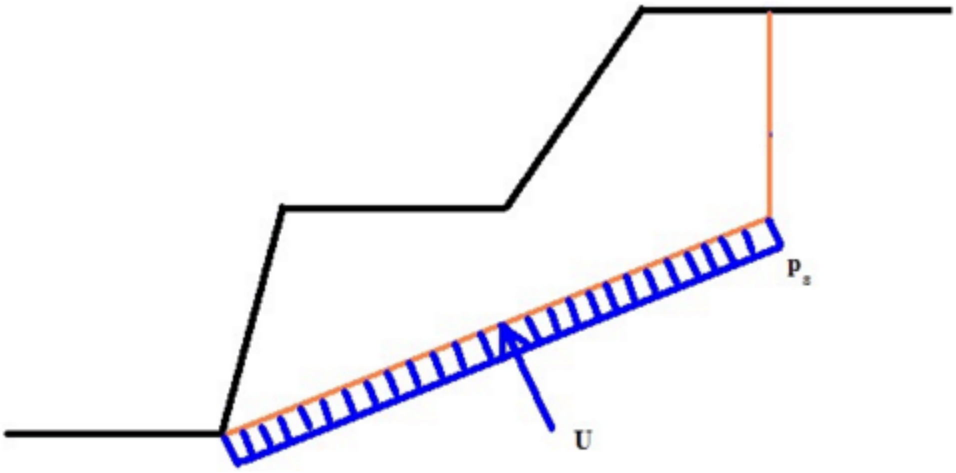
φ - deflection of tension crack from vertical

The value of water pressure acting on the slip surface is equal to zero.

Own Water Force Acting Only on Slip Surface

Own water force acting on slip surface only

The program allows a manual input of the constant value of water pressure p_s [kPa] acting on a slip surface.

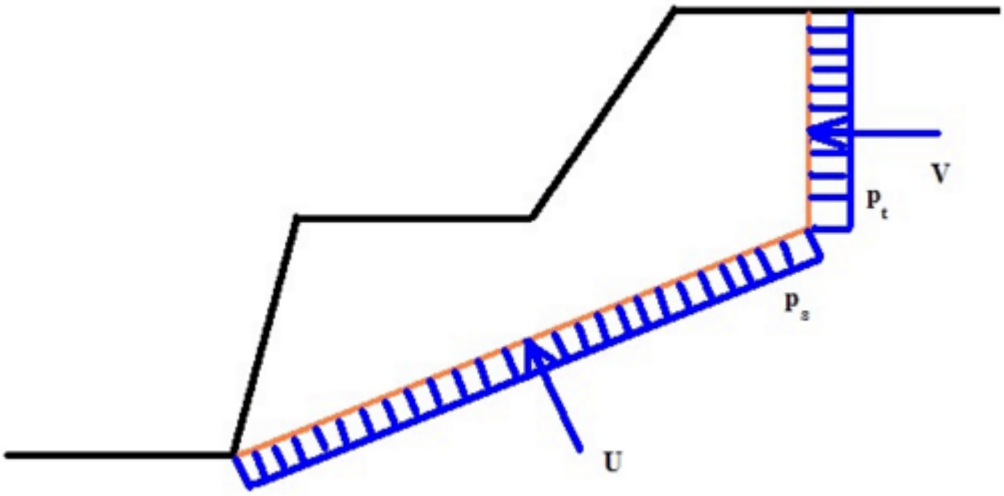


Own value of water pressure acting on slip surface

Own Water Force Behavior

Own water force behavior

The program allows a manual input of the constant value of water pressure p_t [kPa] acting on the slip surface and p_t [kPa] acting on a tension crack.



Own values of water pressure on slip surface and on tension crack

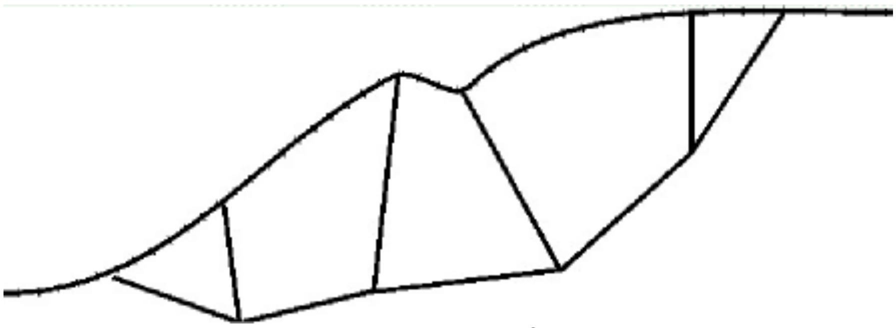
Polygonal Slip Surface

The program performs stability analysis of rock blocks moving along the polygonal slip surface. Owing to the complexity of the general solution the program admits the following assumptions:

- Motion of rock blocks is only translational.
- Blocks translate along the polygonal slip surface formed either by planar planes or planes with moderate waviness.
- Rock blocks are divided by joints with known directions.
- Actual deformation of rock mass inside the blocks is negligible.
- Failure on the polygonal slip surface and along joints is driven by the Mohr-Coulomb failure criterion.
- The same factor of safety is assumed for the entire polygonal slip surface.
- All rock blocks are in contact (the opening of joints is not allowed).

The Mohr-Coulomb shear strength parameters on the slip surface and on joints separating individual blocks are the main input data for the determination of stability of rock blocks. The solution is further influenced by the weight of block (depends on the geometry of block and unit weight of rock), anchorage, surcharge acting on the blocks, influence of water, and seismic effects.

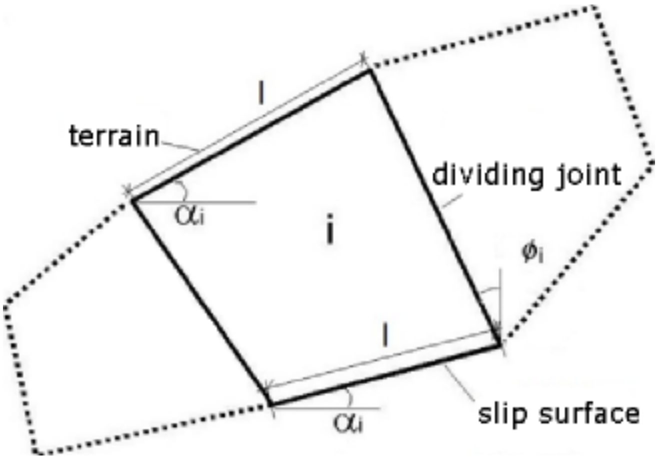
The basic theoretical grounds of the solution are described here.



Polygonal slip surface

Geometry of Rock Block

The block geometry is determined by the gradient α , by the length of a given slip surface l and by the gradient of dividing joints φ separating the subsequent block as well as by the gradient α and the length l of the top face of the external surface of a rock slope (natural profile). Lengths of planes can be defined either by the total length or by the lengths of their horizontal and vertical projections. It is necessary to ensure the condition that all rock blocks are in contact (the opening between joints is not allowed).



Geometry of the i^{th} element

Anchor Forces, Surcharge

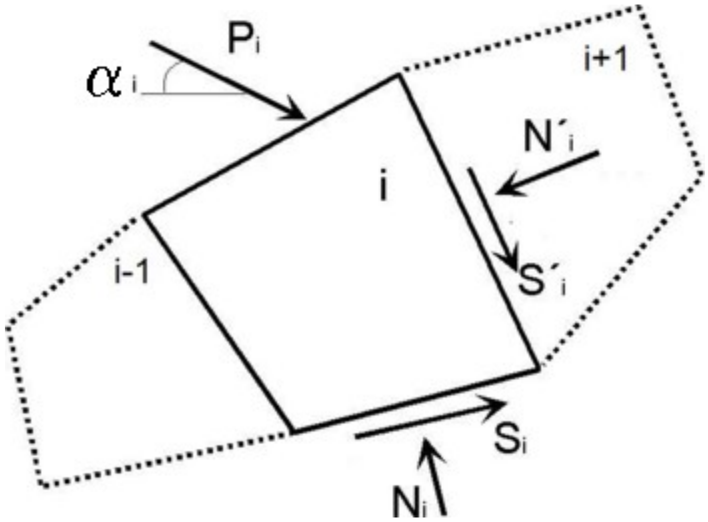
It is possible to introduce anchor forces and surcharges of rock blocks. The resultant of forces P_i $[kN/m]$ acting on the i^{th} block is then determined. All forces acting on the block including the water pressure on the slip surface and the block separating joints are taken into account.

Surcharge acting on the block

It is possible to input surface, strip, and trapezoidal surcharge of terrain. The program then determines its effect on individual rock blocks.

Anchor forces

The applied anchor force is adjusted per 1 m run based on the specified horizontal spacing of anchors.



External forces acting on the i^{th} element

Influence of Water

The influence of water can be considered using the following options: **general shape of GWT**, **horizontal GWT,** or directly by **water acting on the blocks**.

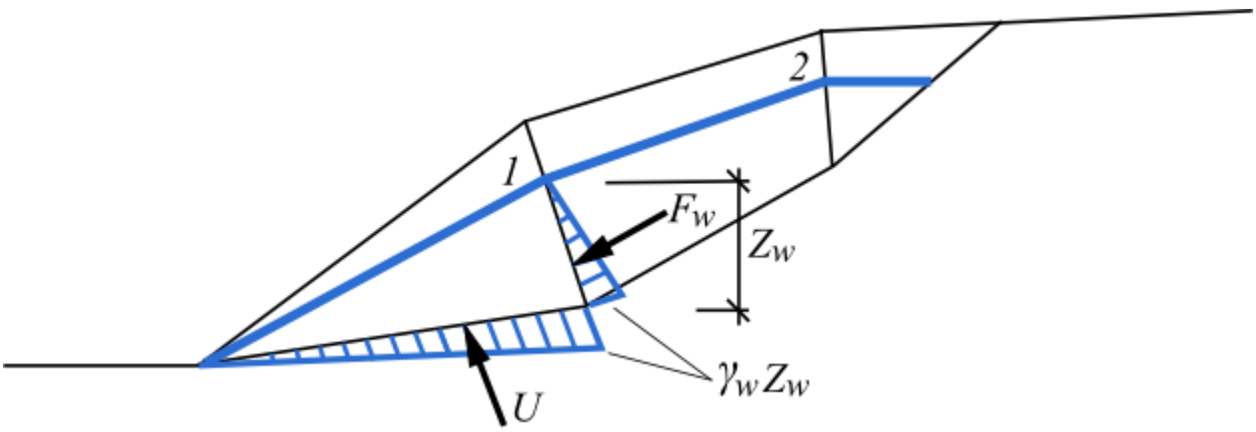
General shape of GWT

The general shape of GWT is entered as a polygon. The pore pressure (stress) on the slip surface is considered linearly according to the equation: $u = \gamma_w * z_w$.

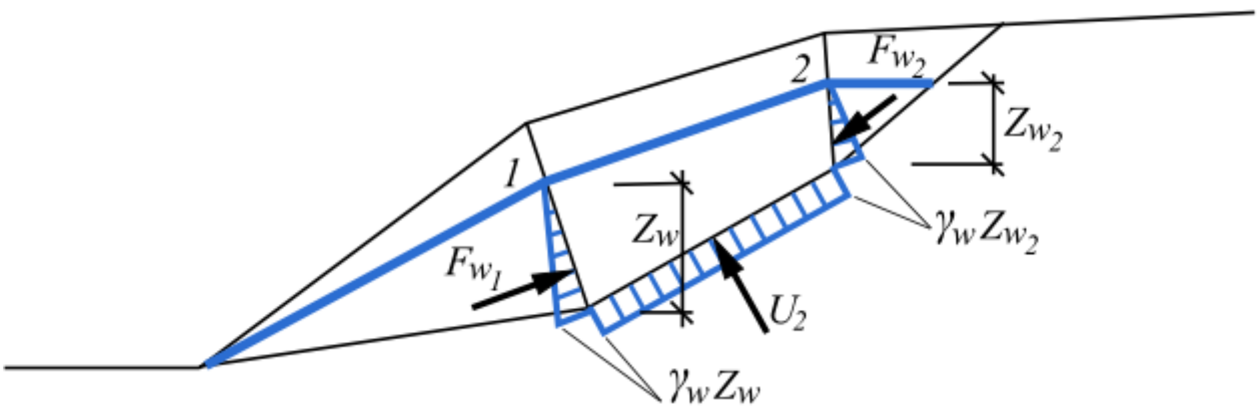
where: γ_w - unit weight of water

z_w - height of GWT above slip surface in the joints

The resultant forces U (force due to water on slip surface) or F_w (force due to water on internal slip surface acting on the submerged part of joint per metre of width) are calculated from the pore pressure load diagrams.



Forces from the water acting on the block - water can flow freely of the gap



Forces from the water acting on the block - water can't flow of the gap

Horizontal GWT

The horizontal GWT is entered by constant height h_w over the heel of the slope (from the origin of the coordinate system). The influence of water is considered from the water level (GWT) to a given point on the vertical.

Water entered on blocks

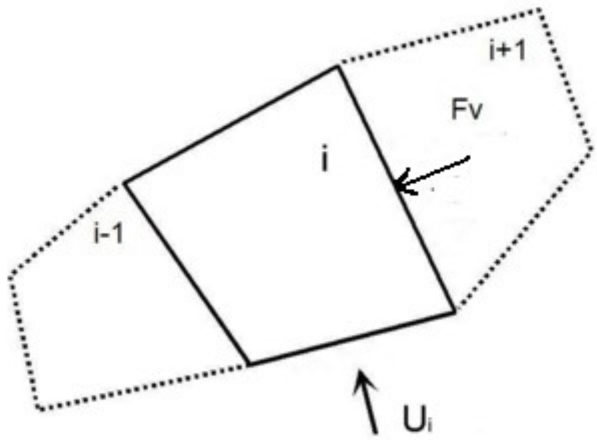
The water pressure along the joints and on the slip surface can be taken into account. It is introduced as an external load:

Force due to water on internal slip surface (water between blocks) F_v

It must be introduced into the analysis whenever the presence of water in the joints between blocks is expected. It is applied as a resultant force F_v in kN (the pressure acting on the immerse part of the joint per 1m run is considered).

Force due to water on external slip surface (uplift pressure) U

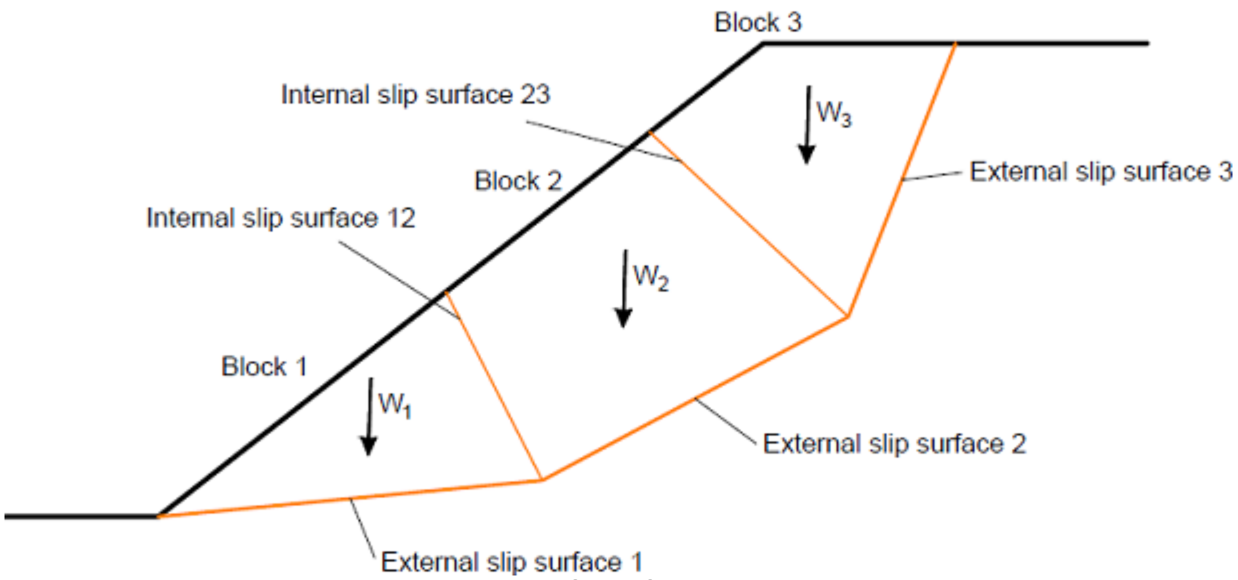
It is defined as hydrostatic pressure on each slip surface of the polygon (external slip surface) separately and introduced as an external load (uplift pressure) U in kN, which can be reduced depending on the slip surface permeability (the pressure acting on the immerse part of the slip surface per 1m run is considered).



Water forces acting on a rock block

Solution Procedure

The principle of calculation of rock slope stability for polygonal slip surface is shown in the following figures.



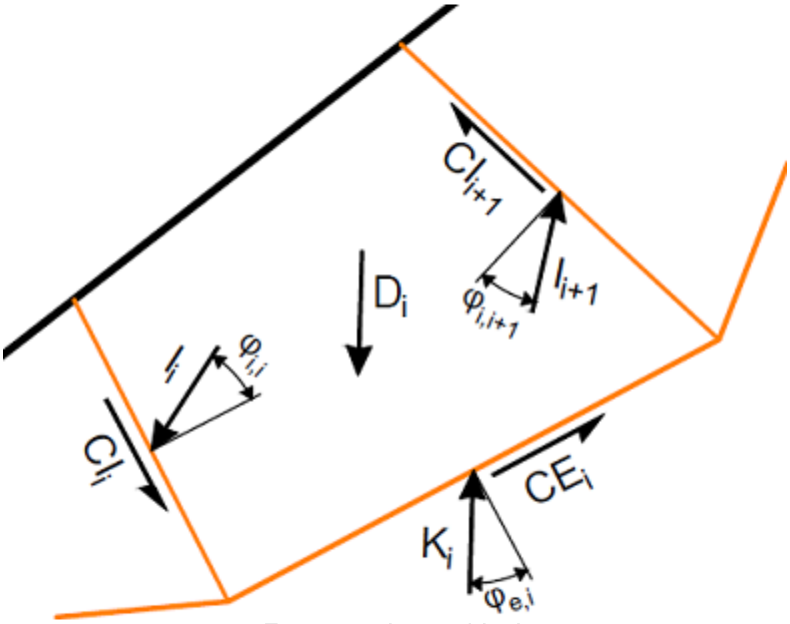
Rock Slope Scheme

The stability analysis is conducted in a sequential manner for all blocks ranging from the uppermost block to the lowest block. The stability of each block is assessed to the required factor of safety and forces acting on the next (lower) block are calculated. The lowest calculated value of stability from all blocks is given as the final stability of the rock slope.

D_i is the vector representing the resultant of all the disturbing forces acting on the block i given by:

$$\vec{D_i} = \vec{W_i} + \vec{P_i} + \vec{E_i} + \vec{U_i} + \vec{V_i} + \vec{H_i} + \vec{B_i}$$

where: W_i - weight of the block
P_i - external forces (loads) acting on the block
E_i - earthquake forces on the block
U_i - force due to water on external slip surface
V_i - force due to water on internal slip surface
H_i - force due to water above the terrain
B_i - anchor force on the block



Forces acting on block

where: D - resultant of all external forces acting on the block
CI - cohesion on internal slip surface
CE - cohesion on external slip surface
I - interaction force
K - reaction force on external slip surface

To assess the stability of the block i the resultant force R is determined. It consists of resultant of external forces D and forces acting on the block from the previous block:

$$\overrightarrow{R_i} = \overrightarrow{D_i} - \overrightarrow{I}_{i+1} - \overrightarrow{CI}_{i+1}$$

If the tensile has occurred on the external slip surface, the stability of the block against slip on the internal slip surface is assessed.

$$T_i = R_{\mathrm{N},i} \tan \varphi_i + CI_i$$

$$\frac{T_i}{R_{\mathrm{T},i}} \geq SF$$

where: T_i - resisting force
φ_i - angle of the internal friction on the internal slip surface
$R_{N,i}$ - normal component of R
$R_{T,i}$ - tangential component of R
CI_i - cohesion on internal slip surface

If the external slip surface is loaded by compression, the factor of safety *FS,* for which the equilibrium of forces acting on the block is fulfilled, is determined. The force R , cohesion on internal and external slip surface must be in equlibrium with the reaction K and interaction force I . The inclination of forces K and I is given by φ_M .

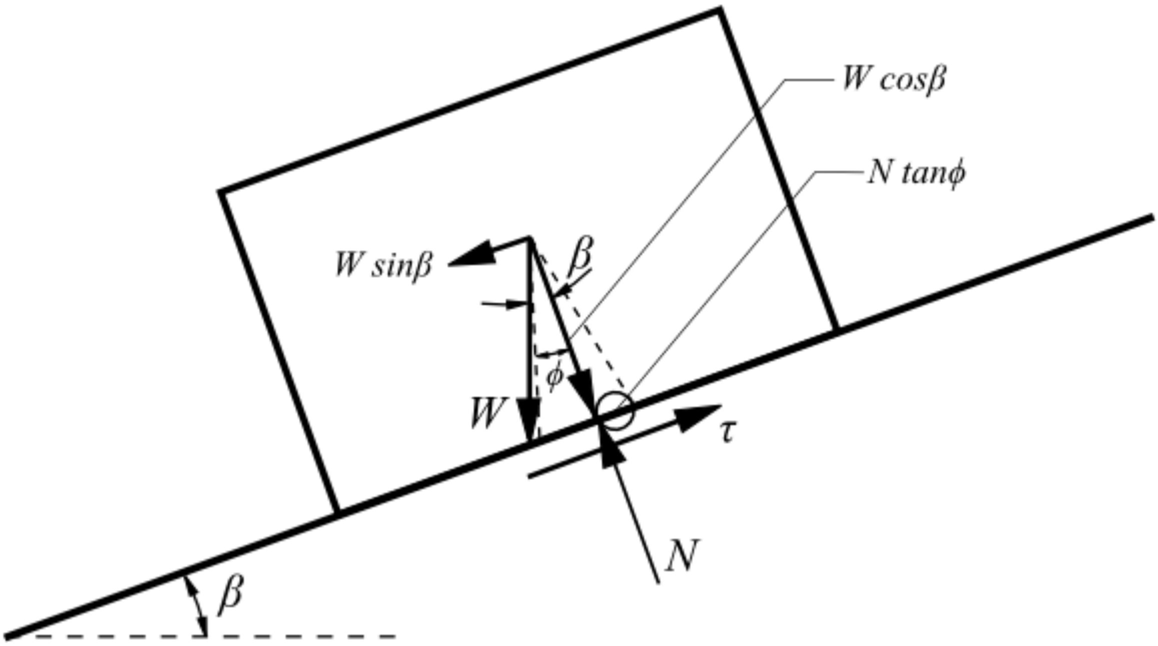
$$\tan \varphi_{\mathrm{M}} = \frac{\tan \varphi}{FS}$$

$$-\left(\vec{R}_i + \frac{\vec{CI}_i}{FS} + \frac{\vec{CE}_i}{FS}\right) = \vec{K}_i + \vec{T}_i$$

The resulting value of SF, when the equilibrium on the block is fulfilled, is compared with the required factor of safety.

Cone Friction Concept

Friction cone concept is a combination of the kinematic and kinetic calculation method, which is used to find potential slip surfaces of failure. The principle of solution is shown in the following figure.



The friction cone concept for a block resting on an inclined plane (polygonal slip surface)

The resistant forces are described using the following condition:

$$N\,tg\,\phi = c\,A + W\,\cos\beta\,tg\,\phi$$

where: A - block area resting on the slip surface

c - cohesion on the sliding surface

W - vector due to self-weight of the block (self-weight resultant of the rock block)

N normal to the slip plane

ϕ - angle of internal friction

Literature:

Goodman, R. E.: Introduction to Rock Mechanics: John Wiley & Sons, New York, 1989, 562 p.

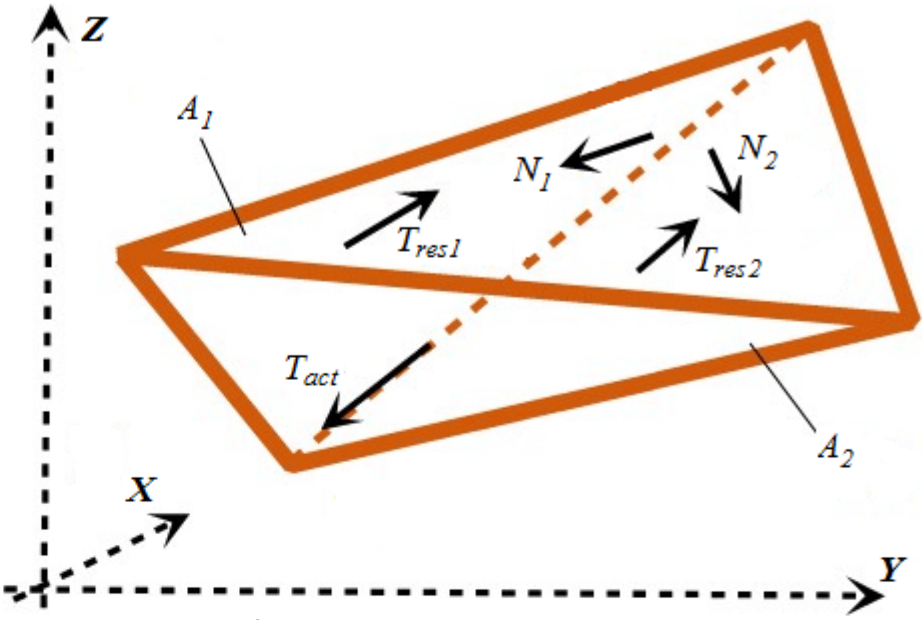
Rock Wedge

The program performs stability analysis of a rock wedge that is wedged in between two surfaces (planes) and slides in the direction of the line of interaction (tray) of these planes. The rock wedge can be specified with a tension crack. The gradient of this intersection must be considerably larger than the angle of internal friction dividing planes, whereas the falling line of both dividing planes must be directed towards the line of intersection. It is further assumed that the tray is located in a stable rock body.

The solution requires determination of **the normal force** N, the **shear force** T_{act} (active) and the **resisting (passive) shear force** T_{res} acting on slip surfaces A_1 and A_2. The active force T_{act} and the normal force N are obtained as a summation of all forces entering the analysis after performing the spatial resolution of these forces.

The Mohr-Coulomb shear strength parameters and the **normal force** N acting on the slip surface are the main input data for the determination of the **resisting shear forces** T_{res}. Calculation of the **active shear force** T_{act} and the **normal force** N is further influenced by the weight of the wedge (depends on the geometry of wedge and unit weight of rock), anchorage of the wedge, surcharge acting on the wedge, influence of water and seismic effects.

The resulting verification can be carried out according to the selected verification methodology based on the input in the "Settings" frame.

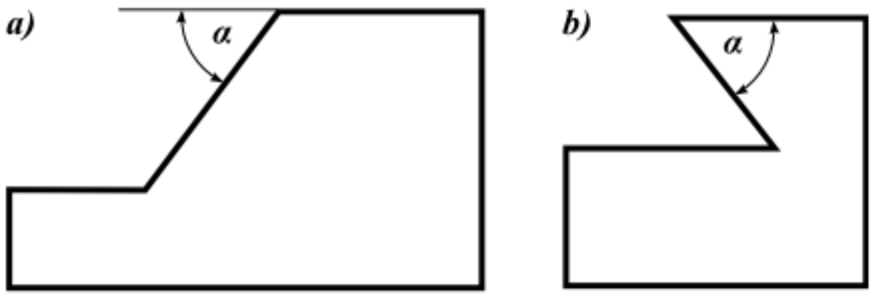


Components acting on a rock wedge

Geometry of Rock Wedge

Entering geometry of a rock wedge using either gradient or falling line gradient direction requires a definition of space orientation of the rock face, terrain (top face), slip surfaces N_1 and N_2 and/or tension crack, such that:

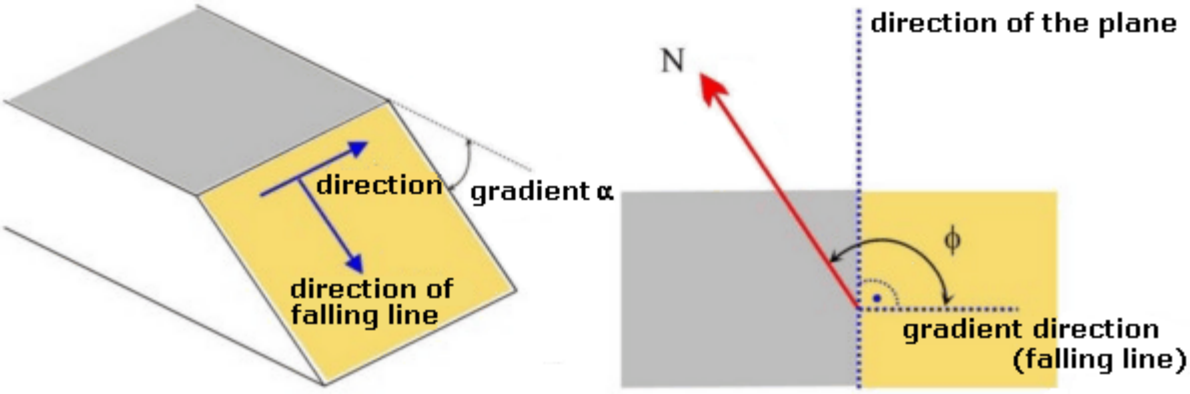
- **Gradient of surface** (gradient angle) is an inclination angle α representing inclination of the surface from horizontal (it may receive values from 0° to 90°). In case of overhanging slope (the edge of the slope is before the slope toe - wall tends from edge to the rock mass), then it must be checked button "**Overhang rock face**" and gradient of the face φ is considered in the half-plane of the rock mass. The program checks the possibility of overturning failure of the rock mass with the overhang rock face. If the option of an earth wedge overturning is realistic, the program notifies the user in the listing of the results. However, the program does not make a real evaluation of the overturning or rotation of the rock wedge.



a) Pendulous rock face b) Overhang rock face

- **Gradient direction** (falling line) is an angle b between the horizontal projection of the line normal to the strike direction measured as an azimuth angle from the north in the clockwise direction (the falling line corresponds to inclination of the plane), it may receive values from 0° to 360°.

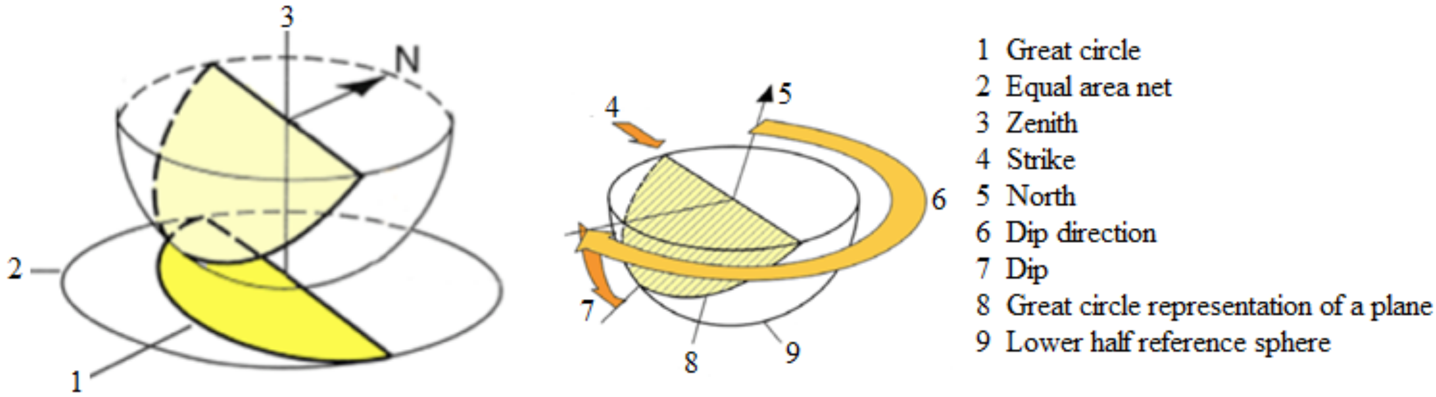
The program when defining space orientation of planes displays these planes using a stereographic projection.



Description of orientation of surfaces - using gradient and direction

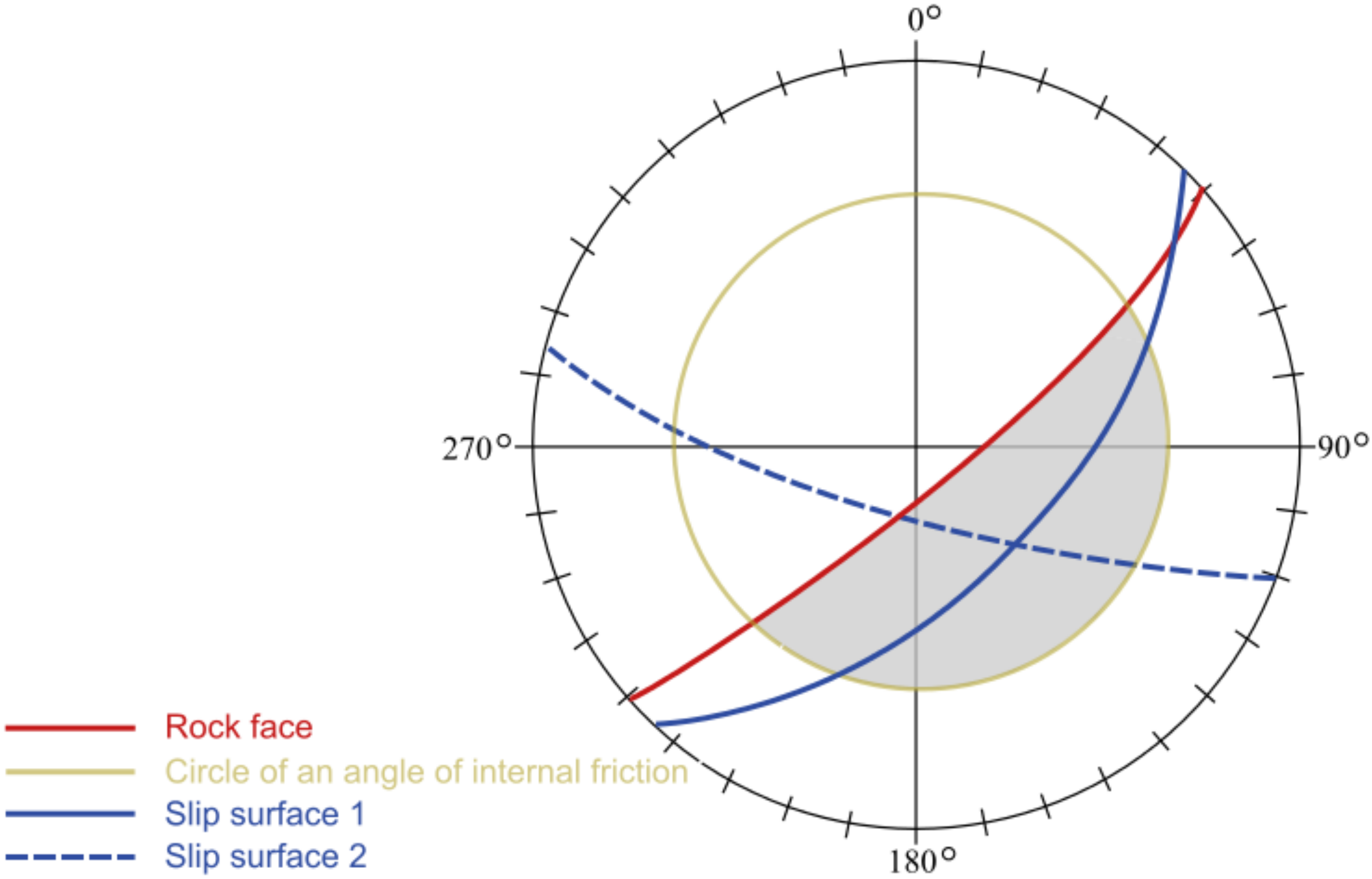
Stereographic Projection

When defining the geometry of the wedge and slip surfaces using space projection, the program displays individual surfaces with the help of great circles (in equal are net) of Lambert's hemispherical projection.



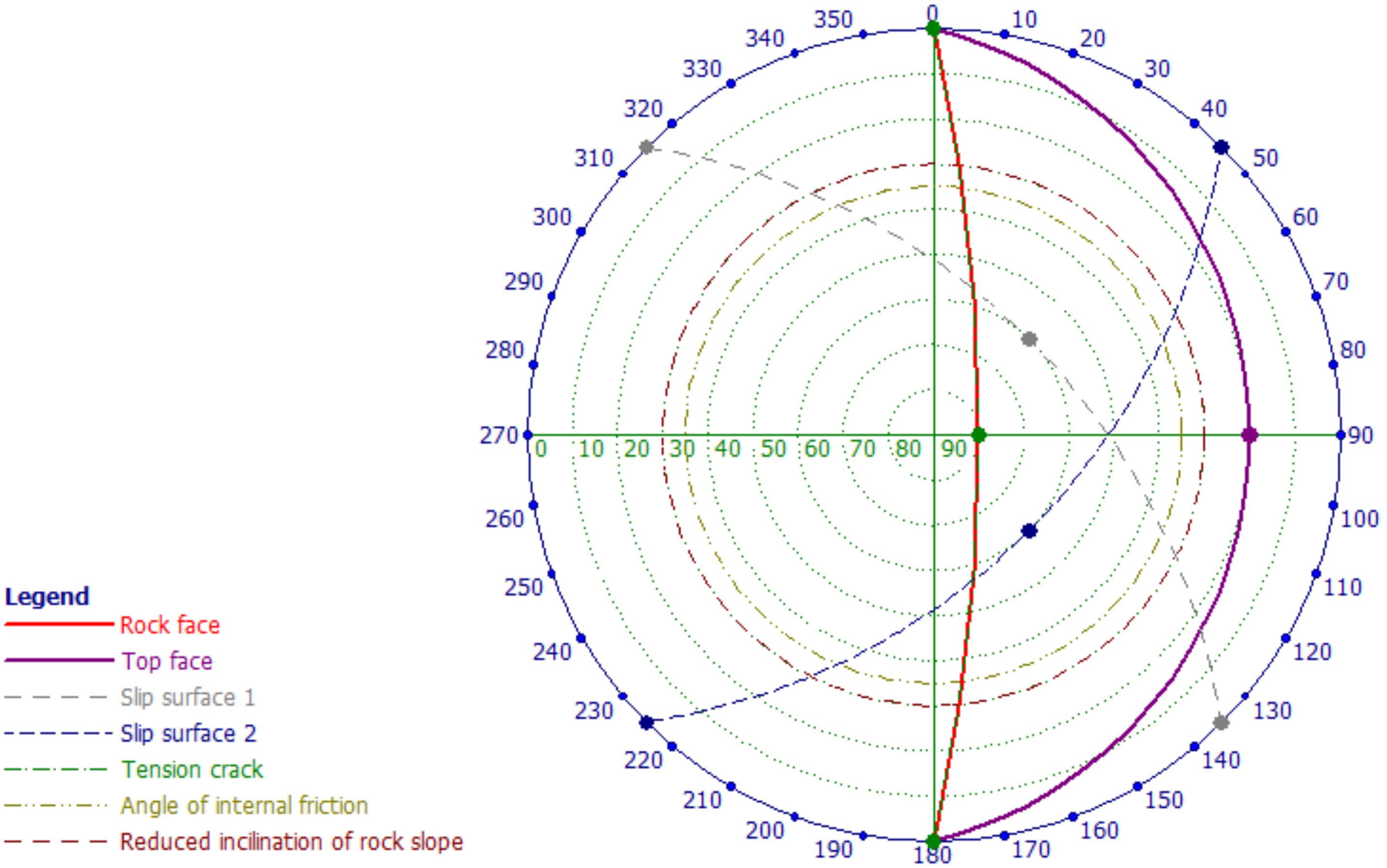
Principle of Stereographic projection (Lambert's projection)

Markland test utilizes a stereographic projection of the great circle for the kinematic possibility of the rock wedge failure. The geometry of the rock wedge is plotted on the stereographic projection using the great circle representing the slope face and discontinuities of slip surfaces 1 and 2 (see figure). The friction circle is plotted on the projection too, the friction angle on the discontinuities is measured from the north and the circle center is the center of the stereograph. The zone between the great circle representing the slope face and the friction circle is a shaded area on the figure. When the plunge of the line of intersection of two discontinuities (1 and 2) is in the shaded area, then the failure is cinematically possible.



Kinematic possibility of the rock wedge (Markland's test plot in a stereographic projection)

View of the geometry of the rock wedge is supplement by elements of Markland's test plot. This makes it possible to assess the stereographic projection of the kinematics of the rock wedge.



Markland's test plot

Influence of water

By default, the program performs the stability analysis of a rock wedge without considering groundwater. If interested in the influence of groundwater on a rock wedge it is necessary to introduce the height of GWT from the line of intersection of slip surfaces and rock face (the GWT takes an arbitrary position over the entire height of a rock wedge). The program assumes that water can flow freely discontinuities located below the GWT (no restrictions, e.g. due to ice blocks, are considered).

The water pressure acts in the direction normal to the slip surfaces against normal components of the passive forces. If the height y_w above the point of maximal pressure P_{max} is equal or larger than $Z/2$ and it is fully contained by the rock wedge, then its value is assumed to be equal to $Z/2$ (case A). If the height y_w above the point of maximal pressure P_{max} is less than $Z/2$ (case B), then its value reduced as:

$$y_w = \left(\frac{1}{2}.L^*.\sin\delta\right).\left(\frac{tg\alpha_1}{tg\delta} - 1\right)$$

where: L^* - length of the line intersection of slip surfaces A_1 , A_2

α_1 - gradient of rock face

δ - gradient intersection line of the slip surfaces

The resulting water pressure on slip surfaces 1 and 2 is given by:

$$P_1 = \frac{1}{3}.P_{max}.A_1^w = \frac{1}{3}.\gamma_w.y_w.A_1^w$$

$$P_2 = \frac{1}{3}.P_{max}.A_2^w = \frac{1}{3}.\gamma_w.y_w.A_2^w$$

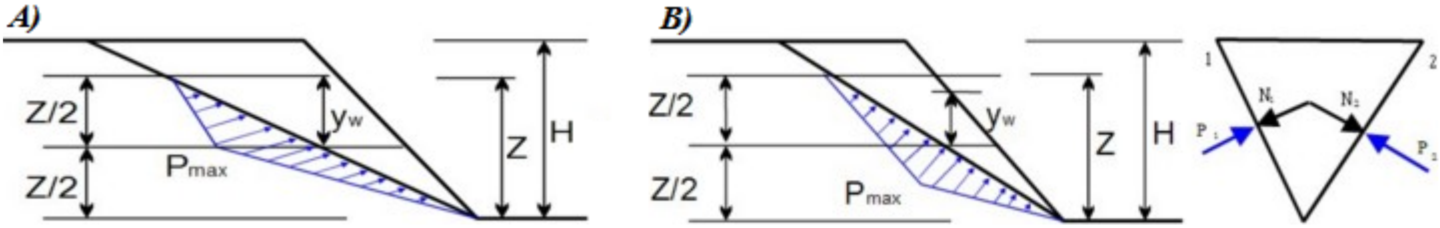
where: Z - height of GWT above the line of intersection of slip surfaces and rock face

P_{max} - maximal water pressure on the line of intersection of slip surfaces

γ_w - unit weight of water (≈ 10 kN/m^3)

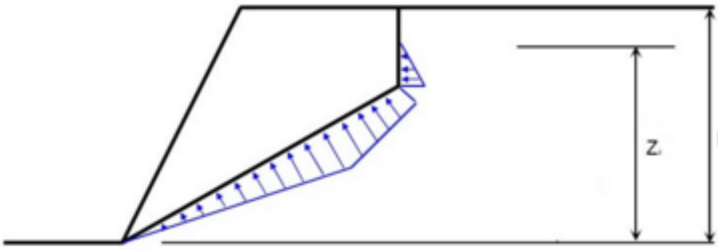
A_1^w - are of the wetted part of the slip surface 1

A_2^w - area of the wetted part of the slip surface 2



Distribution of water pressure on the line of intersection of slip surfaces

If a tension crack is found either entirely or partially below the GWT, then the influence of water pressure is reflected both on slip surfaces 1 and 2 through forces P_1 and P_2 acting on the intersection of these surfaces and on tension crack through force P_3 acting in the direction normal to the tension crack.



Distribution of water pressure when considering GWT in tension crack

Resolution of Acting Forces

Forces acting on a rock wedge (weight of rock wedge, external load, anchor force) are resolved into directions normal to planes A_1 and A_2 (the block is wedged in between these surfaces) and into the direction of their intersection. The resolution of forces results in the normal forces N_1, N_2 acting on planes A_1 and A_2, resisting (passive) forces T_{res1}, T_{res2} acting along planes A_1 and A_2.

This step further generates the **shear (active) force** T_{act} acting in the direction of the line of intersection of slip surfaces. The resulting **shear (active) force** T_{act} is obtained as a sum of individual shear forces $T_{act,i}$.

The resisting (passive) force T_{res} is found by summing up the components T_{res1}, T_{res2} (e.g. due to external load) and friction forces on planes $A1$ and $A2$ due to normal forces:

$$T_{res} = \sum T_{res,1} + \sum T_{res,2} + \sum \left(N_1 tg\varphi_1 + c_1 A_1 \right) + \sum \left(N_2 tg\varphi_2 + c_2 A_2 \right)$$

where:

c_1 - cohesion on slip surface A_1

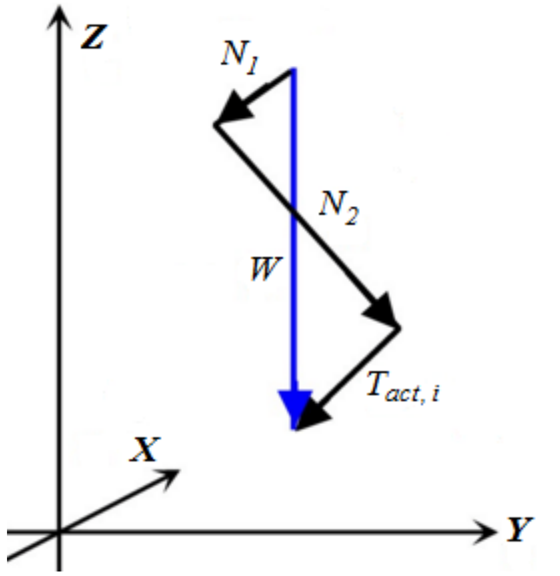
c_2 - cohesion on slip surface A_2

φ_1 - angle of internal friction on the slip surface A_1

φ_2 - angle of internal friction on the slip surface A_2

T_{res1} - resisting forces on slip surface As

T_{res2} - resisting forces on slip surface A_2



Space resolution of self-weight of earth wedge W

Verification

Verification can be carried out either according to the selected verification methodology based on the input in the "Settings" frame.

Verification According to the Factor of Safety

When performing verification according to the factor of safety the program directly determines the value of the factor of safety SF . Verification condition has the form:

$$SF = \frac{T_{res}}{T_{act}} > SF_s$$

where:
- T_{act} - shear forces along the slip surface
- T_{res} - passive forces along the slip surface
- SF - safety factor
- SF_s - required safety factor (input in the "Stability analysis" tab)

Typical values are e.g. - for walls of foundation pits $SF_s =$ 1.1 to 1.25, - for rock cuts of highways $SF_s =$ 1.2 to 1.5, etc.

Verification According to the Theory of Limit States

When performing verification according to the theory of limit states the program reduces material parameters of rocks (angle of internal friction or tangent of the angle of internal friction, cohesion) using partial coefficients entered in the "Stability analysis" tab. Verification condition has the form:

$$T_{act} < \frac{T_{res}}{\gamma_s}$$

where:
- T_{act} - shear forces along the slip surface (active)
- T_{res} - passive forces along the slip surface
- γ_s - reduction coefficient of the overall stability of construction (input in the "Stability analysis")

When analyzing the **polygonal slip surface** the program compares the calculated value with the value corresponding to the fully stressed design (state of equilibrium with zero reserves). Verification condition has the form:

$$SF > SF_s$$

where:
- SF - factor of safety calculated with the reduced material parameters
- SF_s - coefficient of the overall stability of the structure

Rock - Shear Resistance Criteria

The shear strength is the basic criterion to determine the resisting passive forces. The resisting force is given by the following expression:

$$T_{res} = \tau\, l$$

where: τ - shear strength on the slip surface

l - length of the slip surface

The shear strength for the planar slip surface can be written as:

- Mohr - Coulomb
- Hoek - Brown
- Barton - Bandis

Mohr - Coulomb

The shear strength τ $[kPa]$ according to Mohr-Coulomb is given by the equation:

$$\tau = c + \frac{N}{l} tg\varphi$$

where: N - normal force acting on the slip surface

l - length of the slip surface

c - cohesion of rock on the slip surface

φ - angle of internal friction of rock on the slip surface

Approximate ranges of parameters of the Mohr-Coulomb failure criterion for selected soils are given here.

Mohr - Coulomb Parameters

If possible the strength parameters should be determined in-situ measurements. The results of in-situ and laboratory experiments show that the angle of internal friction is found for the majority of discontinuities in the rock mass in the range of $27°$ to $47°$. Approximate values of the angle of internal friction φ and cohesion c for rocks based on the RMR classification are stored in the following table:

Rock class	**I**	**II**	**III**	**IV**	**V**
RMR	100 - 81	80 - 61	60 - 41	40 - 21	< 20
Angle of internal friction φ [°]	> 45	35 - 45	25 - 45	15 - 25	< 15
Cohesion c $[kPa]$	> 400	300- 400	200 - 300	100 - 200	< 100

Hoek - Brown

Hoek-Brown failure criterion describes the failure of a rock mass (based on the performed analyses of hundreds of underground structures and rock slopes) as:

$$\sigma_{1,ef} = \sigma_{3,ef} + \sigma_c \left(\frac{m_b . \sigma_{3,ef}}{\sigma_c} + s \right)^a$$

where: σ_{1ef} - major principal stress during rock failure

σ_{3ef} - minor principal stress during rock failure

σ_c - strength of the intact rock in simple compression

σ_{ci} - uniaxial compressive strength of intact pieces of rock

$m_{b,s}$ - nonlinear material constant depending on the rock quality

a - coefficient depending on the rock breaking

The basic parameters of the modified Hoek-Brown model should be determined from in-situ and laboratory measurements. To become more acquainted with this model, a brief list of ranges of individual parameters is provided. If **rock mass classification using GSI** is known then it is possible to let the program determine the H-B parameters by itself.

For actual analysis, the H-B parameters are transformed into the M-C parameters. The solution procedure then becomes identical to that of the Mohr-Coulomb criterion.

For conversion of Hoek-Brown parameters is used solution according to Hoek, Carranza-Torres, and Corkum (2002) in case of the analysis of rock slope stability:

The angle of internal friction φ :

$$\varphi' = \arcsin\left[\frac{6\,a\,m_b\left(s+m_b\,\sigma'_{3n}\right)^{a-1}}{2(1+a)(2+a)+6\,a\,m_b\left(s+m_b\,\sigma'_{3n}\right)^{a-1}}\right]$$

Cohesion (shear strength) c :

$$c' = \frac{\sigma_{ci}\left[(1+2a)s+(1-a)m_b\,\sigma'_{3n}\right]\left(s+m_b\,\sigma'_{3n}\right)^{a-1}}{(1+a)(2+a)\sqrt{1+\left(6\,a\,m_b\left(s+m_b\,\sigma'_{3n}\right)^{a-1}\right)/\left((1+a)(2+a)\right)}}$$

where:

$$\sigma'_{3n} = \frac{\sigma'_{3\max}}{\sigma_{ci}}$$

The maximum value of a smaller principal stress σ'_{3max} is given by:

$$\frac{\sigma'_{3\max}}{\sigma'_c} = 0{,}72\left(\frac{\sigma'_c}{\gamma\,H}\right)^{-0{,}91}$$

where:
- γ - unit weight of the rock
- H - height of the rock slope
- σ_c - strength of the intact rock in simple compression, or uniaxial compressive strength of intact rock mass

Literature:

Stability analysis of rock slopes with a modified Hoek-Brown failure criterion, ANG Xiao-Li ; LIANG LI ; YIN Jian-Hua International journal for numerical and analytical methods in geomechanics. ISSN 0363-9061, 2004, vol. 28, no2, pp. 181-190.

Hoek E, Carranza-Torres CT, Corkum B.: Hoek–Brown failure criterion—2002 edition. Proceedings of the 5th North American Rock Mechanics Symposium, Toronto, Canada, vol. 1, 2002, pp. 267 – 273.

Parameters Hoek - Brown

Parameter of rock breaking a

Parameter a is an exponent receiving values from 0.5 to 0.65 (for the original Hoek-Brown condition it is equal to 0.5) and depends on the degree of rock breaking.

Nonlinear parameters $m_b = m$, s for $a =$ 0.5

(index r denotes residual values)

	Carbonate rocks with well-developed cleavage - dolomite, limestone, marble	**Argillaceous rocks - mudstone, siltstone, shale, slate**	**Arenaceous rocks - sandstone, quartzite**	**Fine-grained igneous crystalline rocks - andesite, dolerite, basalt, rhyolite**	**Coarse metamorphic and igneous rocks - gabbro, gneiss, granite**
Intact rock material, Laboratory specimens have no discontinuities, $RMR =$ 100, $Q =$ 500	$m =$ 7.00 $s =$ 1.00 $m_r =$ 7.00 $s_r =$ 1.00	$m =$ 10.00 $s =$ 1.00 $m_r =$ 10.00 $s_r =$ 1.00	$m =$ 15.00 $s =$ 1.00 $m_r =$ 15.00 $s =$ 1.00	$m =$ 17.00 $s =$ 1.00 $m_r =$ 17.00 $s =$ 1.00	$m =$ 25.00 $s =$ 1.00 $m_r =$ 25.00 $s =$ 1.00
Very good quality rock mass, Rocks without isolated blocks with non-weathered discontinuities, $RMR =$ 85, $Q =$ 100	$m =$ 2.40 $s =$ 0.082 $m_r =$ 4.10 $s_r =$ 0.189	$m =$ 3.43 $s =$ 0.082 $m_r =$ 5.85 $s_r =$ 0.189	$m =$ 5.14 $s =$ 0.082 $m_r =$ 8.78 $s_r =$ 0.189	$m =$ 5.82 $s =$ 0.082 $m_r =$ 9.95 $s_r =$ 0.189	$m =$ 8.56 $s =$ 0.082 $m_r =$ 14.63 $s_r =$ 0.189

Good quality rock mass, Slightly damaged rocks with non-weathered discontinuities spaced from 1 to 3 m, RMR = 65, Q = 10	m = 0.575 s = 0.00293 m_r = 2.006 s_r = 0.0205	m = 0.821 s = 0.00293 m_r = 2.865 s_r = 0.0205	m = 1.231 s = 0.00293 m_r = 4.298 s_r = 0.0205	m = 1.395 s = 0.00293 m_r = 4.871 s_r = 0.0205	m = 2.052 s = 0.00293 m_r = 7.163 s_r = 0.0205
Fair quality rock mass, Partially weathered discontinuities spaced from 0.3 to 1 m, RMR = 44, Q = 1	m = 0.128 s = 0.00009 m_r = 0.947 s_r = 0.00198	m = 0.183 s = 0.00009 m_r = 1.353 s_r = 0.00198	m = 0.275 s = 0.00009 m_r = 2.030 s_r = 0.00198	m = 0.311 s = 0.00009 m_r = 2.301 s_r = 0.00198	m = 0.458 s = 0.00009 m_r = 3.383 s_r = 0.00198
Poor quality rock mass, Numerous weathered discontinuities spaced from 30 to 500 mm, RMR = 23, Q = 0.1	m = 0.029 s = 0.000003 m_r = 0.447 s_r = 0.00019	m = 0.041 s = 0.000003 m_r = 0.639 s_r = 0.00019	m = 0.061 s = 0.000003 m_r = 0.959 s_r = 0.00019	m = 0.069 s = 0.000003 m_r = 1.087 s_r = 0.00019	m = 0.102 s = 0.000003 m_r = 1.598 s_r = 0.00019
Very poor quality rock mass, Numerous extremely weathered discontinuities with filling spaced by less than 50 mm, fine-grained waste rock, RMR = 3, Q = 0.01	m = 0.007 s = 0.0000001 m_r = 0.219 s_r = 0.00002	m = 0.010 s = 0.0000001 m_r = 0.313 s_r = 0.00002	m = 0.015 s = 0.0000001 m_r = 0.469 s_r = 0.00002	m = 0.017 s = 0.0000001 m_r = 0.532 s_r = 0.00002	m = 0.025 s = 0.0000001 m_r = 0.782 s_r = 0.00002

Strength of rocks in simple compression σc, Poisson's ratio ν and unit weight of rock γ

Rock strength	**Types of rock (examples)**	**Strength** σ_c $[MPa]$	**Poisson's ratio** ν	**Bulk weight of rock** γ $[kN/m^3]$
Solid rock	most hard solid rock, intact, compact and dense quartz rock and basalt, other extraordinary hard rocks	>150	0.10	28.00 - 30.00
Highly hard rock	very hard granite rock, quartz porphyry, very hard granite, hard flinty shale, quartzite, very hard sand rock, and very hard calcite	100 - 150	0.15	26.00 - 27.00
Hard rock	granite, very hard sandstone and calcite, quartzite lode, hard conglomerate, very hard ore, hard limestone, marble, dolomite, pyrite	80 - 100	0.20	25.00 - 26.00
Rock	sandstone, ore, medium sandy shale, flagstone	50 - 80	0.25	24.00
Medium rock	hard mudstone, softer sand rock and calcite, chalky clay	20 - 50	0.25 - 0.30	23.00 - 24.00
Soft rock	shale, soft limestone, chalk, salt rock, frozen ground, anthracite, marl, remoulded sandstone, soft conglomerate, ground with fels	5 - 20	0.3 - 0.35	22.00 - 26.00
Weak soil	compact clay, soil eluvium, black coal	0.5 - 5	0.35 - 0.40	20.00 - 22.0 18.00 - 2

Calculation of Hoek-Brown Parameters

If rock mass is descripted using GSI (Geological Structure Index) is known then it is possible to let the program to determine the H-B parameters as follows:

$$m_b = m_i . e^{(GSI-100/28-14.D)}$$

$$s = e^{(GSI-100/9-3.D)}$$

$$a = \frac{1}{2} + \frac{1}{6}\left(e^{(-GSI/15)} - e^{(-20/3)}\right)$$

where: GSI - Geological Structure Index

D - damage coefficient of rock mass

m_i - strength material constant of the intact rock for peak conditions

Values of damage coefficient D for rock slope

Description of rock mass	**Suggested value of coefficient D**
Small scale blasting in engineering slopes results in modest rock mass damage, particularly if controlled blasting is used. However, stress relief results in some disturbance. (Good blasting).	0.7
Small scale blasting in engineering slopes results in modest rock mass damage, particularly if controlled blasting is used. However, stress relief results in some disturbance. (Poor blasting).	1
Very large open-pit mine slopes significant disturbance due to heavy production blasting and due to stress relief from overburden removal. (Production blasting).	1
In some softer rocks excavation can be carried out by ripping and dozing and the degree of damage to the slope is less. (Mechanical excavation).	0.7

Approximate values of strength material constant of the intact rock $m\ i$ (after Hoek)

Type of rock	**Representative rocks**	$m\ i$ [-]
Limestone rocks with well developed crystalline cleavage	Dolomite, calcite, marble	≈ 7
Consolidated clayey rocks	Mudstone, siltstone, silty shale, slate	≈ 10
Sandy rocks with solid crystals and poorly developed crystalline cleavage	Sandstone, quartzite	≈ 15
Fine-grained igneous crystalline rocks	Andesite, dolerite, diabase, rhyolite	≈ 17
Coarse-grained and metamorphic rocks	Amphibolite, gabbro, gneiss, granite, diorite	≈ 25

Barton - Bandis

The Barton-Bandis shear strength failure criterion for the rock mass takes the following form:

$$\tau = \sigma_n . tg\left[\varphi_b + JRC.log_{10}\left(\frac{JCS}{\sigma_n}\right)\right]$$

where: JRC - joint roughness coefficient

σ_n - normal stress acting on the surface of the rock joint

JCS - joint compressive strength

φ_b - basic angle of internal friction of the slip surface

If possible the shear strength parameters should be determined from in-situ measurements. Approximate ranges of parameters of the Barton-Bandis failure criterion are given here.

Barton - Bandis Parameters

Joint roughness coefficient JRC

If the value of JRC cannot be determined by direct measurements on the joint surface, it is possible to obtain this value from the Barton graph (see figure) showing the variation of the coefficient JRC as a function of the length of profile and roughness depth.

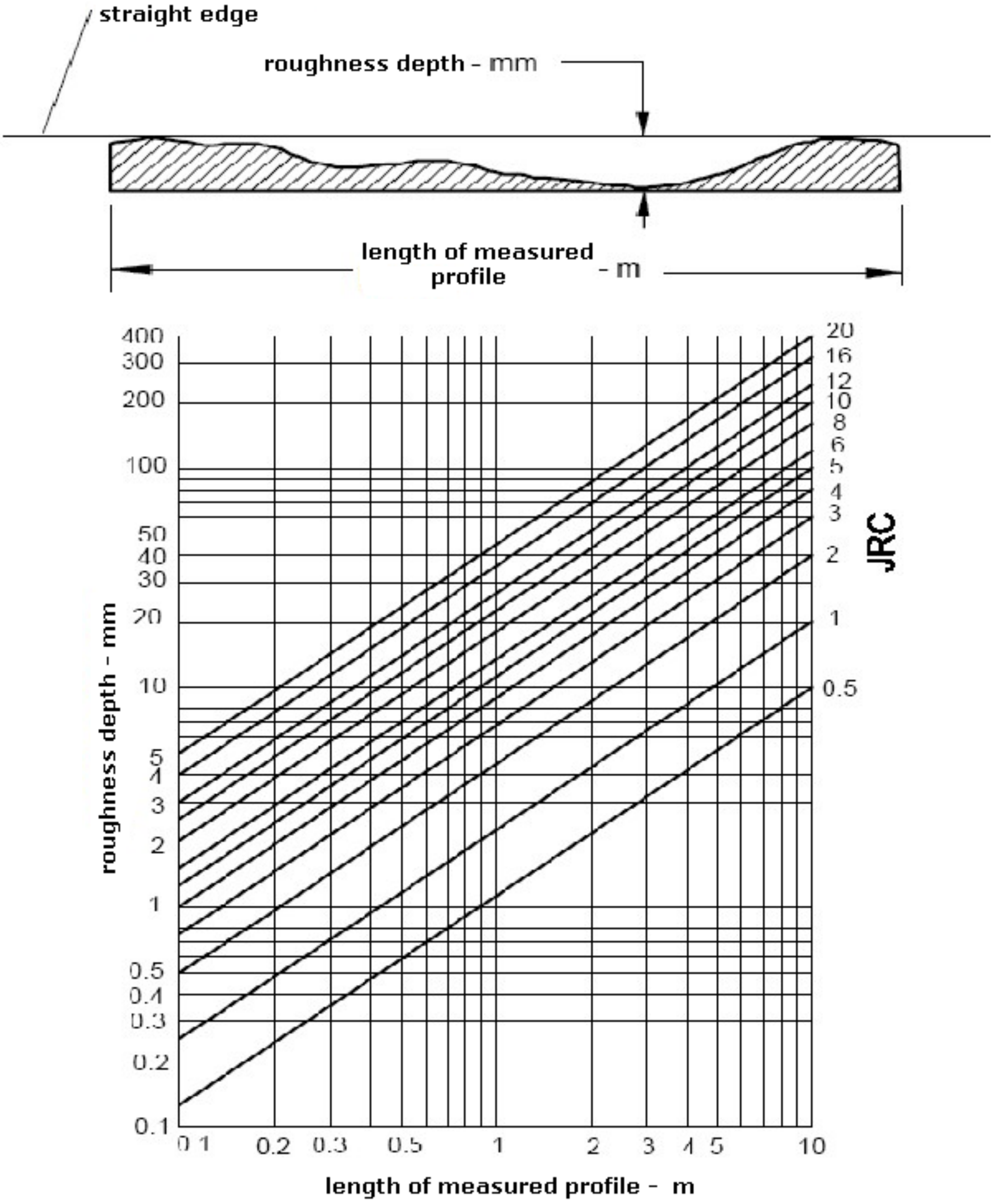
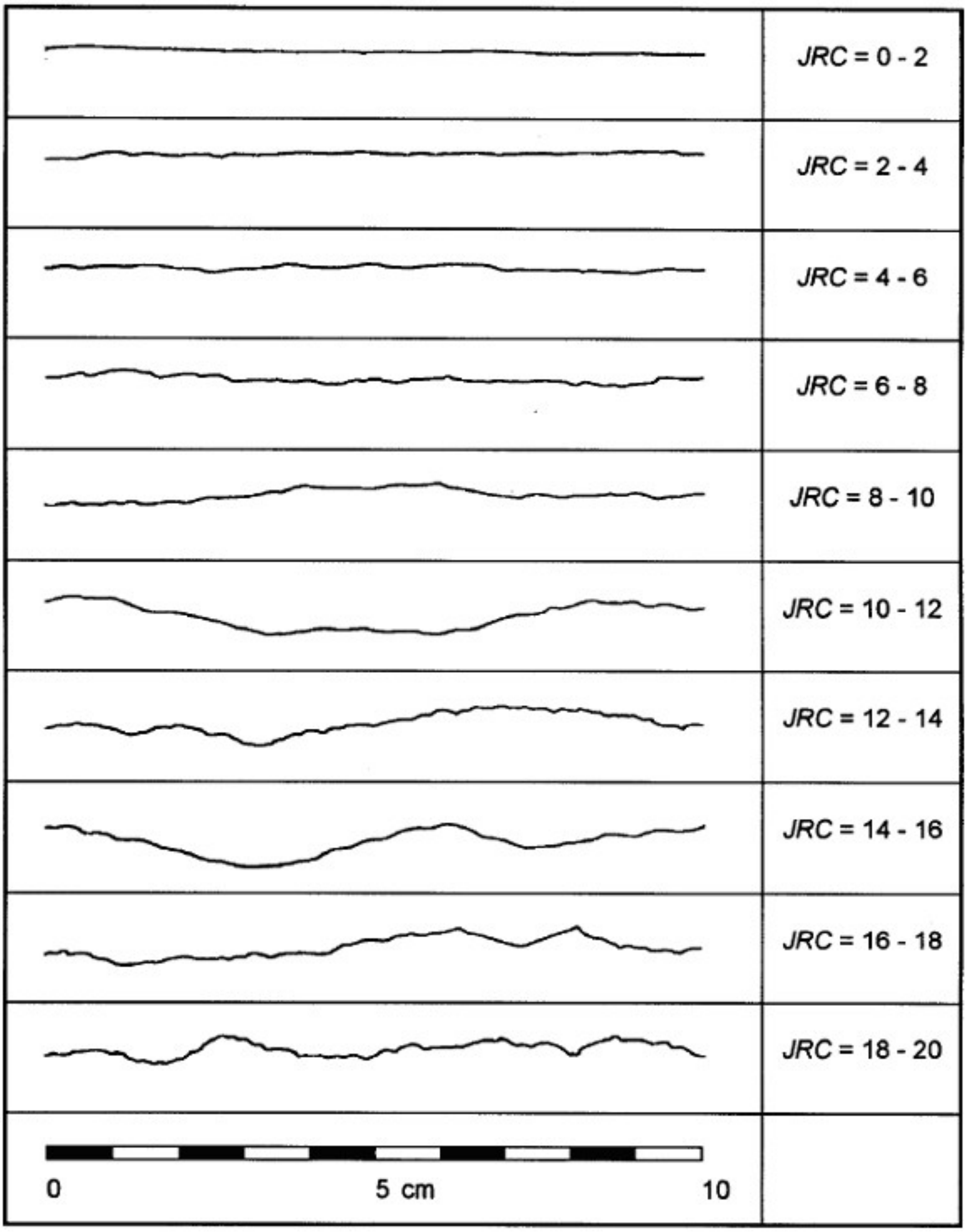


Diagram to determine JRC (after Barton)

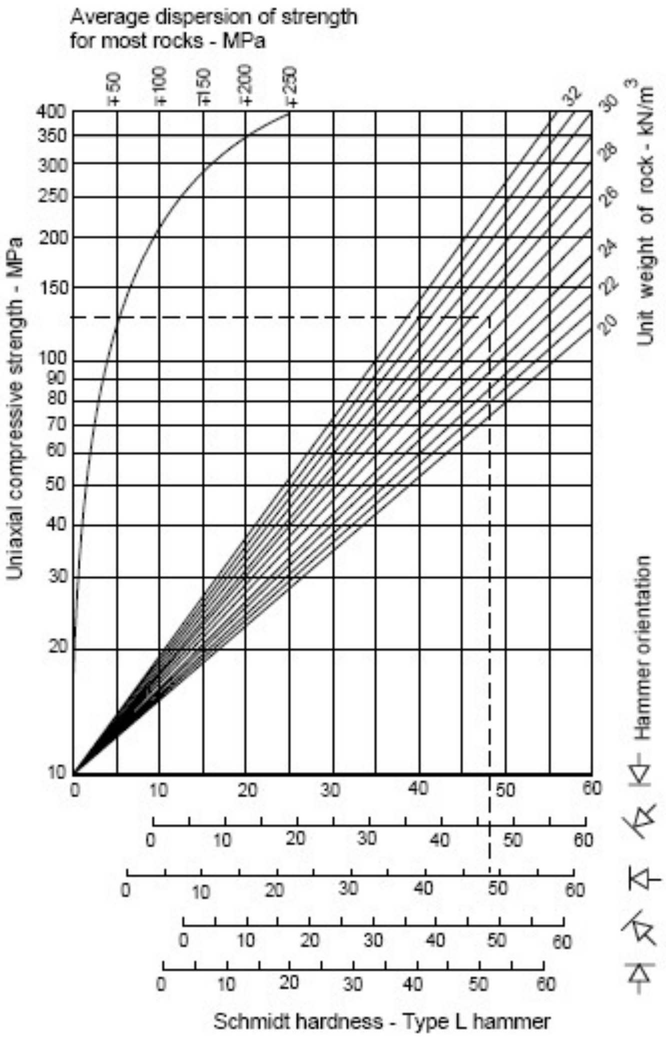
Rock joint roughness profiles showing the typical range of JRC are plotted next.



Rock joint roughness profiles showing the typical range of JRC (Barton & Chubey 1977)

Compressive strength of discontinuity JCS

Methods allowing us to determine the compressive strength of discontinuity (slip surface) JCS are generally recommended by ISRM. The value of JCS can be obtained from the Deere-Miller graph showing its dependence on the rock strength found from the Schmidt hammer measurements, see figure below.



Basic angle of internal friction on slip surface φ_b

The basic value of the angle of internal friction on the surface is approximately equal to the residual value φ_r . Nevertheless, it can be generally measured in laboratories using shear measurement devices (typical area of the specimen is 50$*$50 mm). Typical ranges of the basic angle of internal friction for weathered rock surfaces are 25° to 35° .

Unit Weight of Rocks

Unit weight or rock γ

Rock strength	Rock category (examples)	Unit weight of rock γ $[kN/m^3]$
Solid rock	most hard solid rock, intact, compact and dense quartz rock and basalt, other extraordinary hard rocks	28.0 - 30.0
Highly hard rock	very hard granite rock, quartz porphyry, very hard granite, hard flinty shale, quartzite, very hard sand rock, and very hard calcite	26.0 - 27.0
Hard rock	granite, very hard sandstone and calcite, quartzite lode, hard conglomerate, very hard ore, hard limestone, marble, dolomite, pyrite	25.0 - 26.0
Rock	sandstone, ore, medium sandy shale, flagstone	24.0
Medium rock	hard mudstone, softer sand rock, and calcite, chalky clay	23.0 - 24.0

Soft rock	shale, soft limestone, chalk, salt rock, frozen ground, anthracite, marl, remolded sandstone, soft conglomerate, ground with fels	22.0 - 26.0
Weak soil	compact clay, soil eluvium, black coal	20.0 - 22.0 18.0 - 20.0

Influence of Seismic Effects

The programs takes into account the earthquake effects using two variables - factor of horizontal acceleration K_h and factor of vertical acceleration K_v.

The factor of acceleration is a dimensionless number, which represents the seismic acceleration as a fraction of the gravity acceleration. Earthquake effects are introduced through the seismic force S, which is determined by multiplying the weight of the rock subjected to the earthquake (i.e. rock block) by the factor of acceleration. When assuming seismic effects only in the horizontal direction the seismic force is given by:

$$S = K_h . W$$

where: K_h - factor of the horizontal acceleration

W - weight of the rock body

The seismic force always acts in the center of gravity of the rock body. Usually, only seismic effects in the horizontal direction are considered. Nevertheless, the program also allows for treating the vertical direction (with the help of vertical factor of acceleration K_v). The effects in both directions are then combined.

M_C_S grade	**Horizontal acceleration**	**Factor of horizontal acceleration**
((MSK-64)	$[mm/s^2]$	K_h
1	0.0 - 2.5	0.0 - 0.00025
2	2.5 - 5.0	0.00025 - 0.0005
3	5.0 - 10.0	0.0005 - 0.001
4	10.0 - 25.0	0.001 - 0.0025
5	25.0 -50.0	0.0025 - 0.005
6	50.0 - 100.0	0.005 - 0.01
7	100.0 - 250.0	0.01 - 0.025
8	250.0 - 500.0	0.025 - 0.05
9	500.0 -1000.0	0.05 -0.1
10	1000.0 -2500.0	0.1 -0.25
11	2500.0 -5000.0	0.25 -0.5
12	> 5000.0	> 0.5

The values of factor K_h correspond to individual degrees of earthquake according to M-C-S scale

MSE Wall

The program allows the following verifications:

Verification

The program verifies the external stability of so-called **fictitious structure** consisting of the structure front face and a curve bounding the geo-reinforcements endpoints. The fictitious structure is loaded by calculated forces acting on the structure and checked for **overturning** and **slip** - similarly to the verification of a gravity wall.

Dimensioning

From the calculated forces acting on a structure, the program determines the forces in the checker cross-section. Only the forces above the checked joint (see figure) are taken into account. Reinforcements introduce stabilizing forces which equal to the lower value of the two bearing capacities (against tearing and pull-out). The actual verification for **overturning** and **slip** follows afterwords. The program also allows an automatic verification of the most critical cross-section.

Bearing capacity

The bearing capacity of foundation soil below a **fictitious structure** is verified. The constant stress in the foundation joint is determined from all forces acting on a structure and calculated in the "Verification" frame. In the case of the input foundation the bearing capacity is determined from all forces calculated in the "Dimensioning" frame (the option "**Entire wall**" must be selected).

Slip on georeinforcement

The slip of a **reinforced soil block** along a geo-reinforcement is verified. The reinforced block is bounded the wall front face, the checked geo-reinforcement, a vertical line passing through the geo-reinforcement endpoint and terrain. The block is loaded by an active earth pressure and by stabilizing forces due to geo-reinforcements exceeding the boundary of the reinforced block and by other forces. The program also allows us to automatically verify the sliding along individual reinforcements and find the most critical result.

Internal stability

The strength of reinforcement and pull-out resistance are checked.

Global stability

The program allows us to verify the overall slope stability along a circular slip surface. The slip surface can be automatically optimized, i.e. the program automatically selects the verification along the most critical surface. The actual slope stability analysis can be carried out with the help of two slice methods: Spencer (rigorous, more accurate method) and Bishop (more conservative, simpler, more easily finds the solution satisfying equilibrium conditions).

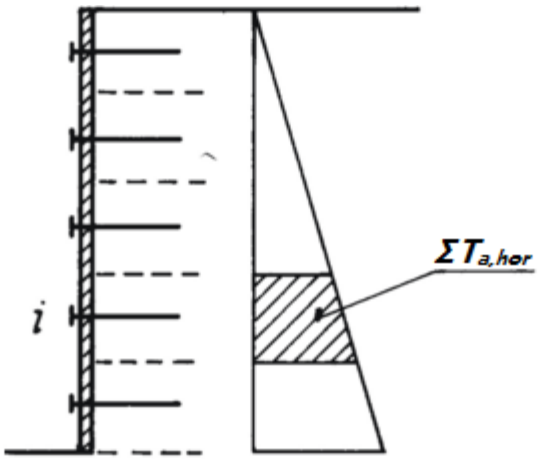
Slope stability

Verification of overall (global) stability by the "**Slope stability**" program.

Internal Stability

Geo-reinforcement force

Determination of forces in geo-reinforcements is performed by splitting and assigning the calculated earth pressure to individual layers. Each reinforcement accommodates part of the active pressure, which acts in the **corresponding layer**, i.e. force developed in the reinforcement $F_x = \Sigma T_{a,hor}$.



Forces transmitted by individual reinforcements

Shape of the slip surface depends on the selected standard of calculation.

Analysis of internal stability varies by type of geo-reinforcements:

- extensible reinforcements (Standard - straight slip surface, AASHTO - Extensible, FHWA NHI-10-024)
- inextensible reinforcements (AASHTO - Inextensible, JTGD30 - 2004 Highway China Code, TB 10025 Railway China Code, BS 8006 - Coherent Gravity Method)

Earth pressure is considered active for extensible reinforcements, or as a combination of pressures for inextensible reinforcements.

Reinforcement strength check

Long-term design strength of geo-reinforcement R_t is calculated from the input parameters of the geo-reinforcement:

$$R_t = \frac{T_{ult}}{RF_{CR} \cdot RF_D \cdot RF_{ID} \cdot FS_{UNC}}$$

where:
- R_t - long-term design strength of reinforcement
- T_{ult} - short-term characteristic strength of geo-reinforcement
- RF_{CR} - reduction coefficient of long-term deformation of reinforcement (determined based on geo-reinforcement lifetime)
- RF_D - reduction coefficient of the durability of reinforcement (determined based on soil pH)

RF_{ID} - reduction coefficient of failure of reinforcement when inserting into the soil (determined based on the grain sizes of soil)

FS_{UNC} - overall coefficient of model uncertainty

Note: The FS_{UNC} coefficient is not taken into account when calculating the **reinforcement strength** in the Redi-Rock Wall program.

Reinforcement bearing capacity against pull-out

The resistance of reinforcement against pull-out from the earth body is calculated from the input parameters of the geo-reinforcement and the normal force acting in the direction normal to its area:

$$T_p = 2 \cdot L \cdot C_i \cdot \sigma_z \cdot \tan\varphi$$

where:

T_p - bearing capacity against tearing

L - reinforcement length (from front face to its end)

C_i - coefficient of interaction between soil and geo-reinforcement

σ_z - vertical geostatic stress

φ - angle of internal friction of soil

Note: In the Redi-Rock Wall program, the **pull-out resistance** calculation takes into account the **scale correction factor** α .

Verification of bearing capacity of reinforcement against pull-out can be carried out according to the factor of safety or the theory of limit states.

Verification - Safety Factor

The main advantage of this verification is lucidity and uniqueness since neither soil parameters nor acting forces are reduced.

Check for tearing:

$$\frac{R_t}{F_x} > SF_{st}$$

where:

F_x - force developed in reinforcement

R_t - long-term design strength of reinforcement

SF_{st} - safety factor for geo-reinforcement strength (input in the "Wall analysis" tab)

Check for pull-out:

$$\frac{T_p}{F_x} > SF_{po}$$

where:

F_x - force developed in reinforcement

T_p - bearing capacity of reinforcement against pull-out

SF_{po} - safety factor for pull out resistance of geo-reinforcement (input in the frame "Wall analysis" tab)

Verification - Limit States

The soil parameters are **reduced** depending on the setting in the "Wall analysis" tab. The result is reinforcement utilization compared with *100 %* .

Check for tearing:

$$\frac{F_x}{R_t} \cdot 100 < 100\%$$

where:

F_x - force developed in reinforcement

R_t - long-term design strength of reinforcement against tearing

Check for pull-out:

$$\frac{F_x}{T_p} \cdot 100 < 100\%$$

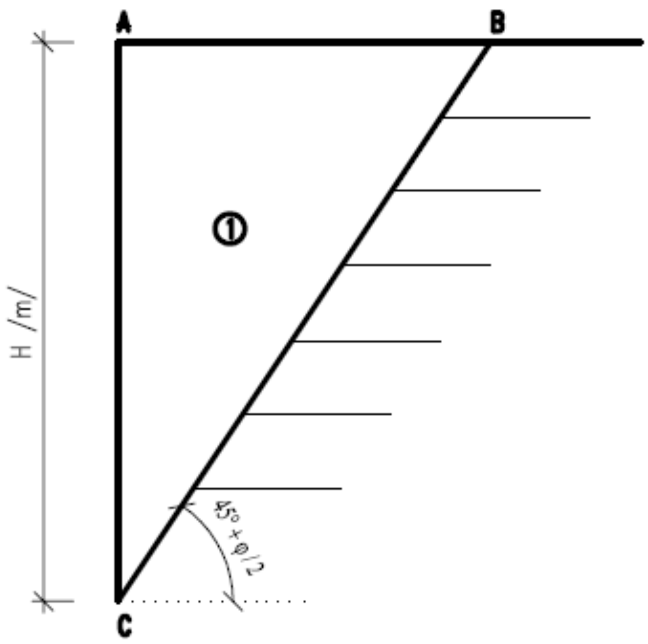
where: F_x - force developed in reinforcement

T_p - bearing capacity of reinforcement against pull-out

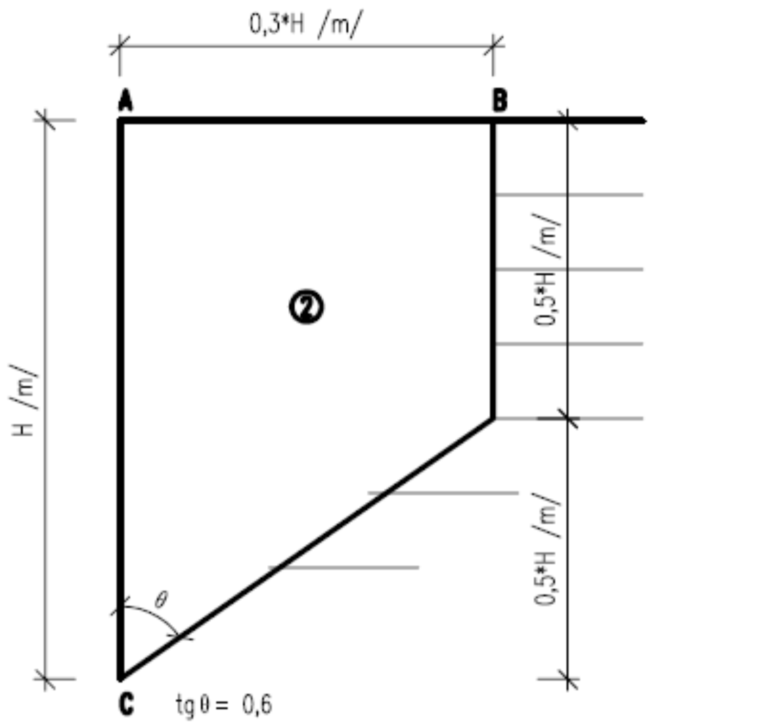
Shapes of Slip Surfaces

Slip surface has a different shape according to the selected standard of calculation:

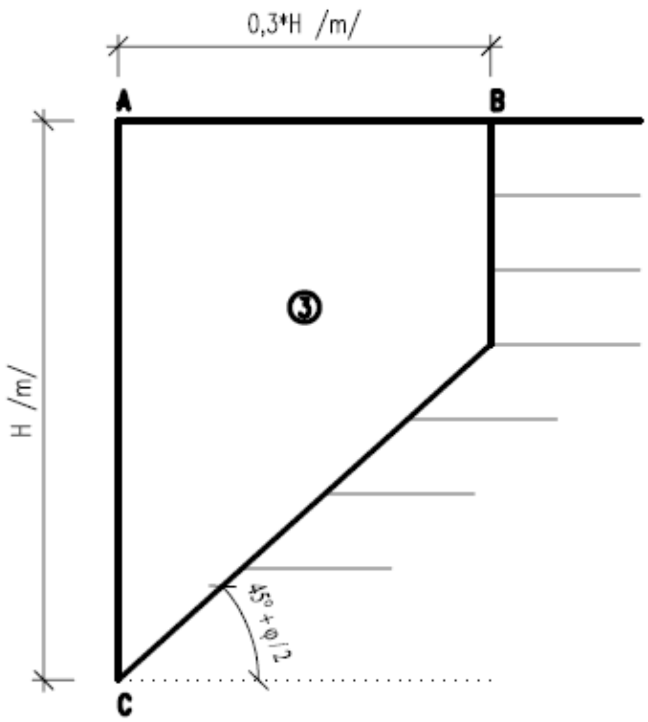
- **Straight slip surface** - Standard, AASHTO - Extensible, FHWA NHI-10-024
- **Broken slip surface** - AASHTO - Inextensible, JTGD30 - 2004 Highway China Code, TB 10025 Railway China Code, BS 8006 - Coherent Gravity Method.



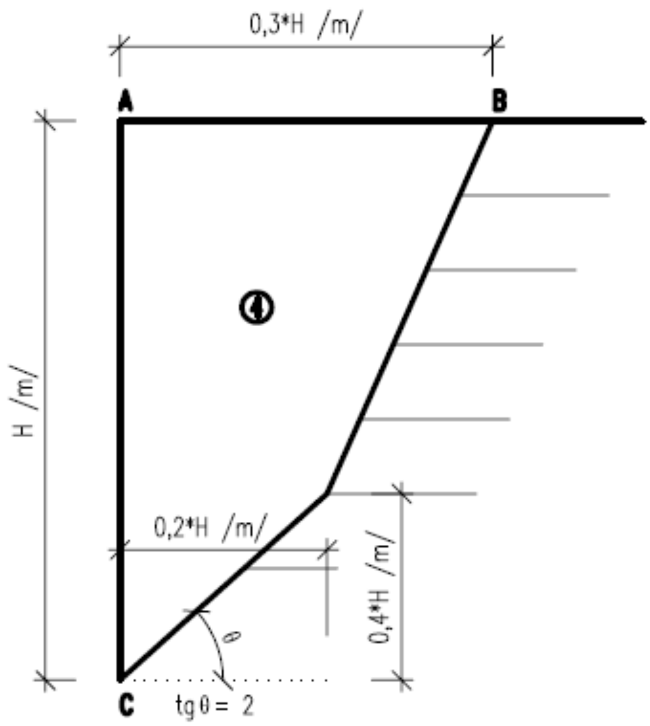
① STANDARD, AASHTO – EXTENSIBLE, FHWA NHI–10–024



② TB 10025 – 2006 RAILWAY CHINA CODE, AASHTO – INEXTENSIBLE



③ JTGD30 – 2004 HIGHWAY CHINA CODE



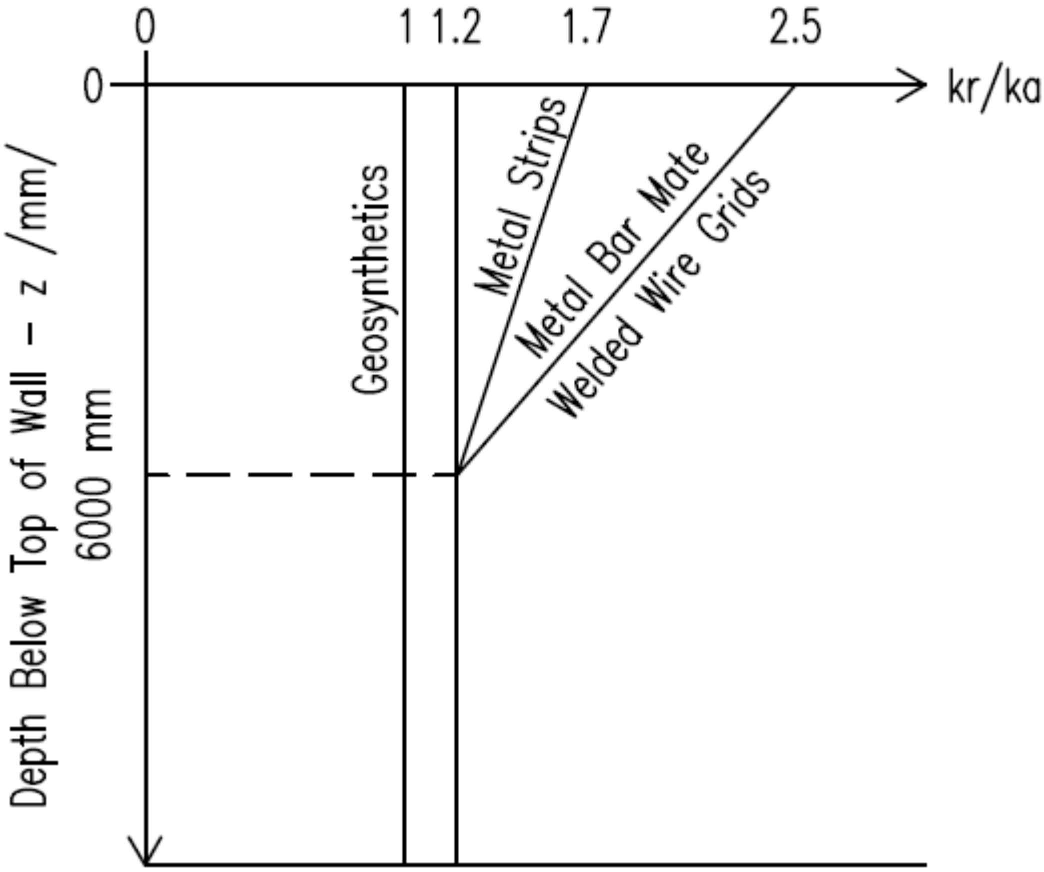
④ BS 8006 – COHERENT GRAVITY METHOD

Shapes of slip surfaces according to the individual standards of calculation

Extensible Reinforcements - Active Earth Pressure

Active earth pressure is considered in the calculation of internal stability for extensible reinforcements (Standard - straight slip surface, AASHTO - Extensible, FHWA NHI-10-024).

The program allows us to multiply the calculated earth pressure by coefficient k_r / k_a (according to the AASHTO standards). Recommended values are shown in the following picture.



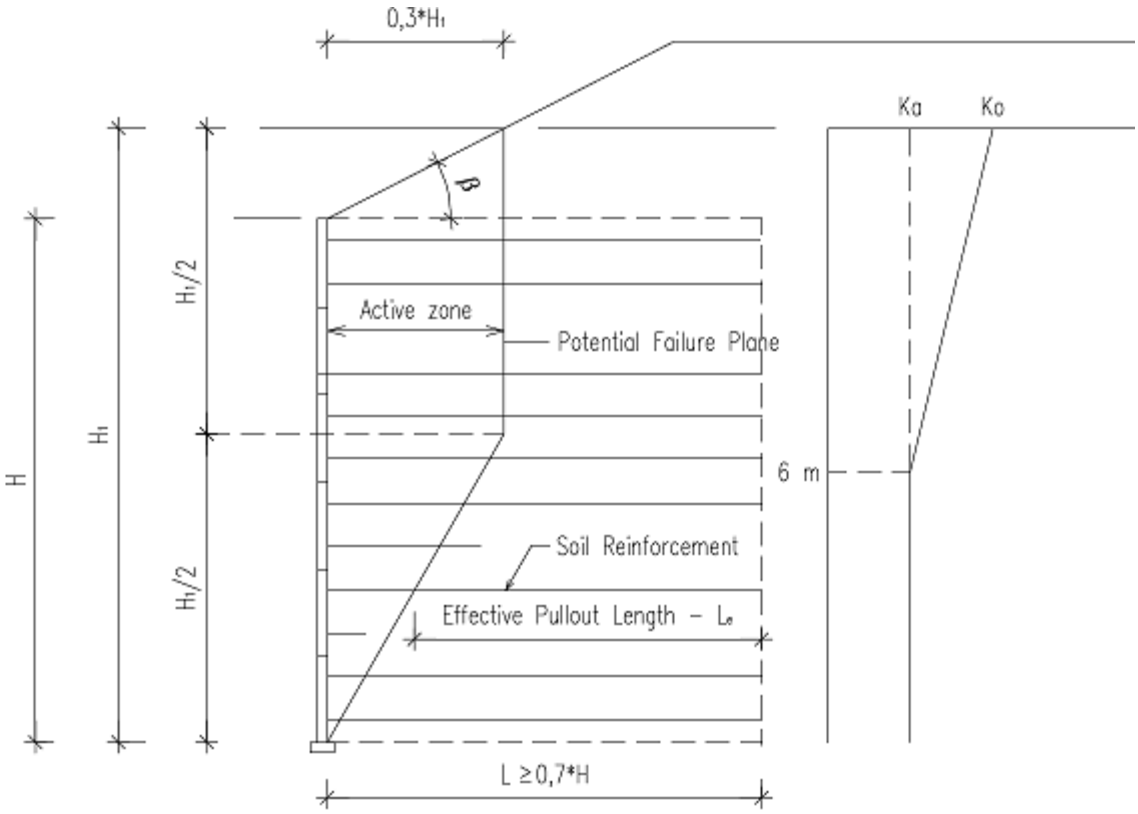
Variation of the coefficient of horizontal stress ratio k_r/k_a with depth for extensible reinforcements

Literature:

AASHTO LRFD Bridge Design Specifications 2004 (SI).

Inextensible Reinforcements - Combination of Earth Pressures

The combination of active earth pressure and earth pressure at rest is considered in the calculation of internal stability for inextensible reinforcements (AASHTO - Inextensible, JTGD30 - 2004 Highway China Code, TB 10025 Railway China Code, BS 8006 - Coherent Gravity Method).



Determination of failure plane location and variation of earth pressure coefficients K_a, K_o with depth for inextensible reinforcements

Literature:

AASHTO Highway Bridges.

Foundation Bearing Capacity

The vertical bearing capacity of the foundation soil is verified according to the theory of limit states using the following inequality:

$$\sigma \leq \frac{R_d}{\gamma_{RV}}$$

or based on the factor of safety as:

$$\frac{R_d}{\sigma} \geq SF_v$$

where:
- σ - extreme design contact stress at the footing bottom
- R_d - design bearing capacity of foundation soil
- γ_{RV} - coefficient of vertical bearing capacity of foundation (for input use the "Spread Footing" tab)
- SF_v - safety factor for vertical bearing capacity

Extreme design contact stress at the footing bottom is given by:

$$\sigma = \frac{V}{A_{ef}}$$

where:
- V - extreme design vertical force
- A_{ef} - effective area of the foundation

The vertical bearing capacity of the foundation soil R_d is determined for three basic types of foundation conditions:

- Drained subsoil
- Undrained subsoil
- Bedrock

The above computations are applicable only for the homogeneous soil. If there is a **non-homogeneous** soil under the footing bottom (or there is groundwater present), then the inserted profile is transformed into a homogeneous one.

Bearing Capacity on Drained Subsoil

For foundation in drained condition, it is possible to select one of the followings approaches to assess the bearing capacity of the foundation:

- standard analysis
- according to CSN 73 1001 "Základová pùda pod plošnými základy" approved 8.6. 1987
- according to Polish standard PN-81 B - 03020 "Grunty budowiane, Posudowienie bezpošrednie budowli, Obliczenia statyczne i projekktowanie" from year 1982
- according to Indian standard IS:6403-1981 "Code of Practice for Determination of Bearing Capacity of Shallow Foundations" from the year 1981
- according to EC 7-1 (EN 1997-1:2003) "Design of geotechnical structures - Part 1: General rules"
- according to NCMA Segmental retaining walls manual, second edition
- according to Chinese standard GB 50007-2002
- according to Russian standard SNiP 2.02.01-83
- according to Danish standard DS/EN 1997-1 DK NA:2013
- according to DPWH standard "Design Guidelines, Criteria and Standards: Volume 2C – Geological and Geotechnical Investigation - Annex G: Geotechnical Formulas"
- according to Meyerhof
- according to Vesic
- according to German standard DIN 4017
- according to Spanish standard CTE DB SE-C
- according to NZ standard B1/VM4
- according to Russian standard SP22.13330.2016

All approaches (with exception of German standard DIN 4017 and Spanish standard CTE DB SE-C) incorporate coefficients due to Brinch - Hansen (see standard analysis) to account for inclined ground surface and inclined footing bottom.

Assuming drained conditions during construction the soil below spread footing deforms including both shear and volumetric deformations. In such a case the strength of the soil is assumed in terms of effective values of the angle of internal friction φ_{ef} and the effective cohesion c_{ef}. It is also assumed that there is effective stress in the soil equal to the total stress (consolidated state). Effective parameters φ_{ef}, c_{ef} represent the peak strength parameters.

Owing to the fact that the choice of drained conditions depends on a number of factors (rate of load, soil permeability, degree of saturation and degree of overconsolidation) it is the designer's responsibility to decide, depending on the actual problem being solved, if the effective parameters should be used.

During **seismic analysis**, the program performs the calculation of bearing capacity for both cases - with and without seismic effect. The resultant bearing capacity is lower from these two values.

Analysis of Seismic Bearing Capacity

Seismic bearing capacity of shallow foundation is calculated according to the following formula:

$$p_{LE} = c.N_{cE} + \gamma.d.N_{qE} + \frac{1}{2}.\gamma.B.N_{\gamma E}$$

where:	p_{LE}	-	seismic bearing capacity
	c	-	cohesion of soil
	γ		unit weight of soil
	d		depth of the footing bottom
	B		width of foundation
	N_{CE}, N_{qE}, $N_{\gamma E}$	-	seismic bearing factors

$$N_{cE} = (N_{qE} - 1).\cot\varphi$$

$$N_{qE} = \frac{K_{PE}}{K_{AE}}$$

$$N_{\gamma E} = \tan\rho_{AE}.(\frac{K_{PE}}{K_{AE}} - 1)$$

where: K_{AE}, K_{PE} - seismic coefficients of earth pressures

ρ_{AE} - angle of failure

φ angle of internal friction

$$K_{AE} = \frac{(\cos\alpha)^2}{\cos\theta.\cos(\delta+\theta).(1+\sqrt{\frac{\sin(\varphi+\delta).\sin\alpha}{\cos(\delta+\theta)}})^2}$$

$$K_{PE} = \frac{(\cos\alpha)^2}{\cos\theta.\cos(\delta+\theta).(1-\sqrt{\frac{\sin(\varphi+\delta).\sin\alpha}{\cos(\delta+\theta)}})^2}$$

$$\rho_{AE} = \alpha + \tan^{-1}(\frac{\sqrt{[1+(\tan\alpha)^2].[1+\tan(\delta+\theta).\cot\alpha]}-\tan\alpha}{1+\tan(\delta+\theta).(\tan\alpha+\cot\alpha)})$$

$$\theta = \tan^{-1}\frac{k_h}{1-k_v}$$

$$\alpha = \varphi - \theta$$

where: k_h - seismic coefficient of horizontal acceleration

k_v - seismic coefficient of vertical acceleration

Note: If the angle α is negative, i.e. $\varphi < \theta$, then the seismic bearing capacity is $p_{LE} = 0$.

References:

Richards Jr., R.; et al. SEISMIC BEARING CAPACITY AND SETTLEMENTS OF FOUNDATION. J.Geotech. Engrg. 1993, 4 (119)

Das, B. M. Principles of Soil Dynamics, 3rd ed.; California State University: Sacramento, 2016.

Standard Analysis

By default, the solution based on J. Brinch - Hansen theory is used, where the bearing capacity of foundation soil is:

$$R_d = c.N_e.s_e.d_e.i_e.b_e.g_e + q_0.N_d.s_d.d_d.i_d.b_d.g_d + \frac{b}{2}.\gamma.N_b.s_b.d_b.i_b.b_b.g_b$$

where:

$$q_0 = \gamma_1.d$$

bearing capacity factors:

$$N_e = (N_d - 1).cotg\varphi \quad \text{for: } \varphi > 0$$

$$N_e = 2 + \pi \quad \text{for: } \varphi = 0$$

$$N_d = tg^2\left(45 + \frac{\varphi}{2}\right).e^{\pi.tg\varphi}$$

$$N_b = 1{,}5(N_d - 1).tg\varphi$$

shape factors:

$$s_e = 1 + 0{,}2.\frac{b}{l}$$

$$s_d = 1 + \frac{b}{l}.sin\varphi$$

$$s_b = 1 - 0{,}3.\frac{b}{l}$$

depth factors:

$$d_e = 1 + 0{,}1.\sqrt{\frac{d}{b}}$$

$$d_d = 1 + 0{,}1.\sqrt{\frac{d}{b}.sin2\varphi}$$

$$d_b = 1$$

inclination factors:

$$i_e = i_d = i_b = (1 - tg\delta)^2$$

base factors:

$$b_e = b_d - \frac{(1 - b_d)}{N_e}.tan\varphi$$

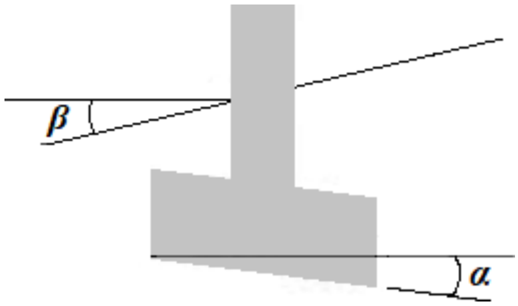
$$b_d = (1 - \alpha.tan\varphi)^2$$

$$b_b = b_d$$

ground factors:

$$g_e = 1 - \frac{2.\beta}{\pi + 2}$$

$$g_d = g_b = (1 - 0{,}5.tg\beta)^5$$



Notation of angles for coefficients b , g

where:	c	-	cohesion of soil
	q_0	-	equivalent uniform load accounting for the influence of foundation depth
	d	-	depth of the footing bottom
	γ_1	-	unit weight of soil above the footing bottom
	b	-	width of foundation
	γ	-	unit weight of soil
	N_e , N_d , N_b	-	coefficient of bearing capacity
	s_e , s_d , s_b	-	coefficients of shape of the foundation
	d_e , d_d , d_b	-	coefficients of influence of foundation depth
	i_e , i_d , i_b	-	coefficients of influence of slope of the load
	b_e , b_d , b_b	-	coefficients of influence of base slope
	g_e , g_d , g_b	-	coefficients of influence of slope of the terrain
	φ	-	angle of internal friction of soil
	l	-	length of foundation
	δ	-	angle of deviation of the resultant force from the vertical direction
	β	-	slope of terrain
	α	-	slope of footing bot

Meyerhof

This solution is based on the theory of G. G. Meyerhof. The calculation is the same as for standard approach, except the following coefficients:

bearing capacity factors:

$$N_b = (N_d - 1).\tan(1{,}4.\varphi)$$

shape factors:

$$s_e = 1 + 0{,}2.K_p.\frac{b}{l}$$

for: $\varphi \leq 10°$

$$s_d = 1$$

for: $\varphi > 10°$

$$s_d = 1 + 0{,}1.K_p.\frac{b}{l}$$

$$s_b = s_d$$

where:

$$K_p = \tan^2(45 + \frac{\varphi}{2})$$

depth factors:

for: $\varphi = 0°$

$$d_e = 1 + 0{,}2.\frac{d}{b}$$

$$d_d = d_b = 1$$

for: $\varphi \neq 0°$

$$d_e = 1 + 0{,}2.\frac{d}{b}.\tan(45 + \frac{\varphi}{2})$$

$$d_d = d_b = 1 + 0{,}1.\frac{d}{b}.\tan(45 + \frac{\varphi}{2})$$

inclination factors:

$$i_e = i_d = (1 - \frac{2.\delta}{\pi})^2$$

$$i_b = (1 - \frac{\delta}{\varphi})^2$$

where:

d	-	depth of the footing bottom
b	-	width of the foundation
l	-	length of the foundation
φ	-	angle of internal friction of the soil
δ	-	angle of deviation of the resultant force from the vertical direction

Literature:

J.-G. Sieffert and Ch. Bay-Gress: Comparison of European bearing capacity calculation method for shallow foundations, Proc. Instn Civ. Engrs Geotech. Engng, 2000, 143, Apr., 65-74

Braja M. Das: Principles of Geotechnical Engineering, Sixth Edition, Thomson India 2006, ISBN: 978-8131502020

Vesic

This solution is based on the theory of A. S. Vesic. The calculation is the same as for standard approach, except the following coefficients:

bearing capacity factors:

$$N_b = 2.(N_d + 1).\tan\varphi$$

shape factors:

$$s_e = 1 + \frac{b}{l}.\frac{N_d}{N_e}$$

$$s_d = 1 + \frac{b}{l}.\tan\varphi$$

$$s_b = 1 - 0{,}4.\frac{b}{l}$$

depth factors:

for: $d/b \leq 1$

$$d_e = 1 + 0{,}4.\frac{d}{b}$$

$$d_d = 1 + 2.\tan\varphi\,.(1 - \sin\varphi).\frac{d}{b}$$

$$d_b = 1$$

for: ***d/b*** $>$ 1

$$d_e = 1 + 0{,}4.\tan^{-1}\frac{d}{b}$$

$$d_d = 1 + 2.\tan\varphi\,.(1 - \sin\varphi).\tan^{-1}\frac{d}{b}$$

$$d_b = 1$$

inclination factors:

for: $\varphi = 0°$

$$i_e = 1 - \frac{m.H}{A.a.N_c}$$

for: $\varphi \neq 0°$

$$i_e = \frac{i_d.N_d - 1}{N_d - 1}$$

$$i_d = (1 - \tan\theta)^m$$

$$i_b = (1 - \tan\theta)^{m+1}$$

where:

$$\tan\theta = \frac{H}{V + A.c.\cot\varphi}$$

$$m = \frac{2 + \frac{b}{l}}{1 + \frac{b}{l}}$$

where:

	d	-	depth of the footing bottom
	b	-	width of foundation
	l	-	length of foundation
	φ	-	angle of internal friction of soil
	c	-	cohesion of soil
	δ	-	angle of deviation of the resultant force from the vertical direction
	H	-	horizontal component of resultant load
	V	-	vertical component of resultant load

Literature:

J.-G. Sieffert and Ch. Bay-Gress: Comparison of European bearing capacity calculation method for shallow foundations, Proc. Instn Civ. Engrs Geotech. Engng, 2000, 143, Apr., 65-74

Braja M. Das: Principles of Geotechnical Engineering, Sixth Edition, Thomson India 2006, ISBN: 978-8131502020

ALEKSANDAR SEDMAK VESIC: ANÁLISIS DE LA CAPACIDAD DE CARGA DE CIMENTACIONES SUPERFICIALES, UNIVERSIDAD NACIONAL AUTÓNOMA DE MÉXICO, 1973

DIN 4017

The solution is based on German standard DIN 4017, where the bearing capacity of foundation soil is given by the formula:

$$R_d = \gamma.b'.N_b.s_b.i_b.g_b.b_b + q_0.d.N_d.s_d.i_d.g_d.b_d + c.N_c.s_c.i_c.g_c.b_c$$

where:

$$g_0 = \gamma_1.d$$

bearing capacity factors:

$$N_b = (N_d - 1).tan\varphi$$

$$N_d = tan^2(45 + \varphi/2).e^{\pi.tan\varphi}$$

$$N_c = (N_d - 1)/tan\varphi$$

for: $\varphi > 0$

$N_c = 2 + \pi$ for: $\varphi = 0$

shape factors:

$s_b = 1 - 0,3.\frac{b'}{l'}$

$s_d = 1 - \frac{b'}{l'}.sin\varphi$

$s_c = \frac{s_d.N_d - 1}{N_d - 1}$ for: $\varphi > 0$

$s_c = 1 + 0,2.\frac{b'}{l'}$ for: $\varphi = 0$

inclination factors for $\varphi >$ 0:

$i_b = 1 - (tan\delta)^{m+1}$ for: $\delta > 0$ *

$i_d = 1 - (tan\delta)^{m}$ for: $\delta > 0$

where:

$m = m_l.cos^2\omega + m_b.sin^2\omega$

$m_l = (2 + \frac{l'}{b'})/(1 + \frac{l'}{b'})$

$m_b = (2 + \frac{b'}{l'})/(1 + \frac{b'}{l'})$

$i_b = cos\delta.(1 - 0,04.\delta)^{0,64+0,028.\varphi}$ for: $\delta < 0$

$i_d = cos\delta.(1 - 0,0244.\delta)^{0,03+0,04.\varphi}$ for: $\delta < 0$

$i_c = \frac{i_d.N_d - 1}{N_d - 1}$

inclination factors for $\varphi =$ 0:

$i_b = 0$

$i_d = 1$

$i_c = 0,5 + 0,5.\sqrt{1 - \frac{T}{A'.c}}$

ground factors:

$g_b = (1 - 0,5.tan\beta)^6$ for: $\varphi > 0$

$g_d = (1 - tan\beta)^{1,9}$ for: $\varphi > 0$

$g_c = \frac{N_d.e^{-0,0349.\beta.tan\varphi} - 1}{N_d - 1}$ for: $\varphi > 0$

$g_b = 0$ for: $\varphi = 0$

$g_d = 1$ for: $\varphi = 0$

$g_c = 1 - 0,4.tan\beta$ for: $\varphi = 0$

base factors:

$b_b = b_d = b_c = e^{-0,045.\alpha.tan\varphi}$ for: $\varphi > 0$

$b_b = 0$ for: $\varphi = 0$

$b_d = 1$ for: $\varphi = 0$

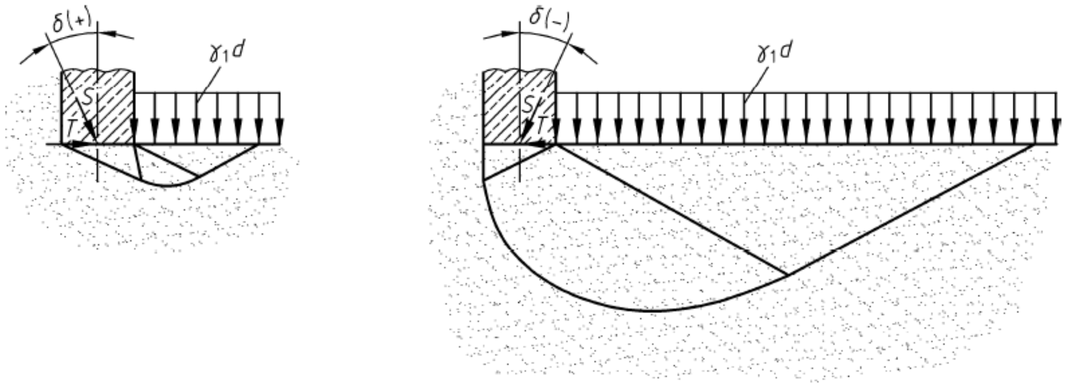
$b_c = 1 - 0,0068.\alpha$ for: $\varphi = 0$

where:

c	-	cohesion of soil
q_0	-	equivalent uniform load accounting for the influence of foundation depth
d	-	depth of the footing bottom
γ_1	-	unit weight of soil above the footing bottom
b	-	width of foundation
γ	-	unit weight of soil
N_c, N_d, N_b	-	bearing capacity factors
s_c, s_d, s_b	-	shape factors
i_c, i_d, i_b	-	inclination factors
g_c, g_d, g_b	-	ground factors
b_c, b_d, b_b	-	base factors
φ	-	angle of internal friction of soil
l	-	length of the foundation

δ	-	angle of deviation of the resultant force from the vertical direction
β	-	slope of the terrain
α	-	slope of the footing bottom
T	-	horizontal component of the resultant load
ω	-	angle between the horizontal component of resultant load T and the longer axis of footing (in the direction l_{ef})

** Note: The program performs analysis for both orientations of load inclination and uses more unfavourable value.*



Orientation of load inclination

Literature:

DIN 4017:2006-03

CTE DB SE-C

The solution is based on Spanish standard CTE DB SE-C, where the bearing capacity of foundation soil is given by the formula:

$$R_k = c_k N_c d_c s_c i_c t_c + q_0 N_q d_q s_q i_q t_q + \frac{1}{2} b_{ef} \gamma N_\gamma d_\gamma s_\gamma i_\gamma t_\gamma$$

where:

$$q_0 = \gamma_1 d$$

bearing capacity factors:

$N_c = (N_q - 1)/tan(\varphi)$ for: $\varphi > 0$

$N_c = \pi + 2$ for: $\varphi = 0$

$N_q = tan^2(45 + \varphi/2)\, e^{\pi tan(\varphi)}$

$N_\gamma = 1,5(N_q - 1)tan(\varphi)$

depth factors:

$d_c = 1 + 0,34arctan(d/b_{ef})$

$d_q = 1 + 2(N_q/N_c).(1 - sin(\varphi))^2.arctan(d/b_{ef})$ for: $\varphi > 0$

$d_q = 1$ for: $\varphi = 0$

$d_\gamma = 1$

$d_c = d_q = d_\gamma = 1$ for: $d < 2,0m$

shape factors:

$s_c = 1 + 0,2(b_ef/l_{ef})$

$s_q = 1 + 1,5.tan(\varphi).(b_{ef}/l_{ef})$

$s_\gamma = 1 - 0,3(b_{ef}/l_{ef})$

inclination factors:

$i_c = \dfrac{i_q N_q - 1}{N_q - 1}$ for: $\varphi > 0$

$i_c = 0,5\left(1 + \sqrt{1 - \dfrac{H}{b_{ef} l_{ef} c_k}}\right)$ for: $\varphi = 0$

$i_q = (1 - 0,7tan(\delta_B))^3.(1 - tan(\delta_L))$

$i_\gamma = (1 - tan(\delta_B))^3.(1 - tan(\delta_L))$

$tan(\delta_B) = H_B/V$

$$tan(\delta_L) = H_L/V$$

$$i_c = i_q = i_\gamma = 1 \quad \text{for: } H \leq 0,1V$$

ground factors:

$$t_c = e^{-2\beta tan(\varphi)}$$

$$t_q = t_\gamma = 1 - sin(\beta)$$

$$t_c = t_q = t_\gamma = 1 \quad \text{for: } \beta \leq 5°$$

where:	c	-	cohesion of soil
	q_0	-	equivalent uniform load accounting for the influence of foundation depth
	d	-	depth of the footing bottom
	γ_1	-	unit weight of soil above the footing bottom
	b_{ef}	-	effective width of foundation
	γ	-	unit weight of soil
	N_c , N_q , N_γ	-	bearing capacity factors
	d_c , d_q , d_γ	-	depth factors
	s_c , s_q , s_γ	-	shape factors
	i_c , i_q , i_γ	-	inclination factors
	t_c , t_q , t_γ	-	ground factors
	φ	-	angle of internal friction of soil
	l_{ef}	-	effective length of the foundation
	δ_B	-	angle of deviation of the resultant force from the vertical in foundation width direction
	δ_L	-	angle of deviation of the resultant force from the vertical in the foundation length direction
	H	-	horizontal component of the resultant load
	H_B	-	horizontal component of the resultant load in the foundation width direction
	H_L	-	horizontal component of resultant load in the foundation length direction
	V	-	vertical component of the resultant load
	β	-	slope of the terrain

Literature: CTE DB SE-C

B1/VM4

This solution is based on the NZ standard B1/VM4. The calculation is the same as for the standard approach, except the following coefficients:

bearing capacity factors:

$$N_b = 2.(N_d - 1).tan\varphi$$

shape factors:

$$s_e = 1 + \frac{b}{l}.\frac{N_d}{N_e}$$

$$s_d = 1 + \frac{b}{l}.\tan\varphi$$

$$s_b = 1 - 0{,}4.\frac{b}{l}$$

depth factors:

for $\varphi = 0$ and $d/b \leq 1$:

$$d_e = 1 + 0{,}4.\frac{d}{b}$$

for $\varphi = 0$ and $d/b > 1$:

$$d_e = 1 + 0{,}4.\tan^{-1}\frac{d}{b}$$

for $\varphi > 0$:

$$d_e = d_d - \frac{1 - d_d}{N_d.tan(\varphi)}$$

for $d/b \leq 1$:

$$d_d = 1 + 2.\tan\varphi.(1 - \sin\varphi)^2.\frac{d}{b}$$

for $d/b > 1$:

$$d_d = 1 + 2.\tan\varphi.(1 - \sin\varphi)^2.\tan^{-1}\frac{d}{b}$$

$$d_b = 1$$

inclination factors:

for $\varphi = 0$:

$$i_e = 0,5 + 1.\sqrt{1 - \frac{H}{A'.c}}$$

$$i_d = 1$$

for $\varphi > 0$ H parallel to L_{ef}:

$$i_d = i_b = 1 - \frac{H}{V + A.c.\cot\varphi}$$

$$i_e = \frac{i_d.N_d - 1}{N_d - 1}$$

for $\varphi > 0$ H parallel to B_{ef}:

$$i_d = (1 - \frac{0{,}7.H}{V + A.c.\cot\varphi})^3$$

$$i_b = (1 - \frac{H}{V + A.c.\cot\varphi})^3$$

$$i_e = \frac{i_d.N_d - 1}{N_d - 1}$$

where:

d	-	depth of the footing bottom
b	-	width of foundation
l	-	length of foundation
φ	-	angle of internal friction of soil
c	-	cohesion of soil
H	-	horizontal component of resultant load
V	-	vertical component of resultant load

Literature:

Acceptable Solutions and Verification Methods For New Zealand Building Code Clause B1 Structure, 1 December 2008

SP22.13330.2016

This solution is based on Russian standard **SP22.13330.2016**, which has two approaches for the calculation of soil bearing capacity:

1) **Service resistance (R)**

$$R = \frac{\gamma_{c1}.\gamma_{c2}}{k}.M_\gamma.k_z.b.\gamma_2 + M_q.d_1.\gamma_1 + M_c.c$$

where:

γ_{c1}, γ_{c2}	-	coefficients of working conditions
k	-	$k = 1$ if the strength characteristics are determined by the direct tests, or $k = 1.1$ (taken from the tables of Annex A)
M_γ, M_q, M_c	-	bearing capacity factors
k_z	-	$k_z = 1$ if $b < 10$ m, else $k_z = 8/b + 0.2$ [m]
b	-	width of foundation
γ_1, γ_2	-	unit weight of soil above/below the footing bottom
c	-	cohesion of soil

d_1 - depth of the footing bottom

Note: Parameters φ , c , γ_2 are taken as weighted average to the depth of z_{SP} , where $z_{SP} = b/2$ if $b <$ 10 m , else z_{SP} = 4+0.1b [m]

2) **Bearing capacity (N)**

$$N = \frac{\gamma_c}{\gamma_n}.(N_\gamma.s_\gamma.b_{ef}.\gamma_2 + N_q.s_q.d.\gamma_1 + N_c.s_c.c)$$

$$s_\gamma = 1 - 0,25.\frac{b_{ef}}{l_{ef}}$$

$$s_q = 1 + 1,5.\frac{b_{ef}}{l_{ef}}$$

$$s_c = 1 + 0,3.\frac{b_{ef}}{l_{ef}}$$

kde:

γ_c	-	coefficients of working conditions
γ_n	-	coefficient of reliability (γ_n = 1.2; γ_n = 1.15; γ_n = 1.1 for structures with level of responsibility I, II, III)
N_γ , N_q , N_c	-	bearing capacity factors
s_γ , s_q , s_c	-	shape factors
b_{ef}	-	effective width of foundation
γ_1 , γ_2	-	unit weight of soil above/below the footing bottom
c	-	cohesion of soil
d	-	depth of the footing bottom

Note: Parameters φ , c , γ_2 are taken along the Prandtl slip surface.

Literature:

SP22.13330.2016.

Coefficients of Working Conditions

Coefficients of working conditions γ_{c1} , γ_{c2} are taken from the following table according to the **type of soil** and **ratio of the length and the height of the structure** L/H :

Soils	Coefficient γ_{c1}	Coefficient γ_{c2} for structures with a rigid structural scheme and L/H	
		4 and more	1,5 and less
Coarse-grained with sand aggregate and sands, except fine and silty	1.4	1.2	1.4
Fine sands	1.3	1.1	1.3
Fine sands with silt (low-moisture)	1.25	1.0	1.2
Fine sands with silt (saturated)	1.1	1.0	1.2
Clay and coarse with clay aggregate consistency index of soil or aggregate $I_L \leq$ 0.25	1.25	1.0	1.1
the same with the 0.25 < $I_L \leq$ 0.5	1.2	1.0	1.0
the same with the I_L > 0.5	1.1	1.0	1.0

Notes:

1. For buildings with a flexible design scheme, the coefficient γ_{c2} is equal to one.

3. For intermediate L/H values, the coefficient γ_{c2} is determined by interpolation.

4. For loose sands γ_{c1} and γ_{c2} are equal to one

The coefficient of working conditions γ_c is taken from the following table according to the **type of soil/rock**:

Soils	Coefficient γ_c
Sands, except silty	1.0
Silty sands, and clay soils in a stable condition	0.9
Clay soils in the unstabilized state	0.85
Rocks	
Non-weathered	1.0
Weathered	0.9
Intensely weathered	0.8

Literature:

SP22.13330.2016.

Bearing capacity factors M

Bearing capacity factors M are taken from the following table according to the angle of internal friction φ :

φ [°]	**Bearing capacity factors** M		
	M_γ	M_q	M_c
0	0.00	1.00	3.14
1	0.01	1.06	3.23
2	0.03	1.12	3.32
3	0.04	1.18	3.41
4	0.06	1.25	3.51
5	0.08	1.32	3.61
6	0.10	1.39	3.71
7	0.12	1.47	3.82
8	0.14	1.55	3.93
9	0.16	1.64	4.05
10	0.18	1.73	4.17
11	0.21	1.83	4.29
12	0.23	1.94	4.42
13	0.26	2.05	4.55
14	0.29	2.17	4.69
15	0.32	2.30	4.84
16	0.36	2.43	4.99
17	0.39	2.57	5.15
18	0.43	2.73	5.31
19	0.47	2.89	4.48
20	0.51	3.06	5.66
21	0.56	3.24	5.84
22	0.61	3.44	6.04
23	0.66	3.65	6.24
24	0.72	3.87	6.45
25	0.78	4.11	6.67
26	0.84	4.37	6.90
27	0.91	4.64	7.14
28	0.98	4.93	7.40
29	1.06	5.25	7.67
30	1.15	5.59	7.95
31	1.24	5.95	8.24
32	1.34	6.34	8.55
33	1.44	6.76	8.88
34	1.55	7.22	9.22
35	1.68	7.71	9.58
36	1.81	8.24	9.97
37	1.95	8.81	10.37
38	2.11	9.44	10.80

39	2.28	10.11	11.25
40	2.46	10.85	11.73
41	2.66	11.64	12.24
42	2.88	12.51	12.79
43	3.12	13.46	13.37
44	3.38	14.50	13.98
45	3.66	15.64	14.64

Literature:

SP22.13330.2016.

Bearing capacity factors N

Bearing capacity factors N are taken from the following table according to the angle of internal friction φ and angle of deviation of the resultant force from the vertical direction δ :

φ [°]		δ [°]									
		0	5	10	15	20	25	30	35	40	45
0	N_γ N_q N_c	0.00 1.00 5.14									
5	N_γ N_q N_c	0.20 1.57 6.49	0.05 1.26 2.93	δ' = 4.9							
10	N_γ N_q N_c	0.60 2.47 8.34	0.42 2.16 6.57	0.12 1.66 3.38	δ' = 9.8						
15	N_γ N_q N_c	1.35 3.94 10.98	1.02 3.45 9.13	0.61 2.84 6.88	0.21 2.06 3.94	δ' = 14.5					
20	N_γ N_q N_c	2.88 6.40 14.84	2.18 5.56 12.53	1.47 4.64 10.02	0.82 3.64 7.26	0.6 2.69 4.65	δ' = 18.9				
25	N_γ N_q N_c	5.87 10.66 20.72	4.50 9.17 17.53	3.18 7.65 14.26	2.00 6.13 10.99	1.05 4.58 7.68	0.58 3.60 5.58	δ' = 22.9			
30	N_γ N_q N_c	12.39 18.40 30.14	9.43 15.63 25.34	6.72 12.94 20.68	4.44 10.37 16.23	2.63 7.96 12.05	1.29 5.67 8.09	0.95 4.95 6.85	δ' = 26.5		
35	N_γ N_q N_c	27.50 33.30 46.12	20.58 27.86 38.36	14.63 22.77 31.09	9.79 18.12 24.45	6.08 13.94 18.48	3.38 10.24 13.19	1.60 7.04 8.63	δ' = 29.8		
40	N_γ N_q N_c	66.01 64.19 75.31	48.30 52.71 61.63	33.84 42.37 49.31	22.56 33.26 38.45	14.18 25.39 29.07	8.26 18.70 21.10	4.30 13.11 14.43	2.79 10.46 11.27	δ' = 32.7	
45	N_γ N_q N_c	177.61 134.87 133.87	126.09	86.20	56.50	32.26	20.73	11.26	5.45	5.22	δ' = 35.2

Literature:

SP22.13330.2016

Bearing Capacity on Undrained Subsoil

One of the following approaches is available to assess the bearing capacity of a foundation if undrained conditions are assumed:

- standard analysis
- according to CSN 73 1001 "Základová pùda pod plošnými základy" approved 8.6. 1987
- according to Indian standard IS:6403-1981 "Code of Practice for Determination of Bearing Capacity of Shallow Foundations" from year 1981
- according to EC 7-1 (EN 1997-1:2003) "Design of geotechnical structures - Part 1: General rules"
- according to Danish standard DS/EN 1997-1 DK NA:2013
- according to Spanish standard CTE DB SE-C
- according to NZ standard B1/VM4

In addition (with exception of Spanish standard CTE DB SE-C) the coefficients due to Brinch - Hansen are used to account for inclined footing bottom (see standard analysis).

In case of undrained conditions it is assumed that during construction the spread footing undergoes an instantaneous settlement accompanied by shear deformations of soil in the absence of volumetric changes. When the structure is completed the soil experiences both primary and secondary consolidation accompanied by volumetric changes. The influence of neutral stress appears in the reduction of soil strength. The strength of the soil is then presented in terms of total values of the angle of internal friction φ_u and the total cohesion c_u (these parameters can be considered as the minimal ones). Depending on the degree of consolidation the value of the total angle of internal friction φ_u ranges from 0 to φ_{ef}, the total cohesion c_u is greater than c_{ef}. Owing to the fact that the choice of undrained conditions depends on a number of factors (rate of load, soil permeability, degree of saturation and degree of overconsolidation) it is the designer's responsibility to decide, depending on the actual problem being solved, if the effective parameters should be used. Nevertheless, the total parameters are generally used for fine-grained soil.

Standard Analysis

The following formula is used by default:

$$R_d = (\pi + 2).c_u.s_c.d_c.i_c.b_c + q$$

with dimensionless coefficients:

$$s_c = 1 + 0{,}2.\frac{b}{l}$$

$$d_c = 1 + 0{,}1.\sqrt{\frac{d}{b}}$$

$$i_c = (1 - tg\delta)^2$$

$$b_c = 1 - \frac{2.\alpha}{\pi + 2}$$

where:

where:	c_u	-	total cohesion of the soil
	b	-	width of the foundation
	l	-	length of the foundation
	d	-	depth of the foundation
	δ	-	angle of deviation of the resultant force from the vertical direction
	α	-	slope of the footing bottom from horizontal direction
	q	-	overburden pressure at the level of foundation base

CTE DB SE-C

The solution is based on Spanish standard CTE DB SE-C, where the bearing capacity of foundation soil is given by the formula:

$$R_k = c_u N_c d_c s_c i_c t_c + q_0$$

where:

$$N_c = 2 + \pi$$

$$d_c = 1 + 0{,}34 arctan(d/b_{ef})$$

$$s_c = 1 + 0{,}2(b_{ef}/l_{ef})$$

$$i_c = 0,5\left(1 + \sqrt{1 - \frac{H}{b_{ef} l_{ef} c_u}}\right)$$

$$t_c = 1$$

$$q_0 = \gamma_1 d$$

for ***d* < 2,0m**:

$$d_c = 1$$

for ***H* ≤ 0,1*V***:

$$i_c = 1$$

where:

	c_u	-	total cohesion of the soil
	b_{ef}	-	effective width of the foundation
	l_{ef}	-	effective length of the foundation
	d	-	depth of the footing bottom
	H	-	horizontal component of the resultant load
	V	-	vertical component of the resultant load
	γ_1	-	unit weight of the soil above the footing bottom
	q_0	-	equivalent uniform load accounting for the influence of the foundation depth

Literature:

CTE DB SE

B1/VM4

The solution is based on **NZ standard B1/VM4**, where the bearing capacity of foundation soil is given by the formula:

$$R_d = N_e . s_e . d_e . i_e . c_u + q$$

bearing capacity factors:

$$N_e = 2 + \pi$$

$$N_d = 1$$

shape factors:

$$s_e = 1 + \frac{b}{l} . \frac{N_d}{N_e}$$

depth factors:

for $d/b \leq 1$:

$$d_e = 1 + 0{,}4 . \frac{d}{b}$$

for $d/b > 1$:

$$d_e = 1 + 0{,}4 . \tan^{-1}\frac{d}{b}$$

inclination factors:

for $\varphi = 0$:

$$i_e = 0,5 + 1 . \sqrt{1 - \frac{H}{A' . c}}$$

where:

	d	-	depth of the footing bottom
	b	-	width of foundation
	l	-	length of foundation
	c	-	undrained shear strength
	H	-	horizontal component of resultant load
	V	-	vertical component of resultant load

Literature:

Acceptable Solutions and Verification Methods For New Zealand Building Code Clause B1 Structure, 1 December 2008

Bearing Capacity of Foundation on Bedrock

The following methods can be used to compute the design bearing capacity R_d of the foundation with a horizontal footing bottom supported by the rock mass composed of rocks or weak rocks:

- Standard approach
- Solution according to CSN 73 1001
- Solution according to EC 7-1

Standard Analysis

The bearing capacity of foundation soil composed of rocks or weak rocks is found from the expression proposed by Xiao-Li Yang and Jian-Hua Yin[1]:

$$R_d = s^{0,5}\sigma_c.N_s + q_0.N_q + \frac{b}{2}.\gamma_2.N_\gamma$$

where:

$$s = e^{\frac{GSI-100}{9-3.D}}$$

$$N_q = \frac{1}{2}.sec^2\left(\frac{\pi}{4}+\frac{\varphi}{2}\right).e^{\left[\left(\frac{2}{3}.\pi-\varphi\right).tg\varphi\right]}$$

$$N_\gamma = \left(N_q - 1\right)\frac{e^{\left[\left(\frac{\pi}{2}-\varphi\right).tg\varphi\right]}}{2.cos\,\varphi}$$

where:			
	s	-	nonlinear parameter depending on rock properties (according to Hoek and Brown)
	GSI	-	Geological Strength Index
	D	-	coefficient reflecting damage of a rock mass
	N_q , N_γ	-	coefficients of bearing capacity depending on the angle of internal friction
	N_s	-	coefficient of the strength of rock depending on GSI and strength parameter m_i
	φ	-	angle of internal friction of the rock
	σ_c	-	uniaxial compressive strength of rock > *0,5 MPa*
	q_0	-	equivalent uniform load accounting for the influence of foundation depth
	γ_2	-	unit weight of soil above the footing bottom
	b	-	width of foundation

[1] Xiao-Li Yang, Jian-Hua Yin: Upper bound solution for ultimate bearing capacity with a modified Hoek-Brown failure criterion, International Journal of Rock Mechanics & Mining Sciences 42 (2005),str. 550-560.

Solution According to CSN 73 1001

The bearing capacity of foundation soil composed of rocks or weak rocks follows from articles 97 - 99 of standard CSN 73 1001 "**Foundation soil below spread footing**" approved 8.6. 1987.

As input parameters the analysis requires the unit weight of soil γ , uniaxial compression strength σ_c , Poisson's ratio ν and deformation modulus E_{def} .

Analysis According to EC 7-1 (EN 1997-1:2003)

The bearing capacity of the foundation R_d with a horizontal footing bottom is determined according to a design method for the derivation of expected bearing capacity of spread footings resting on a bedrock outlined in a supplement G (informative) EC 7-1 (EN 1997-1:2003) "Design of geotechnical structures - Part 1: General rules". For low strength or damaged rocks with closed discontinuities including chalks with low porosity less than *35%* the derivation of expected bearing capacity follows from the classification of rocks into groups of rocks stored in the table below. The analysis further requires an input of discontinuity spacing S_d , unit weight of rock γ , Poisson's ratio ν and uniaxial compressive strength

σ_c . It is assumed that the structure is able to transmit a settlement equal to *0,5 %* of the foundation width. The expected values of bearing capacity for other settlements can be estimated using direct proportion. For weak and broken rocks with opened or filled discontinuities it is recommended to use lower values than the expected ones.

Rock groups

Group	Type of rock
1	Pure limestones and dolomites Carbonate sandstones of low porosity
2	Igneous Oolitic and marly limestones Well cemented sandstones Indurated carbonate mudstones Metamorphic rocks, including slates and schist (flat cleavage / foliation)
3	Very marly limestones Poorly cemented sandstones Slates and schist (steep cleavage / foliation)
4	Uncemented mudstones and shales

Literature:

Eurocode 7:Geotechnical design - Part 1:General rules.

Parameters to Compute Foundation Bearing Capacity

Parameters to compute vertical bearing capacity of a foundation resting on bedrock

The following parameters are used in the program to compute the foundation vertical bearing capacity:

- values of coefficient D reflecting a state of damage of a rock mass
- values of strength parameter m_i
- strength of rocks in simple compression σ_c
- Poisson's ratio of rocks ν
- unit weight of rocks γ

Estimating disturbance coefficient D

Description of rock mass	Suggested value of D
Rock mass, intact strong rock, excavation by blasting or by open TBM	0
Rock mass, poor quality rock, mechanical excavation with minimal disturbance	0
Rock mass, poor rock, mechanical excavation, significant floor heave, temporary invert or horizontal geometry of excavation sequence	0.5
Rock mass, very poor rock often very altered, local damage of surrounding rock (app. 3 m)	0.8
Rock slope or rock outcrop, modification with controlled blasting	0.7
Rock slope or rock outcrop, modification with blasting results to the some disturbance	1.0
Open-pit mines, excavation with blasting	1.0
Open-pit mines, mechanical excavation	0.7

Values of strength parameter m_i

Type of rock	Representative rocks	m_i [-]
Carbonate rocks with well-developed cleavage	Dolomite, limestone, and marble	≈ 7
Lithified argillaceous rocks	Mudstone, siltstone shale, slate	≈ 10
Arenaceous rock with strong crystal and poorly developed crystal cleavage	Sandstone and quartzite	≈ 15
Fine-grained polymineralic igneous crystalline rocks	Andesite, dolerite, diabase, rhyolite	≈ 17

Coarse-grained polymineralic igneous and metamorphic rocks	Amphibolite, gabbro, gneiss, granite and quartz diorite	≈ 25

Uniaxial compressive strength $\sigma\ c$, Poisson's ratio ν and Unit weight of rock γ

Strength of rocks	**Types of rock (examples)**	**Uniaxial compr. strength σ_c $[MPa]$**	**Poisson's ratio ν**	**Unit weight of rock γ $[kN/m^3]$**
Extremely hard rock	Very hard, intact rock strong and solid quartzite, basalt and other extremely hard rock	>150	0.10	28.00 - 30.00
Very hard rock	Very hard granite, quartz porphyry, quartz slate, very hard sandstones and limestones	100 - 150	0.15	26.00 - 27.00
Hard rock	Solid and compact granite, very hard sandstone and limestone, siliceous iron veins, hard pudding stones, very hard iron ores hard calcite, not very hard granite, hard sandstone, marble, dolomite, pyrite	80 - 100	0.20	25.00 - 26.00
Fairly hard rock	Normal sandstone, medium-hard iron ore, sandy shale, flagstone	50 - 80	0.25	24.00
Medium-hard rock	Hard mudstones, not very hard sandstones and calcite, soft flagstone, not very hard shales, dense marl	20 - 50	0.25 - 0.30	23 - 24.00
Fairly weak rock	Soft schist, soft limestones, chalk, rock salt, frost soils, anthracite, normal marl, disturbed sandstones, soft flagstones and soils with aggregates	5 - 20	0.30 - 0.35	22.00 - 26.00
Weak rock	Compact clay, hard soil (eluvium with soil texture)	0.5 - 5	0.35 - 0.40	22.00 - 18.00

Horizontal Bearing Capacity of Foundation

The foundation horizontal bearing capacity is verified according to the theory of limit states using the following inequality:

$$H \leq \frac{R_{dh}}{\gamma_{RH}}$$

or based on the factor of safety as:

$$\frac{R_{dh}}{H} \leq SF$$

where:

$$R_{dh} = Q.\tan\psi_d + a_d.A_{ef} + S_{pd}$$

$$H = \sqrt{H_x^2 + H_y^2}$$

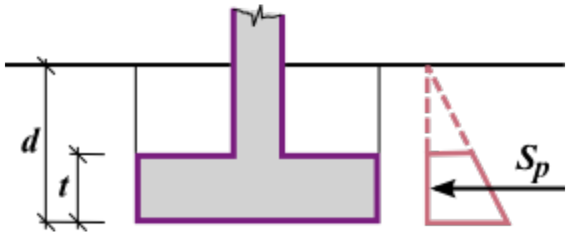
where:

ψ_d	-	angle of friction between the foundation and soil
a_d	-	cohesion between the foundation and soil
A_{ef}	-	effective area of the foundation
S_{pd}	-	earth resistance
H_x, H_y	-	components of the horizontal force
Q	-	extreme design vertical force
γ_{RH}	-	coefficient of the horizontal bearing capacity of the foundation (for input use the "Spread Footing" tab)
SF	-	safety factor

When adopting the analysis methodology according to EN 1997 the term with cohesion ($a_d * A_{ef}$) is excluded for drained conditions whereas the term with friction between foundation and soil ($Q * tan\psi_d$) is excluded for undrained conditions.

The analysis depends on the design angle of internal friction below the footing bottom φ_d, the design value of cohesion below the footing bottom c_d and the design value of earth resistance S_{pd}. If the soil-footing frictional angle and the soil-footing cohesion are less than the values of soil below the footing bottom, then it is necessary to use those values.

The earth resistance is assumed as displayed in the figure:



Earth resistance

The earth resistance S_{pd} is found with the help of the reduction of passive earth pressure or pressure at rest employing influence coefficients:

$$S_{pd} = \frac{S_p}{\gamma_{mR}}$$

where:

S_p	-	passive earth pressure, pressure at rest or reduced passive pressure
γ_{mR}	-	coefficient of reduction of earth resistance (for input used the "Spread Footing" tab) - for the analysis according to CSN it assumes the value $\gamma_{mR} =$ 1.5 for passive pressure, $\gamma_{mR} =$ 1.3 for pressure at rest

Coefficients of earth pressures are clear from the following formulas:

for passive pressure:

$$K_p = tan(45 + 0,5.\varphi_d)$$

for pressure at rest in drained soils:

$$K_0 = 1 - sin\varphi_d$$

for pressure at rest in other soils:

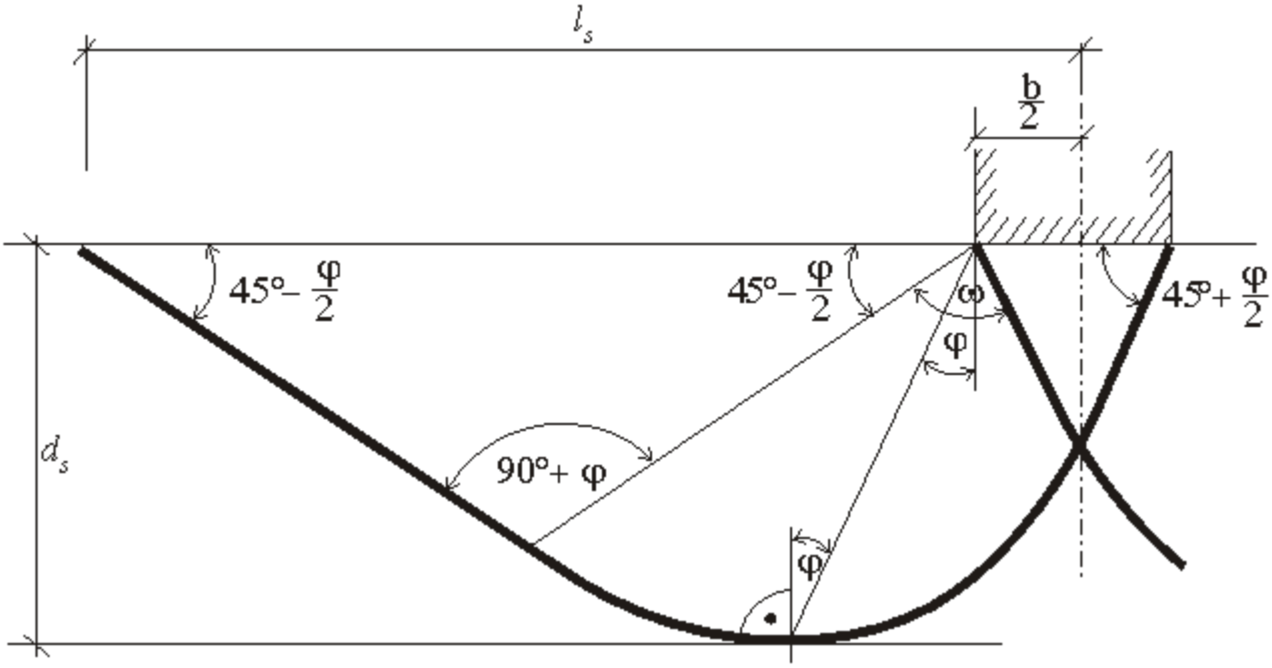
$$K_0 = \frac{\nu}{1-\nu}$$

When determining the **reduced passive pressure,** the resultant force includes contributions due to the passive pressure and pressure at rest.

The passive pressure can be considered, if the deformation needed for its activation does not cause unallowable stresses or deformations in the upper structure.

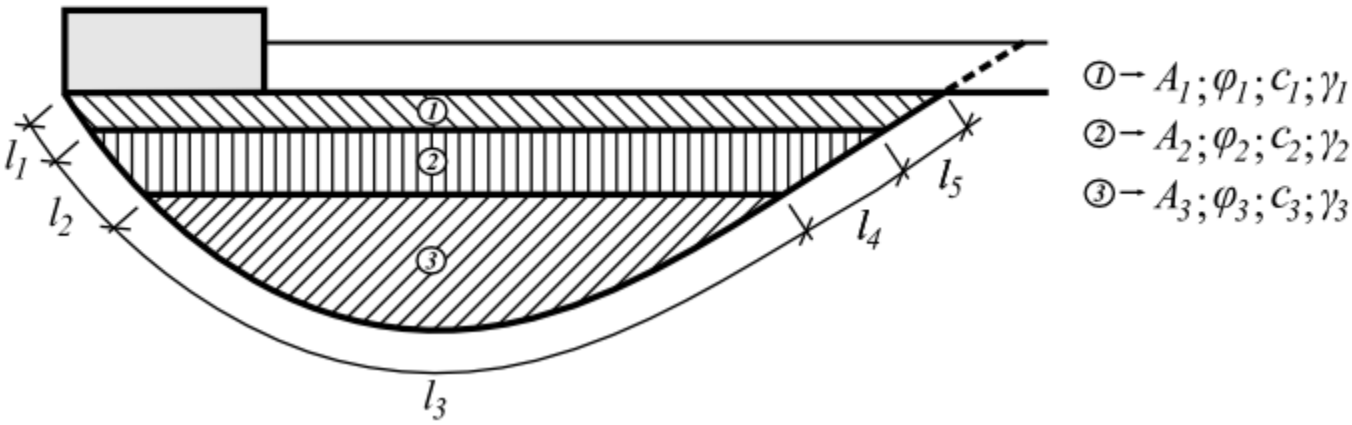
Homogenization of Layered Subsoil

If the soil below the footing bottom is inhomogeneous (or if there is groundwater present) then the input profile is transformed into a homogeneous soil based on the Prandtl slip surface (see Fig.), which represents the type and location of failure of the foundation.



The Prandtl slip surface

Determination of equivalent values of φ (angle of internal friction), c (cohesion of soil) γ (unit weight of soil below footing bottom) is evident from the following formulas. The unit weight of soil above the foundation is derived in the same way.



Procedure for computation of auxiliary values

$$\varphi = \frac{\varphi_1 (l_1 + l_5) + \varphi_2 (l_2 + l_4) + \varphi_3 l_3}{\sum_{i=1}^{5} l_i}$$

$$c = \frac{c_1 (l_1 + l_5) + c_2 (l_2 + l_4) + c_3 l_3}{\sum_{i=1}^{5} l_i}$$

$$\gamma = \frac{\gamma_1 A_1 + \gamma_2 A_2 + \gamma_3 A_3}{\sum_{i=1}^{3} l_i}$$

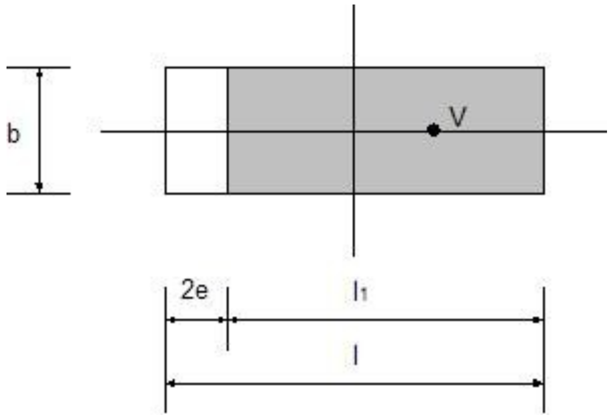
Effective Area

When solving the problem of eccentrically loaded foundations the program offers two options to deal with an affective dimension of the foundation area:

- a rectangular shape of the effective area is assumed
- general shape of the effective area is assumed

Rectangular shape

A simplified solution is used in such cases. In the case of axial eccentricity (bending moment acts in one plane only) the analysis assumes a uniform distribution of contact stress σ applied only over a portion of the foundation l_1, which is less by twice the eccentricity e compared to the total length l.



Determination of effective area in case of axial eccentricity

An effective area ($b*l_1$) is assumed to compute the contact stress so that we have:

$$\sigma = \frac{V}{b\,(l-2e)}$$

In case of a general eccentric load (foundation is loaded by the vertical force V and by bending moments M_1 and M_2 the load is replaced by a single force with given eccentricities:

$$e_1 = \frac{M_1}{V}$$

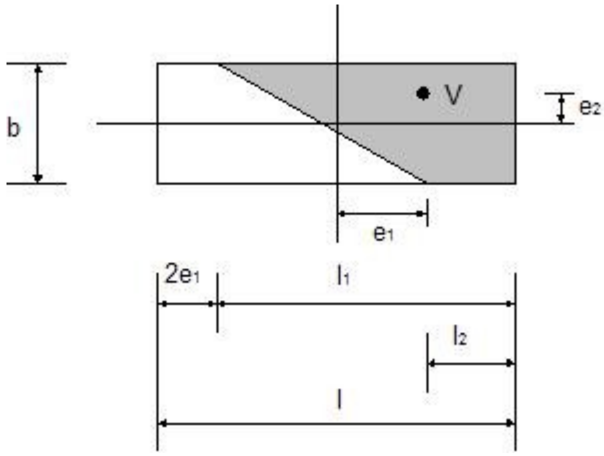
$$e_2 = \frac{M_2}{V}$$

The size of affective area follows from the condition that the force V must act eccentrically:

$$A_{ef} = b_{ef} l_{ef} = (b-2e_2)\,(l-2e_1)$$

General shape of contact stress

In the case of an eccentric load the effective area is determined from the assumption that the resultant force V must act in the center of gravity of the compressive area. The theoretically correct solution appears in Fig.



Determination of contact stress for general eccentricity - general shape

Owing to considerable complexity in determining the exact location of the neutral axis, which in turn is decisive when computing the effective area, the program follows the solution proposed by Highter a Anders1), where the effective areas are derived with the help of graphs.

1) Highter, W.H. - Anders,J.C.: Dimensioning Footings Subjected to Eccentric Loads Journal of Geotechnical Engineering. ASCE, Vol. 111, No GT5, pp 659 - 665.

Determination of Cross-Sectional Internal Forces

Longitudinal reinforcement of a foundation is checked for the load due to bending moment and shear force. The stress in the footing bottom is assumed as **linear**. Stresses in individual directions x, y are determined independently.

When the **linear distribution of stress** in the footing bottom is considered the distribution of stress over the cross-section is provided by:

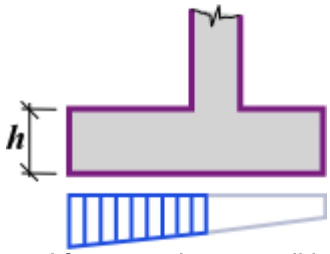
$$\sigma_1 = \frac{N}{d^2}\left(4d - 6\left(\frac{d}{2} - e\right)\right)$$
$$\sigma_2 = \frac{N}{d^2}\left(-2d + 6\left(\frac{d}{2} - e\right)\right)$$

or when excluding tension:

$$\sigma = \frac{2N}{3\left(\frac{d}{2} - e\right)}$$

where: e - eccentricity of normal force N
d - width of foundation
N - normal force acting at the footing bottom

Bending moment and shear force are determined as reaction developed on the cantilever beam as shown in the figure:



Internal forces acting on wall jump

Internal forces in the cross-section corresponding to the **constant distribution of stress** are provided by:

$$M = \frac{1}{2}{d_v}^2\sigma$$
$$Q = d_v\sigma$$
$$\sigma = \frac{N}{d - 2e}$$

where: σ - maximal stress in the footing bottom
d_v - length of jump
e - eccentricity of normal force N
d - width of wall foundation
N - normal force acting at the footing bottom

Verification of Foundation Eccentricity

Verification of foundation eccentricity is carried out for the *1st* LS (foundation bearing capacity) and the *2nd* LS (foundation settlement) analysis.

During analysis the program performs verification for the following cases:

- maximum eccentricity in the direction of base length: $e_x \leq e_{alw}$
- maximum eccentricity in the direction of base width: $e_y \leq e_{alw}$
- maximum overall eccentricity: $e_t \leq e_{alw}$

The value of maximum allowable foundation eccentricity e_{alw} is input in the "**Settings**" frame, tab "Spread Footing".

The value of overall eccentricity e_t is provided by:

$$e_t = \sqrt{{e_x}^2 + {e_y}^2}$$

where: e_x - maximum eccentricity in the direction of base length
e_y - maximum eccentricity in the direction of base width

Procedure for calculating eccentricities needed for the determination of an effective area of off-center (eccentrically) loaded foundation is described in more detail here.

For a shallow foundation resting on a rock foundation or for a concrete slab type of foundation it is necessary in some cases to adopt different values of limit eccentricities.

Analysis of Uplift

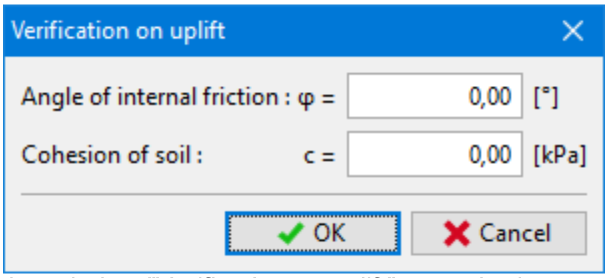
The analysis of a spread footing in tension is performed when the load due to a negative normal force N (the force acts upwards) is assumed. The verification of such a footing is carried out according to the corresponding verification methodology. During the analysis the program compares the maximum tensile force $N_{t,max}$ with the uplift resistance R_t. The program considers the following four methods of the calculation of bearing capacity (uplift resistance R_t) of footing:

- standard approach
- cone method
- DL/T 5219-2005
- EN 50341

Standard Approach

The uplift resistance R_t combines the self-weight of the soil overburden and footing + friction along the footing walls + fictitious (substitute) block of soil above the footing.

The soil properties are determined according to **overburdentype** from the "Foundation" frame.If the chosen type is "**input shape and soil**" or "**from geological profile**" the angle of internal friction φ and cohesion c are taken from the assigned soils. In case of **"input unit weigh"** the "**Bearing cap.**" frame in the "**Verification on uplift**" dialog window allows us to input the angle of internal friction φ and cohesion c of overburden.



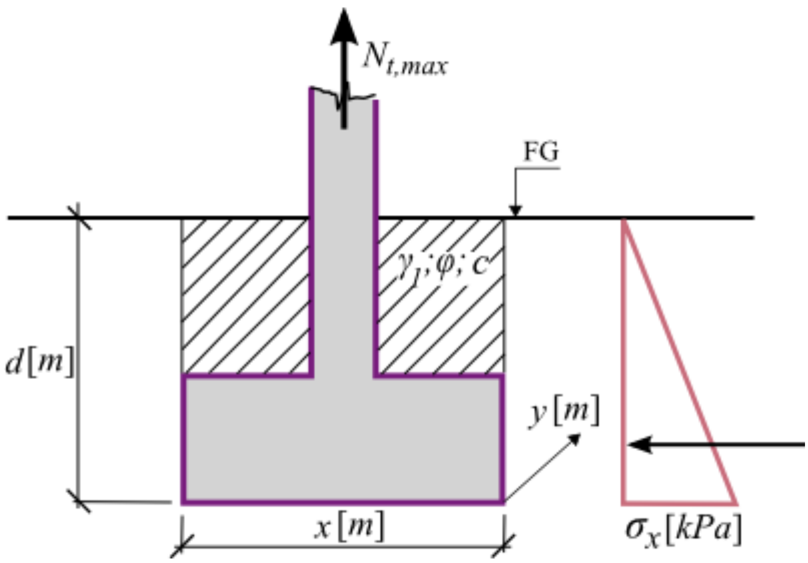
Dialog window "Verification on uplift" - standard approach

The vertical bearing capacity check - spread footing in tension (uplift resistance) follows from:

$$R_t = (\sigma_x tg\varphi_d + c_d)\, dp + G_p$$

where:

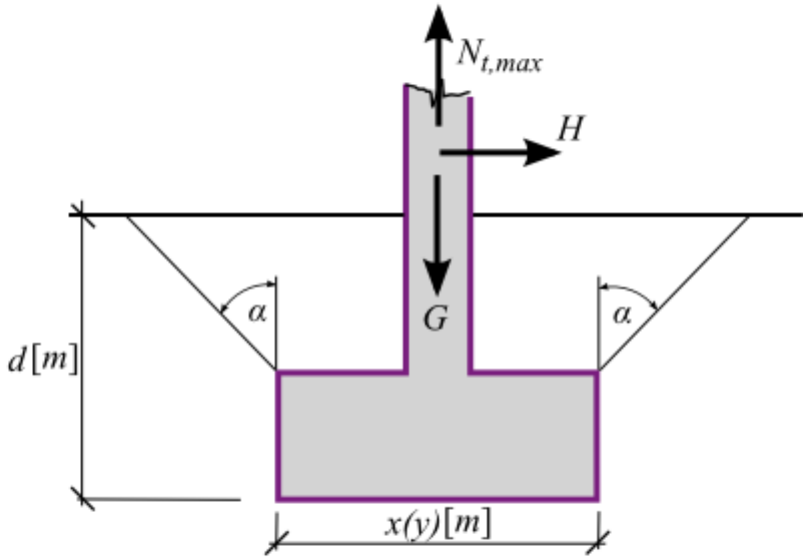
σ_x	-	earth pressure at rest due to overburden
φ_d	-	design angle of internal friction of overburden
c_d	-	design cohesion of overburden
d	-	depth of the footing bottom
p	-	footing perimeter
G_p	-	self-weight of foundation



Verification on uplift - standard approach

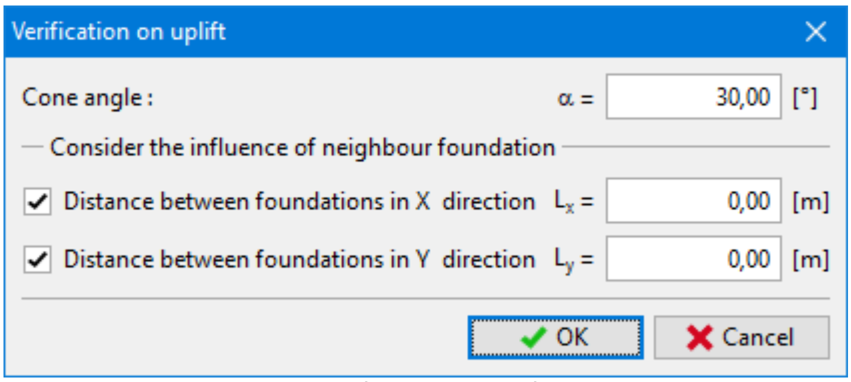
Cone Method

The uplift resistance R_t combines the footing self-weight and the self-weight of the soil overburden in the shape of a cone as evident from the following figure.

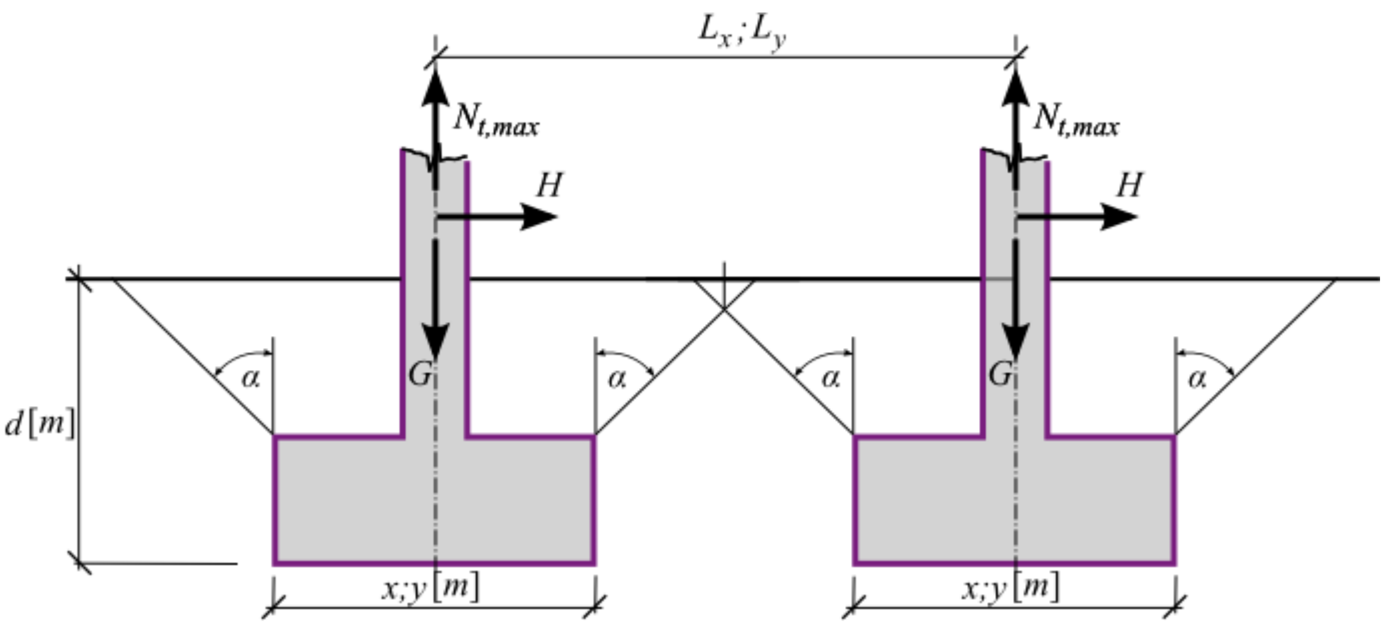


Basis of the cone method

The "**Bearing cap.**" frame in the "**Verification on uplift**" dialog window serves to input the cone angle α . When calculating the uplift resistance R_t it is also possible to account for the influence of a neighbor foundation either in one or both directions that reduces the volume of the soil cone.



Dialog window "Verification on uplift" - cone method



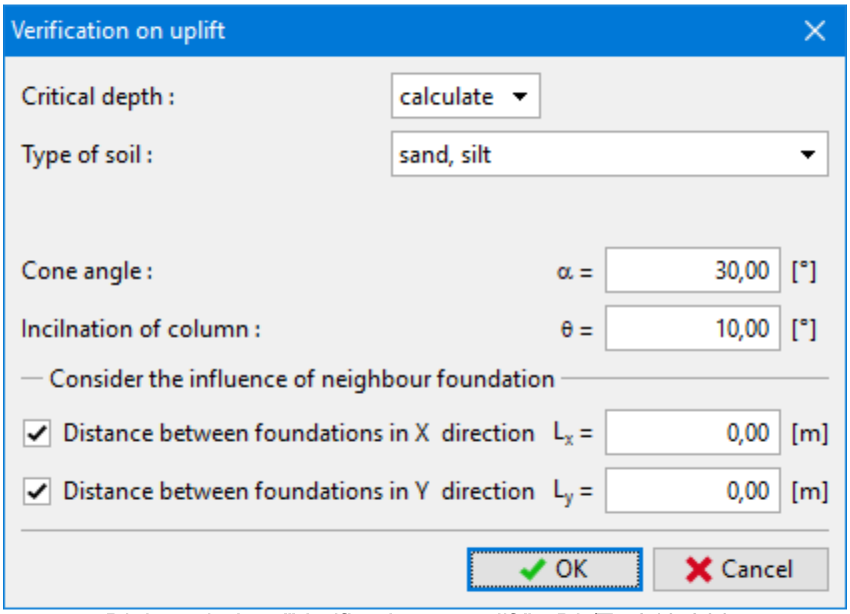
DL/T 5219 - 2005

This type of verification originates from the Chinese standard **DL/T 5219 - 2005**. Unlike the cone method and the standard approach it introduces a critical depth h_c, which depends on the type of soil and the shape of the foundation. The "**Verification on uplift**" dialog window allows either inputting the critical depth h_c directly or it can be determined by the program depending on the input type of soil and the shape of foundation according to table **6.3.1-1 - Critical depth h_c**.

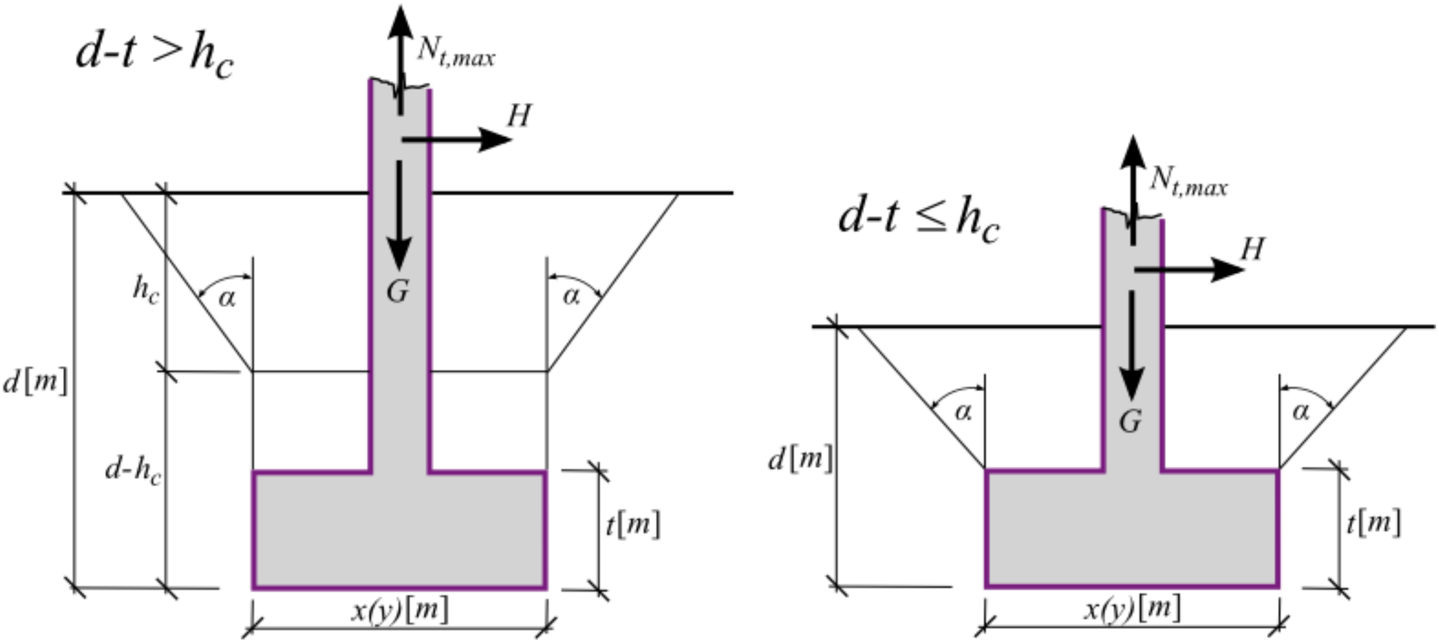
Table 6.3.1-1 with the values of a critical depth based on the Chinese standard DL/T 5219-2005

Type of soil	Natural state of soil	Critical depth h_c for tensile foundation	
		Circular foundation	Square foundation
Sand or Silt	Dense ~ Slightly dense	2.5D	3.0B
Clay	Hard ~ Stiff	2.0D	2.5B
	Plastic	1.5D	2.0B
	Soft - plastic	1.2D	1.5B
Note 1: For a rectangular foundation, if the ration between length L' and width B is smaller than 3, calculate h_c as a circular foundation and $D =$ 0.6*$(B+L')$. Note 2: The soil must be in a natural state.			

The "**Bearing cap.**" frame in the "**Verification on uplift**" dialog window serves to specify as another input parameter the cone angle α. The influence of a neighbor foundation is described in the cone method. The inclination of column θ has no influence on the calculation of bearing capacity of a footing in tension (the uplift resistance).



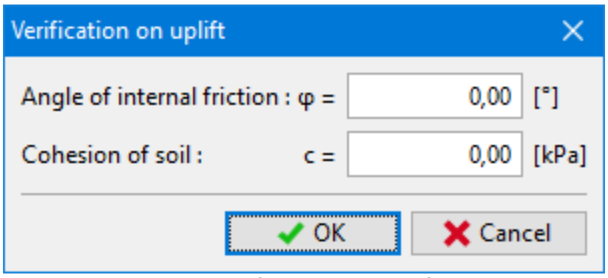
Dialog window "Verification on uplift" - DL/T 5219-2005



Verification of footing in tension according to DL/T 5219-2005

EN 50341

The uplift resistance R_t combines the weight of the soil overburden and footing + friction along the footing walls and soil block above the footing. The soil properties are determined according to the **overburdentype** from the "Foundation" frame.If the chosen type is "**input shape and soil**" or "**from geological profile**" the angle of internal friction φ and cohesion c are taken from the assigned soils. In the case of **"input unit weigh**" the "**Bearing cap.**" frame in the "**Verification on uplift**" dialog window allows us to input the angle of internal friction of overburden φ and cohesion of overburden c .



Dialog window "Verification on uplift" - EN 50341

The **uplift resistance** follows from:

$$R_t = R_s + R_b + G_f + G_b$$

where:
- R_s - bearing capacity along the foundation
- R_b - bearing capacity along the soil block
- G_f - weight of foundation
- G_b - weight of overburden

The **bearing capacity along the foundation** is calculated from:

$$R_s = pt\left(c + \frac{1}{2}K_r\gamma(2d - t)tan(\varphi)\right)$$

$$K_r = 1 - sin(\varphi)$$

where:
- p - footing perimeter
- c - cohesion of the soil
- K_r - coefficient of the earth pressure at rest

γ	-	weight of the soil
d	-	depth of the footing bottom
t	-	footing thickness
φ	-	angle of internal friction of the soil

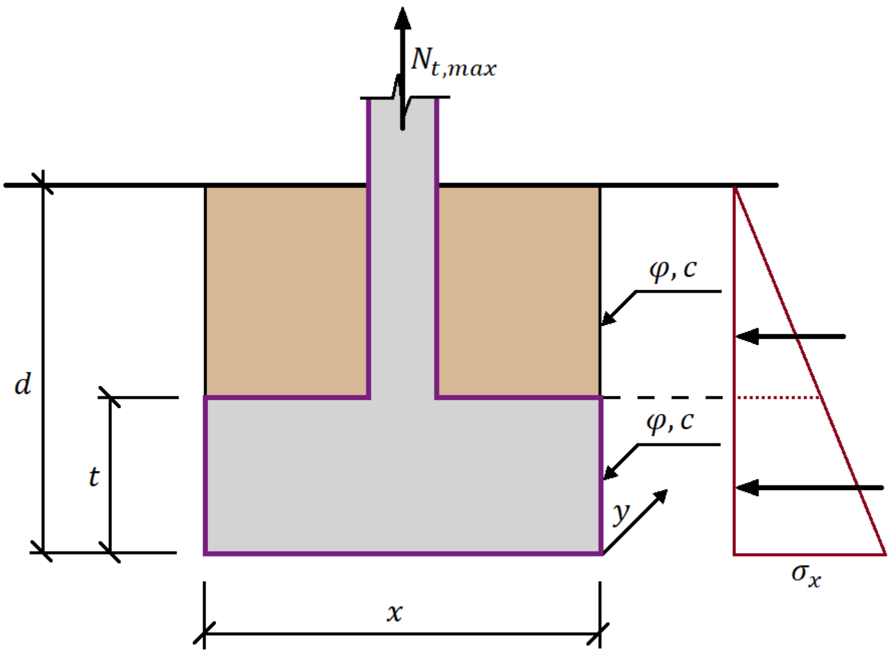
The **bearing capacity along the soil block** is calculated from:

$$R_b = p\,(d-t)\left(\frac{1}{2}K_a\gamma(d-t)tan(\varphi)\right)$$

$$K_a = tan^2(45 - \frac{\varphi}{2})$$

where:

p	-	footing perimeter
d	-	depth of the footing bottom
t	-	footing thickness
K_a	-	coefficient of active earth pressure
γ	-	weight of soil
φ	-	angle of internal friction of the soil



Verification on uplift - EN 50431

Pile Analysis

Analyses available in the program "**Pile**" can be divided into three main groups:

- Analysis of vertical bearing capacity
- Pile settlement
- Analysis of horizontal bearing capacity

Vertical Bearing Capacity

Analysis of pile vertical resistance can be carried out using:

- Analytical solution
- Spring method

Analytical Solution

The analytical solution assumes that the pile total compressive resistance R_c is derived as a sum of the pile base resistance R_b and the pile shaft resistance R_s (developed due to friction of the surrounding soil along the shaft). The following generally accepted methods are implemented into the program:

- NAVFAC DM 7.2
- Tomlinson
- Effective stress method
- CSN 73 1002
- CTE-DB SE-C

For the above-specified methods it is possible to choose one of the following verification methodologies:

- Classical way
- EN 1997-1

When running the **compression pile** analysis, the pile self-weight is introduced depending on the setting in the frame "Load". As for the **tensile pile**, the pile self-weight is always taken into account automatically. Based on the input load the program itself performs the verification analysis for **either compression or tensile pile**.

NAVFAC DM 7.2

Calculation of vertical pile resistance is performed according to the publication: NAVFAC DM 7.2, Foundation and Earth Structures, U.S. Department of the Navy 1984, where all approaches are described in more detail. The analysis provides the pile base resistance R_b and the pile shaft resistance R_s .

For non-cohesive, the program takes into account the critical depth.

Pile Base Resistance

Pile base resistance for **non-cohesive soils** is given by:

$$R_b = \sigma_{efb} .N_q .A_b$$

where: σ_{efb} - effective stress on the pile base

N_q - bearing capacity factor

A_b - area of pile base

The bearing capacity factor N_q is back-calculated by the program; however, its values can be manually modified.

For **cohesive soils,** the following expression holds:

$$R_b = 9.c_u.A_b$$

where: c_u - undrained shear strength at the base

A_b - area of pile base

Pile Shaft Resistance

Pile shaft resistance for **non-cohesive soils** is given by:

$$R_s = \sum_{j=1}^{n} K_j .\sigma_{ef,j} .tg\delta_j .A_{s,j}$$

where: K_j - coefficient of lateral earth pressure in the j^{th} layer

$\sigma_{ef,j}$ - effective strength of soil in the j^{th} layer

δ_j - pile skin friction angle (between pile material and surrounding soil in the j^{th} layer)

A_{sj} - area of the pile shaft in the j^{th} layer

The lateral earth pressure coefficient K is back calculated by the program; however, its values can be manually modified in the "Add new soils" dialog window.

For **cohesive soils,** the following expression holds:

$$R_s = \sum_{j=1}^{n} \alpha_j .c_{u,j} .A_{s,j}$$

where: α_j - skin friction coefficient in the j^{th} layer

$c_{u,j}$ - undrained cohesion in the j^{th} layer

A_{sj} - area of the pile shaft in the j^{th} layer

Bearing Capacity Factor Nq

Reference values of the bearing capacity factor N_q are listed in the table. If jet grouting is used when constructing the pile, then the maximum angle of internal friction φ is equal to 28°.

Bearing capacity factor N_q

Angle of internal friction φ [°]	26	28	30	31	32	33	34	35	36	37	38	39	40
Bearing capacity factor N_q for driven piles	10	15	21	24	29	35	42	50	62	77	86	120	145
Bearing capacity factor N_q for bored piles	5	8	10	12	14	17	21	25	30	38	43	60	72

Literature:

NAVFAC DM 7.2, Foundation and Earth Structures, U.S. Department of the Navy, 1984.

Coefficient of Lateral Earth Pressure K

The soil around a driven pile is compressed during construction and the lateral earth pressure of this soil acting on the pile skin is greater than the earth pressure at rest (given by coefficient K_0) and smaller than the maximum earth pressure (passive earth pressure given by coefficient K_p):

$$K_0 < K < K_p$$

Reference values of the coefficient of lateral earth pressure K are listed later in the table. The coefficient of lateral earth pressure K is approximated as follows:

$$K = \frac{K_a + K_p + K_0}{3}$$

where: K_0 - coefficient of earth pressure at rest

$$K_0 = 1 - sin\varphi$$

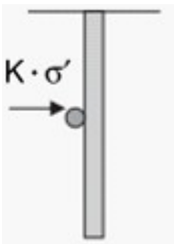
φ - angle of soil shear resistance

K_p - coefficient of passive earth pressure

$$K_p = tg^2\left(45^\circ + \frac{\varphi}{2}\right)$$

K_a - coefficient of active earth pressure

$$K_a = tg^2\left(45^\circ - \frac{\varphi}{2}\right)$$



Pressure on the pile

Reference values of the lateral earth pressure coefficient K

Type of pile	K for compression piles	K for tensile - uplifted piles
Driven H-piles	0.5 - 1.0	0.3 - 0.5
Driven displacement piles (round and square)	1.0 - 1.5	0.6 - 1.0
Driven displacement tapered piles	1.5 - 2.0	1.0 - 1.3
Driven jetted piles	0.4 - 0.9	0.3 - 0.6
Bored piles (less than 70cm)	0.7	0.4

Literature:

NAVFAC DM 7.2, Foundation and Earth Structures, U.S. Department of the Navy, 1984.

Friction Angle on Pile Skin

Reference values of friction angle between the pile skin material and the surrounding non-cohesive soil are listed in the following table:

Friction angle on pile δ [°]

Pile material	δ [°]
Steel piles	20
Timber piles	0.75φ
Steel reinforced concrete piles	0.75φ

where: φ - angle of internal friction of the soil

Literature:

NAVFAC DM 7.2, Foundation and Earth Structures, U.S. Department of the Navy, 1984.

Adhesion Coefficient

Reference values of the adhesion coefficient α are listed in the following table:

Empirical adhesion coefficient α

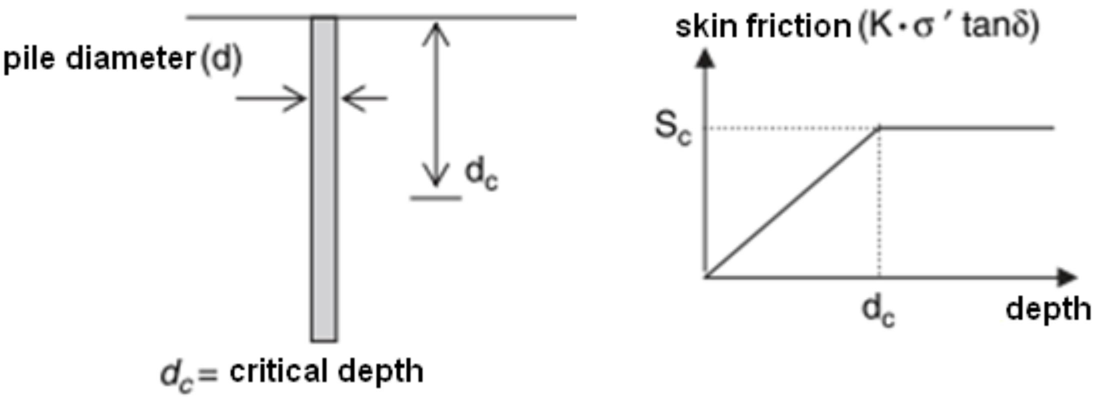
Pile material	Soil consistency	Cohesion range $c\ u$ [kPa]	Adhesion coefficient α [-]
Timber and concrete piles	Very soft	0 - 12	0.00 - 1.00
	Soft	12 - 24	1.00 - 0.96
	Medium stiff	24 - 48	0.96 - 0.75
	Stiff	48 - 96	0.75 - 0.48
	Very stiff	96 - 192	0.48 - 0.33
Steel piles	Very soft	0 - 12	0.00 - 1.00
	Soft	12 - 24	1.00 - 0.92
	Medium stiff	24 - 48	0.92 - 0.70
	Stiff	48 - 96	0.70 - 0.36
	Very stiff	96 - 192	0.36 - 0.19

Literature:

NAVFAC DM 7.2, Foundation and Earth Structures, U.S. Department of the Navy, 1984.

Critical Depth

For **non-cohesive** soils, the skin friction does not increase infinitely with depth as e.g. effective stress, but from a certain so-called **critical depth** it acquires a constant value - see the following figure, where d_c is the critical depth, S_c is the skin friction at critical depth and d is the pile diameter. A similar rule holds also for the pile base resistance in non-cohesive soils, where the same values of the critical depth d_c are considered for simplicity.



Critical depth

The reference value of the critical depth for soft sand is 10d (d is the pile diameter or its width), for medium compact sand and compact sand the values are 15d and 20d, respectively.

The coefficient of critical depth k_{dc} can be specified in the "Ver. capacity". The critical depth follows from:

$$d_c = k_{dc} . d$$

where: k_{dc} - critical depth coefficient

d - pile diameter

CTE-DB SE-C

Calculation of vertical pile resistance is performed according to Spanish standard CTE-DB SE-C. The analysis provides the pile base resistance R_b and the pile shaft resistance R_s .

Pile Base Resistance

Pile base resistance for **non-cohesive soils** is given by:

$$R_b = q_b . A_b$$

where: q_b - unit pile base resistance

A_b - area of pile base

For **cohesive soils,** the following expression holds:

$$R_b = 9 . c_u . A_b$$

where: c_u - undrained shear strength at the base

A_b - area of pile base

Pile Shaft Resistance

Pile shaft resistance is given by:

$$R_s = \sum_{i=1}^{n} \tau_{s,i} . A_{s,i}$$

where: $\tau_{s,i}$ - unit pile shaft resistance in the i^{th} layer

$A_{s,i}$ - area of the pile shaft in the i^{th} layer

Unit pile shaft resistance for **non-cohesive soils** is given by:

$$\tau_s = \sigma_{ef} . K_s . f_s . tan(\varphi) \leq \tau_{s,max}$$

where: σ_{ef} - effective stress in the soil layer

K_s - coefficient of lateral earth pressure

f_s - technology coefficient

φ - angle of internal friction

$\tau_{s,max}$ - maximum limit for unit pile shaft resistance

According to *CTE-DB SE-C* the maximum limit for unit pile shaft resistance in non-cohesive soil is ***120 kPa***.

For **cohesive soils,** the following expression holds:

$$\tau_s = \frac{100 . c_u}{100 + c_u} \leq \tau_{s,max}$$

where: c_u - undrained shear strength of soil

$\tau_{s,max}$ - maximum limit for unit pile shaft resistance

According to *CTE-DB SE-C* the maximum limit for unit pile shaft resistance in cohesive soil is ***100 kPa***.

Unit Pile Base Resistance

Unit pile base resistance for **non-cohesive soils** is given by:

$$q_b = f_b . \sigma_{efb} . N_q \leq q_{b,max}$$

$$N_q = tan^2 (45 + \varphi/2) \, e^{\pi tan(\varphi)}$$

where: f_b - technology coefficient

σ_{efb} - effective stress on the pile base

N_q - bearing capacity factor

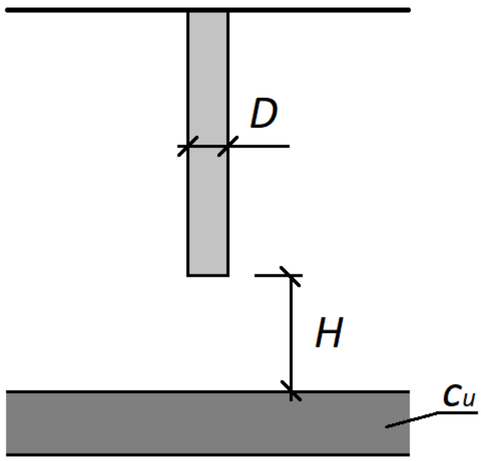
$q_{b,max}$ - maximum limit for unit pile base resistance
φ - angle of internal friction

Unit resistance q_b cannot be higher than the maximum limit of unit pile base resistance $q_{b,max}$. This limit can be input manually or calculated in the "Vertical cap." frame.

According to *CTE-DB SE-C* the maximum limit for unit pile base resistance is **20000 kPa**. If the soft soil layer with lower resistance is under the pile base, the $q_{b,max}$ is calculated according to the formula:

$$q_{b,max} = 6.\left(1 + \frac{H}{D}\right)^2 .c_u$$

where:
D - pile diameter
H - distance of soft soil layer from the pile base
c_u - undrained shear strength of the soft soil



Influence of soft soil layer

Influence of Pile Technology

The technology coefficient f_b has an influence on the **pile base resistance**. Its value is automatically calculated according to **pile technology** or it can be entered manually by the user.

Pile Technology	f_b [-]
bored	2.50
CFA	2.50
driven	3.00

The technology coefficient f_s has an influence on the **pile shaft resistance**. Its value is automatically calculated according to **pile material** or it can be entered manually by the user.

Pile Material	f_s [-]
concrete	1.00
timber	1.00
precast concrete	0.90
steel	0.80

The lateral earth pressure coefficient K_s is calculated according to **pile technology** or it can be entered manually.

Pile Technology	K_s [-]
bored	0.75
CFA	0.75
driven	1.00

Tomlinson

This widely used method adopts undrained shear strength parameters to calculate the pile bearing capacity. It further assumes that the pile shaft resistance depends on the pressure due to overburden surcharge.

The **pile shaft resistance** is given by:

$$R_s = \sum_{j=1}^{n} c_{a,j} . A_{s,j} = \sum_{j=1}^{n} \alpha_j . c_{u,j} . A_{s,j}$$

where:

- $c_{a,j}$ - adhesion in the j^{th} layer (shear stress between the pile skin and the surrounding soil)
- $A_{s,j}$ - area of the pile shaft in the j^{th} layer
- α_j - empirical adhesion coefficient (depends on the type of soil, type of pile, etc.) in the j^{th} layer
- $c_{u,j}$ - undrained cohesion in the j-th layer (undrained shear strength)

The empirical adhesion coefficient α is back-calculated by the program. Its values, however, can be manually adjusted in the "Add new soil" dialog window.

The **pile base resistance** is given by:

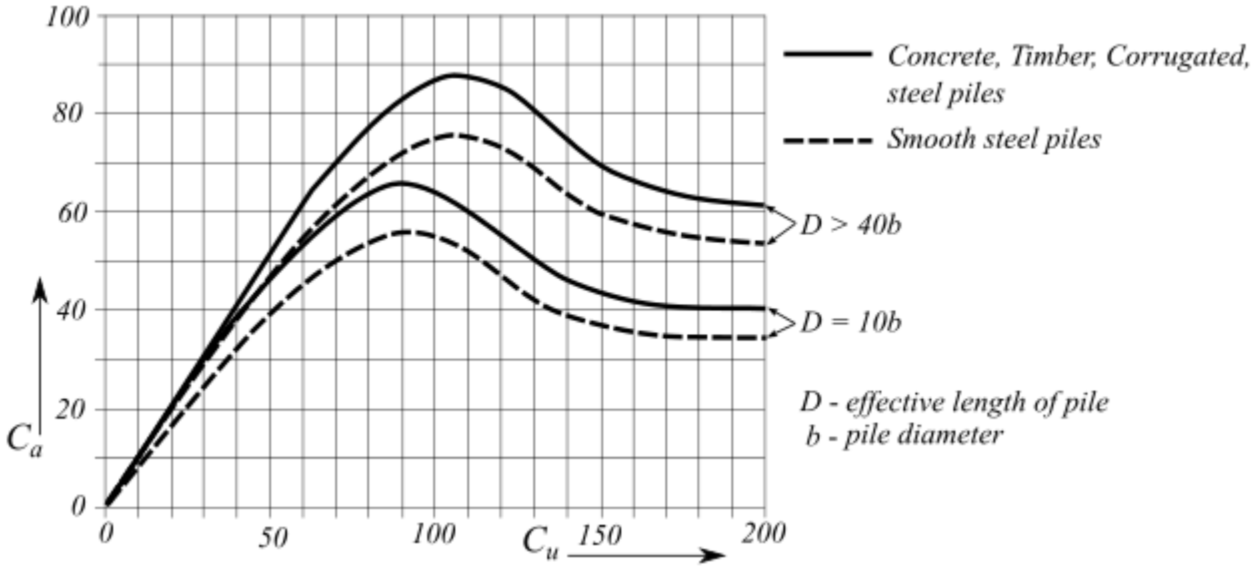
$$R_b = q_b . A_b = 9 . c_u . A_b$$

where:

- q_b - unit pile base resistance
- A_b - pile base area
- c_u - undrained shear strength

Adhesion Coefficient

The empirical adhesion coefficient α takes into account the behavior of soil around the pile skin and depends on the pile material, quality of the pile skin surface, and the type of surrounding soil. The values of this coefficient are introduced into the program employing the following graph taken from M.J. Tomlinson: Pile Design and Construction Practice.



Graph to determine adhesion coefficient

Effective Length

The effective length D determines the pile length, which effectively transfers the load from the pile into the soil. If the entire pile is placed into the resistant soil, in which the load is transferred by skin friction, then the effective length corresponds to the pile length below terrain - see Fig. A. In case of a layered medium, highly compressible layers (in which the load is not transferred into the soil by skin friction) and the layers above are not counted into the effective length D - see Fig. B. The introduction of effective length into the analysis and its magnitude is set in the frame "Vertical capacity".

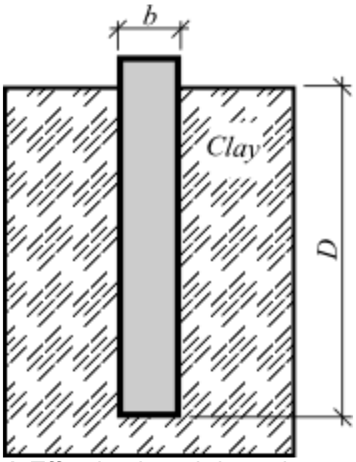


Fig. A Effective length in resistant soils

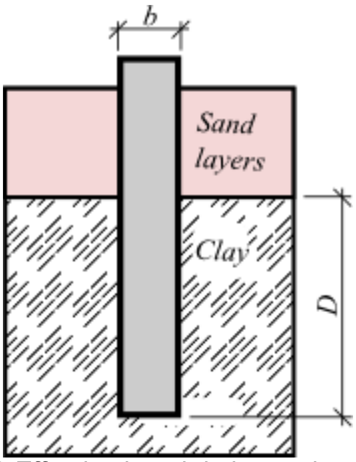


Fig. B Effective length in layered medium

Effective Stress Method

The effective stress method allows us to calculate the vertical bearing capacity of an isolated pile in both cohesive and non-cohesive soils. This method is suitable for drained conditions - i.e. conditions that prevail after sufficient time passed the construction.

The **pile shaft resistance** is given by:

$$R_s = \sum_{j=1}^{n} q_{s,j} . A_{s,j} = \sum_{j=1}^{n} \beta_{p,j} . \sigma_{0,j} . A_{s,j}$$

where:

- $q_{s,j}$ - shaft resistance in the j^{th} layer
- $\beta_{p,j}$ - coefficients according to Bjerrum and Burland in the j^{th} layer
- $\sigma_{0,j}$ - average effective stress due to overburden acting along the pile in the j^{th} layer
- A_{sj} - pile shaft area in the j^{th} layer

The **pile base resistance** is given by:

$$R_b = q_p . A_b = N_p . \sigma_p . A_b$$

where:

- q_p - unit pile base resistance
- A_b - pile base area
- N_p - pile base resistance coefficient (according to Fellenius)
- σ_p - effective stress due to overburden acting at pile base

Coefficients of Pile Bearing Capacity

Recommended ranges of values of coefficients of pile base resistance N_p and coefficient β are listed in the following table. The coefficient β is usually found in the given range, it seldom exceeds the value $1,0$.

Range of coefficients Np and β (Fellenius, 1991)

Type of soil	φ_{ef}	N_p	β
Clay	25 - 30	3 - 30	0.23 - 0.40
Silt	28 - 34	20 - 40	0.27 - 0.50
Sand	32 - 40	30 - 150	0.30 - 0.60
Gravel	35 - 45	60 - 300	0.35 - 0.80

Literature:

Fellenius, B.H.: Foundation Engineering Handbook, Editor H.S. Fang, Van Nostrand Reinhold Publisher, New York, 1991, 511 - 536.

CSN 73 1002

There are two methods implemented in the program to compute the pile vertical bearing capacity following the Commentary to the **CSN 73 1002 standard** "**Pile foundation**":

- **Analysis according to the theory of the 1st group of limit states**

The solution procedure is described in the Commentary to the CSN 73 1002 standard "Pile foundation" in the chapter 3 "Design" part B - general solution according to the theory of the 1st group of limit states (pp. 15). All computational approaches are based on formulas presented therein. The original geostatic stress σ_{or} is assumed from the finished grade. The coefficient of conditions of the behavior of foundation soil is considered for the depth z (measured from the finished grade).

$$z \le 1 \Rightarrow \gamma_{r2} = 1{,}3$$
$$1 < z \le 2 \Rightarrow \gamma_{r2} = 1{,}2$$
$$2 < z \le 3 \Rightarrow \gamma_{r2} = 1{,}1$$
$$3 < z \Rightarrow \gamma_{r2} = 1{,}0$$

The effective pile length used for the computation of skin bearing capacity is reduced by a segment:

$$l_p = \frac{d.N_d^{2/3}}{4}$$

where: d - pile diameter

- **Analysis of pile resting on incompressible subsoil**

Analysis of a pile resting on incompressible subsoil (rocks class R1, R2) is based on part G - Analysis of vertical bearing capacity R_c according to CSN 73 1004 - Commentary to CSN 73 1002 "Pilotové základy". The description begins on page 27 titled "Piles resting on incompressible subsoil". The solution procedures used in the program are identical. The influence coefficient of settlement I_{wp} is interpolated from Table 16, which is also built-in the program.

If checking the option "**analysis according to CSN 73 1002**" in the "Piles" tab the verification analysis is carried out exclusively according to CSN 73 1002 and other coefficients ate not used. Providing this option is not checked the verification is performed based on the selected methodology adopting particular coefficients.

Literature:

Československá státní norma ČSN 73 1002 Pilotové základy, Normalizační institut, Praha, 1987.

Československá státní norma ČSN 73 1004 Velkoprůměrové piloty, Normalizační institut, Praha, 1981.

Verification

Verification of pile **bearing capacity** depends on the verification methodology selected in the "Piles" tab:

- verification according to the factor of safety
- verification according to the theory of limit states
- verification according to EN 1997

Actual analyses (e.g. assessment of the pile base resistance) are the same for both options - they differ only by incorporation of design coefficients, combinations, and in the way of demonstrating the structure safety. Design coefficients (verification parameters) are specified in the "Piles" tab.

If the verification analysis **according to CSN 73 1002** is selected, the verification is carried out exclusively according to the Commentary to CSN 73 1002.

Verification According to the Theory of Limit States

When running the verification analysis according to the theory of limit states, it is possible to introduce the required values of design coefficients in the "Piles" tab.

The program performs verification of the **compression pile** as:

$$R_c = \frac{R_b}{\gamma_b} + \frac{R_s}{\gamma_s} \geq V_d + W_p$$

where:

- R_c - pile compressive resistance
- R_b - pile base resistance
- R_s - pile shaft resistance
- γ_b - partial factor on pile base resistance
- γ_s - partial factor on pile shaft resistance
- V_d - extreme vertical load acting on a pile
- W_p - pile self-weight

For **tension pile** the following verification applies:

$$R_{sdt} = \frac{R_s}{\gamma_{st}} \geq V_d + W_p$$

where:

- R_{sdt} - pile tensile resistance
- R_s - pile shaft resistance
- γ_{st} - partial factor on tensile pile shaft resistance
- V_d - extreme vertical load acting on a pile
- W_p - pile self-weight

Design Coefficients

The "Piles" tab allows us to specify two groups of design (partial) coefficients:

Partial factors on soil parameters

- $\gamma_{m\varphi}$ - reduction coefficient of internal friction
- γ_{mc} - reduction coefficient of cohesion
- $\gamma_{m\gamma}$ - coefficient of unit weight

It is also possible to choose a reduction of $tg\varphi$.

Partial factors on pile resistance

- γ_b - reduction coefficient of base resistance
- γ_s - reduction coefficient of shaft resistance
- γ_t - reduction coefficient of total resistance
- γ_{st} - reduction coefficient of resistance in tension

The values of individual coefficients are listed in corresponding standards.

Verification According to the Safety Factors

When running the verification analysis according to the factor of safety, it is possible to introduce the required value of factor safety SF for the vertical bearing capacity in the "Piles" tab.

The program performs verification of vertical bearing capacity of **compression pile** as:

$$\frac{R_c}{V_d + W_p} > SF_{cp}$$

where:

- V_d - extreme vertical load acting on a pile
- R_c - pile compressive resistance
- W_p - pile self-weight (introduction into the analysis based on the setting in the frame "Load")

and for **tension pile**:

$$\frac{R_{sdt}}{V_d + W_p} > SF_{tp}$$

where: V_d - extreme vertical load acting on a pile

R_{st} - pile tensile resistance

W_p - pile self-weight

Vertical Bearing Capacity - Spring Method

The program module "**Pile - spring method**" is part of the "**Pile**" program. It allows us to calculate the pile vertical bearing capacity in the generally layered subsoil. This analysis provides the load-settlement curve and distributions of forces and displacements developed along the pile.

The main advantage of this module is the availability of the required input parameters of soils around the pile - the user is asked to specify the **angle of internal friction, cohesion, unit weight, and deformation modulus** of a given soil.

The solution procedure in the module "**Pile - spring method**" is based on a semi-analytical approach. The response of soil surrounding the pile follows from the well known solution of layered subsoil as a generalization of the Winkler-Pasternak model. The elastic rigid plastic response in shear is assumed along the pile-soil interface in view of the Mohr-Coulomb failure criterion. The normal stress acting on the pile is determined from the geostatic stress and soil (concrete mixture) pressure at rest.

The **influence of water** in the vicinity of the pile is not only introduced into the shear bearing capacity of the pile skin but also affects the depth of the influence zone below the pile heel.

Providing the pile reaches incompressible subsoil the spring method cannot be used.

The pile settlement can also be influenced by the settlement of the surrounding terrain. In particular, settlement of soil may reduce the pile bearing capacity. The pile settlement increases without increasing load. This phenomenon is modeled in the program as so-called negative skin friction.

The analysis may also account for the influence of the technological process of pile construction on the stiffness of the pile foundation.

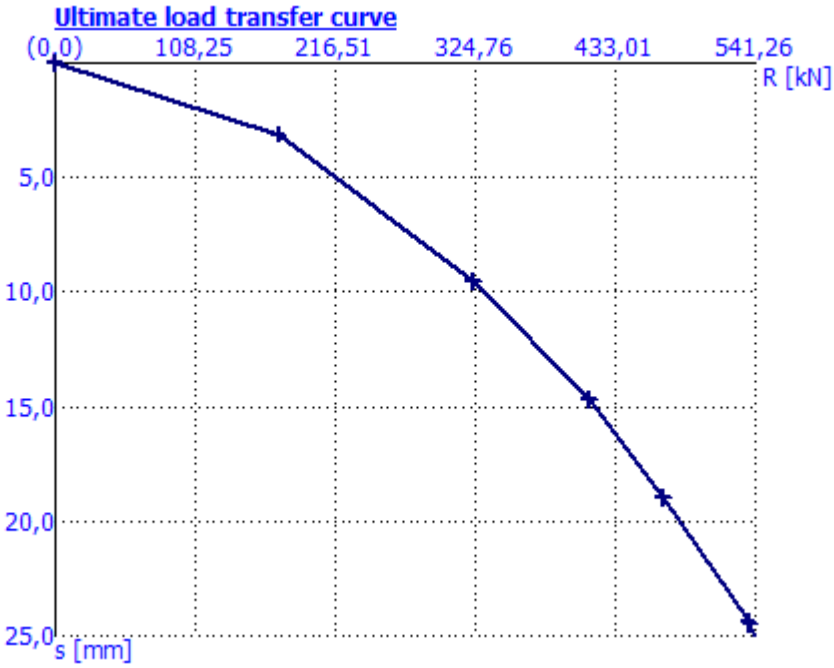
The solution procedure consists of several steps:

1. In the analysis, the pile subdivided into a number of segments. Subdivision into individual segments complies with the condition that the ratio between the pile segment and its diameter should be approximately equal to 2.5. The minimum number of segments is 10.
2. Each segment is in the analysis characterized by a spring. The spring stiffness serves to model both the shear resistance of the skin and at the pile heel the stiffness of soil below the pile heel.
3. For each segment the limit value of shear force T_{lim} transmitted by the skin is determined.
4. The pile is loaded at its top end by increments of the vertical load. For each load increment the magnitude of spring force for each segment is determined. However, it cannot exceed the limit value of skin friction T_{lim}. It is clear that for a certain load level all springs will no longer be able to increase their force and with additional load increase the pile becomes supported by the base spring only. This spring has no restriction on the transmitted force.
5. As a result the analysis provides the load-settlement curve, forces developed in the pile, and a graph showing a variation of shear as a function of deformation at a given location.

Load-Settlement Curve

The Load-settlement curve describes the variation of vertical load Q as a function of the pile settlement.

By default, the program offers the construction of this curve for the maximal value of settlement equal to 25 mm. This magnitude, however, can be adjusted up to the value of 100 mm before running the calculation. An example showing a typical shape of the load-settlement curve appears in the figure.



Load-settlement curve of single pile

Shear Strength of Skin

For each segment of the analyzed pile the program determines the limiting value of the force that can be transmitted by the pile skin at the location of a given segment. Its value depends on the geostatic stress σ_z found at a depth of a given segment.

$$\sigma_z = \sum \gamma . h$$

where: γ - unit weight of soil

h - depth below the ground surface

Summation sign denotes that σ_z is summed over individual layers of the soil.

The allowable shear stress is then given by:

$$\tau = \sigma_z . k . tg\varphi + c$$

where: c - cohesion of soil at the location of the beam

φ - angle of internal friction of soil at the location of the beam

k - coefficient of increase of allowable skin friction due to technology

If the beam is found below the groundwater table, the allowable skin friction is then reduced to receive the form:

$$\tau = (\sigma_z - u) . k . tg\varphi + c$$

where: u - pore pressure below the groundwater table

The allowable shear force then follows from:

$$T_{lim} = O . l . \tau$$

where: O - length of the perimeter of the pile skin

l - length of the pile beam

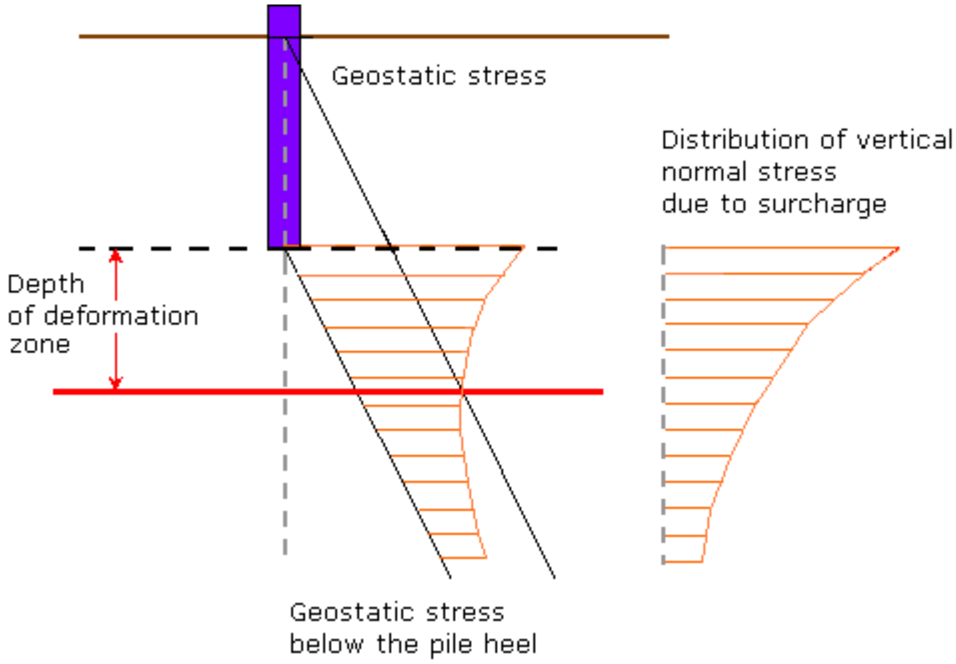
Coefficient of Increase of Limit Skin Friction

A specific input parameter is the coefficient of increase of limit skin friction k due to the applied technology of construction. By default, the value of this coefficient is set equal to 1.0. There is no recommendation by the standard for its specific value - its adjustment depends solely on the practical experiences of the designer. It has been found from the in situ measurements on real piles that the value of k is usually greater than 1.0 and may reach a value of 1.5. Theoretically, however, it may attain values even less than 1.0.

Depth of Deformation Zone

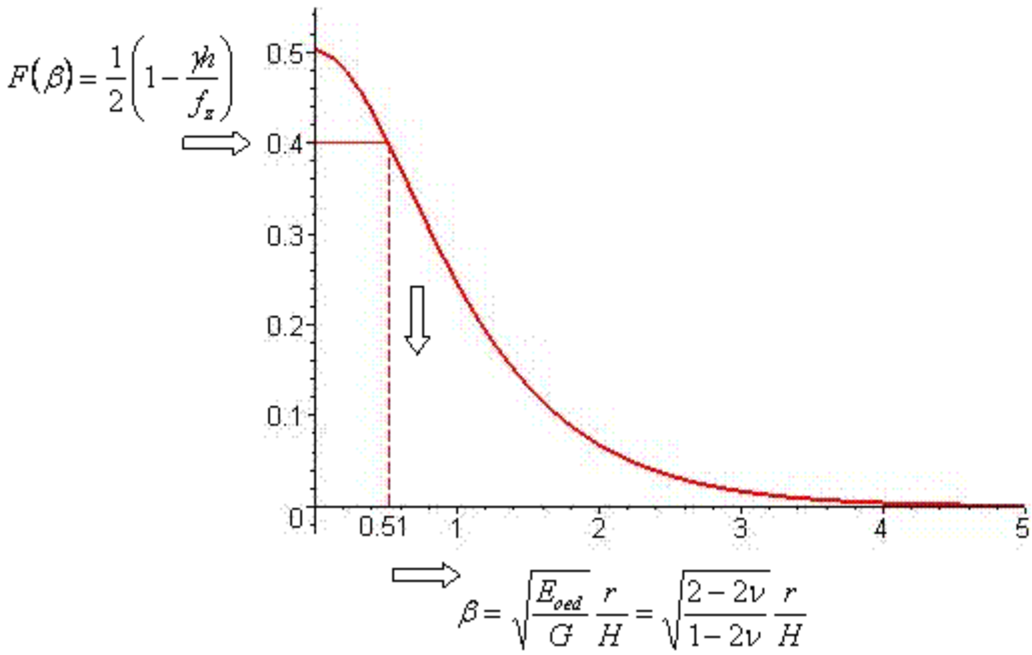
The assumed depth of influence is a variable, which considerably influences the stiffness of soil below the pile heel. It is one of the input parameters for the determination of parameters C_1 and C_2 of the Winkler-Pasternak model. The deeper the influence zone the smaller the stiffness of subsoil. When the depth of influence zone approaches in the limit zero the stiffness of subsoil tends to infinity.

The depth of influence zone depends both on subsoil parameters and magnitude of the applied surcharge, thus on stress below the pile heel. The program assumes that the depth of the influence zone is found in the location, where the stress below the heel equals the geostatic stress. Such an idea is depicted in the following figure:



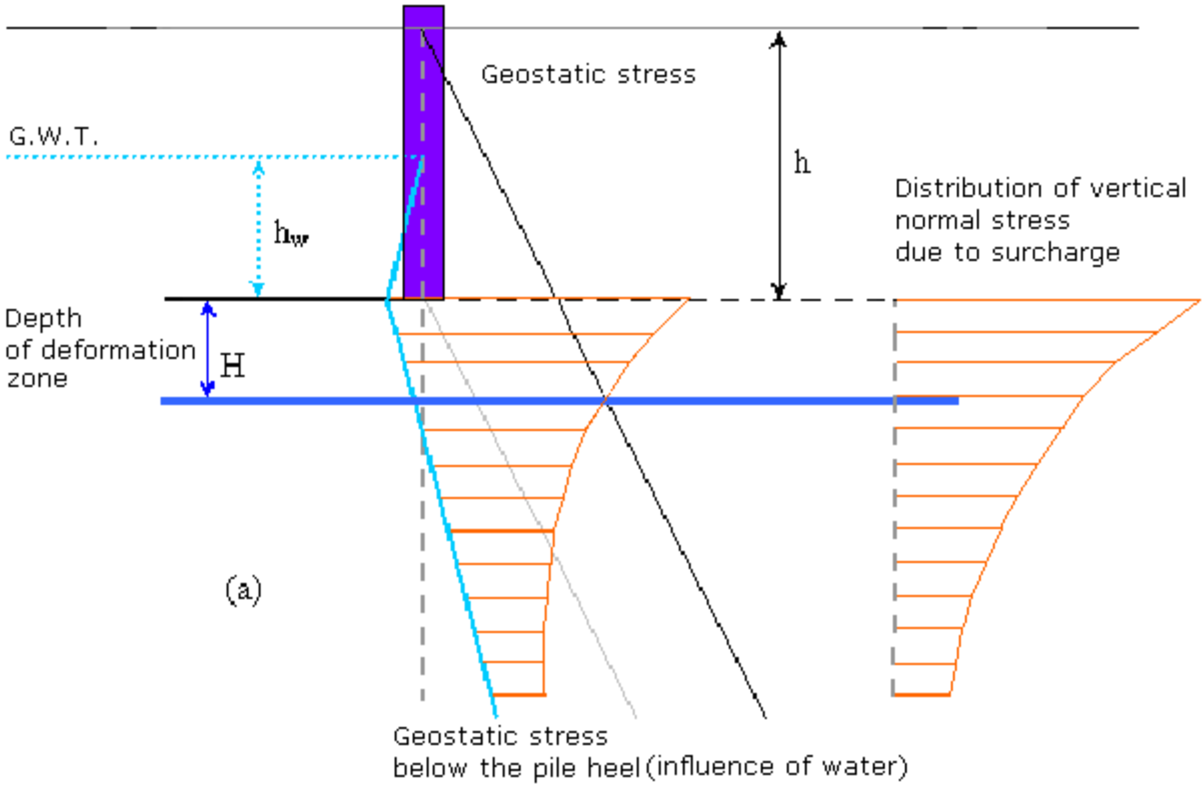
Determination of the depth of influence zone below the pile heel

For digital determination of the depth of influence zone H serves the function $F(\beta)$. Its distribution appears in the figure. This function was derived using the above assumptions and in the program appears in the form of a table. Its application is evident from the following steps. The values of $F(\beta)$ are determined for the current value of stress f_z below the pile heel and for the original geostatic stress γ_h. For this value of $F(\beta)$ we determine the parameter β. This value serves to determine for the actual value Poisson's ratio ν and pile diameter r the corresponding depth of influence zone H.



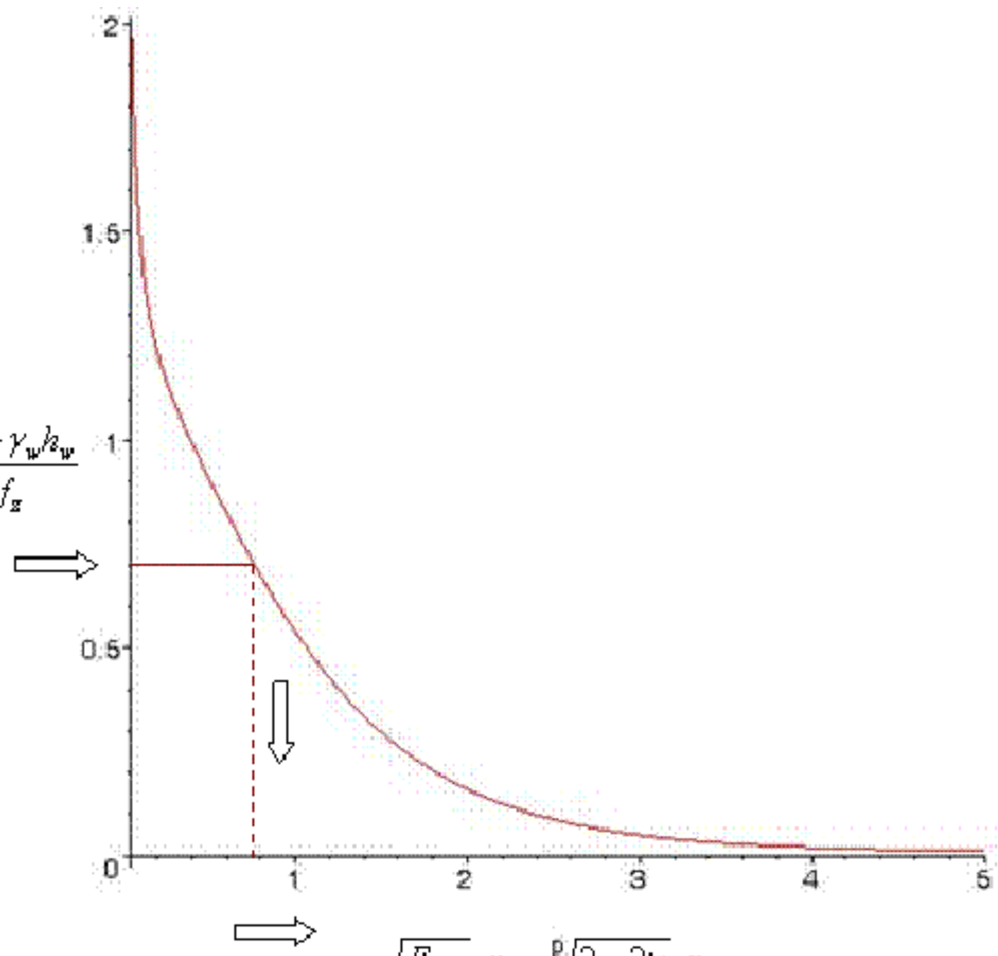
Variation of function $F(\beta)$

The depth of influence zone can be affected by the presence of groundwater. In such s case its determination is outlined in the following figure:



Determination of the depth of influence zone below the pile heel including water

For digital determination of the depth of influence zone $\boldsymbol{H}$ is then used the function $\boldsymbol{G}(\boldsymbol{\beta})$. Its distribution appears in the figure. In the analysis this function is exploited in a similar way as function $\boldsymbol{F}(\boldsymbol{\beta})$. The only difference when determining the values of $\boldsymbol{G}(\boldsymbol{\beta})$ appears in the use of hydrostatic pressure $\gamma_w * h_w$.



$$G(\beta) = 1 - \frac{\gamma h + \gamma_w h_w}{f_z}$$

$$\beta = \sqrt{\frac{E_{oed}}{G}}\frac{r}{H} = \sqrt{\frac{2-2\nu}{1-2\nu}}\frac{r}{H}$$

Variation of function $G(\beta)$

Incompressible Subsoil

At a certain depth below the ground surface it is possible to specify incompressible subsoil. If the pile exceeds this specified depth the spring method cannot be used, because the pile is assumed rigid and therefore no deformation can develop in its surrounding. If there is incompressible subsoil below the pile heel but not deeper than the depth of influence zone below the heel, the depth of influence zone for the stiffness computation is reduced such that the influence zone reaches the incompressible subsoil. This way also the incompressible subsoil below the base increases its stiffness and consequently also the bearing capacity of the pile base. If the incompressible subsoil is found below the depth of the influence zone, it does not influence the analyzed pile.

Negative Skin Friction

Negative skin friction is a phenomenon that arises from a settlement of soil in the vicinity of a pile. The soil deforming around the pile tends to pull the pile downwards thus reducing its bearing capacity for a given pile settlement.

The input parameters for assessing the influence of negative skin friction is the settlement of ground surface w and depth of influence zone of this deformation h . For a uniformly distributed load around the pile the value of w should be measured in the distance equal to three times the pile diameter from its outer face. The value then represents the depth influenced by the ground surface settlement and below which the soil is assumed incompressible with no deformation.

Computation of negative skin friction is carried out first while determining the limit shear forces transmitted by the pile skin T_{lim} . The solution procedure assumes that the soil settlement decreases linearly with depth from the value of w on the ground surface up to 0 at a depth of h . The specific value of the soil settlement is therefore assumed for each level below the ground surface till the depth of h . The forces developed in springs of pile segments due to their deformation are determined and then subtracted from T_{lim} to reduce the bearing capacity of the pile skin.

From the presented theory it is evident that for large settlement w or large depth h the values of T_{lim} may drop down to zero. In extreme cases the negative skin friction may completely eliminate the skin bearing capacity so that the pile is then supported only by the elastic subsoil below the pile heel.

Influence of Technology

The pile bearing capacity is considerably influenced by technological processes applied during construction. The module "**Pile - Spring method**" allows us to specify the technology of pile construction. The mobilized skin friction and the resistance at the pile heel are then reduced with the help of reduction coefficients depending on the selected technology. The values of these coefficients follow from the Dutch standard NEN 6743 Pile foundation.

Apart from technologies offered by the program and corresponding coefficients, the users can input user-defined values.

Shear Resistance on Skin

The shear resistance on pile skin is in the analysis represented by the stiffness of springs supporting individual pile segments. This stiffness is associated with material parameters of the Winkler-Pasternak model C_1 and C_2 . The values of C_1 and C_2 are determined from parameter E_{def} . They depend on the depth of the influence zone, which varies with the pile deformation (settlement). The variability of influence zone is in the analysis determined such that for zero deformation it receives the value of 1**x** the pile diameter and for deformation at the onset of skin failure equals k x the pile diameter, where k is the specified value, resp. 2.5.

The decisive parameter for the determination of magnitudes of C_1 and C_2 is the deformation modulus. Caution must be taken when estimating the value of E_{def} from deformational characteristics of soil using standards. In particular, in case of long piles we are essentially dealing with deep-seated foundations and the soil at the pile heel will certainly experience higher stiffness than that proposed by the standard for spread footings. This holds, particularly for cohesive soils. The most reliable estimates are of course those obtained directly from experimental measurements.

Formulas given below serve to determine the stiffness of springs representing the shear resistance of pile skin as a function of computed parameters of the elastic subsoil. They depend on the shape of cross-section and for the implemented cross-sections they receive the following forms:

Circle:

$$k = 2.\pi.r.\sqrt{C_1 C_2}.\frac{K_1.(\alpha.r)}{K_2.(\alpha.r)}$$

where:	r	-	radius of the pile cross-section
	C_1 , C_2	-	subsoil parameters
	K_1 (α_r), K_2 (α_r)	-	values of the modified Bessel functions

Parameter α attains the value:

$$\alpha = \sqrt{\frac{C_1}{C_2}}$$

Rectangle:

$$k = \left[2.(a+b).\sqrt{C_1 C_2} + 3.C_2\right].k_{red}$$

where a , b are lengths of rectangle edges and C_1 , C_2 are subsoil parameters and k_{red} is the reduction coefficient, which reduces the stiffness with respect to the slenderness of the rectangle.

It receives the following values:

$$k_{red} = 0,6 + 0,4.e^{0,5.\left(1-\frac{b}{a}\right)}$$
$$H \geq 3.a$$
$$k_{red} = 1 - \frac{1 - 0,6 + 0,4.e^{0,5.\left(1-\frac{b}{a}\right)}}{3.a}.H$$
$$H < 3.a$$

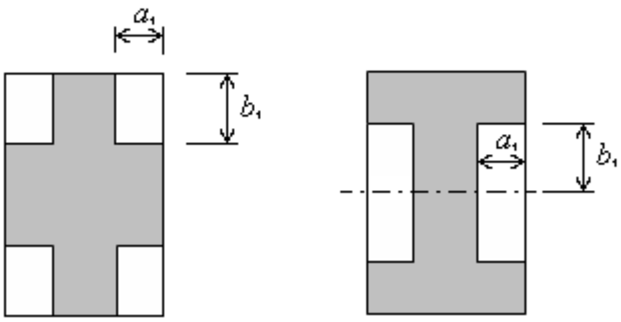
where a is the length of a shorter edge of the rectangle and H is the depth of influence zone.

Cross, **I-section**:

For these cross-sections the stiffness is derived from the stiffness of the rectangular cross-section reduced by subtracting the stiffness corresponding to four "**removed**" parts of the cross-section.

$$k = \left[2.(a+b).\sqrt{C_1.C_2} + 3.C_2\right].k_{red} - 4.\left(1 - e^{-e.\sqrt{a_1^2+b_1^2}}\right).\left[C_1.\frac{a_1.b_1}{9} + C_2\left(\frac{a_1}{3.b_1} + \frac{b_1}{3.a_1}\right)\right]$$

a_1 , b_1 - evident from the following figure



Stiffness of Subsoil Below the Pile Heel

The soil stiffness below the pile heel follows from the value of stiffness of the Winkler model C_1 . The value of C_1 is determined for soil parameters E_{def} and ν at the location of the pile heel. The value of C_1 further depends on the depth of the influence zone beneath the heel.

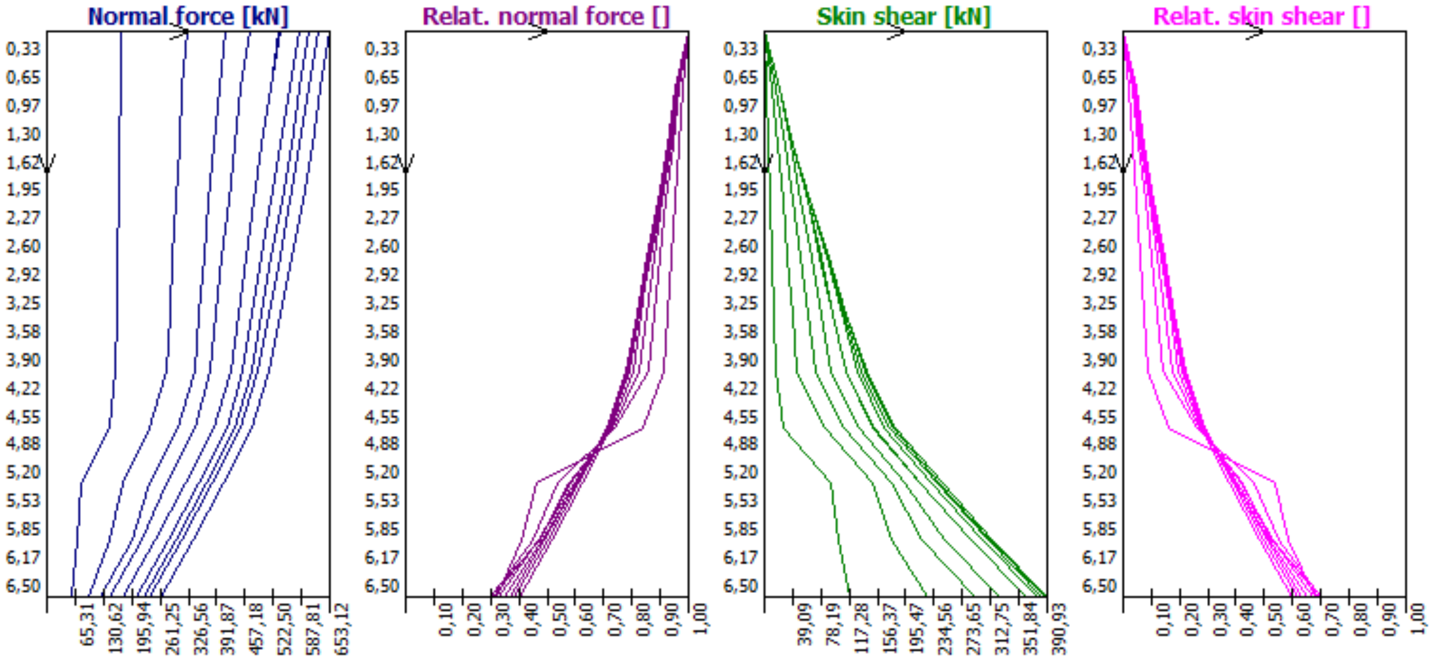
The spring stiffness introduced at the pile base is then provided by:

$$k_r = C_1.A$$

where: A - cross-sectional area at the pile heel

Distributions of Forces Acting on a Pile

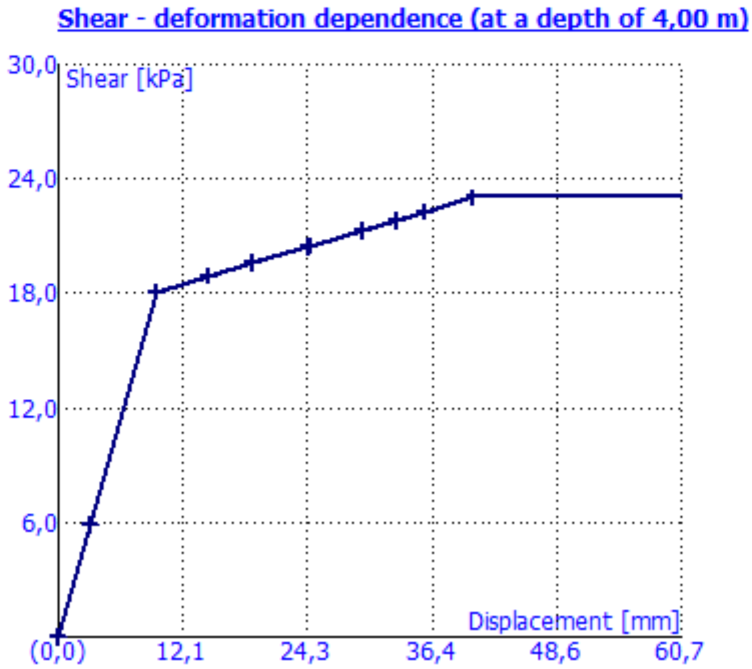
Apart from the load-settlement curve it is also possible to keep track of the distribution of normal force in the pile and the distribution of shear force developed along the pile skin. The normal forces decrease from the top to the bottom as the load is gradually taken by the shear force developed along the pile skin. Unlike the normal force the shear force thus increases from the top to the bottom. Both forces are evaluated in relative values related to the magnitude of the vertical load.



Distribution of internal forces acting on the pile

Dependence of Shear on Deformation

At an arbitrary (selected) depth it is possible to view the distribution of skin friction as a function of displacement (settlement) of a given point of the pile. This graph shows the process of gradual reduction of shear stiffness of pile skin until zero with increasing deformation. This dependency is initially linear, particularly in stage, where the spring force does not exceed the value T_{lim} . When this value is exceeded the spring stiffness starts to gradually decrease manifested by the flattening of the curve.



Dependence of shear on displacement (settlement) of the pile

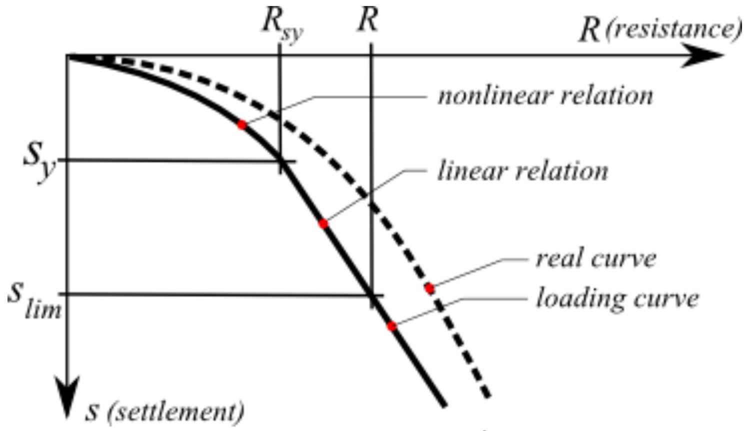
Pile Settlement

Two options are available to perform the pile settlement analysis:

- Nonlinear theory (Masopust)
- Linear theory (Poulos)
- EA-Pfähle

Nonlinear Theory (Masopust)

The nonlinear theory constructs the load-settlement curve assuming that the evolution of settlement as a function of resistance up to full mobilization of skin friction can be represented by a parabola. After that the relationship is linear as displayed in the figure. This method was derived from equations of regression curves constructed on the basis of statistical analysis of the results of static loading tests of piles and for the determination of vertical bearing capacity it employs regression coefficients. Further details are provided herein.



Load-settlement curve of a pile

Literature:

Masopust, J.: Vrtane piloty. 1st edition, Prague, Cenek a Jezek, 1994, 263 p.

Masopust, J., Glisnikova, V.: Zakladani staveb Modul M01. 1st edition, Brno, AN CERM, 2007, 182 p., ISBN 978-80-7204-538-9.

Approach According to Masopust

The load-settlement curve of a single pile is constructed in the following way:

1) The ultimate skin friction q_s is determined as follows:

$$q_s = a - \frac{b}{v_i / d_i}$$

where:

where:	a, b	-	regression coefficient of the specific skin friction
	v_i	-	depth from terrain up to the middle of the i^{th} layer $[m]$
	d_i	-	pile diameter in the i^{th} layer $[m]$

and **pile skin bearing capacity** is provided by:

$$R_s = m_1 . m_2 . \pi . \sum_{i=1}^{n} d_i . h_i . q_{si}$$

where:	m_1	-	load type coefficient
	m_2	-	shaft protection coefficient
	d_i	-	pile diameter in the i^{th} layer $[m]$
	h_i	-	thickness of the i^{th} layer $[m]$
	q_{si}		ultimate skin friction in the i^{th} layer $[MPa]$

2) The pile base bearing capacity q_b follows from:

$$q_b = e - \frac{f}{D / d_b}$$

where:	e, f	-	regression coefficients below the pile base
	D	-	pile length inside soils $[m]$
	d_b	-	pile base diameter $[m]$

3) The proportion of applied load transferred to pile base β is written as:

$$\beta = \frac{q_b}{q_b + 4.\bar{q}_s.D/d_b}$$

where:
- q_b - pile base bearing capacity [MPa]
- $\bar{q}_s$ - weighted average of ultimate skin friction [MPa]
- D - pile length inside soils [m]
- d_b pile base diameter [m]

The **load at the mobilization of skin friction R_{sy}** is then given by:

$$R_{sy} = \frac{R_s}{1-\beta}$$

where:
- R_s - pile skin bearing capacity [N]
- β - proportion of applied load transferred to the pile base [-]

4) The load at the shaft resistance activation (= mobilization of skin friction) R_{sy} is given by:

$$s_y = I_s.\frac{R_{sy}}{0{,}7.d.E_s}$$

where:
- I_s - settlement-influence factor
- R_{sy} - load at the mobilization of skin friction [N]
- d - pile diameter [m]
- E_s - secant modulus of soil along the pile shaft [MPa]

5) The load at the pile base for the prescribed settlement (for limiting settlement of $25\ mm$) follows from:

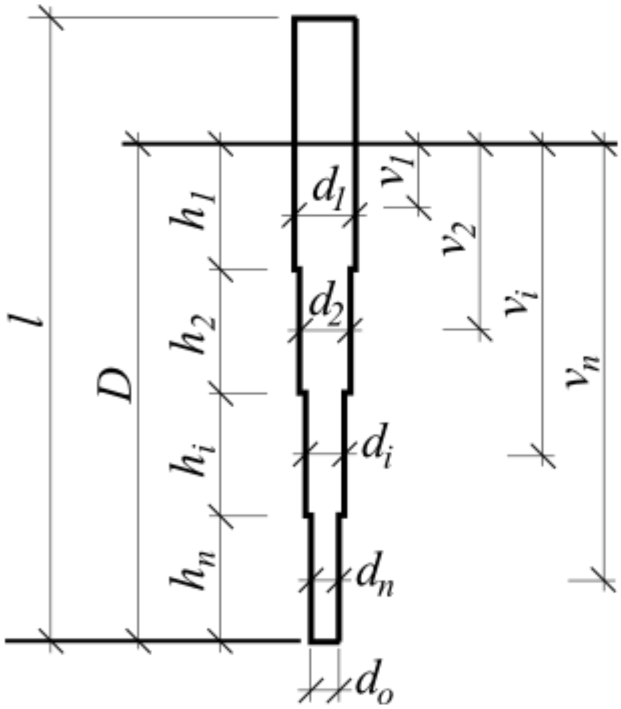
$$R_{b,lim} = \beta.R_{sy}.\frac{s_{lim}}{s_y}$$

where:
- β - proportion of applied load transferred to the pile base [-]
- R_{sy} - load at the mobilization of skin friction [N]
- s_{lim} - limit settlement (usually prescribed as $25\ mm$) [m]
- s_y settlement at shaft resistance activation [m]

and the pile resistance attributed to a given **limit settlement s_{lim}** is then provided by:

$$R_c = R_{b,lim} + R_s$$

where:
- $R_{b,lim}$ - load on the pile base for prescribed settlement [N]
- R_s - pile shaft resistance [N]



Approach according to Masopust

Literature:

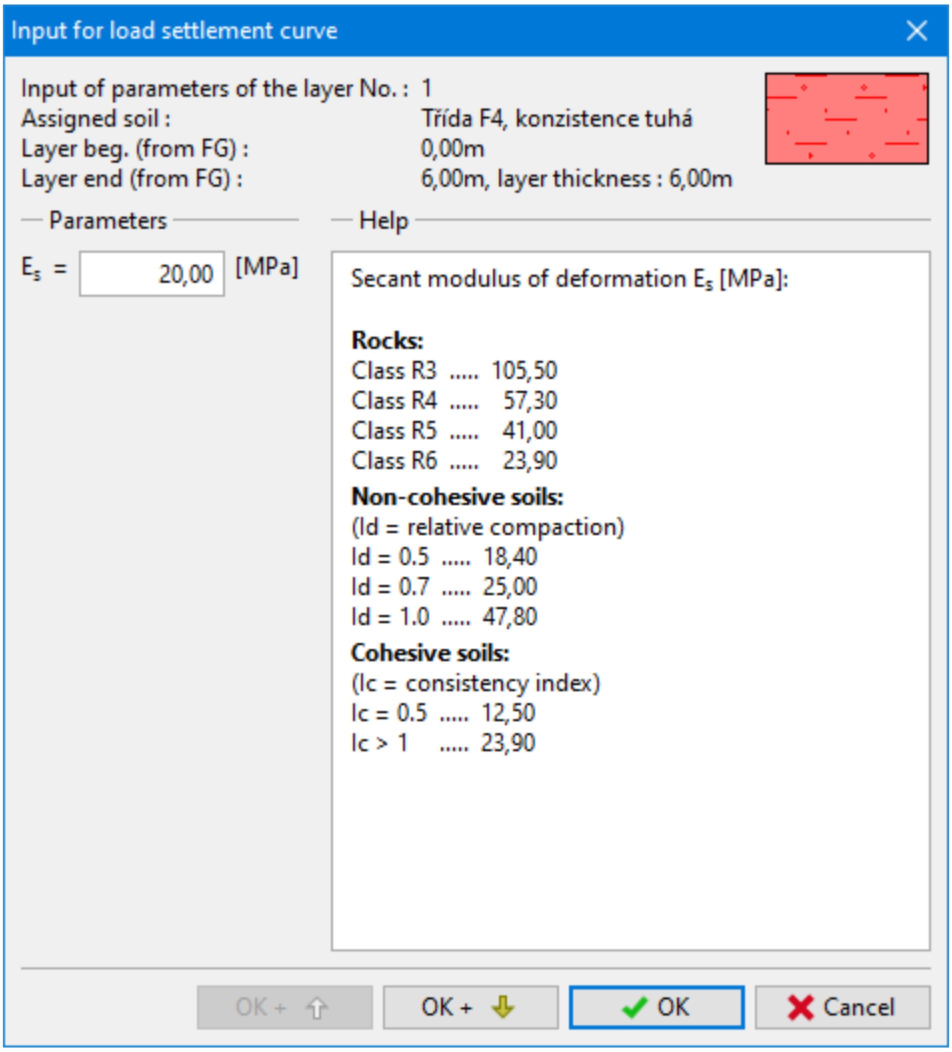
Masopust, J.: Vrtane piloty. 1st edition, Prague, Cenek a Jezek, 1994, 263 p.

Masopust, J., Glisnikova, V.: Zakladani staveb Modul M01. 1st edition, Brno, AN CERM, 2007, 182 p., ISBN 978-80-7204-538-9.

Regression Coefficients

The specific skin friction depends on regression coefficients a , b . The pile base resistance (at the full mobilization of skin friction) depends on the regression coefficients e , f . The values of these regression coefficients were derived from equations of regression curves constructed based on statistical analysis of the results of approximately 350 static loading tests of piles.

The dialog window for entering regression coefficients can be displayed in the frame "Settlement" using the "**Edit a , b** ", "**Edit e , f**" buttons. When editing the dialog window displays the recommended values of regression coefficients for various types of soils and rocks.



*Dialog window "Input for load settlement curve" - input of regression coefficients **a, b (e, f)***

Coefficients m1, m2

Load type coefficient m_1 :

-	for service load	0.7
-	for extreme load	1.0

Shaft protection coefficient m_2 :

-	for concreting in a dry shaft or underwater	1.0
-	for concreting with bentonite slurry	0.9
-	for PVC sheet pile protection (thickness over 0.7 mm)	0.7
-	for sheet pile protection and B system mesh	0.5
-	for steel casing tube protection	0.15

Literature:

Masopust, J., Glisnikova, V.: Zakladani staveb Modul M01. 1st edition, Brno, AN CERM, 2007, 182 p., ISBN 978-80-7204-538-9.

Secant Modulus of Soil Es

Value of secant modulus of soil E_s depends on pile diameter d and depth of individual layers of the soil h_i. The values of this modulus should be determined experimentally from pile-load tests.

For cohesionless soils its value also depends on the index of relative density I_d, for cohesive soils this value depends on the index of consistency I_c. The value of the secant modulus of soil E_s increases with depth.

Issue 60 in [2] states that the cumulative value of this module (applicable to all soil types along the shaft and the base of the large-diameter pile) is given by the following equation:

$$E_s = I_s \frac{Q}{s\,d}$$

where: I_s - settlement-influence factor [-]

d - pile diameter [m]

Q - relevant value of load (force) measured during the pile-loading test [N]

s - relevant value of pile settlement measured during the pile-loading test [m]

Values of secant modules E_{si} for various types of soils and different pile diameters and depths of piles are shown in the following tables [3]. Intermediate values of secant modulus of soil E_s can be interpolated linearly.

Secant modulus of soil E_s located in rocks and weak rocks

h (m)	d (m)								
	0.6			1.0			1.5		
	R3	**R4**	**R5**	**R3**	**R4**	**R5**	**R3**	**R4**	**R5**
1.5	50.3	28.2	20.2	72.3	35.0	24.7	85.5	33.5	22.3
3.0	64.5	43.1	30.8	105.5	57.3	41.0	138.3	58.8	41.2
5.0	-	58.2	41.3	-	75.3	54.8	-	87.9	63.7
10.0	-	87.5	61.6	-	114.5	83.2	-	133.0	97.0

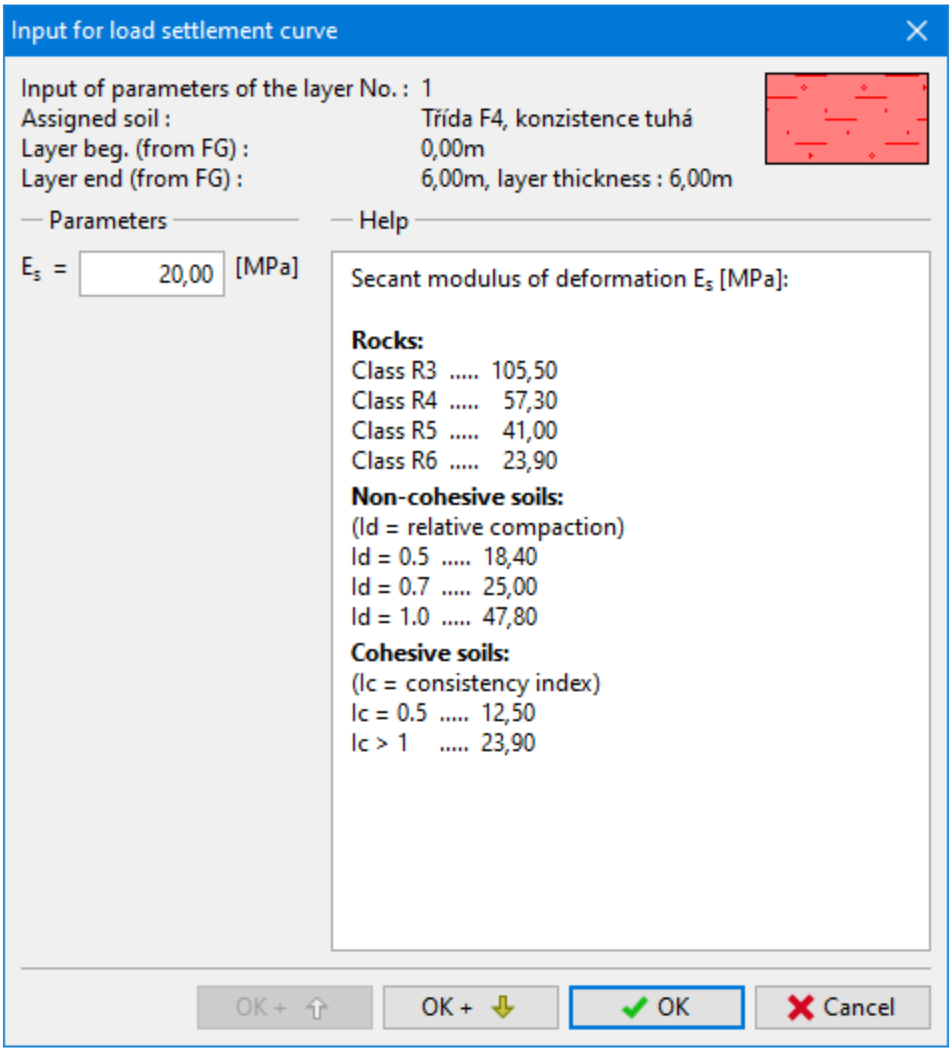
Secant modulus of soil E_s located in cohesionless soils

h (m)	d (m)								
	0.6			1.0			1.5		
	I_d								
	0.5	0.7	0.9	0.5	0.7	0.9	0.5	0.7	0.9
1.5	11.0	13.7	28.3	12.8	15.8	30.6	13.0	15.3	29.0
3.0	15.5	20.2	44.5	18.4	25.0	47.8	19.4	24.5	52.5
5.0	18.8	26.6	56.1	22.8	32.5	69.1	24.5	36.0	78.2
10.0	23.8	36.6	72.1	29.8	47.8	93.4	32.6	54.0	107.3

Secant modulus of soil E_s located in cohesive soils

h (m)	d (m)					
	0.6		1.0		1.5	
	I_c					
	0.5	≥ 1.0	0.5	≥ 1.0	0.5	≥ 1.0
1.5	6.9	13.2	7.9	13.4	8.6	12.3
3.0	10.0	22.0	12.5	23.9	13.7	23.0
5.0	12.5	31.2	15.9	35.4	18.4	36.7
10.0	15.5	44.3	21.3	51.3	24.6	57.4

The dialog window for entering the secant modulus E_s can be displayed in the frame "Settlement" using the "**Edit** E_s" button. When editing the dialog window displays the **recommended values** of the secant modulus of soil E_s according to the pile diameter d and depth of the center of soil layers h_i.



Dialog window "Input for load settlement curve" - Secant modulus of soil E_s [MPa]

Literature:

[1] CSN 73 1002: Pilotove zaklady. Praha, UNM, 1988, 28 p.

[2] CSN 73 1004: Velkoprumerove piloty. Praha, UNM, 1981, 56 p.

[3] Masopust, J., Glisnikova, V.: Zakladani staveb Modul M01. 1st edition, Brno, AN CERM, 2007, 182 p., ISBN 978-80-7204-538-9.

[4] Pochman, R., Simek, J.: Pilotove zaklady - Komentar k CSN 73 1002. 1st edition, Prague, Vydavatelstvi norem, 1989, 80 p.

Settlement-Influence Factor Is

The settlement-influence factor depends on the depth of the pile below the surface of a resistant layer D and the pile diameter d. The settlement-influence factor of pile I_s is given by:

$$I_s = I_0 . R_k . R_h$$

where:

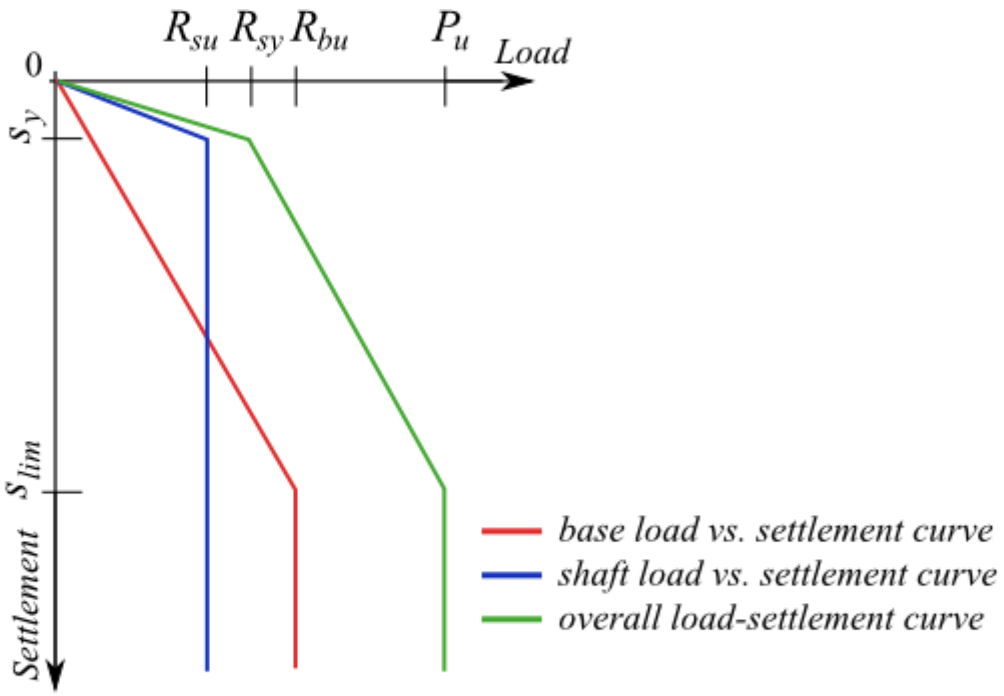
where:	I_0	-	base settlement-influence factor
	R_k	-	correction factor for pile compressibility
	R_h	-	correction factor for the finite depth of layer on a rigid base

Linear Theory (Poulos)

Analysis of the load-settlement curve of a single pile or pile group is based on the solution described in the book **Pile Foundations Analysis and Design** (H. G. Poulos et. E. H. Davis, 1980) and is based on the theory of elasticity and modifications attributed to in-situ measurements. Foundation soil is therefore characterized by the modulus of elasticity E and by the Poisson's ratio ν. This method allows the construction of the load-settlement curve for pile foundations (single pile, pile group).

The basic input parameters of the analysis are pile base bearing capacity R_{bu} and pile skin bearing capacity R_{su}.

Ultimate bearing capacity of pile foundation, respectively ultimate load is given by equation $P_u = R_{su} + R_{bu}$. These values are obtained by the program from the analysis of the vertical bearing capacity of a single pile or pile group and it depends on the selected method of analysis. All partial factors of the analysis are assumed equal to 1.0 so that the resulting resistance is greater than the one obtained from the actual bearing capacity analysis.



Constructing of the load-settlement curve of pile foundation according to Poulos method

Literature:

Poulos, H. G. et. Davis, E. H.: Pile Foundations Analysis and Design. New York: John Wiley and Sons, 1980, chapter 5, pp. 71 - 108.

Settlement of Piles According to Poulos

The basic assumption of the analysis is the determination of the **load at the shaft resistance activation** R_{sy}. At this point the shaft resistance no longer increases, the further load is taken by the pile base only. This force is given by the equation:

$$R_{sy} = \frac{R_s}{1-\beta}$$

where: R_s - pile shaft resistance $[N]$

β - proportion of applied load transferred to the pile base [-]

The **proportion of applied load transferred to pile base** β is provided by:

$$\beta = \beta_0 . C_k . C_b . C_v$$

where: β_0 - base-load proportion for incompressible pile

C_k - correction factor for pile compressibility

C_v - correction factor for Poisson's ratio of soil

C_b - correction factor for the stiffness of bearing stratum

The corresponding value of **settlement** s_y **at shaft resistance activation** R_{sy} is given by:

$$s_y = \frac{I . R_{sy}}{d . E_s}$$

where: I - settlement-influence factor [-]

E_s - average value of secant modulus of soil along the pile shaft $[MPa]$

d - pile diameter $[m]$

R_{sy} - load at the shaft resistance activation $[N]$

The **settlement-influence factor** I is given by:

$$I = I_0 R_k R_b R_v$$

where: I_0 - basic settlement-influence factor

R_k - correction factor for pile compressibility

R_b - correction factor for the stiffness of bearing stratum

R_v - correction factor for Poisson's ratio of soil

The **overall limit settlement** s_{lim} is provided by:

$$s_{lim} = \frac{I.R_{bu}}{\beta.d.E_s}$$

where: I - settlement-influence factor [-]

R_{bu} - ultimate pile base bearing capacity [N]

β - proportion of applied load transferred to the pile base [-]

d - pile diameter [m]

E_s average value of secant modulus of soil along the pile shaft [MPa]

Literature:

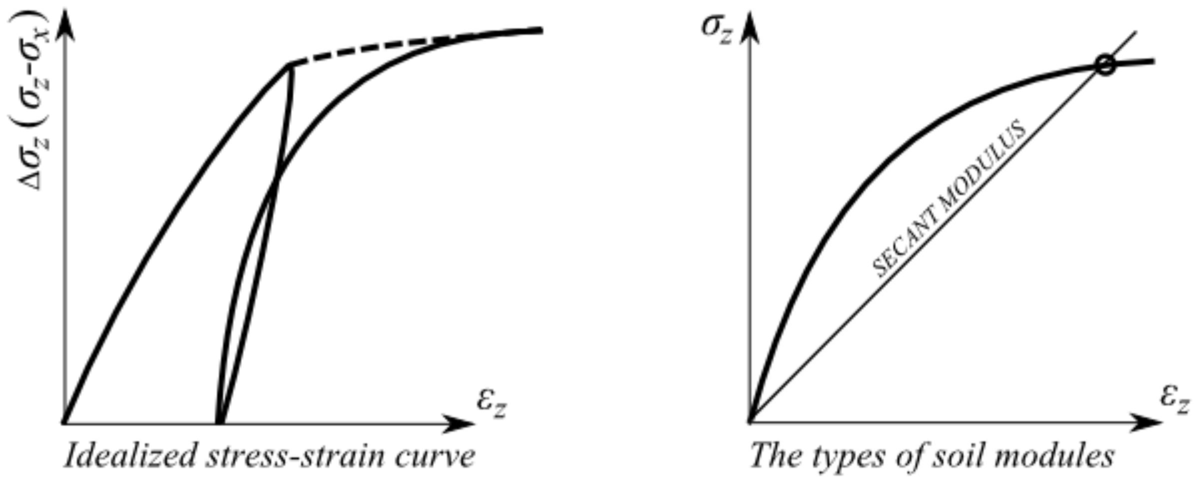
Poulos, H. G. et. Davis, E. H.: Pile Foundations Analysis and Design. New York: John Wiley and Sons, 1980, chapters 5.3 and 5.4, pp. 84 - 100.

Secant Modulus of Soil Es

In the literature, it appears double marking of soil modulus E_s. According to Poulos et. Davis (1980) this parameter is referred to as the **modulus elasticity of soil** (**Young's modulus**), while Briaud (2001) and Gopal Ranjan (2000) named this parameter as the **secant modulus of soil**. Both titles of this modulus E_s has the same meaning. However, the soil behaves elastically only in the field of small strains (generally it is a heterogeneous material), and thus is more appropriate to speak rather about the **secant modulus of soil** E_s.

Modulus of elasticity of soil E is obtained from the deviator stress-axial strain curve. The undrained modulus, E_u is obtained from the undrained triaxial test data while the drained modulus E_d is obtained from the drained test conditions.

At the initial stage of the stress-strain curve is nearly linear dependence, but the elastic strain of soils is very small, due to the overall value of the strain. There are defined several types of modules - **tangent modulus of soil**, **secant modulus of soil,** and **initial tangent modulus**. The introduction of this simplifying assumption is possible to use the theory of elasticity for detecting of stress-strain state in soils.



Distribution of idealized stress-strain curve and determination of individual types of soil modules

Secant modulus of soil E_s is defined as the ratio of the difference in deviator of normal stress to the corresponding axial strain of soil according to the following equation:

$$E_s = \frac{\Delta(\sigma_1 - \sigma_3)}{\Delta\,\varepsilon_E}$$

Lambe et. Whitman (1969) says that the elastic modulus for soil is usually the secant modulus from zero deviators of normal stress to a deviator stress equal to one-half or one-third of the peak deviator stress.

The secant modulus E_s decreases as the strain level increases because the stress-strain curve has a downward curvature. There are three means of obtaining this parameter:

- laboratory triaxial tests (from a calculation based on the tangent modulus of soil)
- pile-load test
- empirical correlations based on previous experience

The typical range of values for the static stress-strain (secant) modulus E_s for selected soils - field values depend on stress history, water content, density (Gopal Ranjan et. Rao, 2000):

Type of soil	**Consistency or Density of soil**	**Modulus** E_s $[MPa]$
Silt	Very soft	0.2 - 2
Clay	Very soft	2 - 15
	Soft	5 - 25
	Firm, medium	15 - 50
	Hard	50 - 100
	Sandy	25 - 250
Loess sand	Silty	7 - 21
	Loose	10 - 24
	Dense	48 - 80
Sand and gravel	Loose	50 - 145
	Dense	100 - 190

Literature:

Briaud, J.-L.: Introduction to Soil Moduli. Geotechnical News, June 2001, BiTech Publishers Ltd, Richmond, B.C., Canada.

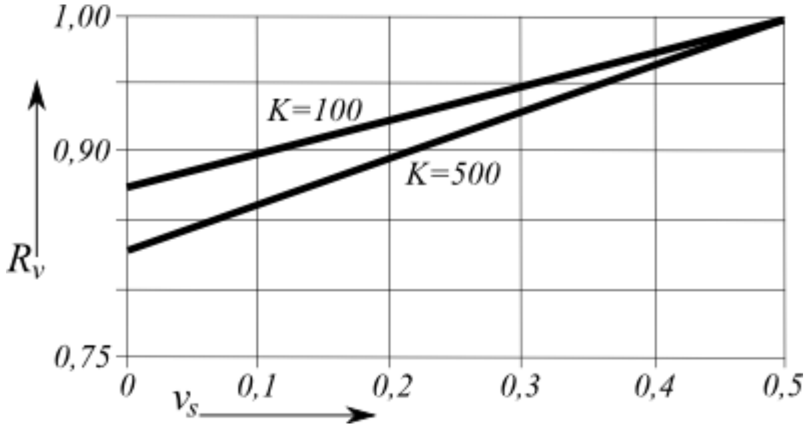
Gopal Ranjan et. A. S. R. Rao: Basic and Applied Soil Mechanics. New Age International, 2000, chapter 10.11, pp. 328 - 330. ISBN: 8122412238, 9788122412239.

Lambe, T. W. et. Whitman, V. R.: Soil Mechanics. New York: John Wiley and Sons, 1969, 576 p. ISBN: 978-0-471-51192-2.

Poulos, H. G. et. Davis, E. H.: Pile Foundations Analysis and Design. New York: John Wiley and Sons, 1980, chapter 5.5, pp. 101 - 104.

Correction Factor for Soil Poisson's Ratio Rv

The correction factor for the influence of Poisson's ratio R_v accounts for the influence of reduction of Poisson's ratio v of soils surrounding the pile on the values of pile settlement for constant modulus of elasticity of these soils. These values are generally presented as a function of Poisson's ratio of the surrounding soil v_s for various pile-stiffness factor K. These graphs are implemented in the program in a digital format.



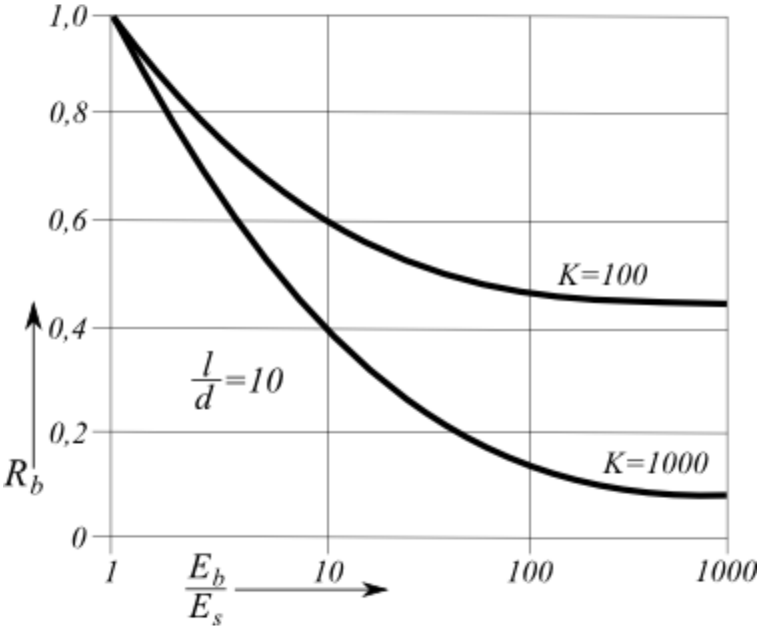
Poisson's ratio correction factor for settlement R_v

Literature:

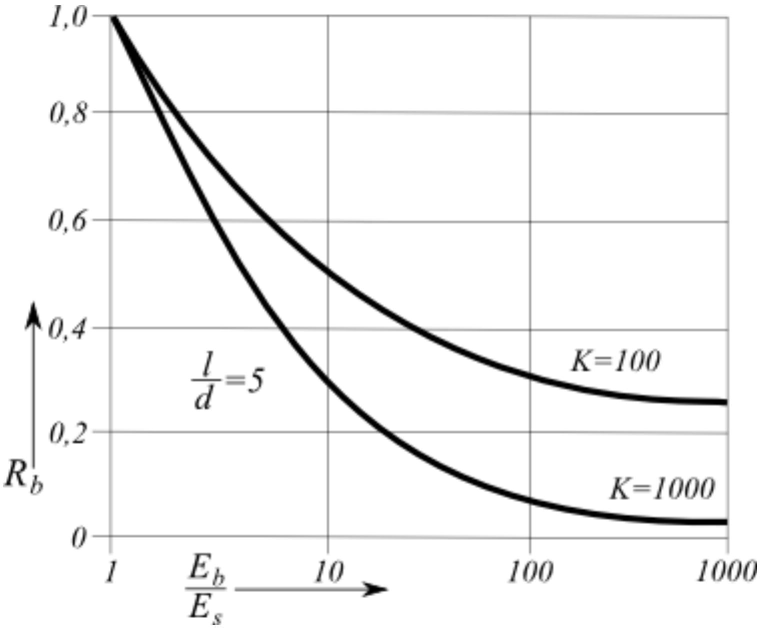
Poulos, H. G. et. Davis, E. H.: Pile Foundations Analysis and Design. New York: John Wiley and Sons, 1980, chapter 5.3.3, pp. 89 (figure 5.21).

Correction Factor for Stiffness of Bearing Stratum Rb

The values of the correction factor R_b are generally presented as a function of the ratio of modulus of elasticity of the soil at the pile base and secant modulus of the surrounding soil (E_b/E_s) for various pile-stiffness factor K and various pile length to pile diameter ratios (l/d). These graphs are implemented in the program in a digital format.



Base modulus correction factor for settlement R_b *(L/d = 10)*



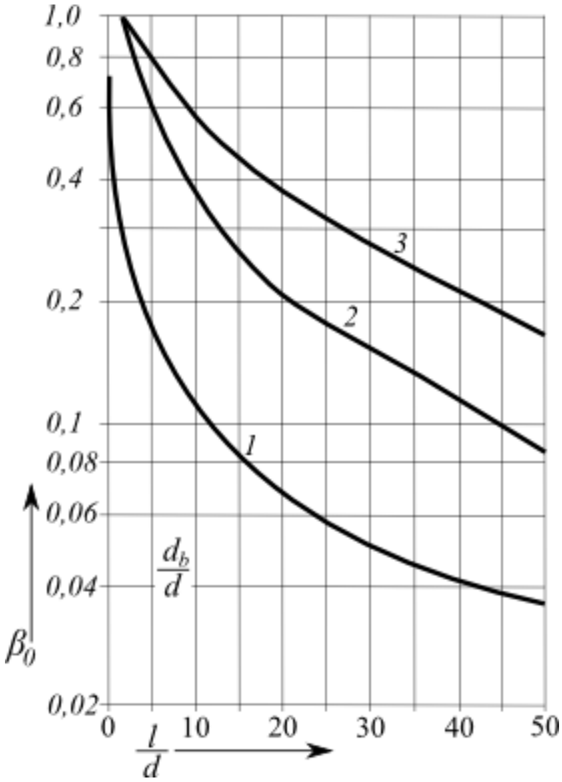
Base modulus correction factor for settlement R_b *(L/d = 5)*

Literature:

Poulos, H. G. et. Davis, E. H.: Pile Foundations Analysis and Design. New York: John Wiley and Sons, 1980, chapter 5.3.3, pp. 90 (figure 5.22).

Base-Load Proportion for Incompressible Pile BETAo

The base-load proportion for incompressible pile β_0 represents the influence of compression of elastic half-space, which adopts the load transferred by the pile from incompressible soil. The values of this coefficient are generally presented as a function of the pile length to pile diameter ratio (l/d) for various pile base diameter to pile diameter ratios (d_b/d). These graphs are implemented in the program in a digital format.



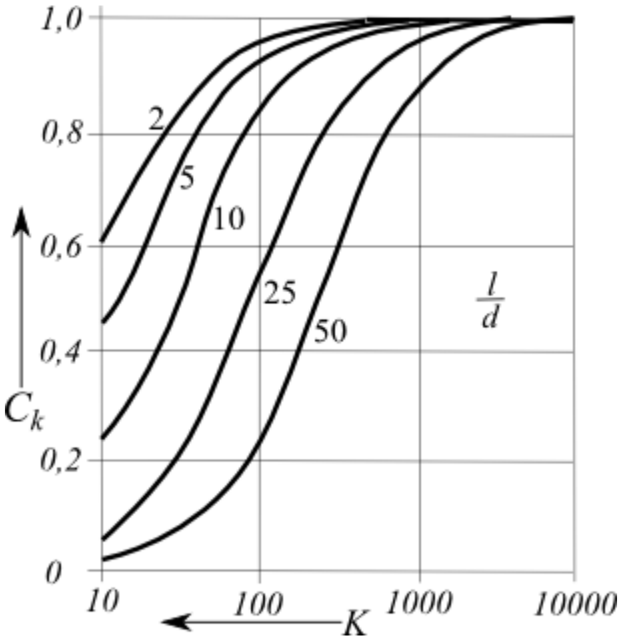
Proportion of baseload β_0

Literature:

Poulos, H. G. et. Davis, E. H.: Pile Foundations Analysis and Design. New York: John Wiley and Sons, 1980, chapter 5.3.3, pp. 86 (figure 5.11).

Correction Factor for Pile Compressibility Ck

The values of the factor C_k are generally presented as a function of the pile-stiffness factor K for various pile length to pile diameter ratios (l/d). These graphs are implemented in the program in a digital format.



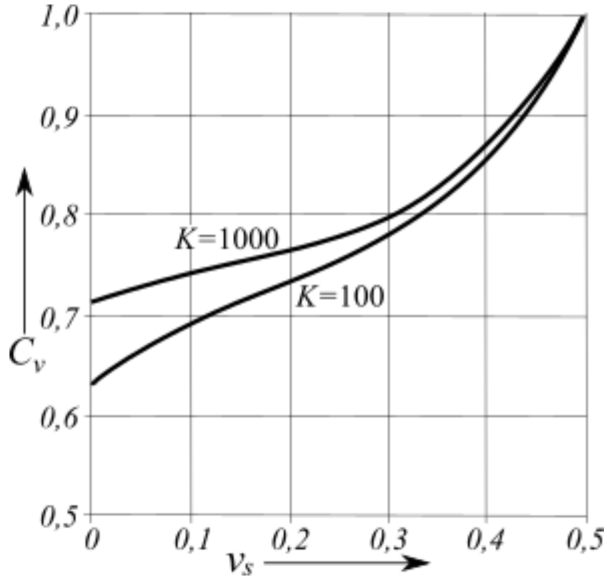
Compressibility correction factor for baseload C_k

Literature:

Poulos, H. G. et. Davis, E. H.: Pile Foundations Analysis and Design. New York: John Wiley and Sons, 1980, chapter 5.3.3, pp. 86 (figure 5.12).

Correction Factor for Poisson's Ratio of Soil Cv

The values of the factor C_v are generally presented as a function of Poisson's ratio of the surrounding soil v_s for various pile-stiffness factor K. These graphs are implemented in the program in a digital format.



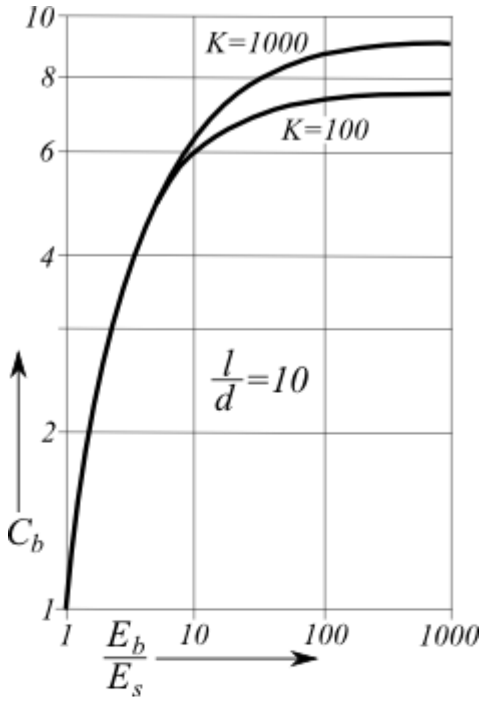
Poisson's ratio correction factor for baseload C_v

Literature:

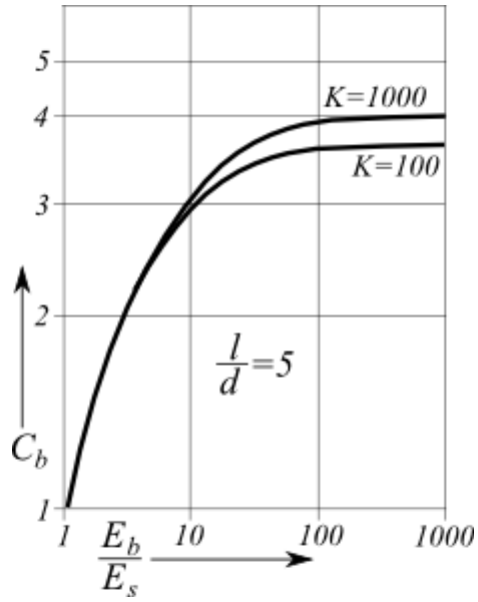
Poulos, H. G. et. Davis, E. H.: Pile Foundations Analysis and Design. New York: John Wiley and Sons, 1980, chapter 5.3.3, pp. 86 (figure 5.13).

Correction Factor for Stiffness of Bearing Stratum Cb

The values of the factor C_b are generally presented as a function of the ratio of modulus of elasticity of soil at the pile base and secant modulus of the surrounding soil (E_b / E_s) for various pile-stiffness factor K and various pile length to pile diameter ratios (l / d). These graphs are implemented in the program in a digital format.



Base modulus correction factor for baseload C_b (L/d = 10)



Base modulus correction factor for baseload C_b (L/d = 5)

Literature:

Poulos, H. G. et. Davis, E. H.: Pile Foundations Analysis and Design. New York: John Wiley and Sons, 1980, chapter 5.3.3, pp. 87 - 88 (figure 5.14).

Pile-Stiffness Factor K

The pile-stiffness factor is defined as:

$$K = \frac{E_p . R_a}{E_s}$$

where: E_p - elastic modulus of pile material [MPa]

E_s - average value of secant modulus of soil along the pile shaft [MPa]

R_a - ratio of an area of pile section to the area bounded by pile outer-circumference [-]

$$R_a = \frac{A_1}{A_2}$$

where: A_1 - average area of cross-section of a pile [m^2]

A_2 - area of pile shaft [m^2]

(for stiff piles $R_a = 1$)

EA-Pfähle

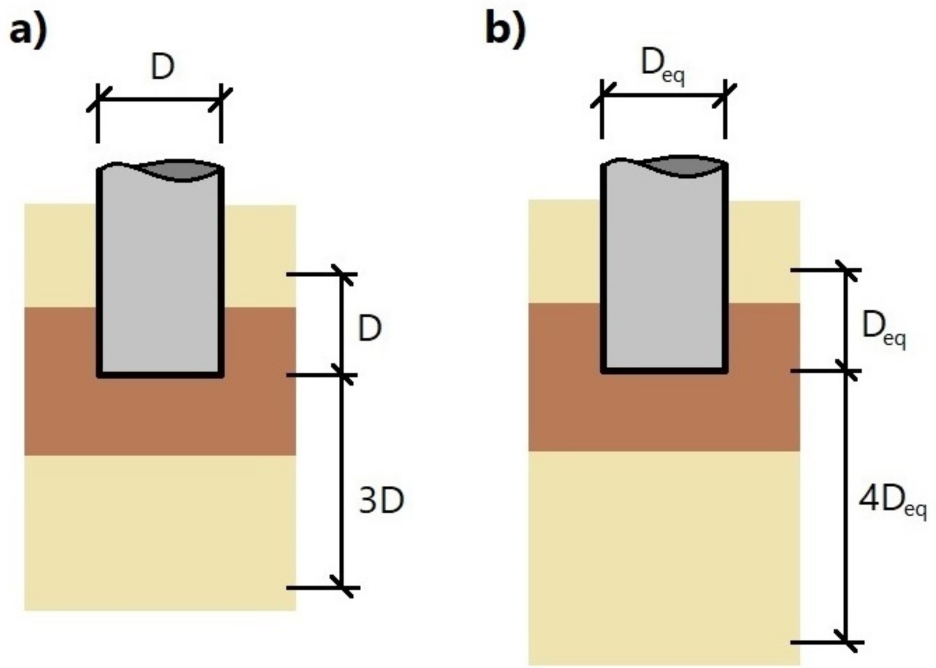
The calculation of load-settlement curve of **single pile** is based on the solution described in **EA-Pfähle** manual.

The main input for the calculation are empirical values of shaft resistance q_s and base resistance q_b , that are based on the analysis of the results of static loading tests of piles.

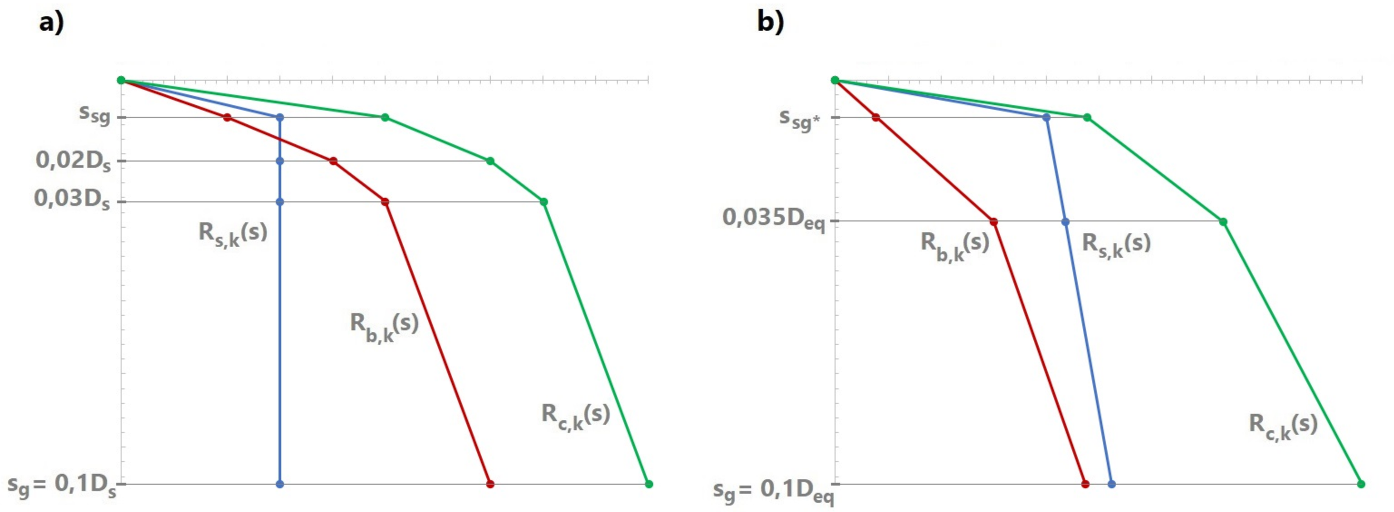
The resistance is determined according to **pile technology** (bored, driven, CFA) and according to the **soil type** (cohesive, non-cohesive). The values can be calculated automatically or entered in the Settlement frame.

In case of **automatic calculation**, the values of q_s are automatically assigned according to soil in the layer. For q_b values, a **fictitious profile** is created at the pile base level. The resultant resistance is calculated as a weighted average of q_b values from the individual layers.

Further details are provided herein.



Fictitious profile according to pile technology - a) bored and CFA piles, b) driven piles



Load-settlement curve according to EA-Pfähle - a) bored and CFA piles, b) driven piles

Literature:

German Geotechnical Society: Recommendations on Piling (EA-Pfähle). Berlin: Ernst & Sohn, 2012.

Analysis According to EA-Pfähle

The load-settlement curve of a single pile is constructed in the following way:

1) The shaft bearing capacity $R_{s,k}$ is determined as follows:

- for **bored** and **CFA** piles

$$R_{s,k}(s_{sg}) = \sum_{i=1}^{n} q_{s,k,i}.A_{s,i}$$

- for **driven** piles

$$R_{s,k}(s_{sg^*}) = \sum_{i=1}^{n} \eta_s.q_{s,k,i}.A_{s,i}$$

where: $q_{s,k,i}$ - shaft resistance in i -th layer [kPa]

$A_{s,i}$	-	shaft area in i -th layer [m^2]
s_{sg} ,s_{sg*}	-	settlement at shaft resistance activation
η_s	-	shaft correction factor

subsequently, the settlement condition is assessed:

- for **bored** and **CFA** piles

$$s_{sg} = 0,5.R_{s,k}(s_{sg}) + 0,5 \leq 30mm$$

- for **driven** piles

$$s_{sg^*} = 0,5.R_{s,k}(s_{sg^*}) \leq 10mm$$

where:

s_{sg} ,s_{sg*}	-	settlement at shaft resistance activation [mm]
$R_{s,k}$	-	shaft bearing capacity for settlement s_{sg} resp. s_{sg*} [MN]

If condition does not satisfy, the load curve cannot be constructed.

2) The base bearing capacity $R_{b,k}$ is calculated in required levels of s/D :

- for **bored** and **CFA** piles

$$R_{b,k} = q_{b,k}.A_b$$

- for **driven** piles

$$R_{b,k} = \eta_b.q_{b,k}.A_b$$

where:

$q_{b,k}$	-	base resistance in level s/D resp. s/D_{eq} [kPa]
A_b	-	base area [m^2]
η_b	-	base correction factor
s	-	settlement
D ,D_{eq}	-	base diameter

3) The total pile resistance $R_{c,k}$ is given by:

$$R_{c,k} = R_{s,k}(s_g) + R_{b,k}(s_g)$$

where:

$R_{s,k}$	-	shaft bearing capacity for limit settlement s_g [kPa]
$R_{b,k}$	-	base bearing capacity for limit settlement s_g [kPa]
s_g	-	limit settlement 0,1D resp. 0,1D_{eq}

The pile bearing capacity is assessed for **compressive loads**. The pile loaded by **tensile loads** is not verified.

The pile verification is carried out according the theory of limit states:

$$R_{c,d} = \frac{Rc,k}{\gamma_t} \geq V_d + W_p$$

where:

$R_{c,d}$	-	design value of total bearing capacity
$R_{c,k}$	-	characteristic value of total baring capacity
γ_t	-	partial factor on total resistance of pile
V_d	-	extreme vertical load acting on a pile
W_p	-	pile self-weight

Literature:

German Geotechnical Society: Recommendations on Piling (EA-Pfähle). Berlin: Ernst & Sohn, 2012.

Base Resistance

Determination of base resistance q_b should be part of the geological survey.

Recommended values of resistance q_b [kPa] can be found in professional literature for specific soils and structure types:

- **for bored piles**:

	q_b	
	non-cohesive soil	cohesive soil

	q_c			c_u		
s/D	**7,5**	**15**	**25**	**100**	**150**	**250**
0,02	550-800	1050-1400	1750-2300	350-450	600-750	950-1200
0,03	700-1050	1350-1800	2250-2950	450-550	700-900	1200-1450
s_g **=0,10**	1600-2300	3000-4000	4000-5300	800-1000	1200-1500	1600-2000

- **for driven piles**:

	q_b					
	non-cohesive soil			cohesive soil		
	q_c			c_u		
s/D_{eq}	**7,5**	**15**	**25**	**100**	**150**	**250**
0,035	2200-5000	4000-6500	4500-7500	350-450	550-700	800-950
s_g **=0,10**	4200-6000	7600-10200	8750-11500	600-750	850-1100	1150-1500

where:
- q_c - cone penetration resistance, CPT $[MPa]$
- c_u - total cohesion of the soil $[kPa]$
- s - pile settlement
- s_g - limit settlement
- D, D_{eq} - pile diameter

Intermediate values may be linearly interpolated.

Literature:

EA-Pfähle, ISBN: 978-3-433-03005-9.

Shaft Resistance

Determination of shaft resistance q_s should be part of the geological survey.

Recommended values of resistance q_s $[kPa]$ can be found in professional literature for specific soils and structure types:

- **for bored piles**:

q_s					
non-cohesive soil			cohesive soil		
q_c			c_u		
7,5	**15**	**25**	**60**	**150**	**250**
55-80	105-140	130-170	30-40	50-65	65-85

- **for driven piles**:

	q_s					
	non-cohesive soil			cohesive soil		
	q_c			c_u		
s/D_{eq}	**7,5**	**15**	**25**	**60**	**150**	**250**
s_{sg*}	30-40	65-90	85-120	20-30	35-50	45-65
s_g **=0,1**D_{eq}	40-60	95-125	125-160	20-35	40-60	55-80

where:
- q_c - cone penetration resistance, CPT $[MPa]$
- c_u - total cohesion of the soil $[kPa]$
- s_{sg*} - settlement at shaft resistance activation
- s_g - limit settlement
- D_{eq} - pile diameter

Intermediate values may be linearly interpolated.

Literature:

EA-Pfähle, ISBN: 978-3-433-03005

Correction Factors for Driven Piles

The correction factors for **base** η_b and **shaft** η_s may be calculated or input for **driven piles** in the "Geometry" frame.

Pile type	η_b [-]	η_s [-]
Reinforced concrete and prestressed concrete	1,00	1,00
Steel tube - open (0,30 m ≤ D_b ≤ 1,60 m)	$0{,}95.e^{-1{,}2Db}$	$1{,}1.e^{-0{,}63Db}$
Steel tube - closed (D_b ≤ 0,80 m)	0,80	0,60

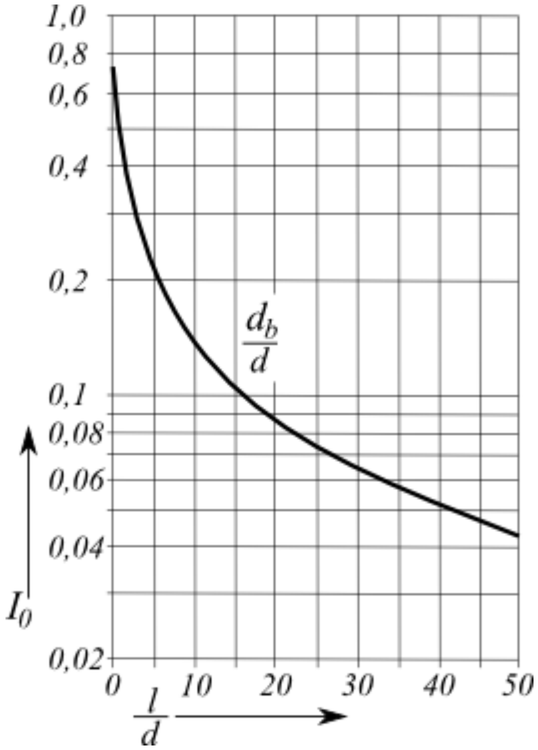
where: D_b - equivalent pile base diameter [m]

Literature:

German Geotechnical Society: Recommendations on Piling (EA-Pfähle). Berlin: Ernst & Sohn, 2012.

Basic Settlement-Influence Factor Io

The basic settlement-influence factor I_o depends on the pile length l and diameter d and the values of this coefficient are generally provided by the following graph also showing their ranges:



Basic settlement-influence factor I_o

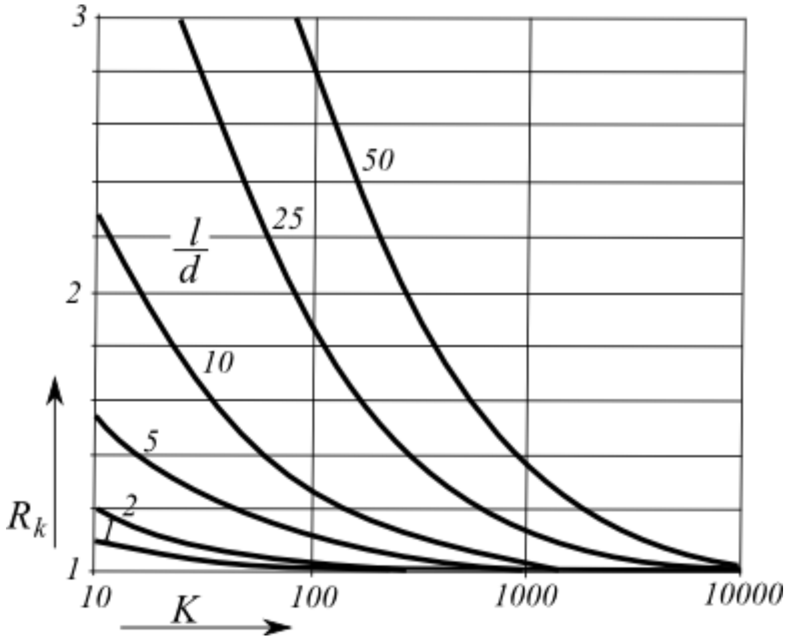
These graphs are implemented in the program in a digital format.

Literature:

Poulos, H. G. et. Davis, E. H.: Pile Foundations Analysis and Design. New York: John Wiley and Sons, 1980, chapter 5.3.3, pp. 89 (figure 5.18).

Correction Factor for Pile Compressibility Rk

The correction factor R_k represents the pile stiffness in dependence on the pile-stiffness factor K for various ratios of the pile length to pile diameter (l/d). Its values are provided by the following graphs, that are implemented in the program in a digital format.



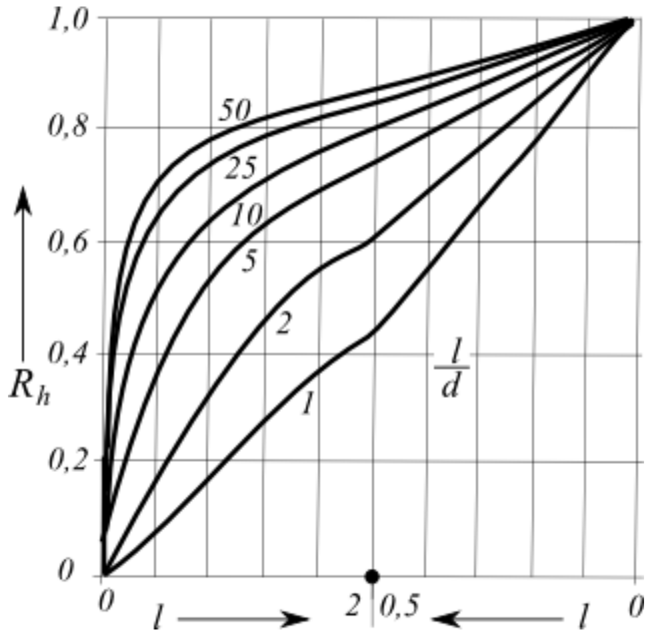
Compressibility correction factor for settlement R_k

Literature:

Poulos, H. G. et. Davis, E. H.: Pile Foundations Analysis and Design. New York: John Wiley and Sons, 1980, chapter 5.3.3, pp. 89 (figure 5.19).

Correction Factor for Finite Depth of Layer on a Rigid Base Rh

The correction factor R_h represents the influence of the incompressible layer below the pile base. Its values are presented in the literature graphically for various pile length to pile diameter ratios (l/d) and ratios of pile length to thickness of compressible layer above the incompressible layer (l/h or h/l). These graphs are implemented in the program in a digital format.



Depth correction factor for settlement R_h

Literature:

Poulos, H. G. et. Davis, E. H.: Pile Foundations Analysis and Design. New York: John Wiley and Sons, 1980, chapter 5.3.3, pp. 89 (figure 5.20).

Horizontal Bearing Capacity - Elastic Subsoil (p-y Method)

Horizontal bearing capacity of a pile, dimensioning

The horizontally loaded pile is analyzed using the finite element method as a beam on elastic Winkler foundation. The soil parameters along the pile are represented by the modulus of subsoil reaction. By default, the pile is subdivided into 30 segments. For each segment the program determines the values of the modulus of the subsoil reaction, internal forces, and deformation (displacements). The program also allows for dimensioning of the steel-reinforced concrete pile based on the method specified in the frame "Settings" and on the parameters input in the "Piles" tab.

The program also enables to analyze a pile loaded by the **prescribed displacements** (translation or rotation of the pile head). In such a case the analysis is carried out only with the prescribed displacement. The input mechanical load is excluded.

The following options for inputting the **modulus of subsoil reaction** are available in the program:

- **by distribution** (distribution of the modulus of subsoil reaction along the pile is specified)
- constant distribution
- linear distribution (Bowles)
- according to CSN 73 1004
- according to Matlock and Reese
- according to Vesic

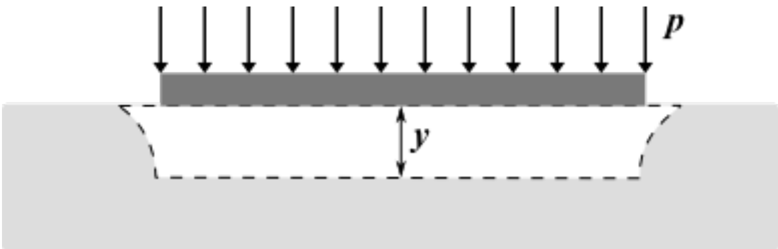
In general, the modulus of the subsoil reaction corresponds to the spring stiffness in the Winkler model. This model describes the settlement of a rigid slab as a function of the applied load. The corresponding relationship is represented by the following formula:

$$p = ky$$

where: p - load acting along a slab-soil interface

k - stiffness of a Winkler spring

y - translation (displacement) of slab into subsoil



Definition of the modulus of subsoil reaction

Constant Distribution of Modulus of Subsoil Reaction

The modulus of the subsoil reaction of the i^{th} layer is provided by:

$$k_h = \frac{3E_{def}}{2r}$$

where: E_{def} - deformation modulus of soil [MPa]

r - reduced width of pile [m], which is given by equation:

$$r = d + 2d\tan\beta$$

where: d - pile diameter [m]

β - angle of dispersion - is input conserning the angle of internal friction in the range of $\varphi/4$ to φ

Literature:

Pochman, R., Simek, J.: Pilotove zaklady - Komentar k CSN 73 1002. 1st edition, Prague, Vydavatelstvi norem, 1989, 80 p.

Linear Modulus of Subsoil Reaction

The modulus of subsoil reaction at a depth z is given by the equation:

$$k_h = k\left(0.308 + 1.584\frac{d}{l}\right)\frac{z}{rl}$$

where: d - pile diameter [m]

l - length of the pile [m]

k - soil parameter (modulus) according to Bowles [MN/m^3]

r - reduced width of the pile [m], which is given by the equation:

$$r = d + 2d \tan \beta$$

where: d - pile diameter $[m]$

β - angle of dispersion - is input with respect to the angle of internal friction in the range of $\varphi/4$ to φ

A representative range of values of lateral modulus k $[MN/m^3]$ according to Bowles:

dense sandy gravel	220 - 400
medium dense gravel	155 - 300
medium-graded sand	110 - 280
fine sand	80 - 200
stiff clay	60 - 220
saturated stiff clay	30 - 110
plastic clay	40 - 140
saturated plastic clay	10 - 80
soft clay	2 - 40

Literature:

Bowles, J. E.: Foundations Analysis and Design. 5th edition, New York: McGraw-Hill Book Company, 1997, ISBN 0-07-118844-4, chapter 16-15.2, s. 941 (table 16-4).

Pochman, R., Simek, J.: Pilotove zaklady - Komentar k CSN 73 1002. 1st edition, Prague, Vydavatelstvi norem, 1989, 80 p.

Modulus of Subsoil Reaction According to CSN 73 1004

The modulus of subsoil reaction for **cohesive soil** assumes the form:

$$k_h = \frac{2E_{def}}{3d}$$

where: E_{def} - deformation modulus of soil $[MPa]$

d - pile diameter $[m]$

For **cohesionless soil,** it is given by:

$$k_h = n_h \frac{z}{d}$$

where: n_h - modulus of horizontal compressibility $[MN/m^3]$

d - pile diameter $[m]$

z depth of a given section from original grade $[m]$

Approximate values of modulus of horizontal compressibility n_h for cohesionless soils:

Soil	n_h $[MN/m^3]$		
Relative density of soil I_D [-]	0.3	0.5	0.9
Dry sand and gravel Wet sand and gravel	2.5 1.5	7.0 4.5	18.0 11.0

Literature:

CSN 73 1004: Velkoprumerove piloty. Praha, UNM, 1981, 56 p.

Masopust, J.: Vrtane piloty. 1st edition, Prague, Cenek a Jezek, 1994, 263 p.

Modulus of Subsoil Reaction According to Matlock and Reese

This method is applicable for **cohesionless soils**. The modulus of the subsoil reaction is given by the equation:

$$k_h = n_h \frac{z}{d}$$

where: n_h - modulus of horizontal compressibility $[MN/m^3]$

d - pile diameter $[m]$

z - depth of a given section from finished grade $[m]$

Approximate values of modulus of horizontal compressibility n_h for cohesionless soils:

Soil - density	n_h $[MN/m^3]$

Dry sand and gravel - loose - medium dense - dense	 1.8 - 2.2 5.5 - 7.0 15.0 - 18.0
Wet sand and gravel - loose - medium dense - dense	 1.0 - 1.4 3.5 - 4.5 9.0 - 12.0

Literature:

Reese, L. C. et. Matlock, H.: Non-Dimensional Solutions for Laterally Loaded Piles with Soil Modulus Assumed Proportional to Depth. University of Texas, Austin, 1956.

Reese, L. C. et. Matlock, H.: Generalized Solutions for Laterally Loaded Piles. Journal of the Soil Mechanics and Foundations Division, ASCE 86, No. 5, 1960, pp. 63 - 91.

Reese, L. C. et. Matlock, H.: Foundation analysis of offshore pile-supported structures. Proceedings of the 5th International Conference, ISSMFE, Paris, Vol. 2, 1961, pp. 91-7

Modulus of Subsoil Reaction According to Vesic

The modulus of subsoil reaction is provided by:

$$k_h = \frac{0.65}{d} \sqrt[12]{\frac{E_s}{E_p}\frac{d^4}{I_p}} \frac{E_s}{1-\nu^2}$$

where:

E_p	-	modulus of elasticity of pile [MPa]
I_p	-	moment of inertia of pile [m^4]
E_s	-	modulus of elasticity of soil [MPa]
d	-	pile diameter [m]
ν	-	Poisson's ratio [-]

Literature:

Poulos, H. G. et. Davis, E. H.: Pile Foundations Analysis and Design. New York: John Wiley and Sons, 1980, chapter 8.2.3, pp. 174 (equation 8.43).

Vesic, A. S.: Bending of Beams Resting on Isotropic Elastic Solid. JSMFD, ASCE, vol. 87, 1961, EM 2: pp. 35 - 53.

Vesic, A.S.: Design of Pile Foundations. National Cooperative Highway Research Program Synthesis 42, Transportation Research Board, Washington D.C., 1977.

Pile Horizontal Bearing Capacity - Brom’s Method

Analysis of a single pile according to Broms is described in Broms, 1964. This method exclusively assumes a pile in the **homogeneous soil**. Thus the analysis method does not allow for layered subsoil. The type of analysis of the pile horizontal bearing capacity is specified in the "**Settings**" frame, tab "Piles".

When adopting the Broms method for the analysis of horizontal bearing capacity the program disregards up now input soil layers. The soil parameters are specified in the "Horizontal bearing capacity" frame based on the **type of soil** (cohesive, cohesionless).

The input parameters for the analysis of pile horizontal bearing capacity are the **pile material characteristics** (modulus of elasticity and strength of a given material), **pile geometry** (pile length l and its diameter d), and also the **pile load** due to shear force and bending moment.

The coefficient of pile stiffness β for cohesive soils is given by:

$$\beta = \left(\frac{k_h d}{4EI}\right)^{\frac{1}{4}}$$

where:

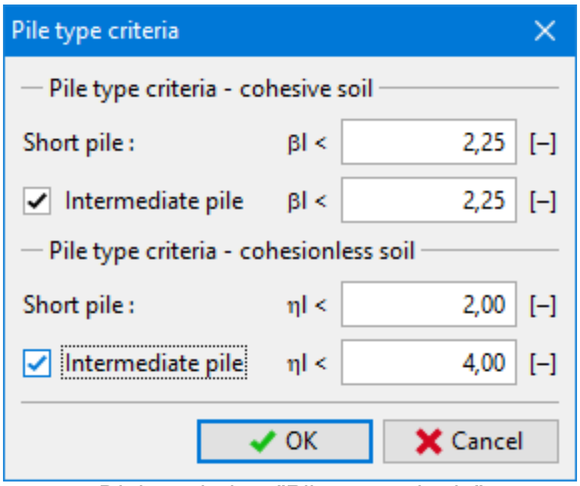
$E*I$	-	bending stiffness of pile section [MNm^2]
k_h	-	modulus of subsoil reaction [MNm^3]
d	-	diameter of a single pile [m] - in case of a pile with a **circular variable cross-section** the calculation of parameter β assumes a constant value of the pile diameter d_1 input in the "Geometry" frame

The coefficient of pile stiffness η for cohesionless soils follows from:

$$\eta = \left(\frac{n_h}{EI}\right)^{\frac{1}{5}}$$

where: E^*I - bending stiffness of pile section [MNm^2]
n_h - coefficient of soil modulus variation [MNm^3]

The program automatically determines whether to consider a long or a short pile based on the values β^*l (for **cohesive soils**) and η^*l (for **cohesionless soils**), respectively. Because literature offers different criteria for different types of piles, the program allows the user to define them. For an **intermediate** pile length the verification analysis considers both short and long piles and then the program automatically chooses the result with the lowest value of the pile horizontal bearing capacity Q_u.



Dialog window "Pile type criteria"

Type of pile criteria (long, short, medium) are considered according to the following conditions:

- **free head**: for long piles, it holds $\beta^*l >$ 2,5; for short piles then $\beta^*l <$ 2,5
- **restrained**: for long piles, it holds $\beta^*l >$ 1,5; for short piles then $\beta^*l <$ 1,5

Type of pile (pile head support) can be considered in two ways:

- **free head** - rotation at pile head is not constrained
- **restrained** - pile is constrained against rotation at its head. In such cases we typically deal with piles that are part of a planar pile grid or a pile group.

Another important input parameter is the **flexure bearing capacity**. This quantity is automatically back-calculated by the program using the following relation:

$$M_u = \gamma_k f W_y$$

where: W_y - section modulus of the pile section [m^3]
f - strength of the pile material [MPa]
γ_k - reduction coefficient of cross-section strength [-] - the cross-section bearing capacity is according to different standards pre-multiplied by different safety coefficients. This coefficient enables to adapt the program to these standards.

In case of a **steel-reinforced concrete pile** the flexure bearing capacity, M_u depends on the amount of designed steel.

The reduction coefficient of bearing capacity γ_{Qu} reduces the overall magnitude of the **single pile horizontal bearing capacity** as:

$$Q_{u,red} = \frac{Q_u}{\gamma_{Q_u}}$$

where: Q_u - horizontal bearing capacity of a single pile [kN]
γ_{Qu} - reduction coefficient of bearing capacity [-]

The result of an analysis is horizontal bearing capacity of a single pile Q_u, respectively $Q_{u,red}$ and displacement of a pile at the terrain surface u.

Literature:

[1] BROMS, BENGT. B.: Lateral Resistance of Piles in Cohesive Soils. Proceedings of the American Society of Civil Engineers, Journal of the Soil Mechanics and Foundations Division, Vol. 90, SM2, 1964.

[2] BROMS, BENGT. B.: Lateral Resistance of Piles in Cohesionless Soils. Proceedings of the American Society of Civil Engineers, Journal of the Soil Mechanics and Foundations Division, vol. 90 SM3, 1964.

Pile CPT

The Pile CPT program allows:

- bearing capacity calculation from CPTs
- settlement calculation from CPTs
- bearing capacity calculation from SPTs

Analyses from CPTs

The Pile CPT program allows us to verify the bearing capacity and settlement of a single pile or a group of piles based on the results of penetration tests - CPTs.

The main objective is to determine the base and shaft bearing capacities. This analysis can be carried according to the following standards and approaches:

- EN 1997-2
- NEN 6743
- LCPC (Bustamante)
- Schmertmann
- NBN EN1997-1 ANB

For all methods the essential input parameters are dimensionless coefficients adjusting the magnitude of bearing capacity and shaft friction, respectively. Different notation of these parameters can be encountered in various publications. The following notation is used in program Pile CPT:

α_p - pile base coefficient

α_s - shaft friction coefficient

These coefficients are automatically calculated based on the type pile and the surrounding soil - these parameters can be, however, also specified manually (α_p can be entered in the "Geometry" input mode, α_s as a soil parameter).

When analyzing rectangular piles the pile shape coefficient s is introduced to reduce the base bearing capacity. When analyzing piles with enlargement the expanded pile base coefficient β is introduced to adjust the expanded base bearing capacity. When calculating the base bearing capacity the program automatically accounts for the influence of the change of terrain elevation.

The program allows for the calculation of the load-settlement curve and pile settlement for a given load. This analysis adopts the values of the calculated base and shaft bearing capacities and follows the NEN 6743 standard. A negative skin friction can also be taken into account when calculating pile settlement.

Verification of a pile bearing capacity depends on the verification methodology selected in the "Pile CPT" tab.

Bearing Capacity

The maximum bearing capacity of a single pile based on the values of tip resistance q_c of the i^{th} cone penetration test is given by:

$$F_{\max,i} = F_{\max,base,i} + F_{\max,shaft,i}$$

where:

$F_{max,i}$ - maximum bearing capacity of the pile from i^{th} CPT

$F_{max,base,i}$ - maximum base resistance from i^{th} CPT

$F_{max,shaft,i}$ - maximum shaft resistance from i^{th} CPT

If performing the analysis according to the **factor of safety** or the **limit states** theory, the bearing capacity of a single pile is obtained as an arithmetic average of n calculated bearing capacities:

$$F_{\max} = \frac{\sum_{i=1}^{n} F_{\max,i}}{n}$$

If performing the analysis according to EN 1997-2, or NBN EN1997-1 ANB, final bearing capacity depends on correlation coefficients ξ_3 and ξ_4.

If performing the analysis according to the NEN 6743 standard then the approach for more CPTs is different and follows directly the NEN 6743 standard (article 5.3.2.2).

The maximum pile base resistance $F_{max,base}$ is provided by:

$$F_{\max,base} = A_{base}\, p_{\max,base}$$

where: A_{base} - pile base cross-sectional area

$p_{max, base}$ - maximum pressure at pile base from CPT results

The maximum shaft resistance $F_{max,shaft}$ is provided by:

$$F_{\max, shaft} = O_p \int_0^{\Delta L} p_{\max, shaft} dz$$

where: O_p - pile periphery in bearing soil

$p_{max, shaft}$ - maximum force on shaft (friction) from CPT results

ΔL - pile length, either length of activated shaft friction or length of expanded base

z - vertical dimension along pile axis

The actual calculation of the maximum pressure at pile base $p_{max,base}$ and the maximum force developed along the shaft $p_{max,shaft}$ (determined according to the selected type of analysis selected in the "Pile CPT" tab).

EN 1997-2

The **EN 1997-2** standard determines the maximum pressure at the pile base (maximum resistance) $p_{max, base}$ from the corresponding i^{th} penetration test as follows:

$$p_{\max, base} = 0{,}5\, \alpha_p\, \beta\, s \left(\frac{q_{c,I,mean} + q_{c,II,mean}}{2} + q_{c,III,mean} \right)$$

where: $q_{c, I, mean}$ - mean from values $q_{c, I}$ (see Addendum D.7 in EN 1997-2)

$q_{c, II, mean}$ - mean of the minimum cone tip resistances $q_{c, II}$ (see Addendum B4 in EN 1997-3)

$q_{c, III, mean}$ - mean of the cone tip resistance $q_{c, III}$ (see Addendum B4 in EN 1997-3)

α_p - pile base coefficient (pile class factor)

s - pile shape coefficient

β - expanded pile base coefficient

The **maximum value of penetration pressure** q_c is limited by the value of 15 MPa . In cohesionless soils the analysis takes into account the influence of overconsolidation (OCR).

The **maximum shaft friction** (shaft resistance) $p_{max, shaft}$ is given by:

$$p_{\max, shaft} = \alpha_s\, q_{c,z,a}$$

where: α_s - shaft friction coefficient

$q_{c, z, a}$ - tip resistance at depth h

Literature:

EN 1997-2 Geotechnical design. Ground investigation and testing.

NEN 6743

The NEN 6743 "Piled Foundations" standard determines the **maximum pressure at pile base** $p_{max, base}$ from the corresponding i^{th} penetration test as follows:

$$p_{\max, base} = 0{,}5\, \alpha_p\, \beta\, s \left(\frac{q_{c,I,mean} + q_{c,II,mean}}{2} + q_{c,III,mean} \right)$$

where: $q_{c, I, mean}$ - mean of the cone tip resistance $q_{c, I}$ (see article 5.3.3.3 in NEN 6743 standard)

$q_{c, II, mean}$ - mean of the minimum cone tip resistance $q_{c, II}$ (see article 5.3.3.3 in NEN 6743 standard)

$q_{c, III ,mean}$ - mean of the cone tip resistance $q_{c, III}$ (see article 5.3.3.3 in NEN 6743 standard)

α_p - pile base coefficient

s - pile shape coefficient

β - expanded pile base coefficient

The **maximum value of penetration pressure** q_c is limited by the value of 15 MPa . In cohesionless soils the analysis takes into account the influence of overconsolidation (OCR).

The **maximum shaft friction** $p\ max,\ shaft$ is given by:

$$p_{\max, shaft} = \alpha_s \, q_{c,z,a}$$

where: α_s - shaft friction coefficient

$q_{c,\ z,\ a}$ - tip resistance at depth h

Literature:

NEN 6743:1991/A1:1997, Geotechniek - Berekeningsmethode voor funderingen op palen - Drukpalen.

LCPC (Bustamante)

The LCPC - Laboratoire Central des Ponts et Chausees method (also known as Bustamante method based on the works of Bustamante and Gianeselli) determines the **maximum pressure at pile base** $p_{max,\ base}$ as follows:

$$P_{\max, base} = \alpha_p \, q_{c,eq}$$

where: α_p - pile base coefficient

$q_{c,\ eq}$ - equivalent average cone tip resistence

The **maximum shaft friction** $p_{max,\ shaft}$ is given by:

$$P_{\max, shaft} = \alpha_s \, q_{c,z,a}$$

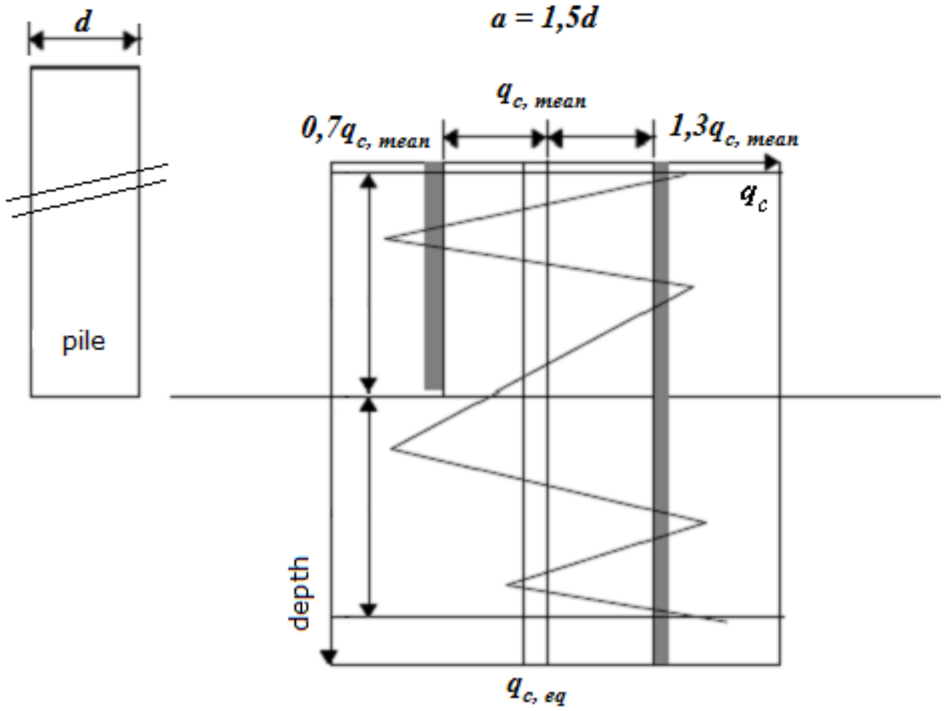
where: α_s - shaft friction coefficient

$q_{c,\ z,\ a}$ - tip resistance

Determination of Equivalent Average Cone Tip Resistance

An equivalent average cone tip resistance is obtained in the following way:

1) Calculate the average tip resistance $q_{c,\ mean}$ at the tip of the pile by averaging q_c values over a zone ranging from 1.5d below the pile tip to 1.5d above the pile tip (d is the pile diameter)
2) Eliminate q_c values in the zone which are higher than 1.3 multiple of the mean of the cone tip resistance $q_{c,\ mean}$ and those are lower than 0.7 multiple of the mean of the cone tip resistance $q_{c,\ mean}$ as shown in figure
3) Calculate the equivalent average cone tip resistance $q_{c,\ eq}$ by averaging the remaining cone tip resistance q_c values over the same zone that were not eliminated (i.e. from values in the range 0.7 to 1.3 multiple of the cone tip resistance $q_{c,\ mean}$).



Determination of equivalent average cone tip resistance $q_{c,\ eq}$

Literature:

Tom Lunne, Peter K. Robertson, John J.M. Powell: Cone Penetration Testing in Geotechnical Practice, Spon Press, 1997, London.

Schmertmann

The Schmertmann method determines the maximum pressure at pile base $p_{max,\ base}$ as follows:

$$P_{max,toe} = \alpha_p . q_{upr}$$

where: α_p - pile base coefficient

q_{upr} - modified equivalent average cone tip resistance

$$q_{upr} = \frac{q_{c1} + q_{c2}}{2}$$

where: q_{c1} , q_{c2} - minimum value of the mean of cone tip resistance

In cohesionless soils the analysis takes into account the influence of overconsolidation (OCR).

The **maximum shaft friction** $p\ max,shaft$ is given by following formulas:

- **for cohesionless soils:**

$$P_{max,shaft} = K\left[0{,}5.\left(\bar{f}_s . A_s\right)_{0\ to\ 8d} + \left(\bar{f}_s . A_s\right)_{8\,d\ to\ D}\right]$$

where: K - correlation coefficient of skin friction

$\bar{f}_s$ - mean value of penetrometer sleeve local friction f_s in the interval given by bracket subscript

A_s - area of the pile shaft surface in a given interval

d - diameter of pile

D - embedded pile length

- **for cohesive soils:**

$$P_{max,shaft} = \sum_i \alpha_{s,i} . \bar{f}_{s,i} . A_{s,i}$$

where: $\alpha_{s,i}$ - shaft friction coefficient according to Tomlinson in the i^{th} layer

$\bar{f}_{s,i}$ - mean value of penetrometer sleeve local friction f_s in the i^{th} layer

$A_{s,i}$ - area of the pile shaft surface in the i^{th} layer

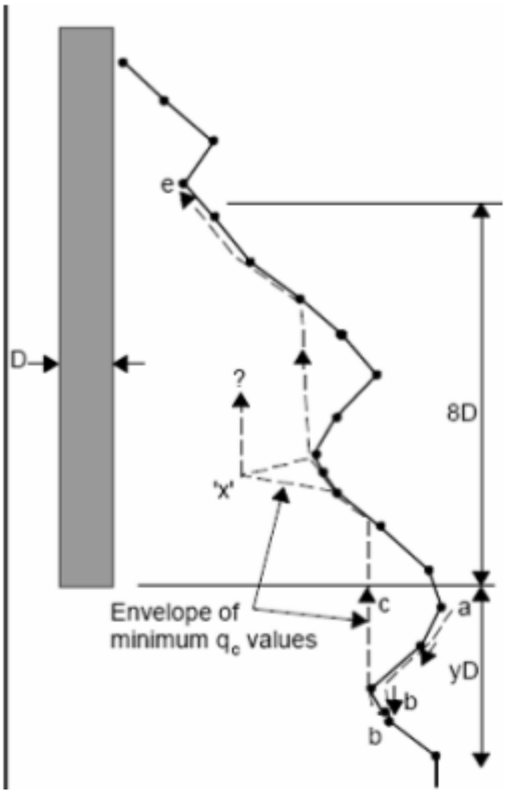
Literature:

Schmertmann J.H.: Guidelines for Cone Penetration Test, Performance and deign, U.S. Departments of Transportation, report No. FHWA-TS-78-209, Washington,D.C., 1978.

Determination of Average Cone Tip Resistance

The minimum mean value of the cone tip resistance q_c s determined by the minimum value of the mean of the cone tip resistance q_c over the influenced zone ranging from 0.7d to 4d below the pile base (d is the pile diameter). The minimum mean value of the cone tip resistance q_{c2} is determined over the influence zone extending from 8d above the pile base (d is the pile diameter). The procedure for obtaining the mean value of the cone tip resistance q_{c1} , q_{c2} is as follows (see figure):

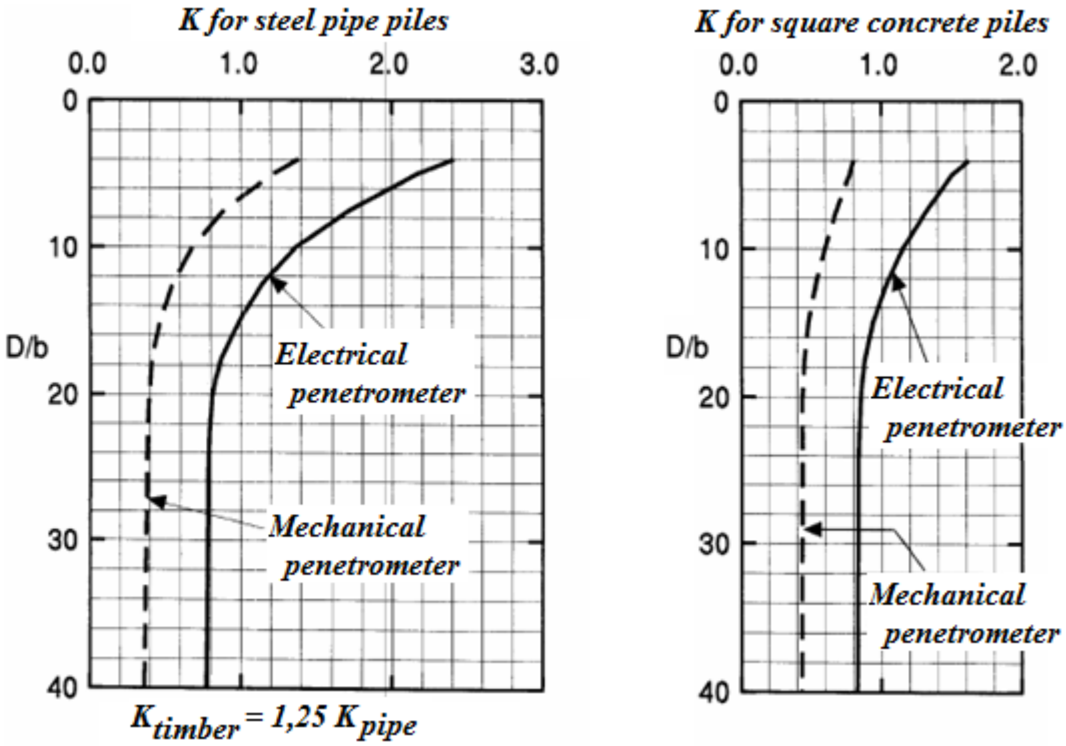
1) determine two averages of the cone stress within the zone below the pile base, one for a zone depth of 0.7d and one for 0.4d along the path "**a**" through "**b**". The smaller of the two is retained. (The zone height 0.7d applies to where the cone stress increases with depth below the pile base).
2) determine the smallest cone stress within the zone used for Step 1
3) determine the average of the two values per Steps 1 and 2. Step 4 is determining the average cone stress in the zone
4) determine the average cone stress in the zone 8d above the pile base that gives the value q_c . Finally, the average of Step 3 and Step 4 values is determined.



Determination of average cone tip resistance q_{c1}, q_{c2}

Correlation Coefficient K

The correlation coefficient of skin friction K is entered in the "Pile CPT" tab. The value of this coefficient is equal to the ratio of unit pile shaft resistance and unit penetrometer sleeve local friction. Correlation coefficient K can be expressed for example by the function of embedded pile length - see the following graphs.



Function of embedded pile length (D - embedded pile length, b - pile width or diameter)

Literature:

FHWA HI 97-013: Design and Construction of Driven Pile Foundations, Workshop manual - Volume 1, National Highway institute.

NBN EN 1997-1 ANB

The **NBN EN 1997-1 ANB** standard determines the maximum pressure at the pile base (maximum resistance) $p_{max,\,base}$ from the corresponding i^{th} penetration test as follows:

$$p_{max,base} = \alpha_p.\varepsilon_p.\beta.s.q_p$$

where:	α_p	- pile base coefficient
	ε_p	- scale factor
	β	- expanded pile base coefficient
	s	- pile shape coefficient
	q_p	- unit pile base resistance according to the De Beer method

The **Scale factor ε_p** is determined as follows:

- for tertiary OC-clay:

$$\varepsilon_p = max(1 - 0,01.(\frac{D_{b,eq}}{D_c} - 1); 0,476)$$

- for other soil types:

$$\varepsilon_p = 1$$

where:	$D_{b,eq}$	- equivalent pile base diameter
	D_c	- diameter of the CPT-cone

The **maximum shaft friction** (shaft resistance) $p\ max,\ shaft$ is given by formula:

$$p_{max,shaft} = \sum_{i=1}^{n} \alpha_{s,i}.q_{s,i}$$

where:	$\alpha_{s,\,i}$	- shaft friction coefficient in the i th layer
	$q_{s,i}$	- average shaft friction in the i th layer
	n	- number of layers along the pile

Average shaft friction $q_{s,i}$ is determined as follows:

$$q_{si} = 1000.\eta_{pi}.q_{c,m,i}$$

where:	$q_{c,m,i}$	- average cone resistance (qc) in the i th layer
	η_{pi}	- empirical friction coefficient

Empirical friction coefficient η_{pi} (or max. average shaft friction) are determined according to the soil type as follows:

Soil type	$q\ c,m,i$ [MPa]	η_{pi} [-] or $q_{s,i}$ [kPa]
Clay	1 to 4,5	- η_{pi} = 1/30
	> 4,5	$q_{s,i}$ = 150
Silt	1 to 6	η_{pi} = 1/60
	> 6	$q_{s,i}$ = 100
Sandy Clay	1 to 10	η_{pi} = 1/80
	> 10	$q_{s,i}$ = 125
Sand	1 to 10	η_{pi} = 1/90
	10 to 20	$q_{s,i}$ = 110+4($q_{c,m,i}$ - 10)
	> 20	$q_{s,i}$ = 150

Literature:

RICHTLIJNEN VOOR DE TOEPASSING VAN DE EUROCODE 7 IN BELGIË VOLGENS DE NBN EN 1997-1 ANB (WTCB Rapport nr. 20 - November 2020).

De Beer Method

This method determines the **unit pile base resistance**. It is based on the so-called "scale effect" for CPT-cone and pile base.

The method is described by the following steps which are performed in the standard 0,2 m interval along the pile length.

1) The calculation of the angle of internal friction φ according to the formula (1)

$$\frac{q_c}{\sigma_{eff}} = 1,3.e^{2\pi.tg\varphi}.(45+\varphi/2)$$

where: q_c - cone tip resistance

σ_{eff} - effective stress

φ - angle of internal friction

2) The calculation of the β_c and β_p angles according to the formulas (2) and (3).

$$\frac{h}{d_c} = \frac{tg(\pi/4+\varphi/2).e^{\pi/2.tg\varphi}.sin\beta_c.e^{\beta_c.tg\varphi}}{1+\delta.sin(2\pi)}$$

$$\frac{h}{D} = \frac{tg(\pi/4+\varphi/2).e^{\pi/2.tg\varphi}.sin\beta_p.e^{\beta_p.tg\varphi}}{1+\delta.sin(2\pi)}$$

Note: Both angles β_c a β_p can be maximally 90°.

where: h - measurement depth

d_c - diameter of the CPT-cone

D - pile base diameter

δ - pile base shape coefficient

For the circular and square piles:

$$\delta = 1$$

For the rectangular piles:

$$\delta = \frac{B}{L}$$

where: B - shorter side of the pile base

L - longer side of the pile base

3) Calculation of homogeneous values $q_{p(1)}$ according to the formula (4).

$$q_{p(1)} = \frac{q_c}{e^{2.(\beta_c-\beta_p).tg\varphi}}$$

4) Calculation of downward values $q_{p,\,j+1}$ according to the formula (5).

$$q_{p,j+1} = q_{p,j} + \frac{a}{h_{crit}}.\frac{d_c}{D}.\left[\frac{\sigma_{eff,j}+\frac{D}{d_c}.\frac{\gamma.h_{crit}}{2}}{\sigma_{eff,j}+\frac{\gamma.h_{crit}}{2}}.q_{p(1),crit} - q_{p,j}\right]$$

where: a - measurement interval

h_{crit} - critical depth

γ - unit weight of soil

$q_{p(1),\,crit}$ - homogeneous value in the critical depth h_{crit}

The critical depth h_{crit} depends on the pile base diameter D and angle of internal friction φ :

For piles $D <$ 0,4 m

- $h_{crit} =$ 0,2 m

For piles 0,4 $m \leq D \leq$ 0,6 m

- $h_{crit} =$ 0,2 m for $\varphi <$ 32,5°

- $h_{crit} =$ 0,2 m and 0,4 m for $\varphi \geq$ 32,5°

For piles $D \geq$ 0,6 m

- $h_{crit} =$ 0,2 m for $\varphi <$ 32,5°

- $h_{crit} =$ 0,2 m and 0,4 m for 32,5° $\leq \varphi <$ 37,5°

- h_{crit} = 0,2 m and 0,4 m a 0,6 m for $\varphi \geq$ 37,5°

Note: The final downward value is minimum for all relevant critical depths.

Note: The downward value must be smaller or equal to the homogeneous value.

5) Calculation of upward values $q_{p,\,q+1}$ according to the formula (6).

$$q_{p,q+1} = q_{p,q} + [(q_{p,j+1})_{q+1} - q_{p,q}] \cdot \frac{d_c}{D}$$

Note: The upward value must be smaller or equal to the downward value.

6) Calculation of blended (mixed) $q_{r,b}$ values as an average from n upward values $q_{p,\,q+1}$. The unit pile base resistance is the blended value in the pile base level.

$$n = \frac{D}{0,2} + 1$$

Note: The blended *value must be smaller or equal to the homogeneous value.*

Literature:

Prof. Dr. ir E. DE BEER: METHODES DE DEDUCTION DE LA CAPACITE PORTANTE D´UN PIEU A PARTIR DES RESULTATS DES ESSAIS DE PENETRATION (MEMOIRES - VERHANDELINGEN), online: https://www.bggg-gbms.be/l/library/download/urn:uuid:05c758bb-bf51-4eb7-9dc0-5a03b293f7e0/overige+publicaties.pdf?format=

Equivalent Pile Base Diameter

The equivalent pile base diameter $D_{b,eq}$ is defined as follows in the **NBN EN 1997-1 ANB** standard:

- for **circular piles**

$$D_{b,eq} = D$$

- for **square** or **rectangular piles**

- if $b \leq$ 1,5a

$$D_{b,eq} = \sqrt{\frac{4.a.b}{\pi}}$$

- if $b >$ 1,5a

$$D_{b,eq} = \sqrt{\frac{6.a^2}{\pi}}$$

where: D - pile base diameter
a - shorter side of the pile base
b - longer side of the pile base

Negative Skin Friction

Negative skin friction is an effect that arises as a result of the settlement of soil around the pile. A soil deforming around the pile tends to pull the pile down thus reducing its bearing capacity. In extreme cases this effect may eliminate the influence of shaft friction. The pile is then supported only by elastic subsoil below the pile base.

The **negative skin friction** $F_{s,nk,rep}$ is given by:

$$F_{s,nk,rep} = O_p \cdot \sum_{i=1}^{n} h_i . K_{0,i,rep} . tan(\delta_{i,rep}) \frac{\sigma_{v,i-1,rep} + \sigma_{v,i,rep}}{2} + p_{i,a,rep} - \Delta\sigma_{i,v,w,rep}$$

where: O_p - pile periphery
n - number of layers in the negative friction zone
h_i - depth of i^{th} layer
$K_{0,i,rep}$ - representative value of the coefficient of earth pressure at rest
$\delta_{i,rep}$ - friction between soil and pile at i^{th} layer

$$\delta_{i,rep} = 0,75.\varphi_{i,rep}$$

$\varphi_{i,rep}$ - representative value of the angle of internal friction at i^{th} layer

$\sigma_{v,i-1,rep}$ - horizontal stress in soil at i- 1 layer

$\sigma_{v,1,rep}$ - horizontal stress in soil at i^{th} layer

$p_{i,a,rep}$ - surcharge at i^{th} layer

$\Delta\sigma_{i,v,w,rep}$ - change of vertical stress σ_v at i^{th} layer

the following relation holds: $K_{0,i,rep}.tan(\delta_{i,rep}) > 0,25$

If a slip surface is defined then the value of negative skin friction $F_{s,nk,rep}$ is provided by:

$$F_{s,nk,rep} = O_p.\sum_{i=1}^{n} h_i.c_{i,rep}$$

where: O_p - pile periphery

h_i - depth of i^{th} layer

$c_{i,rep}$ - representative cohesion of slip surface

- for bitumen $10*10^3\ N/m^2$
- for bentonite $20*10^3\ N/m^2$
- for synthetic material $50*10^3\ N/m^2$

The value of representative cohesion along a slip surface can also be introduced directly by the user.

Shaft Friction Coefficient ALFAs

The coefficient reducing the shaft friction α_s differs based on the applied method and the type of soil. The values of these coefficients are built into the program according to **EN 1997-2 and NEN 6743** standards.

The values **for sands** and **sands with gravel** are listed in the following table:

Piles	**NEN 6743** α_s [-]	**EN 1997-2** α_s [-]
prefabricated driven piles or steel piles	0.010	0.010
Franki piles	0.014	0.012
driven wooden piles	0.012	0.012
vibrating or vibropressed	0.012	0.012
cast in place screw piles	0.009	0.009
prefabricated screw piles	0.009	0.009
cast in place screw piles with additional grouting	0.006	0.006
prefabricated screw piles with additional grouting	0.006	0.006
steel tubular piles	0.0075	0.0075
Continuous Flight Auger piles (CFA)	0.006	0.006
bored piles or piles sheeted by bentonite suspense	0.006	0.006
bored piles with steel casing	0.005	0.005

For **very coarse-grained sands** and **gravels** the above values are reduced in both methods by a reduction coefficient (coarse-grained sand 0.75, gravel 0.5).

For **peat** the value of α_s = 0 is considered.

For **clay and silt** the values of α_s according to the **EN 1997-2** are listed in the following table:

Type of soil	q_c [MPa]	α_s [-]
clay	> 3	< 0.030
clay	< 3	< 0.020
silt		< 0.025

For **clay and silt** the values of α_s according to the **NEN 6743** are listed in the following table:

q_c [MPa]	α_s [-]
> 1	0.035
< 1	0.0 depth to quintuple of pile diameter
	0.025 depth from 5 to 20 multiple of pile diameter

	0,035 depth over 20 multiple of pile diameter

If the **LCPC (Bustamante)** method is used the shaft friction coefficient α_s is used depending on the tip resistance q_c (orientation values are available in the following table).

Orientation values of the shaft friction coefficient α_s based on the cone tip resistance q_c

LCPC (Bustamante) Soil type	Cone stress (tip resistance) q_c $[MPa]$	α_s for piles of type "A"	α_s for piles of type "B"	Maximum shaft resistance $[kPa]$
Clay	< 1	0.033	0.033	15
	$1 < q_c < 5$	0.025	0.011	35
	$5 < q_c$	0.017	0.008	35
Sand	$q_c < 5$	0.010	0.008	35
	$5 < q_c < 12$	0.010	0.005	80
	$12 < q_c$	0.007	0.005	120

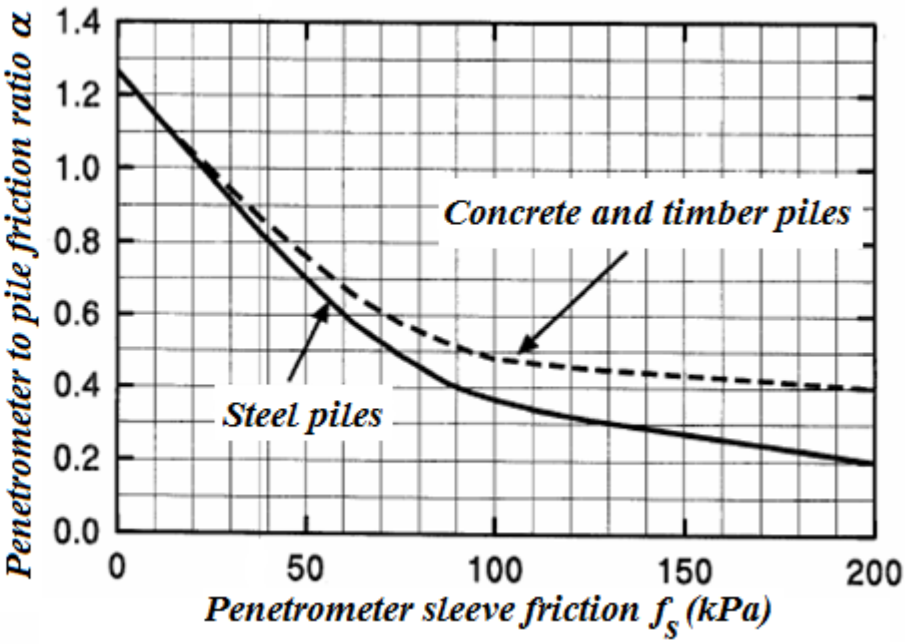
Type "A" includes these types of technology of installation of piles:

- **screw** (cast in place, prefabricated, cast in place with additional grouting, prefabricated with additional grouting, CFA piles, bored or piles sheeted by bentonite suspense)
- **driven prefabricated**

Type "B" includes these types of technology of installation of piles:

- **driven steel**
- **Franki piles**
- **vibrating**
- **steel tubular**
- **bored with steel casing**

When using **Schmertmann** method, coefficient α_s reducing shaft friction according to Tomlinson is considered. Values used in the program are derived from the following graph mentioned in publication M. J. Tomlinson: Pile Design and Construction Practice (1994).



For the **NBN EN 1997-1 ABN** standard, the coefficients are taken as follows:

Piles	α_s [-]	$\alpha_{s,tert.\ clay}$ [-]
prefabricated driven piles or steel piles	0.6	0.6
Franki piles	1.0	0.9
driven wooden piles	1.15	1.15
vibrating or vibropressed	0.6	0.6

cast in place screw piles	0.6	0.6
prefabricated screw piles	0.6	0.6
cast in place screw piles with additional grouting	0.6	0.6
prefabricated screw piles with additional grouting	0.6	0.6
steel tubular piles	0.6	0.6
Continuous Flight Auger piles (CFA)	0.5	0.3
bored piles or piles sheeted by bentonite suspense	0.5	0.5
bored piles with steel casing	0.5	0.3

Literature:

Tomlinson M. J.: Pile Design and Construction Practice, 4th edition, Taylor and Francis, 1994, ISBN 0 419 18450 3.

RICHTLIJNEN VOOR DE TOEPASSING VAN DE EUROCODE 7 IN BELGIË VOLGENS DE NBN EN 1997-1 ANB (WTCB Rapport nr. 20 - November 2020)

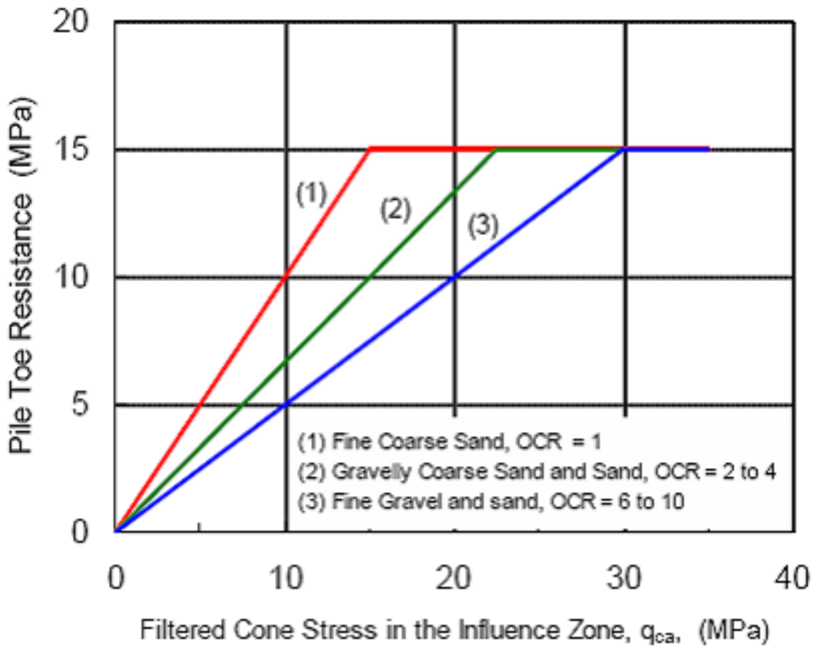
Influence of Overconsolidation (OCR)

For sand and gravel the maximum pressure at pile base $p_{max,base}$ (determined according to the selected type of analysis selected in the "Pile CPT" tab) is reduced depending on the value of overconsolidation OCR (defined as a soil parameter in the frame "Soils") as follows:

Analysis according to EC 7-3, NEN 6743:

- for all cohesionless soils the maximum pressure at pile base $p_{max,base}$ is 15 MPa
- for $OCR \leq$ 2 no reduction is performed
- for $2 < OCR \leq$ 4 the maximum pressure at pile base $p_{max,base}$ is multiplied by 0.67
- for $OCR >$ 4 the maximum pressure at pile base $p_{max,base}$ is multiplied by 0,50

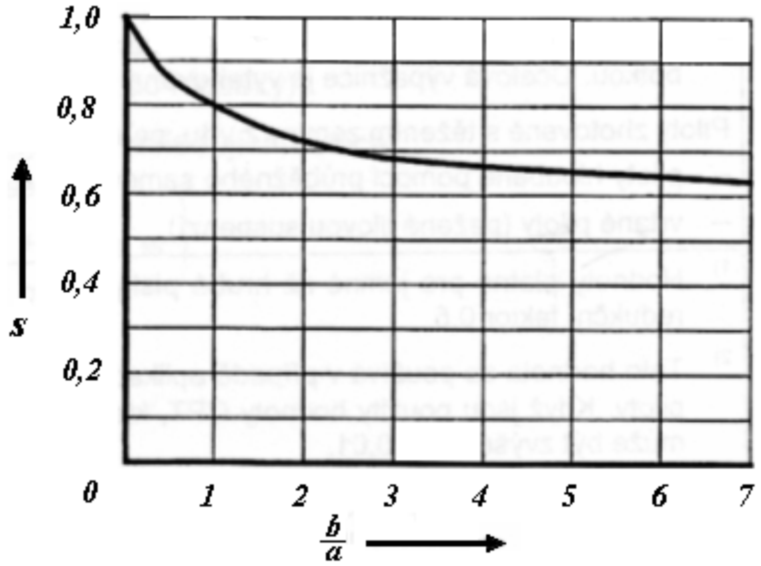
When using the Schmertmann method the reduction is performed according to the following graph:



Reduction of equivalent mean cone tip resistance according to OCR (Schmertman)

Coefficient of Influence of Pile Shape s

This coefficient represents the influence of a **rectangular** pile cross-section, the b/a ratio in particular. Its values are evident from the following graph (function of b/a):



Graph to determine pile shape coefficient s *(*a *- length of the smallest side,* b *- the largest side)*

For the **NBN EN 1997-1 ABN** standard, the shape coefficient s is defined as follows:

- for **circular piles**

$$s = 1$$

- for **square** or **rectangular piles**

$$s = \frac{1 + 0,3.\frac{a}{b}}{1,3}$$

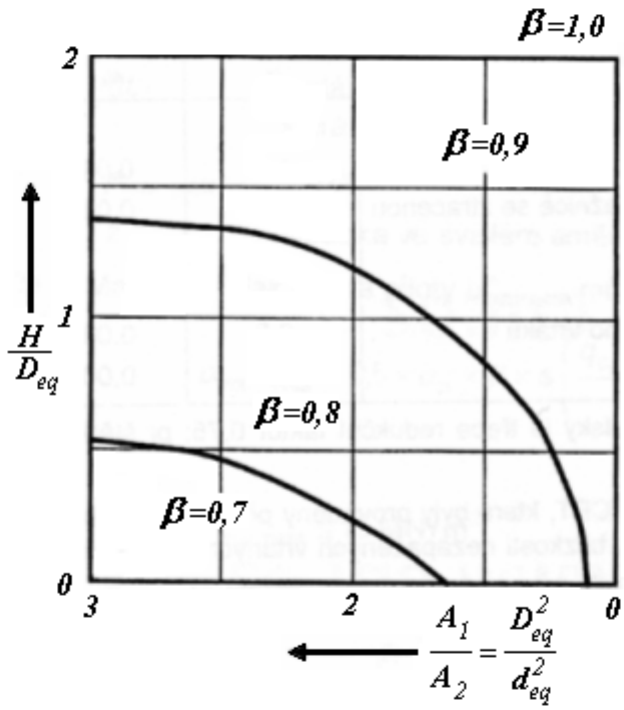
where: a - shorter side of the pile base

b - longer side of the pile base

Coefficient of Influence of Pile Widened Base BETA

This coefficient denoted as β represents the influence of an expanded pile base, its values are evident from the following figure:

(as a function $\frac{D_{eq}^2}{d_{eq}^2}$ of $\frac{H}{D_{eq}}$):



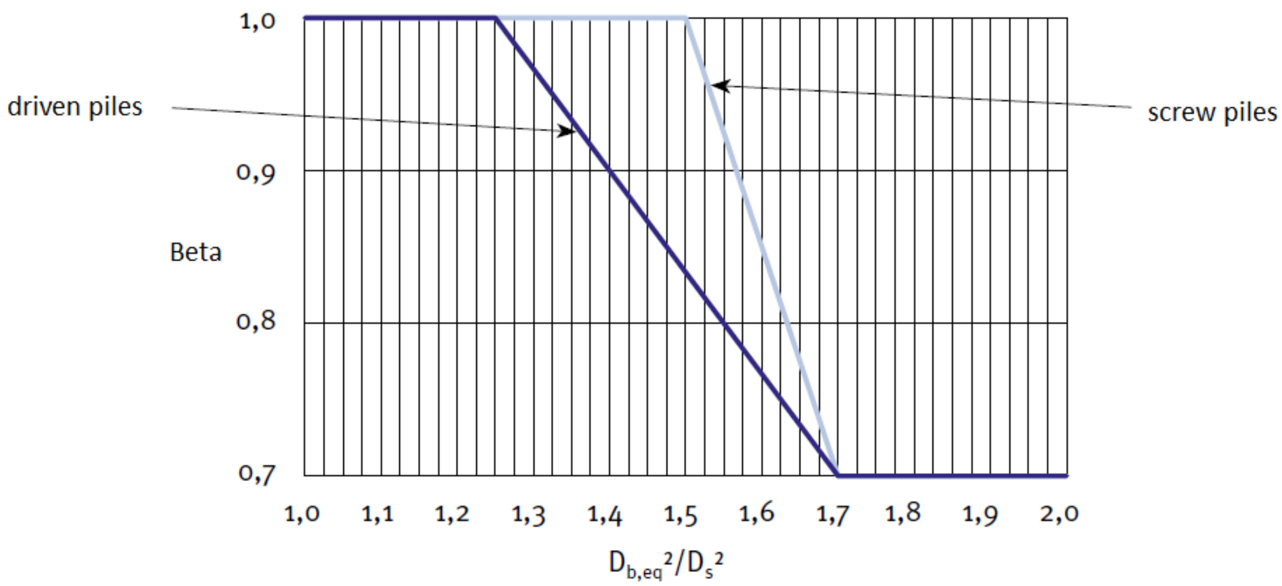
Graph to determine the coefficient β

where: H - pile length $[m]$

D_{eq} - equivalent pile diameter at pile base $[m]$

d_{eq} - equivalent pile shank diameter $[m]$

For the **NBN EN 1997-1 ABN** standard, the coefficient β is interpolated according to the following chart:



where: $D_{b,eq}$ - equivalent pile base diameter $[m]$

D_s - shaft pile diameter $[m]$

Coefficient of Reduction of a Pile Base Bearing Capacity ALFA p

The coefficient of reduction of pile base bearing capacity α_p identifies the type of pile. Its values are determined from one of the available calculation methods or they can be entered manually by the user.

For **NEN 6743** and **EN 1997-2** methods the following built-in values of the coefficient α_p are available:

Piles	αp [-]
prefabricated driven piles or steel piles	1.0
Franki pile	1.0
driven wooden pile	1.0
Vibrating	1.0
cast in place screw piles	0.9
prefabricated screw pile	0.8
cast in place screw piles with additional grouting	0.9
prefabricated screw pile with additional grouting	0.8
steel tubular piles	1.0
continuous Flight Auger (CFA)	0.8
bored piles or piles sheeted by bentonite suspense	0.5
bored piles with steel casing	0.5

For **LCPC** and **Schmertmann** the coefficient is back-calculated based on the value of cone resistance q_c (the values are presented in the following table):

Orientation values of the coefficient α_p based on the cone resistance q_c

LCPC (Bustamante) Soil type	**Cone resistance** q_c $[MPa]$	α_p **for bored piles**	α_p **for driven piles**
Clay	< 1	0.04	0.50
	1 < q_c < 5	0.35	0.45
	5 < q_c	0.45	0.55
Sand	q_c < 12	0.40	0.50

	$13 < q_c$	0.30	0.40

For **NBN EN 1997-1 ANB** standard, the following built-in values of the coefficient α_p are available:

Piles	$\alpha_{p, tert. clay}$ [-]	α_p [-]
prefabricated driven piles or steel piles	1,0	1,0
Franki pile	1,0	1,0
driven wooden pile	1,0	1,0
Vibrating	1,0	1,0
cast in place screw piles	0,8	0,5
prefabricated screw pile	0,8	0,5
cast in place screw piles with additional grouting	0,8	0,5
prefabricated screw pile with additional grouting	0,8	0,5
steel tubular piles	1,0	1,0
continuous Flight Auger (CFA)	0,8	0,5
bored piles or piles sheeted by bentonite suspense	0,8	0,5
bored piles with steel casing	0,8	0,5

Pile Group

Analysis of a group of piles depends on the **structure stiffness**. The basic assumption is that for a stiff structure all piles experience the same settlement, while for a compliant structure each pile deforms independently - no interaction is assumed.

The **maximum bearing capacity of a rigid pile foundation** is given by:

$$F_{r,found,\max} = M.F_{r,\max,rep}$$

where: M - number of piles in the pile foundation

$F_{r,max,rep}$ - single pile bearing capacity in the pile foundation

If adopting the NEN6743 standard then a coefficient of capacity reduction ξ is introduced into the analysis depending on the number piles M and the number of CPTs (article 5.3.2.1). The maximum **bearing capacity of a compliant pile foundation** is determined according to the bearing capacity of the most stressed pile in the group as:

$$F_{r,found,\max} = \max\left(F_{r,i}\right)$$

where: $F_{r,i}$ - bearing capacity of the pile in the group

Calculation of Pile Base Settlement

The magnitude of pile head settlement $w_{1,d}$ is determined as follows:

$$w_{1,d} = w_{toe,d} + w_{el,d}$$

where: $w_{base,d}$ - pile base settlement due to acting force

$$w_{toe,d} = w_{toe,d,1} + w_{toe,d,2}$$

$w_{base,d,1}$ - pile base settlement due to force acting at the base

$w_{base,d,2}$ - pile base settlement due to force acting on the shaft

$w_{el,d}$ - pile settlement due to elastic compression

The magnitudes of settlements $w_{base,.d,1}$ and $w_{base,d21}$ are determined from built-in graphs according to the NEN6743 standard. The value $w_{el,d}$ is given by:

$$w_{el,d} = \frac{LF_{mean,d}}{A_{plast}E_{p,mat,d}}$$

where: L - pile length

$F_{mean,d}$ - mean of force acting on the pile

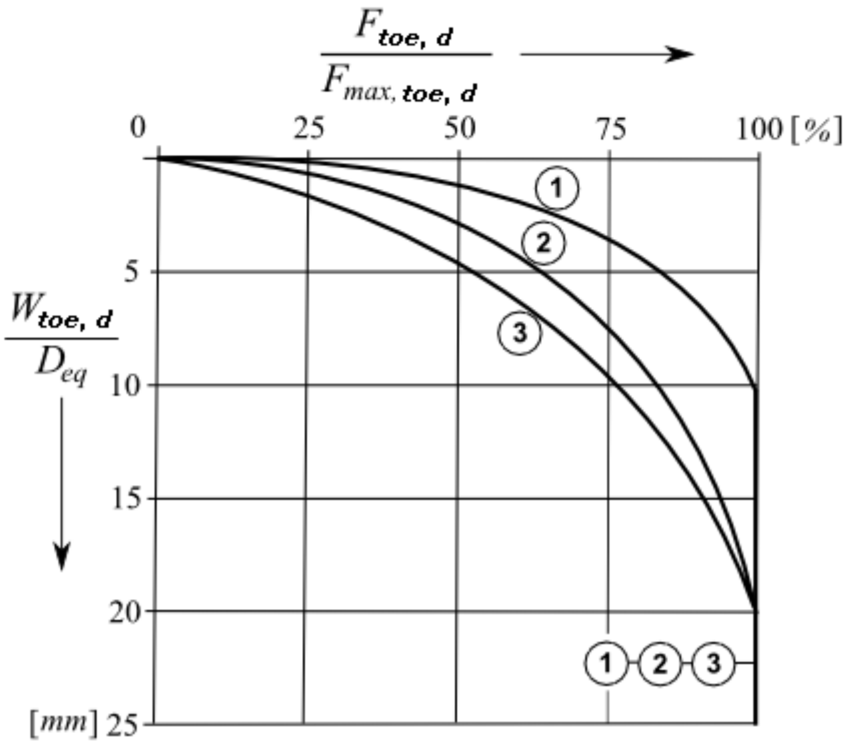
A_{plast} - pile shank cross-sectional area

$E_{p,mat,d}$ - modulus of elasticity of the pile material

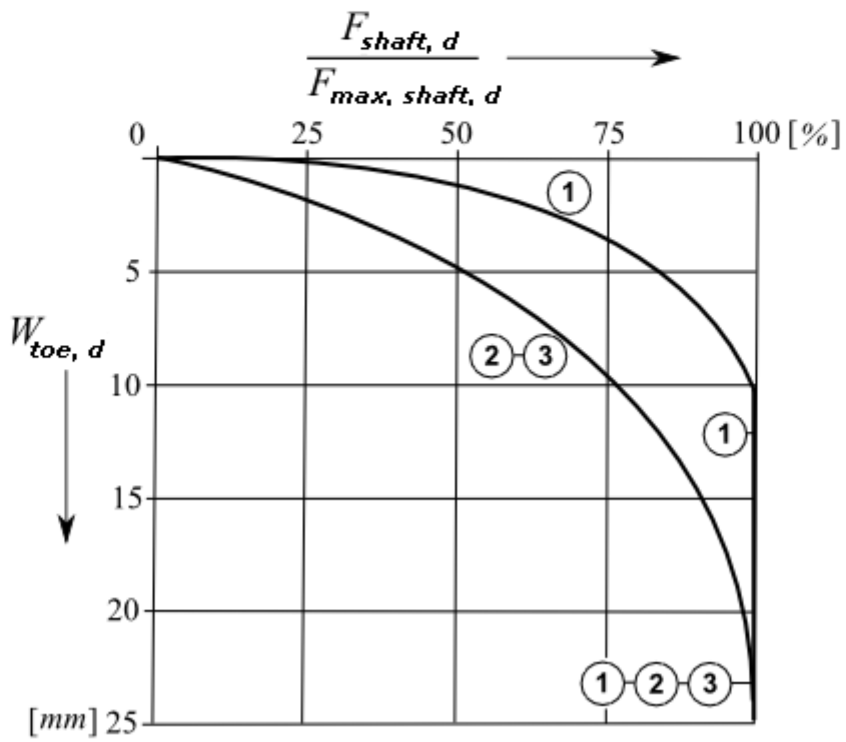
Graphs to Calculate Settlement

Graphs to calculate settlement are taken from the NEN6743 standard (article 6.2.1), which allow us to determine:

- Pile settlement due to base vertical force (pile settlement in percentage of the equivalent pile diameter plotted as a function of the base vertical force given in percentage of the maximum base resistance $F_{max,base}$).
- Pile settlement due to shaft force (pile settlement in mm plotted as a function of the shaft force given in percentage of the maximum shaft resistance $F_{max,shaft}$).



Graph to determine $w_{base,d,1}$ (1 - driven piles, 2 - continuous auger, 3 - bored piles)

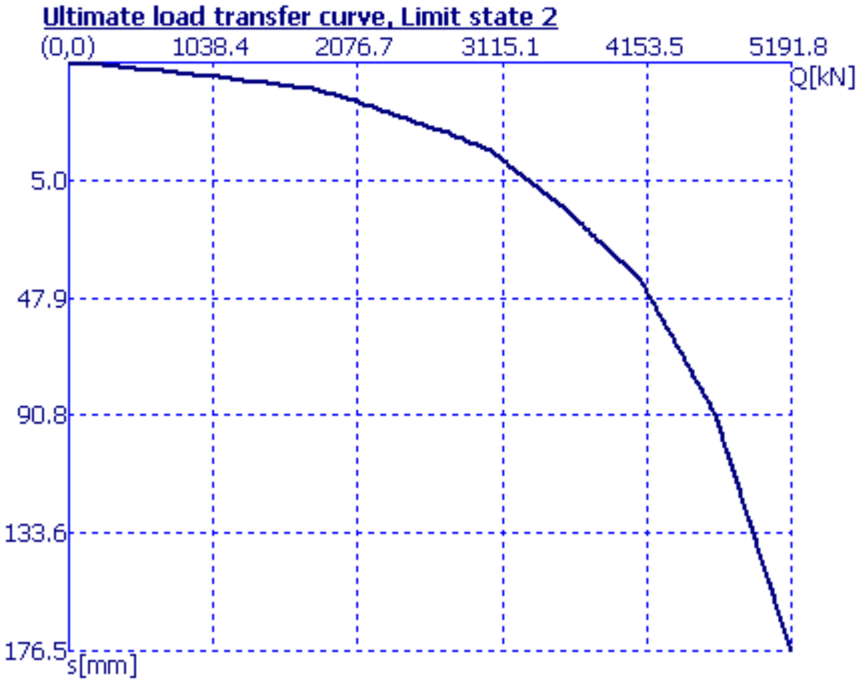


Graph to determine $w_{base,d,2}$ (1 - driven piles, 2 - continuous auger, 3 - bored piles)

Calculation of Load-Settlement Curve

One of the program outputs is a load diagram of the vertically loaded pile - **load-settlement curve**, which plots the pile vertical settlement as a function of the applied load.

The load-settlement curve is determined as a sum of the settlements due to forces at the pile base and on the shaft derived from graphs used to calculate the pile settlement. A typical example of the load-settlement curve appears in the following figure.



Load-settlement curve

Verification

Verification of pile (or group of piles) **bearing capacity** depends on the verification methodology selected in the "Pile CPT" tab:

- verification according to the EN 1997-2
- verification according to the factor of safety or the theory of limit states
- verification according to the NEN 6743

In **settlement** calculation it is possible to use either a load-settlement curve or a load-displacement curve when adopting the NEN 6743 standard.

Verification According to EN 1997-2

The program determines the base and shaft bearing capacities. The result is n values of total bearing capacities for n CPTs:

$$R_{c,i} = R_{b,i} + R_{s,i}$$

$$R_{cd,i} = \frac{R_{b,i}}{\gamma_b} + \frac{R_{s,i}}{\gamma_s}$$

where:

$R_{c,i}$ - bearing capacity from i^{th} CPT

$R_{cd,i}$ - design bearing capacity from i^{th} CPT

$R_{b,i}$ - base bearing capacity from i^{th} CPT

$R_{s,i}$ - shaft bearing capacity from i^{th} CPT

γ_b - partial factor on base resistance (it's defined in the "Piles CPT" tab)

γ_s - partial factor on shaft resistance (it's defined in the "Piles CPT" tab)

Total bearing capacity is determined from:

$$R_c = \min\left(\frac{R_{c,\min}}{\xi_4} \; ; \; \frac{R_{c,mean}}{\xi_3}\right)\frac{1}{\gamma_{cal}}$$

x

$$R_{cd} = \min\left(\frac{R_{cd,\min}}{\xi_4} ; \frac{R_{cd,mean}}{\xi_3}\right)\frac{1}{\gamma_{cal}}$$

where: $R_{c,\,min}$ - minimal bearing capacity

$R_{c,\,mean}$ - average value of the bearing capacity from all CPTs

Partial factor on model uncertainty γ_{cal} is defined in the "Settings" frame (default value is 1).

Correlation coefficients $\xi\,3$ and ξ_4 are either set automatically according to the number of CPTs, or are specified in the frame "Settings". For constructions with sufficient stiffness and resistance the correlation coefficients can be reduced by value 1.1 (result cannot be less than 1.0 after dividing).

Verification of pile is given by the following formula:

$$F_{s,d} < R_{c,d}$$

where: $F_{s,d}$ - design load

$R_{c,d}$ - design pile bearing capacity

Correlation Coefficients for Evaluating of Bearing Capacity of Piles from CPTs

Correlation coefficients ξ for evaluating of bearing capacity from CPTs (n - number of CPTs)

ξ for n =	1	2	3	4	5	7	10
ξ_3	1.40	1.35	1.33	1.31	1.29	1.27	1.25
ξ_4	1.40	1.27	1.23	1.20	1.15	1.12	1.08

For the NBN EN 1997-1 ABN standard, the correlation coefficients are defined according to the density of CPTs on the construction site.

The coefficient ξ_3 is taken as follows:

NUMBER OF PILES	CPT DENSITY				
	1 CPT per 10 m^2	1 CPT per 50 m^2	1 CPT per 100 m^2	1 CPT per 300 m^2	1 CPT per 1000 m^2
1-3	1,25	1,29	1,32	1,36	1,40
4-10	1,15	1,19	1,21	1,25	1,29
> 10	1,14	1,17	1,20	1,24	1,27

The coefficient ξ_4 is taken as follows:

NUMBER OF PILES	CPT DENSITY				
	1 CPT per 10 m^2	1 CPT per 50 m^2	1 CPT per 100 m^2	1 CPT per 300 m^2	1 CPT per 1000 m^2
1-3	1,08	1,17	1,23	1,31	1,40
4-10	1,00	1,07	1,13	1,21	1,29
> 10	1,00	1,06	1,12	1,20	1,27

Verification According to the Safety Factor

The verification analysis according to the factor of safety is selected in the "Pile CPT" tab. This frame also allows us to define the required factor of safety for bearing capacity. Pile verification then assumes the form:

$$\frac{F_{r,d}}{F_{s,d}} > SF_b$$

where: $F_{s,d}$ - pile load

SF_b - safety factor for bearing capacity

$F_{r,d}$ - pile bearing capacity

Verification According to Limit States

The verification according to limit states is selected in the "Pile CPT" tab, including the setting of the pile bearing capacity reduction coefficient. When using the NEN 6743 standard the program automatically performs the verification analysis as specified by this standard and therefore the frame "**Settings**" is not accessible. Pile verification for the **first limit state** assumes the formula:

$$F_{s,d} < \frac{F_{r,d}}{\gamma_t}$$

where:

$F_{s,d}$	-	design pile load
γ_t	-	reduction coefficient of bearing capacity
$F_{r,d}$	-	design pile bearing capacity

Analyses from SPTs

The pile bearing capacity is calculated from standard penetration tests (SPTs) and the **soil profile**.

The calculation method is selected in the "Settings" frame:

- Décourt-Quaresma
- Aoki-Velloso

Décourt-Quaresma Method

Total pile resistance is given by:

$$R_c = R_b + R_s$$

where:

R_c	-	total pile resistance
R_b	-	pile base resistance
R_s	-	pile shaft resistance

The **pile base resistance**:

$$R_b = \alpha . K_{dq} . N . A_b$$

where:

α	-	pile base coefficient
K_{dq}	-	soil parameter
N	-	avarage value of SPT blow counts at the pile base level
A_b	-	base area

The **pile shaft resistance**:

$$R_s = \sum \beta_i . 10 \left(\frac{N_i}{3} + 1 \right) . A_{s,i}$$

where:

β_i	-	pile shaft coefficient at i th layer
N_i	-	avarage value of SPT blow counts at i th layer
$A_{s,i}$	-	shaft area at i th layer

Literature:

Cintra J.C.A., Aoki N. : Fundações por Estacas Projeto Geotécnico, Oficina de Textos, 2010, ISBN 978-85-7975-004-5

Soil Parameter Kdq

The value of K_{dq} parameter is automatically calculated according to the **soil type** or it can be entered manually by the user.

Table of K_{dq} parameter

Soil type	$K\,dq$ $[kPa]$
Sand	400
Sand clayey	400
Sand clayey-silty	400
Sand silty-clayey	400
Sand silty	400
Clay	120
Clay sandy	120
Clay sandy-silty	120
Clay silty-sandy	120
Clay silty	120
Silt	200
Silt sandy-clayey	250
Silt sandy	250
Silt clayey-sandy	200
Silt clayey	200

Coefficients of Resistance α a β

The **pile base coefficient** α depends on the pile type. Its value is automatically calculated according to the **pile and soil types** or it can be entered manually by the user.

Piles	α for sand	α for clay	α for silt
prefabricated driven piles or steel piles	1.0	1.0	1.0
Franki piles	1.0	1.0	1.0
driven wooden piles	1.0	1.0	1.0
vibrating or vibropressed	1.0	1.0	1.0
cast in place screw piles	1.0	1.0	1.0
prefabricated screw piles	1.0	1.0	1.0
cast in place screw piles with additional grouting	1.0	1.0	1.0
prefabricated screw piles with additional grouting	1.0	1.0	1.0
steel tubular piles	1.0	1.0	1.0
Continuous Flight Auger piles (CFA)	0.3	0.3	0.3
bored piles or piles sheeted by bentonite suspense	0.5	0.85	0.6
bored piles with steel casing	0.5	0.85	0.6
root pile	0.5	0.85	0.6
strauss pile	0.5	0.85	0.6

The **pile shaft coefficient** β is automatically determined according to the **pile and soil types** or it can be entered manually by the user.

Piles	β for sand	β for clay	β for silt
prefabricated driven piles or steel piles	1.0	1.0	1.0
Franki piles	1.0	1.0	1.0
driven wooden piles	1.0	1.0	1.0
vibrating or vibropressed	1.0	1.0	1.0
cast in place screw piles	1.0	1.0	1.0
prefabricated screw piles	1.0	1.0	1.0
cast in place screw piles with additional grouting	1.0	1.0	1.0
prefabricated screw piles with additional grouting	1.0	1.0	1.0
steel tubular piles	1.0	1.0	1.0
Continuous Flight Auger piles (CFA)	1.0	1.0	1.0
bored piles or piles sheeted by bentonite suspense	0.6	0.9	0.75
bored piles with steel casing	0.5	0.8	0.65
root pile	1.5	1.5	1.5
strauss pile	0.5	0.8	0.65

Aoki-Velloso Method

Total pile resistance is given by:

$$R_c = R_b + R_s$$

where: R_c - total pile resistance

R_b - pile base resistance

R_s - pile shaft resistance

The **pile base resistance**:

$$R_b = \frac{K_{av}.N}{F_1}.A_b$$

where: K_{av} - soil parameter

N - avarage value of SPT blow counts at the pile base level

F_1 - pile base technology coefficient

A_b - base area

The **pile shaft resistance**:

$$R_s = \sum \alpha_i.\frac{K_{av,i}.N_i}{F_2}.A_{s,i}$$

where: α_i - pile shaft resistance parameter at i th layer

$K_{av,i}$ - soil parameter at i th layer

N_i - avarage value of SPT blow counts at i th layer

F_2 - pile shaft technology coefficient

$A_{s,i}$ - shaft area at i th layer

Literature:

Cintra J.C.A., Aoki N. : Fundações por Estacas Projeto Geotécnico, Oficina de Textos, 2010, ISBN 978-85-7975-004-5.

Soil Parameters Kav And α

The values of K_{av} and α parameters are automatically calculated according to **soil type** or they can be entered manually by the user.

Table of K_{av} and α parameters

Soil type	K_{av} [kPa]	α [%]
Sand	1000	1.4
Sand clayey	600	3.0
Sand clayey-silty	500	2.8
Sand silty-clayey	700	2.4
Sand silty	800	2.0
Clay	200	6.0
Clay sandy	350	2.4
Clay sandy-silty	300	2.8
Clay silty-sandy	330	3.0
Clay silty	220	4.0
Silt	400	3.0
Silt sandy-clayey	450	2.8
Silt sandy	550	2.2
Silt clayey-sandy	250	3.0
Silt clayey	230	3.4

Pile Technology Coefficients

The coefficient F_1 reduces the pile base resistance and coefficient F_2 reduces pile shaft resistance. Values can be entered manually by the user or automatically calculated according to the **pile type** from the following table:

Piles	F_1	F_2
prefabricated driven piles or steel piles	1.2	2.3
Franki piles	2.3	3.0
driven wooden piles	5.0	5.0

vibrating or vibropressed	2.3	3.2
cast in place screw piles	3.0	4.0
prefabricated screw piles	2.5	4.0
cast in place screw piles with additional grouting	3.0	3.5
prefabricated screw piles with additional grouting	2.5	3.5
steel tubular piles	1.8	3.5
Continuous Flight Auger piles (CFA)	3.0	3.8
bored piles or piles sheeted by bentonite suspense	3.5	7.0
bored piles with steel casing	4.0	3.9
root pile	2,2	2,4
strauss pile	4,2	3,9

Spread Footing CPT

This program is used to design and verify spread footings based on data from field tests (CPT, SPT, PMT).

The program performs an analysis for:

- vertical bearing capacity
- horizontal bearing capacity
- settlement
- longitudinal and shear reinforcement (punching)

Bearing capacity analysis

The bearing capacity of the foundation soil can be calculated according to the CPTs, SPTs, or PMTs and verified according to the factor of safety as:

$$\frac{R_d}{\sigma} \geq SF_v$$

where: σ - extreme contact stress at the footing bottom

R_d - bearing capacity of foundation soil

SF_v - safety factor for vertical bearing capacity (for input use the "Analysis" frame)

Extreme design contact stress at the footing bottom is given as:

$$\sigma = \frac{V}{A_{ef}}$$

where: V - extreme vertical force

A_{ef} - effective area of the foundation

The bearing capacity R_d determined from the CPTs:

- "Meyerhof" method
- "Schmertmann" method
- "Skempton" method

The bearing capacity R_d determined from the SPTs:

- "Meyerhof" method

The bearing capacity R_d determined from the PMTs:

- "NF P94-261"

Meyerhof Method (CPT)

This solution for **cohesionless soils** use a **Meyerhof** theory, where the **bearing capacity of foundation soil** is given by a formula:

$$R_d = q_c \frac{b_{ef}}{40} \left(C_{w_1} + C_{w_2} \frac{d}{b_{ef}} \right) R_i$$

It is recommended to use a safety factor $FS = 3$ when calculating the bearing capacity using this method.

where: R_d - bearing capacity of foundation soil

q_c - average value of cone penetration resistance measured at depths from footing base to 1,5$*b_{ef}$ below the footing base

b_{ef} - effective footing width

C_{w1}, C_{w2} - GWT influence factors

d - depth of the footing bottom

R_i - load inclination factor

The formula is derived for imperial units [tsf, ft] - the program calculates automatically in the units used in the program.

GWT influence factors C_{w1} and C_{w2} are determined as follows:

$h_{GWT} = 0$ (water in the terrain level) -> $C_{w1} = C_{w2} = 0{,}5$

$h_{GWT} = d$ (water in the depth of the footing bottom) -> $C_{w1} = 0{,}5$; $C_{w2} = 1$

$h_{GWT} > d + 1{,}5 * b_{ef}$ -> $C_{w1} = C_{w2} = 1$

where: h_{GWT} - depth of groundwater table from the terrain

Intermediate values C_{w1} and C_{w2} are interpolated.

Load inclination factor R_i is interpolated according to the following table:

H/V	R_i		
	$d/b_{ef} = 0$	$d/b_{ef} = 1$	$d/b_{ef} = 5$
0.10	0.75	0.80	0.85
0.15	0.65	0.75	0.80
0.20	0.55	0.65	0.70
0.25	0.50	0.55	0.65
0.30	0.40	0.50	0.55
0.35	0.35	0.45	0.50
0.40	0.30	0.35	0.45
0.45	0.25	0.30	0.40
0.50	0.20	0.25	0.30
0.55	0.15	0.20	0.25
0.60	0.10	0.15	0.20

where: H - horizontal component of the resultant load
V - vertical component of the resultant load

This method should not be used for ratio $H/V > 0{,}6$.

The influence of inclined terrain and inclined footing bottom is considered in the same way as in the Schmertmann method.

Literature:

FHWA-SA-91-043: THE CONE PENETROMETER TEST

Bridge Engineering Handbook (Wai-Fah Chen, Lian Duan, 1999)

Schmertmann Method (CPT)

This solution for **cohesionless soils** use a **Schmertmann** theory, where the **bearing capacity of foundation soil** is given by a formula:

$$R_d = K_q \gamma d N_q + \frac{1}{2} K_\gamma \gamma b_{ef} N_\gamma$$

It is recommended to use a safety factor $FS = 3$ when calculating the bearing capacity using this method.

where: R_d - bearing capacity of foundation soil

K_q , K_γ	-	reduction factors
γ	-	unit weight of the soil in the footing bottom
b_{ef}	-	effective footing width
N_q , N_γ	-	bearing capacity factors
d	-	depth of the footing bottom

The formula is derived for imperial units [tsf , tcf , ft] - the program calculates automatically in the units used in the program.

$$N_q = N_\gamma = 1,25\sqrt{q_{c1}q_{c1}}$$

where: q_{c1} - average value of cone penetration resistance measured at depths from the footing base to 0.5*b_{ef} below the footing base

q_{c2} - average value of cone penetration resistance measured from the depth 0.5*b_{ef} below the footing base to 1.5*b_{ef} below the footing base

$$K_q = i_q s_q d_q b_q g_q$$
$$K_\gamma = i_\gamma s_\gamma b_\gamma g_\gamma$$

where: i_q , i_γ - load inclination factors

s_q , s_γ - footing shape factors

d_q - footing depth factor

b_q , b_γ - footing bottom inclination factor

g_q , g_γ - ground inclination factor

Inclination factors i_q and i_γ are determined as follows:

$$i_q = \left(1 - \frac{H}{V}\right)^2$$

$$i_\gamma = \left(1 - \frac{H}{V}\right)^{m-1}$$

$$m_\gamma = m_l cos^2\theta + m_b sin^2\theta$$

$$m_l = \frac{2 + \frac{l_{ef}}{b_{ef}}}{1 + \frac{l_{ef}}{b_{ef}}}$$

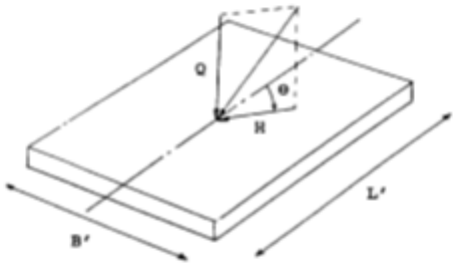
$$m_b = \frac{2 + \frac{l_{ef}}{b_{ef}}}{1 + \frac{b_{ef}}{l_{ef}}}$$

where: H - horizontal component of resultant load

V - vertical component of resultant load

l_{ef} - effective footing length

θ - angle between horizontal component of the resultant load H and the longer axis of footing (in the direction l_{ef})



determination of angle θ

This method should not be used for ratio $H/V >$ 0.4.

Footing shape factors s_q and s_γ are determined as follows:

$$s_q = 1 + \left(\frac{b_{ef}}{l_{ef}}\right) tg\varphi$$

$$s_\gamma = 1 - 0,4\frac{b_{ef}}{l_{ef}}$$

where: φ - angle of internal friction in the footing bottom

Footing depth factor d_q is determined as follows:

$$d_q = 1 + 2tg\varphi\,(1 - sin\varphi)^2\,\frac{d}{b_{ef}}$$

Footing bottom inclination factor b_q and b_γ are determined as follows:

$$b_q = b_\gamma = (1 - \eta tg\varphi)^2$$

where: η - angle of the footing bottom inclination

Ground inclination factor g_q and g_γ are determined as follows:

$$g_q = g_\gamma = (1 - tg\beta)^2$$

where: β - angle of ground inclination

Literature:

FHWA-SA-91-043: THE CONE PENETROMETER TEST

Bridge Engineering Handbook (Wai-Fah Chen, Lian Duan, 1999)

Skempton Method (CPT)

This solution for **cohesive soils** use a **Skempton** theory, where the **bearing capacity of foundation soil** is given by a formula:

$$R_d = K_c.N_c.S_u + \gamma.d$$

It is recommended to use a safety factor $FS =$ 3 when calculating the bearing capacity using this method.

Where: R_d - bearing capacity of foundation soil

K_c - load inclination factor

N_c - Skempton bearing capacity factor

S_u - undrained shear strength

γ - unit weight of the soil in the footing bottom

d - depth of the footing bottom

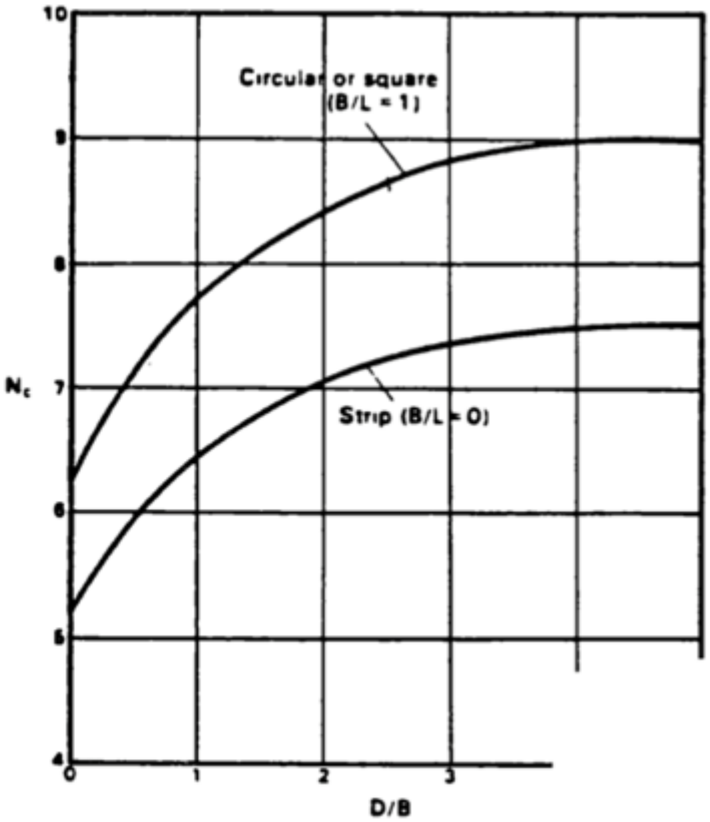
The formula is derived for imperial units [tsf , tcf , ft] - the program calculates automatically in the units used in the program.

$$K_c = \left(1 - 1{,}3.\frac{H}{V}\right)$$

Where: H - horizontal component of resultant load
V - vertical component of resultant load

This method should not be used for ratio $H/V >$ 0,4.

Skempton bearing capacity factor N_c depends on the ratio of effective width b_{ef} and effective length l_{ef} and on the ratio of the footing bottom depth d and effective width b_{ef}.



Determination of N_c factor (From Peck, Hanson a Thornburn, 1974)

Undrained shear strength S_u is determined as follows:

$$S_u = \frac{(q_c - \sigma_{vo})}{N_k}$$

Where: q_c - average value of cone penetration resistance
σ_{v0} - average value of vertical stress measured at depths from footing base to $1{,}5.b\ ef$ below the footing base
N_k - Cone factor (range $<10;20>$)

$$q_c = \sqrt{q_{c1} \cdot q_{c2}}$$

Where: q_{c1} - average value of cone penetration resistance measured at depths from footing base to $0{,}5.b\ ef$ below the footing base
q_{c2} - average value of cone penetration resistance measured from the depth $0{,}5.b\ ef$ below the footing base to $1{,}5.b\ ef$ below the footing base

The influence of inclined terrain and inclined footing bottom is considered in the same way as in the Schmertmann method.

Literature:

FHWA-SA-91-043: THE CONE PENETROMETER TEST

Bridge Engineering Handbook (Wai-Fah Chen, Lian Duan, 1999)

Meyerhof Method (SPT)

This solution for **cohesive** and **cohesionless soils** use a **Meyerhof** theory, where the **bearing capacity of foundation soil** is given by a formula:

$$R_d = N\frac{b_{ef}}{10}\left(C_{w_1} + C_{w_2}\frac{d}{b_{ef}}\right)R_i$$

It is recommended to use a safety factor $FS = 3$ when calculating the bearing capacity using this method.

Where:
- R_d - bearing capacity of foundation soil
- N - average value of SPT blow counts measured at depths from the footing base to $1.5*b_{ef}$ below the footing base
- b_{ef} - effective footing width
- C_{w1}, C_{w2} - GWT influence factors
- d - depth of the footing bottom
- R_i - load inclination factor

The formula is derived for imperial units [tsf, ft] - the program calculates automatically in the units used in the program.

In saturated very fine or silty sands, the measured SPT blow count for $N_i > 15$ should be corrected as follows:

$$N_{cor,i} = 15 + 0,5\,(N_i - 15)$$

This corellation can be performed automatically in the frame "Analysis".

GWT influence factors C_{w1} and C_{w2} are determined as follows:

$h_{GWT} = 0$ (water in the terrain level) -> $C_{w1} = C_{w2} = 0.5$

$h_{GWT} = d$ (water in the depth of the footing bottom) -> $C_{w1} = 0.5$; $C_{w2} = 1$

$h_{GWT} > d + 1,5*b_{ef}$ -> $C_{w1} = C_{w2} = 1$

Where: h_{GWT} - depth of groundwater table from the terrain

Intermediate values C_{w1} and C_{w2} are interpolated.

Load inclination factor R_i is interpolated according to the following table:

H/V	R_i		
	$d/b_{ef} = 0$	$d/b_{ef} = 1$	$d/b_{ef} = 5$
0.10	0.75	0.80	0.85
0.15	0.65	0.75	0.80
0.20	0.55	0.65	0.70
0.25	0.50	0.55	0.65
0.30	0.40	0.50	0.55
0.35	0.35	0.45	0.50
0.40	0.30	0.35	0.45
0.45	0.25	0.30	0.40
0.50	0.20	0.25	0.30
0.55	0.15	0.20	0.25
0.60	0.10	0.15	0.20

Where:
- H - horizontal component of resultant load
- V - vertical component of resultant load

This method should not be used for ratio $H/V > 0.6$.

The influence of inclined terrain and inclined footing bottom is considered in the same way as in the Schmertmann method.

Literature:

Bridge Engineering Handbook (Wai-Fah Chen, Lian Duan, 1999)

NF P94-261 (PMT)

This solution for **cohesive** and **cohesionless soils** use theory from the French standard **NF P94-261**, where the **bearing capacity of foundation soil** is given by a formula:

$$R_d = p^*_{le} k_p i_\delta i_g$$

It is recommended to use a safety factor $FS =$ 3 when calculating the bearing capacity using this method.

Where: R_d - bearing capacity of foundation soil

p^*_{le} - equivalent net limit pressure

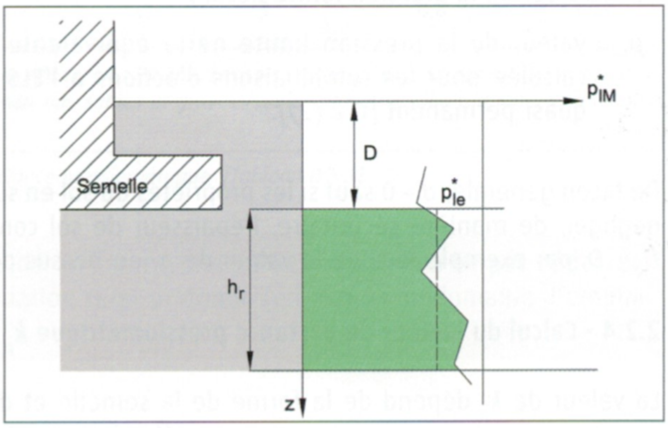
k_p - bearing capacity factor

i_δ - load inclination factor

i_g - ground inclination factor

Equivalent net limit pressure p_{le}^* is a geometric average of values of pressiometric limit pressure from the footing bottom to the depth of h_r.

$$p^*_{le} = \sqrt[n]{\prod_{i=1}^{n} p^*_{l;k}}$$



Depth h_r depends on the type of load (design/service), the geometry of foundation and eccentricity of the load:

<table>
<tr><th></th><th></th><th>design load</th><th>service load</th></tr>
<tr><td rowspan="2">Strip footing</td><td>$\left(1-\frac{2e}{B}\right) \geq \frac{1}{2}$</td><td>$h_r = 1.5B$</td><td rowspan="6">$h_r = 1.5B$</td></tr>
<tr><td>$\left(1-\frac{2e}{B}\right) < \frac{1}{2}$</td><td>$h_r = 3B - 6e$</td></tr>
<tr><td rowspan="2">Circle footing</td><td>$\left(1-\frac{2e}{B}\right) \geq \frac{9}{16}$</td><td>$h_r = 1.5B$</td></tr>
<tr><td>$\left(1-\frac{2e}{B}\right) < \frac{9}{16}$</td><td>$h_r = \frac{8B}{3} - \frac{16e}{3}$</td></tr>
<tr><td rowspan="2">Spread footing</td><td>$\left(1-\frac{2e_B}{B}\right)\left(1-\frac{2e_L}{L}\right) \geq \frac{1}{2}$</td><td>$h_r = 1.5B$</td></tr>
<tr><td>$\left(1-\frac{2e_B}{B}\right)\left(1-\frac{2e_L}{L}\right) < \frac{1}{2}$</td><td>$h_r = min(3B - 6e_B; 3L - 6e_L; 1.5B)$</td></tr>
</table>

Bearing capacity factor k_p depends on the type of soil, geometry of the foundation (ratio L/B) and equivalent embedding depth D_e.

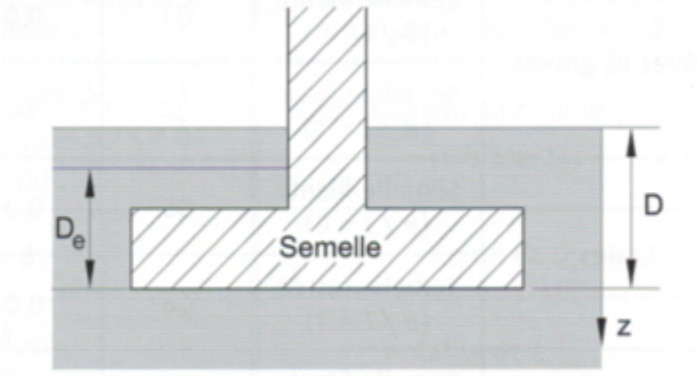
Equivalent embedding depth D_e is calculated as follows:

$$D_e = \frac{1}{p_{le}^*} \int_0^n p_{LM,i}(z) dz$$

or:

$$D_e = \sum_1^n z_i \frac{p_{LM,i}}{p_{le}^*}$$

Values $p_{LM,i}$ for determining equivalent embedding depth D_e are measured between terrain and foundation bottom.



If $D_e/B < 2$:

$$k_p = k_{pmax}$$

Calculation k_p for strip, circle or square foundation:

$$k_{p;B/L} = k_{p0} + (a + b\frac{D_e}{B})(1 - e^{-c\frac{D_e}{B}})$$

Calculation k_p for spread footing (ratio L/B)

$$k_{p;B/L} = k_{p;B/L=0} + (1 - \frac{B}{L}) + k_{p;B/L=1}\frac{B}{L}$$

Coefficients a , b , c , k_{p0} and $k\,pmax$ depend on the type of soil according to the table:

Soil category	Variation curve of the bearing factor		Expression of k_p				
			a	b	c	k_{p0} ($De/B=0$)	kp_{max}
Clays & silts	Strip footing ($B/L=0$)	Q1	0.20	0.02	1.30	0.80	1.022
	Square footing ($B/L=1$)	Q2	0.30	0.02	1.50	0.80	1.123
Sand & gravels	Strip footing ($B/L=0$)	Q3	0.30	0.05	2.00	1.00	1.393
	Square footing ($B/L=1$)	Q4	0.22	0.18	5.00	1.00	1.580
Chalk	Strip footing ($B/L=0$)	Q5	0.28	0.22	2.80	0.80	1.517
	Square footing ($B/L=1$)	Q6	0.35	0.31	3.00	0.80	1.768
Marls Calcareous...	Strip footing ($B/L=0$)	Q7	0.20	0.20	3.00	0.80	1.399
	Square footing ($B/L=1$)	Q8	0.20	0.30	3.00	0.80	1.598

Load inclination factor i_δ is calculated as follows:

$$\delta_d = arctan(\frac{H_d}{V_d})$$

If $\delta_d < \pi/4$:

$$i_{\delta;f} = \left(1 - \frac{2\delta_d}{\pi}\right)^2 - \frac{2\delta_d}{\pi}\left(2 - 3\frac{2\delta_d}{\pi}\right) e^{\frac{-D_e}{B}}$$

If $\delta_d \geq \pi/4$:

$$i_{\delta;f} = \left(1 - \frac{2\delta_d}{\pi}\right)^2 - \left(1 - \frac{2\delta_d}{\pi}\right)^2 e^{\frac{-D_e}{B}}$$

Literature:

NF P94-261

Horizontal bearing capacity

The foundation **horizontal bearing capacity** is verified according to the factor of safety:

$$\frac{R_{dh}}{H} \leq SF$$

where R_{dh} depends on the subsoil type (selected in the "Settings" frame):

- Analysis for **drained conditions:**

$$R_{dh} = Q.\tan\Psi$$

- Analysis for **undrained conditions:**

$$R_{dh} = \min(0{,}4.V_d; A_{ef}.a)$$

and:

$$H = \sqrt{H_x^2 + H_y^2}$$

Where:	ψ	-	angle of friction between the foundation and the soil
	a	-	cohesion between the foundation and the soil
	A_{ef}	-	effective area of the foundation
	H_x, H_y	-	components of the horizontal force
	Q	-	extreme design vertical force
	SF	-	safety factor for the horizontal bearing capacity (input in the "Analysis" frame)

The angle of friction between foundation and soil ψ is recommended:

- For cast in place foundations:

$$\Psi = \varphi$$

- For prefabricated foundations:

$$\Psi = 2/3.\varphi$$

where:	φ	-	angle of internal friction of soil in footing bottom

Literature:

NF P94-261

Settlement analysis

The settlement of spread footing can be calculated according to the CPT, SPT, or PMT tests.

The settlement determined from the CPT tests:

- "Schmertmann" method

The settlement determined from the SPT tests:

- "NAVFAC DM7" method

The settlement determined from the PMT tests:

- "NF P94-261"

Schmertmann Method (CPT)

Settlement of spread footing using CPT tests according to **Schmertmann** theory is based on the formula:

$$s = C_1 . C_2 . \sigma_{ol} . \sum_{1}^{N} \frac{I_{zi} . h_i}{x . q_{ci}}$$

Where: s - settlement of spread footing

C_1 - correction factor for footing depth

C_2 - correction factor for creep settlement

σ_{ol} - stress in the footing bottom

I_{zi} - strain influence factor at the center of the ith sublayer

h_i - thickness of the i^{th} sublayer

χ - modulus factor

q_{ci} - average value of cone penetration resistance in the ith sublayer

The formula is derived for imperial units [*tsf* , *ft*] - the program calculates automatically in the units used in the program.

Correction factor for footing depth C_1 is determined as follows:

$$C_1 = 1 - 0{,}5 . \left(\frac{\sigma_{or}}{\sigma_{ol}}\right)$$

Where: σ_{or} - geostatic stress in the footing bottom

Correction factor for creep settlement C_2 is determined as follows:

$$C_2 = 1 + 0{,}2 . log_{10}\left(\frac{t_{YR}}{0{,}1}\right)$$

Where: t_{yr} - time in years after the application of load

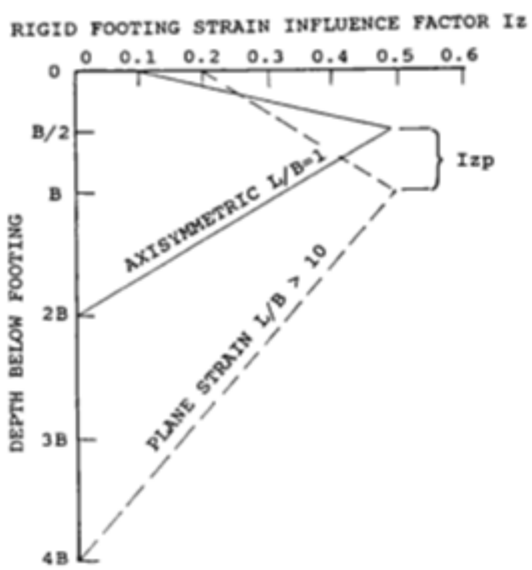
Modulus factor χ depends on the ratio of footing length l and footing width b :

l/b = 1 -> χ = 2,5

l/b = 10 -> χ = 3,5

Intermediate values χ are interpolated.

Strain influence factor I_z is located on the horizontal axis of the following chart. The depth below the footing base is located on the vertical axis.



Determination of I_z factor (Schmertmann, 1978)

Diagram of the factor I_z also depends on the ratio of footing length l and footing width b.

For the ratio $l/b = 1$ maximum value of strain influence factor I_{zp} is in the depth $b/2$ below the footing bottom and overall settlement is calculated to the depth $2b$ below the footing bottom.

For the ratio $l/b = 10$ maximum value of strain influence factor I_{zp} is in the depth b below the footing bottom and overall settlement is calculated to the depth $4b$ below the footing bottom.

The ,aximum value of strain influence factor I_{zp} is determined as follows:

$$I_{zp} = 0{,}5 + 0{,}1.\sqrt{\frac{\sigma_{ol}}{\sigma_{zp}}}$$

Where: σ_{zp} - geostatic stress in the depth of $b/2$ (for ratio $l/b = 1$) or b ($l/b =$ 10) below the footing bottom

Strain influence factor I_z is interpolated for intermediate values of ratio l/b ($1 < l/b <$ 10).

Literature:

Schmertmann, J.H. (1970). Static cone to compute static settlement over sand. ASCE Journal of SoilMechanics & Foundations Division, 96 (3), 1011-1043.

Schmertmann, J.H., Hartmann, J.P. and Brown, P.R. (1978). Improved strain influence factor diagrams,ASCE Journal of the Geotechnical Engineering Division, 104 (GT8), 113

NAVFAC DM7 Method (SPT)

Settlement of spread footing using SPT tests according to **NAVFAC DM7** method is based on the formula:

$$s = \frac{C\sigma}{K_v}\left(\frac{b}{b+1}\right)^2 C_w$$

Where:
- s - settlement of spread footing
- C - footing width factor
- σ - stress in the footing bottom
- K_v - modulus of subgrade reaction
- b - footing width
- C_w - GWT influence factor

The formula is derived for imperial units [tsf, tcf, ft] - the program calculates automatically in the units used in the program.

Footing width factor C is determined as follows:

$b <$ 20 ft -> $C =$ 4

$b >$ 40 ft -> $C =$ 2

Intermediate values C are interpolated.

GWT influence factor C_w is determined as follows:

GWT is located between the terrain level and depth of $1{,}5.b_{ef}$ below the footing bottom:

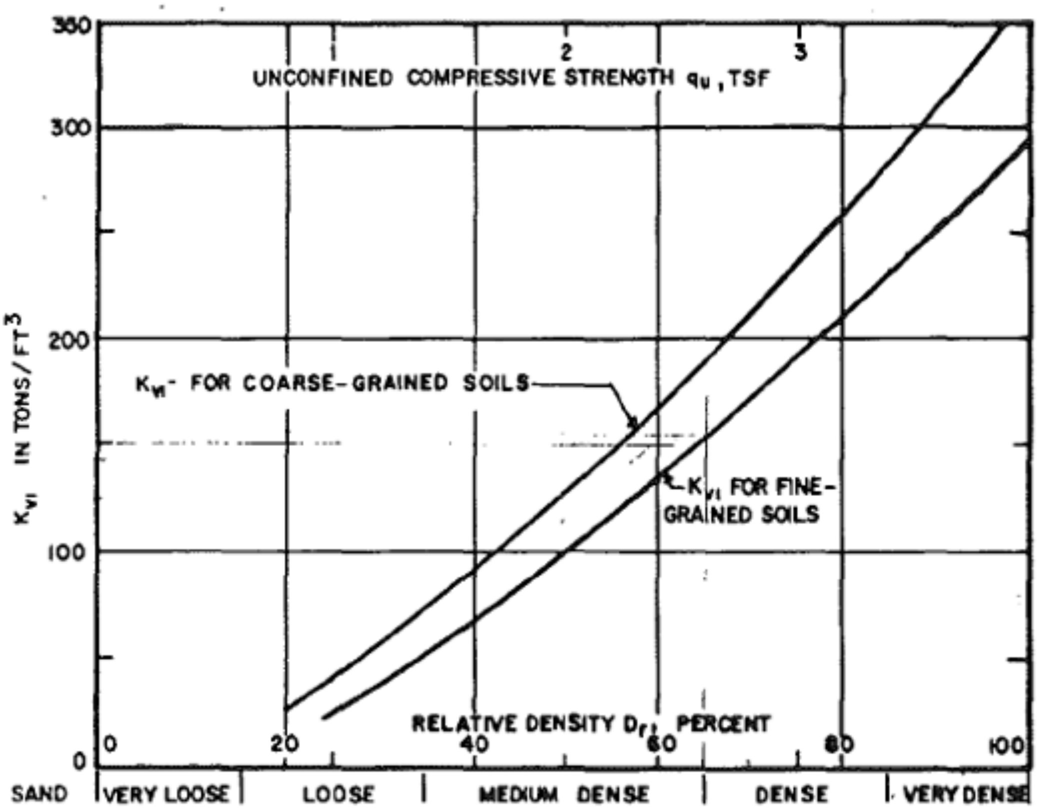
$$C_w = 2.0 - \frac{h_{HPV} - d}{1.5b}$$
$$C_w \leq 2.0$$

GWT is located below the depth of $1{,}5.b_{ef}$ below the footing bottom:

$$C_w = 1.0$$

Where:
- h_{GWT} - depth of groundwater table from the terrain
- d - depth of the footing bottom

Modulus of subgrade reaction K_v depends on relative density factor I_D:



Determination of modulus K_v (Navfac, 1982)

Relative density factor I_D is determined as follows:

$\sigma_{zp} \leq$ 1.5 [ksf]:

$$I_D = \sqrt{\frac{N}{20\left(1 + 2\sigma_{zp}\right)}}$$

$\sigma_{zp} >$ 1.5 [ksf]:

$$I_D = \sqrt{\frac{N}{20\left(3.25 + 0.5\sigma_{zp}\right)}}$$

Where: N - uncorrected blow count at a depth of $b/2$ under the footing bottom

σ_{zp} - geostatic stress at a depth of $b/2$ under the footing bottom

The formula is derived for imperial units [ksf] - the program calculates automatically in the units used in the program.

Literature:

Navfac, 1982

NF P94-261 (PMT)

Settlement of spread footing using PMTs according to **NF P94-261** method is based on the formula:

$$s = \frac{\alpha}{9E_c} q\lambda_c B + \frac{2}{9E_d} qB_0 \left(\lambda_d \frac{B}{B_0}\right)^{\alpha}$$

where: s - overall settlement

E_c , E_d - equivalent pressiometric modulus

α - rheological coefficient

q - stress in the footing bottom

λ_c , λ_d - shape coefficients

B - footing width

B_0 - reference width (0,6 m)

Approximate values of rheological coefficient α for soils:

	Clay	Silt	Sand	Gravel
Overconsolidated	1	2/3	1/2	1/3
Normally consolidated	2/3	1/2	1/3	1/4
Non-consolidated	1/2	1/2	1/3	1/4

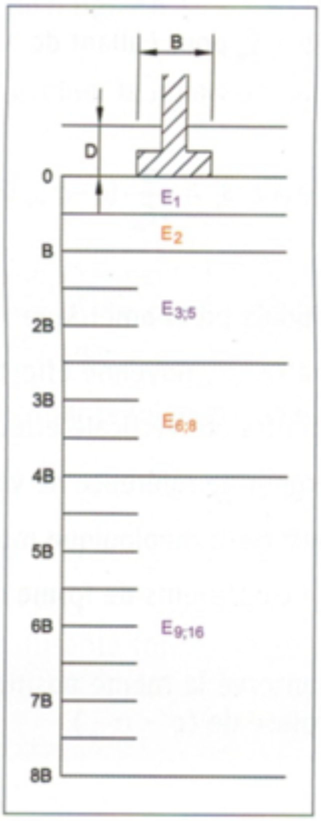
Approximate values of rheological coefficient α for rocks:

Rocks	
Type	α
Almost intact	2/3
Normally fractured	1/2
Very fractured	1/3
Very altered	2/3

Shape coefficients λ_c , λ_d are interpolated according to the following table:

L/B	circle	square	2	3	5	20
λ_d	1	1,12	1,53	1,78	2,14	2,65
λ_c	1	1,1	1,2	1,3	1,4	1,5

Equivalent pressiometric modulus E_c and $E\ d$ are determined according to the pressiometric modulus measured in layers of $B/2$ thickness below the foundation bottom.



Modul E_c is determined as an average pressiometric modulus in the first layer of $B/2$ thickness below the foundation bottom.

$$E_c = E_1$$

The determination of modulus E_d depends on the overall depth of PMT (d_{PMT}).

$$D_{PMT} \geq 8B$$

$$E_d = \frac{1}{\frac{0{,}25}{E_1} + \frac{0{,}3}{E_2} + \frac{0{,}25}{E_{3,5}} + \frac{0{,}1}{E_{6,8}} + \frac{0{,}1}{E_{9,16}}}$$

$$4B \leq D_{PMT} < 8B$$

$$E_d = \frac{1}{\frac{0{,}25}{E_1} + \frac{0{,}3}{E_2} + \frac{0{,}25}{E_{3,5}} + \frac{0{,}2}{E_{6,8}}}$$

$$2,5B \leq D_{PMT} < 4B$$

$$E_d = \frac{1}{\frac{0{,}25}{E_1} + \frac{0{,}3}{E_2} + \frac{0{,}4}{E_{3,5}}}$$

Literature:

CSN EN 1997-2 Annex E

NF P94-261

Menard, L. (1975): "The Menard Pressuremeter: Interpretation and Application of the Pressuremeter Test Results to Foundations Design", Sols-Soils, No. 26, Paris, France.

Pile Group

Analyses performed in the "**Pile Group**" program can be divided into two groups:

- Analytical solution - calculation of the vertical bearing capacity of a pile group for cohesive and cohesionless soils and the determination of settlement
- Analysis of a pile group using the spring method together with the determination of reinforcement of piles

Analytical Solution

Analysis of the vertical bearing capacity of a pile group can be performed for:

- cohesionless soil (analysis for drained conditions)
- cohesive soil (analysis for undrained conditions)

The actual verification analysis is carried out according to the factors of safety or the theory of limit states.

The verification is performed for the **vertical load** only. Load due to moments and shear forces is not considered. To account for horizontal actions of the pile group calls for choosing the spring method in the frame "Settings".

The analytical methods also allow for calculating the pile group settlement.

Cohesionless Soil (Analysis for Drained Conditions)

The same methods as for the analysis of an isolated pile are used to calculate the vertical bearing capacity of a pile group:

- NAVFAC DM 7.2
- Effective stress
- CSN 73 1002

The pile group vertical bearing capacity is provided by:

$$R_g = \sum R_c = nR_c\eta_g$$

where:

	n	-	number of piles in a group
	R_c	-	vertical bearing capacity of an isolated pile
	η_g	-	pile group efficiency

The actual verification analysis is carried out according to the factors of safety or the theory of limit states.

Efficiency of a Pile Group

UFC 3-220-01A

- $\eta_g \approx 0{,}7$ for the axial spacing of piles in the group: 3d
- $\eta_g \approx 1{,}0$ for the axial spacing of piles in the group: 6d

La Barré (CSN 73 1002):

$$\eta_g = 1 - \psi\left[\frac{(n_x - 1)n_y + (n_y - 1)n_x}{90 n_x n_y}\right]$$

$$\psi = arctg\frac{d}{s}$$

where:

- n_x - number of piles in the x direction
- n_y - number of piles in the y direction
- ψ - angle having tangent $tg\psi = d/s$, expressed in degrees
- s - axial spacing of piles
- d - diameter of piles

Seiler-Keeney formula:

$$\eta_g = \left[1 - 0{,}479\left(\frac{s}{s^2 - 0{,}093}\right)\left(\frac{n_x + n_y - 2}{n_x + n_y - 1}\right)\right] + \frac{0{,}3}{n_x + n_y}$$

where:

- n_x - number of piles in the x direction
- n_y - number of piles in the y direction
- s - axial spacing of piles

Input efficiency

User-defined input of the degree of efficiency in the range of 0.5 - 1.0.

Literature:

Pochman, R.; Simek, J.: Pilotove zaklady - Komentar k CSN 73 1002. 1st edition, Prague, Vydavatelstvi norem, 1989, 80 p.

Unified Facilities Criteria (UFC 3-220-01A): Design of deep foundations - Technical instructions, Chapter 5-3, 1997.

Venkatramaiah, C.: Geotechnical Engineering. Second edition, New Delhi (India): New Age International Publishers, 1995.

Cohesive Soil (Analysis for Undrained Conditions)

UFC 3-220-01A:

The bearing capacity of an earth block is provided by:

$$R_g = 2 \cdot l \cdot (b_x + b_y) \cdot c_{us} + N_{cg} \cdot c_{ub} \cdot b_x \cdot b_y$$

where:

$$N_{cg} = 5 \cdot \left[\left(1 + 0{,}2 \cdot \frac{b_x}{b_y}\right) \cdot \left(1 + 0{,}2 \cdot \frac{l}{b_x}\right)\right] \quad \text{for condition: } \frac{l}{b_x} \le 2{,}5$$

$$N_{cg} = 7{,}5 \cdot \left(1 + 0{,}2 \cdot \frac{b_x}{b_y}\right) \quad \text{for condition: } \frac{l}{b_x} > 2{,}5$$

Masopust:

The bearing capacity of an earth block is provided by:

$$R_g = 0,5.(2.(B_x + B_y).l.c_{us} + N_{cg}.B_x.B_y.c_{ub}$$

$$N_{cg} = 5.(1 + \frac{l}{5.B_x}).(1 + \frac{l}{5.B_y})$$

where:

- l - length of piles
- b_x, b_y - plane dimensions of the base of an earth body in the form of a block
- c_{us} - average undrained shear strength along the piles ($\varphi_u \approx 0$)
- c_{ub} - undrained shear strength at the base of piles
- N_{cg} - cohesion group bearing capacity factor

b_x - minimum width of pile group (shorter layout size of the pile cap)

Note: The earth body is represented by a block with its base given by a plane containing feet of individual piles and having vertical walls found in the distance of one pile diameter from the axes of outer piles. This earth block subjected to overall load caused by the pile group resists by shear along the walls - **skin friction** and by bearing capacity at its **base**.

The actual verification analysis is carried out according to the factors of safety or the theory of limit states.

Literature:

UFC 3-220-01A

MASOPUST, Jan. Navrhování základových a pažicích konstrukcí: příručka k ČSN EN 1997. Praha: Pro Českou komoru autorizovaných inženýrů a techniků činných ve výstavbě vydalo Informační centrum ČKAIT, 2012. ISBN 978-80-87438-31-2.

Analysis According to the Safety Factor

When performing the analysis according to the factor of safety the program carries out the verification analysis for a **pile group in compression**:

$$\frac{R_g}{V_d + W_p} > SF_{cp}$$

where:

R_g - vertical bearing capacity of a pile group

V_d - maximum vertical force (including the pile cap self-weight)

W_p - self-weight of piles (only when the option "**Consider the self-weight of pile**" is checked)

SF_{cp} - factor of safety for a pile group in compression

Analysis According to the Theory of Limit States

When performing the analysis according to the theory of limit states the program carries out the verification analysis for a pile group in a **cohesionless soil**:

$$R_g = n\frac{R_c}{\gamma_t}\eta_g \geq V_d + W_p$$

where:

R_g - vertical bearing capacity of a pile group

n - number of piles in the group

R_c - vertical bearing capacity of an isolated pile ($R_b + R_s$)

γ_t reduction coefficient of total resistance

η_g - pile group efficiency

V_d - maximum vertical force (including the pile cap self-weight)

W_p - self-weight of piles (only when the option "**Consider the self-weight of pile**" is checked)

When performing the analysis according to the theory of limit states the program carries out the verification analysis for a pile group in a **cohesive soil**:

$$\frac{R_g}{\gamma_t} \geq V_d + W_p$$

where:

R_g - vertical bearing capacity of a pile group

V_d - maximum vertical force (including the pile cap self-weight)

W_p - self-weight of piles (only when the option "**Consider the self-weight of pile**" is checked)

γ_t - reduction coefficient of total resistance

When performing the verification analysis according to EN 1997-1 the pile group vertical bearing capacity in a cohesive soil is reduced by the coefficient of base resistance ($\gamma_t = \gamma_b$).

Pile Group Settlement

Cohesionless soil

The analysis of a pile group in a cohesionless soil is developed based on the linear theory of settlement (Poulos). The load-settlement curve for a pile group and the value of the total settlement s_g is increased by the so-called **group settlement factor** g_f.

An immediate settlement of the pile group increased by the group settlement factor is provided by:

$$s_g = g_f \cdot s_0$$

$$g_f = \sqrt{\frac{b_x}{d}}$$

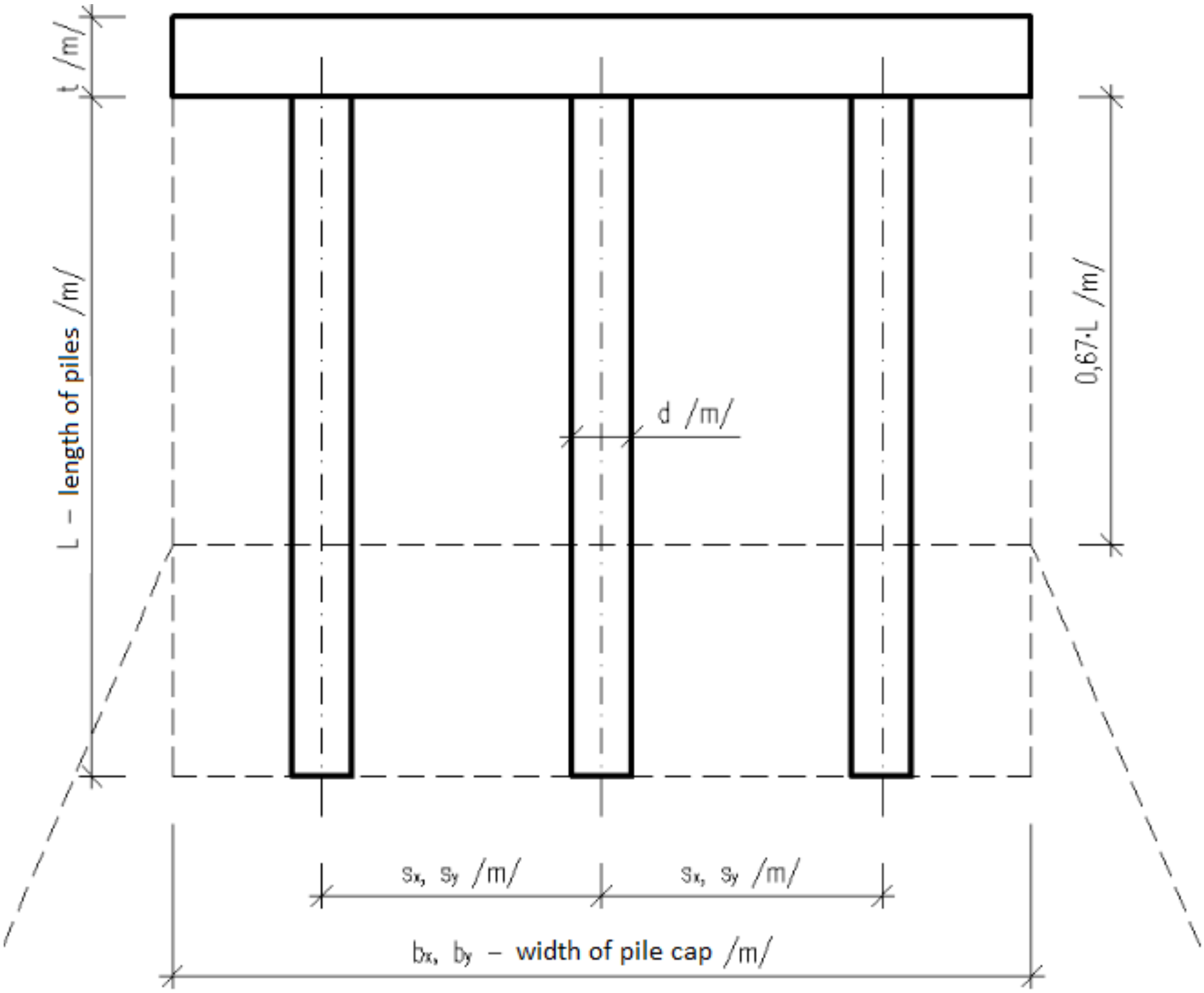
where:

s_g	-	pile group settlement
g_f	-	group settlement factor for a cohesionless soil (according to Pile Buck Inc. 1992)
s_0	-	settlement of a single pile (determined, e.g. from the load-settlement curve)
d	-	pile diameter
b_x	-	minimum width of pile group

Cohesive soil

The pile group settlement in a cohesive soil is determined as the settlement of a substitute foundation at a depth of $0{,}67 * L$, having a width B and a length B'.

Analyses to calculate settlement are described in more detail in "Settlement analysis".

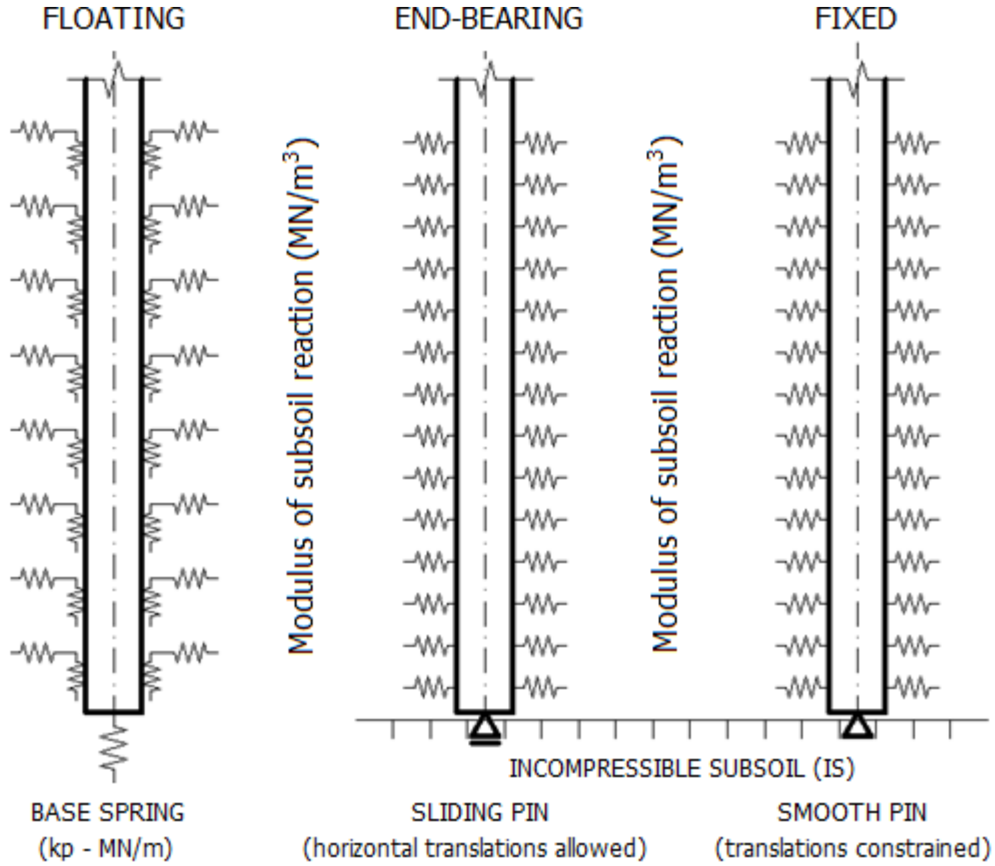


Scheme of substitute foundation - settlement of pile group in cohesive soil

Spring Method

The pile group is analyzed using the Finite Element Method. The pile cap is considered infinitely stiff. A general load is applied in the center of the cap and can be imported from an arbitrary program that performs a static analysis.

The piles analyzed according to figure:



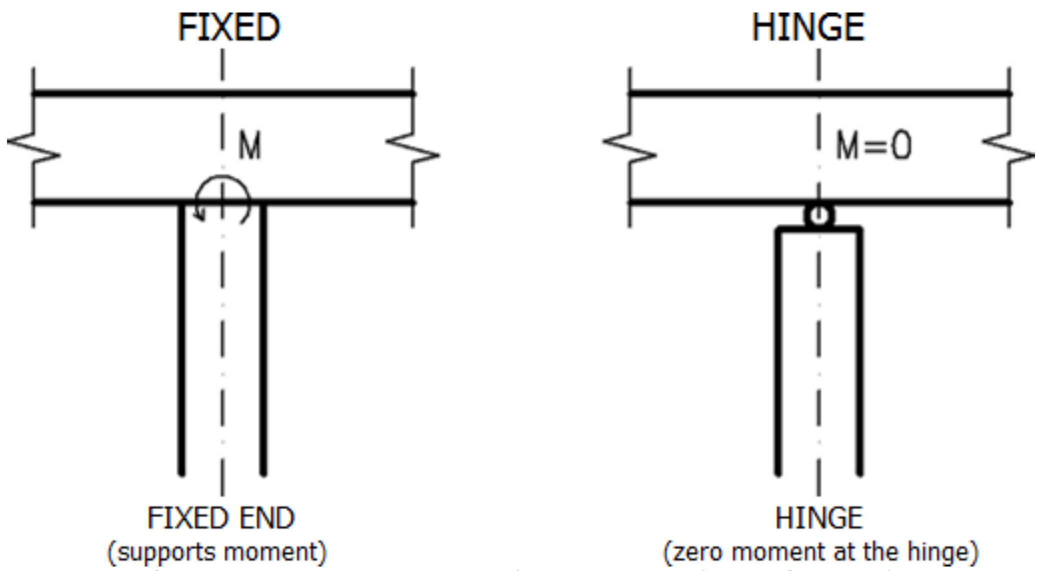
Modeling piles

Four options to model piles are available in the frame "Settings":

1. Floating piles - compute the stiffness of springs from soil parameters
2. Floating piles - input the stiffness of springs
3. Piles resting on the rock subgrade
4. Piles fixed into the rock subgrade

All options require inputting the "Horizontal modulus of subsoil reaction" characterizing the pile behavior in the transverse direction. Floating piles further require determining the stiffness of vertical springs. The program allows for back calculation of this stiffness from the available soil parameters and the typical load. They can also be inputted directly in the frame "Vertical springs".

Either hinge or fixed-type connection of a pile to the pile cap can be considered.



Connecting pile to the pile-cap (selected in the frame "Settings")

The analysis of the structure is performed using the finite element method (FEM). Each pile is divided into ten elements. For each element the program defines the magnitude of horizontal and vertical springs. In comparison to a single pile, the stiffness of horizontal and vertical springs are further reduced for both the inner and outer piles - the horizontal stiffness is reduced by the coefficients equal to $0,5$ and $0,25$ for the outer and inner piles, respectively; the shear stiffness is reduced by the coefficients equal to $0,5$ and $0,1$ for the outer and inner piles, respectively. These reductions well represent the real behavior of a pile group. The springs at the pile-base are not reduced.

Calculation of Stiffness of Vertical Springs

When back calculating the stiffnesses of vertical springs it is necessary to input a **typical load** in the frame "Vertical springs" that will serve to determine the spring stiffnesses. This load should be selected such as to characterize the structure behavior as close as possible.

The stiffnesses are determined as follows:

1. Typical load is applied to individual piles
2. The stiffness of shear vertical springs distributed along the pile is calculated depending on the soil parameters.
3. The stiffness of vertical spring at the pile-base is calculated depending on the stiffness of the subsoil below the pile-base and the depth of influence zone. For tensile piles this stiffness is equal to zero.

These stiffnesses are further adjusted according to their location in the pile group - the shear stiffness is reduced by the coefficients equal to 0.5 and 0.1 for the outer and inner piles, respectively.

Micropile

The program performs verification analysis of micropiles (reinforced by steel tube)

- based on the limit states
- based on the factor of safety

Both the root section and micropile tube (micropile cross-section) are examined for both cases. When examining the micropile tube the analysis may include expected lifetime of the micropile.

Verification Based on Safety Factor

The program performs verification analysis of the micropile tube and root:

Verification of the cross-section (tube)

Both, internal stability of section and coupled section bearing capacity, are verified.

1. Internal stability of section

$$\frac{N_{cr}}{N_{max}} > SF_f$$

where:

N_{cr} - standard critical normal force, calculated in dependence on the method set in the "Micropiles" tab

N_{max} - maximal normal force entered in the frame "Load"

SF_f - critical force safety factor entered in the "Micropiles" tab

2. Coupled section bearing capacity

$$\frac{R_s}{\sigma_s} > SF_s$$

where:

R_s - standard strength of steel, entered in the frame "Material"

σ_s - stress in steel, calculated according to the way of load (section loaded only by normal force or by a combination of bending moment and normal force)

SF_s - safety factor of section resistance, entered in the "Micropiles" tab

Verification of the root

$$\frac{Q}{N_{max}} > SF_r$$

where:

Q - standard root bearing capacity, calculated in dependence on used method (see "Bearing capacity of the micropile root section")

N_{max} - maximal normal force entered in the frame "Load"

SF_r - root resistance safety factor entered in the "Micropiles" tab

Verification Based on Limit States

The program performs verification analysis of the micropile tube and root:

Verification of the cross-section (tube)

Both, internal stability of section and coupled section bearing capacity, are verified.

1. Internal stability of section

$$N_{\max} < N_{crd}$$

where: N_{max} - maximal normal force entered in the frame "Load"

N_{crd} - design critical normal force

$$N_{crd} = \frac{N_{cr}}{\gamma_{mf}}$$

where: N_{cr} - standard critical normal force, calculated in dependence on the method set in the "Micropiles" tab

γ_{mf} - reduction coefficient of critical force entered in the "Micropiles" tab (limit states)

2. Coupled section bearing capacity

$$\sigma_s < R_{sd}$$

where: σ_s - stress in steel, calculated according to the way of load (section loaded only by normal force or by a combination of bending moment and normal force)

R_{sd} - design strength of steel

$$R_{sd} = \frac{R_s}{\gamma_{ss}}$$

where: R_s - standard strength of steel entered in the frame "Material"

γ_{ss} - reliability coefficient of steel entered in the "Micropiles" tab (limit states)

Verification of the root

$$N_{\max} < Q_{rd}$$

where: N_{max} - maximal normal force entered in the frame "Load"

Q_{rd} - design root bearing capacity

$$Q_{rd} = \frac{Q}{\gamma_r}$$

where: Q - standard root bearing capacity, calculated in dependence on the used method (see "Bearing capacity of the micropile root section")

γ_r - reduction coefficient of root resistance entered in the "Micropiles" tab (limit states)

Verification of the Micropile Tube

When calculating the tube bearing capacity (micropile cross-section) the program differentiates between a micropile loaded in tension or in compression.

In case of tension the program determines coupled section bearing capacity (strength of cement mixture is not considered).

In case of compression the program examines both, a coupled section bearing capacity and the internal stability of the section, depending on the method set in the "Micropiles" tab.

Coupled Section Bearing Capacity

In case of coupled section bearing capacity, the micropile tube is examined against the failure due to load caused by normal force or by a combination of bending moment and normal force.

When determining the coupled section bearing capacity it is possible to involve the influence of the expected lifetime of the micropile.

Micropile Lifetime

The micropile lifetime is introduced by reducing the area of the reinforcing tube using the reduction coefficient of the influence of corrosion of steel tube r_e and coefficient F_{ut} taking into account the connection of the micropile and the surrounding soil.

$$A_a = \frac{\pi}{4}\left[\left(D-2.r_e\right)^2-\left(D-2.t\right)^2.\right]F_{ut}$$

where:

D	-	external diameter of reinforcing tube
t	-	wall thickness of reinforcing tube
F_{ut}	-	coefficient taking into account the connection of micropile and surrounding soil (0,5 -1,0)
r_e	-	coefficient of influence of corrosion of steel tube

Literature:

BS EN 14199:2005 Execution of special geotechnical works. Micropiles British-Adopted European Standard / 30-Mar-2005 / 52 pages ISBN: 0580457249.

Coefficient of the Influence of Corrosion

Coefficient of influence of corrosion of steel tube r_e [mm] (based on EN 14199)

Type of soil	Required lifetime of micropile [$years$]				
	5	25	50	75	100
Soils in natural deposition	0.00	0.30	0.60	0.90	1.20
Soils in natural deposition contaminated	0.15	0.75	1.50	2.25	3.00
Organic soils	0.20	1.00	1.75	2.50	3.25
Loose soils	0.18	0.70	1.20	1.70	2.20
Special soils (containing soluble salts)	0.50	2.00	3.25	4.50	5.75

Note: Values of the coefficient of influence of corrosion of steel tube r_e are for intermediate values.

Bearing Capacity of Cross Section Loaded by Normal Force

Tension normal force

In case of tension force, the stress in steel part of cross section is calculated using the following formula:

$$\sigma_s = \frac{N}{A_s}$$

where:

σ_s	-	stress in steel
N	-	normal force acting in section
A_s	-	area of the steel part of the micropile cross section

Compressive normal force

Bearing capacity of the cross section in compression, reduced by buckling coefficient, is determined as:

$$N_{c,u} = \chi.\left(A_s.R_{sd} + A_c.R_{cd}\right)$$

where:

χ	-	buckling coefficient
A_s	-	area of the steel part of the micropile cross section
A_c	-	area of the cement mixture part of the micropile cross section
R_{sd}	-	design strength of steel
R_{cd}	-	design strength of cement mixture in compression

Design strengths are equal to standard values in the verification based on the factor of safety.

Design strengths of steel and cement mixture are calculated in the verification based on the theory of limit states as follows:

$$R_{sd} = \frac{R_s}{\gamma_{ss}}$$

$$R_{cd} = \frac{R_c}{\gamma_{sc}}$$

where:

R_s - standard strength of steel, entered in the "Material" frame

γ_{ss} - reduction coefficient of steel strength, entered in the "Micropiles" tab

R_c - standard strength of cement mixture in compression, entered in the "Material" frame

γ_{sc} - reduction coefficient for cement mixture, entered in the "Micropiles" tab

The stress in the steel part of the cross section is determined as:

$$\sigma_s = \frac{N}{N_{c,u}} . R_{sd}$$

where:

N - normal force acting in section

$N_{c,u}$ - bearing capacity of the cross section in compression, reduced by influence of buckling

R_{sd} - design strength of steel

Bearing Capacity of Cross Section Loaded by Combination of Bending Moment and Normal Force

A cross-section loaded by a combination of bending moment and normal force requires the determination of a neutral axis, dividing the cross-section into a **tensile** and **compressed** part. When searching the position of the neutral axis, the influence of buckling is included, i.e. normal force is increased by dividing it by coefficient of buckling χ . The neutral axis is searched following the procedure known from the dimensioning of concrete cross-sections, reinforced by steel, as a limit equilibrium method. Compression is transmitted by a part of a steel tube and cement mixture filling. Tension is taken by the remaining part of the steel tube, the cement mixture in tension, is not considered.

The bearing capacity in bending is determined by the following formula:

$$M_u = R_{sd} . \left(A_{s,t} \, t_{s,t} + A_{s,c} . t_{s,c} \right) + R_{cd} . A_{c,c} . t_{c,c}$$

where:

R_{sd} - design strength of steel

$A_{s,t}$ - area of the tensile part of the steel micropile cross-section

$A_{s,c}$ - area of the compressed part of the steel micropile cross-section

$A_{c,c}$ - area of the compressed part of the cement mixture cross-section

$t_{s,t}$ - location of the center of tensile steel part

$t_{s,c}$ - location of the center of compressed steel part

$t_{c,c}$ - location of the center of compressed cement mixture part

R_{cd} - design strength of cement mixture in compression

Design strengths are equal to standard values in the verification based on the factor of safety.

Design strengths of steel and cement mixture are calculated in the verification based on the theory of limit states as follows:

$$R_{sd} = \frac{R_s}{\gamma_{ss}}$$

$$R_{cd} = \frac{R_c}{\gamma_{sc}}$$

where:

R_s - standard strength of steel, entered in the frame "Material"

γ_{ss} - reliability coefficient of steel, entered in the "Micropiles" tab

R_c - standard strength of cement mixture in compression, entered in the frame "Material"

γ_{sc} - reliability coefficient of cement mixture, entered in the "Micropiles" tab

The stress in the steel part of the cross-section is determined as:

$$\sigma_s = \frac{M}{M_u} . R_{sd}$$

where:
- M - bending moment acting in section
- M_u - bearing capacity in bending
- R_{sd} - design strength of steel

Influence of Buckling

The analysis is preceded by the determination of characteristics of an ideal cross-section, in which the effect of the cement mixture cross-section is transformed into steel. The slenderness of element is determined as:

$$\lambda = \frac{l_{cr}}{i}$$

where:
- l_{cr} - element buckling length
- i - radius of gyration of the ideal cross-section

$$l_{cr} = \sqrt{\frac{E.I.\pi^2}{N_{cr}}}$$

where:
- E - modulus of elasticity of the ideal cross-section
- I - moment of inertia of the ideal cross-section
- N_{cr} - standard critical normal force, calculated in dependence on the method set in the "Micropiles" tab

Recounted slenderness λ_p is determined next:

$$\lambda_p = \lambda \sqrt{\frac{R_{sd}}{210}}$$

where:
- R_{sd} - design strength of steel (in calculation based on the factor of safety design strength is equal to the standard strength)

$$R_{sd} = \frac{R_s}{\gamma_{ss}}$$

where:
- R_s - standard strength of steel, entered in the frame "Material"
- γ_{ss} - reliability coefficient of steel, entered in the "Micropiles" tab (limit states)

Buckling coefficient χ is determined according to slenderness λ_p with the help of the following formulas:

$$\chi = \frac{1}{2}\left[1{,}26 + \left(\frac{93}{\lambda_p}\right)^2\right] - \sqrt{\frac{1}{4}\left[1{,}26 + \left(\frac{93}{\lambda_p}\right)^2\right]^2 - \left(\frac{93}{\lambda_p}\right)^2} \qquad for: \lambda_p \leq 250$$

$$\chi = \left\{\frac{1}{2}\left[1{,}26 + \left(\frac{93}{250}\right)^2\right] - \sqrt{\frac{1}{4}\left[1{,}26 + \left(\frac{93}{250}\right)^2\right]^2 - \left(\frac{93}{250}\right)^2}\right\}\left(\frac{250}{\lambda_p}\right)^2 \qquad for: \lambda_p > 250$$

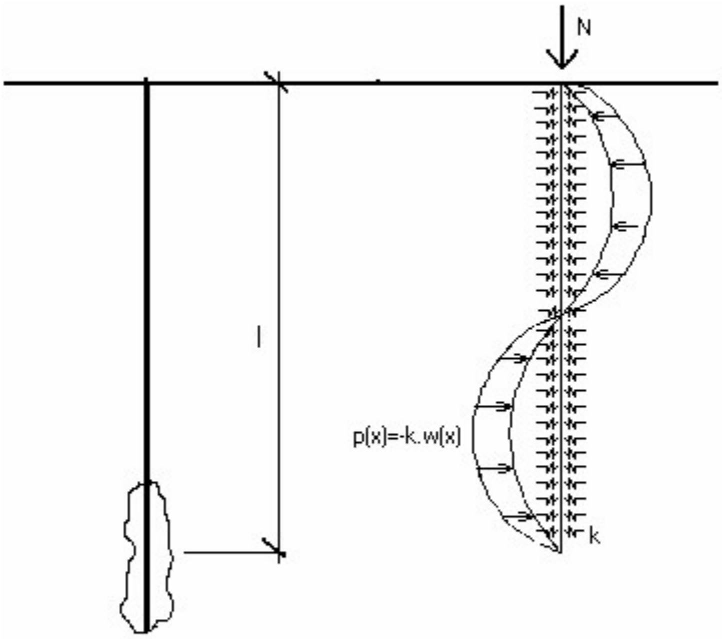
Internal Stability of Section

The internal stability of a section, examines the failure of a micropile due to buckling into the surrounding soil. The crucial step for the determination of the internal stability of the section is the calculation of the normal force N_{cr} that depends on the micropile length, the surrounding soil, and other effects. User can choose one of the following solution methods in the "Micropiles" tab for calculating critical normal force N_{cr}:

- Geometric method (Euler)
- Salas theory
- Véas-Souche theory

Geometric Method (Euler)

The soil surrounding the micropile is represented in the program by the modulus of subsoil reaction E_p (Winkler constant k) defined by the user in the frame "Verification of cross-section". A model of a structure is displayed in the figure.



Model of structure

For a micropile in compression it is expected that a varying number of half-waves occurs depending on the geometry and stiffness of the structure and surrounding soil, respectively. The solution to this case arises from the equation of bending of a straight beam.

$$w'' = -\frac{M + N.w}{E.I}$$

After some manipulations the bending equation can be expressed as:

$$w_{(x)} = C_1.\cos(A.x) + C_2.\sin(A.x) + C_3.\cos(B.x) + C_4.\sin(B.x)$$

where:

$$A = \sqrt{\frac{\alpha^2}{2} + \sqrt{\frac{\alpha^4}{2} - 4.\beta^4}}$$

$$B = \sqrt{\frac{\alpha^2}{2} - \sqrt{\frac{\alpha^4}{2} - 4.\beta^4}}$$

$$\alpha = \sqrt{\frac{N}{E.I}}$$

$$\beta = \sqrt[4]{\frac{k}{4.E.I}}$$

Integration constants C_1-C_4 are found from four boundary conditions depending on the assumed endpoint supports.

The critical force N_{cr} is calculated from the formula (in [1]):

$$N_{cr}(l_p) = E_i I_i \frac{\pi^2}{l_p^2} n^2 + E_r \frac{l_p^2}{\pi^2} n^{-2} \qquad (1)$$

where:
- E_i - modulus of elasticity of a ideal cross-section
- I_i - moment of inertia of a ideal cross-section
- l_p - effective micropile length (free length + 1/2 root length)
- E_r - reaction of soil in the horizontal direction
- n - number of half-waves

The critical force N_{cr} is taken as a minimum of function (1). It is achieved for the length of half-wave:

$$\frac{l_p}{n} = \pi \sqrt[4]{\frac{E_i I_i}{E_r}} \qquad (2)$$

The number of half-waves n is based on formula (2):

$$n = \frac{l_p}{\pi} \sqrt[4]{\frac{E_r}{E_i I_i}} \qquad (3)$$

If the part of micropile is above the terrain (pile head offset), the reduced values n a E_r are calculated as follows:

$$n_{red} = \frac{l_v}{l_p} + \left(1 - \frac{l_v}{l_p}\right) n \qquad (4)$$

$$E_{r,red} = E_r \left(1 - \frac{l_v}{l_p}\right) \qquad (5)$$

where: l_v - length of micropile above the terrain

Assuming **hinge-hinge** condition,the following equation is used:

$$N_{cr} = E_i I_i \frac{\pi^2}{l_p^2} n_{red}^2 + E_{r,red} \frac{l_p^2}{\pi^2} n_{red}^{-2} \qquad (6)$$

Assuming **hinge-fixed** condition, the following equation is used:

$$N_{cr} = E_i I_i \frac{2\pi^2}{l_p^2} n_{red}^2 + E_{r,red} \frac{l_p^2}{2\pi^2} n_{red}^{-2} \qquad (7)$$

Literature:

[1] *Timoshenko, S. P.: Theory of Elastic Stability, New York, 1936*

Salas Theory

The critical force N_{cr} for basic support conditions in the micropile head (determining micropile deflection) follows from:

$$N_{cr} = \pi^2 . \frac{E_a . I_a}{\left(l + l_{ef}\right)^2} . A$$

where:
- $E_a * I_a$ - bending stiffness of micropile reinforcing tube
- l - free length of micropile length
- l_{ef} - length of the fictitious fixed end
- A - constant reflecting the type of support in micropile head

$$l_{ef} = 1,2 . f . l_e$$

where:
- f - coefficient depending on the ratio of modulus of elasticity of soil in micropile head and base
- l_e - elastic length of micropile given by:

$$l_e = \left(\frac{3.E_a I_a}{E_l} \right)^{\frac{1}{4}}$$

where: $E_a * I_a$ - bending stiffness of micropile reinforcing tube

E_l - modulus of elasticity of soil in the micropile base

Literature:

Jiménez Salas J.A. a kol:Geotecnica y Cimientos III, Capitulo 3, Rueda, Madrid (Spanish).

Constant A Reflecting the Type of Support in the Micropile Head

Constant A reflecting the type of support in the micropile head

Type of support in the micropile head	A [-]
Hinged	2.045
Free	0.25
Fixed	4.0
Horizontally movable	1.0

Coefficient f

Coefficient f

E_o / E_l [1] [-]	f [-]
0	1.70
0.5	1.25
1	1.00

[1] E_o - the modulus of elasticity of soil below terrain surface (at the micropile head)

E_l - the modulus of elasticity of soil at the micropile root

Véas-Souche Theory

The critical force N_{cr} is calculated according to the formula [1]:

$$N_{\mathrm{cr}}(l_{\mathrm{p}}) = \frac{\pi^2 EI}{\left(\frac{l_{\mathrm{p}}}{m}\right)^2} + \frac{E_{\mathrm{r}} \left(\frac{l_{\mathrm{p}}}{m}\right)^2}{\pi^2} \qquad (1)$$

where: l_p - effective micropile length (free length + $1/2$ root length)

EI - bending stiffness of micropile reinforcing tube

E_r - reaction of soil in the horizontal direction

m - number of half-waves

The critical force N_{cr} is taken as a minimum of function (1). It is achieved for the length of half-wave:

$$\frac{l_{\mathrm{p}}}{m} = \pi \sqrt[4]{\frac{EI}{E_{\mathrm{r}}}} \qquad (2)$$

The number of half-waves n is based on formula (2):

$$m = \frac{l_{\mathrm{p}}}{\pi} \sqrt[4]{\frac{E_{\mathrm{r}}}{EI}} \qquad (3)$$

Let's introduce the dimensionless parameter ω , calculated according to the formula:

$$\omega = \frac{l_{\mathrm{v}}}{l_{\mathrm{p}}} \qquad (4)$$

where: l_v - length of micropile above the terrain

and another dimensionless quantify, taken according to the formula:

$$\frac{N_{cr}\, l_p}{\pi^2 EI} \qquad (5)$$

Relations between formulas (3), (4) and (5) are described in the article [2] using the charts. The critical force N_{cr} is calculated from the dimensionless quantify (5), which is determined from the charts for known values of ω and m .

Literature:

[1] *Timoshenko, S. P.: Theory of Elastic Stability, New York, 1936*

[2] *Véase, Souche: Étude du fla,berment de pieux partiellernent immergés dans offrant latéralement une réaction élastique pure, Annales de l'ITBTP, No. 423, Sene Soils et Foundations, 187, mars - avril 1984, str. 38 - 60 (in French)*

Modulus of Horizontal Reaction of Subsoil

The soil surrounding the micropile can be represented using horizontal springs along micropile characterized by the Winkler constant k_h . For buckling of micropile into the soil in the direction of the x axis it is possible to write:

$$p_h = k_h.x = E_p.x$$

where: p_h - reaction of soil caused by the displacement of micropile in direction of x axis (soil in compression)

k_h - stiffness of Winkler spring (modulus of subsoil reaction E_p)

x - shift of a micropile in the direction of x axis

Providing we consider the reaction of soil to pressing of micropile per one meter run of the micropile we arrive at:

$$p_h = E_r.x$$

where: E_r - reaction of soil in the horizontal direction

p_h - reaction of soil caused by the shift of micropile in direction of x axis per one meter run of micropile

x shift of micropile in the direction of x axis

The above equations identify the relation between the modulus of subsoil reaction E_p $[kN/m^3]$ and the reaction of soil in the horizontal direction E_r $[kN/m^2]$ (assuming constant E_r in the soil):

$$E_r = k_h.D = E_p.D$$

where: D - diameter of micropile

k_h - stiffness of Winkler spring (modulus of subsoil reaction E_p)

The reaction of soil in the horizontal direction E_r can be post calculated based on the knowledge of the pressuremeter modulus E_m .

Calculation of the Modulus of Horizontal Reaction of Subsoil Er

The modulus of a horizontal reaction of subsoil can be determined when knowing the pressuremeter modulus E_m and coefficient α_p as:

$$E_r = E_m \cdot \frac{6}{\frac{4}{3}(2,65)^{\alpha_p} + \alpha_p}$$

where: E_m - pressuremeter (Menard) modulus $[MPa]$

α_p - rheological factor of soil (see the table below)

Reference values E_m and P_{lim}

Soils		E_m $[Mpa]$	P_{lim} $[MPa]$
cohesionless	loose	0 - 3.5	0 - 0.5
	medium dense	3.5 - 12	0.5 - 1.5
	dense	12 - 22.5	0.5 - 2.5
	very dense	> 22.5	> 2.5

cohesive	slush	0 - 2.5	0 - 0.2
	soft	2.5 - 5	0.2 - 0.4
	stiff	5 - 12	0.4 - 0.8
	solid	12 - 25	0.8 - 1.6
	hard	> 25	> 16

Values of rheological factor α_p for various soil conditions

Type of soil	Peat	Clay, silt		Sediment		Sand		Sand and gravel	
	α_p	E_m / P_{lim}	α_p	E_m / P_{lim}	α_p	E_m / P_{lim}	α_p	E_m / P_{lim}	α_p
preconsolidated	1	> 16	1.0	> 14	0.67	> 12	0.5	> 10	0.33
normally consolidated	1	9 - 16	0.67	8 - 14	0.5	7 - 12	0.33	6 - 10	0.25
underconsolidated	-	7 - 9	0.5	5 - 8	0.5	5 - 7	0.5	-	0.25

Literature:

Menard, L. F.: Proceedings of the 6th International Conference on Soil Mechanics and Foundation Engineering, Montreal, Vol. 2, 1965, pp. 295 - 299 (table 2.29 a 2.30).

Values of the Modulus of Subsoil Reaction Ep

Values of the modulus of subsoil reaction $E_p = k_h$ $[MN/m^3]$

Soil	E_p Min/Max $[MN/m^3]$	Average value $k_h = E\,p$ $[MN/m^3]$
soft clay	2 - 5	3.5
stiff clay	3 - 8	5.5
solid clay	6 - 16	11
sand naturally wet loose	6 - 13	9.5
sand naturally wet medium dense	20 - 40	30
sand naturally wet dense	45 - 90	67.5
sand aquiferous loose	4 - 8	6
sand aquiferous medium dense	10 - 20	15
sand aquiferous dense	30 - 60	45
sandy clay soft	3 - 6	4.5
sand clay stiff	5 - 9	7
sandy clay solid	8 - 17	12.5
clayey sand wet loose	4 - 9	6.5
clayey sand wet medium dense	12 - 32	22
clayey sand wet dense	24 - 44	34
clayey sand aquiferous loose	3.5 - 6.5	5
clayey sand aquiferous medium dense	7 - 11	9
clayey sand aquiferous dense	11.5 - 13.5	12.5

Bearing Capacity of the Micropile Root Section

The micropile bearing capacity can be determined computationally using one of the approaches available in the literature and standards. The program "**Micropile**" provides a set of methods representing the basic approaches to the solution of bearing capacity of the micropile root. The analysis is accrued out according to setting in the "Micropiles" tab employing one of the following procedures:

Lizzi theory - average limit friction on root skin is specified

Littlejohn theory - grouting pressure is specified

Zweck theory - method depends on the geostatic stress and soil parameters of the surrounding soil

Bowles theory method depends on the geostatic stress and soil parameters of the surrounding soil

Véas theory - the way the micropile is built and soil parameters of surrounding soil are specified

root in rock - rock parameters of surrounding soil are specified

Bustamante - method depends on the parameters of SPT or pressuremeter tests (PMT)

Lizzi Theory

The Lizzi method is currently the most popular method used. The root bearing capacity is provided by:

$$Q = \pi . d . l . \tau_m . J$$

where:
- d - root diameter
- l - root length
- τ_m - average limit skin friction
- J - coefficient reflecting the influence of borehole

Coefficient J reflects the influence of the borehole diameter - it ranges from 1.0 for hole up to 100 mm and 0.8 for the hole from 200 mm.

Average limit skin friction of the micropile root can be found in the literature. The program contains three tables with reference values of limit skin friction. The first one is created by the program authors using various literature sources, the second one contains values of τ_m according to DIN 4128, and the third one includes values published in by Klein and Mišov (Inženýrské stavby, 1984). The third table contains measured values of skin friction of anchor roots for various soils, root diameters, number if groutings, etc. - using this table yields rather realistic results.

Literature:

Lizzi, F. (1982). "The pali radice (root piles)". Symposium on soil and rockimprovement techniques including geotextiles, reinforced earth and modern pilingmethods, Bangkok, D-3.

Skin Friction of the Micropile Root

Reference values of limit skin friction (recommended by the authors)

Soil	**Skin friction** $[kPa]$
soft clay	40 - 60
stiff clay	65 - 85
solid clay	130 - 170
sand naturally wet, loose	110 - 150
sand naturally wet, medium dense	140 - 180
sand naturally wet, dense	170 - 230
aquiferous sand, loose	80 - 130
aquiferous sand, medium dense	120 - 160
aquiferous sand, dense	160 - 200
sandy clay, soft	50 - 70
sandy clay, stiff	75 - 95
sandy clay, solid	125 - 165
clayey sand, wet, loose	90 - 135
clayey sand, wet, medium dense	135 - 165
clayey sand, wet, dense	150 - 170
clayey sand, aquiferous, loose	80 - 105
clayey sand, aquiferous, medium dense	90 - 130
clayey sand, aquiferous, dense	115 - 155

Values of limit skin friction according to DIN 4128

Soil	**Average limit skin friction**	
	piles in compression $[kPa]$	**piles in tension** $[kPa]$
medium to coarse-grain sand	200	100
sand and gravel sand	150	80
cohesive soils	100	50

Recommended parameters of anchor roots (Mišove, Klein, Inženýrské stavby 5/1986)

Type of support of micropile in head	**Final grouting press.** $[MPa]$	**Number of groutings**	**Root diameter** $[mm]$	**Root length** $[m]$	**Skin friction** $[kPa]$
bedrock	-	0	120	5 - 3	1000 - 1600

semirock	0.5 - 3.0	0 - 1	120 - 220	7 - 3	300 - 1000
gravel, injectable soils	1,0	1 - 2	250 - 400	7 - 5	250 - 320
gravel, non-injectable soils	2.0 - 4.0	1 - 2	280 - 350	7 - 5	230
medium and fine-grain sand	1.5 - 4.0	2 - 3	220 - 350	12 - 7	150 - 180
cohesive stiff and solid soils	1.5 - 3.0	1 - 3	200 - 280	17 - 8	130 - 190
cohesive solid to rigid plastic soils	1.0 - 2.5	2 - 3	150 - 400	20 - 9	100 - 130
cohesive soft plastic soils	0.5 - 2.0	3 - 4	300 - 450	27 - 13.5	50 - 70

Littlejohn Theory

When using the Littlejohn method the root bearing capacity is provided by:

$$Q = \pi . d . l . p_i$$

where: d - root diameter

l - root length

p_i - magnitude of grouting pressure

It follows from experimental measurements of micropiles that their bearing capacity also depends on the course of grouting and on the grouting pressure (grouting course often governs the micropile bearing capacity). The bearing capacity considerably increases with repeated grouting. Grouting pressures range from 0.1 to 3 Mpa , in some cases they may reach up to 8 Mpa . The Littlejohn method gives the bearing capacity directly proportional to the grouting pressure.

Literature:

LITTLEJOHN, G. S. y BRUCE, D. A. (1975).: "Rock Anchors - State of the Art. Part 1. Design". En Ground Engineering, Vol. 8, N° 4.

Zweck Theory

The Zweck and Bowles methods were developed for the analysis of anchor roots - they depend mainly on the geostatic stress in the location of the micropile root. These methods arise from the same priciples - the pressure magnitude is however reduced using the coefficient of pressure at rest K_o .

$$Q = \pi . d . l . \frac{1 + K_o}{2} . \sigma_z . \tan \varphi$$

$$K_o = 1 - \sin \varphi$$

where: d - root diameter

l - root length

K_o - magnitude of pressure at rest

σ_z - average geostatic stress at the micropile root

φ - average value of friction angle at the micropile root

Bowles Theory

The Bowles solution allows incorporating the influence of the cohesion on the root bearing capacity - therefore it is more suitable for cohesive soils.

$$Q = \pi . d . l . \sigma_z . K_o . \tan \varphi + \pi . d . l . c$$

$$K_o = 1 - \sin \varphi$$

where: d - root diameter

l - root length

K_o - coefficient of pressure at rest

σ_z - average geostatic stress at the micropile root

φ - average magnitude of the angle of internal friction at the micropile root

Literature:

J.E. Bowles - Foundation Analysis and Design, McGraw Hill book Company.

Véas Theory

This solution takes into account the effect of geostatic stress at the micropile root and course of grouting.

Bearing capacity of the micropile root is provided by:

$$Q = R_{bk} + R_{sk}$$

where: R_{bk} - bearing capacity of the micropile root

R_{sk} - skin bearing capacity of the micropile root

Micropile skin bearing capacity:

$$R_{sk} = \sum_{i-1}^{n} A_{si} . q_{si}$$

where: n - number of layers passed by the micropile root

A_{si} - area of the wall of the micropile base in the i^{th} layer

q_{si} - skin friction in the i^{th} layer

Bearing capacity of the micropile root is provided by:

$$R_{bk} = 0{,}15 . R_{sk}$$

Skin friction q_s at a depth of z below the terrain surface:

$$q_s(z) = \frac{c}{F_c} + \sigma_h(z) \frac{tg\delta}{F_\varphi}$$

where: z - depth z below the terrain surface, where the magnitude of skin friction is determined

c - effective cohesion of soil at a depth of z

δ - friction angle along the interface of the micropile root and the soil at a depth of z :

$$\delta \in \left\langle \tfrac{2}{3} \varphi' ; \varphi' \right\rangle$$

φ' - effective angle of internal friction of soil at a depth of z

$\sigma_h(z)$ - horizontal component of geostatic stress at a depth of z :

- for grouting course of type IR and IRS (with monitoring of grouting pressure) and depth $z \geq 5$ m:

$$\sigma_h(z) = K_o . \sigma_v(z) + \frac{p_i}{3}$$

- other cases:

$$\sigma_h(z) = K_o . \sigma_v(z)$$

K_o - coefficient of earth pressure at rest

- for normally consolidated soils:

$$K_o = 1 - \sin \varphi'$$

- for overconsolidated soils:

$$K_o = (1 - \sin \varphi) . \sqrt{OCR}$$

$\sigma_v(z)$ - vertical component of geostatic stress at a depth of z

p_i - grouting pressure for grouting course of type IR and IRS and depth $z \geq 5\ m$, in other cases $p_i = 0$

F_c, F_φ - coefficients of type of application of micropile

Literature:

Véas, Souche : Étude du flambement de pieux partiellement immergés dans un milieu offrant latéralement une réaction élastique pure, Annales de l'ITBTP, No. 423, Sene Soils et Foundations, 187, mars - avril 1984, p. 38 - 60 (French).

Coefficients of Type of Application of Micropile

Coefficients of type of application of micropile

Type of application of micropile	F_c [-]	F_φ [-]
Newly constructed foundations	1.50	1.50
Existing foundations	1.20	1.20

Bearing Capacity of the Root in Rock

This solution is suitable for the micropile root reaching into rocks with index $RQD > 60$ or having the strength in simple compression $\sigma_c > 20\ MPa$ (ISRM < III). The root bearing capacity is given by:

$$Q = A_s . q_{sr} + A_b . q_{br}$$

where:
A_s - area of the wall of micropile root
q_{sr} - skin friction in rock
A_b - area of the micropile root
q_{br} - bearing capacity of the micropile root in rock

Literature:

Guía para el proyecto y la ejecución de micropilotes en obras de carretera, Ministerio de fomento, 2005 (Spanish).

Skin Friction and Bearing Capacity of the Micropile Root in Rock

Skin friction in rock q_{sr} and bearing capacity of the micropile root in rock q_{br}

Type of rock	q_{sr} [MPa]	q_{br} [MPa] 1)1
Sediments	0.15 - 0.40	$0.07\sigma_c$
Slates and phyllites	0.20 - 0.30	$0.07\sigma_c$
Sandstones	0.30 - 0.45	$0.07\sigma_c$
Lime stones and dolomites	0.40 - 0.50	$0.10\sigma_c$
granites, basalts	0.40 - 0.60	$0.10\sigma_c$

1) σ_c - strength in simple tension MPa

Bustamante (SPT, Pressuremeter PMT)

The analysis of the bearing capacity of the micropile root section is based on the results of standard penetration tests (SPT) or pressuremeter tests (PMT).

The magnitude of **skin friction of the micropile root** q_s [MPa] is available from the graphs according to Bustamante, which depends on the type of soil and the injection technology.

Shaft resistance of the micropile root R_s follows from:

$$R_s = \sum \pi d_r l_r q_s$$

where:
d_r - diameter of the micropile root
l_r - length of the micropile root
q_s - skin friction of the micropile root (value determined from the graph)

Base resistance of the micropile root R_b may not be considered in the analysis or is taken as:

$$R_b = 0{,}15 R_s$$

where: R_s - Shaft resistance of the micropile root

Base resistance of the micropile root R_b is assumed in the program in the form:

$$R_b = A_p k_p p_{LM}$$

where: A_p - cross-section area of the base of the micropile root

k_p - soil factor in the vicinity of the base of the micropile root

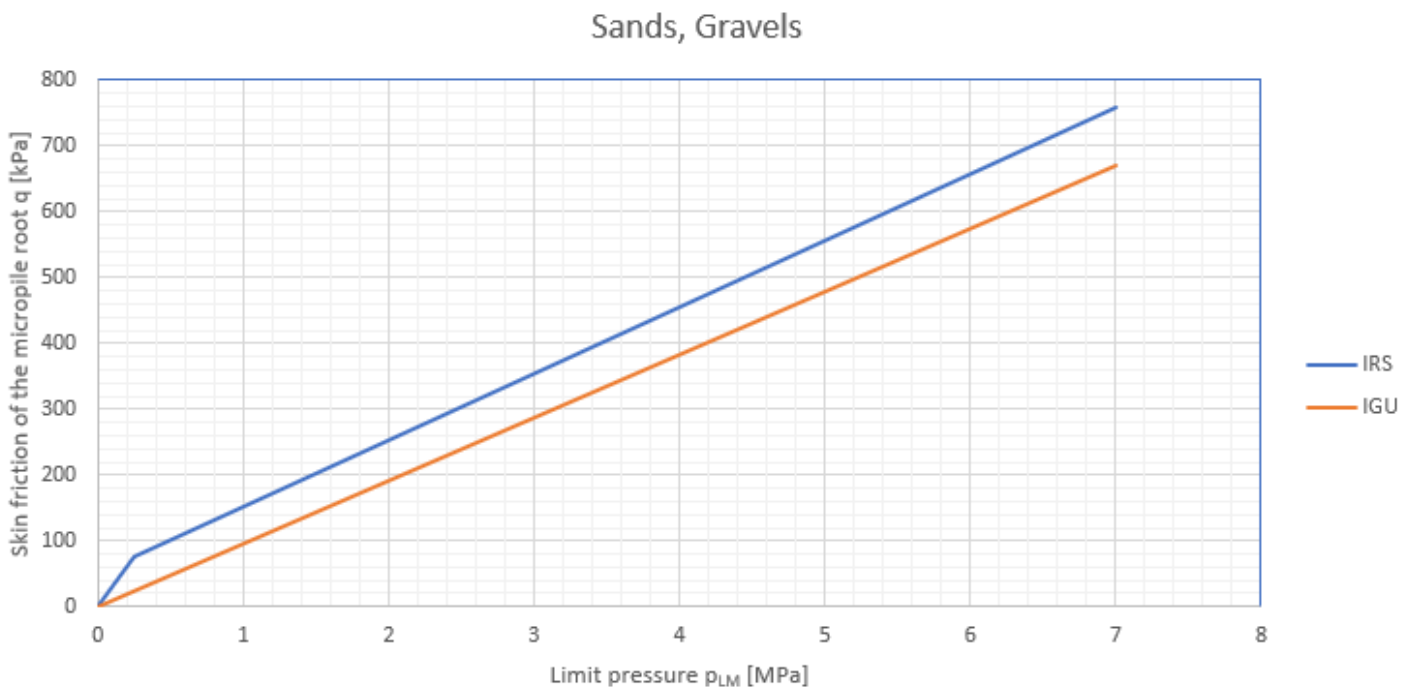
p_{LM} - limit pressure according to Menard

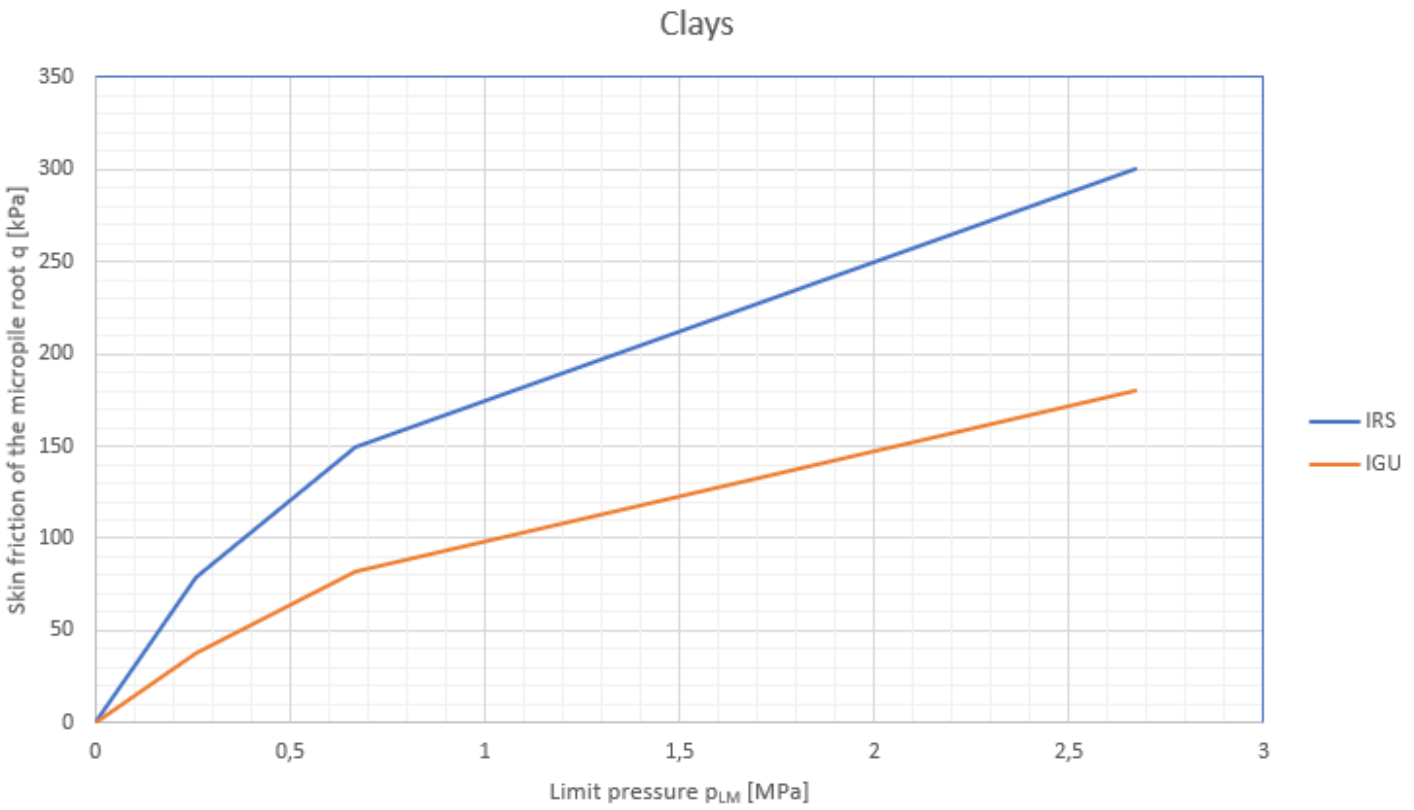
Skin Friction of the Micropile Root - Graphs

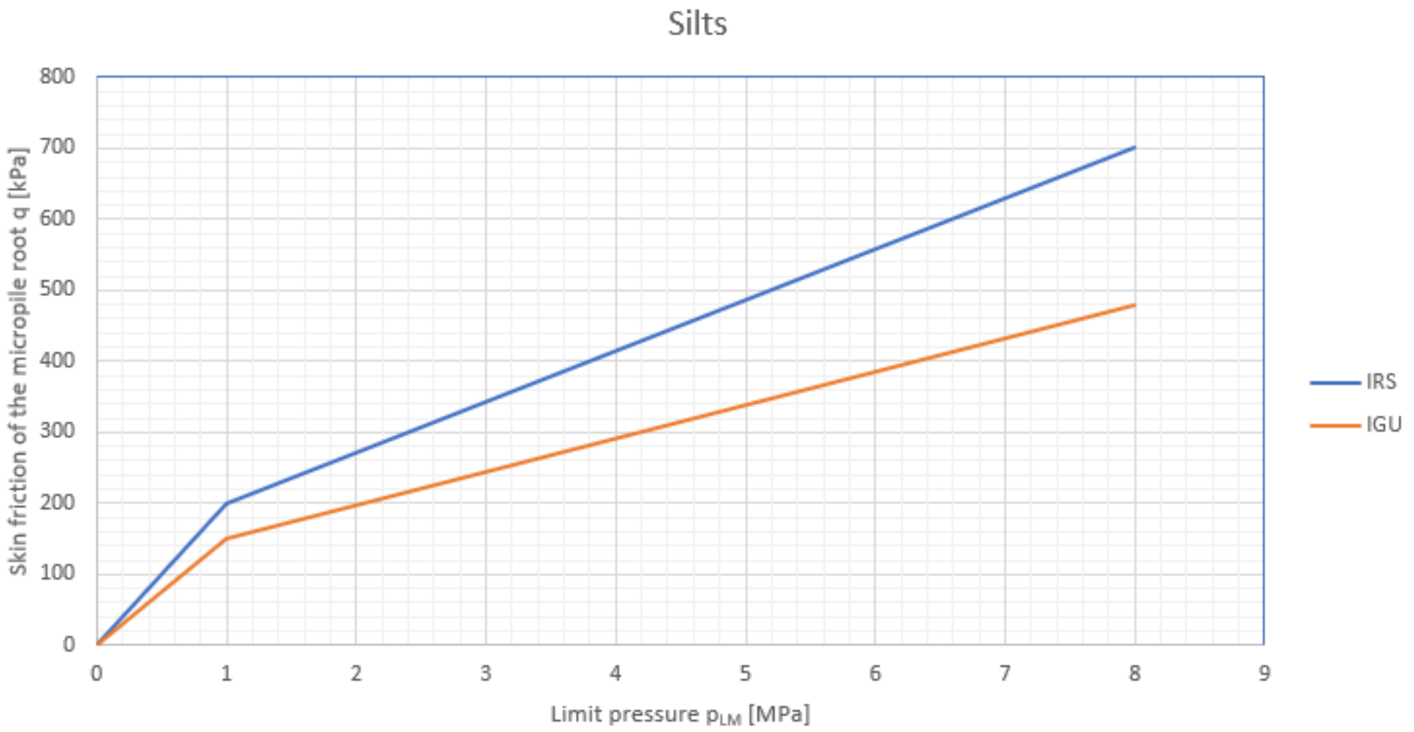
The analysis of shaft resistance of the micropile root R_s depends considerably on the **type of injection** of micropile root. The following options of injection are considered in the program:

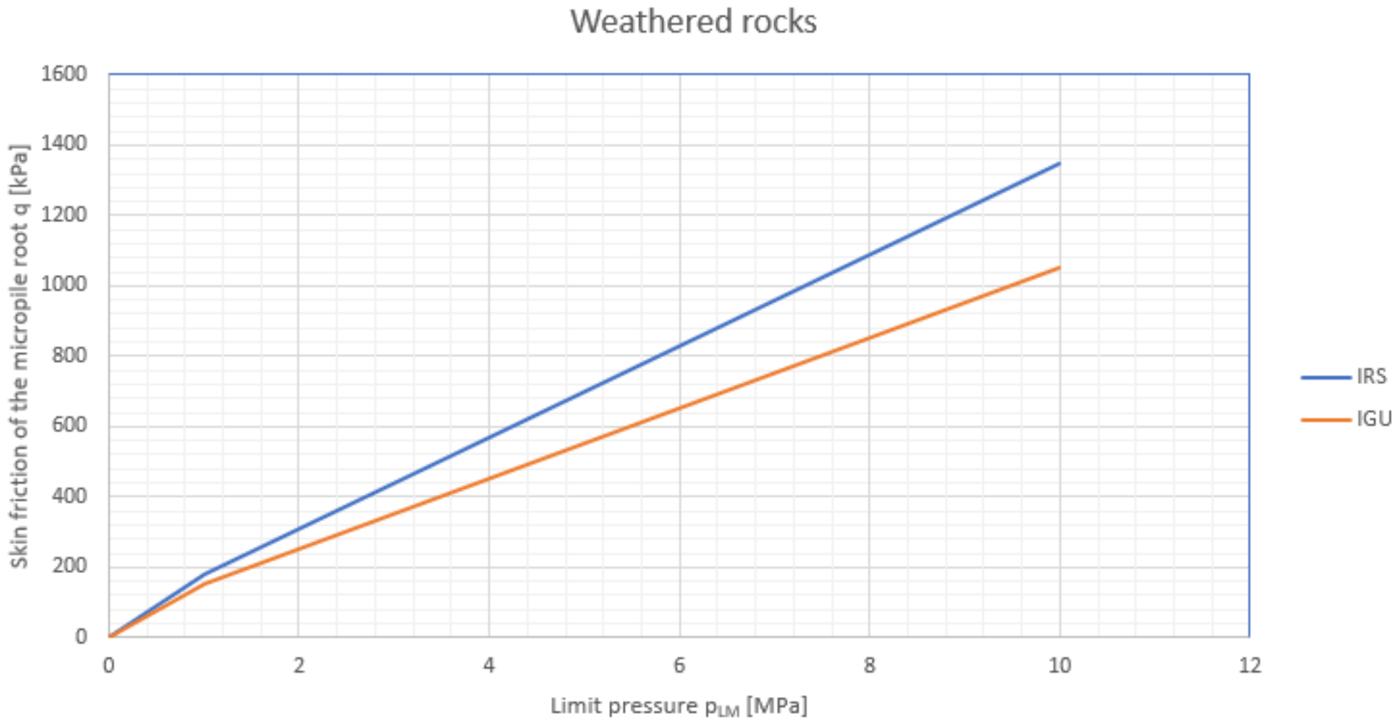
- **IRS**: repeated selective injection of the micropile root over sleeves performed locally (**Tube-á-Manchette**),
- **IGU**: unified global pressure injection (**Looped Tube Systems**).

The following graphs for the analysis of **skin friction of the micropile root** q_s $[MPa]$ are built in the program:




Clays
Skin friction of the micropile root q [kPa]
Limit pressure p_{LM} [MPa]
IRS
IGU
0
50
100
150
200
250
300
350
0,5
1
1,5
2
2,5
3


Silts
Skin friction of the micropile root q [kPa]
Limit pressure p_{LM} [MPa]
IRS
IGU
0
100
200
300
400
500
600
700
800
1
2
3
4
5
6
7
8
9



The displayed graphs consider on the horizontal axis the limit pressure p_{LM} determined from the pressuremeter tests (PMT). In case of SPT tests, the same graphs are used, but the limit pressure p_{LM} $[MPa]$ is then determined as the n **-multiple** of the number of blows N for the interval of penetration depth $d =$ 0.3 m , i.e. SPT $[N/$ 0.3 $m]$. For individual types of soils the values of limit pressure p_{LM} according to Menard are as follows:

- **sand, gravel, silt** and **weak rock:** $p_{LM} = SPT/$ 20,
- **clays**: $p_{LM} = SPT/$ 15.

For example, for the sandy soil and the value of the multiple of number of blows $SPT =$ 120 the limit pressure is given by $p_{LM} = SPT/$ 20 = 120 / 20 = 6.0 MPa .

Next, for example, for the clayey soil and the value of a multiple number of blows $SPT =$ 30 the limit pressure is provided by $p_{LM} = SPT/$ 15 = 30/ 15 = 2.0 MPa .

The vertical axis provides the value of skin friction of the micropile root q_s depending on the value of limit pressure p_{LM} and the applied type of injection (**IRS** or **IGU**, respectively).

Field Testing

The "Stratigraphy" program and some other GEO5 programs use as input parameters for the analysis of these types of **field tests**:

- CPT - **Cone penetration tests** - "Pile CPT" and "Spread footing CPT"
- DPT - **Dynamic probe test**
- SPT - **Standard penetration tests -** "Micropile", "Pile CPT" and "Spread footing CPT"
- PMT - **Pressuremeter tests** - "Sheeting Check", "Anti-Slide Pile", "Micropile" and "Spread footing CPT"
- DMT - **Dilatometric tests** - "Spread footing", "Sheeting Check" and "Anti-Slide Pile"

CPT (Cone Penetration Tests)

The **cone penetration test (CPT, CPTu)** is based on pushing a penetration cone using a system of penetration rods with constant velocity (20 - 25 mm/s) into the soil. During the penetration test the values of the **cone resistance**q_c and the **local skin friction**f_s , respectively are recorded. The cone resistance thus represents in general the resistance against penetration of a cone spike into the soil (subsoil). The diameter of the tip of the cone spike is typically in the range of 25 - 50 mm .

The **cone resistance** q_c represents the ratio of the measured force on the cone tip Q_c and the area of the normal projection of the cone tip A_c .

The **local skin friction** f_s represents the ratio of the measured force on the friction of sleeve F_s and the area of its skin A_s .

The result of the cone penetration test is its distribution plotted as a graph.

Cone penetration tests (CPT) serve as an input parameter for the analyses in the "Pile CPT" and "Spread Footing CPT" programs and for stratigraphic modeling in the "Stratigraphy" program.

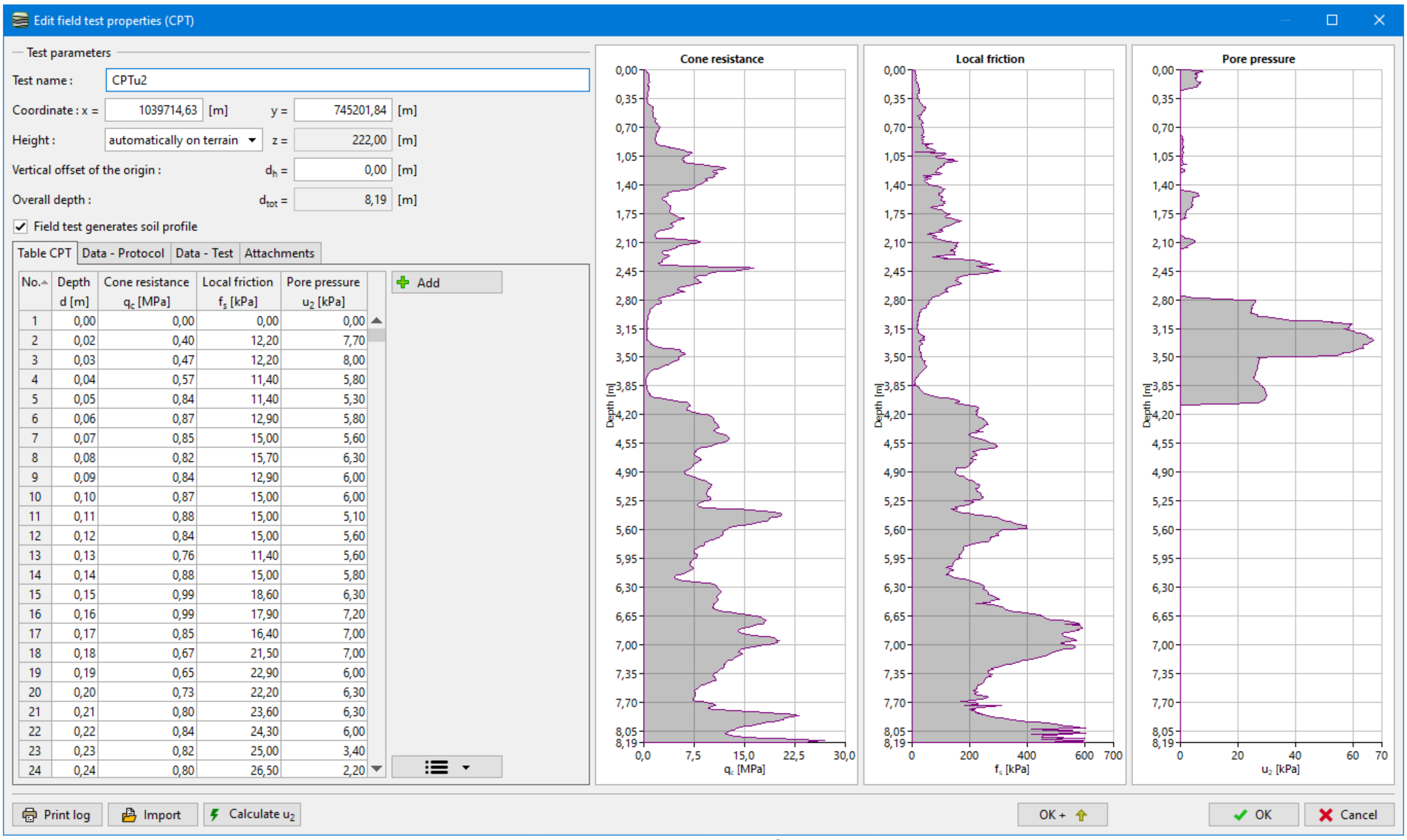
The CPT can be imported into the program using the "**Import**" button. The procedure of an import of table data (*.txt) is more described herein.

The name of the test and the vertical offset of the origin of the CPT is entered in the "**New field test**" dialog window. In the "**Stratigraphy**" program, the input of coordinates x , y , z is required.

The values of measured cone resistance q_c are entered in the table.

When calculating the pile bearing capacity according to Schmertmann's theory, local friction f_s must be entered.

When performing the classification of soils, the values of pore pressure u_2 must be entered. If the pore pressure was not measured (CPTu), it can be calculated from input GWT by pressing the button "**Calculate u2**".



Dialog window "New field test"

Literature:

EN ISO 22476-1: Geotechnical investigation and testing - Field testing. Part 1: Electrical cone and piezocone penetration test, 2013.

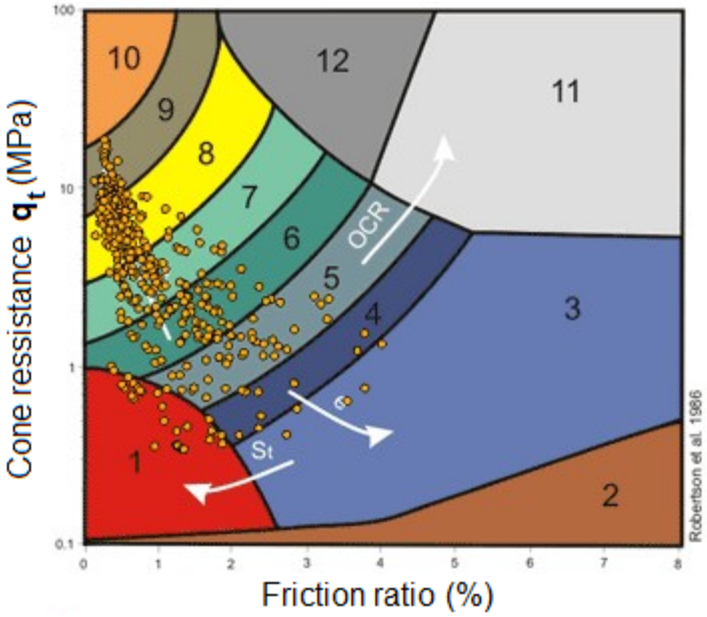
EN ISO 22476-12: Geotechnical investigation and testing - Field testing. Part 12: Mechanical cone penetration test (CPTM), 2009.

Roy E. Hunt: Geotechnical Engineering Investigation Handbook, Second Edition (CRC Press, 2005)

Classification of Soils According to Robertson

During the classification of soils according to Robertson (1986 or 2010) it is not necessary to input parameters of soils, the program performs this step automatically with their assignment to the geological profile. For this reason, the assessment of the performed CPT is very fast and especially clear.

Classification of soils according to Robertson (1986 or 2010) is based on the measured values of penetration resistance q_c , local skin friction f_s , pore pressure u_2 respectively. Based on the **corrected value of the cone resistance** $q_t = q_c + u_2 * (1 - a)$, or percentage ratio q_c / p_a and friction ratio $R_f = f_s / q_t$ program automatically performs the assignment of soil behavior type (SBT) according to the following graphs. p_a - atmospheric pressure = 100 kPa (= 1 tsf).

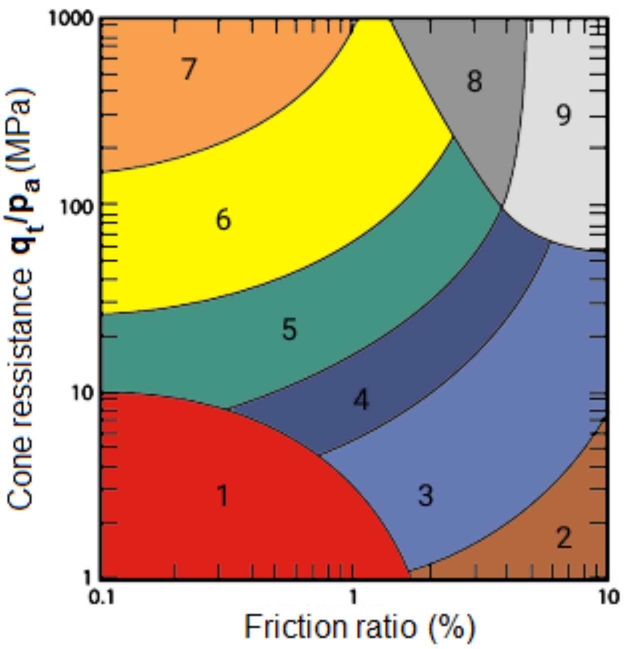


Non-normalized CPT Soil Behavior Type (SBT) chart according to Robertson, 1986 (source: Robertson et al., 1986)

Soil classification according to Robertson, 1986 (source: Robertson et al., 1986)

Zone	**Soil Behavior Type (SBT)**
1	Sensitive fine-grained
2	Organic material
3	Clay
4	Silty Clay to clay
5	Clayey silt to silty clay
6	Sandy silt to clayey silt
7	Silty sand to sandy silt
8	Sand to silty sand
9	Sand
10	Gravelly sand to sand
11	Very stiff fine-grained *
12	Sand to clayey sand *

* Overconsolidated or cemented soil



Non-normalized CPT Soil Behavior Type (SBT) chart according to Robertson, 2010 (source: [4], Figure 21, pp. 26)

Soil classification according to Robertson, 2010 (source: [6], Figure 21, pp. 26)

Zone	Soil Behavior Type (SBT)
1	Sensitive, fine-grained
2	Organic soils - clay
3	Clay - silty clay to clay
4	Silt mixtures - clayey silt to silty clay
5	Sand mixtures - silty sand to sandy silt
6	Sands - clean sand to silty sand
7	Gravelly sand to dense sand
8	Very stiff sand to clayey sand *
9	Very stiff fine-grained *

* Heavily overconsolidated or cemented

A newer classification of soils according to Robertson (2010) contains a smaller number of individual classes of soils than the original soil classification from 1986. However, the classification of soils according to Robertson (2010) is now more accurate and more used in the world.

If it's chosen option "**calculate**" for unit weight of soil in the frame "**Soil Classification**", then the **unit weight of soil** γ is determined by the following formula:

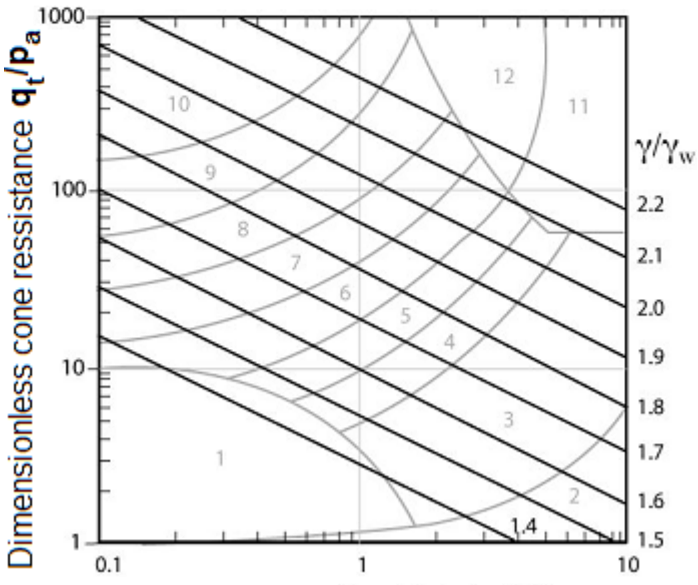
$$\frac{\gamma}{\gamma_w} = 0.27\left(\log R_f\right) + 0.36\left(\log \frac{q_t}{p_a}\right) + 1.236$$

where: γ_w - unit weight of water (≈10) [kN/m^3]

p_a - atmospheric pressure (≈100) [kPa]

$$R_f = \frac{f_s}{q_t} 100\ \%$$

where: R_f - friction ratio between skin friction and cone resistance



Dimensionless soil unit weight γ/γ_w based on CPTs (source: [4], Figure 28, pp. 36)

The input of the thickness of soil layers influences what is the minimum thickness of the layer of the i^{th} soil. In case of the **zero layers of soil** there are assigned all layers of soils based on the soil classification according to Robertson (1986 or 2010) into the geological profile.

When the input is a **non-zero minimum thickness of layer**, then the number of layers of soil is reduced in the geological profile. The layout and number of the soil layer, affect the vertical bearing capacity and settlement of pile or spread footing investigated by CPT.

Literature:

[1] EN ISO 22476-1: Geotechnical investigation and testing - Field testing. Part 1: Electrical cone and piezocone penetration test, 2013.

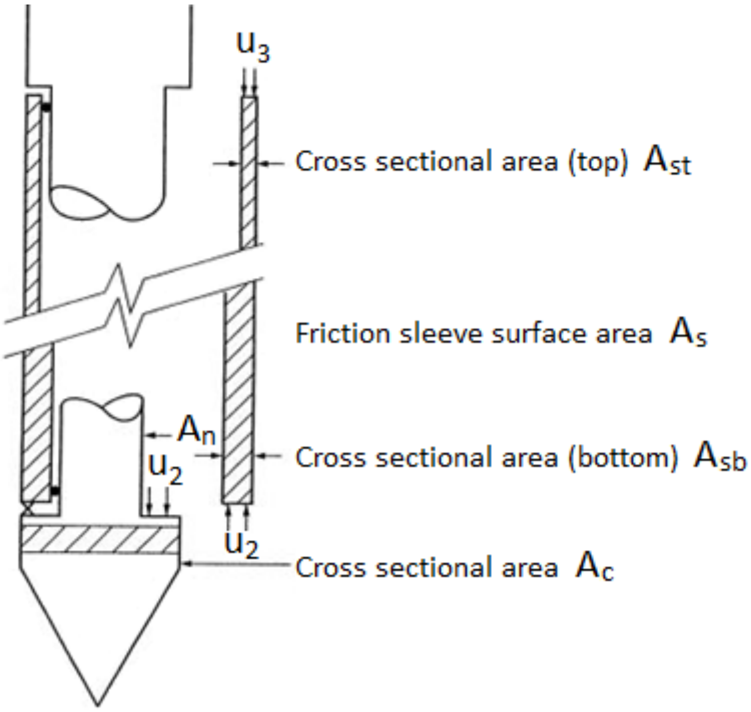
[2] EN ISO 22476-12: Geotechnical investigation and testing - Field testing. Part 12: Mechanical cone penetration test (CPTM), 2009.

[3] Robertson, P. K.: Interpretation of Cone Penetration Tests – a unified approach. Canadian Geotechnical Journal, 2009, No. 46, pp. 1337 – 1355.

[4] Robertson, P. K. and Cabal, K. L.: Guide to Cone Penetration Testing for Geotechnical Engineering. Gregg Drilling & Testing, Inc., USA, 6th edition, 2014, 133 p.

Coefficient of Penetrometer (Net Area Ratio)

This **coefficient** α [-] represents the **net area ratio** which is determined from calibration measurement at the laboratory (the effort to eliminate the adverse effects of friction sleeve and unequal cone tip). Typical values of this coefficient are in the range from 0.7 to 0.85.



Unequal end area effects on cone tip and friction sleeve (source: [4], Figure 20, pp. 22)

Literature:

[1] EN ISO 22476-1: Geotechnical investigation and testing - Field testing. Part 1: Electrical cone and piezocone penetration test, 2013.

[2] EN ISO 22476-12: Geotechnical investigation and testing - Field testing. Part 12: Mechanical cone penetration test (CPTM), 2009.

[3] Robertson, P. K.: Interpretation of Cone Penetration Tests – a unified approach. Canadian Geotechnical Journal, 2009, No. 46, pp. 1337 – 1355.

[4] Robertson, P. K. and Cabal, K. L.: Guide to Cone Penetration Testing for Geotechnical Engineering. Gregg Drilling & Testing, Inc., USA, 6th edition, 2014, 133 p.

Import CPT

The program "**Pile CPT**", "**Spread Footing CPT**" and "**Stratigraphy**" allows importing the CPT results in several formats.

The "**Import CPT**" dialog window contains a table with the list of imported tests. The combo lists serve to select the type of file and the desired system of units.

***.txt, *.xlsx, *.csv, *.ods**	-	a general text or table format
***.spe**	-	a data format used in Czech and Slovak Republic, originally from the GeProDo software
***.cpt**	-	a text file standard particularly for Netherlands (used e.g., in programs Geodelft M-Serie), which serves to input elevations of individual points and values of penetration resistance (may contain more CPTs)
***.cpt**	-	a text file Geotech AB CPT
***.cpt**	-	a text file Gouda Geo CPT
***.cpt**	-	a text file Hogentogler CPT

***.gef** - GEF (Geotechnical Exchange Format) is a general language structure for storing and transferring of geotechnical information.
Detailed information available online:
https://publicwiki.deltares.nl/display/STREAM/GEF-CPT

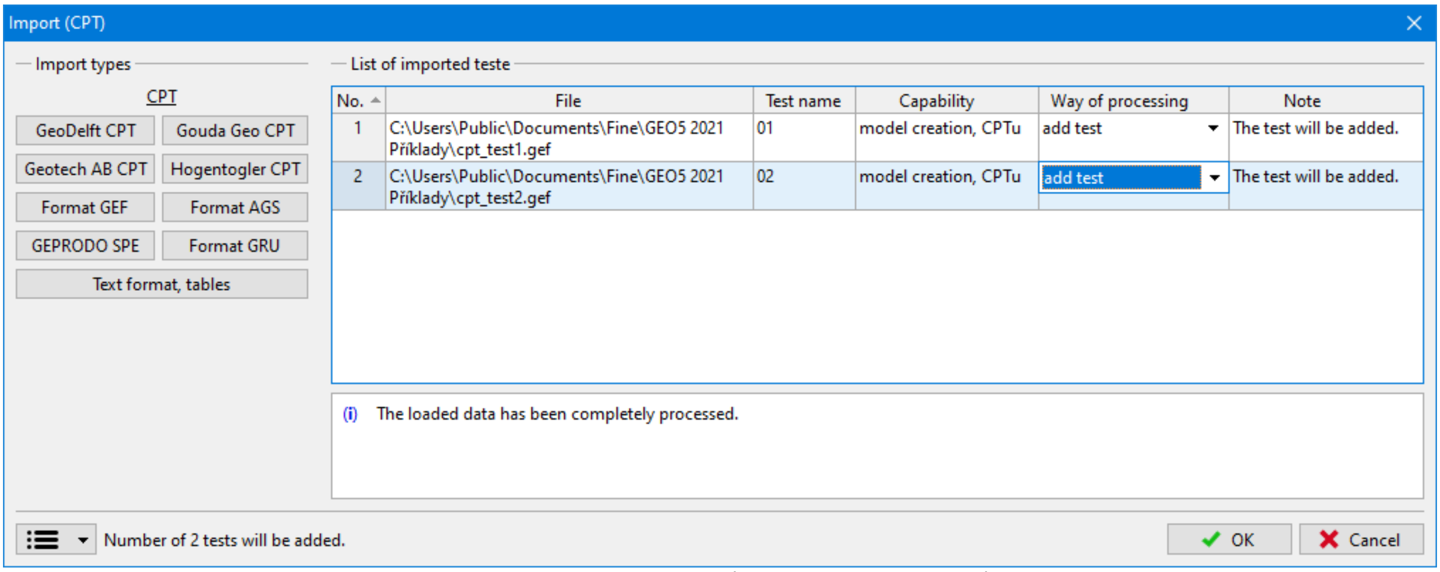
***.ags** a format used for transferring of geotechnical information in Great Britain.
Detailed information available online:
http://www.agsdataformat.com/datatransferv4/intro.php

***.GRU** - a data format used in Poland

A TXT format allows for selecting a particular **system of units** to store data of the test. When importing the program automatically converts the adopted system of units to the one used in the program.

For a correct calculation the test must be introduced into the soil body - therefore the window requires us to input an **elevation of the original ground**. The particular test is then inserted into the soil body according to its specified elevation. If no elevation is given the origin of the test is automatically placed on the **original ground**.

Providing you use a certain standard of a CPT text file not supported by the program, feel free to contact us at **hotline@fine.cz** - it will be introduced into the forthcoming version.



Dialog window "Import" (cone penetration test)

DPT (Dynamic Penetration Test)

The result of the **dynamic probe test (DPT)** is a number of blows N, needed to penetrate a cone device to the soil or rock by a so-called **interval of penetration depth**.

The **Interval of penetration depth** differs according to the type of dynamic probe test:

- **DPL** – dynamic probing light
- **DPM** – dynamic probing medium
- **DPH** – dynamic probing heavy
- **DPSH - A** – dynamic probing superheavy
- **DPSH - B** – dynamic probing superheavy

Theinterval of penetration depth is 100 mm for **DPL**, **DPM**, and **DPH** and100or200 mm for **DPSH- A** or **DPSH -B** tests.

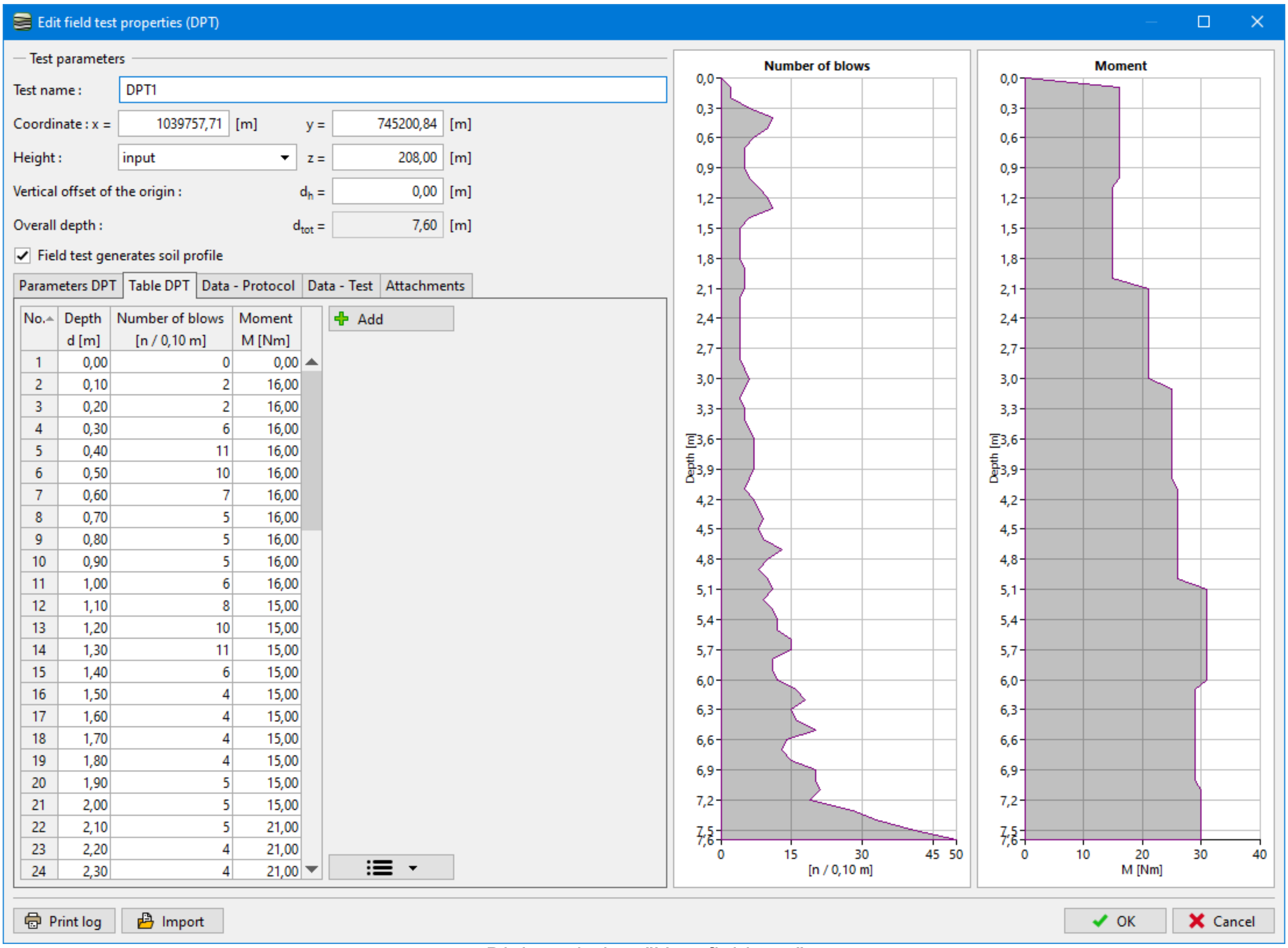
The necessary torque **moment** to rotation by **1,5 turns** or to reaching its maximum value should also be recorded and written down at least after every 1,0 m of the test. This value serves to eliminate the shear friction.

The result of DPT is its process plotted as a graph. The evaluation of DPTs are used as input parameters for the stratigraphic modeling in the "Stratigraphy" program.

The results of DPTs can be imported into the program as .txt data.

The name of the test and the vertical offset of the origin of the DPT is entered in the "**New field test**" dialog window. In the "**Stratigraphy**" program, the input of coordinations x, y, z is required.

The **number of blows** and **moment** is entered in the table.



Dialog window "New field test"

Literature:

EN ISO 22476-3: Geotechnical investigation and testing - Field testing. Part 2: Dynamic probing test, 2005.

Roy E. Hunt: Geotechnical Engineering Investigation Handbook, Second Edition (CRC Press, 2005)

SPT (Standard Penetration Test)

The result of the **Standard penetration test (SPT)** is a number of blows N (penetration resistance), needed to penetrate a sampling device to the soil or rock by a so-called **Interval of penetration depth** 0,3 m (1 ft).

The number of blows N measured during the SPT test is correlated for the reason of various testing devices and for the influence of the weight of overburden in sand. **Corrected** (correlated) **value N_{60} is used in calculations**.

The correlated number of blows N_{60} is calculated according to this equation:

$$N_{60} = \frac{E_r}{60} C_N \lambda N$$

where:

- E_r - Energetic ratio of the testing device
- N - number of blows measured during SPT test
- C_N - correlation for vertical stress
- λ - user's correlation

The energetic ratio of testing device E_r represents the ratio of real energy E_{meas} and the calculated energy E_{theor} of the ram. It is not necessary to use a correlation for a standard SPT device (Mohr), because the efficiency of the machine is 60 $\%$ and correlation is performed to this value (the values measured during SPT test are used).

Correlation C_N for vertical stress $\sigma'V$ - represents the influence of weight of overburden in sands. The values of the correction factor C_N greater than 1.5 should not be used (according to EN ISO 22476-3 recommendations).

Table of built-in types of correlations

Type	**Type of consolidation**	**Relative compactness I_p [%]**	**Correlation factor C_N**
Type 1 - EN ISO 22476-3 (Tab. A2)	Normally consolidated	40 - 60	$C_N = \frac{200}{100 + \sigma'_V}$
Type 2 - EN ISO 22476-3 (Tab. A2)		60 - 80	$C_N = \frac{300}{200 + \sigma'_V}$
Type 3 - EN ISO 22476-3 (Tab. A2)	Over-consolidated	-	$C_N = \frac{170}{70 + \sigma'_V}$
Type 4 - EN ISO 22476-3	Normally consolidated sands	-	$C_N = \sqrt{\frac{98}{\sigma'_V}}$
Type 5 - FHWA (1998), Peck (1974)	-	-	$C_N = \left(\frac{100}{\sigma'_V}\right)^{0,4}$

where: σ'_V - Effective vertical stress

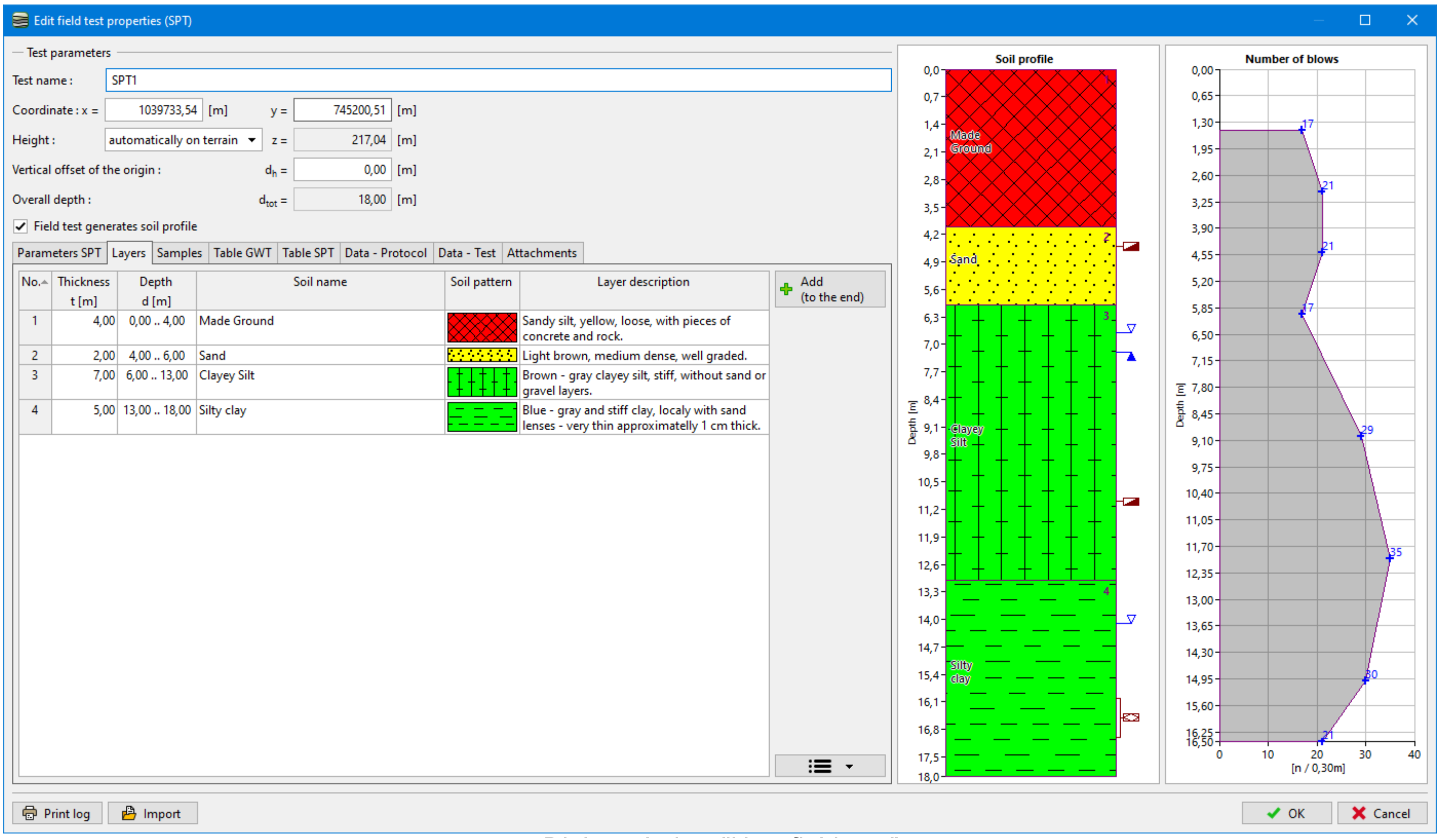
User's correlation λ [-] - represents the loss of energy due to the length of the system of rods, the impact of borehole diameter, or the influence of the sampling device.

The result of **SPT** is its process plotted as a graph. The evaluation of standard penetration tests are used as input parameters for the analyses in the "Micropile" and "Spread Footing CPT" programs and for stratigraphic modeling in the "Stratigraphy" program.

Standard penetration tests can be imported into the program as .txt data.

The name of the test and the vertical offset of the origin of the SPT is entered in the "**New field test**" dialog window. In the "**Stratigraphy**" program, the input of coordinations x, y, z is required.

The **number of blows** is entered in the table.



Dialog window "New field test"

Literature:

EN ISO 22476-3: Geotechnical investigation and testing - Field testing. Part 3: Standard penetration test, 2005.

Roy E. Hunt: Geotechnical Engineering Investigation Handbook, Second Edition (CRC Press, 2005)

PMT (Pressuremeter Test)

The **Pressuremeter test (PMT)** consists of a pressuremeter probe placed in the tested soil and gradually filled with water. The subsequent swelling of soil or rock around the hole is determined as a dependence of the measured volume of water on the pressure increment that is gradually increased in a priori defined time intervals.

The pressuremeter test provides the following parameters as a function of depth:

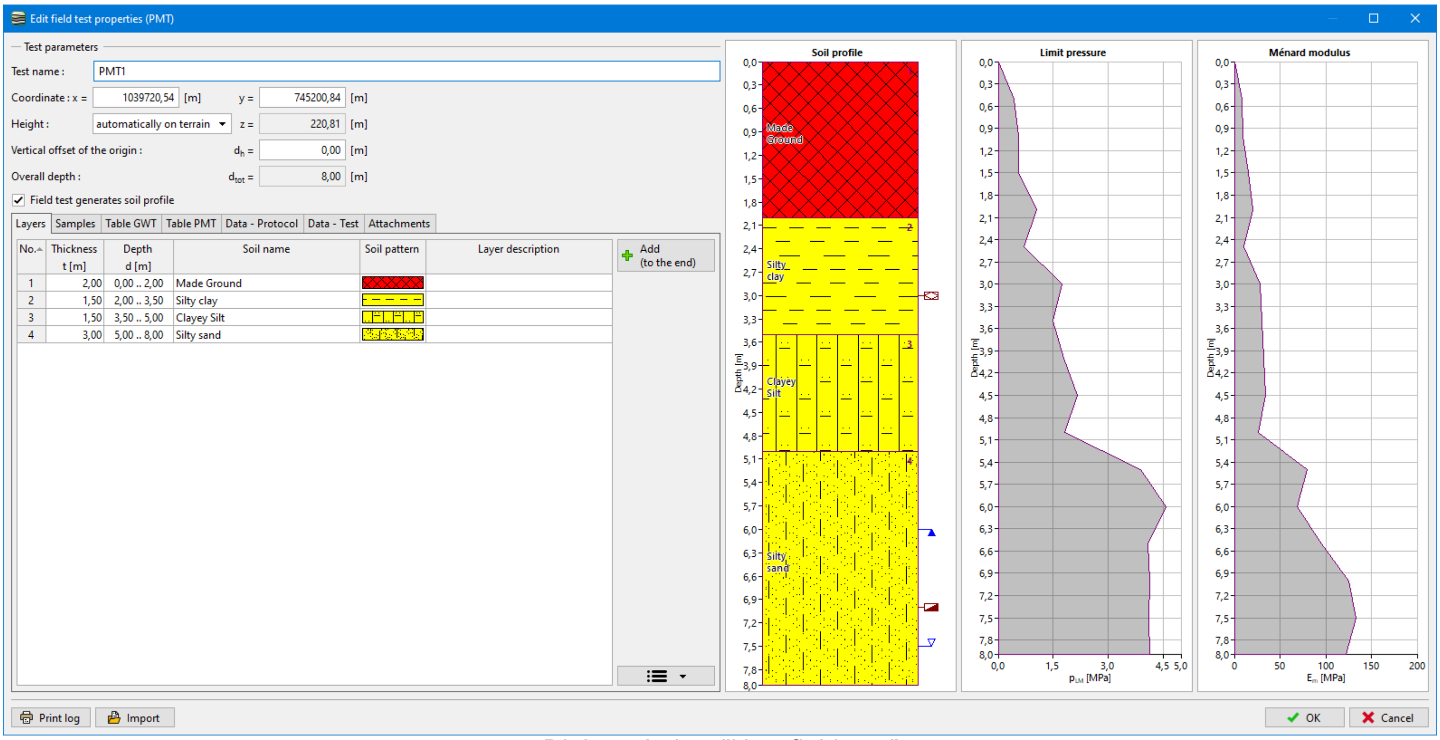
- **pressuremeter (Menard) modulus** E_m - is obtained from the pressuremeter test and depends on the type of **sheath of the probe** (rubber sleeve, perforated casing).
- **limit pressure** p_{LM} - represents an increment of water pressure in the testing probe depending on the volume change of soil or rock, respectively.

The result of the pressuremeter test is its process plotted as a graph. The evaluation of pressuremeter tests (PMT) is used as input parameter for the analyses in the "Sheeting Check", "Anti-Slide Pile", "Micropile" and "Spread Footing CPT" programs and for stratigraphic modeling in the "Stratigraphy" program.

The results of pressuremeter tests (PMT) can be imported into the program as .txt data.

The name of the test and the depth and the vertical offset of the origin of the PMT is entered in the "**New field test**" dialog window. In the "Stratigraphy" program, the input of coordinations x , y , z is required.

The limit pressure p_{LM} and the Menard modulus E_m are entered in the table.



Dialog window "New field test"

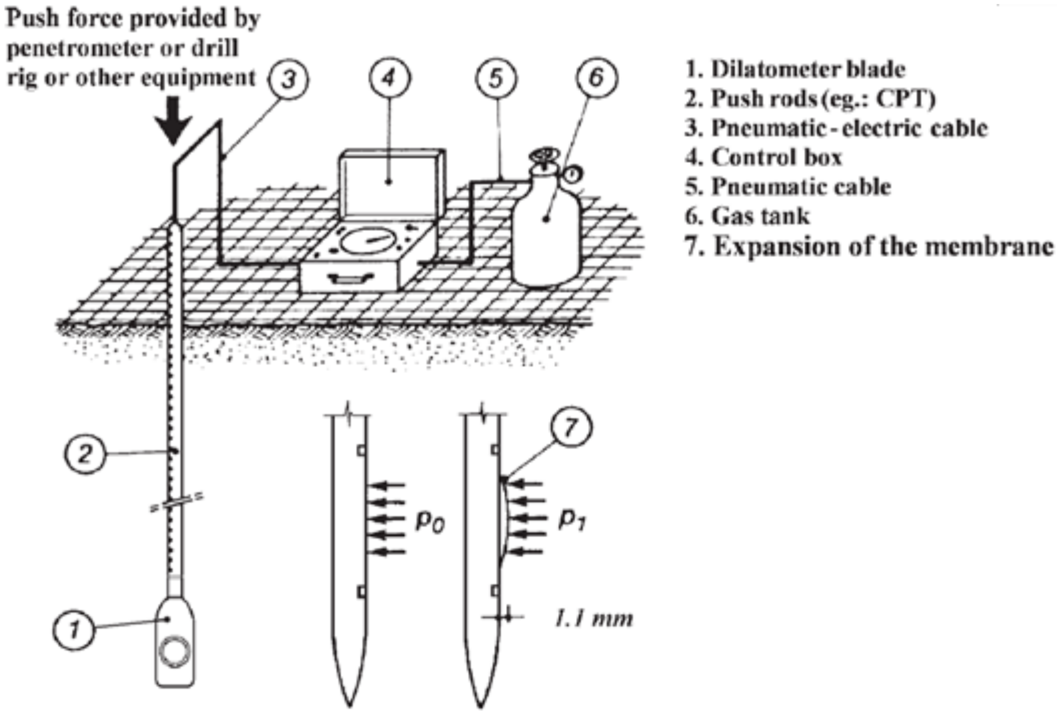
Literature:

EN ISO 22476-4: Geotechnical investigation and testing - Field testing. Part 4: Menard pressuremeter test, 2005.

Roy E. Hunt: Geotechnical Engineering Investigation Handbook, Second Edition (CRC Press, 2005)

DMT (Dilatometric Test)

The **dilatometric test (DMT)** is performed by using a dilatometric, which operates on the principle of verification of values by using the displacements of the inductive sensors (with a sensitivity of up to 0.001 mm). The advantage of these tests is a more accurate description of the displacement and deformation of foundation soil.



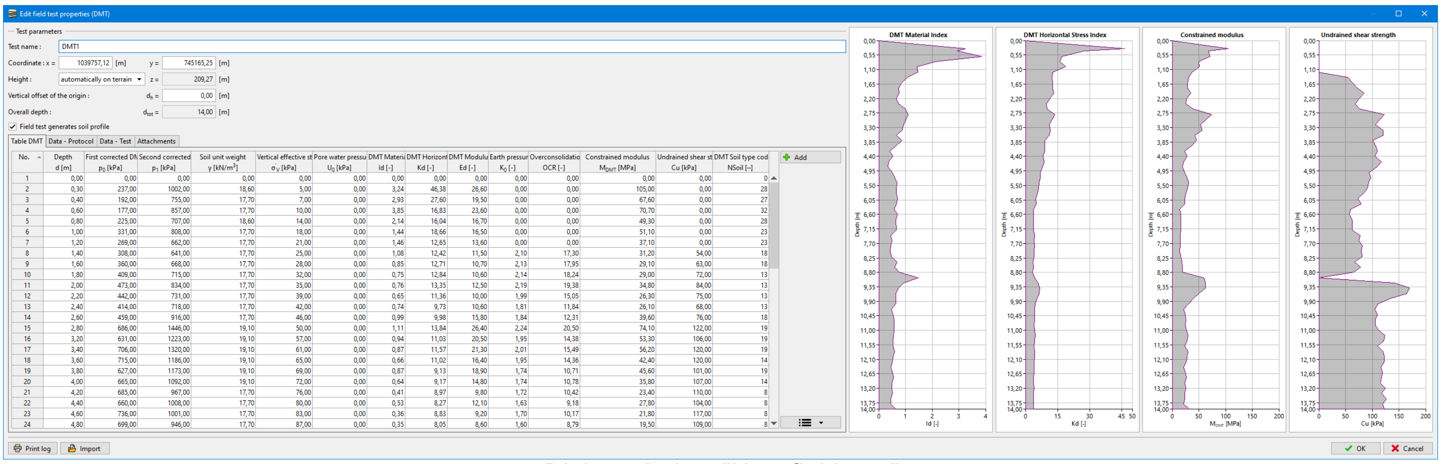
General Layout of Dilatometer Test (source: [1], Figure 2, pp. 10)

The result of the dilatometric test is its process plotted as a graph. The evaluation of dilatometric tests (DMT) is used as input parameter for the analyses in the "Spread Footing", "Sheeting Check" and "Anti-Slide Pile" programs and for stratigraphic modeling in the "Stratigraphy" program.

The results of the dilatometric test (DMT) are imported into the program by inserting the file in format **UNI** (***.uni**). It is a **standardized and universal format** for import of the measured data obtained from dilatometric tests, which is used in the world.

The name of the test and the vertical offset of the origin of the DMT is entered in the "**New field test**" dialog window. In the "**Stratigraphy**" program, the input of coordinations x , y , z is required.

The constrained soil modulus M_{DMT} is entered in the table.



Dialog window "New field test"

Literature:

Marchetti, S., Monaco, P., Totani, G. & Calabrese, M.: The Flat Dilatometer Test (DMT) in soil investigations. A Report by the ISSMGE Committee TC16, University of L'Aquila, Italy, 2001, 48 p.

Roy E. Hunt: Geotechnical Engineering Investigation Handbook, Second Edition (CRC Press, 2005)

EN ISO 22476-11: Geotechnical investigation and testing - Field testing. Part 11: Flat dilatometer test

Settlement Analysis

One of the following methods is available to compute settlement:

- Using the oedometric modulus
- Using the compression constant

- Using the compression index
- According to NEN (Buismann, Ladde)
- Using the Soft soil model
- According to Janbu theory
- Using the DMT (constrained modulus)

The program offers two options to constrain the depth of the influence zone:

- Exploiting the theory of structural strength
- Using the percentage of the magnitude of geostatic stress

The theory of elasticity (Boussinesq theory) is employed to determine stress in a soil state in all methods available for the settlement analysis.

General theories of settlement serve as bases in all the above methods.

When computing settlement below the footing bottom the programs first calculate the stress in the footing bottom and then determine the overall settlement and rotation of foundation.

The general approach in all theories draws on subdividing the subsoil into layers of different thickness based on the depth below the footing bottom or ground surface. Vertical deformation of each layer is then computed - the overall settlement is then defined as a sum of partial settlements of individual layers within the influence zone (deformations below the influence zone are either zero or neglected):

$$s = \sum \Delta s_i$$

where: s - settlement

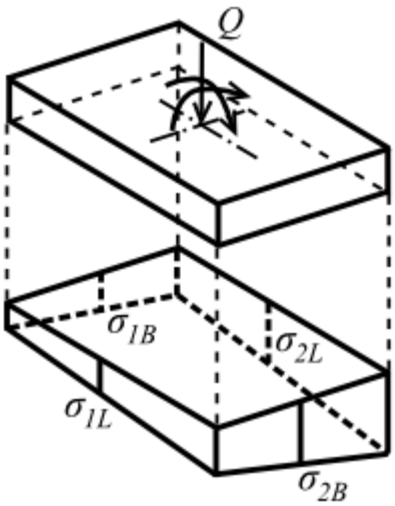
s_i - settlement of the i^{th} layer

Stress in the Footing Bottom

The stress in the footing bottom can be assumed as:

- **rectangular** (uniform in the footing bottom)
- **general** (trapezoidal) with different edge values

The general distribution of stress follows from the figure:



Stress in the footing bottom

where:

$$\sigma_{1B,z} = \frac{Q}{l.b} \pm \frac{Q.e_b}{W_b} \quad e_b = \frac{M_x + H_y.t + N.p_x}{Q} \quad W_b = \frac{1}{6}.l.b^2$$

$$\sigma_{1L,z} = \frac{Q}{l.b} \pm \frac{Q.e_t}{W_t} \quad e_t = \frac{-M_y + H_x.t + N.p_y}{Q} \quad W_t = \frac{1}{6}.l.b^2$$

where: Q - vertical load of footing

l,b - footing width and length

e_b - load eccentricity

M - moment acting on the footing

H - horizontal force

N - normal force at the eccentric footing

p - column axis offset from the footing center

If in some points the stress becomes negative, the program continues with adjusted dimensions $b{*}l$ while excluding tension from the analysis. Before computing the stress distribution due to surcharge the stress in the footing bottom is reduced by the geostatic stress in the following way:

$$\sigma_{ol} = max(\sigma_{ol} - \sigma_{or,sp};0)$$

There are three options in the program to specify the geostatic stress in the footing bottom:

- **From the original ground** It is therefore considered, whether the footing bottom in the open pit measured from the original ground is free of stress for the time less than needed for soil bulking and subsequent loss of stress in the subsoil.
- **From the finished grade** The same assumptions as above apply.
- **Not considered at all**.

Overall Settlement and Rotation of Foundation

The foundation settlement is substantially influenced by the overall stiffness of the system represented by foundation structure and foundation soil given by:

$$k = \frac{E_{basic}.t^3}{E_{def,av}.l^3}$$

where: E_{basic} - modulus of elasticity of footing

t - foundation thickness

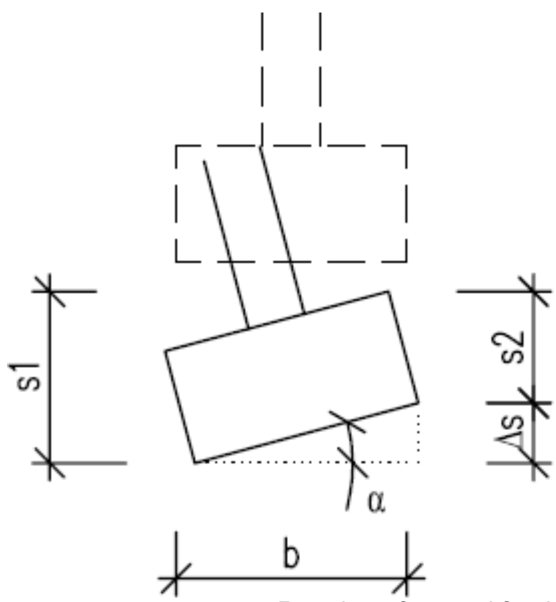
$E_{def,av}$ - weighted average of the deformation modulus up to a depth of the influence zone

l - footing dimension in the direction of searched stiffness

For $k >$ 1 the foundation is assumed to be rigid and as a representative point for the determination of its settlement is assumed the **characteristic point** (distant by 0.37 times the foundation dimension from its axis).

For $k <$ 1 the foundation structure is assumed to be compliant and as a representative point for the determination of foundation, the settlement is assumed the **foundation center point**.

The **foundation rotation** is determined as a difference in settlements of individual edges centers.



$$\Delta s = s_1 - s_2$$

$$rotation = \frac{\Delta s}{b}(\tan{*}1000)$$

$$\alpha = \arctan\frac{\Delta s}{b}[deg]$$

Rotation of spread footing - principle calculation

Influence of Foundation Depth and Incompressible Subsoil

When computing settlement it is possible to account for the **influence of the foundation depth** by introducing the reduction coefficient κ_1 :

for strip footing:

$$\kappa_1 = 1 + 0{,}61.arctg\frac{d}{z}$$

for spread footing:

$$\kappa_1 = 1 + 0{,}35.arctg\left(1{,}55.\frac{d}{z}\right)$$

where: d - depth of the footing bottom

z - depth under footing bottom

Influence of incompressible layer is introduced into the analysis by the reduction coefficient κ_2 :

$$\kappa_2 = 1 - e^{\left(\frac{z_{ic}}{z}.ln0{,}25 + ln0{,}8\right)}$$

where: z_{ic} - depth of rigid base under footing bottom

z - depth under footing bottom

Incorporating the above coefficients allows the **transformation** of the vertical component of stress σ_z such that the actual depth is replaced by a **substitute value** z_r given by:

$$z_r = \kappa_1.\kappa_2.z$$

where: κ_1 - coefficient of footing bottom depth

κ_2 - coefficient of the rigid base

z - depth under footing bottom

Influence of Sand-Gravel Cushion

If the sand-gravel cushion is specified below the spread footing, the material parameters X in individual layers are computed in the following way:

For layer $h_{a,i}$:

$$X_i < X_c$$

where: X_i - material parameters at i^{th} layer

X_c - material parameters of sand-gravel cushion

For layer $h_{b,i}$:

$$X_i = \frac{(A_i - A_c).X_c + A_i.X_c}{A_i}$$

$$A_i = b_i.l_i$$

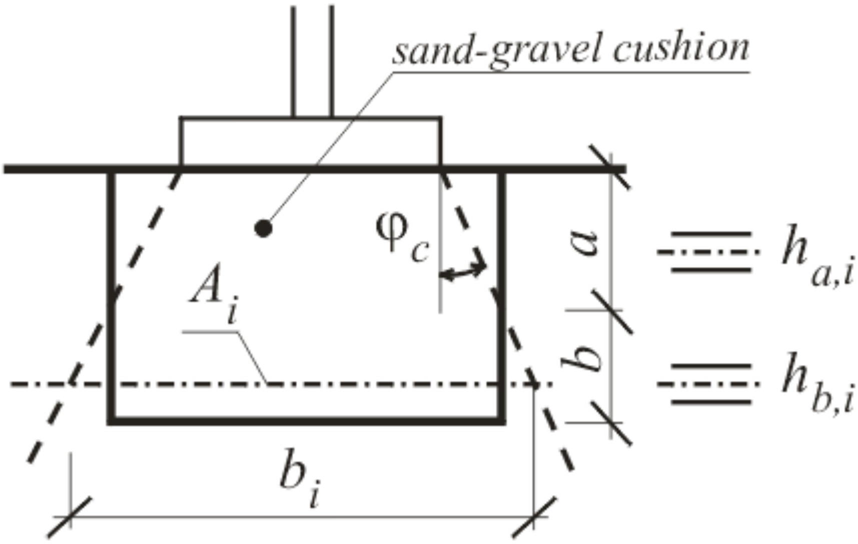
where: A_c - area of sand-gravel cushion

X_c - material parameters of sand-gravel cushion

$X_{b,i}$ - material parameters of b,i layer

b_i - cushion widths in the i^{th} layer

l_i - cushion length in the i^{th} layer



Analysis X_i in the sand-gravel cushion

Analysis Using the Oedometric Modulus

The equation to compute compression of an i^{th} soil layer below foundation having a thickness h arises from the definition of oedometric modulus E_{oed} :

$$s_i = \sum \frac{\sigma_{z,i} h_i}{E_{oed,i}}$$

where:

	$\sigma_{z,i}$	-	vertical component of incremental stress in the middle of the i^{th} layer
	h_i	-	thickness of the i^{th} layer
	$E_{oed,i}$	-	oedometric modulus of the i^{th} layer

The oedometric modulus E_{oed} can be specified for each soil either as constant or with the help of an oedometric curve (σ_{ef}/ε relation). When using the oedometric curve the program assumes for each layer the value of E_{oed} corresponding to a given range of original and final stress. If the value of oedometric modulus E_{oed} is not available, it is possible to input the deformation modulus E_{def} and the program carries out the respective transformation.

$$E_{oed} = \frac{E_{def}}{\beta}$$

where:

$$\beta = 1 - \frac{2.\nu^2}{1-\nu}$$

where:

	ν	-	Poisson's ratio
	E_{def}	-	deformation modulus

Analysis Using the Compression Constant

The equation to compute compression of an i^{th} soil layer below foundation having a thickness h arises from the definition of compression constant C :

$$s = \frac{h_i}{C_i} . ln \frac{\sigma_{or,i} + \sigma_{z,i}}{\sigma_{or,i}}$$

where:

	$\sigma_{or,i}$	-	vertical component of original geostatic stress in the middle of i^{th} layer
	$\sigma_{z,i}$	-	vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression
	h_i	-	thickness of the i^{th} layer
	C_i	-	compression constant in the i^{th} layer

The program allows for inputting either the compression constant C_i or the compression constant C_{10} (the program itself carries out the transformation).

Analysis Using the Compression Index

Equation for settlement when employing the compression index C_c of the i^{th} layer arises from the formula:

$$s_i = C_{c,i} \frac{h_i}{1+e_0} . log \frac{\sigma_{or,i} + \sigma_{z,i}}{\sigma_{or,i}}$$

where:

- $\sigma_{or,i}$ - vertical component of geostatic stress in the middle of i^{th} layer
- $\sigma_{z,i}$ - vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression
- e_o - initial void ratio
- h_i - thickness of the i^{th} layer
- $C_{c,i}$ - compression index in the i^{th} layer

Analysis According to NEN (Buismann, Ladd)

This method computes both the primary and secondary settlement. When computing the method accounts for overconsolidated soils and differentiates between two possible cases:

- Sum of the current vertical effective stress in the soil and stress due to external surcharge is less than the pre-consolidation pressure so that only additional surcharge is considered.
- Sum of the current vertical effective stress in the soil and stress due to external surcharge is greater than the pre-consolidation pressure so that the primary consolidation is set on again. The primary settlement is then larger when compared to the first case.

Primary settlement

Primary settlement of the i^{th} layer of overconsolidated soil (OCR> 1) is provided by:

for: $\sigma_{or} + \sigma_z \leq \sigma p$ (sum of the current vertical stress and its increment is less than the preconsolidation pressure):

$$s_i = C_{r,i} \frac{h_i}{1+e_0} . log \frac{\sigma_{or,i} + \sigma_{z,i}}{\sigma_{or,i}}$$

for: $\sigma_{or} + \sigma_z > \sigma p$ (sum of the current vertical stress and its increment is greater than the preconsolidation pressure):

$$s_i = C_{r,i} \frac{h_i}{1+e_0} . log \frac{\sigma_{p,i}}{\sigma_{or,i}} + C_{c,i} \frac{h_i}{1+e_0} . log \frac{\sigma_{or,i} + \sigma_{z,i}}{\sigma_{p,i}}$$

where:

- $\sigma_{or,i}$ - vertical component of geostatic stress in the middle of i^{th} layer
- $\sigma_{z,i}$ - vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression
- $\sigma_{p,i}$ - pre-consolidation pressure in the i^{th} layer
- e_o - initial void ratio
- h_i - thickness of the i^{th} layer
- $C_{c,i}$ - compression index in the i^{th} layer
- $C_{r,i}$ - recompression index in the i^{th} layer

Primary settlement of the i^{th} layer of normally consolidated soil (OCR= 1) reads:

$$s_i = C_{c,i} \frac{h_i}{1+e_0} . log \frac{\sigma_{or,i} + \sigma_{z,i}}{\sigma_{or,i}}$$

where:

- $\sigma_{or,i}$ - vertical component of geostatic stress in the middle of i^{th} layer
- $\sigma_{z,i}$ - vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression
- e_o - initial void ratio
- h_i - thickness of the i^{th} layer

$C_{c,i}$ - compression index the i^{th} layer

Secondary settlement

Secondary settlement of the i^{th} layer assumes the form:

for: $\sigma_{or} + \sigma_z \leq \sigma p$ (sum of the current vertical stress and its increment is less than the preconsolidation pressure):

$$s_{i,d} = C_{\alpha r,i}.h_i.\left(log\frac{t_s}{t_p}\right)$$

for: $\sigma_{or} + \sigma_z > \sigma_p$ (sum of the current vertical stress and its increment is greater than the preconsolidation pressure):

$$s_{i,d} = C_{\alpha,i}.h_i.\left(log\frac{t_s}{t_p}\right)$$

where:

- h_i - thickness of the i^{th} layer
- $C_{\alpha r,i}$ - secondary compression index below preconsolidation pressure in the i^{th} layer
- C_α - index of secondary compression in the i^{th} layer
- t_p - time to terminate primary consolidation
- t_s - time required for secondary settlement

If we specify the value of pre-consolidation index of secondary compression the same as for the index of secondary compression, the program does not take into account in the computation of secondary settlement the effect of pre-consolidation pressure.

Literature:

Netherlandish standard NEN6740, 1991, Geotechniek TGB1990 Basisen en belastingen, Nederlands normalisatie-Institut.

Analysis Using the Soft Soil Model

The analysis employs the modified compression index λ and is based on the Soft soil elastic-plastic model developed in the University of Cambridge. The soil deformation assumes the volumetric strain to be linearly dependent on the change of effective mean stress ε plotted in the natural logarithmic scale. The settlement of the i^{th} layer is then provided by:

$$s_i = \lambda_i.h_i.ln\frac{\sigma_{or,i} + \sigma_{z,i}}{\sigma_{or,i}}$$

where:

- $\sigma_{or,i}$ - vertical component of geostatic stress in the middle of i^{th} layer
- $\sigma_{z,i}$ - vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression
- h_i - thickness of the i^{th} layer
- λ - modified compression index in the i^{th} layer

The analysis requires inputting the modified compression index λ usually obtained from triaxial laboratory measurements.

If the modified compression index λ is not known, it is possible to specify the compression index C_C together with an average value of the void ratio e (if you do not know it, it is sufficient to provide the initial void ratio e_o) and the program then performs an approximate computation of the modified compression index λ using the available information.

Literature:

Burland J.B. The yielding and dilatation of clay (correspondence), Géotechnique, 15 (2),1965, str. 211-214.

Analysis According to the Janbu Theory

It is based on principles of nonlinear elastic deformation, where the stress-strain relationship is described by a function of two dimensionless parameters unique for a given soil. The parameters are the exponent j and the Janbu modulus m. Equations describing the settlement are obtained by specifying ε from the definition of deformation modulus E_t and by subsequent integration. The program allows the user to compute settlement for the following types of soil:

- Cohesionless soils
- Coarse-grained soil
- Sands and silts
- Overconsolidated sands and silts
- Cohesive soils

- Overconsolidated cohesive soils

Literature:

Method of settlement computation for various types of soils, Soil Mechanics and foundation engineering, Springer, 7 (3), 1970, str, 201-206.

Analysis for Cohesionless Soils

For cohesionless soils the stress exponent is not equal to zero. For layered subsoil the resulting settlement equals to the sum of partial settlements of individual layers:

$$s_i = \frac{h_i}{m_i.j_i}\left[\left(\frac{\sigma_{or,i}+\sigma_{z,i}}{100}\right)^j - \left(\frac{\sigma_{or,i}}{100}\right)^j\right]$$

where:

	$\sigma_{or,i}$	-	vertical component of geostatic stress in the middle of i^{th} layer
	$\sigma_{z,i}$	-	vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression
	j_i	-	stress exponent in the i^{th} layer
	m_i	-	Janbu modulus in the i^{th} layer
	h_i	-	thickness of the i^{th} layer

Analysis for Coarse-Grained Soils

For dense coarse-grained soils (e.g. ice soil) the stress-deformation (settlement) relationship is usually assumed as "elastic", i.e. the stress exponent j is equal to one. Thus for $j =$ 1 and the reference stress $\sigma_r =$ 100 kPa the resulting settlement equals to the sum of partial settlements of individual layers:

$$s_i = \frac{h_i}{100.m_i}.(\sigma_{z,i})$$

where:

	$\sigma_{z,i}$	-	vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression - i.e. change of effective stress
	m_i	-	Janbu modulus in the i^{th} layer
	h_i	-	thickness of the i^{th} layer

Analysis for Sands and Silts

For sands and silts the stress exponent j receives the value around 0,5, for the reference stress $\sigma_r =$ 100 kPa the resulting settlement equals the sum of partial settlements of individual layers. It can be derived from the following formula:

$$s_i = \frac{h_i}{5.m_i}.\left(\sqrt{\sigma_{or,i}+\sigma_{z,i}} - \sqrt{\sigma_{or,i}}\right)$$

where:

	$\sigma_{or,i}$	vertical component of geostatic stress in the middle of i^{th} layer
	$\sigma_{z,i}$	vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression
	m_i	Janbu modulus in the i^{th} layer
	h_i	thickness of the i^{th} layer

Analysis for Overconsolidated Sands and Silts

Providing the final stress in the soil exceeds the pre-consolidation pressure ($\sigma_{or}+\sigma_z > \sigma_p$), the settlement of layered subsoil is found from the following equation:

$$s_i = \frac{h_i}{5.m_{r,i}}.\left(\sqrt{\sigma_{p,i}} - \sqrt{\sigma_{or,i}}\right) + \frac{h_i}{5.m_i}.\left(\sqrt{\sigma_{or,i}+\sigma_{z,i}} - \sqrt{\sigma_{p,i}}\right)$$

where:

	$\sigma_{or,i}$	-	vertical component of geostatic stress in the middle of i^{th} layer
	$\sigma_{p,i}$	-	pre-consolidation pressure in the i^{th} layer
	$\sigma_{z,i}$	-	vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression
	m_i	-	Janbu modulus in the i^{th} layer

$m_{r,i}$	-	Janbu modulus of recompression in the i^{th} layer
h_i	-	thickness of the i^{th} layer

If the stress due to surcharge does not cause the final stress to exceed the pre-consolidation pressure ($\sigma_{or}+\sigma_z \leq \sigma_p$), it is possible to assume the following forms of equations for the computation of settlement of layered sand or silt subsoil:

$$s_i = \frac{h_i}{5.m_{r,i}}.\left(\sqrt{\sigma_{or,i}+\sigma_{z,i}} - \sqrt{\sigma_{or,i}}\right)$$

where:

$\sigma_{or,i}$	-	vertical component of geostatic stress in the middle of i^{th} layer
$\sigma_{p,i}$	-	preconsolidation pressure in the i^{th} layer
$\sigma_{z,i}$	-	vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression
$m_{r,i}$	-	Janbu modulus of recompression in the i^{th} layer
h_i	-	thickness of the i^{th} layer

Analysis for Cohesive Soils

In the case of cohesive soils the stress exponent is equal to zero. For normally consolidated soils we obtain from the definition of the tangent modulus of deformation (by modification and subsequent integration) E_t equation for the settlement of layered subsoil formed by cohesive soils in the form:

$$s_i = \frac{h_i}{m_i}.\ln\frac{\sigma_{or,i}+\sigma_{z,i}}{\sigma_{or,i}}$$

where:

$\sigma_{or,i}$	-	vertical component of geostatic stress in the middle of i^{th} layer
$\sigma_{z,i}$	-	vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression
m_i	-	Janbu modulus in the i^{th} layer
h_i	-	thickness of the i^{th} layer

Analysis for Overconsolidated Cohesive Soils

Most cohesive soils in the original order except very young or organic clays are overconsolidated. If the final stress in the soil exceeds overconsolidation stress ($\sigma_{or}+\sigma_z > \sigma_p$) than the settlement of the layered subsoil composites from cohesive soils is computed from following relation:

for: $\sigma_{or}+\sigma_z > \sigma_p$

$$s_i = \frac{h_i}{m_{r,i}}.\ln\frac{\sigma_{p,i}}{\sigma_{or,i}} + \frac{h_i}{m_i}.\frac{\sigma_{or,i}+\sigma_{z,i}}{\sigma_{p,i}}$$

for: $\sigma_{or}+\sigma_z \leq \sigma_p$

$$s_i = \frac{h_i}{m_{r,i}}.\frac{\sigma_{or,i}+\sigma_{z,i}}{\sigma_{or,i}}$$

where:

$\sigma_{or,i}$	-	vertical component of geostatic stress in the middle of i^{th} layer
$\sigma_{p,i}$	-	pre-consolidation pressure in the i^{th} layer
$\sigma_{z,i}$	-	vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression
m_i	-	Janbu modulus in the i^{th} layer
$m_{r,i}$	-	Janbu modulus of recompression in the i^{th} layer
h_i	-	thickness of the i^{th} layer

Settlement Analysis Using DMT (Constrained Soil Modulus)

Constrained modulus M_{DMT} is defined as the vertical drained confined tangent modulus at σ_{vo} . Modulus M_{DMT} is obtained from dilatometric tests (DMT).

If the value of the constrained modulus M_{DMT} is not available, it is possible to input the coefficient of volume compressibility m_V (determined from the oedometric test) and the program carries out the respective transformation:

$$M_{DMT} = \frac{1}{m_V}$$

where: M_{DMT} - constrained modulus

m_V - coefficient of volume compressibility

The analysis employs the constrained modulus M_{DMT} or coefficient of volume compressibility m_V and is based on Marchetti's theory. This approach is based on linear elasticity, provides a settlement proportional to the load, and is unable to provide non-linear predictions.

The settlement of the i^{th} layer is then provided by:

$$s_i = \frac{\sigma_{z,i}.h_i}{M_{DMT}}$$

where: $\sigma_{z,i}$ - vertical component of incremental stress in the middle of i^{th} layer

h_i - thickness of the i^{th} layer

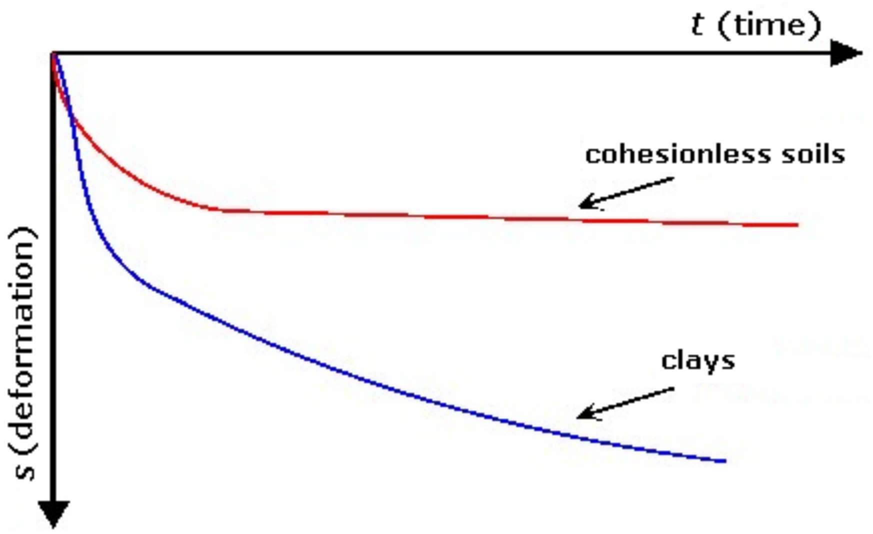
M_{DMT} - constrained modulus

Literature:

Marchetti, S., Monaco, P., Totani, G. & Calabrese, M.: The Flat Dilatometric Test (DMT) in soil investigations. A Report by the ISSMGE Committee TC16, University of L'Aquila, Italy, 2001, 48 p.

Theory of Settlement

If the stress change in the soil or in the currently build earth structure, caused by the ground surface surcharge, is known, it is possible to determine the soil deformation. The soil deformation is generally inclined and its vertical component is termed the settlement. In general, the settlement is non-stationary dependent on time, which means that it does not occur immediately after introducing the surcharge, but it rather depends on the consolidation characteristics of a soil. Permeable, less compressible soils (sand, gravel) deform fast, while saturated, low permeability clayey soils experience gradual deformation called consolidation.



Time dependent settlement of soils

Applied load yields settlement, which can be subdivided based on time-dependent response into three separate components:

- Instantaneous settlement (initial)
- Primary settlement (consolidation)
- Secondary settlement (creep)

Instantaneous settlement

During instantaneous settlement the soil experiences only shear deformation resulting into change in shape without volumetric deformation. The loss of pore pressure in the soil is zero.

Primary settlement

This stage of soil deformation is characterized by skeleton deformation due to motion and compression of grains manifested by volume changes. If the pores are filled with water (particularly in case of low permeability soils), the water will be carried away from squeezed pores into locations with lower pressure (the soil will undergo consolidation). The consolidation primary settlement is, therefore, time-dependent and is terminated by reaching zero pore pressure.

Secondary settlement

When the primary consolidation is over the skeleton deformation will no longer cause the change in pore pressure (theoretically at infinite time). With increasing pressure the grains may become so closely packed that they will start to deform themselves and the volumetric changes will continue - this is referred to as creep deformation of skeleton or secondary consolidation (settlement). Unlike the primary consolidation the secondary consolidation proceeds under constant effective stress. Particularly in case of soft plastic or squash soils the secondary consolidation should not be neglected - in case of overconsolidated soils it may represent the app. 10$\%$ of the overall settlement, for normally consolidated soils even app. 20$\%$.

Primary Settlement

The final primary settlement s is often substituted by the term settlement. Most of the computational approaches can be attached to one of the two groups:

- **Linear elastic deformation**
- **Nonlinear elastic deformation**

Linear elastic deformation

The linear stress-strain relationship follows the Hook law:

$$\varepsilon = \frac{\Delta\sigma_{ef}}{E}$$

where:
- ε - induced deformation of the soil layer
- $\Delta\sigma_{ef}$ - induced change of effective stress in the soil layer
- E - Young's modulus in the soil layer
- ν - Poisson's ratio

The applicability of Young's modulus E of elasticity is substantiated only in cases, in which the stressed soil is allowed to stretch in the horizontal direction. This, however, is acceptable only for small spread foundations. When applying the load over a larger area, the stressed soil cannot, except for its edges, to deform sideways and experiences therefore only a vertical (one-dimensional) strain related to the oedometric modulus E_{oed}, that is larger than the elastic modulus E .

The settlement of a soil layer s is determined by multiplying the deformation of a soil layer ε by the layer thickness (height) H_o :

$$s = \varepsilon . H_o$$

where:
- ε - deformation of the soil layer
- H_o - thickness of the soil layer

In case of layered subsoil we get the total settlement by summing up settlements of individual layers:

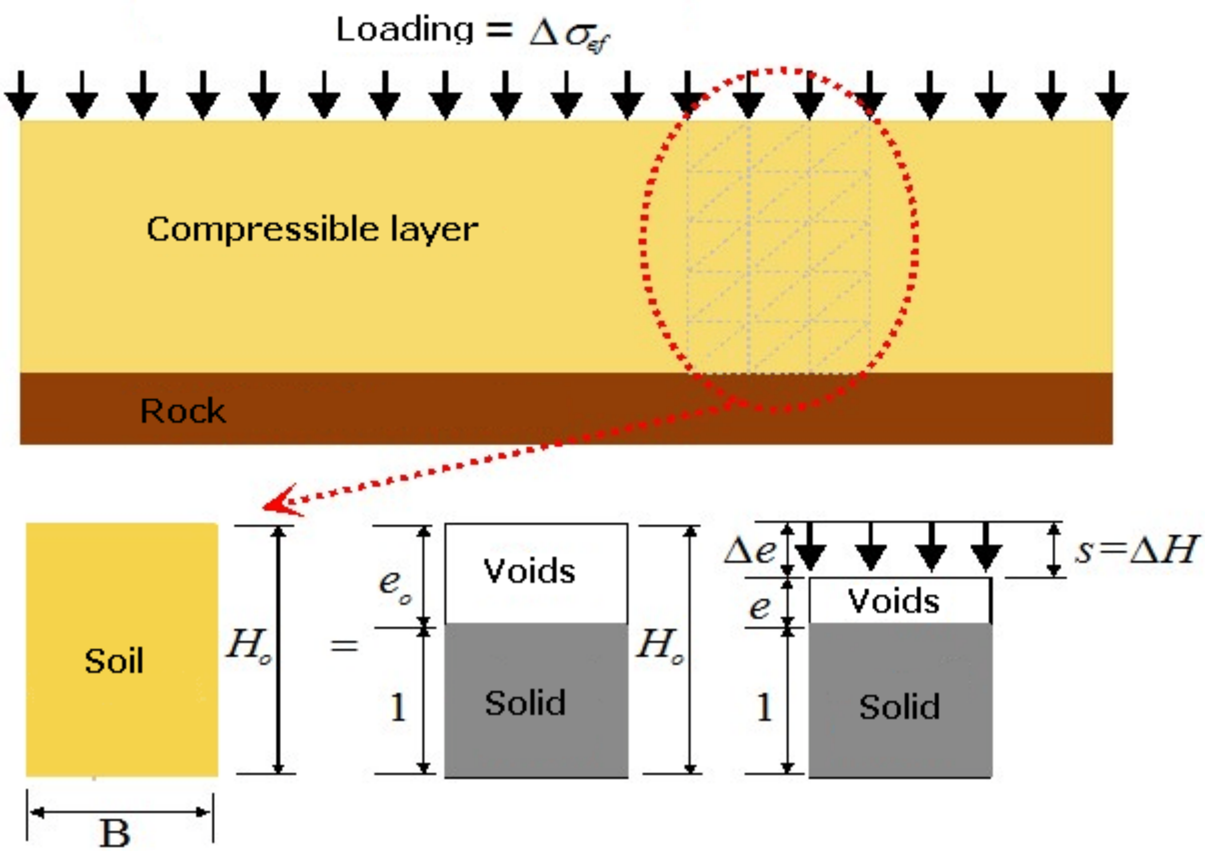
$$s = \sum s_i = \sum \varepsilon_i . H_{oi}$$

where:
- s - settlement of the layered subsoil
- ε_i - deformation of the i^{th} soil layer
- H_{oi} - thickness of the i^{th} soil layer

Nonlinear elastic deformation

For most soils the stress-strain relationship is nonlinear and often influenced by the load history. This nonlinearity cannot be neglected, particularly when computing the settlement of fine-grained soils (silts, clays). Clearly, the procedure based on the application of Young's modulus of elasticity is not generally applicable. Even if employing the stress-dependent oedometric modulus of deformation, it will not be possible to receive reasonable estimates of the behavior of certain overconsolidated soils. Nonlinear elastic deformation is generally modeled using the void ratio and deformation characteristics derived from one-dimensional deformation of a soil sample (e.g. compression constant, compression index, etc.).

The procedure for the computation of settlement of a compressible saturated soil layer using the void e is described on the following soil element having the height H_o and the width $B =$ 1 m :



Analysis of settlement from the phase diagram

Because the soil is a three-phase medium (it contains solid particles and pore filled with fluid and gas) it is possible to describe the solid particles (rock particles and mineral grains) by their volume V_s (and assumed to be equal to unity), while the porous phase can be described using the void ratio e .

The soil element is subjected on its upper surface to a uniform load q causing the change in stress inside the sample and also the vertical displacement ΔH , which in turn leads to the reduction of pores V_p and therefore also to the reduction of void ratio (from its original value e_o to a new value e). The vertical strain ε of a soil sample is given by the ratio of ΔH to the original sample height H_o , and can be expressed using the void ratio e :

$$\varepsilon = \frac{\Delta H}{H_o} = \frac{s}{H_o} = \frac{\Delta e}{1+e}$$

where:

ε	-	vertical relative compression
ΔH	-	vertical deformation
H_o	-	origin height of the element
s	-	settlement
e	-	void ratio
Δe	-	change of void ratio

By modifying this equation we arrive at the formula describing the sample settlement with the help of void ratio:

$$s = \frac{\Delta e}{1+e} . H_o = \varepsilon . H_o$$

where:

ε	-	vertical relative compression
H_o	-	origin height of the element
s	-	settlement
e	-	void ratio
Δe	-	change of void ratio

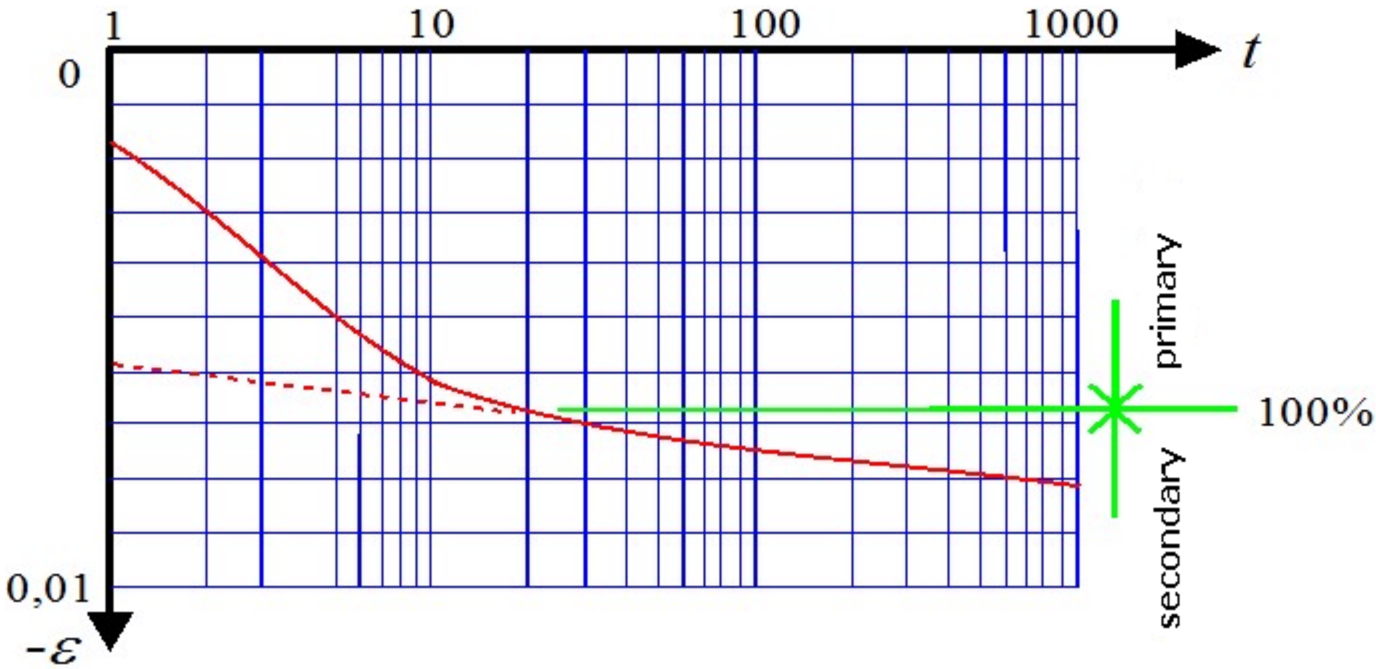
Secondary Settlement

To describe a gradual creep of soil during secondary settlement the program employs the Buissman method (it incorporates the index of secondary compression C_α derived by Ladd). From observations suggesting that the soil deformation follows a linear path when plotted in the semi-logarithmic scale against time Buissman proposed the variation of ε due to long-term stress in the form:

$$\varepsilon = \varepsilon_p + \varepsilon_s . log\left(\frac{t}{t_o}\right)$$

where:

ε	-	total deformation
ε_p	-	deformation associated with primary consolidation
ε_s	-	deformation associated with secondary consolidation
t	-	time of consolidation
t_0	-	reference time



Time-dependent variation of strain (primary and secondary consolidation)

Consolidation Analysis

The program allows you to set the analysis of consolidation in the frame "Settings". Consolidated layer, formed by impermeable, resp. lower permeable soil subsequently settles with increasing time. Consolidation affects values of pore pressure. Soil parameters influencing consolidation analysis are entered in the frame "Soils", other consolidation parameters are set in the frame "Analysis" in individual construction stages.

Consolidation coefficient depended on the soil properties, is calculated:

$$c_v = \frac{E_{oed} . k}{\gamma_w}$$

where:

E_{oed}	-	oedometric modulus of deformation
k	-	coefficient of permeability
γ_w	-	unit weight of water

When the consolidated layer is composed of a non-homogeneous soil, coefficient c_v is evaluated as an average of soil coefficients.

Consolidation analysis is also influenced by time factors, which are dependent on the path of water outflow. This path is equal to the thickness of the consolidated layer in case of only one direction outflow (upwards or downwards) or half of the thickness in case of both directions outflow (upwards and downwards). The real-time factor is evaluated according to the following formula:

$$T_v = \frac{c_v . t}{H^2}$$

where: c_v - consolidation coefficient
t - real-time
H - drainage path

The time factor of build duration is influenced by the duration of load action. When the entire load is introduced at the beginning of the stage, build time is equal to zero. When load linearly increases during stage duration, then build time is equal to the time of stage duration. The time factor of build duration is calculated by the formula:

$$T_c = \frac{c_v.t_c}{H^2}$$

where: c_v - consolidation coefficient
t_c - build time
H - drainage path

Degree of consolidation is evaluated by the following formulas:

for: $T_v \le T_c$

$$U_{av} = \frac{T_v}{T_c}\left\{1 - \frac{2}{T_v}\sum_{m=0}^{m=\infty}\frac{1}{M^4}\left[1-\exp\left(-M^2.T_v\right)\right]\right\}$$

for: $T_v > T_c$

$$U_{av} = 1 - \frac{2}{T_c}\sum_{m=0}^{m=\infty}\frac{1}{M^4}\left[\exp\left(M^2.T_c\right)-1\right]\exp\left(-M^2.T_v\right)$$

where:

$$M = (2m+1)\frac{\pi}{2}$$

where: T_v - real-time factor
T_c - time factor of build duration

The original value of deformation in a consolidated layer in certain construction stage is multiplied by the corresponding degree of consolidation U_{av} to obtain result value of deformation:

$$\varepsilon_{fin} = U_{av}.\varepsilon$$

where: ε_{fin} - result value of deformation
ε - original value of deformation
U_{av} - degree of consolidation

Consolidation analysis also influences pore pressure values in the consolidated layer. In the time of introducing the load action, pore pressure values are the highest. When time increases to theoretical infinity, pore pressure decreases to zero.

Pore pressure:

for: $T_v \le T_c$

$$u = \sum_{m=0}^{m=\infty}\frac{2.u_0}{M^3.T_c}.\sin\frac{M.z}{H}\left[1-\exp\left(-M^2.T_v\right)\right]$$

for: $T_v > T_c$

$$u = \sum_{m=0}^{m=\infty}\frac{2.u_0}{M^3.T_c}.\left[\exp\left(M^2.T_c\right)-1\right]\sin\frac{M.z}{H}\exp\left(-M^2.T_v\right)$$

where:

$$M = (2m+1)\frac{\pi}{2}$$

where: T_v - real-time factor
T_c - time factor of build duration
H - drainage path
z - depth, where the value of pore pressure is evaluated
u_0 - change of effective stress compared to the previous stage (load)

Literature:

Braja M. Das. Advanced Soil Mechanics; Taylor & Francis: London, 2008.pp278 - 316Verruijt A. Soil Mechanics, Delft

University of Technology, 2010, pp97-123.

http://geo.verruijt.net/software/SoilMechBook.pdf.

Determination of the Influence Zone Depth

From the theoretical point of view when applying a load on the ground surface we may expect the change of stress in subsoil into an infinite depth. The soil, however, deforms only up to a certain depth - within so-called influenced zone.

The program offers two options to specify the influence zone:

- Using the theory of structural strength
- By specifying a certain percentage of the primary geostatic stress

Theory of Structural Strength

The structural strength represents the resistance of soil against deformation for a load at the onset of failure of its internal structure. With decreasing coefficient m the soil responds tends to be linear.

If the structural strength is accounted for during settlement analysis, then:

a) the influence zone is characterized by the depth below the footing bottom at which the increment of vertical stress σ_z becomes equal to the structural strength of soil (determined by multiplying the original geostatic stress σ_{or} by the coefficient m):

$$\sigma_z = m.\sigma_{or}$$

where: m - coefficient of structural strength

σ_{or} - original geostatic stress

b) when computing the settlement of a layer, the increment of vertical stress σ_z due to surcharge and reduced by the structural strength of the soil is provided by:

$$\sigma_z - m.\sigma_{or}$$

where: m - coefficient of structural strength

σ_{or} - original geostatic stress

σ_z - incremental stress in the middle layer

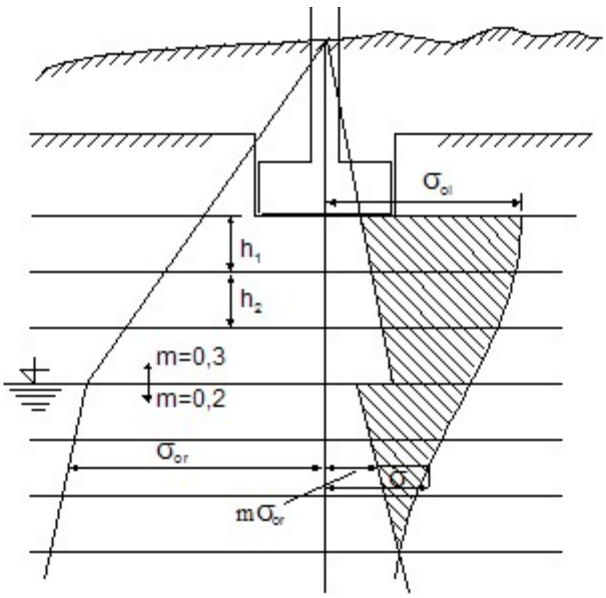
and the settlement s then follows from the stress denoted in figure by hatching and is given by:

$$s = f(\sigma_z, m, \sigma_{or})$$

where: m - coefficient of structural strength

σ_{or} - original geostatic stress

σ_z - incremental stress in the middle layer



Depth of influence zone based on theory of structural strength (area of effective surcharge is hatched)

Method of Restriction of the Primary Stress Magnitude

If we assume in the settlement analysis the constrains in terms of the percentage of primary geostatic stress, then:

a) the influence zone is represented by the depth below the footing bottom where the incremental stress σ_z reaches a certain percentage of the original geostatic stress:

$$\sigma_z = x\%.\sigma_{or}$$

where: $x\%$ - considered magnitude of the geostatic stress

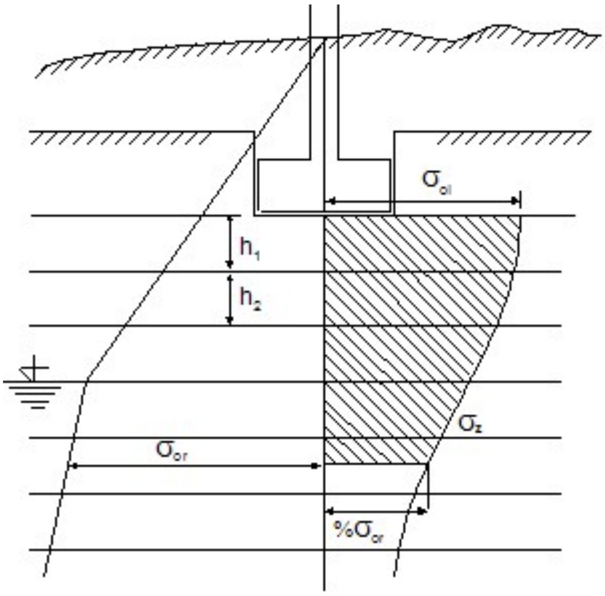
σ_o - geostatic stress

b) the settlement s is derived from stress value denoted in figure by hatching and it receives the form:

$$s = f(\sigma_z, \sigma_{or})$$

where: σ_z - incremenatal stress

σ_{or} - geostatic stress



Depth of influence zone given by constraining the magnitude of primary stress

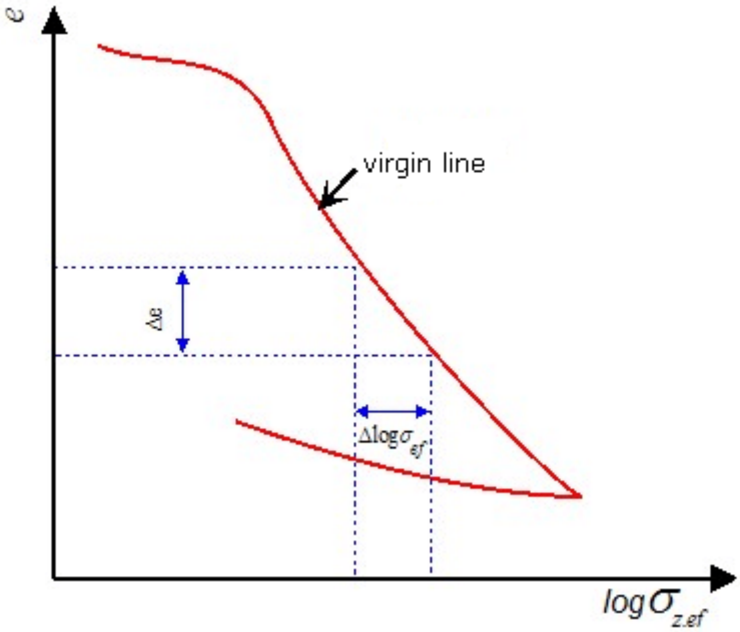
Characteristics of Settlement Analyses

Depending on the selected solution method the program employs for the computation of settlement the following characteristics that may differ by the type of experiment needed for their determination or in the way of representation of measured variables:

- Compression index C_c
- Oedometric modulus E_{oed}
- Deformation modulus E_{def}
- Compression constant C
- Compression constant C_{10}
- Void ratio e
- Recompression index C_r
- Janbu characteristics
- Correcting coefficient m
- Modified compression index λ
- Index of secondary compression C_α
- Overconsolidation index of secondary compression $C_{\alpha r}$

Compression Index

It describes the variation of the void ratio e as a function of the change of effective stress σ_{ef} plotted in the logarithmic scale:



Void ratio e versus effective stress σ_{ef}

It, therefore, represents a deformation characteristic of overconsolidated soil:

$$C_c = \frac{\Delta e}{\Delta \log \sigma_{ef}}$$

where: Δe - variation of void ratio

$\Delta log\sigma_{ef}$ - variation of effective stress

Range of compression index C_c (Naval Facilities Engineering Command Soil MechanicsDESIGN MANUAL 7.01)

A typical range of the compression index is from 0.1 to 10. Approximate values for homogeneous sand for the load range from 95 kPa to 3926 kPa attain the values from 0.05 to 0.06 for loose state and 0.02 to 0.03 for the dense state. For silts this value is 0.20.

For lightly overconsolidated clays and silts tested in USA Louisiana Kaufmann and Shermann (1964) present the following values:

Soil	**Effective consolidation stress** σ_{cef} $[kPa]$	**Final effective stress in the soil** σ_{ef} $[kPa]$	**Compression index** Cc [-]
CL soft clay	160	200	0.34
CL hard clay	170	250	0.44
ML silt of low plasticity	230	350	0.16
CH clay of high plasticity	280	350	0.84
CH soft clay with silt layers	340	290	0.52

Prof. Juan M.Pestana-Nascimento (University of California, Berkeley) offers the following typical values of the compression index C_c:

Soil	**Compression index** Cc [-]
Normal consolidated clays	0.20 - 0.50
Chicago clay with silt (CL)	0.15 - 0.30
Boston blue clay (CL)	0.30 - 0.50
Vickburgs clay - dray falls into lumps (CH)	0.3 - 0.6
Swedish clay (CL - CH)	1 - 3
Canada clay from Leda (CL - CH)	1 - 4
Mexico City clay (MH)	7 - 10
Organic clays (OH)	4 and more
Peats (Pt)	10 - 15

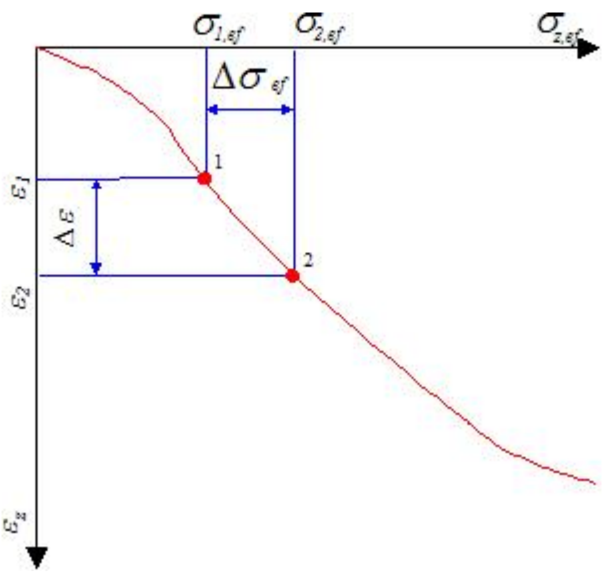
Organic silts and clayey silts (ML - MH)	1.5 - 4.0
San Francisco sediments (CL)	0.4 - 1.2
Clay in the old San Francisco Bay	0.7 - 0.9
Bangkok clay (CH)	0.4

In addition, there are empirical expressions available to determine approximate values of C_c for silts, clays and organic soils; their applicability, however, is more or less local:

Soil	Equations	Reference
Transformed clays	$C_c = 0{,}007(w_z - 7\%)$	Skempton 1944
Clays	$C_c = 1{,}15.(e_0 - 0{,}35)$	Nishida 1956
Brazilian clays Sao Paulo clays	$C_c = 0{,}256 + 0{,}43.(e_0 - 0{,}84)$ $C_c = 0{,}0046(w_z - 9\%)$	Cozzolino 1961
New York clays	$C_c = 0{,}009(w_z - 10\%)$	Terzaghi a Peck 1948
Clays of low plasticity	$C_c = 0{,}75.(w_0 - 0{,}50)$	Sowers 1970
Taipei clays and silts	$C_c = 0{,}54.(e_0 - 0{,}23)$ $C_c = 0{,}007(w_z - 7\%)$	Moh a kol. 1989
Clays	$C_c = 2{,}203.\rho_c.e_0.\left(1 - \left(\frac{0{,}4}{e_0}\right)^2\right)$ $C_c = \frac{a.w_z}{100}.\left(1 - \left(\frac{20}{w_z}\right)^2\right)$	Pestana 1994

Oedometric Modulus

If the results from oedometric test are represented in terms of oedometric curve ($\Delta\varepsilon = f(\Delta\sigma_{ef})$), it becomes evident that for each point on the curve we receive a different ratio $\sigma_{ef} / \varepsilon$.



Determination of oedometric modulus E_{oed}

If the stress-strain curve is replaced for a certain interval of two neighboring stresses σ_{1ef} - σ_{2ef} by a secant line, it is acceptable to assume a linear behavior of soil within this interval and represent the soil compressibility by as $\Delta\sigma_{ef}/\Delta\varepsilon$ - called the oedometric modulus of deformation. The oedometric modulus of deformation is, therefore, a secant modulus linked to a certain stress interval σ_{1ef} - σ_{2ef} selected on the stress-strain diagram $\Delta\varepsilon = (\Delta\sigma_{ef})$:

$$E_{oed} = \frac{\Delta\sigma_{ef}}{\Delta\varepsilon} = \frac{\sigma_{2,ef} - \sigma_{1,ef}}{\varepsilon_2 - \varepsilon_1}$$

In general, the oedometric modulus of deformation E_{oed} tends to decrease its value with the increasing stress interval. Therefore we should consider for each layer a specific value of E_{oed} pertinent to a given stress interval (from original to final stress state). This is reflected in the program by the way of inputting E_{oed}, where it is possible to specify for each soil the respective oedometric curve (σ_{ef}/ε diagram).

Practical experience, however, suggests (e.g. for clays) several orders of magnitude difference between the value of E_{oed} derived from the deformation modulus E_{def} and that provided by the in situ measured loading curve.

The relation between E_{def} and E_{oed} is provided by:

$$E_{oed} = \frac{E_{def}}{\beta}$$

$$\beta = 1 - \frac{2\,\nu^2}{1-\nu}$$

where: ν - Poisson's ratio
E_{def} - deformation modulus

Approximate range of values of oedometric modulus of deformation E_{oed} for individual soils and typical stress range (prof. I. Vanicek: Soil mechanics):

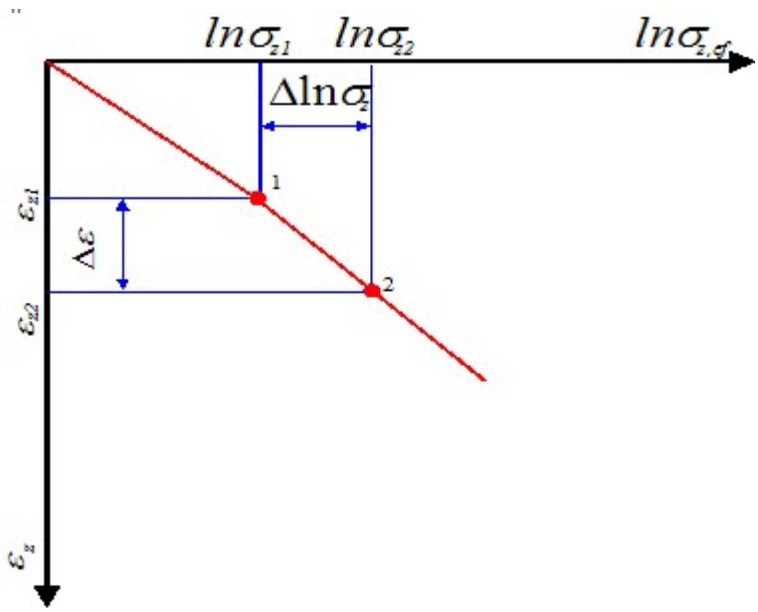
Soil	**Oedometric modulus E_{oed} [MPa]**
gravels	60 - 600
medium dense sands to dense sands	7 - 130
cohesive	2 - 30

Literature:

Vanicek, I.: Geomechanika 10: mechanika zemin. 3th edition, Prague, CTU, 2000, 229 s., ISBN 80-01-01437-1.

Compression Constant

When plotting the effective vertical stress against the vertical strain in the semi-logarithmic scale we often arrive at a linear dependency.



*Detemination of compression constant **C***

The slope of this curve is one of the soil parameters particularly in the case of a one-dimensional deformation and is referred to as the compression constant C:

$$C = \frac{1}{\Delta\varepsilon}.ln\frac{\sigma_{2,ef}}{\sigma_{1,ef}}$$

where: σ_{1ef} - initial effective stress of soil in oedometer
σ_{2ef} - final effective stress of soil in oedometer

Margins of compression constant C *(J.Šimek: Mechanika zemin)*

Soil	**Compression constant C [-]**
Loess silt	15 - 45
Clay	30 - 120
Silts	60 - 150
Medium dense and dense sands	150 - 200
Sand with gravel	> 250

Compression Constant 10

In engineering practice the natural logarithm with base is sometimes replaced by logarithm with base 10 when plotting the stress σ_{ef}. In this case it is common to denote the compression constant with subscript 10: C_{10}. Since it holds:

$$log(x) = \frac{ln(x)}{2,3}$$

it is possible to derive a relationship between compression constant C and C_{10}:

$$C_{10} = \frac{C}{2,3}$$

Arnold Verruijt (Soil Mechanics) offers the following values of compression constant:

Soil	C	C_{10}
Sand	50 - 500	20 - 200
Silt	25 - 125	10 - 50
Clay	10 - 100	4 - 40
Peat	2 - 25	1 - 10

Literature:

Arnold Verruijt: Soil mechanics, Delft University of Technology, 2001, 2006, http://geo.verruijt.net/.

Void Ratio

The void ratio e describes porosity of soil and is provided by:

$$e = \frac{V_p}{V_s}$$

where: V_p - volume of voids

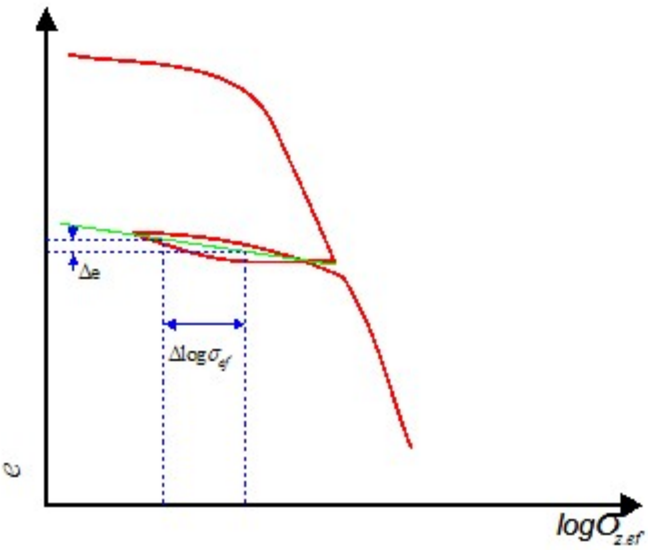
V_s - volume of solid grains

Ranges of void ratio e *(Braja M. DAS: Principles of Foundation Engineering)*

Soil	**Void ratio e [-]**
Poorly graded sand with loose density	*0.8*
Well graded dense sand	*0.45*
Loose density sand with angular particles	*0.65*
Dense density sand with angular particles	*0.4*
Stiff clay	*0.6*
Soft clay	*0.9 - 1.4*
Loess	*0.9*
Soft organic clay	*2.5 - 3.2*
Glacial till	*0.3*

Recompression Index

The recompression index C_r is determined from the graph representing the variation of void ratio e as a function of the effective stress σ_{ef} plotted in the logarithmic scale for unloading - reloading sequence:



Determination of recompression index C_r

$$C_r = \frac{\Delta e}{\Delta \log \sigma_{ef}}$$

where: Δe - change of void ratio for the unloading-reloading curve

$\Delta \log \sigma_{ef}$ - change of effective stress for the unloading-reloading curve

If no results from either laboratory or in situ measurements are available, the recompression index C_r can be approximately derived from:

$$C_r \cong \frac{1}{5} \sim \frac{1}{10} C_c$$

where: C_c - compression constant

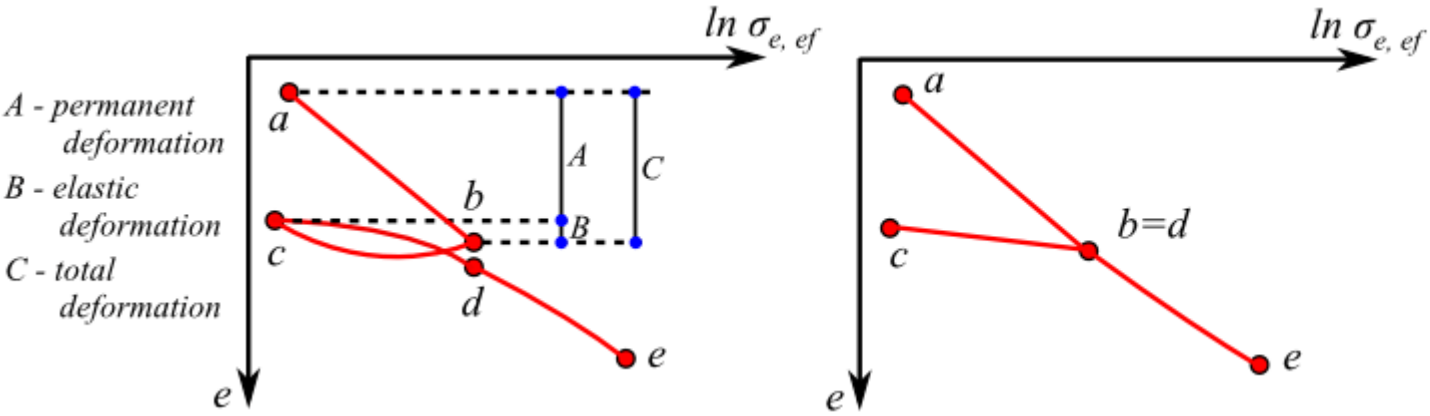
Janbu Characteristics

Values of the Janbu modulus m **and of stress exponent** j (according to Canadian Foundation Engineering Manual 1992)

Soil	**Janbu modulus** m	**Stress index** j
Very dense to dense till, glacial till	1000 - 300	1
Gravel	400 - 40	0.5
Dense sand	400 - 250	0.5
Medium condition sand	250 - 150	0.5
Loose sand	150 - 100	0.5
Dense silt	200 - 80	0.5
Medium condition silt	80 - 60	0.5
Loose silt	60 - 40	0.5
Hard to very stiff clay	60 - 20	0
Medium to stiff clay	20 - 10	0
Soft clayey silt	10 - 5	0
Soft marine clays	20 - 5	0
Organic clays	20 - 5	0
Peats	5 - 1	0

Influence of Load History

The load history has a substantial influence on the distribution of the deformation curve and therefore also on the values of deformation characteristics. The following figure displays the deformation curve ($\Delta e = f(\Delta \sigma_{ef})$ diagram) derived from oedometric loading test corresponding, e.g. to natural dense sandy soil.



Load history a) Deformation curve for clayey soils from oedometric test b) Simplified interpretation of deformation curve

The soil sample was gradually loaded to reach the stress level σ_{bef}, the stress-strain relationship (σ_{bef} - ε) within the section ***a-b*** is linear and is denoted as primary or virgin (i.e., relative compression is encountered). Upon exceeding the stress level σ_{bef} the sample was elastically unloaded and the soil moved up the ***b-c*** section of the deformation curve. Upon reloading the soil moved down the ***b-c*** section till reaching the original stress σ_{bef} prior to unloading. When loading beyond σ_{bef} the deformation curve approaches asymptotically within the ***d-e*** section the primary line accompanied by inelastic deformation of a soil sample. Such a complex stress-strain curve is often simplified by the idealized deformation curve (fig. b). Such a curve characterizes so-called overconsolidated soils, which were in the past subjected large stresses and subsequently unloaded. The overconsolidation ratio (**OCR**) then represents the ratio between the maximum preconsolidation stress the soil has ever experienced and the current vertical stress. Overconsolidated soils typically follow the deformation curve given by points ***c-d-e*** . The change in slope along this line (given app. by point ***d***) corresponds either to the vertical geostatic stress σ_o (normally consolidated soils) or to preconsolidation pressure σ_c (overconsolidated soils). This point influences the soil deformation, which is smaller within the ***c-d*** section when compared to the ***d-e*** section (where for the large degree of overconsolidation the soil deformation increases). Additional deformation characteristics such as deformation modulus upon unloading E_e , one-dimensional swelling index C_e , recompression index C_r , etc. were introduced to describe such a complex soil behavior. Currently the most often used parameter is the recompression index C_r suitable for the computation of settlement of overconsolidated soils.

Coefficient m

The correction coefficient of surcharge due to structural strength m determines the structural strength of soil.

Values of the correction coefficient of surcharge *m*

Type of fundamental soil	m
Very compressible fine soils class F1 -F8 - with deformation modulus $E_{def} < 4\ MPa$ - non overconsoluded - soft to hard consistency (all 3 attributes must be fulfiled), filling, made - ground secondary and tertiary sediments rocks class R1, R2	0.1
fine soils class F1-F8, not belonging to coefficient m = 0.1 or 0.4 or 0.6 sands and gravels class S1, S2, G1, G2 under GWT rock class R3, R4	0.2
Sands and gravels class S1, S2, G1, G2 above GWT sands and gravels with clay, silt or fine soil admixture soils class S3, S4, S5, G3, G4, G5 rocks class R5, R6	0.3
eluvium of igneous and metamorphic rocks	0.4

Modified Compression Index

The analysis employing the Soft soil model builds on the elastic-plastic model developed in the university in Cambridge. Here, the vertical deformation of soil ε assumes linear dependence on the logarithmic variation of effective stress in soil. The application of this model requires an introduction of the modified compression index λ usually obtained from triaxial tests.

If the modified compression index λ is not available from laboratory measurements, it can be approximately found from the compression index C_C:

$$\lambda = \frac{C_c}{2{,}3.(1+e)}$$

where: C_C - compression index

e - average void ratio (if this value is not available, it can be approximately substituted by the initial void ratio e_0)

Index of Secondary Compression

The index of secondary compression is proportional to the logarithm of time and the slope of primary consolidation (it is strongly dependent on the final effective stress in soil):

$$C_\alpha = \frac{\Delta\varepsilon}{\log t_2 - \log t_1}$$

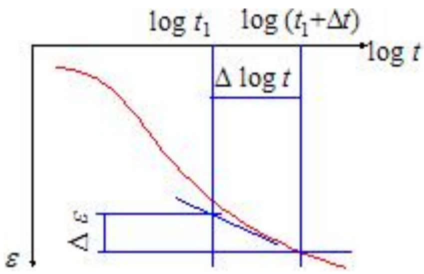
where: C_α - index of secondary compression

α - deformation of a soil layer

t_1 - initial time of a period of monitoring (measured from the start of consolidation)

t_2 - final time of a period of monitoring

Determining the value of index of secondary compression C_α requires either laboratory (e.g. one-dimensional consolidation in oedometer) or in-situ measurements:



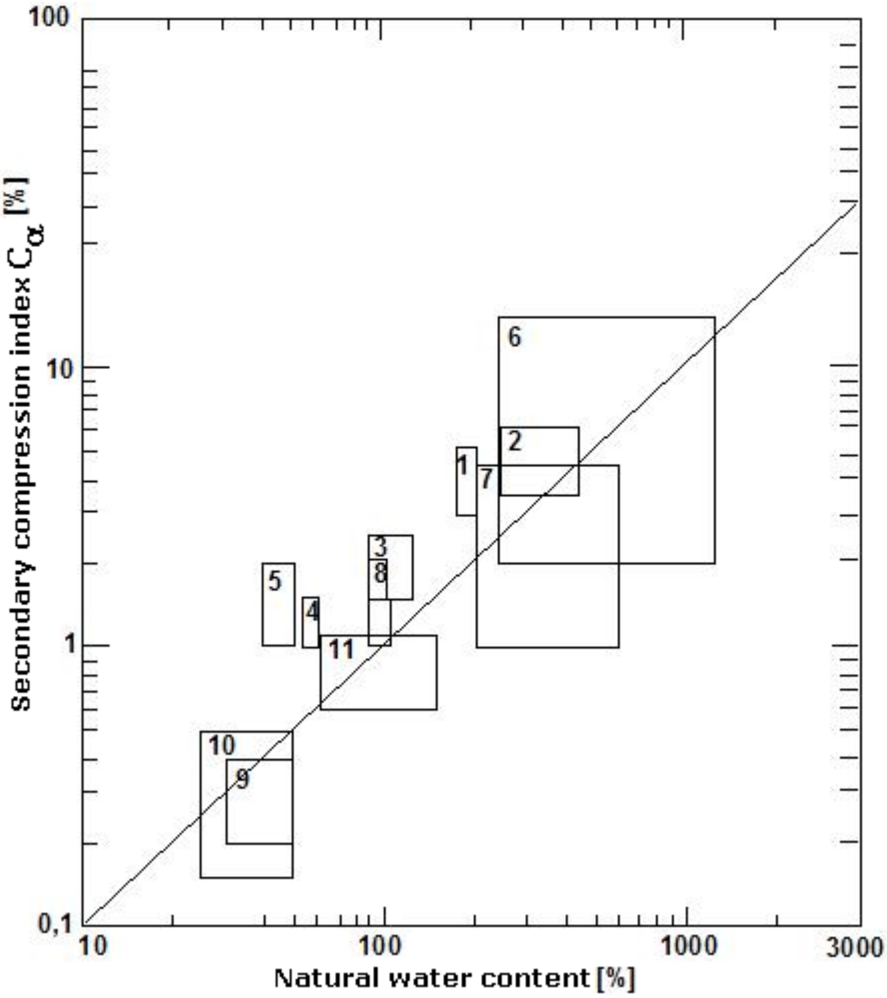
Determination of index of secondary compression C_α

Ranges of values of the index of secondary compression C_α

sand	0.00003 - 0.00006
silty loess	0.0004
clay	0.01

The ratio between the index of secondary compression C_α and the compression index C_c is approximately constant for most of the normally consolidated clays for load typical in engineering practice. Its average value is 0.05.

Variation of natural moisture of soil as a function of the index of secondary compression C_α derived by Mesri appears in the figure:



Variation of natural moisture of soil as a function of the index of secondary compression C_α after Mesri

1 Whangamarino clay
2 Mexico City clay
3 Calcareous organic silt
4 Leda clay
5 Norwegian plastic clay
6 Amorphous and fibrous peat
7 Canadian muskeg
8 Organic marine deposits
9 Boston blue clay
10 Chicago blue clay
11 Organic silty clay

Overconsolidation Index of Secondary Compression

The overconsolidation index of secondary compression depends on laboratory measurements (e.g. one-dimensional consolidation) and is proportional to the logarithm of time and slope of virgin consolidation line providing the preconsolidation pressure was not exceeded:

$$C_{\alpha r} = \frac{\Delta \varepsilon}{log t_2 - log t_1}$$

where:
- $C_{\alpha r}$ - overconsolidation index of secondary compression
- ε - deformation of a soil layer
- t_1 - initial time of a period of monitoring (measured from the onset of consolidation)
- t_2 - final time of a period of monitoring

Ground Loss

Analyses performed in the program "**Ground Loss**" can be divided into the following groups:

- Analysis of the shape of subsidence trough above excavations

- Analysis of failure of buildings

The failure analysis of buildings is based on the shape of subsidence trough.

Analysis of Subsidence Trough

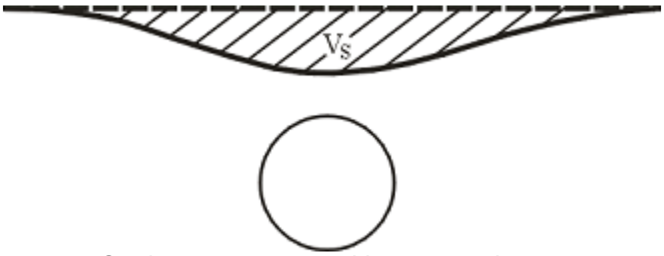
The analysis of subsidence trough consists of several sequential steps:

- Determination of the **maximum settlement** and **dimensions of subsidence trough** for individual excavations
- Analysis of the shape of subsidence trough
- Back calculation of the shape and dimensions of subsidence trough providing it is calculated at a given depth below the terrain surface
- Determination of the overall shape of subsidence trough for more excavations
- Post-processing of other variables (horizontal deformation, slope)

The analysis of maximum settlement and dimensions of subsidence trough can be carried out using either the theory of volume loss or the classic theories (Peck, Fazekas, Limanov).

Volume Loss

The volume loss method is a semi-empirical method based partially on theoretical grounds. The method introduces, although indirectly, the basic parameters of excavation into the analysis (these include mechanical parameters of a medium, technological effects of excavation, excavation lining etc) using 2 comprehensive parameters (**coefficient k for determination of inflection point** and **a percentage of volume loss VL**). These parameters uniquely define the shape of subsidence trough and are determined empirically from years of experience.



Settlement expressed in terms volumes

The maximum settlement S_{max} , and location of inflection point L_{inf} are provided by the following expressions:

$$L_{inf} = kZ$$

$$S_{max} = \frac{A.VL}{100} \frac{1}{\sqrt{2\pi L_{inf}}}$$

where: A - excavation area

Z - depth of center point of excavation

k - coefficient to calculate inflection point (material constant)

VL - percentage of volume loss

The roof deformation u_a follows from:

$$u_a = \frac{2r - \sqrt{4r^2 - \frac{4r^2 VL}{100}}}{2}$$

where: r - excavation radius

VL - percentage of volume loss

Literature:

http://www.groundloss.com/

Recommended Values of Parameters for Volume Loss Analysis

Data needed for the determination of subsidence trough using the volume loss method:

Coefficient to calculate inflection point k

Soil or rock	k
cohesionless soil	0.3

normally consolidated clay	0.5
overconsolidated clay	0.6 - 0.7
clay slate	0.6 - 0.8
quartzite	0.8 - 0.9

Percentage of volume loss VL

Technology	VL
TBM	0.5 - 1
Sequential excavation method	0.8 - 1.5

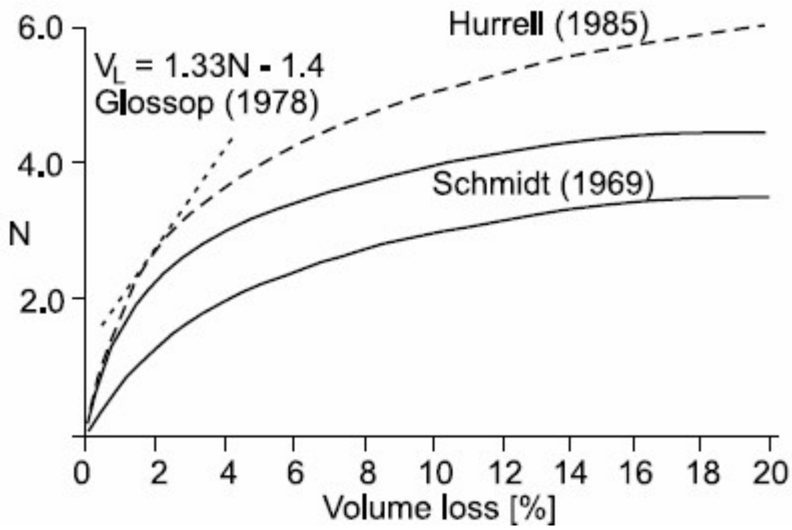
Several relationships were also derived to determine the value of lost volume VL based on stability ratio N defined by Broms and Bennermarkem:

$$N = \frac{\sigma_v \sigma_t}{S_n}$$

where:

- σ_v - overall stress along excavation axis
- σ_t - excavation lining resistance (if lining is installed)
- S_u - undrained stiffness of clay

For $N <$ 2 the soil/rock in the vicinity of excavation is assumed elastic and stable. For $N \in < 2,4$ local plastic zones begin to develop in the vicinity of excavation, for $N \in < 4,6$ a large plastic zone develops around the excavation and for $N =$ 6 the loss of stability of tunnel face occurs. The figure shows the dependence of stability ration and lost volume VL .



Literature:

Broms, B.B., Bennemark, H., 1967. Stability of clay at vertical openings. ASCE, Journal of Soil Mechanics and Foundation Engineering Division, SMI 93, 71-94.

Classic Theory

Convergence analysis of excavation and calculation of the maximum settlement in a **homogeneous body** are the same for all classic theories. The subsidence trough analyses then differ depending on the assumed theory (Peck, Fazekas, Limanov).

When calculating settlement the program first determines the radial load of a circular excavation as:

$$p = \sigma_z \frac{1 + K_r}{2}$$

where:

- σ_z - geostatic stress in the center of the excavation
- K_r - coefficient of pressure at rest of cohesive soil

The roof u_a and the bottom u_b deformations of excavation follow from:

$$u_a = (1 + \nu) \frac{p}{E} r \frac{Z + (1 - 2\nu) r}{Z - r}$$

$$u_b = -(1 + \nu) \frac{p}{E} r \frac{Z - (1 - 2\nu) r}{Z + r}$$

where: Z - depth of center point of excavation

r - excavation radius

E - modulus of elasticity of rock/soil in the vicinity of the excavation

ν - Poisson's ratio of rock/soil in the vicinity of the excavation

The maximum terrain settlement and the length of subsidence trough are determined as follows:

$$S_{max} = \left(1 - \nu^2\right) \frac{p}{E} \frac{4r^2 Z}{Z^2 - r^2}$$

$$L = 2\sqrt{Z^2 - r^2}$$

where: Z - depth of center point of excavation

r - excavation radius

E - modulus of elasticity of rock/soil in the vicinity of the excavation

ν - Poisson's number of rock/soil in the vicinity of the excavation

When the **tunnel roof displacement is prescribed** the maximum settlement is provided by the following expression:

$$S_{max} = 4u_a \frac{rZ\left(1 - \nu\right)}{\left(Z + r\right)\left(Z + r + 2\nu r\right)}$$

where: Z - depth of center point of excavation

r - excavation radius

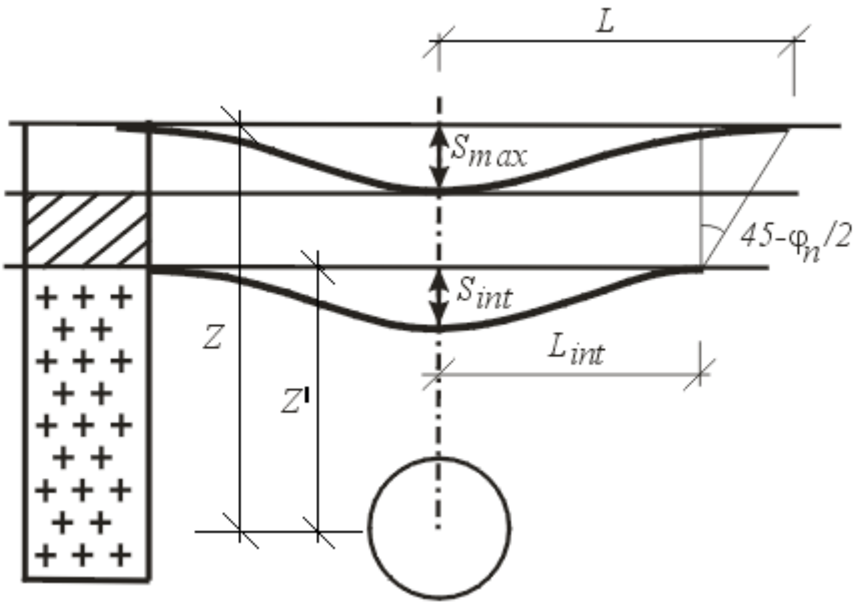
u_a - tunnel roof displacement

ν - Poisson's number of rock/soil in the vicinity of the excavation

Analysis for Layered Subsoil

When determining a settlement of layered subsoil the program first calculates the settlement at the interface between the first layer above excavation and other layers of overburden S_{int} and determines the length of subsidence trough along the layer interfaces. In this case, the approach complies with the one used for a homogeneous soil.

Next (as shown in Figure) the program determines the length of subsidence trough L at the terrain surface.



Analysis of settlement for layered subsoil

The next computation differs depending on the selected analysis theory:

Solution according to Limanov

Limanov described the horizontal displacement above excavation with the help of lost area F :

$$S_{max} = \frac{F}{L}$$

where: L - length of subsidence trough

F - volume loss of soil per 1m run determined from:

$$F = S_{int} \pi \frac{L_{int}}{2}$$

where: L_{int} - length of subsidence trough along interfaces above the excavation

S_{int} - settlement of respective interface

Solution according to Fazekas

Fazekas described the horizontal displacement above excavation using the following expression:

$$S_{max} = S_{int} \frac{L_{int}}{L}$$

where: L - length of subsidence trough

L_{int} - length of subsidence trough along interfaces above the excavation

S_{int} - settlement of respective interface

Solution according to Peck

Peck described the horizontal displacement above excavation using the following expression:

$$S_{max} = S_{int} \frac{L_{int}}{L_{inf}} \frac{\pi}{5}$$

where: L_{int} - length of subsidence trough along interfaces above the excavation

S_{int} - settlement of respective interface

L_{inf} - distance of inflection point of subsidence trough from excavation axis at terrain surface

Literature:

Széchy, Károly, The art of tunelling, Budapest : Akadémiai Kiadó, 1966.

Shape of Subsidence Trough

The program offers two particular shapes of subsidence troughs - according to Gauss or Aversin.

Curve based on Gauss

A number of studies carried out both in the USA and Great Britain proved that the transverse shape of subsidence trough can be well approximated using the Gauss function. This assumption then allows us to determine the horizontal displacement at a distance x from the vertical axis of symmetry as:

$$S_i = S_{max} e^{\left(\frac{-x_i^2}{2L_{inf}^2}\right)}$$

where: S_i - settlement at point with coordinate x_i

S_{max} - maximum terrain settlement

L_{inf} - distance of inflection point

Curve based on Aversin

Aversin derived, based on visual inspection and measurements of underground structures in Russia, the following expression for the shape of subsidence trough:

$$S_i = S_{max} \left(1 - \frac{x_i}{L}\right)^4 e^{\left(\frac{4x_i}{L}\right)}$$

where: S_i - settlement at point with coordinate x_i

S_{max} - maximum terrain settlement

L - reach of subsidence trough

Literature:

Széchy, Károly, The art of tunelling, Budapest : Akadémiai Kiadó, 1966.

Coefficient of Calculation of Inflection Point

When the classical methods are used the input coefficient k_{inf} allows the determination of the inflection point location based on $L_{inf} = L / k_{inf}$. In this case, the coefficient k_{inf} represents a very important input parameter strongly influencing the shape and slope of subsidence trough. Its value depends on the average soil or rock, respectively, in overburden - literature offers the values of k_{inf} in the range 2.1 - 4.0.

Based on a series of FEM calculations the following values are recommended:

gravel soil G1-G3	$k_{inf} =$ 3.5
sand and gravel soil S1-S5, G4, G5, rocks R5-R6	$k_{inf} =$ 3.0
fine-grained soil F1-F4	$k_{inf} =$ 2.5
fine-grained soil F5-F8	$k_{inf} =$ 2.1

The coefficient for calculation of inflection point is input in the frame "Settings".

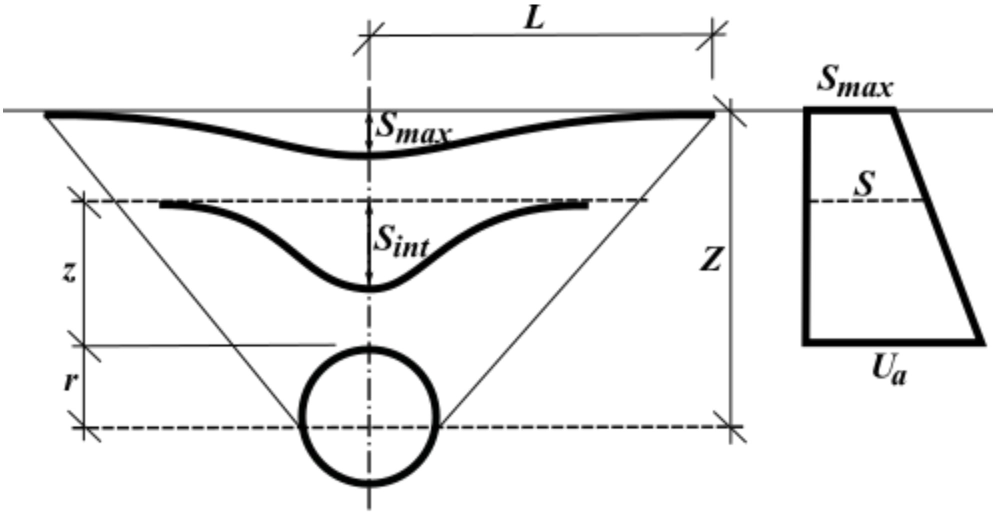
Subsidence Trough with Several Excavations

The principle of superposition is used when calculating the settlement caused by structured or multiple excavations. Based on input parameters the program first determines subsidence troughs and horizontal displacements for individual excavations. The overall subsidence trough is determined subsequently.

Other variables, horizontal strain, and gradient of subsidence trough, are post-processed from the overall subsidence trough.

Analysis of Subsidence Trough in Depth

Linear interpolation between the maximal value of the settlement S_{max} at a terrain surface and the displacement of roof excavation u_a is used to calculate the maximum settlement S at a depth h below the terrain surface in a homogeneous body.



Analysis of subsidence trough at a depth

The width of subsidence trough at an overburden l is provided by:

$$l = \frac{(L-r)(z+r)}{Z} + r$$

where:

- L - length of subsidence trough at terrain surface
- r - excavation radius
- Z - depth of the center point
- z - analysis depth

The values l and S are then used to determine the shape of subsidence trough in the overburden above an excavation.

Calculation of Other Variables

A vertical settlement is accompanied by the evolution of horizontal displacements which may cause damage to nearby buildings. The horizontal displacement can be derived from the vertical settlement providing the resulting displacement vectors are directed into the center of excavation. In such a case the horizontal displacement of the soil is provided by the following equation:

$$S_x = -\frac{s(x)}{Z-r}$$

where:

x	-	distance of point x from the axis of excavation
$s(x)$	-	settlement at point x
Z	-	depth of center point of excavation
r	-	excavation radius

The horizontal displacements are determined in a differential way along the x axis and in the transverse direction they can be expressed using the following equation:

$$\varepsilon_x = -\frac{s(x)}{Z-r}\left(\frac{x^2}{L_{inf}^2}-1\right)$$

where:

x	-	distance of point x from the axis of excavation
$s(x)$	-	settlement at point x
Z	-	depth of center point of excavation
L_{inf}	-	distance of inflection point
r	-	excavation radius

Analysis of Failure of Buildings

The program first determines the shape and dimensions of subsidence trough and then performs analysis of their influence on buildings.

The program offers four types of analysis:

- Determination of tensile cracks
- Determination of gradient damage
- Determination of a relative deflection of buildings (hogging, sagging)
- Analysis of the input section of a building

Tensile Cracks

One of the causes responsible for the damage of buildings is the horizontal tensile strain. The program highlights individual parts of a building with a color pattern that corresponds to a given class of damage. The maximum value of the tensile strain is provided in the text output.

The program offers predefined zones of damage for masonry buildings. These values can be modified in the frame "Stage settings". Considerable experience with a number of tunnels excavated below build-up areas allowed for elaborating the relationship between the shape of subsidence trough and damage of buildings to such precision that based on this it is now possible to estimate an extent of compensations for possible damage caused by excavation with accuracy acceptable for both preparation of contractual documents and for contractors preparing proposals for excavation of tunnels.

Recommended values for masonry buildings from one to six floors are given in the following table.

Horizontal strains (per mille)

Proportional h.s. (per mille)	Damage	Description
0.2 - 0.5	Microcracks	Microcracks
0.5 - 0.75	Little damage - superficial	Cracks in plaster
0.75 - 1.0	Little damage	Small cracks in walls
1.0 - 1.8	Medium damage, functional	Cracks in walls, problems with windows and doors
1.8 -	Large damage	Wide open cracks in bearing walls and beams

Gradient Damage

One of the causes leading to the damage of buildings is the slope subsidence trough. The program highlights individual parts of a building with a color pattern that corresponds to a given class of damage. The maximum value of the tensile strain is provided in the text output.

The program offers predefined zones of damage for masonry buildings. These values can be modified in the frame "Stage settings". Considerable experience with a number of tunnels excavated below build-up areas allowed for elaborating the relationship between the shape of subsidence trough and damage of buildings to such precision that based on this it is now possible to estimate an extent of compensations for possible damage caused by excavation with accuracy acceptable for both preparation of contractual documents and for contractors preparing proposals for excavation of tunnels.

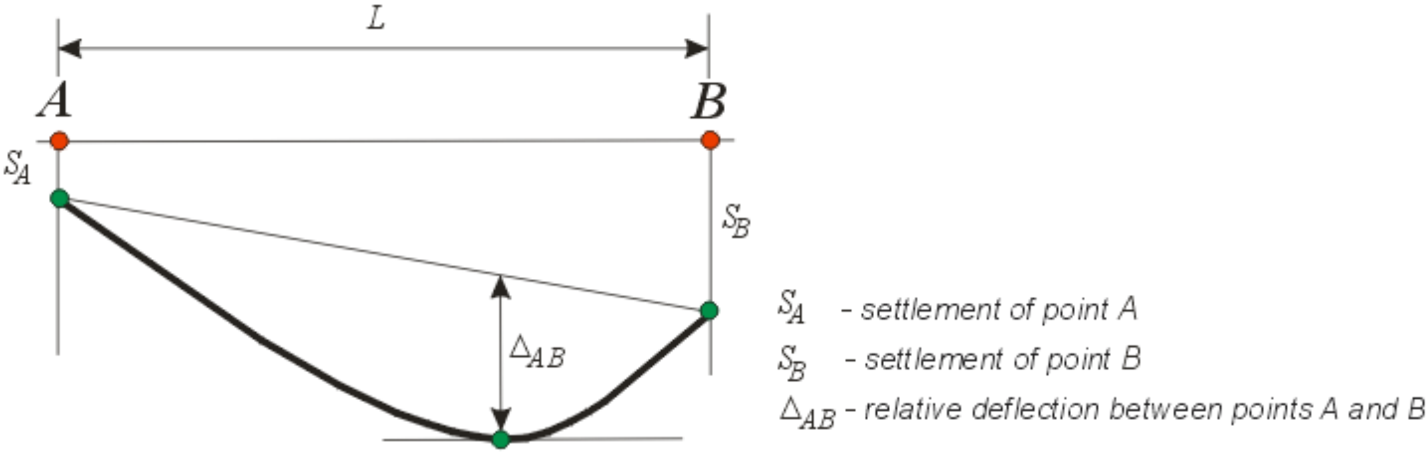
Recommended values for masonry buildings from one to six floors are given in the following table.

Gradient

Gradient	Damage	Description
1:1200 - 800	Microcracks	Microcracks
1:800 - 500	Little damage - superficial	Cracks in plaster
1:500 - 300	Little damage	Small cracks in walls
1:300 - 150	Medium damage, functional	Cracks in walls, problems with windows and doors
1:150 - 0	Large damage	Wide-open cracks in bearing walls and beams

Relative Deflection

Definition of the term relative deflection is evident from the figure. The program searches regions on buildings with the maximum relative deflection both upwards and downwards. Clearly, from the damage of the building point of view the most critical is the relative deflection upwards leading to the "**tensile opening**" of building.



Relative deflection

Verification of the maximum relative deflection is left to the user - the following tables list the ultimate values recommended by literature.

Type of structure	**Type of damage**	**Ultimate relative deflection** Δ/l			
		Burland and Wroth	Meyerhof	Polshin a Tokar	ÈSN 73 1001
Unreinforced bearing walls	Cracks in walls	For $L/H =$ 1 - 0.0004 For $L/H =$ 5 - 0.0008	0.0004	0.0004	0.0015
	Cracks in bearing structures	For $L/H =$ 1 - 0.0002 For $L/H =$ 5 - 0.0004	-	-	-

Failure of a Section of a Building

In a given section the program determines the following variables:

- maximum tensile strain
- maximum gradient
- maximum relative deflection

- relative gradient between input points of a building

Evaluation of the analyzed section is left to the user - the following tables list the recommended ultimate values of relative rotation and deflection.

Type of structure	**Type of damage**	**Ultimate relative gradient**				
		Skempton	Meyerhof	Polshin a Tokar	Bjerrum	CSN 73 1001
Frame structures and reinforced bearing walls	Structural	1/150	1/250	1/200	1/150	
	Cracks in walls	1/300	1/500	1/500	1/500	1/500

Type of structure	**Type of damage**	**Ultimate relative deflection** Δ / l			
		Burland and Wroth	Meyerhof	Polshin a Tokar	CSN 73 1001
Unreinforced bearing walls	Cracks in walls	For $L/H =$ 1 - 0.0004 For $L/H =$ 5 - 0.0008	0.0004	0.0004	0.0015
	Cracks in bearing structures.	For $L/H =$ 1 - 0.0002 For $L/H =$ 5 - 0.0004	-	-	-

Concrete Structures

Concrete structures can be analyzed according to the following standards:

- EN 1992-1-1 (EC 2) or EN 1992-2
- CSN 73 1201R
- CSN 73 6206 (only for Abutment)
- PN-B-03264:2002
- BS 8110:1997
- IS 456
- ACI 318-19
- AS 3600-2018
- SNiP 52-101-2003
- GB 50010-2010
- NZS 3101-2006
- CSA A23.3-14
- NBR 6118-2014

EN 1992-1-1 (EC2) or EN 1992-2

This help contains the following methods:

- Materials, coefficients, notation
- Standard values of coefficients
- Verification of rectangular cross-section made from plain concrete
- Verification of rectangular RC cross-section
- Verification of circular RC cross-section
- Verification of spread footing for punching shear
- Design of longitudinal reinforcement for slabs
- Design of shear reinforcement for slabs
- Verification of Crack Width

Materials, Coefficients, Notation

The following notation for material parameters is used:

f_{ck} - characteristic value of cylindrical compressive strength of concrete

f_{cd} - design compressive strength of concrete

f_{cm} - average value of compressive strength of concrete

f_{ctm} average value of tensile strength of concrete

$f_{ctk0,05}$ - lower value of the characteristic tensile strength of concrete

f_{ctd} - design tensile strength of concrete

f_{yk} - characteristic tensile strength of steel bar

f_{yd} - design tensile strength of steel bar

The characteristic compressive strength of concrete is the basic input parameter given by the class of concrete - it serves to derive the remaining coefficients of reliability (Tbl. 3.1).

$$f_{cd} = \alpha_{cc} . \frac{f_{ck}}{\gamma_c}$$

$$f_{cm} = f_{ck} + 8$$

$$f_{ctm} = 0{,}3.(f_{ck})^{\frac{2}{3}}$$ for: $f_{ck} \leq$ 50 MPa

$$f_{ctm} = 2{,}12.ln\left(1 + \frac{f_{cm}}{10}\right)$$ for: $f_{ck} >$ 50 MPa

$$f_{ctk,005} = 0{,}7.f_{ctm}$$

$$f_{ctd} = \alpha_{ct} . \frac{f_{ctk,005}}{\gamma_c}$$

$$E_{cm} = 22.\left(\frac{f_{cm}}{10}\right)^{0,3}$$

$$f_{yd} = \frac{f_{yk}}{\gamma_s}$$

Standard values of coefficients α_{cc}, γ_c, α_{ct}, γ_s are built-in the program - these values can also be input by the user depending on the **selected National annex**.

The most common notation for geometrical parameters:

b - cross-section width

h - cross-section depth

d - effective depth of cross-section

z - lever arm (arm of internal forces)

Standard Values of Coefficients

The standard contains a number of coefficients, which can be adjusted in **National annexes**. The table provides a description of individual coefficients, their values, and corresponding article of the standard. In some cases the formula contains a variable, which has no symbol in the standard - in such a case the variable in the **expression is denoted by** X.

Coefficient	Value	Annotations	Article
γ_c	1.5		2.4.2.4
γ_s	1.15		2.4.2.4
α_{cc}	1		3.1.6
α_{ct}	1		3.1.6
$\alpha_{cc,pl}$	0.8		12.3.1
$\alpha_{ct,pl}$	0.8		12.3.1
k	1.5		12.6.3
ρ_{min}	0.0013		9.2.1.1
X	0.26	$\rho_{min} = X . \frac{f_{ctm}}{f_{yk}}$	9.2.1.1

ρ_{max}	0.04		9.2.1.1
ρ_{min}	0.002		9.5.2
X	0.1	$\rho_{\min} = \frac{X\,N_{Ed}}{f_{yd}\,A_s}$	9.5.2
ρ_{max}	0.04		9.5.2
X	0.18	$C_{Rd,c} = \frac{X}{\gamma_c}$	6.2.2
v_{min}	-	$0{,}035.k^{\frac{3}{2}}.f_{ck}^{\frac{1}{2}}$	6.2.2
X	0.5	$v_{max} = X.v.f_{cd}$	6.2.2
v	-	$v = 0{,}6.\left(1 - \frac{f_{ck}}{250}\right)$	6.2.2
$\cot\theta_{min}$	1		6.2.3
$\cot\theta_{max}$	2.5		6.2.3

National Annex Czech Republic (CSN EN 1992-1-1 - 2010)

Coefficient	Value	Annotations	Article
$\alpha_{ct,pl}$	0.7		12.3.1

other values are standard

National Annex Slovakia (STN EN 1992-1-1 - 2008)

all values are standard

National Annex Poland (PN EN 1992-1-1 - 2008)

Coefficient	Value	Annotations	Article
γ_c	1.4		2.4.2.4
$\cot\theta_{max}$	2.0		6.2.3

other values are standard

National Annex Norway (NS EN 1992-1-1 - 2004)

Coefficient	Value	Annotations	Article
α_{cc}	0.85		3.1.6
α_{ct}	0.85		3.1.6

other values are standard

National Annex Romania (BDS EN 1992-1-1 - 2005)

all values are standard

National Annex Italy (UNI EN 1992-1-1 - 2005)

Coefficient	Value	Annotations	Article
α_{cc}	*0,85*		3.1.6
v	*0,5 for ≤C70/85*	$v = 0{,}6.\left(1 - \frac{f_{ck}}{250}\right)$ for other	6.2.2

other values are standard

National Annex United Kingdom (BS EN 1992-1-1 - 2004)

Coefficient	Value	Annotations	Article
α_{cc}	0,85		3.1.6
$\alpha_{cc,pl}$	*0,6*		12.3.1
$\alpha_{ct,pl}$	*0,6*		12.3.1

other values are standard

EN 1992-2 - 2007

Coefficient	Value	Annotations	Article
α_{cc}	0.85		3.1.6

other values are standard

Verification of Rectangular Cross Section Made of Plain Concrete

The cross-section is rectangular, loaded by the bending moment M_{Ed}, normal force N_{Ed} (applied in the cross-section centroid), and by the shear force V_{Ed}. The shear strength is provided by (Art. 12.6.3):

$$V_{Rd} = \frac{f_{cvd} \cdot A_{cc}}{k}$$

where: A_{cc} - compressed area of concrete

$$f_{cvd} = \sqrt{f_{ctd}^2 + \sigma_{cp} \cdot f_{ctd} - \left(\frac{Max(0\,;\sigma_{cp} - \sigma_{c,lim})}{2}\right)^2}$$

$$\sigma_{cp} = \frac{N_{Ed}}{A_{cc}}$$

$$\sigma_{c,lim} = f_{cd} - 2.\sqrt{f_{ctd} \cdot (f_{cd} + f_{ctd})}$$

Standard value of the coefficient k is built-in the program (Art. 12.6.3) - this value can also be adjusted in the program based on the **selected National annex**.

Strength of concrete cross-section subject to the combination of bending moment and normal force is derived from the following expressions (Art. 12.6.1) depending on the normal force eccentricity e :

As the greater of:

$$N_{Rd} = b.x.\eta.f_{cd}$$

$$N_{Rd} = Min\left(\frac{b.h.f_{ctd}}{\frac{6.e}{h} - 1}\,;\frac{b.h.f_{cd}}{\frac{6.e}{h} + 1}\right)$$

Formula expresses the strength with the linear stress-strain diagram of cross-section without the crack.

$$\eta = 1{,}0 - \frac{Max(f_{ck};50) - 50}{200}$$

$$x = h - 2.e$$

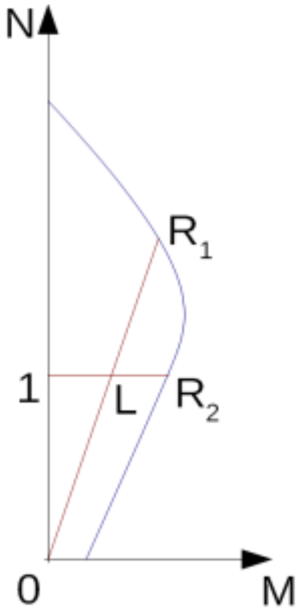
$$e = Max\left(abs\left(\frac{M_{Ed}}{N_{Ed}}\right); \frac{h}{30}; 20mm\right)$$

Minimal values of eccentricity are from article 6.1(3).

$$f_{cd} = \alpha_{cc,pl} \cdot \frac{f_{ck}}{\gamma_c}$$

$$f_{ctd} = \alpha_{ct,pl} \cdot \frac{f_{ctk,005}}{\gamma_c}$$

Standard values of coefficients $\alpha_{cc,pl}$, $\alpha_{ct,pl}$, γ_c are built-in the program - these values can also be input by the user depending on the **selected National annex**.



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R_1|$ or $|1L|/|1R_2|$. Where L is load, R_1 is strength with prescribed eccentricity, and R_2 is strength with prescribed normal force.

Verification of Rectangular RC Cross Section

The cross-section is rectangular, unilaterally reinforced, and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation. The maximum allowable strain of concrete in compression is 0.002 to 0.0035. Compression reinforcement is not taken into account. Minimum eccentricity is applied: (Art. 6.1(3)):

$$e_0 = \text{Min}\left[\frac{h}{30}; 20mm\right]$$

The computed degree of reinforcement is checked using the following expressions (Art. 9.2.1.1):

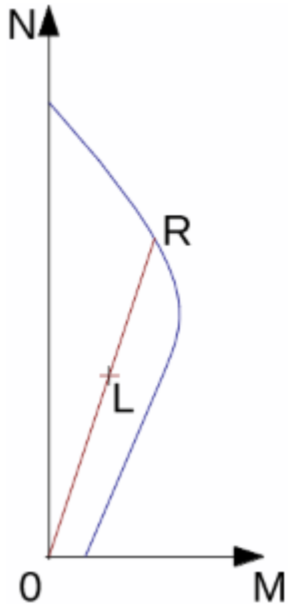
$$\rho_{min} \leq \rho \leq \rho_{max}$$

$$\rho = \frac{A_s}{bd}$$

$$\rho_{min} = \text{Max}\left(0.0013; 0.26\frac{f_{ctm}}{f_{yk}}\right)$$

$$\rho_{max} = 0.04$$

Standard values of coefficients ρ_{min}, ρ_{max} are built-in the program - these values can also be input by the user depending on the **selected National annex**.



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R|$. Where L is load and R is strength with prescribed eccentricity.

Bending without normal force

The cross-section is rectangular, reinforced on one side, and loaded by the bending moment M_{Ed} .

The permissible moment for a given area of reinforcements A_s reads (Art. 6.1, Art. 3.1.7(3)):

$$M_{Rd} = \lambda x b \eta f_{cd}\left(d - \frac{\lambda}{2}x\right)$$

$$x = \frac{A_s f_{yd}}{\lambda b \eta f_{cd}}$$

$$\lambda = 0.8 - \frac{\mathrm{Max}\left(f_{ck};50\right) - 50}{400}$$

$$\eta = 1.0 - \frac{\mathrm{Max}\left(f_{ck};50\right) - 50}{200}$$

The limit location of the neutral axis is found from (Art. 5.6.3(2)):

$x_{max} =$ 0.45d for concrete C40/45 and lower

$x_{max} =$ 0.35d for concrete C45/50 and higher

Shear

First, the program computes the ultimate shear strength of concrete $V_{Rd,c}$ (Art. 6.2.2(1)).

$$V_{Rd,c} = \mathrm{Max}\left[\nu_{min}; C_{Rd,c}k(100\rho_l f_{ck})^{\frac{1}{3}}\right] bd$$

where:

$$k = 1 + \sqrt{\frac{200}{d}} \leq 2.0$$

$$\rho_l = \frac{A_{sl}}{bd} \leq 0.02$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength $V_{Rd,max}$ is checked (Art. 6.2.3(3)).

$$V_{Rd,max} = bz\nu f_{cd}/(\cot\theta + \tan\theta)$$

Next, the necessary reinforcement area is given by (Art. 6.2.3(3)):

$$A_{sw,l} = \frac{V_{Ed}}{z\, f_{ywd}\, \cot\theta}$$

Standard values of coefficients ν , ν_{max} , $cot\ \theta_{min}$, $cot\ \theta_{max}$ are built-in the program - these values can also be input by the user depending on the **selected National annex**.

Verification of Circular RC Cross Section

The program verifies a reinforced concrete pile using the method of limit deformation. The maximum allowable strain of concrete in compression is 0.002 - 0.0035. Concrete strength $\eta * f_{cd}$ is reduced by ten percent due to the shape of cross-section (Art. 3.1.7).

The degree of reinforcement is checked using the formula:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

- **Pile** (Art. 9.8.5)

$$\rho = \frac{4A_s}{\pi d^2}$$

$A_c \leq 0.5m^2$	$\rho_{min} = 0.005$
$0.5m^2 < A_c \leq 1m^2$	$\rho_{min} = 0.0025m^2/A_c$
$A_c \geq 1m^2$	$\rho_{min} = 0.0025$

where: A_c - cross-section area of the pile

$$\rho_{max} = 0.04$$

- **Column** - check for dominant compression (Art. 9.5.2)

$$\rho = \frac{4A_s}{\pi d^2}$$

$$\rho_{min} = \mathrm{Max}\left(0.002; \frac{0.1N_{Ed}}{f_{yd}A_s}\right)$$

$$\rho_{max} = 0.04$$

- **Beam** - check for dominant bending (Art. 9.2.1.1)

$$\rho = 0.5\frac{4A_s}{\pi d^2}$$

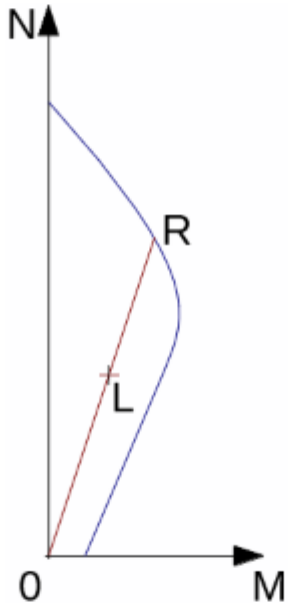
$$\rho_{min} = \mathrm{Max}\left(0.0013; 0.26\frac{f_{ctm}}{f_{yk}}\right)$$

$$\rho_{max} = 0.04$$

where: d - pile diameter

A_s - cross-sectional area of reinforcement

Standard values of coefficients ρ_{min} , ρ_{max} are built-in the program - these values can also be input by the user depending on the **selected National annex**.



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L| / |0R|$. Where L is load and R is strength with prescribed eccentricity.

Shear

First, the program computes the ultimate shear strength of concrete $V_{Rd,c}$ (Art. 6.2.2(1)). Formulas are from Art. 6.2.2(1), where the section width (b_w) is replaced by 0.88d and effective depth (d) is replaced 0.8d.

$$V_{Rd,c} = \mathrm{Max}\left[\nu_{min}; C_{Rd,c}k(100\rho_l f_{ck})^{\frac{1}{3}}\right] 0.704d^2$$

where:

$$k = 1 + \sqrt{\frac{200}{0.8d}} \leq 2.0$$

$$\rho_l = 0.33\frac{A_{sl}}{0.25\pi d^2} \leq 0.02$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength $V_{Rd,max}$ and strength of reinforced section $V_{Rd,s}$ are checked (Art. 6.2.3(3)).

$$V_{Rd,max} = (0.88d)(0.72d)\nu f_{cd}/(\cot\theta + \tan\theta)$$

$$V_{Rd,s} = \frac{A_{sw}}{s}0.72df_{ywd}\cot\theta$$

$$V_{Rd} = \mathrm{Max}(V_{Rdc}; \mathrm{Min}(V_{Rd,max}; V_{Rd,s}))$$

Standard values of coefficients ν , ν_{max} , $\cot\theta_{min}$, $\cot\theta_{max}$ are built-in the program - these values can also be input by the user depending on the **selected National annex.**

Verification of Spread Footing for Punching Shear

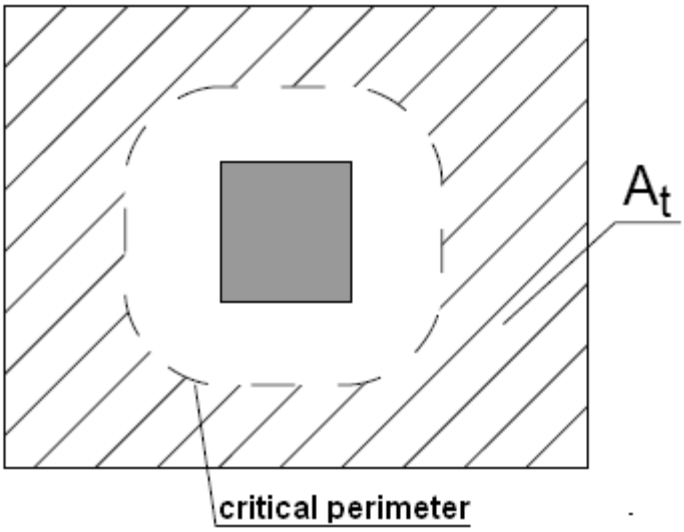
It is loaded by the prescribed moments M_{Edx} , M_{Edy} and by the shear force V_{Ed} provided by:

$$V_{Ed} = \frac{VA_t}{A}$$

where: A - area of footing

V - assigned vertical force developed in column

A_t - hatched area in fig.



Dimensioning of shear reinforcement area A_t

The program constructs **control sections** at **distances "a "** from $0{,}5d$ to $2d$ in case of **footing without shear reinforcement**. In case of **reinforced footing**, the distances are from $0{,}5d$ to $4d$, where d **is the effective depth of footing**. The shear reinforcement is considered in control sections, which are in the distance of less than $2d$ from the column. The control sections are considered in intervals of $0{,}25d$.

The load stress v_{Ed} in each control section is found using 6.4.3 (3),

$$v_{Ed} = \beta \frac{V_{Ed}}{u_i d}$$

$$\beta = 1 + k \frac{M_{Ed}}{V_{Ed}} \frac{u_1}{W_1}$$

$$v_{Rd,c} = \mathrm{Max}(C_{Rd,c} k (100 \rho_l f_{ck})^{1/3}; v_{min}) 2d/a$$

the punching shear resistance of footing without shear reinforcement $v_{Rd,c}$ follows from 6.4.4 (2)

$$k = 1 + \sqrt{\frac{200}{d}} \leq 2.0$$

$$\rho_l = \sqrt{\rho_{lx} \rho_{ly}} \leq 0.02$$

$$C_{Rd,c} = 0.18/\gamma_c$$

and if necessary the punching shear resistance of reinforced footing $v_{Rd,cs}$ is given by 6.4.5 (1).

$$v_{Rd,cs} = \mathrm{Min}(0.75 v_{Rd,c} + 0.75 \frac{A_{sw} f_{ywd,eff}}{ud}; k_{max} v_{Rd,c})$$

$$f_{ywd,eff} = 250 + 0.25d \leq f_{ywd}$$

Furthermore, the **compression chord resistance** at the column perimeter $v_{Rd,max}$ is calculated according to 6.4.5 (3). $v_{Rd,max}$ depends on column dimensions and the footing thickness.

$$v_{Rd,max} = 0.4 \nu f_{cd}$$

For $v_{Ed} < v_{Rd,c}$ no shear reinforcement is needed.

For $v_{Ed} > v_{Rd,c}$ and $v_{Ed} < v_{Rd,max}$ the shear reinforcement must be introduced.

For $v_{Ed} > v_{Rd,max}$ the shear reinforcement cannot be designed. It is therefore necessary to increase the cross-section height.

The control section ratio of load and resistance with the highest utilization is considered as critical and marked in the program.

Standard values of coefficients ν , v_{min} are built-in the program - these values can also be input by the user depending on the **selected National annex**.

Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment M_{Ed}. The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross-section. First, the program determines the location of the neutral axis as (Art. 3.1.7, Art. 6.1):

$$x = \frac{d - \sqrt{d^2 - \frac{M_{Ed}}{0{,}5\,b.\eta.f_{cd}}}}{\lambda}$$

Providing the location of the neutral axis is less than the allowable one ($x < x_{max}$), the program determines the area of tensile reinforcement A_{st} from the expression:

$$A_{st} = \frac{\lambda.\eta.b.x.f_{cd}}{f_{yd}}$$

Providing the location of the neutral axis is greater than the allowable one ($x > x_{max}$), the program determines the areas of both compressive A_{sc} and tensile A_{st} reinforcement from the expressions:

$$A_{sc} = \frac{M - F_{c,max}\left(d - 0{,}5.\lambda.x_{max}\right)}{f_{yd}.z}$$

$$A_{st} = \frac{F_{c,max} + A_{sc}.f_{yd}}{f_{yd}}$$

$$F_{c.max} = \lambda.\eta.b.x_{max}.f_{cd}$$

The limit location of the neutral axis is found from (Art. 5.6.3(2)):

$x_{max} = 0.45d$ for concrete C40/45 and lower

$x_{max} = 0.35d$ for concrete C45/50 and higher

The computed degree of reinforcement is checked using the following expressions (Art. 9.3.1.1):

$$\rho_{min} \leq \rho \leq \rho_{max}$$

$$\rho = \frac{A_s}{b.d}$$

$$\rho_{min} = Max\left(0{,}0013\ ;\ 0{,}26.\frac{f_{ctm}}{f_{yk}}\right)$$

$$\rho_{max} = 0{,}04$$

Standard values of coefficients ρ_{min}, ρ_{max} are built-in the program - these values can also be input by the user depending on the **selected National annex**.

If the maximum degree of total reinforcement ρ_{max} is exceeded, the program informs the user that the longitudinal reinforcement cannot be designed for a given cross-section.

Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and bends, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force transmitted by concrete $V_{Rd,c}$ (Art. 6.2.2(1)), and the maximal allowable shear force $V_{Rd,max}$ (Art. 6.2.3(3)).

$$V_{Rd,c} = \mathrm{Max}\left[\nu_{min}; C_{Rd,c}k(100\rho_l f_{ck})^{\frac{1}{3}}\right] d$$

where:

$$k = 1 + \sqrt{\frac{200}{d}} \leq 2.0$$

$$\rho_l = \frac{A_{sl}}{bd} \leq 0.02$$

$$V_{Rd,max} = z\nu f_{cd}/(\cot\theta + \tan\theta)$$

As for stirrups the necessary reinforcement area is given by (Art. 6.2.3(3)):

$$A_{sw,l} = \frac{V_{Ed}}{z\, f_{ywd}\, \cot\theta}$$

As for bends the necessary reinforcement area is given by (Art. 6.2.3(4)):

$$A_{sw,l} = \frac{V_{Ed}}{z\, f_{ywd} \sin\alpha(\cot\theta + \cot\alpha)}$$

Standard values of coefficients ν , ν_{min} are built-in the program - these values can also be input by the user depending on the **selected National annex**.

Verification of Crack Width

Crack width is evaluated according to chapter 7.3.4 of standard.

First, the maximal tension stress in concrete is calculated at the ideal section. If the stress is less than concrete tension strength f_{ctm} than cracks don't develop.

If not fulfilled then the crack width is determined according to:

$$w_k = s_{r,max}(\varepsilon_{sm} - \varepsilon_{cm})$$

where

$$\varepsilon_{sm} - \varepsilon_{cm} = \frac{\sigma - k_t \frac{f_{ctm}}{\rho_{p,eff}}(1 + \alpha_e \rho_{p,eff})}{E_s} \geq 0.6\frac{\sigma_s}{E_s}$$

where σ_s is stress in tensile reinforcement determined at the ideal section with crack

$$\alpha_e = \frac{E_s}{E_{cm}}$$

$$\rho_{p,eff} = \frac{A_s}{A_{c,eff}}$$

$$A_{c,eff} = b \times \mathrm{Min}\,(2.5(h-d), (h-x)/3, h/2)$$

$$k_t = 0.4$$

If the distance of the reinforcement bar is less or equal than $5(c+\phi/2)$:

$$s_{r,max} = k_3 c + k_1 k_2 k_4 \o / \rho_{p,eff}$$

where: k_1 = 0,8

k_2 = 0,5

k_3 = 3,4

k_4 = 0,425

c - cover

ϕ - reinforcement bar diameter

If the distance of reinforcement bars is greater than $5(c+\phi/2)$:

$$s_{r,max} = 1.3(h-x)$$

CSN 73 1201 R

This help contains the following methods:

- Materials, coefficients, notation
- Verification of rectangular cross-section made from plain concrete
- Verification of rectangular RC cross-section
- Verification of circular RC cross-section
- Verification of spread footing for punching shear
- Design of longitudinal reinforcement for slabs
- Design of shear reinforcement for slabs

Materials, Coefficients, Notation

The following notation for material parameters is used:

R_{bd} - design compressive strength of concrete

R_{btd} - design tensile strength of concrete

γ_u - coefficient of the shape of cross-section

z - lever arm (arm of internal forces)

Coefficient γ_u is given by equation (Art. 5.2.2):

$$\gamma_u = Max\left(1 - \frac{20}{1000.h + 50}\ ; 0{,}85\right)$$

The most common notation for geometrical parameters:

b - cross-section width

h - cross-section depth

h_e - effective depth of the cross-section

z - lever arm (arm of internal forces)

Verification of Rectangular Cross Section Made of Plain Concrete

The cross-section is rectangular, loaded by the bending moment M, normal force N (applied in the cross-section centroid), and by the shear force Q. The cross-section bearing capacity subjected to bending moment is given by (Art. 5.2.5):

$$M_u = \frac{b.h^2}{6}.R_{btd}.\gamma_u$$

The shear strength is provided by (Art. 5.3.3, Appendix 9):

$$Q_u = \frac{1}{3}.\kappa_h.\kappa_n.b.h.R_{btd}$$

$$\kappa_h = Max\left(1\ ; 1{,}4 - \frac{2}{3}h\right)$$

$$\kappa_n = Min\left(2\ ; 1 + 0{,}2.\frac{N}{b.h.R_{btd}}\right)$$

Strength of concrete cross-section subject to the combination of bending moment and normal force is derived from the following expressions depending on the normal force eccentricity e (Art. 5.2.5):

for:

$$e < 0{,}9.a_{gc} \Rightarrow N_u = b.x.R_{bd}.\gamma_u$$

$$e > 0{,}9.a_{gc} \Rightarrow N_u = \frac{b.h.R_{btd}.\gamma_u}{\frac{6.e}{h} - 1}$$

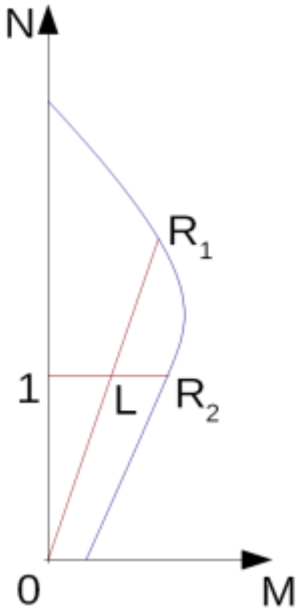
$$x_u = h - 2.e$$

$$e = \frac{abs(M)}{N}$$

$$a_{gc} = \frac{h}{2}$$

The ultimate bearing capacity is checked using the following formula (Art. 5.2.5.5):

$$N_{lim} = 0{,}8.b.x_u.R_{btd}.\gamma_u$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R_1|$ or $|1L|/|1R_2|$. Where L is load, R_1 is strength with prescribed eccentricity, and R_2 is strength with prescribed normal force.

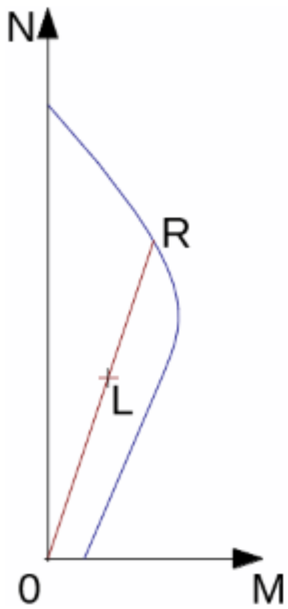
Verification of Rectangular RC Cross Section

The cross-section is rectangular, unilaterally reinforced, and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation (Art. 5.2.8). The maximum allowable strain of concrete in compression is 0.0025. Compression reinforcement is not taken into account.

The computed degree of reinforcement is checked using the following expressions (Art. 3.1.4.3, Art. 3.1.4.6):

$$\mu_{st,min} = \frac{R_{btd}}{3.R_{sd}} < \mu_{st} < 0{,}03 = \mu_{st,max}$$

$$\mu_{st} = \frac{A_s}{b.h}$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R|$. Where L is load and R is strength with prescribed eccentricity.

Bending without normal force

The cross-section is rectangular, reinforced on one side, and loaded by the bending moment M_d.

The ultimate moment is provided by (Art. 5.2.7):

$$M_u = b.x_u R_{bd} \cdot \left(h_e - \frac{x_u}{2} \right) \cdot \gamma_u$$

$$x_u = \frac{A_s . R_{sd}}{b.R_{bd}}$$

The program further checks whether the location of the neutral axis x is less than the limit location of neutral axis x_{lim} given by (Art. 5.2.7.1):

$$x_{lim} = Min \left(0{,}533; \frac{1}{1{,}25 + \frac{R_{sd}}{420}} \right)$$

Shear

First, the program computes the ultimate shear strength of concrete Q_{bu} (Art. 5.3.3, Appendix 9).

$$Q_{bu} = \frac{1}{3} b\, h \kappa_q R_{btd}$$

where:

for:	$h \geq 0.3m$	is: $\kappa_q = 1.25$
for:	$h > 0.15m$	je $\kappa_q = 1.50$
for:	$h < 0.15m$	je $\kappa_q = 1.60$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength Q_{max} is checked (Art. 5.3.2.1).

$$Q_{\max} = \frac{1}{3} b\, h\, Min \left(R_{bd} \,; 18 \right)$$

Next, the necessary reinforcement area is given by (Art. 5.3.4):

$$A_b = \frac{Q - Q_{bu}}{R_{swd}\, c} b$$

where (Art. 5.3.5):

$$c = Max \left(\frac{1{,}2.b.R_{btd}.h_e^2}{Q - Q_{bu}} \,; z \right)$$

The magnitude of c is bounded by the following expression:

$$c < 0{,}18 \frac{R_{bd}.h}{\kappa_q . R_{btd}}$$

Verification of Circular RC Cross Section

The program verifies a reinforced concrete pile using the method of limit deformation (Art. 5.2.8). The maximum allowable strain of concrete in compression is 0.0025. The degree of reinforcement is checked using the formula:

- **Column** - check for dominant compression (Art. 3.1.4.3, Art. 3.1.4.6)

$$\mu_{st,min} = 0.0008 \leq \mu_{st} \leq 0.04 = \mu_{st,max}$$

$$\mu_{st} = \frac{4A_s}{\pi d^2}$$

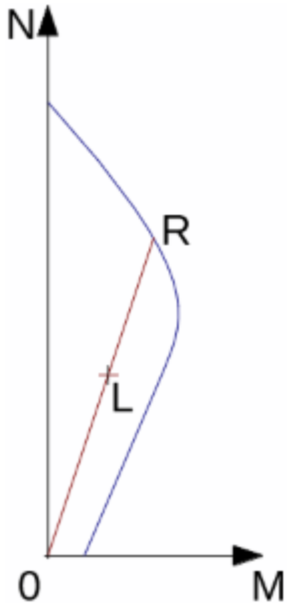
- **Beam** - check for dominant bending

$$\mu_{st,min} = \frac{R_{btd}}{3R_{sd}} \leq \mu_{st} \leq 0.03 = \mu_{st,max}$$

$$\mu_{st} = 0.5\frac{4A_s}{\pi d^2}$$

where: d - pile diameter

A_s - reinforcement area



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R|$. Where L is load and R is strength with prescribed eccentricity.

Shear

First, the program computes the ultimate shear strength of concrete Q_{bu} (Art. 5.3.3, Appendix 9).

$$Q_{bu} = \frac{1}{3}(0.88d)(0.88d)\kappa_q R_{btd}$$

where:

for: $0.88d \geq 0.3m$ is: $\kappa_q = 1.25$

for: $0.88d > 0.15m$ is: $\kappa_q = 1.50$

for: $0.88d < 0.15m$ is: $\kappa_q = 1.60$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength Q_{max} and strength of reinforced section Q_u are checked (Art. 5.3.2.1).

$$Q_{max} = \frac{1}{3}(0.88d)(0.88d)\mathrm{Min}\,(R_{bd}; 18)$$

$$Q_u = Q_{bu} + A_b R_{swd} c$$

where (Art. 5.3.5):

$$c = \mathrm{Max}\left(\frac{1.2(0.88d)R_{btd}(0.8d)^2}{Q - Q_{bu}}; 0.9(0.8d)\right)$$

The magnitude of c is bounded by the following expression:

$$c \leq 0.18 \frac{R_{bd}(0.88d)}{\kappa_q R_{btd}}$$

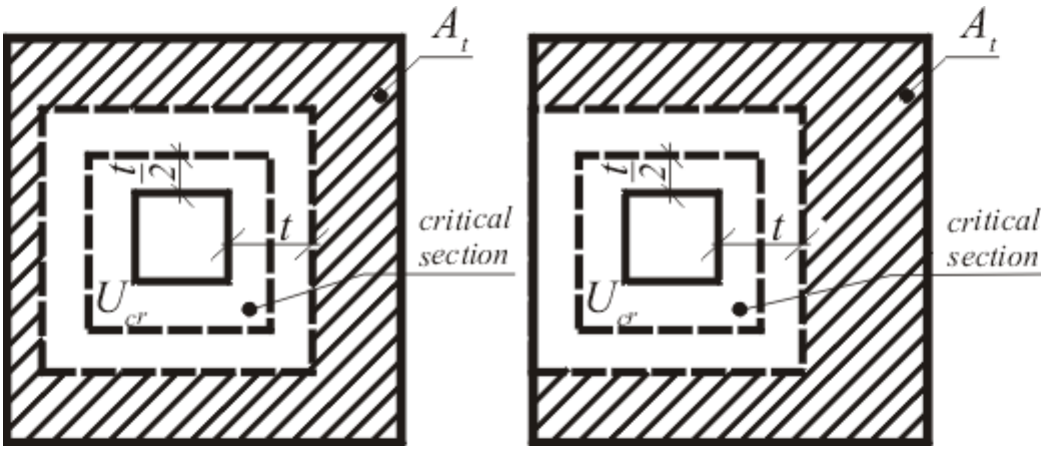
Verification of Spread Footing for Punching Shear

The program allows us to verify spread footing for punching shear or for the design of shear reinforcement. The critical section loaded in shear U_{cr} is distant from the column edge by one half of the footing thickness. It is loaded by the prescribed moments M_x, M_y and by the shear force Q_r provided by:

$$Q_{d\,max} = \frac{Q.A_t}{A}$$

where:

- A - area of footing
- Q - assigned vertical force developed in column
- A_t - hatched area in fig.



Dimensioning of shear reinforcement area A_t

The program computes the maximal shear force Q_{dmax} developed in the critical section, the shear force transmitted by concrete with no shear reinforcement Q_{bu}, and the maximal allowable force Q_{max}:

$$Q_{bu} = 0{,}42.\kappa_h.\kappa_n.\kappa_s.t.R_{btd}$$

$$Q_{max} = 2.Q_{bu}$$

where for: $\mu_s > \mu_{min}$ is: $\kappa_s = Min[1 + 50.(\mu_s - \mu_{min}) ; 1{,}5]$ or else: $\kappa_s = 1$

$$\kappa_h = Max\left[1{,}4 - \frac{2}{3}.h ; 1\right]$$

$$\kappa_n = 1$$

For $Q_{dmax} < Q_{bu}$ no shear reinforcement is needed.

For $Q_{dmax} > Q_{bu}$ and $Q_{dmax} < Q_{max}$ the shear reinforcement must be introduced. The ultimate shear force is given by:

$$Q_u = Q_{su} + Q_{bu}$$

$$Q_{su} = \frac{A_s.R_{sd}.\sin\alpha}{U_{cr}}$$

where:

- U_{cr} - critical cross-section span
- α - is angle of bends
- A_s - overall area of bends in footing

For $Q_{dmax} > Q_{max}$ the shear reinforcement cannot be designed. It is therefore necessary to increase the cross-section height.

Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment M. The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross-section. First, the program determines the location of neutral axis as:

$$x = \frac{h_e - \sqrt{h_e^2 - \frac{M}{0{,}5.b.\gamma_u.R_{bd}}}}{0{,}8}$$

Providing the location of the neutral axis is less than the allowable one ($x < x_{lim}$), the program determines the area of tensile reinforcement A_{st} from the expression:

$$A_{st} = \frac{0{,}8.b.x.R_{bd}}{R_{sd}}$$

Providing the location of the neutral axis is greater than the allowable one ($x > x_{lim}$), the program determines the areas of both compressive A_{sc} and tensile A_{st} reinforcement from the expressions:

$$A_{sc} = \frac{\frac{M}{\gamma_u} - N_{max}\left(h_e - 0{,}5.0{,}8.x_{lim}\right)}{R_{sd}.z}$$

$$A_{st} = \frac{N_{max} + A_{sc}.R_{scd}}{R_{sd}}$$

$$N_{max} = x_{lim}.0{,}8.b.R_{bd}$$

The limit location of the neutral axis is found from:

$$x_{max} = Min\left(0{,}533\ ;\ \frac{1}{1{,}25 + \frac{R_{sd}}{420}}\right).h_e$$

The computed degree of reinforcement is checked using the following expressions:

$$\mu_{st,min} = \frac{R_{btd}}{3.R_{sd}} < \mu_{st} < 0{,}03 = \mu_{st,max}$$

$$\mu_{st} = \frac{A_s}{b.h}$$

If the maximum degree of tensile reinforcement ($\mu_{st,max} =$ 0.03) or total reinforcement ($\mu_{max} =$ 0.04), respectively, is exceeded, the program informs the user that the longitudinal reinforcement cannot be designed for a given cross-section.

Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and bends, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force transmitted by concrete Q_{bu} and the maximum allowable shear force Q_{max}.

$$Q_{bu} = \frac{1}{3}b.h.\kappa_q.R_{btd}$$

$$Q_{max} = \frac{1}{3}b.h.Min\left(R_{bd}\ ;18\right)$$

where:

for:	$h \geq$ 0.3m	is: $\kappa_q =$ 1.25
for:	$h >$ 0.15m	je $\kappa_q =$ 1.50
for:	$h <$ 0.15m	je $\kappa_q =$ 1.60

As for stirrups, the necessary reinforcement area is given by:

$$A_b = \frac{Q - Q_{bu}}{R_{swd}.c}$$

As for bends the necessary reinforcement area is given by:

$$A_b = \frac{Q - Q_{bu}}{R_{swd}\left(c.\sin\alpha + 0{,}8.h_e.\cos\alpha\right)}$$

where:

$$c = Max\left(\frac{1{,}2.b.R_{btd}.h_e^2}{Q - Q_{bu}}\ ;z\right)$$

The magnitude of c is bounded by the following expression:

$$c < 0{,}18\frac{R_{bd}.h}{\kappa_q.R_{btd}}$$

CSN 73 6206

When selecting "**CSN 73 6206**", frame **"Analysis methods"**, the verification analysis of decisive joints is performed according to the standard CSN 73 6206 "Design of concrete and steel reinforced concrete bridge structures", including changes a-10/1989 a Z2/1994. The program allows us to verify cross sections from plain concrete or single-ended steel reinforced concrete. All calculations related to concrete are carried out using the **theory of allowable stresses**.

The main difference when compared to other standards appears in the dimensioning of concrete joints where the earth pressure is computed **always without reduction of input parameters** independently of the input in the frame "Settings".

When performing the verification analysis of cross sections made either from plain or steel reinforced concrete it is possible input the **coefficient of allowable stress** according to art. 47 **CSN 73 6206** to increase the material allowable stress.

The following joints can be verified by the program:

Abutment stem - foundation, construction joint - the cross-section can be made either from plain or steel reinforced concrete. The joint is verified for the load due to normal force and bending moment. The allowable stresses of concrete, steel and concrete in concentric pressure are checked. In case of reinforced concrete the program also checks the degree of reinforcement, cross sections from plain concrete are then checked for overturning ($h/2e <$ 1.35) and translation ($N*f <$ 1.5); friction concrete-concrete is assumed as $f =$ 0.5).

Closure wall - bearing block - the cross-section is verified for the load due to normal force and bending moment. The steel reinforced concrete cross-section is always assumed. The allowable stresses of concrete and steel and the degree of reinforcement are checked.

Wing wall - abutment - the joint can be made either from concrete or steel reinforced concrete. The allowable stresses of concrete, steel and concrete in concentric pressure are checked. In case of reinforced concrete the program also checks the degree of reinforcement.

Front jump of abutment foundation - the front jump of abutment is verified according to its projection. In case of jump projection $v < 0.5h_z$ (h_z is the height of foundation jump) the program checks the magnitude of stress in principal tension due to forces developed in the above-foundation joint. The stress is determined as:

$$\sigma = 0{,}15.\frac{N}{d - 2.\frac{M}{N}}$$

where:

where:	d	-	width of above-foundation joint
	M,N	-	moment and normal force in above-foundation joint

In case of jump projection $v > 0.5h_z$ the jump is analyzed as cantilever bended by the reaction (stress) of foundation soil. The joint can be made either from concrete or steel reinforced concrete. The allowable stresses of concrete, steel and concrete in concentric pressure are checked. In case of reinforced concrete the program also checks the degree of reinforcement.

PN-B-03264:2002

This help contains the following methods:

Materials, Coefficients, Notation

The following notation for material parameters is used:

f_{ck} - characteristic compressive strength of concrete

f_{cd} - design compressive strength of concrete

f_{ctk} - characteristic tensile strength of concrete

f_{ctd} - design tensile strength of concrete

f_{yk} - characteristic tensile strength of steel bar

f_{yd} - design tensile strength of steel bar

f_{ctm} - mean tensile strength of steel bar

$$f_{cd} = \frac{f_{ck}}{\gamma_c}.\alpha_{cc}$$

$$f_{ctd} = \frac{0,7.f_{ctm}}{\gamma_c}.\alpha_{ct}$$

$$E_{cm} = 11000.(f_{ck} + 8)^{0,3}$$

$$f_{ctm} = 0,3.(f_{ck})^{\frac{2}{3}}$$

where:

$\alpha_{cc} = 1$

$\alpha_{ct} = 1$

$\gamma_c = 1.5$ - for reinforced concrete structures

$\gamma_c = 1.8$ - for concrete strustures

The most common notation for geometrical parameters:

where:

b - cross-section width

h - cross-section depth

d - effective depth of cross-section

z - lever arm (arm of internal forces)

Verification of Rectangular Cross Section Made of Plain Concrete

The cross-section is rectangular, loaded by the bending moment M_{Sd}, normal force N_{Sd} (applied in the cross-section centroid), and by the shear force V_{Sd}. The cross-section bearing capacity subjected to bending moment is given by:

$$M_{Rd} = \frac{b.h^2}{6}.f_{ctd}$$

The shear strength is provided by:

$$V_{Rd,1} = 0,35.f_{ctd}.k.1,2.b.d$$

where:

$$k = Max(1,6 - d\ ;1)$$

Strength of concrete cross-section subject to the combination of bending moment and normal force is derived from the following expressions depending on the normal force eccentricity e :

As the greater of:

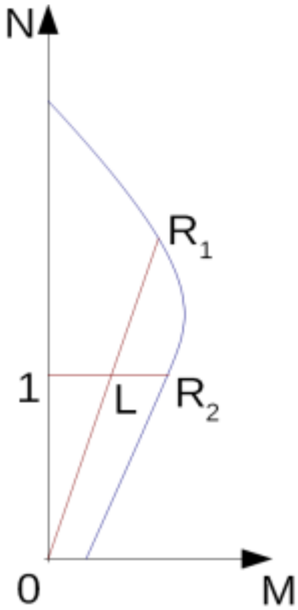
$$N_{Rd} = b.x.f_{cd}$$

$$N_{Rd} = Min\left(\frac{b.h.f_{ctd}}{\frac{6.e}{h} - 1} ; \frac{b.h.f_{cd}}{\frac{6.e}{h} + 1} \right)$$

where:

$$x = h - 2.e$$

$$e = abs\left(\frac{M}{N} \right)$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L| / |0R_1|$ or $|1L| / |1R_2|$. Where L is load, R_1 is strength with prescribed eccentricity, and R_2 is strength with prescribed normal force.

Verification of Rectangular RC Cross Section

The cross-section is rectangular, unilaterally reinforced, and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation. The maximum allowable strain of concrete in compression is 0.002 - 0.0035. Compression reinforcement is not taken into account.

The computed degree of reinforcement is checked using the following expressions:

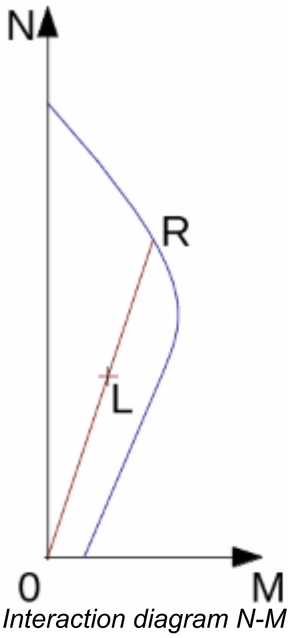
$$\rho_{min} \le \rho \le \rho_{max}$$

where:

$$\rho = \frac{A_s}{b.d}$$

$$\rho_{min} = Max\left(0{,}0013 ; 0{,}26.\frac{f_{ctm}}{f_{yk}} \right)$$

$$\rho_{max} = 0{,}04$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L| / |0R|$. Where L is load and R is strength with prescribed eccentricity.

Bending without normal force

The cross-section is rectangular, reinforced on one side and loaded by the bending moment M_{Sd} .

The permissible moment for a given area of reinforcements A_s reads:

$$M_{rd} = 0{,}8.x.b.f_{cd}.(d - 0{,}4.x)$$

$$x = \frac{A_s \cdot f_{yd}}{0{,}8.b.f_{cd}}$$

The program further checks whether the location of the neutral axis x is less than the limit location of neutral axis x_{lim} given by:

$$x_{max} = \frac{\varepsilon_{cu}}{\varepsilon_{cu} + \varepsilon_{yd}}.d$$

where:

$$\varepsilon_{cv} = 0{,}0035$$

$$\varepsilon_{yd} = \frac{f_{yd}}{E_s}$$

Shear

First, the program computes the ultimate shear strength of concrete V_{Rd1} .

$$V_{Rd1} = 0{,}35\, k\, f_{ctd}\, (1{,}2 + 40\, \rho_L)\, d\, b$$

where:

$$k = 1{,}6 - d$$

$$\rho_L = \frac{A_{sL}}{b\, d} \leq 0{,}01$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength V_{Rd2} is checked.

$$V_{Rd2} = 0{,}5\, v\, f_{cd}\, z\, b$$

where:

$$v = 0{,}6\left(1 - \frac{f_{ck}}{250}\right)$$

Next, the necessary reinforcement area is given by:

$$A_{sw1} = \frac{V_{Ed}}{f_{ywd1}\, z} b$$

Verification of Circular RC Cross Section

The program verifies a reinforced concrete pile using the method of limit deformation. The maximum allowable strain of concrete in compression is 0.002 - 0.0035. The degree of reinforcement is checked using the formula:

$$\rho_{min} \le \rho \le \rho_{max}$$

- **Column** - check for dominant compression

$$\rho = \frac{4.A_s}{\pi.d^2}$$

$$\rho_{min} = Max\left(0{,}003\ ;\ \frac{0{,}15\, N_{Ed}}{f_{yd}\, A_s}\right)$$

$$\rho_{max} = 0{,}04$$

- **Beam** - check for dominant bending

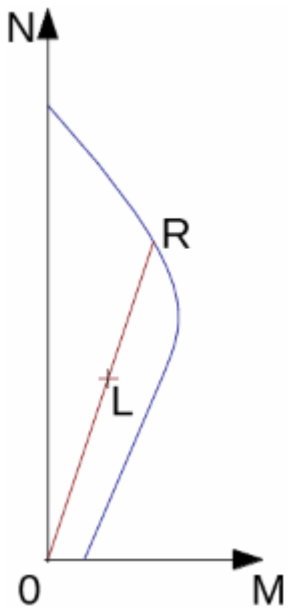
$$\rho = 0{,}5\, \frac{4\, A_s}{\pi\, d^2}$$

$$\rho_{min} = Max\left(0{,}0013\ ;\ 0{,}26.\frac{f_{ctm}}{f_{yk}}\right)$$

$$\rho_{max} = 0{,}04$$

where: d - pile diameter

A_s - reinforcement area



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R|$. Where L is load and R is strength with prescribed eccentricity.

Verification of Spread Footing for Punching Shear

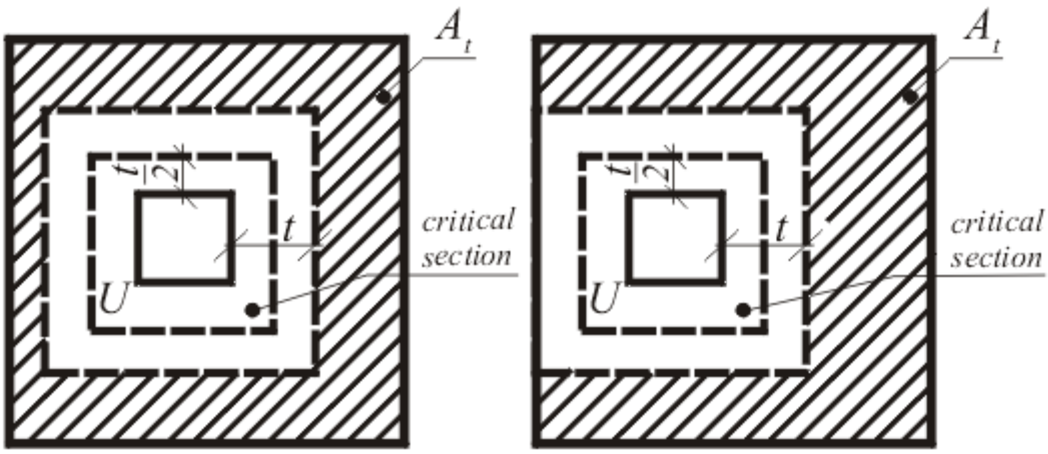
The critical section loaded in shear u is distant from the column edge by one half of the footing thickness. It is loaded by the prescribed moments M_x, M_y and by the shear force N_{Sd} provided by:

$$N_{Sd} = \frac{Q.A_t}{A}$$

where: A - area of footing

V - assigned vertical force developed in column

A_t - hatched area in fig.



Dimensioning of shear reinforcement area A_t

The program computes the maximal shear force N_{Sd} developed in the critical section, the shear force transmitted by concrete with no shear reinforcement N_{Rd1} and the maximal allowable force $N_{Rd,max}$:

$$N_{Rd} = f_{ctd}.d$$
$$N_{Rd,max} = 1{,}4.N_{Rd}$$

For $N_{Sd} < N_{Rd}$ no shear reinforcement is needed.

For $N_{Sd} > N_{Rd}$ and $N_{Sd} < N_{Rd,max}$ the shear reinforcement must be introduced. The ultimate shear force is given by:

$$N_{Rd} = \frac{\sum A_{sw}.f_{yd}.\sin\alpha}{u}$$

where: u - critical cross-section span

α - is angle of bends

A_{sw} - overall area of bends in footing

For $N_{Sd} > N_{Rd,max}$ the shear reinforcement cannot be designed. It is therefore necessary to increase the cross-section height.

Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment M_{Sd}. The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross section. First, the program determines the location of the neutral axis as:

$$x = \frac{d - \sqrt{d^2 - \frac{M_{sd}}{0{,}5.b.f_{cd}}}}{0{,}8}$$

Providing the location of the neutral axis is less than the allowable one ($x < x_{max}$), the program determines the area of tensile reinforcement A_{st} from the expression:

$$A_{st} = \frac{0{,}8.b.x.f_{cd}}{f_{yd}}$$

Providing the location of the neutral axis is greater than the allowable one ($x > x_{max}$), the program determines the areas of both compressive A_{sc} and tensile A_{st} reinforcement from the expressions:

$$A_{sc} = \frac{M - F_{c,max}(d - 0{,}4.x_{max})}{f_{yd}.z}$$

$$A_{st} = \frac{F_{c,max} + A_{sc}.f_{yd}}{f_{yd}}$$

$$F_{c,max} = 0{,}8.x_{max}.b.f_{cd}$$

The limit location of the neutral axis is found from:

$$x_{max} = \frac{\varepsilon_{cu}}{\varepsilon_{cu} + \varepsilon_{yd}}.d$$

where:

$$\varepsilon_{cu} = 0{,}0035$$

$$\varepsilon_{yd} = \frac{f_{yd}}{E_s}$$

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} \le \rho \le \rho_{max}$$

where:

$$\rho = \frac{A_s}{b.d}$$

$$\rho_{min} = Max\left(0{,}0013\ ;\ 0{,}26.\frac{f_{ctm}}{f_{yk}}\right)$$

$$\rho_{max} = 0{,}04$$

If the maximum degree of total reinforcement ρ_{max} is exceeded, the program informs the user that the longitudinal reinforcement cannot be designed for a given cross-section.

Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and bends, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force transmitted by concrete V_{Rd1} and the maximum allowable shear force V_{Rd2}.

$$V_{Rd1} = 0{,}35.k.f_{ctd}.(1{,}2 + 40.\rho_L).d$$

where:

$$k = 1{,}6 - d$$

$$\rho_L = \frac{A_{sL}}{b.d} \le 0{,}01$$

$$V_{Rd2} = 0{,}5.v.f_{cd}.z$$

where:

$$v = 0{,}6.\left(1 - \frac{f_{ck}}{250}\right)$$

As for stirrups the necessary reinforcement area is given by:

$$A_{sw1} = \frac{V_{Ed}}{f_{ywd1}.z}$$

As for bends the necessary reinforcement area is given by:

$$A_{sw2} = \frac{V_{Ed}}{f_{ywd2}.z.\sin\alpha.(1+\cot\alpha)}$$

BS 8110:1997

This help contains the following methods:

- Materials, Coefficients, Notation
- Verification of Rectangular cross-section made of Plain Concrete
- Verification of Rectangular RC Cross-section
- Verification of Circular RC Cross-section
- Verification of Spread Footing for Punching Shear
- Design of Longitudinal Reinforcement for Slabs
- Design of Shear Reinforcement for Slabs

Materials, Coefficients, Notation

The following notation for material parameters is used:

f_{cu} - characteristic strength of concrete

f_y - characteristic strength of reinforcement

f_{yd} - design strength of steel in tension

$$f_{yd} = \frac{f_y}{1{,}05}$$

The characteristic compressive strength of concrete is the basic input parameter given by the class of concrete.

The most common notation for geometrical parameters:

b - cross-section width

h - cross-section depth

d - effective depth of cross section

z - lever arm (arm of internal forces)

All computations are carried out according to the theory of limit states.

Verification of Rectangular Cross Section Made of Plain Concrete

The cross-section is rectangular, loaded by the bending moment M, normal force N (applied in the cross-section centroid), and by the shear force V.

Strength of concrete cross-section subject to the combination of bending moment and normal force with eccentricity e is derived from the following expressions:

$$N_u = x.0{,}45.f_{cu}$$

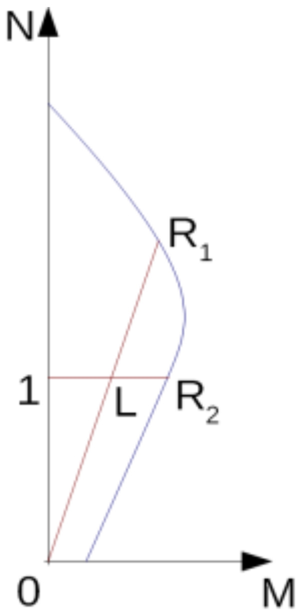
$$x = h - 2.e$$

$$e = Max\left(\frac{abs(M)}{N}\ ;0{,}05.h\ ;20mm\right)$$

The shear strength is provided by:

$$V_u = v_c.b.h$$

where: v_c - is the design value of shear stress in concrete for degree of longitudinal reinforcement $\rho = 0$ (see: Verification of spread footing for punching shear)



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R_1|$ or $|1L|/|1R_2|$. Where L is loading, R_1 is strength with prescribed, eccentricity and R_2 is strength with prescribed normal force.

Verification of Rectangular RC Cross Section

The cross-section is rectangular, unilaterally reinforced, and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation. The maximum allowable strain of concrete in compression is 0,002 - 0,0035. Compression reinforcement is not taken into account. Minimum eccentricity is applied:

$$e_0 = Min(0{,}05.h\ ;\ 20mm)$$

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} \le \rho \le \rho_{max}$$

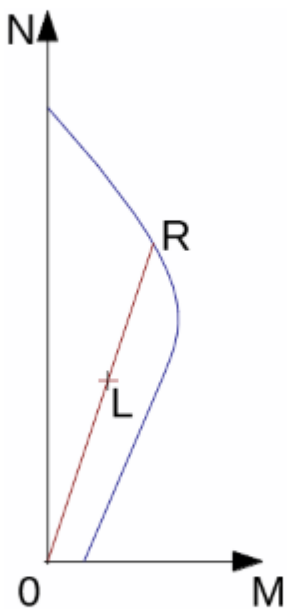
where:

$$\rho = \frac{A_s}{b.d}$$

$$\rho_{max} = 0{,}04$$

$\rho_{min} = 0{,}0013$ - for $f_y =$ 460 N/mm^2

$\rho_{min} = 0{,}0024$ - for $f_y =$ 250 N/mm^2



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L| / |0R|$. Where L is load and R is strength with prescribed eccentricity.

Bending without normal force

The cross-section is rectangular, reinforced on one side, and loaded by the bending moment M_u .

The permissible moment for a given area of reinforcements A_s reads:

$$M_u = b.F_c.(d - 0{,}45.x)$$
$$F_c = 0{,}402\, f_{cu}\, x$$
$$x = \frac{A_s\, f_{yd}}{b\, 0{,}402\, f_{cu}}$$

The program further checks whether the location of the neutral axis x is less than the limit location of neutral axis x_{max} given by:

$$x_{\max} = 0{,}5.d$$

Shear

First, the program computes the ultimate shear strength of concrete V_c .

$$V_c = v_c\, d\, b$$

where:

$$v_c = \frac{0{,}79 \left(\frac{100\, A_s}{b\, h}\right)^{\frac{1}{3}} \left(\frac{400}{d}\right)^{\frac{1}{4}}}{1{,}25}$$

The v_c values are for f_{cu} above 25 N/mm^2 multiplied by $(f_{cu} / 25)^{1/3}$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength V_{max} is checked.

$$V_{\max} = Min\left(5\ ;\ 0{,}8\sqrt{f_{cu}}\right) d\, b$$

Next, the necessary reinforcement area is given by:

$$A_{sl} = \frac{V - V_c}{0{,}95\, f_{yv}\,(d - d')}\, b$$

where:

$$f_{yv} \leq 460\, MPa$$

Verification of Circular RC Cross Section

The program verifies a reinforced concrete pile using the method of limit deformation. The maximum allowable strain of concrete in compression is 0,002 - 0,0035.

The degree of reinforcement is checked using the formula:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

- **Column** - check for dominant compression

$$\rho = \frac{4.A_s}{\pi.d^2}$$

ρ_{max} = 0,04

ρ_{min} = 0,0013 - for f_y = 460 N/mm^2

ρ_{min} = 0,0024 - for f_y = 250 N/mm^2

- **Beam** - check for dominant bending

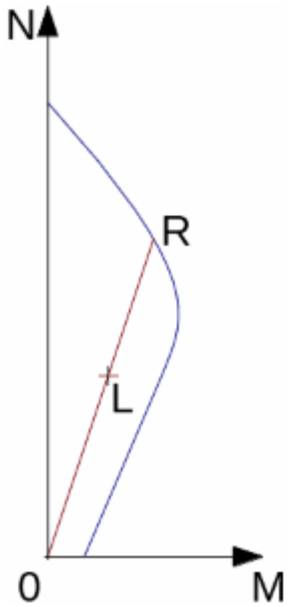
$$\rho = 0{,}5\,\frac{4\,A_s}{\pi\,d^2}$$

$\rho_{max} =$ 0,06

$\rho_{min} =$ 0,004

where: d - pile diameter

A_s - reinforcement area



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|\,/\,|0R|$. Where L is load and R is strength with prescribed eccentricity.

Verification of Spread Footing for Punching Shear

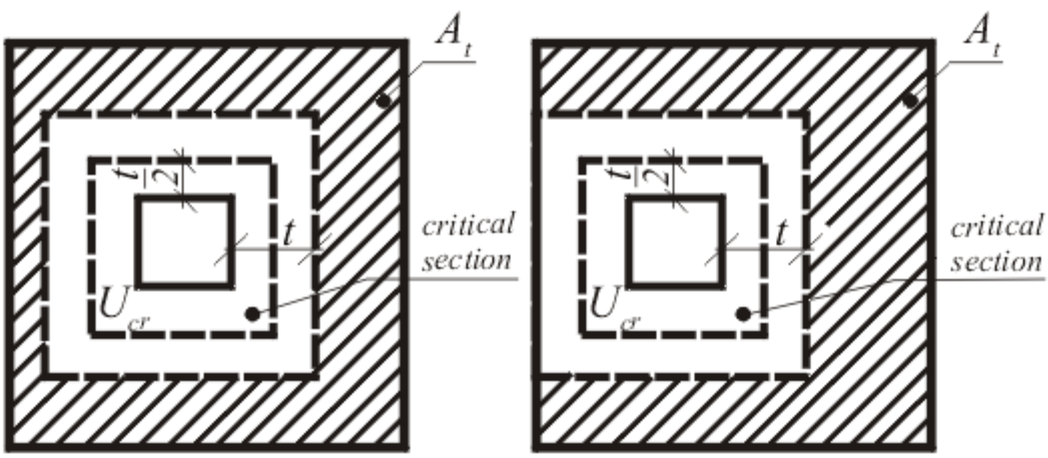
The critical section loaded in shear U_{cr} is distant from the column edge by one half of the footing thickness. It is loaded by the prescribed moments M_x , M_y and by the shear force V provided by:

$$V = \frac{Q.A_t}{A}$$

where: A - area of footing

Q - assigned vertical force developed in column

A_t - hatched area in fig.



Dimensioning of shear reinforcement area At

The program computes the maximum shear force V developed in the critical section, the shear force transmitted by concrete with no shear reinforcement V_c, and the maximal allowable force V_u:

$$V_c = v_c.d$$
$$V_u = v_u.d$$

where:

$$v_c = \frac{0{,}79.\left(\frac{100.A_s}{b.h}\right)^{\frac{1}{3}}.\left(\frac{400}{d}\right)^{\frac{1}{4}}}{1{,}25}$$

The v_c values are for f_{cu} above 25 N/mm^2 multiplied by $(f_{cu}/25)^{1/3}$

$v_u = 0{,}8.\sqrt{f_{cu}}$ or 5 N/m^2

where: v_u - is ultimate shear stress

For $V < V_c$ no shear reinforcement is needed.

For $V > V_c$ and $V_c < V_u$ it is necessary to design shear reinforcement. The permissible shear force is given by:

$$V_{rd} = V_c + V_{us}$$
$$V_{us} = \frac{\sum 0{,}95 A_{us}.f_{yv}.\sin\alpha}{u}$$
$$f_{yv} = Min(f_y\ ;460)$$

where:
- u - critical cross-section span
- α - angle of bends
- A_{us} - overall area of bends in footing

For $V > V_u$ the shear reinforcement cannot be designed. It is therefore necessary to increase the cross-section depth.

Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment M_d. The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross-section. First, the program determines the location of the neutral axis as:

$$x = \frac{d - \sqrt{d^2 - \frac{2.M_d}{0{,}402.b.f_{cu}}}}{0{,}9}$$

Providing the location of the neutral axis is less than the allowable one ($x < x_{max}$), the program determines the area of tensile reinforcement A_{st} from the expression:

$$A_{st} = \frac{0{,}402.b.f_{cu}.0{,}9.x}{f_{yd}}$$

Providing the location of the neutral axis is greater than the allowable one ($x > x_{max}$), the program determines the areas of both compressive A_{sc} and tensile A_{st} reinforcement from the expressions:

$$A_{sc} = \frac{M - F_{c,max}(d - 0{,}45.x_{max})}{f_{yd}.z}$$
$$A_{st} = \frac{F_{c,max} + A_{sc}.f_{yd}}{f_{yd}}$$
$$F_{c,max} = 0{,}9.x_{max}.0{,}67.\frac{f_{cu}}{1{,}5}$$

The limit location of the neutral axis is found from:

$$x_{u,lim} = 0{,}5$$

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

where:

$$\rho = \frac{A_s}{b.d}$$

$$\rho_{max} = 0{,}04$$

$$\rho_{min} = 0{,}0013 \quad \text{- for } f_y = 460\ N/mm^2$$

$$\rho_{min} = 0{,}0024 \quad \text{- for } f_y = 250\ N/mm^2$$

If the maximum degree of reinforcement ρ_{max} is exceeded, the program informs the user that the longitudinal reinforcement cannot be designed for a given cross-section.

Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and bends, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force transmitted by concrete V_c and the maximum allowable shear force V_{max}.

$$V_c = v_c.d$$

where:

$$v_c = \frac{0{,}79.\left(\frac{100.A_s}{b.h}\right)^{\frac{1}{3}}.\left(\frac{400}{d}\right)^{\frac{1}{4}}}{1{,}25}$$

The v_c values are for f_{cu} above 25 N/mm^2 multiplied by $(f_{cu}/25)^{1/3}$

$$V_{max} = Min\left(5\ ;\ 0{,}8.\sqrt{f_{cu}}\right)d$$

As for stirrups, the necessary reinforcement area is given by:

$$A_{sl} = \frac{V - V_c}{0{,}95.f_{yv}.(d - d')}$$

As for bends, the necessary reinforcement area is given by:

$$A_{sb} = \frac{V - V_c}{0{,}95.f_{yv}.(sin\,\beta + cos\,\beta).(d - d')}$$

where:

$$f_{yv} \leq 460MPa$$

IS 456

This help contains the following methods:

- Materials, coefficients, notation
- Verification of rectangular cross-section made of plain concrete
- Verification of rectangular RC cross-section
- Verification of circular RC cross-section
- Verification of spread footing for punching shear
- Design of longitudinal reinforcement for slabs
- Design of shear reinforcement for slabs

Materials, Coefficients, Notation

The following notation for material parameters is used:

f_{ck} - characteristic cube compressive strength of concrete

f_{cd} - design compressive strength of concrete

f_{ctk} - characteristic tensile strength of concrete

f_{ctd} - design tensile strength of concrete

f_y - characteristic strength of steel bar

f_{yd} - design tensile strength of steel bar

The characteristic compressive strength of concrete is the basic input parameter given by the class of concrete - it serves to derive the remaining coefficients of reliability.

$$f_{cd} = 0{,}67.\frac{f_{ck}}{1{,}5}$$

$$f_{ctk} = 0{,}7.\sqrt{f_{ck}}$$

$$f_{ctd} = \frac{f_{ctk}}{1{,}5}$$

$$E_c = 5000.\sqrt{f_{ck}}$$

$$f_{yd} = \frac{f_y}{1{,}15}$$

The most common notation for geometrical parameters:

b - cross-section width

h - cross-section depth

d - effective depth of cross section

z - lever arm (arm of internal forces)

All computations are carried out according to the theory of limit states.

Verification of Rectangular Cross Sections Made of Plain Concrete

The cross-section is rectangular, loaded by the bending moment M, normal force N (applied in the cross-section centroid) and by the shear force V:

$$M_{rd} = \frac{b\,h^2}{6} f_{ctd}$$

The shear strength is provided by:

$$V_{rd} = \tau_c\, b\, h$$

where: τ_c - is the design value of stress in concrete obtained from table 19 of the IS456 standard for the degree of longitudinal reinforcement $\rho =$ 0.

Strength of concrete cross-section subject to the combination of bending moment and normal force with eccentricity e is derived from the following expressions:

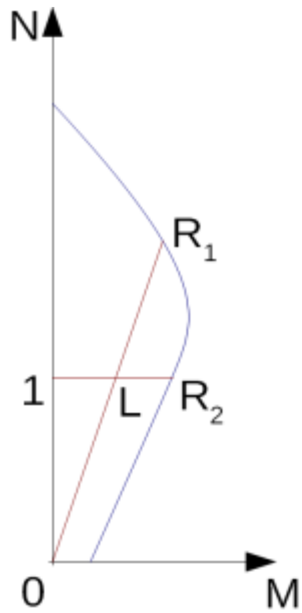
$$P_{rd} = b\, x\, f_{cd}$$

$$P_{rd} = Min\left(\frac{b\,h\,f_{ctd}}{\frac{6\,e}{h} - 1} \;;\; \frac{b\,h\,f_{cd}}{\frac{6\,e}{h} + 1} \right)$$

where:

$$x = h - 2\,e$$

$$e = \frac{abs(M_u)}{P_u}$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R_1|$ or $|1L|/|1R_2|$. Where L is load, R_1 is strength with prescribed eccentricity, and R_2 is strength with prescribed normal force.

Verification of Rectangular RC Cross Section

The cross-section is rectangular, unilaterally reinforced, and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation. The maximum allowable strain of concrete in compression is 0.002 - 0.0035. Compression reinforcement is not taken into account. Minimum eccentricity is applied:

$$e_0 = Max\left(\frac{h}{30}; 20mm\right)$$

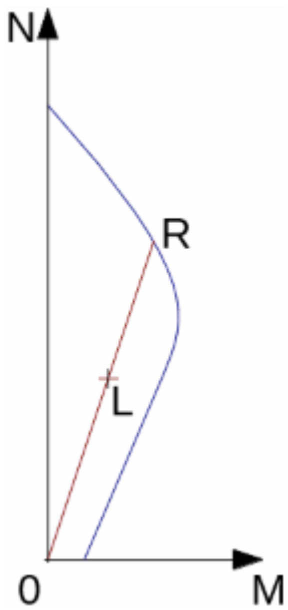
The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

$$\rho = \frac{A_s}{b\,d}$$

$$\rho_{min} = \frac{0{,}85}{f_y}$$

$$\rho_{max} = 0{,}04$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R|$. Where L is load and R is strength with prescribed eccentricity.

Bending without normal force

The cross-section is rectangular, reinforced on one side, and loaded by the bending moment M.

The permissible moment for a given area of reinforcements A_s reads:

$$M_{rd} = b\,F_c\,(d - 0{,}42\,x)$$
$$F_c = 0{,}36\,f_{ck}\,x$$
$$x = \frac{A_s\,f_{yd}}{b\,0{,}36\,f_{ck}}$$

The program further checks whether the location of the neutral axis x is less than the limit location of neutral axis x_{max} given by:

$x_{max} = 0.53d$ - for steel Fe 250

$x_{max} = 0.48d$ - for steel Fe 400

$x_{max} = 0.46d$ - for steel Fe 500

Shear

First, the program computes the ultimate shear strength of concrete V_{uc}.

$$V_{uc} = \tau_c\,d\,b$$

where: τ_c is determined according to table 19 standard IS 456 : 2000.

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength $V_{uc,max}$ is checked.

$$V_{uc,\max} = \tau_{c,\max}\,d\,b$$

where: $\tau_{c,max}$ is determined according to table 20 standard IS 456 : 2000.

Next, the necessary reinforcement area is given by:

$$A_{sv} = \frac{V_u - V_{uc}}{0{,}87\,f_y\,d}\,b$$

where:

$$f_y \le 415\,MPa$$

Verification of Circular RC Cross Section

The program verifies a reinforced concrete pile using the method of limit deformation. The maximum allowable strain of concrete in compression is 0.002 - 0.0035.

The degree of reinforcement is checked using the formula:

$$\rho_{min} \le \rho \le \rho_{max}$$

- **Column** - check for dominant compression

$$\rho = \frac{4.A_s}{\pi.d^2}$$
$$\rho_{\min} = 0{,}008$$
$$\rho_{max} = 0{,}04$$

- **Beam** - check for dominant bending

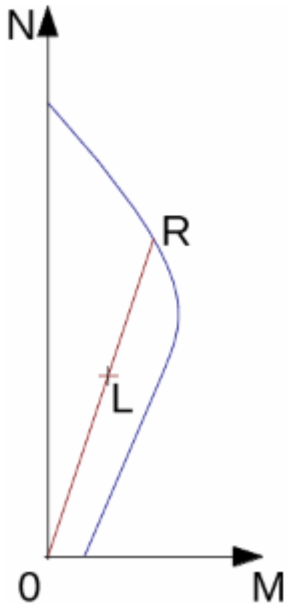
$$\rho = 0{,}5\,\frac{4\,A_s}{\pi\,d^2}$$

$$\rho_{min} = \frac{0{,}85}{f_y}$$

$$\rho_{max} = 0{,}04$$

where: d - pile diameter

A_s - reinforcement area



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R|$. Where L is load and R is strength with prescribed eccentricity.

Verification of Spread Footing for Punching Shear

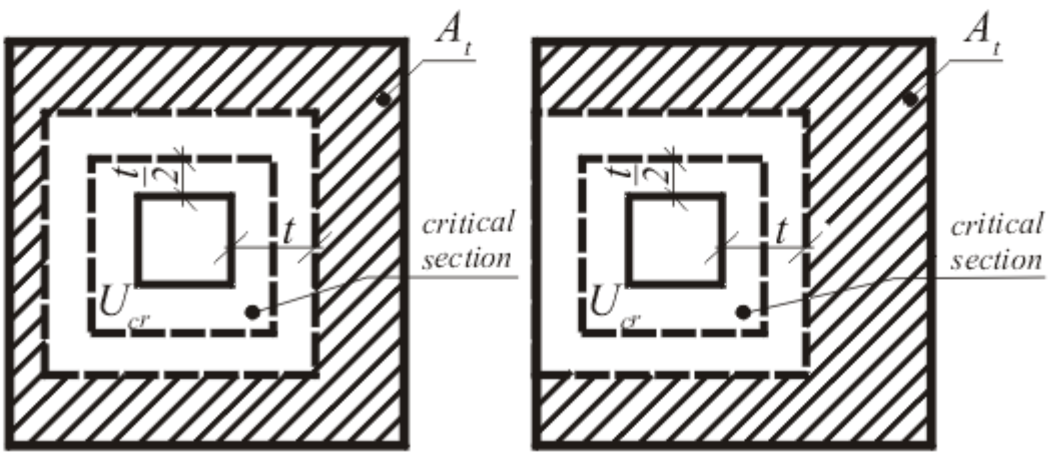
The critical section loaded in shear U_{cr} is distant from the column edge by one half of the footing thickness. It is loaded by the prescribed moments M_x, M_y and by the shear force V_r provided by:

$$V_r = \frac{Q.A_t}{A}$$

where: A - area of footing

Q - assigned vertical force developed in column

A_t - hatched area in fig.



Dimensioning of shear reinforcement area At

The program computes the maximum shear force V developed in the critical section, the shear force transmitted by concrete with no shear reinforcement V_c, and the maximal allowable force V_{max}:

$$V_c = \tau_{rd}.k_s.h$$
$$V_{max} = 1{,}5.V_c$$

where:

$$\tau_c = 0{,}25.\sqrt{f_{ctk}}$$
$$k_s = Min\left(0{,}5 + \frac{c_x}{c_y}\ ;1\right)$$

where: c_x, c_y - are dimensions of footing column

For $V < V_c$ no shear reinforcement is needed.

For $V > V_c$ and $V < V_{max}$ it is necessary to design shear reinforcement. The permissable shear force is given by:

$$V_{rd,3} = \frac{1}{2}.V_c + V_{us}$$
$$V_{us} = \frac{\sum 0{,}87\,A_{sv}.f_{yd}.sin\alpha}{u}$$

where:

u	-	critical cross-section span
α	-	is angle of bends
A_{sv}	-	overall area of bends in footing

For $V > V_{max}$ the shear reinforcement cannot be designed. It is therefore necessary to increase the cross-section depth.

Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment M_{rd}. The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross-section. First, the program determines the location of the neutral axis as:

$$x = \frac{d - \sqrt{d^2 - \frac{M_{rd}}{0{,}96.b.f_{cd}}}}{0{,}84}$$

Providing the location of the neutral axis is less than the allowable one ($x < x_{max}$), the program determines the area of tensile reinforcement A_{st} from the expression:

$$A_{st} = 0{,}36.b.x.f_{ck}$$

Providing the location of the neutral axis is greater than the allowable one ($x > x_{max}$), the program determines the areas of both compressive A_{sc} and tensile A_{st} reinforcement from the expressions:

$$A_{sc} = \frac{M - F_{c,max}\left(d - 0{,}42.x_{max}\right)}{f_{yd}.z}$$
$$A_{st} = \frac{F_{c,max} + A_{sc}.f_{yd}}{f_{yd}}$$
$$F_{c.max} = 0{,}36.x_{max}.b.f_{ck}$$

The limit location of the neutral axis is found from:

$x_{max} =$ 0.53d	for steel Fe 250
$x_{max} =$ 0.48d	for steel Fe 400
$x_{max} =$ 0.46d	for steel Fe 500

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} \le \rho \le \rho_{max}$$

$$\rho = \frac{A_s}{b.d}$$

$$\rho_{min} = \frac{0,85}{f_y}$$

$$\rho_{max} = 0,04$$

If the maximum degree of tensile reinforcement ($\rho_{t,max} =$ 0.04) or total reinforcement ($\rho_{max} =$ 0.08), respectively, is exceeded, the program informs the user that the longitudinal reinforcement cannot be designed for a given cross-section.

Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and bends, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force transmitted by concrete V_{cu} and the maximum allowable shear force $V_{uc,max}$.

$$V_{uc} = \tau_c.d$$

where: τ_c is determined according to table 19 standard IS 456 : 2000.

$$V_{uc,max} = \tau_{c,max}.d$$

where: $\tau_{c,max}$ is determined according to table 20 standard IS 456 : 2000.

As for stirrups, the necessary reinforcement area is given by:

$$A_{sv} = \frac{V_u - V_{uc}}{0,87.f_y.d}$$

As for bends, the necessary reinforcement area is given by:

$$A_{sv} = \frac{V_u - V_{uc}}{0,87.f_y.(sin\,\alpha + cos\,\alpha).d}$$

where:

$$f_y \le 415\,MPa$$

ACI 318-19

This help contains the following methods:

- Materials, coefficients, notation
- Verification of rectangular cross-section made of plain concrete
- Verification of rectangular RC cross-section
- Verification of circular RC cross-section
- Verification of spread footing for punching shear
- Design of longitudinal reinforcement for slabs
- Design of shear reinforcement for slabs

Materials, Coefficients, Notation

The following notation for material parameters is used:

f'_c - specified compressive strength of concrete

E_c - modulus of elasticity of concrete

f_y - specified yield strength of reinforcing steel

The modulus of elasticity is provided by (Art. 19.2.2.1b):

$$E_c = 57000\sqrt{f'_c}$$

The most common notation for geometrical parameters:

b - cross-section width

h - cross-section depth

d - effective depth of cross section

Verification of Rectangular Cross Section Made of Plain Concrete

The cross-section is rectangular, loaded by the bending moment M, normal force P (applied in the cross-section centroid), and by the shear force V_n.

The shear strength is provided by (Art. 22.5.5, Art. 14.5.5.1):

$$\phi V_n \geq V_u$$
$$V_n = \frac{4}{3}\sqrt{f'_c}bh$$
$$\phi = 0.6$$

Strength of concrete cross-section subject to the combination of bending moment and normal force is derived from the following expressions (Art. 21.2, Art. 14.5.2-4):

for compression side:

$$\frac{M_u}{\phi M_n} + \frac{P_u}{\phi P_n} \leq 1$$

where:

$$P_n = 0.60 f'_c bh$$
$$M_n = 0.85 f'_c S_m$$
$$S_m = \frac{bh^2}{6}$$
$$\phi = 0.6$$

for tension side:

$$\frac{M_u}{S_m} - \frac{P_u}{bh} \leq 5\phi\sqrt{f'_c}$$

where:

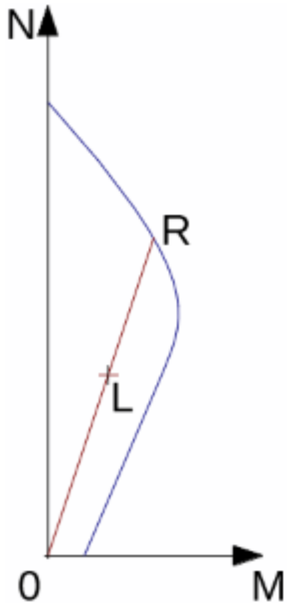
$$\phi = 0.6$$

Verification of Rectangular RC Cross Section

The cross-section is rectangular, unilaterally reinforced and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation (Art. 22.2). The maximum allowable strain of concrete in compression is 0.003.

The computed degree of reinforcement is checked using the following expressions (Art. 9.6.1.2):

$$\rho_{min} \leq \rho \leq \rho_{max}$$
$$\rho = \frac{A_s}{bd}$$
$$\rho_{min} = \frac{\mathrm{Max}\left(3\sqrt{f'_c}; 200\right)}{f_y}$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R|$. Where L is load and R is strength with prescribed eccentricity.

Bending without normal force

The cross-section is rectangular, reinforced on one side, and loaded by the bending moment M_u .

The ultimate moment is provided by (Ch. 9.3, Art. 21.2.2, Art. 22.2.2.4.3):

$$M_u \leq \phi M_n$$

$$M_n = A_s f_y \left(d - \frac{a}{2}\right)$$

$$a = \frac{A_s f_y}{0.85 f_c' b}$$

$$c = \frac{a}{\beta_1}$$

$$\beta_1 = 0.85 - 0.05 \frac{f_c' - 4000}{1000}$$

$$0.65 \leq \beta_1 \leq 0.85$$

$$\phi = 0.65 + 0.25 \frac{(\varepsilon_t - \varepsilon_{ty})}{0.003}$$

$$0.65 \leq \phi \leq 0.9$$

The limit location of the neutral axis is found from (Art. 9.3.1.1):

$$c_{max} = \frac{0.003}{0.003 + \varepsilon_{ty}}$$

Shear

First, the program computes the ultimate shear strength of concrete V_c (Art. 22.5.5.1).

$$V_c = \left[\mathrm{Max}(2; 8\sqrt[3]{\rho_w})\sqrt{f_c'} + \frac{N_u}{6A_g}\right] b_w d \le 5\sqrt{f_c'} b_w d$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength V_{max} is checked (Art. 22.5.12).

$$V_{max} = V_c + 8\sqrt{f_c'} b_w d$$

Next, the necessary reinforcement area is given by (Art. 22.5.8.5.3):

$$A_v = \frac{V_u - \phi V_c}{\phi f_{yt} d} b_w$$

where (Art. 20.2.2.4, Art. 21.2.1):

$$f_{yt} \le 60000\mathrm{psi}$$
$$\phi = 0.75$$

Verification of Circular RC Cross Section

The program verifies a reinforced concrete pile using the method of limit deformation (Art. 22.2). The maximum allowable strain of concrete in compression is $0{,}003$. The degree of reinforcement is checked using the formula:

$$\rho_{min} \le \rho \le \rho_{max}$$

- **Column** - check for dominant compression (Art. 10.6.1)

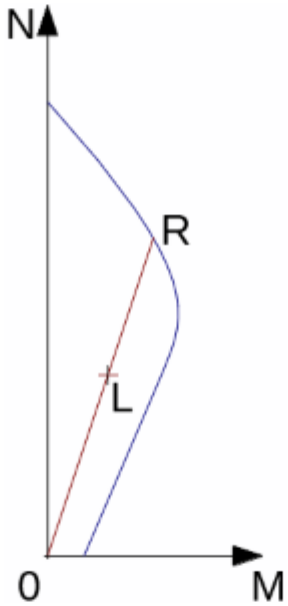
$$\rho = \frac{4A_s}{\pi d^2}$$
$$\rho_{min} = 0.01$$
$$\rho_{max} = 0.08$$

- **Beam** - check for dominant bending (Art. 9.6.1.2)

$$\rho = 0.5\frac{4A_s}{\pi d^2}$$
$$\rho_{min} = \frac{\mathrm{Max}\left(3\sqrt{f_c'}; 200\right)}{f_y}$$

where: d - pile diameter

A_s - reinforcement area



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R|$. Where L is load and R is strength with prescribed eccentricity.

Shear

First, the program computes the ultimate shear strength of concrete V_c (Art. 22.5.5.1).

$$V_c = \left[\mathrm{Max}(2; 8\sqrt[3]{\rho_w})\sqrt{f_c'} + \frac{N_u}{6A_g} \right] 0.8d^2 \leq 5\sqrt{f_c'}0.8d^2$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength V_{max} and strength of reinforced section V_s are checked (Art. 22.5.8.5.3).

$$V_{max} = V_c + 8\sqrt{f_c'}b_w d$$

$$V_s = \frac{A_v f_{yt} 0.8d}{s}$$

where (Art. 20.2.2.4, Art. 21.2.1):

$$f_{yt} \leq 60000\mathrm{psi}$$

$$\phi = 0.75$$

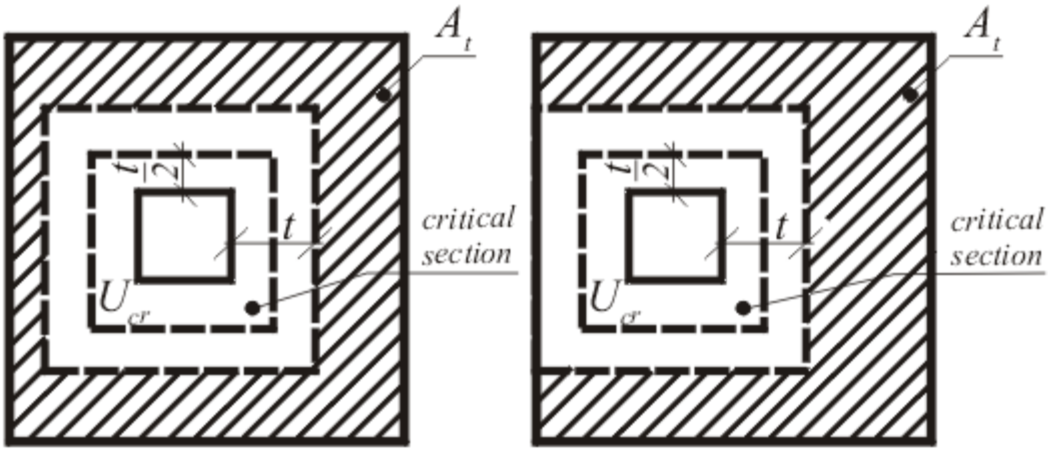
Verification of Spread Footing for Punching Shear

The program allows us to verify spread footing for punching shear or for the design of shear reinforcement. The critical section loaded in shear b_o is distant from the column edge by one half of the footing thickness. It is loaded by the prescribed moments M_x , M_y and by the shear force V_u provided by:

$$V_u = \frac{V A_t}{A}$$

where:

A	-	area of footing
V	-	assigned vertical force developed in column
A_t	-	hatched area in fig**.**



Dimensioning of shear reinforcement area A_t

The program computes the maximal shear force V_u developed in the critical section, the shear force transmitted by concrete with no shear reinforcement V_c as minimum of values (Art. 22.6.5.2):

$$V_c = \left(2 + \frac{4}{\beta}\right)\lambda_s\sqrt{f_c'}b_o d$$

where β_c is the ratio of the log side to short side od column.

$$V_c = \left(2 + \frac{\alpha_s d}{b_o}\right)\lambda_s\sqrt{f_c'}b_o d$$

where α_s = 40 - inner column

30 - edge column

20 - corner column

$$V_c = 4\lambda_s\sqrt{f_c'}b_o d$$

and the maximal allowable force V_{max} (Art. 22.6.6.3):

$$V_{max} = 6\sqrt{f_c'}b_o d$$

For $V_u < \phi \times V_c$ no shear reinforcement is needed.

For $V_u > \phi \times V_c$ and $V_u < \phi \times V_{max}$ the shear reinforcement must be introduced. The ultimate shear force is given by (Art. 22.6.6):

$$V_n = \phi\left(2\lambda_s\sqrt{f_c'}b_o d + A_v f_{yt}\sin\alpha\right)$$

$$\phi = 0.75$$

where: b_o - critical cross-section span

α - is angle of bends

A_v - overall area of bends in footing

For $V_u > \phi \times V_{max}$ the shear reinforcement cannot be designed. It is therefore necessary to increase the cross-section height.

Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment M. The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross-section. First, the program determines the location of the neutral axis as:

$$x = d - \sqrt{d^2 - \frac{2M_d}{0.85\phi b f_c'}}$$

Providing the location of the neutral axis is less than the allowable one ($c < c_{max}$), the program determines the area of tensile reinforcement A_{st} from the expression:

$$A_{st} = \frac{0.85 f_c' b x \beta_1}{f_y}$$

Providing the location of the neutral axis is greater than the allowable one ($c > c_{max}$), the program determines the areas of both compressive A_{sc} and tensile A_{st} reinforcement from the expressions:

$$A_{sc} = \frac{\frac{M}{\phi} - F_{c,max}(d - 0.45x_{max})}{f_{yd} z}$$

$$A_{st} = \frac{M - A_{sc} f_y}{f_y}$$

$$F_{c,max} = 0.85 b f_c'$$

where:

$$\phi = 0.65 + 0.25\frac{(\varepsilon_t - \varepsilon_{ty})}{0.003}$$

$$0.65 \le \phi \le 0.9$$

The limit location of the neutral axis is found from (Art.9.3.1.1):

$$c_{max} = \frac{0.003}{0.003 + \varepsilon_{ty}}$$

The computed degree of reinforcement is checked using the following expressions (Art. 7.6.1.1):

$$\rho_{min} \le \rho \le \rho_{max}$$

$$\rho = \frac{A_s}{bd}$$

$$\rho_{min} = 0.0018$$

Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and bends, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force transmitted by concrete V_c (Art. 22.5.5) and the maximum allowable shear force V_{max} (Art. 22.5.1.2).

$$V_c = \left[\mathrm{Max}(2; 8\sqrt[3]{\rho_w})\sqrt{f_c'} + \frac{N_u}{6A_g}\right] b_w d \le 5\sqrt{f_c'} b_w d$$

$$V_{max} = V_c + 8\sqrt{f'_c} b_w d$$

As for stirrups, the necessary reinforcement area is given by (Art. 22.5.8.5.3):

$$A_v = \frac{V_u - \phi V_c}{\phi f_{yt} d}$$

As for bends, the necessary reinforcement area is given by (Art. 22.5.8.5.4):

$$A_v = \frac{V_u - \phi V_c}{\phi f_{yt} (\sin\alpha + \cos\alpha) d}$$

where (Art. 20.2.2.4, Art. 21.2.1):

$$f_{yt} \leq 60000\text{psi}$$
$$\phi = 0.75$$

AS 3600 - 2018

This help contains the following methods:

- Materials, coefficients, notation
- Verification of rectangular cross-section made of plain concrete
- Verification of rectangular RC cross-section
- Verification of circular RC cross-section
- Verification of spread footing for punching shear
- Design of longitudinal reinforcement for slabs
- Design of shear reinforcement for slabs

Materials, Coefficients, Notation

The following notation for material parameters is used:

f'_c - characteristic compressive cylinder strength of concrete at 28 days

E_c - mean value of the modulus of elasticity of concrete at 28 days

$f'_{ct,f}$ - characteristic flexural tensile strength of concrete

f'_{ct} - characteristic principal tensile strength of concrete

f_{sy} - yield strength of reinforcing steel

$$f'_{ct,f} = 0.6\sqrt{f'_c}$$
$$f'_{ct} = 0.36\sqrt{f'_c}$$

The characteristic compressive strength of concrete is the basic input parameter given by the class of concrete.

The most common notation for geometrical parameters:

b - cross-section width

D - cross-section depth

d - effective depth of cross-section

z - lever arm (arm of internal forces)

All computations are carried out according to the theory of limit states.

Verification of Rectangular Cross Sections Made of Plain Concrete

The cross-section is rectangular, loaded by the bending moment M, normal force N (applied in the cross-section centroid) and by the shear force V. The cross-section bearing capacity subjected to bending moment is given by (Art. 20.4.2, 2.2.2):

$$M < \phi M_{uo}$$
$$M_{uo} = \frac{bD^2}{6} f'_{ct,f}$$
$$\phi = 0.6$$

The shear strength is provided by (Art. 20.4.3):

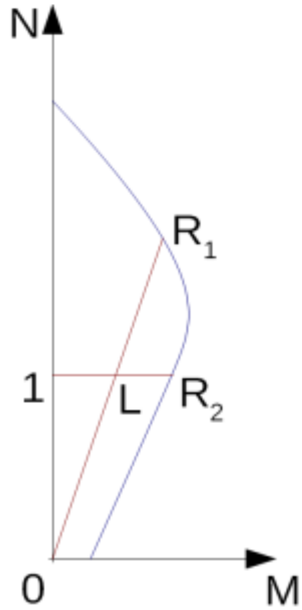
$$V < \phi V_{uc}$$
$$V_{uc} = 0.15bD(f'_c)^{\frac{1}{3}}$$
$$\phi = 0.6$$

Strength of concrete cross-section subject to the combination of bending moment and normal force is derived from the following expressions depending on the normal force eccentricity $e \geq 0.05D$ (Art. 10.1.2, 20.4.2, 2.2.2):

$$N < \phi N_{uo}$$
$$N_{uo} = \mathrm{Min}\left(\frac{bDf'_{ct,f}}{\frac{6e}{D} - 1}; \frac{bDf'_c}{\frac{6e}{D} + 1}\right)$$

where:

$$e = \left|\frac{M}{N}\right|$$
$$\phi = 0.6$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L| / |0R_1|$ or $|1L| / |1R_2|$. Where L is load, R_1 is strength with prescribed eccentricity, and R_2 is strength with prescribed normal force.

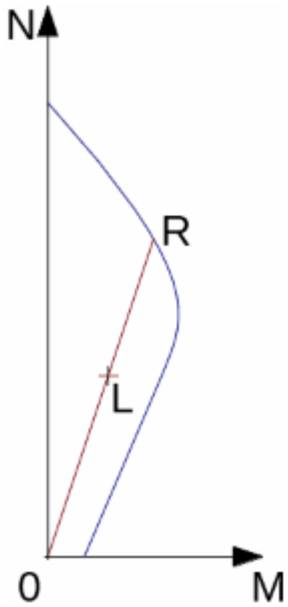
Verification of Rectangular RC Cross Section

The cross-section is rectangular, unilaterally reinforced and loaded by the bending moment, and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation (Art. 8.1). The maximum allowable strain of concrete in compression is 0.003. Compression reinforcement is not taken into account.

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} = \left[0.2\left(\frac{D}{d}\right)^2 \frac{f_{ct,f}}{f_{sy}}\right] < \rho < 0.04 = \rho_{max}$$

$$\rho = \frac{A_s}{bd}$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R|$. Where L is load and R is strength with prescribed excentricity.

Bending without normal force

The cross-section is rectangular, reinforced on one side, and loaded by the bending moment M.

The permissible moment for a given area of reinforcements A_s reads:

$$M < \phi M_{uo}$$

for L class reinforcement (Art. 2.2.2):

$$\phi = 0.65$$

for N class reinforcement:

$$\phi = 0.65 \leq \left(1.24 - \frac{13k_u o}{12}\right) \leq 0.85$$

where

$$M_{uo} = A_s f_{sy}\left(d - 0.5\gamma k_u d\right)$$

$$\gamma k_u d = \frac{A_s f_{sy}}{b\alpha_2 f'_c}$$

$$\alpha_2 = \mathrm{Max}\left(0.67; 0.85 - 0.0015 f'_c\right)$$

$$\gamma = \mathrm{Max}\left(0.67; 0.97 - 0.0025 f'_c\right)$$

The program further checks whether the neutral axis parameter k_u is less than the limit value (Art 8.1.5) :

$$k_u \leq 0.36$$

$$k_u = \frac{A_s f_{sy}}{b\alpha_2 f'_c \gamma d}$$

Shear

First, the program computes the ultimate shear strength of concrete V_{uc} (Art. 8.2.4.1).

$$V_{uc} = k_v b_v d_v \sqrt{f'_c}$$

where (Art. 8.2.1.9, Art. 8.2.4.3):

$$d_v = \mathrm{Max}\left(0.72D; 0.9d\right)$$

$$\sqrt{f'_c} \leq 8\text{MPa}$$

$$k_v = \text{Min}\left(0.1; \frac{200}{1000 + 1.3d_v}\right)$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength $V_{u,max}$ is checked (Art. 8.2.3.3).

$$V_{u,max} = 0.55\left[f'_c b d_v \frac{\cot(\theta_v)}{1 + \cot(\theta_v)^2}\right]$$

$$\theta_v = 36°$$

Next, the necessary reinforcement area is given by (Art. 8.2.5.2):

$$A_{sv} = \frac{V - \phi V_{uc}}{\phi f_{sy,f} d_v \cot(\theta_v)} b$$

where:

$$\phi = 0.75$$

Verification of Circular RC Cross Section

The program verifies a reinforced concrete pile using the method of limit deformation (Art 10.6.1). The maximum allowable strain of concrete in compression is 0.003.

The degree of reinforcement is checked using the formula:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

- **Column** - check for dominant compression (Art. 10.7.1)

$$\rho = \frac{4A_s}{\pi d^2}$$

$$\rho_{min} = 0.01$$

$$\rho_{max} = 0.04$$

- **Beam** - check for dominant bending (Art. 8.1.6.1)

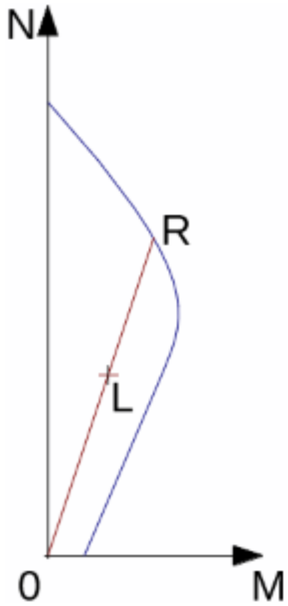
$$\rho = 0.5\frac{4A_s}{\pi d^2}$$

$$\rho_{min} = \left[0.2\left(\frac{D}{d}\right)^2 \frac{f_{ct,f}}{f_{sy}}\right]$$

$$\rho_{max} = 0.04$$

where: d - pile diameter

A_s - cross sectional area of reinforcement



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R|$. Where L is load and R is strength with prescribed excentricity.

Shear

$$V < \phi V_u$$
$$\phi = 0.75$$

First, the program computes the ultimate shear strength of concrete V_c. Formulas are from Art. 8.2.4.1, where the section width (b_v) is replaced by 0.88d and effective depth (d_v) is replaced 0.8 * 0.9 * 0.9d.

$$V_{uc} = 0.57 k_v d^2 \sqrt{f_c'}$$

where (Art. 8.2.1.9, Art. 8.2.4.3):

$$\sqrt{f_c'} \leq 8\text{MPa}$$
$$k_v = \text{Min}\left(0.1; \frac{200}{1000 + 1.3 d_v}\right)$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength $V_{u,max}$ and strength of reinforced section V_{us} are checked (Art. 8.2.5.2).

$$V_{u,max} = 0.55\left[f_c' 0.57 d^2 \frac{\cot(\theta_v)}{1 + \cot(\theta_v)^2}\right]$$
$$\theta_v = 36°$$
$$V_{us} = V_{uc} + \frac{A_{sv}}{s} f_{sy,f} d_v \cot(\theta_v)$$

Verification of Spread Footing for Punching Shear

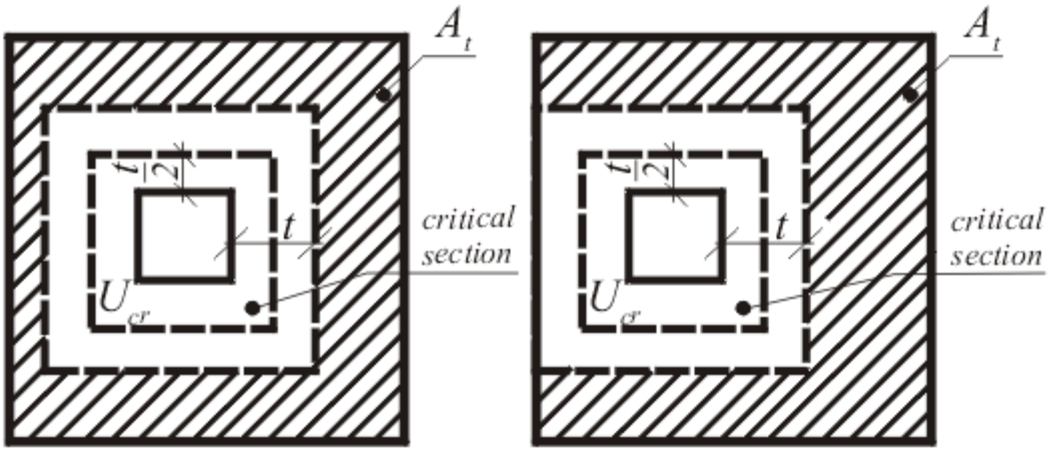
The program allows us to verify spread footing for punching shear. The critical section loaded in shear U_{cr} is distant from the column edge by one half of the footing thickness. It is loaded by the prescribed moments M_x, M_y and by the shear force V^* provided by:

$$V^* = \frac{V A_t}{A}$$

where: A - area of footing

V - assigned vertical force developed in column

A_t - hatched area in fig.



Dimensioning of shear reinforcement area At

The program computes the maximal shear force V^* developed in the critical section, the shear force transmitted by concrete with no shear reinforcement V_{uc} as a minimum of values (Art. 9.3.3):

$$V^* \leq \phi V_u$$
$$\phi = 0.75$$
$$V_{uc} = \frac{V_{uo}}{1 + u\frac{M_v^*}{8V^* a d_{om}}}$$

where:

$$V_{uo} = u d_{om} f_{cv}$$
$$f_{cv} = \mathrm{Min}\left(0.34; 0.17 + \left(1 + \frac{2}{\beta_h}\right)\right)\sqrt{f_c'}$$

where β_c is the ratio of the longest overall dimension of the effective loaded area, Y, to the overall dimension, X, measured perpendicular to Y.

and the maximal allowable force V_{umax} (Art. 9.3.4):

$$V_{umax} = 3V_{umin}\sqrt{\frac{x}{y}}$$
$$V_{umin} = \frac{1.2V_{uo}}{1 + u\frac{M_v^*}{2V^* a^2}}$$
$$V_{umin} = \frac{1.2V_{uo}}{1 + u\frac{M_v^*}{2V^* a^2}}$$

where:

- u - critical cross-section span
- x - short dimension of cross section
- y - long dimension of cross section
- a - the dimension of the critical shear perimeter measured parallel to the direction of M^*_v
- M^*_v - the bending moment transferred from the slab to support in the direction being considered

For $V^* < V_{uc}$ no shear reinforcement is needed.

For $V^* > V_{uc}$ and $V^* < V_{umax}$ the shear reinforcement must be introduced. The ultimate shear force is given by (Art. 9.3.4):

$$V_{us} = V_{umin}\sqrt{\frac{\frac{A_{sw}}{s}}{A_{swmin}}}$$

$$A_{swmin} = \frac{0.2 y_1}{f_{sy,f}}$$

Area of reinforcement must satisfy condition (Art. 9.3.4):

$$\frac{A_{sw}}{s} \geq A_{swmin}$$

For $V^* > V_{umax}$ the shear reinforcement cannot be designed. It is therefore necessary to increase the cross-section height.

The analysis is carried out independently in directions x and y, as the decisive one the lower value of V_u is accepted.

Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment M. The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross-section. First, the program determines the location of the neutral axis as (Art. 8.1):

$$x = \frac{d - \sqrt{d^2 - \frac{M}{0.5 \phi b \alpha_2 f'_c}}}{\gamma}$$

where:

$$\alpha_2 = \text{Max}\,(0.67; 0.85 - 0.0015 f'_c)$$
$$\gamma = \text{Max}\,(0.67; 0.97 - 0.0025 f'_c)$$

for L class reinforcement (Art. 2.2.2):

$$\phi = 0.65$$

for N class reinforcement:

$$\phi = 0.65 \leq \left(1.24 - \frac{13 k_u o}{12}\right) \leq 0.85$$

Providing the location of the neutral axis is less than the allowable one (x_{max}), the program determines the area of tensile reinforcement A_{st} from the expression:

$$A_{st} = \frac{\gamma \alpha_2 b x f'_c}{f_{sy}}$$

Providing the location of the neutral axis is greater than the allowable one (x_{max}), the program determines the areas of both compressive (A_{sc}) and tensile (A_{st}) reinforcement from the expressions:

$$A_{sc} = \frac{\frac{M}{\phi} - F_{x,max}(d - 0.5 \gamma x_{max})}{f_{sy} z}$$

$$A_{st} = \frac{F_{x,max} + A_{sc} f_{sy}}{f_{sy}}$$

$$F_{x,max} = \gamma \alpha_2 b x_{max} f'_c$$

The limit location of neutral axis x_{max} is found from (Art. 8.1.5):

$$x_{max} = 0.36d$$

The computed degree of reinforcement is checked using the following expressions (Art. 10.5.1.2):

$$\rho_{min} = \left[0.2 \left(\frac{D}{d}\right)^2 \frac{f_{ct,f}}{f_{sy}}\right] < \rho < 0.04 = \rho_{max}$$

$$\rho = \frac{A_s}{bd}$$

If the maximum degree of total reinforcement ρ_{max} is exceeded, the program informs the user that the longitudinal reinforcement cannot be designed for a given cross-section.

Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and bends, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force transmitted by concrete V_{uc} (Art. 8.2.4.1) and the maximum allowable shear force $V_{u,max}$ (Art. 8.2.3.3).

$$V < \phi V_u$$
$$\phi = 0.75$$

where:

$$V_{uc} = k_v b_v d_v \sqrt{f_c'}$$
$$\sqrt{f_c'} \leq 8\text{MPa}$$
$$d_v = \text{Max}\left(0.72D; 0.9d\right)$$
$$k_v = \text{Min}\left(0.1; \frac{200}{1000 + 1.3d_v}\right)$$
$$V_{u,max} = 0.55\left[f_c' b d_v \frac{\cot(\theta_v)}{1 + \cot(\theta_v)^2}\right]$$
$$\theta_v = 36°$$

As for stirrups, the necessary reinforcement area is given by (Art. 8.2.5.2):

$$A_{sv} = \frac{V - \phi V_{uc}}{\phi f_{sy,f} d_v \cot(\theta_v)}$$

As for bends, the necessary reinforcement area is given by (Art. 8.2.5.2):

$$A_{sv} = \frac{V - \phi V_{uc}}{\phi f_{sy,f} d_v (\sin(\alpha_v)\cot(\theta_v) + \cos(\alpha_v))}$$

SNiP 52-101-2003

This help contains the following methods:

- Materials, coefficients, notation
- Verification of rectangular cross-section made of plain concrete
- Verification of rectangular RC cross-section
- Verification of circular RC cross-section
- Verification of spread footing for punching shear
- Design of longitudinal reinforcement for slabs
- Design of shear reinforcement for slabs

Materials, Coefficients, Notation

The following notation for material parameters is used:

R_b - design compressive strength of concrete

R_{bt} - design tensile strength of concrete

R_{sc} - design compressive strength of steel

Rs - design tensile strength of steel bar

The most common notation for geometrical parameters:

b - cross section width

h - cross section depth

h_0 - effective depth of cross section

z - lever arm (arm of internal forces)

Verification of Rectangular Cross Section Made of Plain Concrete

The cross-section is rectangular, loaded by the bending moment M, normal force N (applied in the cross-section centroid), and by the shear force Q. The cross-section bearing capacity subjected to bending moment is given by:

$$M_{ult} = \frac{b.h^2}{6}.R_{bt}$$

The shear strength is provided by:

$$Q_{ult} = 1{,}5.b.h..R_b$$

Strength of concrete cross-section subject to the combination of bending moment and normal force is derived from the following expressions depending on the normal force eccentricity e:

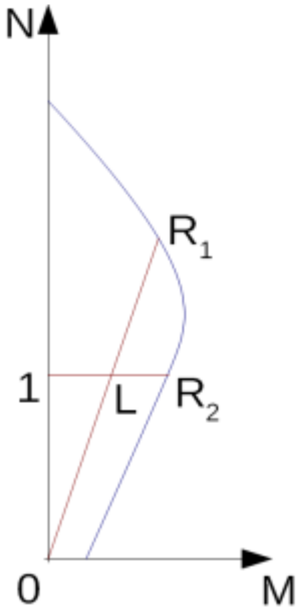
for:

$$N_{ult} = b.x.R_b$$

$$N_{ult} = Min\left(\frac{b.h.R_{bt}}{\frac{6.e}{h} - 1} \; ; \; \frac{b.h.R_b.}{\frac{6.e}{h} + 1} \right)$$

$$x_u = h - 2.e$$

$$e = \frac{abs(M)}{N}$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R_1|$ or $|1L|/|1R_2|$. Where L is load, R_1 is strength with prescribed eccentricity and R_2 is strength with prescribed normal force.

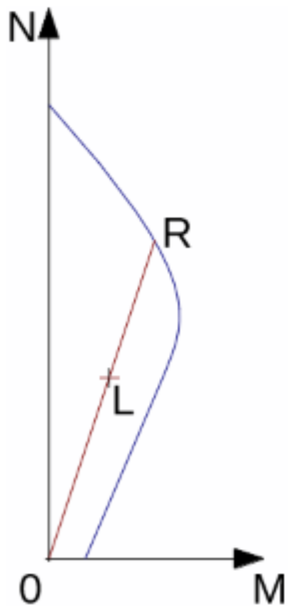
Verification of Rectangular RC Cross Section

The cross-section is rectangular, unilaterally reinforced, and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation. The maximum allowable strain of concrete in compression is 0.002 to 0.0035. Compression reinforcement is not taken into account.

The computed degree of reinforcement is checked using the following expressions:

$$\mu_{st,\min} = 0{,}001 < \mu_{st}$$

$$\mu_{st} = \frac{A_s}{b\,h_0}$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R|$. Where L is load and R is strength with prescribed eccentricity.

Bending without normal force

The cross-section is rectangular, reinforced on one side, and loaded by the bending moment M.

The ultimate moment is provided by:

$$M_{ult} = b\,x\,R_b\left(h_0 - \frac{x}{2}\right)$$

$$x = \frac{A_s R_s}{b\,R_b}$$

The program further checks whether the location of the neutral axis x is less than the limit location of neutral axis x_R given by:

$$x_R = \frac{0{,}8\,h_0}{1 + \frac{R_s}{700}}$$

Shear

First, the program computes the ultimate shear strength of concrete Q_b.

$$Q_b = 2{,}5\,R_{bt}\,h_0\,b$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength Q_{max} is checked.

$$Q_{max} = 0{,}3\,R_b\,h_0\,b$$

Next, the necessary reinforcement area is given by:

$$A_{sw} = \frac{Q - 1{,}5\,R_{bt}\,h_0}{0{,}75\,R_{sw}\,h_0}\,b$$

where:

$$R_{sw} = Min\left(0{,}8\,R_s\;;\;300\,MPa\right)$$

Verification of Circular RC Cross Section

The program verifies a reinforced concrete pile using the method of limit deformation. The maximum allowable strain of concrete in compression is 0.0015 - 0.0035. The degree of reinforcement is checked using the formula:

- **Column** - check for dominant compression

$$\mu_{st,min} = 0{,}001 < \mu_{st}$$

$$\mu_{st} = \frac{4.A_s}{\pi.d^2}$$

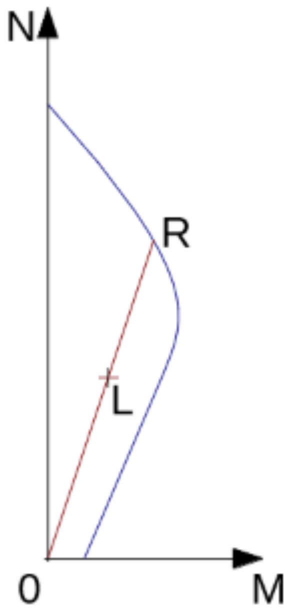
- **Beam** - check for dominant bending

$$\mu_{st,min} = 0{,}001 < \mu_{st}$$

$$\mu_{st} = 0{,}5\,\frac{4\,A_s}{\pi\,d^2}$$

where: d - pile diameter

A_s - reinforcement area



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R|$. Where L is load and R is strength with prescribed eccentricity.

Verification of Spread Footing for Punching Shear

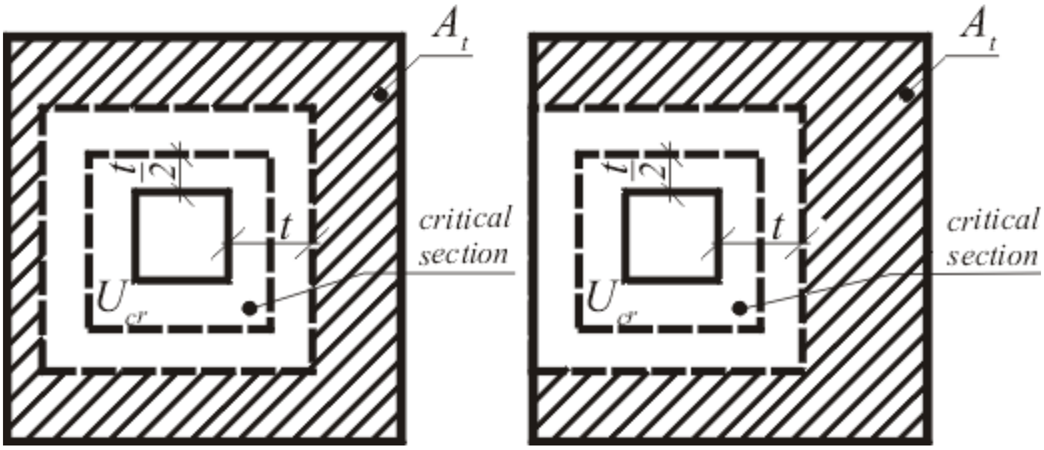
The program allows us to verify spread footing for punching shear or for the design of shear reinforcement. The critical section loaded in shear U_{cr} is distant from the column edge by one half of the footing thickness. It is loaded by the prescribed moments M_x, M_y and by the shear force F provided by:

$$F = \frac{Q.A_t}{A}$$

where: A - area of footing

Q - assigned vertical force developed in column

A_t - hatched area in fig.



Dimensioning of shear reinforcement area A_t

The program computes the maximal shear force F developed in the critical section, the shear force transmitted by concrete with no shear reinforcement $F_{b,ult}$, and the maximal allowable force $F_{ult,max}$:

$$F_{b,ult} = R_{bt}.h_0$$
$$F_{ult,max} = 2.F_{b,ult}$$

For $F < F_{b,ult}$ no shear reinforcement is needed.

For $F > F_{b,ult}$ and $F < F_{ult,max}$ the shear reinforcement must be introduced. The ultimate shear force is given by:

$$F_{ult} = F_{b,ult} + F_{sw,ult}$$
$$F_{sw,ult} = \frac{0{,}8.A_s.R_{sw}.\sin\alpha}{V_{cr}}$$

where: V_{cr} - critical cross-section span

α - is angle of bends

A_s - overall area of bends in footing

For $F > F_{ult,max}$ the shear reinforcement cannot be designed. It is therefore necessary to increase the cross-section height.

Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment M. The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross-section. First, the program determines the location of the neutral axis as:

$$x = h_0 - \sqrt{h_0^2 - \frac{M}{0{,}5.b.R_b}}$$

Providing the location of the neutral axis is less than the allowable one ($x < x_{max}$), the program determines the area of tensile reinforcement A_{st} from the expression:

$$A_{st} = \frac{b.x.R_{bd}}{R_{sd}}$$

Providing the location of the neutral axis is greater than the allowable one ($x > x_{max}$), the program determines the areas of both compressive A_{sc} and tensile A_{st} reinforcement from the expressions:

$$A_{sc} = \frac{M - F_{c,max}\left(h_e - 0{,}5.x_{max}\right)}{R_{sd}.z}$$
$$A_{st} = \frac{F_{c,max} + A_{sc}.R_{sd}}{R_{sd}}$$
$$F_{c,max} = x_{max}.b.R_{bd}$$

The limit location of the neutral axis is found from:

$$x_{max} = 0{,}533.h_e$$

The computed degree of reinforcement is checked using the following expressions:

$$\mu_{st,\min} = 0{,}001 < \mu_{st}$$

$$\mu_{st} = \frac{A_s}{b\,h_0}$$

Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and bends, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force transmitted by concrete Q_b and the maximum allowable shear force Q_{max}.

$$Q_b = 2{,}5\,R_{bt}\,h_0$$

$$Q_{\max} = 0{,}3\,R_b\,h_0$$

As for stirrups, the necessary reinforcement area is given by:

$$A_{sw} = \frac{Q - 1{,}5\,R_{bt}\,h_0}{0{,}75\,R_{sw}\,h_0}$$

As for bends, the necessary reinforcement area is given by:

$$A_{sw} = \frac{Q - 1{,}5\,R_{bt}\,h_0}{0{,}75\,R_{sw}\,h_0\,\sin\alpha}$$

where:

$$R_{sw} = Min\left(0{,}8\,R_s\ ;\ 300\,MPa\right)$$

GB 50010-2010

This help contains the following methods:

- Materials, coefficients, notation
- Verification of rectangular cross-section made of plain concrete
- Verification of rectangular RC cross-section
- Verification of circular RC cross-section
- Verification of spread footing for punching shear
- Design of longitudinal reinforcement for slabs
- Design of shear reinforcement for slabs

Materials, Coefficients, Notation

The following notation for material parameters is used:

f_c - design compressive strength of concrete

f_t - design axial tensile strength of concrete

f'_y - design compressive strength of steel bar

f_y - design tensile strength of steel bar

The most common notation for geometrical parameters:

b - cross-section width

h - cross-section depth

h_0 - effective depth of cross section

Verification of Rectangular Cross Section Made of Plain Concrete

The cross-section is rectangular, loaded by the bending moment M, normal force N (applied in the cross-section centroid) and by the shear force V. The cross-section bearing capacity subjected to bending moment is given by (Art.

D.3):

$$M_u = \frac{bh^2}{6}\gamma f_{ct}$$

where (Art. 7.2.4, Art. D.2.2):

$$\gamma = \left(0.7 + \frac{120}{h}\right)1.55$$

$$400\text{mm} \leq h \leq 1600\text{mm}$$

$$f_{ct} = 0.55 f_t$$

The shear strength is provided by (Art. 6.3.3):

$$V_u = 0.7 f_t bh$$

Strength of concrete cross-section subject to the combination of bending moment and normal force is derived from the following expressions depending on the normal force eccentricity e_0 (Art. D.2.1):

As the greater of:

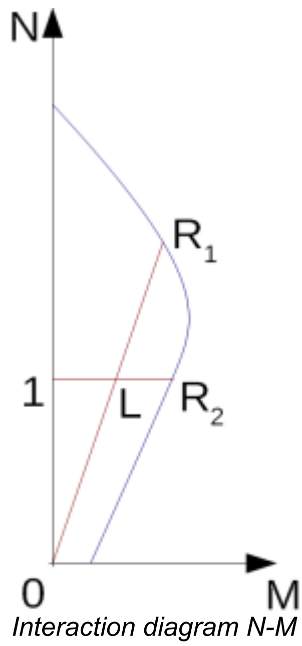
$$e_0 \leq 0.45h \Rightarrow N_u = f_{cc}b(h - 2e_0)$$

$$N_u = \text{Min}\left(\frac{bh\gamma f_{ct}}{\frac{6e_0}{h} - 1}; \frac{bhf_c}{\frac{6e_0}{h} + 1}\right)$$

where (Art. D.2.1):

$$f_{cc} = 0.85 f_c$$

$$e_0 = \left|\frac{M}{N}\right|$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R_1|$ or $|1L|/|1R_2|$. Where L is load, R_1 is strength with prescribed eccentricity, and R_2 is strength with prescribed normal force.

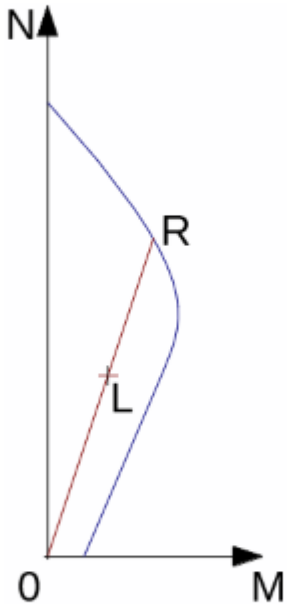
Verification of Rectangular RC Cross Section

The cross-section is rectangular, unilaterally reinforced, and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation (Art. 6.2.1). The maximum allowable strain of concrete in compression is 0.002 to 0.0033. Compression reinforcement is not taken into account.

The computed degree of reinforcement is checked using the following expressions (Art. 8.5.1):

$$\rho_{min} = \text{Max}\left(0.002; 0.45\frac{f_t}{f_y}\right) \leq \rho$$

$$\rho = \frac{A_s}{bh_0}$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L| / |0R|$. Where L is load and R is strength with prescribed eccentricity.

Bending without normal force

The cross-section is rectangular, reinforced on one side, and loaded by the bending moment M.

The ultimate moment is provided by (Art. 6.2.10):

$$M_u = \alpha_1 f_c bx\left(h_0 - \frac{x}{2}\right)$$

$$x = \frac{f_y A_s}{\alpha_1 f_c b}$$

$\alpha_1 =$ 1 for: ≤ **C50**

$\alpha_1 =$ 0.94 for: ≥ **C80**, intermediate values are obtained using linear interpolation method (Art. 6.2.6).

The program further checks whether the depth of the compression zone x is less than the limit depth of compression zone $\xi_b * h_0$ given by (Art. 6.2.7):

$$\xi_b = \frac{\beta_1}{a + \frac{f_y}{E_s \varepsilon_{cu}}}$$

$\beta_1 =$ 0.8 for: ≤ **C50**

$\beta_1 =$ 0.74 for: ≥ **C80**, intermediate values are obtained using linear interpolation method (Art. 6.2.6).

Shear

First, the program computes the ultimate shear strength of concrete V_c:

For flexural element (Art. 6.3.3):

$$V_c = 0.7 f_t bh$$

For compression element (Art. 6.3.12):

$$V_c = \frac{1.75}{\frac{M}{Vh_0} + 1} f_t bh + 0.07\,\text{Min}(N; 0.3 f_c A)$$

For tension element (Art. 6.3.14):

$$V_c = \frac{1.75}{\frac{M}{Vh_0}+1} f_t bh - 0.2N$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength V_{max} is checked (Art. 6.3.1).

for $h_0/b \leq 4$

$$V_{max} = 0.25\beta_c f_c bh_0$$

for $h_0/b \geq 6$

$$V_{max} = 0.2\beta_c f_c bh_0$$

intermediate values are obtained using a linear interpolation method

$\beta_c =$ 1 for: ≤ **C50**

$\beta_c =$ 0.8 for: ≥ **C80**, intermediate values are obtained using linear interpolation method.

Next, the necessary reinforcement area is given by (Art. 6.3.4).

$$A_{sv} = \frac{V - V_c}{f_{yv} h_0}$$

$$f_{yv} = \mathrm{Min}\,(360\mathrm{MPa}, f_y)$$

Verification of Circular RC Cross Section

The program verifies a reinforced concrete pile using the method of limit deformation (Art. 6.2.1). The maximum allowable strain of concrete in compression is 0.002 - 0.0033. The degree of reinforcement is checked using the formula:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

- **Column** - check for dominant compression (Art. 8.5.1, Art. 9.3.1)

$$\rho = \frac{4A_s}{\pi d^2}$$

for steel strength grade greater or equal to 500 ***MPa***

$$\rho_{min} = 0.005$$

for steel strength grade greater or equal to 400 ***MPa***

$$\rho_{min} = 0.0055$$

for steel strength grade less than 335 ***MPa***

$$\rho_{min} = 0.006$$

ρ_{min} is increased by 0.001 for concrete strength grade greater than **C60**

$$\rho_{max} = 0.05$$

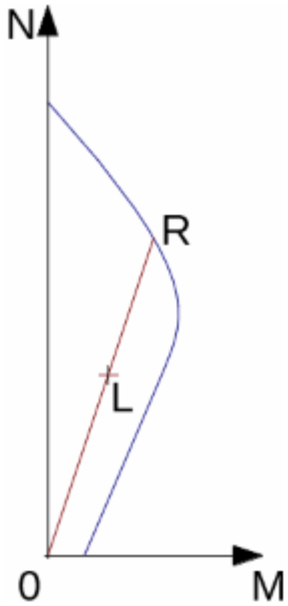
- **Beam** - check for dominant bending (Art. 8.5.1)

$$\rho = 0.5\frac{4A_s}{\pi d^2}$$

$$\rho_{min} = \mathrm{Max}\left(0.002; 0.45\frac{f_t}{f_y}\right)$$

where: d - pile diameter

A_s - reinforcement area



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L| / |0R|$. Where L is load and R is strength with prescribed eccentricity.

Shear

First, the program computes the ultimate shear strength of concrete V_c:

For flexural element (Art. 6.3.3, Art. 6.3.15):

$$V_c = 0.7 f_t (0.88d)(0.8d)$$

For compression element (Art. 6.3.12, Art. 6.3.15):

$$V_c = \frac{1.75}{\frac{M}{Vh_0} + 1} f_t (0.88d)(0.8d) + 0.07 \operatorname{Min}(N; 0.3 f_c A)$$

For tension element (Art. 6.3.14, Art. 6.3.15):

$$V_c = \frac{1.75}{\frac{M}{Vh_0} + 1} f_t (0.88d)(0.8d) - 0.2N$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength V_{max} (Art. 6.3.1, Art. 6.3.15) and strength of reinforced section V_s are checked (Art. 6.3.4, Art. 6.3.12, Art. 6.3.14, Art. 6.3.15).

for $h_0/b \leq 4$

$$V_{max} = 0.25 \beta_c f_c (0.88d)(0.8d)$$

for $h_0/b \geq 6$

$$V_{max} = 0.2 \beta_c f_c (0.88d)(0.8d)$$

$\beta_c =$ 1 for: ≤ **C50**

$\beta_c =$ 0.8 for: ≥ **C80**, intermediate values are obtained using linear interpolation method.

$$V_{cs} = V_c + f_{yv} \frac{A_{sv}}{s} (0.8d)$$

$$f_{yv} = \operatorname{Min}\,(360\text{MPa}, f_y)$$

Verification of Spread Footing for Punching Shear

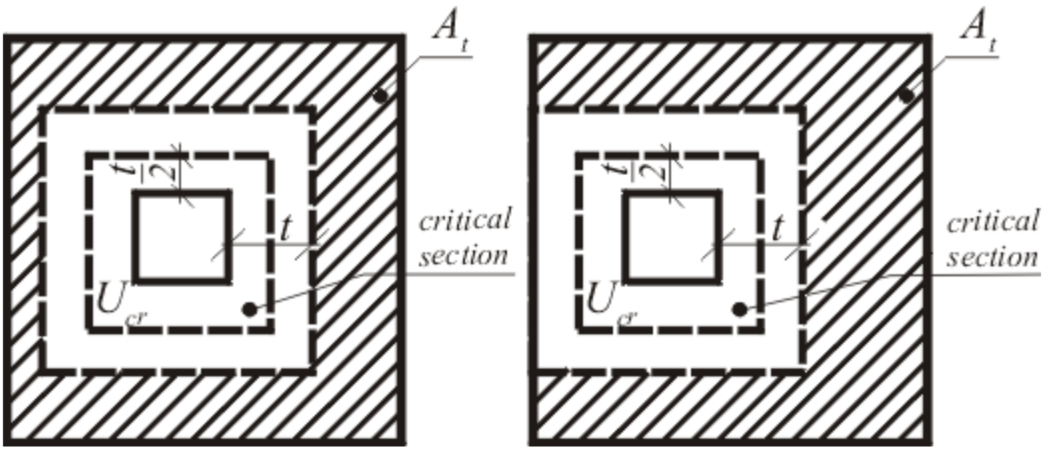
The program allows us to verify spread footing for punching shear or for the design of shear reinforcement. The critical section loaded in shear U_{cr} is distant from the column edge by one half of the footing thickness. It is loaded by the prescribed moments M_x, M_y and by the shear force F_l provided by:

$$F_l = \frac{QA_t}{A}$$

where: A - area of footing

Q - assigned vertical force developed in column

A_t - hatched area in fig.



Dimensioning of shear reinforcement area A_t

The program computes the maximal shear force F_l developed in the critical section (the influence of unbalanced bending moment is added according to Appendix F of standard GB50010-2010), the shear force transmitted by concrete with no shear reinforcement F_c (Art. 6.5.1), and the maximal allowable force F_{max} (Art. 6.5.3). The shear forces are related to the unit length of the critical section.

$$F_c = 0.7\beta_h f_t \eta h_0$$

$\beta_h = 0.9$ for $h \geq 2000mm$

$\beta_h = 1$ for $h \leq 800mm$, intermediate values are obtained using linear interpolation method.

$$\eta = \mathrm{Min}\left(\eta_1, \eta_2\right)$$
$$\eta_1 = 0.4 + \frac{1.2}{\beta_s}$$

where β_s is the size ratio of the long side and short side of the action area.

$$2 \leq \beta_s \leq 4$$
$$\eta_2 = 0.5 + \frac{\alpha_s h_0}{4u_m}$$

where α_s: 40 - for interior column

30 - for edge column

20 - for corner column

$$F_{max} = 1.2 f_t \eta h_0$$

For $F_l < F_c$ no shear reinforcement is needed.

For $F_l > F_c$ and $F_l < F_{max}$ the shear reinforcement must be introduced. The ultimate shear force is given by:

$$F_u = 0.5 f_t \eta h_0 + 0.8 f_{yv} A_{sbu} \sin\alpha$$
$$f_{yv} = \mathrm{Min}\left(360\mathrm{MPa}, f_y\right)$$

where: u_m - critical cross-section span

α - is angle of bends

A_{sbu} - area of bends in unit length of critical section

For $F_l > F_{max}$ the shear reinforcement cannot be designed. It is therefore necessary to increase the cross-section height.

Additional check according to article 8.2.9 of standard GB50007-2011 is done for narrow footing or strip footing.

$$V_c = 0.7\beta_h f_t b_0 h_0$$

where:

$$\beta_h = \left(\frac{800}{h_0}\right)^{\frac{1}{4}}$$

$$800\text{mm} \leq h_0 \leq 2000\text{mm}$$

b_0 is an average width of the footing.

Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment M. The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross-section. First, the program determines the depth of the compression zone as (Art. 6.2.10):

$$x = h_0 - \sqrt{h_0^2 - \frac{M}{0.5b\alpha_1 f_c}}$$

Providing the depth of compression zone is less than the allowable one ($x < \xi_b h_0$), the program determines the area of tensile reinforcement A_{st} from the expression:

$$A_{st} = \frac{\alpha_1 b x f_c}{f_y}$$

Providing the depth of compression zone is greater than the allowable one ($x > \xi_b h_0$), the program determines the areas of both compressive A_{sc} and tensile A_{st} reinforcement from the expressions:

$$A_{sc} = \frac{\dfrac{M}{h_0 - 0.5\xi_b h_0} - F_{c,max}}{f_y'}$$

$$A_{st} = \frac{F_{c,max} + A_{sc} f_y'}{f_y}$$

$$F_{c,max} = \alpha_1 b \xi_b h_0 f_c$$

The limit depth of compression zone $\xi_b h_0$ is found from (Art. 6.2.7):

$$\xi_b = \frac{\beta_1}{1 + \dfrac{f_y}{E_s \varepsilon_{cu}}}$$

The computed degree of reinforcement is checked using the following expressions (Art. 8.5.1):

$$\rho_{min} = \text{Max}\left(0.0015; 0.45\frac{f_t}{f_y}\right) \leq \rho$$

$$\rho = \frac{A_s}{bh_0}$$

Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and bends, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force transmitted by concrete V_c (Art. 6.3.3) and the maximum allowable shear force V_{max} (Art. 6.3.1).

$$V_c = 0.7\beta_h f_t b h_0$$

where:

$$\beta_h = \left(\frac{800}{h_0}\right)^{\frac{1}{4}}$$

$$800\text{mm} \leq h_0 \leq 2000\text{mm}$$

for $h_0/b \leq 4$

$$V_{max} = 0.25\beta_c f_c b h_0$$

for $h_0/b \geq 6$

$$V_{max} = 0.2\beta_c f_c b h_0$$

intermediate values are obtained using linear interpolation method

$\beta_c = 1$ for: ≤ **C50**

$\beta_c = 0.8$ for: ≥ **C80**, intermediate values are obtained using linear interpolation method.

As for stirrups, the necessary reinforcement area is given by (Art. 6.3.4):

$$A_{sv} = \frac{V - V_c}{f_{yv} h_0}$$

$$f_{yv} = \text{Min}\left(360\text{MPa}, f_y\right)$$

As for bends, the necessary reinforcement area is given by (Art. 6.3.5):

$$A_{sb} = \frac{V - V_c}{0.8 f_{yv} h_0 \sin \alpha_s}$$

NZS 3101-2006

This help contains the following computationals methods:

- Materials, coefficients, notation
- Verification of rectangular cross-section made of plain concrete
- Verification of rectangular RC cross-section
- Verification of circular RC cross-section
- Verification of spread footing for punching shear
- Design of longitudinal reinforcement for slabs
- Design of shear reinforcement for slabs

Materials, Coefficients, Notation

The following notation for material parameters is used:

f'_c - specified compressive strength of concrete

E_c - modulus of elasticity of concrete

f_y - lower characteristic yield strength of reinforcing steel

$$E_c = \left[3320\sqrt{f'_c} + 6900\right]$$

The characteristic compressive strength of concrete is the basic input parameter given by the class of concrete.

The most common notation for geometrical parameters:

b - cross-section width

h - cross-section depth

d - effective depth of cross section

z - lever arm (arm of internal forces)

All computations are carried out according to the theory of limit states.

Verification of Rectangular Cross Section Made of Plain Concrete

The cross-section is rectangular, loaded by the bending moment M^*, normal force N^* (applied in the cross-section centroid) and by the shear force V^*. The shear strength is provided by:

$$V^* \leq \phi V_n$$

where:

$$V_n = v_c.b.h$$

for cross sections with height smaller than 200mm

$$v_c = 0.17\sqrt{f_c'}$$

for cross sections with height greater than 400mm

$$v_c = 0.08\sqrt{f_c'}$$

intermediate values are obtained using linear interpolation method.

f_c' is limited to value $50MPa$.

$$\phi = 0.75$$

Strength of concrete cross-section subject to the combination of bending moment and normal force is derived from the following expressions depending on the normal force eccentricity e :

$$N^* \leq \phi N_n$$

Where N_n is determined as the greater of:

$$N_n = bx\alpha_1 f_c'$$

$$N_n = Min\left(\frac{bh0.36\sqrt{f_c'}}{\frac{6e}{h}-1}; \frac{bhf_c'}{\frac{6e}{h}+1}\right)$$

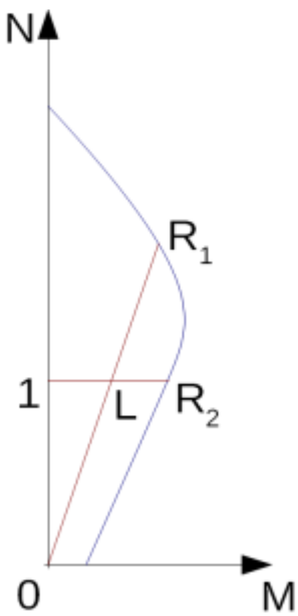
for $f_c' < 55MPa$ is $\alpha_1 = 0.85$

for concrete with greater strength is

$$\alpha_1 = \text{Max}(0.85 - 0.004(f_c' - 55); 0.75)$$

$$x = h - 2e$$

$$\phi = 0.6$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R_1|$ or $|1L|/|1R_2|$. Where L is load, R_1 is strength with prescribed eccentricity, and R_2 is strength with prescribed normal force.

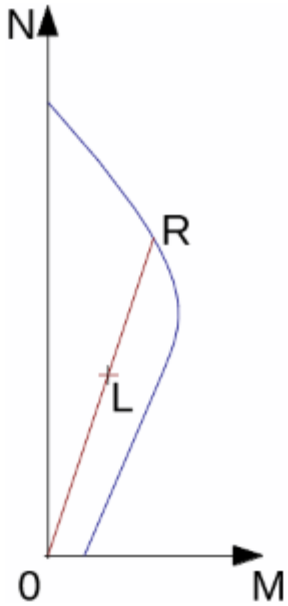
Verification of Rectangular RC Cross Section

The cross-section is rectangular, unilaterally reinforced, and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation. The maximum allowable strain of concrete in compression is 0.003. Compression reinforcement is not taken into account.

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} = \frac{\text{Max}(0.25\sqrt{f_c'}; 1.4)}{f_y} < \rho < 0.04 = \rho_{max}$$

$$\rho = \frac{A_s}{bd}$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L| / |0R|$. Where L is load and R is strength with prescribed eccentricity.

Bending without normal force

The cross-section is rectangular, reinforced on one side, and loaded by the bending moment M^*.

The ultimate moment is provided by:

$$M^* \leq \phi M_n$$
$$\phi = 0.85$$
$$M_n = \beta_1 c b \alpha_1 f_c'(d - 0.5\beta_1 c)$$
$$c = \frac{A_s f_y}{\beta_1 b \alpha_1 f_c'}$$

for $f_c' < 55\ MPa$ is $\alpha_1 = 0.85$

for concrete with greater strength is

$$\alpha_1 = \text{Max}(0.85 - 0.004(f_c' - 55); 0.75)$$

for $f_c' < 30\ MPa$ is $\beta_1 = 0.85$

for concrete with greater strength is

$$\beta_1 = \text{Max}(0.65; 0.85 - 0.008(f_c' - 30))$$

The program further checks whether the location of the neutral axis c is less than the limit location of neutral axis $0.75 * c_b$ given by:

$$c_b = \frac{0.003}{0.003 + \frac{f_y}{E_s}}$$

Shear

$$V^* \leq \phi V_n$$

where:

$$\phi = 0.75$$

First, the program computes the ultimate shear strength of concrete V_c.

$$V_c = v_c b d$$

for cross-sections with a height smaller than 200 mm

$$v_c = 0.17\sqrt{f_c'}$$

v_c is computed according to the following formulas for cross-sections with a height greater than 400 mm, intermediate values are obtained using a linear interpolation method.

$$v_c = k_n k_d v_b$$

$$k_n = 1 + \frac{N^*}{bhf_c'}$$

$$k_d = \text{Max}(0.9; \text{Min}(1; (400/d)^{0.25}))$$

$$v_b = \text{Min}(0.2; \text{Max}(0.08; 0.07 + 10\rho_w))\sqrt{f_c'}$$

where ρ_w is the degree of reinforcement and f_c' is limited to value 50 MPa.

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength V_{max} is checked.

$$V_{max} = \text{Min}(8\text{MPa}; 0.2f_c')bd$$

Next, the necessary reinforcement area is given by:

$$A_v = \frac{V^* - \phi V_c}{f_{yt} d} b$$

Verification of Circular RC Cross Section

The program verifies a reinforced concrete pile using the method of limit deformation. The maximum allowable strain of concrete in compression is 0.003.

The degree of reinforcement is checked using the formula:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

- **Pile**

$$\rho = \frac{4A_s}{\pi d^2}$$

$A_g <$ 0.5m^2 $\rho_{min} =$ 2.4$/f_y$

$A_g >$ 2m^2 $\rho_{min} =$ 1.2$/f_y$

where: A_g - cross-section area of the pile

intermediate values are calculated according to:

$$\rho_{min} = \frac{2.4}{f_y\sqrt{2A_g}}$$

$$\rho_{max} = 0.08$$

- **Column** - check for dominant compression

$$\rho = \frac{4A_s}{\pi d^2}$$

$$\rho_{min} = 0.008$$

$$\rho_{max} = 0.08$$

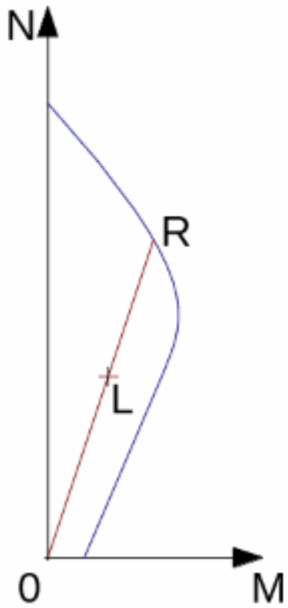
- **Beam** - check for dominant bending

$$\rho = 0.5\frac{4A_s}{\pi d^2}$$

$$\rho_{min} = \frac{\mathrm{Max}(0.25\sqrt{f_c'}; 1.4)}{f_y}$$
$$\rho_{max} = 0.04$$

where: d - pile diameter

A_s - cross-sectional area of reinforcement



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R|$. Where L is load and R is strength with prescribed eccentricity.

Shear

$$V^* \leq \phi V_n$$

where:

$$\phi = 0.75$$

First, the program computes the ultimate shear strength of concrete V_c .

$$V_c = v_c b d$$

for cross-sections with height smaller than 200mm

$$v_c = 0.17\sqrt{f_c'}$$

v_c is computed according to the following formulas for cross-sections with height greater than 400mm, intermediate values are obtained using a linear interpolation method.

$$v_c = k_n k_d v_b$$
$$k_n = 1 + \frac{N^*}{bhf_c'}$$
$$k_d = \mathrm{Max}(0.9; \mathrm{Min}(1; (400/d)^{0.25}))$$
$$v_b = \mathrm{Min}(0.2; \mathrm{Max}(0.08; 0.07 + 10\rho_w))\sqrt{f_c'}$$

where ρ_w is the degree of reinforcement and f_c' is limited to value 50 MPa .

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength V_{max} and strength of reinforced section V_s are checked.

$$V_{max} = \mathrm{Min}(8\mathrm{MPa}; 0.2f_c')bd$$
$$V_{cs} = \phi V_c + \phi A_v f_{yt}(0.8d)$$

Verification of Spread Footing for Punching Shear

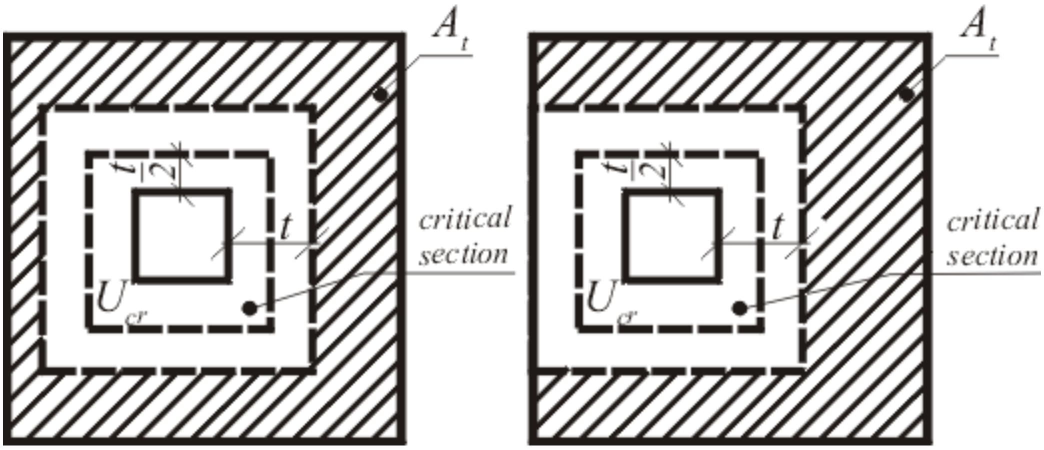
The program allows us to verify spread footing for punching shear or for the design of shear reinforcement. The critical section loaded in shear b_o is distant from the column edge by one half of the footing thickness. It is loaded by the prescribed moments M^*_x, M^*_y and by the shear force V^* provided by:

$$V^* = \frac{V A_t}{A}$$

where: A - area of footing

V - assigned vertical force developed in column

A_t - hatched area in fig.



Dimensioning of shear reinforcement area A_t

The program computes the maximal shear force V^* developed in the critical section, the shear force transmitted by concrete with no shear reinforcement V_c, and the maximal allowable force V_{max}:

$$V_c = v_c b_o d$$

where:

$$v_c = \frac{1}{6} k_{ds} \left(1 + \mathrm{Min}\left(1; \frac{2}{\beta_c}; \frac{\alpha_s d}{b_o}\right)\right) \sqrt{f'_c}$$

where α_s: 20 - for interior column

15 - for edge column

10 - for corner column

β_c c is the ratio of the long side to the short side of the critical section

$$k_{ds} = \sqrt{\frac{0.2}{d}} \cdots \langle 0.5; 1 \rangle$$

$$V_{max} = 0.5 \sqrt{f'_c}$$

For $V^* < \phi V_c$ no shear reinforcement is needed.

For $V^* > \phi V_c$ and $V^* < \phi V_{max}$ the shear reinforcement must be introduced. The ultimate shear force is given by:

$$V_n = \mathrm{Min}\left(V_c; \frac{1}{6}\sqrt{f'_c}\right) + A_v f_{yv} sin\alpha$$

where: b_o - critical cross-section span

α - is angle of bends

A_v - overall area of bends in footing

For $V^* > \phi V_{max}$ the shear reinforcement cannot be designed. It is therefore necessary to increase the cross-section height.

Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment M^*. The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross-section. First, the program determines the location of the neutral axis as:

$$c = \frac{d - \sqrt{d^2 - \dfrac{M^*}{0.5b\alpha_1 f_c'}}}{\beta_1}$$

Providing the location of the neutral axis is less than the allowable one ($c < 0.75c_b$), the program determines the area of tensile reinforcement A_{st} from the expression:

$$A_{st} = \frac{\beta_1 \alpha_1 bc f_c'}{f_y}$$

Providing the location of the neutral axis is greater than the allowable one ($c > 0.75c_b$), the program determines the areas of both compressive A_{sc} and tensile A_{st} reinforcement from the expressions:

$$A_{sc} = \frac{M^* - F_{c,max}(d - 0.5\beta_1 0.75c_b)}{f_y z}$$

$$A_{st} = \frac{F_{c,max} + A_{sc} f_y}{f_y}$$

$$F_{c,max} = \beta_1 \alpha_1 b 0.75 c_b f_c'$$

The limit location of the neutral axis is found from:

$$c_b = \frac{0.003}{0.003 + \dfrac{f_y}{E_s}}$$

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} = \frac{\mathrm{Max}(0.25\sqrt{f_c'}; 1.4)}{f_y} < \rho < 0.04 = \rho_{max}$$

If the maximum degree of total reinforcement ρ_{max} is exceeded, the program informs the user that the longitudinal reinforcement cannot be designed for a given cross-section.

Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and bends, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force transmitted by concrete V_c and the maximum allowable shear force V_{max}.

$$V^* \leq \phi V_n$$

where:

$$\phi = 0.75$$

$$V_c = v_c bd$$

for cross-sections with a height smaller than 200mm

$$v_c = 0.17\sqrt{f_c'}$$

v_c is computed according to the following formulas for cross-sections with a height greater than 400mm, intermediate values are obtained using a linear interpolation method.

$$v_c = k_d v_b$$

$$k_d = \mathrm{Max}(0.9; \mathrm{Min}(1; (400/d)^{0.25}))$$

$$v_b = \mathrm{Min}(0.2; \mathrm{Max}(0.08; 0.07 + 10\rho_w))\sqrt{f_c'}$$

where ρ_w is the degree of reinforcement and f_c' is limited to value $50MPa$.

$$V_{max} = \mathrm{Min}(8\mathrm{MPa}; 0.2f_c^{'})bd$$

As for stirrups, the necessary reinforcement area is given by:

$$A_v = \frac{V^* - \phi V_c}{f_{yt}d}$$

As for bends, the necessary reinforcement area is given by:

$$A_v = \frac{V^* - \phi V_c}{f_{yt}dsin\alpha}$$

CSA A23.3-14

This help contains the following methods:

- Materials, coefficients, notation
- Verification of rectangular cross-section made of plain concrete
- Verification of rectangular RC cross-section
- Verification of circular RC cross-section
- Verification of spread footing for punching shear
- Design of longitudinal reinforcement for slabs
- Design of shear reinforcement for slabs

Materials, Coefficients, Notation

The following notation for material parameters is used:

f'_c - specified compressive strength of concrete

E_c - modulus of elasticity of concrete

f_r - modulus of rupture of concrete

f_y - specified yield strength of reinforcing steel

$$E_c = \left[3320\sqrt{f_c'} + 6900\right]$$

$$f_r = 0.6\sqrt{f_c'}$$

The most common notation for geometrical parameters:

b - cross-section width

h - cross-section depth

d - effective depth of cross section

Verification of Rectangular Cross Section Made of Plain Concrete

The cross-section is rectangular, loaded by the bending moment M_f, normal force P_f (applied in the cross-section centroid), and by the shear force V_f. The cross-section bearing capacity subjected to bending moment is given by (Art. 22.6.5):

$$M_r = \frac{bh^2}{6}0.37\phi_c\frac{f_r}{0.6}$$

where (Art. 8.4.2, Art. 8.6.4):

$$\phi_c = 0.65$$

$$f_r = 0.6\sqrt{f_c'}$$

The shear strength is provided by (Art. 22.6.5):

$$V_r = \frac{2}{3}0.18\phi_c\sqrt{f_c'}bh$$

where:

Strength of concrete cross-section subject to the combination of bending moment and normal force is derived from the following expressions depending on the normal force eccentricity $e \geq 0.1h$ (Art. 22.6.5):

As the greater of:

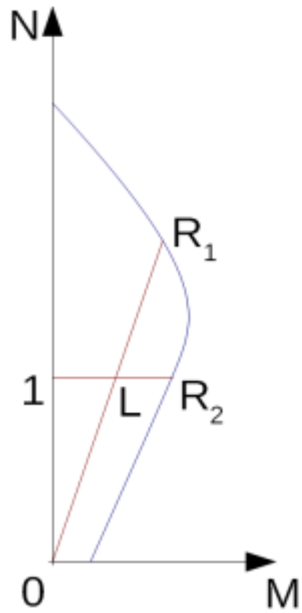
$$P_r = 0.45\alpha_1\phi_c f'_c b(h-2e)$$

$$P_r = \text{Min}\left(\frac{bh0.37\phi_c\frac{f_r}{0.6}}{\frac{6e}{h}-1};\frac{bh0.75\phi_c f'_c}{\frac{6e}{h}+1}\right)$$

where (Art. 8.4.2):

$$f_r = 0.6\sqrt{f'_c}$$

$$e = \left|\frac{M_f}{P_f}\right|$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R_1|$ or $|1L|/|1R_2|$. Where L is load, R_1 is strength with prescribed eccentricity, and R_2 is strength with prescribed normal force.

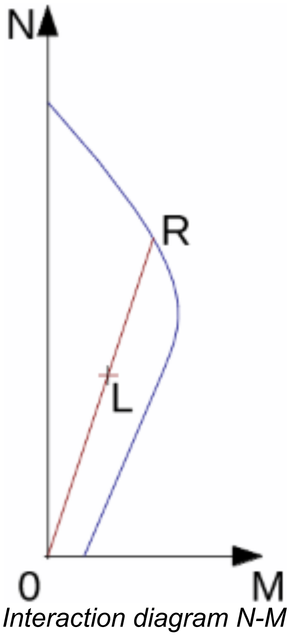
Verification of Rectangular RC Cross Section

The cross-section is rectangular, unilaterally reinforced, and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation (Art. 10.1). The maximum allowable strain of concrete in compression is 0.0035. Compression reinforcement is not taken into account.

The computed degree of reinforcement is checked using the following expressions (Art. 10.5.1.2):

$$\rho_{min} = \frac{0.2\sqrt{f'_c}}{f_y} < \rho < 0.04 = \rho_{max}$$

$$\rho = \frac{A_s}{bd}$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R|$. Where L is load and R is strength with prescribed eccentricity.

Bending without normal force

The cross-section is rectangular, reinforced on one side, and loaded by the bending moment M_f.

The ultimate moment is provided by (Art. 10.1.7, 8.4.2, 8.4.3):

$$M_r \leq M_f$$
$$M_r = \beta_1 c b \alpha_1 \phi_c f'_c (d - 0.5\beta_1 c)$$
$$c = \frac{A_s \phi_s f_y}{\beta_1 b \alpha_1 \phi_c f'_c}$$
$$\phi_c = 0.65$$
$$\phi_s = 0.85$$
$$\alpha_1 = \mathrm{Max}(0.85 - 0.0015 f'_c; 0.67)$$
$$\beta_1 = \mathrm{Max}(0.65; 0.85 - 0.008(f'_c - 30))$$
$$\beta_1 = \mathrm{Max}(0.97 - 0.0025 f'_c; 0.67)$$

The program further checks whether the location of the neutral axis c is less than the limit location of neutral axis c_{max} given by (Art. 10.5.2):

$$c_{max} = \frac{700}{700 + f_y} d$$

Shear

$$V_f \leq V_r$$

First, the program computes the ultimate shear strength of concrete V_c (Art. 11.3.4):

$$V_c = 0.18 \phi_c \sqrt{f'_c} b d_v$$

where

$$\sqrt{f'_c} \leq 8\mathrm{MPa}$$
$$d_v = 0.9d$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength V_{max} is checked (Art. 11.3.3):

$$V_{r,max} = 0.25 \phi_c f'_c b d_v$$

Next, the necessary reinforcement area is given by (Art. 11.3.5.1):

$$A_v = \frac{V_f - V_c}{\phi_s f_y d_v} b$$

Verification of Circular RC Cross Section

The program verifies a reinforced concrete pile using the method of limit deformation (Art 10.1). The maximum allowable strain of concrete in compression is 0.0035.

The degree of reinforcement is checked using the formula:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

- **Column** - check for dominant compression (Art. 10.9.1, 10.9.2)

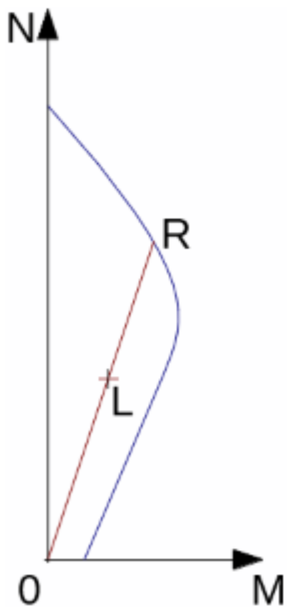
$$\rho = \frac{4A_s}{\pi d^2}$$
$$\rho_{min} = 0.01$$
$$\rho_{max} = 0.08$$

- **Beam** - check for dominant bending (Art. 10.5.1)

$$\rho = 0.5\frac{4A_s}{\pi d^2}$$
$$\rho_{min} = \frac{0.2\sqrt{f_c'}}{f_y}$$
$$\rho_{max} = 0.04$$

where: d - pile diameter

A_s - cross sectional area of reinforcement



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R|$. Where L is load and R is strength with prescribed eccentricity.

Shear

$$V_f \leq V_r$$

First, the program computes the ultimate shear strength of concrete V_c. Formulas are from Art. 11.3.4, where the section width (b) is replaced by $0.88\times d$ and effective depth (d_v) is replaced $0.8\times0.9\times d$.

$$V_c = 0.18\phi_c\sqrt{f_c'}0.6336d^2$$

where

$$\sqrt{f_c'} \leq 8\text{MPa}$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength V_{max} and strength of reinforced section V_s are checked (Art. 11.5.1).

$$V_{r,max} = 0.25\phi_c f_c' 0.6336d^2$$

$$V_s = \frac{A_v \phi_s f_y 0.72d}{s}$$

Verification of Spread Footing for Punching Shear

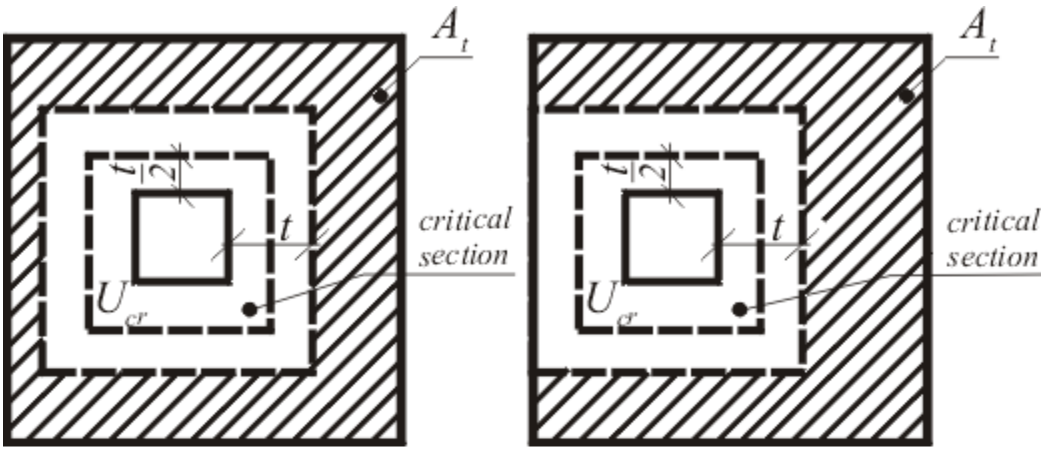
The program allows us to verify spread footing for punching shear or for the design of shear reinforcement. The critical section loaded in shear b_o is distant from the column edge by one half of the footing thickness. It is loaded by the prescribed moments M_x, M_y and by the shear force V_f provided by:

$$V_f = \frac{VA_t}{A}$$

where: A - area of footing

V - assigned vertical force developed in column

A_t - hatched area in fig.



Dimensioning of shear reinforcement area A_t

The program computes the maximal shear force V_f developed in the critical section, the shear force transmitted by concrete with no shear reinforcement V_c as a minimum value (Art. 13.3.4.1):

$$V_c = \left(1 + \frac{2}{\beta_c}\right) 0.19\phi_c \sqrt{f_c'} b_o d$$

where β_c is the ratio of a long side to short side od column.

$$V_c = \left(\frac{\alpha_s d}{b_o} + 0.19\right) \phi_c \sqrt{f_c'} b_o d$$

where α_s = 4 - inner column

3 - edge column

2 - corner column

$$V_c = 0.38\phi_c \sqrt{f_c'} b_o d$$

where

$$\sqrt{f_c'} \leq 8\text{MPa}$$

and the maximal allowable force V_{max} (Art. 13.3.9.2):

$$V_{max} = 0.55\phi_c \sqrt{f_c'} b_o d$$

For $V_f < V_c$ no shear reinforcement is needed.

For $V_f > V_c$ and $V_f < V_{max}$ the shear reinforcement must be introduced. The ultimate shear force is given by (Art. 13.3.9.4):

$$V_s = \left(0.19\phi_c\sqrt{f_c'}b_o d + A_{vs} f_{yv}\sin\alpha\right)$$

where: b_o - critical cross-section span

α - is angle of bends

A_v - overall area of bends in footing

For $V_f > V_{max}$ the shear reinforcement cannot be designed. It is therefore necessary to increase the cross-section height.

Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment M_f. The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross-section. First, the program determines the location of the neutral axis as (Art 10.1):

$$c = \frac{d - \sqrt{d^2 - \frac{M_f}{0.5b\alpha_1 f_c'}}}{\beta_1}$$

Providing the location of the neutral axis is less than the allowable one (c_{max}), the program determines the area of tensile reinforcement A_{st} from the expression:

$$A_{st} = \frac{\beta_1\alpha_1 bc f_c'}{f_y}$$

Providing the location of the neutral axis is greater than the allowable one (c_{max}), the program determines the areas of both compressive A_{sc} and tensile A_{st} reinforcement from the expressions:

$$A_{sc} = \frac{M_f - F_{c,max}(d - 0.5\beta_1 c_{max})}{f_y z}$$

$$A_{st} = \frac{F_{c,max} + A_{sc} f_y}{f_y}$$

$$F_{c,max} = \beta_1\alpha_1 b c_{max} f_c'$$

The limit location of neutral axis c_{max} is found from (Art. 10.5.2):

$$c_{max} = \frac{700}{700 + f_y} d$$

The computed degree of reinforcement is checked using the following expressions (Art. 10.5.1.2):

$$\rho_{min} = \frac{0.2\sqrt{f_c'}}{f_y} < \rho < 0.04 = \rho_{max}$$

$$\rho = \frac{A_s}{bd}$$

If the maximum degree of total reinforcement ρ_{max} is exceeded, the program informs the user that the longitudinal reinforcement cannot be designed for a given cross-section.

Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and bends, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force transmitted by concrete V_c (Art. 11.3.4) and the maximum allowable shear force V_{max} (Art. 11.3.3).

$$V_f \le V_r$$

where:

$$V_c = 0.18\phi_c\sqrt{f_c'}bd_v$$
$$\sqrt{f_c'} \leq 8\text{MPa}$$
$$d_v = 0.9d$$
$$V_{r,max} = 0.25\phi_c f_c' bd_v$$

As for stirrups, the necessary reinforcement area is given by (Art. 11.3.5.1):

$$A_v = \frac{V_f - V_c}{\phi_s f_y d_v}$$

As for bends, the necessary reinforcement area is given by (Art. 11.3.5.1):

$$A_v = \frac{V_f - V_c}{\phi_s f_y d_v \sin\alpha}$$

NBR 6118-2014

This help contains the following methods:

- Materials, coefficients, notation
- Verification of rectangular cross-section made of plain concrete
- Verification of rectangular RC cross-section
- Verification of circular RC cross-section
- Verification of spread footing for punching shear
- Design of longitudinal reinforcement for slabs
- Design of shear reinforcement for slabs

Materials, Coefficients, Notation

The following notation for material parameters is used:

f_{ck} - characteristic value of cylindrical compressive strength of concrete

f_{cd} - design compressive strength of concrete

f_{ctm} average value of tensile strength of concrete

f_{ctd} - design axial tensile strength of concrete

f_{yk} - characteristic tension strength of steel bar

f_{yd} - design tension strength of steel bar

The characteristic compressive strength of concrete is the basic input parameter given by the class of concrete - it serves to derive the remaining coefficients of reliability (Ch. 8.2).

$$f_{cd} = \frac{f_{ck}}{\gamma_c}$$
$$\gamma_c = 1.4$$
$$f_{yd} = \frac{f_{yk}}{\gamma_s}$$
$$\gamma_s = 1.15$$

for $f_{ck} \leq 50\ MPa$

$$E_{ci} = \alpha_E 5600\sqrt{f_{ck}}$$

for $f_{ck} > 50\ MPa$

$$E_{ci} = \alpha_E 21500\sqrt[3]{0.1f_{ck} + 1.25}$$
$$\alpha_E = 1$$
$$E_{cs} = \alpha_i E_{ci}$$
$$\alpha_i = 0.8 + 0.2\frac{f_{ck}}{80} \leq 1$$

$$G = \frac{E_{cs}}{2.4}$$

for $f_{ck} \leq$ 50 MPa

$$f_{ctm} = 0.3 f_{ck}^{2/3}$$

for $f_{ck} >$ 50 MPa

$$f_{ctm} = 2.12 \ln(1 + 0.11 f_{ck})$$

The most common notation for geometrical parameters:

b - cross-section width

h - cross-section depth

d - effective depth of cross section

z - lever arm (arm of internal forces)

Verification of Rectangular Cross Section Made of Plain Concrete

The cross-section is rectangular, loaded by the bending moment M_{Sd}, normal force N_{Sd} (applied in the cross-section centroid) and by the shear force V_{Sd}. The cross-section bearing capacity subjected to bending moment is given by (Art. 24.5.2):

$$M_{Rd} = \frac{bh^2}{6} f_{ctd}$$

where:

$$f_{ctd} = 0.85(0.7 f_{ctm}) / \gamma_C$$
$$\gamma_C = 1.68$$

The shear strength is provided by (Art. 24.5.2.3, 24.5.5.1):

$$V_{Rd} = \tau_{wRd} A_{cc}$$

where: A_{cc} - compressed area of concrete

$$\tau_{wRd} = 0.3 f_{ctd} \,\mathrm{Min}\left(1 + 3\frac{N_{Sd}}{bh}; 2\right)$$

Strength of concrete cross-section subject to the combination of bending moment and normal force is derived from the following expressions depending on the normal force eccentricity e (Art. 24.5.2):

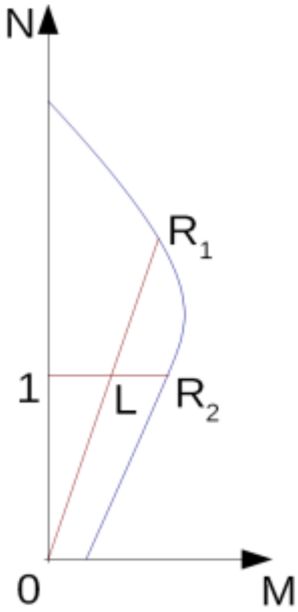
As the greater of:

$$N_{Rd} = f_{cd} b(h - 2e)$$

$$N_{Rd} = \mathrm{Min}\left(\frac{bh f_{ctd}}{\frac{6e}{h} - 1}; \frac{bh f_{cd}}{\frac{6e}{h} + 1}\right)$$

where:

$$f_{cd} = 0.85 f_{ck} / \gamma_C$$
$$f_{ctd} = 0.85(0.7 f_{ctm}) / \gamma_C$$
$$\gamma_C = 1.68$$
$$e = \left|\frac{M_{Sd}}{N_{Sd}}\right|$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L| / |0R_1|$ or $|1L| / |1R_2|$. Where L is load, R_1 is strength with prescribed eccentricity, and R_2 is strength with prescribed normal force.

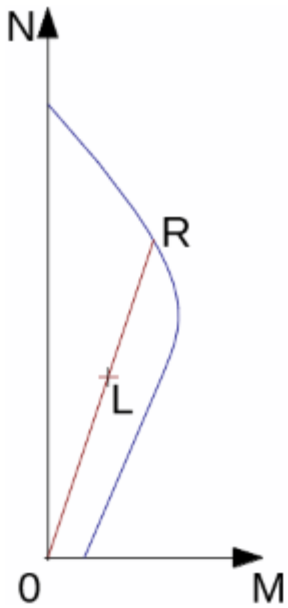
Verification of Rectangular RC Cross Section

The cross-section is rectangular, unilaterally reinforced, and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation (Art. 17.2). The maximum allowable strain of concrete in compression is 0.0035 (Art. 8.2.10.1). Compression reinforcement is not taken into account.

The computed degree of reinforcement is checked using the following expressions (Art. 17.3.5.2.1, 17.3.5.2.4, ρ_{min} taken from table 17.3):

$$\rho_{min} < \rho < 0.04 = \rho_{max}$$

$$\rho = \frac{A_s}{bd}$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L| / |0R|$. Where L is load and R is strength with prescribed eccentricity.

Bending without normal force

The cross-section is rectangular, reinforced on one side, and loaded by the bending moment M_{Sd}.

The ultimate moment is provided by (Art. 17.2.2):

$$M_{Sd} \leq M_{Rd}$$
$$M_{Rd} = \lambda x b \alpha_c f_{cd}(d - 0.5\lambda x)$$
$$x = \frac{A_s f_{yd}}{\lambda b \alpha_c f_{cd}}$$
$$\lambda = 0.8; f_{ck} \leq 50\text{MPa}$$
$$\lambda = 0.8 - (f_{ck} - 50)/400; f_{ck} > 50\text{MPa}$$
$$\alpha_c = 0.85; f_{ck} \leq 50\text{MPa}$$
$$\alpha_c = 0.85[1 - (f_{ck} - 50)/200]; f_{ck} > 50\text{MPa}$$

The program further checks whether the location of the neutral axis x is less than the limit location of neutral axis x_{max} given by:

$$x_{max} = \frac{\epsilon_{cu}}{\epsilon_{cu} + \epsilon_{yd}} d$$

Shear

$$V_{Sd} \leq V_{Rd}$$

First, the program computes the ultimate shear strength of concrete V_{Rd1} (Art. 17.4.2.2 - Model I):

$$V_{Rd1} = 0.6 f_{ctd} b_w d$$

where

$$f_{ctd} = f_{ctk,inf}/\gamma_C$$
$$f_{ctk,inf} = 0.7 f_{ctm}$$
$$\gamma_C = 1.4$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength V_{Rd2} is checked (Art. 17.4.2.2):

$$V_{Rd2} = 0.27\alpha_{v2} f_{cd} b_w d$$
$$\alpha_{v2} = (1 - f_{ck}/250)$$

Next, the necessary reinforcement area is given by (Art. 17.4.2.2):

$$A_{sw} = \frac{V_{Sd} - V_{Rd1}}{0.9 f_{ywd} d} b$$

Verification of Circular RC Cross Section

The program verifies a reinforced concrete pile using the method of limit deformation (Art 17.2). The maximum allowable strain of concrete in compression is 0.0035.

The degree of reinforcement is checked using the formula:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

- **Column** - check for dominant compression (Art. 17.3.5.3)

$$\rho = \frac{4A_s}{\pi d^2}$$
$$\rho_{min} = 0.15 \frac{N_d}{f_{yd} A_c} \geq 0.004$$
$$\rho_{max} = 0.08$$

- **Beam** - check for dominant bending (Art. 17.3.5.2)

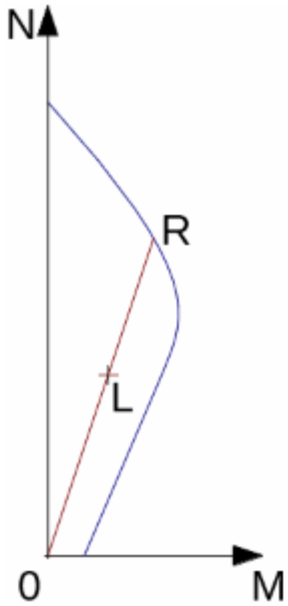
$$\rho = 0.5 \frac{4A_s}{\pi d^2}$$

ρ_{min} obtained from table 17.3.

$$\rho_{max} = 0.04$$

where: d - pile diameter

A_s - cross sectional area of reinforcement



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L| / |0R|$. Where L is load and R is strength with prescribed excentricity.

Shear

$$V_{Sd} \leq V_{Rd}$$

First, the program computes the ultimate shear strength of concrete V_{Rd1}. Formulas are from Art. 17.4.2.2 - Model I, where the section width (b_w) is replaced by 0.88d and effective depth (d) is replaced 0.8* 0.9d.

$$V_{Rd1} = 0.6 f_{ctd} 0.6336 d^2$$

where

$$f_{ctd} = f_{ctk,inf} / \gamma_C$$
$$f_{ctk,inf} = 0.7 f_{ctm}$$
$$\gamma_C = 1.4$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength V_{max} and strength of reinforced section V_s are checked (Art. 17.4.2.2).

$$V_{Rd2} = 0.27 \alpha_{v2} f_{cd} 0.6336 d^2$$
$$\alpha_{v2} = (1 - f_{ck}/250)$$
$$V_{Rd3} = V_{Rd1} + \frac{A_{sw} 0.9 f_{ywd} 0.72 d}{s}$$

Verification of Spread Footing for Punching Shear

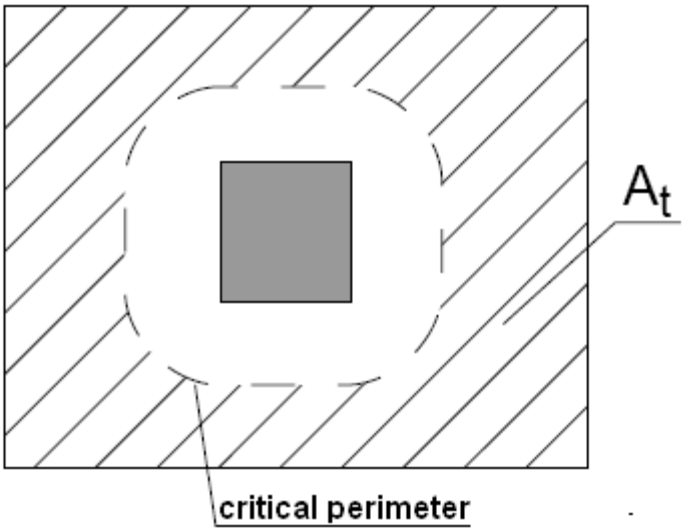
It is loaded by the prescribed moments M_{Sdx}, M_{Sdy} and by the shear force F_{Sd} provided by:

$$F_{Sd} = \frac{V A_t}{A}$$

where: A - area of footing

V - assigned vertical force developed in column

A_t - hatched area in fig.



Dimensioning of shear reinforcement area A_t

The program constructs **control sections** at **distances "a"** from $0{,}5d$ to $2d$ in case of **footing without shear reinforcement**. In case of **reinforced footing**, the distances are from $0{,}5d$ to $4d$, where d **is the effective depth of footing**. The shear reinforcement is considered in control sections, which are in the distance of less than $2d$ from the column. The control sections are considered in intervals of $0{,}25d$.

The load stress τ_{Sd} in each control section is found using 19.5.2,

$$\tau_{Sd} = \frac{F_{Sd}}{ud} + \frac{K M_{Sd}}{W_p d}$$

$$W_p = \int_0^u |e| \, \mathrm{d}l$$

K is obtained from table 19.2

the punching shear resistance of footing without shear reinforcement τ_{Rd1} follows from 19.5.3.2

$$\tau_{Rd1} = 0.13(1 + \sqrt{20/d})(100\rho f_{ck})^{(1/3)} 2d/a$$

$$\rho = \sqrt{\rho_x \rho_y}$$

and if necessary the punching shear resistance of reinforced footing τ_{Rd3} is given by 19.4.2, 15.5.4.

$$\tau_{Rd3} = 0.10(1 + \sqrt{20/d})(100\rho f_{ck})^{(1/3)} 2d/a + 0.75\frac{A_{sw} f_{ywd} \sin\alpha}{ud}$$

Furthermore, the **compression chord resistance** at the column perimeter τ_{Rd2} is calculated according to 19.5.3.1. τ_{Rd2} depends on column dimensions and the footing thickness.

$$\tau_{Rd2} = 0.27\alpha_v f_{cd}$$

$$\alpha_{v2} = (1 - f_{ck}/250)$$

For $\tau_{Sd} < \tau_{Rd1}$ no shear reinforcement is needed.

For $\tau_{Sd} > \tau_{Rd1}$ and $\tau_{Sd} < \tau_{Rd2}$ the shear reinforcement must be introduced.

For $\tau_{Sd} > \tau_{Rd2}$ the shear reinforcement cannot be designed. It is therefore necessary to increase the cross-section height.

The control section with the ratio of load and resistance with the highest utilization is considered as critical and marked in the program.

Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment M_{Sd}. The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross section. First, the program determines the location of neutral axis as (Art 17.2):

$$x = \frac{d - \sqrt{d^2 - \frac{M_f}{0.5b\alpha_c f_{cd}}}}{\lambda}$$

Providing the location of neutral axis is less than the allowable one (x_{max}), the program determines the area of tensile reinforcement A_{st} from the expression:

$$A_{st} = \frac{\lambda\alpha_c b x f_{cd}}{f_{yd}}$$

Providing the location of neutral axis is greater than the allowable one (x_{max}), the program determines the areas of both compressive A_{sc} and tensile A_{st} reinforcement from the expressions:

$$A_{sc} = \frac{M_{Sd} - F_{c,max}(d - 0.5\lambda x_{max})}{f_{yd} z}$$

$$A_{st} = \frac{F_{c,max} + A_{sc} f_{yd}}{f_{yd}}$$

$$F_{c,max} = \lambda\alpha_c b x_{max} f_{cd}$$

$$\lambda = 0.8; f_{ck} \leq 50\text{MPa}$$

$$\lambda = 0.8 - (f_{ck} - 50)/400; f_{ck} > 50\text{MPa}$$

$$\alpha_c = 0.85; f_{ck} \leq 50\text{MPa}$$

$$\alpha_c = 0.85[1 - (f_{ck} - 50)/200]; f_{ck} > 50\text{MPa}$$

The limit location of neutral axis x_{max} is found from:

$$x_{max} = \frac{\epsilon_{cu}}{\epsilon_{cu} + \epsilon_{yd}} d$$

The computed degree of reinforcement is checked using the following expressions (Art. 10.5.1.2):

$$\rho_{min} < \rho < 0.04 = \rho_{max}$$

$$\rho = \frac{A_s}{bd}$$

If the maximum degree of total reinforcement ρ_{max} is exceeded, the program informs the user that the longitudinal reinforcement cannot be designed for a given cross-section.

Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and bends, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force transmitted by concrete V_{Rd1} (Art. 17.4.2.2 - Model I) and the maximum allowable shear force V_{Rd2} (Art. 17.4.2.2).

$$V_{Sd} \leq V_{Rd}$$

where:

$$V_{Rd1} = \tau_{Rd} k(1.2 + 40\rho_1) b_w d$$

$$\tau_{Rd} = 0.25 f_{ctd}$$

$$f_{ctd} = f_{ctk,inf}/\gamma_C$$

$$f_{ctk,inf} = 0.7 f_{ctm}$$

$$\gamma_C = 1.4$$

$$V_{Rd2} = 0.27\alpha_{v2} f_{cd} b_w d$$

$$\alpha_{v2} = (1 - f_{ck}/250)$$

As for stirrups, the necessary reinforcement area is given by (Art. 17.4.2.2):

$$A_{sw} = \frac{V_{Sd} - V_{Rd1}}{0.9 f_{ywd} d}$$

As for bends, the necessary reinforcement area is given by (Art. 17.4.2.2):

$$A_{sw} = \frac{V_{Sd} - V_{Rd1}}{0.9 f_{ywd} d \sin\alpha}$$

Steel Cross Sections Verification

Verification of steel cross sections is carried out for two cases of load:

1. for the maximum value of bending moment and the corresponding shear force ($M_{max} + Q$)
2. for the maximum shear force and the corresponding bending moment ($Q_{max} + M$)

In both cases, the load enters to the assessment with an influence of normal force, which is defined separately. Its value is identical for both of load cases. Internal forces are, prior to analysis, pre-multiplied by the reduction coefficient of bearing capacity. This coefficient represents the degree of uncertainty of the determination of theoretical values of internal forces and as thus introduces into the analysis with such values certain reliability. The value of this coefficient is determined solely by the user.

The "**Sheeting Check**" program exploits following types of analyses for steel cross-section check:

- Verification according to EN 1993-1-1 (EC 3)
- Verification according to CSN 73 1401
- Verification according to a safety factor
- Verification according to limit states
- Verification according to GB 50017-2003

Each cross-section is checked for three types of load:

1. Check for bending moment and normal force

The analysis checks the normal stress σ developed at the edge of cross-section given by:

$$\sigma = \frac{M}{W} + \frac{N}{A}$$

where:
- M - bending moment
- W - elastic modulus of cross-section
- N - normal force
- A - area of cross-section

2. Check for shear

The analysis checks the shear stress τ at the cross-section center of gravity written as:

$$\tau = \frac{Q\,S}{I\,t}$$

where:
- Q - shear force
- S - 1st moment of area
- I - moment of inertia
- t - width (thickness) of cross-section at its center of gravity

3. Check for state of plane stress for the combination of stresses σ_1 and τ_1 at the point of critical load

Equivalent stress for plane stress conditions is defined as:

$$\sigma_k = \sqrt{\sigma_1^2 + 3\tau_1^2}$$

All verifications are carried out assuming elastic response of the material, plasticity is not taken into consideration.

Verification of steel I-sections

Internal forces provide by the "**Sheeting Check**" program are considered per 1 m run of the structure width. Therefore, the units of shear force Q are kN/m and of bending moments M are kNm/m. For dimensioning of individual I-section these forces are, prior to verification analysis, automatically multiplied by their spacing a [m] to get their values at the cross-section center of gravity, i.e. shear force Q in kN and bending moment M in kNm. The normal stress σ is checked at the outer face of flange. The shear stress τ is checked at the center of gravity, thus at the center of web height. The equivalent stress σ_k is checked in the web at the flange-web connection (cut 1).

Verification of sheet pile wall

The verification analysis is carried out for a wall section of a unit length. All cross sectional parameters are therefore determined not for individual sheet piles, but for a wall section of a unit length. The normal stress σ is checked at the outer face of back of sheet piles. The shear stress τ is checked at the web center of gravity, thus for sheet piles of **U** shape at the location of locks and for sheet piles of **Z** shape at the center of inclined sheet pile webs. The equivalent stress σ_k is checked in the sheet pile web at the location of connection of the back of sheet piles (cut 1).

EN 1993-1-1 (EC3)

Check for bending and stress caused by normal force

The design resistance for bending is given by:

$$M_{c,Rd} = \frac{W f_y}{\gamma_{M0}}$$

where: W - elastic modulus of cross-section
f_y - steel yield stress
γ_{M0} - partial factor for resistance of a cross-section

The design resistance for compression is given by:

$$N_{c,Rd} = \frac{A f_y}{\gamma_{M0}}$$

where: A - area of cross-section
f_y - steel yield stress
γ_{M0} - partial factor for resistance of a cross-section

The design values of bending moment and normal force are checked according to

$$\frac{M}{M_{c,Rd}} + \frac{N}{N_{c,Rd}} \leq 1{,}0$$

and the value of utilization is provided by:

$$\left(\frac{M}{M_{c,Rd}} + \frac{N}{N_{c,Rd}}\right) 100\%$$

Check for shear

The design shear resistance is given by:

$$V_{c,Rd} = \frac{I t}{S} \frac{f_y}{\sqrt{3}\, \gamma_{M0}}$$

where: I - moment of inertia
t - section thickness at the center of gravity
S - 1st moment of area
f_y - steel yield stress
γ_{M0} - partial factor for resistance of a cross-section

The design value of the shear force is checked according to:

$$\frac{Q}{V_{c,Rd}} \leq 1{,}0$$

and the value of utilization is provided by:

$$\frac{Q}{V_{c,Rd}} 100\%$$

State of plane stress verification:

The state of plane stress is checked exploiting the following conditions:

$$\left(\frac{\sigma_1}{f_y/\gamma_{M0}}\right)^2 + 3\left(\frac{\tau_1}{f_y/\gamma_{M0}}\right)^2 \leq 1{,}0$$

where: σ_1 - normal stress

τ_1 - shear stress

The value of utilization is provided by:

$$\sqrt{\left(\frac{\sigma_1}{f_y/\gamma_{M0}}\right)^2 + 3\left(\frac{\tau_1}{f_y/\gamma_{M0}}\right)^2}\,100\%$$

Literature:

Eurocode 3: Design of steel structures - Part 1-1: General rules and rules for buildings.

CSN 731401

The CSN 73 1401 standard (from year 1998) adopts as the material parameter the steel design strength R_d. If this value is not determined for the used steel directly, it is back calculated from the steel yield stress as:

$$R_d = R_y/\gamma_m$$

where: γ_m - material coefficient taking the value of 1.15 pro $R_y \leq$ 300 MPa and 1.25 for $R_y >$ 300 MPa

Check for bending

The normal stress σ is checked based on the following expression:

$$\sigma \leq R_d$$

and the value of utilization is provided by:

$$\frac{\sigma}{R_d}100\%$$

Check for shear

The shear stress τ is checked based on the following expression:

$$\tau \leq 0{,}6R_d$$

and the value of utilization is provided by:

$$\frac{\tau}{0{,}6R_d}100\%$$

State of plane stress verification

The state of plane stress is checked exploiting the following conditions:

$$\sqrt{\sigma_1^2 + 3\tau_1^2} \leq 1{,}1R_d$$

where: σ_1 - normal stress

τ_1 - shear stress in the verified section

and the value of utilization is provided by:

$$\frac{\sqrt{\sigma_1^2 + 3\tau_1^2}}{1{,}1R_d}100\%$$

Literature:

CSN 73 1401 (1998): Design of steel structures.

Verification According to the Safety Factor

Check for bending

The normal stress σ is checked based on the following expression:

$$\frac{f_y}{\sigma} \geq SF_s$$

where: f_y - steel yield stress

SF_s - safety factor for bearing capacity of steel cross-section

and the value of utilization is provided by:

$$\frac{SF_s}{f_y/\sigma} 100\%$$

Check for shear

The shear stress τ is checked based on the following expression:

$$\frac{f_y}{\sqrt{3}\tau} \geq SF_s$$

and the value of utilization is provided by:

$$\frac{SF_s}{f_y/\left(\sqrt{3}\tau\right)} 100\%$$

State of plane stress verification

The state of plane stress is checked exploiting the following conditions:

$$\frac{f_y}{\sqrt{\sigma_1^2 + 3\tau_1^2}} \geq SF_s$$

where: σ_1 - normal stress

τ_1 - shear stress in the verified section

and the value of utilization is provided by:

$$\frac{SF_s}{f_y/\sqrt{\sigma_1^2 + 3\tau_1^2}} 100\%$$

Verification According to the Theory of Limit States

When performing the analysis according to the theory of limit states the steel yield stress f_y is reduced by the coefficient of material reliability γ_{ss}.

Check for bending

The normal stress σ is checked based on the following expression:

$$\sigma \leq \frac{f_y}{\gamma_{ss}}$$

and the value of utilization is provided by:

$$\frac{\sigma}{f_y/\gamma_{ss}} 100\%$$

Check for shear

The shear stress τ is checked based on the following expression:

$$\tau \leq \frac{f_y}{\sqrt{3}\gamma_{ss}}$$

and the value of utilization is provided by:

$$\frac{\tau}{f_y/\sqrt{3}\gamma_{ss}} 100\%$$

State of plane stress verification

The state of plane stress is checked exploiting the following conditions:

$$\sqrt{\sigma_1^2 + 3\tau_1^2} \leq \frac{f_y}{\gamma_{ss}}$$

where: σ_1 - normal stress
τ_1 - shear stress in the verified section

and the value of utilization is provided by:

$$\frac{\sqrt{\sigma_1^2 + 3\tau_1^2}}{f_y / \gamma_{ss}} 100\,\%$$

GB 50017-2003

The GB 50017-2003 standard adopts as the material parameter the steel design compressive, tension, and bending strength f and shear strength f_y. If this value is not determined for the used steel directly, it is back calculated from the steel yield stress f_y as:

$$f = \frac{f_y}{\gamma_R}$$

$$f_v = \frac{f}{\sqrt{3}}$$

where: γ_R - resistance sub coefficient, which is 1,087 for $f_y \leq$ 240 MPa and 1,111 for $f_y >$ 240 MPa

Check for bending with the influence of normal force

The bending stress with the influence of normal force is checked according to this expression:

$$\frac{M}{\gamma_x W} + \frac{N}{A} \leq f$$

where: A - area of cross-section
W - elastic modulus of cross-section
γ_x - section plasticity develop factor
f - design strength of steel

Section plasticity develop factor γ_x depends on shape of cross section. For I-sections, sheet piles and casing is considered as $\gamma_x =$ 1,05. The value of utilization is provided by:

$$\frac{\frac{M}{\gamma_x W} + \frac{N}{A}}{f} 100^o/_o$$

Check for shear

The shear stress τ is checked based on the following expression:

$$\tau \leq f_v$$

and the value of utilization is provided by:

$$\frac{\tau}{f_v} 100^o/_o$$

State of plane stress verification

The state of plane stress is checked exploiting the following conditions:

$$\sqrt{\sigma^2 + 3\tau^2} \leq \beta_1 f$$

where: σ_1 - normal stress
τ_1 - shear stress in the verified section
β_1 - strength design value increase coefficient, which is 1,1

The value of utilization is provided by:

$$\frac{\sqrt{\sigma^2+3\tau^2}}{\beta_1 f}100^o/_o$$

Concrete Cross Section with Steel Profile Verification

Verification of the concrete cross-section reinforced by steel section is carried out for two load cases:

1. for the maximum value of bending moment and the corresponding shear force ($M_{max}+Q$)
2. for the maximum shear force and the corresponding bending moment ($Q_{max}+M$)

In both cases, the load enters to the assessment with an influence of normal force, which is defined separately. Its value is identical for both of load cases. Internal forces are, prior to analysis, pre-multiplied by the reduction coefficient of bearing capacity. This coefficient represents the degree of uncertainty of the determination of theoretical values of internal forces and as thus introduces into the analysis with such values certain reliability. The value of this coefficient is determined solely by the user.

Verification of cross-section

Internal forces provide by the "**Sheeting Check**" program are considered per 1 m run of the structure width. Therefore, the units of shear force Q are kN/m and of bending moments M are kNm/m. For dimensioning of individual sections these forces are, prior to verification analysis, automatically multiplied by their spacing a [m] to get their values at the cross-section center of gravity, i.e. shear force Q in kN and bending moment M in kNm.

The program performs dimensioning of concrete cross-sections reinforced by steel profile only according to EN 1994-1-1.

Verification according to EN-1994-1-1

Standard EN 1994-1-1 refers to standards EC2 and EC3 when checking the concrete cross-section reinforced by the steel section. Partial bearing capacities of the steel section are calculated according to EN 1993-1-1 and partial bearing capacities of the concrete cross-section are calculated according to EN 1992-1-1, resp. EN 1992-2.

Check for shear

Shear bearing capacity of cross-section is given by:

$$V_{\mathrm{Rd}}=V_{\mathrm{pl,a,Rd}}+V_{\mathrm{Rd,c}}$$

where: $V_{pl,a,Rd}$ - design shear resistance of steel cross-section
$V_{Rd,c}$ - design shear resistance of the concrete part

$V_{pl,a,Rd}$ is defined according to EN 1993-1-1, chpt. 6.2.6 as:

$$V_{\mathrm{pl,a,Rd}}=A_{\mathrm{V}}f_{\mathrm{yd}}/\sqrt{3}$$

where: A_V - shear area of steel cross-section
f_{yd} - design steel yield stress

$V_{Rd,c}$ is defined according to EN 1992-1-1, chpt. 6.2 as:

$$V_{\mathrm{Rd,c}}=\left[0.035k^{3/2}f_{\mathrm{ck}}^{1/2}+k_1\sigma_{\mathrm{cp}}\right]b_{\mathrm{w}}d$$

$$k=1+\sqrt{\frac{200}{d[\mathrm{mm}]}}\leq 2.0$$

where: f_{ck} - characteristic compressive strength of concrete
k_1 - coefficient with recommended value 0,15
σ_{cp} - compressive stress in concrete
b_w - cross-section width
d - cross-section height

Design shear resistance is checked according to:

$$\frac{Q}{V_{\mathrm{Rd}}}\leq 1.0$$

Utilization is given by:

$$\frac{Q}{V_{\mathrm{Rd}}}100\%$$

If the shear utilization value is greater than 50%, further compression and bending checks are expected with the reduced design strength of steel $(1\text{-}\rho)f_{yd}$ on those parts of the steel cross-section that transfer the shear. Reduction coefficient is given by:

$$\rho = (2Q/V_{\mathrm{Rd}} - 1)^2$$

Check for compressive

Compressive resistance is calculated according to EN 1994-1-1, chpt. 6.7.3.2. The calculation considers effect of design strength reduction of steel due to shear stress. For concrete cross-section out off the steel section, reduced concrete stregth is assumed. Full concrete strength is calculated in the part where concrete fills closed steel section. Compressive resistance of cross-section is calculated as:

$$N_{\mathrm{pl,Rd}} = (A_{\mathrm{a}} - \rho A_{\mathrm{V}})\, f_{\mathrm{yd}} + (0,85 A_{1,\mathrm{c}} + A_{2,\mathrm{c}})\, f_{\mathrm{c,d}}$$

where:
- A_a - area of steel cross-section
- ρ - strength reduction coefficient due to shear
- A_V - shear area of steel cross-section
- f_{yd} - design steel yield stress
- $A_{1,c}$ - concrete area out of steel cross-section
- $A_{2,c}$ - concrete area inside steel cross-section
- f_{cd} - design compressive strength of concrete

Design resistance for compression is checked according to:

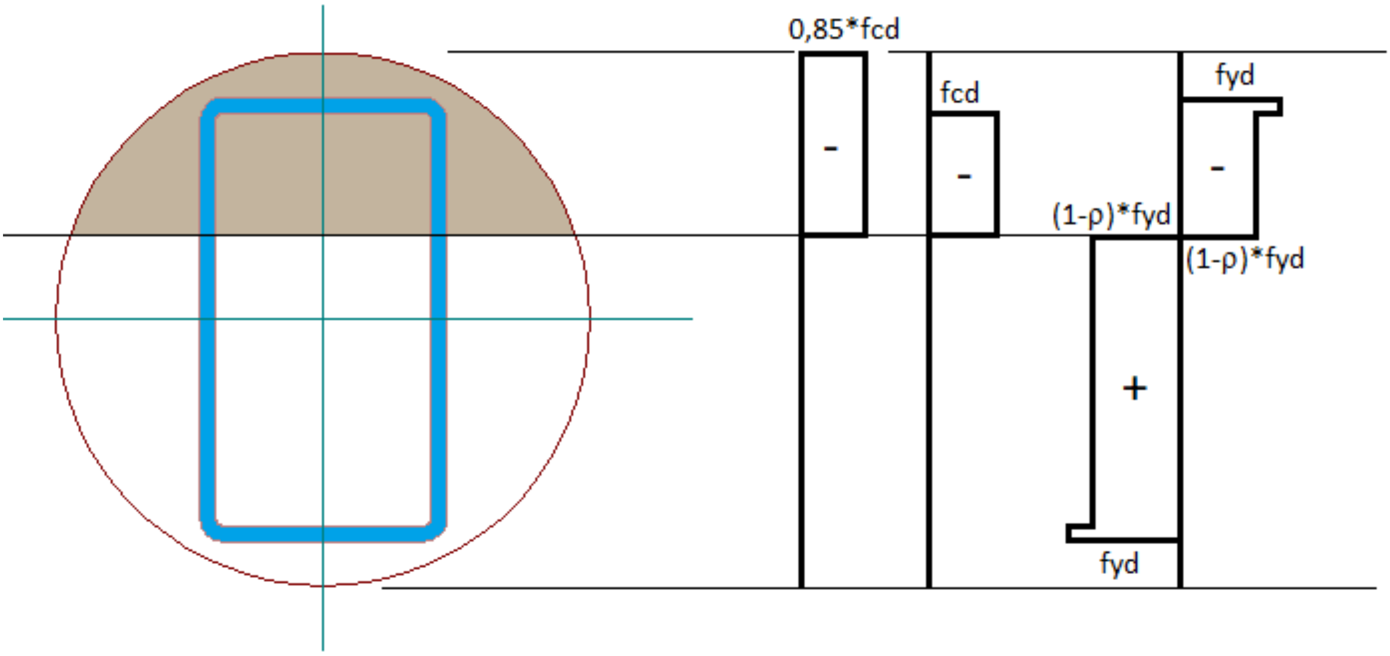
$$\frac{N}{N_{\mathrm{pl,Rd}}} \leq 1,0$$

Utilization is given by:

$$\frac{N}{N_{\mathrm{pl,Rd}}} 100\%$$

Check for bending

The bending resistance of cross-section is determined from the interaction diagram in accordance with EN 1994-1-1, chpt. 6.7.3.2, article (5). The limits of interaction diagram are compressive resistance $N_{pl,Rd}$ and bending resistance $M_{pl,Rd}$. It is assumed that the concrete does not act in tensile and the stress distribution according to theory of plasticity. The scheme for determining the moment $M_{pl,Rd}$ is shown in the figure below. The reduced concrete strength 0.85*f_{cd} is assumed on the concrete part out of the steel section and full strength f_{cd} is assumed for concrete inside the steel section. The reduced strength of the steel $(1\text{-}\rho)$*f_{yd} is assumed on the parts of steel section that transfer the shear and other parts are considered with full strength f_{yd}. Using the default values $N_{pl,Rd}$ and $M_{pl,Rd}$ the interaction diagram is calculated and the corresponding value of bending resistance $M_{pl,N,Rd}$ is determined for the applied normal force N.



Scheme of bending resistance determination $M_{pl,Rd}$

Design resistance for bending is checked according to:

$$\frac{M}{M_{\mathrm{pl,N,Rd}}} \leq \alpha_{\mathrm{M}}$$

Coefficient α_M has a value of 0.9 for steel yield stress f_y < 400 MPa and a value of 0.8 for steel yield stress f_y ≥ 400 MPa .

Utilization is given by:

$$\frac{M}{\alpha_{\mathrm{M}} M_{\mathrm{pl,N,Rd}}} 100\%$$

Timber Cross Section Verification

Verification of timber cross-sections is carried out for a load of bending moment, normal force and shear force. Verification is carried out for a chosen load or for the critical load. The critical load is the load with a maximum value of utilization.

Cross-section can be checked by these ways:

- Verification according to EN 1995-1-1 (EC 5)
- Verification according to safety factor
- Verification according to limit states

Each cross-section is checked for two types of load:

1. Check for bending moment and normal force

The analysis checks the bending moment stress σ_m developed at the edge of cross-section given by:

$$\sigma_{\mathrm{m}} = \frac{M}{W}$$

where: M - bending moment
W - elastic modulus of cross-section

and the normal force stress σ_n given by:

$$\sigma_{\mathrm{n}} = \frac{N}{A}$$

where: N - normal force
A - area of cross-section

The normal force can be either tension or compression.

2. Check for shear

The analysis checks the shear stress τ at the cross-section center of gravity written as:

$$\tau = \frac{Q\,S}{I\,t}$$

where: Q - shear force
S - 1st moment of area
I - moment of inertia
t - width (thickness) of cross-section at its center of gravity

EN 1995-1-1 (EC5)

Check for bending and compression

The design bending strength of timber is given by:

$$f_{\mathrm{m,d}} = k_{\mathrm{mod}} \frac{f_{\mathrm{m,k}}}{\gamma_{\mathrm{M}}}$$

where: $f_{m,k}$ - characteristic bending strength of timber
k_{mod} - modification factor for the duration of load and moisture content

γ_M - partial factor for material properties

The design compressive strength of timber is given by:

$$f_{c,0,d} = k_{mod} \frac{f_{c,k}}{\gamma_M}$$

where: $f_{c,k}$ - characteristic compressive strength of timber
k_{mod} - modification factor for the duration of load and moisture content
γ_M - partial factor for material properties

The bearing capacity is checked according to:

$$\left(\frac{\sigma_{c,0,d}}{f_{c,0,d}}\right)^2 + \frac{\sigma_{m,d}}{f_{m,d}} \leq 1$$

where: $\sigma_{c,0,d}$ - design compressive stress
$\sigma_{m,d}$ - design bending stress

The value of utilization is provided by:

$$\left[\left(\frac{\sigma_{c,0,d}}{f_{c,0,d}}\right)^2 + \frac{\sigma_{m,d}}{f_{m,d}}\right] 100\,\%$$

Check for bending and tension

The design tensile strength of timber is given by:

$$f_{t,d} = k_{mod} \frac{f_{t,k}}{\gamma_M}$$

where: $f_{t,k}$ - characteristic tensile strength of timber
k_{mod} - modification factor for duration of load and moisture content
γ_M - partial factor for material properties

The bearing capacity is checked according to:

$$\frac{\sigma_{t,0,d}}{f_{t,0,d}} + \frac{\sigma_{m,d}}{f_{m,d}} \leq 1$$

where: $\sigma_{t,0,d}$ - design tensile stress
$\sigma_{m,d}$ - design bending stress

The value of utilization is provided by:

$$\left(\frac{\sigma_{t,0,d}}{f_{t,0,d}} + \frac{\sigma_{m,d}}{f_{m,d}}\right) 100\,\%$$

Check for shear

The design shear strength of timber is given by:

$$f_{v,d} = k_{mod} \frac{f_{v,k}}{\gamma_M}$$

where: $f_{v,k}$ - characteristic shear strength of timber
k_{mod} - modification factor for the duration of load and moisture content
γ_M - partial factor for material properties

The bearing capacity is checked according to:

$$\frac{\tau_d}{k_{cr}} \leq f_{v,d}$$

where: τ_d - design shear stress

k_{cr} - modification factor for shear resistance

The value of utilization is provided by:

$$\frac{\tau_d}{k_{cr}\, f_{v,d}} 100\,\%$$

Verification According to the Safety Factor

Check for bending and stress caused by the normal force

The normal stress checking is based on the following expression:

$$\left(\frac{\sigma_n}{f_n} + \frac{\sigma_m}{f_m}\right)^{-1} \geq SF_s$$

where:
σ_n - normal stress caused by tension or compression
σ_m - bending stress
f_n - tensile resp. compressive strength of timber
f_m - bending strength of timber
SF_s safety factor for bearing capacity of timber cross-section

The value of utilization is provided by:

$$SF_s\left(\frac{\sigma_n}{f_n} + \frac{\sigma_m}{f_m}\right) 100\,\%$$

Check for shear

The shear stress τ checking is based on the following expression:

$$\frac{f_v}{\tau} \geq SF_s$$

where: f_v - shear strength of timber

The value of utilization is provided by:

$$\frac{SF_s}{f_v/\tau} 100\,\%$$

Verification According to the Theory of Limit States

When performing the analysis according to the theory of limit states the strengths of timber are reduced by the coefficient of material reliability γ_s.

Check for bending and stress caused by the normal force

The normal stress checking is based on the following expression:

$$\frac{\sigma_n}{f_n/\gamma_s} + \frac{\sigma_m}{f_m/\gamma_s} \leq 1$$

where:
σ_n - normal stress caused by tension or compression
σ_m - bending stress
f_n - tensile resp. compressive strength of timber
f_m - bending strength of timber

The value of utilization is provided by:

$$\left(\frac{\sigma_c}{f_c/\gamma_s} + \frac{\sigma_m}{f_m/\gamma_s}\right) 100\%$$

Check for shear

The shear stress τ checking is based on the following expression:

$$\tau \leq \frac{f_v}{\gamma_s}$$

where: f_v - shear strength of timber

The value of utilization is provided by:

$$\frac{\tau}{f_v/\gamma_s} 100\%$$

Masonry Cross Sections Verification

Masonry structures can be checked by following standards:

- AS 3700 - Masonry Wall program
- EN 1996-1-1 - Masonry Wall program
- EN 1996-1-1 - Masonry cross-section check-in Gravity Wall program
- GB 50003-2011 - Masonry cross-section check-in Gravity Wall program
- JTG D61-2005 - Masonry cross-section check-in Gravity Wall program

AS 3700 - Masonry Wall

Reinforced masonry is verified for load due to bending moment, shear force and combination of compressive normal force and bending moment. When load due normal force is considered, it is necessary to specify also the slenderness ratio S_r.

Design for members in compression and bending

$$F_d \leq 0{,}85.\phi.k_s\left(f'_m.A_b + f_{sy}.A_s\right)$$
$$f'_m = 0{,}35.f'_{mb}$$
$$f'_{mb} = 1{,}3.\sqrt{f'_{uc}}$$

where:
- F_d - the design compression force acting on the cross-section
- ϕ - the capacity reduction factor - 0.75
- k_s - a reduction factor is taken as 1.18 - 0.03*S_r but not greater than 1.0
- f'_{uc} - the characteristic unconfined compressive strength of masonry
- f'_m - the characteristic compressive strength of masonry
- A_b - the bedded area of the masonry cross-section
- f_{sy} - the design yield strength of reinforcement
- A_s - the total cross-sectional area of main reinforcement

Design for members in bending

$$M_d \leq \phi.f_{sy}.A_{sd}.d\left(1 - \frac{0{,}6.f_{sy}.A_{sd}}{1{,}3.f'_m.d}\right)$$
$$f'_m = 0{,}35.f'_{mb}$$
$$f'_{mb} = 1{,}3.\sqrt{f'_{uc}}$$

where:
- M_d - the design bending moment acting on the cross-section of the member
- ϕ - the capacity reduction factor - 0.75
- f_{sy} - the design yield strength of reinforcement
- A_{sd} - the portion of the cross-sectional area of the main tensile reinforcement used for design purposes in a reinforced masonry member
  the lesser of $\frac{0{,}29.1{,}3.f'_m.d}{f_{sy}}$ and A_{st}
- f'_m - the characteristic compressive strength of masonry
- d - the effective depth of the reinforced masonry member
- f_{uc} - the characteristic unconfined compressive strength of masonry

Out-of-plane shear in wall

A reinforced wall subject to out-of-plane shear shall be such that:

$$V_d \leq \phi.(f'_{vm}.d + f_{sv}.A_{st})$$

but not more than:

$$4.\phi.f'_{vm}.d$$

where: V_d - the design shear force acting on the cross-section of the masonry wall
ϕ - the capacity reduction factor - 0.75
f'_{vm} - the characteristic shear strength of reinforced masonry - 0.35 ***Mpa***
d - the effective depth of the reinforced masonry wall
f_{vs} - the design shear strength of the main reinforcement - 17.5 ***Mpa***
f_{sy} - the design yield strength of reinforcement
A_{st} - the cross-sectional area of fully anchored longitudinal reinforcement in the tension zone of the cross-section

EN 1996-1-1 - Masonry Wall

The reinforced masonry is verified for the load caused by the combination of the compressive normal force and the bending moment and for the load due to the shear force.

Verification for pressure and bending

Analysis assumptions (Chapter 6.6):

- plane cross-sections remain plane
- the strain of steel is the same as the strain of the attached masonry
- the tensile strength of masonry is assumed to equal to zero
- the limit strain of masonry in compression is 0.0035
- the limit strain of steel in tension is 0.01
- variation of stress as a function of the strain of masonry is assumed parabolic-rectangular
- variation of stress as a function of the strain of steel is assumed bounded by a horizontal upper branch
- the properties of filling concrete are considered the same as the properties of masonry (it is necessary to use the worse of the two materials)
- design strength of masonry (concrete) is provided by:

$$f_d = \frac{f_k}{\gamma_M}$$

where: f_k - characteristic strength of masonry (concrete)
γ_M - 1.8

- If the slenderness ratio given by the ratio of the height and the width of the wall is greater than 12, the effect of the II-nd order theory is considered by including an additional design bending moment given by:

$$M_{ad} = \frac{N_{Ed} h_{ef}^2}{2000t}$$

where: N_{Ed} - design value of the normal force
h_{ef} - buckling height of the wall
t - wall thickness

- If the slenderness ratio is greater than 27, then it is not possible to perform the analysis and it is necessary by changing geometry to obtain a more favorable slenderness ratio.

Verification for shear

Chapter 6.7, Appendix J

$$V_{Ed} \leq f_{vd} t l$$

$$f_{vd} = \frac{\mathrm{Min}(f_{vk} + 17.5\rho, 0.7)}{\gamma_M}$$

$$\rho = \frac{A_s}{bd}$$

where: V_{Ed} - design value of the shear force
f_{vd} - design value of the shear strength of masonry (concrete)
ρ - longitudinal reinforcement ratio
t - wall thickness

l - wall length - 1 running meter

EN 1996-1-1 - Gravity Wall

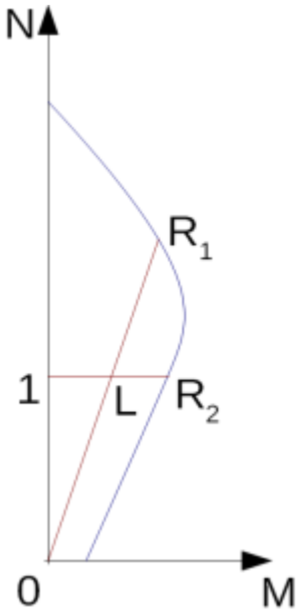
The masonry is verified for the load caused by the combination of the compressive normal force and the bending moment and for the load due to the shear force.

Verification of compression bearing capacity

Chapter 6.1.2.1

$$N_{Ed} \leq N_{Rd}$$
$$N_{Rd} = A_c f_k / \gamma_M$$
$$A_c = bh\left(1 - 2\frac{e}{h}\right)$$
$$e = \frac{M_{Ed}}{N_{Ed}}$$

where:
- N_{Ed} - design value of the normal force
- N_{Rd} - compression bearing capacity
- A_c - compressed area of cross-section
- f_k - characteristic value of compressive strength of masonry
- γ_M - partial factor of masonry
- b - width of cross-section
- h - height of cross-section
- e - excentricity of normal force
- M_{Ed} - design value of bending moment



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as $|0L|/|0R_1|$ or $|1L|/|1R_2|$. Where L is load, R_1 is strength with prescribed excentricity, and R_2 is strength with prescribed normal force.

Verification of shear bearing capacity

Chapter 6.2

$$V_{Ed} \leq V_{Rd}$$
$$V_{Rd} = A_c f_{vk} / \gamma_M$$
$$f_{vk} = \mathrm{Min}\left(f_{vko} + 0.4\frac{N_{Ed}}{A_c}; 0.065 f_b\right)$$

where:
- V_{Ed} - design value of shear force
- V_{Rd} - shear bearing capacity

f_{vk}	-	characteristic value of shear strength of masonry
f_{vko}	-	characteristic value of the original shear strength of masonry
f_b	-	compressive strength of masonry unit

GB 50003-2011 - Gravity Wall

The masonry is verified for the load caused by the combination of the compressive normal force and the bending moment and for the load due to the shear force.

Verification of compression bearing capacity

Non-seismic design situation (Art 5.1.1):

$$\gamma_0 N \leq \varphi f A$$

Seismic design situation (Art 10.1):

$$N \leq \varphi f A / \gamma_{RE}$$

where:

γ_0	-	coefficient of the importance of structure
N	-	design value of the normal force
f	-	design value of compressive strength of masonry
A	-	area of cross-section
φ	-	influence factor due to eccentricity of normal force and the depth-thickness ratio of the structure
γ_{RE}	-	seismic adjusting coefficient for compressive strength of masonry

φ is provided by:

When $\beta \leq$ 3 (Art D.0.1-1)

$$\varphi = \frac{1}{1 + 12\left(\frac{e}{B}\right)^2}$$

When $\beta >$ 3 (Art D.0.1-2, D.0.1-3)

$$\varphi = \frac{1}{1 + 12\left[\frac{e}{B} + \sqrt{\frac{1}{12}\left(\frac{1}{\varphi_0} - 1\right)}\right]^2}$$

$$\varphi_0 = \frac{1}{1 + \alpha\beta^2}$$

where:

e	-	eccentricity of a normal force acting on the cross-section
B	-	depth of the cross-section
φ_0	-	stability coefficient of the structure loaded with axial pressure
α	-	coefficient due to strength grade of mortar
β	-	depth-thickness ratio of the structure

β is provided by:

$$\beta = \gamma_\beta \frac{2H}{B}$$

$$\varphi_0 = \frac{1}{1 + \alpha\beta^2}$$

where:

γ_β	-	adjusting coefficient of depth-thickness ratio based on the type of masonry material
H	-	height of the structure above cross-section

Verification of shear bearing capacity

Non-seismic design situation (Art. 5.5.1-1):

$$\gamma_0 V \leq (f_v + \alpha\mu\sigma_0) A$$

Seismic design situation (Art. 10.1):

$$V \leq (f_v + \alpha\mu\sigma_0) A / \gamma_{RE}$$

When $\gamma_G \leq$ 1.2 (Art 5.5.1-2):

$$\mu = 0.26 - 0.082 \frac{\mathrm{Min}\,(0.8f;\sigma_0)}{f}$$

When $\gamma_G \geq$ 1.35 (Art 5.5.1-3):

$$\mu = 0.23 - 0.065 \frac{\mathrm{Min}\,(0.8f;\sigma_0)}{f}$$

Intermediate values are interpolated.

where:

- γ_0 - coefficient of importance of the structure
- V - design value of shear force
- f_v - design value of shear strength of masonry
- A - area of the cross-section
- σ_0 - average value of normal stress on the cross-section
- f - design value of compressive strength of masonry
- γ_G - partial factor for permanent actions
- α - correction factor; when $\gamma_G \leq$ 1.2: $\alpha =$ 0.64; $\gamma_G \geq$ 1.35: $\alpha =$ 0.66 Intermediate values are interpolated
- μ - influence factor for shear-compression load
- γ_{RE} - seismic adjusting coefficient for shear strength of masonry

Plastic Sheet Pile Verification

Verification of vinyl sheet piles

Verification of vinyl sheet piles is performed for bending and shear. Bending is verified for maximum moment M_{max}, shear is verified for maximum shear force Q_{max}. Internal forces are calculated per unit.

Calculated internal forces can be multiplied by the reduction **coefficient of bearing capacity** before the verification. This coefficient reflects the degree of uncertainty of determining the theoretical values of the internal forces and brings into calculation security.

Bending verification:

Bearing capacity for bending is calculated according to the formula:

$$\frac{M_{\max}}{M_u} \leq 1{,}0$$

where:

- M_{max} - maximum value of bending moment
- M_u - bending bearing capacity

The value of bending bearing capacity M_u is determined per unit and given by the manufacturer of profiles.

Utilization of cross-section is calculated according to the formula:

$$\frac{M_{\max}}{M_u} 100\,\%$$

Shear verification:

Shear bearing capacity is calculated according to the formula:

$$\frac{Q_{\max}}{Q_u} \leq 1{,}0$$

where:

- Q_{max} - maximum value of shear force
- Q_u - shear bearing capacity

The value of shear bearing capacity Q_u is determined per unit and given by the manufacturer of profiles.

Utilization of cross-section is calculated according to the formula:

$$\frac{Q_{\max}}{Q_u} 100\,\%$$

The vinyl strength:

The bearing capacities of the profile are given for the material with ultimate tension strength f_u = 6300 psi (43,44 MPa). The user can change this value. If the strength is changed by the user, the values of bearing capacities are recalculated in a ratio corresponding to the changed value of strength.

Literature:

Implemented sheet piles in our catalog

- *Catalog Everlast Synthetic Products, LLC*

(online: https://everlastseawalls.com/seawall-products/vinyl-sheet-piling)

Soil Mix Verification

Verification of soil mix wall is performed for **bending** and **shear**.

Calculated internal forces are determined in the same way as for lagging and can be multiplied by the reduction **coefficient of bearing capacity** before the verification. This coefficient reflects the degree of uncertainty of determining the theoretical values of the internal forces and brings the safety into the calculation.

Bending verification:

Bearing capacity for bending is calculated according to the formula:

$$\frac{\sigma_{Ed}}{UCS} \leq 1$$

where: σ_{Ed} - normal stress in cross-section
UCS - compressive strength of soil mix

Utilization of cross-section is calculated according to the formula:

$$\frac{\sigma_{Ed}}{UCS}.100$$

Normal stress in cross-section is calculated according to the formula:

$$\sigma_{Ed} = \frac{N_{Ed}}{0,5.h}$$

where: N_{Ed} - normal force
h - wall thickness

Normal force is calculated according to the formula:

$$N_{Ed} = \frac{M_{Ed}}{h_{red}.cos(\alpha)}$$

where: M_{Ed} - bending moment
h_{red} - arc rise (calculated as 2/3 of steel cross-section height, or inputted manually)

Angle α is calculated according to the formula:

$$\alpha = arctan(\frac{4.h_{red}}{d})$$

where: d - spacing of profiles

Shear verification:

Shear bearing capacity is calculated according to the formula:

$$\frac{\tau_{Ed}}{\tau} \leq 1$$

where: τ_{Ed} - shear stress in cross-section
τ - shear resistance of soil mix

Utilization of cross-section is calculated according to the formula:

$$\frac{\tau_{Ed}}{\tau}.100$$

Normal stress in cross-section is calculated according to the formula:

$$\tau_{Ed} = \frac{V_{Ed}}{0,5.h}$$

where: V_{Ed} - shear force

The compressive strength UCS is the basic material input for soil mix. Other parameters can be calculated according to the UCS, or they can be inputted manually:

Shear resistance τ is calculated as follows (intermediate values are interpolated):

for:
- UCS = 0,0 MPa — τ = 0,5 . UCS
- UCS = 1,0 MPa — τ = 0,4 . UCS
- UCS = 1,5 MPa — τ = 0,35 . UCS
- UCS = 3,5 MPa — τ = 0,3 . UCS
- UCS ≥ 4,0 MPa — τ = 0,2 . UCS

Modulus of elasticity E is calculated as follows (intermediate values are interpolated):

for:
- UCS = 0,0 MPa — E = 50 . UCS
- UCS = 2,0 MPa — E = 300 . UCS
- UCS = 5,0 MPa — E = 1000 . UCS

Shear modulus G is calculated according to the formula:

$$G = 0,4.E$$

Literature:

Nicolas Denies, Noel Huybrechts: Deep mixing method for the construction of earth and water retaining walls, Rilem Technical Letters (2017) 2:1-9

A.Porbaha: State of the art in deep mixing technology. Part IV: design considerations, Ground Improvement (2000) 3, 111-125