

Table of contents

2. Gravity wall (file: „Demo01.gtz”)	2
2.1 Basic example	2
2.2 Inserting input data	3
2.2.1 Geometry of structure	3
2.2.2 Profile.....	5
2.2.3 Soils and rocks	8
2.2.4 Assignment	14
2.2.5 Material of a structure	15
2.2.6 Terrain profile	17
2.2.7 Ground water	17
2.2.8 Surcharge	18
2.2.9 Front face resistance (FF resistance)	20
2.2.10 Applied forces	21
2.2.11 Setting parameters controlling the analysis process	21
2.3 Analyses and outputs	22
2.3.1 Verification analysis	22
2.3.2 Dimensioning.....	24
2.3.3 Verification analysis of external stability	26
2.3.4 Data check	26
2.3.5 Text outputs	27
2.3.6 Graphical outputs	28

2. Gravity wall (file: „Demo01.gtz”)

This chapter outlines the design and analysis of the gravity wall. It enables the user to get familiar with all the details associated with this module such that it becomes simple to extend the knowledge gained from this example to other modules:

- Basic input of construction - used in **Cantilever wall, Earth pressures, Gabion, Spread footing, Pile, Abutment, Sheeting design, Sheeting verification**
- Input of soils and rocks – used in **all GEO 4** modules
- Text and graphic outputs - used in **all GEO 4** modules
- Materials – used in **Cantilever wall, Spread footing, Abutment, Plate**
- Verification of external stability – used in **Cantilever wall, Gabion, Sheeting verification**

2.1 Basic example

In this example we desing and verify the gravity wall. Basic input parameters (wall geometry, surcharge, terrain profile, etc.) are displayed in **Fig. 2.1**.

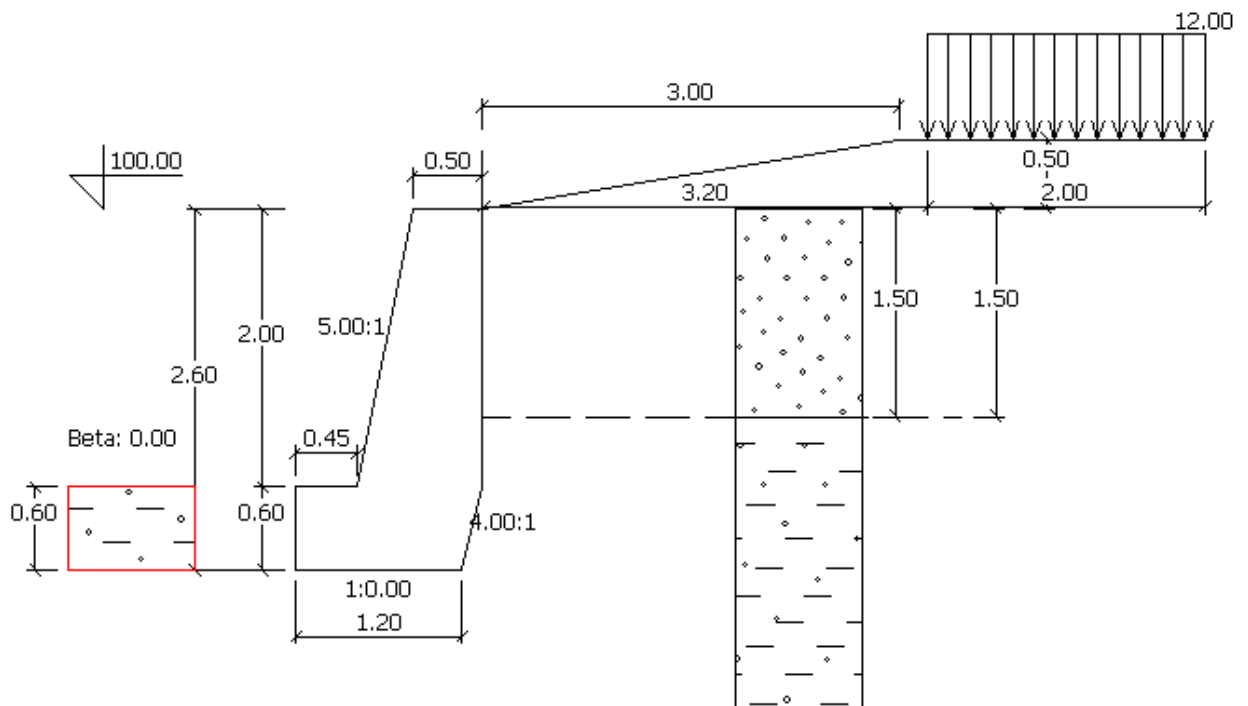


Fig. 2.1 – Basic example – gravity wall

Material properties

Wall: Concrete B20, bulk weight $\gamma = 25 \text{ kN/m}^3$

Soils: Sandy clay, saturation $< 0,8$, consistency stiff ($\varphi_{ef} = 24.5 \text{ dg.}$, $c = 14 \text{ kPa}$, $\gamma = 18.5 \text{ kN/m}^3$, $\delta = 15 \text{ dg.}$)

Sandy silt, saturation $< 0,8$, consistency stiff ($\varphi_{ef} = 29.5 \text{ dg.}$, $c = 0 \text{ kPa}$, $\gamma = 17.5 \text{ kN/m}^3$, $\delta = 15 \text{ dg.}$)

2.2 Inserting input data

2.2.1 Geometry of structure

To enter the geometry of a structure you begin by opening a new file and its saving as e.g. "Demo01" (see the basic user's guide "A tour of the program environment", chapter 2.1 "Handling files"). Using the menu (item "Data") or with the help of the vertical toolbar you select the mode "Geometry". Corresponding screen status is shown in Fig. 2.2.

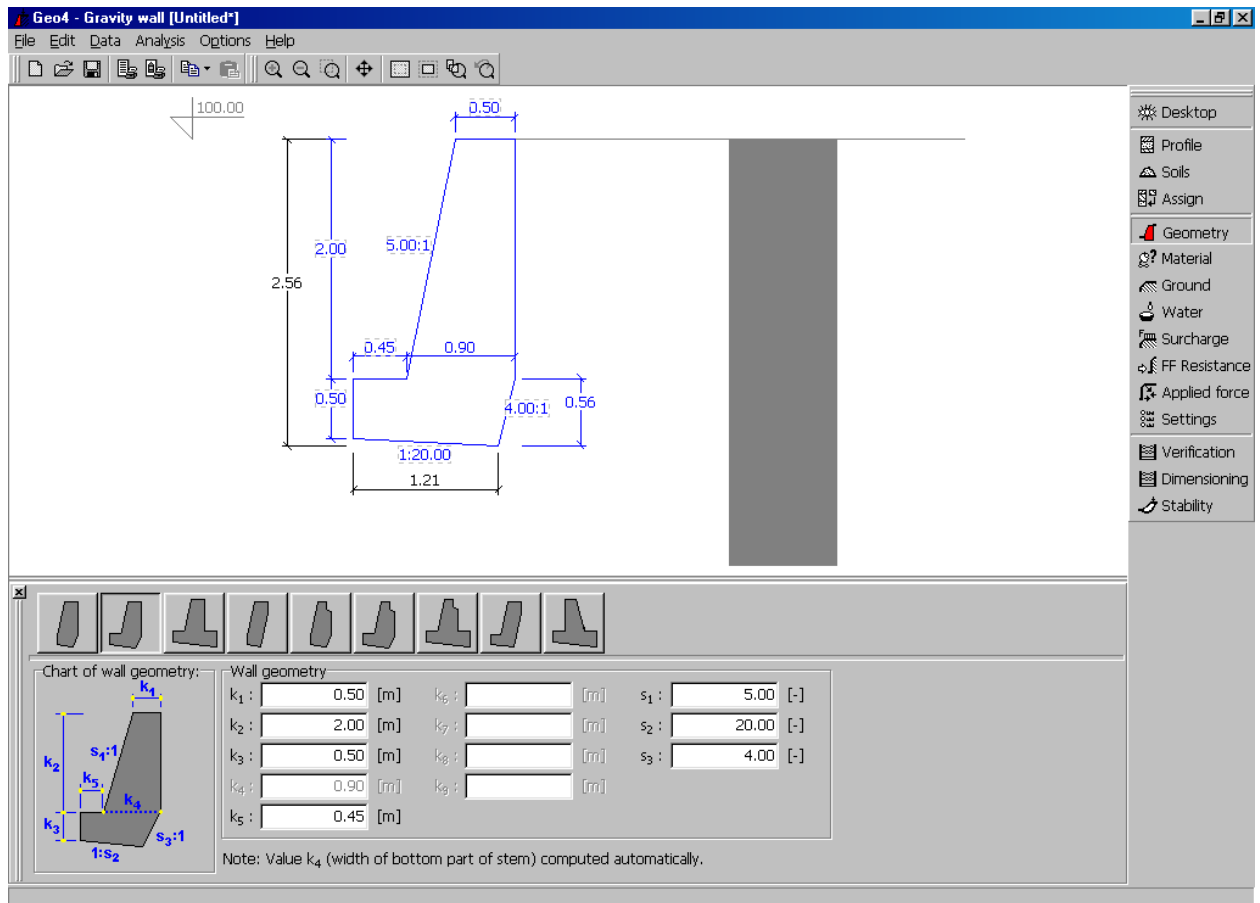


Fig. 2.2 – Geometry input

After selecting the basic shape of a structure on the toolbar below the desktop you may continue by putting in the specific dimensions of the wall. In general, it is possible to proceed in two different ways. The simplest one is to edit individual dimensions directly on the desktop, which displays the shape of the selected structure. In particular, using the left mouse button you click on a number, which corresponds to the selected dimension (Fig. 2.3). Once this field becomes active you are allowed inserting a new value.

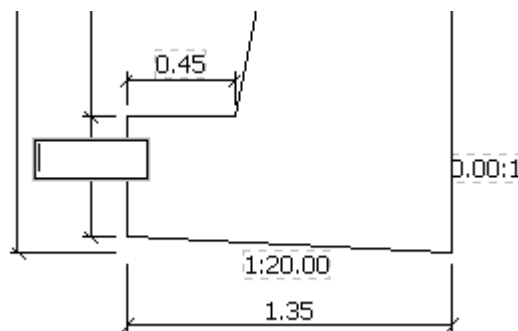


Fig. 2.3 – Edit of dimension on the desktop

The second option is to put the desired dimensions into the Wall geometry check boxes (**Fig. 2.2**). Individual dimensions are marked with letters k with indexes 1 to 9. Data can be put only into the active boxes. That depends on the type of wall. In this particular example you select dimensions k1, k2, k3 and k5, while the dimension k4 is computed automatically. To enter slopes of inclined edges you use side ratios s1 to s3 (see the Wall geometry chart in **Fig. 2.2**). Horizontal and vertical edges are characterized by the zero value of s ($s_i=0$).

Complementary information

Provided that no value is put in or the value has no meaning the program prompts an error message (see **Fig. 2.4**)

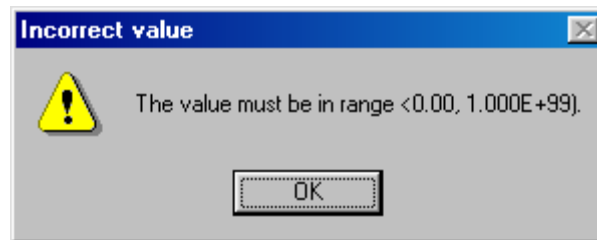


Fig. 2.4 – Error message when inserting dimensions of a structure

Note:

The whole structure might not be displayed within the drawing window when handling the data input. In that case you may use the “**Scale and shift**” button on a toolbar menu (see **Fig. 2.5**), which adjusts the dimensions of a drawing such that all-relevant objects are visible. The toolbar menu contains also other items, which serve to adjust the drawing dimensions (zoom, shift, etc.). They will be explained in detail whenever necessary.



Fig. 2.5 – Toolbar menu with buttons to adjust the drawing of a structure

Note:

When handling an arbitrary data input only a part of the drawing, pertinent to that particular input mode, is visible. In particular, when selecting for example the geometry data input mode, only those parameters, which determine the dimensions of a structure (wall profile and its dimensions), are active. Other elements of the drawing are inactive (colored in gray) and cannot be edited. On the contrary, when switching to a different mode, e.g. “**Profile**”, the geometry parameters become inactive and only the basic wall dimensions remain visible. Therefore, if you wish to display all already inserted parameters use the mode “**Desktop**”.

The mode “Desktop“, apart from the drawing option, offers another important function. Particularly, using this mode you may select which parts of a drawing and in what color should be displayed. **Fig. 2.6** shows an example of a drawing setup for the geometry of a structure.

It is evident from **Fig. 2.6** that the option “Structure” allows you to set up drawing of either overall or detailed dimensions (or both), plotting of point marks, description of points and their numbers as well as the line thickness used for drawing. As shown in **Fig. 2.6** certain drawing options are turned off to reduce complexity of subsequent plots associated with the present example.

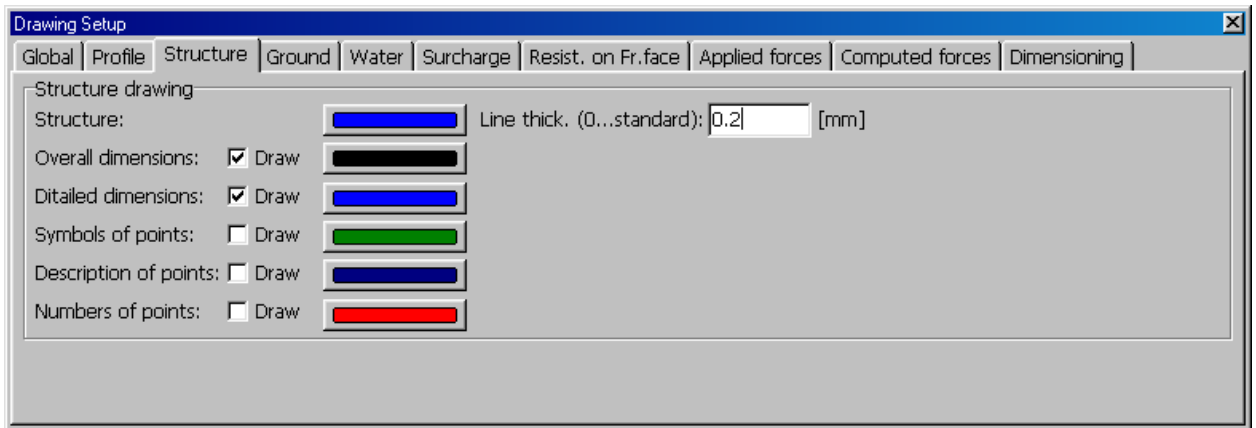


Fig. 2.6 –Setting the geometry of a structure drawing

Complementary information

2.2.2 Profile

The next step is to put in the ground profile and parameters of soils behind the wall. First you insert the thickness of individual layers. To do so select the mode “**Profile**” by clicking on an appropriate menu item on the toolbar (see **Fig. 2.7**).

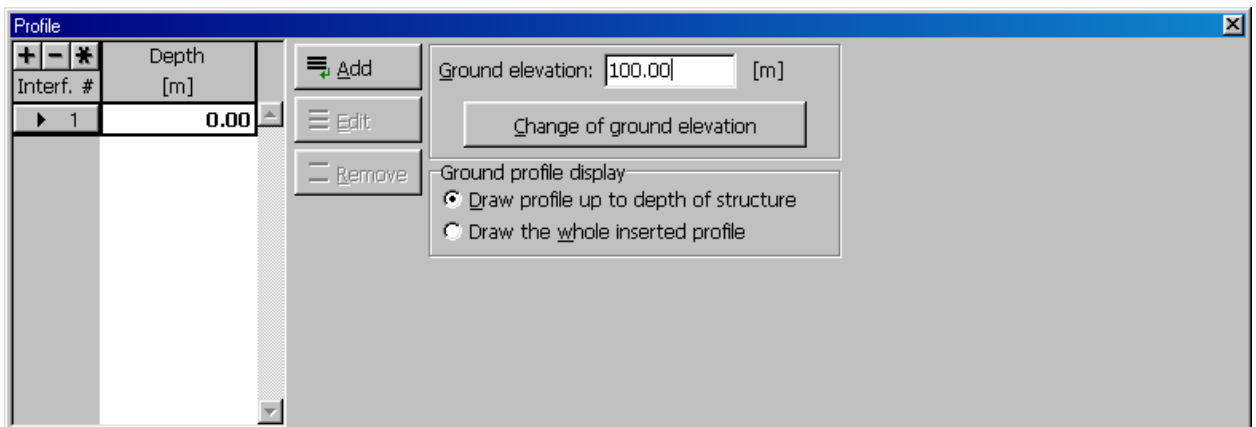


Fig. 2.7 – Ground profile input

The ground profile is put in from the upper point of the wall towards the depth. You begin by clicking on the item “Add”. When the “**New interface**” dialog box appears (see **Fig. 2.8**) you insert the thickness of the first layer. Press the item “Add” to store this value in the list of selected interfaces (see **Fig. 2.9**).

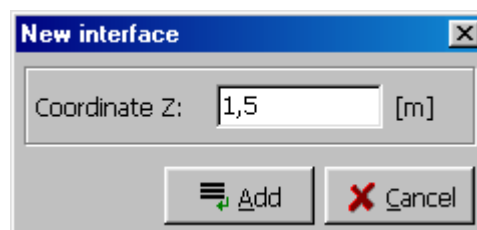


Fig. 2.8 – Thickness of a layer input

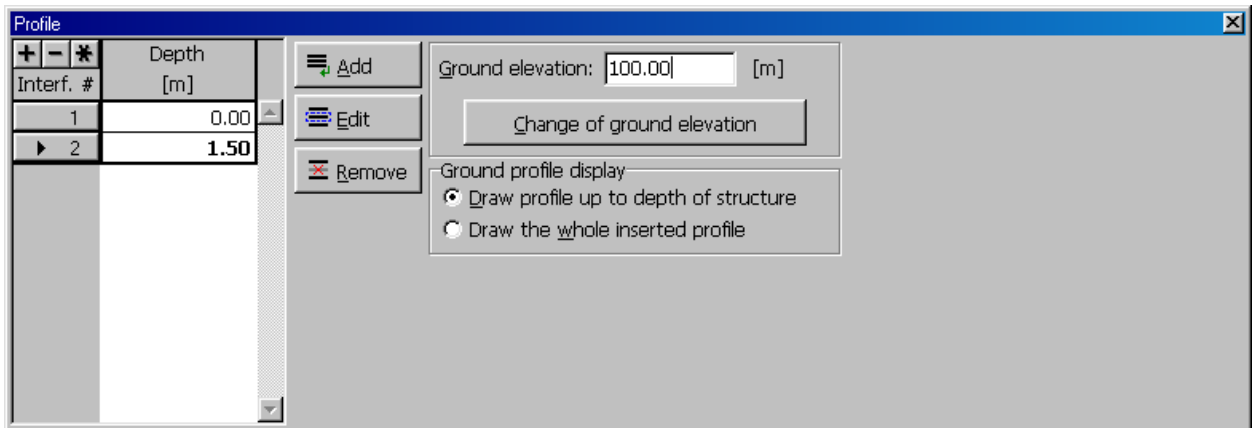


Fig. 2.9 – List of inserted interfaces

This dialog box remains open as long as you continue with the above procedure. Once the required number of interfaces is inserted you may close the box by clicking on the “Cancel” button. The result appears in **Fig. 2.10**.

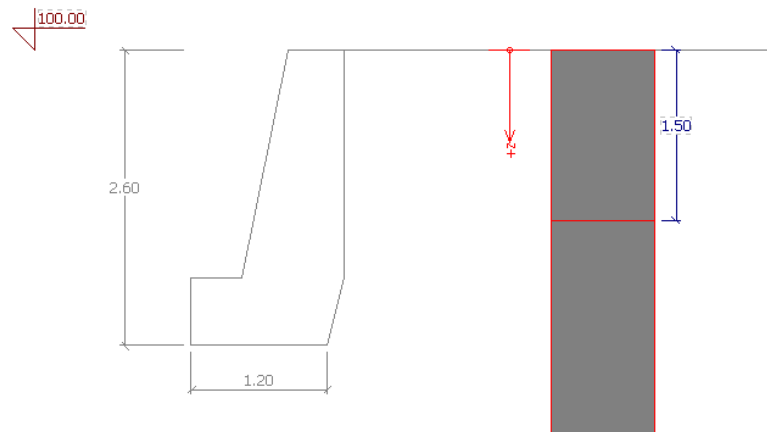


Fig. 2.10 – Profile layers input

Complementary information

If at least one interface (layer) is put in you may continue with its modification. Using the item “Edit” you are able to change the layer thickness or when pressing the “Remove” button the current interface delete. Inserted values are immediately displayed on the Desktop together with the corresponding dimensions. Providing you select the item “Edit” to adjust the layer thickness, the “Edit interface” dialog box appears and the value can be modified. (see **Fig. 2.11**).

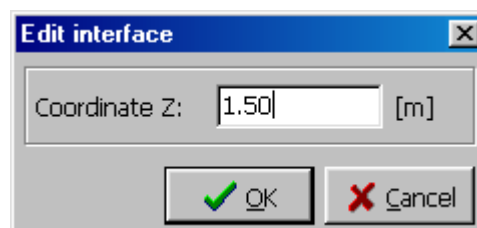


Fig. 2.11 – Dialog box to modify the thickness of a layer in the ground profile

If you wish to remove one or more already inserted interfaces you may proceed by clicking on the item “**Remove**”. Program then opens a new “**Confirm**” dialog box asking to confirm this step. If you select **yes** the particular interface is deleted (see **Fig. 2.12**).

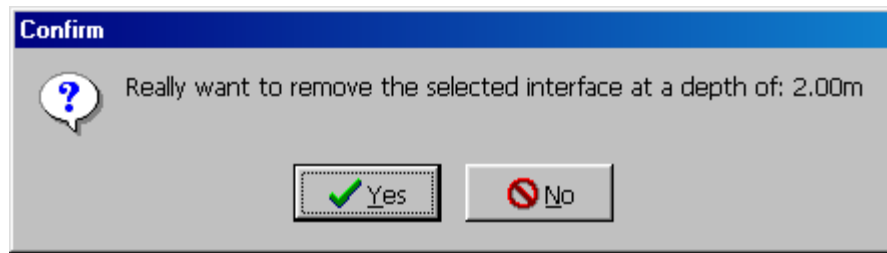


Fig. 2.12 – Confirmation dialog box when removing a layer

When creating a new interface, you enter the coordinate of an already existing one, the program prompts an error message (see **Fig. 2.13**).

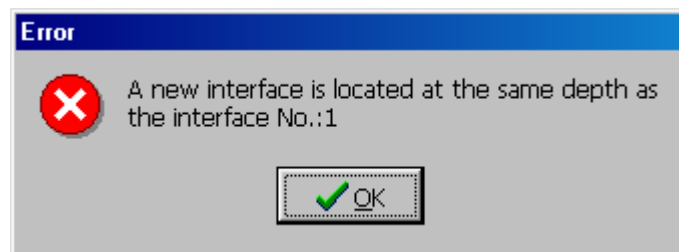


Fig. 2.13 – Error message when inserting an interface

Once the dimensions appear in the drawing window you are also able to edit individual interfaces by clicking on the desired interface directly in the drawing (you are not allowed to either add or remove a layer from the desktop) the same way as we have shown when explaining the input of the geometry of a structure.

Note:

When inserting parameters of the ground profile you encountered for the first time a new element of the program, editable list. We shall now briefly describe its function, as it will be used in other modes of the program as well. An example of such a list with a description of its parts is shown in **Fig. 2.14**.

Interf. #	Depth [m]
1	0.00
▶ 2	1.50

Fig. 2.14 – Using the list

The list is composed of rows and columns. You can select rows either individually or all together. To select a single row you click on its heading. To select a group you use a set of buttons located at the left upper corner of the table. For example, to remove several interfaces at once you first select the desired interfaces and then with the help of the “Remove” button you take them out.

Complementary information 

2.2.3 Soils and rocks

Before assigning individual soils or rocks to corresponding layers you have to first prepare their list. To that end, use the mode “Soils” (see **Fig. 2.15**). The material characteristics of individual soils or rocks are then put in using the “New soil” dialog box (see **Fig. 2.16**).

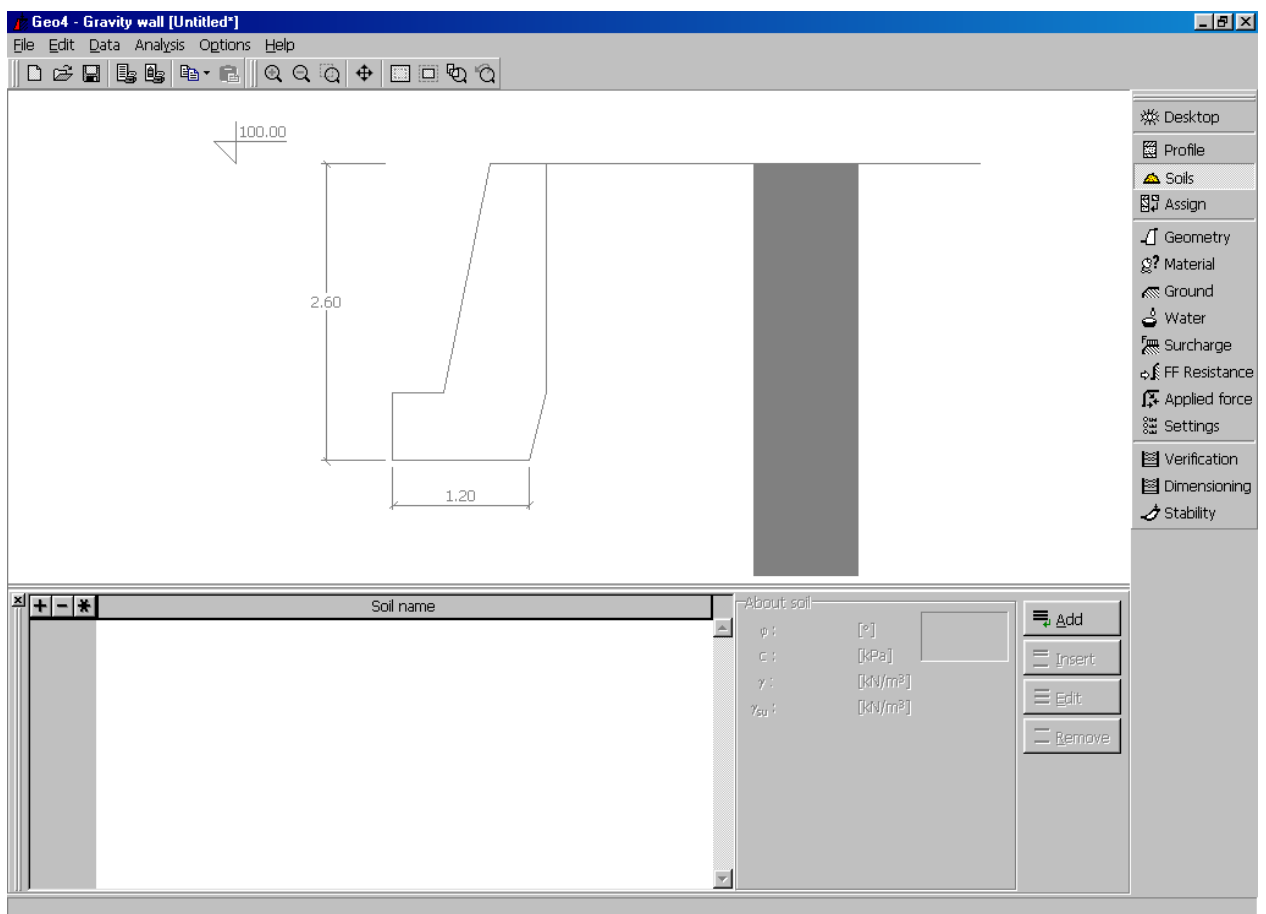
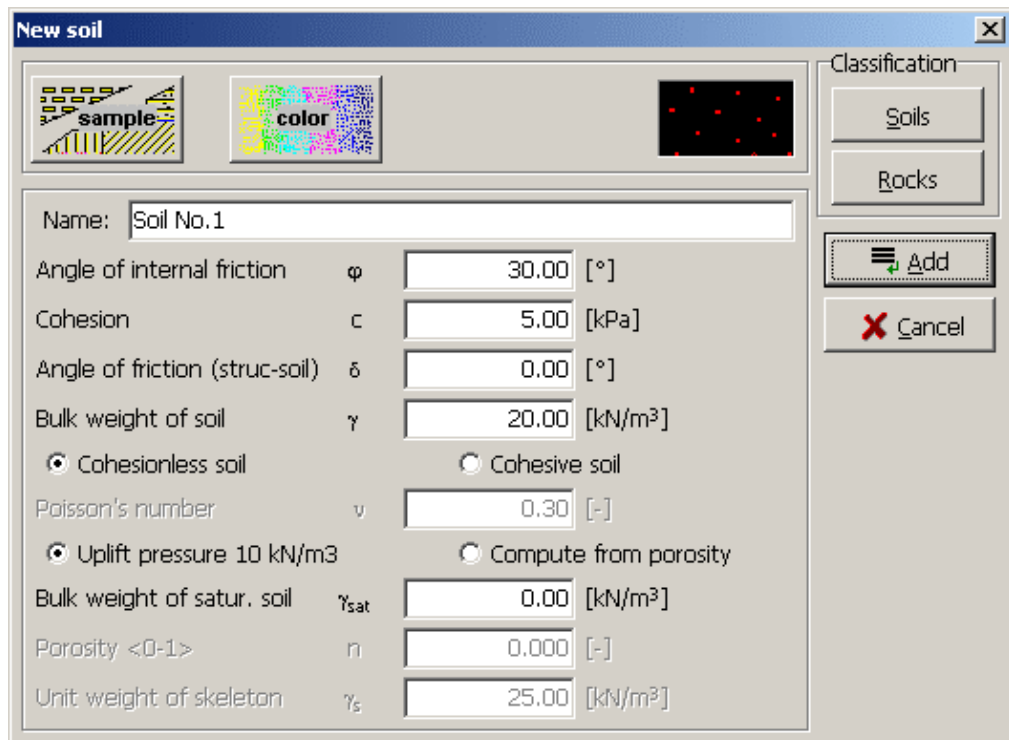


Fig. 2.15 – Mode to insert various types of soils and rocks



New soil

sample color

Name: Soil No.1

Angle of internal friction ϕ 30.00 [°]

Cohesion c 5.00 [kPa]

Angle of friction (struc-soil) δ 0.00 [°]

Bulk weight of soil γ 20.00 [kN/m³]

Cohesionless soil Cohesive soil

Poisson's number ν 0.30 [-]

Uplift pressure 10 kN/m³ Compute from porosity

Bulk weight of satur. soil γ_{sat} 0.00 [kN/m³]

Porosity <0-1> n 0.000 [-]

Unit weight of skeleton γ_s 25.00 [kN/m³]

Classification

Soils

Rocks

Add

Cancel

Fig. 2.16 – Dialog box to insert material characteristics of a new soil

In this dialog box, fill in the name of a soil or rock and select a sample and its color by clicking on the “Sample” (see Fig. 2.17) and “Color” (see Fig. 2.18) buttons. Finally, insert the required input data.

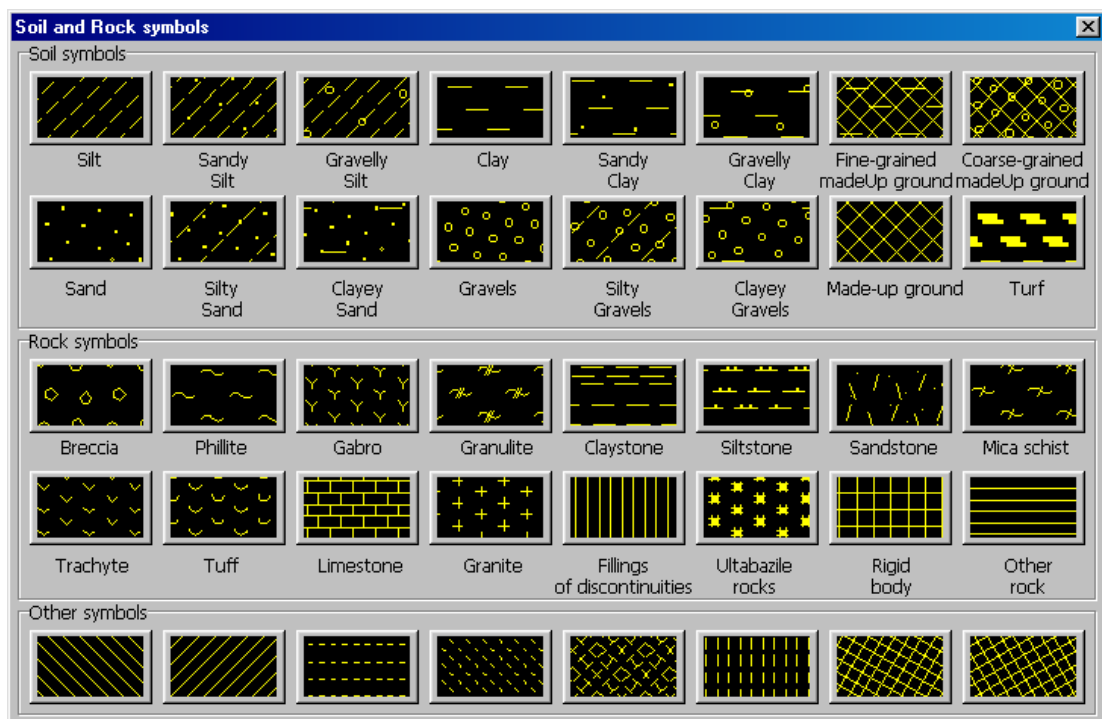


Fig. 2.17 – Dialog box to assign samples of soils or rocks

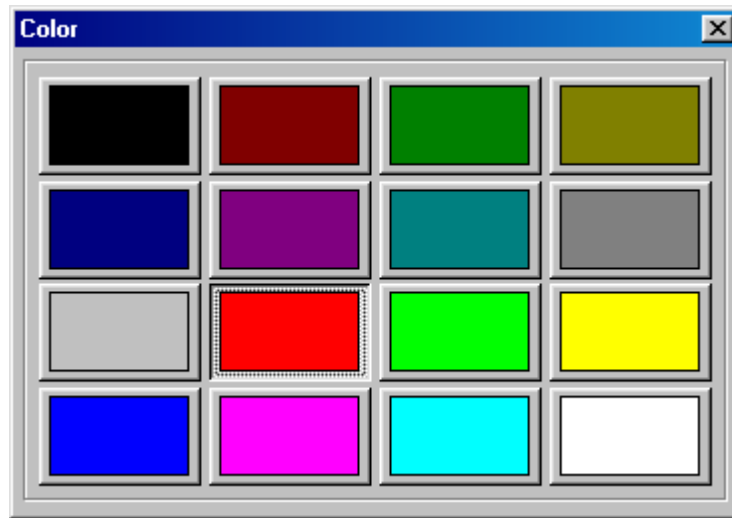


Fig. 2.18 – Dialog box to select color of a soil or rock sample

Using the button “**Add**” you store the sample into the editable list. The dialog box then remains opened and you may continue to put in new values for the next soil. This procedure allows you to create an entire list of soils and rocks for a given profile. An example of the list of soils (mode “**Soils**”) appears in **Fig. 2.19**. This list will be used hereafter.

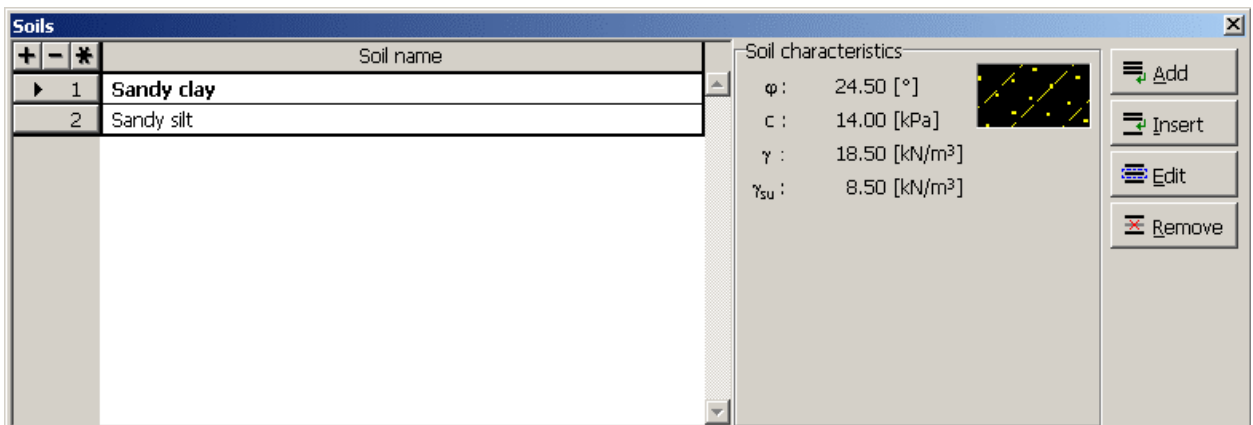


Fig. 2.19 –List of soils

Complementary information

Another option, which exploits catalogues of soils and rocks created according to the Czech standard CSN 73 1001, is available when preparing the list of soils or rocks. To open catalogues you select either “**Soils**” or “**Rocks**” item from one of the dialog boxes “**New soil**” or “**Edit soil**” designed to insert material parameters of soils. After clicking on one of the buttons either the “**Classification of soils**” or “**Classification of rocks**” dialog box appears (see **Figs. 2.20** and **2.21**).

Classification of soils

Soil classification
Sandy silt

Consistency / Density
Stiff consistency, $S_r < 0.8$ (Easy to penetrate by nail, degree of saturation < 0.8)

Standard characteristics of soils

Sandy silt, consistency very stiff $S_r < 0.8$

Soil parameters	Mark	Unit	Value
Poisson's ratio	ny	[-]	0.35
Coeff.	beta	[-]	0.62
Bulk weight	gamma	[kN/m ³]	18.0
Deformation modulus	Edef	[MPa]	12 - 15
Effective parameters:			
Angle of intern. friction	Fi,eff	[dg.]	24 - 29
Cohesion of soil	C,eff	[kPa]	20 - 40
Total parameters:			
Angle of intern. friction	Fi,u	[dg.]	12 - 15
Cohesion of soil	C,u	[kPa]	60 - 70
Design strength:			
Width of footing < 3 m	Rd	[kPa]	275
Coeff. of struc. strength	m	[-]	0.3

Manual OK OK+Assign Cancel

Fig. 2.20 – Catalogue of soils

Classification of rocks

Rock classification
R1, very high strength (Rock is hard to batter with a hammer)

Type of deformation
Brittle deformation - $E_{def}/\sigma_c > 500$

Rock parameters

Class R1, brittle deformation

Discontinuities	Edef	ny	SigmaC
	[MPa]	[-]	[MPa]
fine	25000	0.10	150
small	25000	0.10	150
medium	25000	0.10	150
large	6000	0.10	150
very large	1500	0.10	150
extreme	600	0.10	150
Coeff. of structural strength $m = 0.1$			

OK OK+Assign Cancel

Fig. 2.21 – Catalogue of rocks

In the catalogue, you first select the desired soil or rock from the drop-down list. By clicking on either “OK” or “OK+Assign” button you import the values of the soil material parameters into the “New soil” dialog box. Values of soil parameters provided by the standard are displayed next to entry fields of corresponding material properties (see Fig. 2.22). When using the “OK+Assign” button the entry field is automatically filled with an average value of a given material parameter.

The 'Edit soil' dialog box displays the following parameters and values:

Parameter	Symbol	Value	Unit
Name		Sandy clay	
Angle of internal friction	ϕ	24.50	[°]
Cohesion	c	14.00	[kPa]
Angle of friction (struc-soil)	δ	15.00	[°]
Bulk weight of soil	γ	18.50	[kN/m ³]
<input checked="" type="radio"/> Uplift pressure 10 kN/m ³ <input type="radio"/> Compute from porosity			
Bulk weight of satur. soil	γ_{sat}	18.50	[kN/m ³]
Porosity <0-1>	n	0.000	[-]
Unit weight of skeleton	γ_s	25.00	[kN/m ³]

Classification buttons: Soils, Rocks. Action buttons: OK + ↑, OK + ↓, OK, Cancel.

Fig. 2.22 – Values provided by the catalogue

Warning:

Only parameters defined in the standard are passed on with nonzero values. The remaining ones are set to zero. The crucial point when classifying soils is to check whether the imported data are meaningful and geotechnically correct.

When inserting the bulk weight of a saturated soil (used to determine the bulk weight of soils below the GWT) it is possible to keep the value $\gamma_{sat} = 0$. The program then automatically assigns $\gamma_{sat} = \gamma$. If a nonzero value of γ_{sat} less than γ is inserted the program prompts an error message (Fig. 2.23) and the input data must be corrected.

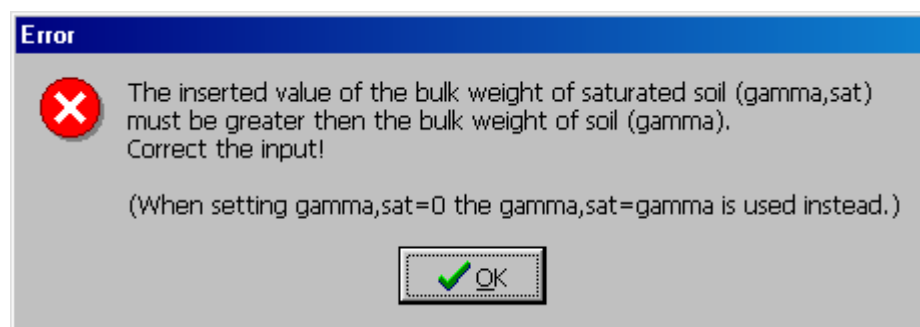


Fig. 2.23 – Error message when an incorrect value of γ_{sat} is put in

Buttons designed to modify the data input in the mode “Soil” become active immediately after storing material parameters of at least one soil or rock into the list. In particular, the “Insert” button opens the “Inserted soil” dialog box, where you may insert the required soil employing the previously described procedure. Then, by clicking on the “Insert” button the soil is included in the table. The item (soil or rock) is placed in the list in front of the soil you selected last prior to opening the dialog box. Using the “Edit” button you open the “Edit soil” dialog box designed to modify parameters of soils and rocks already included in the table.

Fig. 2.24 – Dialog box to modify parameters of already inserted soils

This particular dialog box is rather advantageous, as it allows you to scroll through the entire list of soils and rocks in the table using the “OK +↑ “ and “OK +↓” buttons (see **Fig. 2.24**) without going back into the table and selecting the next item from the list. The last “Remove” button serves to remove one or more soils (rocks) from the list. **Figs. 2.25** and **2.26** then display the dialog box “Confirm “ in which this step must be confirmed. After that the item is deleted.

Fig. 2.25 – Conformation dialog box to remove a soil – only one item is selected

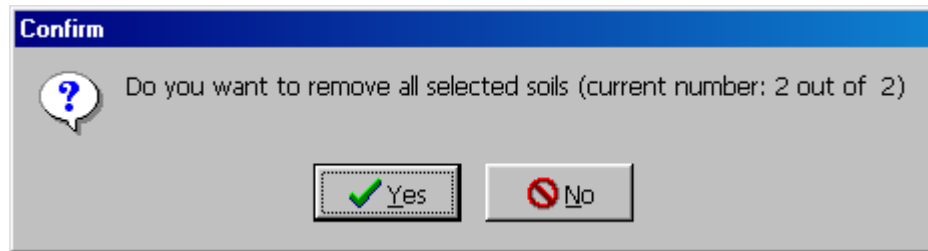


Fig. 2.26 – Confirmation dialog box to remove a soil – more items are selected

If you used, in this stage of input process, the “Add” button instead of “Insert” a new item (soil or rock) would be placed at the end of the list. Basic information of the soil (or rock) currently marked in the heading of rows is displayed next to list (see **Fig. 2.19**).

Complementary information 

2.2.4 Assignment

To continue with this example switch to mode “Assign”. Note that the list of soils previously created in the mode “Soils” is already available.

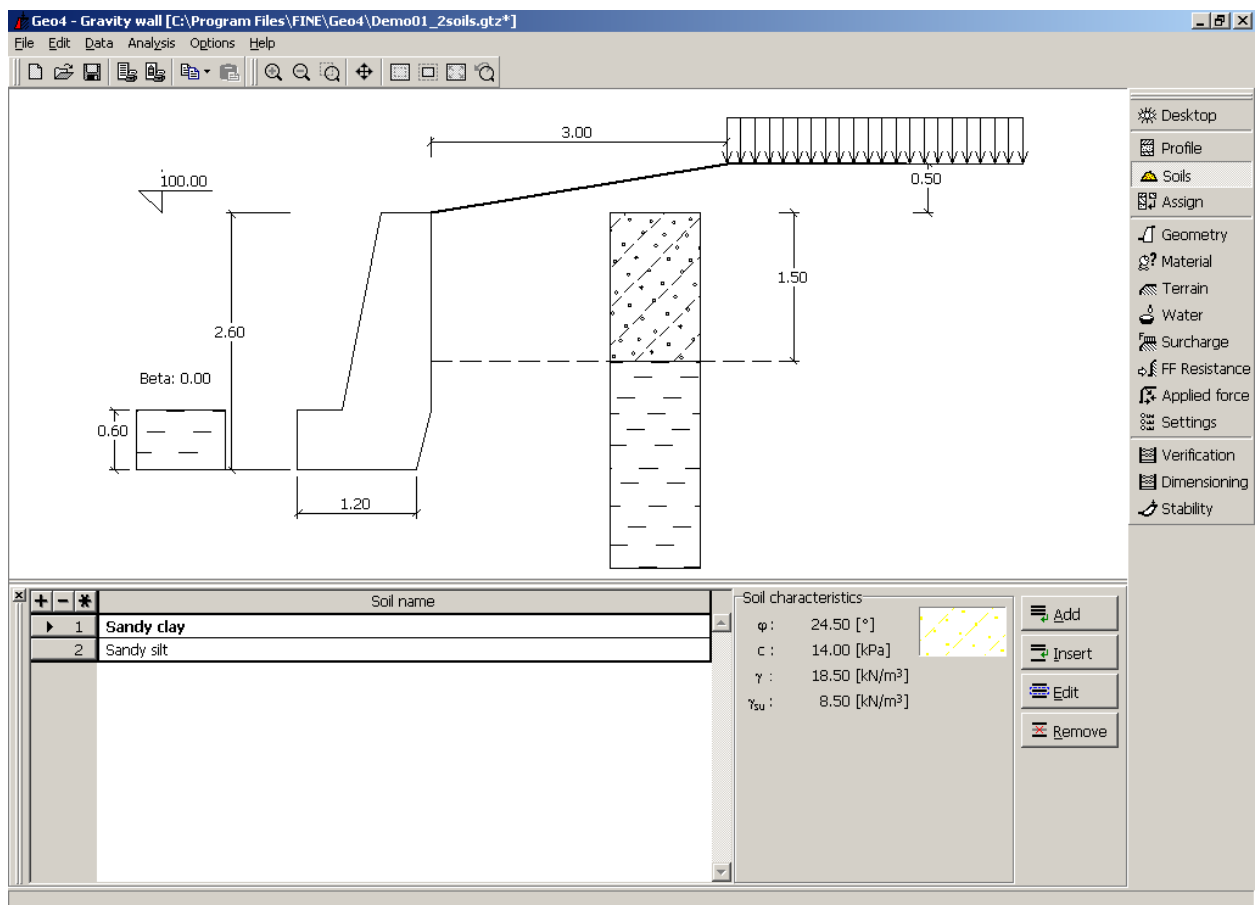


Fig. 2.27 – Dialog box to assign soils and rocks to individual layers.

Note from **Fig. 2.27** that individual soils stored in the table are shown on a bar above the list as well. Two options are available to assign a soil to a given layer in the ground profile. When

selecting the first option you begin by clicking on the button located above the list associated with the soil in the list you wish to assign. To assign the soil you mark the corresponding layer directly on the desktop (see Fig. 2.27). The list of soils offers the second option (note that individual rows correspond to layers of the ground profile). In particular, in a given row you open a drop-down list of soils (see Fig. 2.28) and select, for a given layer, the required type of soil. This option is usually preferable when the layer, depicted on the desktop, is too narrow and it is difficult to hit it with a pointer. Fig. 2.28 provides the result of this step-by-step procedure.

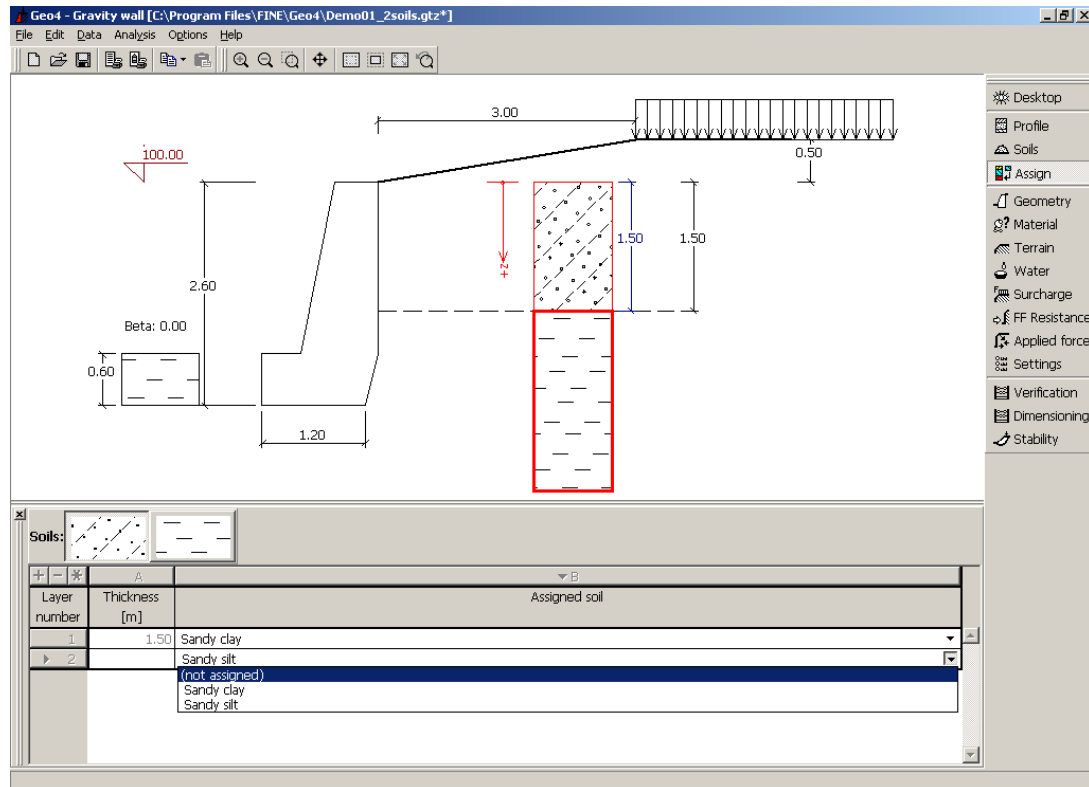


Fig. 2.28 – Soils assigned to layers of the ground profile

2.2.5 Material of a structure

To choose material of a structure we proceed in the following way. First, select the “Material” insert mode. The dialog box shown in Fig. 2.29 appears.

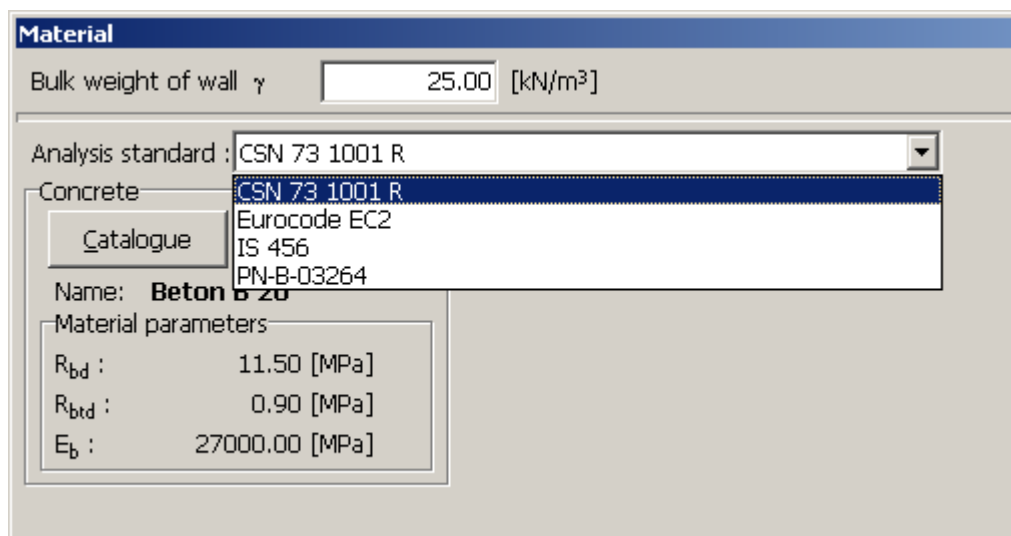


Fig. 2.29 – Dialog box to insert material properties of a structure

You begin by putting the value of the bulk weight of a structure into the corresponding entry field. In this window you select the standard (PN, IS, BS, CSN, EC and other)– all materials and analyses of the concrete structures will be computed according to this standard.

Then, using the “**Catalogue**” button (see **Fig. 2.30**) open the “**Catalogue of materials**” dialog box and select the desired type of concrete (concrete **B 20** is assumed in this example).

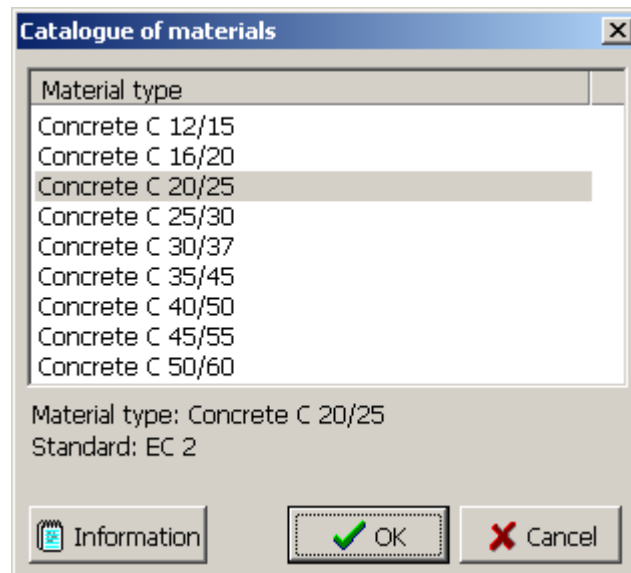


Fig. 2.30 – Catalogue of materials

Complementary information

The program makes also possible to insert your own material characteristics. To that end, click on the “**Own**” button to open the “**Edit material – Concrete**” dialog box (see **Fig. 2.31**), which serves to assign your own values to material parameters.

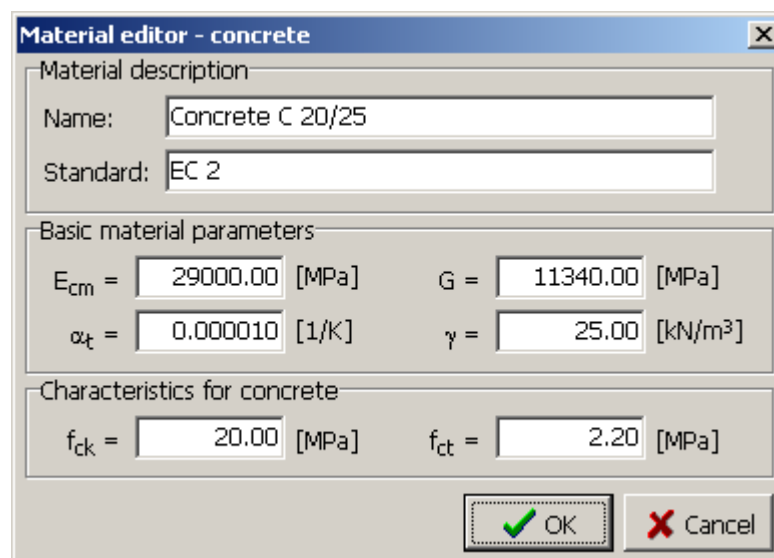


Fig. 2.31 – Dialog box to enter your own material properties of concrete

It is worth to note that the material input is not displayed anywhere on the desktop but it of course affects the analysis of that particular structure.

Complementary information 

2.2.6 Terrain profile

Fig. 2.32 introduces the dialog box designed to determine the profile of the terrain surrounding the structure. To complete this task you first select the basic profile of the terrain using the predefined charts located on the bar below the desktop. Individual entry fields within the dialog box serve to specify individual parameters. In this particular case the terrain profile is determined using the length and height of an embankment.

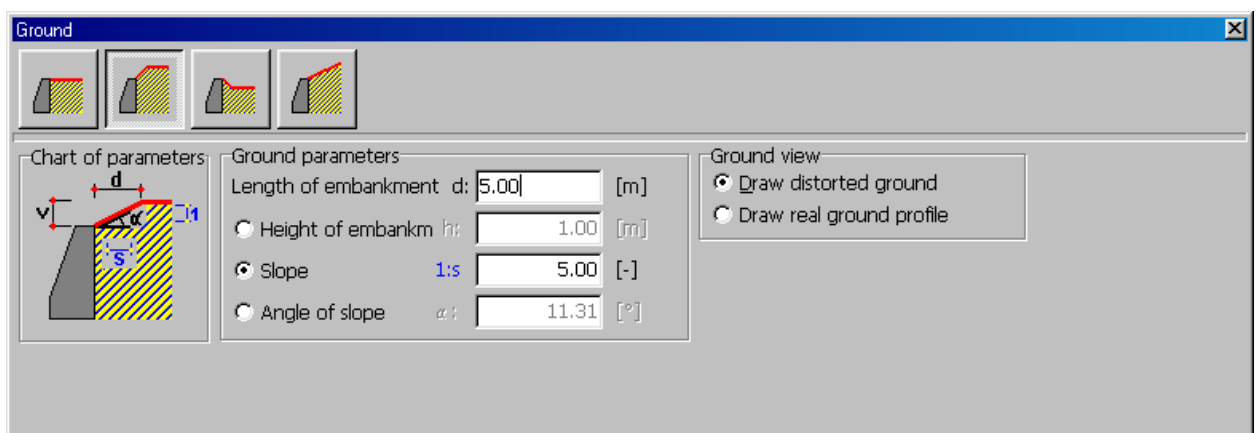


Fig. 2.32 – Mode “Terrain” – dialog box to determine the terrain profile

2.2.7 Ground water

To incorporate an influence of ground water into your analysis switch to mode “**Water**” (see **Fig. 2.33**). From the toolbar below the desktop then select the required type. Parameters relevant to that particular item become active. In our example you are prompted to enter the height of the ground water table behind the structure.

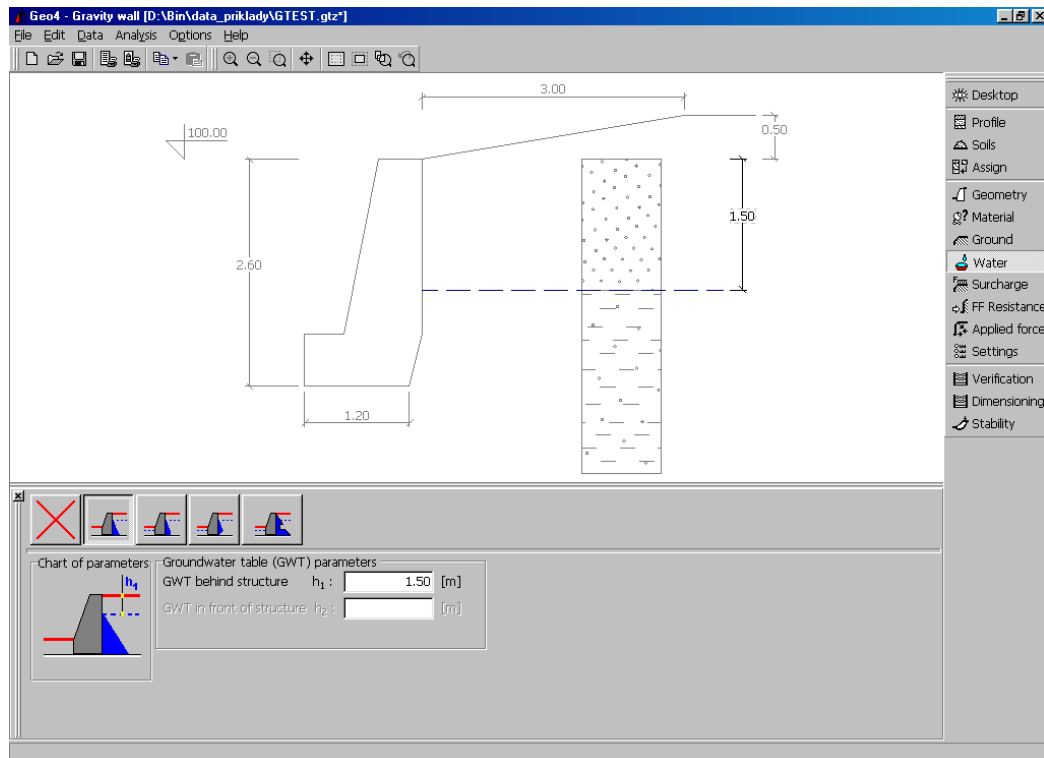


Fig. 2.33 – Entering the ground water table

2.2.8 Surcharge

To enter a new surcharge, switch to mode “**Surcharge**”. Similarly, as in the previous modes, you begin by clicking on the “**Add**” button, thus opening the “**New surcharge**” dialog box (see **Fig. 2.34**). Once again you first select the type of a surcharge by clicking on one of the items offered in the dialog box. The next step is to fill in the relevant entry fields. To store them into the list you press the “**Add**” button. To continue with the next surcharge you just repeat the above procedure. To stay with this example select strip loading. Inserted values are displayed in **Fig. 2.34**. See also **Fig. 2.35** showing the setup displayed in the mode “**Surcharge**”.

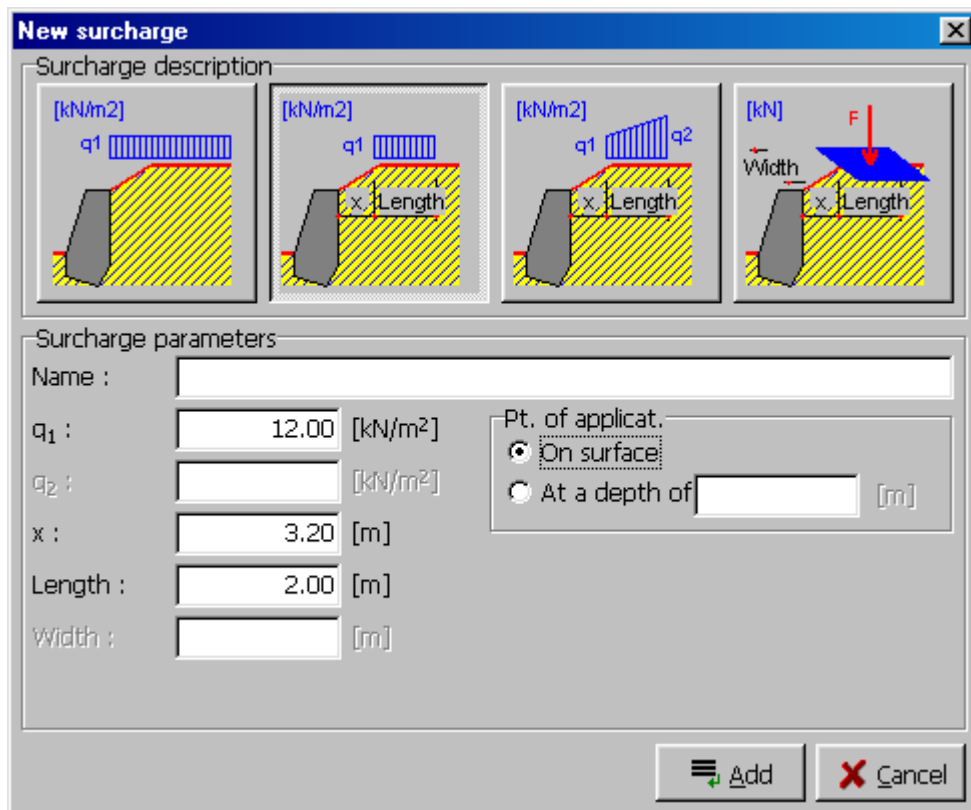


Fig. 2.34 – Dialog box to insert a new surcharge

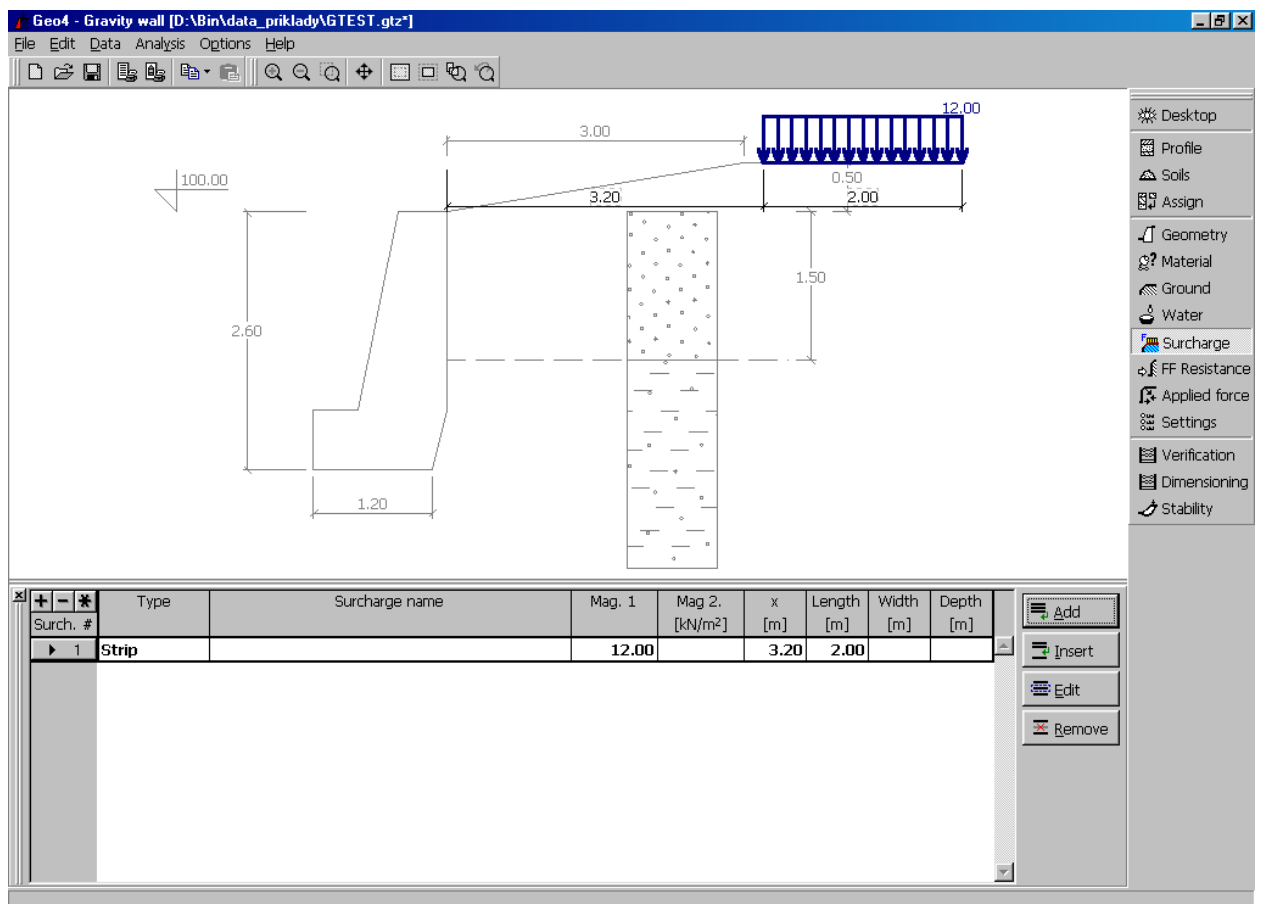


Fig. 2.35 – Inserting data relevant to mode surcharge

Complementary information

Procedure, which allows you to modify individual items in the list of surcharges with the help of the **Edit**, **Insert**, and **Remove** buttons, has been already described in **chapter 2.4 Soils and rocks** (complementary information). This procedure is identical for all lists. The proceeding paragraph yet describes graphical options you may exploit when working with lists:

- 1) To select surcharge click on a given surcharge plotted on the desktop using the left mouse button
- 2) Double clicking on a given surcharge plotted on the desktop opens the editing dialog box
- 3) Surcharges selected in the list (using buttons marked with the surcharge number) are colored in green
- 4) When pressing the “**Remove**” button, the selected surcharge is colored in red
- 5) All surcharges can be edited in the graphical regime by clicking on corresponding dimensions and then changing their values

Complementary information

2.2.9 Front face resistance (FF resistance)

Fig. 2.36 shows the dialog box designed to set up a resistance developed on the front face of a structure. As usual you first select the earth resistance type (passive, active, or at rest) and fill in the required parameters. In the present example select the pressure at rest.

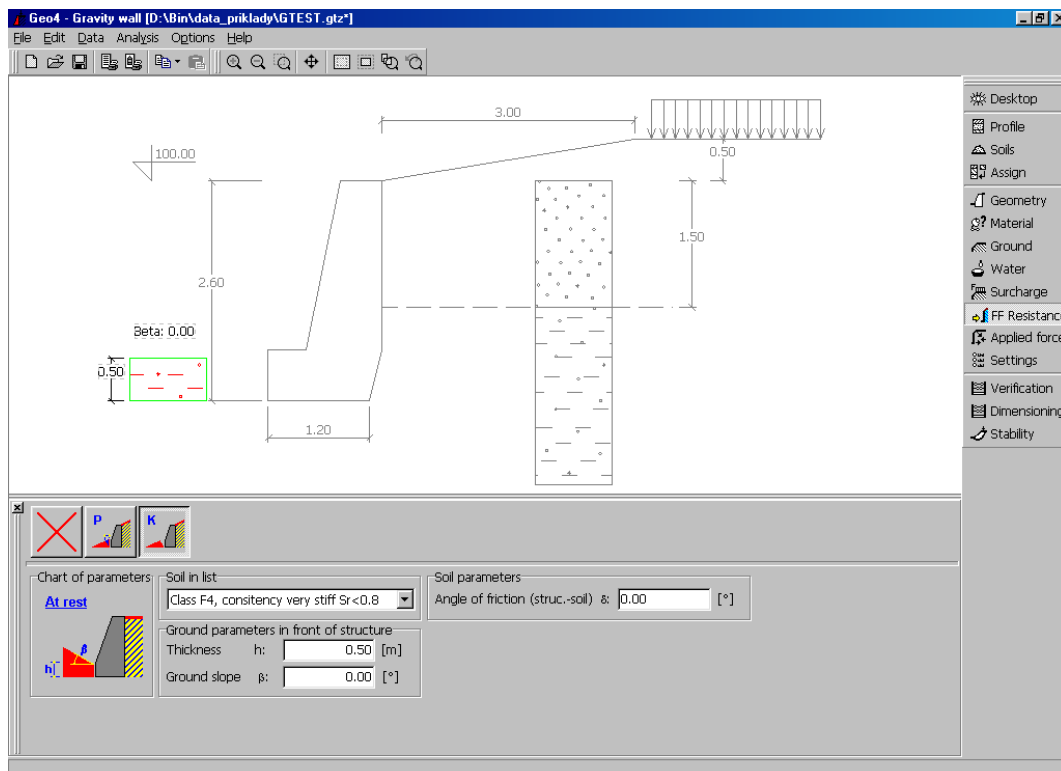


Fig. 2.36 – Inserting data relevant to mode FF resistance

2.2.10 Applied forces

In this mode, you are able to put in an arbitrary number of forces acting on the structure such as anchors, safety fences, forces acting on the wall crest, etc. In this example, however, they will not be considered. Nevertheless, to apply such forces you switch to mode “**Applied forces**” and then continue by following the procedure described in **Section 2.2.8** (surcharge). The “**Add**” button opens the “**New force**” dialog box (see **Fig. 2.37**), in which you may insert the required force parameters. This procedure can be repeated as many times as necessary. The coordinate system accompanying an input of additional forces is displayed on the desktop.

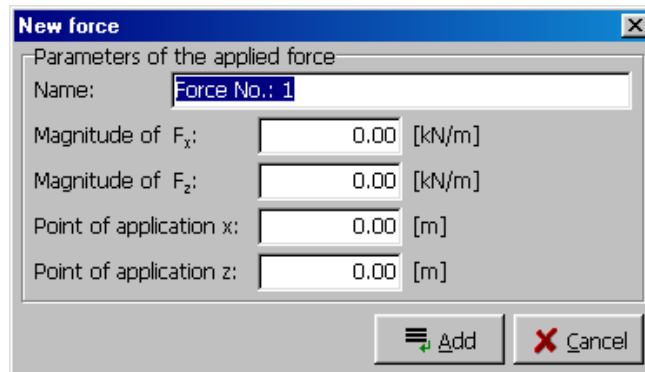


Fig. 2.37 – Dialog box to insert applied forces

2.2.11 Setting parameters controlling the analysis process

This mode allows you to control the solution process by selecting the desired type of analysis. You may choose the standard limit state analysis or carry out the analysis according to the classical theory without reduction of input parameters. In the last option you may adjust the analysis by setting up the reduction coefficients. Details are provided in the theoretical manual.

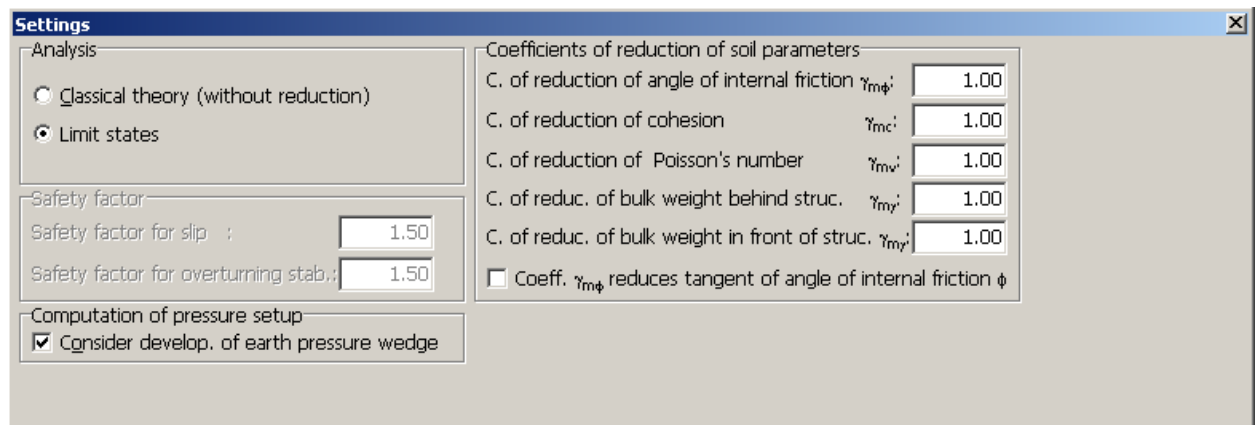


Fig. 2.38 – Setting analysis parameters

2.3 Analyses and outputs

2.3.1 Verification analysis

This mode serves to check the entire wall with respect to the overturning and sliding stability and bearing capacity of the foundation soil. Entering this regime results in computation of all forces acting on the structure (gravity force, earth pressures, forces due to surcharge, etc.). These forces are usually termed the action effects. Magnitudes and locations of these forces are listed in the table located in the left bottom corner of the screen. An example of a structure with forces is plotted in **Fig. 2.39**.

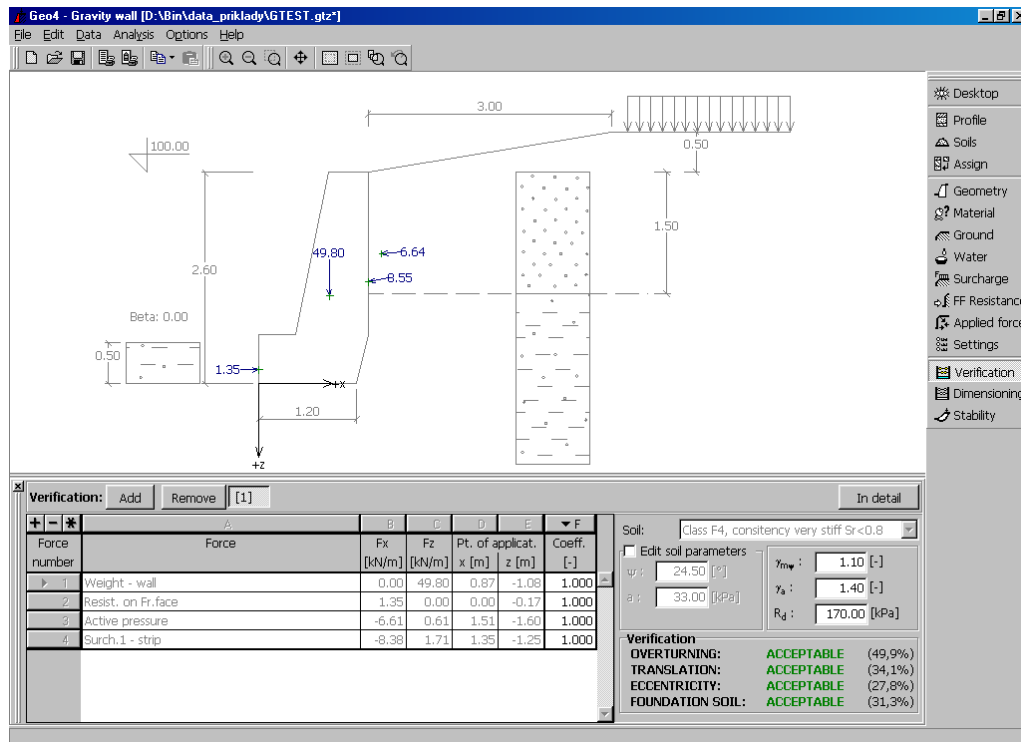


Fig. 2.39 – Results after verification and their layout on the desktop

Column “F” contains coefficients to adjust magnitudes of individual forces. The program allows you to edit these coefficients directly in the table. All coefficients are initially set to **1.00**, but you are able to modify these coefficients as necessary. For example, if you wish to see results derived when excluding the front face resistance from the analysis you simply set the relevant coefficient to zero and the entire verification analysis is carried out again. Note that in such a case the force corresponding to the front face resistance is not displayed on the desktop (see **Fig. 2.40**).

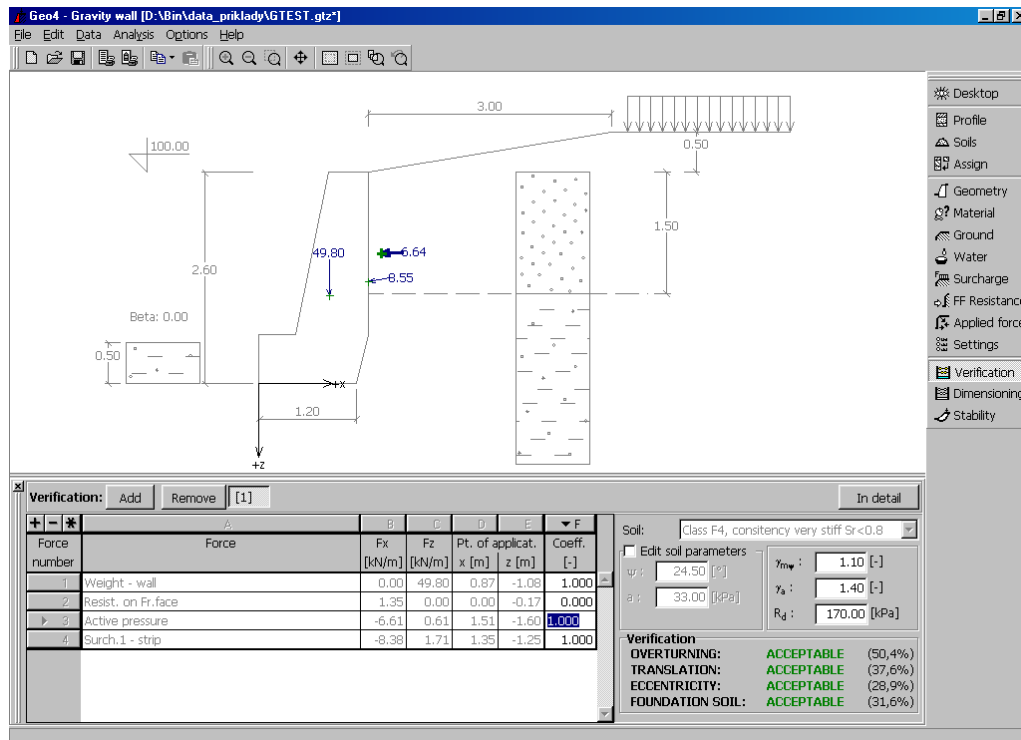


Fig. 2.40 – Results after verification analysis and their layout on the desktop

In addition, you can modify parameters and bearing capacity (R_d) of the foundation soil together with the coefficients of reduction of friction and cohesion between foundation and soil. Corresponding entry fields are located on the right of the work area. Concise results of the verification analysis, pertinent to new input data, appear on the right bottom corner of the work area. To get more information click on the “In detail” button (see Fig. 2.41).

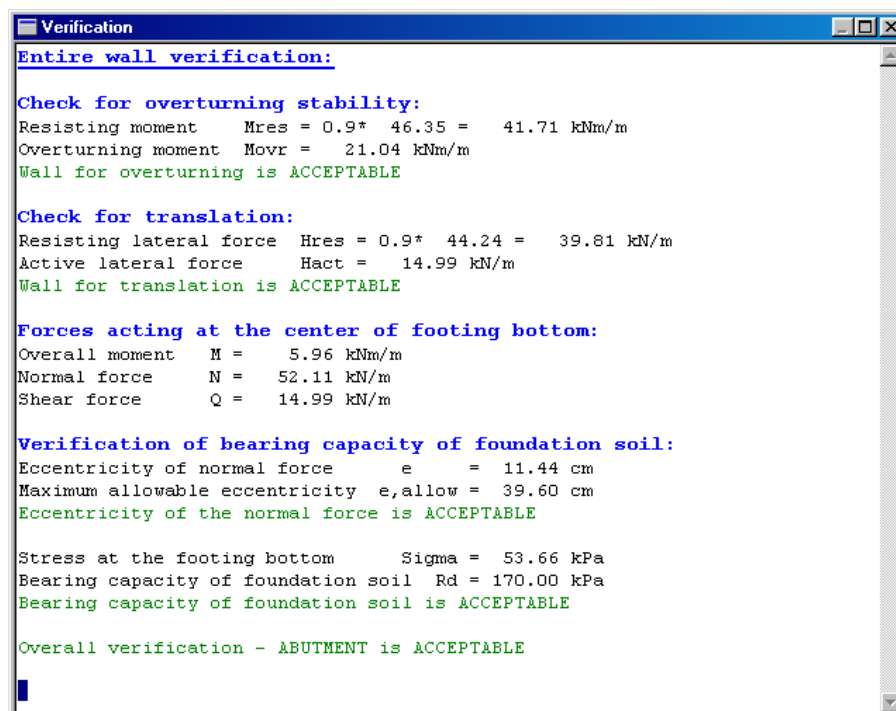


Fig. 2.41 – Detailed description of results of the verification analysis

The proper meaning of buttons “Add”, “Remove”, [1] is explained in the next section.

2.3.2 Dimensioning

Switch to mode “**Dimensioning**” using the appropriate item on the toolbar. Internal forces acting on the structure are again automatically evaluated. To adjust levels of the applied load proceed as in the previous example. An arbitrary number of analyses can be carried out in this mode. Default setup corresponds to the verification analysis of the wall stem design (see Fig. 2.42).

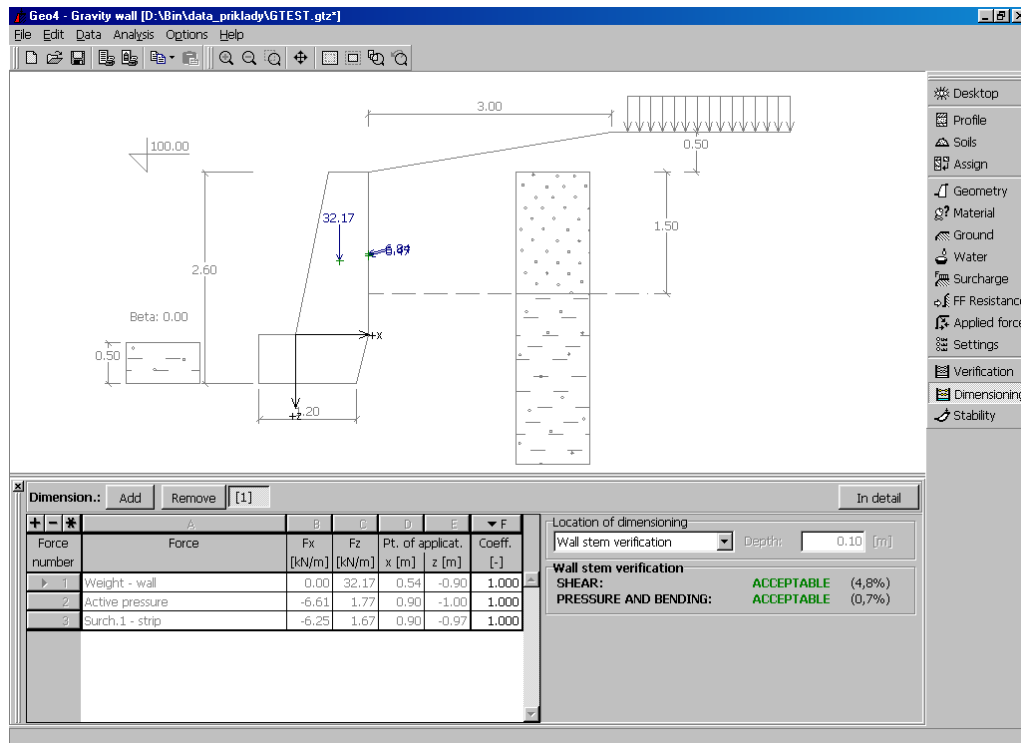


Fig. 2.42 – Detailed results of dimensioning analysis No. 1

The “**Add**” button allows you to add another analysis (individual analyses are displayed on the bar as [1], [2], [...] and are mutually independent). Using the drop-down list select the item “**Wall stem check**”, which automatically calls the analysis solver. Results then appear on the bottom of the work area (see Fig. 2.43).

To move between individual analyses, use the [1], [2] buttons. The “**Remove**” button can be used to remove the current analysis.

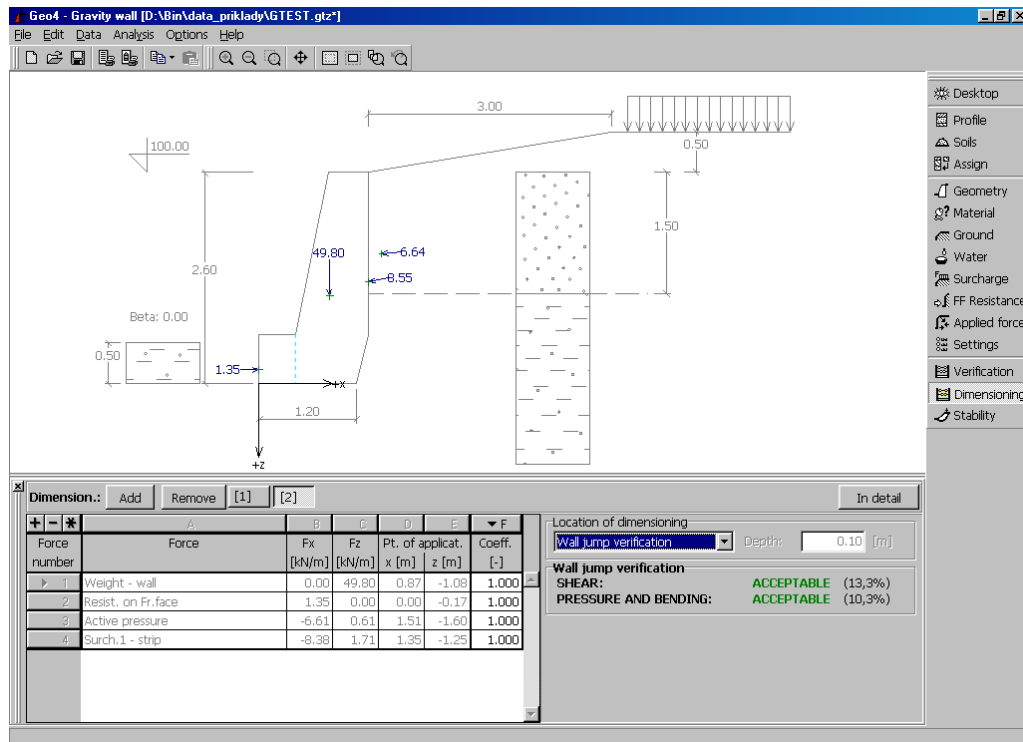


Fig. 2.43 – Detailed results of dimensioning analysis No. 2

Verification analysis at the construction joint is performed similarly. Analysis results are shown in Fig. 2.44.

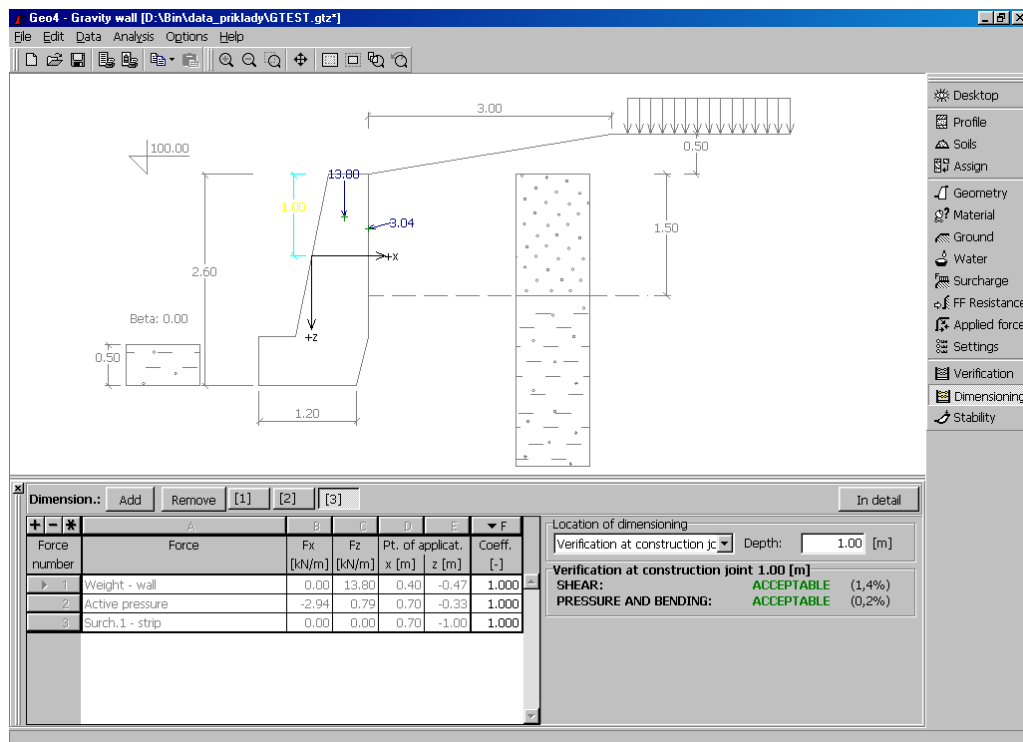


Fig. 2.44 – Detailed results of dimensioning analysis No. 3

2.3.3 Verification analysis of external stability

Providing you have the module “**Stability of slopes**” you may calculate the external stability of the wall structure. To activate this module, click on the item “**Stability**” on the vertical toolbar (see Fig. 2.45). The “**OK**” button then imports the results derived from the stability analysis back to the module “**Gravity wall**”. However, when changing the input data in the module “**Gravity wall**” the previously obtained results are automatically deleted and the stability analysis must be run again.

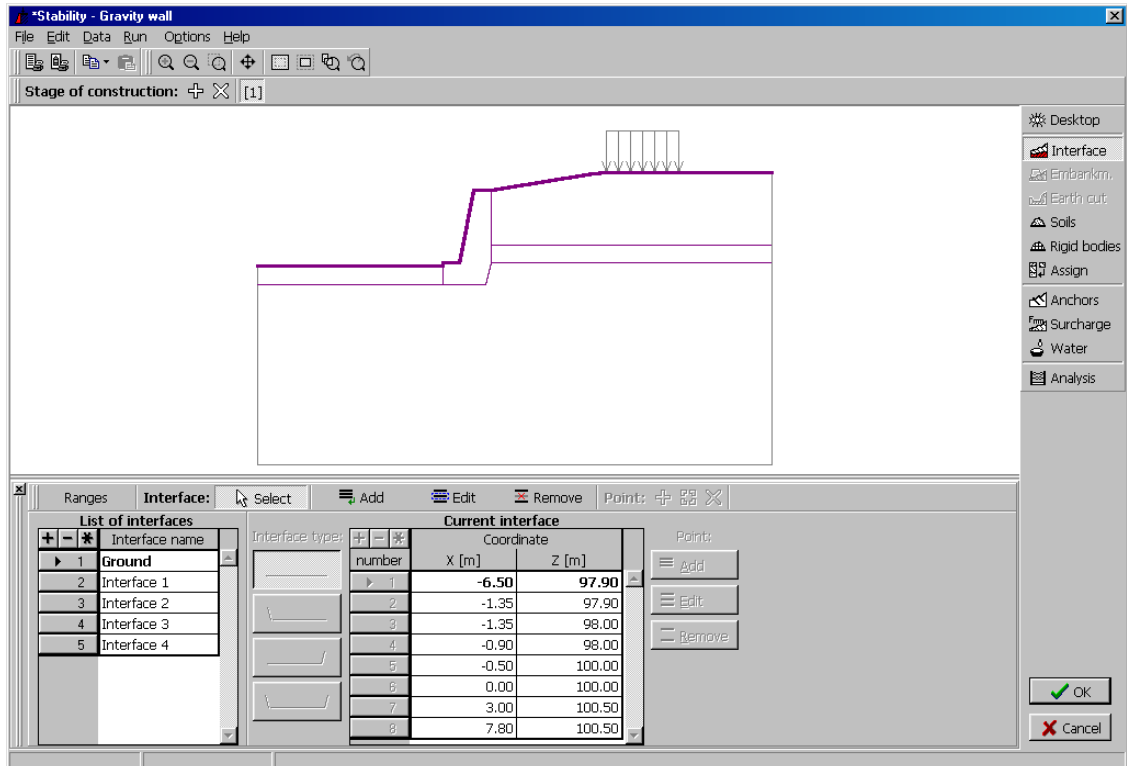


Fig. 2.45 – Verification analysis of external stability

2.3.4 Data check

Before each analysis (verification analysis, dimensioning) the program checks all input data. If the input data are not properly set (e.g. no soil is assigned to a certain layer in the ground profile) the program prompts an error message (see Fig. 2.46) and the user is asked to correct them.

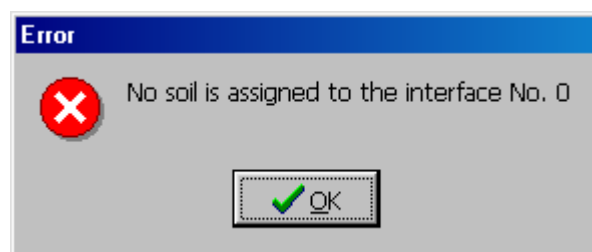


Fig. 2.46 – Example of an error message during the input data check

2.3.5 Text outputs

From the menu item “File” select “Text printout” or click on an appropriate button on the toolbar to open the text printout dialog box (see Fig. 2.47). On the left part of the work area, mark items you wish to print out and then click on the button “Generate” to create an output protocol. Since the protocol is generated not before pressing this button it always complies with the current data. The basic user's guide “A tour of the program environment”, (chapter 2, section 2.5.1. “Text printouts”), provides a thorough discussion on handling the text output dialog box.

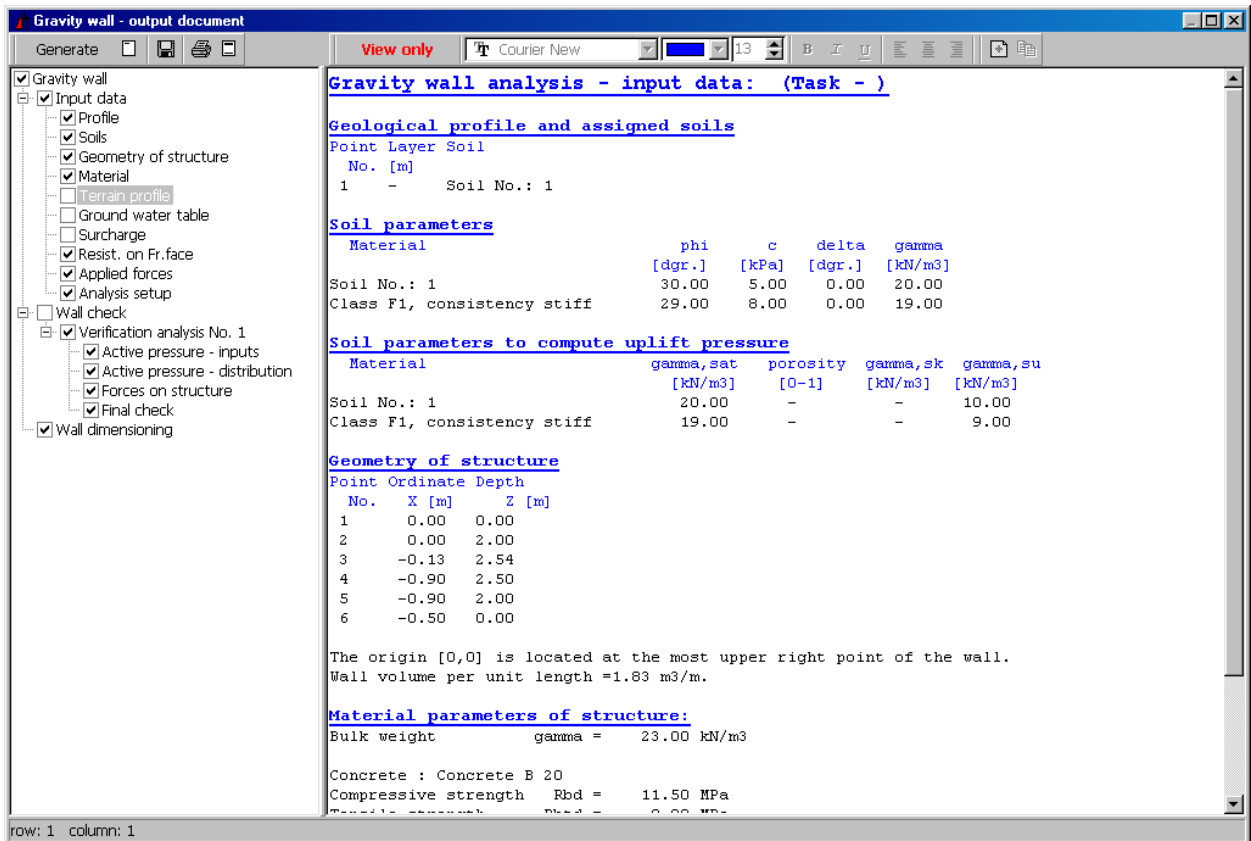


Fig. 2.47 – Text outputs setting

2.3.6 Graphical outputs

To request a graphical printout you may proceed in two different ways. In general, plots displayed on the desktop are printed out using the “**Graphic printout**” button located on the toolbar or the same item from the menu. The final step just requires the printer parameters setup (see **Fig. 2.48**) and printout confirmation.

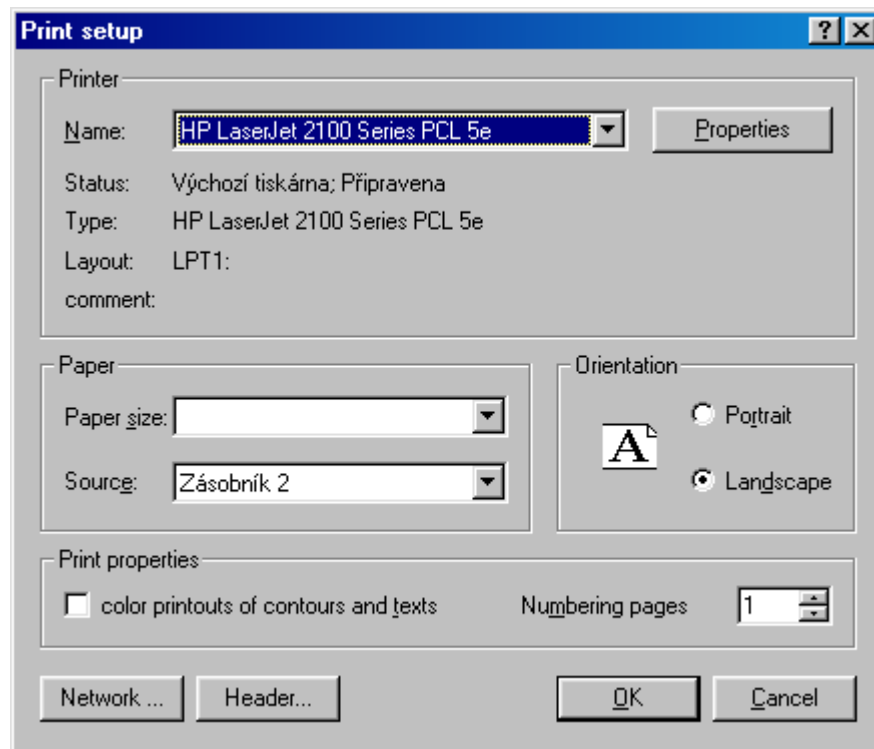


Fig. 2.48 – Graphic printout

The second option exploits the “**Copy picture**” button (see the same command in the menu). This command copies the selected picture to the clipboard. The “**Insert**” button then allows you to import the picture into an arbitrary editor (e.g. Microsoft Word, see **Fig. 2.49**).

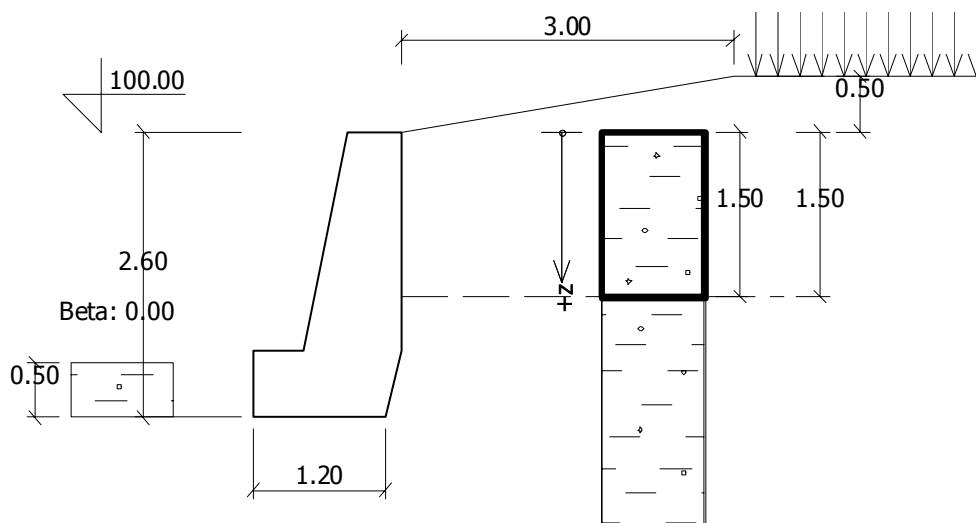


Fig. 2.49 – Picture copied to the clipboard