Table of contents

3. Stability of earth slopes (file: demo01.gst)

This chapter helps you to become familiar with essential features of the module **"Stability of slopes"**. The quickest way, as in the previous chapter, is to start with an example.

The information presented in this chapter is useful for other GEO4 programs as well:

- **interface input 2D** - used in **Beam, Rock stability, Settlement, FEM**
- **stages of construction** – used in **Sheeting verification, Settlement,.FEM**

3.1 Problem definition and input data

In this example, you wish to check the slope stability for a given structure and if necessary to take actions for its enhancement. The current slope is displayed in **Fig. 3.1**. **Fig. 3.2** then shows an additional surcharge caused by the structure. Input data are listed on the next page.

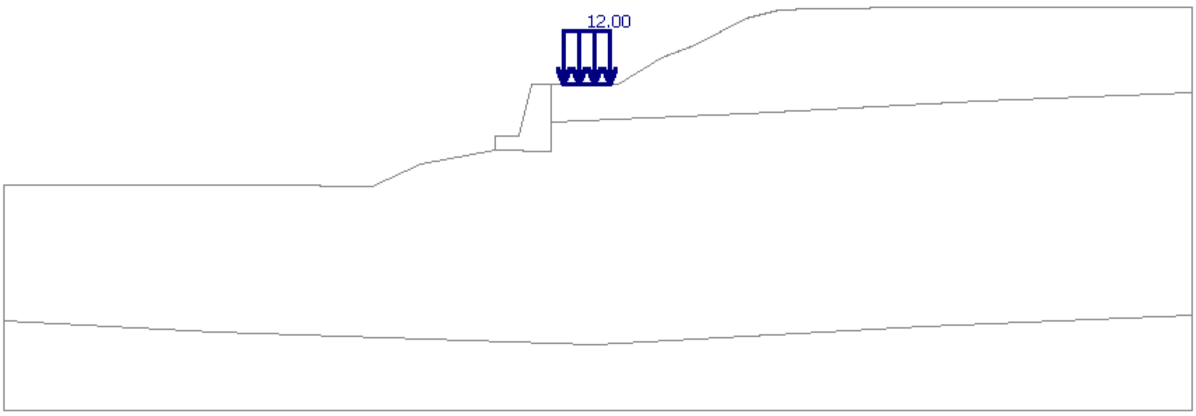

Fig. 3.1 – Problem definition – current state

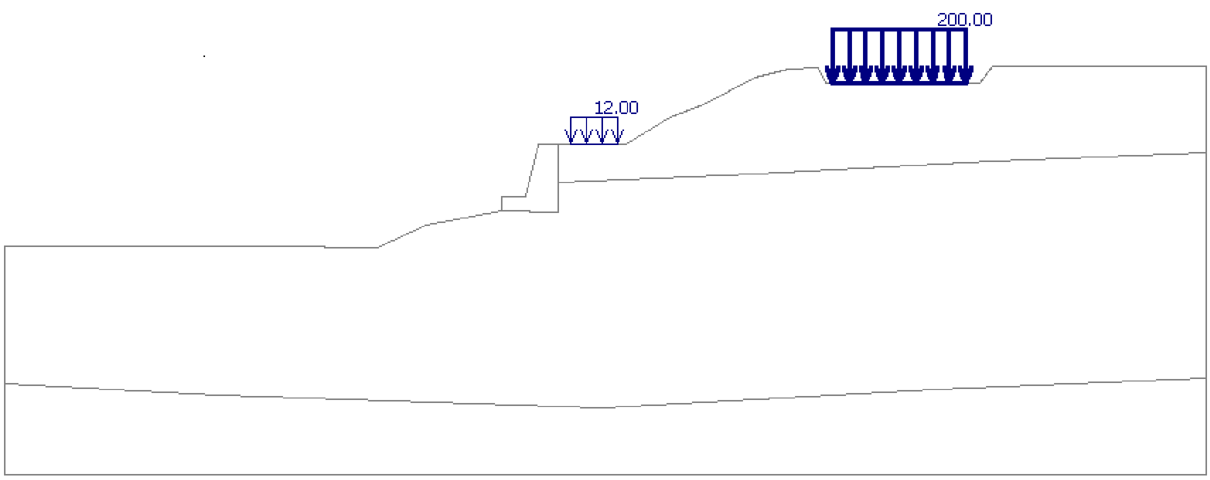

Fig. 3.2 – Problem definition – new state with a surcharge

Soil parameters

Material	fi [dgr.]	c [kPa]	gamma [kN/m3]	gamma,sat [kN/m3]
Class F5 stiff consistency	21.00	12.00	20.00	22.00
Class F3 stiff consistency	26.50	16.00	18.00	18.00
Class R4, brittle deformation	40.00	50.00	19.00	22.00

Ground coordinates:

Assigned soil: Rigid body Nr.1

Point No.	Depth [m]	X-coord. [m]
1	-20.00	115.32
2	0.00	115.32
3	7.89	115.20
4	11.54	116.85
5	17.25	118.00
6	17.25	119.00
7	19.00	119.00
8	20.00	122.98
9	26.50	122.98
10	29.80	124.92
11	32.39	125.92
12	36.16	127.92
13	38.69	128.51
14	41.22	128.69
15	50.00	128.75
16	70.00	128.75

Interface btw. layers No.1:

Assigned soil: Class F5 stiff consistency

Point No.	Depth [m]	X-coord. [m]
1	17.20	117.99
2	21.50	117.90
3	21.50	122.98

Interface btw. layers No.2:

Assigned soil: Class F3 stiff consistency

Point No.	Depth [m]	X-coord. [m]
1	-20.00	115.32
2	0.00	115.32
3	7.89	115.20
4	11.54	116.85
5	17.20	117.99
6	21.50	117.90
7	21.50	120.02
8	36.18	120.75
9	53.99	121.70
10	70.00	122.34

Interface btw. layers No.3:

Assigned soil: Class R4, brittle deformation

Point No.	Depth [m]	X-coord. [m]
1	-20.00	105.06
2	-3.99	104.21
3	24.73	103.26
4	49.75	104.63
5	70.00	105.48

3.2 First construction stage – original state

3.2.1 Interface input

Using the vertical toolbar switch to mode **"Interface"** (see **Fig. 3.3**). Selecting this item opens the **"Profile"** dialog box on the work area. It lists inserted interfaces. In addition, the **"Interface"** toolbar located above the profile dialog box serves to either add new or modify existing data. Both tools are shown here as floating windows. The default setting, however, corresponds to the location at the bottom of the work area (see **Fig. 3.5**).

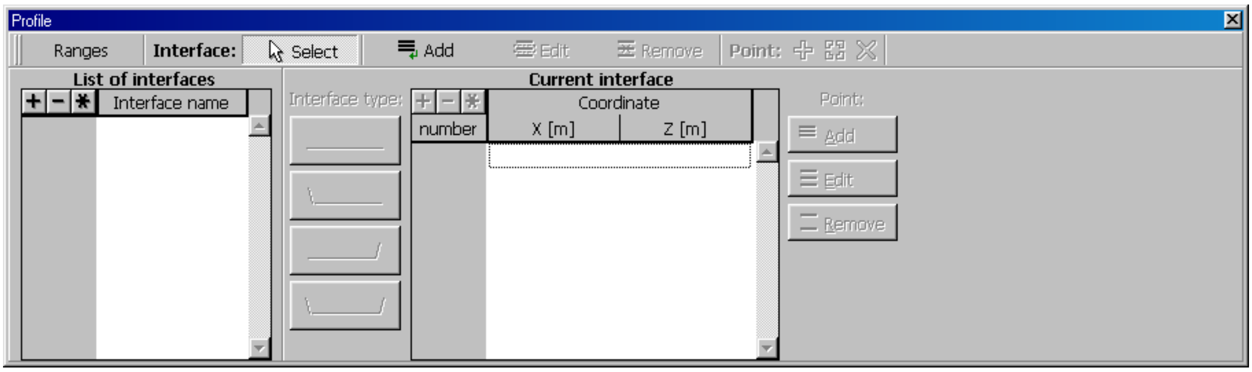

Fig. 3.3 – Dialog box and toolbar to insert an interface

The first step is to set dimensions of the work area. To select the minimum and maximum dimensions of the slope, click on the **"Ranges"** button, which opens the **"World coordinates"** dialog box, **Fig. 3.4**.

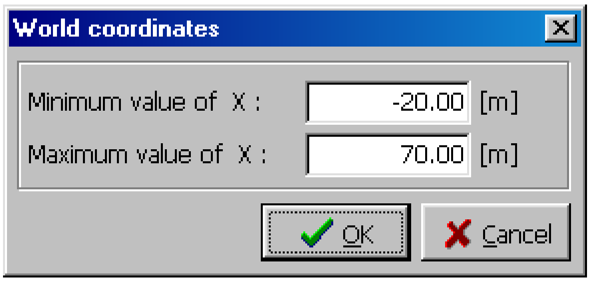

Fig. 3.4 – Dialog box to enter dimensions of the work area

You can modify the current setting any time – program then automatically cuts-off or prolongs individual layers, respectively.

You may now proceed to insert new interfaces. Begin with the ground profile. To activate the **"Interface"** toolbar, click on the **"Add"** button.

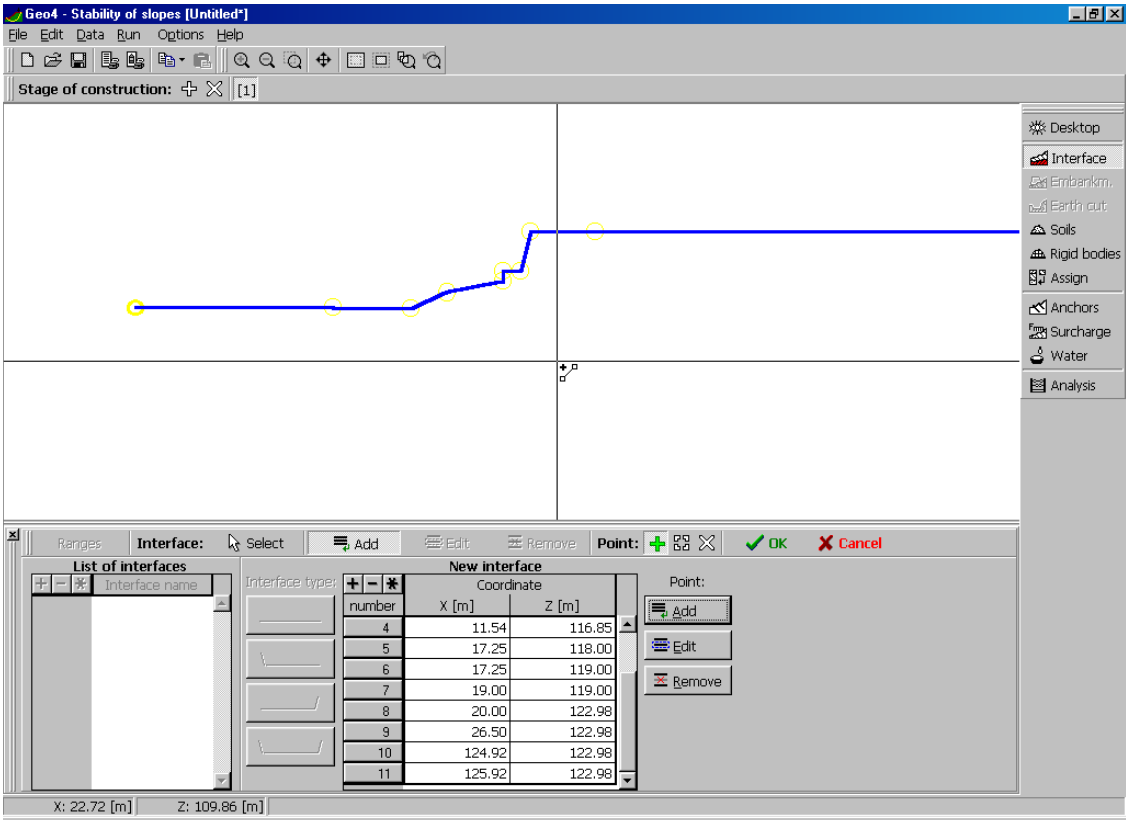

Fig. 3.5 – Inserting and editing an interface

To insert interface points you may proceed in two ways:

1) By clicking the left mouse button in the drawing window (the pointer changes into a crosshair to indicate an insert mode, **Fig. 3.5**)

2) By setting point coordinates directly in the dialog box (see **Fig. 3.6**)

When selecting the first option, you determine the coordinates of a point only approximately (the cursor current position is indicated on the left bottom corner of the work area). A straight line is drawn between a new and the last but one inserted points. At the same time, the coordinates of each point are stored into the list **"Current interface"** (or **"Edit interface"**). Using the **"Edit"** button then allows you to adjust the coordinate values as necessary.
When using the dialog box, you may enter the desired values directly. As above, these values are stored into the list. Lines connecting individual points are immediately plotted on the desktop.

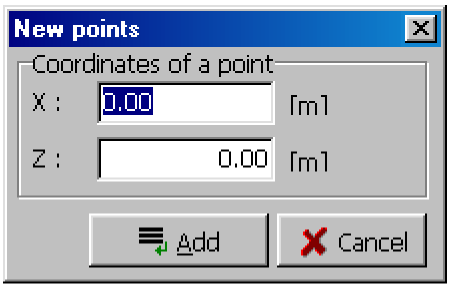

Fig. 3.6 – Dialog box to insert coordinates of interface points

The toolbar buttons **"Add"** and **"Edit"** activate another three buttons (see **Fig. 3.7**), which can be used to modify coordinates of points on the interface.

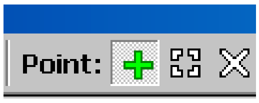

Fig. 3.7 – Buttons for modifying the interface

Complementary information

Fig. 3.8 shows a dialog box designed to edit coordinates of a point. Using the **"OK+←"** and **"OK+→"** buttons, you can move along the entire interface – the point, you wish to edit, does not have to be picked up each time from the list.

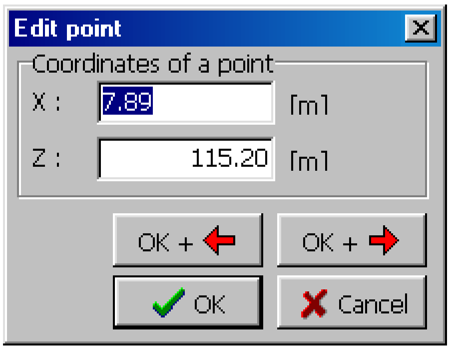

Fig. 3.8 – Dialog box to edit coordinates of interface points

None of the inserted points will be stored unless you select the **"OK"** button to exit the insert mode. If you use the **"Cancel"** button, the program opens a warning message dialog box asking to confirm this step (see **Fig. 3.9**).

Fig. 3.9 – Dialog box to confirm canceling the interface insert mode

If the x-coordinate of the point you wish to put in already exists, the program opens the **"New point at the interface"** dialog box (see **Fig. 3.10**), in which you have to chose positioning of a new point – a vertical line then appears on the desktop.

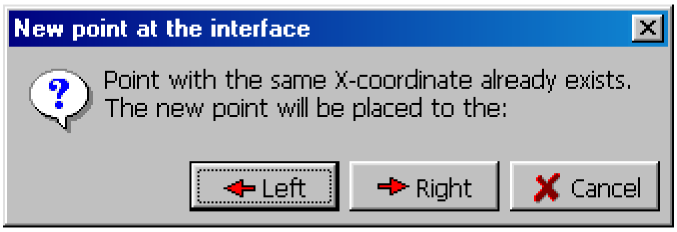

Fig. 3.10 – Positioning of a point when inserting an interface

Neither two identical points nor three points having the same x-coordinate can be assigned to a single interface. In such a case the program prompts an error message and the point is not stored into the list.

In addition, the program advises through a number of error message dialog boxes whenever the action you want to take is invalid or meaningless (see **Fig 3.11**).
Obviously you may remove the entire interface all at once. To do so you first designate the desired interface in the list of interfaces and then, to delete interface, click on the **"Remove"** button on the toolbar. As usual, you are asked to confirm this action first (see **Fig. 3.12**).

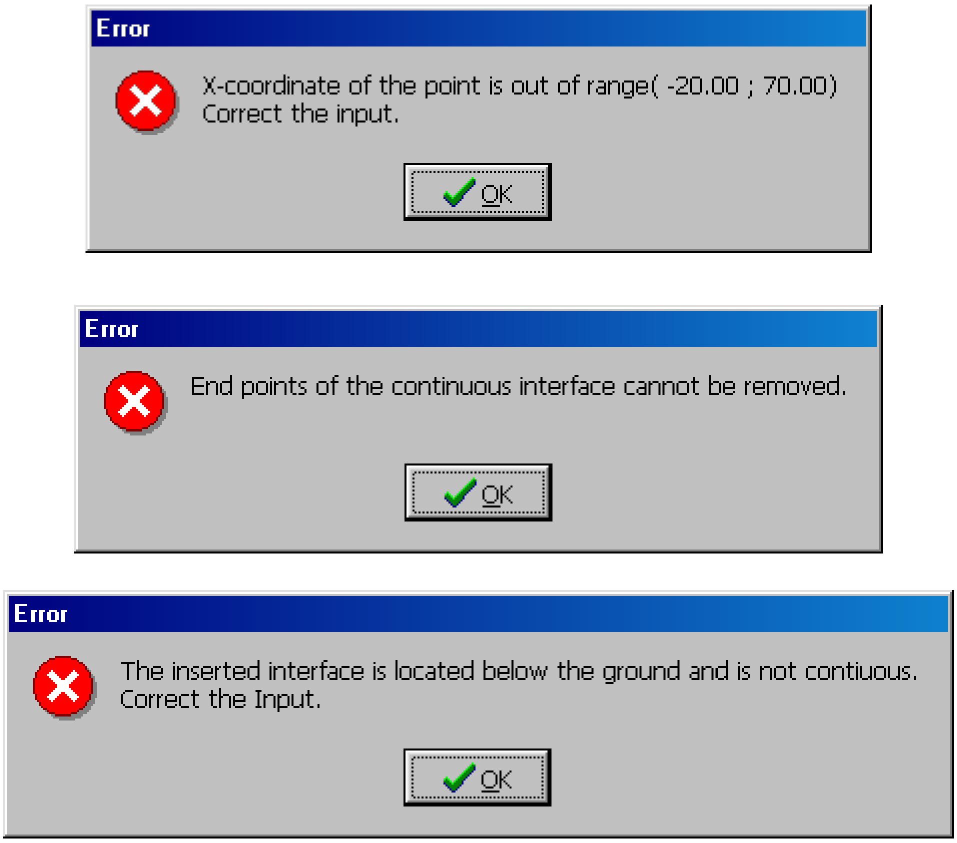

Fig. 3.11 – Error messages when inserting and editing an interface

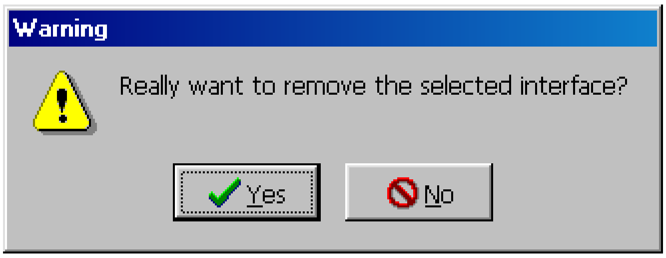

Fig. 3.12 – Confirmation dialog box when removing the entire interface

When designating points directly in the list of coordinates, the program does not allow you to delete the end points of the interface (see **Fig. 3.11**).

You may now proceed with inserting the next interface – the wall geometry.

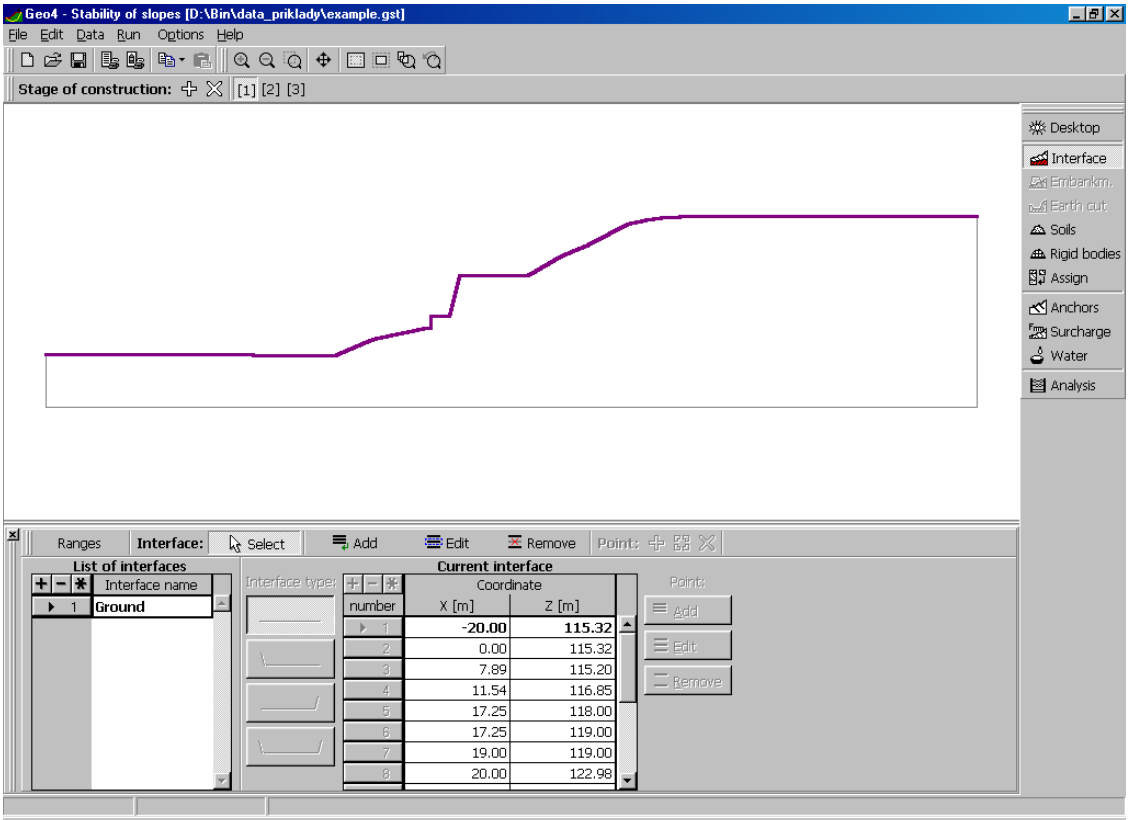

Fig. 3.13 – Dialog box and toolbar to insert an interface

First, using the left mouse button design an approximate shape of a wall (the end points of a new interface will immediately merge with the existing one) and then confirm the interface input. You may then exploit the editing capabilities already discussed and adjust the coordinate values as necessary. Only the x-coordinate of end points can be edited – an interface represents a bounding (on the contrary, only the z-coordinate of end points can be modified when working with a continuous interface).

Complementary information

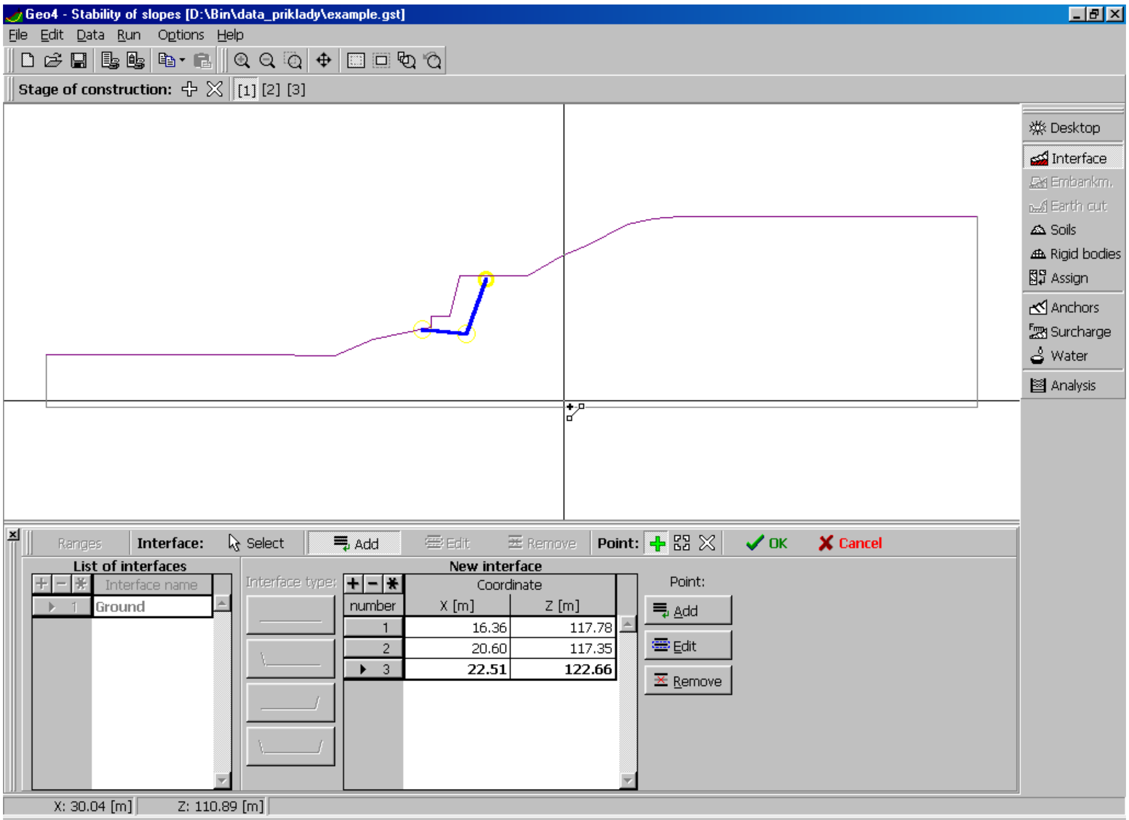

Fig. 3.14 – Designing an approximate shape of a wall

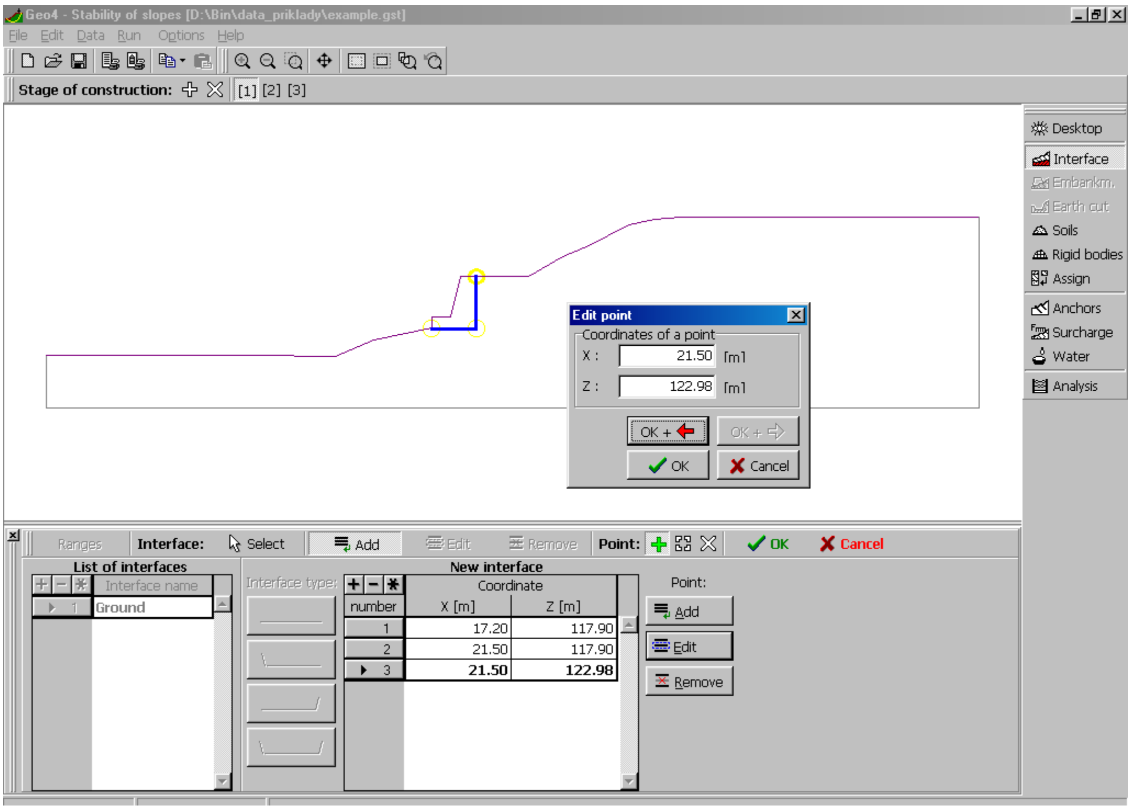

Fig. 3.15 – Correcting an input using the dialog box

Remaining interfaces are inserted in the same way.

Complementary information

The program allows you to switch between various types of interfaces (continuous, bounded, bounded from the left, bounded from the right) using a set of buttons displayed in **Fig. 3.16**.

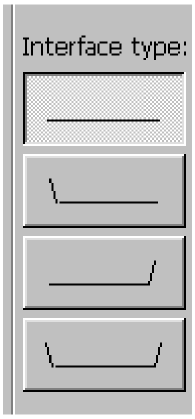

Fig. 3.16 – Predefined types of interfaces

As evident from **Fig. 3.17**, it is extremely simple to add a new interface using the left mouse button. Once you are done the program adjusts a new interface to attain a proper shape.

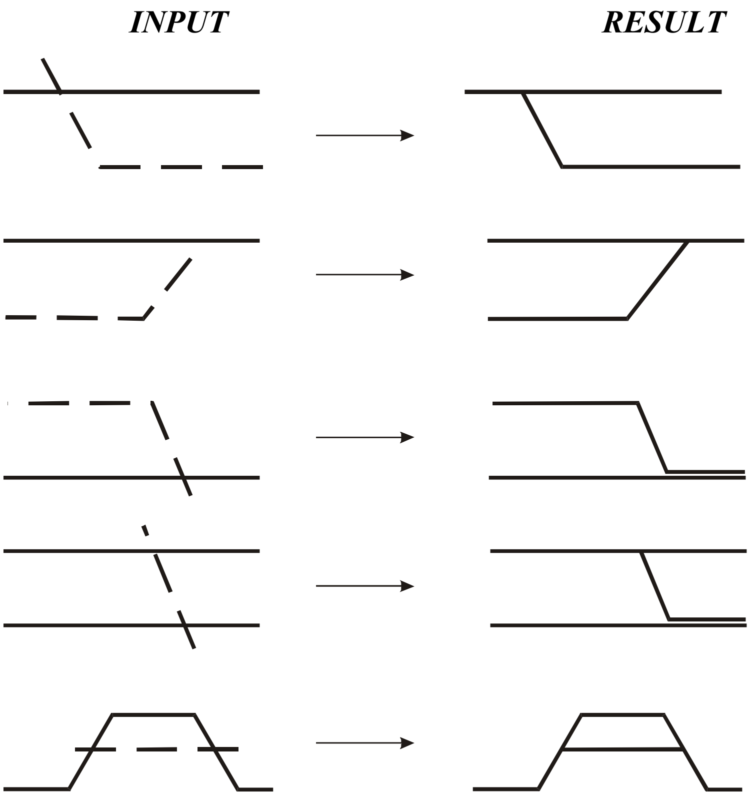

Fig 3.17 – Inserting an interface using mouse

Complementary information

3.2.2 Inserting soil and rigid body parameters

Inserting soil parameters and assigning particular soils to individual layers of the ground profile was discussed in detail in chapter 2. Similarly, you may proceed when putting in data pertained to rigid bodies. The **"Assign soils"** dialog box serves to this purpose (see **Fig. 3.19**).

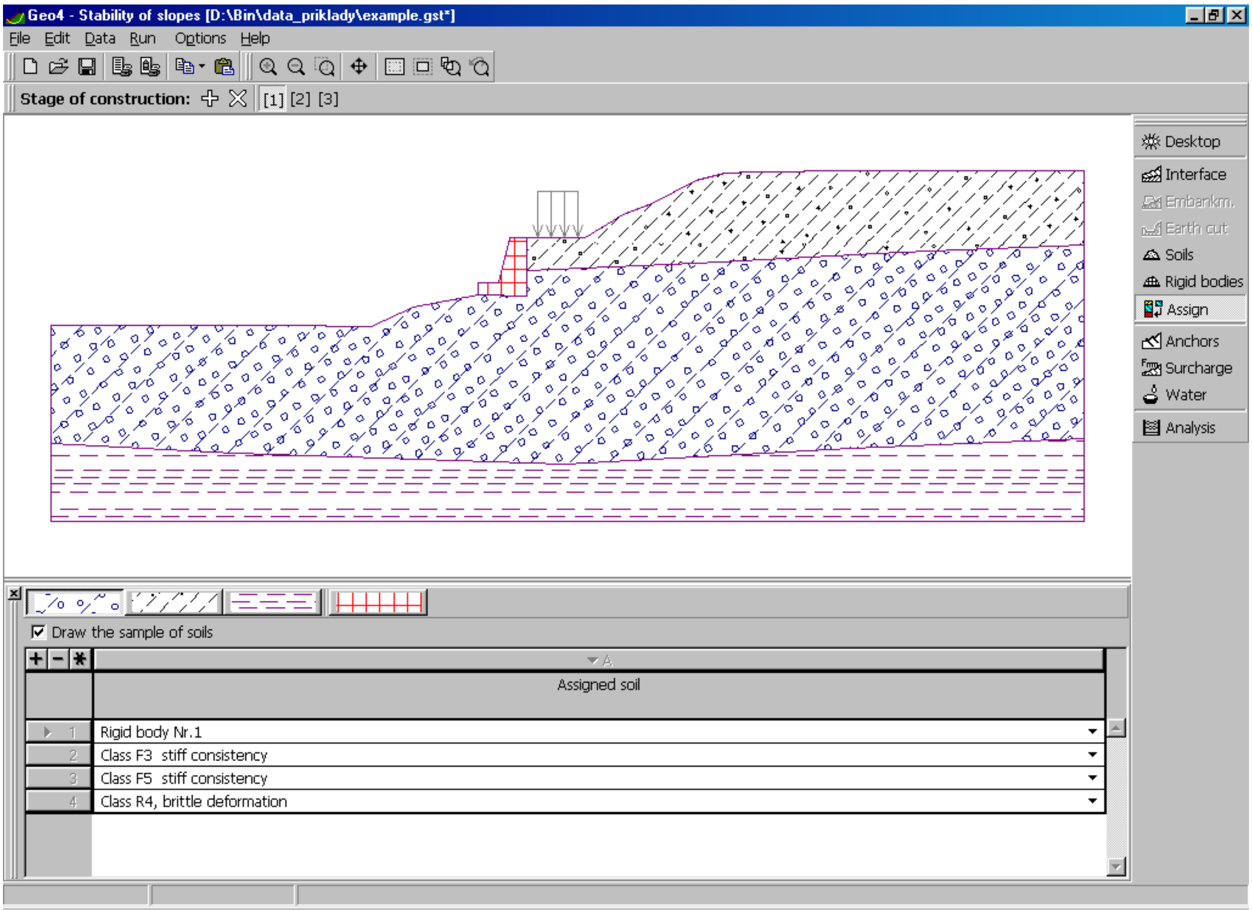

Fig. 3.18 Dialog box to assign soils and rigid bodies

3.2.3 Inserting ground water parameters

There is no water considered in the present example. Nevertheless, the program allows you to enter ground water characteristics in three different ways.

1) The ground water table is put in as a continuous interface both below and above the ground.

2) Coefficients Ru are specified in terms of isolines. The first isoline always coincides with the ground. To insert values of coefficients Ru use the "List of interfaces" table located at the left bottom part of the work area. Values between isolines are linearly interpolated.

3) Pore pressure values are specified in terms of isolines. The first isoline always coincides with the ground. To insert remaining isolines you may follow the procedure described for interfaces. To insert pore pressure values use the "List of interfaces" table located at the left bottom part of the work area. Values between isolines are linearly interpolated.

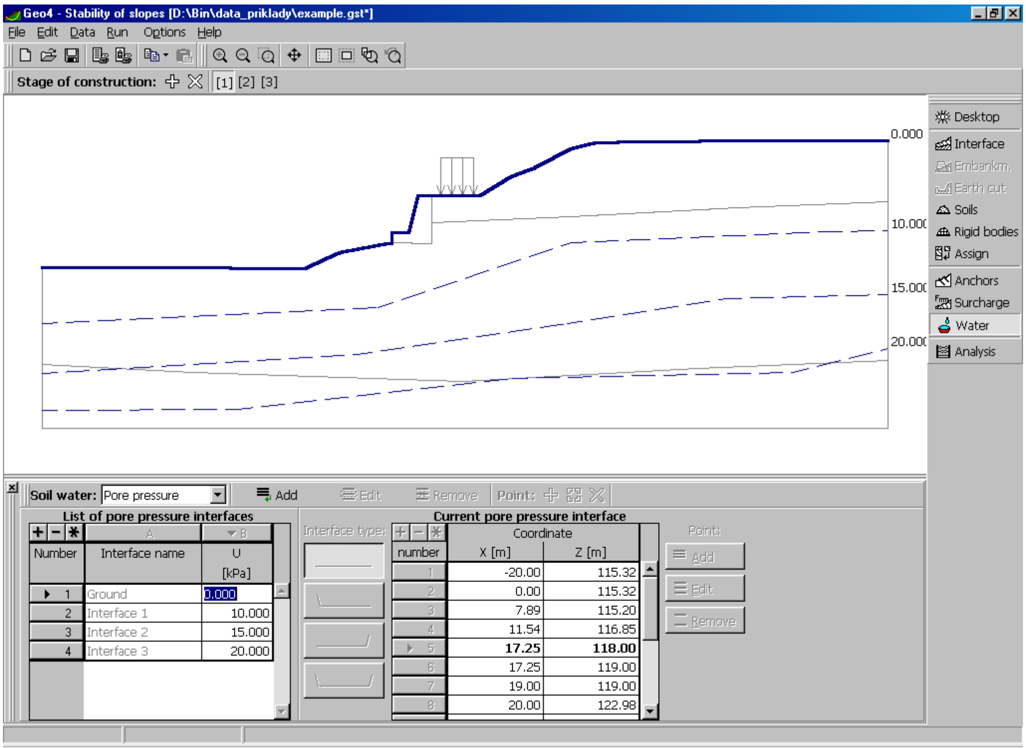

Fig. 3.19 Ground water influence – pore pressure

3.2.4 Inserting a surcharge

For details on setting up sucharge characteristics you are referred to chapter 2 (see also **Fig. 3.20**).

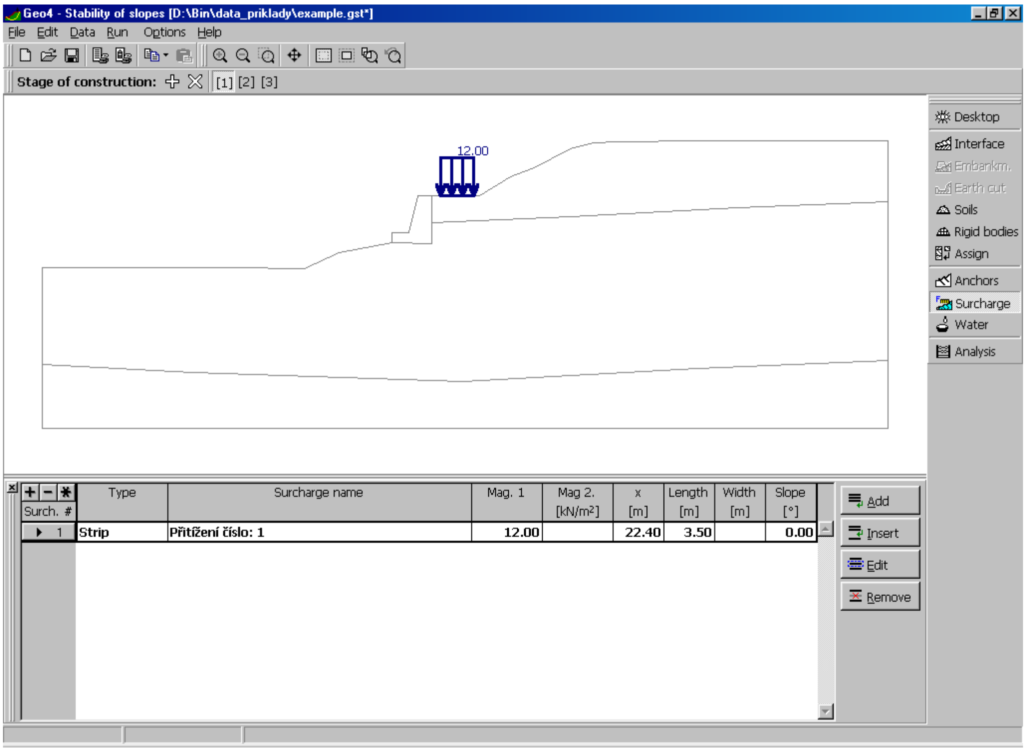

Fig. 3.20 – Inserting surcharge mode

To maintain compatibility with 1D programs (Gravity wall, Earth pressure…), you are also able to specify an area surcharge. Such a loading is specified here as a ground surcharge spread over x-min and x-max coordinates (0, *Max-X*).

3.2.5 Analysis

The **"Analysis"** window allows you to carry out an arbitrary number of analyses of a given slope assuming both the circular and polygonal slip surfaces (see **Fig. 3.21**). Let's first perform the stability analysis using a circular slip surface. Thus, clicking the **"Insert"** button switch to a circular type of a slip surface. Clicking the mouse button then select points to specify the slip surface location.

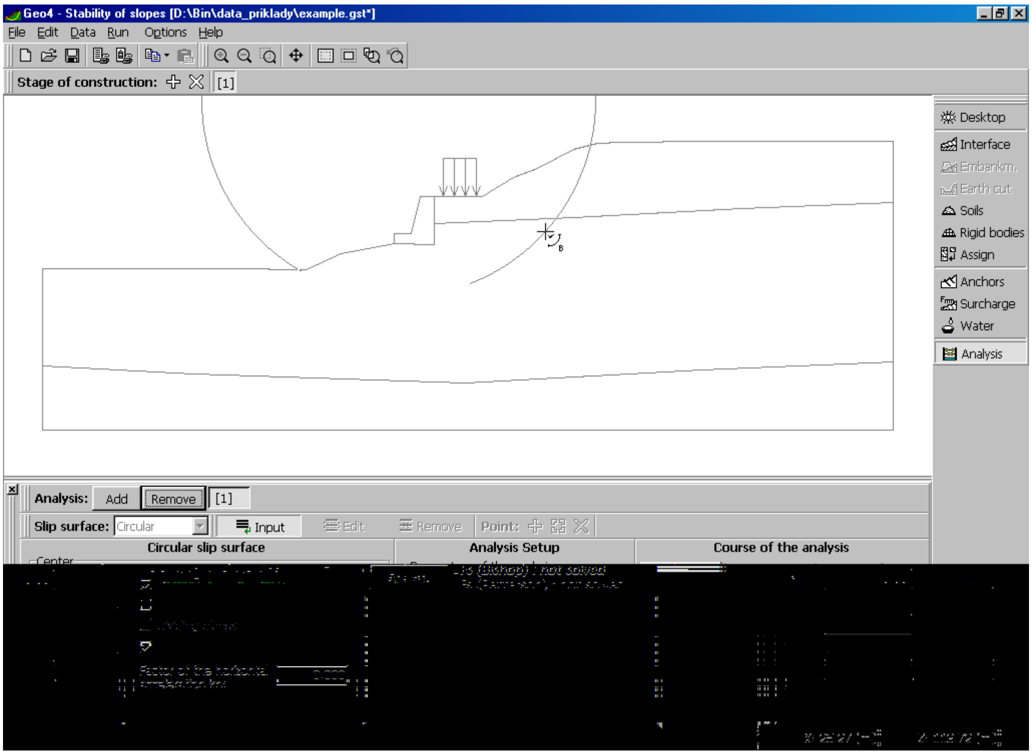

Fig. 3.21 – Mode Analysis – inserting a circular slip surface

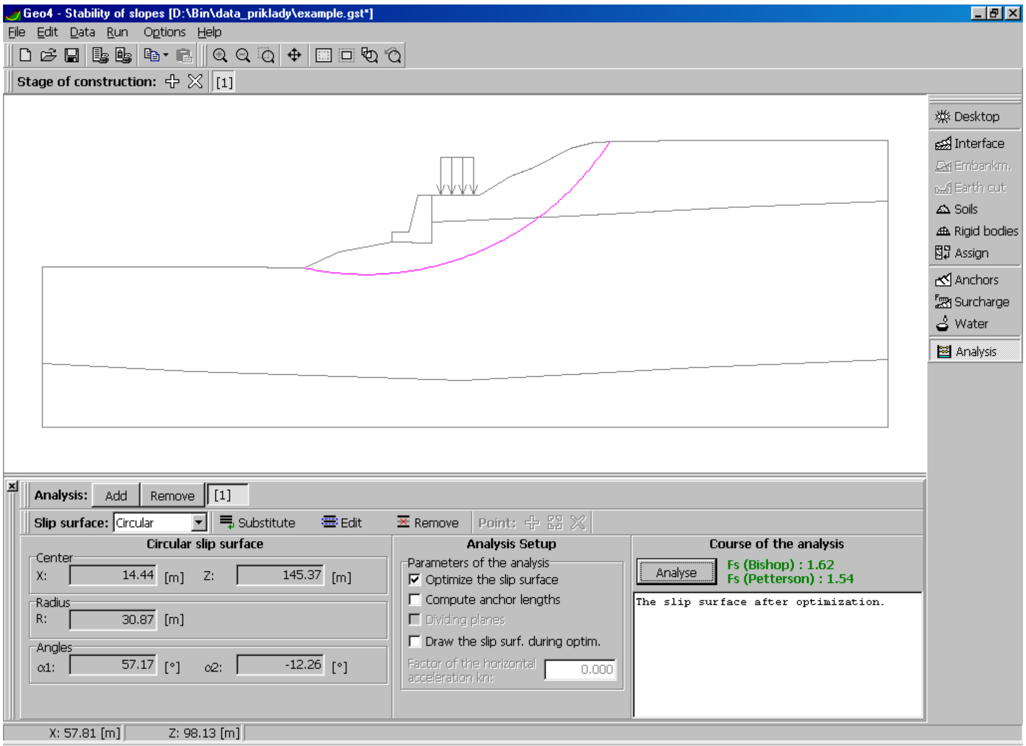

Fig. 3.22 – An optimized circular slip surface

The program is now ready to perform the stability analysis. Set analysis parameters to **"Optimize the slip surface"** and press **"Analyse"**. Results after optimization are shown in **Fig. 3.22**. To enhance the computation efficiency you may turn a slip surface animation off.

Then, clicking the **"Add"** button, add a new analysis and specify a polygonal slip surface. Hint: use the optimized circular slip surface as guidance (see **Fig. 3.23**). Finally, run the stability analysis with optimization. Results appear in **Fig. 3.24**.

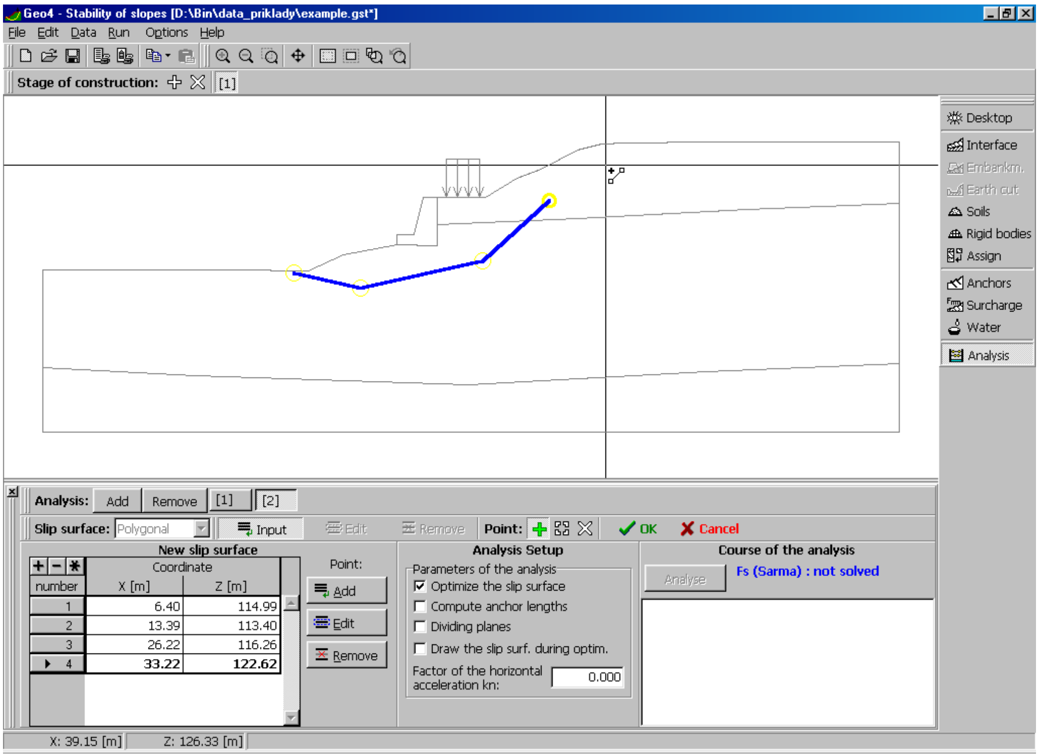

Fig. 3.23 – Inserting a polygonal slip surface

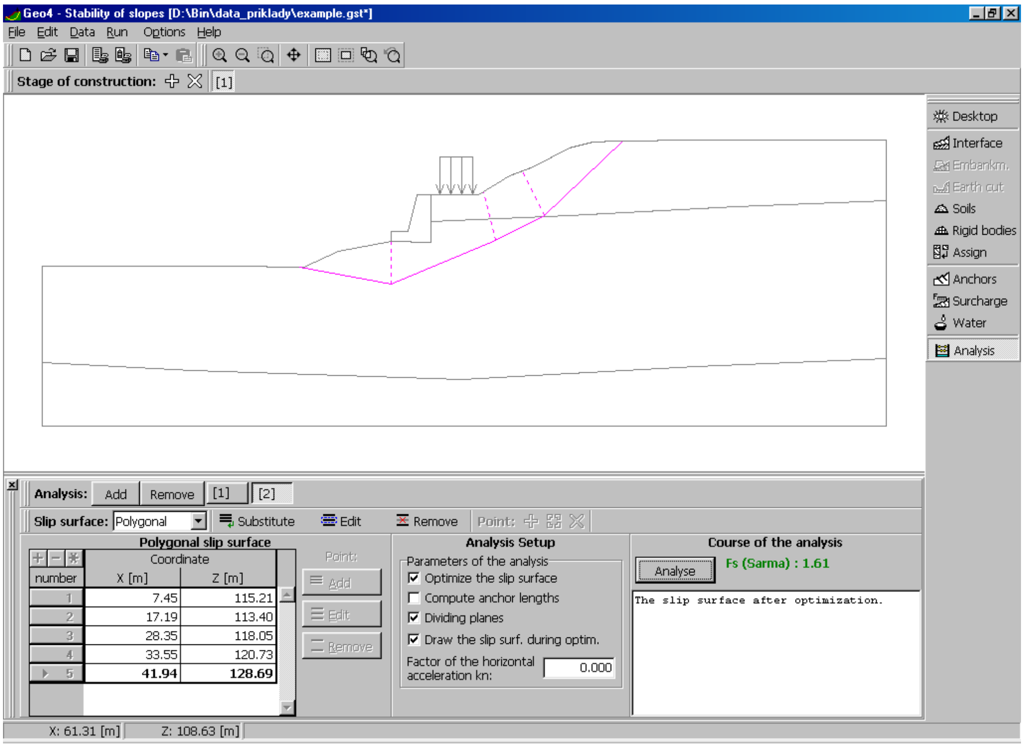

Fig. 3.24 – An optimized polygonal slip surface

Since the resulting safety factors are 1.62 according to Bishop and 1.61 according to Sarma, respectively, we may conclude that the present slope is stable.

3.3 Inserting a new stage of construction

This section describes an important feature of the module, which allows you to model a step-by-step construction of an earth structure within one task through successive construction stages. Note that the **"Sheeting-design"** and **"Settlement"** modules posses the same capability. The **"Stages of construction"** toolbar, located above the drawing window, is designed to handle individual construction stages.

Clicking the **"+"** button you add a new construction stage while the **"x"** button serves to remove the last (not the current one) stage of construction. To move between individual stages, use the **("1", "2", "3")** buttons.
In the first stage (the original state), you may insert and verify an entire structure, while in the subsequent stages you are only able to modify the original structure and run the stability analysis. You are not allowed to change the soil material parameters and layer interfaces. The **"Earth cut"** and **"Embankment"** buttons are available to modify the terrain profile.

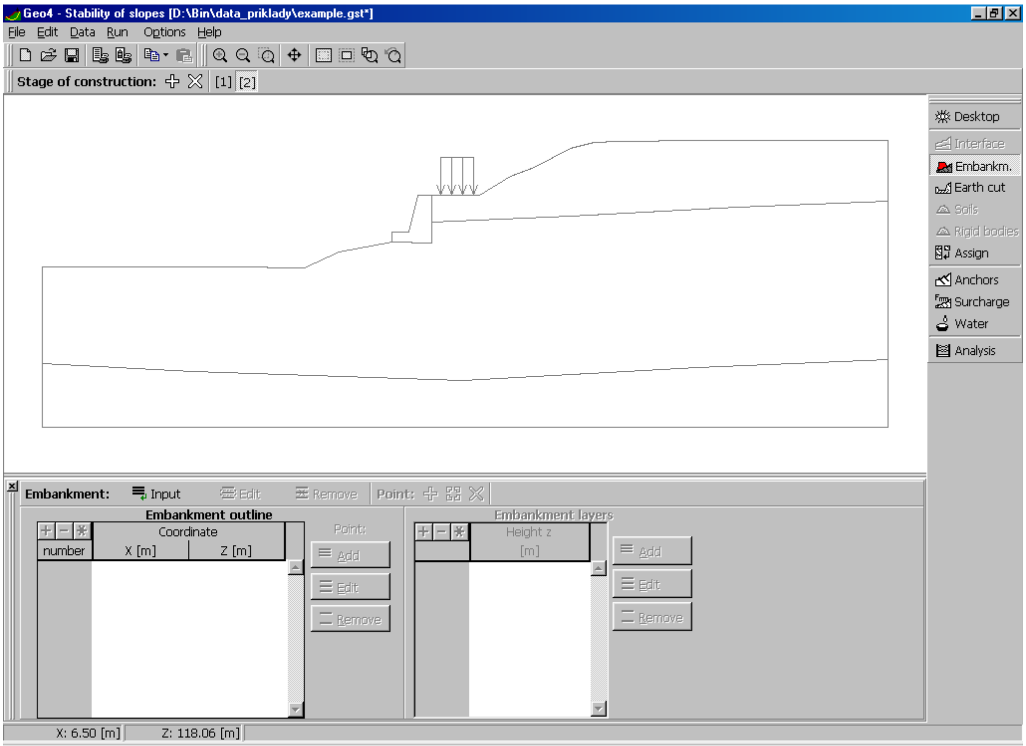

Fig. 3.25 – The second stage of construction

A new stage of construction, until modified, coincides with the previous one. There are certain connections between inserted construction stages – e.g. between individual anchors or surcharges, respectively. If you change, for example, the geometry of a certain anchor in one stage, it is automatically changed in all subsequent construction stages. In those stages you are now only able to either modify its magnitude or take it out.

3.3.1 Modifying the original grade

To exercise the above procedure insert the second stage of construction and then, by creating an earth cut for a new structure, modify the profile of the original grade. To that end, switch to mode **"Earth cut"** and insert the cut dimensions as shown in Fig. **3.26**.

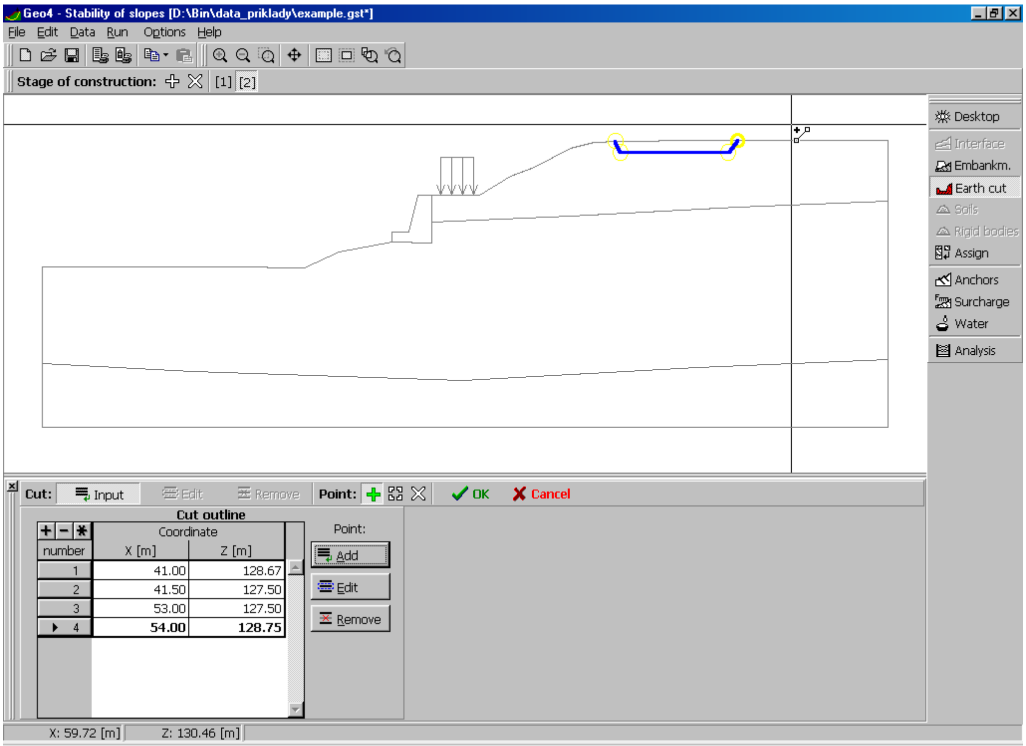

Fig. 3.26 – Inserting an earth cut

Once the cut is created, you may now simulate the loading caused by the construction work through an additional surcharge (see Fig. **3.27**, Fig. **3.28**).

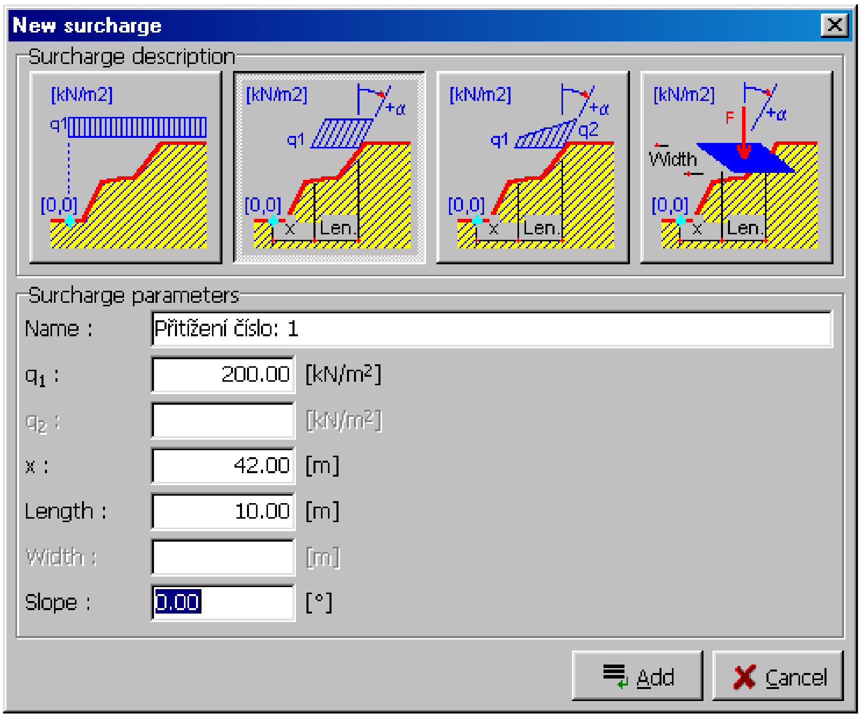

Fig. 3.27 – Inserting a surcharge due to structure

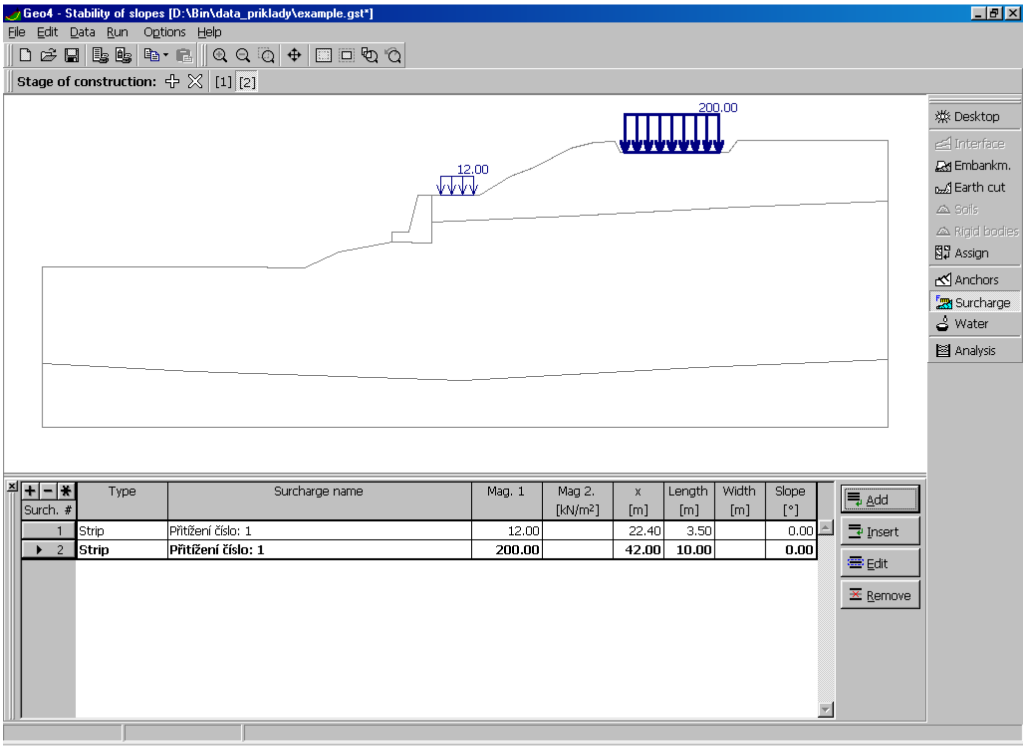

Fig. 3.28 – The second stage of construction after modification

3.3.2 Analyzing the second stage

By clicking the **"Analysis"** button on the vertical toolbar you activate the **"Analysis"** dialog window. As shown in **Fig. 3.29**, the already computed slip surfaces are passed onto the current construction stage.

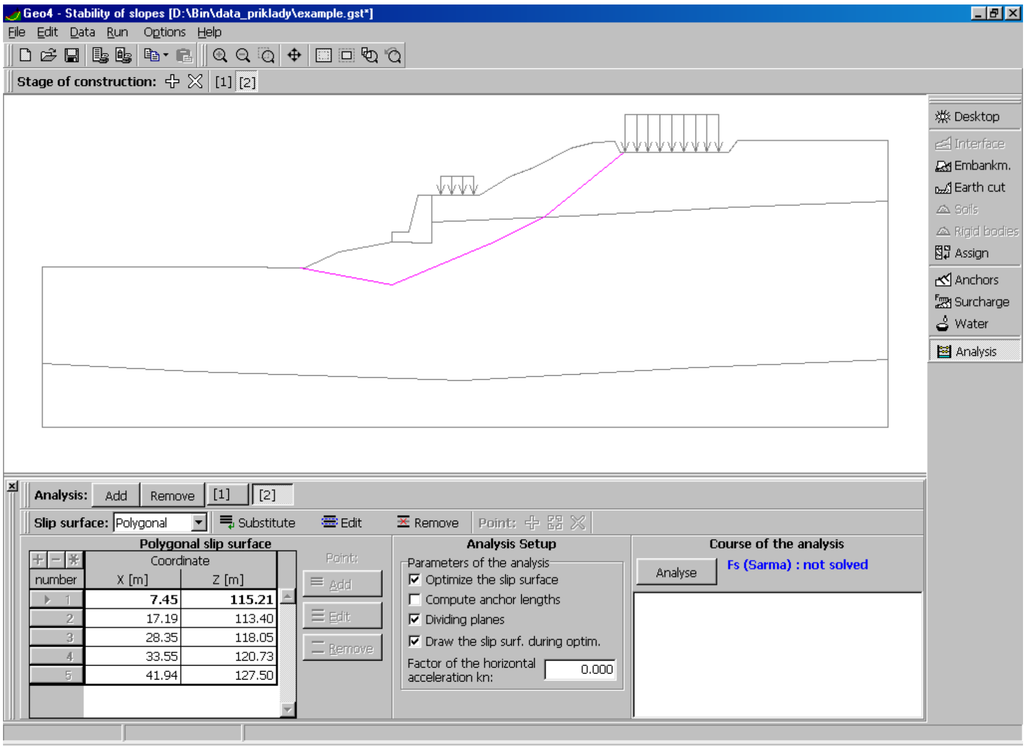

Fig. 3.29 – Analyzing the second stage

An optimization of these surfaces for a new setup can be performed exactly as discussed in **section 3.2.5**. Results suggest (**Fig. 3.30, 3.31**) that for the present configuration the slope stability is no longer acceptable (Bishop – 1.53, Sarma – 1.38, Petterson – 1.47). Therefore, certain measures must be taken to enhance its stability.

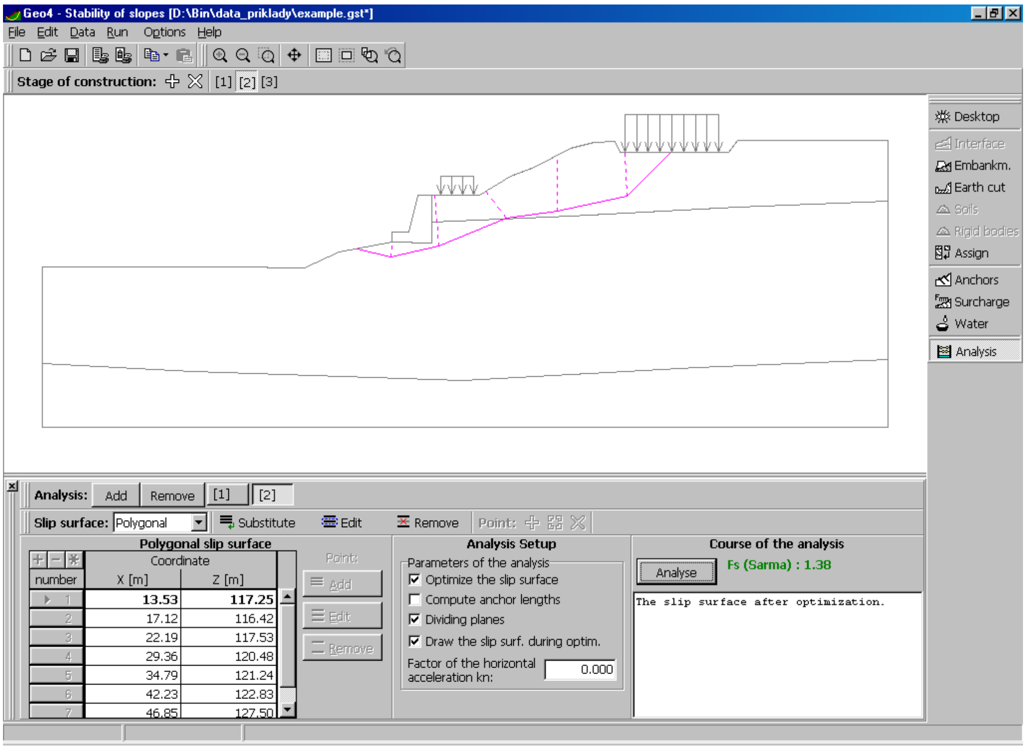

Fig. 3.30 – Analysis of the second stage according to Sarma

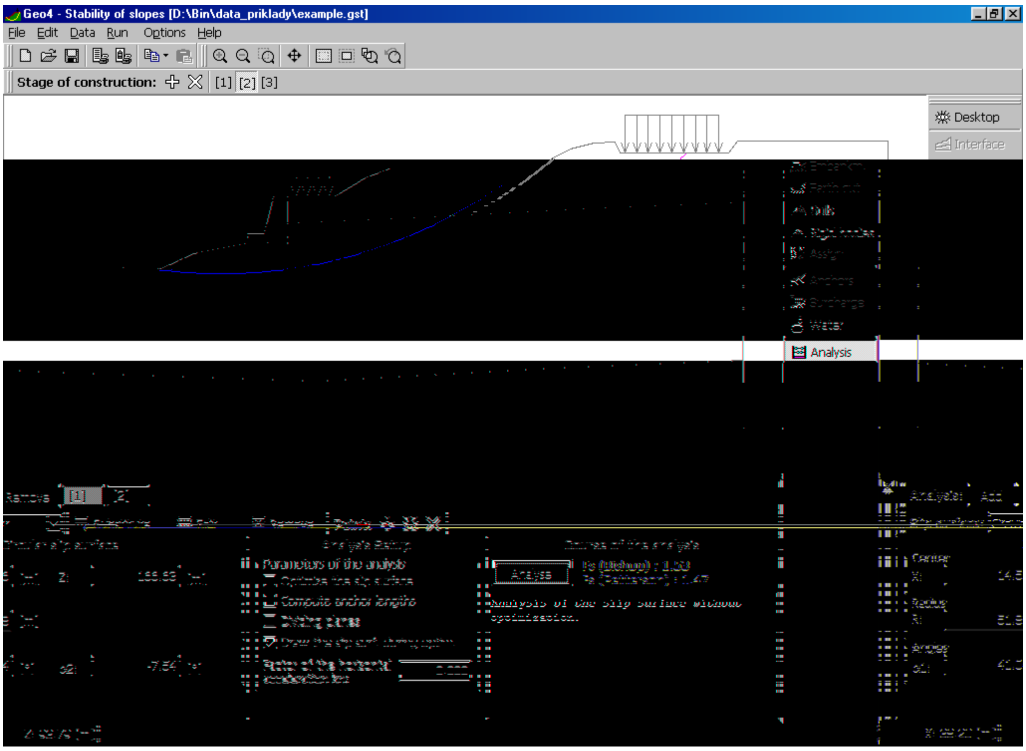

Fig. 3.31 - Analysis of the second stage according to Bishop

3.4 The third stage of construction

The third stage of construction is added in order to enhance the slope stability using anchors.

3.4.1 Inserting anchors

Switch to the **"Anchors"** dialog window and click the **"Add"** button. The pointer changes into the crosshair. Click the left mouse button to select the anchor starting point and **"drag"** the anchor up to the desired length.

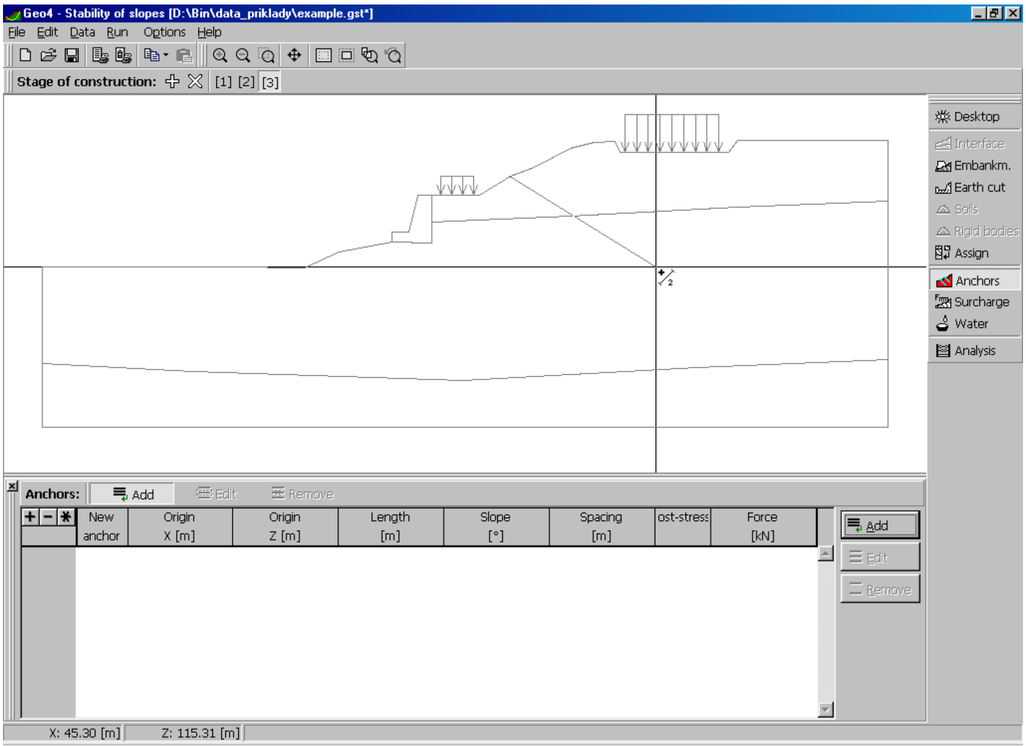

Fig. 3.32 – Inserting an anchor

After putting the end point of an anchor in, the **"New Anchor"** dialog box appears (see **Fig. 3.33**). This dialog window allows you to adjust various anchor parameters. It also serves to insert the magnitude of the anchor force and spacing of anchors. Resulting arrangement appears in Fig. **3.34**.

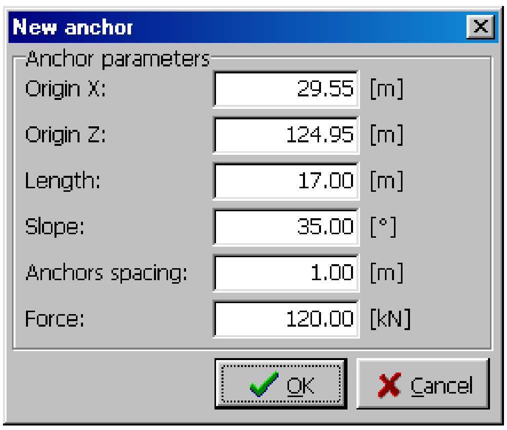

Fig. 3.33 – Parameters of an anchor

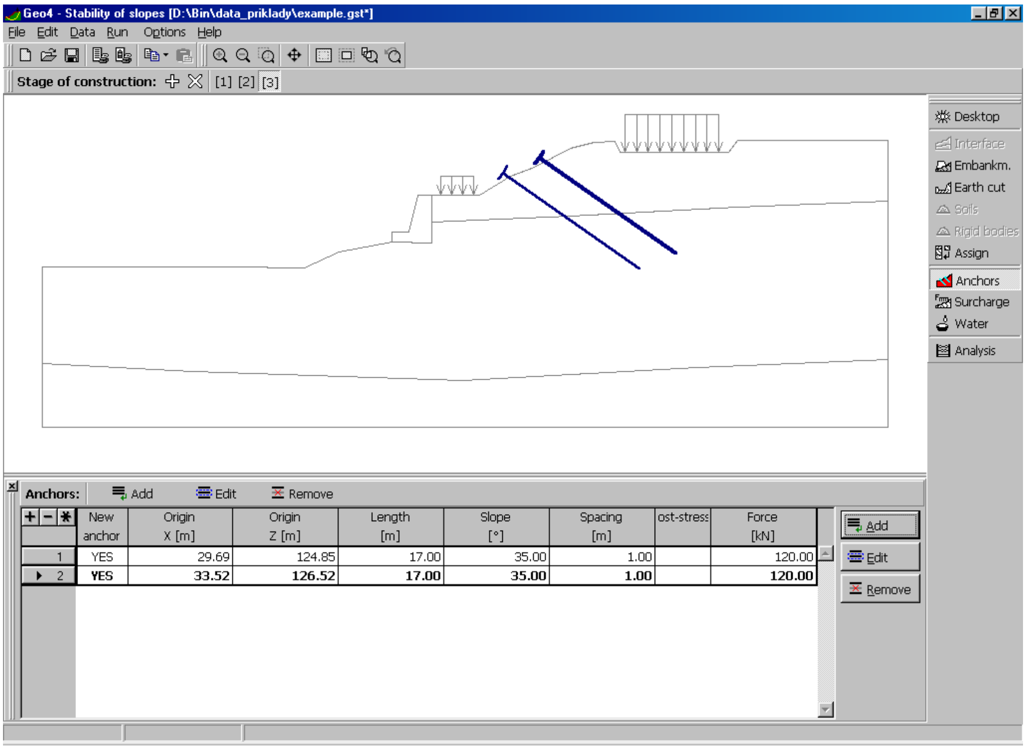

Fig. 3.34 – Arrangement of anchors

3.4.2 Resulting verification analysis

As before, run the optimization process. The resulting safety factors after optimization are Sarma -1.60 (**Fig. 3.35**), Bishop – 1.59 and Petteron – 1.50 (**Fig. 3.36**), respectively. The slope is stable.

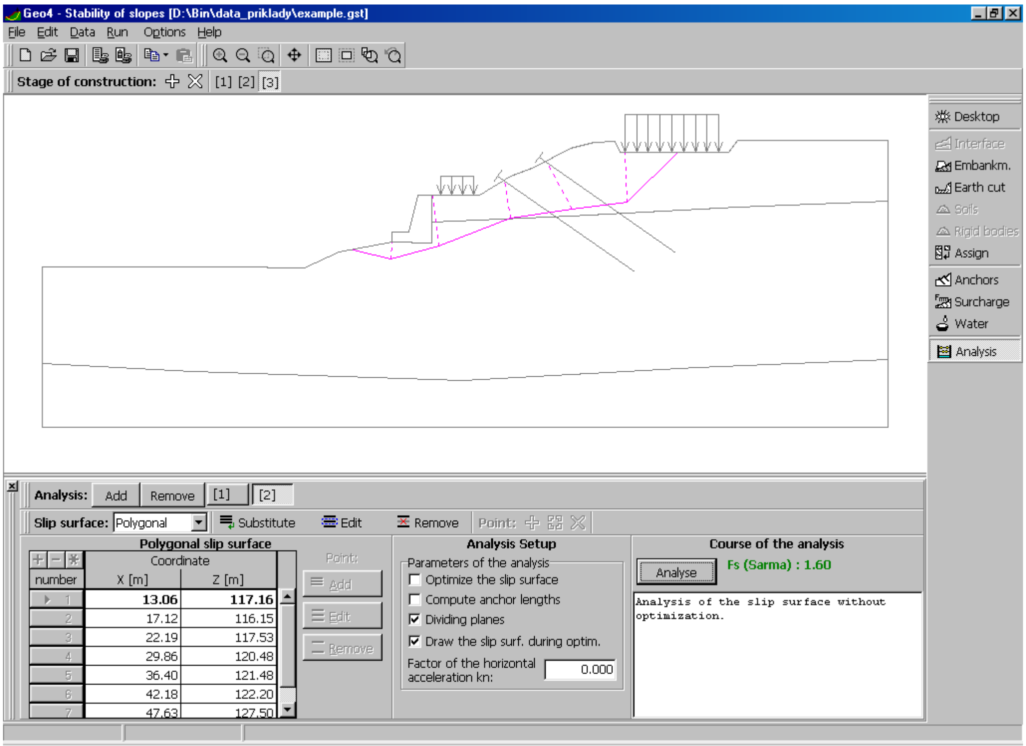

Fig. 3.35 – An optimized polygonal slip surface after enhancing the slope stability

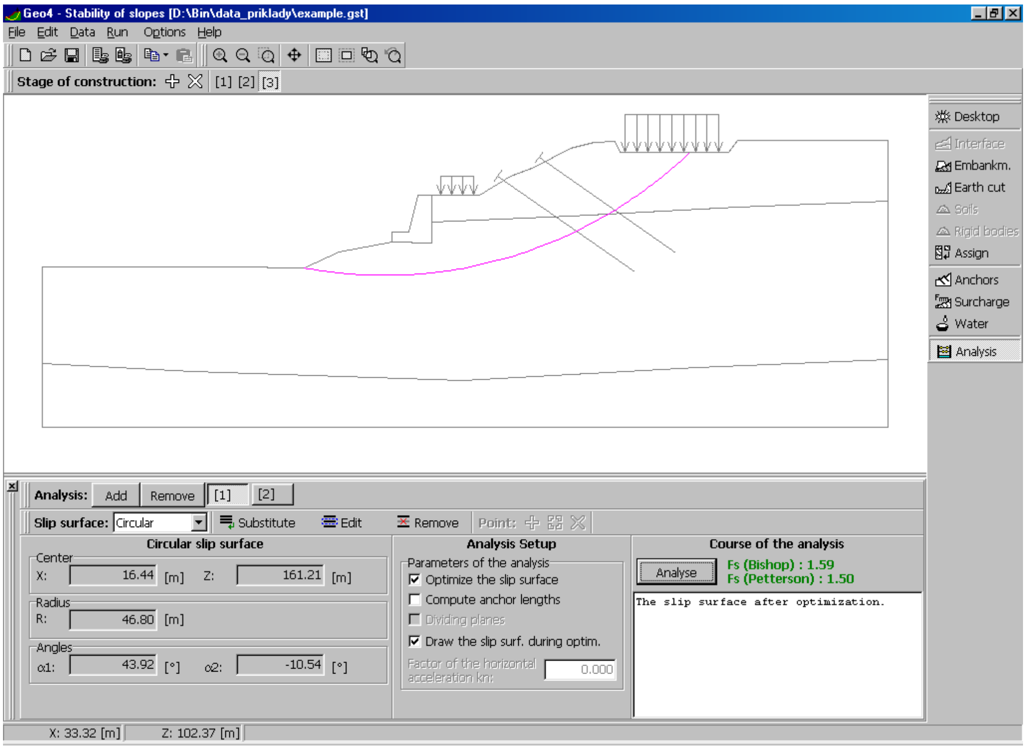

Fig. 3.36 – An optimized circular slip surface after enhancing the slope stability

The program makes also possible to compute a minimum required length of an anchor (see the **"Compute anchor lengths"** check box in the **"Analysis setup"** dialog box). In such a case the program assumes an infinite length of the anchor (the anchor force always enters the stability analysis). The required anchor length is then given as a distance from the anchor starting point to an intersection of the anchor with the most unfavorable slip surface. However, the final verification analysis must be carried out with the real lengths of anchors. In addition, it is necessary to check the stability of slip surfaces behind the anchor roots.