Numerical solution to a sheeting wall structure
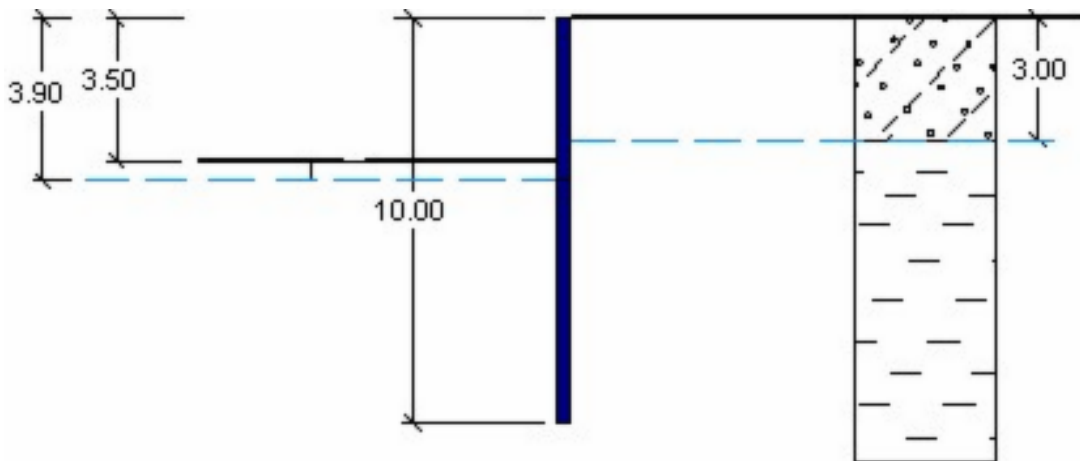
Program: FEM
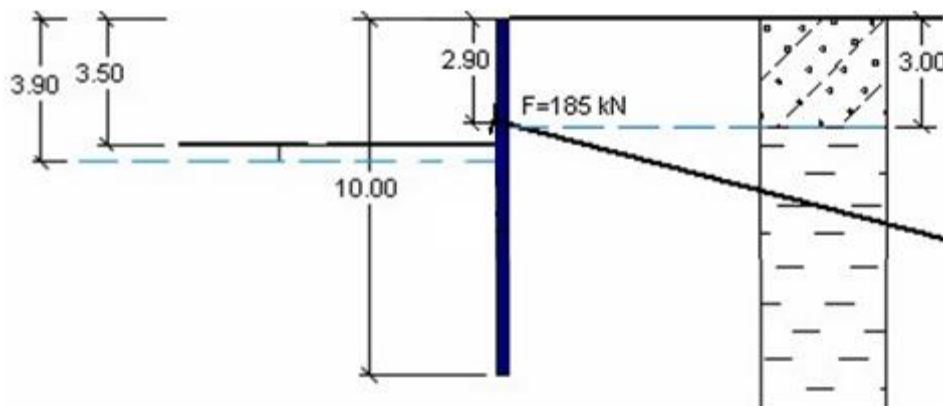
File: Demo_manual_24.gmk

The objective of this manual is to analyse the deformations of an anchored sheet pile wall and determine the diagrams of internal forces using the Finite Element Method.
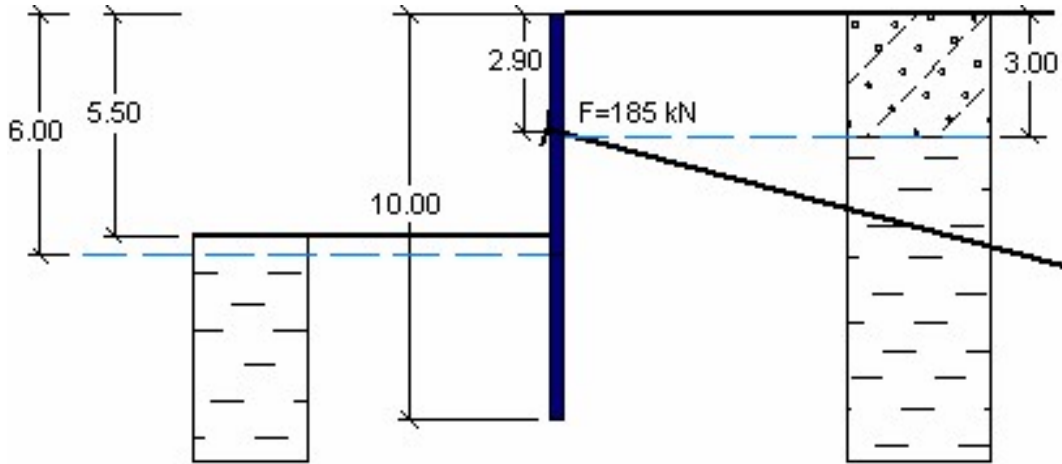
Task specification

Determine the state of stress (deformations) of an anchored sheet pile wall consisting of $500 \times 340 \times 9,7$ mm VL 503 interlocking piles; the structure scheme for individual construction stages is shown in the diagrams below. Determine the internal forces acting along the anchored wall length. Sheet piles are made from EN 10 025: Fe 360 steel. The wall structure is 10 m long (high).

Construction stage 2 – extracting soil up to the depth of 3.5 m

Construction stage 3 – adding the anchor

Construction stage 4 – extracting soil up to the depth of 5.5 m

The geological profile consists of two soil types with the following parameters:

- *0.0 to 3.0 m*:   Silty sand (SM – medium dense soil),

- *down from 3 m:*   Low plasticity clay (CL, CI – stiff consistency).

Soil parameters / Classification	Silty sand (SM)	Low plasticity clay (CL, CI)
Unit weight of soil: γ $[kN/m^3]$	18	21
Modulus of elasticity: E $[MPa]$	10	4,5
Unloading modulus of elasticity: E_{ur} [MPa]	20	13
Poisson´s ratio: ν $[-]$	0,3	0,4
Cohesion of soil: c_{eff} $[kPa]$	5	12
Angle of internal friction: ϕ_{eff} $[°]$	29	19
Dilation angle: ψ $[°]$	0	0
Saturated unit weight: γ_{sat} $[kN/m^3]$	20	23

Table with the soil parameters – anchored sheet pile wall

Solution

We will use the GEO 5 – FEM program to analyse this problem. We will describe the solution in this example step by step in the text below:

- − Topology: setting and modelling the problem (interface, contacts, increasing density of lines)

- − Construction stage 1: primary geostatic stress, specification of point monitors

- − Construction stage 2: activation of regions, specification of beams, analysis of deformations, internal forces

- − Construction stage 3: specification of anchors

- − Construction stage 4: excavation of soil, analysis results + monitors.

- − Assessment of results: comparison, conclusion.
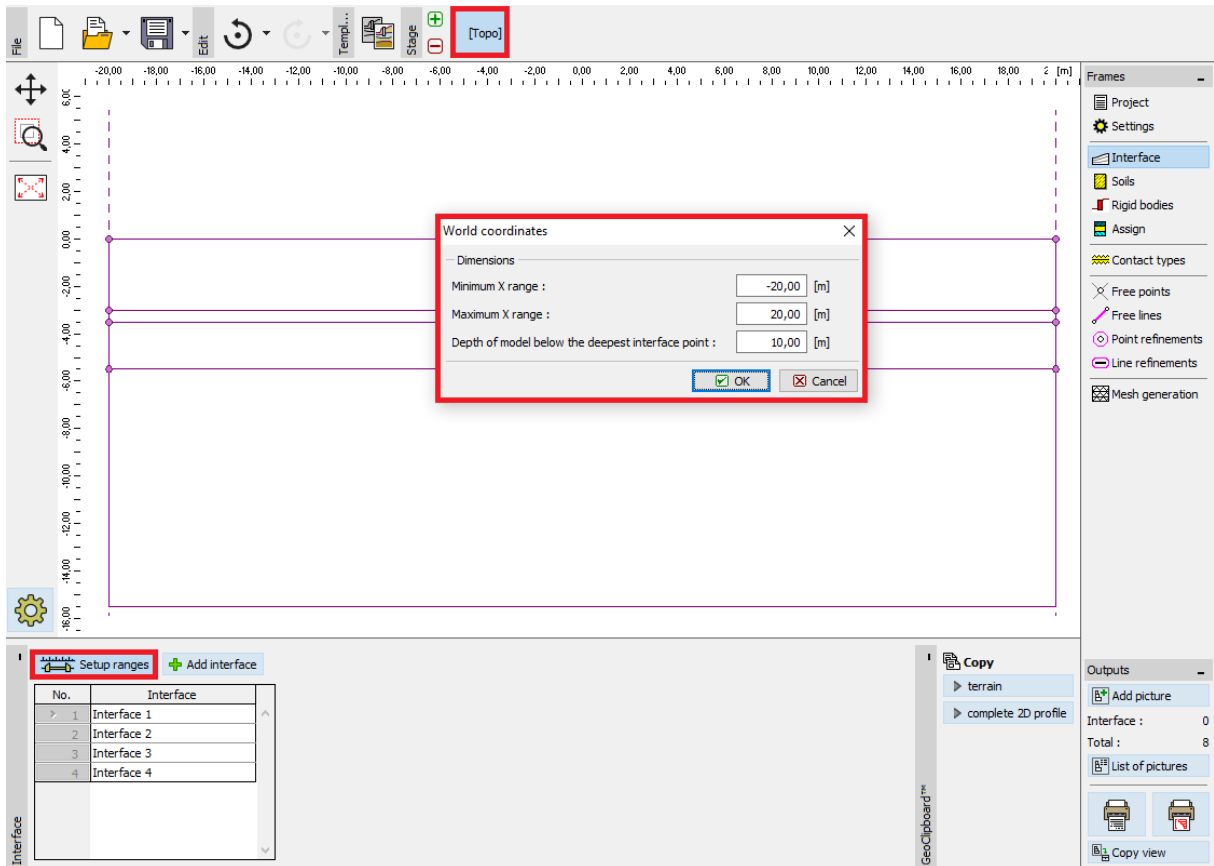
Topology: problem settings

We will leave the analysis method for construction stage 1 at "geostatic stress". We will consider the analysis type as *plane strain*.
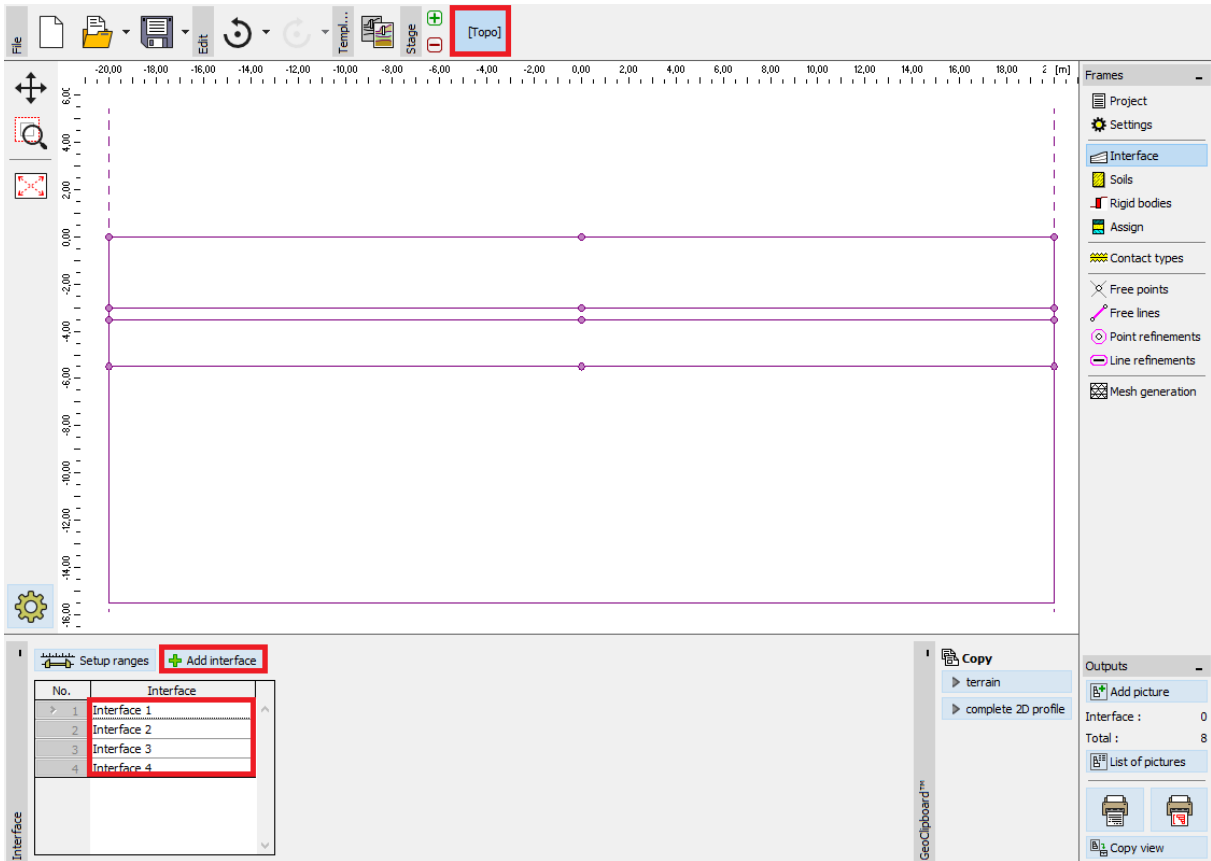

"Settings" frame

Further we will set the world coordinates; we will choose sufficient dimensions of the world so that the results are not affected by conditions at the edge. For our problem we will choose model dimensions $\langle-20\ m;\ 20\ m\rangle$, we will set the depth from the lowest interface point to 10 m.

"World coordinates" dialogue window

When diaphragm wall structures are being designed, it is necessary to define the depths up to which the soil will be excavated at individual construction stages as soil interfaces. In this particular case we will therefore set the terrain surface level to 0.0 m and horizontal interfaces at the levels of – 3.0 m, – 3.5 m and – 5.5 m. The point with coordinates [0.0; 0.0] forms the top of the diaphragm wall.
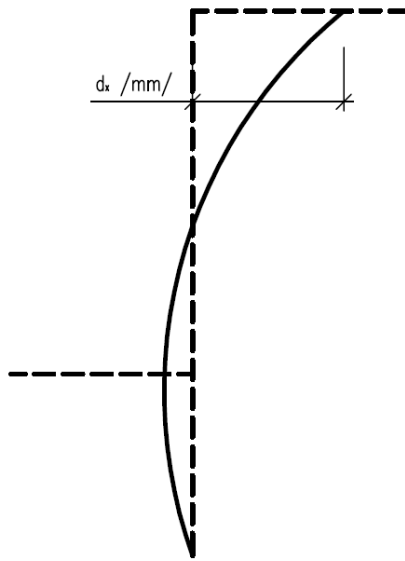

"Interface" frame

Now we will specify the respective parameters of the soil and subsequently we will assign the soil to the created region. We will choose the Modified Mohr-Coulomb model (see the note).

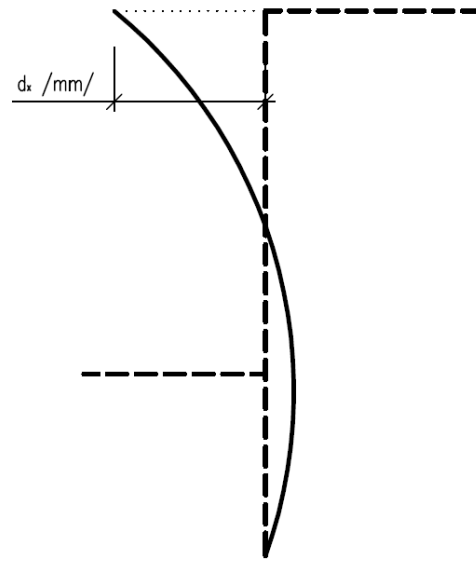*"Add new soils" dialogue window*

"Add new soils" dialogue window

Note: When sheeting structures are being designed, it is necessary to introduce contact elements between the soil and the beam. Solving problems without contact elements leads to totally unrealistic results (for more details visit Help – F1).

Solution without contacts
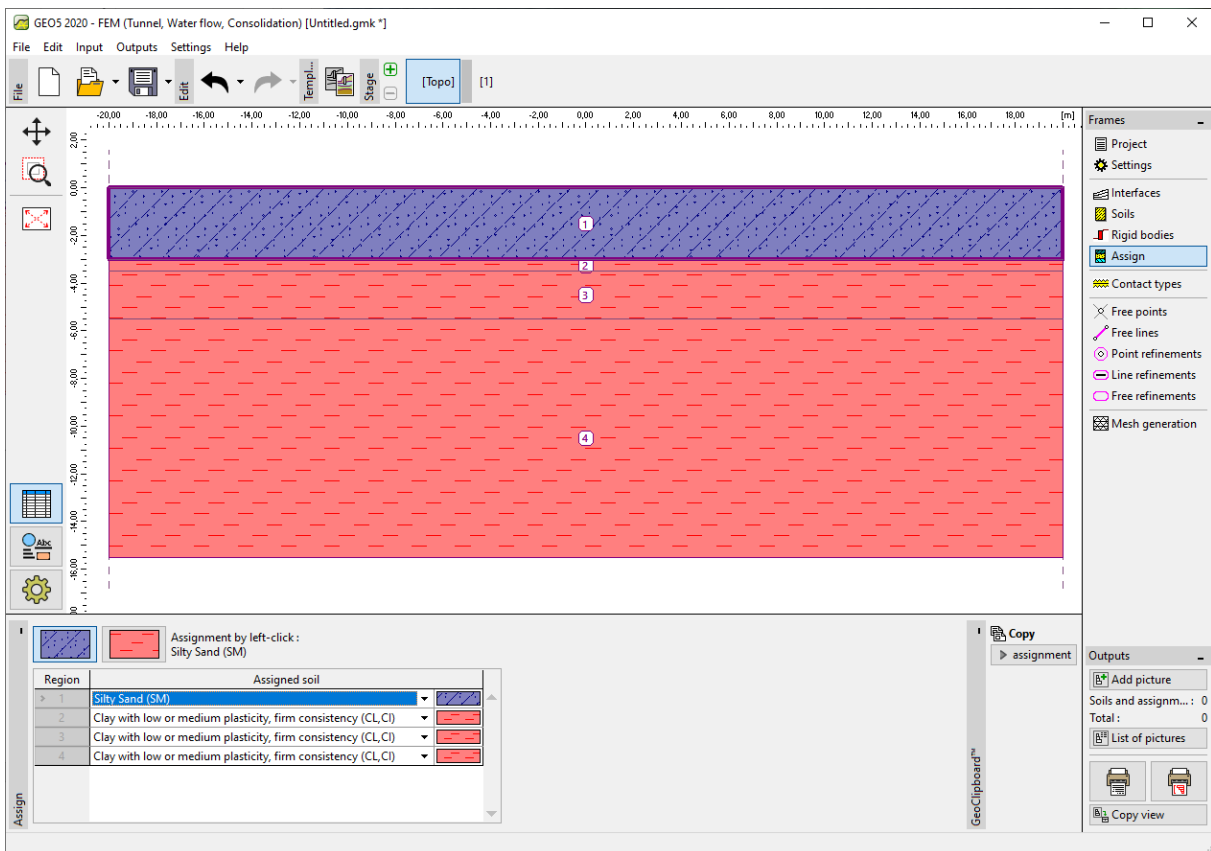
Solution with contacts

deformation of sheeting to a soil
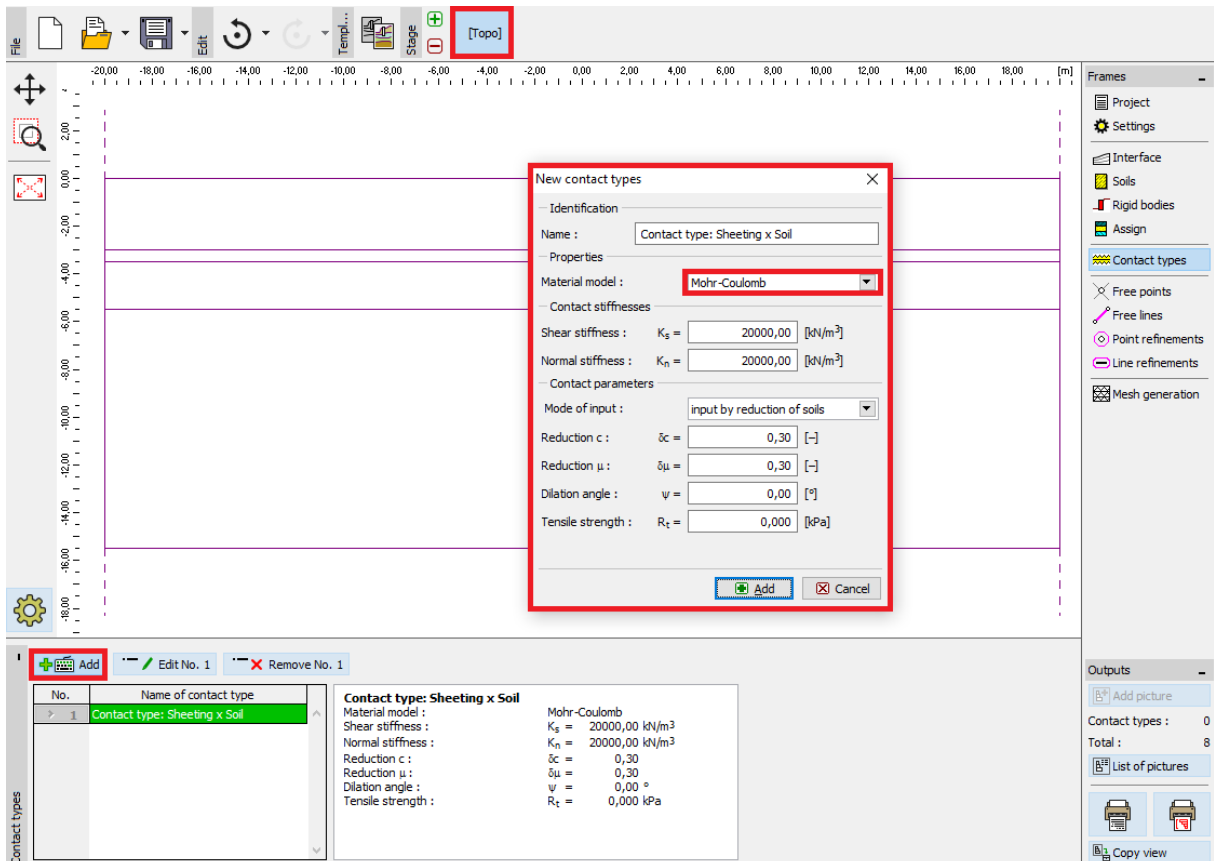(wrong)

deformation of sheeting from a soil
(right)

The application of a suitable material model to a numerical analysis of sheeting structures

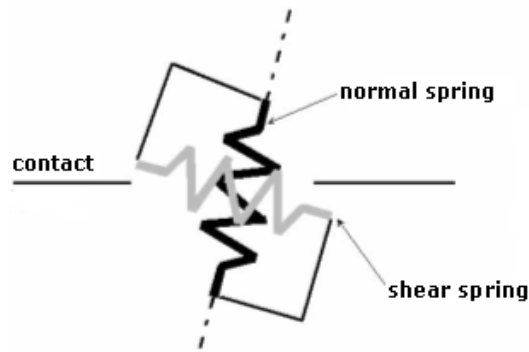The following figure shows the assignment of a soil to the geological profile.

"Assign" frame

The next step lies in setting the contact parameters (using the "Add" button). When sheeting structures are being analysed, it is always necessary to define the contact with the non-linear material model for beam elements. In this particular case we will select the "Mohr-Coulomb" option to obtain realistic results. We assume the reduction of soil parameters on the contact to be $\delta c = \delta \mu = 0.3$ and will keep standard values of the contact stiffness $K_s = K_n = 20\ 000\ kN/m^3$.

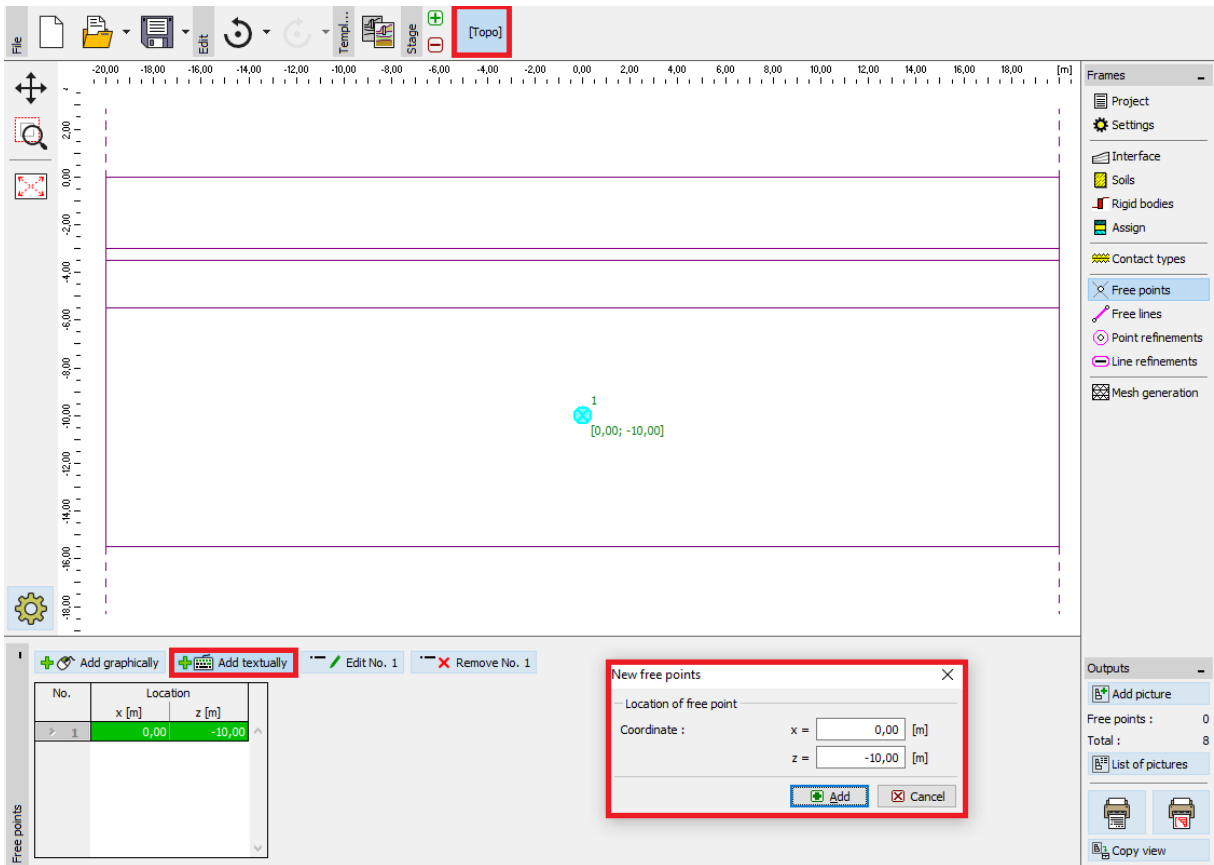*"New contact types" dialogue window*

Note: Contact elements are used in analyses when it is necessary to allow for the interaction between the structure and the surrounding environment – an interface between two totally different materials (soil – sheeting). A typical example of the use of contact elements is the modelling of sheeting structures, retaining walls or tunnel linings, where we use the contact element for simulating a thin area of soil or rock in which intense stressing, most of all shear stress, takes place. The contacts can be introduced even between individual soil interfaces. The contact element is an element with zero thickness, expressing the relationship between contact stresses and relative changes in displacements along the contact (for more details visit Help – F1).
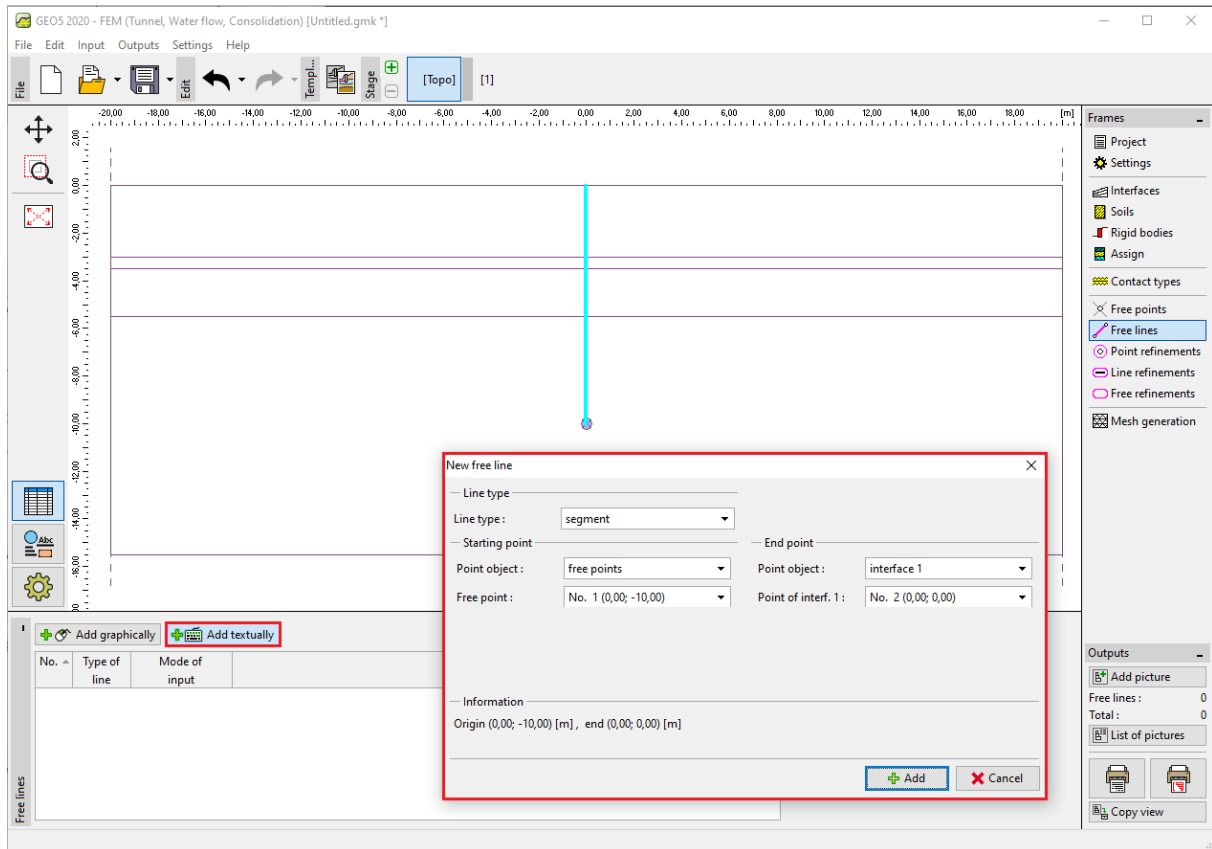
Schematic representation of a contact element

Note: Despite the fact that the selection of parameter K_s is not important in the case of a completely plastic behaviour of the contact, the magnitude of this quantity is crucial for a successful solution to the non-linear problem in question. Too high stiffness values (over $100\ 000\ \ kN/m^3$) may lead to an oscillation of the numerical solution. On the contrary, very low parameter values K_s and K_n (under $10\ 000\ \ kN/m^3$) may lead to unrealistic deformations of structures. However, the values of contact stresses K_s and K_n themselves are not significantly affected by the selection of stiffness K_s and K_n (for more details visit Help – F1).

Subsequently we will set the sheeting structure geometry in "Free points" and "Free lines" frames. The principle of setting free points and free lines was described in more detail in the previous chapter *23. Collector lining analysis*.
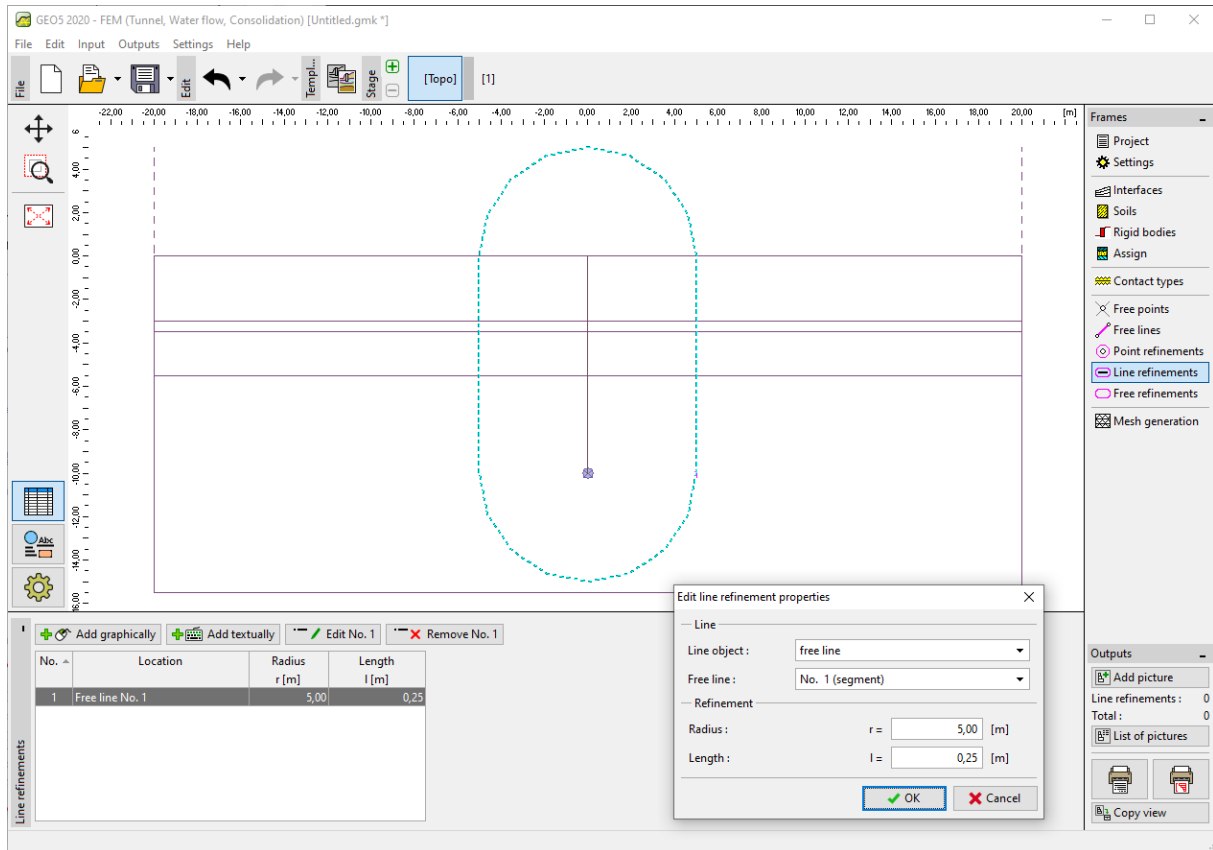

"New free points" dialogue window

First we will set a new free point with coordinates [0.0; – 10.0]. The free line forming the sheeting wall will originate by connecting this point with the terrain interface point (for more details visit Help – F1).
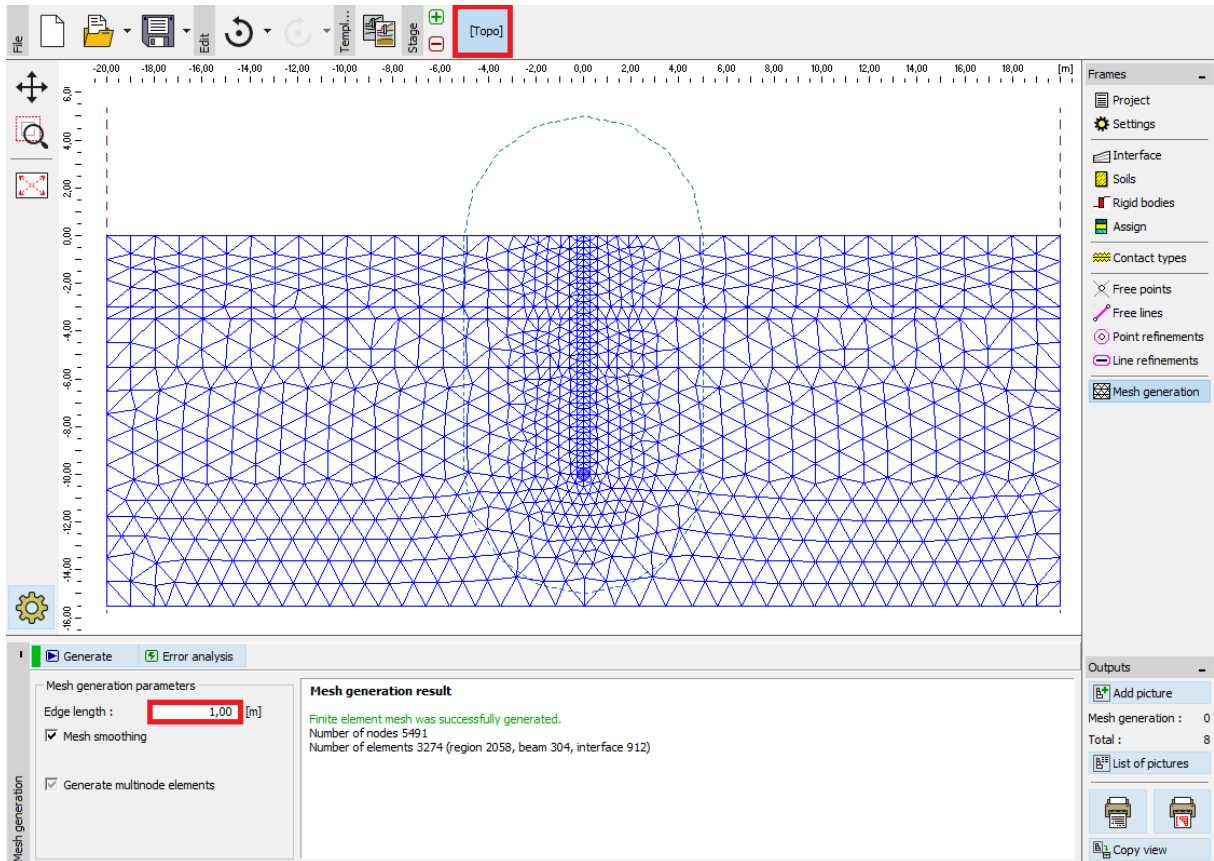

"New free lines" dialogue window

The last step in setting the topology is the generation of the finite element mesh. It is reasonable to refine FE mesh in the sheeting wall surroundings. In the "New line refinements" we will select the respective extent radius $r = 5.0\ m$ and the element edge length $l = 0.25\ m$.

"New line refinements" dialogue window

Then we will go over to the "Mesh generation" frame and generate a mesh with the element edge length of 1.0m (using "**Generate**" button). The program will automatically smoothen the refined FE mesh.
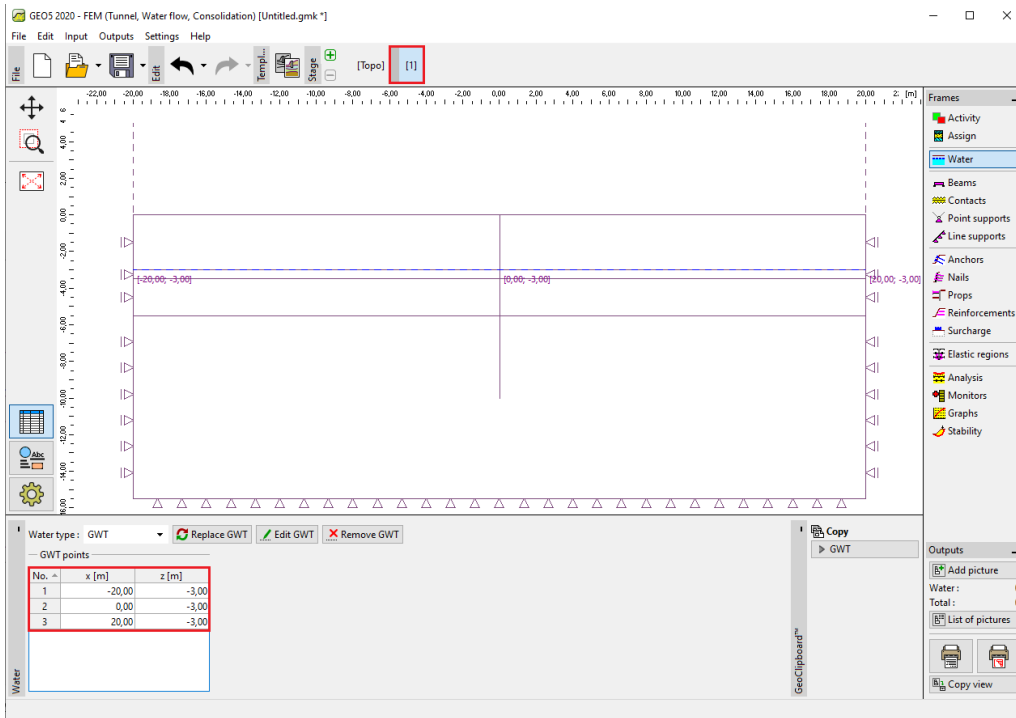
Note: Subsequently we will visually check whether the finite element mesh density is adequate to the extent and complexity of the given problem (for more details visit Help – F1). The increased mesh density contributes to the stabilisation of the non-linear analysis similarly to the effect of the decrease in shear stiffness.

"Mesh generation" frame – element edge length of 1 m (with locally increased mesh density)
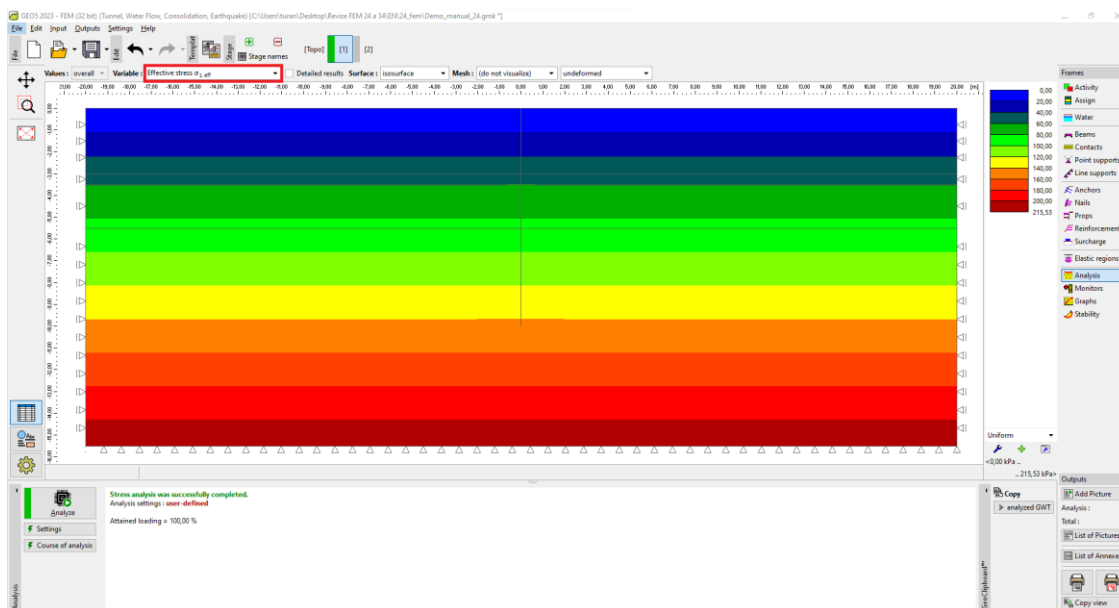
Construction stage 1: primary geostatic stress

After generating the mesh, we will go over to construction stage 1 and set the ground water table level (hereinafter referred to as the GWT) at the depth of 3.0 m under the terrain surface (see the *diagram*).
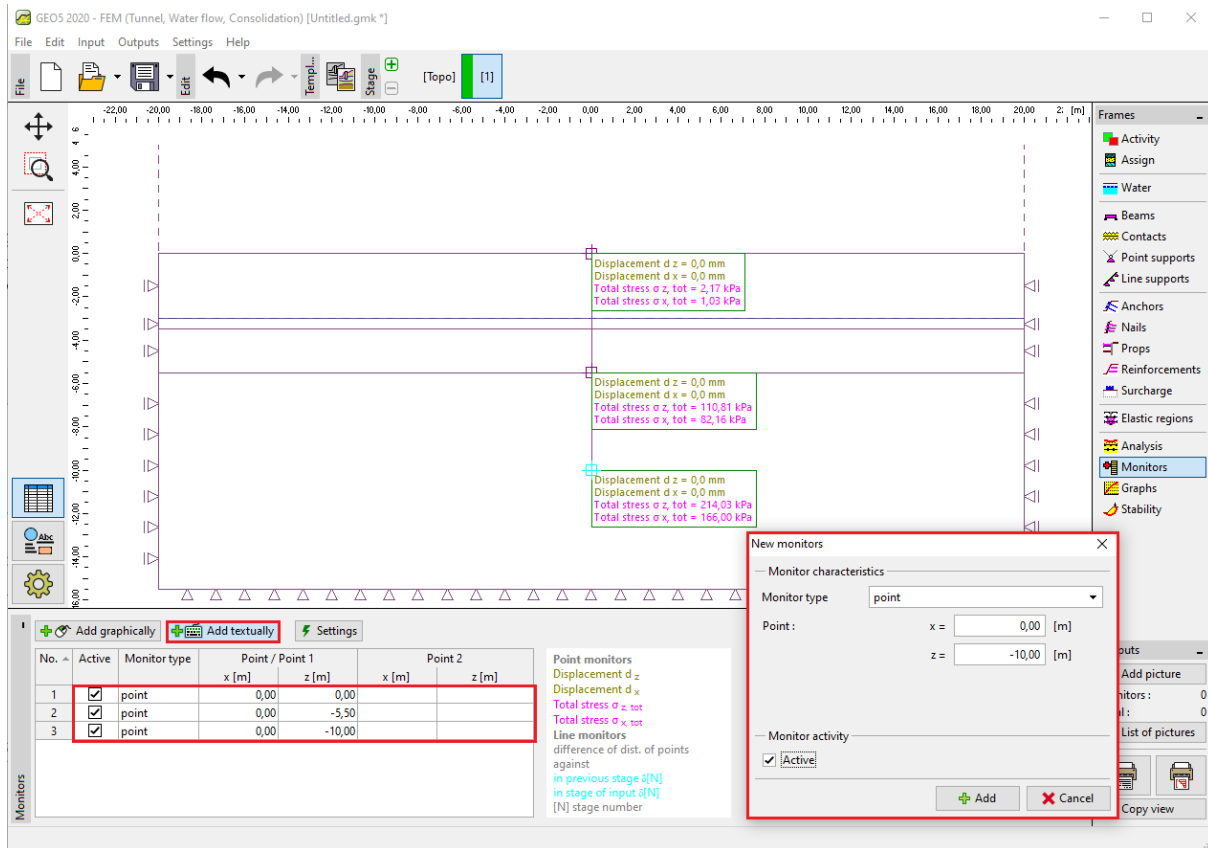

"Water" frame – Construction stage 1 (GWT at the depth of 3.0 m)

We will carry out the analysis of primary geostatic stress. We will leave the "Standard" analysis setting in place (for more details visit Help – F1).

"Analysis" frame – Construction stage 1 (Vertical geostatic stress $\sigma_{z,ef}$)

For the purpose of observing the values of certain quantities (during the course of the analysis of individual construction stages) we will define so-called *point monitors* in the program (using the "Add" button). We will select the monitored locations at the points which will represent the head and toe of the sheeting wall being modelled, i.e. [0.0; 0.0] and [0.0; − 10.0] and further in the region of the excavation of soil at the construction pit bottom [0.0; − 5.5].

"Monitors" frame – Construction stage 1 (point monitors)

Note: We will edit the individual values of the quantities we want to visualise in the results using the "Settings" button (at the bottom right of the screen). When analysing sheeting structure, we are most of all interested in the change in geostatic stress and the magnitude of the vertical or lateral displacements.

Construction stage 2: modelling of beam elements

In this construction stage we will first go over to the "Beams" frame and model the sheet pile wall. We will define the following relationships: the location, material and steel class, cross-section type (VL 503), the support of beam ends and contacts (for more details visit Help – F1).

"New beams" dialogue window – Construction stage 2

Then we will model the excavation of soil in the "Activation" frame – we will set the given regions in the program with the mouse cursor as inactive (for more details visit F1).

"Activity" frame – Construction stage 2

Note: It is obvious from the preceding figure that the automatic structure corrector built in the program divided the interfaces of soils crossed by the wall into individual circumscribed regions (for more details visit Help – F1).

Further, in the "Water" frame we introduce the change in the GWT according to the following figure. The other parameters will remain unchanged.

"Water" frame – Construction stage 2 (changes in the course of the GWT)

Now we will carry out the analysis of construction stage 2 and will examine the results for the diagrams of internal forces along the beam, equivalent plastic strain and the deformed structure.

"Analysis" frame – Construction stage 2 (settlement d_x – deformed structure)

"Analysis" frame – Construction stage 2 (equivalent deviatoric strain $E_{d,pl.}$

and displacement vectors)

"Analysis" frame – Construction stage 2 (distribution of bending moment M)

Construction stage 3: settings for anchors

We will add construction stage 3 and then we will push the "Add" button in "Anchors" frame and, in the "New anchors" dialogu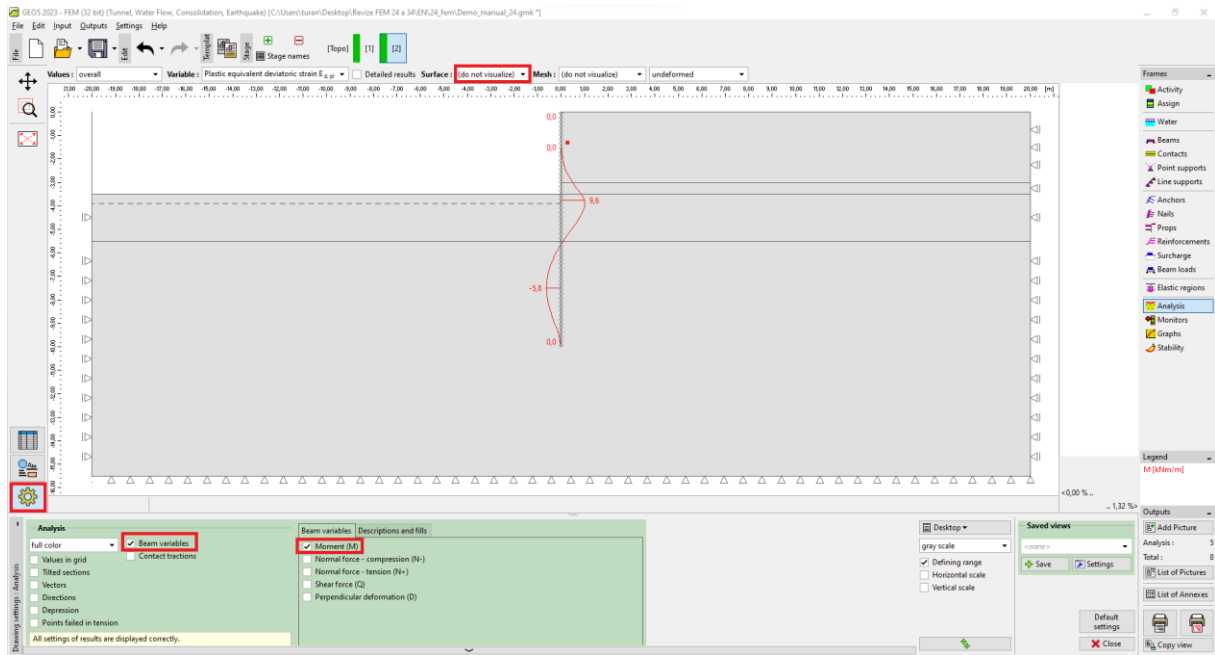e window set a steel anchor with pre-stressing force $F = 185\ kN$. We will consider the anchor to be at a depth of 2.9 m under the terrain surface – we will set the anchor head coordinates by the point [0.0; − 2.9].
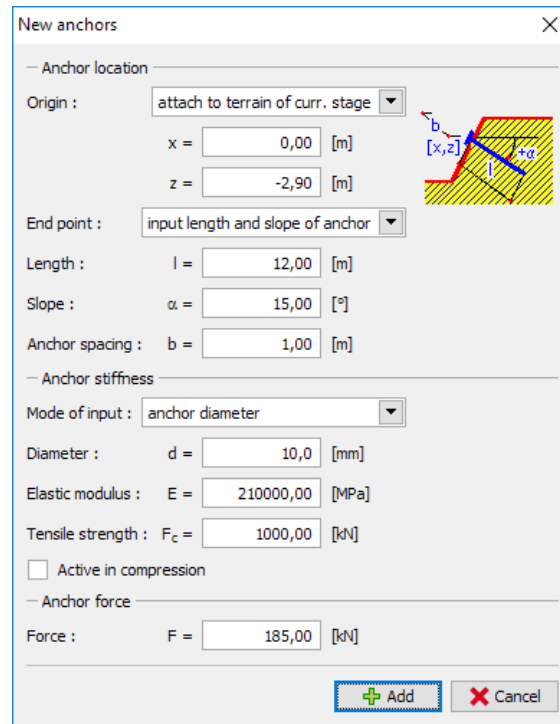
Note: Anchors are modelled in the program by means of an elastic rod element with constant normal stiffness. A failure of the anchoring element is controlled by specifying the maximum force. The anchor is fixed to the soil at two points – at the beginning and at the end. No interaction between the soil and the reinforcing element is assumed along the anchor length (for more details visit Help – F1).

We will assume the following anchor parameters for this particular problem:

− Anchor length: $l = 12\ m$,

− Anchor slope: $\alpha = 15\ °$,

− Anchor diameter: $d = 10\ mm$,

− Anchor spacing: $b = 1\ m$.

Note: Anchor stiffness is defined in the analysis by the modulus of elasticity, anchor cross-sectional area and spacing of anchors. It is necessary to realise that in the case of plane-strain deformation, discrete anchors are replaced by a 1 m-wide membrane. Another important input data of anchors is

the pre-stressing force and the anchor breaking force. In this particular case we will not consider the possibility of the reinforcement element breaking, therefore we will set the break force F_c to a sufficiently high value (for more details visit Help – F1).
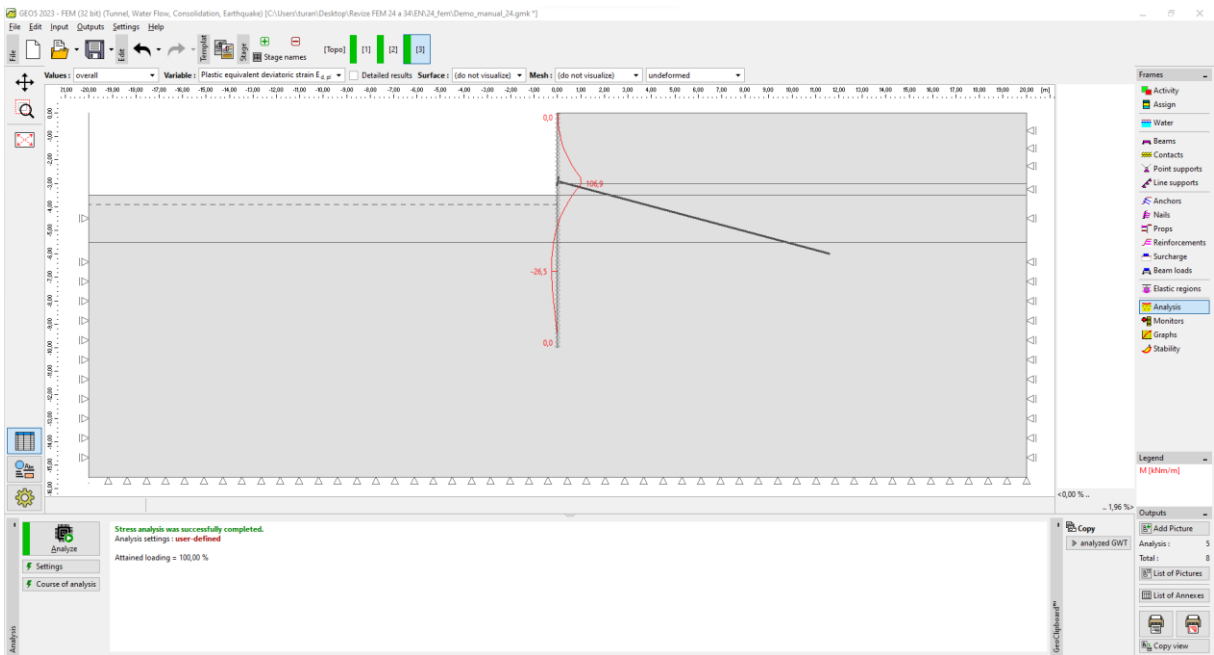

"New anchors" dialogue window – Construction stage 3

Note: The anchor becomes deformed during the course of the analysis. As a result of the deformation of the anchor and the surrounding massif, the pre-stressing force set for the anchor can drop. Therefore, if we want to achieve the concrete pre-stressing force, it is necessary to introduce additional stress into the anchor during the next stage or to set sufficiently higher pre-stressing force (the resultant force in the anchor after the analysis is indicated in the diagram at the anchor head, under the pre-stressing force set). At the subsequent construction stages the anchor parameters cannot be changed; it is only possible to add stress to achieve the new pre-stressing force or to remove the entire anchor from the structure.

The embedment of the anchor in ground mass should be sufficiently tough (fixing to an element) so that an unrealistic pulling of the anchor does not occur when significant plastic strains develop in the vicinity of the anchor root (fixing to a node, too great increase in density in the root surroundings), causing unrealistic loss of the pre-stressing force.
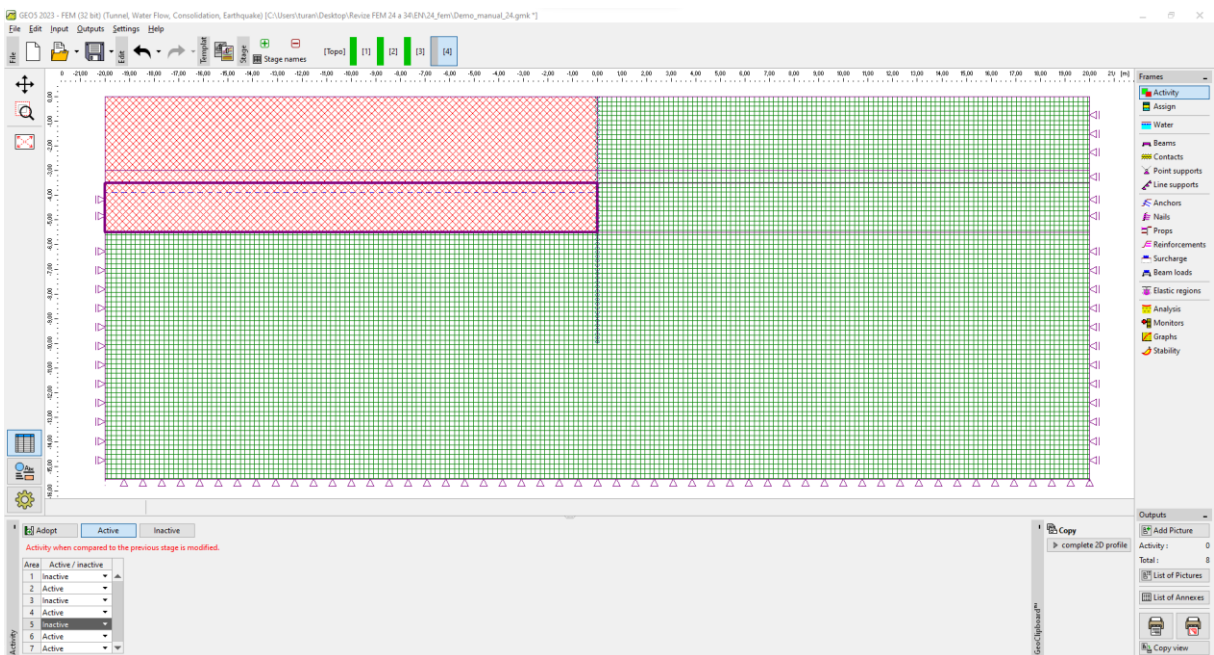
Now we will carry out the analysis of construction stage 3 and again examine the results of the numerical solution (similarly to the previous construction stage).
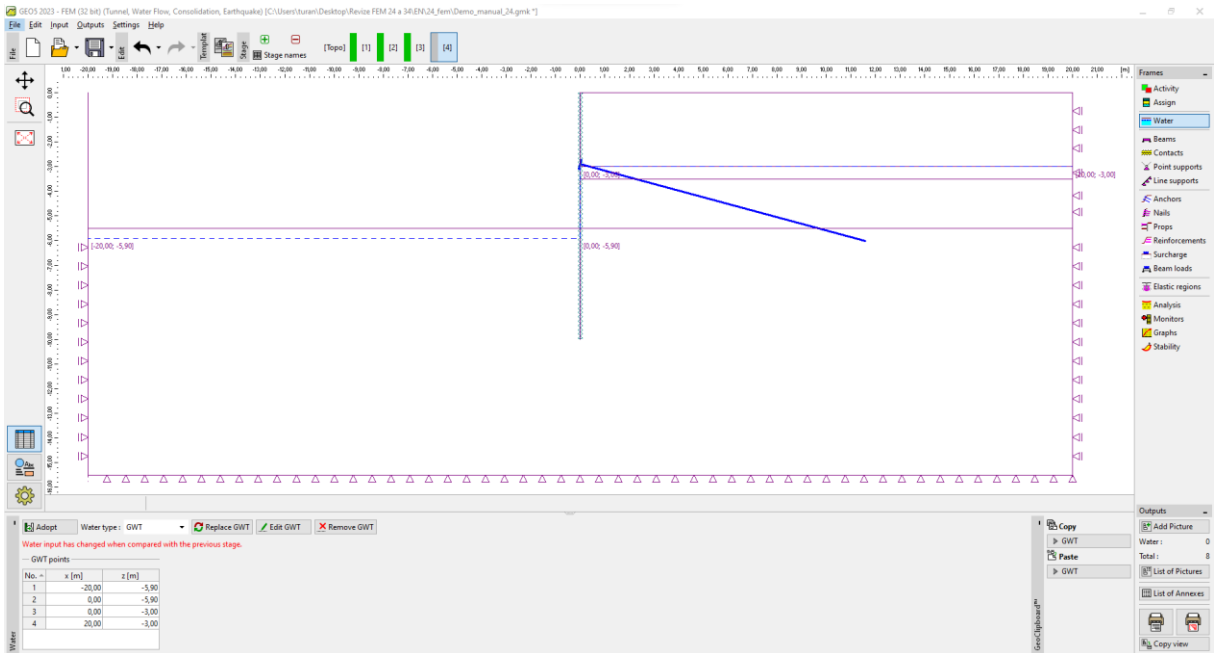

"Analysis" frame – construction stage 3

Construction stage 4: extracting remaining soil

We will add construction stage 4 and extract the remaining soil. Firstly, we will select the given region with the mouse cursor and click the "Inactive" button.
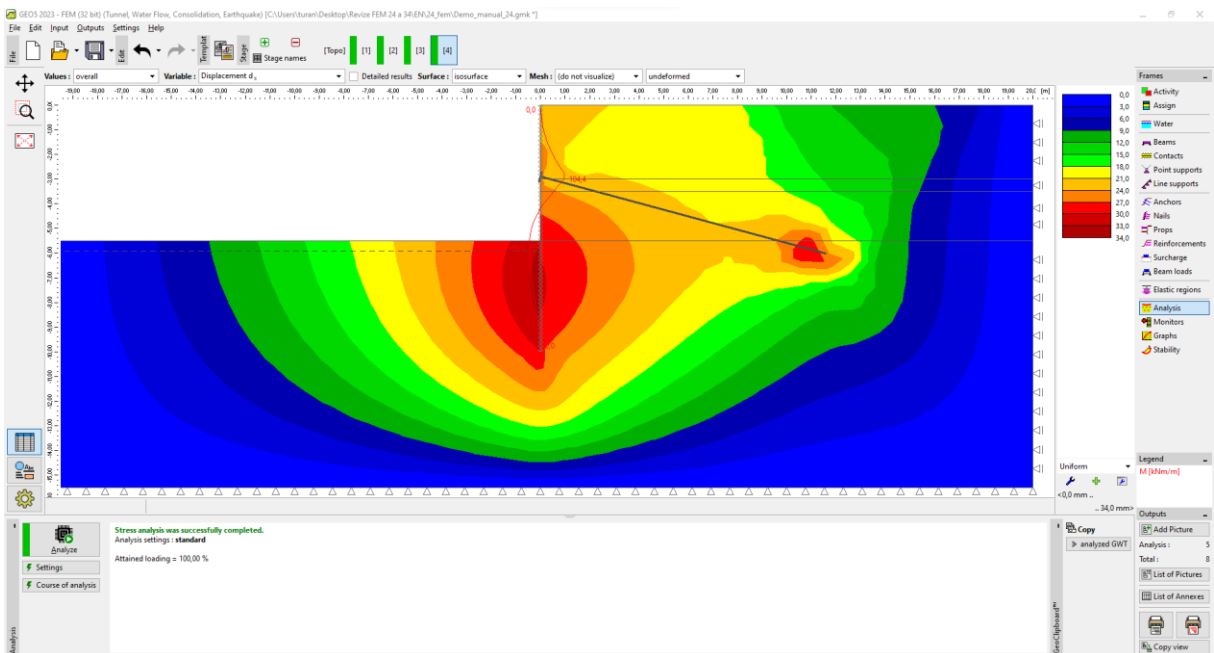

"Activity" frame – Construction stage 4

In the last step of setting construction stage 4 we will change the ground water table level according the diagram below. The other input parameters will remain unchanged.
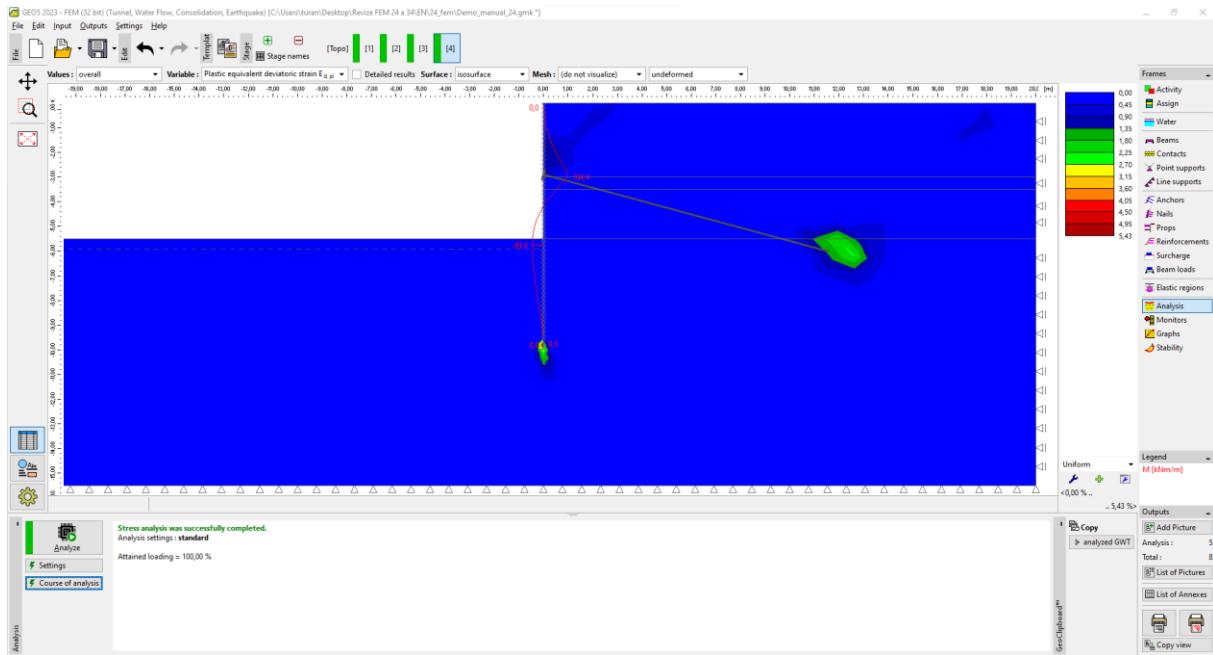

"Water" frame – Construction stage 4 (change in the GWT level)
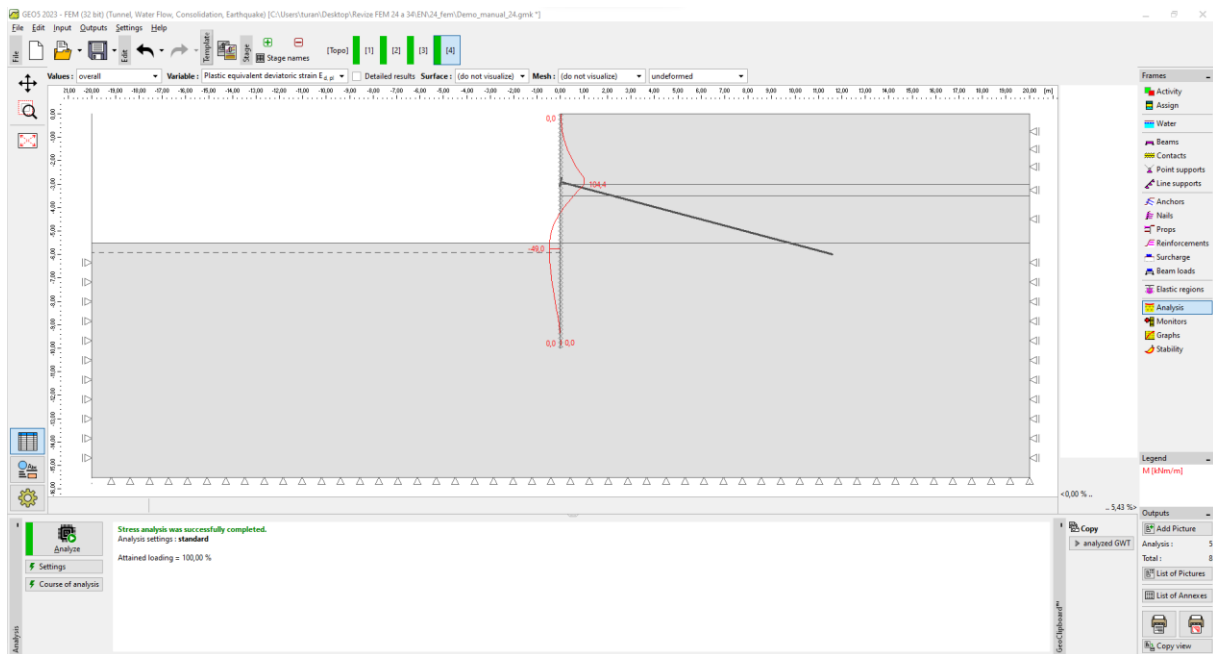

"Analysis" frame – Construction stage 4 (settlement d_x – deformed structure)

It follows from this figure that the maximum lateral displacement in the vicinity of the sheeting wall formed by steel sheet piles is 34 mm.

"Analysis" frame – Construction stage 4 (equivalent plastic strain $\varepsilon_{eq.,pl.}$)

It is obvious from the plotted equivalent plastic strains that the largest plastic strains in the soil are developed in the vicinity of the sheeting wall toe. In the second stage, soil was plasticised in the vicinity of the anchor location (for more details visit Help – F1).

"Analysis" frame – Construction stage 4 (distribution of bending moments M)

We will identify local extremes in the diagram of the curves for bending moments along the sheeting wall length; we will record them in the table, which is presented in the last part of this chapter.

Now we will examine the results for monitors and determine deformations at the sheeting pile wall head.

"Monitors" frame – Construction stage 4 (Point monitors)

Assessment of results:

The following table shows the extremes of internal forces along the sheet pile wall height for construction stages 2 and 3. They are the values of bending moments. We carried out this analysis first for the Modified Mohr-Coulomb material model with locally increased density of mesh using the line-refinement option. Then we compared these results with the GEO 5 – Sheeting assessment program.

Material model / program	Stage 2 $M \ [kNm/m]$	Stage 4 – field $M \ [kNm/m]$	Stage 4 – anchor $M \ [kNm/m]$
MCM (Modified M-C)	9.60	-49.00	104.4
Sheeting Check * (analytical solution)	29.16	− 28.91	110.57

Summary of results – bending moments along the sheeting structure length (height)

Note *: For the analytical solution we considered the analysis of the sub-grade of horizontal reaction module according to Schmitt (for more details visit Help – F1). We defined the supplementary parameters as follows:

– **Soil class SM, medium dense**: pressure at rest analysis – cohesionless soil,

  angle of friction between structure and soil $\delta = 17$ °,

  soil deformation modulus $E_{def} = 10 \ MPa$.

– **Soil class CL, stiff consistency**: pressure at rest – cohesive soil ($\nu = 0.4$),

  angle of friction between structure and soil $\delta = 14$ °,

  soil deformation modulus $E_{def} = 4.5 \ MPa$.

We considered the analysis setting as "Standard – Limit states". The analysis of earth pressures was carried out without reducing the soil parameters. Further we did not take into consideration the value of the minimum dimensioning pressure (for more details visit Help – F1).

Conclusion

The following conclusions can be drawn from the numerical analysis results:

– The locally increased density of the FE mesh in the surroundings of lines leads to a more accurate determination of the internal forces results.

– It is necessary for analyses of sheeting walls to use contact elements and non-linear material models, allowing for the development of plastic strains and giving a truer picture of the real behaviour of structures in the surrounding ground mass.

– Maximum equivalent plastic strains $E_{d.,pl.}$ represent the potential locations of a failure (as a result of exceeding the material yield condition).