

Verification of a multi-anchored wall

Program: Sheeting check

File: Demo_manual_07.gp2

In this chapter, we will show you how to design and verify a multi-anchored wall. This sheeting was realized in the construction of the Prosek station, of the metro C line in Prague.

You can find more information about this project in the following documents:

- [Leaflet](#)
- [Research paper comparing the calculated results with the monitored results](#)

Introduction

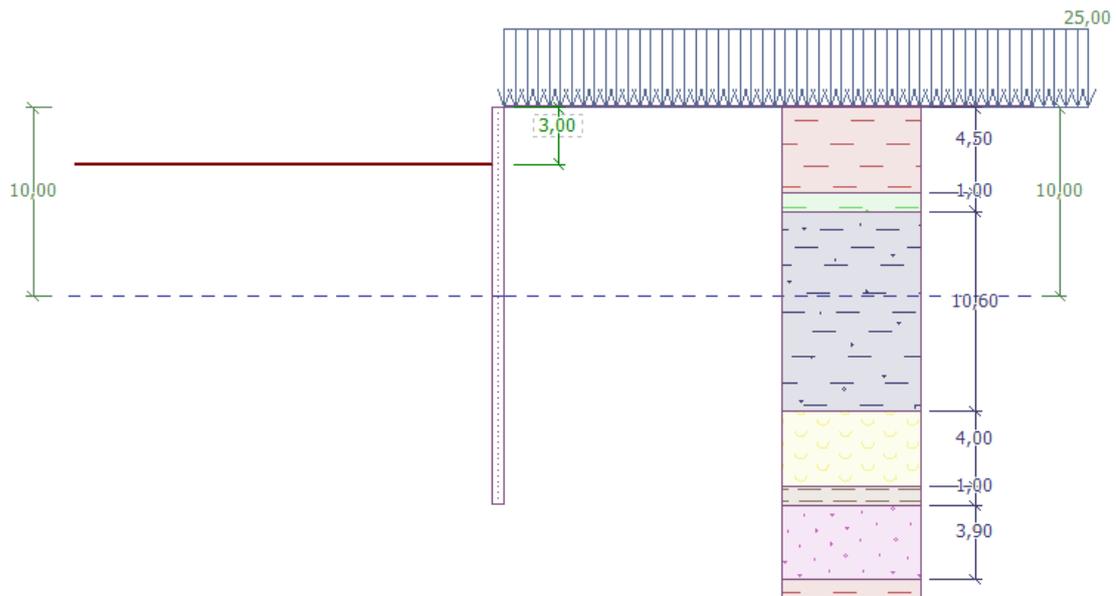
The basic assumption with method of dependent pressures is that the soil or rock in the vicinity of the wall do behave as an ideally elastic-plastic Winkler material. This material is determined not only by the modulus of subsoil reaction k_h , which characterizes the deformation in the elastic region, but also by additional limiting deformations. When these deformations are exceeded, the material behaves ideally plastic.

The following assumptions are made:

- The pressure acting on a wall may attain an arbitrary value between the values of the active and passive pressure – but it cannot fall outside of these boundaries.
- The pressure at rest acts on an undeformed structure ($w = 0$).

Assignment

Verify a multi-anchored wall made from steel soldier piles I 400 with a length of $l = 21$ m. The depth of the ditch is $h = 15$ m. The terrain is horizontal. The surcharge acts at the surface and is permanent with the size of $q = 25 \text{ kN/m}^2$. The GWT behind the construction is 10 m below the surface. The spacing between the centers of the steel profiles is $a = 2$ m.



Scheme of the wall anchored in multiple layers – Construction stage 1

Soil	Thickness of the layer of soil [m]	γ [kN/m ³]	φ_{ef} [°]	c_{ef} [kPa]	δ [°]	ν [-]
F6	4,5	19,5	20	16	7,5	0,4
F4	1	19,5	22	14	7,5	0,35
R3	10,6	22	40	100	15	0,25
R5 (1)	4	19	24	20	7,5	0,3
R5 (2)	1	21	30	35	14	0,25
R5 (3)	3,9	21	40	100	15	0,2

Table with the soil and rock parameters

The unit weight of the soil γ will be the same as the unit weight of the saturated soil γ_{sat} . Stress state is considered to be **effective**, the pressure at rest is calculated for **cohesive** soils and the calculation of uplift is selected as **standard** for each soil.

All anchors have a diameter $d = 32 \text{ mm}$ and modulus of elasticity $E = 210 \text{ GPa}$. The anchor spacing is $b = 4 \text{ m}$.

Anchor no.	Depth z [m]	Free length l [m]	Root l_k [m]	Slope α [°]	Anchor force F [kN]	Stage construction for a new anchor
1	2,5	13	6	15	300	2
2	5,5	10	6	17,5	350	4
3	8,5	7	6	20	400	6
4	11	6	4	22,5	500	8
5	13	5	3	25	550	10

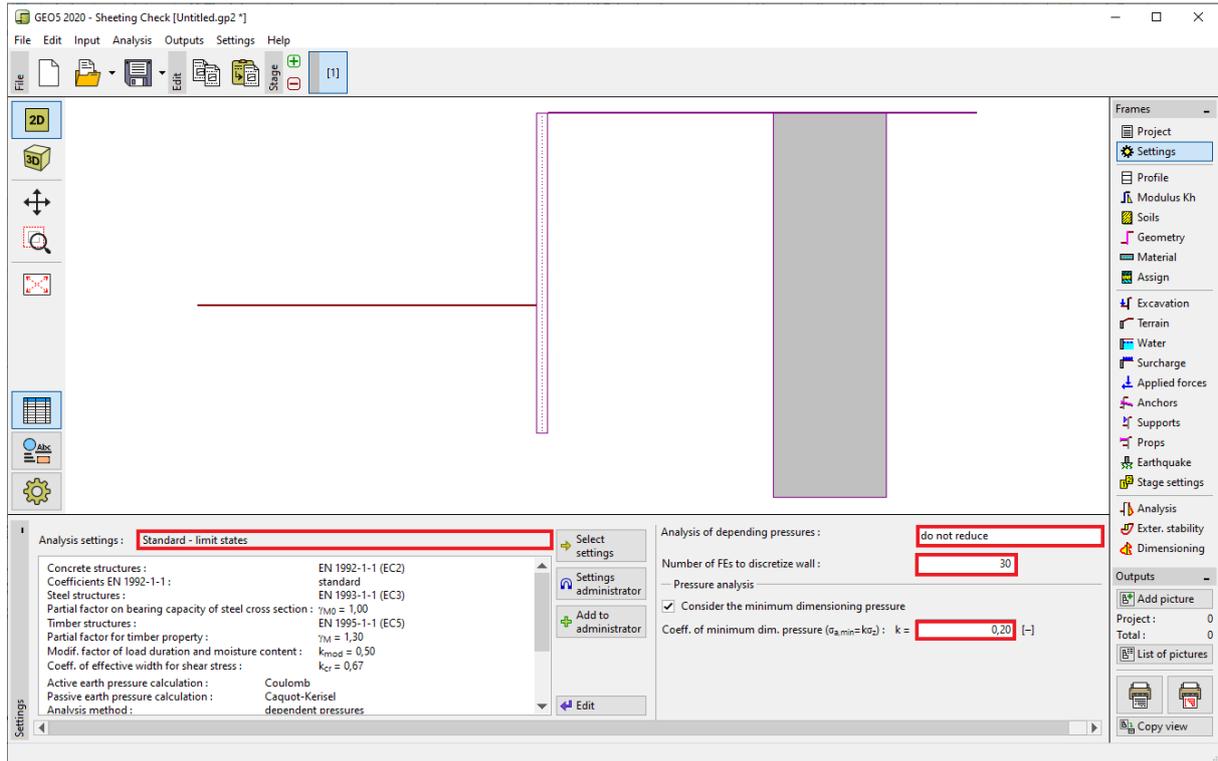
Table with positions and dimensions of the anchors

The Kh Modulus is linearly growing with the increasing depth up to the depth of 5 m. Its value reaches 10 MN/m^3 there. From this depth onwards, its value is constant.

Solution

To solve this task, use the GEO5 “Sheeting Check” program. The analysis will be performed without a reduction of the input data, so the real behavior of the structure will be observed.

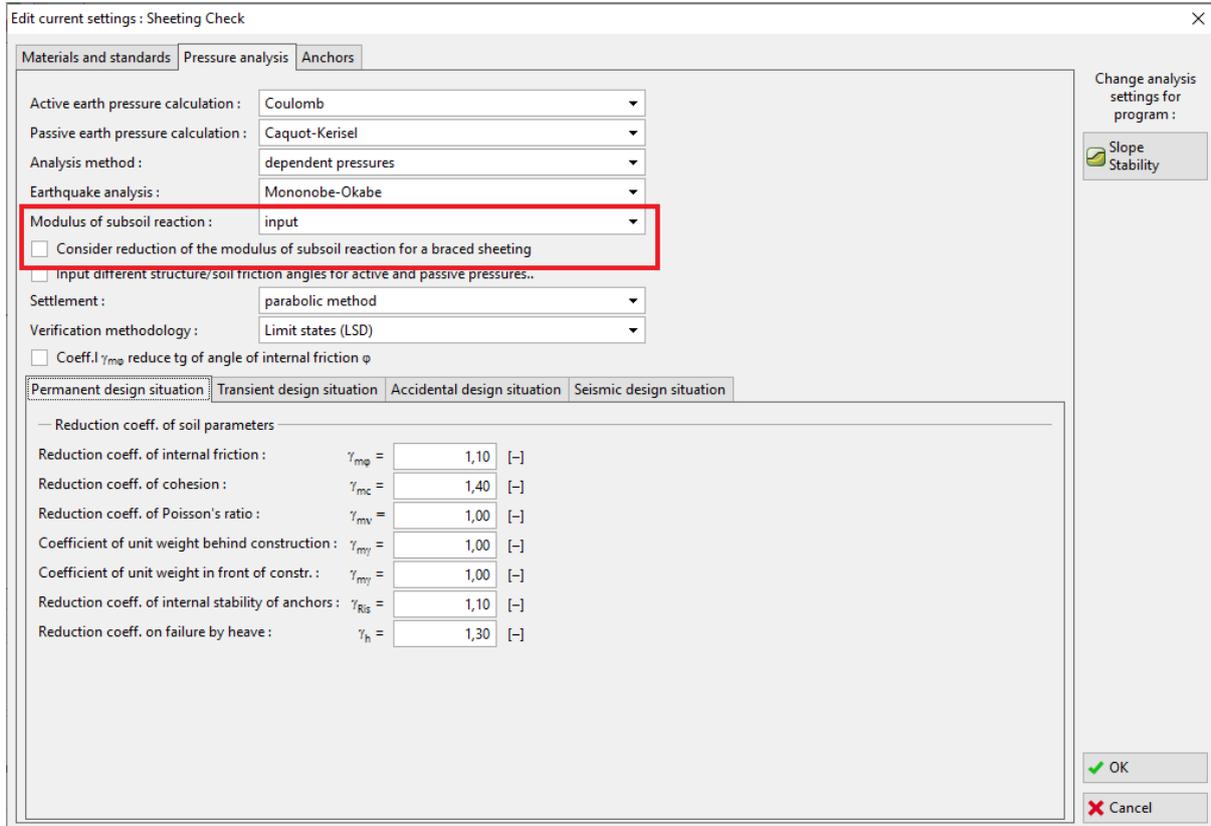
In the “Settings” frame select option no. 2 “Standard – limit states”. We will consider the minimum dimensioning pressure $k = 0,2$. Then, change the number of FEs to discretize the wall to 30 (*see figure*).



Frame “Settings”

Note: For more complex tasks (e.g. multiple anchored walls), the authors of the program recommend computing the limit pressures without reducing the soil input parameters - without reducing the size of the appropriate partial factors for earth pressures. The method of dependent pressures without the reduction of the soil input parameters better shows the real behavior of the structure (the user receives the real values of displacement) and this calculation is similar to the numerical solution by FEM (see program help – F1).

Then open the “Edit current settings” dialog window using the “Edit” button and select “input” as the method for modulus of subsoil reaction. Also uncheck the possibility “Consider reduction of the modulus reaction for braced sheeting” (more info in program help – press F1).



“Edit current settings” Dialog window

In the “Profile”, “Soils” and “Assign” frame define the geological profile of the task according to the table and information in the assignment above. First, in the “Profile” frame, add 4 new interfaces at the depths shown in the figure below.

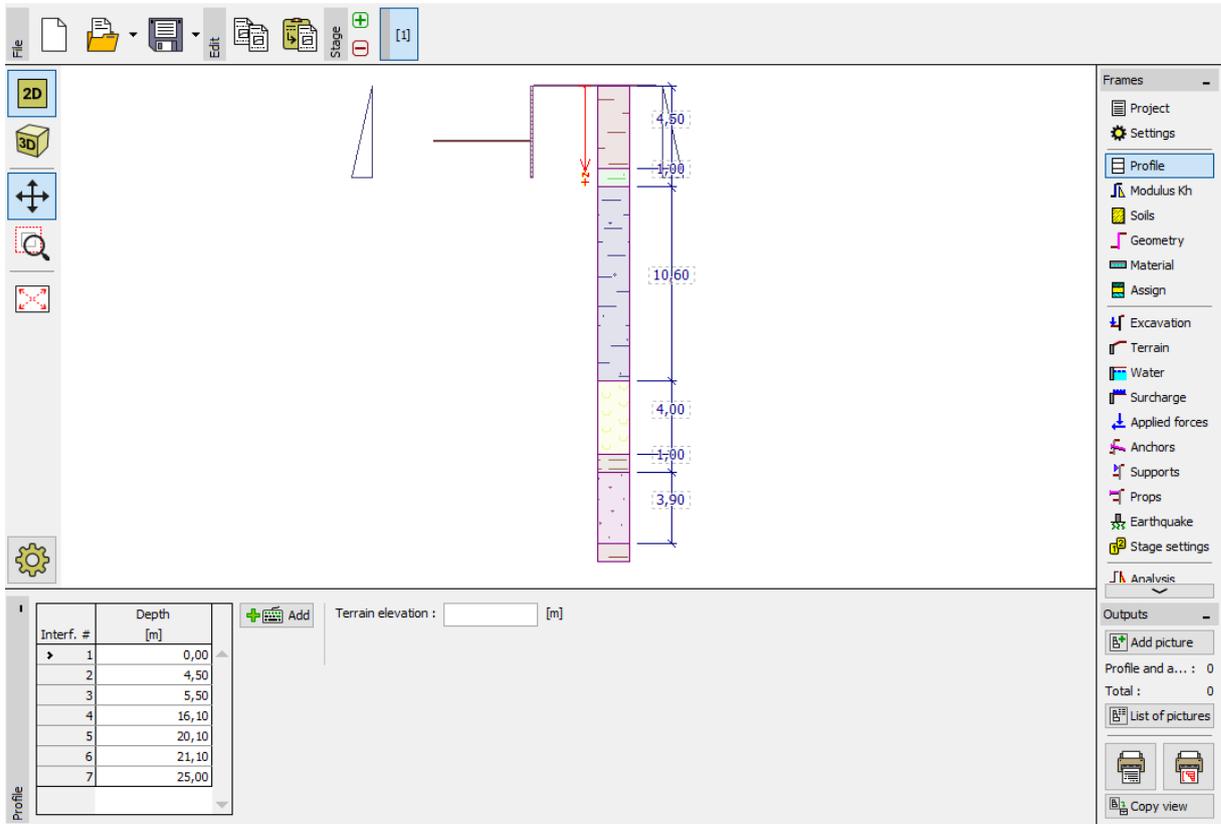
The screenshot displays the GEO5 software interface. At the top, there is a menu bar with options like File, Edit, and Stage. Below the menu bar is a toolbar with icons for 2D and 3D views, navigation, and settings. The main workspace shows a vertical profile with several segments and their depths: 4,50, 4,20, 10,60, 4,00, 4,00, and 3,90. A red arrow points to the 4,20 segment. On the right side, there is a 'Frames' panel with a list of frames including Project, Settings, Profile, Modulus Kh, Soils, Geometry, Material, Assign, Excavation, Terrain, Water, Surcharge, Applied forces, Anchors, Supports, Props, Earthquake, Stage settings, and Analysis. Below the 'Frames' panel is an 'Outputs' panel with buttons for 'Add picture', 'List of pictures', and 'Copy view'. At the bottom left, there is a 'Profile' table with columns for 'Interf. #' and 'Depth [m]'. The table contains the following data:

Interf. #	Depth [m]
1	0,00
2	4,50
3	5,50
4	16,10
5	20,10
6	21,10
7	25,00

Below the table, there is an 'Add' button and a 'Terrain elevation' input field with a unit of [m].

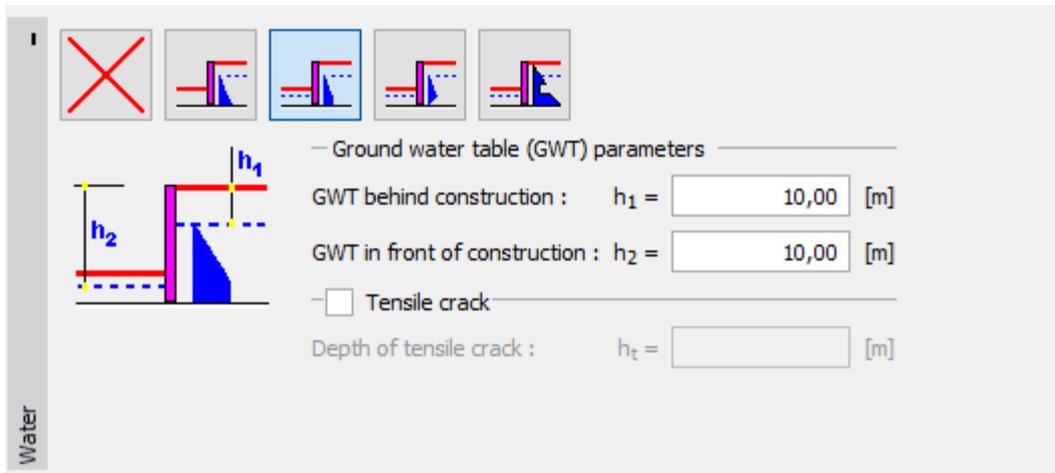
“Profile” Frame – add a new interface

Then, in the “Soils” frame, add 6 new soils with the parameters described in the table above and in the “Assign” frame, assign them to the profile.



“Profile” Frame – soils assigned to profile

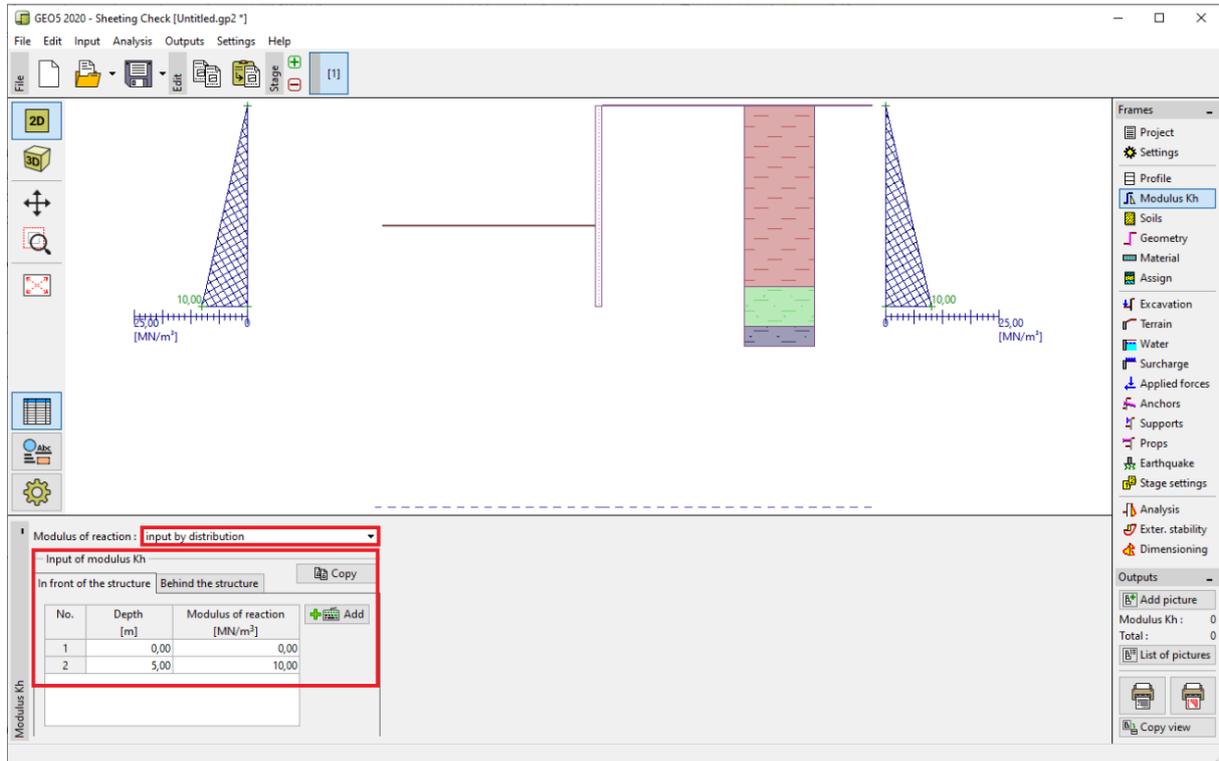
Lastly, define the GWT in the “Water” frame. In the first construction stage, the values in front and behind the construction will be 10 m.



“Water” Frame – define GWT parameters

In the “Modulus Kh” frame input the value of Kh by distribution – linearly growing to a depth of 5 m and then constant with the value of 10 MN/m³. The modulus is defined to the length of the structure.

If the length is changed, the modulus is automatically corrected. In this case, it will continue constantly to the overall length of the structure (21 m). If the value of Kh isn’t known, we could use the known parameters of the soils to calculate it (for example using the Schmitt method – based on E_{osd} or E_{def}). More information about the modulus Kh can be found in the program help – F1.



“Modulus Kh” Frame

Then, in the “Geometry” frame define the parameters of the braced sheeting – the type of the wall and the section length $l = 21$ m. Click on the “Add” button and from the I-sections database, select the **I (IPN) 400** section. The spacing between the centers of the steel profiles will be defined as $a = 2$ m. We will also change the coefficient of pressure reduction below the ditch bottom to **0,5**.

Note: The coefficient of reduction of earth pressures below the excavation reduces the pressures in the soil. For normal retaining walls this is equal to 1,0. For braced sheeting it is less than or equal to 1. It depends on the size and the spacing of the braces (more info in the program help – F1).

New section [X]

Type of wall : Steel I-section

Cross-section name : I-cross-section : I(IPN) 400; a = 2,00 m User def.

Section length : $l =$ 21,00 [m]

Coeff. of pressure reduc. below ditch bottom : input 0,50 [-]

— Geometry —

Spacing of centers : $a =$ 2,00 [m]

— Cross-section —

Name : **I(IPN) 400**



— Information —

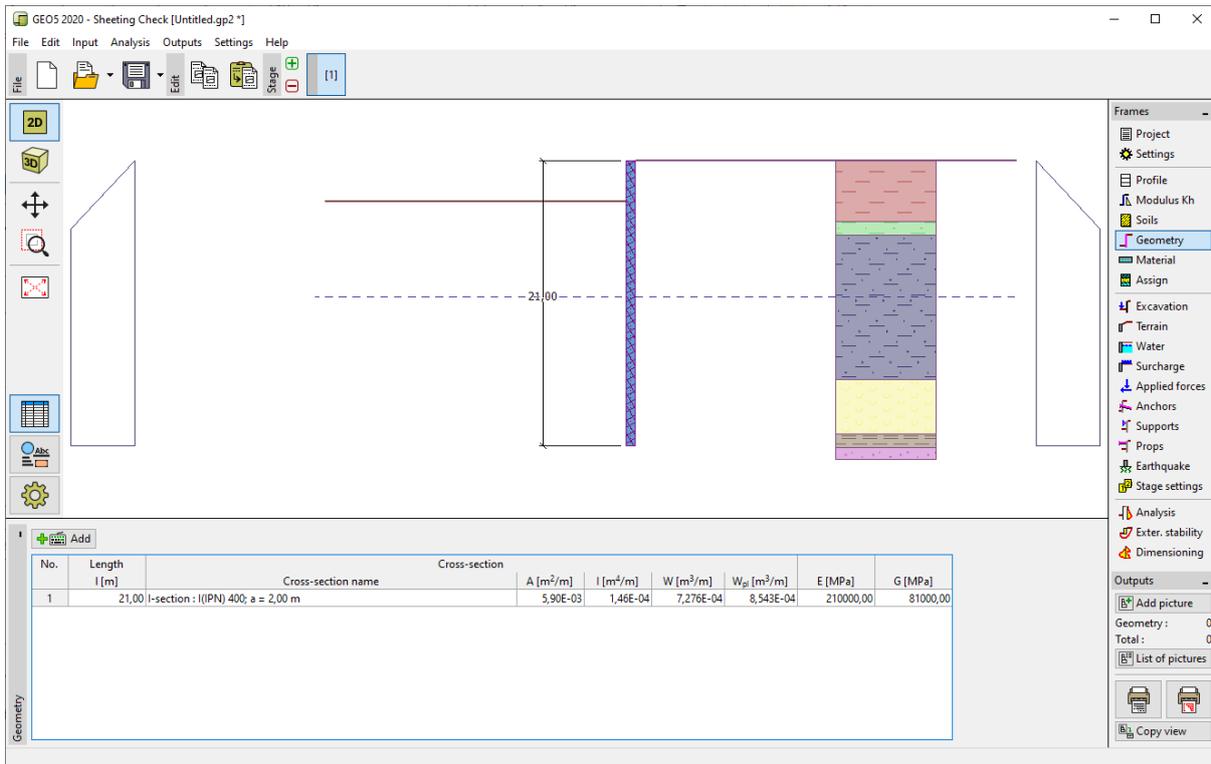
$A = 5,90E-03$ [m²/m] $I = 1,46E-04$ [m⁴/m]

$W_{y1} = 7,276E-04$ [m³/m] $W_{pl,y} = 8,543E-04$ [m³/m]

“New section” Dialog window

GEOS 2020 - Sheeting Check [Untitled.gp2 *]

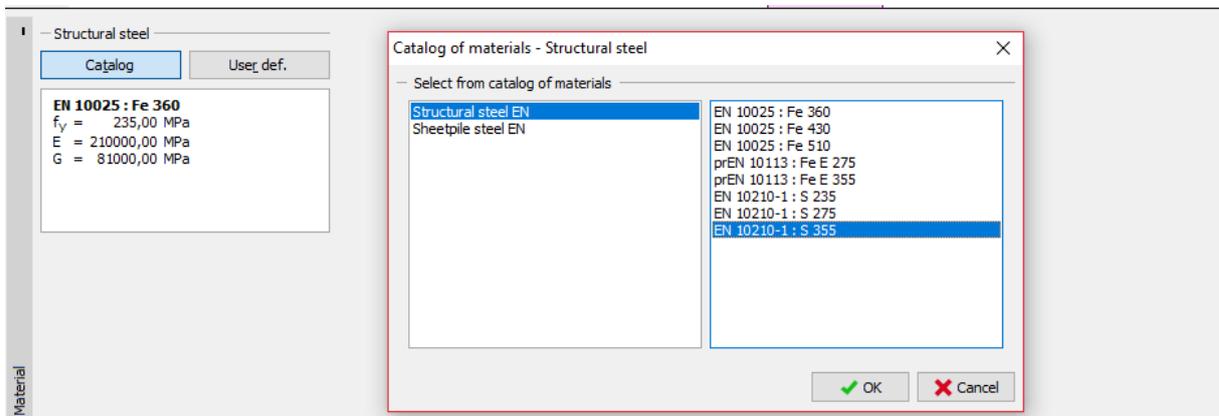
File Edit Input Analysis Outputs Settings Help



No.	Length l [m]	Cross-section name	Cross-section				E [MPa]	G [MPa]
			A [m ² /m]	I [m ⁴ /m]	W [m ³ /m]	W _p [m ³ /m]		
1	21,00	I-section : I(IPN) 400; a = 2,00 m	5,90E-03	1,46E-04	7,276E-04	8,543E-04	210000,00	81000,00

“Geometry” Frame – add a new section

In the “Material” frame, select the appropriate class of steel for the structure from the catalog. In this case, select the **EN 10210-1: S 355** type.

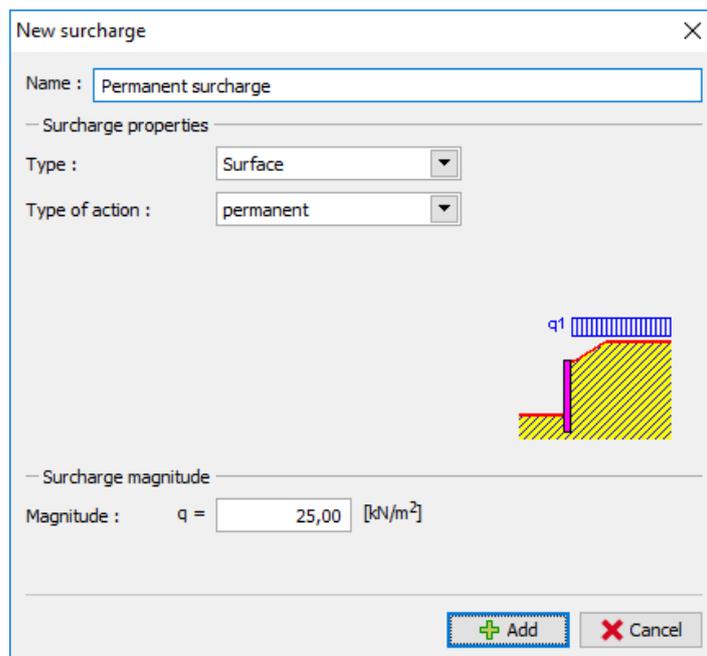


“Catalog of materials” Dialog window

Now, we will describe building the wall stage by stage. It is necessary to model the task in stages, to reflect how it will be constructed in reality. In each stage it is necessary to look at the values of internal forces and displacement.

If the braced sheeting is not stable in any stage of the construction or if the analyzed deformation is too large, then we need to change the structure – for example make the wall embedment longer, make the ditch shallower, increase the anchor forces etc.

In the first stage of the construction we will define the permanent surface surcharge in the frame “Surcharge” as $q = 25 \text{ kN/m}^2$.



“Surcharge” Frame – “New surcharge” dialog window

In construction stage 1, the ditch is made to a depth of $h = 3$ m. The **GWT** behind and in front of the structure is at a depth of $h_1 = h_2 = 10$ m beneath the surface (ground level). Perform the analysis and then add a new stage. In stage 2, add a new anchor in the “Anchors” frame it will be placed at a depth of $z = 2,5$ m.

New anchor ✕

Anchor type : ▾

Name :

— Anchor parameters —

Depth : $z =$ [m]

Free length : $l =$ [m]

Root length : $l_k =$ [m]

Slope : $\alpha =$ [°]

Spacing : $b =$ [m]

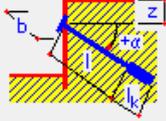
— Stiffness —

Type of input : ▾

Diameter : $d_s =$ [mm]

Elasticity modulus : $E =$ [MPa]

Pre-stressing force : $F =$ [kN]



“Anchors” Frame – add a new anchor (construction stage 2)

The screenshot displays the GEO5 2020 software interface. The main workspace shows a 2D cross-section of a retaining wall with an anchor. The anchor is placed at a depth of 2.50 m. The software window title is "GEO5 2020 - Sheeting Check [C:\Users\tomas\Desktop>manual 7.gp2 *]". The interface includes a menu bar (File, Edit, Input, Analysis, Outputs, Settings, Help), a toolbar, a central workspace with a 2D/3D view, a right-hand "Frames" panel, and a bottom "Anchors" table.

No.	new	Anchor post-stressed	Depth z (m)	Anchor type	Name	Spacing b (m)	Force F (kN)
1	Yes		2,50	not specified	Anchor no. 1	4,00	300,00

"Anchors" Frame – Construction stage 2

Perform the analysis and add another construction stage. In the 3rd stage of the construction, go to the "Excavation" frame and change the depth of the ditch to $h = 6,5$ m. There will be no anchors added in this stage. Again, perform the analysis and add a 4th construction stage. In the 4th stage, add a new anchor placed at a depth of $z = 5,5$ m. The GWL has stayed the same so far.

New anchor [X]

Anchor type :

Name :

— Anchor parameters —

Depth : $z =$ [m]

Free length : $l =$ [m]

Root length : $l_k =$ [m]

Slope : $\alpha =$ [°]

Spacing : $b =$ [m]

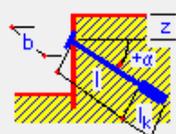
— Stiffness —

Type of input :

Diameter : $d_s =$ [mm]

Elasticity modulus : $E =$ [MPa]

Pre-stressing force : $F =$ [kN]

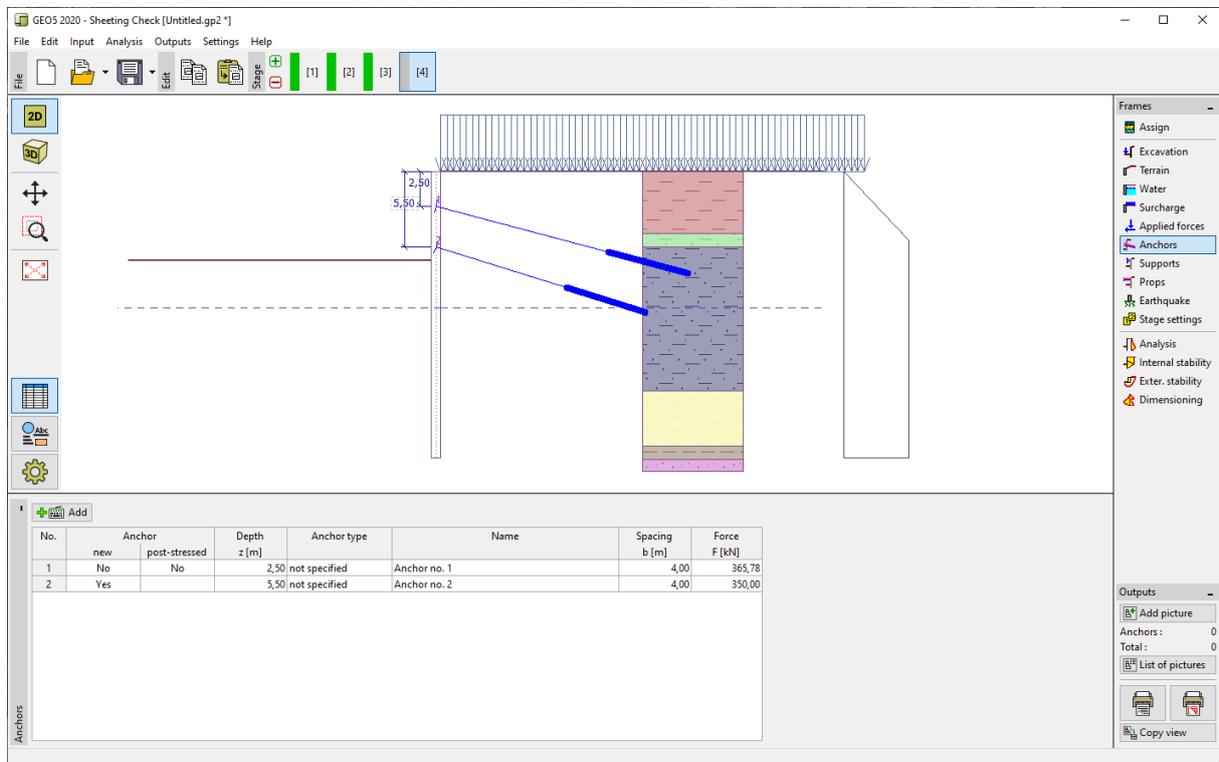


“Anchors” Frame – add a new anchor (construction stage 4)

GEOS 2020 - Sheeting Check [Untitled.gp2 1]

File Edit Input Analysis Outputs Settings Help

File [Icons] Stage [1] [2] [3] [4]



Anchors

No.	new	Anchor post-stressed	Depth z [m]	Anchor type	Name	Spacing b [m]	Force F [kN]
1	No	No	2,50	not specified	Anchor no. 1	4,00	365,78
2	Yes	Yes	5,50	not specified	Anchor no. 2	4,00	350,00

Outputs

Anchors: 0

Total: 0

“Anchors” Frame – Construction stage 4

Add a 5th construction stage. In the 5th construction stage, change the depth of the ditch to $h = 9$ m. Then, add another construction stage. In the 6th stage, add a new anchor at a depth of $z = 8,5$ m. The depth of the GW is unchanged.

New anchor ✕

Anchor type :

Name :

— Anchor parameters —

Depth : $z =$ [m]

Free length : $l =$ [m]

Root length : $l_k =$ [m]

Slope : $\alpha =$ [°]

Spacing : $b =$ [m]

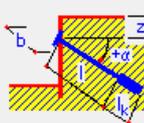
— Stiffness —

Type of input :

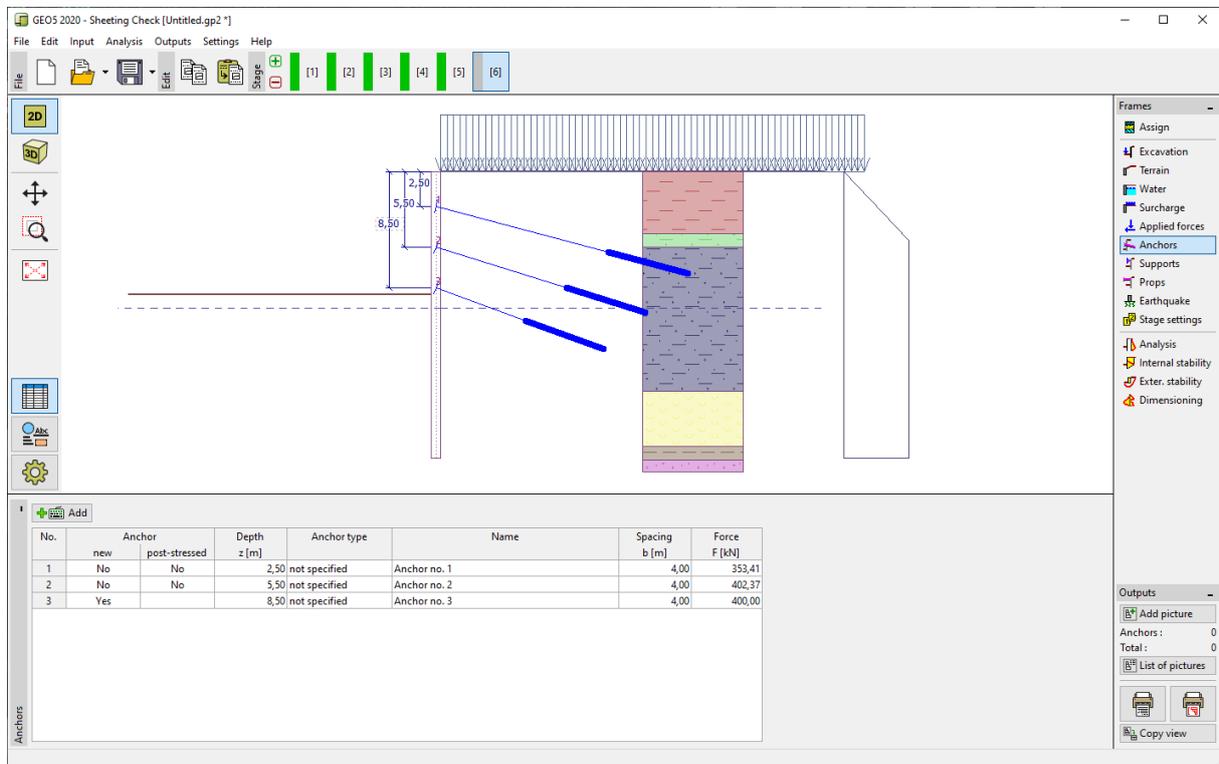
Diameter : $d_s =$ [mm]

Elasticity modulus : $E =$ [MPa]

Pre-stressing force : $F =$ [kN]



“Anchors” Frame – add a new anchor (construction stage 6)



No.	new	Anchor post-stressed	Depth z [m]	Anchor type	Name	Spacing b [m]	Force F [kN]
1	No	No	2,50	not specified	Anchor no.1	4,00	353,41
2	No	No	5,50	not specified	Anchor no.2	4,00	402,37
3	Yes	No	8,50	not specified	Anchor no.3	4,00	400,00

“Anchors” Frame – Construction stage 6

Add another construction stage. In the 7th construction stage, change the depth of the excavation to $h = 11,5$ m. In the frame “Water”, also change the GWT in front of the wall to $h_2 = 12$ m. The GWT behind the structure is unchanged. Add the 8th construction stage. In the 8th construction stage, add a new Check anchor at a depth of $z = 11$ m.

New anchor ✕

Anchor type :

Name :

— Anchor parameters

Depth : $z =$ [m]

Free length : $l =$ [m]

Root length : $l_k =$ [m]

Slope : $\alpha =$ [°]

Spacing : $b =$ [m]

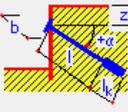
— Stiffness

Type of input :

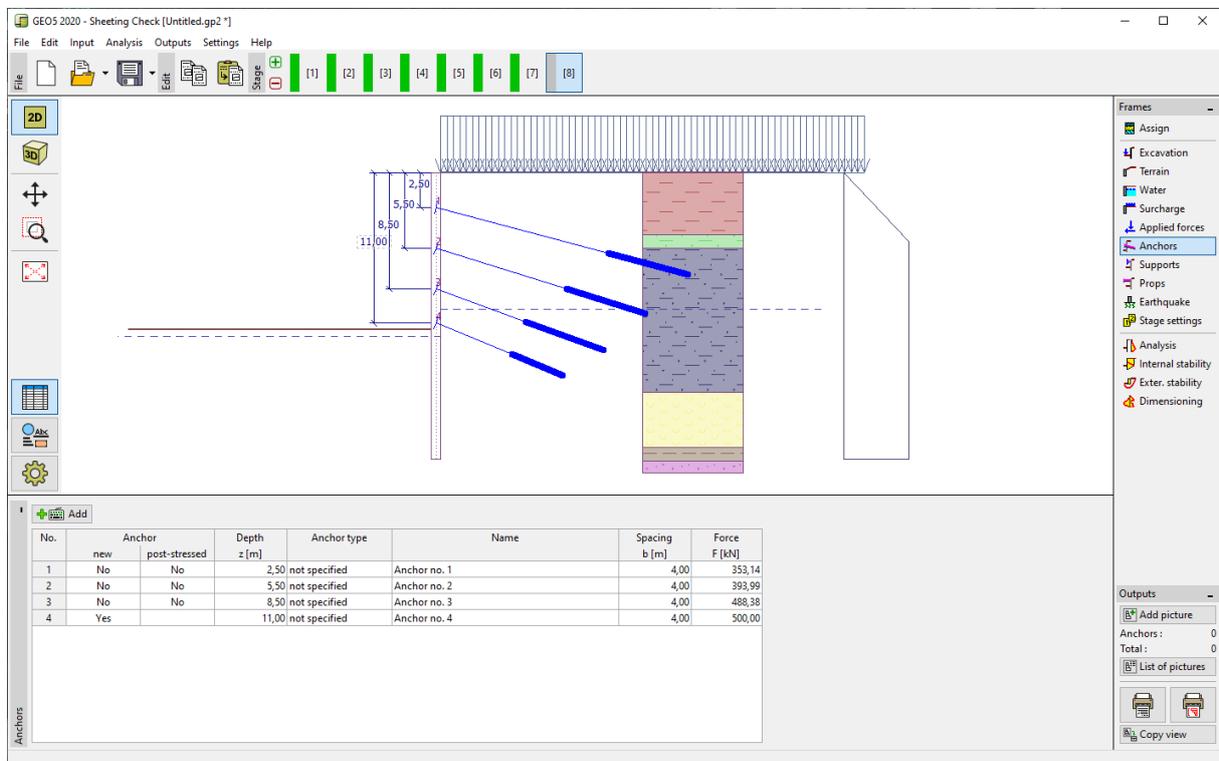
Diameter : $d_s =$ [mm]

Elasticity modulus : $E =$ [MPa]

Pre-stressing force : $F =$ [kN]



“Anchors” Frame – add a new anchor (construction stage 8)



The screenshot shows the 'Anchors' frame in GEO5 2020. The main window displays a cross-section of a retaining wall with four anchors installed at different depths. The software interface includes a menu bar, a toolbar, and a right-hand panel with various analysis and output options.

No.	new	Anchor post-stressed	Depth z [m]	Anchor type	Name	Spacing b [m]	Force F [kN]
1	No	No	2,50	not specified	Anchor no. 1	4,00	353,14
2	No	No	5,50	not specified	Anchor no. 2	4,00	393,99
3	No	No	8,50	not specified	Anchor no. 3	4,00	488,38
4	Yes		11,00	not specified	Anchor no. 4	4,00	500,00

“Anchors” Frame—Construction stage 8

Add another construction stage. In the 9th construction stage, the ditch is excavated to a depth of $h = 13,5$ m. Change the GWT in front of the structure to $h_2 = 15,5$ m. Then, add another construction stage. In the 10th stage, add a new anchor at a depth of $z = 13$ m.

New anchor ✕

Anchor type :

Name :

— Anchor parameters —

Depth : $z =$ [m]

Free length : $l =$ [m]

Root length : $l_k =$ [m]

Slope : $\alpha =$ [°]

Spacing : $b =$ [m]

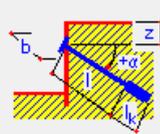
— Stiffness —

Type of input :

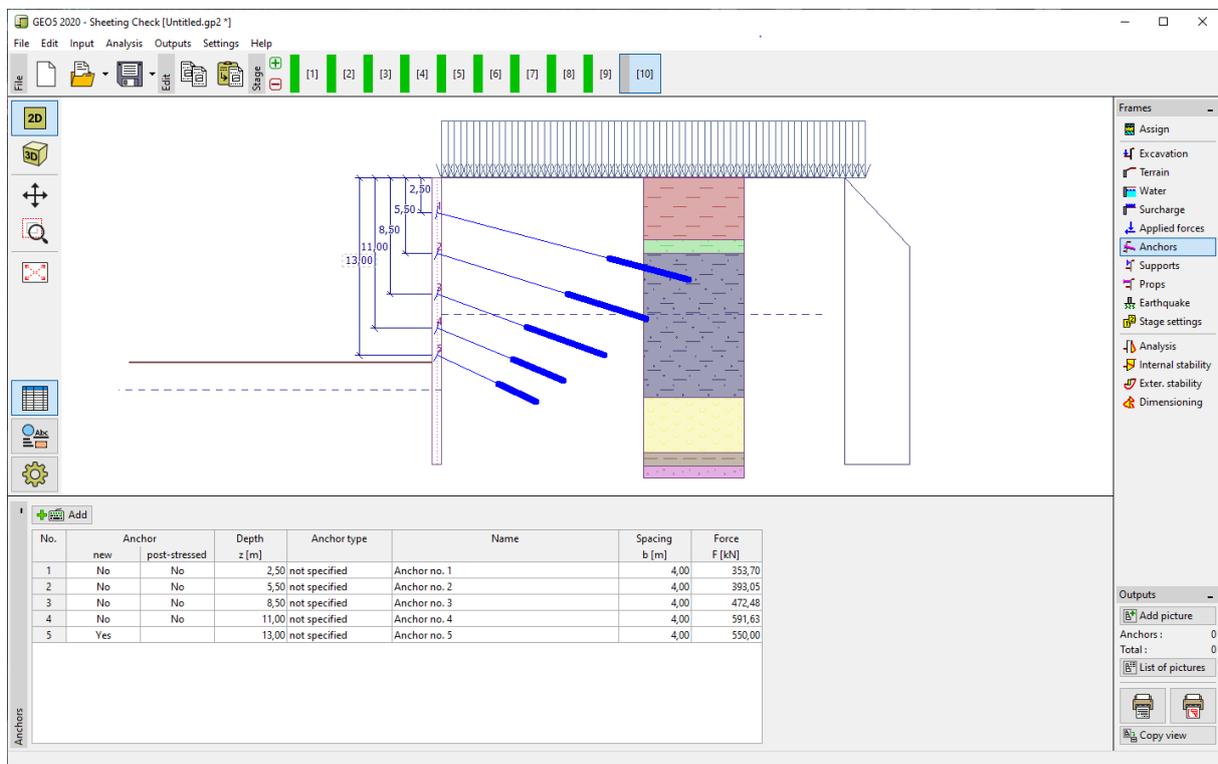
Diameter : $d_s =$ [mm]

Elasticity modulus : $E =$ [MPa]

Pre-stressing force : $F =$ [kN]



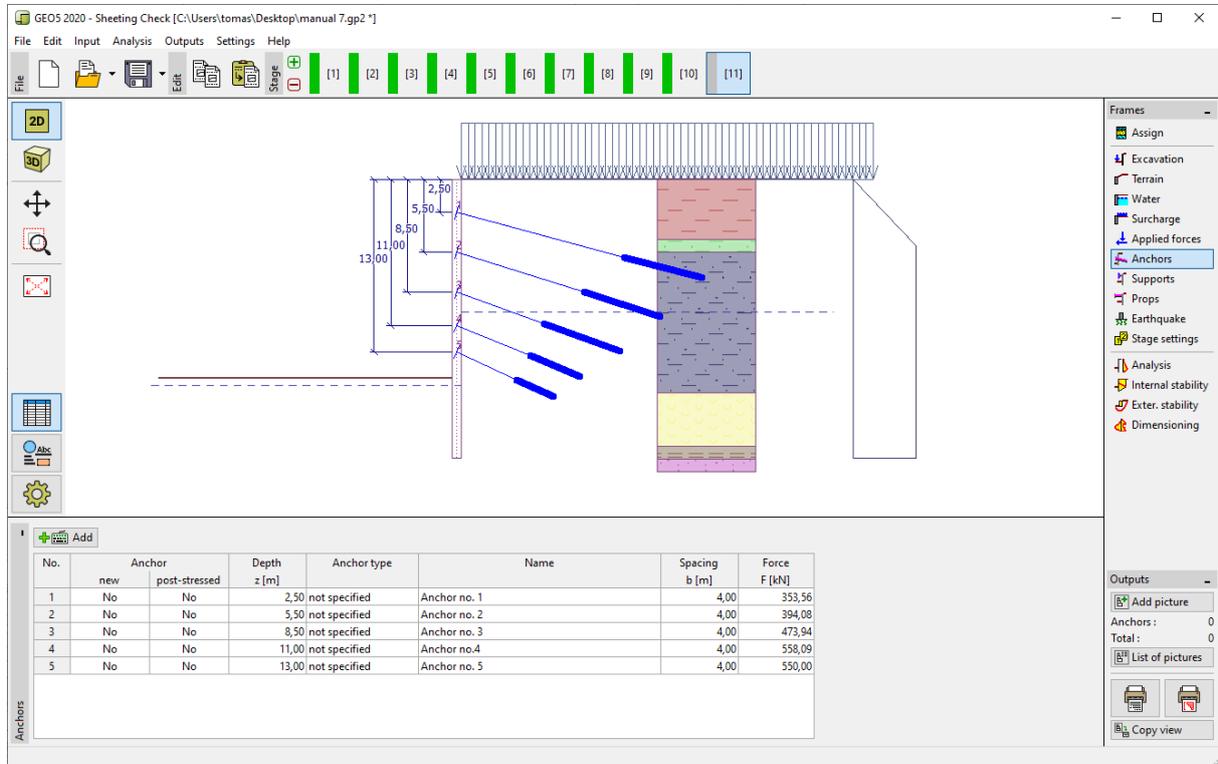
“Anchors” Frame – add a new anchor (construction stage 10)



No.	new	Anchor post-stressed	Depth z [m]	Anchor type	Name	Spacing b [m]	Force F [kN]
1	No	No	2,50	not specified	Anchor no. 1	4,00	353,70
2	No	No	5,50	not specified	Anchor no. 2	4,00	393,05
3	No	No	8,50	not specified	Anchor no. 3	4,00	472,48
4	No	No	11,00	not specified	Anchor no. 4	4,00	591,63
5	Yes	No	13,00	not specified	Anchor no. 5	4,00	550,00

“Anchors” – Construction stage 10

In the 11th and last construction stage, change the depth of the excavation to $h = 15$ m. We will not add any new anchors. The GWT has not changed since the 9th stage of the construction (in front of the wall it's at a depth of $h_2 = 15,5$ m, behind the wall it's at a depth of $h_1 = 10$ m).

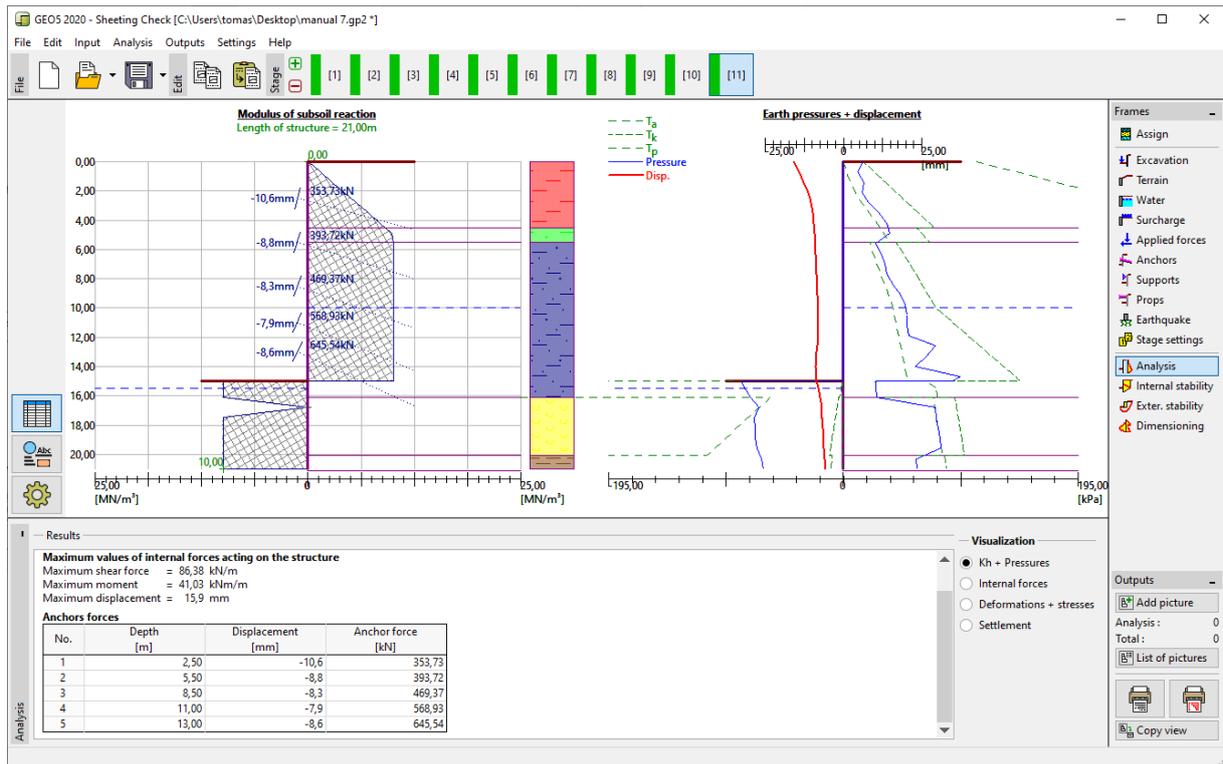


“Anchors” Frame – Construction stage 11

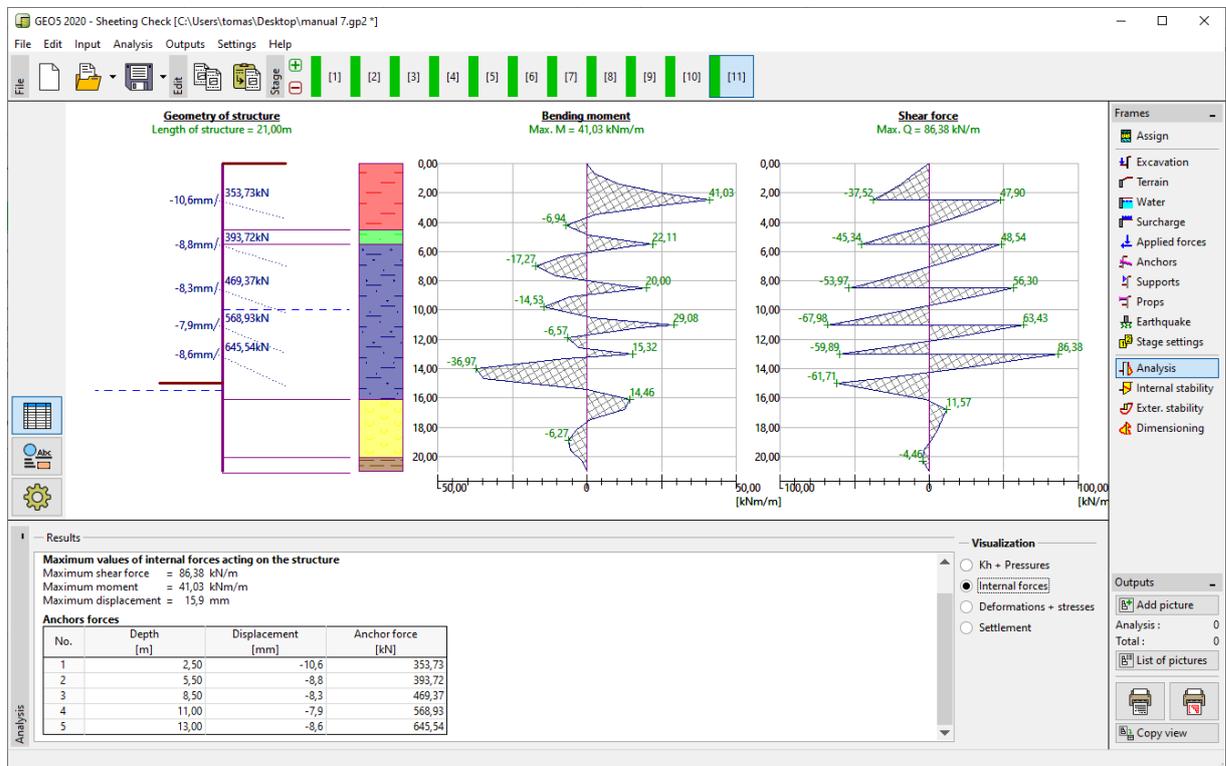
Note: Due to a deformation of the structure the forces in the anchors are changing. These changes depend on the stiffness of the anchors and the deformation of the anchor's head. The force can decrease (due to a loss of pre-stress force) or increase. The forces can be pre-stressed in any stage of the construction to the required force.

Results of the analysis

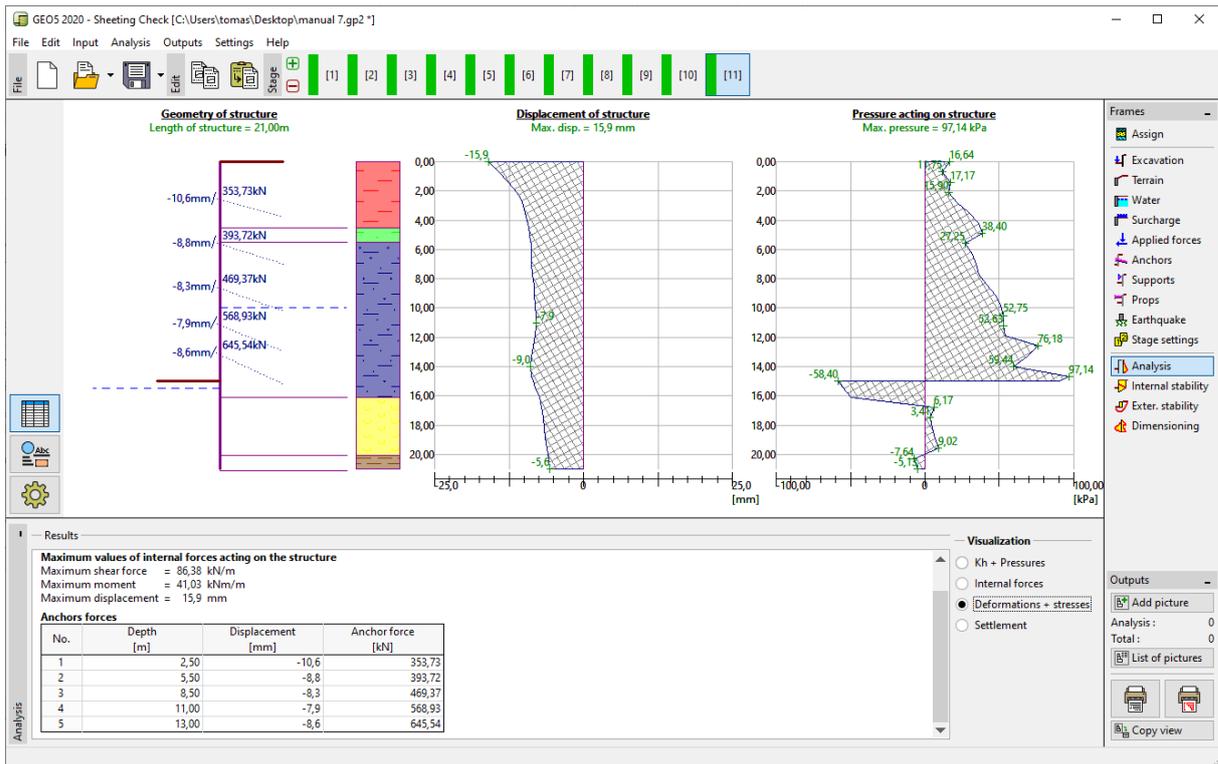
In the pictures below the results of the analysis are shown (analysis of internal forces – bending moment and shear force, displacement of the structure and earth pressure) for the last 11th construction stage.



“Analysis” Frame – Construction stage 11 (modulus of subsoil reaction)



“Analysis” Frame – Construction stage 11 (Internal forces)



“Analysis” Frame – Construction stage 11 (Displacement of structure + earth pressures)

All the stages are analyzed. That means that the structure of the braced sheeting is stable and functional in all construction stages. The size of the displacement must also be checked, as well, as that the anchor force does not exceed the bearing capacity of the anchor (The user must check this manually as this is not automatically checked by the Sheeting Check program). For the last 11th construction stage the results are as follows:

- Max. shear force: $Q_{\max} = 86,38 \text{ kN/m}$
- Max. bending moment: $M_{\max} = 41,03 \text{ kNm/m}$
- Max. earth pressure: $\sigma_x = 97,04 \text{ kPa}$
- Max. displacement: $u_{\max} = 15,9 \text{ mm}$

Verification of a cross-section of the structure

In the last, 11th construction stage, open up the frame “Dimensioning”. Here you can see the maximum and minimum values of the variables (envelopes of internal forces).

- Maximum shear force (minimum): $Q_{\max,\min} = 86,38 \text{ kN/m}$
- Maximum bending moment (minimum): $M_{\max,\min} = 50,73 \text{ kNm/m}$

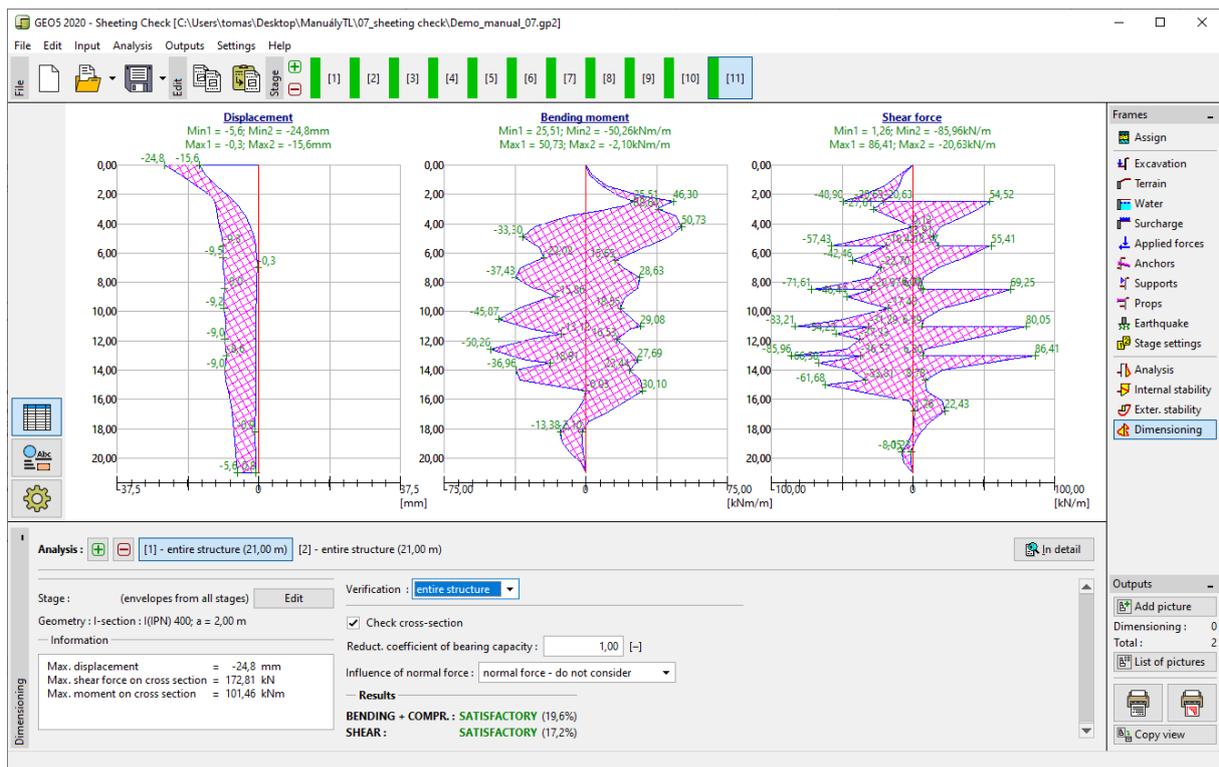
The internal forces are calculated per one meter (foot) of the construction. For an actual design of the soldier beams (steel I-section) we have to multiply these values by the spacing between the profiles $a = 2 \text{ m}$, to obtain the internal forces in the cross-section.

- Max. shear force for dimensioning: $Q_{Ed} = 86,38 * 2,0 = 172,76 \text{ kN}$
- Max. bending moment for dimensioning: $M_{Ed} = 50,73 * 2,0 = 101,46 \text{ kNm}$

The program performs the assessment of soldier beams (steel I-section) using the extreme values of internal forces according to EN 1993-1-1 (EC 3).

For the time being, we leave the reduction coefficient of bearing capacity at 1,0. In this case the results are as follows:

- Bearing capacity of the cross-section: $M_{Rd} = 516,61 \text{ kNm} \geq M_{Ed,max} = 101,46 \text{ kNm}$.
- Total utilization of the steel I-section: **19,6 %** I-section satisfies the analysis criteria.



“Dimensioning” Frame – Construction stage 11 (Assessment of steel I- section I 400)

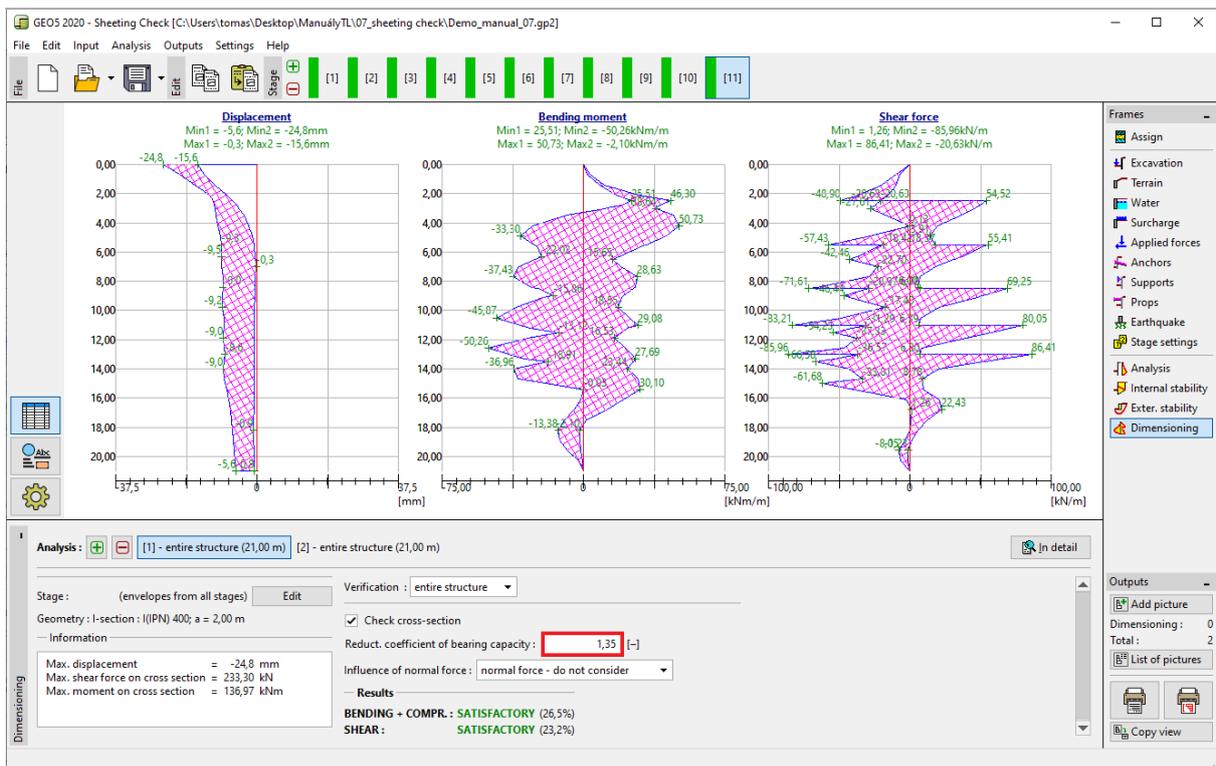
In the calculation we kept the size of the limit earth pressures non-reduced, so the load is lower than it should be according to the EN 1997-1. However, the internal forces suit the real behaviour of the structure. Changes in earth pressures lead to improvements in safety, but also distort the results of the analysis. That’s why, for the assessment of the steel section we introduced a custom value of the reduction coefficient of bearing capacity.

Note: The EN 1997-1 standard assumes the partial factor for a permanent load to be $\gamma_G = 1,35$, for a variable load it equals to $\gamma_Q = 1,5$. However, in this case, all of the surcharge and the load act as permanent, we therefore consider the partial factor γ_G to be 1,35.

For a combination of permanent and variable loads, we have to determine the value of the design partial factor estimates, ranging from 1,35 to 1,5 of the ratios depending on the components of the load, which is prevalent.

Now, we will change the reduction coefficient of bearing capacity to 1,35. We multiply the internal forces acting on the cross-section of the soldier beam by this partial factor. In this case the internal forces are as follows:

- Max. shear force for dimensioning: $Q_{Ed} = 86,38 * 2,0 * 1,35 = 233,30 \text{ kN}$
- Max. bending moment for dimensioning: $M_{Ed} = 50,73 * 2,0 * 1,35 = 136,97 \text{ kNm}$



Frame "Dimensioning" – Construction stage 11 (New assessment of steel I-section I 400)

In this case (an assessment with the influence of the reduction coefficient of bearing capacity as 1,35) the results are:

- Bearing capacity of the cross-section: $M_{c,Rd} = 516,61 \text{ kNm} \geq M_{Ed,max} = 136,97 \text{ kNm}$
- Total utilization of the steel I-section: **26,5 %** OK

- Shear verification: $V_{c,Rd} = 1005,29 \text{ kN} \geq Q_{Ed,max} = 233,30 \text{ kN}$
- Total utilization of the steel I-section: **23,2 %** OK

Analysis of the internal stability

Go to the frame “Internal stability” in the last construction stage and look at the maximum allowable force in each anchor.

Note: This is the way verification is done. At first, we iterate the force in the anchor, resulting in an equilibrium of all forces acting on the earth wedge. This earth wedge is bordered by the construction, the terrain, the centre of the roots of the anchors and the theoretical heel of the structure (more info can be found in the program help – F1). If an anchor is not satisfactory the best way to resolve the issue is to make it longer or to decrease the pre-stressed force.

The calculation shows us the maximum force in the anchor (row no. 5) and the total utilization of the anchor:

- Internal stability: **10,66 %** $F = 645,54 \text{ kN} < F_{max} = 6056,23 \text{ kN}$ OK

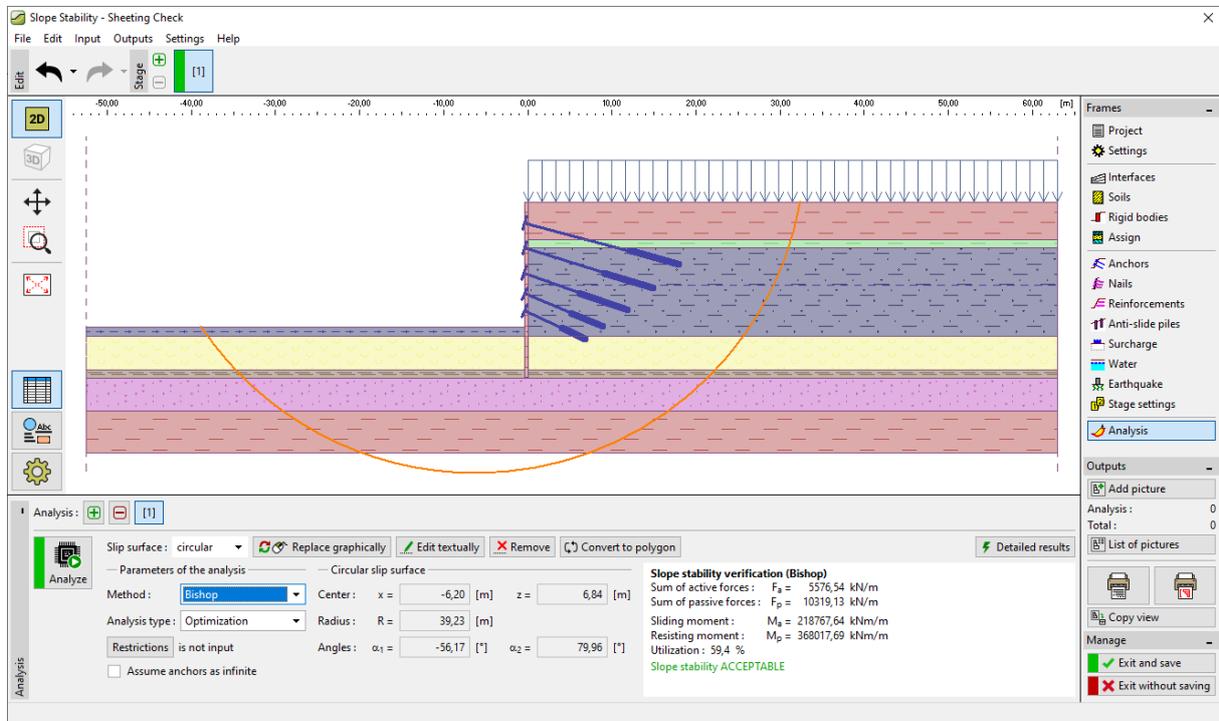
No.	Anchor force [kN]	Max. allow. force [kN]	Verification
1	353,73	6841,55	is satisfactory
2	393,72	6133,74	is satisfactory
3	469,36	5345,08	is satisfactory
4	568,86	5442,13	is satisfactory
5	645,53	6056,23	is satisfactory

Results
 Verification of decisive row of anchors: [In detail](#)
 Anchor No. : 5
 Specified anchor force = 645,53 kN
 Computed max. allowable anchor force = 6056,23 kN
SATISFACTORY

“Internal stability” Frame – Construction stage 11

Verification of the external (overall) stability

The last required analysis is the external stability. Clicking on the “External stability” frame automatically opens the “Slope stability” program, where we go to the “Analysis” frame. There, after clicking on the button “Analyze”, we compute the overall stability.



“Slope Stability” program – Bishop method with an optimization of circular slip surface

Conclusion, completion of the results:

The structure was successfully designed with a maximum deformation of 24,8 mm. This is satisfactory for this type of construction. Additionally, the limits of forces in the anchors were not exceeded.

– Bearing capacity:	26,5 %	516,61 kNm ≥ 136,97 kNm	Satisfactory
– Internal stability:	10,66 %	F = 645,54 kN < F_{max} = 6056,23 kN	Satisfactory
– Overall stability:	59,4 %	Method – Bishop (optimization)	Satisfactory

The designed braced sheeting satisfies the evaluation criteria.