fine civil engineering software

geotechnical software suite

GEO5

Engineering manuals

Part 3

www.finesoftware.eu

Engineering manuals for GEO5 programs

Part 3

Chapter 1 - 12, refer to Engineering Manual Part 1

Chapter 13 - 19, refer to Engineering Manual Part 2

Chapter 20. Finite Element Method (FEM) – Introduction

The objective of this chapter is to explain basic terms of the particular field of problems and practical application of GEO 5 – FEM program to solve some geotechnical problems.

GEO 5 – FEM program allows for the modelling of various types of problems and analyses. The text below explains basic terms and general procedures in a more detailed way – individual analysis modules are described in other chapters.

From the **problem type** aspect, GEO 5 – FEM program distinguishes two basic cases:

– ***a planar problem***: this analysis module is suitable for solving linear structures (a tunnel, embankment, open cut, dam etc.), for which it applies that their longitudinal dimension is by an order of magnitude larger than the lateral dimensions of the area being solved.

  In such a case it is proper to relate the analysis to 1 *rm* of the structure and solve the problem *under the assumption of plane-strain deformation*. Deformations in planes parallel to the longitudinal axis of the structure can be disregarded in this case. Thus the development of deformations and stresses in the massif is taken into consideration only in the plane perpendicular to the longitudinal axis and, as a result of lateral contraction, also the normal stress in the longitudinal direction. Regarding beam elements, it is the case of solving a 1*m* wide plate strip (for more details visit Help – F1).

– ***axial symmetry***: this analysis module is suitable for solving rotationally symmetric problems. This assumption must be satisfied by both the geometrical arrangement of the structure and loading. A suitable example is the analysis of a vertically loaded single pile, circular excavation or pumping groundwater from a circular borehole.

  Similarly to the case of the plane-strain deformation, this is the case of a generally three-dimensional problem, which, however, can be again converted into the planar problem solution. The solution is then related to $1\,rad$ of the $x\,(r)$-radius arch. The axis of symmetry always represents the $x\,(r)$ coordinate origin.

Shear components of deformation in the direction of rotation can be disregarded. The evolution of a circumferential normal component of stress and deformation is also taken into consideration, in addition to the components of stress and deformation in the cross-sectional plane (for more details visit Help – F1).

From the **type of analysis** point of view, the program allows the following cases to be solved using individual modules:

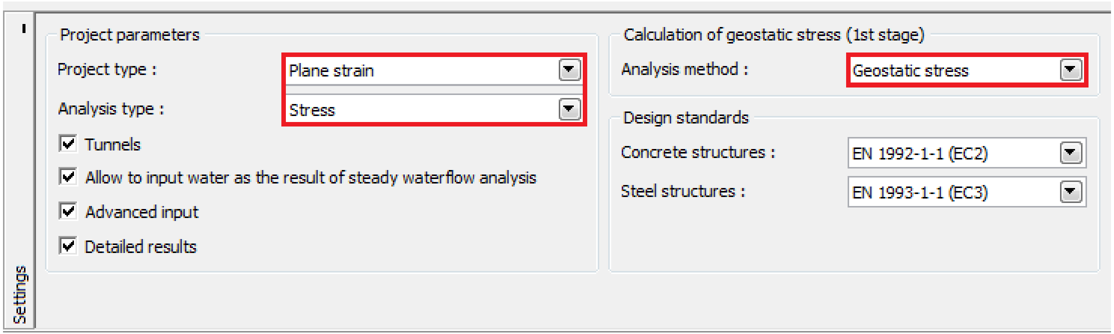
- ***stress*:** it serves to solve basic geotechnical problems in ground environment and a rock mass (e.g. for determining the vertical or horizontal geostatic stress, pore pressure, deformations, volumetric changes and sub-grade deformations, as well as analysing internal forces along a diaphragm wall structure length (height) etc.).

- ***steady flow***: it assumes a zero change in the saturation degree with time; individual construction stages are completely independent of each other (in contrast with the unsteady flow).

- ***unsteady flow***: this analysis module allows for determination of the evolution of pore pressures (the total head) and the current degree of saturation with time possible. In this case, the analysis methodology is similar to that of the stress analysis.

- ***slope stability***: during the analysis, this program reduces the input values of the internal friction angle φ_{ef} or the soil cohesion c_{ef} and seeks the onset of failure associated with the evolution of the critical region of localized plastic deformation. The result is the factor of safety corresponding to classical slope stability analysis methods. The definition and input of a model in this regime is completely identical with that of the "Stress" module.

- ***tunnels***: this module allows for performing the analysis of underground excavation (modelling of the 3D effect of excavation attributed to the New Austrian Tunnelling Method), accounting for degradation of beams, temperature-induced loads acting on beams, swelling-induced loads acting within specified regions and monitoring of results.



„Settings" frame

GEO 5 – FEM program incorporates the possibility of so-called advanced specification, where it is possible to define in more detail complementary input parameters of soils for individual material models, to develop mixed meshes with multi-nodal elements and visualise larger numbers of output variables.

Note: The standard setting assumes drained boundary conditions. In such the case the analysis assumes steady-state conditions, where the skeleton deformation has no influence on the development of pore pressures. In such a case pressures have only the character of external loads and do not vary during the course of the analysis. In the case of undrained conditions, where the entire boundary of the particular area behaves as being fully impermeable, we solve the opposite case, i.e. the fully coupled problem of the skeleton deformations and pore pressures under the assumption that all changes are immediate and the influence of time does not take effect.

The method of determining the initial stress (1^{st} construction stage) is assumed in the program as:

– ***geostatic stress***: The standard method for the analysis of the vertical geostatic stress is based on the following relationship:

$$\sigma_z = \sum_{i=1}^{n} \gamma_i \cdot h_i \ [kPa]$$

where: γ_i – bulk density of soil in i^{th} layer,

h_i – thickness of i^{th} layer.

– K_0 ***procedure***: It is used in the cases where the user needs to define another *initial lateral stress*. For example, the actual lateral stress in overconsolidated soils can be significantly higher than that in normally consolidated soils (for more details visit Help – F1). The lateral pressure coefficient K_0 is set as a soil parameter. If this parameter is not specified, it is provided by:

$$K_0 = \frac{\nu}{1-\nu}$$

where: ν – Poisson's ratio.

Input parameters of soils also depend on the selected material model for stress or flow analysis, respectively. The most important input parameters for stress analysis comprise the modulus of elasticity E and Poisson's ratio ν (to be specified for all models), and for non-linear models, the angle of internal friction φ_{ef} and the soil cohesion c_{ef}. The steady-state condition after the redistribution of pore pressures is assumed in the program and *effective parameters* of shear strength of soils or rocks are therefore used during the analysis (for more details visit Help – F1).

The selection of the material model and subsequent setting of soil parameters belong among the most important and at the same time most problematic tasks when the FEM modelling of a structure is being carried out. Material models try to plausibly describe the behaviour of soil or rock). They can be divided into two basic groups, namely **linear** and **non-linear** models.

Note: The correct selection of the material model is absolutely necessary for giving a true picture of the real behaviour of the structure. Non-linear models are necessary for the majority of structures (e.g. the analysis of a sheeting wall structure using a linear model of soil will give completely unrealistic results), but the application of linear models can be in many cases very proper, simplifying the entire analysis. A simplified procedure recommended for the modelling of problems using the FEM is presented in Help – F1).

Linear models give relatively fast, but not very accurate, assessment of the real material behaviour. They can be used in the cases where we are interested only in the stress or deformations of the ground mass, not in the area and mode of the potential failure.

They can also be used in the cases where only a local failure develops, having no fundamental influence on the development of a global failure, but which may result into the premature termination of the analysis in the program (for more details visit Help – F1).

The group of linear models comprises:

- ***elastic model***: It uses conversion relationships between the stress and deformation given by Hooke's law (within the linear elasticity scope).

- ***modified elastic model***: It makes incorporating the influence of surcharge loading or unloading into the analysis possible, using the secant modulus E_{def} and the unloading/reloading modulus E_{ur}.

Although, if we try to obtain a plausible description of the ground mass behaviour or if we are interested in the distribution location of areas of potential failures, it is necessary to adopt non-linear models. Basic non-linear models can be again divided into two groups. The first group of models is based on the classical Coulomb failure condition. It comprises the **Drucker-Prager**, **Mohr-Coulomb** and **Modified Mohr-Coulomb models**. Using these models, it is also possible to model hardening or softening of soils. A common feature of these models lies in the unlimited elastic deformation under the assumption of geostatic stress (for more details visit Help – F1).

The second group of material models, which are based on the notion of the critical state of soil, is represented by the **Modified Cam-clay, Generalized Cam-clay** and **Hypoplastic clay models**. These models provide a significantly better picture of the non-linear response of soil to external loading. Individual material models differ not only in their parameters, but also in the assumptions.

The boundary between the linear (elastic) and non-linear (plastic) response of the material is formed by the surface of plasticity. The mathematical expression of the surface of plasticity represents a certain failure condition (plasticity function). Exceeding this condition leads to the development of permanent (irreversible) plastic strains.

Note: Apart from the basic material parameters adopted for linear models, the non-linear models require introducing certain strength-related characteristics of soils, which are necessary for the formulation of yield conditions. ***The onset of evolution plastic strain*** *depends on the value of angle of internal friction φ and cohesion c. The dilatancy angle ψ controls informs magnitude of the plastic volumetric strain (for more details visit Help – F1).*

The selection of a material model suitable for the analysis of geotechnical structures adheres first of all to the character of the soil/rock environment. In the Finite Element Method-based process of comprehensive modelling of more complex problems, the selection of the numerical model represents an absolutely essential influence on specifying the input data and assessing the analysis results.

Working with interface, model dimensions:

The description of the work with the specification of individual interfaces is in a more detailed way presented in the program help (see F1). The input data essential for numerical analyses using the FEM are the so-called *world coordinates* (specifying the magnitude of the area being solved), where it is necessary, especially for stability analyses, to secure sufficient neighbourhood (the interface width) to obtain meaningful results.

Note: The depth of the FE mesh is also very important. It is possible to visualise the end of the mesh as an incompressible sub-grade. If there is no incompressible sub-grade specified for the particular geological profile, it is possible to assume that internal forces will disappear at a certain distance from the loading spot or from the contact of the structure with the sub-grade; no deformation will therefore develop. The world boundaries of the problem being solved are subsequently defined to be at this distance (for more details visit Help – F1).

The interface can be imported from other programs of the GEO 5 system via the clipboard. The program further allows the import and export of interfaces in *DXF format and the import of interfaces in gINT format. To simplify the specification of interface points (geometry) it is possible to use the so-called interface corrector (for more details visit Help – F1).

Mesh generation:

The successful generation of a mesh is the last item of the process of setting the *structure topolog*y (interfaces between soil layers, lines of structures, parameters of soils and rocks, contacts etc.). Individual construction stages are modelled and analysed subsequently. When the mesh is being generated, the program automatically generates also the *standard boundary conditions*. The standard setting of the boundary conditions comprises:

- smooth pin at the mesh nodes found along the horizontal bottom edge,
- sliding pin at the mesh nodes along the left-hand and right-hand vertical edges.

The GEO 5 – FEM program has a built-in *automatic corrector of the specified* geometry. This means that the program itself finds intersections of lines and all closed areas and develops an adequate model before the generation of the finite element mesh (for more details visit Help – F1).

The newly developed areas can be subsequently removed from the analysis or new soil can be assigned to them. The user will become aware of the main advantages of this system particularly when analysing tunnels and braced structures. Specifying even a very complicated structure becomes a very simple and fast matter (for more details visit Help – F1).

The correctly generated finite element mesh is the basic condition for obtaining results regarding the real behaviour of the structure. GEO 5 – FEM program has a built-in automatic mesh generator, which significantly simplifies this task. Despite this fact it is necessary to adhere to certain rules:

- The denser the mesh the more accurate results. On the other hand, the problem analysis becomes significantly slower. The objective therefore is to find the optimum density of the mesh. It depends both on the user experience and the finite element used.

– The mesh of elements should be sufficiently dense, especially in the area where large stress gradients can be expected (spot support, sharp corners, underground excavations etc.). It is perform mesh refinement around individual points or lines. It is necessary for the density-refining radius to be at least 3 to 5 times larger than the density in the density refining centre and both values at the points (the density and radius) to be in a reasonable proportion to the mesh density prescribed for the surrounding area. In this way smooth transition between the areas with different density will be secured.

Note: Singular lines must be dealt with similarly. In the cases of more complicated problems it is advisable to carry out a preliminary analysis of a course finite element mesh and on the basis of the analysis of generated results, to carry out a local refinement of the mesh density (for more details visit Help – F1).

The program uses six-node triangular elements with automatic mesh smoothing by default (for more details visit Help – F1).

Construction stage:
When the structure topology specification and the finite element mesh generation are finished, the analyses are then carried out for individual construction stages.

The construction stages correspond to the gradual building of the construction, and the correct specification and sequence are very important. The analysis of each stage (with the exception of stability analyses) is based on the **results of the preceding stage.** Information about individual construction objects and their properties is maintained between the construction stages. The rule of properties inheritance is applied between the construction stages when a stage is being edited or specified (for more details visit Help – F1).

The first construction stage (**the analysis of the primary geostatic stress**) represents the initial state of the massif before the construction commencement. For that reason the analysis result refers to the stress in soil or rock mass, not to deformations.

Note: A principal problem of the FEM analysis usually lies in the non-convergence of some of the construction stages. If the results of one phase are not available, the analysis of the subsequent stage is impossible. As for the correct modelling, the program authors recommend to follow a recommended procedure for the modelling of construction process (for more details visit Help – F1).

Analysis settings and description:

During the course of the analysis the program tries to find (iterate) such a solution at which the equilibrium conditions in the massif are met for the specified boundary conditions. The iteration process and the analysis convergence can be followed on the screen (for more details visit Help – F1).

The analysis can be interrupted at any time; in such a case the results are available for the last successful convergence of the loading increment. Correct results can be obtained when the 100 per cent loading is reached. Nevertheless, it may happen that the analysis passes off only up to a certain percentage of the loading, which means that the program has not succeeded in finding the correct solution and the analysis does not converge (for more details visit Help – F1). In such the case it is possible to change the standard analysis parameters setting or to carry out some modifications of the model:

– increasing the structure stiffness,
– reducing the applied loads,
– reduce the soil excavation area,
– improving soil characteristics,
– changing the material model of soils in plasticity locations,
– adding beam reinforcing elements or tensional elements,
– adding boundary conditions,
– changing the iteration course in the analysis settings (e.g. increasing the number of iterations).

Note: The visualisation of plastic strains showing critical locations with the anticipated development of failure surfaces can provide the advice why the analysis does not converge (for more details visit Help – F1).

The program provides a default set of analysis parameters ensuring sufficient accuracy and speed. However, an experienced user may want to change certain parameters or test their influence on the analysis accuracy (for more details visit Help – F1).

Note: The authors of the program recommend that any changes in the setting of analysis parameters should be approached very carefully, after thoroughly studying the problems. Incorrectly selected settings may cause not only incorrect iteration of the solution and slowing the analysis down, but also inaccurate results (for more details visit Help – F1).

It is possible in the analysis stage to change the following parameters in the "Settings" window:

- ***the solving method (and its settings)***: Newton-Rapshon method – NRM,
  Arc-length method – ALM.

- ***the stiffness matrix***: the initial stress method; the full or modified NRM.

- ***the initial computation step***: the ratio between the load in the particular loading step and the prescribed overall load (the standard setting value is 25 per cent of the overall load).

- ***the maximum number of iterations***: for achieving equilibrium within the framework of the given loading increment.

- ***the convergence criterion***: the setting of the error tolerance (changes in the vector standard) for displacement, unbalanced forces and internal energy.

- ***the line search method***: the determination of the weight coefficient η for meeting the equilibrium condition, resulting in the acceleration or dampening of the computation process.

- ***plasticity***: the setting of the tolerance for the error in the return to the yield surface, expressing the accuracy required for meeting the yield condition.

Note: Individual analysis settings, including basic equations for meeting the equilibrium conditions or yield conditions, are described in more detail in the program Help (for more details visit Help – F1).

Visualisation and interpretation of results:

The visualisation and interpretation of results is one of the most important parts of the program. GEO 5 – FEM program allows several basic types of graphical outputs:

- the drawing of the deformed structure
- planar representation of quantities in the ground mass (it is possible to represent absolute values or values compared with another construction stage)
- internal forces in beams and on contacts
- forces in anchors and reactions
- settlement trough
- tilted cross-sections or vectors and directions of the quantities

Note: The program uses certain coordinate systems for the representation of results. All outputs and selected results can be printed in the analysis protocol (for more details visit Help – F1).

Some results cannot be drawn concurrently because of the necessity for transparency and comprehensibility. It is, for example, impossible to draw a deformed structure and, at the same time, the distribution of internal forces along a beam. It is always necessary to choose only one variant. The program gives notice at the bottom of the dialogue window in the case of setting inadmissible combinations of outputs.

The program makes the setting of an arbitrary quantity of point monitors and line monitors possible at any place of the structure or outside the structure. ***Monitors*** have several functions:

- visualising the values of quantities at a particular point (the point monitor),
- visualising the values of the difference at a distance between two points compared with the previous construction stage $d\,[N]$, or compared with the setting phase where N , is the number of the construction stage (*the line monitor*).

List of chapters relating to the FEM:

- *Chapter 20*: Finite Element Method (FEM) – Introduction.
- *Chapter 21*: Terrain settlement analysis.
- *Chapter 22*: Settlement of a circular silo foundation.
- *Chapter 23*: Collector lining analysis.
- *Chapter 24*: Numerical solution to a sheeting wall structure.
- *Chapter 25*: Slope stability assessment.
- *Chapter 26*: Numerical modelling of tunnel excavation adopting the NATM method.
- *Chapter 27*: Steady water flow analysis – Body of the embankment.
- *Chapter 28*: Steady water flow analysis – Flowing in and out of a well.
- *Chapter 29*: Unsteady water flow analysis – Filling of a dam reservoir.

Chapter 21. Terrain settlement analysis

This example contains the solution to terrain settlement under surcharge loading using the Finite Element Method.

Task specification:

Determine the terrain settlement induced by strip surcharge loading $q = 250\,kPa$ along the length of 4.0 m and the total settlement after subsequent unloading. The geological profile is homogeneous; the soil parameters are as follows:

- Unit weight of soil: $\gamma = 19.0\,kN/m^3$
- Modulus of elasticity: $E = 15.0\ MPa$
- Poisson's ratio: $\nu = 0.35$
- Soil cohesion: $c_{ef} = 8.0\ kPa$
- Angle of internal friction: $\varphi_{ef} = 29.0\ °$
- Unit weight of saturated soil: $\gamma_{sat} = 21.0\,kN/m^3$

Regarding the *modified elastic model*, the input parameters of soils will be considered as follows:

- Modulus of soil deformation: $E_{def} = 15.0\ MPa$
- Unloading modulus: $E_{ur} = 45.0\ MPa$

Compare the analysis of the settlement or the total vertical deformation value $d_z\ [mm]$ with other material models (we will not take the Clam-Clay model and Hypoplastic model for clays into consideration because the soil mass is formed by cohesionless soil).

Note: The Mohr-Coulomb and Drucker Prager models are used in the engineering praxis even for cohesive soils because they are based on the shear failure and use common input parameters of soils and rocks (φ, c).

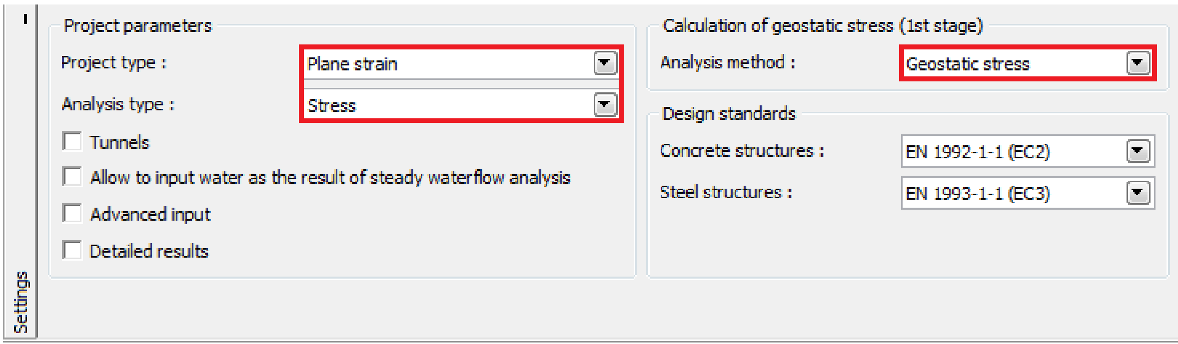
Solution:

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

- Topology: settings for and modelling of the problem,
- Construction stage 1: geostatic stress analysis,
- Construction stage 2: introduction of surcharge loading, terrain settlement analysis,
- Construction stage 3: terrain surface unloading, terrain settlement analysis,
- Assessment of results (conclusion).

Topology: settings for and modelling of the problem

First we will go over to the problem settings, where we will characterise the problem type itself, the type of the analysis and the primary stress analysis method.



Problem settings – Problem characteristics; primary stress analysis

We will not switch on the Tunnels, Advanced input and detailed results options – they are intended for experienced users of finite elements or for another type of problems. Their description exceeds the scope and purpose of this manual.

*Note: The **planar problem (plane-strain conditions assumed)** is suitable for solutions to linear structures (a tunnel, embankment, open cut, dam etc.), for which it applies that their longitudinal dimension is by an order of magnitude larger than the lateral dimensions of the area being solved – zero deformations are assumed in the direction of the y axis. The analysis is carried out under the assumption of plane-strain deformation (for more details visit Help – F1). The other problem type (axial symmetry) is solved in the subsequent chapter.*

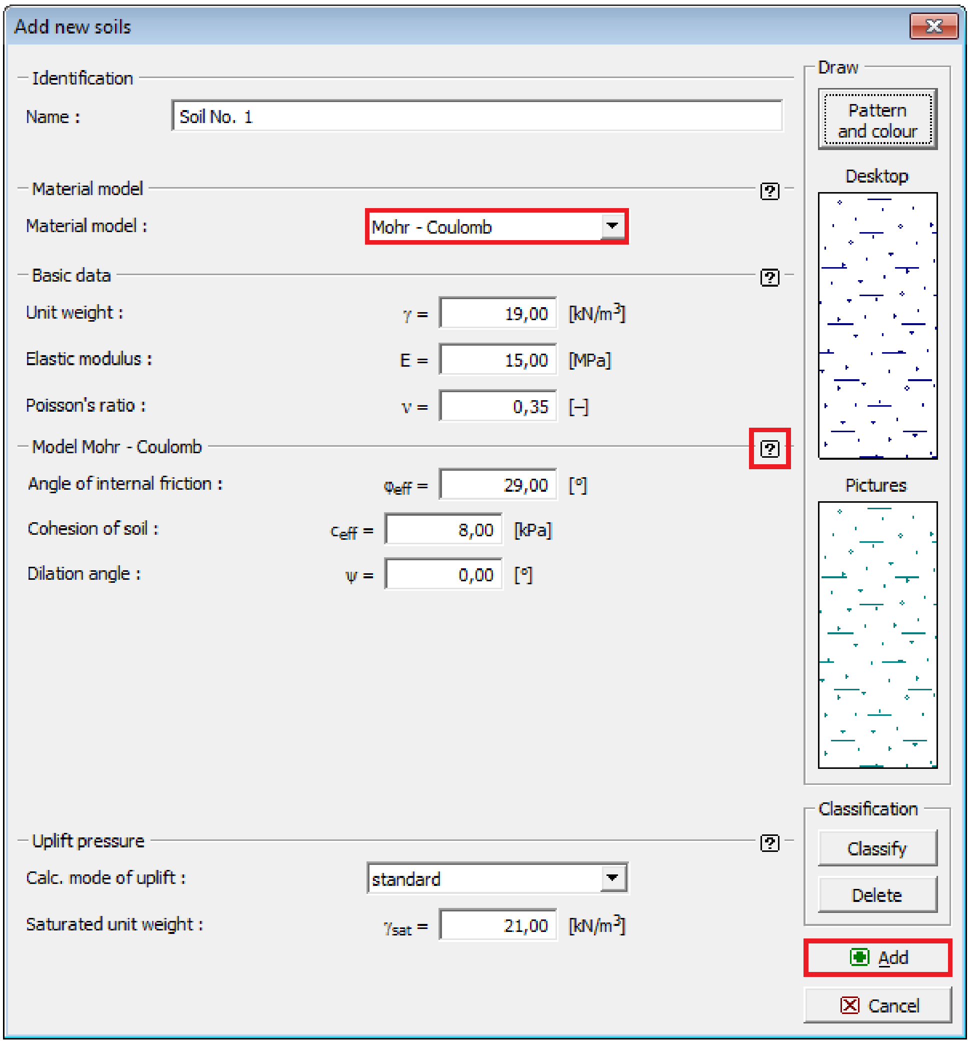
*Note: the **Stress** type of the analysis deals with stresses and deformations within the area being solved. It is the basic analysis type; other analysis types and other options (flow, slope stability) will be described separately in other chapters.*

*Note: Two options are available for the **primary stress** analysis (for construction stage 1):*

- ***geostatic stress**: It is a standard method for the geostatic stress analysis, taking into consideration the dead weight of soils and horizontal stresses and horizontal stresses according to the theory of elasticity. The lateral pressure coefficient is then given by* $K_0 = \frac{\nu}{1-\nu}$.

- K_0 ***procedure** (according to Jáky, for overconsolidated soils etc.)*.

As for the present analysis, we choose the **Mohr-Coulomb** model of soil (the comparison for various models is presented at the end of this example) and specify the particular soil parameters. This non-linear model will allow us to follow the development of plastic strains or the distribution of potential failure zones.

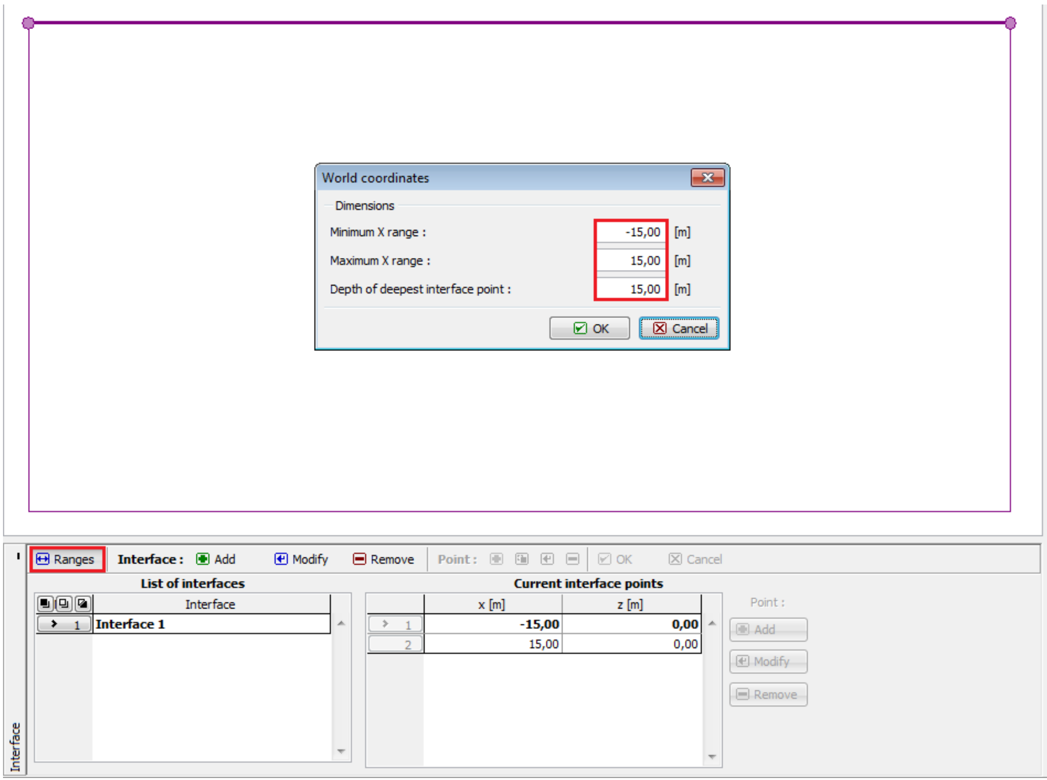


Setting soil parameters

Note: The elastic model assumes the soil behaviour according to Hooke's law (ideally elastic material). The main advantage of this model is that it always analyses the results to the very end. The disadvantage is that the soil behaves in this way only when the loading magnitude is small – it is therefore unsuitable for real structures. On the other hand, it is suitable for modelling areas in which we do not expect plastic failures of material (e.g. gabion walls, stiff sub-grade etc.) or for the verification of a basic numerical model.

Note: The modified elastic model allows for differences in the behaviour of soil under surcharge loading and unloading, however, more realistic results are obtained using more advanced constitutive relationships (Mohr-Coulomb, Drucker-Prager), which already take plasticity into consideration.

Then we will set the world coordinates (the magnitude of the numerical model of the problem being solved) and the terrain interface. We will choose the world coordinates to be sufficient for the results not to be affected by conditions at the edge. For our particular problem we will choose the model dimensions $\langle -15\,m; 15\,m \rangle$ and will set the thickness of 15.0 m for the layer to be examined. We will set the terrain coordinates – one point with the coordinates $[0,0]$ will be sufficient, the program will add the interface automatically.



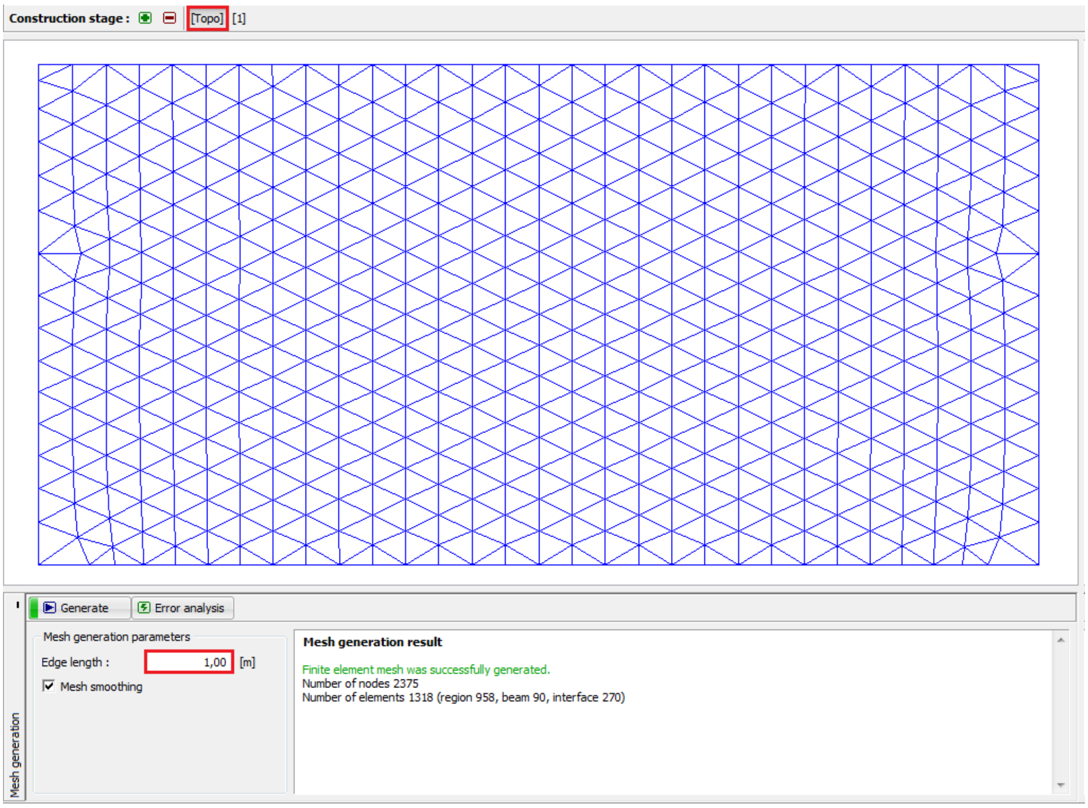
Specified terrain shape – current interface points, world coordinates

Note: The guidance values of recommended dimensions to set the model boundaries for individual solution cases are presented and described in more details in the program Help (for more details visit Help – F1).

Subsequently we will assign the soil to the area developed in this way.

We will leave out other frames for the specification of contact types, free points and lines; they are meaningless for our problem.

The next step is the generation of the Finite Element (hereinafter referred to as FE) mesh. For the mesh generation parameters, we will choose the length of 1.0m for the elements edge (the edge length is chosen depending on the problem dimensions and variability). We will check the option Mesh Smoothing and push the **Generate** button. The program will automatically generate and smooth the FE mesh. We will verify whether the mesh density is adequate taking into consideration the problem magnitude.



Generation of the finite element mesh – Topology (triangular mesh)

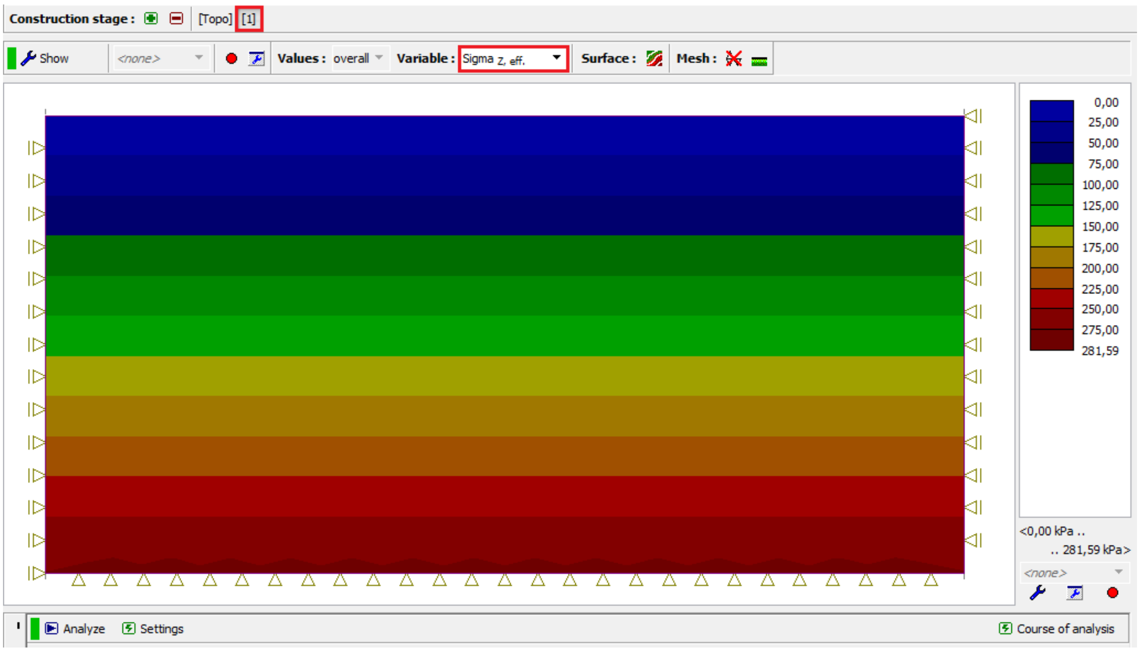
Note: The standard triangular mesh with six-node elements is suitable for the majority of geotechnical problems. In the advanced input mode, the program allows also for other mesh types (mixed, triangular) to be generated – this is intended for experienced FEM users.

Note: The correctly generated finite element mesh is the basic condition for achieving results reasonably well representing the real behaviour of the structure. The FE mesh significantly affects the values obtained because the FEM analysis primarily determines the values of nodal displacements. The remaining variables (stresses, strains) are determined from these values subsequently.

Unfortunately, it is impossible to provide a general rule for the correct mesh density because of the fact that individual problems are different. For beginners to the FEM analysis we recommend that first a courser mesh should be chosen, the problem analysis be carried out and then several other variants containing the smoothing of the mesh or its parts be tried. (It is also possible to refine the mesh density around the points or lines – more details are contained in the other chapters of EM). It applies in general that the courser the mesh the stiffer model behaviour (the resultant settlement value is smaller).

Construction stage 1: primary stress analysis

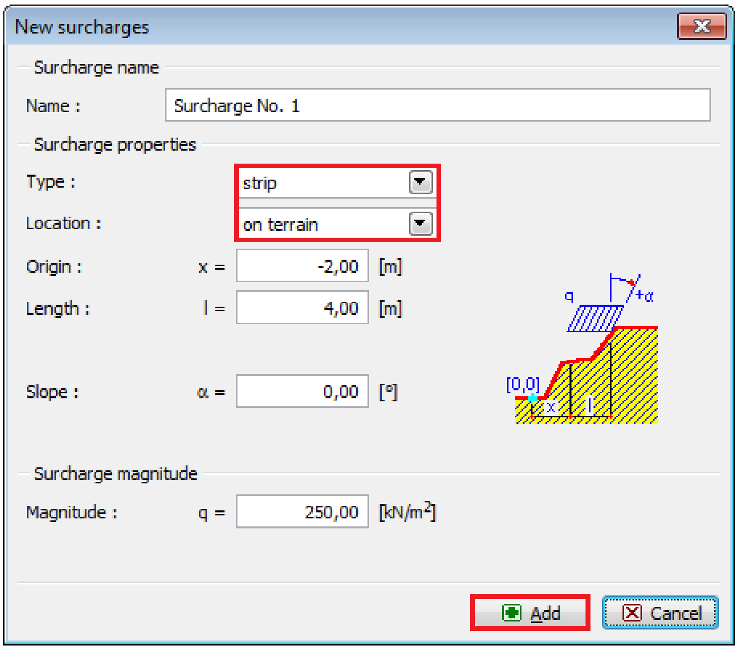
When the FE mesh generation is finished, we will switch to Phase 1 (using the tool bar on the left side of the screen) and then we will carry out the geostatic stress analysis by pushing the "**Analyse**" button. Subsequently we will examine the results for the geostatic stress $\sigma_{z,eff}$ $[kPa]$.



Construction stage 1 analysis – primary geostatic stress

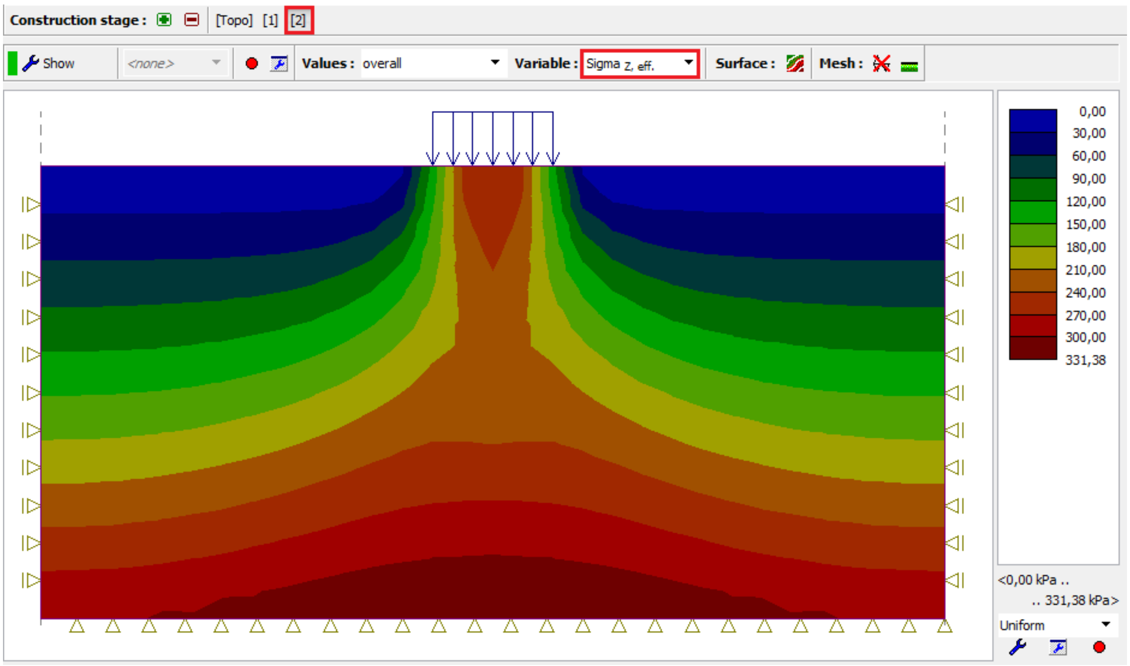
Construction stage 2: introduction of surcharge loading

In the next step we will add construction stage 2. Subsequently we will define the surcharge load acting on the terrain surface and will set the relevant characteristics. Then we will confirm everything by the **"Add"** button.



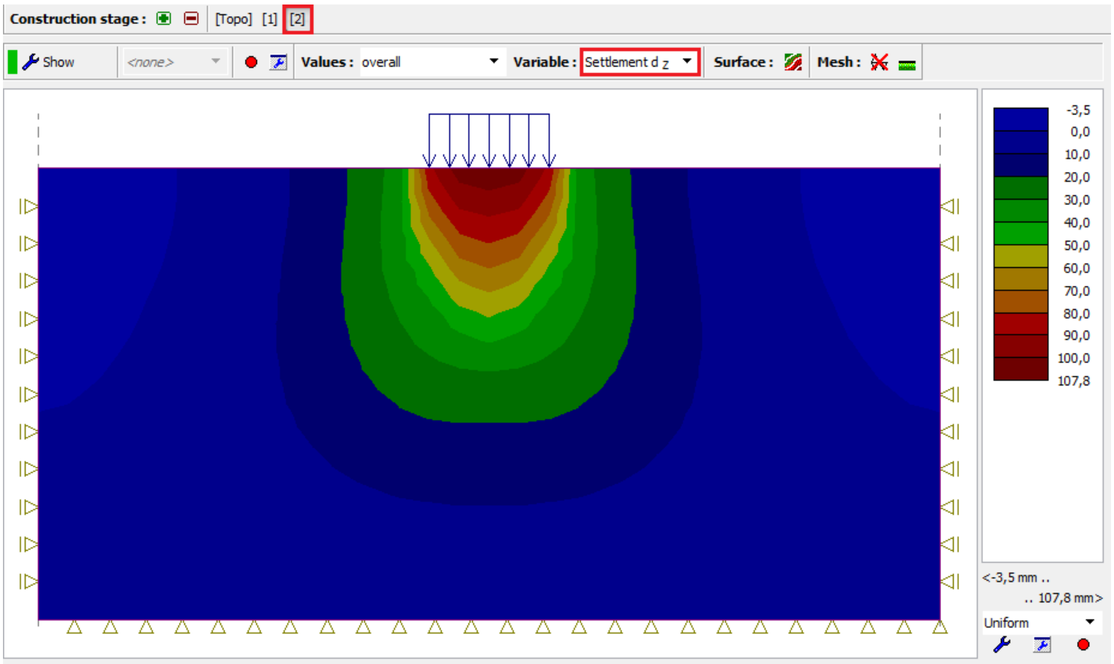
Setting new surcharges

At this construction stage we will again carry out the analysis and examine the results, first for the vertical normal stress $\sigma_{z,eff}$ $[kPa]$.



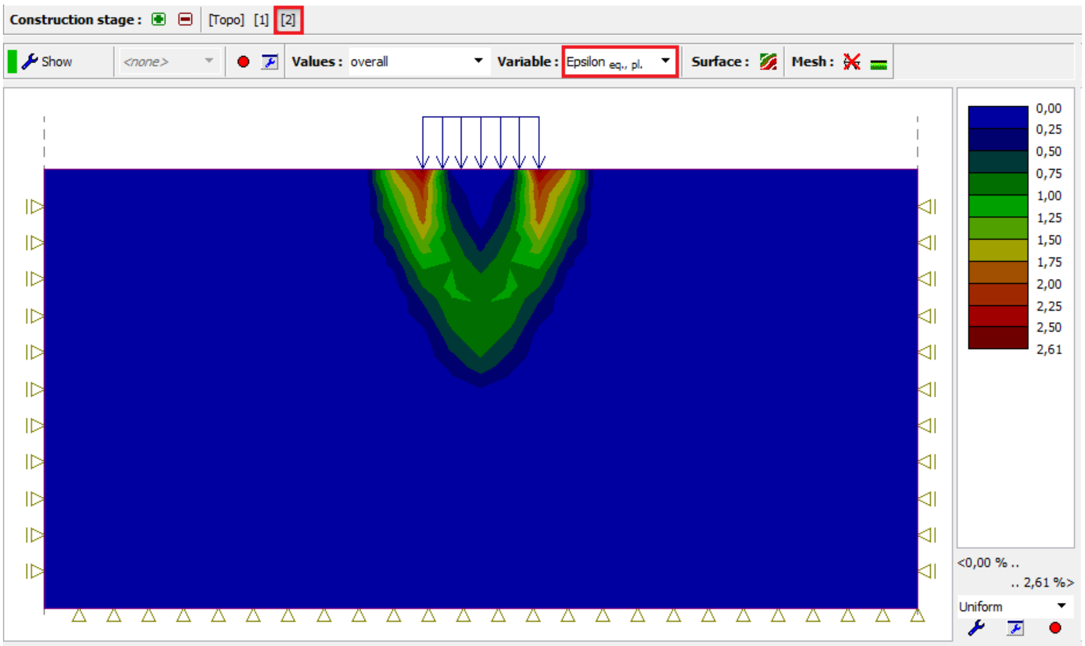
Construction stage 2 analysis – vertical normal stress $\sigma_{z,eff}$ $[kPa]$

Then we will switch to the visualisation for drawing the vertical settlement d_z $[mm]$. It follows from the drawing that the maximum vertical deformation amounts to 107.8 mm.



Construction stage 2 analysis – surcharge-induced vertical deformation d_z $[mm]$

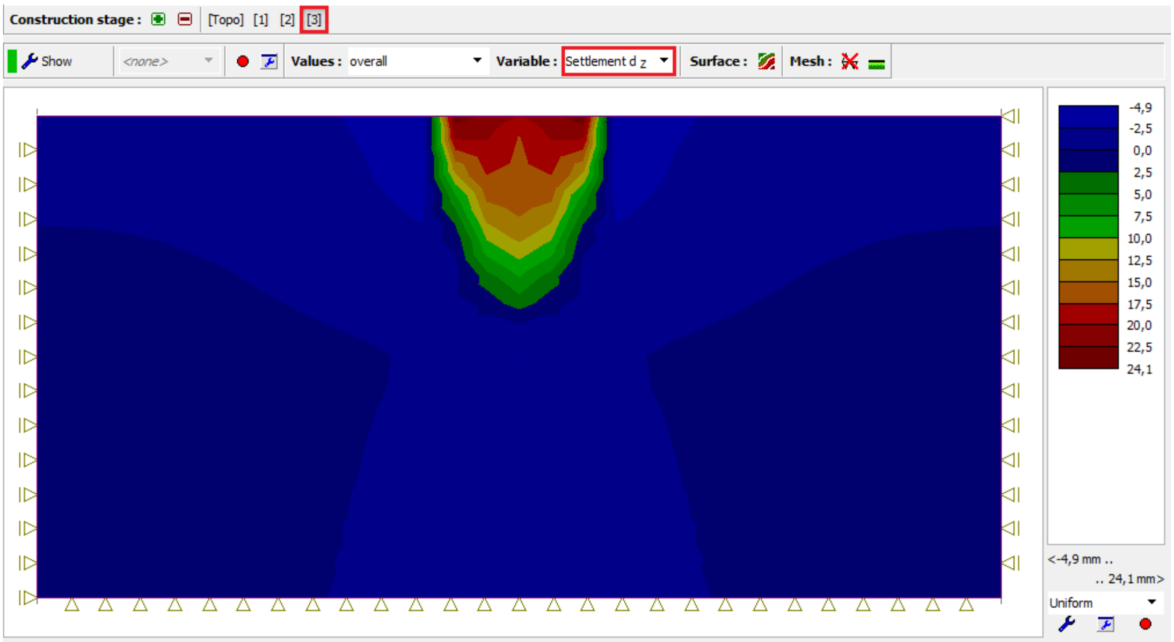
When a FE problem is being examined, an important output parameter is given by equivalent plastic strains (for non-linear models). They represent the locations where the yield condition was exceeded, i.e. the soil is in the state of plastic deformation, exhibiting permanent plastic strains.



Construction stage 2 analysis – equivalent plastic strain ratio $\varepsilon_{eq.,pl.}$ $[\%]$

Construction stage 3: terrain surface unloading

In the next step we will add construction stage 3. At this construction stage we do not consider the terrain surcharge. Then we will again carry out the analysis and will determine the values of stress and deformations. The total settlement after the terrain surface unloading amounts to 24.1 mm (for a triangular FE mesh).



Construction stage 3 analysis – vertical deformation induced by the surcharge $d_z\ [mm]$

This completes the basic analysis. We will carry out also other comparative analyses for another density of the mesh (with the length of the edges of elements of 1.5 m and 2.0 m) and the other material models.

Assessment of results:

The following table presents the results for the total settlement $d_z\ [mm]$ at the same example, but for different material models of GEO 5 – FEM program, with various lengths of the triangular element mesh edge.

Material model / program	Mesh spacing $[m]$	Stage 2 $d_z\ [mm]$	Stage 3 $d_z\ [mm]$	Note
Elastic	1.0	88.3	0	---
Elastic	1.5	88.4	0	---
Elastic	2.0	83.1	0	---
ELM	1.0	88.2	58.8	---
ELM	1.5	88.3	58.9	---
ELM	2.0	83.1	55.4	---
DP	1.0	123.0	36.9	---
DP	1.5	120.7	32.5	---
DP	2.0	120.5	42.2	---
MC	1.0	107.8	24.1	---
MC	1.5	106.9	19.7	---
MC	2.0	106.7	28.3	
MCM	1.0	97.5	11.1	---
MCM	1.5	96.3	7.8	---
MCM	2.0	96.0	16.4	
Settlement	---	153.6	---	CSN 73 1001

Total settlement results – summary

Note: For the purpose of the analytical solution in ***GEO 5 – Settlement*** *program we took into consideration the settlement analysis according to the oedometric modulus (in accord with CSN 73 1001 standard). We defined the modulus of deformation of soil as* $E_{def} = 5.0\ MPa$ *and the structural strength coefficient as* $m = 0.2$.

Conclusion:

It is possible to deduce several following conclusions from the summary table of the total settlement:

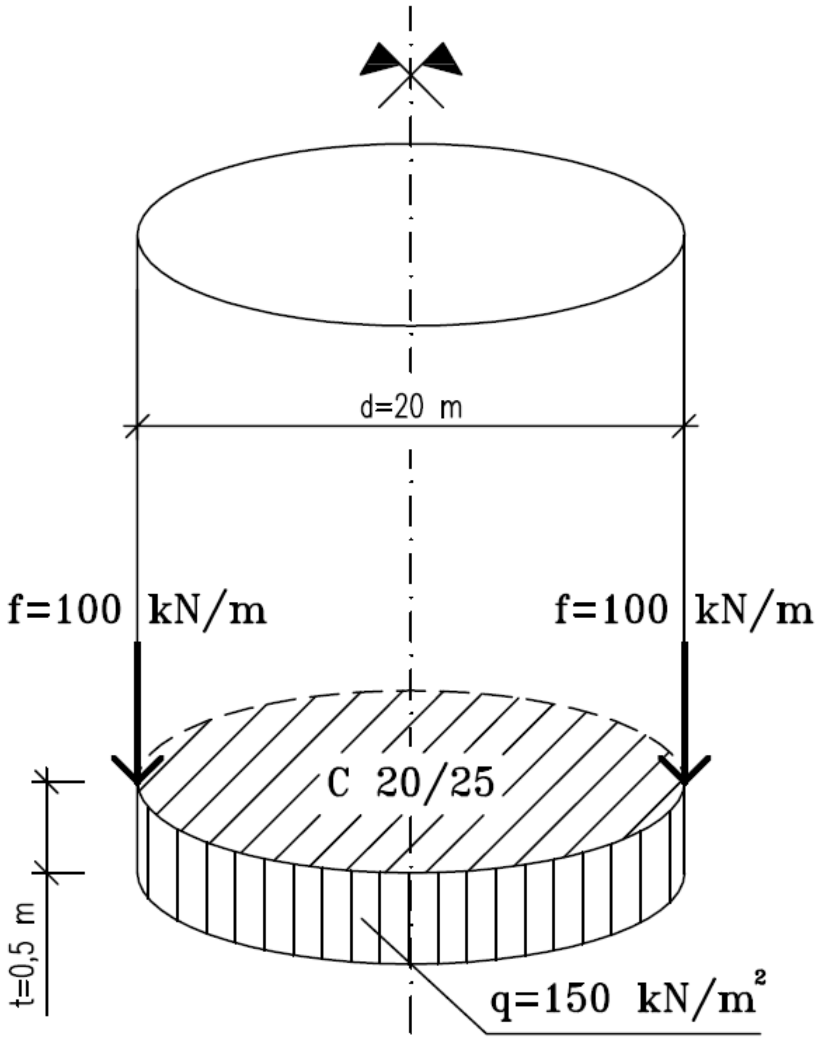
- A denser mesh leads to larger plastic strains and larger settlement values.
- Drucker-Prager model is in this particular case slightly more compliant than the classical Mohr-Coulomb model or the Modified Mohr-Coulomb material model.

Chapter 22. Settlement of a circular silo foundation

The objective of this manual is to describe the solution to a circular silo foundation settlement using the Finite Element Method and the Axial Symmetry module.

Task specification:

Determine the settlement of a circular silo foundation (thickness of 0.5 m and diameter of 20.0 m) induced by the complete filling of the silo, i.e. the surcharge $q = 150\,kPa$. Further determine the total settlement of the silo after subsequent emptying. The geological profile, including parameters of soils, is identical with the profile described in the previous task (chapter *21. Analysis of terrain settlement induced by strip surcharge loading*). Apply the **axial symmetry** to this particular case. The circular silo foundation is made from a matured reinforced concrete, class C 20/25.



Task specification chart – a circular silo foundation

In this case, the values of the total vertical deformation, i.e. settlement d_z $[mm]$, will be derived from the Mohr-Coulomb material model only. The comparison of the other material models with various mesh densities was carried out in the previous chapter 21. *Analysis of terrain settlement induced by strip surcharge loading*.

Solution:

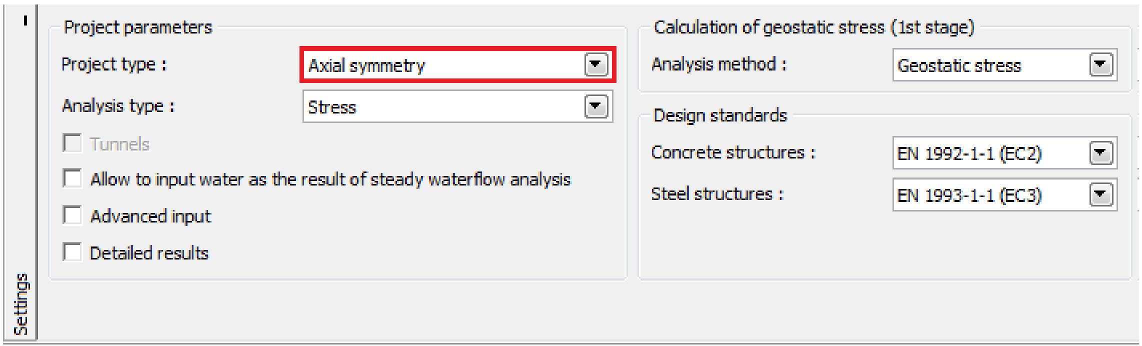
The analysis will be performed using the GEO 5 – FEM program. The following paragraphs provide the step by step description of the solution procedure:

- Topology: settings for and modelling of the problem (free points),
- Construction stage 1: primary geostatic stress,
- Construction stage 2: modelling of and loading on beam elements, settlement analysis,
- Construction stage 3: unloaded terrain surface settlement (deformation) analysis, internal forces,
- Assessment of results: comparison, conclusion.

Note: To solve this task we shall represent the silo foundation made of reinforced concrete by beam elements free of contact elements thus assuming a perfect bond between the foundation and the soil. The issue of contact elements will be analysed in more detail in Chapter 24. Numerical solution to a sheeting wall structure.

Topology: problem settings

We will select the "Axial symmetry" option in the Project type of the "Settings" frame. The other data will remain the same.

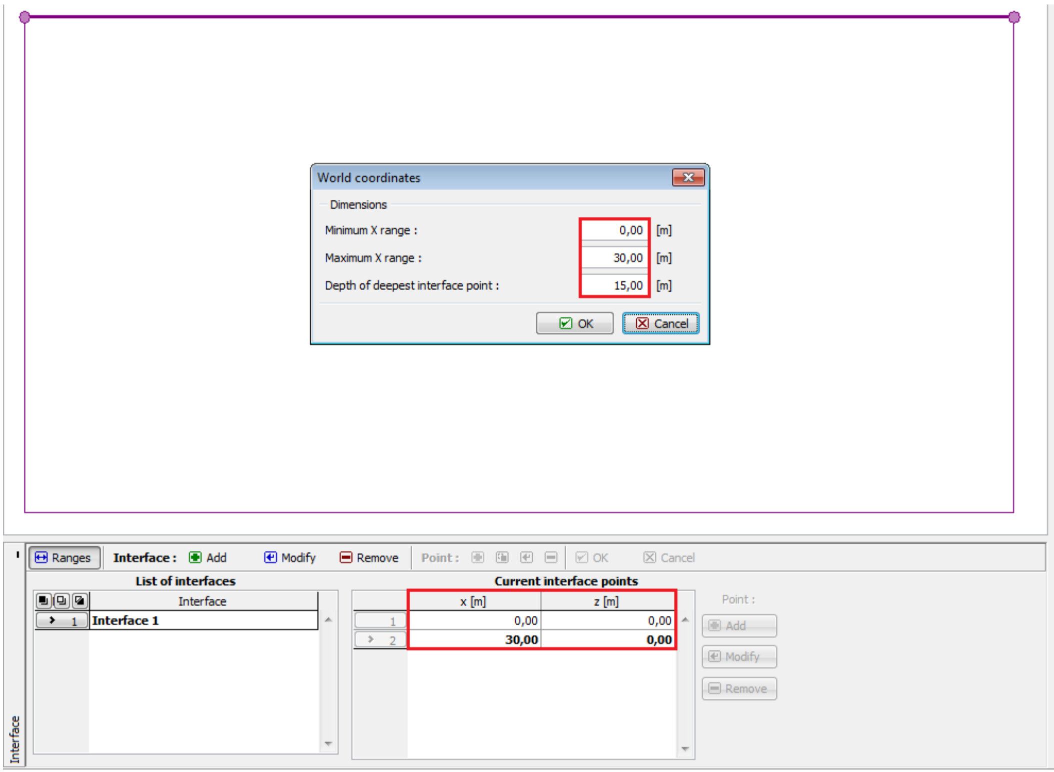


"Settings" frame

Note: ***Axial symmetry*** *is suitable for solving rotationally symmetric problems. This assumption must comply both with the geometrical arrangement of the structure and the loading. The solution to this problem – a circular silo foundation – is therefore a suitable example.*

The solution is related to 1rad of the $x(r)$-radius arch. The axis of symmetry always represents the $x(r)$ coordinate origin. Shear components of deformation in the direction of rotation can be disregarded. The evolution of a circumferential normal component of stress and strain (hoop stress and strain) is also taken into consideration, in addition to the components of stress and strain in the cross-sectional plane (for more details visit Help – F1).

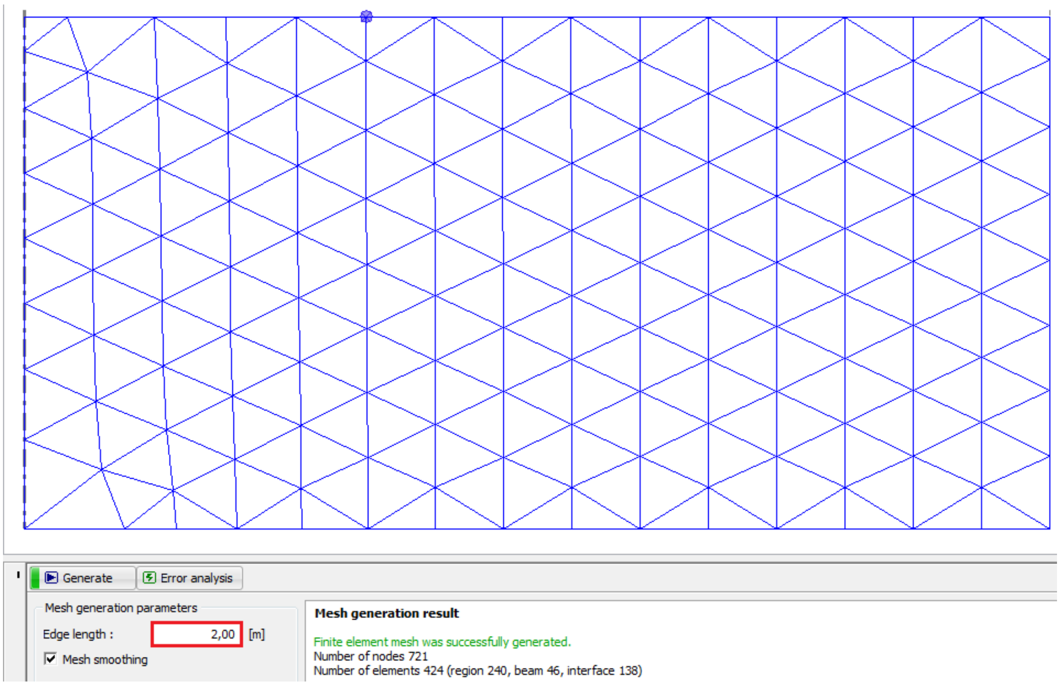
In the "Interface" frame, we will first set the new world coordinates. Then we will set the coordinates of the first point of the interface to $[0{,}0]$. The next point of the interface (at the edge) will be added by the program automatically.



"Interface" frame + "World coordinates" dialogue window

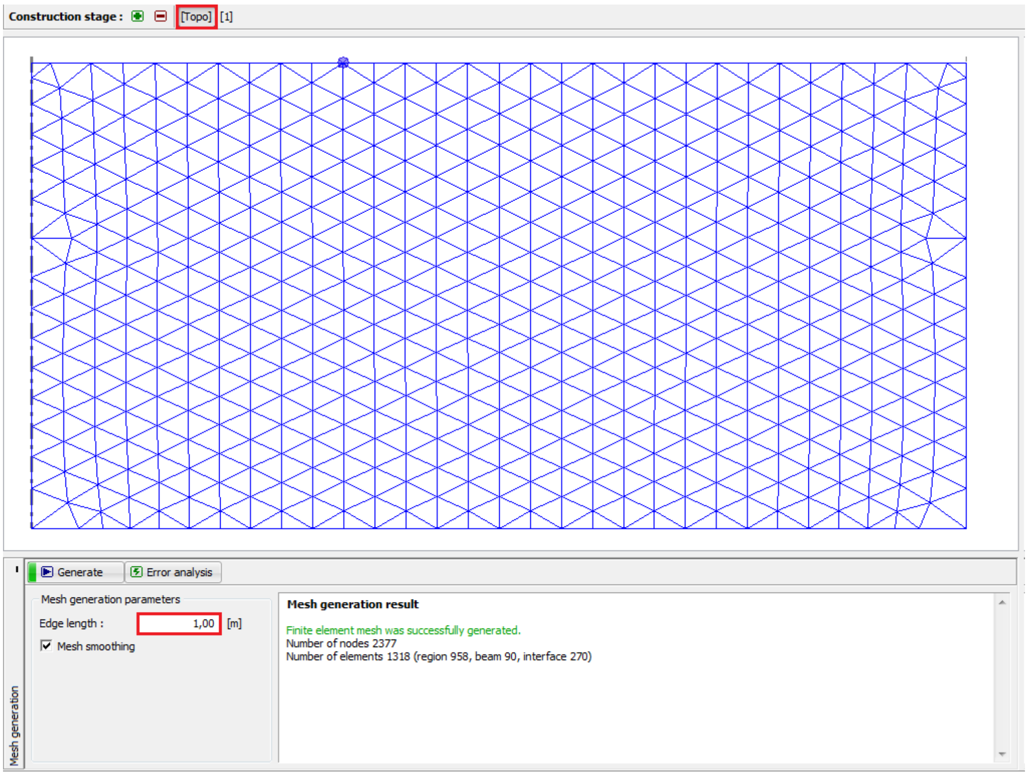
Then we will define the parameters of soils and assign them to the interface region No. 1. In this particular case neither rigid bodies nor contact types will be considered.

For the mesh generation, we will first set the elements edge length equal to 2.0 m.



"Mesh generation" frame – Triangular mesh with the element edge length of 2.0 m

Upon examining the generated mesh we may conclude that for a given problem the resulting mesh is too coarse. For that reason we will change the length of the edges of mesh elements to 1.0 m.

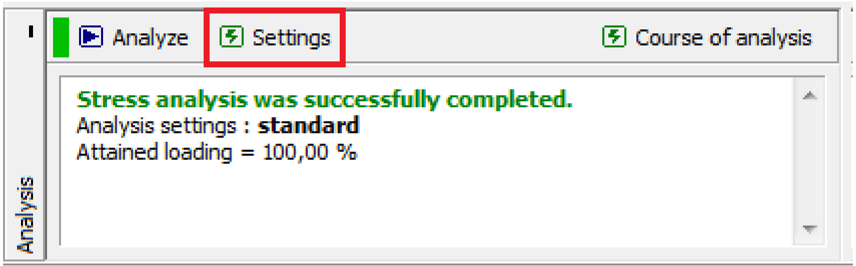


"Mesh generation" frame – Triangular mesh with the element edge length of 1.0 m

Note: It would be reasonable for the area under the circular silo foundation being solved to refine the density of mesh using line refinement (for more details visit Help – F1). We will describe this function in more detail in the following chapter 23. Collector lining analysis.

Construction stage 1: primary geostatic stress

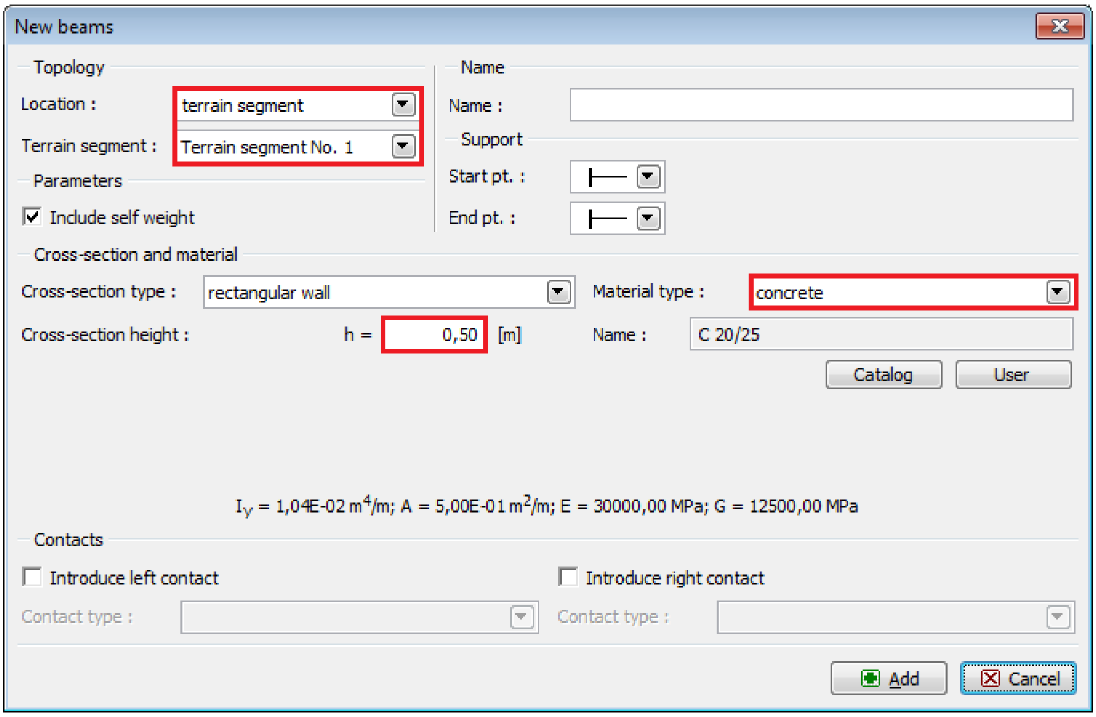
After generating the FE mesh, we will switch over to construction stage 1 and will carry out the primary geostatic stress analysis. We will leave the analysis settings as "Standard" (for more details visit Help – F1).



"Analysis" frame – Construction stage 1

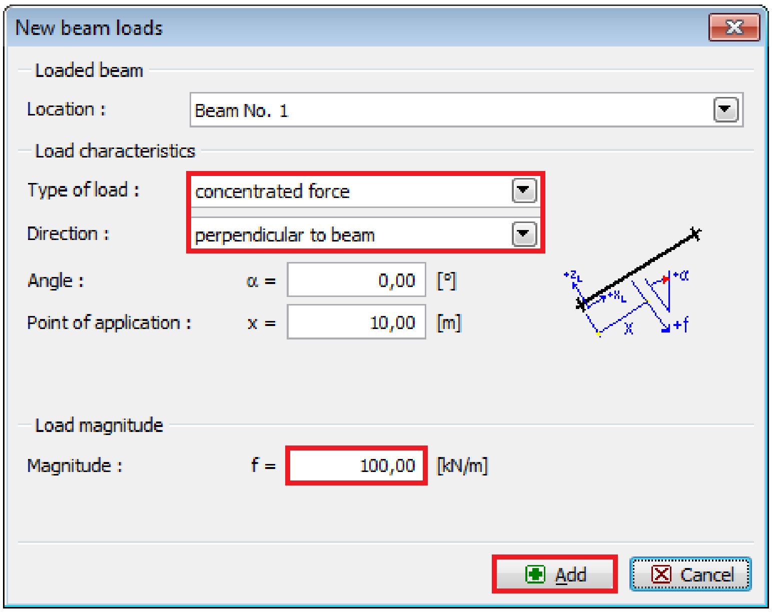
Construction stage 2: modelling of and loading on beam elements

In the next step we will add construction stage 2. Then, in the "Beams" frame, we will define the following parameters: beam location, material and concrete class, cross-section height (0.5m) and beam ends supports (for more details visit Help – F1).



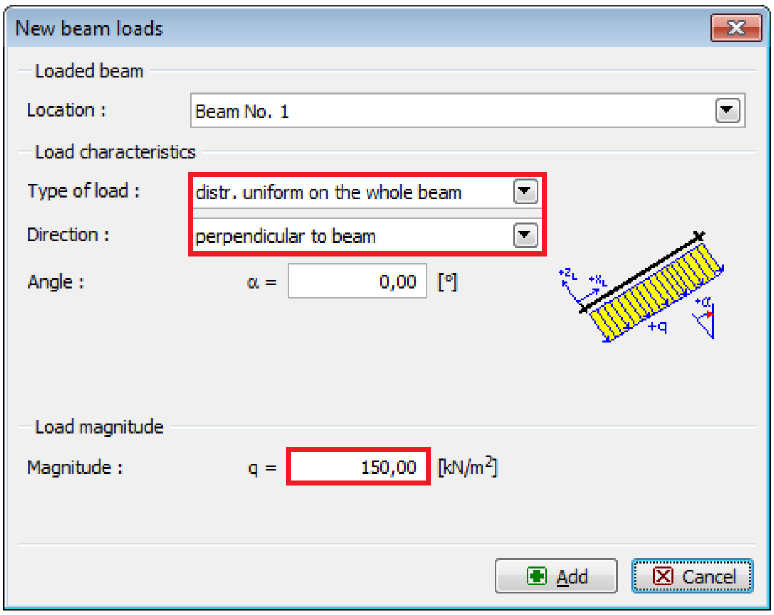
"New beams" dialogue window – Construction stage 2

Subsequently we will proceed to the "Beam loads" frame, where we will set the load magnitude $f = 100\ kN/m$; we will consider it as the weight of the circular silo walls acting on the foundation.

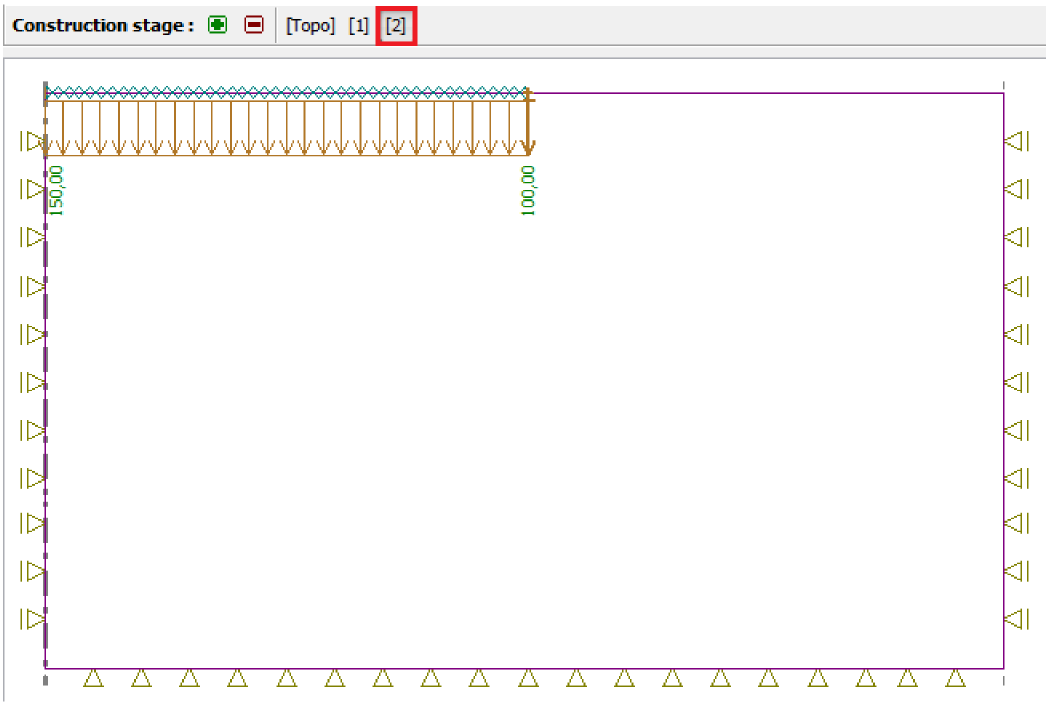


"New beam loads" dialogue window – loads induced by walls, acting on the circular foundation

Further we will set the value of the uniform continuous load to $q = 150\ kN/m^2$, representing the filling of the circular silo, acting on its bottom or the upper edge of the foundation.

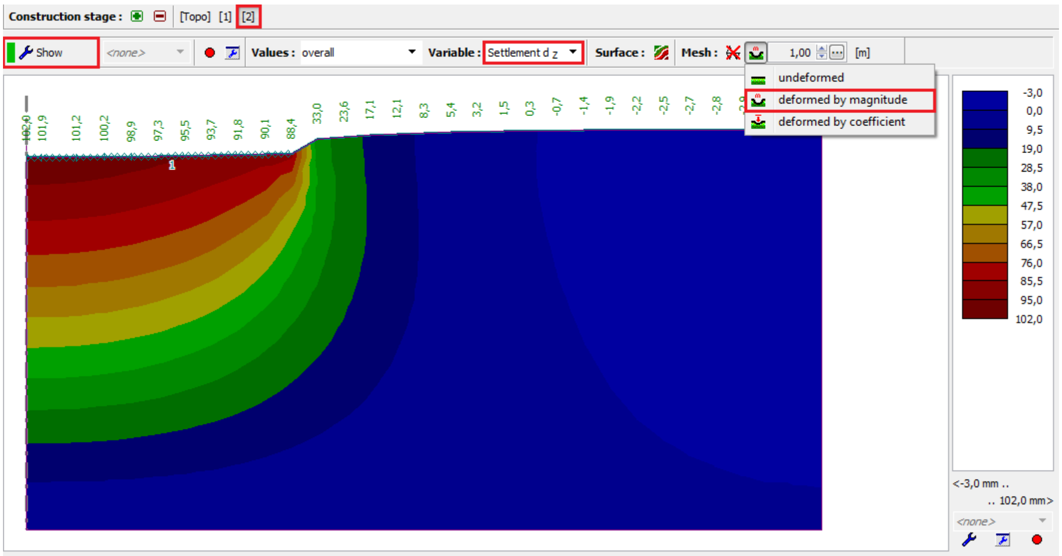


"New beam loads" dialogue window – load on the circular foundation induced by filling the silo



"Beams loads" frame – Construction stage 2

At this stage, we will again carry out the analysis and will examine the results for settlement d_z $[mm]$. It follows from the picture that the maximum vertical displacement is 102.0 mm. To better understand the structure behaviour we will visualise the deformed mesh (the button in the upper part of the screen).

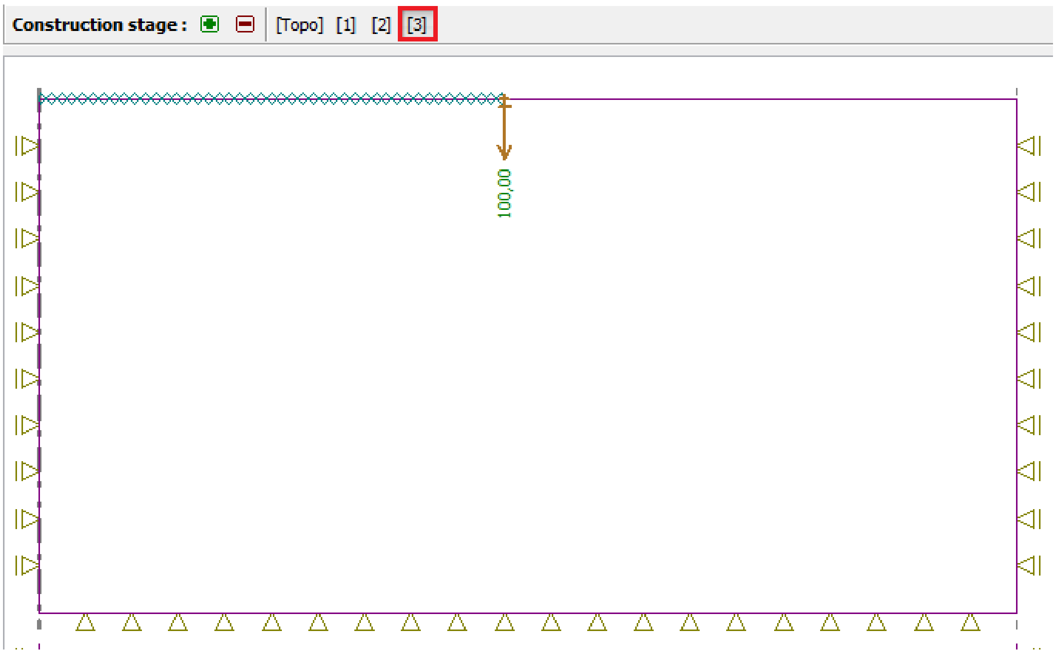


"Analysis" frame – Construction stage 2 (vertical deformation d_z and the depression)

We will click on the "Show" button and check options "Plot" and "Values" in the "Depression" flag (for more details visit Help – F1).

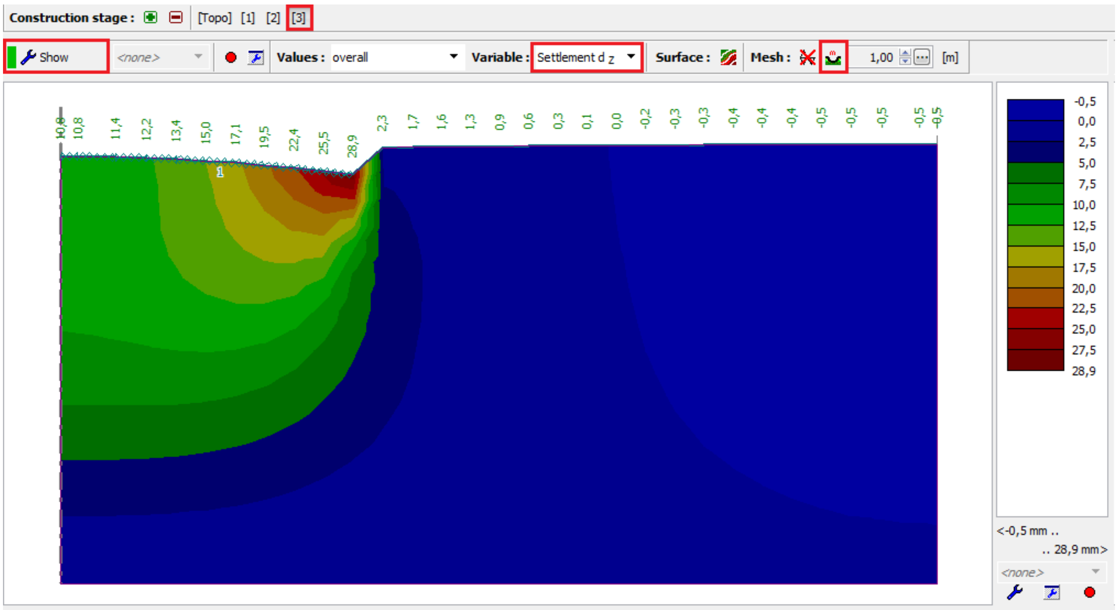
Construction stage 3: unloaded terrain surface settlement, internal forces

As a next step we will add construction stage 3. At this construction stage we will remove the uniform continuous load. Further on we will take into consideration only the beam loading induced by the circular silo walls, which is identical with that determined in the previous construction stage, i.e. $f = 100\ kN/m$.



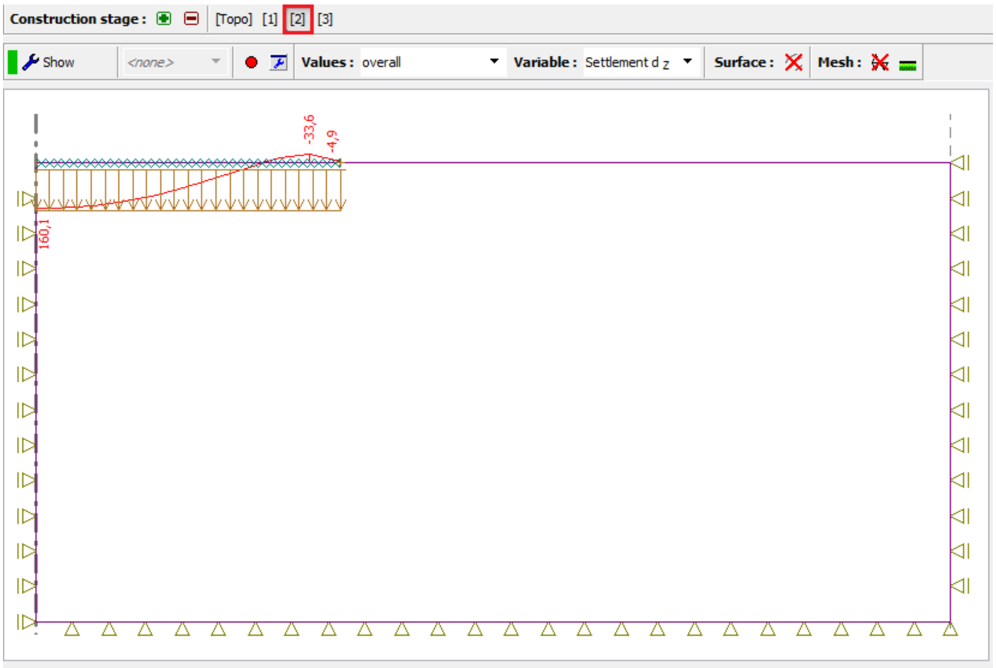
„Beam loads" frame – Construction stage 3

Then we will repeat the analysis and will determine the values of displacements. The total settlement value after the terrain surface unloading d_z is 28.9 mm.

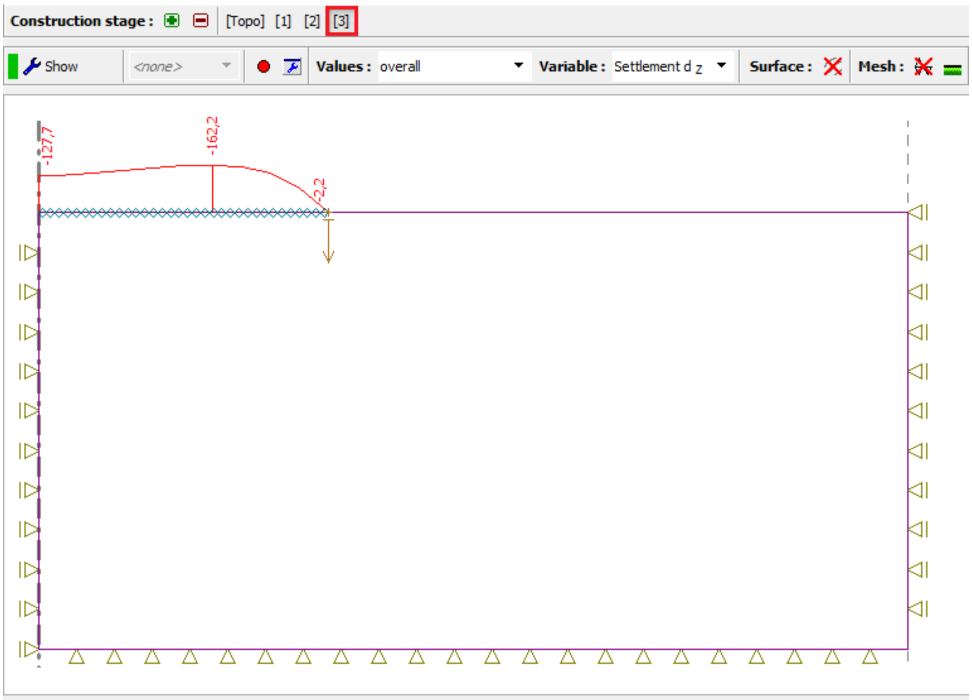


"Analysis" frame – Construction stage 3 (vertical deformation d_z with the settlement trough)

Now we will examine the radial moment diagrams $M_r\ [kNm/m]$ for construction stages 2 or 3 (using the "Show" button in the "Distributions" flag) and will record the magnitude of local extremes in a table. The main structural reinforcement of the circular silo foundation can be designed and assessed for these values in an arbitrary static program (e.g. FIN EC – CONCRETE 2D).



"Analysis" frame – Construction stage 2 (variation of radial moment M_r)



"Analysis" frame – Construction stage 3 (variation of radial moment M_r)

Assessment of results:

The following table presents the results for the total settlement d_z $[mm]$ and radial moments M_r $[kNm/m]$ for construction stages 2 and 3, in which we modelled the loading or unloading of the circular silo foundation. We carried out the analysis for the Mohr-Coulomb material model with the triangular element mesh having edge length equal to 1.0 m.

Material Model	**Stage 2** d_z $[mm]$	**Stage 3** d_z $[mm]$	**Stage 2** M_r $[kNm/m]$	**Stage 3** M_r $[kNm/m]$
Mohr-Coulomb (1.0 m)	102.0	28.9	+ 160.1	---
			– 33.6	– 162.2

Results for total settlement d_z and radial moments M_r for individual construction stages

Conclusion:

Several following conclusions can be drawn from the results for the quantities being examined:

- When the silo is full (as a result of the action of the uniform continuous load), a positive bending moment prevails along the beam length, where its bottom fibres are stretched.
- After emptying the silo (as a result of the subsequent unloading), the circular foundation is loaded only by the silo walls. A negative bending moment prevails along the beam length, where its upper fibres are stretched.

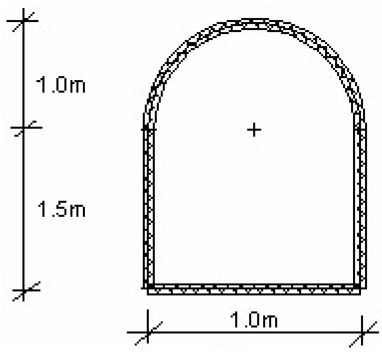
Chapter 23. Collector lining analysis

The objective of this example is to perform the analysis of a mined collector with emphases on the lining response using the Finite Element Method.

Task specification:

Determine the lining response of a mined collector; its dimensions can be seen in the following chart. Determine internal forces acting on the collector lining. The collector lining (0.1 m thick) is made from reinforced concrete, class C 20/25, the bottom is at the depth of 12.0 m. The geological profile is homogeneous; the soil parameters are as follows:

- Unit weight of soil: $\gamma = 20.0\, kN/m^3$
- Modulus of elasticity: $E = 12.0\ MPa$
- Poisson's ratio: $\nu = 0.40$
- Effective soil cohesion: $c_{ef} = 12.0\ kPa$
- Effective angle of internal friction: $\varphi_{ef} = 21.0\ °$
- Unit weight of saturated soil: $\gamma_{sat} = 22.0\, kN/m^3$



Task specification chart – mined collector

We will determine the values of displacements and internal forces only for the elastic model because we do not expect the development of plastic deformations. We will subsequently use the Mohr-Coulomb material model for the verification of yield condition.

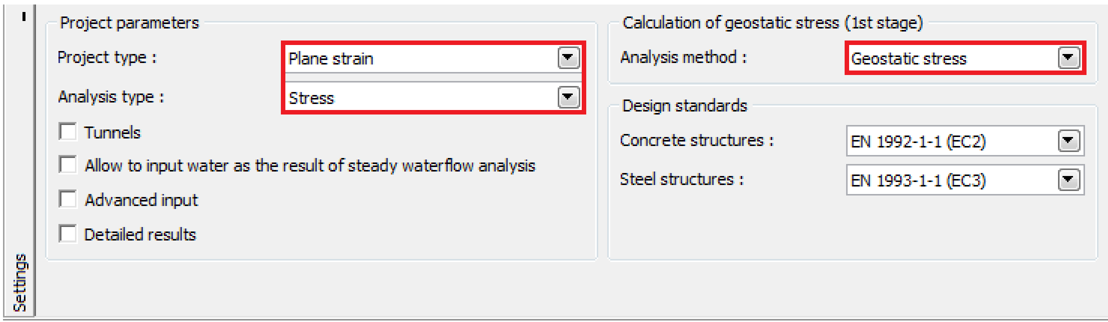
Solution:

To analyse this task we adopt the GEO 5 – FEM program. The following paragraphs provide the step by step description of the solution procedure:

- Topology: settings for and modelling of the problem (interface, free points and lines – refining the density)
- Construction stage 1: primary geostatic stress,
- Construction stage 2: modelling beam elements, analysing displacements, internal forces
- Assessment of results: comparison, conclusion.

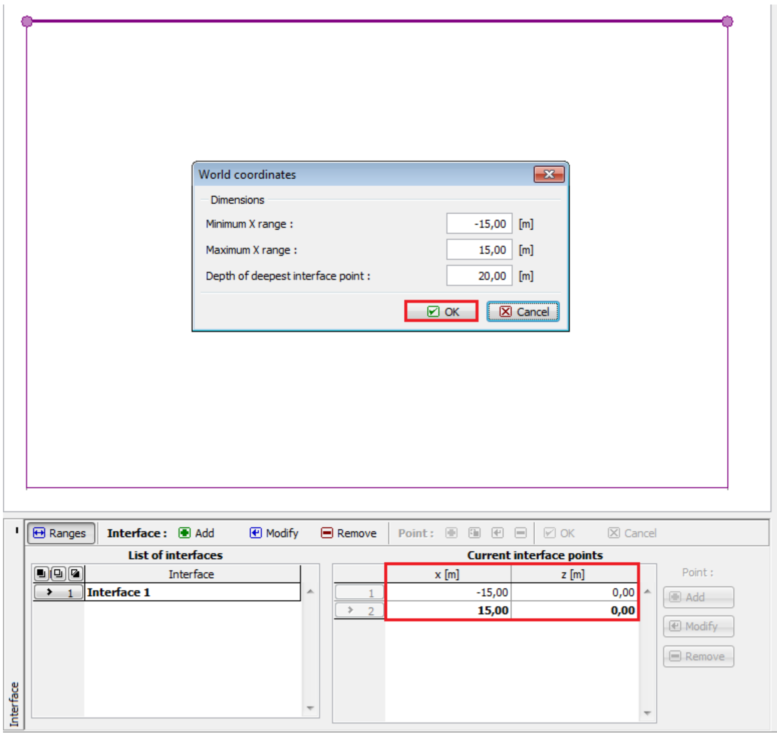
Topology: problem settings

In the "Settings" frame we select the option geostatic stress to perform the analysis of the construction stage 1. We will consider the problem or the analysis type as *plane strain*.



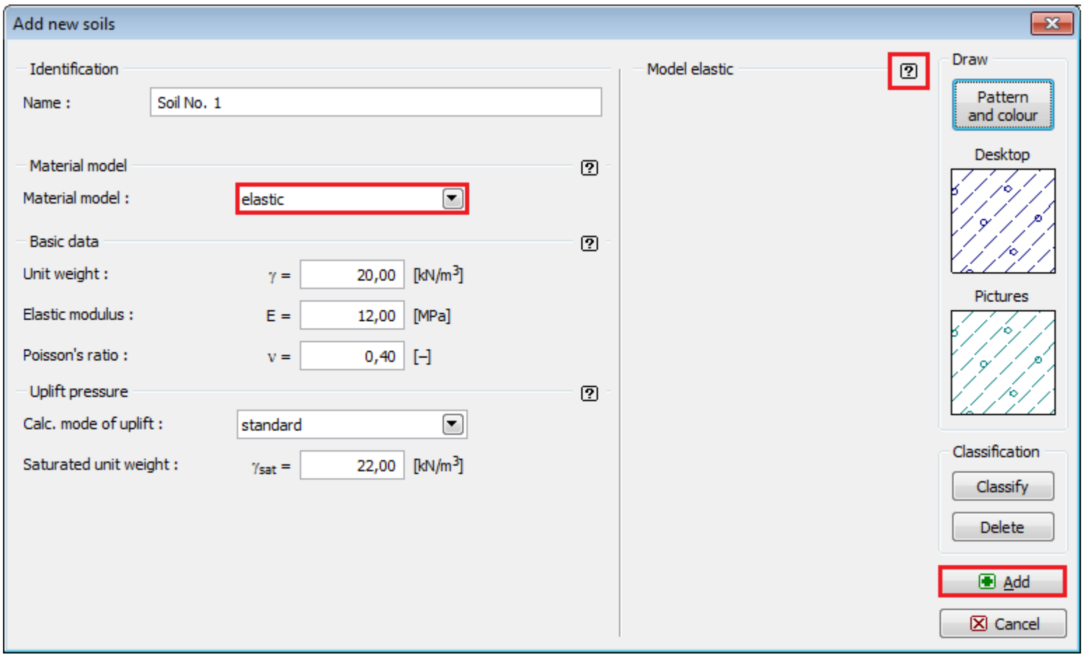
"Settings" frame

Further we will set the world coordinates and the terrain interface. We will choose sufficiently large world dimensions so that the results are not affected by the conditions at the edge. For our particular problem we will choose the model dimensions $\langle -15\ m; 15\ m \rangle$; we will set the depth of 20.0 m for the layer to be examined.



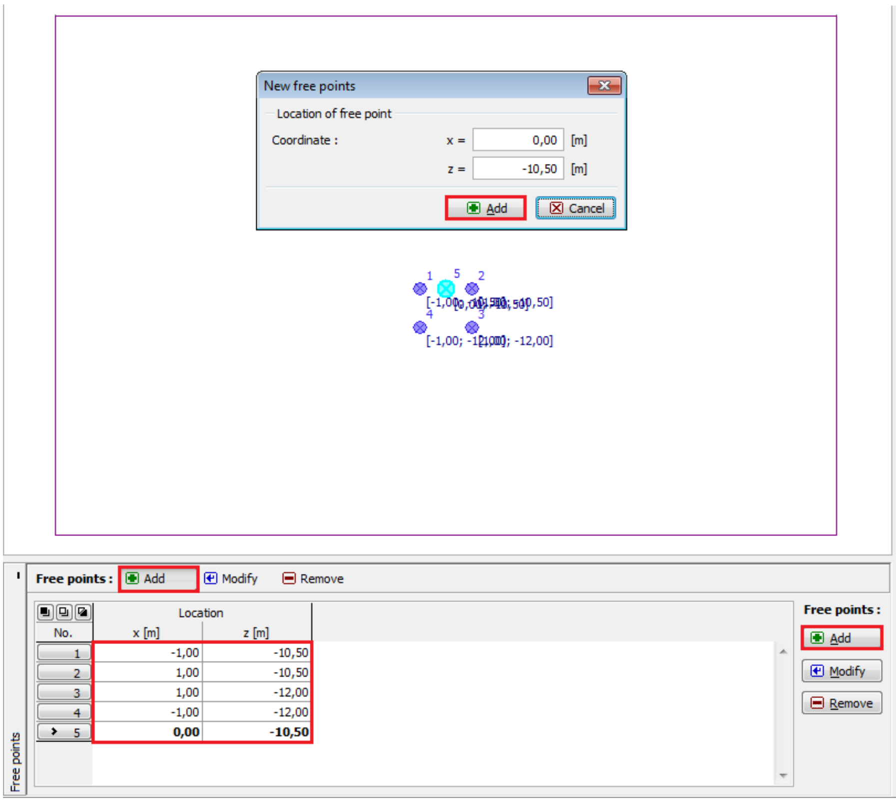
"Interface" frame + "World coordinates" dialogue window

Now we will specify respective soil parameters including the material model and subsequently we will assign the soil to the created region (for more details visit Help – F1).



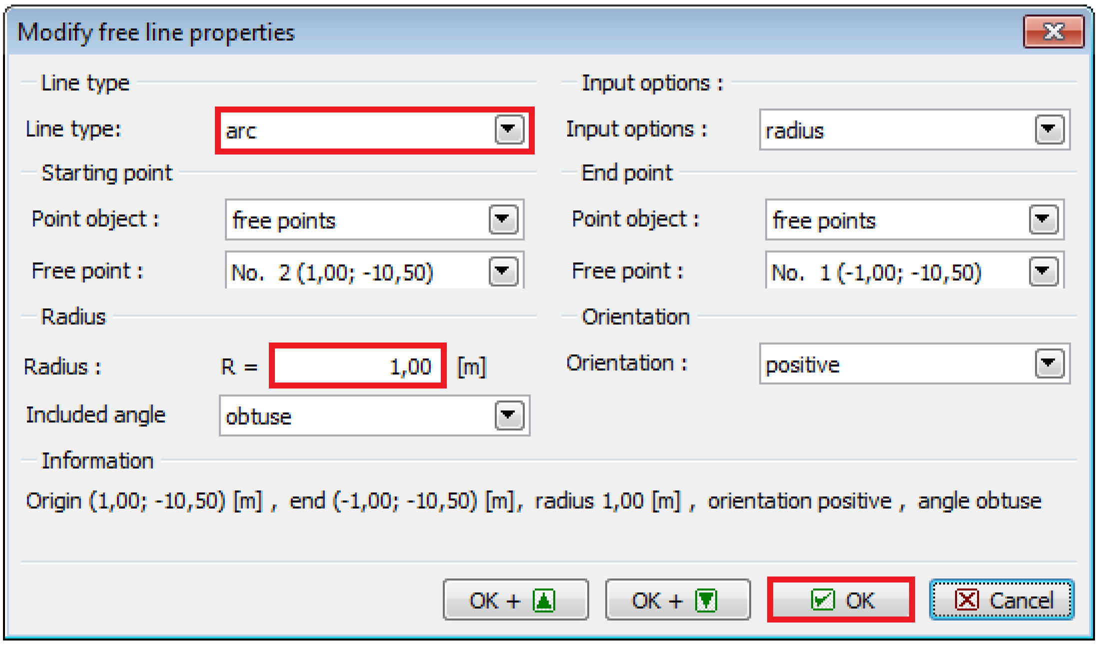
"Add new soils" dialogue window

The next step is to set the structure geometry. First we will define coordinates of free points ("Add" button), forming the collector corners (for more details visit Help – F1).



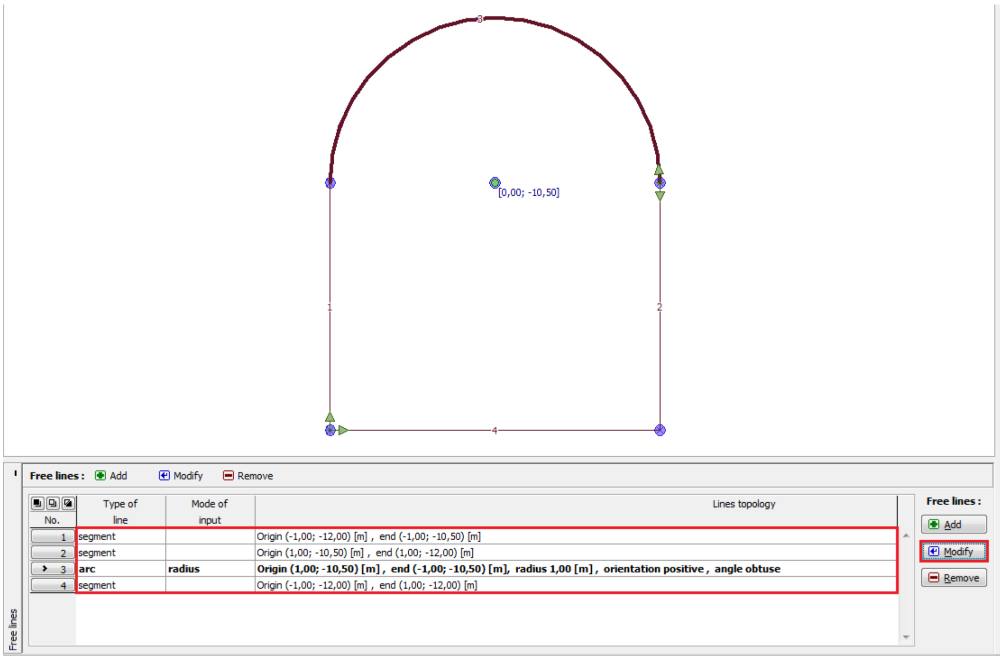
"Free points" frame + "New free points" dialogue window

Subsequently we will click on "Add" button in "Free lines" frame and will connect the given points on the screen by respective lines by means of the cursor (for more details visit Help – F1). To set the arc with the radius $R = 1.0\ m$, we must change the line type (using the "Edit" button).



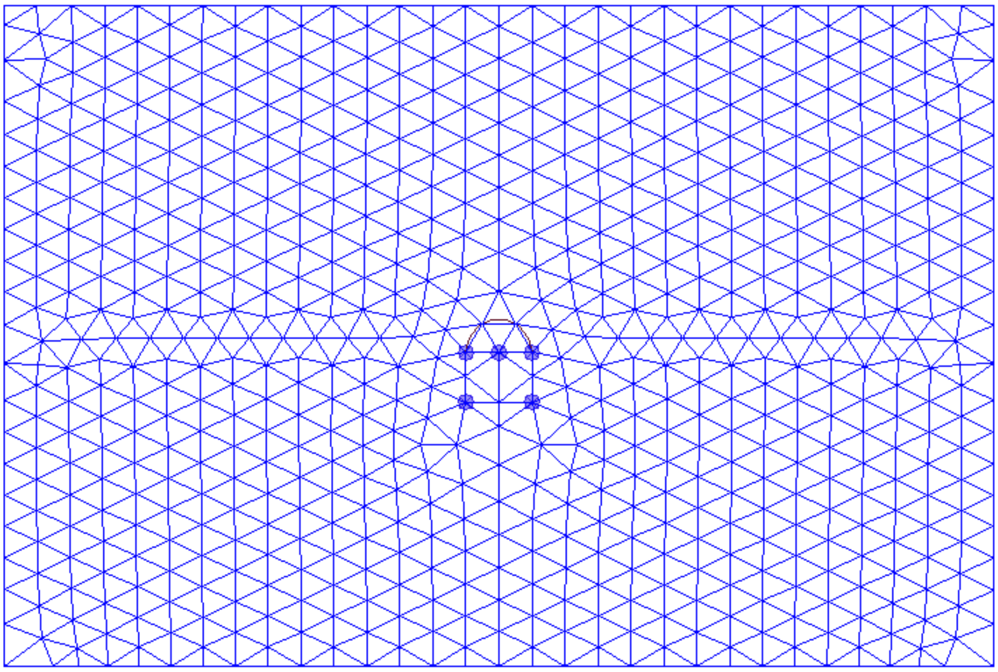
"Modify free line properties" dialogue window

We will examine the resultant structure outline. Through this step, the collector geometry setting will be finished and we will go over to the FE mesh generation (for more details visit Help – F1).



"Free lines" frame

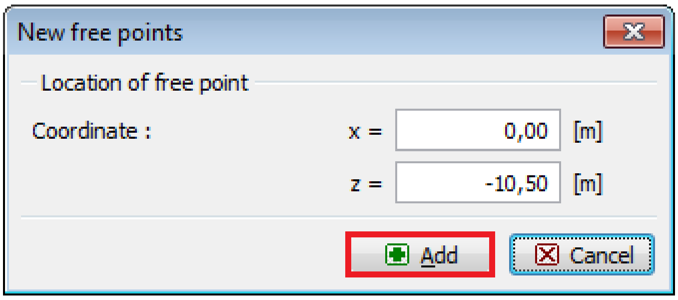
For the mesh generation parameters we will choose the element edge length of 1.0 m and push the "**Generate**" button. The program will automatically generate and smooth the FE mesh.



"Mesh generation" frame – element edge length of 1.0 m (without local refinement)

It is obvious at first glance that the generated mesh in the collector surroundings is very coarse. We will therefore increase the density. We can refine the mesh density either around lines or around free points. The following procedure is suitable for the refining of density around the collector lining (in general the excavation):

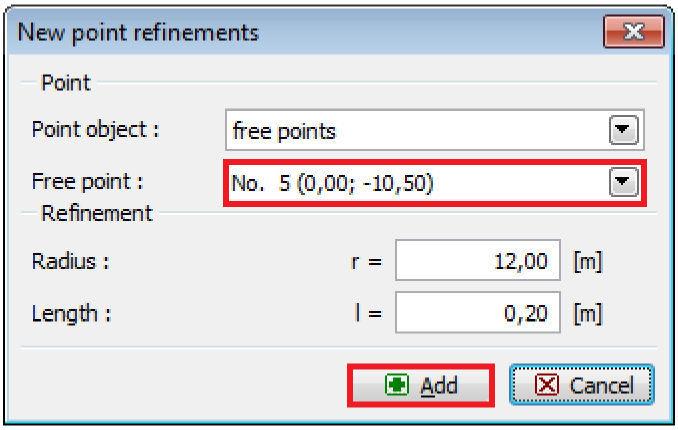
– we will specify a free point in the vicinity of the excavation centre,
– we will refine the density around this point.



"New free points" dialogue window

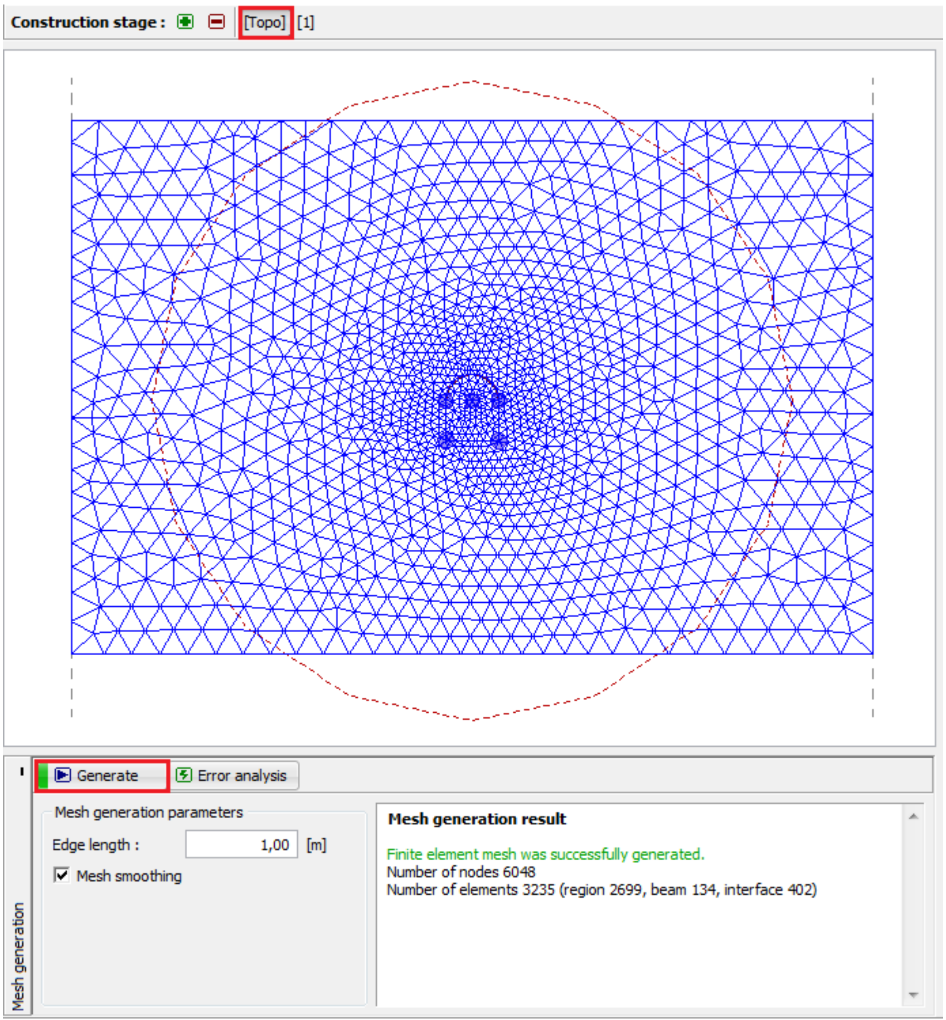
Note: Internal forces in beams are analysed at individual points of the mesh and it is therefore necessary to sufficiently refine the free lines and points of the FE mesh (for more details visit Help – F1).

To refine the finite element mesh density, we will specify the respective radius $r = 12.0\ m$ and the element edge length $l = 0.2\ m$. Then we will return to "Mesh generation" frame and generate the FE mesh again.



"New point refinements" dialogue window

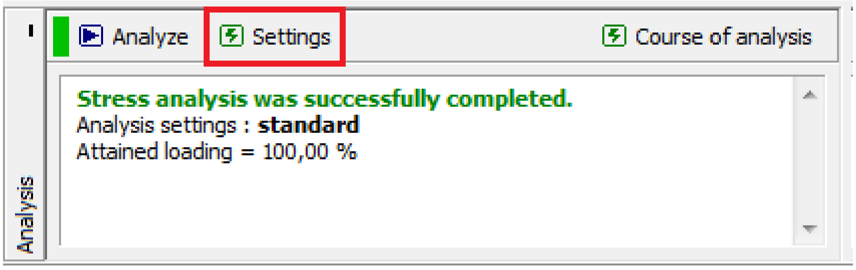
Note: The elements mesh should be sufficiently dense especially in the region where large stress gradients can be expected (point support, sharp corners, underground excavations etc.). It is necessary for this density-refining radius to be at least 3 to 5 times larger than the density in the density refining centre and both values (the density and radius) at the points to be in a reasonable proportion to the mesh density prescribed for the surrounding region. In this way smooth transition between the regions with different density will be secured (for more details visit Help – F1).



"Mesh generation" frame – FE edge length of 1.0 m (with locally increased mesh density in the collector surrounding)

Construction stage 1: primary geostatic stress

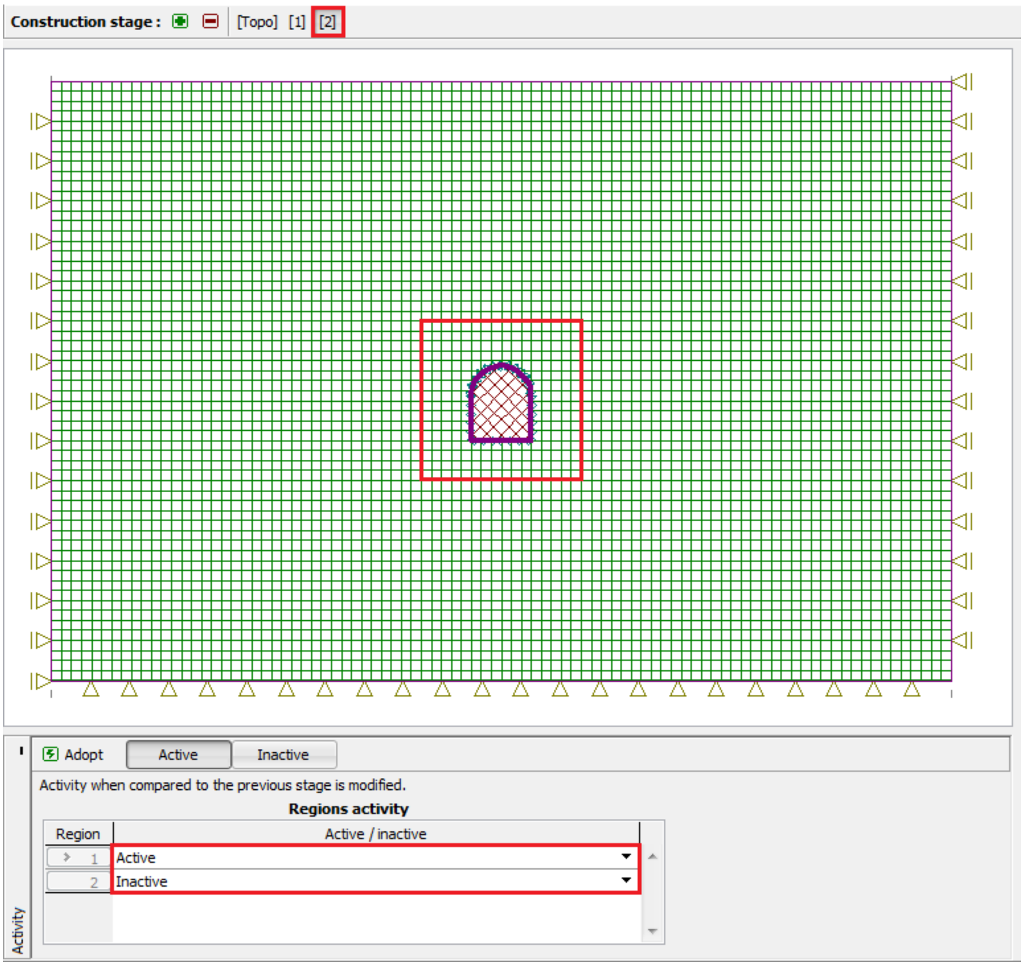
After subsequent generation, the mesh in the collector vicinity looks significantly better. Now we will go over to construction stage 1 and will carry out the analysis of the primary geostatic stress. We will maintain the "Standard" analysis setting (for more details visit Help – F1).



"Analysis" frame – Construction stage 1

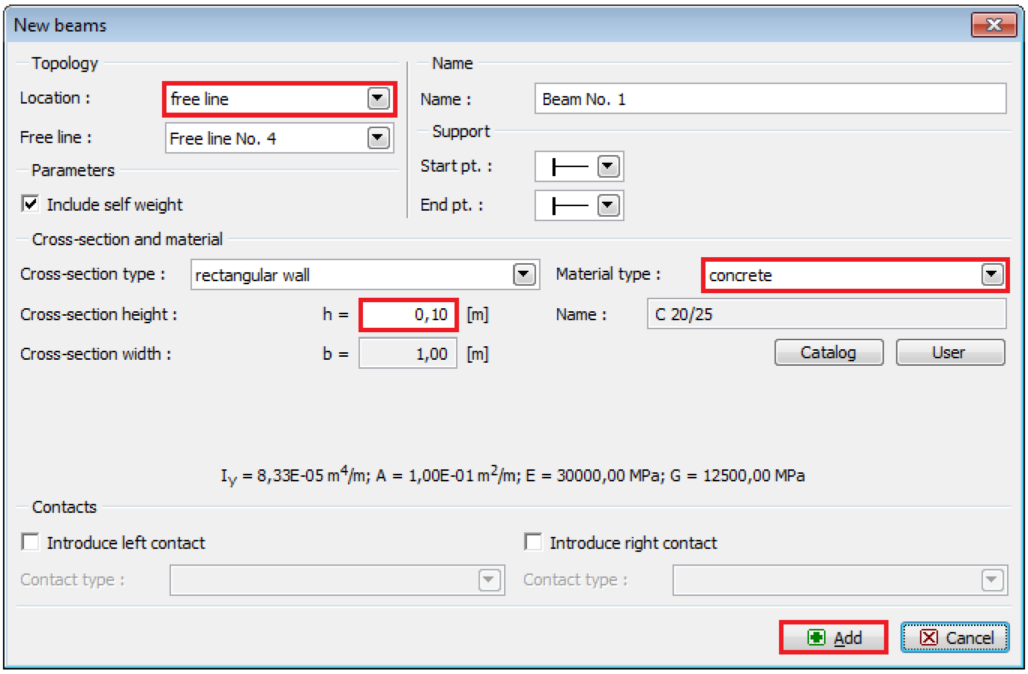
Construction stage 2: modelling of beam elements

In the "Activity" frame, we will first model the excavation of soil from the collector cross-section – we will set the particular region as inactive (for more details visit Help – F1).



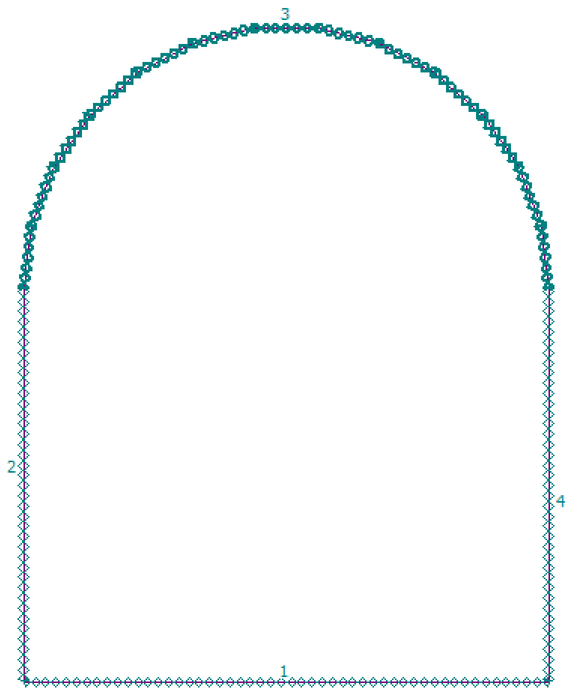
"Activity" frame – Construction stage 2

Then we will go over to "Beams" frame and will model the mined collector lining. We will define the following parameters – the beam location (we take all free lines into consideration), material and concrete class, cross-section height (0.1 m) and the supports of beam ends (for more details visit Help – F1).



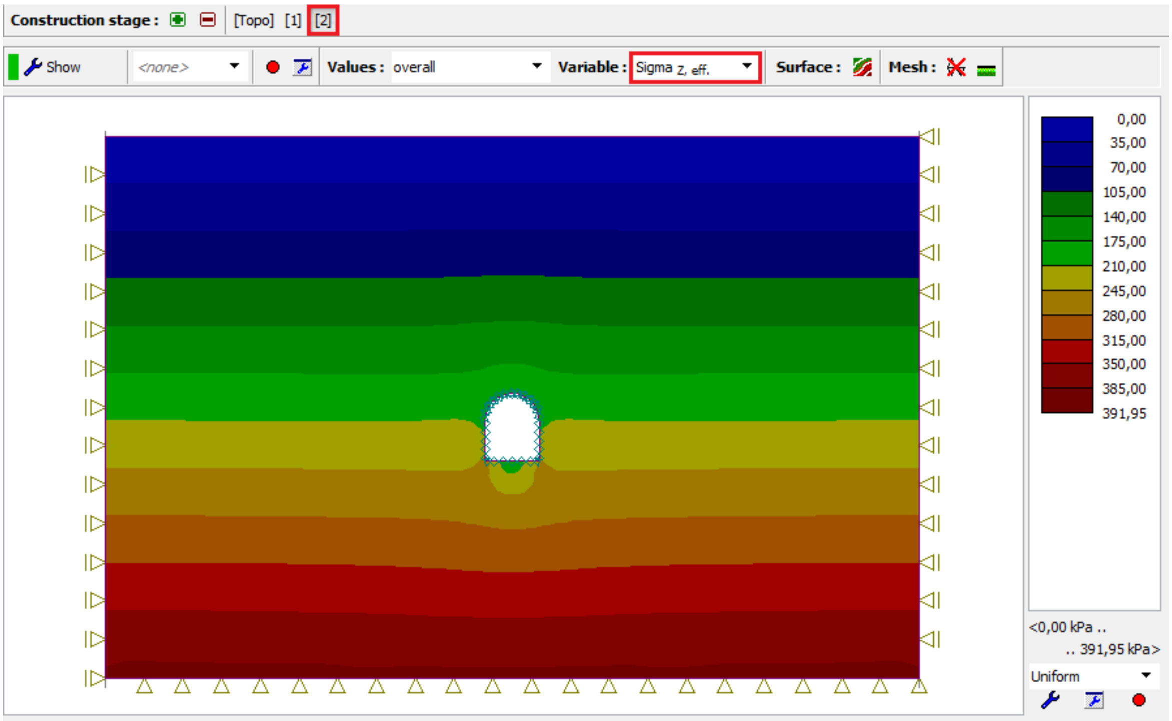
"New beams" dialogue window – Construction stage 2

All beam elements of the collector are drawn in the diagram below.



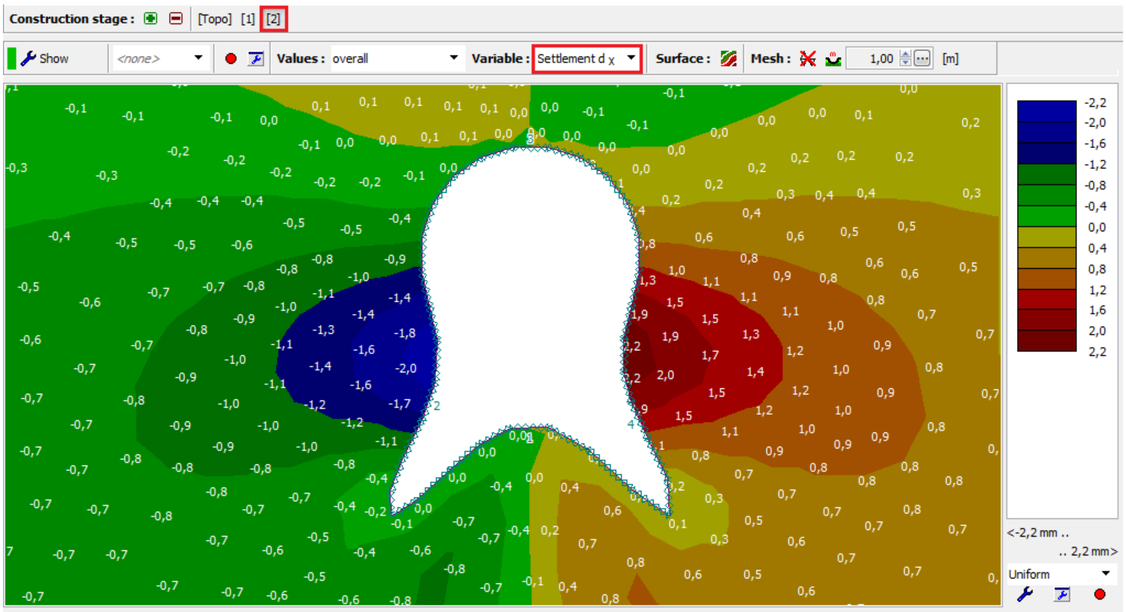
"Beams" frame – Construction stage 2

Now we will carry out the analysis and will visualise the results for the vertical geostatic stress $\sigma_{z,ef}$ $[kPa]$, lateral displacement d_x $[mm]$ and internal forces in the collector lining.



"Analysis" frame – Construction stage 2 (vertical geostatic stress $\sigma_{z,ef}$)

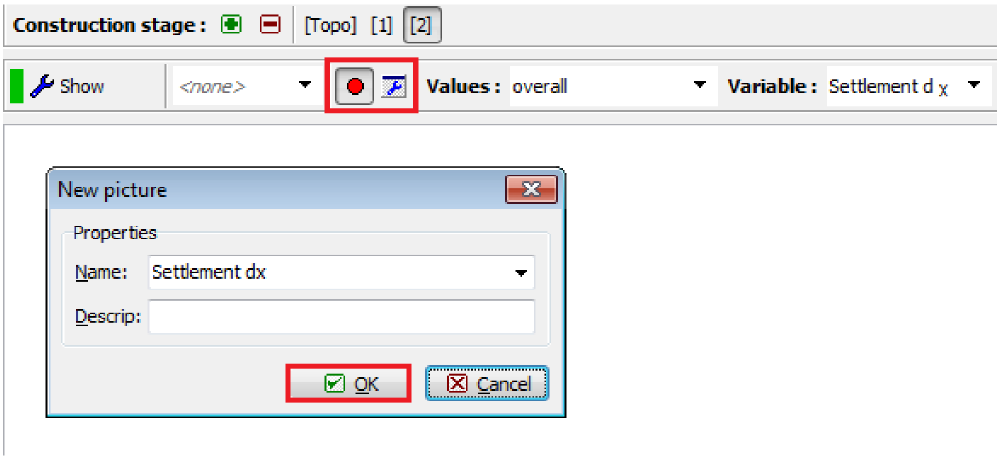
It follows from the picture that the maximum horizontal displacement is 2.2 mm (the collector behaves as a rigid body). To better understand the structure behaviour, we will visualise the deformed mesh (the button in the upper part of screen).



"Analysis" frame – Construction stage 2 (horizontal displacement

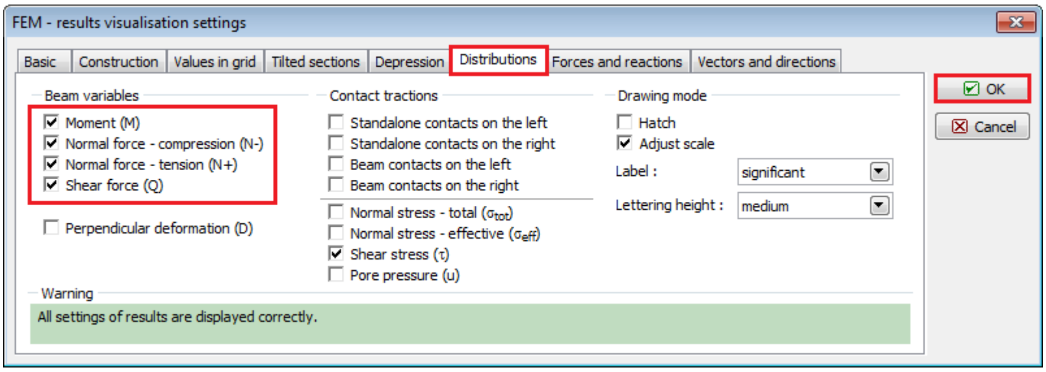
d_x after excavating the soil)

Note: Individual current views presented on the screen can also be downloaded as independent objects (using the red button in the upper left part of the desktop, on the horizontal tool bar). They can also be managed afterwards. In this way the visualization of results is significantly accelerated (for more details visit Help – F1).



Horizontal tool bar + "New picture" dialogue window

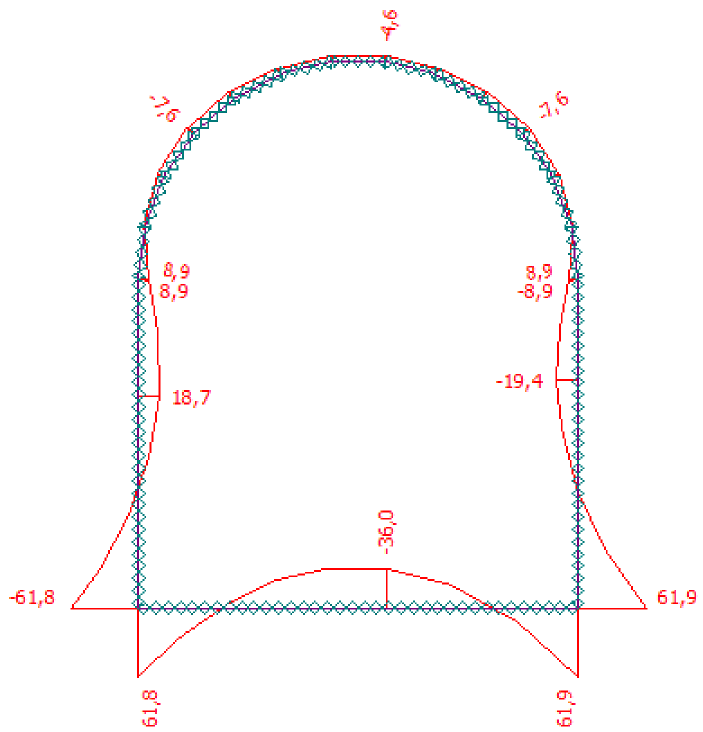
Now we will examine the diagrams for bending moments $M\ [kNm/m]$, shear forces $Q\ [kN/m]$ and normal compressive forces $N^-\ [kN/m]$ for construction stage 2 (using "Show" button in "Distributions" flag).



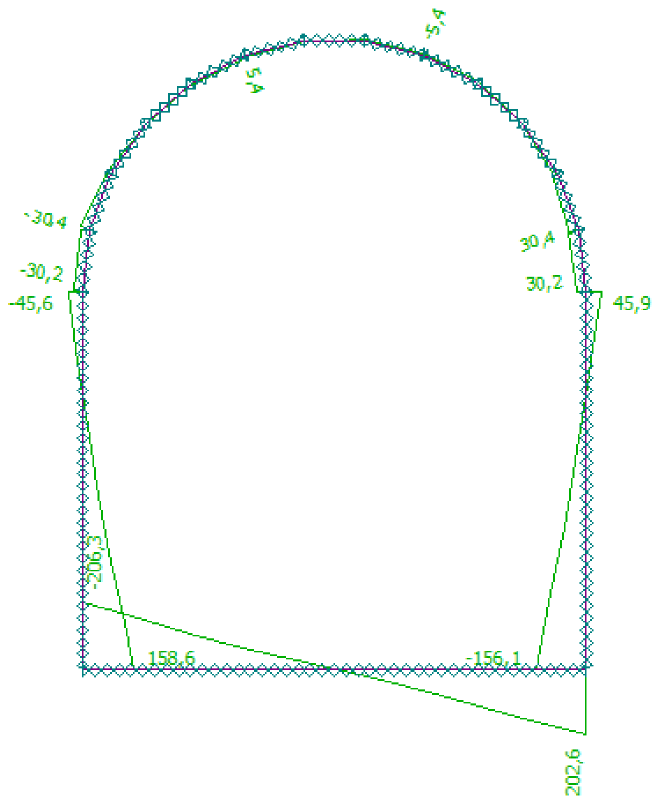
"FEM – results visualisation settings" dialogue window

Note: To maintain sufficient clarity and comprehensibility some results cannot be displayed concurrently. It is, for example, impossible to draw a deformed structure concurrently with diagrams for internal forces along the beam; it is always necessary to choose only one variant. The program gives notice at the bottom of the dialogue window in the case of setting inadmissible combinations of outputs (for more details visit Help – F1).

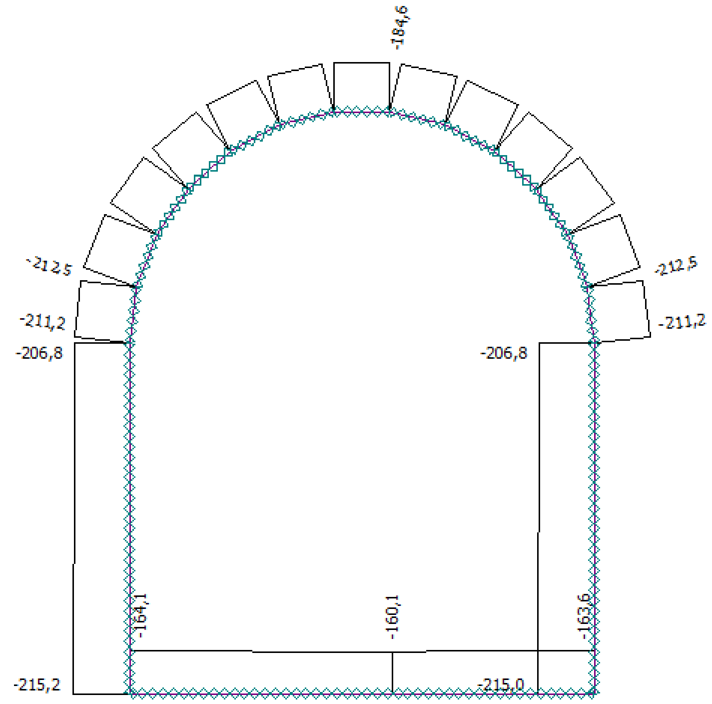
The collector lining reinforcement can be designed and assed for these values in any arbitrary static program (e.g. FIN EC – CONCRETE 2D). We will record the results in a summary table.



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



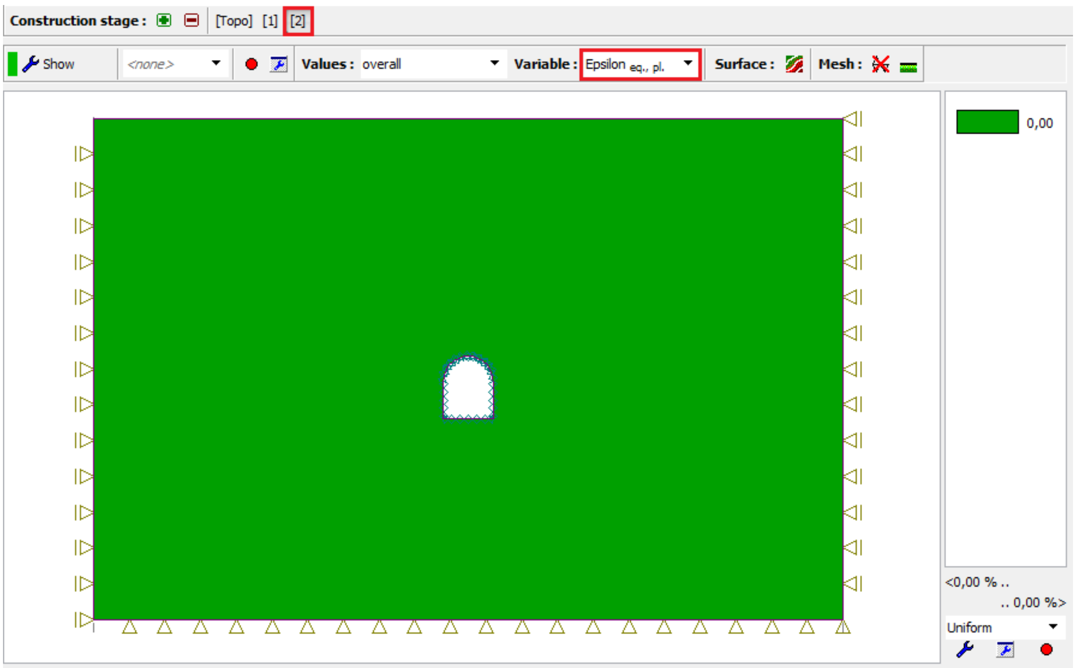
"Analysis" frame – Construction stage 2 (variation shear force Q)



"Analysis" frame – Construction stage 2 (variation of normal compressive force N^-)

Verification of the yield condition: Mohr-Coulomb material model

Now we will verify whether plastic strains develop when adopting non-linear models. We will return back to "Topology" mode and change the material model to „Mohr-Coulomb" in "Soils". After completing the analysis, we will examine the equivalent plastic strains.



"Analysis" frame – Construction stage 2 (equivalent plastic strains $\varepsilon_{eq.,pl.}$ according to the MC model)

It follows from the previous diagram that the yield condition for the Mohr-Coulomb model is not exceeded – equivalent plastic strains $\varepsilon_{eq.,pl.}$ are zero, which corresponds to the structure behaviour determined according to the elastic material model. Resulting values of displacements, geostatic stress and internal forces are therefore identical.

Assessment of results:

The following table presents the values of extreme internal forces along beams (the collector lining) for construction stage 2 (the values of bending moments, shear forces and normal forces). We carried this analysis out for an elastic material model with locally increased density of triangular elements.

Material model	**Construction stage 2**		
	$N^{-}\ [kN/m]$	$M\ [kNm/m]$	$Q\ [kN/m]$
Elastic	– 160.1	+ 61.9	+ 202.6
	– 215.2	– 61.8	– 206.3

Curves for internal forces along beams (extremes) – Construction stage 2

Conclusion:

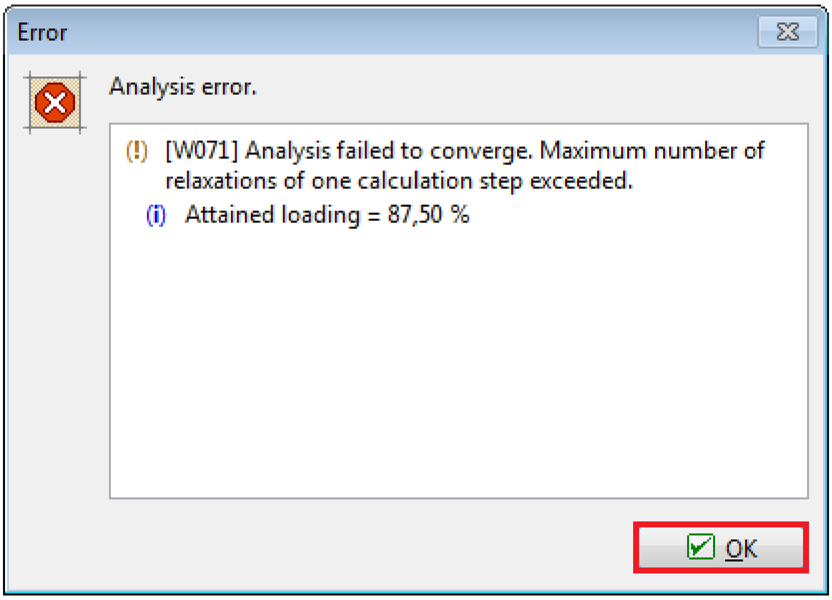
The following conclusions can be drawn from the numerical analysis results:

- Local density refinement of finite element mesh leads to more accurate results.
- If non-linear material models (e.g. Mohr-Coulomb) lead to zero values of equivalent plastic strains $\varepsilon_{eq.,pl.}$, the structure behaves elastically and the results of internal forces, displacements and stresses are identical for both model types.

Note: The analysis we have carried out is in reality based on an unrealistic assumption that the lining acts concurrently with excavating the soil. This procedure would be suitable for structures carried out by jacking through soft soils (jacking of complete structures into soil). In reality the ground mass is unloaded and deformed in the direction of the excavated space when the soil is being excavated. A real example of tunnel modelling is described in Chapter 26. Numerical modelling of tunnel excavation adopting the NATM method.

If, in our particular case, the lining had not been activated immediately (it can be modelled as another stage without specifying beam elements), the excavation would have collapsed; for the elastic model this is presented by large deformations, whilst the program will not find the solution in the case of the non-linear model.

Analysis without beam elements (settlement d_z according to the elastic model)



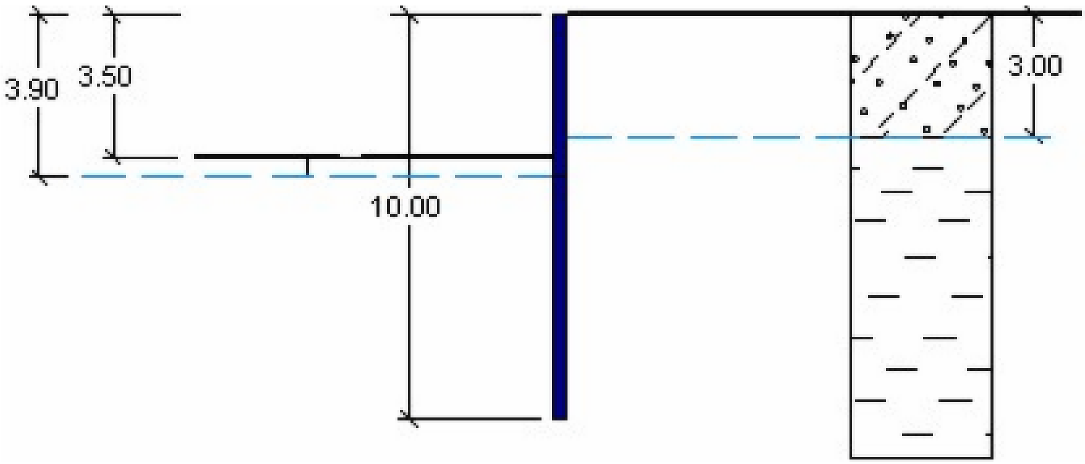
"Error" dialogue window – analysis without using beam elements (according to the MC model)

Chapter 24. Numerical solution to a sheeting wall structure

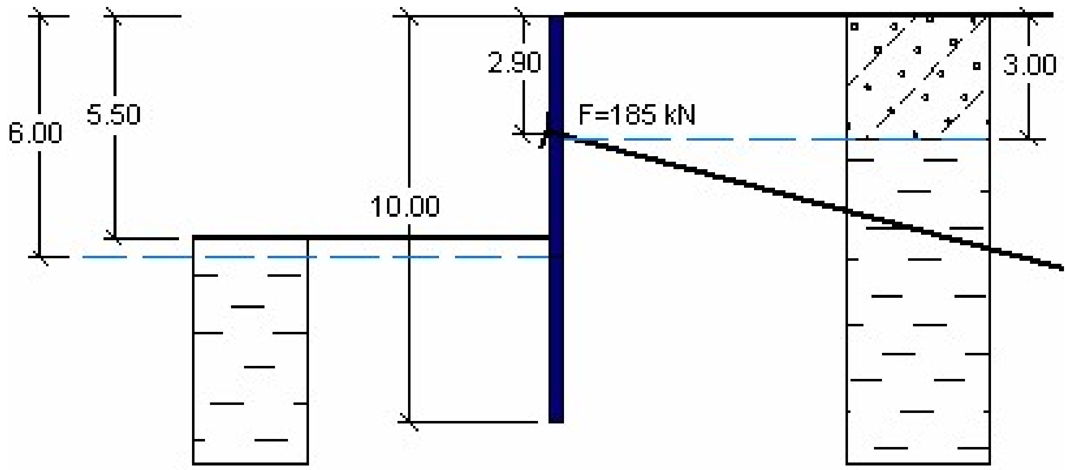
The objective of this manual is to analyse deformations of an anchored sheet pile wall and determine diagrams of internal forces using the Finite Element Method.

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 diagrams below. Determine internal forces acting along the anchored wall length. Sheet piles are 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 and 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: $\gamma \left[kN/m^3\right]$	18	21
Modulus of elasticity: $E \left[MPa\right]$	10	4,5
Poisson´s ratio: $\nu \left[-\right]$	0,3	0,4
Cohesion of soil: $c_{eff} \left[kPa\right]$	5	12
Angle of internal friction: $\varphi_{eff} \left[^\circ\right]$	29	19
Dilation angle: $\psi \left[^\circ\right]$	0	0
Saturated unit weight: $\gamma_{sat} \left[kN/m^3\right]$	20	23

Table with the soil parameters – anchored sheet pile wall

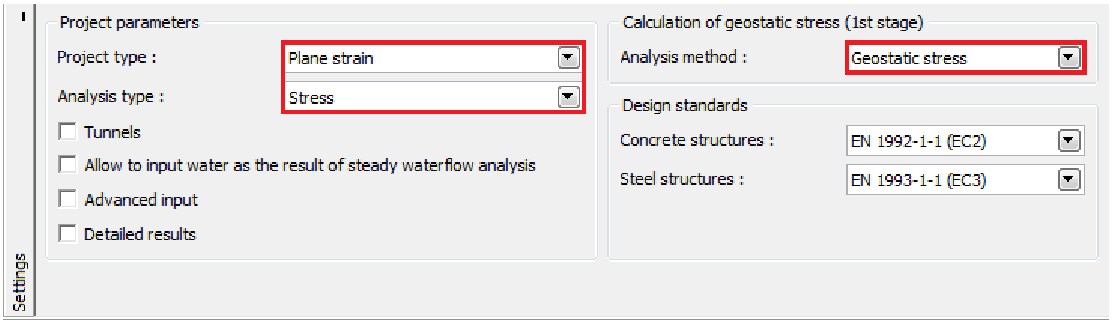
Solution:

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

- Topology: settings for and modelling of 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: excavation of soil, specification of anchors, analysis results + monitors.
- Assessment of results: comparison, conclusion.

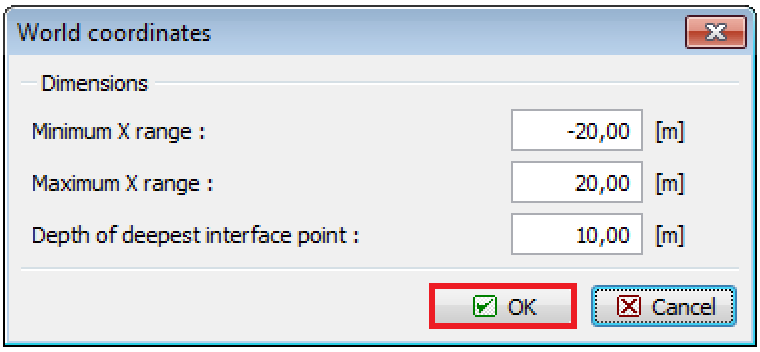
Topology: problem settings

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



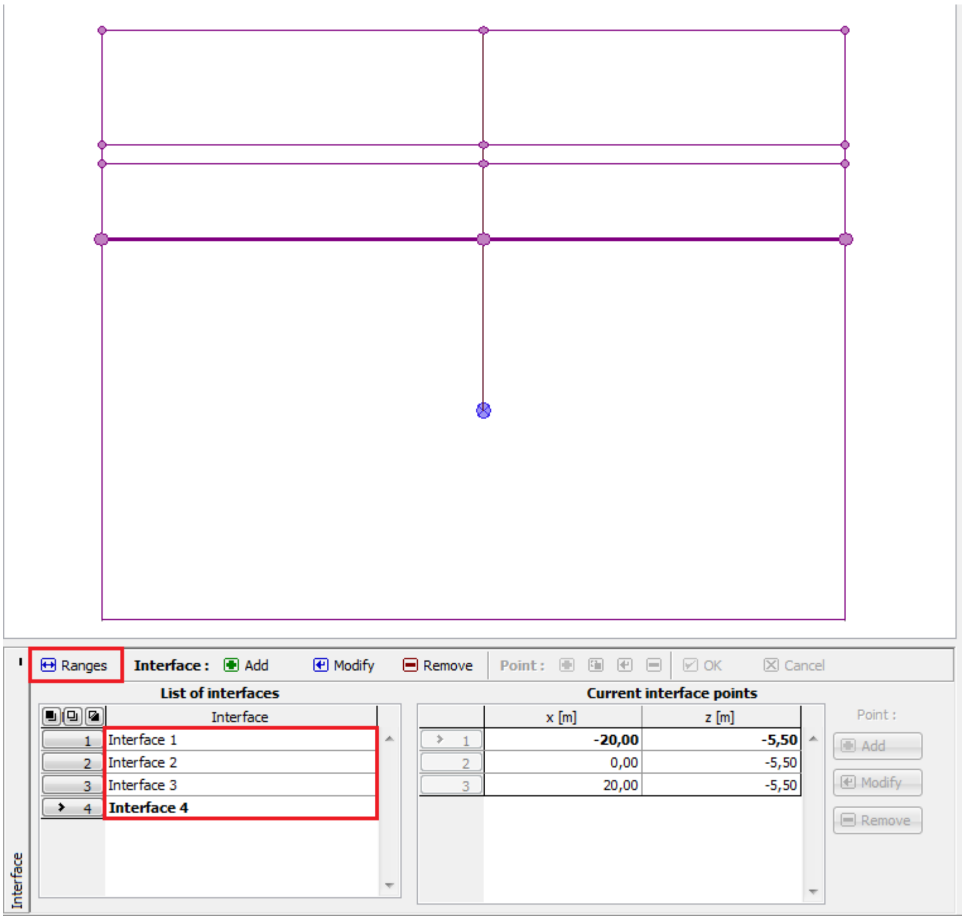
"Settings" frame

Further we will set the world coordinates; we will choose sufficient dimensions of the world so that 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.



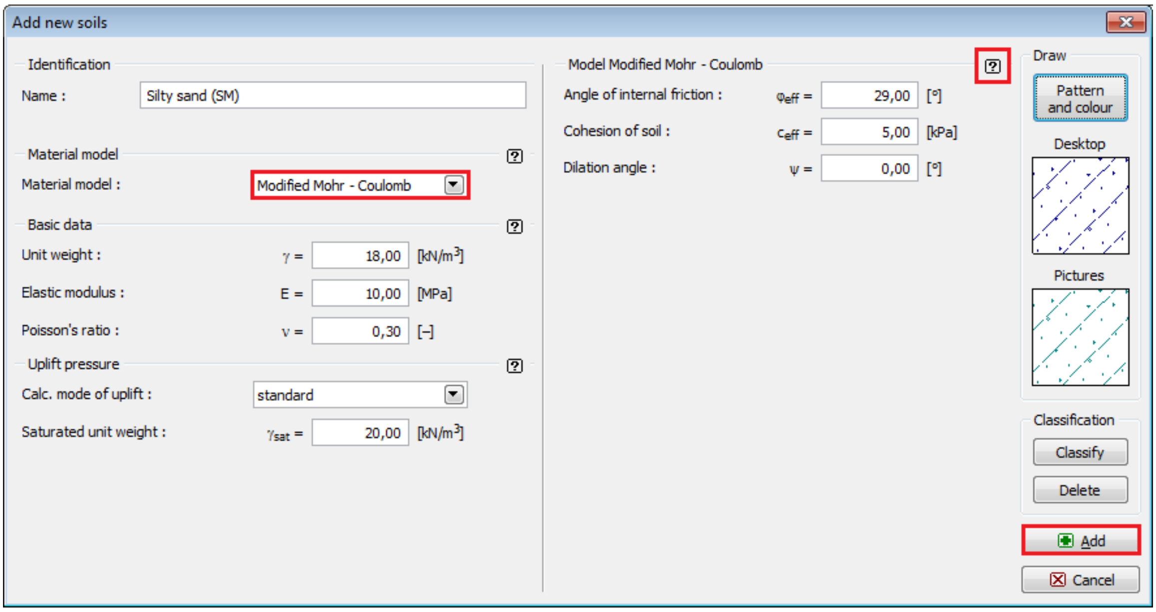
"World coordinates" dialogue window

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.



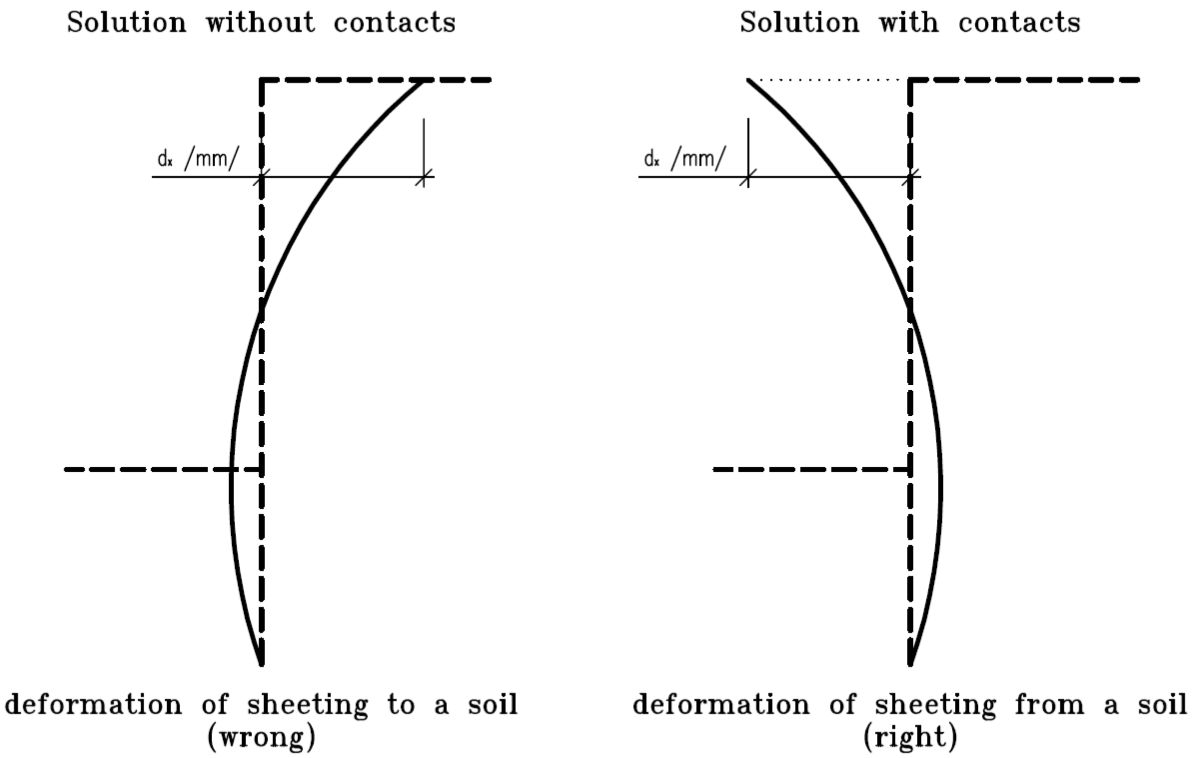
"Interface" frame

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



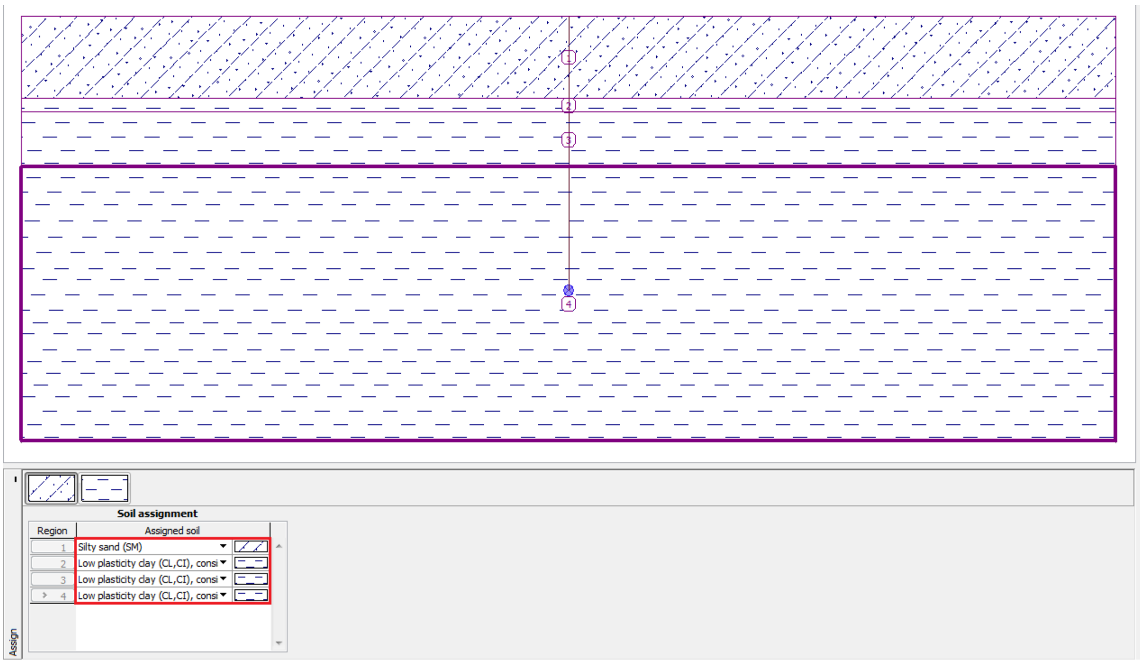
"Add new soils" dialogue window

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).



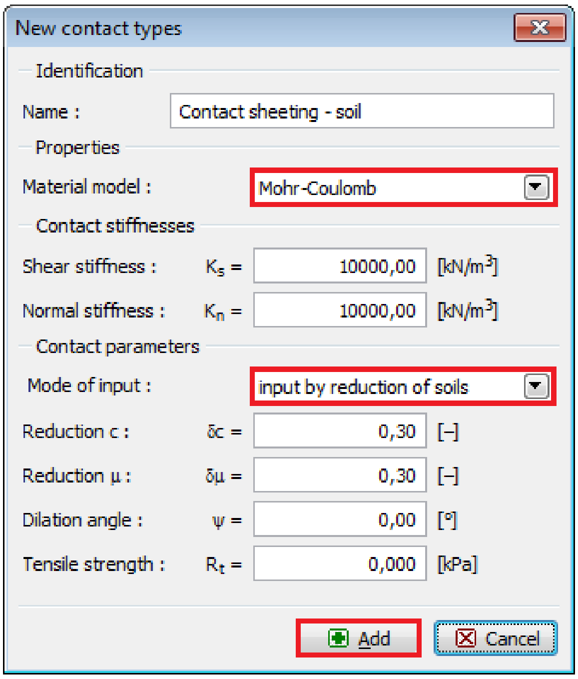
The application of a suitable material model to a numerical analysis of sheeting structures

The following figure presents the assigning of soil to the geological profile.



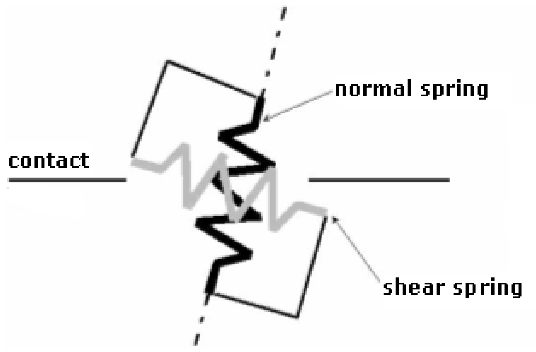
"Assign" frame

The next step lies in setting 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 $K_s = K_n = 10\,000\ kN/m^3$.



"New contact types" dialogue window

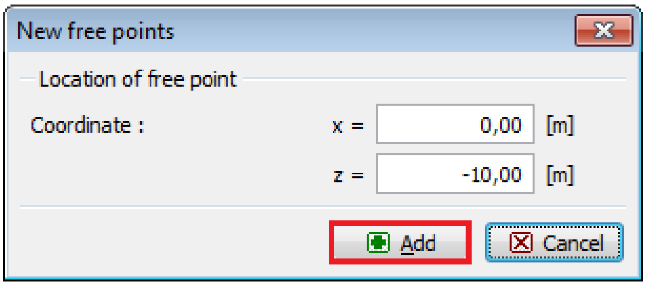
Note: Contact elements are used in the analyses where it is necessary to allow for the interaction between the structure and the surrounding environment – an interface between two totally different materials (soil – sheeting). A typical example of the use of contact elements is the modelling of sheeting 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

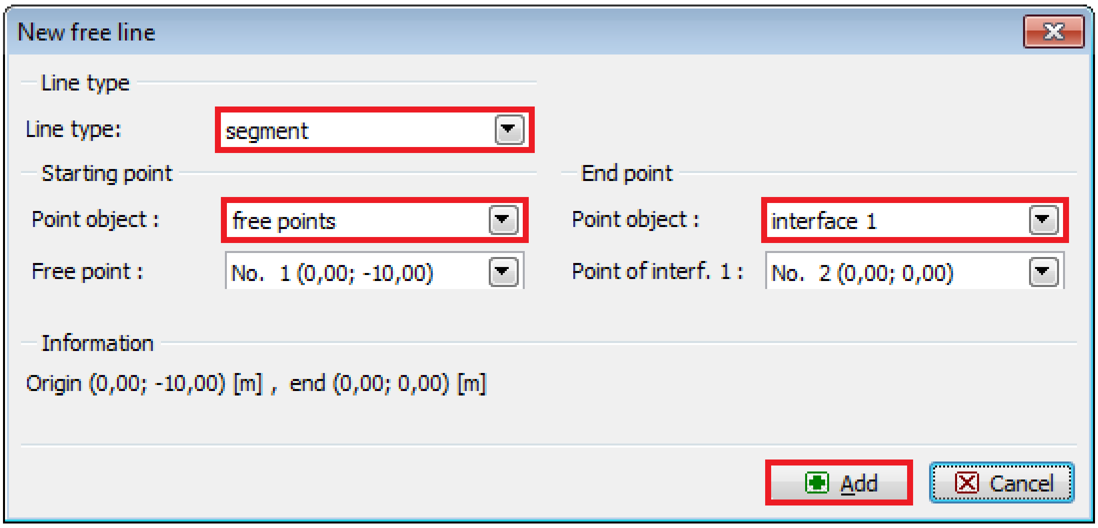
Note: Despite the fact that the selection of parameter K_s *is not important in the case of completely plastic behaviour of the contact, the magnitude of this quantity is crucial for the successful solution to the non-linear problem in question. Too high stiffness values (over* $100\,000\ kN/m^3$ *) may lead to oscillation of the numerical solution. On the contrary, too 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*.



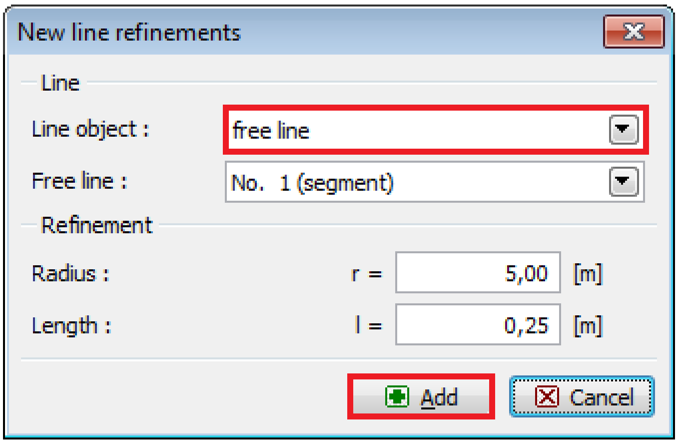
"New free points" dialogue window

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

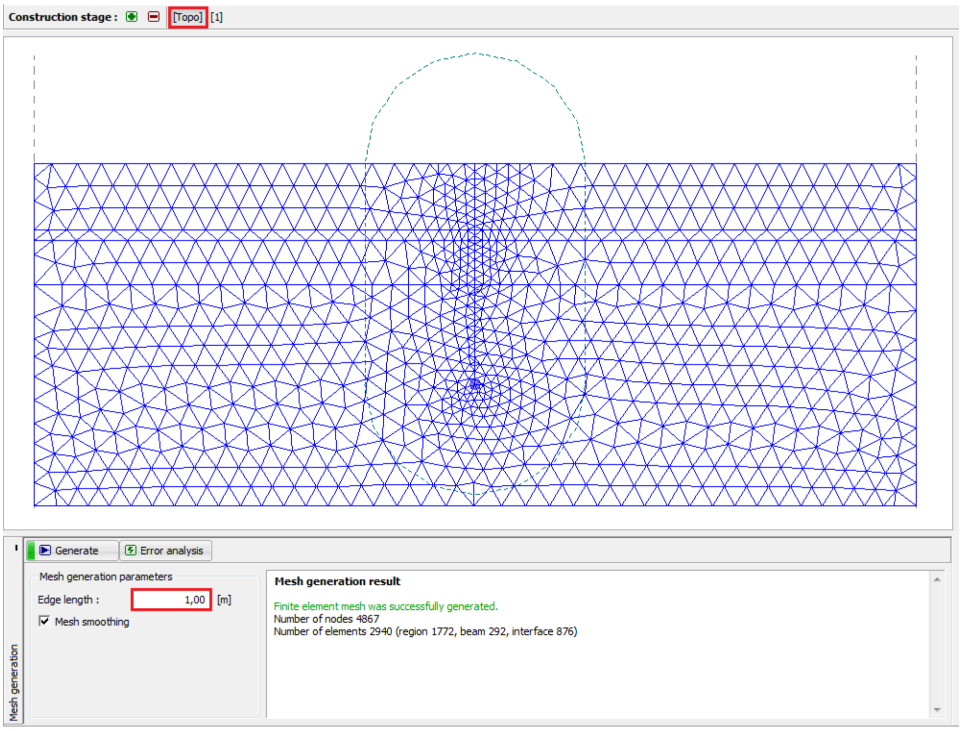
The last step in setting the topology is the generation of the finite element mesh. It is reasonable to refine FE mesh in the sheeting wall surroundings. In the "New line refinements" we will select the respective extent radius $r = 5.0\ m$ and the element edge length $l = 0.25\ m$.



"New line refinements" dialogue window

Then we will go over to "Mesh generation" frame and will generate a mesh with the element edge length of 1.0m (using "**Generate**" button). The program will automatically smooth the refined FE mesh.

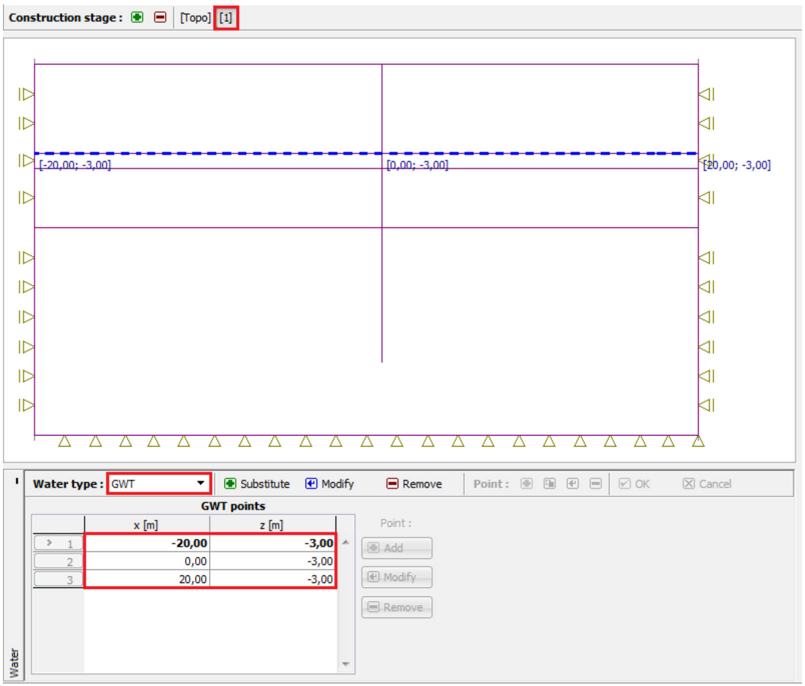
Note: Subsequently we will visually check whether the finite element mesh density is adequate to the extent and complexity of the given problem (for more details visit Help – F1). The increased mesh density contributes to the stabilisation of the non-linear analysis similarly to the effect of the decrease in shear stiffness.



"Mesh generation" frame – element edge length of 1 m (with locally increased mesh density)

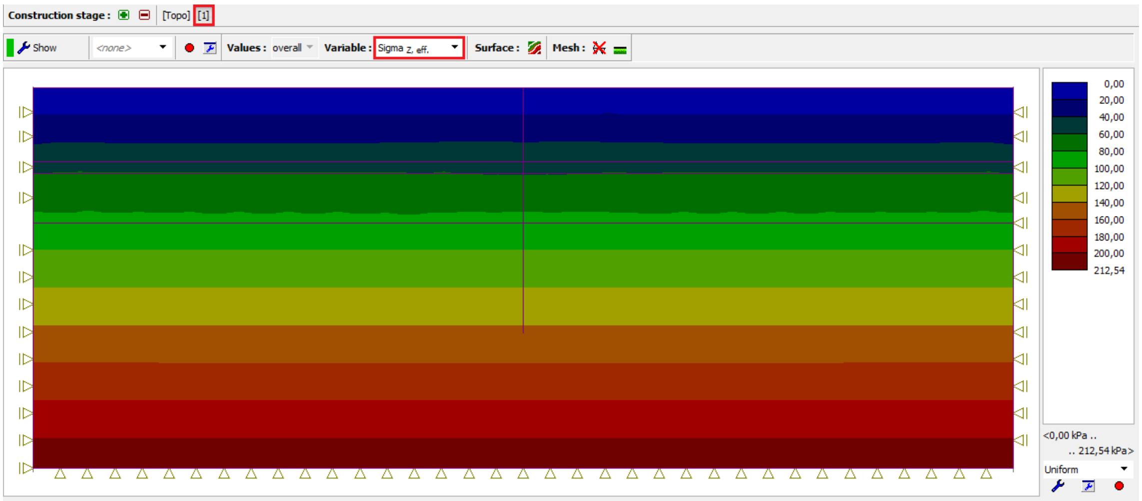
Construction stage 1: primary geostatic stress

After generating the mesh we will go over to construction stage 1 and will 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*).



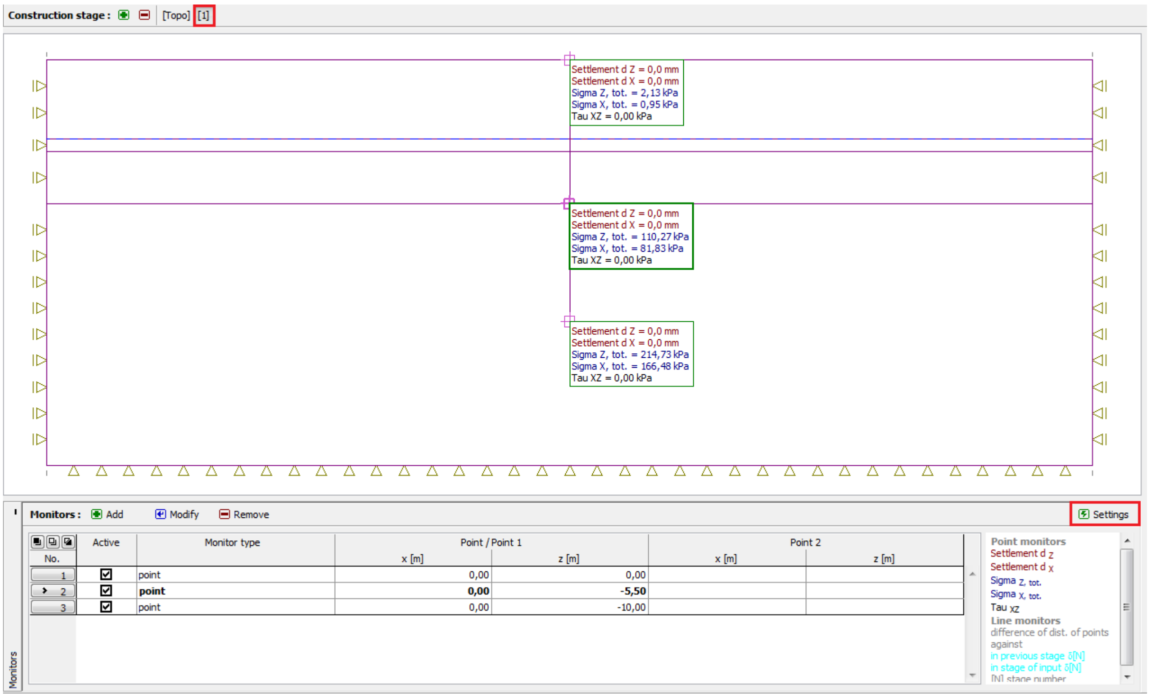
"Water" frame – Construction stage 1 (GWT at the depth of 3.0 m)

We will carry out the analysis of the 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 values of certain quantities (during the course of the analysis of individual construction stages) we will define the 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].

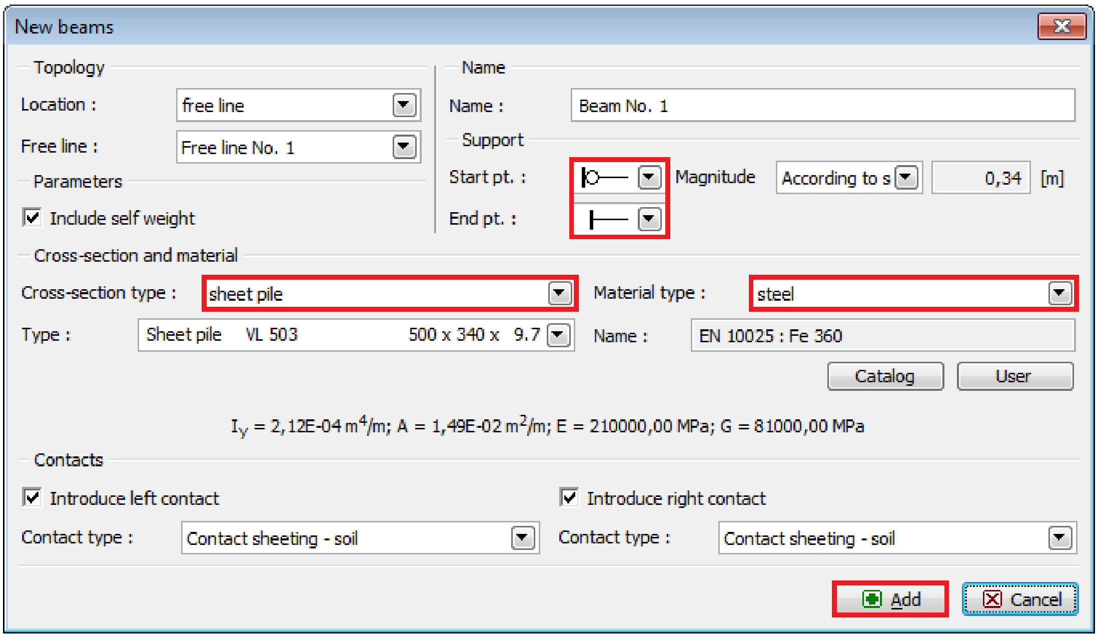


“Monitors” frame – Construction stage 1 (point monitors)

Note: We will edit the individual values of quantities we want to visualise in the results using "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

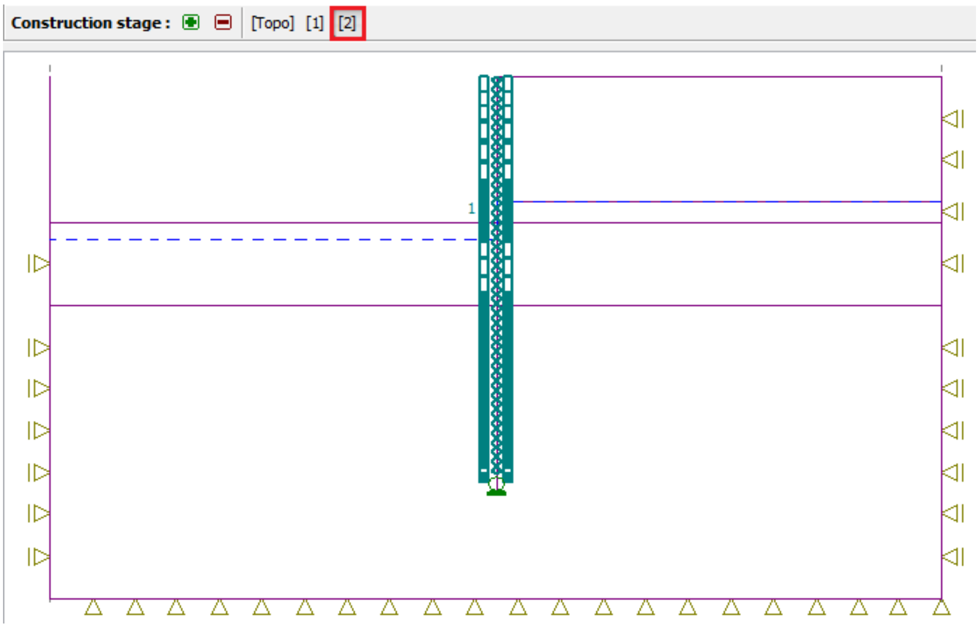
At this construction stage we will first go over to "Beams" frame and will model the sheet pile wall. We will define the following relationships: the location, material and steel class, cross-section type (VL 503), the support of beam ends and contacts (for more details visit Help – F1).



"New beams" dialogue window – Construction stage 2

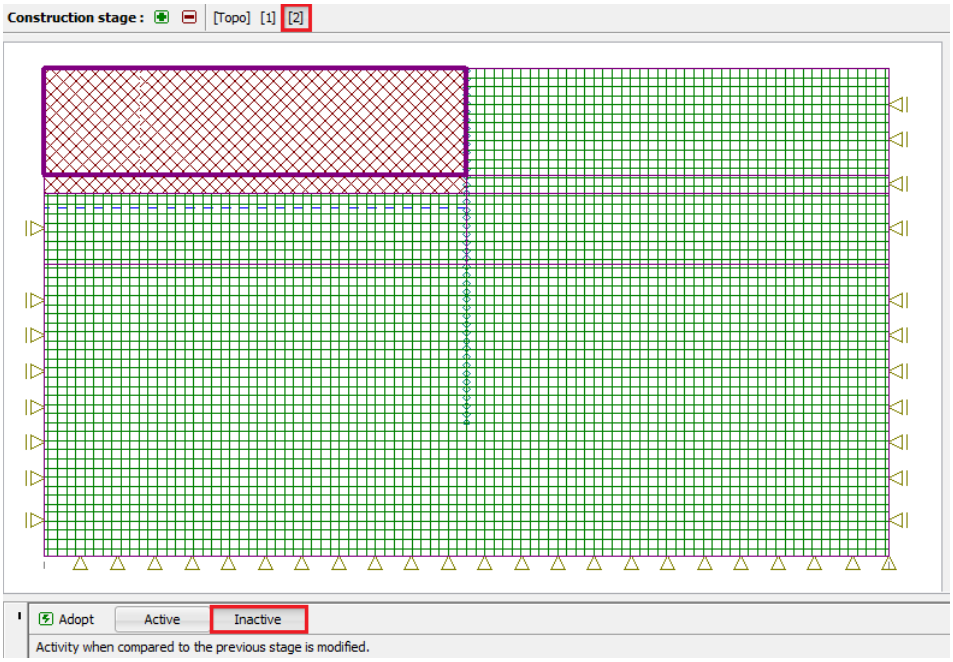
Note: We assume a rotational degree of freedom at the toe of the wall. This boundary condition will ensure for us that there will be zero bending moment at the sheet pile wall toe (for more details visit Help – F1.

The following figure presents a beam element together with contacts.



"Beams" frame – Construction stage 2

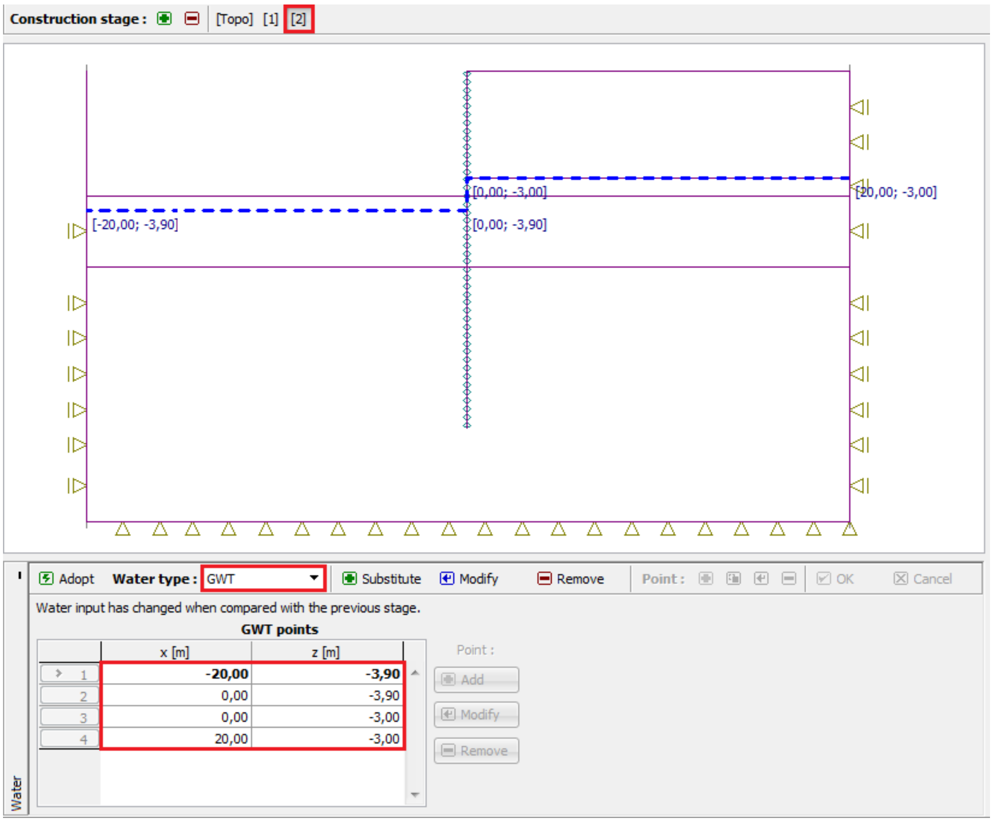
Then we will model the excavation of soil in "Activation" frame – we will set the given regions in the program with the mouse cursor as inactive (for more details visit F1).



"Activation" frame – Construction stage 2

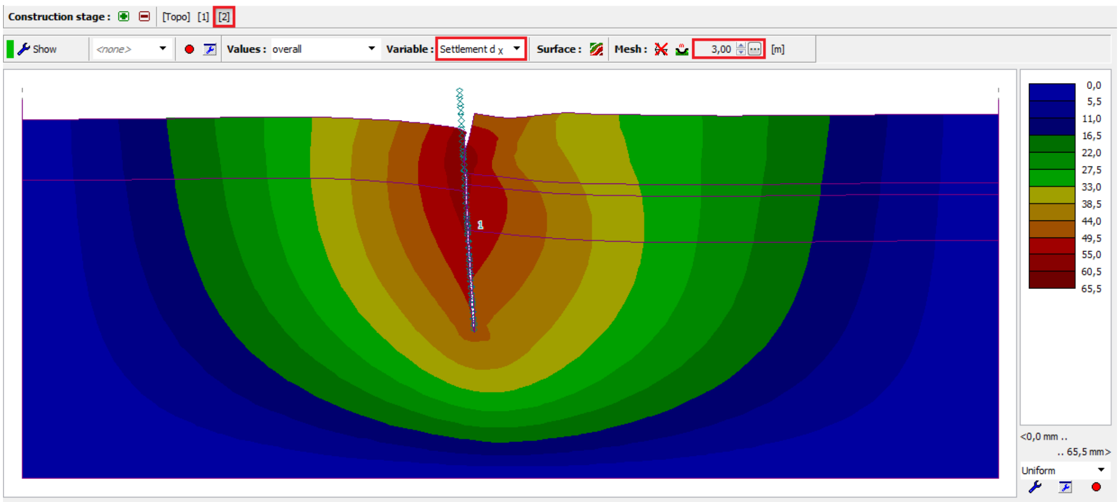
Note: It is obvious from the preceding figure that the automatic structure corrector built in the program divided the interfaces of soils crossed by the wall into individual circumscribed regions (for more details visit Help – F1).

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

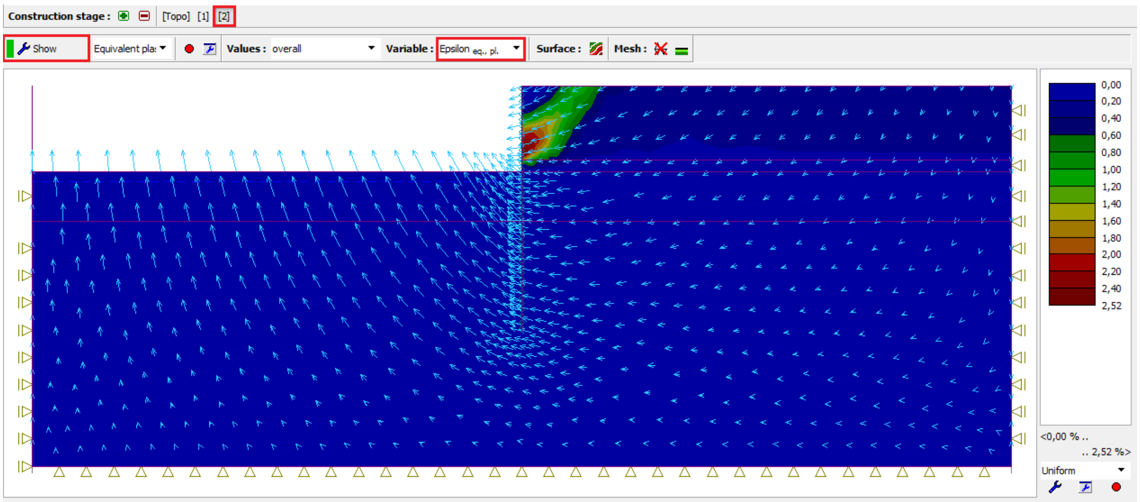


"Water" frame – Construction stage 2 (changes in the course of the GWT)

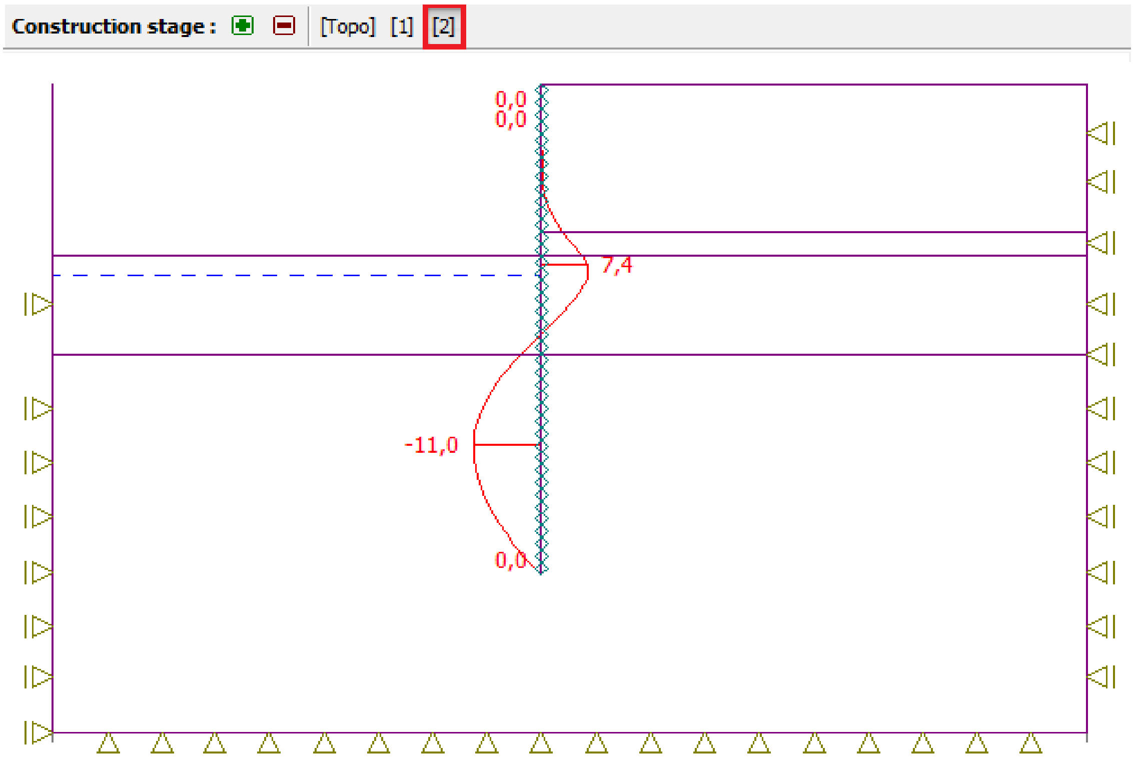
Now we will carry out the analysis of construction stage 2 and will examine the results for the diagrams of internal forces along the beam, equivalent plastic strain and the deformed structure.



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



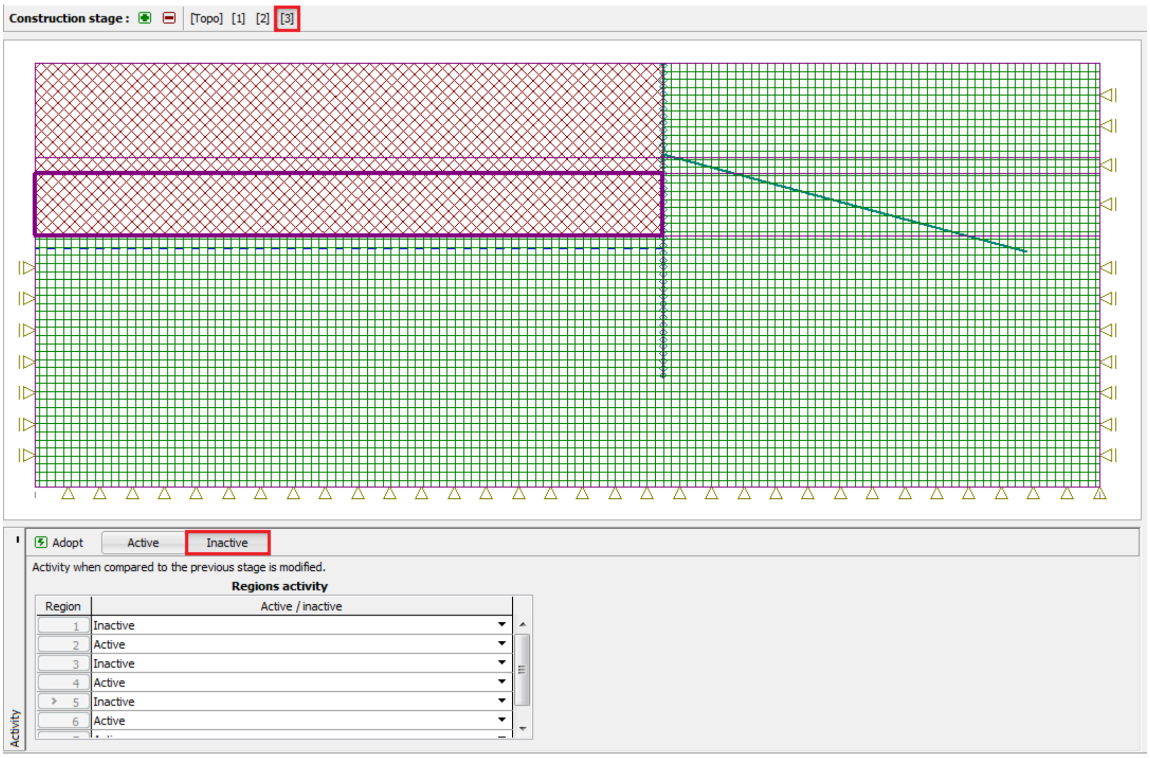
"Analysis" frame – Construction stage 2 (equivalent plastic strain $\varepsilon_{eq.,pl.}$ and displacement vectors)



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

Construction stage 3: settings for anchors

We will add construction stage 3 and will extract the remaining soil. First we will select the given region with the mouse cursor and click the "Inactive" button.



"Activation" frame – Construction stage 3

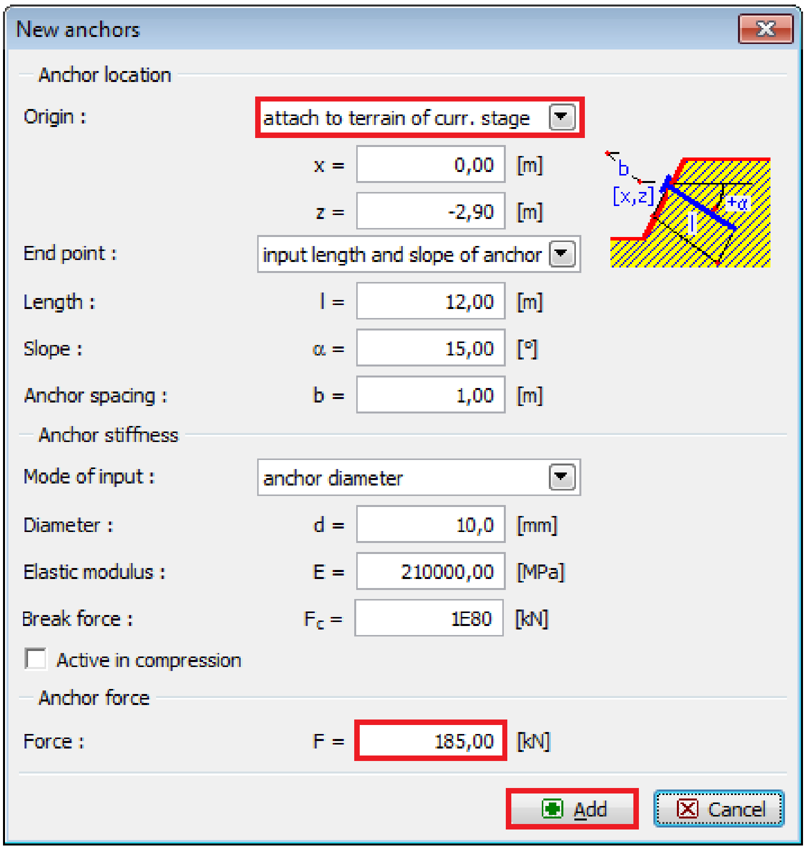
Then we will push the "Add" button in "Anchors" frame and, in the "New anchors" dialogue window, we will set a steel anchor with pre-stressing force $F = 185\ kN$. We will consider the anchor to be at the depth of 2.9 m under the terrain surface – we will set the anchor head coordinates by the point [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 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 take the possibility of the reinforcement element breaking, therefore we will set break force F_c to a sufficiently high value (for more details visit Help – F1).

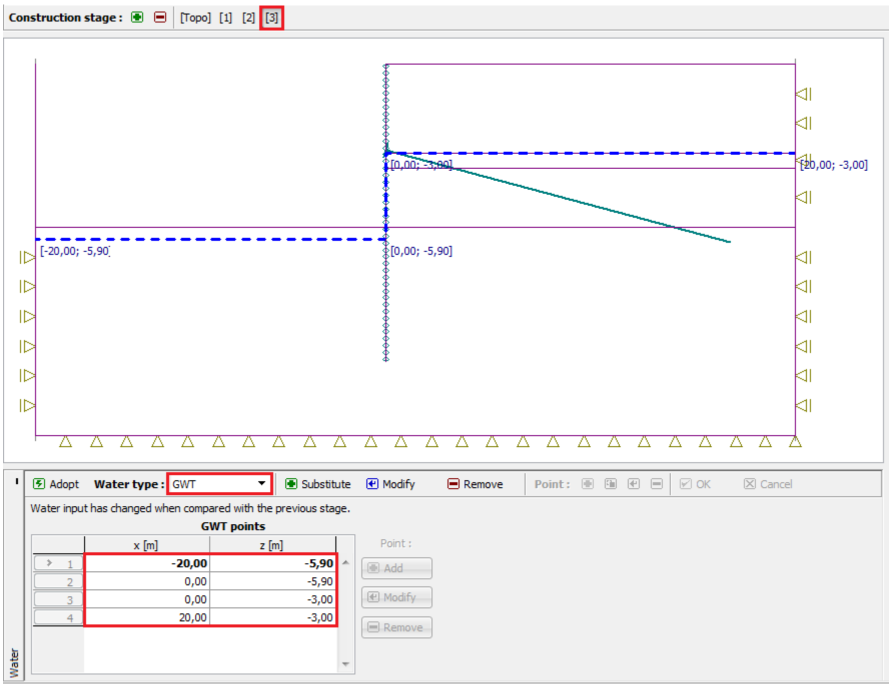


"New anchors" dialogue window – Construction stage 3

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.

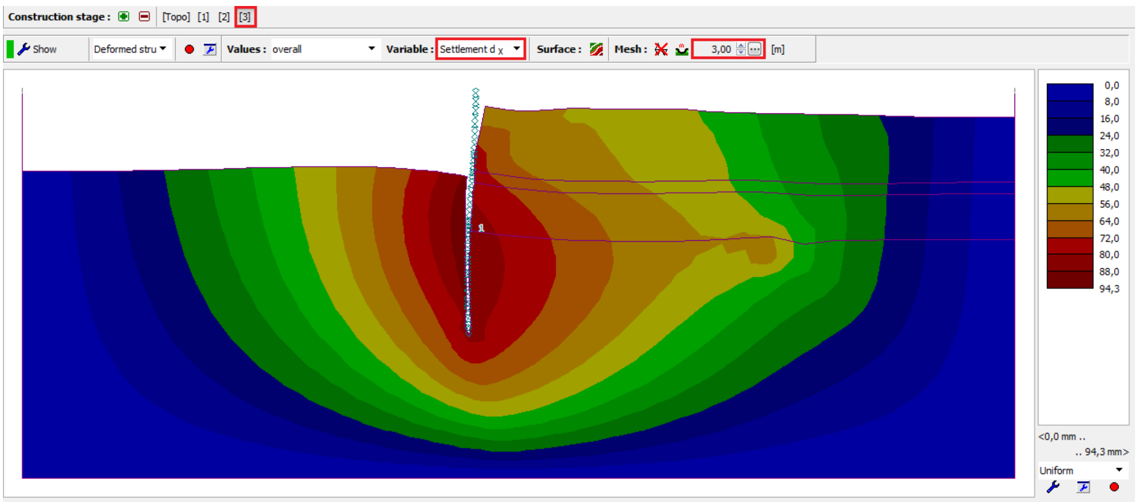
The embedment of the anchor in ground mass should be sufficiently tough (fixing to an element) so that unrealistic pulling of the anchor out does not occur when significant plastic strains develop in the vicinity of the anchor root (fixing to a node, too great increase in density in the root surroundings), causing unrealistic loss of the pre-stressing force.

In the last step of setting construction stage 3 we will change the ground water table level according the diagram below. The other input parameters will remain unchanged.



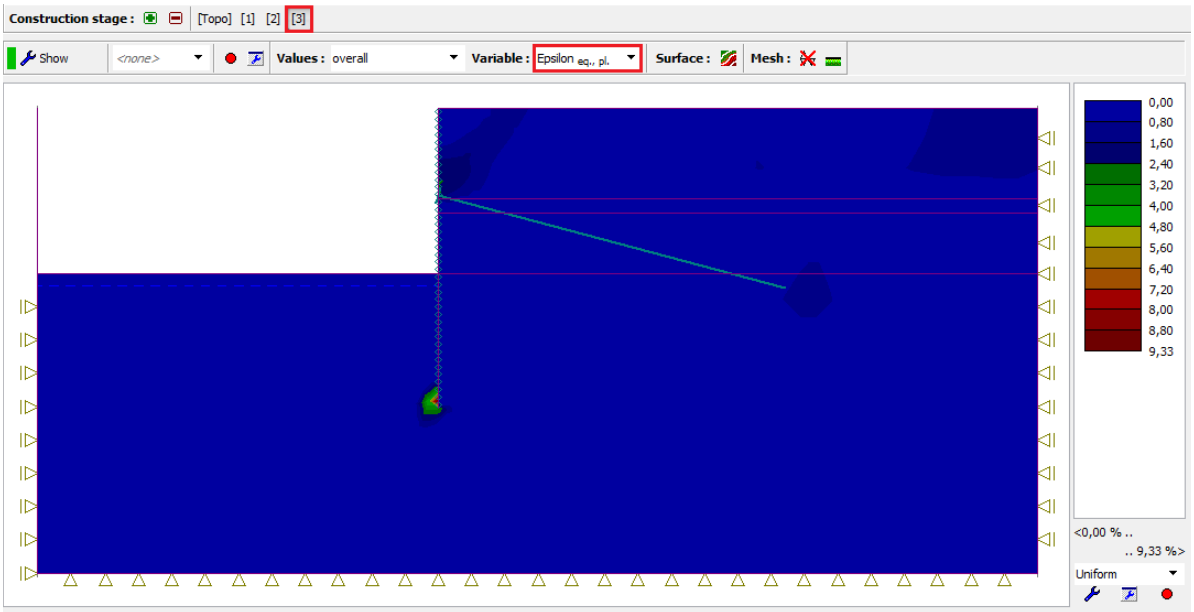
"Water" frame – Construction stage 3 (change in the GWT level)

Now we will carry out the analysis of construction stage 3 and will again examine the results of the numerical solution (similarly to the previous construction stage).



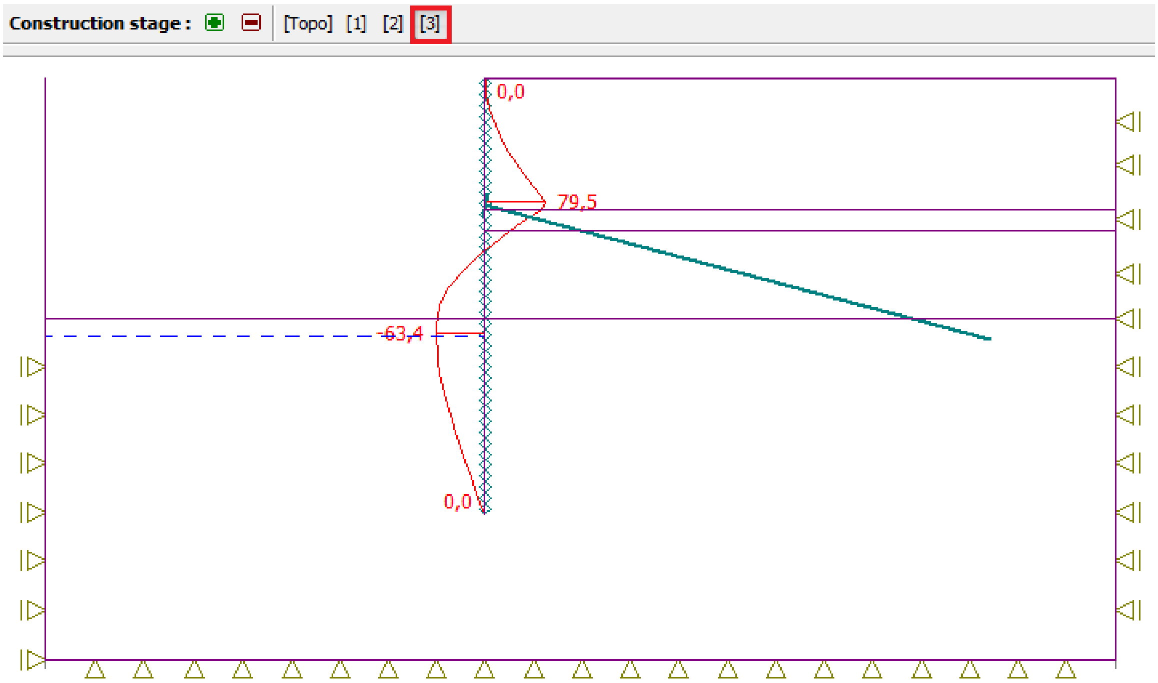
"Analysis" frame – Construction stage 3 (settlement d_x – deformed structure)

It follows from this figure that the maximum lateral displacement in the vicinity of the sheeting wall formed by steel sheet piles is $d_x = 94.3\ mm$.



"Analysis" frame – Construction stage 3 (equivalent plastic strain $\varepsilon_{eq.,pl.}$)

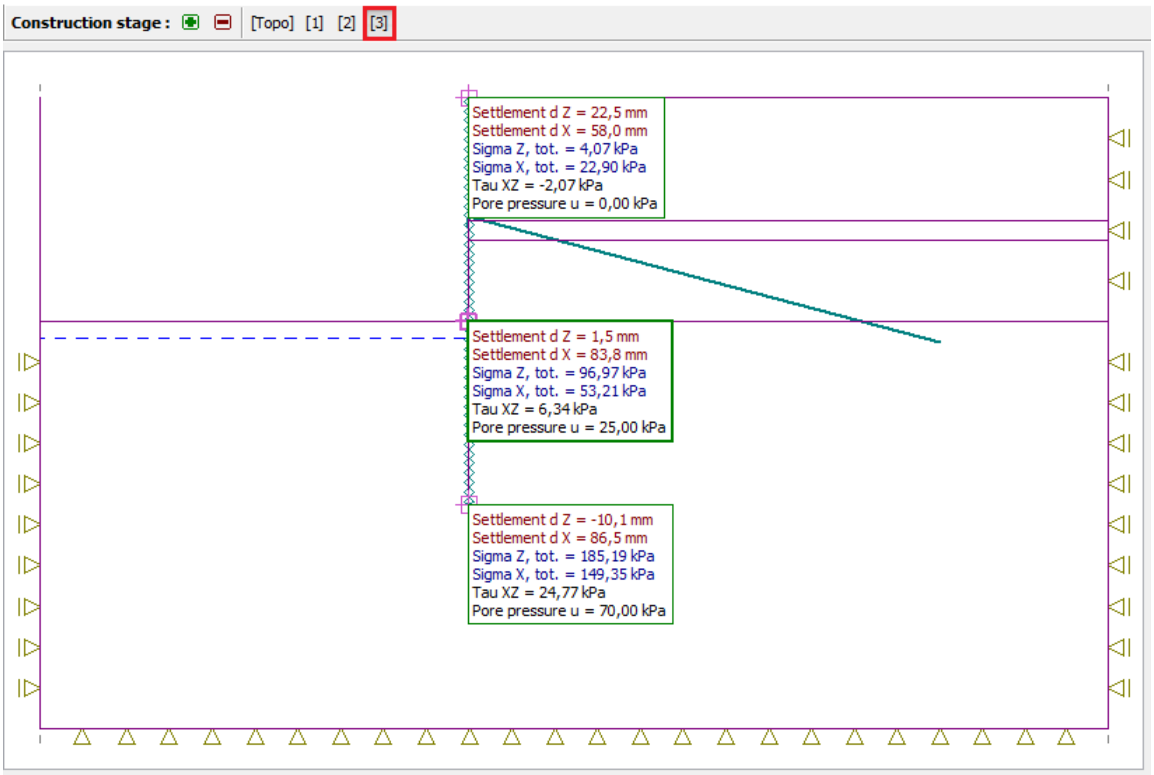
It is obvious from the plotted equivalent plastic strains that the largest plastic strains are developed in the soil in the vicinity of the sheeting wall toe. In the previous stage, soil was plasticised in the vicinity of the anchor location (for more details visit Help – F1).



"Analysis" frame – Construction stage 3 (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 will determine deformations at the sheeting pile wall head.



"Monitors" frame – Construction stage 3 (Point monitors)

Assessment of results:

The following table presents 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 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 GEO 5 – Sheeting assessment program.

Material model / program	**Stage 2** $M\ [kNm/m]$	**Stage 3 – field** $M\ [kNm/m]$	**Stage 3 – anchor** $M\ [kNm/m]$
MC (Mohr-Coulomb)	14.0	– 54.1	78.1
MCM (Modified M-C)	7.4	– 63.4	79.5
Sheeting assessment * (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:

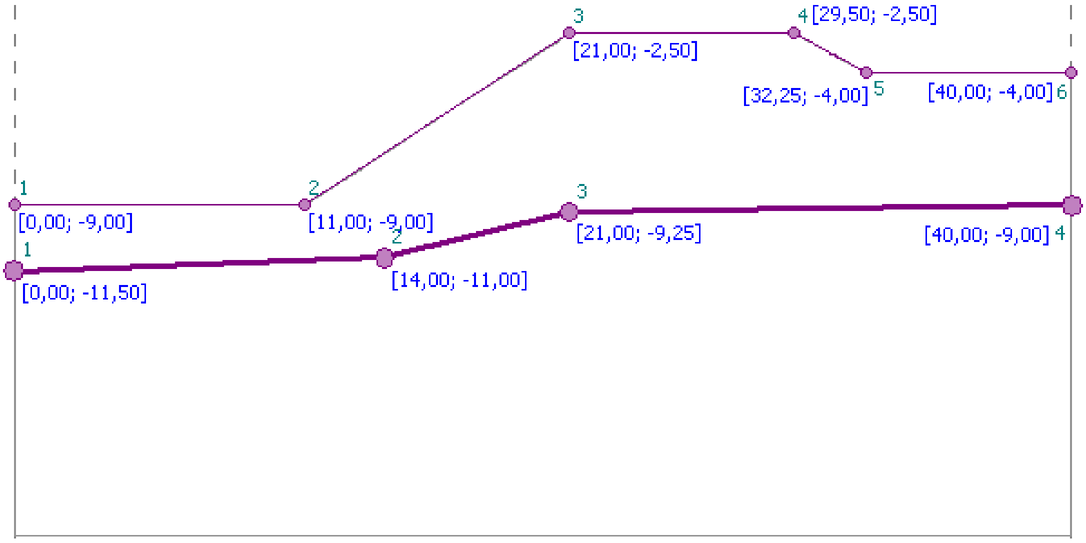
- The locally increased density of the FE mesh in the surroundings of lines leads to more accurate determination of the internal forces results.
- It is necessary for analyses of sheeting walls to use contact elements and non-linear material models, allowing for the development of plastic strains and giving truer picture of the real behaviour of structures in the surrounding ground mass.
- Maximum equivalent plastic strains $\varepsilon_{eq.,pl.}$ represent the potential locations of failure (as a result of exceeding the material yield condition).

Chapter 25. Slope stability assessment

The objective of this manual is to analyse the slope stability degree (factor of safety) using the Finite Element Method.

Task specification:

Determine the slope stability degree, first without the action of strip surcharge and then under the strip surcharge $q = 35{,}0\ kN/m^2$. The slope geometry scheme for all construction stages (including individual interface points) is shown in the following diagram. Then carry out the slope stabilisation by means of pre-stressed anchors.



Scheme of the slope being modelled – individual interface points

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

Soil parameters / Classification	Soil No. 1	Soil No. 2 – R4
Unit weight of soil: $\gamma\ [kN/m^3]$	18	20
Modulus of elasticity: $E\ [MPa]$	21	300
Poisson´s ratio: $\nu\ [-]$	0.3	0.2
Cohesion of soil: $c_{eff}\ [kPa]$	9	120
Angle of internal friction: $\varphi_{eff}\ [°]$	23	38
Dilation angle: $\psi\ [°]$	0	0
Saturated unit weight: $\gamma_{sat}\ [kN/m^3]$	20	22

Table with the soil parameters – verification of slope stability

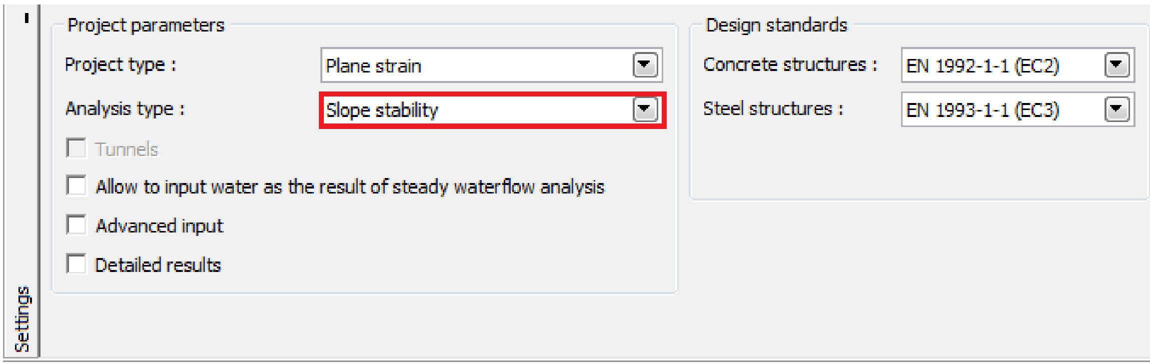
Solution:

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

– Topology: settings for and modelling of the problem (interface, mesh generation),
– Construction stage 1: analysis of the factor of safety of the original slope without the strip surcharge,
– Construction stage 2: analysis of the factor of safety of the slope under the strip surcharge,
– Construction stage 3: slope stabilisation by means of anchors; slope stability analysis.
– Assessment of results: comparison, conclusion.

Topology: problem settings

We will set the analysis type in "Settings" frame with the *Slope stability* option. We will leave the other parameters unchanged.

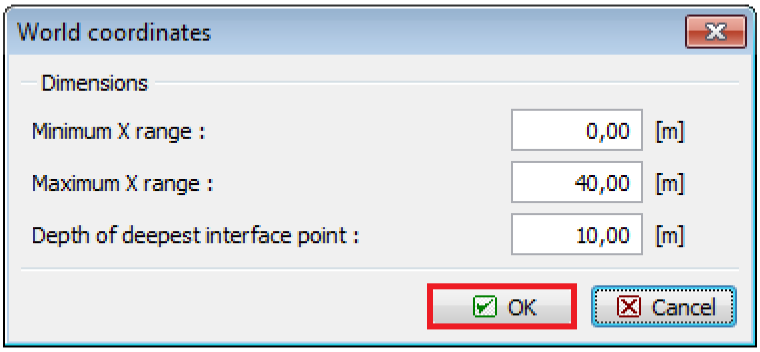


"Settings" frame

Note: The setting and development of the model in the "Slope stability" mode is completely identical with the "Stress" mode. The slope stability analysis is performed by pushing the "Analyze" button. Individual analyses of the slope stability at the construction stages are totally independent and have no relationship with previous stages and analyses (for more details visit Help – F1).

Further we will set the world coordinates. We will choose the world dimensions sufficiently large so that the results are not affected by conditions at the edge. For our particular problem we will choose the model coordinates $\langle 0\ m; 40\ m \rangle$ and set the depth up from the lowest interface point to 10 m.

Then we will define the interface points for individual soil layers that are defined in the following table.



"World coordinates" dialogue window

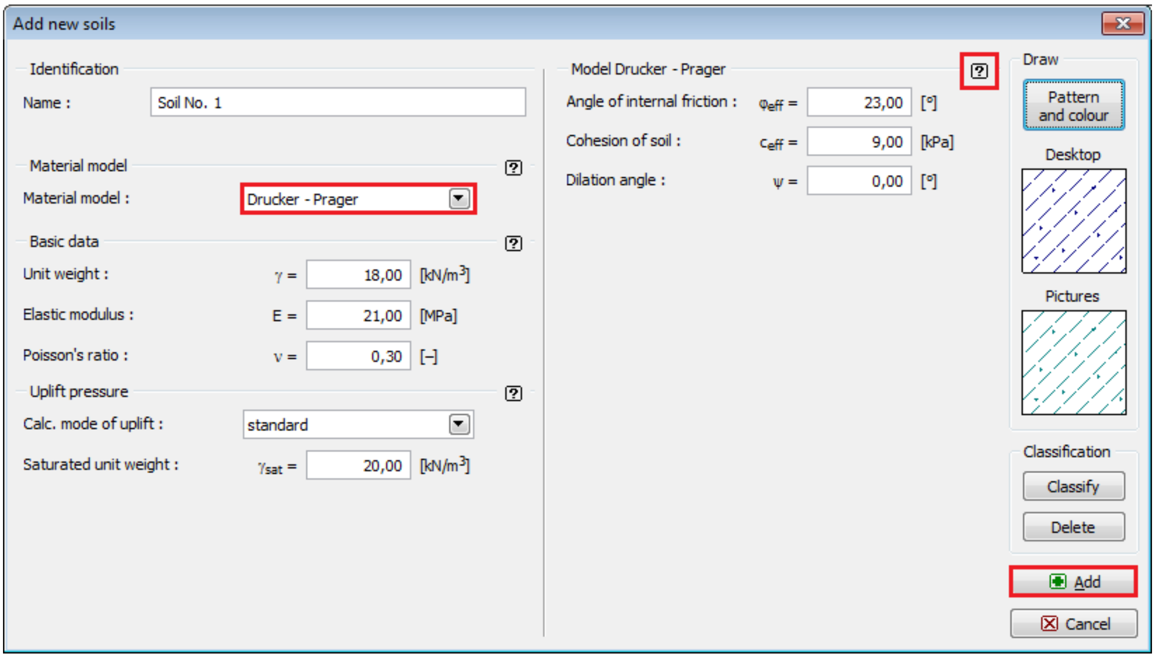
	Interface 1		Interface 2	
	x [m]	z [m]	x [m]	z [m]
1	0,00	-9,00	0,00	-11,50
2	11,00	-9,00	14,00	-11,00
3	21,00	-2,50	21,00	-9,25
4	29,50	-2,50	40,00	-9,00
5	32,25	-4,00		
6	40,00	-4,00		

List of points for individual soil layers interfaces

Now we will set respective parameters of soils and subsequently will assign soils to individual areas. We will choose the Drucker-Prager model (for more see the *note*). We will consider the dilation angle ψ for both soil layers equal to zero, i.e. the material does not change its volume when exposed to shear stress (for more details visit Help – F1).

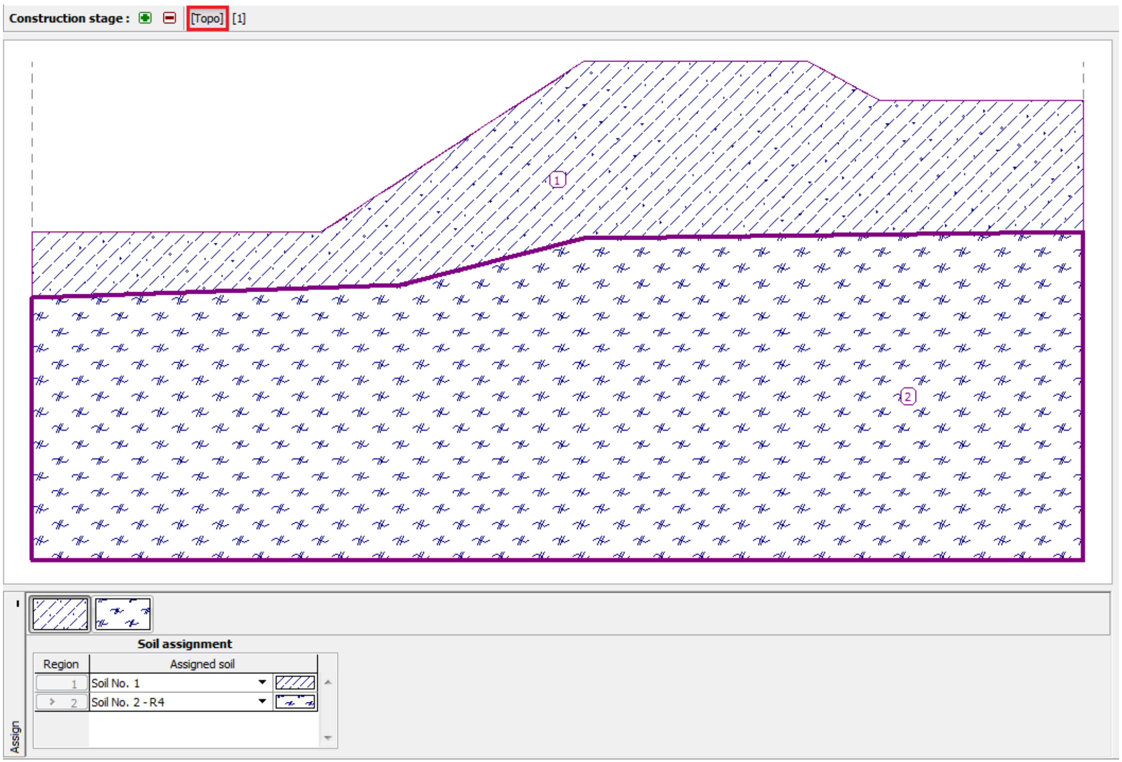
Note: It is necessary for the slope stability analysis to choose a non-linear soil model, which assumes the development of plastic strains and is formulated on the basis of strength parameters of soils c and φ.

In this particular case we chose the Drucker-Prager material model, because of the more yielding response of the structure compared with the classical Mohr-Coulomb model (for more details visit Help – F1). The comparison of results achieved using individual non-linear material models is presented in the table at the end of this example.



"Add new soils" dialogue window

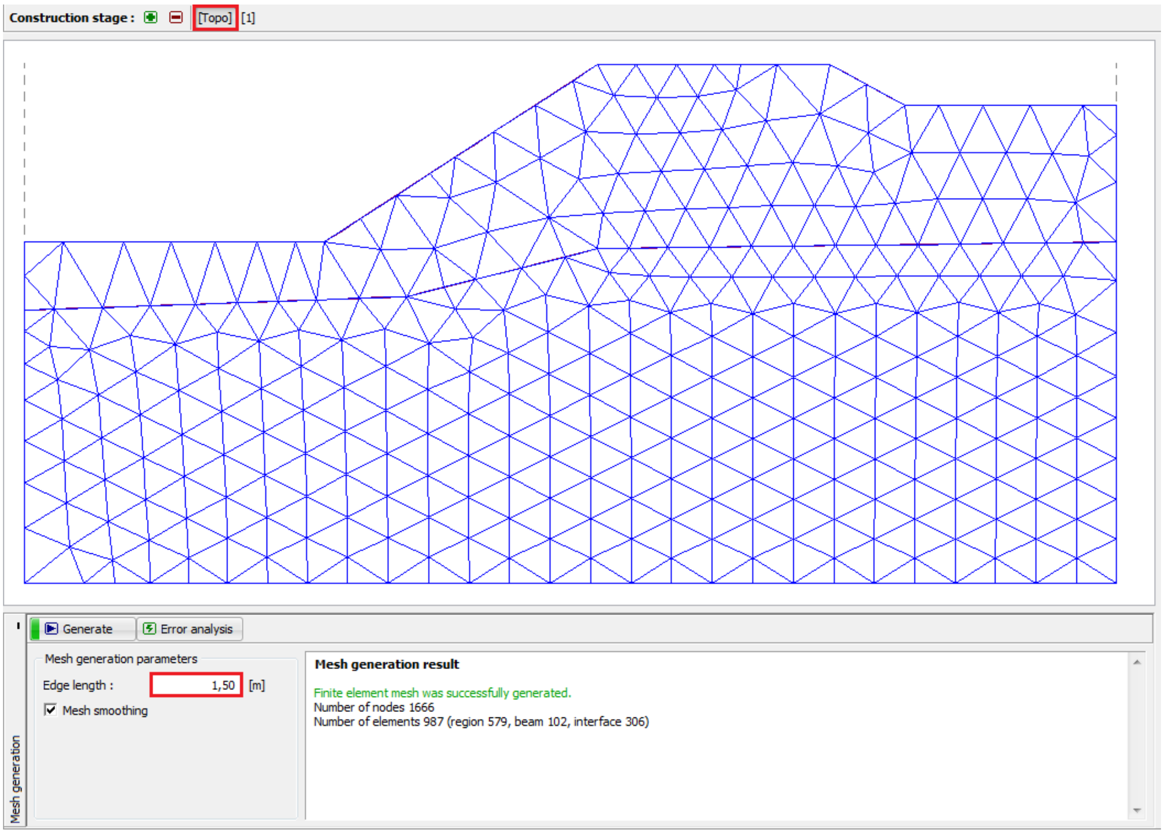
The following diagram shows the assignment of soils to the geological profile.



"Assign" frame

This is the last step in the process of setting the topology is the generation of the finite element mesh. The mesh density significantly affects the resultant degree of stability (factor of safety) and it is therefore always necessary to choose a sufficiently fine mesh.

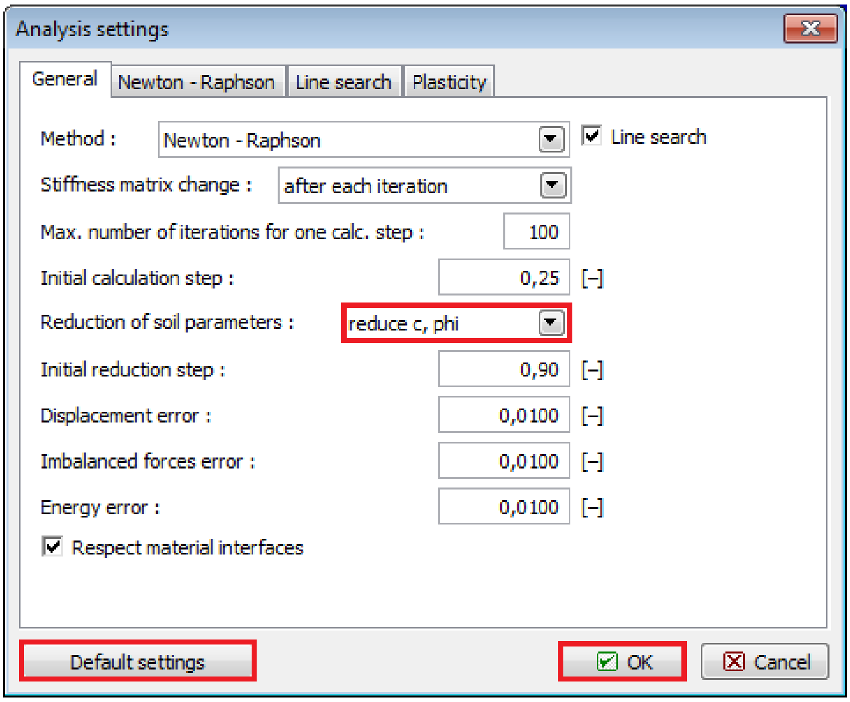
For this particular example we will choose the element edge length of 1.5 m and will generate the mesh (using "**Generate**" button). The results are obtained with the GEO 5 – FEM program for the mesh with the element edge lengths of 1.0, 1.5 and 2.0 m presented in a table at the end of this chapter.



"Mesh generation" frame – element edge length of 1.5 m

Construction stage 1: analysis of the degree of stability (factor of safety)

After generating the FE mesh we move to construction stage 1 and will perform the analysis (by pushing the "Analyse" button). We will keep the analysis setting as "Standard".



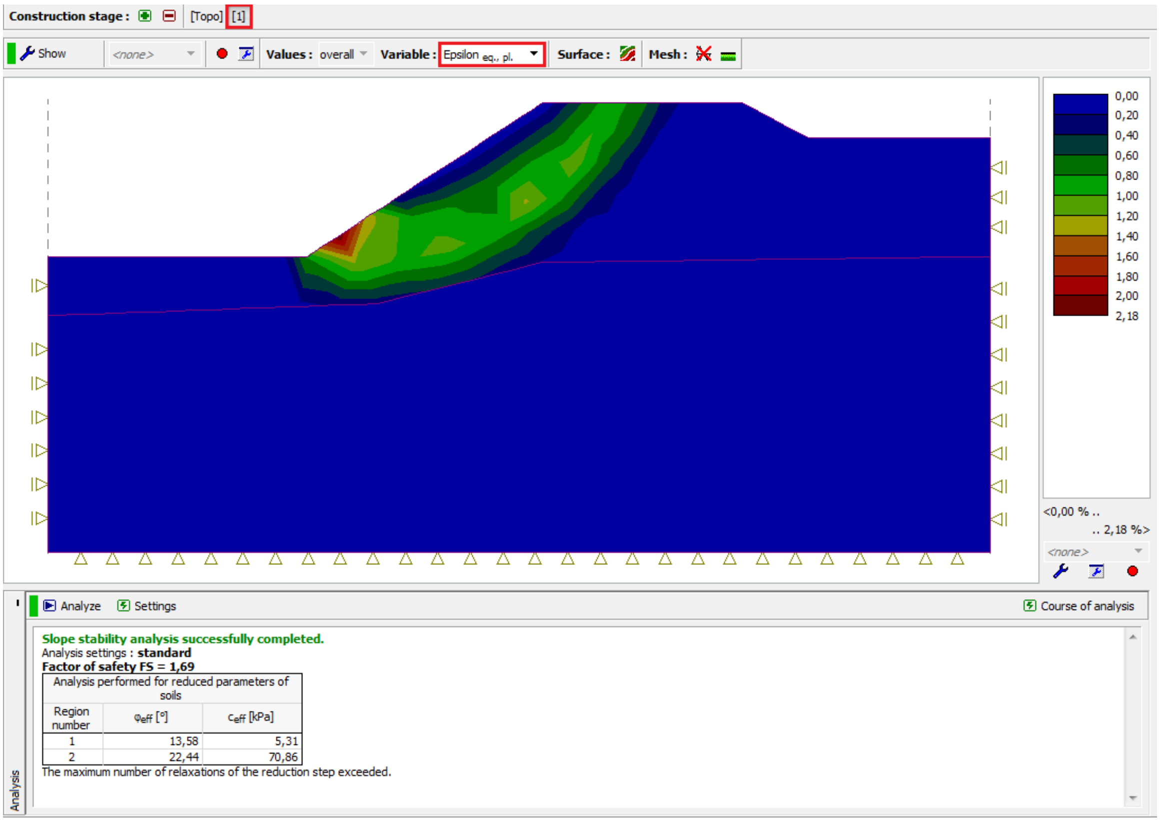
"Analysis setting" dialogue window

Note: The slope stability analysis itself is based on the c, φ ***soil strength-related parameters reduction****. The factor of safety is defined within the framework of this method as a parameter to be applied to reducing the actual values of parameters* c, φ *leading to the loss of stability (for more details visit Help – F1). The slope stability degree is defined in the program by the relationship:* $FS = \tan\varphi^{s} / \tan\varphi^{p}$,

where: φ^{s} *– real value of internal friction angle,*

φ^{p} *– value of internal friction angle at failure.*

One of outputs very useful for stability analyses is the plot of displacement vectors and equivalent plastic strain $\varepsilon_{eq.,pl.}$. Plastic strains show the shape and magnitude of the potential failure surface (see the *following diagram*).

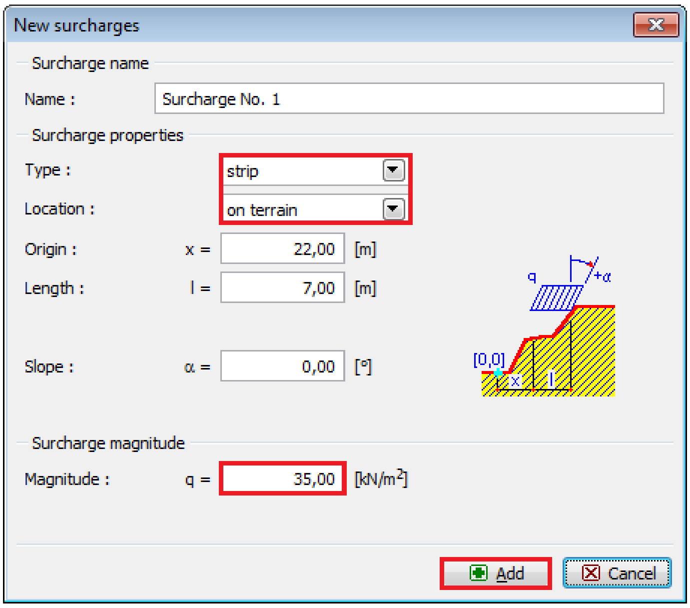


"Analysis" frame – Construction stage 1 (equivalent plastic strains $\varepsilon_{eq.,pl.}$)

Note: The "Stability" mode program allows for plotting the displacements (in the Z and X directions) and strains (total or plastic) only. The structure deformation corresponds to the analysis state for reduced soil parameters, therefore it has nothing in common with the real deformation – it only gives a picture of the entire slope behaviour or the structure at the moment of failure (for more details visit Help – F1).

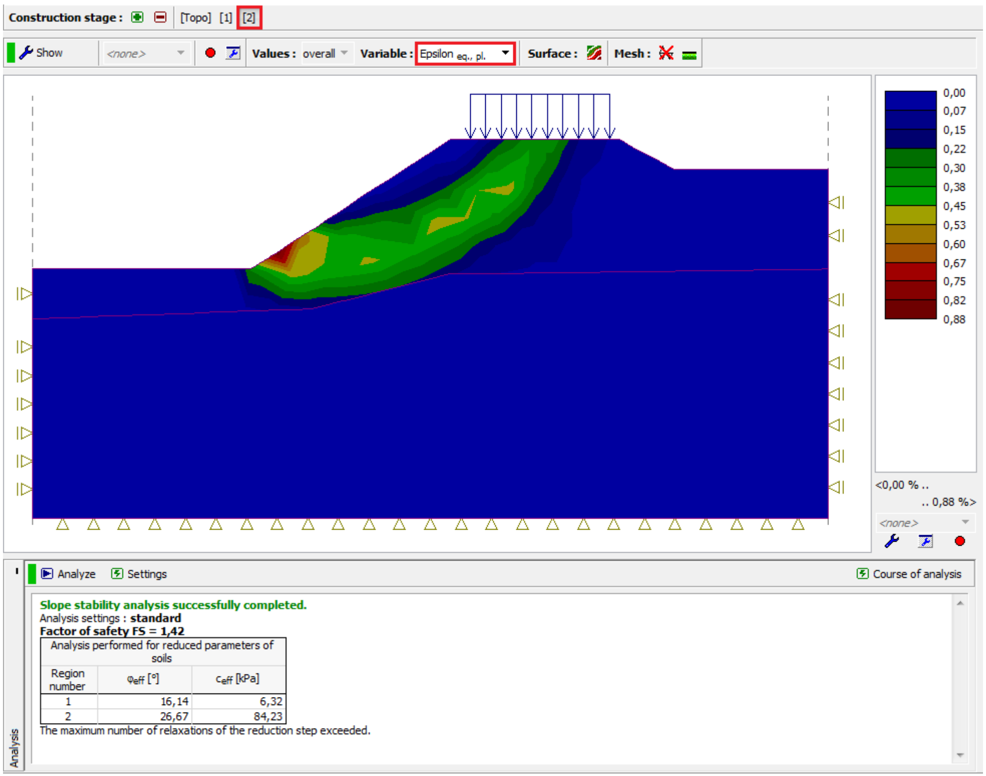
Construction stage 2: adding the slope surcharge, analysis

At this construction stage we will first go over to "Surcharge" frame to define the following parameters – the surcharge characteristics and magnitude.



"New surcharges" dialogue window – Construction stage 2

Now we will carry out the analysis of stage 2 and will examine the equivalent plastic strains.



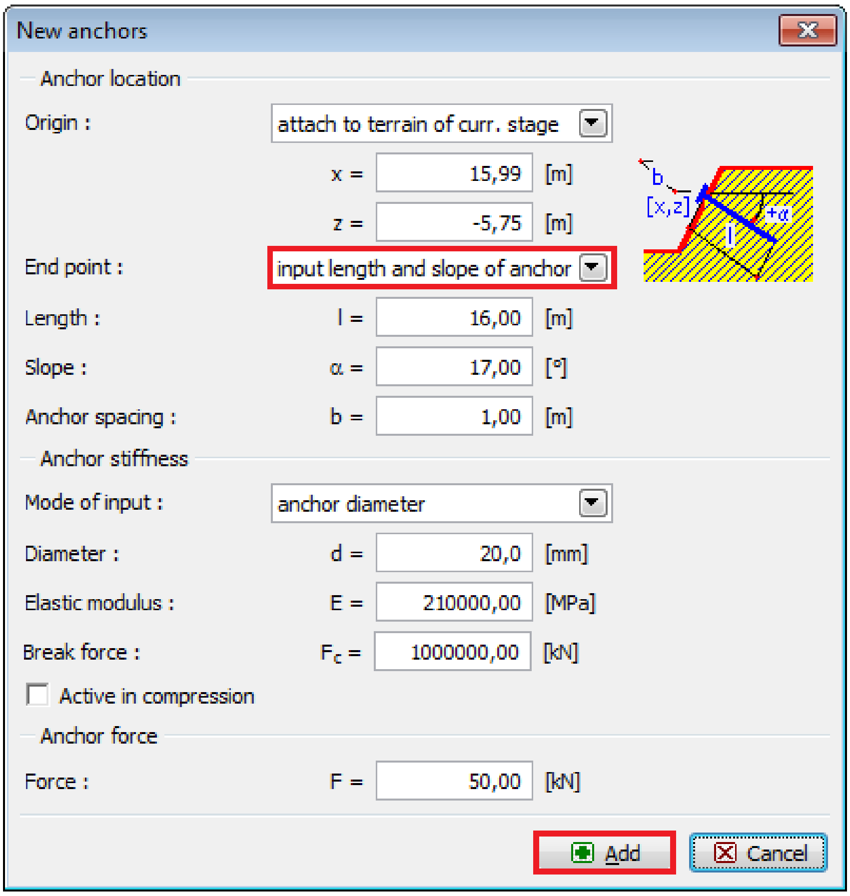
Region number	φ_{eff} [°]	c_{eff} [kPa]
1	16,14	6,32
2	26,67	84,23

"Analysis" frame – Construction stage 2 (equivalent plastic strains $\varepsilon_{eq.,pl.}$)

Construction stage 3: slope stabilisation with anchors, analysis

We will add construction stage 3. Then we will push "Add" button in "Anchors" frame and will set a steel anchor with the pre-stressing force $F = 50\ kN$ in "New anchors" dialogue window. We will consider the following anchor parameters for this problem:

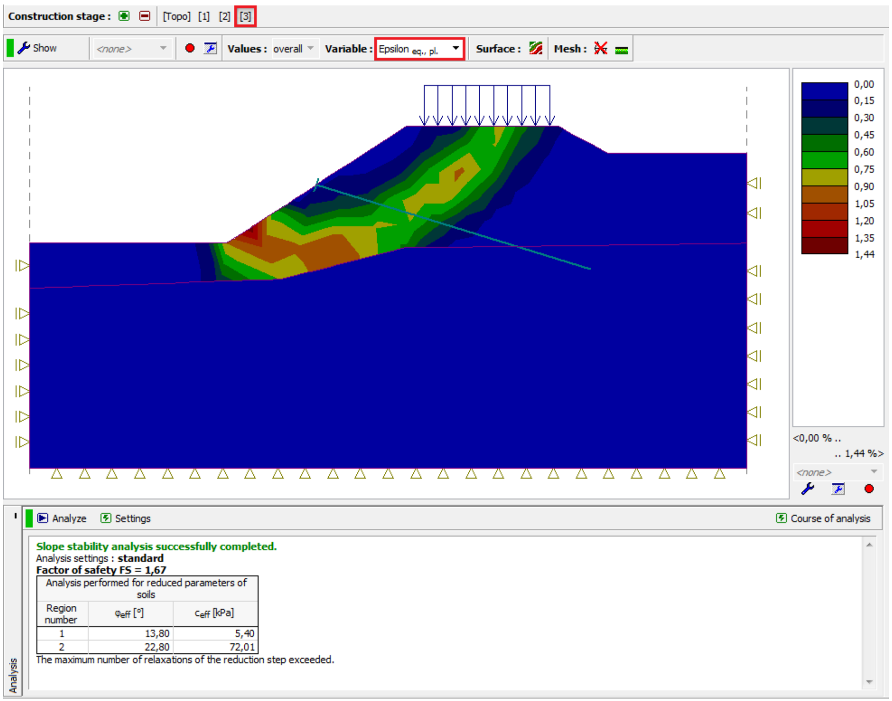
- Anchor length: $l = 16\ m$,
- Anchor slope: $\alpha = 17\ °$,
- Anchor diameter: $d = 20\ mm$,
- Anchor spacing: $b = 1\ m$.



"New anchors" dialogue window – Construction stage 3

Note: At the slope stability analysis, pre-stressed anchors enter the analysis itself as surcharge loading induced by the force acting at the anchor head – the anchor stiffness therefore has no effect on stability. However, soil at the anchor head may get plasticised. It is therefore necessary after the analysis is finished to verify the location and reality of plastic strains representing the shear surface. In the case of the soil under the anchor head getting plasticised, it is necessary to edit the model (for more details visit Help – F1).

The other input parameters will remain unchanged. Now we will carry out the analysis of construction stage 3 and will again examine the solution results (similarly to the previous construction stage).



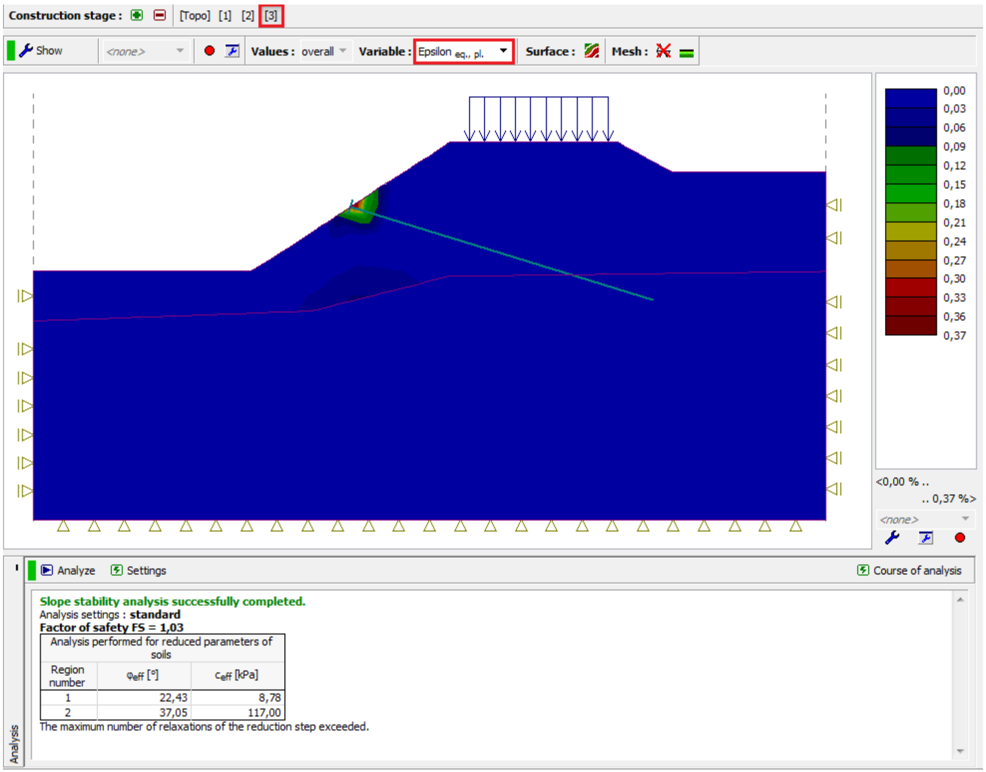
"Analysis" frame – Construction stage 3 (equivalent plastic strains $\varepsilon_{eq.,pl.}$)

This step concludes the initial analyses. We will record the resultant values for the slope stability degree in a summary table; now we will carry out the assessment of the given problem for other material models (Mohr-Coulomb and Modified Mohr-Coulomb).

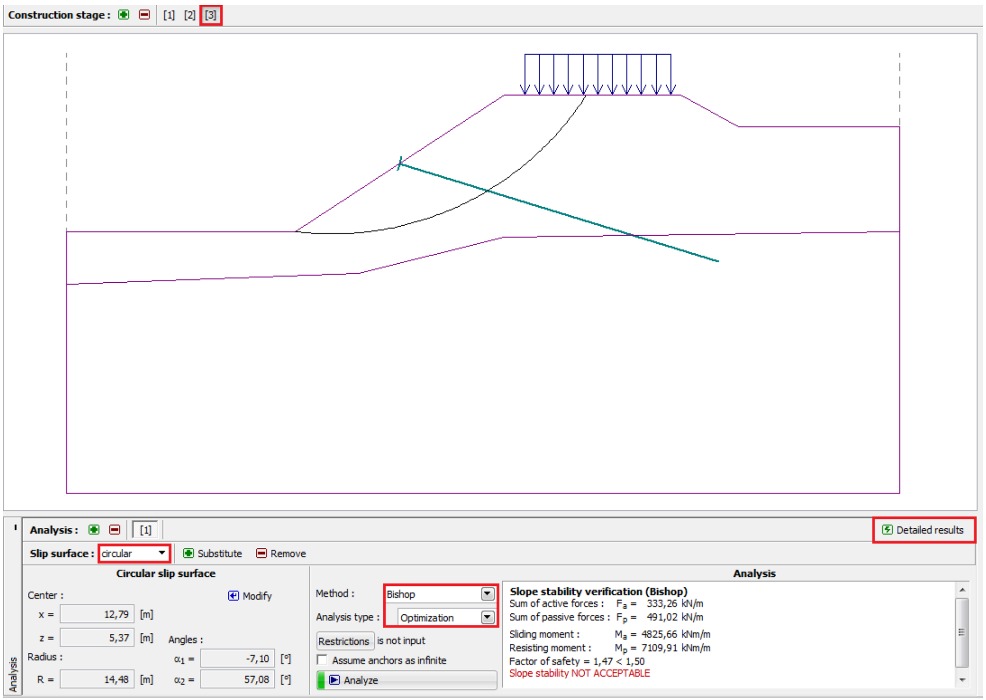
Note: The verification of the shear surface shape is very important in some cases, because a local failure of the structure may take place even in different areas than we expect (for more details visit Help – F1). In the following diagram, we can see the evolution of the localized plastic zone of soil in the vicinity of the anchor head during the course of the analysis with the mesh density of 1.0m for Drucker-Prager model. If this case happens, it is reasonable to edit the structure model, for example as follows:

- *to increase the mesh element edge length,*
- *to introduce a soil with higher strength-related parameters c, φ at the anchor head,*

– *to define beam elements at the anchor head (the distribution of load into the soil will be improved).*



"Analysis" frame – Construction stage 3 (localized plasticity of soil under the anchor head, DP model with 1.0 m mesh)



"Analysis" frame – Construction stage 3 (analytical solution according to Bishop with optimisation of the circular slip surface)

Assessment of results:

The following table presents the slope stability degree (factor of safety) results for individual construction stages. We carried out the calculation for some non-linear material models in GEO 5 – FEM program and various FE mesh densities. For comparison, we also present the results achieved by GEO 5 – Slope stability program (according to Bishop and Spencer).

Material model	**Mesh spacing** $[m]$	**Stage 1** *FS*	**Stage 2** *FS*	**Stage 3** *FS*	**Note**
DP	1.0	1.67	1.44	1.03 *	* Soil under the anchor head gets plasticised.
DP	1.5	1.69	1.42	1.67	
DP	2.0	1.74	1.48	1.69	
MC	1.0	1.56	1.35	0.90 *	* Soil under the anchor head gets plasticised.
MC	1.5	1.58	1.35	1.56	
MC	2.0	1.60	1.41	1.56	
MCM	1.0	1.78	1.56	1.14 *	* Soil under the anchor head gets plasticised.
MCM	1.5	1.81	1.54	1.78	
MCM	2.0	1.85	1.60	1.81	
BISHOP (analytical solution)	---	1.51	1.33	1.47	see below
SPENCER (analytical solution)	---	1.51	1.32	1.52	see below

Summary of results – slope stability degree (factor of safety)

Note: We considered the analysis settings as "Standard – Factors of safety". We carried out the analysis first according to Bishop and then according to Spencer with the optimisation of a circular shear surface (without limitation).

Conclusion:

The following conclusion can be drawn from the numerical solution results:

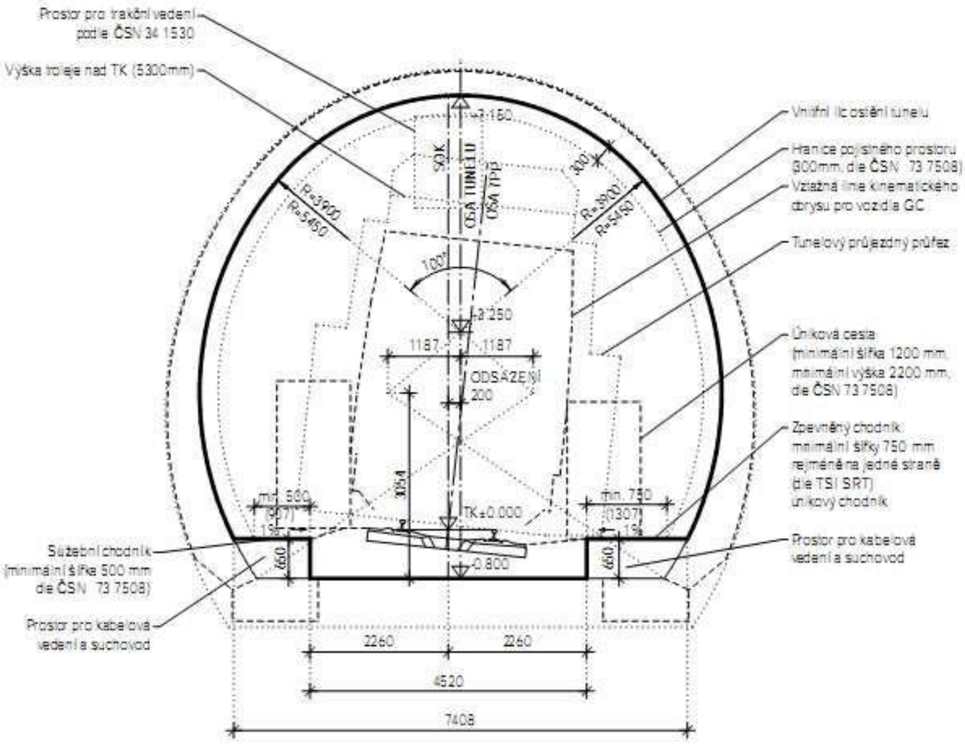
- The locally increased density of the FE mesh leads to more accurate results; on the other hand, the duration of the analysis for each construction stage is extended.
- It is necessary for the analyses to use non-linear material models, allowing for the development of plastic strains.
- Maximum equivalent plastic strains $\varepsilon_{eq.,pl.}$ express the locations where a potential shear failure surface is located.
- Drucker-Prager material model features slightly more yielding response than Mohr-Coulomb model.

Chapter 26. Numerical modelling of tunnel excavation using the NATM method

The objective of this manual is to describe numerical modelling of a single-track railway tunnel using the Finite Element Method.

Problem specification:

Develop a model and assess the primary lining of a single-track railway tunnel for speeds ranging from 160 to 230km/h. The tunnel cross-section is designed on the basis of the respective SZDC (Railway Infrastructure Administration, state organisation) Standard Sheet see *the picture*).



Net tunnel profile for a single-track railway tunnel according to SZDC Standard Sheet

The tunnel will be driven using a conventional tunnelling method (the New Austrian Tunnelling Method, the Sequential Excavation Method) with the excavation sequence consisting of top heading, bench and invert (the so-called horizontal sequence). The overburden is about 14 meters high. The 200 mm thick primary lining is in C 20/25 sprayed concrete. The excavation crown is supported with hydraulically expanded rockbolts (HUIS, type WIBOLT EXP) with the capacity of 120kN. We assume on the basis of the assessment of the stages of survey operations that geological layers are parallel with the terrain surface. The composition of the geological profile is obvious from *Table 1*.

Table 1 – Parameters of soils and rocks

Soil, rock (specification)	Profile $[m]$	γ $[kN/m^3]$	φ_{ef} $[°]$	c_{ef} $[kPa]$	ν $[-]$	E_{def} $[MPa]$	E $[MPa]$
Silty sand (S4 / SM)	0 – 3	19,5	29	10	0,3	10	30
Silty gravel (G4 / GM)	3 – 5	19,5	33	8	0,3	70	210
Heavily weathered slate (R5)	5 – 10	24	29	30	0,33	45	135
Slightly weathered slate (R3)	over 10	26	38	250	0,25	350	1050
Anchored region (R5)	–	24	29	63	0,33	45	135

Solution:

We will apply the GEO 5 – FEM program to the analysis of this problem. In the text below, we will step by step describe the solution to this example:

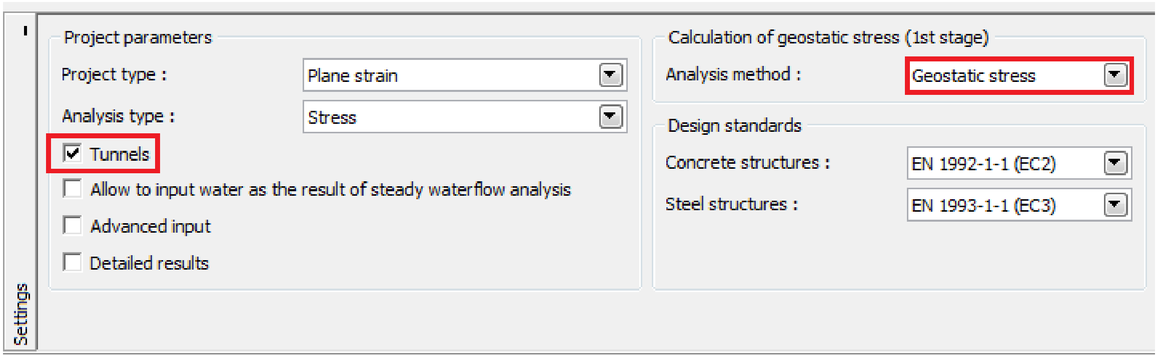
- Topology: the problem settings and modelling (contact elements, modelling of the lining)
- Modelling the construction procedure: primary tunnel lining material, Excavation steps
- Construction stage 1: primary geostatic stress state of rock massif
- Construction stage 2: modelling of the top heading excavation, activation of the unsupported excavated opening
- Construction stage 3: supporting the top heading vault with an immature concrete primary lining
- Construction stage 4: improving material characteristics of mature concrete (top heading)
- Construction stage 5: modelling of the tunnel bench excavation, activation of the unsupported excavated opening
- Construction stage 6: supporting the bench side-walls with an immature concrete primary lining
- Construction stage 7: improving material characteristics of mature concrete (bench)
- Results, conclusion: terrain surface settlement trough, rock massif deformation, distribution of internal forces and displacement of the primary tunnel lining, forces in anchors.

Note: The modelling in GEO5 – FEM program consists of two parts: In the first part, it is necessary to define the magnitude of the numerical model itself in the topology mode, specify the interfaces between soils and rocks, define the tunnel structure geometry by means of points and lines and assign them to respective interfaces of the model (for more details visit Help – F1).

In the second part, individual construction stages are defined and calculations themselves are carried out. During the course of individual stages, our aim is to model the actual course of the construction of the particular underground structure by activating, deactivating or changing materials in pre-circumscribed regions of the model, by adding or removing beam elements representing the structures (e.g. the tunnel lining) and by changing their parameters (material, dimensions). We will achieve a numerical model for which we assume that its behaviour will be identical with the future behaviour of the real underground structure and which will be used for dimensioning of the tunnel structure.

Topology (Part 1): specifying the problem (profile) interfaces and soil parameters

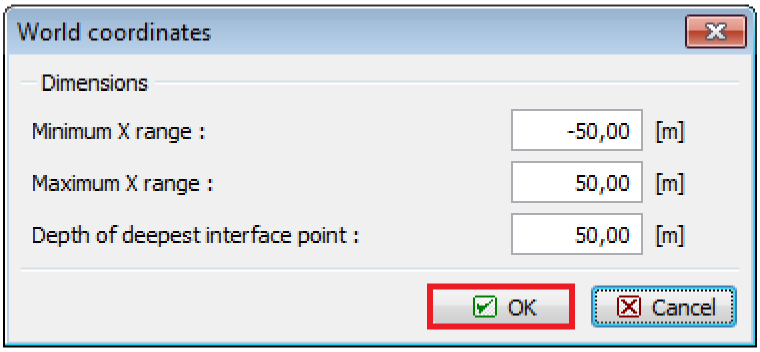
In the "Settings" frame, we will leave the analysis of the Construction Stage 1 to be the geostatic stress state. The "*Stress*" analysis type is applied. In addition, we will switch the "Tunnels" mode, which allows us to model the realistic course of the primary tunnel lining construction.



Frame „Settings"

Note: In the case of the "Tunnels" mode selection, it is possible to use the program for modelling, for example, the Excavations (modelling the 3D effect of the excavation face using the New Austrian Tunnelling Method), specifying and analysing the degradation of beams, the thermal loads acting of beams and regions, the loads acting on the regions induced by swelling and conducting the monitoring of results (for more details visit Help – F1).

Further we will specify the magnitude of the numerical model of the problem being solved and the terrain interface. Regarding this particular problem, we will select the model dimensions $\langle -50\text{ m}; 50\text{ m}\rangle$ and specify the thickness of the layer being investigated at 50 m.



Dialogue window „World coordinates“

Note: The interface of the problem being analysed, or the coordinates of the world, must be selected sufficiently great so that the stress state and rock massif deformations in the location of the structure being analysed (or in the regions of interest) are not affected by boundary conditions of the model. The guidance values of the recommended dimensions of the boundaries of the models are presented for individual cases of the solutions and described in a more detailed way in the program Help (for more details visit F1).

Interface 1		Interface 2		Interface 3		Interface 4	
x [m]	z [m]	x [m]	z [m]	x [m]	z [m]	x [m]	z [m]
-50,0	22,0	-50,0	19,0	-50,0	17,0	-50,0	12,0
50,0	22,0	50,0	19,0	50,0	17,0	50,0	12,0

List of points for individual interfaces between soils and rocks

In the “Soils” frame, we will define the parameters of soil or rock layers as well as the parameters of the rock in the region in which the rockbolts are located (see the *note*). We applied the Mohr-Coulomb material model to the modelling of the problem. It allows us to take the regions of local or global failures into consideration (for more details visit Help).

Note: The rockbolts are introduced into the numerical model using a method in which the region of the rock mass reinforced with rockbolts in the vicinity of the excavated opening corresponding to the length of rockbolts is replaced with rock exhibiting better material parameters. In such the cases an increase in the rock cohesion is usually assumed. The overall cohesion of rock increased by the action of rockbolts is given by:

$$c_{h+s} = c_h + c_s \; [\mathrm{kPa}]$$

where: c_{h+s} *overall cohesion of rock increased by the action of rockbolts,*

c_h *original cohesion of rock,*

c_s *increase in the cohesion by the action of rockbolts.*

The increase in cohesion by the action of rockbolts is calculated in accordance with the following relationship:

$$c_s = \frac{N_u}{A_k} \cdot \frac{1+\sin\varphi_{ef}}{2\cdot\cos\varphi_{ef}} \cdot \frac{1}{\gamma_{kc}} = \frac{120}{2.058} \cdot \frac{1+\sin 29°}{2\cdot\cos 29°} \cdot \frac{1}{1,5} = 33.0 \text{ kPa}$$

where: N_u *rockbolt capacity* $[\mathrm{kN}]$*,*

A_k *region allotted to one rockbolt* $[\mathrm{m}^2]$*,*

φ_{ef} *angle of internal friction of rock* $[°]$*,*

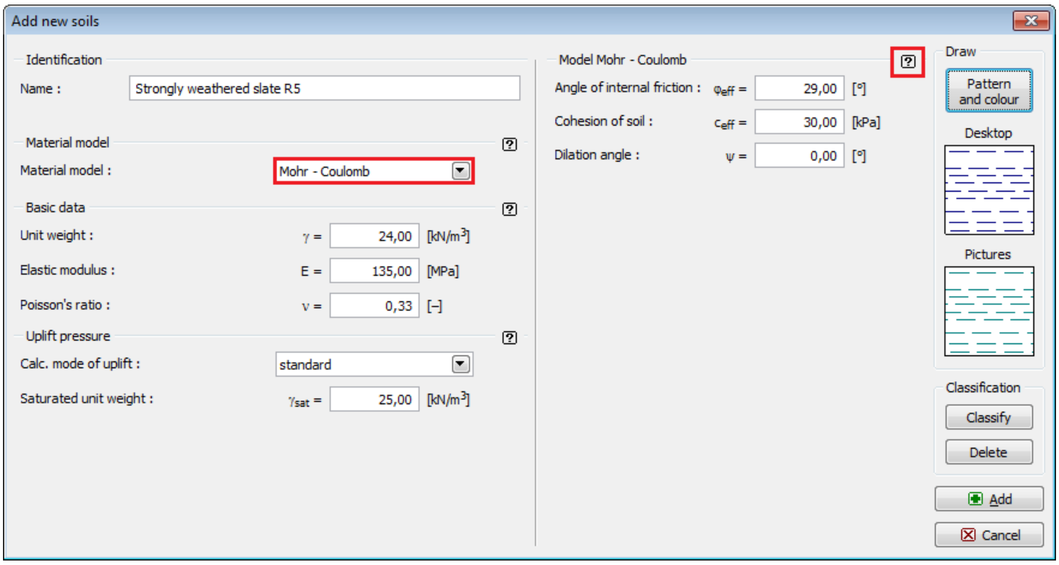
γ_{kc} *anchoring reliability factor* $[-]$.

In this problem, we take ***10 pieces of HUIS rockbolts with the capacity of 120 kN, spaced at 3.5 m,*** *into consideration. The resultant shear strength, or cohesion, in the region reinforced with anchors corresponds to R5 rock type:*

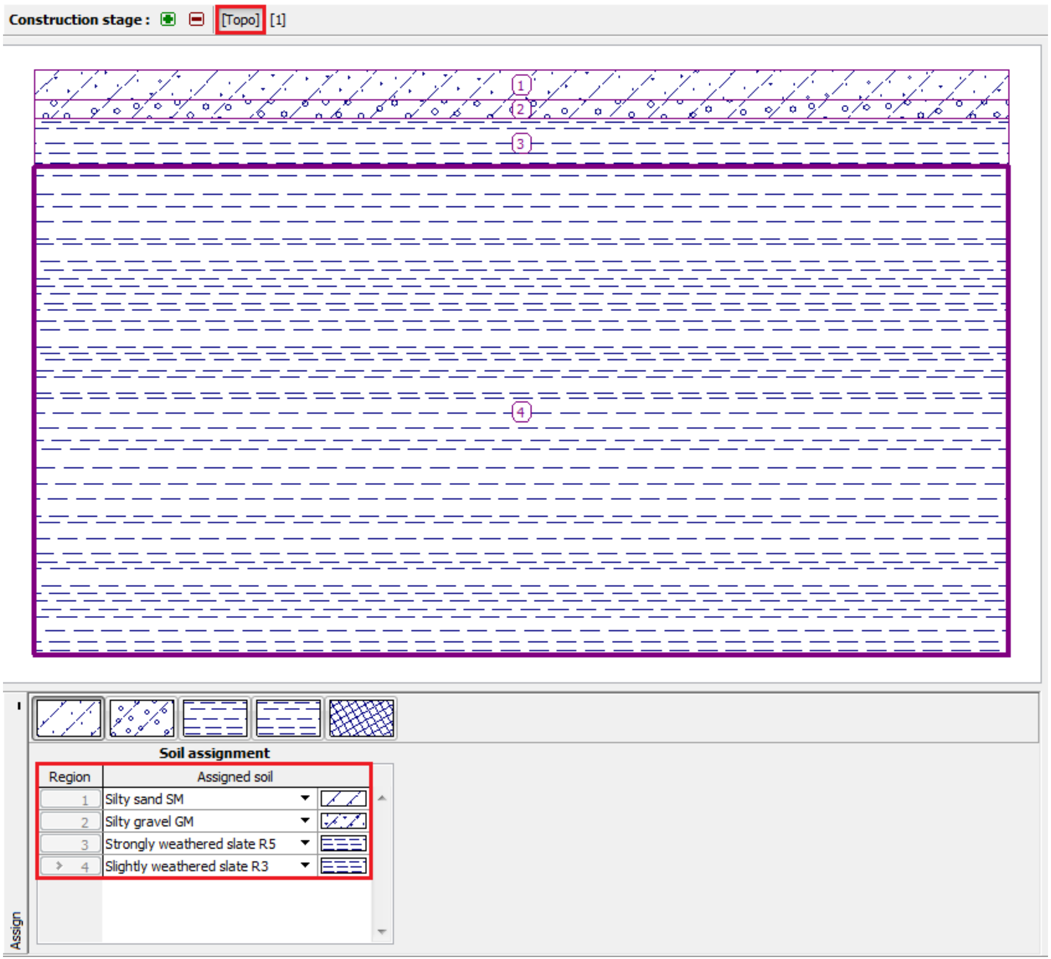
$$c_{h+s} = c_h + c_s = 30 + 33 = 63 \text{ kPa}$$

The modulus of elasticity $E\,[\mathrm{MPa}]$ was not determined directly by the geological survey. For that reason its value was derived from the modulus of deformation $E_{def}\,[\mathrm{MPa}]$ using a general relationship $E = 3 \cdot E_{def}$.

We will take zero dilatancy angle $\psi\,[°]$ into consideration for all layers of soils and rocks. Subsequently we will assign the soils and rocks into individual regions (see the following *picture*).



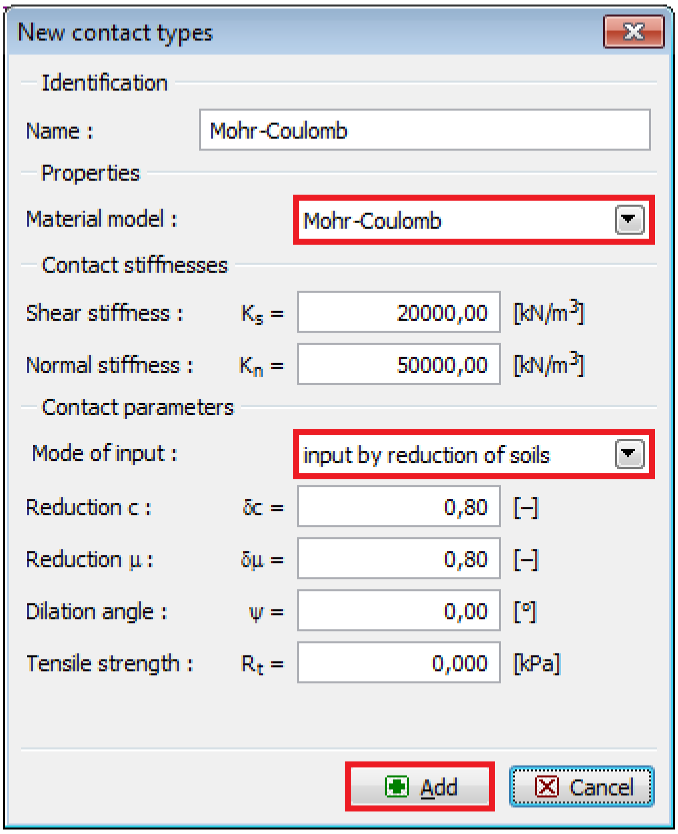
Dialogue window „Add new soils“



Frame „Assign“

The next step is to specify the type of contact elements which are introduced at the interface between the lining and the soil or rock in the "Contact types" frame. We assume the following parameters of contacts to exist at the interface:

- Shear strength: $K_s = 20{,}000 \ \mathrm{kN/m^3}$,
- Normal stiffness: $K_n = 50{,}000 \ \mathrm{kN/m^3}$,
- Reduction for soils: $\delta c = \delta\mu = 0.8$.



Dialogue window „New contact types"

Note: The contact elements allow for making provisions for the interaction between materials found along the interface between the soil and the structure, or between individual soil types etc. The thickness of a contact element is zero. The element expresses the relationship between contact stresses and relative changes in displacements along the contact (for more details visit Help – F1).

In this case we consider the contact elements to be located at the interface between the primary lining and rock, i.e., we consider a certain possibility of the primary lining shifting on the surface of the excavated opening.

Contact elements are generally introduced in less competent soils; they can be disregarded, with some caution, for fresh unbroken rocks (in the cases of tunnel structures). The problems and the method of introducing contact elements have been described in a more detailed way in Chapter 24 Numerical solution to a bracing structure (for more details see http://www.finesoftware.eu/support/engineering-manuals/). The guidance values of stiffness K_s and K_n $\left[\mathrm{kN/m^3}\right]$ *are presented in Help (for more details visit F1).*

In this way the basic specification of the problem (the interface modelling, soil parameters, and contact types) is finished. Now we will pass to the modelling of the primary tunnel lining and, subsequently, to the specification of the region reinforced with anchors.

Topology (Part 2): modelling of the lining and the region reinforced with rockbolts

We will pass to the "Lining" frame and, using the "Add" button, will specify the primary tunnel lining points, the excavation geometry and its location in the region being solved. We will assume the lining thickness of 200 mm, taking into consideration the particular structure type (the lining of a railway tunnel is being modelled).

Note: We can specify the tunnel lining in the program by means of individual points and lines or we can define it as the so-called macro-element. The advantage of the latter solution lies in the fact that when the geological profile changes, we can shift the lining as a whole in an arbitrary way (horizontally or vertically). For more details visit our Help – F1.

The excavated cross-section geometry is specified in the "Lining – FEM" module by means of ten free points (see *Table* 2), which are interconnected by free lines (see *Table 3*).

Table 2 – Free points of excavation (primary lining)

Point No.	Location $x\,[\mathrm{m}]$	Location $y\,[\mathrm{m}]$
1	4.81	2.25
2	3.41	6.11
3	-3.41	6.11

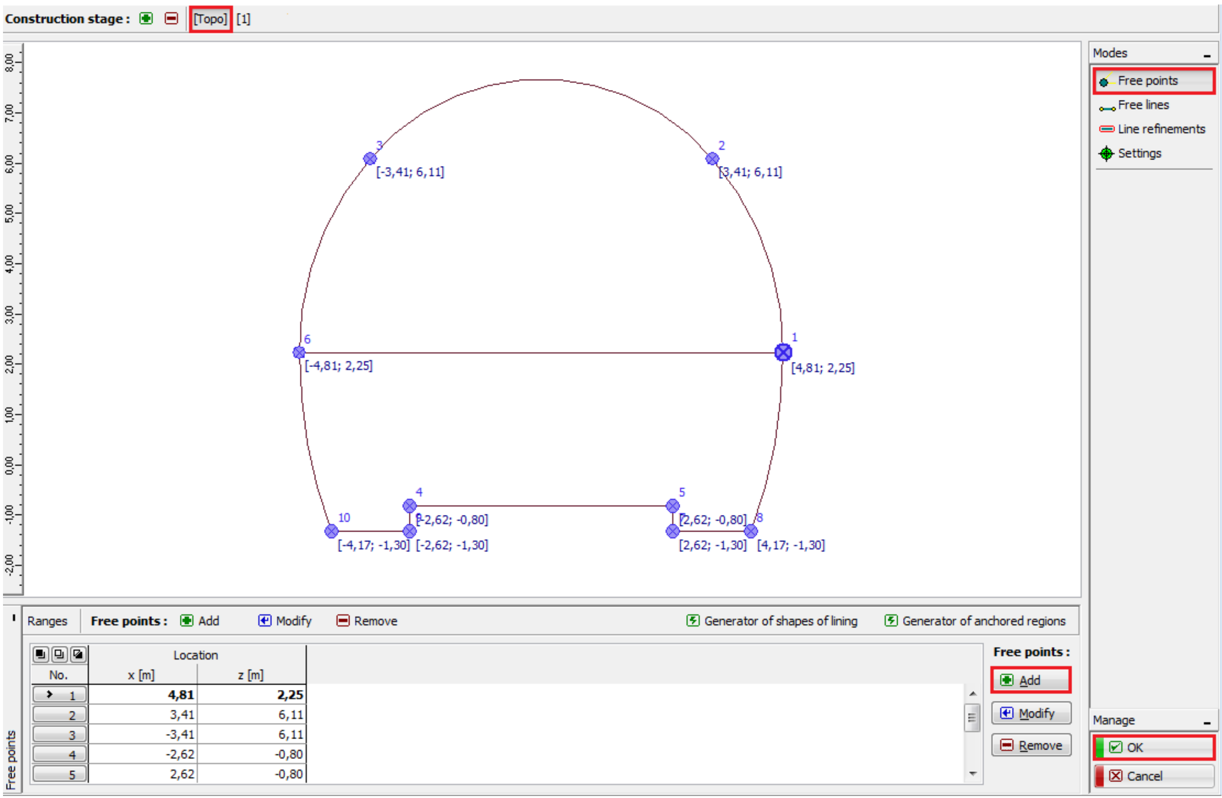
4	-2.62	-0.80
5	2.62	-0.80
6	-4.81	2.25
7	2.62	-1.30
8	4.17	-1.30
9	-2.62	-1.30
10	-4.17	-1.30

Note: The generators of the lining geometries generate respective elements according to the parameters. The elements are subsequently worked with independently, without any possibility of subsequent changing their parameters. If the generation parameters are correct, the current graphical form of the generated elements is shown when the parameters are being modified (for more details visit Help – F1).

Table 3 – Free lines of excavation (primary lining)

Line No.	Type of line	Mode of input	Lines topology
1	arc	centre	Origin – point 1, end – point 2 centre (-1,19; 2,25), orientation positive
2	arc	centre	Origin – point 2, end – point 3, centre (0,00; 3,25), orientation positive
3	segment	–	Origin – point 4, end – point 5
4	arc	centre	Origin – point 3, end – point 6, centre (1,19; 2,25), orientation positive
5	segment	–	Origin – point 7, end – point 8
6	arc	centre	Origin – point 1, end – point 8, centre (-5,39; 2,25), orientation negative
7	segment	–	Origin – point 5, end – point 7
8	segment	–	Origin – point 9, end – point 10
9	arc	centre	Origin – point 10, end – point 6, centre (5,39; 2,25), orientation negative
10	segment	–	Origin – point 4, end – point 9
11	segment	–	Origin – point 6, end – point 1

The free points on the excavation contour line for the primary lining are drawn in the following picture.



Module „Lining – FEM" – Free points of excavation (with horizontal sequencing)

The excavation crown support using rockbolts is taken into consideration during the construction of the underground structure. This support is usually modelled in the engineering practice as the improvement of parameters of the rock which is found in the particular region. For that reason it is in addition necessary in this case to specify the region reinforced with rockbolts – by means of free points (see *Table 4*) and free lines (see *Table 5*).

Table 4 – Free points near anchored region with hydraulically expanded rockbolts

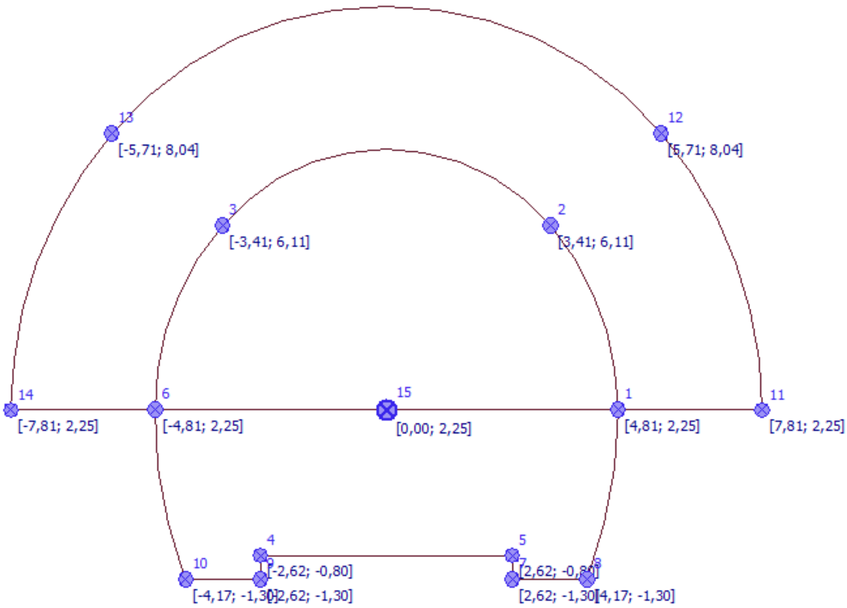
Point No.	Location x [m]	Location y [m]
11	7.81	2.25
12	5.71	8.04
13	-5.71	8.04
14	-7.81	2.25

Table 5 – Free lines near anchored region with hydraulically expanded rockbolts

Line No.	Type of line	Mode of input	Lines topology
12	arc	radius	Origin – point 14, end – point 13 Radius – 9,0 m, orientation – negative Angle – acute
13	arc	radius	Origin – point 13, end – point 12 Radius – 7,45 m, orientation – negative Angle – acute
14	arc	radius	Origin – point 12, end – point 11 Radius – 9,0 m, orientation – negative Angle – acute
15	segment	–	Origin – point 14, end – point 6
16	segment	–	Origin – point 11, end – point 1

Further, we will add a new free point No. 15 with the coordinates $[0,0;\ 2,25]$ in the “Lining” module and will subsequently increase the density of the mesh of finite elements around it (see *Topology –Part 3*).

Now we will examine the resultant geometry of the primary lining of the tunnel being modelled, together with the region reinforced with rockbolts. We will place the lining in the space of the region being solved to the coordinate system origin, i.e. to the $[0,0]$ coordinate, using the “Settings” frame. We will confirm the end of the specification of the points in the “Lining” module by means of the “OK” button.



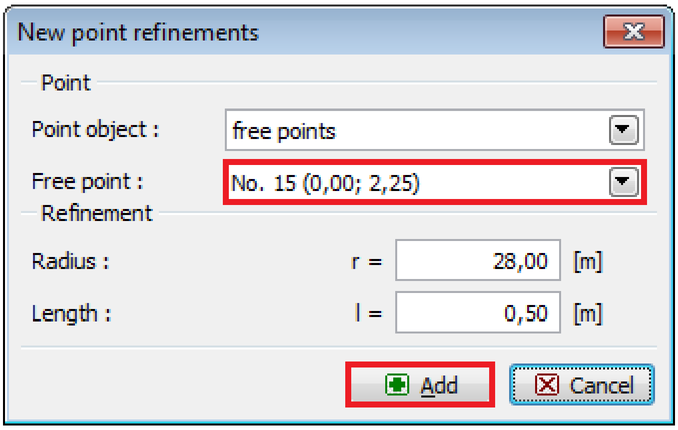
Free points near anchored region and free points of primary lining

In the last part of the structure topology settings we will pass to generating the finite element mesh and increasing its density.

Topology (Part 3): generation of the finite element mesh and increasing its density

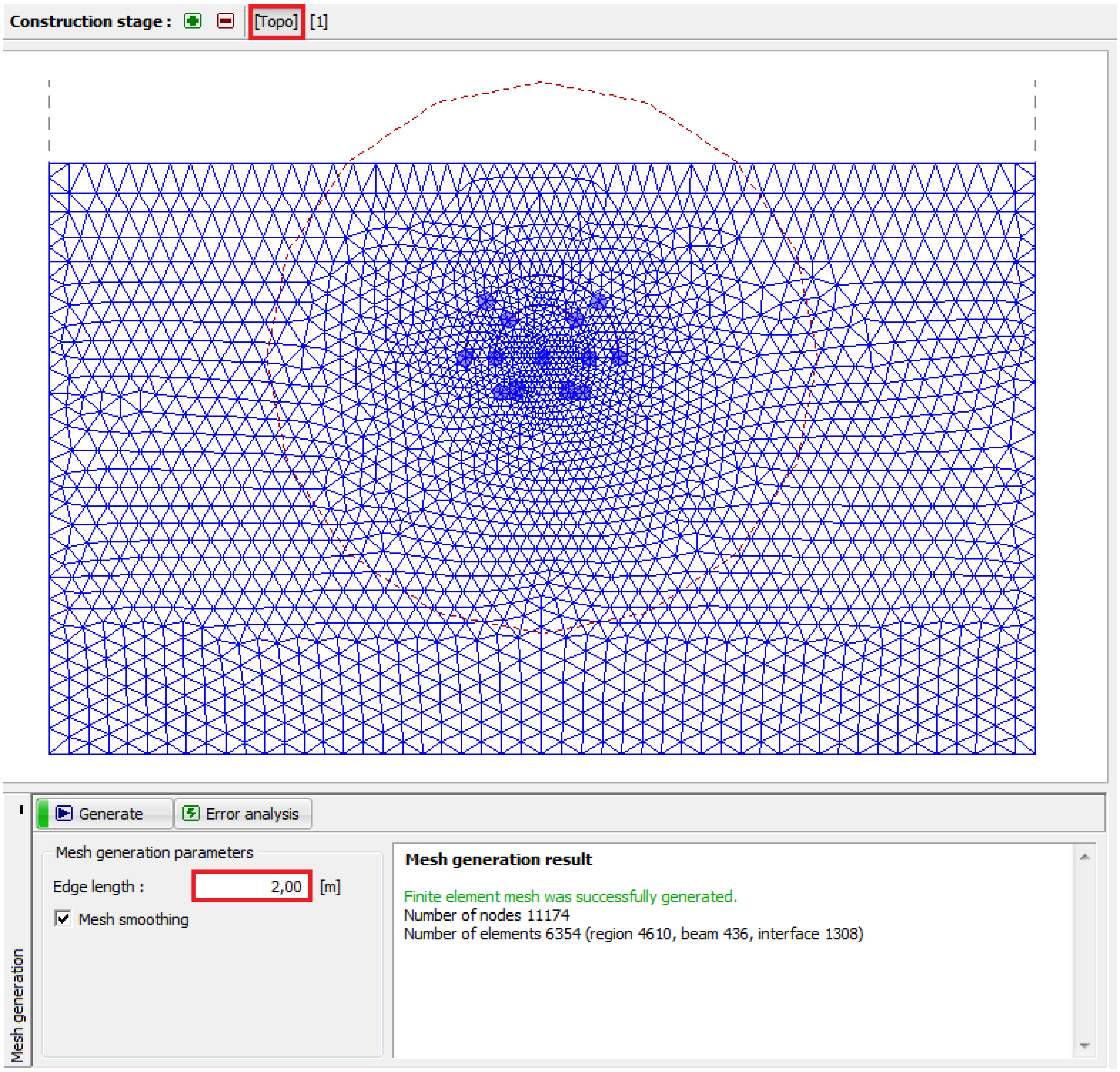
The finite element mesh significantly affects the resultant calculation values. Prior to the mesh generation itself, we will increase the density in the excavated space (around point No. 15) with the element edge length $l = 0.5\ \mathrm{m}$ and the reach radius $r = 28\ \mathrm{m}$.

Note: Through this step we will provide a sufficiently dense mesh in the surroundings of the region of interest (the excavated opening). The process of increasing the density of free points or lines has been described in more detail in Chapter 23. Collector lining analysis (for more details see http://www.finesoftware.eu/support/engineering-manuals/).



Dialogue window „New point refinements“

Subsequently we will pass directly to the generation of the FEM mesh. In the “Mesh generation” frame, we will set the length of the elements edge at 2.0 m and will select the “Mesh smoothing” option.



Frame „Mesh generation" – Refinement of points near excavation region (length 0.5 m)

Notes on the modelling of the construction procedure:

In this part of the manual we present for easy reference important notes relating to the construction procedure itself – the primary tunnel lining material, the excavation sequence (individual excavations). This information is useful for the numerical modelling of our example because some input data is repeated (e.g. the excavations).

Note: The construction stages take into consideration the tunnelling process. To be able to compile individual construction stages, we need to know the primary tunnel lining material, the excavation sequence and hydro-geological conditions to be encountered during the course of the tunnel excavation.

*We will design the **primary lining** to be 200 mm thick, in C 20/25 sprayed concrete. We will introduce into the numerical model only sprayed concrete and the influence of the strength, or the modulus of elasticity, increasing with time (see Table 6).*

Table 6 – Values of modulus of elasticity for sprayed concrete (development with time)

Maturation of sprayed concrete	Modulus of elasticity E_{cm} $[\mathrm{MPa}]$	Shear modulus G $[\mathrm{MPa}]$
Immature concrete	2 900	1 134
Mature concrete	29 000	11 340

Note: The tunnel excavation is modelled as a 2D problem, which does not fully allow for the spatial changes in the stress state of the rock mass which take place during the course of the excavation in the region of the excavation face. During the course of the excavation operation, the temporarily unsupported excavated opening is supported by the face-advance core (the longitudinal and transverse rock arch) and by the part of the excavated opening previously provided with the support. This behaviour can be described only by a 3D model; in a 2D model the behaviour in the direction of excavation progress is solved only approximately.

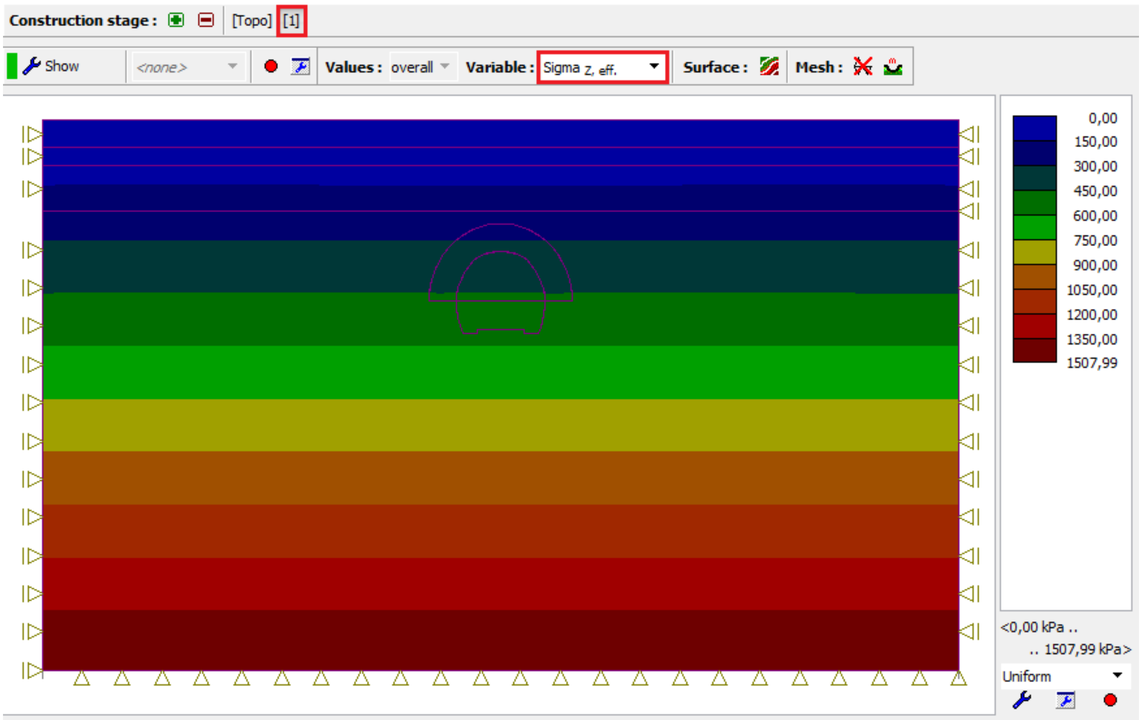
The method which is most frequently used in the engineering practice (generally called the λ method or the β method), assumes that the primary stress state of the massif, i.e. the original stress σ_0 acting before the excavation in the surroundings of the future excavated opening, gradually changes with time according the relationship $(1-\beta)\cdot\sigma_0$ (for the primary stress state $\beta = 1$). If we model a change in the primary stress at 2 calculation stages (construction stages), the unsupported excavated opening is loaded by the value of $(1-\beta)\cdot\sigma_0$ at the first stage and the remaining load of $\beta\cdot\sigma_0$

is taken into consideration at the second stage.

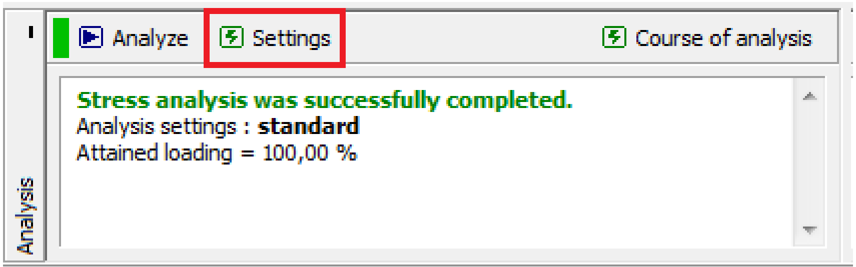
In the case of the sequential excavation system, this procedure has to be applied separately to each partial heading. The value of coefficient β *depends on rock mass geology, the advance per round and the size of the excavated profile; it is relatively hard to determine. In GEO5 – FEM, this method is represented by the so-called Excavation. We have estimated its value for the purpose of numerical modelling as* $\beta = 0.6$ *for the single-track profile for both the top heading and bench.*

Construction stage 1: primary geostatic stress state

After generating the FE mesh we will switch to construction stage 1 and will carry out the analysis of the primary geostatic stress state of the massif. We will maintain the "Standard" analysis setting for all construction stages (for more details visit Help – F1).



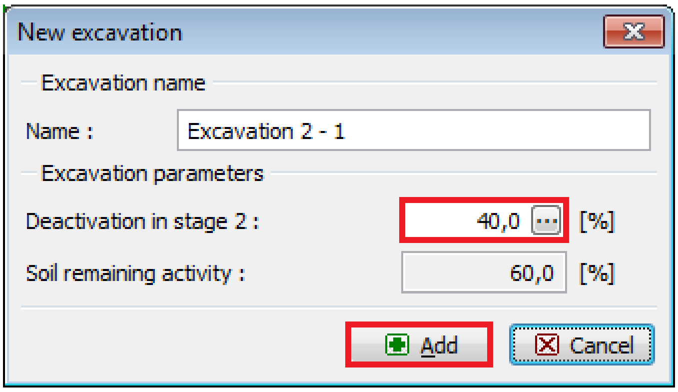
Frame „Analysis" – Construction stage 1 (primary geostatic stress $\sigma_{z,eff}$ *)*



Frame „Analysis" – Construction stage 1

Construction stage 2: top heading excavation, activation of the unsupported excavation

In the next step, we will add construction stage 2. Then we will model the top heading excavation in the "Activation" frame and will carry out the Excavation for region No. 6 (using the "Add" button).



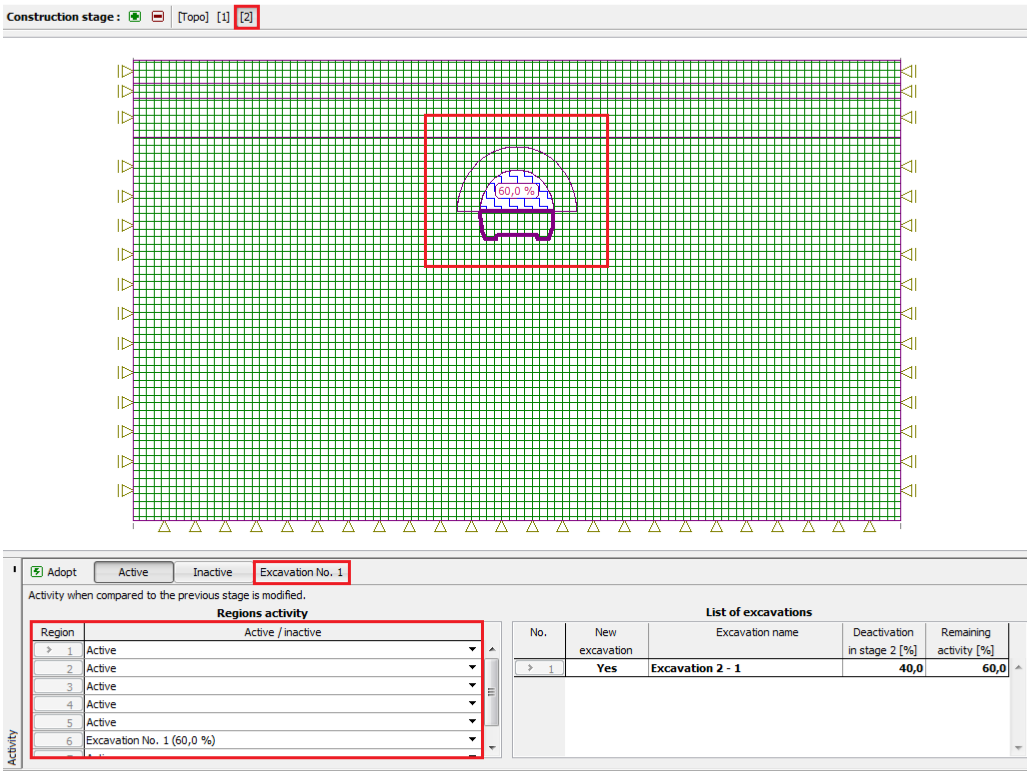
Dialogue window „New excavation" – Construction stage 2

Note: In the engineering practice the excavation sequence (individual excavations) is specified by the percentage of the rock deactivation relative to the remaining rock action. In this example we take the following proportions of the excavations for individual tunnel construction stages:

- top heading excavation, activation of the unsupported excavated opening: 40 % / 60 %,
- top heading vault support with the immature concrete primary lining: 30 % / 30 %,
- improvement of material properties of mature concrete (top heading): 30 % / 0 %.
- modelling of tunnel bench excavation, activation of the unsupported excavated opening: 40 % / 60 %,
- supporting of bench sidewalls by immature concrete primary lining: 30 % / 30 %,
- improvement of material properties of mature concrete (bench): 30 % / 0 %.

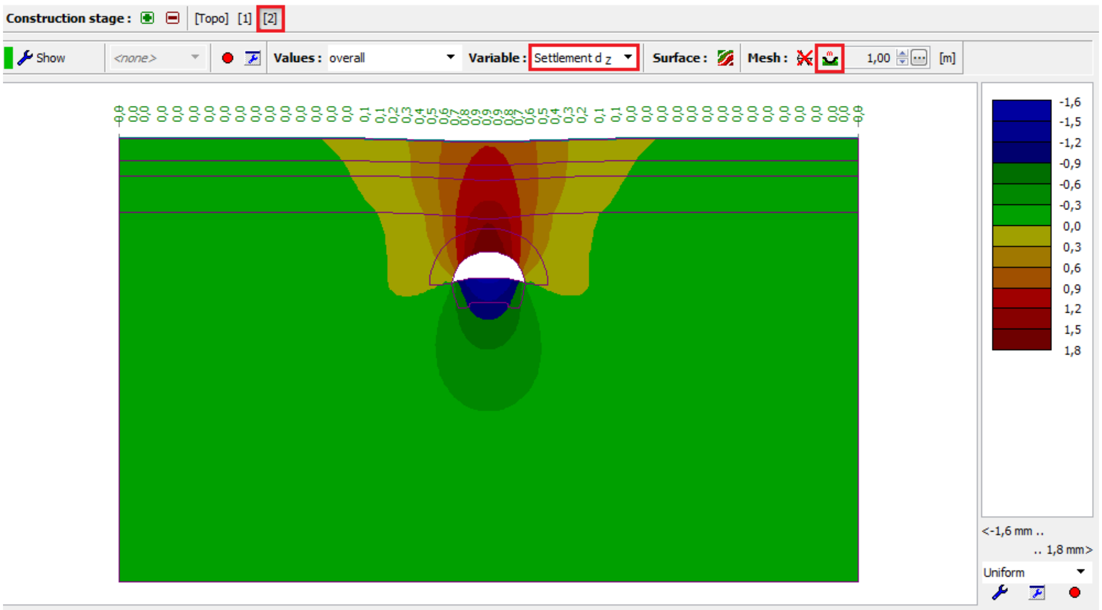
The above-mentioned percentage proportions are based on years of experience and provide relatively reliable results. In the program it is possible to set various percentage proportions of the Excavations for particular construction stages (e.g. 25/75, 30/45, 30/15 a 15/0) for the top heading or bench excavation.

In essence, it is the case of the activation of the percentage proportion of the load acting on the unsupported top heading. At this stage we take the 40% deactivation of soil (see *the picture)* into consideration.



Frame „Activity“ – Construction stage 2 (activity of 40 % of loading on excavation of top heading)

Now we will carry out the analysis and examine the results for vertical displacement d_z $[\mathrm{mm}]$. For better understanding of the excavation behaviour, we will show the deformed mesh and the settlement trough.



Frame „Analysis“ – Construction stage 2 (vertical displacement d_z with settlement trough)

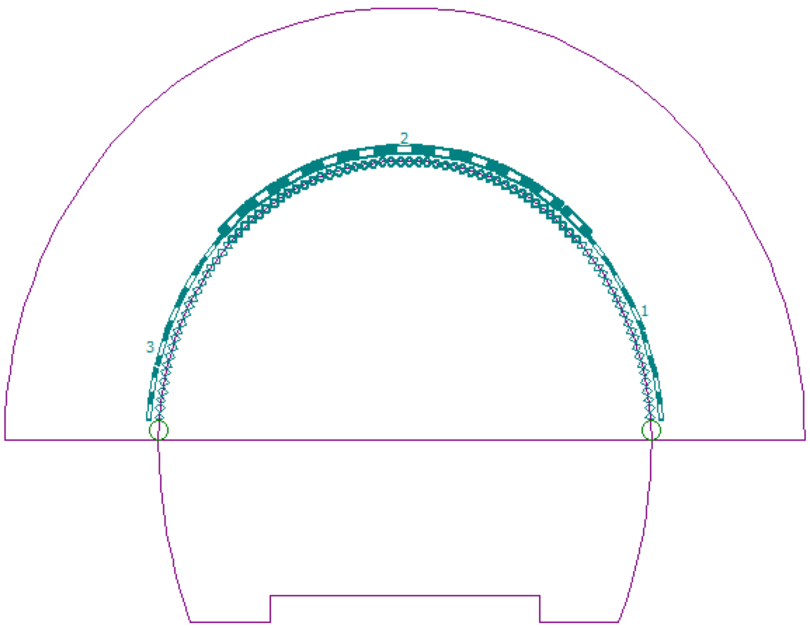
Construction stage 3: supporting the top heading vault with an immature concrete primary lining

In the next step we will add construction stage 3. First of all we will model the top heading vault support with a 200 mm thick immature concrete primary lining in the "Beams" frame.

No.	Beam new	Location	Support [m] Start pt.	End pt.	Include self weight	Cross-section / Degradation in current stage [%]	Material / Beam current activity [%]	Contacts left	right
1	Yes	Free line No. 1	○–	⊢	Yes	1,00 (b) x 0,20 (h) m	SB 25 (C 20/25); E = 2900,00 MPa; G = 1134,00 MPa; α = 0,000010 1/K; γ = 25,00 kN/m3	(not inputted)	Mohr-Coulomb
2	Yes	Free line No. 2	⊢	⊢	Yes	1,00 (b) x 0,20 (h) m	SB 25 (C 20/25); E = 2900,00 MPa; G = 1134,00 MPa; α = 0,000010 1/K; γ = 25,00 kN/m3	(not inputted)	Mohr-Coulomb
3	Yes	Free line No. 4	⊢	○–	Yes	1,00 (b) x 0,20 (h) m	SB 25 (C 20/25); E = 2900,00 MPa; G = 1134,00 MPa; α = 0,000010 1/K; γ = 25,00 kN/m3	(not inputted)	Mohr-Coulomb

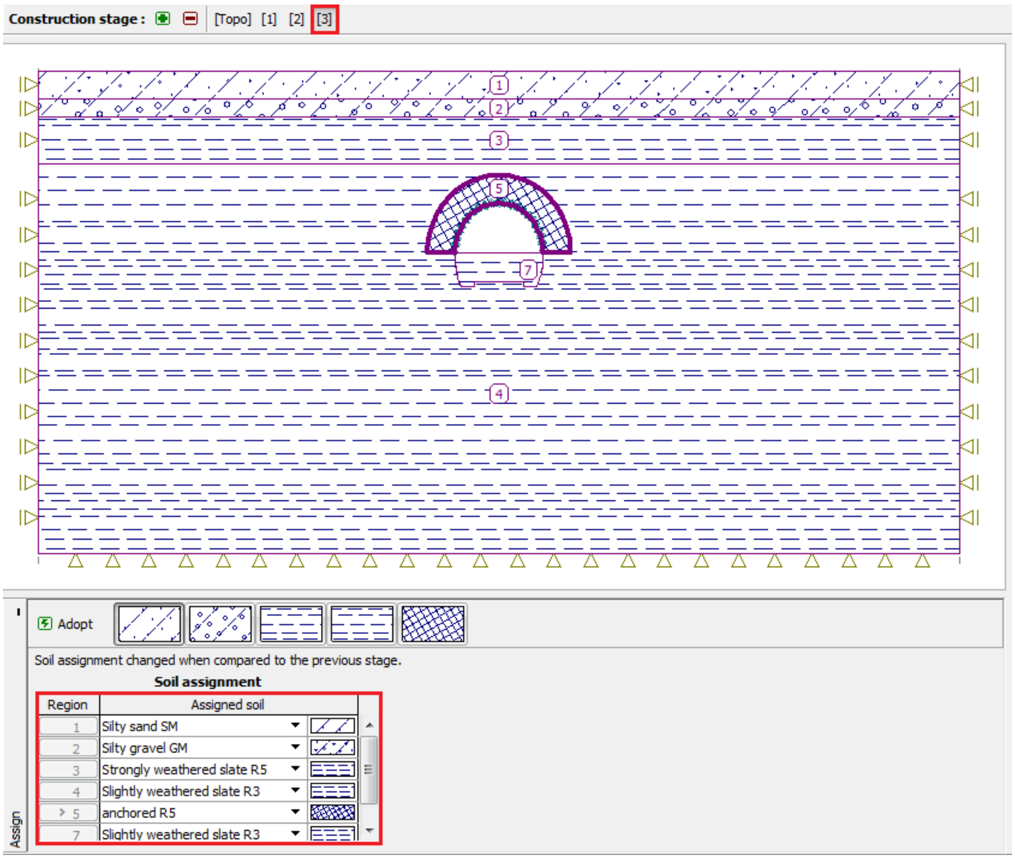
Input of primary lining of top heading with new beams – Construction stage 3 (immature concrete)

Note: We consider the beams to have a rotational degree of freedom at both ends, which means that bending moments at the bottom ends of the beams are zero. In some cases the bearing of the ends of beams is modelled by means of a special type of the so-called ***foot****, which ensures the stability and convergence in the analysis (for more details visit Help – F1).*



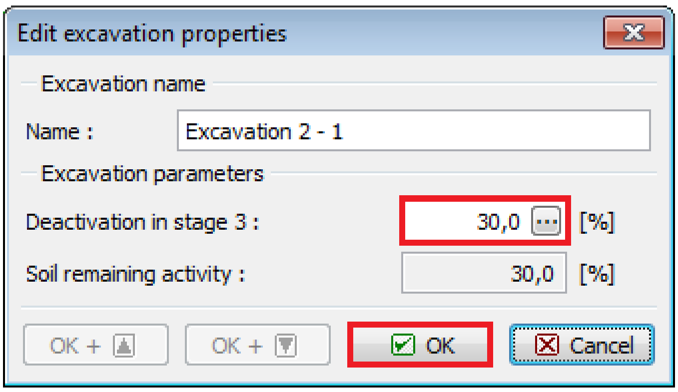
Frame „Beams" – Construction stage 3 (beams of primary lining)

In the "Assign" frame we will change the rock in the region No. 5 (to the "anchored R5" option), in which we will take the anchoring with hydraulically expanded steel rockbolts into consideration (see the *picture*).



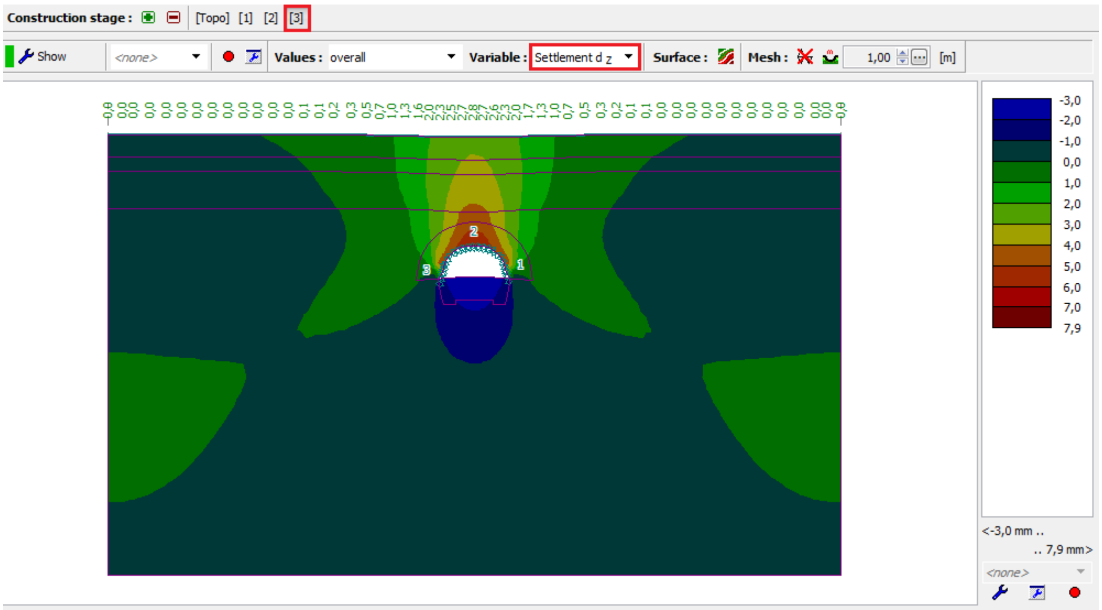
Frame „Assign" – Constr. stage 3 (region anchored with hydraulically expanded rockbolts)

Further on we will pass to the activation of rockbolts in the rock mass reinforced with anchoring in the surroundings of the top heading excavation and will edit the excavation properties by adding 30 % of the load (using the "Edit" button).



Dialogue window „Edit excavation properties" – Construction stage 3

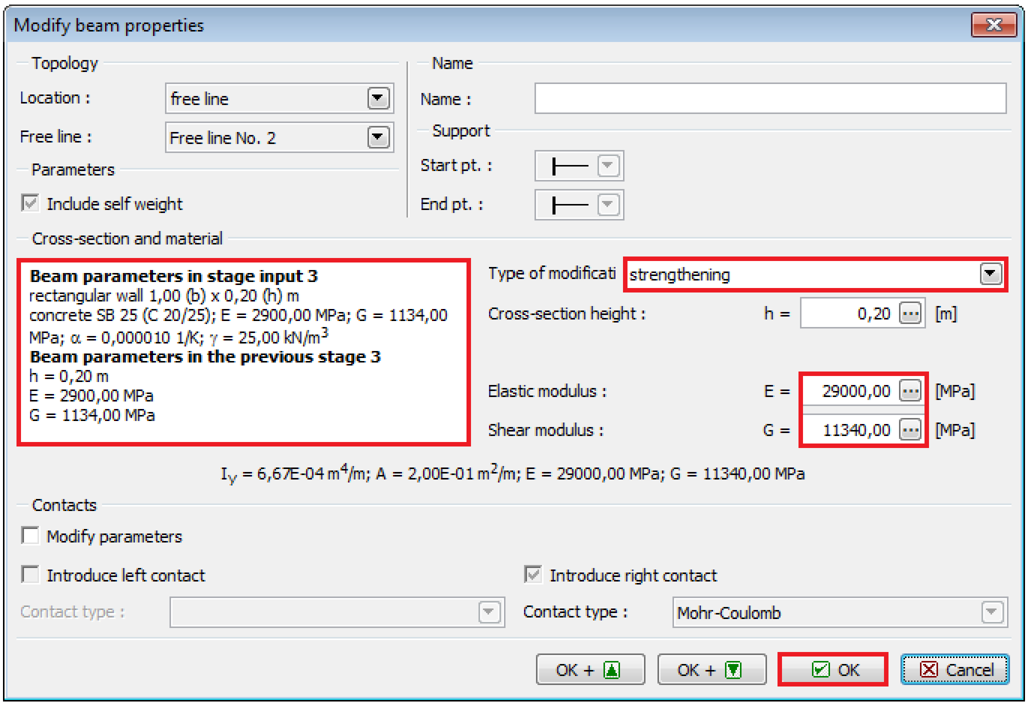
Then we will again carry out the analysis.



Frame „Analysis“ – Construction stage 3 (vertical displacement d_z with settlement trough)

Construction stage 4: improving material characteristics of mature concrete (top heading)

At construction stage 4, we will improve material characteristics of the already mature concrete supporting he top heading. In the dialogue window “Modify beam properties”, we will select the “Strengthening” option and will set respective values of the modules of elasticity. We will leave the other parameters unchanged.

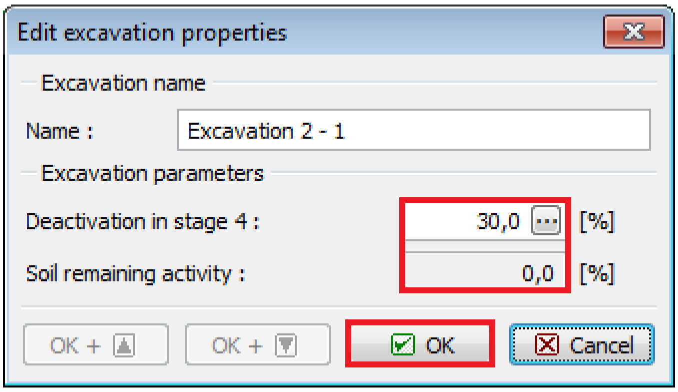


Dialog window „Modify beam properties“ – Construction stage 4 (beam no. 2)

No.	Beam		Location	Support [m]		Include self weight	Cross-section / Degradation in current stage [%]	Material / Beam current activity [%]
	new	modified		Start pt.	End pt.			
1	No	Yes	Free line No. 1	○—	⊢	Yes	↑ h = 0,20 m	↑ E = 29000,00 MPa; G = 11340,00 MPa
2	No	Yes	Free line No. 2	⊢	⊢	Yes	↑ h = 0,20 m	↑ E = 29000,00 MPa; G = 11340,00 MPa
3	No	Yes	Free line No. 4	⊢	○—	Yes	↑ h = 0,20 m	↑ E = 29000,00 MPa; G = 11340,00 MPa

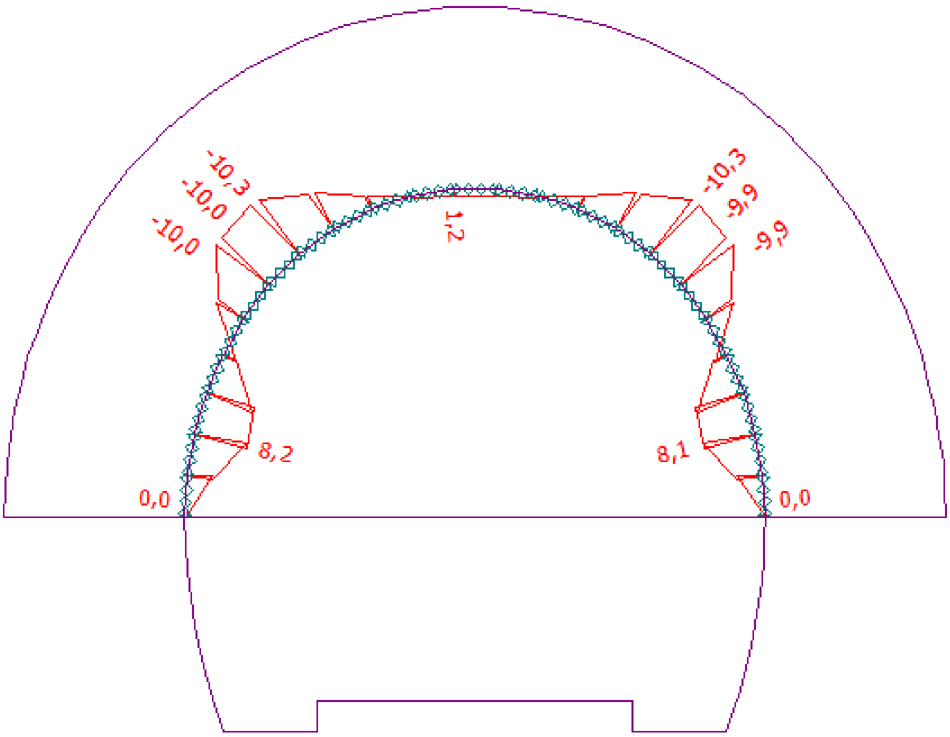
Modify properties of primary lining (top heading) – Construction stage 4 (mature sprayed concrete)

We will activate the remaining 30 % of the load acting on the rock mass. The procedure for editing the properties of the excavation is similar to that used at the previous construction stages.



Dialogue window „Edit excavation properties" – Construction stage 4

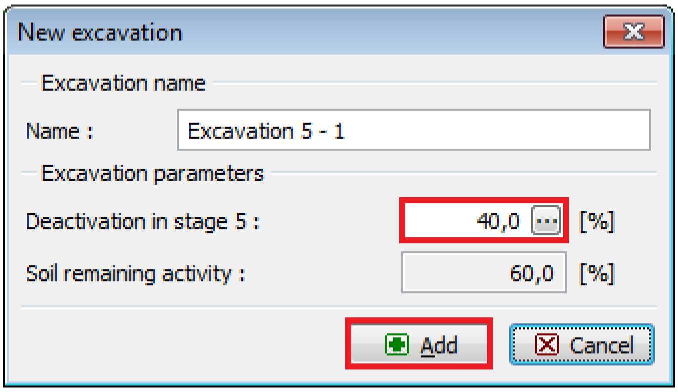
Subsequently we will carry out the analysis and examine the bending moment curve along top heading contour line.



Frame „Analysis" – Construction stage 4 (bending moment M $[\mathrm{kNm/m}]$)

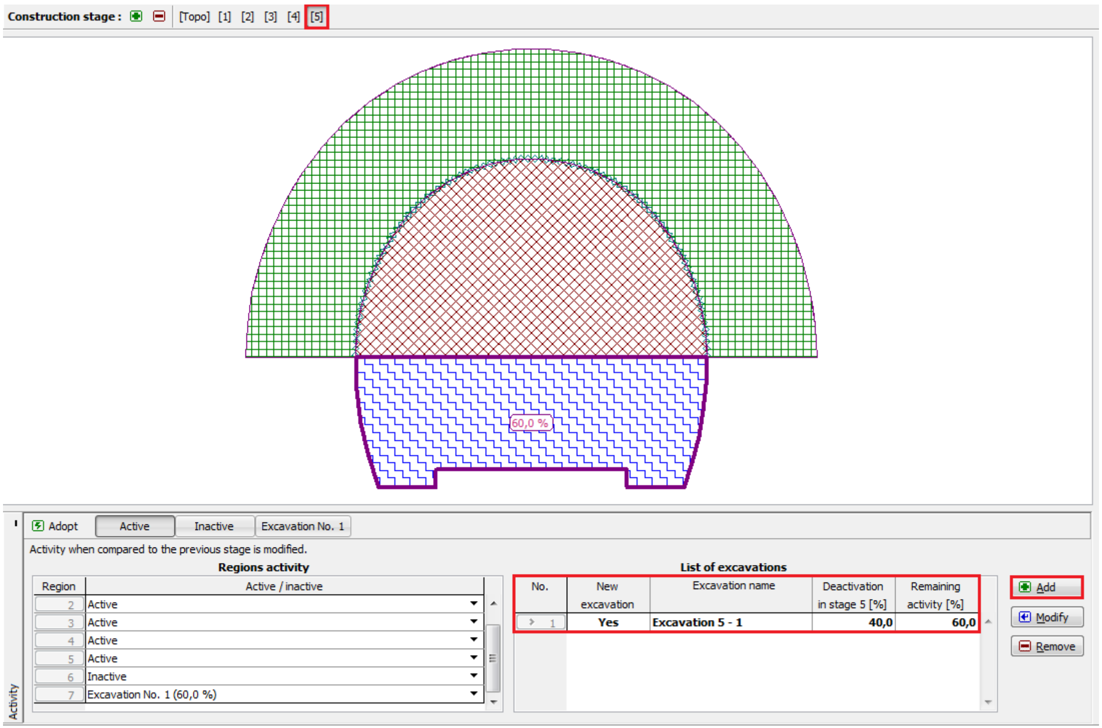
Construction stage 5: modelling of the tunnel bench excavation, activation of the unsupported excavated opening

In the next step, we will add construction stage 5. At this construction stage we will take the deactivation of soil, or the action of 40% of the load, into consideration. The remaining action of soil, or the massif in the vicinity of the tunnel bench excavation, is therefore 60 %.



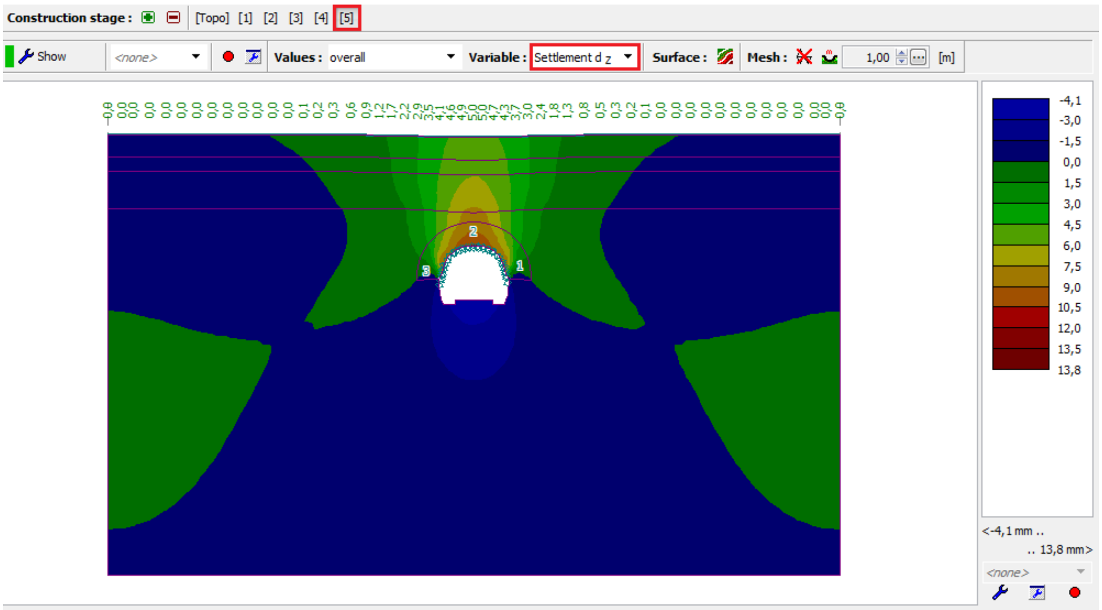
Dialogue window „New excavation“ – Construction stage 5

Note: The procedure for the modelling of this problem to be applied to the subsequent construction stages is similar. First of all, the tunnel bench primary lining is carried out using immature sprayed concrete. The next percentage proportion of the load will be activated subsequently. At the following stage the material characteristics of the already mature sprayed concrete will be improved and the remaining proportion of the load will be activated.



Frame „Activity“ – Constr. stage 5 (activity of 40 % of loading on excav. of tunnel bench)

Subsequently we will carry out the analysis.



Frame „Analysis“ – Construction stage 5 (vertical displacement d_z with settlement trough)

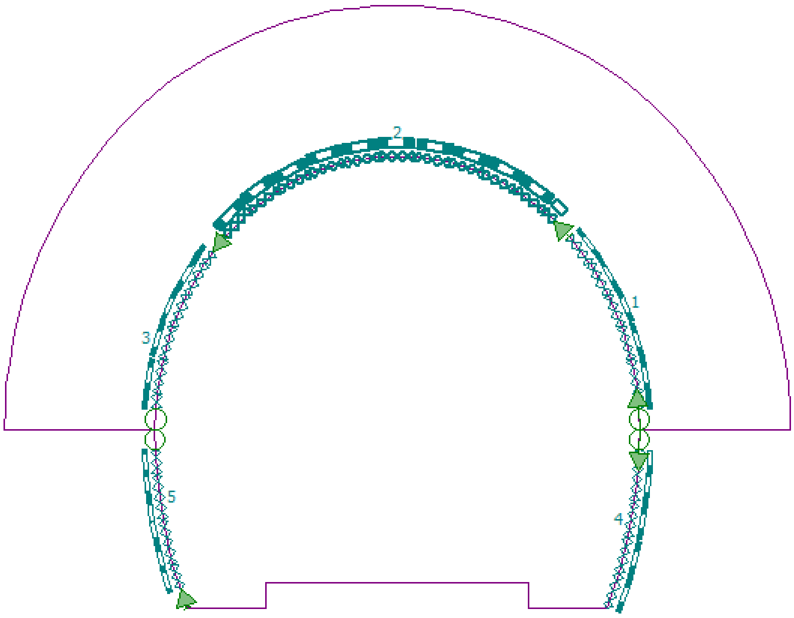
Construction stage 6: supporting the bench side-walls with an immature concrete primary lining

At construction stage 6 we will set the support of the bench side-walls with a 200 mm thick immature sprayed concrete primary lining. The top heading lining will remain unchanged at the subsequent stages.

No.	Beam new	Beam modified	Location	Support [m] Start pt.	Support [m] End pt.	Include self weight	Cross-section / Degradation in current stage [%]	Material / Beam current activity [%]	Contacts left	Contacts right
1	No	No	Free line No. 2	○—	⊢	Yes	without modification	without modification	(not inputted)	Mohr-Coulomb
2	No	No	Free line No. 1	⊢	⊢	Yes	without modification	without modification	(not inputted)	Mohr-Coulomb
3	No	No	Free line No. 4	⊢	○—	Yes	without modification	without modification	(not inputted)	Mohr-Coulomb
4	Yes		Free line No. 6	○—	⊢	Yes	1,00 (b) x 0,20 (h) m	SB 25 (C 20/25) young concrete; E = 2900,00 MPa; G = 1134,00 MPa; α = 0,000010 1/K; γ = 25,00 kN/m³	Mohr-Coulomb	(not inputted)
5	Yes		Free line No. 9	⊢	○—	Yes	1,00 (b) x 0,20 (h) m	SB 25 (C 20/25) young concrete; E = 2900,00 MPa; G = 1134,00 MPa; α = 0,000010 1/K; γ = 25,00 kN/m³	Mohr-Coulomb	(not inputted)

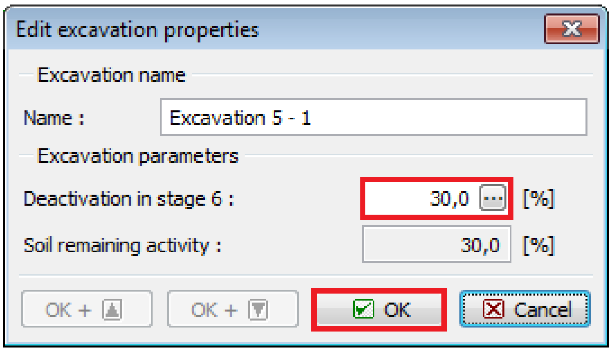
Input of primary lining of tunnel bench with new beams – Constr. stage 6 (immature concrete)

Note: We again consider the beams to have a rotational degree of freedom at both ends; the contact between the top heading and the bench is not capable to transferring the loading by a bending moment (it is not the case of fully continuous joints). The dimensions of the bench cross-section are identical with those of the top heading walls, i.e. $b = 1.0 \text{ m}$, $h = 0.2 \text{ m}$. *But we must set the contacts at the new beams in a reverse way (for more details see the picture) because the orientation of the beams (bench side-walls) is negative.*



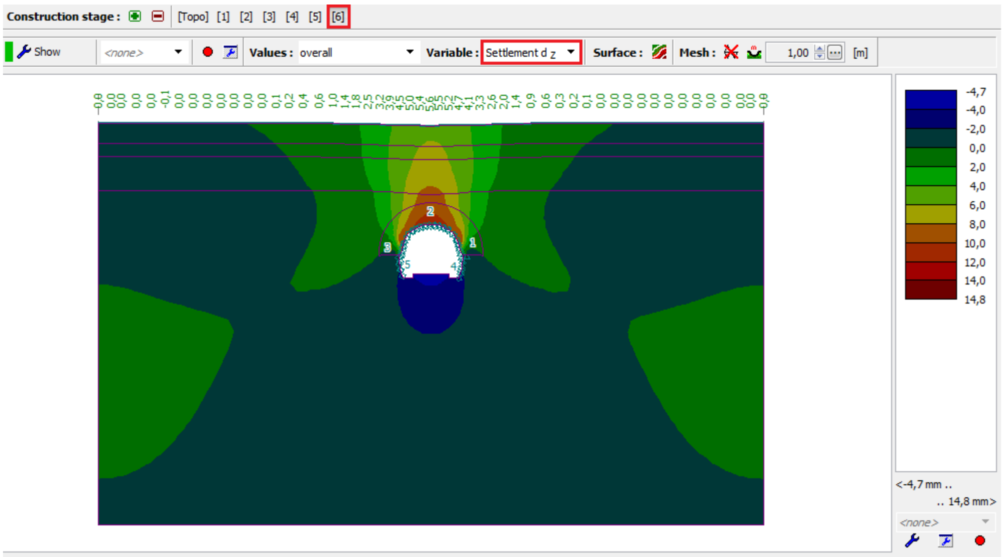
Frame „Beams" – Construction stage 6 (overall view of the primary tunnel lining)

At this stage, we will activate additional 30 % of the loading induced by the rock massif.



Dialogue window „Edit excavation properties" – Construction stage 6

In the last part of this stage we will again carry out the analysis.



Frame „Analysis" – Construction stage 6 (vertical displacement d_z with settlement trough)

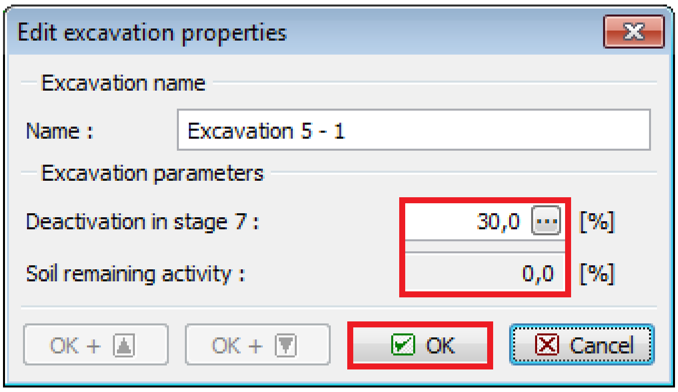
Construction stage 7: improving material characteristics of mature concrete (bench)

At the last construction stage we will improve the material characteristics of the already mature concrete supporting the tunnel bench excavation.

No.	Beam new	Beam modified	Location	Support [m] Start pt.	Support [m] End pt.	Include self weight	Cross-section / Degradation in current stage [%]	Material / Beam current activity [%]
1	No	No	Free line No. 2	○—	⊢	Yes	without modification	without modification
2	No	No	Free line No. 1	⊢	⊢	Yes	without modification	without modification
3	No	No	Free line No. 4	⊢	○—	Yes	without modification	without modification
4	No	Yes	Free line No. 6	○—	⊢	Yes	↑ h = 0,20 m	↑ E = 29000,00 MPa; G = 11340,00 MPa
5	No	Yes	Free line No. 9	⊢	○—	Yes	↑ h = 0,20 m	↑ E = 29000,00 MPa; G = 11340,00 MPa

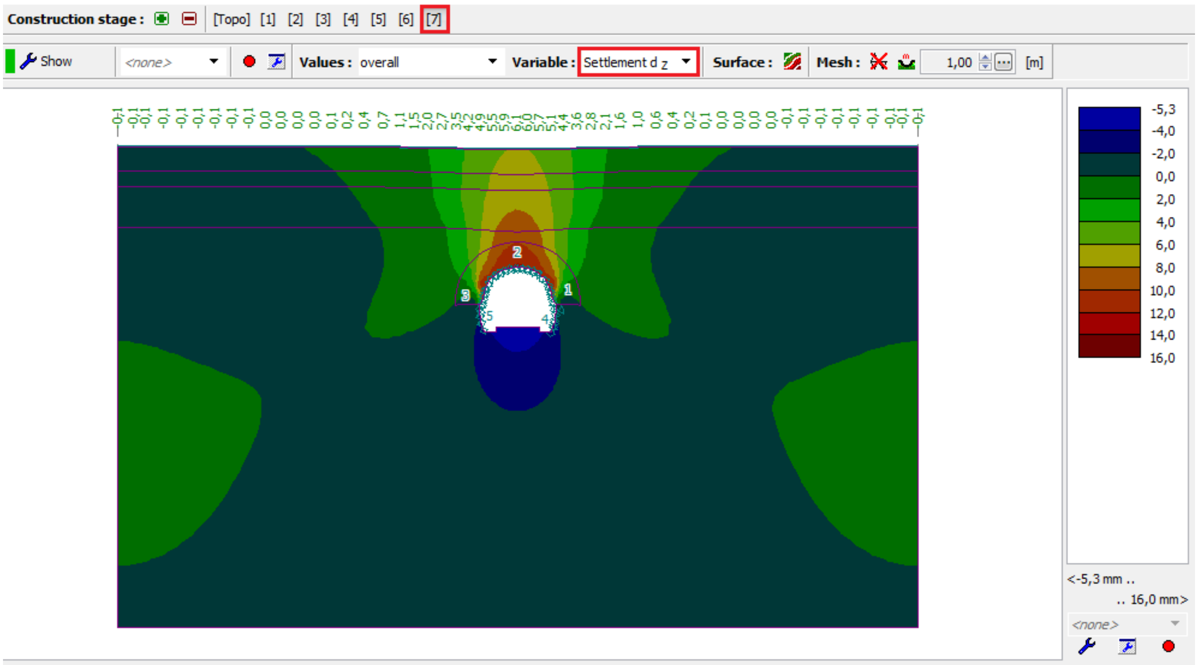
Modify properties of primary lining (tunnel bench) – Stage 7 (mature sprayed concrete)

The settings procedure for increasing the capacity of the beams is similar to that used at construction stage 4. We will activate remaining 30% of the load induced by the rock massif. By taking this step we remove all soil from the excavation space and the loading therefore acts on the primary tunnel lining (inclusive of the top heading and bench walls) at 100 %. Subsequently we will carry out the analysis of the last construction stage.



Dialogue window „Edit excavation properties" – Construction stage 7

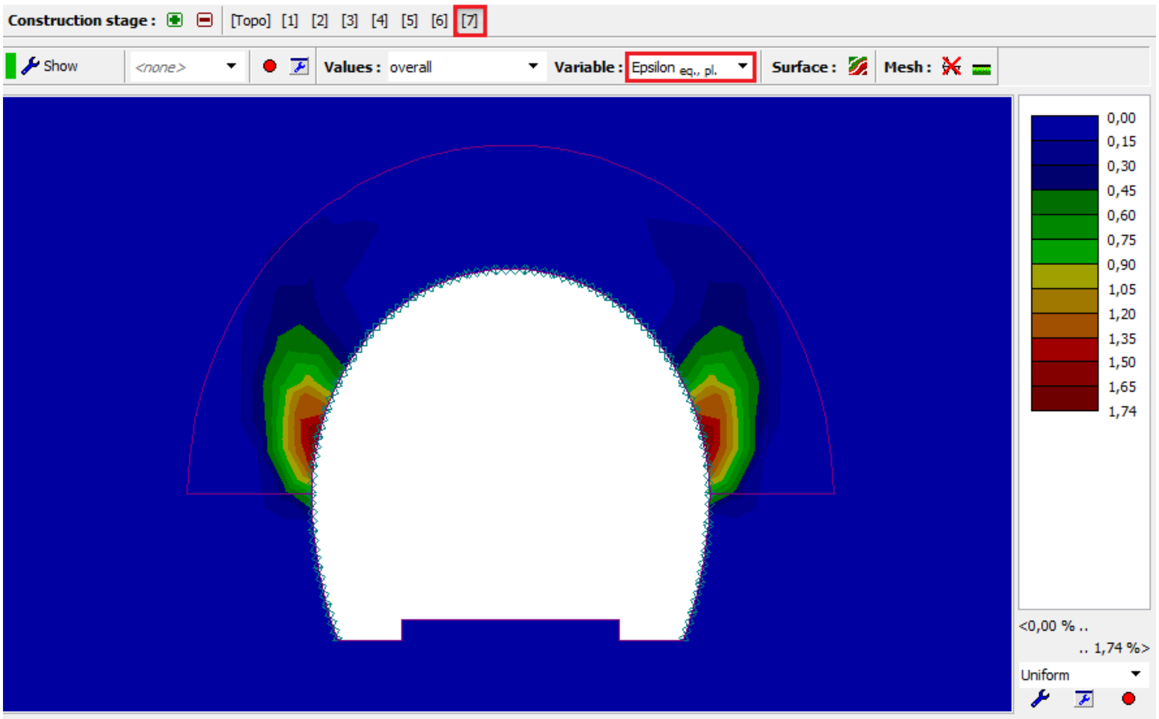
Now we will carry out the analysis of the last construction stage.



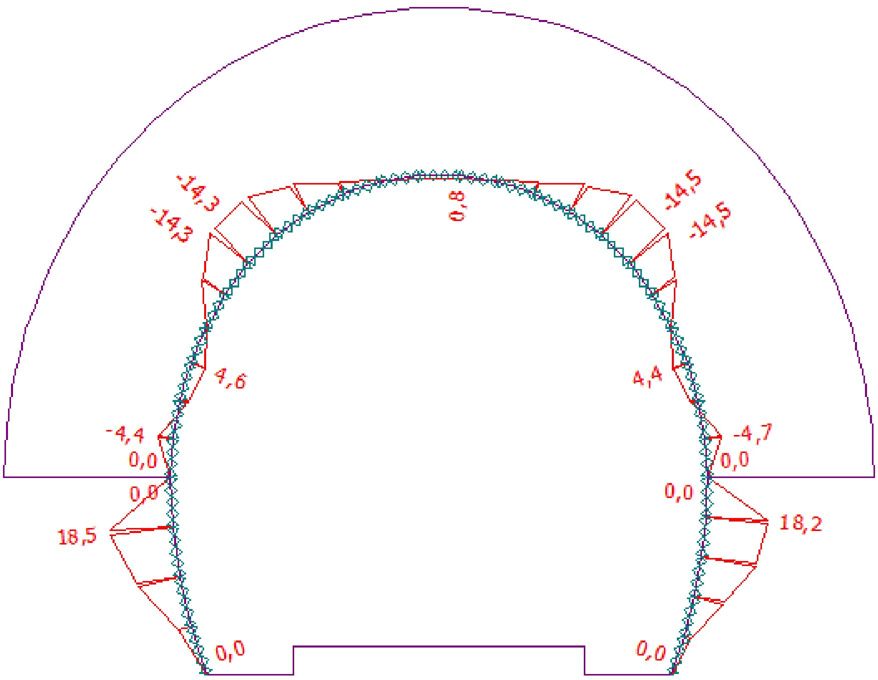
Frame „Analysis" – Construction stage 7 (vertical displacement d_z with settlement trough)

Further on at this stage, we will show equivalent plastic deformations $\varepsilon_{eq.,pl.}$ and the distribution of internal forces for bending moments and normal forces (the "Show" button, "Distribution" tab sheet). We will record the results in a summary table.

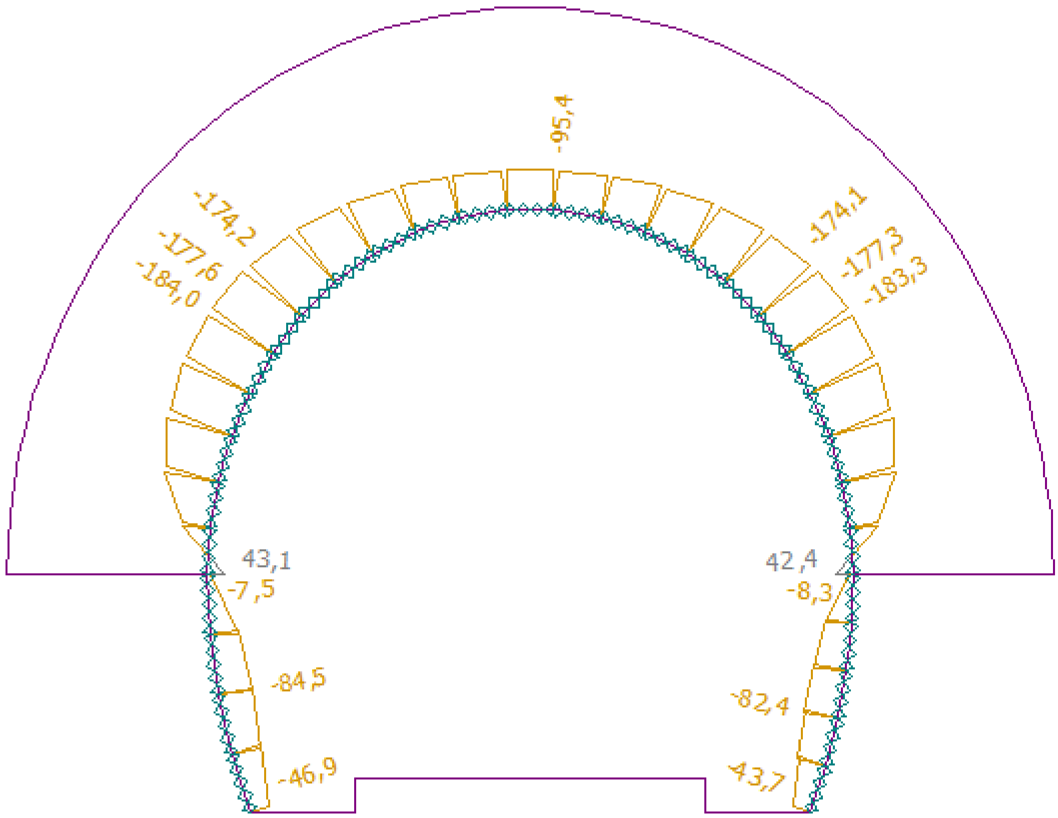
It follows from the picture on *page 27* that equivalent plastic $\varepsilon_{eq.,pl.}$ are not zero, which fact corresponds to the structure behaviour according to the non-linear material model (Mohr – Coulomb).



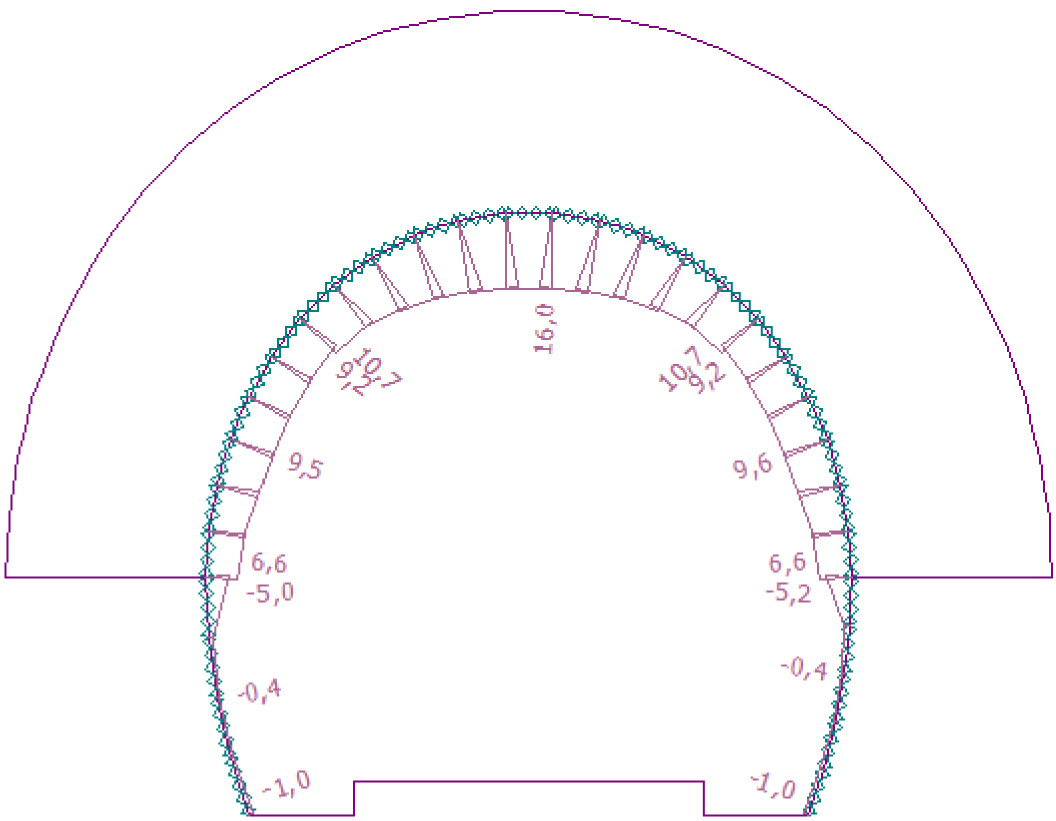
Frame „Analysis" – Construction stage 7 (equivalent plastic strain $\varepsilon_{eq.,pl.}$ *according to Mohr-Coulomb model)*



Frame „Analysis" – Construction stage 7 (bending moment M $[\mathrm{kNm/m}]$*)*



Frame „Analysis“ – Construction stage 7 (normal force – compression N^{-} $[\mathrm{kN/m}]$)



Frame „Analysis“ – Construction stage 7 (perpendicular deformation)

Assessment of the results:

The following table shows the values of extremes of internal forces (bending moments, shear forces and normal forces) acting on the beams (the primary tunnel lining) for construction stage 7. We carried out this analysis for the plastic material model (Mohr – Coulomb) with the locally increased density of the triangular elements.

Material model	**Construction stage 7 – Internal forces**		
	$N\,[\mathrm{kN/m}]$	$M\,[\mathrm{kNm/m}]$	$Q\,[\mathrm{kN/m}]$
Mohr – Coulomb	– 184.0	– 14.5	– 33.0
	+ 43.1	+ 18.5	+ 31.3

Extreme values of internal forces in primary lining – Construction stage 7

This table presents overall values of vertical and horizontal displacements $d_z, d_x\,[\mathrm{mm}]$ of the primary tunnel lining for individual construction stages.

Construction stage	**Values of overall displacement** $d_z, d_x\,[\mathrm{mm}]$			
	$d_{z,\min}$	$d_{z,\max}$	$d_{x,\min}$	$d_{x,\max}$
1	–	–	–	–
2	– 1.6	+ 1.8	– 0.46	+ 0.46
3	– 3.0	+ 7.9	– 2.2	+ 2.2
4	– 4.5	+ 13.7	– 3.6	+ 3.6
5	– 4.1	+ 13.8	– 3.9	+ 4.0
6	– 4.7	+ 14.8	– 4.9	+ 4.9
7	– 5.3	+ 16.0	– 5.8	+ 5.8

Values of displacement d_z, d_x of primary lining (extremes) – All construction stages

Conclusion:

In this problem we demonstrated the modelling of the primary lining of a real tunnel using the Finite Element Method. The tunnel is driven using the NATM. The tunnel excavation is divided into certain parts. When the rock is being removed, the massif is being unburdened and the soil or rock is being deformed, with the contour displacement direction heading toward the excavated opening interior.

The primary lining *is reinforced with KARI mesh (concrete reinforcing steel mesh welded from steel rods 8mm in diameter, with 150 x 150 mm mesh dimensions) and steel lattice girders with 3 load-bearing rods. The introduction of KARI mesh into the FEM numerical model (the homogenisation of concrete and reinforcement) is disputable; it is mostly taken into consideration only after the separate assessment of the lining.*

The primary tunnel lining would be subsequently assessed on the calculated extremes of internal forces, using a structural analysis software (e.g. FIN EC – CONCRETE 2D), as a combination of the section stressing by a bending moment and a normal force (according to the interaction diagram).

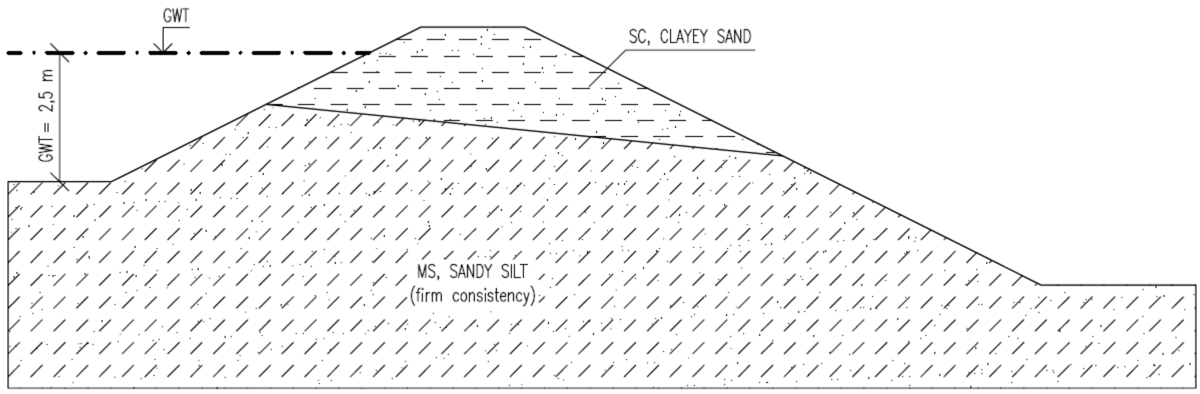
Note: The analysis of an underground structure without the use of beam and contact elements according to a linear material model (with elastic behaviour) was described in Chapter 23. Collector lining analysis (see http://www.finesoftware.eu/support/engineering-manuals/).

Chapter 27. Steady water flow analysis – Body of the embankment

This problem example describes the solution to stationary or steady, flow through an embankment body using the Finite Element Method.

Problem specification:

Carry out the analysis of steady flow through an embankment using the van Genuchten model. In this analysis, determine the distribution of pore pressures u_{wf}, which are used as input parameters for stability calculations. In addition, determine the saturation degree S and the overall balance of the inflow on the upstream side of the dam, or the outflow at the embankment base. The water table on the upstream side of the embankment is at the level of 2.5 m above its bottom. On the outflow side, water can flow freely. The embankment geometry scheme is obvious from the picture below.



Scheme of the embankment body – task for steady water flow analysis

The embankment consists of two soil layers, which have the following parameters:

Parameters of soils	CS, clayey sand (medium dense)	MS, sandy silt (firm consistency)
Permeability coeff. in direct. „ X “: $k_{x,sat}$ $[\mathrm{m/day}]$	$3{,}1 \cdot 10^{-1}$	$2{,}27 \cdot 10^{-2}$
Permeability coeff. in direct. „ Z “: $k_{z,sat}$ $[\mathrm{m/day}]$	$3.1 \cdot 10^{-1}$	$2.27 \cdot 10^{-2}$
Initial void ratio: e_0 $[-]$	0.75	0.60
Transition zone model (material model)	van Genuchten	van Genuchten
Model parameter: $\delta \equiv \alpha$ $[1/\mathrm{m}]$	1.0	1.0

Table with soil parameters – steady water flow analysis (the embankment body)

Note: The filtration coefficient of individual soil layers $k\,[\mathrm{m/day}]$ was determined on the basis of the assessment of laboratory permeability meter tests according to ***ISO/TS 17892-11:2004 (EN)*** *Geotechnical investigation and testing – Laboratory testing of soil – Part 11: Determination of permeability by constant and falling head.*

The initial void ratio e_0 is used for the determination of soil porosity $n = e_0/(1+e_0)$ and, subsequently, for the determination of the velocity of the flow of the fluid seeping solely through pores $v = v^{sw}/n$ $[\mathrm{m/s}]$. The flow velocity $v^{sw} = q/A$ is determined according to Darcy's Law; it expresses the value of the average volumetric flow rate $q\,[\mathrm{m^3/s}]$ through the entire area of passage $A\,[\mathrm{m^2}]$. It applies in general that soil porosity of 50 % corresponds to the void ratio $e = 1$ (for more details visit Help – F1).

Guidance values of the model parameter $\delta \equiv \alpha\,[1/\mathrm{m}]$, depending on grain-size classes of individual soil types according to USDA and FAO for the van Genuchten model, are presented in the theoretical part of the GEO 5 – FEM program Help (for more details visit F1).

Note: From version 18 of GEO5 – FEM on, the model parameter $\delta = [1/\mathrm{m}]$ is denoted in the program as $\alpha\,[1/\mathrm{m}]$.

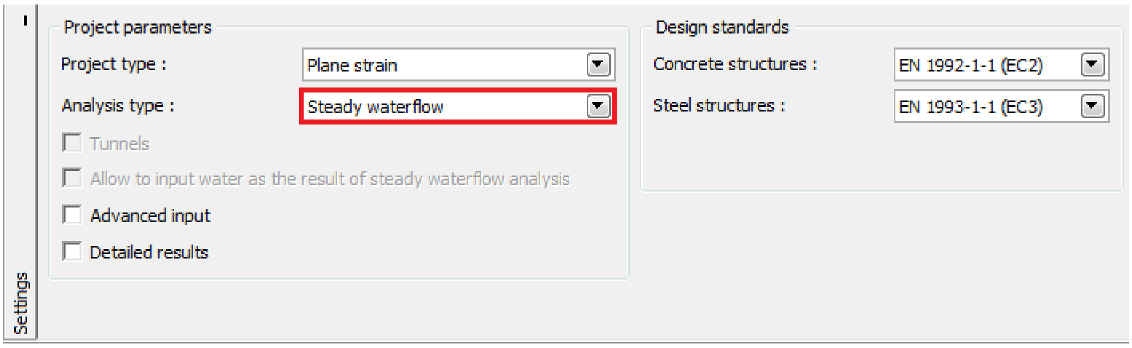
Solution:

We will apply the GEO 5 – FEM program to this problem analysis. In the following text we will step by step describe the solution to this example:

- Topology: the problem settings and modelling (interfaces, soils, mesh generation);
- Construction stage 1: settings of the linear flow – boundary conditions, calculation of quantities;
- Assessment of results: conclusion.

Topology: problem settings

In the "Settings" frame, we will select the analysis type using the *steady water flow* option.

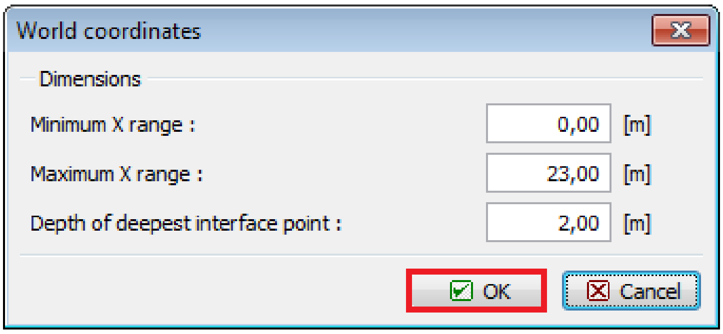


Frame „Settings"

Note: in the cases of both steady and unsteady flow it is in general the description of the flow through an unsaturated, or partly saturated, environment. Flowing through a fully saturated environment exists only under the water table. Flowing through a partly saturated environment (above the water table) follows an appropriate material model (for more details visit Help – F1).

A zero change in the saturation degree with time is assumed in the case of the steady flow; individual construction stages are fully independent of each other – as opposed to the unsteady flow. The introduction of relevant boundary conditions is also part of the analysis. Even if it is the case of the steady flow, it is a non-linear problem (e.g. solving an unconfined aquifer problem) requiring the application of the Newton-Raphson iterative method (for more details visit Help – F1).

Further, we will set the world coordinates. For our problem, we will choose the model dimensions of $\langle 0\ m; 23\ m \rangle$ and will set the depth from the lowest point of the interface at 2 m.



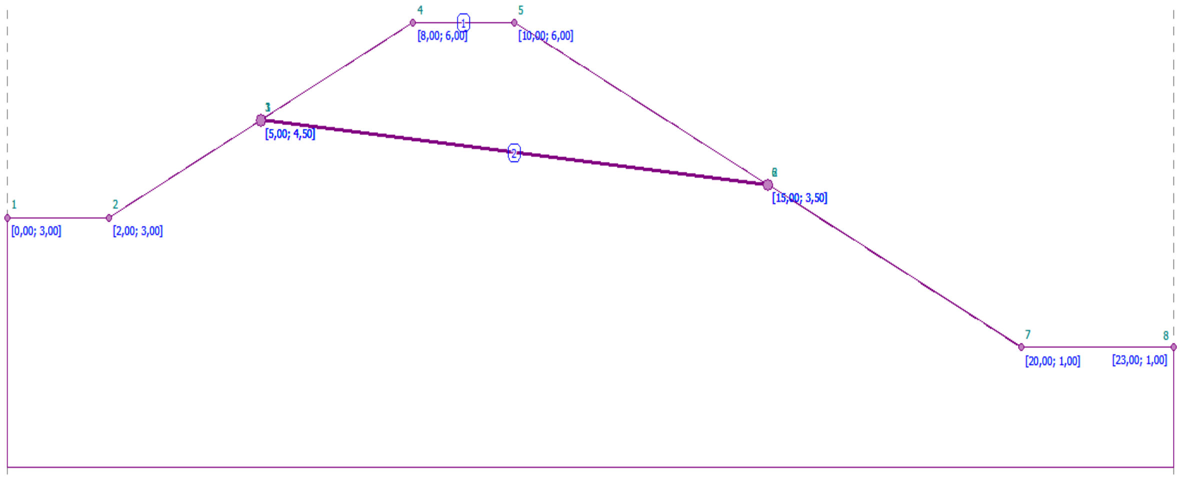
Dialogue window „World coordinates"

Subsequently, we will define the interface points for individual soil layers which are presented in the following table.

	Interface 1		Interface 2	
	x [m]	z [m]	x [m]	z [m]
1	0,00	3,00	5,00	4,50
2	2,00	3,00	15,00	3,50
3	5,00	4,50		
4	8,00	6,00		
5	10,00	6,00		
6	15,00	3,50		
7	20,00	1,00		
8	23,00	1,00		

List of points for individual interfaces between soil layers

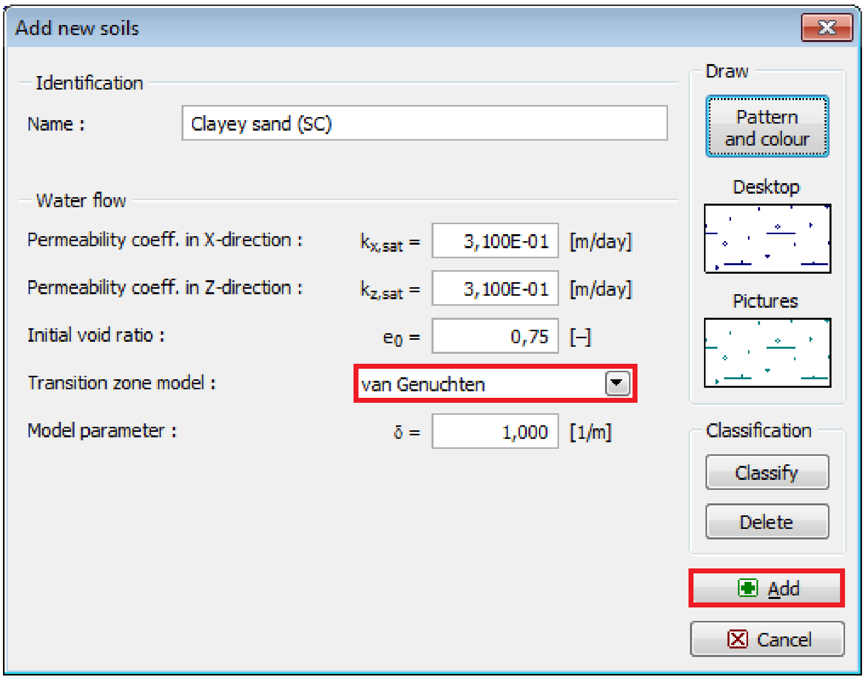
The scheme of the embankment body geometry inclusive of individual points of the interface is presented in the following picture.



Frame „Interface“

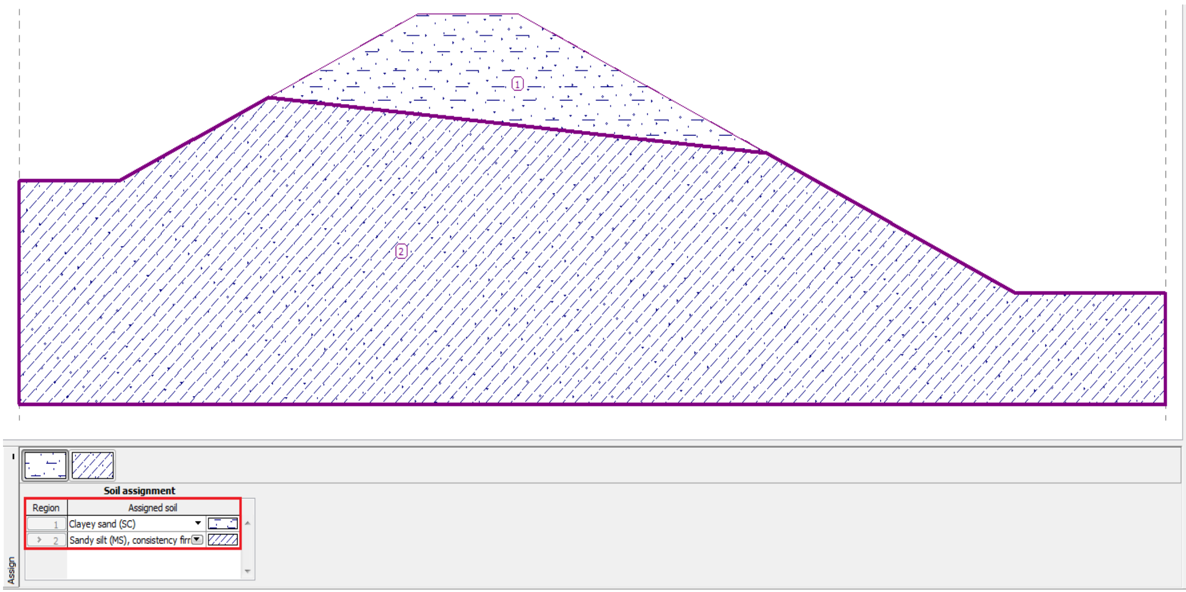
Now we will set respective soil parameters and, subsequently, will assign the soils to individual regions. We will choose the van Genuchten flow model for the solution of this problem.

Note: The relationship between the velocity of the flowing fluid and the change in the hydraulic gradient, or the pore pressure, follows the Darcy’s Law. The current version of the GEO 5 – FEM program (v5.17) assumes constant values of filtration coefficients. The dependence between the permeability coefficient and the pore pressure is not taken into consideration (for more details visit Help - F1).



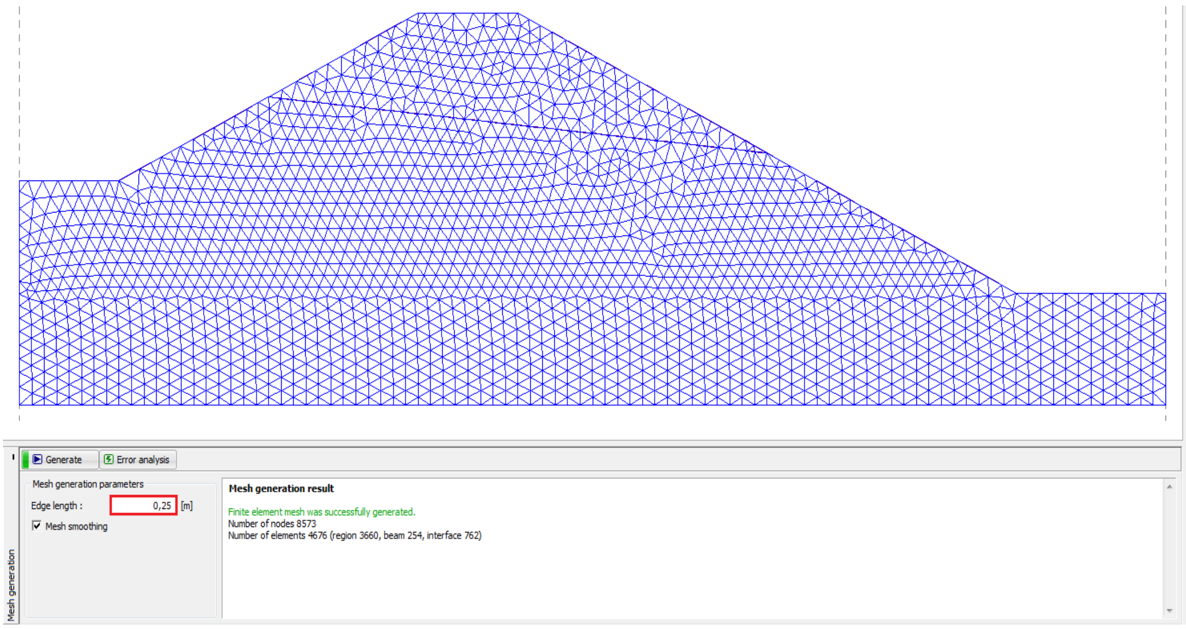
Dialogue window „Add new soils"

The following picture presents the assigning of soils to the embankment body.



Frame „Assign"

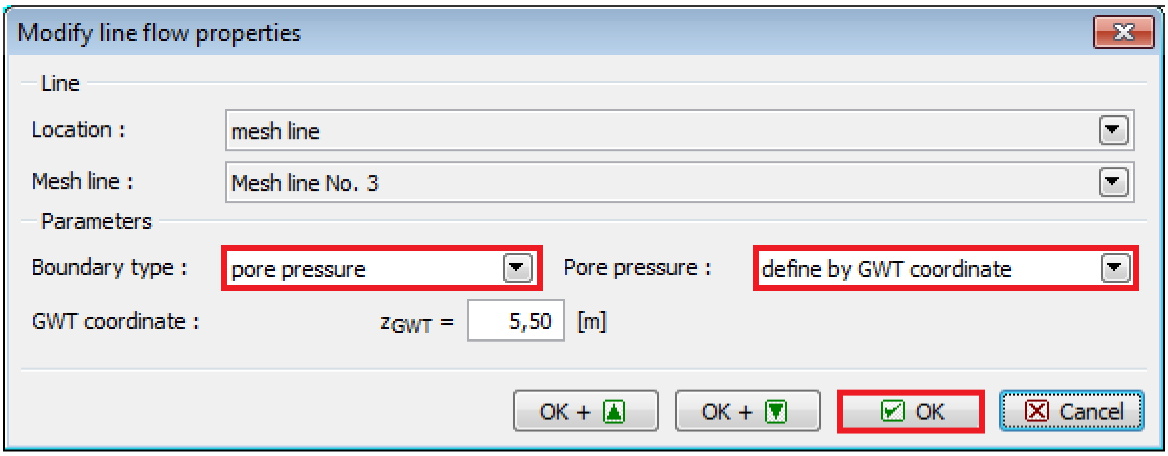
The last step in the topology settings is the generation of the finite element mesh. For this problem we will choose the length of the edge of elements of 0.25 m and will generate the mesh (using the **"Generate"** button). By visual examination we can see that the FE mesh density is sufficient.



Frame „Mesh generation“ – edge length 0.25 m

Construction stage 1: boundary conditions, steady water flow analysis

After generating the FE mesh, we will pass to construction stage 1 and, in the “Line flows” frame; we will define boundary conditions on all lines bounding the set embankment body. On the upstream side (lines 3, 5, 6 and 7) pore pressure values are defined by the water table level, by the coordinate $z_{GWT} = 5.5\ \mathrm{m}$. On the outflow side (lines 1, 9, 10 and 11), where we are not able to define the location of the outflow point unambiguously, we will define this boundary as a seepage surface.



Dialogue window „Modify line flow properties“

Note: The location of the outflow point is determined by the program automatically, depending on the embankment body geometry, boundary conditions and hydraulic parameters of soil (for more details visit Help – F1).

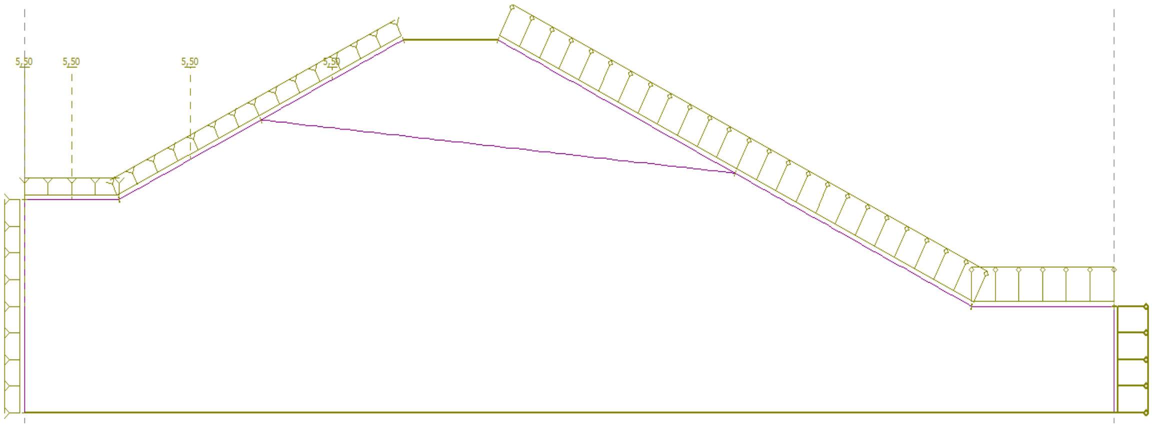
Location	Boundary type	Parameters
Mesh line no. 1	Seepage	–
Mesh line no. 2	Impermeable	–
Mesh line no. 3	Pore pressure	$z_{WT} = 5.5$ m
Mesh line no. 5	Pore pressure	$z_{WT} = 5.5$ m
Mesh line no. 6	Pore pressure	$z_{WT} = 5.5$ m
Mesh line no. 7	Pore pressure	$z_{WT} = 5.5$ m
Mesh line no. 8	Impermeable	–
Mesh line no. 9	Seepage	–
Mesh line no. 10	Seepage	–
Mesh line no. 11	Seepage	–

Boundary conditions for steady water flow analysis – embankment body (line flows)

Note: The boundary conditions for the water flow on the lines can be as follow:

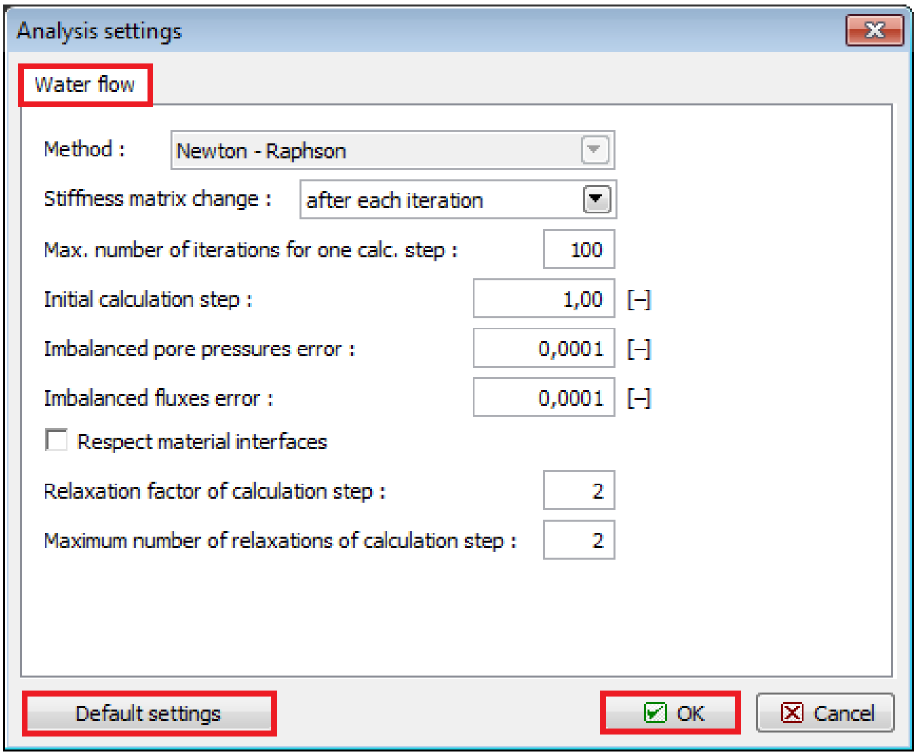
- ***Impermeable***
- ***Permeable**: the pore pressure on the line is equal to zero.*
- ***Pore pressure**: It is possible in the program either to set the distribution of the pore pressure* $p\,[\mathrm{kPa}]$ *numerically or to define the pore pressure distribution by specifying the water table level, i.e. to prescribe the hydraulic head* $h\,[\mathrm{m}]$.
- ***Inflow, or outflow, on the line*** $q\,[\mathrm{m/day}]$*: the rate of the fluid flow to/from the region is prescribed in the program. The default setting corresponds to the impermeable boundary, which complies with the condition* $q = 0$.
- ***Seepage surface**: this boundary condition is introduced in the case where the boundary cannot be divided unambiguously into a part with the pore pressure prescribed and a part with the inflow or outflow prescribed. In such the case the analysis is conducted in two steps. The respective interface is determined in the first step, whilst the problem analysis, with the boundary conditions known, is carried out in the second step. In some cases it is necessary to repeat both steps several times (for more details visit Help - F1).*

Now we will examine the embankment body including the determined boundary conditions (lines of flows along individual edges).



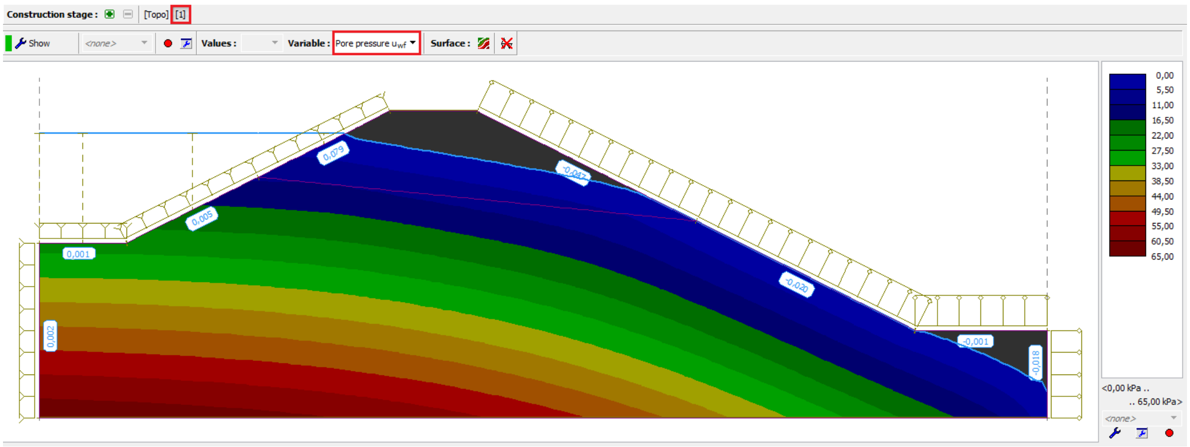
Frame „Flow lines“

Subsequently we will pass to the “Analysis” frame and will solve the given problem (by pushing the “Analyse” button). We will maintain the “Standard” analysis settings.



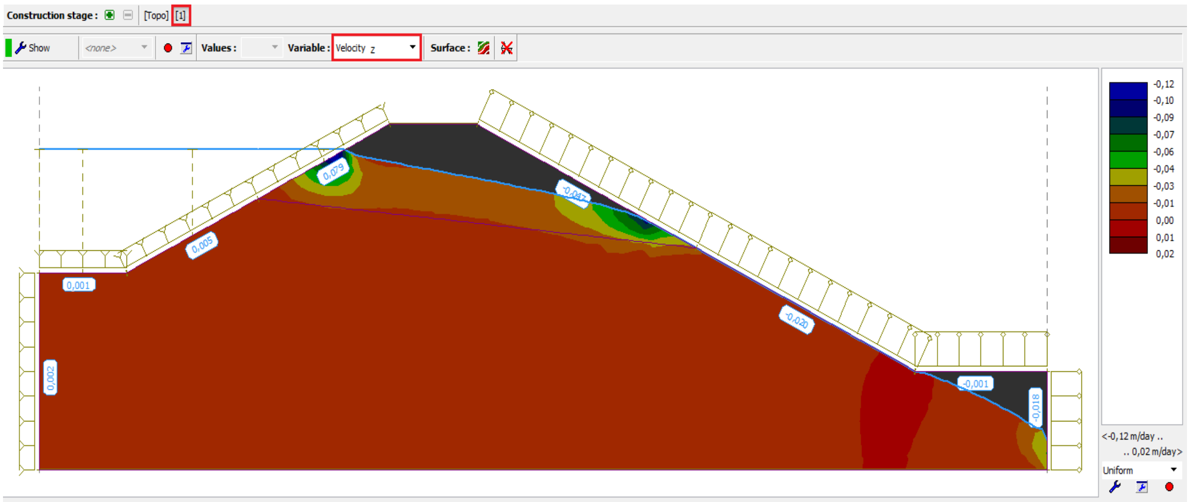
Dialogue window „Analysis settings“

The analysis result contains the distribution of pore pressures, rates and directions of flows, the information on the magnitude of flows into the massif and the outflows from the massif, respectively. We will gradually examine the results for these quantities in the program and subsequently will record their values.

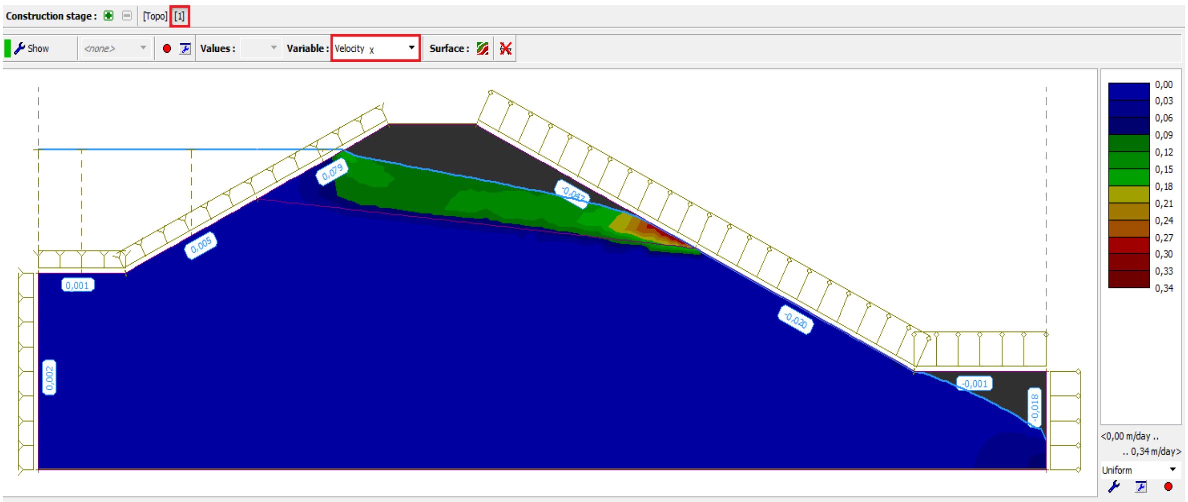


Frame „Analysis" – Construction stage 1 (pore pressure u_{wf})

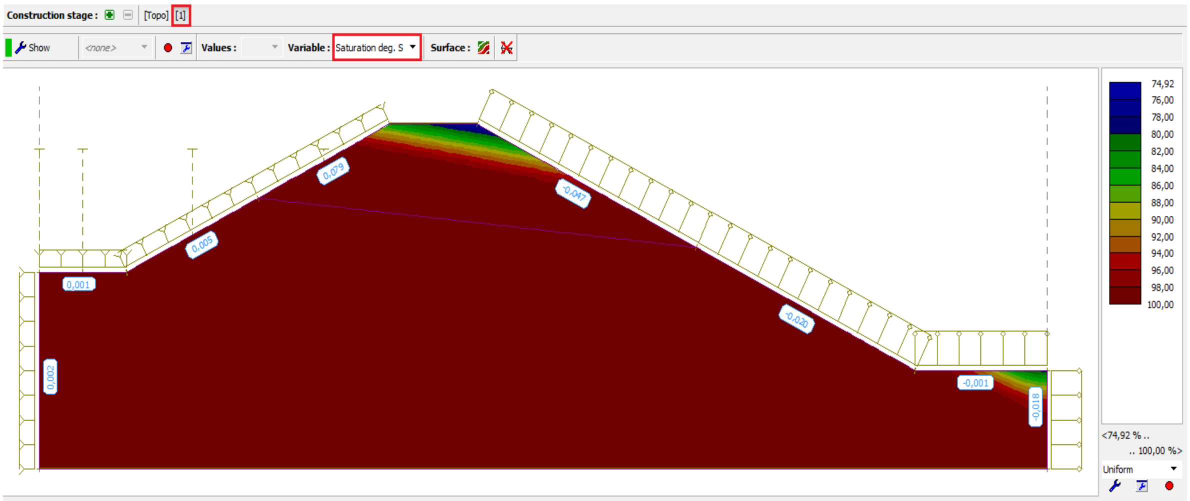
Note: The program does not depict the values of negative pressures in the region of partly saturated soil (for more details visit Help – F1).



Frame „Analysis" – Construction stage 1 (velocity v_z)

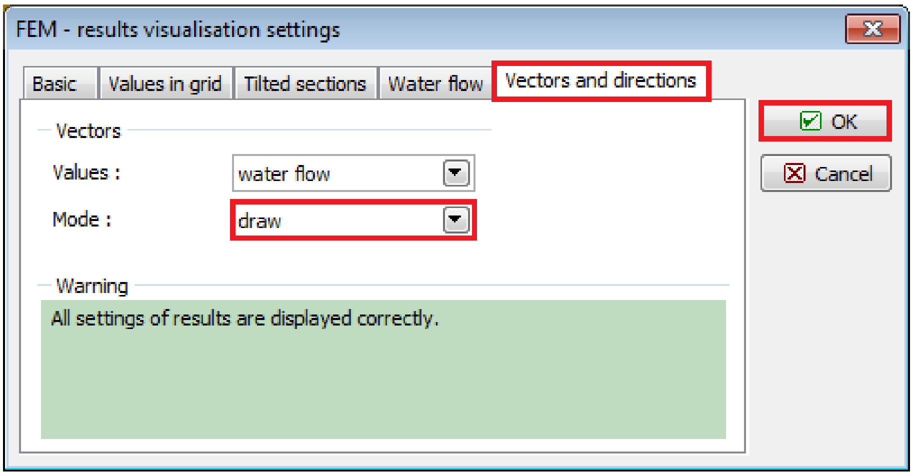


Frame „Analysis" – Construction stage 1 (velocity v_x)

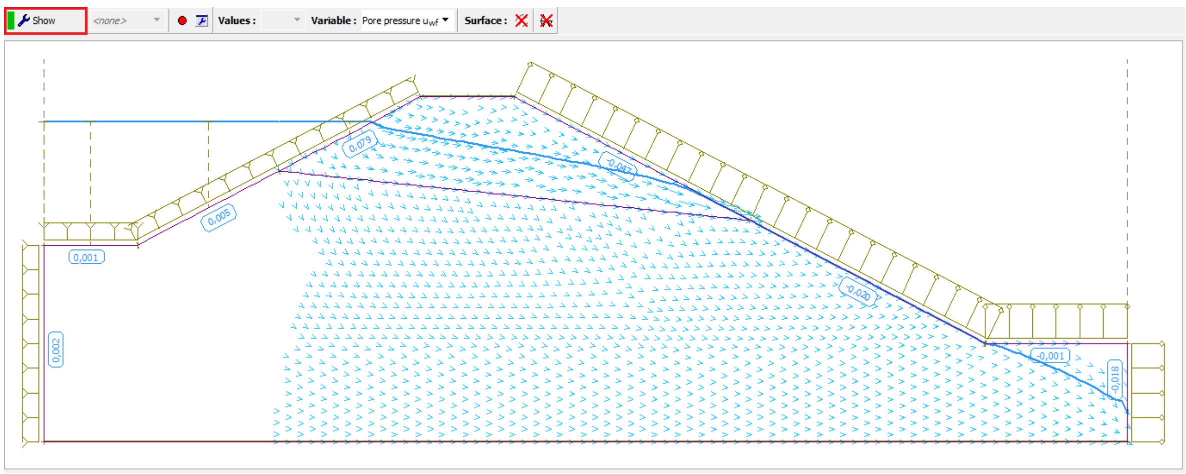


Frame „Analysis" – Construction stage 1 (saturation degree S)

Now we will examine the vectors and directions of the steady flow (using the "Show" button in the "Vectors and directions" tab sheet).



Dialogue window „FEM – results visualisation settings"



Frame „Analysis" – Construction stage 1 (water flow vectors and directions)

Assessment of the results:

In this problem we applied the van Genuchten material model of flow. We carried out the solution to the steady water flow through the embankment body for a triangular mesh with the edges of elements 0.25 m long; we obtained the following results:

- distribution of pore pressures: $u_{wf} = 0 \div 65.0\ \text{kPa}$,
- velocity in the Z direction: $v_z = -0.12 \div 0.02\ \text{m/day}$,
- velocity in the X direction: $v_x = 0 \div 0.34\ \text{m/day}$,
- degree of saturation: $S = 74.92 \div 100\ \%$.

Now we will read from the pictures the balance of the overall inflow and outflow, respectively, for individual flow lines. These values can also be obtained from the text output generated by the program.

Location	Inflow $[\text{m}^3/\text{den / m}]$	Outflow $[\text{m}^3/\text{den / m}]$
Flow line no. 1	0.045	–
Flow line no. 3	–	– 0.08
Flow line no. 4	–	– 0.005
Flow line no. 5	–	– 0.001
Flow line no. 6	–	– 0.002
Flow line no. 8	0.018	–
Flow line no. 9	0.001	–
Flow line no. 10	0.02	–
TOTAL	**0.084**	**– 0.088**

Calculated overall inflow and outflow – steady water flow analysis (embankment body)

Conclusion:

In this problem we have demonstrated the solution to the steady (stationary) water flow through an embankment body. By knowing the course and distribution of pore pressures, the overall balance of the inflow and outflow, or possibly the saturation degree, we will obtain much better idea of the water regime and the real behaviour of the ground body (embankment). The calculated results can be used further, e.g. for determining the overall stability of the earth structure or for hydro-geological purposes (designing flood prevention measures, water supply and water management, power generation etc.).