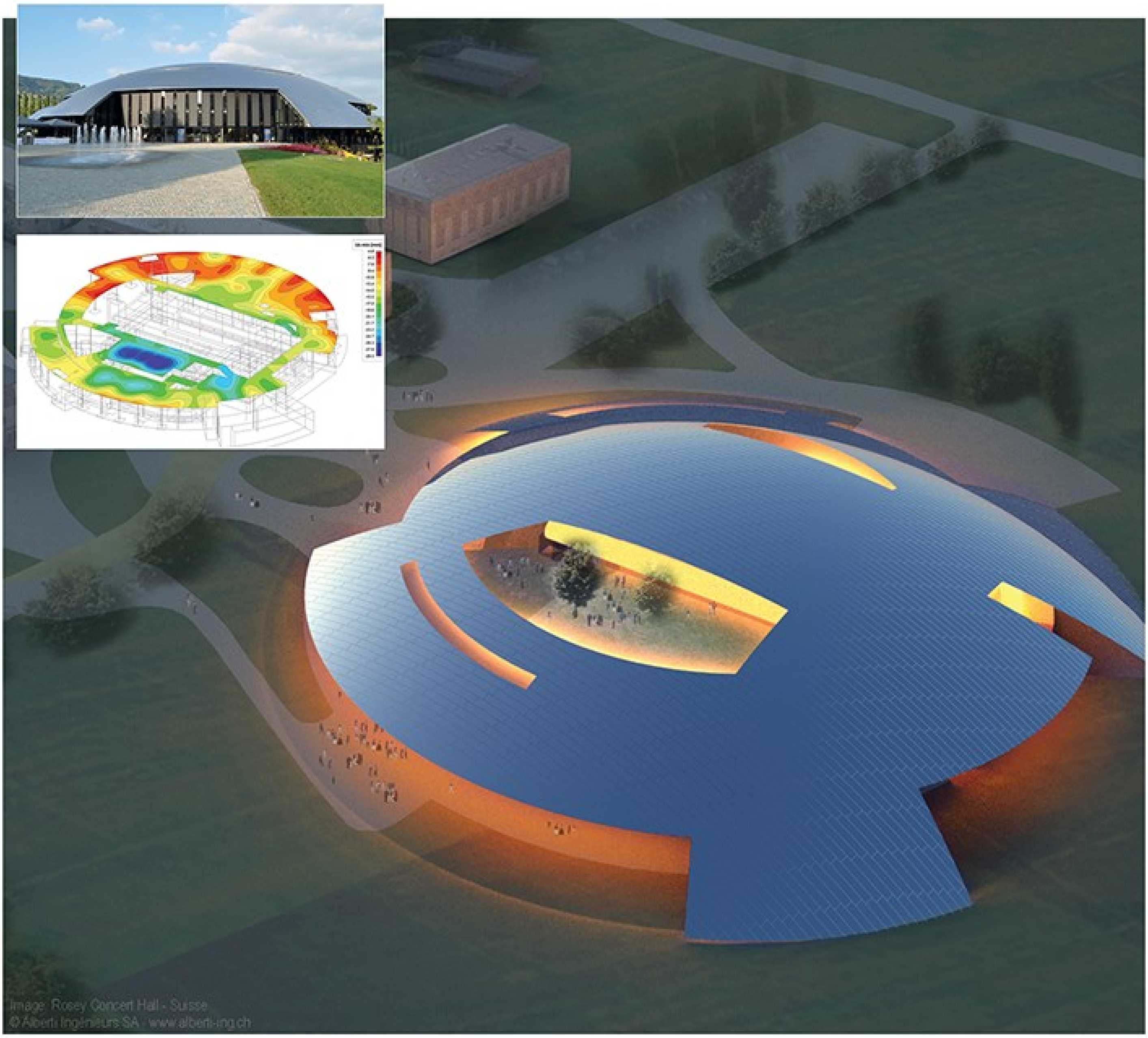


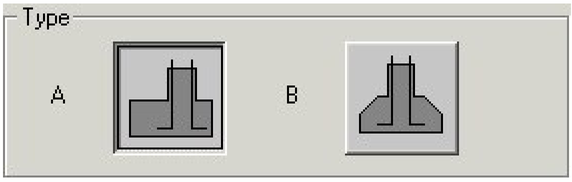
Foundations in Scia Engineer

Foundations, Piles, Subsoil, Soil-In, Geologic profiles

Foundation blocks and strips

Foundation block

The user can choose from two variants of foundation block.



The parameters are:

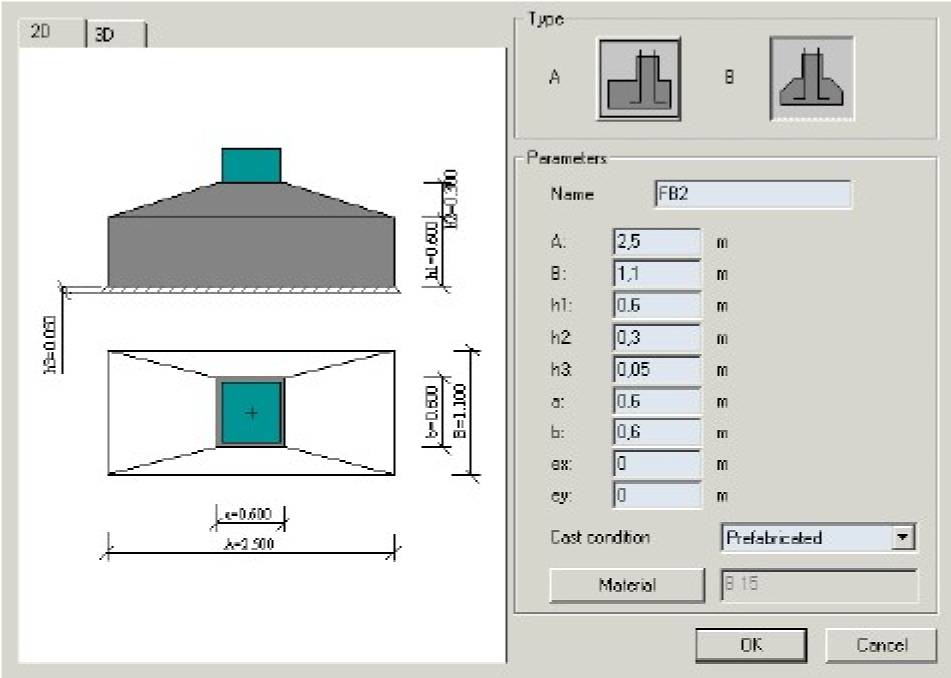
Name	The name is used for the identification of the foundation block.
Type	Specifies the shape of the foundation block.
Dimensions	The input of dimensions can be performed in a dialogue with self-explanatory interactive drawing of the block. That means that the user may click on a dimension line in the drawing and the corresponding item of the dialogue gets the focus. Therefore, it's very simple to specify the dimensions of the foundation block.
Eccentricity	The foundation bock may be either symmetrical or some eccentricity in one or both plan directions may be specified.
Cast conditions	Specifies the production of the foundation block.
Material	This item defines the material of the foundation block.

The foundation block editing dialogue makes it possible to display the foundation block in 2D or 3D mode.

- The 2D mode shows side view, plan view and dimension lines for all input values.
- The 3D mode enables the user to make a good visualisation of the defined foundation block

The above-mentioned properties are defined in the editing dialogue for the foundation block. The editing dialogue can be opened from the Foundation block manager.

In addition, another important parameter of the foundation block support must be defined. It is the soil that is below the footing surface. This last parameter is defined in the property dialogue of support, i.e. it is defined at the moment the support is being inserted into the model.



Foundation Blocks

Requirements

In order to be able to design the pad foundation, it is necessary to switch on the following functionality in the Project data dialogue:

- Subsoil,
- Subsoil > Pad foundation.

Partial safety factors

The partial safety factors for the combination are defined in the Manager for National annexes. It can be opened from the Basic project data dialogue.

Available are factors for **Set B** of the EN-ULS (STR/GEO) combination defined in EN 1990 [Ref.4]. In addition, for Geotechnical analysis, also **Set C** needs to be supported. Therefore the Combination Setup is expanded as follows:

The set C uses the following defaults:

Safety Factor	Default (Set C)
Partial factor permanent action - unfavourable	1,00
Partial factor permanent action - favourable	1,00
Partial factor for prestress action - favourable	1,00
Partial factor for prestress action - unfavourable	1,20
Partial factor leading variable action	1,30
Partial factor accompanying variable action	1,30
Partial factor for shrinkage action	1,00

Note that the **Reduction factor** input field is NOT available for this Set C.

New ULS Combination

A new combination is added: EN-ULS (STR/GEO) Set C

This combination follows exactly the same rules as the EN-ULS (STR/GEO) Set B except for the following differences:

The Combination Setup for ‘6.10’ or ‘6.10a & 6.10b’ is not used for this combination. For this combination always ‘6.10’ is used.

The safety factors are taken from the column Set C of the Combination Setup.

Automatic Classes

By default Scia Engineer creates the following automatic classes:

- All ULS
- All SLS
- All ULS & SLS

A new class is generated automatically: **GEO.** This class contains all combinations of the following types:

EN-ULS (STR/GEO) Set B

EN-ULS (STR/GEO) Set C

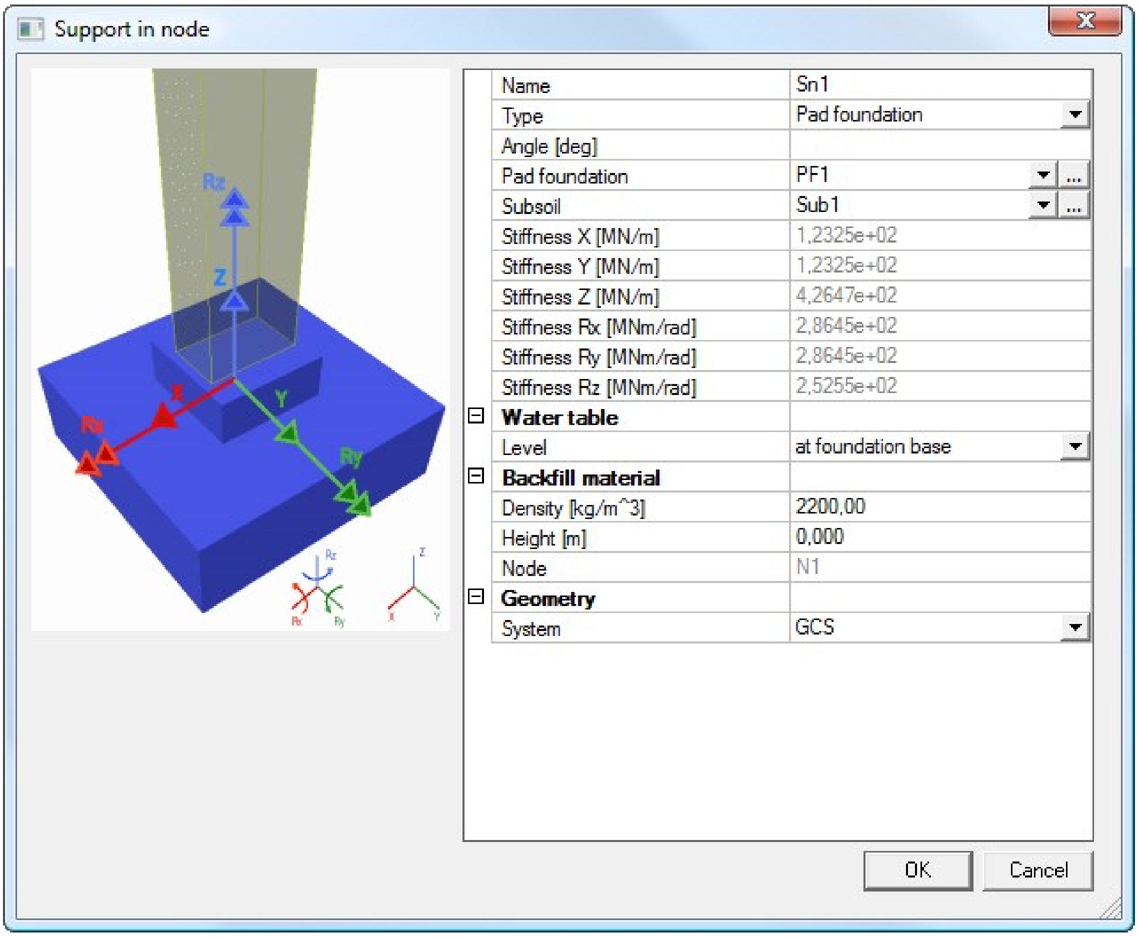
The class is only generated in case the functionality ‘**Subsoil**’ is activated in the Project Data.

In case none of these combination types are available, the class is not generated.

In case only one of these combination types is available, the class is generated with only those combinations.

The Set C combination is specifically used for Geotechnical Design according to **Design Approach 1**. Therefore, it should not be added to the default classes for ULS combinations but only to the Soil class.

Pad Foundation Input



Water table

The group **Water table** contains the water **Level:**

- no influence,
- at foundation base,
- at ground level.

By default the level is set to **No influence**.

Backfill material

Backfill material contains the following items:

density = Specifies the density of the soil above the foundation block or strip.

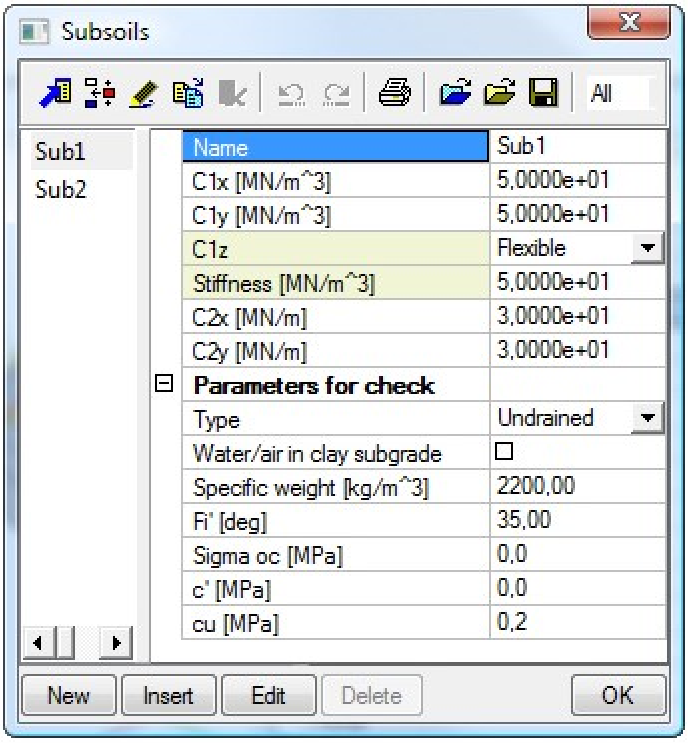
height = It defines the height of the upper soil layer. The height is measured from the top-surface of the foundation block.

The **Height [m]** input field in the **Backfill material** group allows for the input of both positive AND negative values. Negative values are required to indicate that the soil is lower than the top of the foundation block.

Stiffnesses

Stiffnesses are calculated by the program. The formulas applied can be seen in the Theoretical background: page 16 - Annex: Pad Foundation Stiffness

Subsoil parameters



In comparison with previous versions of the program, some Subsoil parameters has been renamed in order to comply with the EN convention:

Old text	New text
Fic [deg]	Fi' [deg]
Cc[MPa]	c'[MPa]
Ccu [Mpa]	cu [Mpa]

In addition, a new checkbox **Water/air in clay subgrade** is added.

Geotechnics Service

Service **Geotechnics** contains items:

- Geotechnics Setup
- Pad foundation - Pad foundation stability.

Geotechnics Setup

The **Geotechnics Setup** has the same layout as Setup dialogues for Steel, Aluminium, etc. We kindly refer you to the theoretical background manual for detailed information.

Support Reaction Elimination Factors

The group **Support Reaction Elimination Factors** allows the user to eliminate specific support reactions by inputting a multiplication factor. These input fields are limited to values between 0 and 1.

By default all reactions are used (factors 1.00). These factors can be used in case the user for example models only a pad foundation and omits other foundation elements like a ring beam. The user can then specify that only 50% of a reaction should be used to design the pad foundation since the other 50% goes into the ring beam.

Maximum value of eccentricity

The **Maximum value of eccentricity** group allows the user to specify the maximum allowed eccentricity in function of the width The user can choose between 1/3, 1/6 and No Limit

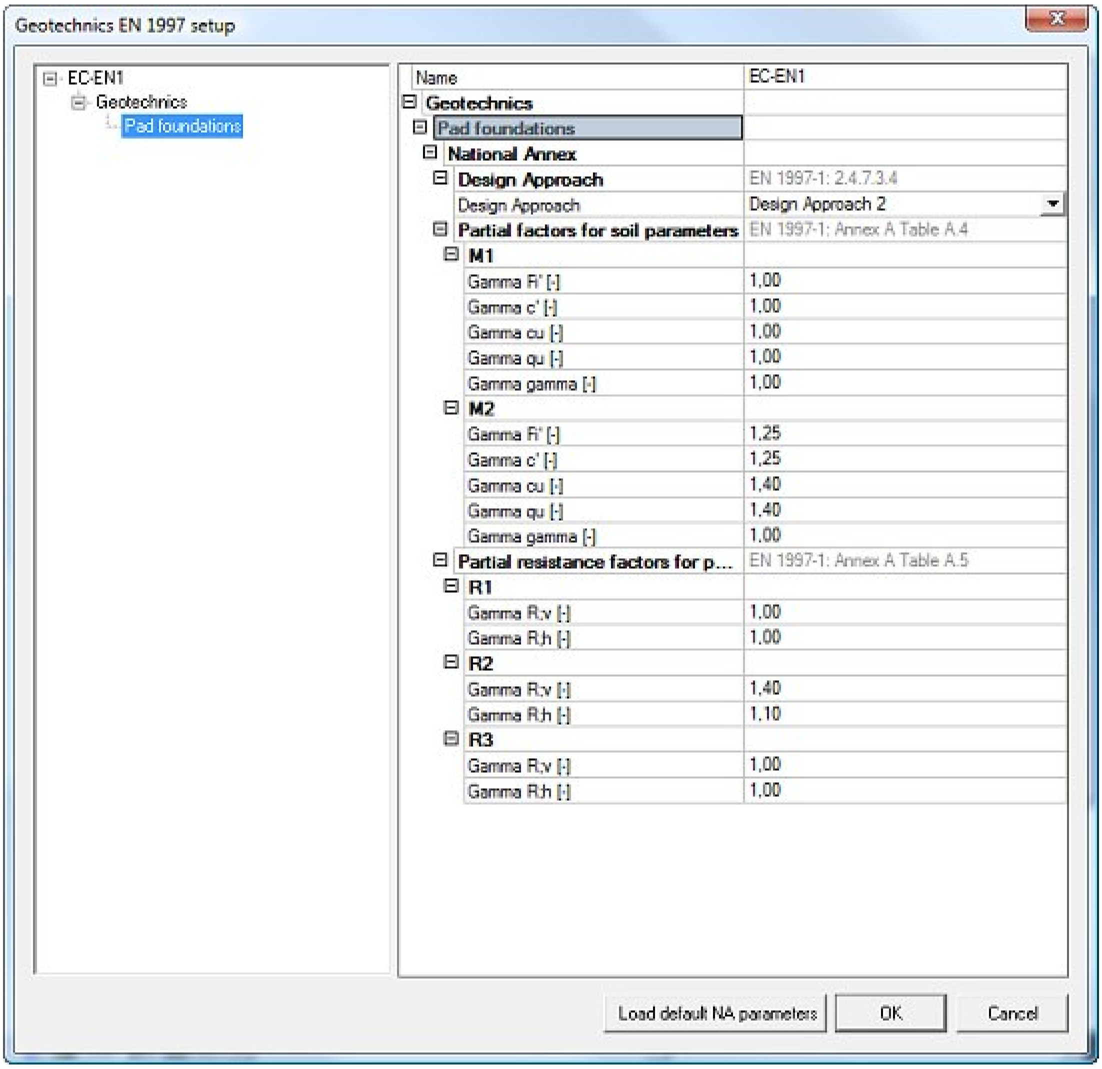
Known soil capacity

The checkbox **Known soil capacity, use Sigma oc** can be used by the user to override the EN 1997-1 bearing calculation. Instead, the inputted Sigma oc value of the subsoil is used..

By default this checkbox is de-activated.

Note that this analysis is NOT according to EN 1997-1.

In addition, under the **National Annex** button in the **Project data dialogue** the parameters relating to the national annex can be specified:



The group Design Approach EN 1997-1: 2.4.7.3.4 contains three options:

The Design Approach determines which set of combinations, safety factors and resistance factors have to be used.

Note: the Design Approach is given here and not on the general tab since the National Annex can give different approaches for different foundation types (for example approach 1 for pad foundations and approach 3 for pile foundations etc).

The group Partial factors for soil parameters contains the safety factors according to Table A.4. Two sets are shown: M1 & M2. These sets have the following default values:

Safety Factor	M1	M2
Gamma Fi’	1,00	1,25
Gamma c’	1,00	1,25
Gamma cu	1,00	1,40
Gamma qu	1,00	1,40
Gamma gamma	1,00	1,00

The group **Partial resistance factors for pad foundations** contains the resistance factors according to Table A.5. Three sets are shown: **R1**, **R2** & **R3**. These sets have the following default values:

Resistance Factor	R1	R2	R3
Gamma R;v	1,00	1,40	1,00
Gamma R;h	1,00	1,10	1,00

Pad Foundation Stability Check Service

The property window for the Pad foundation stability check allow for the check to be executed ONLY for Result Classes.

By default the class **GEO** is used.

In case there is no class in the project the class field is empty.

The Selection in this case refers to Pad Foundation entities.

The action buttons perform ‘Refresh’, ‘Preview’ and ‘AutoDesign’.

When executing the check, the safety and resistance factors which are applied depend on the **Design Approach** set in the Setup.

For **Design Approach 1** the class for which the check is executed needs to contain at least one combination of each of the following types:

- **EN-ULS (STR/GEO) Set B**
- **EN-ULS (STR/GEO) Set C**

In case the class for which the user wishes to execute the check does not comply with this requirement, the check is not executed and a warning dialog is shown:

“Note: For Design Approach 1 it is required that the Result Class contains at least one combination of each of the following types:

- EN-ULS (STR/GEO) Set B
- EN-ULS (STR/GEO) Set C

The selected class does not meet this requirement; please modify the contents of the class.”

For **Design Approaches 2 & 3** there is no requirement for the content of the class.

Pad Foundation Check

In general three separate checks are executed:

- Bearing Check
- Sliding Check
- Eccentricity Check

In a special case, instead of the three above a so called Uplift Check is executed.

Before any check can be executed, the required safety and resistance factors need to be determined depending on the chosen Design Approach.

In addition the vertical design loading $\mathbf{V_d}$, horizontal design loading $\mathbf{H_d}$ and effective geometry of the pad need to be determined.

In the following paragraphs the checks are specified:

Bearing Check

The Bearing check is executed according to [Ref.1] **art. 6.5.2** and **Annex D**

$$V_d \leq R_d$$

The Bearing resistance $\mathbf{R_d}$ depends on the fact whether the soil condition is drained or undrained.

In case the user ‘knows’ the soil capacity, $\mathbf{R_d}$ is read directly from the input data instead of being calculated.

Sliding Check

The Sliding check is executed according to [Ref.1] **art. 6.5.3**

$$H_d \leq R_d + R_{p,d}$$

The Sliding resistance $\mathbf{R_d}$ depends on the fact whether the soil condition is drained or undrained.

The value $\mathbf{R_{p,d}}$ specifies the positive effect of the earth pressure at the side of the foundation. Since this effect cannot be relied upon, this value is taken as zero [Ref.2].

Eccentricity Check

EC7 specifies in **art. 6.5.4** that special precautions are required for loads with large eccentricities.

According to [Ref.3] this is done by checking if the design load is within a critical ellipse or critical diamond.

Uplift Check

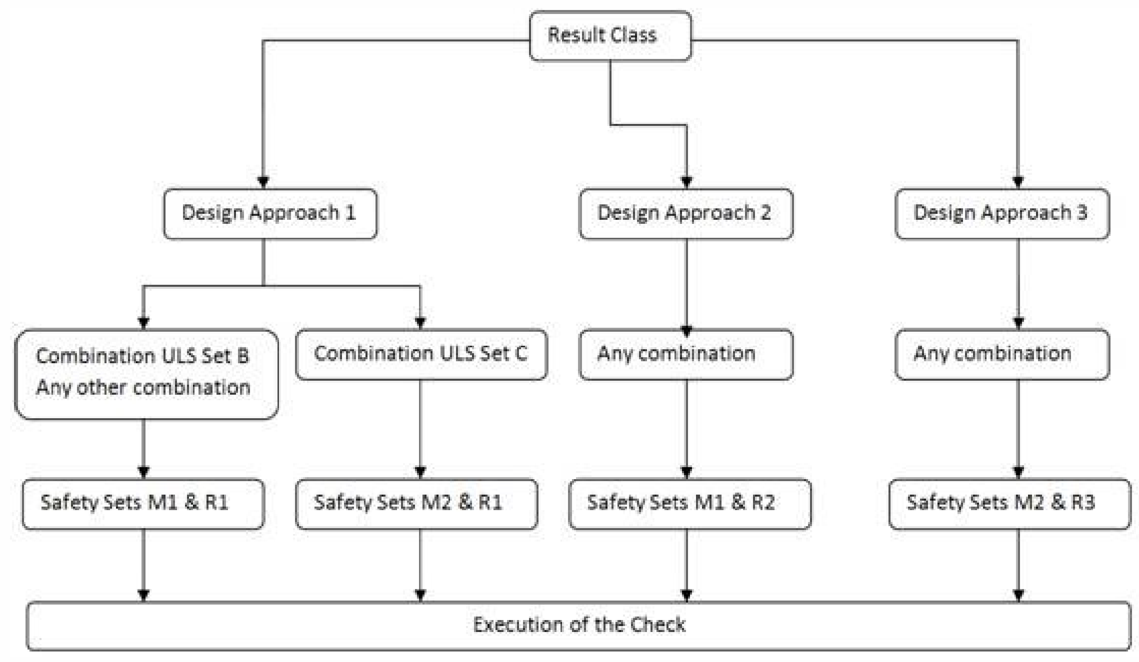
In case the vertical design loading $\mathbf{V_d}$ is negative, it implies that the pad foundation is in tension and may thus be ‘uplifted’ from the ground.

It is important to note that this check is executed INSTEAD of the three above checks.

Determination of Design Values

The check is executed for a Result Class.

Depending on the **Design Approach** set in the Geotechnical Design Setup, the sets of safety factors are read from the setup as follows:



For **Design Approach 1** the safety sets depend on the combination type.

For combinations of type **EN-ULS (STR/GEO) Set B** sets **M1** & **R1** are used.

For combinations of type **EN-ULS (STR/GEO) Set C** sets **M2** & **R1** are used.

For any other combination sets **M1** & **R1** are used.

For **Design Approach 2**, in all cases sets **M1** & **R2** are used.

For **Design Approach 3**, in all cases sets **M2** & **R3** are used.

Note 1: A code combination is internally split into different linear combinations. The check is executed for these linear combinations. It is thus important to retrieve from which code combination type a linear combination was generated in order to know which safety set to apply.

Note 2: The Result Class may off course also contain load cases or non-linear combinations. These are seen as 'Any combination' for the check.

Using the above information, the safety factors can be read from the Geotechnics Service and the design values for the soil properties can be determined:

Design Value	Formula
φ'_d	$= \operatorname{atan}\left[\frac{\tan(\varphi')}{\gamma_{\varphi'}}\right]$ With: φ' read from Subsoil Library $\gamma_{\varphi'}$ read from Geotechnics Setup
c'_d	$= \frac{c'}{\gamma_{c'}}$ With: c' read from Subsoil Library $\gamma_{c'}$ read from Geotechnics Setup

$c_{u,d}$	$= \frac{c_u}{\gamma_{cu}}$ With: c_u read from Subsoil Library γ_{cu} read from Geotechnics Setup
γ'_d	$= \frac{\gamma'}{\gamma_\gamma}$ With: γ' specific weight read from Library γ_γ read from Geotechnics Setup
$\gamma_{Backfill,d}$	$= \frac{\gamma_{Backfill}}{\gamma_\gamma}$ With: $\gamma_{Backfill}$ weight read from Pad foundation input Data γ_γ read from Geotechnics Setup

A final safety factor which needs to be determined concerns the safety factor for the weight of the pad foundation and the backfill material. This safety factor is taken as the safety factor for a permanent load for the combination under consideration i.e. $\mathbf{g_G}$.

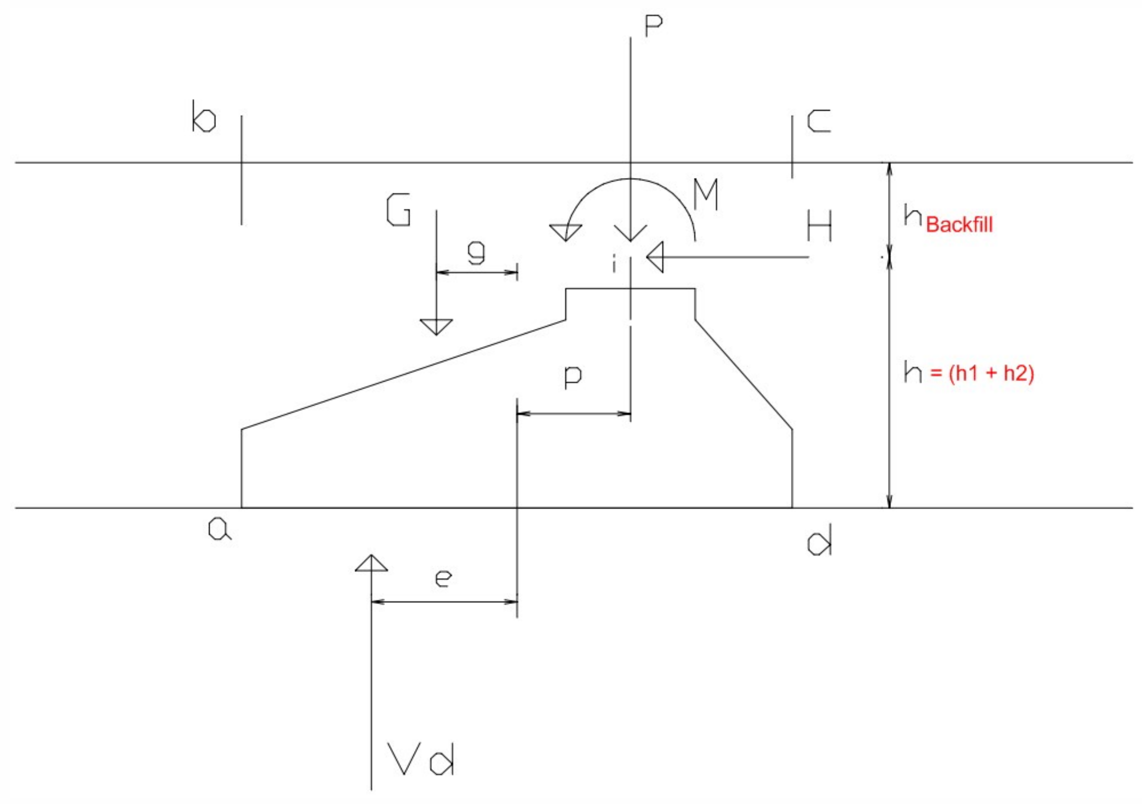
A permanent load can be seen as favourable or as unfavourable. The corresponding safety factor is determined as follows.

- The safety factor of the first permanent load case within an exploded combination is taken as $\mathbf{g_G}$. In this way, the correct value is found for any type of combination (code, linear, envelope, non-linear, …).
- In case the exploded combination does not have a permanent load case, $\mathbf{g_G}$ is taken as **1,00**.

Determination of Effective Geometry

The next step in the check concerns the determination of the effective geometry of the pad foundation.

The following picture illustrates the different actions working on the foundation.



In this picture the following notations are used:

Action	Info
G	Weight of the foundation and of any backfill material inside the area of abcd.
g	Load application point for load **G** referenced to the center point of the foundation base
P	Vertical **Rz** reaction of the support
p	Load application point for load **P** referenced to the center point of the foundation base. This is read as the load eccentricities ex and ey from the Pad Foundation library.
H	Horizontal **Rx** or **Ry** reaction of the support
h	=(h1 + h2) Load application point of the horizontal load H referenced to the foundation base. With h1 and h2 read from the Pad Foundation Library.
M	Moment **Mx** or **My** reaction of the support
V_d	= G + P Ultimate load vertical to the foundation base including the weight of the foundation and any backfill material.
e	Load application point for load $\mathbf{V_d}$ referenced to the center point of the foundation base

The reaction forces **Rx**, **Ry**, **Rz**, **Mx**, **My** need to be multiplied with the **Support Reaction Elimination factors**.

The eccentricity **e** can be calculated as follows:

$$e = \frac{M + G * g + H * h - P * p}{V_d}$$

For a general 3D case this formula is written as:

$$e_x = \frac{M_y + G * g_x + H_x * h - P * p_x}{V_d}$$

$$e_y = \frac{M_x + G * g_y + H_y * h - P * p_y}{V_d}$$

Weight G

The weight G consists of three parts:

1) The weight of the foundation block, $\mathbf{G_{Block}}$

This depends on the shape of the block (prismatic or pyramidal), dimensions and also the density g_{Block} of the block material.

All these data can be read from the Pad Foundation Library.

The density of the block depends on the **Water table level**.

· Water level	· Block Density
No influence	γ_{Block}
at foundation base	γ_{Block}
at ground level	$(\gamma_{Block} - \gamma_W)$

The Water Density $\mathbf{g_W}$ is taken as **9,81 kN/m³**

2) The weight of the backfill around h2, $\mathbf{G_{Backfill,Around}}$

This depends on the shape of the block (prismatic or pyramidal), dimensions and also the density of the backfill material.

All block data can be read from the Pad Foundation Library.

The density of the backfill depends on the **Water table level**.

· Water level	· Backfill Density
No influence	$\gamma_{Backfill,d}$
at foundation base	$\gamma_{Backfill,d}$
at ground level	$(\gamma_{Backfill,d} - \gamma_W)$

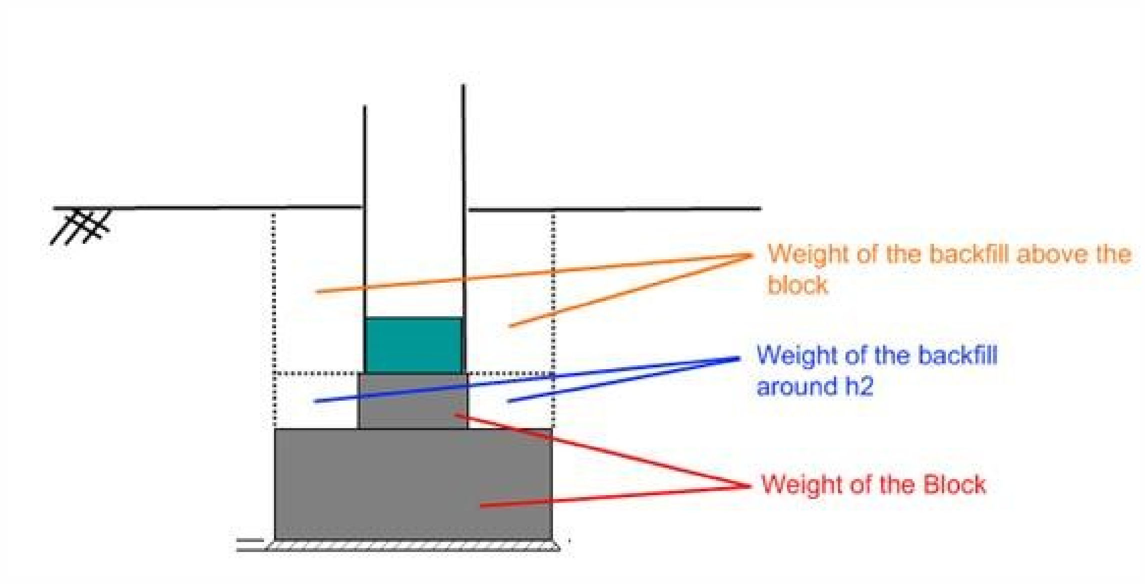
The Water Density $\mathbf{g_W}$ is taken as **9,81 kN/m³**

3) The weight of the backfill above the foundation block, $\mathbf{G_{Backfill,Above}}$

This depends on the height and density of the backfill as specified in the input of the Pad Foundation.

Note that the height of the backfill material can also be negative. Negative values are required to indicate that the soil is lower than the top of the foundation block.

The three parts are illustrated on the following picture:



The design value of the total weight G can then be calculated as follows:

$G_d = \mathbf{g_G} * [G_{Block} + G_{Backfill,Around} + G_{Backfill,Above}]$

With $\mathbf{g_G}$ the safety factor of the permanent loading for the combination under consideration.

Distances gx & gy

Using the weight and the volume, the center of gravity of the block and backfill can be determined. The distances gx and gy are then calculated from this centroid to the center point of the foundation base.

Effective Geometry

As a final step, using the eccentricities ex and ey the effective geometry of the foundation base can be calculated as follows:

L1 = A – 2 * |ex|

L2 = B – 2 * |ey|

With A & B read from the Pad Foundation library.

B' = min (L1 ; L2)

L' = max (L1 ; L2)

A' = B' * L'

In case B' < 0 m or L' < 0 m the geometry is incorrect.

In this case, the check is not executed and a warning is given on the output:

"Warning: The check cannot be executed due to incorrect effective geometry dimensions. Please review the base dimensions of the pad foundation!"

Undrained Bearing Resistance

The formulas in this paragraph are used in case the **Type** field in the Subsoil Library is set to **Undrained**.

The design value of the undrained bearing resistance is calculated as follows:

$$R_d = \frac{[(\pi + 2) * c_{ud} * b_c * s_c * i_c + q] * A'}{\gamma_{R,v}}$$

Value	Formula
c_{ud}	As specified earlier in this document.
b_c	Inclination of the foundation base (always horizontal base in SE) = 1,00
s_c	Shape of the foundation (rectangular shape) $= 1 + 0{,}2 * \frac{B'}{L'}$
i_c	Inclination of the load, caused by horizontal load H_d $= \frac{1}{2}\left[1 + \sqrt{1 - \frac{H_d}{A' * c_{ud}}}\right]$ and $H_d \leq A' * c_{ud}$ in case $H_d > A' * c_{ud}$ the value of $\mathbf{i_c}$ can be set to **0,5**
H_d	Resulting horizontal load $= \sqrt{H_x^2 + H_y^2}$
H_x	Horizontal support reaction Rx.
H_y	Horizontal support reaction Ry.
B'	Effective width.
L'	Effective length.
A'	Effective area.
q	Overburden at the foundation base [Ref.5] $=(h1 + h2 + h_{backfill}) * g_{Backfill,d}$ With: h1 & h2 read from the Pad Foundation Library $h_{backfill}$ read from the Pad Foundation input $g_{Backfill,d}$ as defined earlier in this document.
$g_{R,v}$	Resistance factor read from the Geotechnics Setup

Drained Bearing Resistance

The formulas in this paragraph are used in case the **Type** field in the Subsoil Library is set to **Drained**.

The design value of the drained bearing resistance is calculated as follows:

$$R_d = \frac{\left[c'_d * N_c * b_c * s_c * i_c + q'_d * N_q * b_q * s_q * i_q + 0{,}5 * \gamma'_d * B' * N_\gamma * b_\gamma * s_\gamma * i_\gamma\right] * A'}{\gamma_{R,v}}$$

Value	Formula
c_d'	As specified earlier in this document.

N_c	Bearing resistance factor $= (N_q - 1) * \cot(\varphi'_d)$
N_q	Bearing resistance factor $= e^{\pi * \tan(\varphi'_d)} * \tan^2(45 + \frac{\varphi'_d}{2})$
Ng	Bearing resistance factor $= 2 * (N_q - 1) * \tan(\varphi'_d)$
b_c	Inclination of the foundation base (always horizontal base in SE) = 1,00
b_q	Inclination of the foundation base (always horizontal base in SE) = 1,00
b_γ	Inclination of the foundation base (always horizontal base in SE) = 1,00
s_c	Shape of the foundation (rectangular shape) $= \frac{s_q * N_q - 1}{N_q - 1}$
s_q	Shape of the foundation (rectangular shape) $= 1 + \left(\frac{B'}{L'}\right) * \sin(\varphi'_d)$
s_γ	Shape of the foundation (rectangular shape) $= 1 - 0,3 * \frac{B'}{L'}$
i_c	Inclination of the load, caused by horizontal load H_d $= i_q - \frac{(1 - i_q)}{N_c * \tan(\varphi'_d)}$
i_q	Inclination of the load, caused by horizontal load H_d $= \left[1 - \frac{H_d}{V_d + A' * c'_d * cot(\varphi'_d)}\right]^m$
ig	Inclination of the load, caused by horizontal load H_d $= \left[1 - \frac{H_d}{V_d + A' * c'_d * cot(\varphi'_d)}\right]^{m+1}$

m	$= m_L * \cos^2(\theta) + m_B * \sin^2(\theta)$
m_L	$= \frac{\left[2 + \left(\frac{L'}{B'}\right)\right]}{\left[1 + \left(\frac{L'}{B'}\right)\right]}$
m_B	$= \frac{\left[2 + \left(\frac{B'}{L'}\right)\right]}{\left[1 + \left(\frac{B'}{L'}\right)\right]}$
q	Angle of the horizontal load H_d with the direction L'
φ'_d	As specified earlier in this document.
B'	Effective width as defined earlier in this document.
L'	Effective length as defined earlier in this document.
A'	Effective area as defined earlier in this document.
H_d	Resulting horizontal load $= \sqrt{H_x^2 + H_y^2}$
H_x	Horizontal support reaction Rx.
H_y	Horizontal support reaction Ry.
V_d	As specified earlier in this document.
q'_d	Effective overburden at the foundation base [Ref.5] $=(h1 + h2 + h_{backfill})^* \gamma'_t$ With: h1 & h2 read from the Pad Foundation Library $h_{backfill}$ read from the Pad Foundation input γ'_t is depending on the water level as follows: <table><tr><td>Water level</td><td>γ'_t</td></tr><tr><td>No influence</td><td>$\gamma_{Backfill,d}$</td></tr><tr><td>at foundation base</td><td>$\gamma_{Backfill,d}$</td></tr><tr><td>at ground level</td><td>$(\gamma_{Backfill,d} - \gamma_W)$</td></tr></table> $\gamma_{Backfill,d}$ as defined earlier in this document. γ_W is taken as **9,81 kN/m³**
γ'_d	Effective weight density of the soil below the foundation level depending on the water level as follows: · Water level · γ'_d

	No influence γ'_d at foundation base $(\gamma'_d - \gamma_W)$ at ground level $(\gamma'_d - \gamma_W)$ γ'_d as defined earlier in this document. γ_W is taken as **9,81 kN/m³**
$\gamma_{R,v}$	Resistance factor read from the Geotechnics Setup

Known Soil Capacity Bearing Resistance

In case the Soil capacity is known, this value can be used directly instead of using the EC7 bearing resistance calculation.

This procedure is applied in case the checkbox **Known soil capacity, use Sigma oc** is activated in the Geotechnical Design Setup.

The design value of the bearing resistance is calculated as follows:

$$R_d = A' * \sigma_{od}$$

Value	Formula
A'	Effective area as defined earlier in this document.
σ_{od}	Design value of the admissible soil capacity, taken as $\mathbf{s_{oc}}$
σ_{oc}	Read from the Subsoil Library

Sliding Resistance

The sliding resistance is dependent on the condition of the subsoil.

a) In case the **Type** field in the Subsoil Library is set to **Undrained**.

$$R_d = \frac{A' * c_{ud}}{\gamma_{R,h}}$$

Value	Formula
c_{ud}	As specified earlier in this document.
A'	Effective area as defined earlier in this document.
$\gamma_{R,h}$	Resistance factor read from the Geotechnics Setup

In case the checkbox **Water/air in clay subgrade** in the Subsoil Library is activated, the value of $\mathbf{R_d}$ is limited as follows:

$$R_d \leq 0{,}4 * V_d$$

Value	Formula
V_d	As specified earlier in this document.

b) In case the **Type** field in the Subsoil Library is set to **Drained**.

$$R_d = \frac{V_d * \tan(\delta_d)}{\gamma_{R,h}}$$

Value	Formula
V_d	As specified earlier in this document.
δ_d	Design friction angle at the foundation base Dependent on the **Cast condition** specified in the Pad Foundation Library: · Cast Condition · dd Prefabricated $\frac{2}{3} * \varphi'_d$ In situ φ'_d
φ'_d	As specified earlier in this document.
$\gamma_{R,h}$	Resistance factor read from the Geotechnics Setup

Eccentricity check

To prevent special precautions according to **art. 6.5.4** the eccentricity of the load shall not exceed **1/3** or **1/6** of the width.

The maximal value of the eccentricity is defined in the Geotechnical Design Setup.

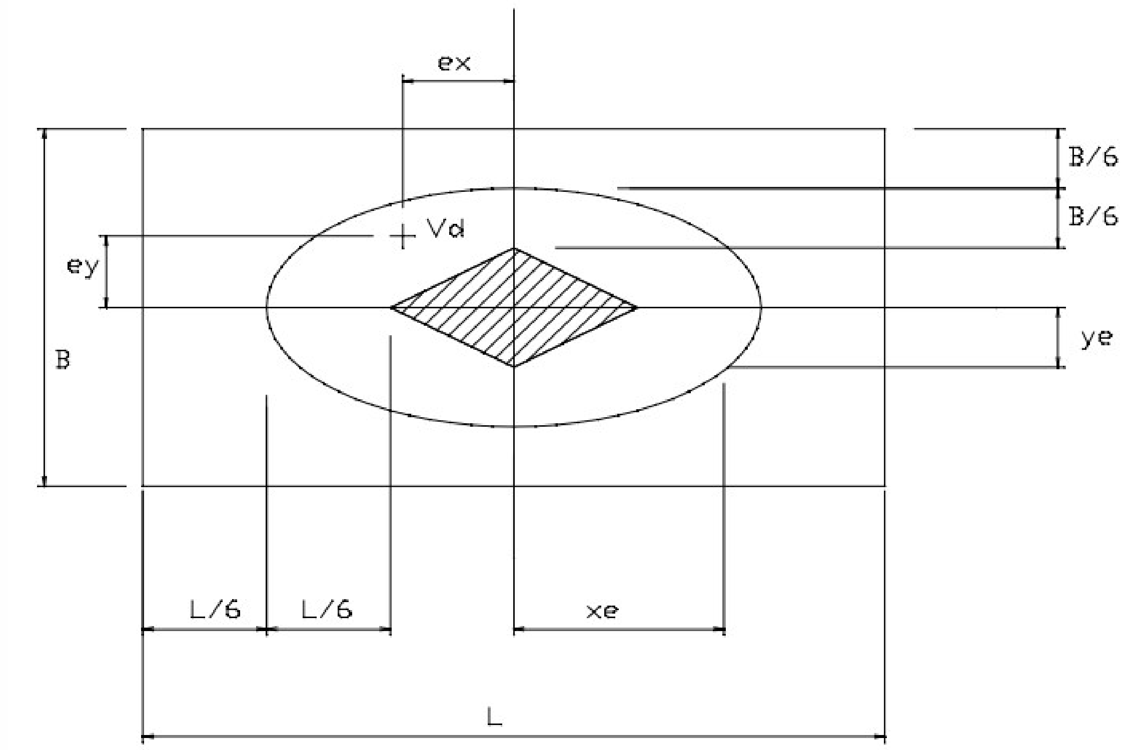
a) In case the maximal eccentricity is set to **1/3**

$$\left(\frac{e_x}{A}\right)^2 + \left(\frac{e_y}{B}\right)^2 \leq \frac{1}{9}$$

b) In case the maximal eccentricity is set to **1/6**

$$\frac{e_x}{A} + \frac{e_y}{B} \leq \frac{1}{6}$$

Value	Formula
e_x	As specified earlier in this document.
e_y	As specified earlier in this document.
A	Read from Pad Foundation Library
B	Read from Pad Foundation Library



c) In case the maximal eccentricity is set to **No limit**

In this case there is no limit i.e. any eccentricity is allowed. The unity check is set to **0,00**.

Uplift Check

In case the vertical design loading $\mathbf{V_d}$ is negative, it implies that the pad foundation is in tension and may thus be 'uplifted' from the ground.

The uplift check can be written out as follows:

$$|P| \leq G_d$$

Value	Formula
P	The vertical **Rz** reaction as specified earlier in this document.
G_d	The weight of the foundation and any backfill as specified earlier in this document.

Output

Both a Brief and Detailed output are available with table composer support.

Unity check values exceeding **1.00** is shown in bold.

AutoDesign

For optimizing the pad foundation, the sensitivity optimization and grid are used as implemented for Steel.

Pad foundation optimization is added both in the Pad Foundation Stability Check Service as well as in the Overall Optimization Solver.

The maximal check limit is configurable for each of the three main checks:

- **Maximal check on bearing**
- **Maximal check on sliding**
- **Maximal check on eccentricity**

By default, the maximal value for each of the three checks is set to **1,00**. Note that all three input fields should only allow the input of positive values.

For each of the three checks a **Maximal unity check** field is shown which shows the current value of the unity check.

The picture shows the pad foundation geometry and is taken from the Pad Foundation Library. In the same way as it is done in steel, during the optimization, the dimensional changes are shown in the picture.

The **Change Pad foundation** button opens the Pad Foundation Library and allows the user to modify the pad foundation or select a different one. It functions in the same way as the Edit button in the steel optimization.

The **Next down** and **Next up** buttons function in the same way as for steel, the selected grid parameter is modified up or down one step.

The **Search for optimal** button function in the same way as for steel, the selected grid parameters are optimized.

The **Direction** combo functions in the same way as for steel. The user can choose to set it to ‘Up & down’ so the AutoDesign works in two directions or the user can set it to ‘Only up’ so the AutoDesign can only increase the parameters. By default the combo is on ‘Up & down’.

The **Parameter** combo allows the user to set which parameter(s) should be optimized. The user can choose any of the pad foundation dimensions or set the combo to Advanced AutoDesign which allows the optimization of several parameters together (Sensitivity).

The parameters are **A**, **B**, **h1**, **h2**, **h3**, **a**, **b**, **ex** and **ey**.

By default the combo shows the **A** parameter.

The optimization grid has the same layout as used for steel except for the ‘Sort By’ column which is removed.

The grid itself functions in the same way as the grid for steel.

In the same way as for steel, the user can assign a Dimension List to a parameter. During AutoDesign, only values from the list are used.

The **Set value** button can be used to modify a selected parameter from the grid. In the same way as for steel, the dialog which appears depends on whether the list has been assigned to the selected parameter or not.

Using **Select/Deselect All** the user can quickly select or deselect all parameters in the grid.

As in steel, parameters can be set in relation to one another. The user can use the **Test relations** button to check if no loops have been made.

Important remark: in steel, several validity tests are run when starting the AutoDesign, for example the test on relations is done automatically. The same tests are also executed automatically for pad foundations.

The Advanced AutoDesign itself uses the Sensitivity algorithm:

- Within each iteration, each parameter is changed separately with its step and it is evaluated which change has the most influence on the Pad foundation utilization. This one is then taken and applied.

The procedure is then repeated in the next iteration until the Pad foundation has a unity check below 1,00.

- When below 1,00 the same procedure is applied (changing each parameter separately with its step) but now the goal is to get as close to 1,00 as possible and not to go over 1,00. This means that the parameters will now be decreased in value.

- In the end, a situation is reached where no parameter can be decreased with its step anymore since this will lead to a unity check above 1,00. This is then the most optimal solution.

References

[1]	EN 1997-1, *Eurocode 7: Geotechnical design – Part 1: General rules*, CEN, 2004.
[2]	Frank R., Baudoin C., Driscoll R., Kavvadas M., Krebs Ovesen N., Orr T., Schuppener B., *Designer's Guide to EN 1997-1 Eurocode 7: Geotechnical design – Part 1: General rules*, Thomas Telford, 2004.
[3]	Schneider K.-J., Bautabellen für Ingenieure, 13. Auflage, Werner Verlag, 1998.
[4]	EN 1990, *Eurocode – Basis of Structural Design*, CEN, 2002.
[5]	Lambe T., Whitman R., *Soil Mechanics*, MIT, John Wiley & Sons, Inc, 1969.

Foundation strip

A foundation strip is used as a kind of linear supporting. It is defined by its width and by the properties of soil below the footing surface.

All the properties of the foundation strip are defined in the property dialogue of support, i.e. at the moment the support is being inserted into the model.

Upper soil of foundation block

Supporting of a structure defined by means of a foundation structure (i.e. foundation block) is defined not only by the dimensions of the foundation structure and properties of the soil below the footing surface, but also by the characteristics of the upper soil.

Density	Specifies the density of the soil above the foundation block or strip.
Height	Defines the height of the upper soil layer. The height is measured from the top-surface of the foundation block.

In addition, the user may specify the level of underground water that also influences the characteristics of the support.

Note: The upper soil parameters are taken into account if the foundation block is checked for stability.

Defining a new foundation block type

The procedure for the definition of a new foundation block type

1. Open the Foundation block manager:
    1. either: use tree menu item Library > Foundation blocks,
    2. or: use menu item Libraries > Foundation blocks.
2. Click button [New].
3. A new foundation block is created and it is added to the list of defined types.
4. Click button [Edit].
5. The editing dialogue appears on the screen.
6. Select the type you want to define.

7. Input the dimensions of the block.
8. Choose or define block's material.
9. Confirm with button [OK].
10. Close the Foundation block manager

> Note: If no foundation block has been defined so far and the user opens the Foundation block manager, the program may automatically open the Foundation block editing dialogue directly. Once the editing dialogue is closed, the Foundation manager appears on the screen and the user may follow to procedure given above.

Inserting the foundation block into model

As the foundation block is a type of support it can be inserted like a standard point support. Therefore, the procedure for the insertion of a point support can be applied. The only difference is, that the user has to specify some additional parameters that are unique for this support type.

> Note: If no subsoil and no type of foundation block has been defined and the user tries to insert a support of Foundation block type, the program automatically creates a default subsoil type and a default foundation block type. It is up to the user to edit these entities and input proper values of their characteristics.

Defining a new foundation strip

As the foundation strip is a type of linear support it can be inserted like a standard linear support. Therefore, the procedure for the insertion of a linear support can be applied. The only difference is, that the user has to specify some additional parameters that are unique for this support type.

> Note: If no subsoil has been defined and the user tries to insert a support of Foundation strip type, the program automatically creates a default subsoil type. It is up to the user to edit this entities and input proper characteristics of it.

Subsoil

Introduction to subsoil

Supports of a "foundation" type, i.e. foundation block and foundation strip, are laid on the soil that forms the base for the structure. The parameters of this soil must be defined in order to allow the program to perform accurate calculations.

In Scia Engineer the "under-foundation" soil is called subsoil and can be defined using functions:

- either: tree menu function Library > Subsoils,
- or: menu function Libraries > Subsoils.

Once at least one subsoil type is defined, it can be used for the definition of foundation blocks or foundation strips.

Defining a new subsoil type

A new subsoil type can be defined by means of the Subsoils manager. It is one of the numerous Scia Engineer database managers.

The procedure for the definition of a new subsoil type

1. Open the Subsoils manager:
    1. either: tree menu function Library > Subsoils,
    2. or: menu function Libraries > Subsoils.
2. Click button [New] to create a new subsoil entity.
3. The new subsoil type is added to the list of defined subsoils.
4. Click button [Edit].
5. The editing dialogue is opened on the screen.
6. Input required values for individual parameters.
7. Confirm the parameters with button [OK].
8. Repeat steps 2 to 7 as many times as required.
9. Close the Subsoils manager.

Defining subsoil parameters

The definition of subsoil parameters can be done in the editing dialogue for subsoil. The editing dialogue is accessible via the Subsoils manager.

Parameters of subsoil

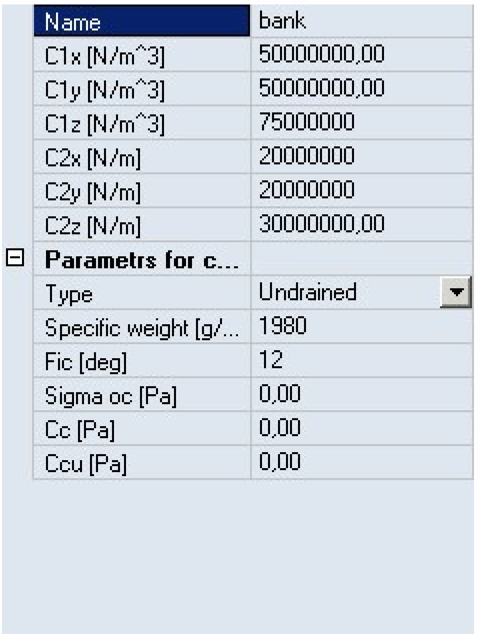
Constants C1 and C2 for directions X, Y, Z	C parameters representing the subsoil properties. (see also Subsoil parameters for subsoil under a beam).

Parameters for check	Here the parameters necessary for check of the subsoil to a technical standards are defined.

Parameters for check

These data are used only for the stability check of a foundation block.

Density	Soil density
Fic	The value of the angle of the shearing resistance in terms of effective stress.
Cc	The value of the cohesion intercept in terms of effective stress.
Ccu	The value of the undrained shear strength.
Sigma oc	The admissible ground stress (optional).
Type	The soil can be Undrained or Drained.



Subsoil parameters for subsoil under a 1D member

The parameters of subsoil for subsoil defined under a 1D member are:

C1x	resistance of environment against ux (deformation in local x direction)
C1y	resistance of environment against uy (deformation in local y direction)
C1z	resistance of environment against uz (deformation in local z direction)
C2x	resistance of environment against dux/dx
C2y	resistance of environment against duy/dx
C2z	resistance of environment against duz/dx

Note: In the complete set of 6 parameters C, four parameters are significant and, if available, can be determined from the C parameters of subsoil 2D model in EPW Soilin module and from the stiffness of boundary bonding "k" modelling the effect of settlement basin:

C*1x (MN/m2)	= b (m) C1x (MN/m3)
C*1y (MN/m2)	= b (m) C1y (MN/m3)
C*1z (MN/m2)	= b (m) C1z (MN/m3) + 2 k (MN/m2)
C*2z (MN)	= b (m) C2x (MN/m)
	where b is the width of the member.

It is not recommended to use the remaining two parameters. Reliable experimental data are not available for C2x and C2y.

Subsoil parameters for subsoil under a slab

The parameters of subsoil for subsoil defined under a slab are:

C1z	resistance of environment against wP (mm) [C1z in MN/m3]
C2x	resistance of environment against wP/xP (mm/m) [C2x in MN/m]
C2y	resistance of environment against wP/yP (mm/m) [C2y in MN/m]
C1x	resistance of environment against uP (mm) [C1x in MN/m3]
C1y	resistance of environment against vP (mm) [C1y in MN/m3]

Note: Usually, C2x is considered equal to C2y and C1x equal to C1y.

Note: See also chapter Model data > Foundation > Subsoil.

Using the subsoil

Subsoil is used as a parameter for the definition of "foundation structures". That means as a parameter for foundation blocks and foundation strips. Both of these foundation structures are a kind of point or linear support.

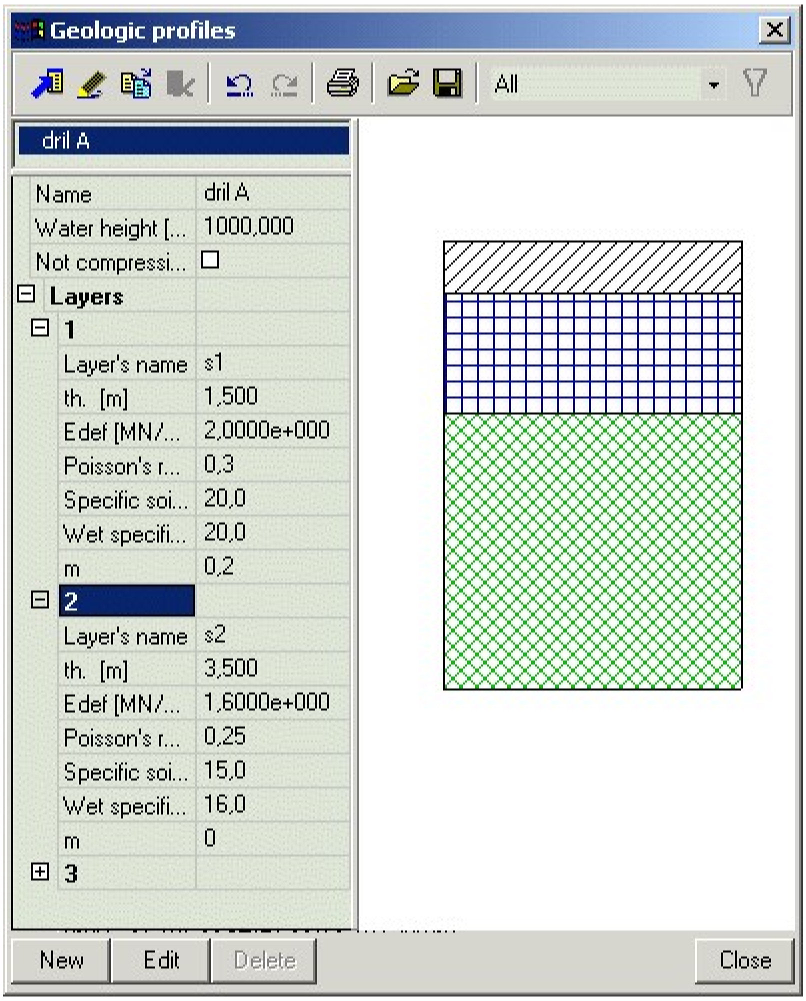
Consequently, the subsoil type used for particular foundation block or foundation strip is adjusted in the property dialogue of a support.

Geologic profile

Geologic profile manager

Geologic profile manager is a standard database manager. Its operation is therefore quite straightforward.

It may look like:



You can perform all common operations with geologic profiles:

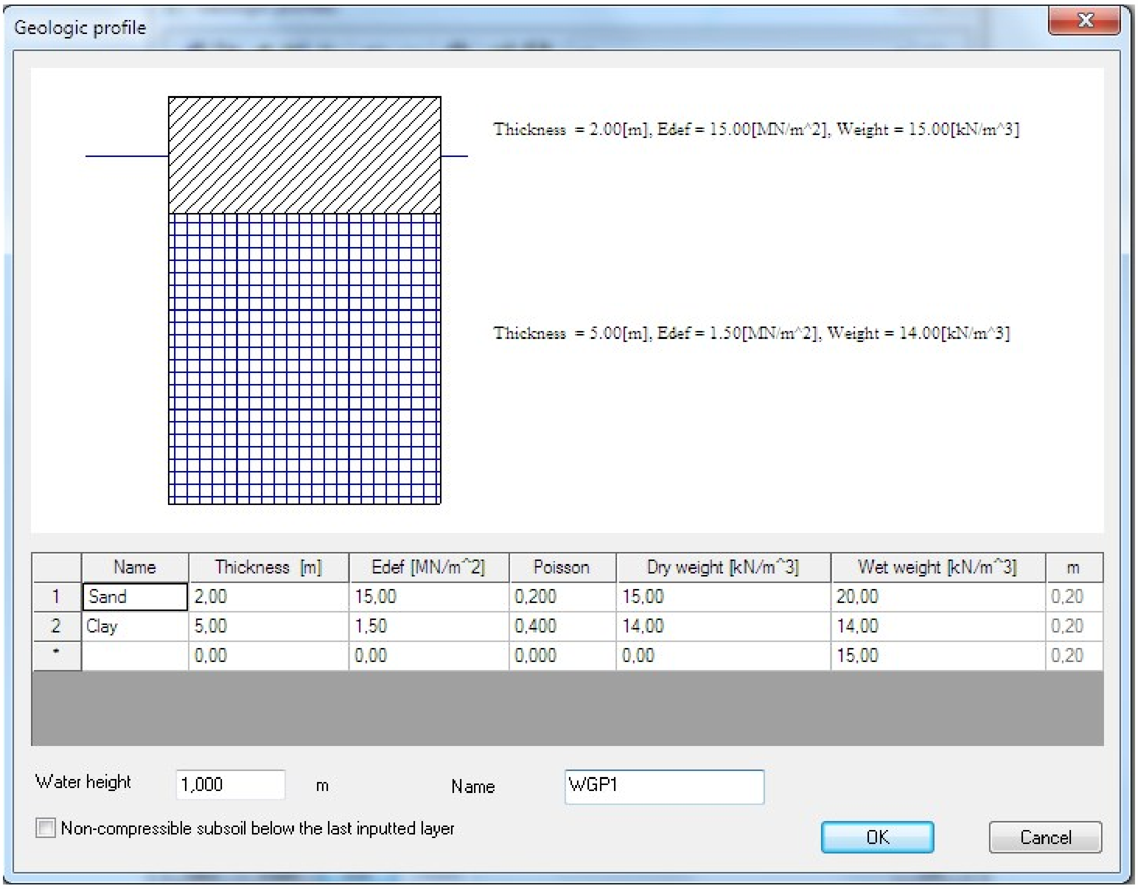
- define a new one,
- edit the existing one,
- make a copy of the existing one,
- delete the existing one (unless it is used in the model),
- print or save the information about it,
- read it from your disk (if you have saved it some time ago).

The Geologic profile manager can be opened via:

- tree menu Library > Geologic profiles,
- menu Libraries > Geologic profiles,
- it is also opened automatically whenever some entity that requires a geologic profile as a parameter is being input and no geologic profile has been defined yet.

Defining a new geologic profile

A new geologic profile can be input in the Geologic profile manager. Function New of the manager opens the Geologic profile dialogue.



General geologic profile parameters

Water level	Defines the level of underground water. The water level influences the parameters of the soil.
Name	Specifies the name of the geologic profile.
Not compressible subsoil	In ON, the program applies coefficient of depth reduction k2 in compliance with CSN 73 1001, art. 80. Numerically it means that the damping of stress component sz in the half-space is slowed down. All components of elastic-half-space-stress-tensor are calculated in this reduced depth. It is just an approximate calculation, not an exact solution of the elastic layer. The difference is however negligible in comparison with other inacuracies.

Layer-related parameters

name	name of the layer
thickness	thickness of the layer
E def	module of deformation For geotechnical categories 1 and 2 the indicative value from e.g. CSN 73 1001 can be used, for category 3 a survey should be carried out to provide for the value.
Poisson's ratio	coefficient of transverse deformation An indicative value or experimentally found value can be used. (range: 0 – 0.5)
specific soil weight	specific soil weight for dry soil normally within the range from 18 to 23 kN/m^3
wet specific soil weight	specific soil weight for wet soil
m	structural strength coefficient Dimensionless value in the formula for settlement according to CSN 73 1001. $s = \sum_{i=1}^{n} \frac{\sigma_{z,i} - m_i \sigma_{or,i}}{E_{oed,i}} h_i$ Table 10 in the standard states indicative values for various soils in the range from 0.1 to 0.5. For category 3 it is advisable to consult the engineer who carried out the survey of the locality in question. For other codes (other than CSN) this coefficient is equal to 0.2.

Geologic profile must be defined up to such a depth where the effective stress is still active, otherwise the program does not have enough information.

Edit the content of the table

Delete or insert new row:

1. Click on the row header.
2. Display the right click menu.
3. Select the option Delete line, Insert line or Delete all lines.

Copy and paste the content from the clipboard

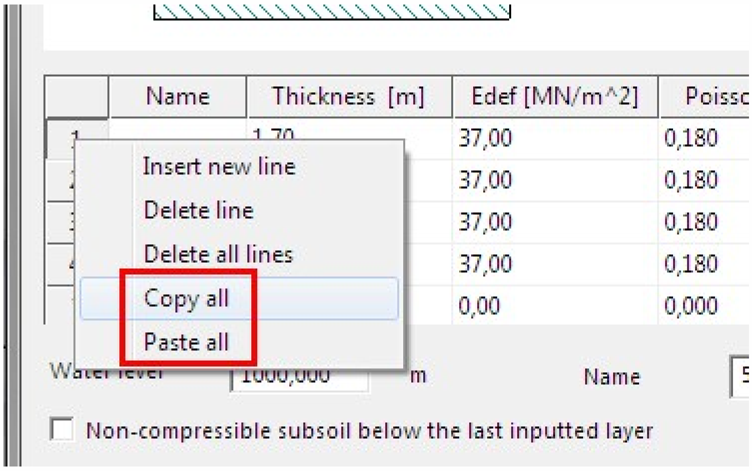
Copy content to the clipboard:

1. Click on the row header.
2. Display the right click menu.

3. Select the option Copy all.
4. Paste the content of the table to any editor (e.g. MS Excel, MS Word, OpenOffice, Notepad etc.).

Paste content to the table:

1. Click on the row header.
2. Display the right click menu.
3. Select the option Paste all.



Editing the existing geologic profile

An existing geologic profile can be edited in the Geologic profile manager. Function Edit of the manager opens the Geologic profile dialogue. The dialogue is described in chapter Defining a new geologic profile.

Geologic areas

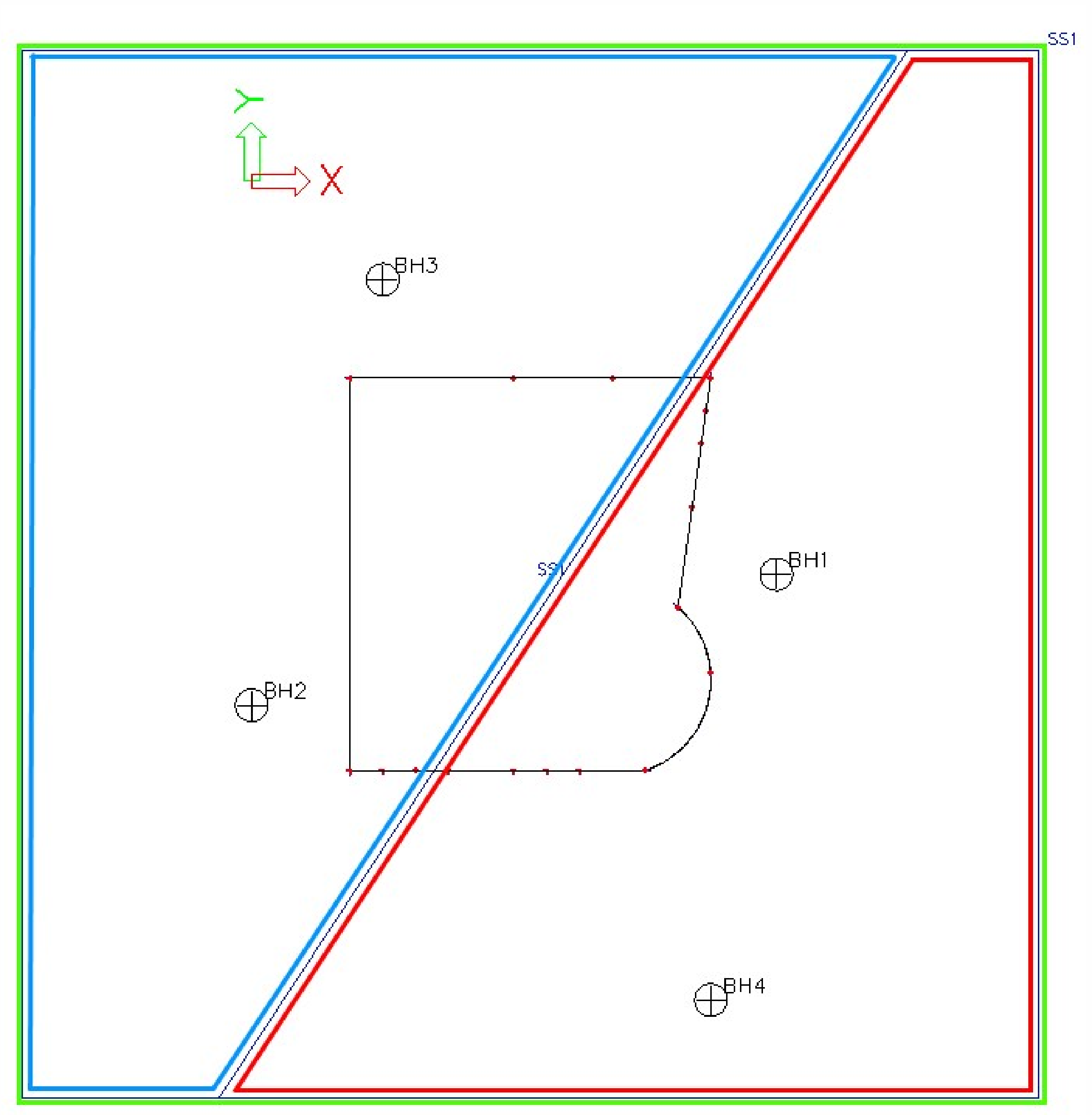
Introduction to geologic areas

The 3D model with defined subsoil and geologic profiles displays the subsoil surface. This surface defines the area where soil properties between boreholes is inter- and extra- polated.

Be to able to define a geologic fault, the basic surface polygon has been divided to the separated areas which are inter- and extra- polated but the first area doesn't affect the next one.

Different number of layers in the geologic profile may be used in different areas. For example 5 layers in all boreholes in area 1 and 8 layers in all boreholes in area 2.

The line between two geologic areas is **geologic fault**.



Green - basic outline of the subsoil surface
Red - Geologic area 1

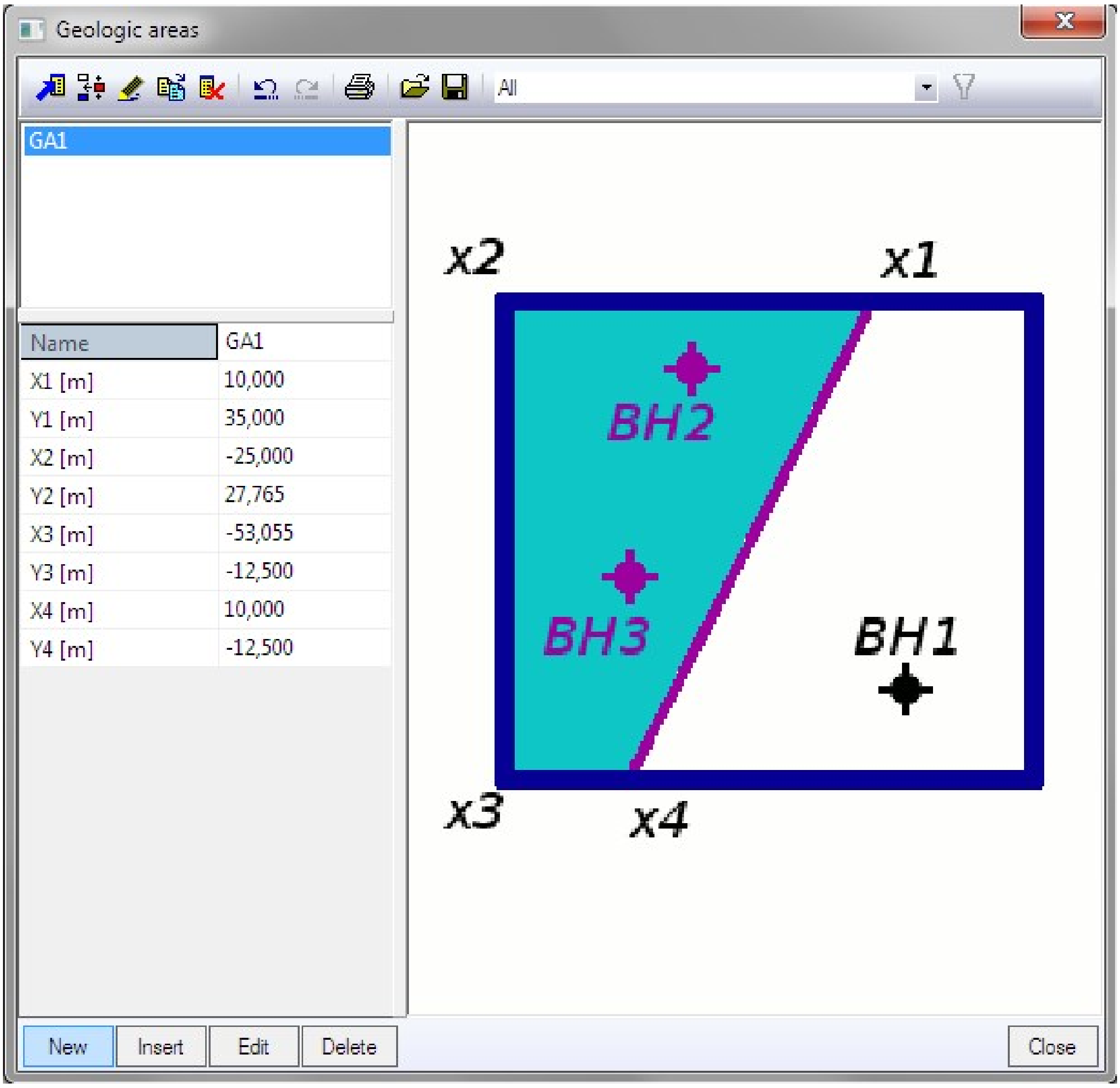
Blue - Geologic area 2
Red-Blue line - Geologic fault

Defining a new geologic area

Geologic area library is a standard database which contains the geometry - 4 points.

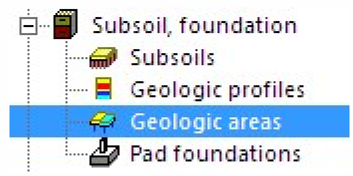
You can perform all common operations with geologic profiles:

- define a new one,
- edit the existing one,
- make a copy of the existing one,
- delete the existing one (unless it is used in the model),
- print or save the information about it,
- read it from your disk (if you have saved it some time ago).



In Scia Engineer the geologic area can be defined using functions:

- either: tree menu function Library > Subsoils > Geologic area,
- or: menu function Libraries > Subsoils > Geologic area.



Editing the existing geologic area

The procedure to edit the existing geologic area

1. Open:
    - either: menu function Libraries > Subsoils > Geologic area.
    - or: Select the Subsoil surface in the model and use the action button "Edit geologic areas" in the property window.
2. The library is opened.
3. Change required coordinates.
4. The changes are immediately taken into account.

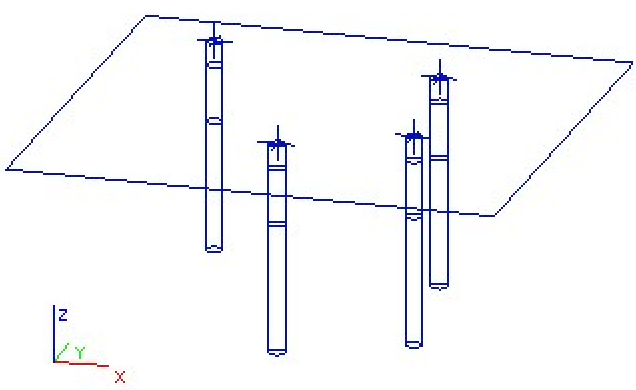
Boreholes

Introduction to boreholes

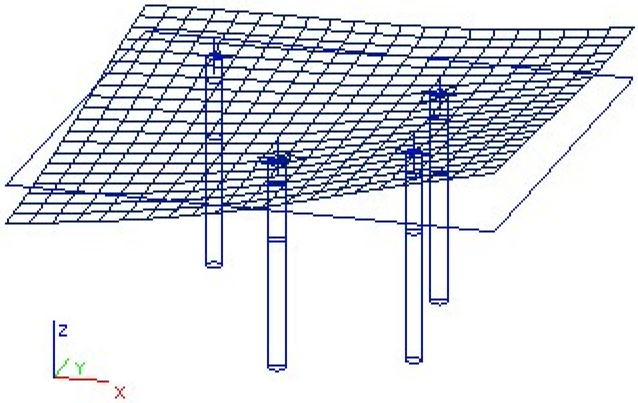
Boreholes together with geologic profiles provide the program with information relating to the composition of foundation soil. Both data are necessary to calculate the interaction between the structure and the soil below it.

A borehole is fully defined by the (i) corresponding geologic profile, (ii) location and (iii) altitude. Usually a set of boreholes will be defined and thus they can be used to calculate and display the surface of the land in their surrounding. This surface can be used for impressive presentations of your project. The surface itself is not taken into account during the calculation.

The following picture shows an example of defined boreholes. The rectangle represents the patch of land over which the soil properties can be inter- and extra- polated.

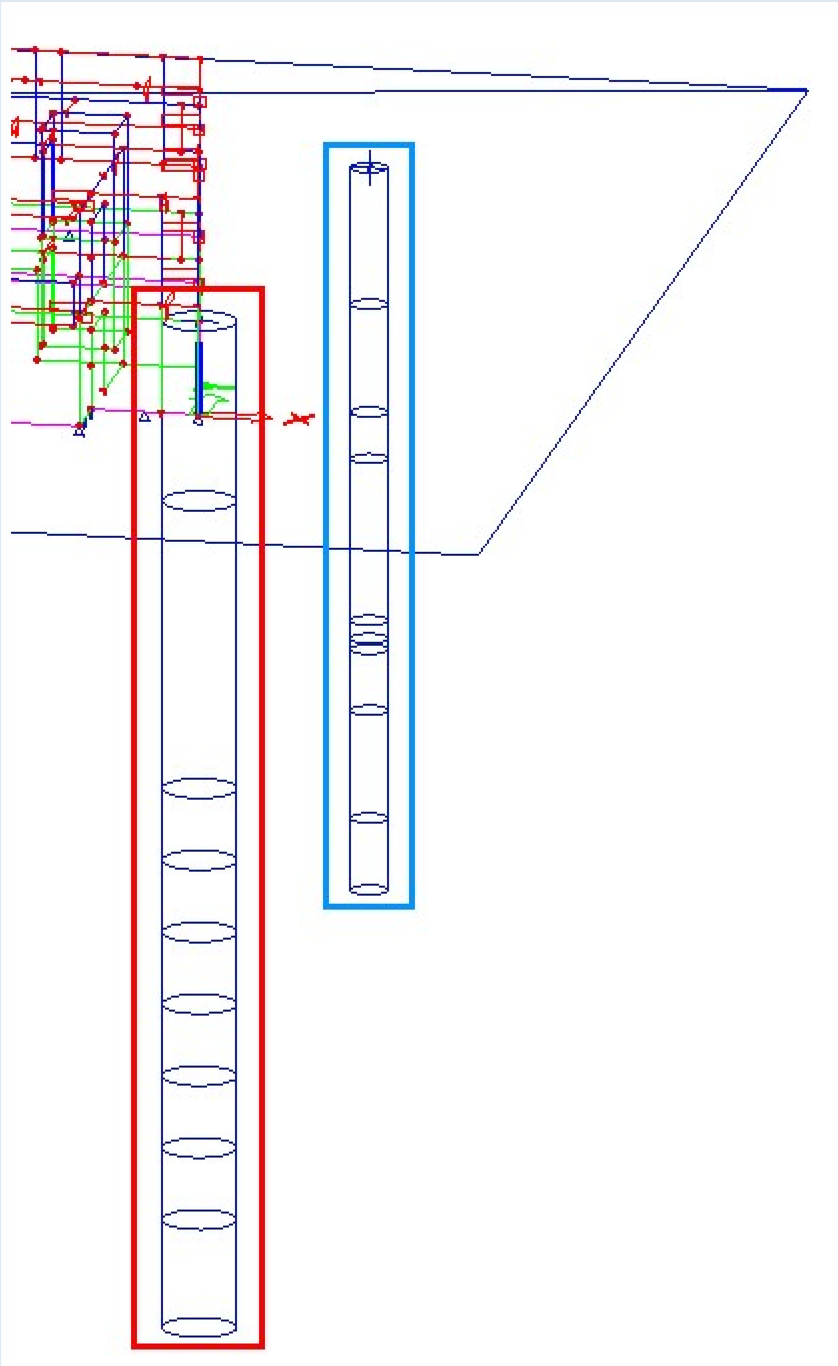


Next picture than shows the calculated surface.



Sand-gravel pile

There is a possibility to use the borehole as a sand-gravel pile. The sand-gravel pile consists from geologic profile and a geometry which defines its outline. The sand-gravel-pile outlines has the same behaviour as geologic fault.



The red colour shows a sand-gravel-pile with diameter 1m.
The blue colour shows a standard borehole.
Both are displayed inside the subsoil surface outline.

Inserting a new borehole

Procedure to define a new borehole

1. Open service Structure.
2. Start function Borehole profile.
3. Fill in the parameters.
4. Confirm with [OK].
5. Define the location of the new borehole or boreholes.

Borehole parameters

Name	Identifies the borehole profile.
Coord X, Y, Z	Coordinates of the inserting point of the borehole.
Results only	When the calculation is performed, you can obtain a table of settlement. The values of settlement are calculated in places where boreholes are located. The borehole itself (the corresponding geologic profile) is also used as an input value for the calculation of interaction between the structure and the soil. However, it is possible to exclude some boreholes from the input data and use them only as the location for the calculation of results – settlement. If this parameter is ON, the geologic profile defined in the borehole is ignored, the conditions in this place are interpolated from surrounding boreholes, but final settlement is calculated in this location.
Geologic profile	Specifies the geologic profile corresponding to the location of the borehole.
Sand-gravel pile	Defines if the borehole is used as a sand-gravel pile.
Radius	Specifies the radius of sand-gravel pile.

Editing the existing borehole

The procedure to edit the existing borehole

1. Select the required borehole.
2. The property window shows the parameters of the borehole.
3. Change required parameters.
4. The changes are immediately taken into account.

After the modification (especially the modification of the position) of the borehole, it may be necessary to refresh the surface (supposing it was displayed before the changes).

Deleting the existing borehole

A borehole that has been defined by mistake or is no longer necessary for any other reason can be deleted like any other entity in the model.

The procedure to delete borehole

1. Select the required borehole.
2. Delete it using:
    1. press key [Del] on your keyboard,
    2. invoke the pop-up menu and select function Delete,
    3. use menu function Modify > Delete.

Displaying or hiding the existing boreholes

Display (or we can say visibility) of boreholes is controlled by view parameter Subsoil > Borehole profiles.

The procedure to display (hide) the boreholes

1. Open dialogue View parameters settings.
    1. via pop-up menu function Set view parameters for all.
    2. via button of the toolbar of the graphical window.
2. Set Subsoil > Borehole profiles ON (or OFF to hide).
3. Confirm with [OK].

Displaying the earth surface

The surface is calculated and displayed across the area that is defined by two conditions:

- all defined boreholes lie inside the area,
- the distance from the outline of the structure to the border of the area is at least 10 metres in any direction.

The geologic area is displayed together with the surface outline.

Display (or we can say visibility) of surface is controlled by view parameter Subsoil > Surface.

The procedure to display (hide) the surface

1. Open dialogue View parameters settings.
    1. via pop-up menu function Set view parameters for all.
    2. via button of the toolbar of the graphical window.
2. Set Subsoil > Surface ON (or OFF to hide).
3. Confirm with [OK].

Refreshing the earth surface

After some changes to the boreholes (or after some other modifications of the model) the surface may disappear. The reason is the that change performed requires the regeneration of the surface and the regeneration of it is not automatic (mainly for speed-related reasons).

The procedure to refresh the outline and/ or surface

1. If the outline is not displayed, display it.
2. Select the outline.
3. The property window shows some basic information about it and also offers two action buttons.
4. Press [Refresh outline] to refresh the outline.
5. The outline may change if new boreholes have been added after the last refresh of the surface or if some existing boreholes has been moved to new locations.
6. Press [Refresh surface] to refresh the surface.

Soil-In

Introduction

The analysis of foundation structures is challenged by the problem of modelling of the part of the foundation that is in contact with subsoil. The best solution is to use 2D model of the subsoil that properly represents the deformation properties of the whole under-foundation massif by means of surface model. The properties of such model are expressed by what is called interaction parameters marked C. These parameters are assigned directly to structure elements that are in contact with the subsoil and they influence the stiffness matrix.

To simplify the matter, we may imagine that C is the characteristics of elastic, more precisely pseudo-elastic, links, or surface spring constants that change according to the actual state of the analysed system. We may also use the professional slang that calls it "support on C parameters", which is the generalisation of standard Winkler idea of the supporting in the form of thick liquid g = C1 (MNm-3) or in the form of infinitely dense system of vertical springs. The generalisation is very important and deals mainly with the consideration of significant shear distribution in the subsoil that is neglected by Winkler model. The parameters of the interaction between the foundation and the subsoil depends on the distribution and loading level, or the contact stress between the structure surface and the surrounding subsoil, on the geometry of the footing surface and on mechanical properties of the soil.

Calculation module Soil-in takes account of all the mentioned dependencies.

As the C parameters influence the contact stress and vice versa – the distribution of the contact stress have impact on the settlement of the footing surface and thus the C parameters, it is necessary to use an iterative solution.

The influence of subsoil in the vicinity of the structure

The modelling of the interaction between a structure and subsoil requires that the influence of the subsoil outside of the structure be taken into account. This outside-subsoil supports the edges of the foundation slab due to shear stiffness. In the past, special procedures were recommended to model this phenomenon. The current versions of Scia Engineer employ a sophisticated solution whose principle is described in the following paragraph.

The program automatically adds to the edge of the analysed foundation slab springs that approximately substitute the effect of what is termed support elements (1 to 2 metre wide strip located along the edges of the foundation slab, the thickness of this strip is almost zero). The solution obtained through this approach takes into account the effect of the subsoil outside (in the vicinity) of the analysed foundation slab.

In comparison with a solution without such springs, the results obtained with the springs gives smaller deformation of the foundation-slab edges which means larger bending moments in the foundation slab.

The springs oriented in the global z-direction are assigned to all edge nodes except the situation when a node already has another spring assigned or if a rotation of a node is specified. In that case, the program assumes that the user has already defined a special type of support and that it is not wanted to alter that special configuration automatically on the background.

These exceptions can be used to deliberately suppress the implementation of edge-springs along certain lines. The user can define very small line springs along required lines (edges) and thus eliminate the effect of the surrounding subsoil (e.g. if a sheet pile wall is installed).

Soil-in output

The results from the soil-in iteration are the C parameters C_{1z}, C_{2x} and C_{2y}.

Parameters C_{1x} and C_{1y} are always defined by the user.

C1z - resistance of environment against wP (mm) [C1z in MN/m3]

C2x - resistance of environment against wP/xP (mm/m) [C2x in MN/m]

C2y - resistance of environment against wP/yP (mm/m) [C2y in MN/m]

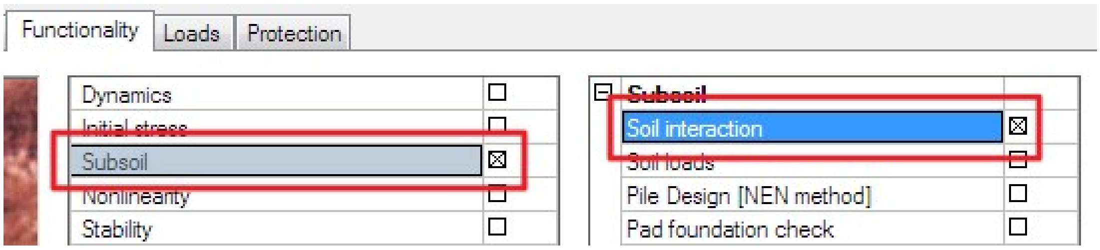
C1x - resistance of environment against uP (mm) [C1x in MN/m3]

C1y - resistance of environment against vP (mm) [C1y in MN/m3]

Usually, C2x is considered equal to C2y and C1x equal to C1y, because the calculation is done by so called isotropic variant of the calculation of C2 parameter.

The soil-in calculation is available when the specific functionality is active.

Check Subsoil on the left part and the Soil interaction on the right part of the functionality tab:



The Soil interaction is available only for Plate XY and General XYZ type of project.

C parameters (explanation from theoretical background of FEM solver)

C_1 - Parameters of the interaction of the foundation with the surface 2D model of the subsoil in physical relation containing components of displacement u, v, w.

Winkler formula for vertical components:

$$\sigma_z = r\,[\mathrm{kPa}] \quad = C_{1z}^{S}\,[\mathrm{MNm}^{-3}]\cdot w\,[\mathrm{mm}]$$

Winkler formula for horizontal shear components:

$$\tau_{zx} = s_x\,[\mathrm{kPa}] \quad = C_{1x}^{S}\,[\mathrm{MNm}^{-3}]\cdot u\,[\mathrm{mm}]$$

$$\tau_{zy} = s_y\,[\mathrm{kPa}] \quad = C_{1y}^{S}\,[\mathrm{MNm}^{-3}]\cdot v\,[\mathrm{mm}]$$

C_2- Parameters of interaction of the foundation with the surface 2D model of the subsoil in physical relations containing the first derivative of settlement.

Paternal formula for shear forces:

$$t_x\,[\mathrm{kNm}^{-1}] = C_{2x}^{S}\,[\mathrm{MNm}^{-1}]\cdot \partial w/\partial x\,[\mathrm{mm/m}]$$

$$t_y\,[\mathrm{kNm}^{-1}] = C_{2y}^{S}\,[\mathrm{MNm}^{-1}]\cdot \partial w/\partial y\,[\mathrm{mm/m}]$$

C_{1z} - foundation compression modulus of the Winkler type, expressing resistance to the vertical displacement of the subsoil surface.

C_{2x}, C_{2y},C_{2xy} - foundation shear modulus expressing resistance to the shear components in the x and y direction of the subsoil surface, generally different in positive and negative shears g_{xy}, g_{yz}(dilatancy and contractancy effects).

Pasternak G_p modulus is our C_2.

Pasternak differential equation:

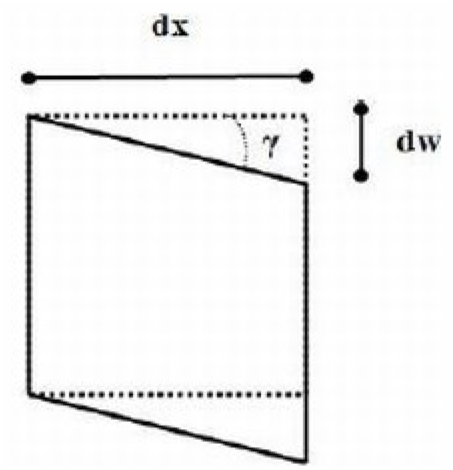
$$p = wk - G_p \frac{d^2w}{dx^2}$$

p - pressure

k - modulus of sub-grade reaction

G_p - shear modulus of the shear layer and it is related to rotation in the differential equation.

Rotation of the surface = bevel dw/dx (see the picture)



w - displacement

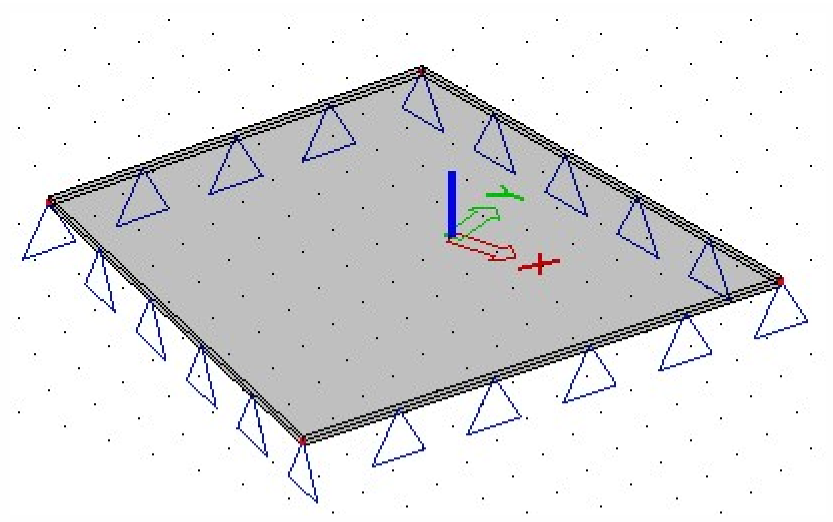
g - shear strain

Support on surface

The interaction between the structure and subsoil is calculated if the structure is put on a support of "Soilin" type.

The procedure to define a new Soilin support

1. Create the structure to be supported.
2. Open service Structure.
3. Start function Support > Surface (el. foundation).
4. Adjust the parameters (see chapter Surface support on slab).
5. Confirm with [OK].
6. Select the slab (groundslab) or slabs that should be supported with this type of support

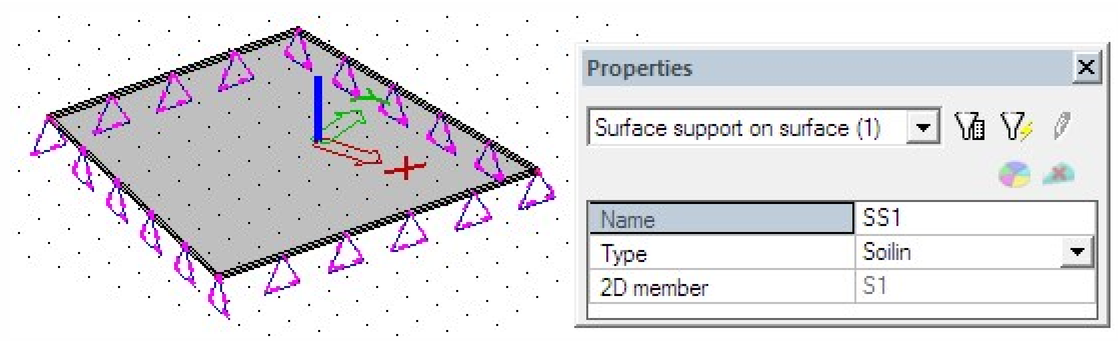


If the ground-slab is not horizontal, one should be aware of the following:

The correct calculation of C parameters assumes that the structure that is in contact with subsoil is more or less horizontal. Technically speaking, the inclination of the footing surface up to 5 to 8 degrees can be allowed. Program is capable of dealing with footing surface in several z-levels, but the results are acceptable only if the z-levels are within certain limits – see the following literature (in Czech):

- Kolář V.: Matematické modelování geomechanických úloh. Skriptum pro postgraduální studium FAST VUT Brno, 1990, 60 str.
- Buček J., Kolář V., Obruča J: Manuál k programu SOILIN, FEM consulting Brno, 1993
- Buček J., Kolář V.: Iterační výpočet NE-XX - SOILIN, FEM consulting Brno, 1995
- Kolář V.: Statické výpočty základových konstrukcí. Knižnice Aktualit České matice technické Praha, ed. plán 1994.
- Kolář V.: Teoretický manuál FEM-Z k programům DEFOR a NE-XX, seminář FEM consulting s.r.o., 5. - 6.10.1993 v Brně.

The surface support properties



Name: Specifies the name of the support.

Type: Defines the type of support – see below.

Subsoil: If necessary for the selected type, this item specifies the subsoil parameters.

Type

Individual:
A particular subsoil type is assigned to the slab.
The subsoil is defined by means of C parameters. These user-defined C parameters are used for the calculation (of e.g contact stress in the footing surface)

Soil-in:
For such a support, the interaction of the structure with the foundation subsoil is carried out by means of SOIL-IN module.
Parameters C1z, C2x, C2y are calculated by SOIL-IN module.

Both:
Both of the above mentioned types are combined on the same slab.
The user defines which C parameters will be user-defined and which ones will be calculated by SOIL-IN module. Parameters can be defined in subsoil properties. Those C parameters that are input in the subsoil-property dialogue as zero, will be calculated by the SOIL-IN module. Nonzero parameters will be taken as they are input.

The additional springs are automatically added on the edges if soilin calculation doesn't recognize additional plates around the support. See chapter Advanced tips.

Subsoil in the 3D model

The subsoil in the 3D window is defined as a soil surface and soil borehole. The geologic profile is defined for each soil borehole. The position and the composition of the geologic profiles provide information about subsoil.

Soil borehole

The borehole is available in the project only when the functionality Soil interaction is checked.

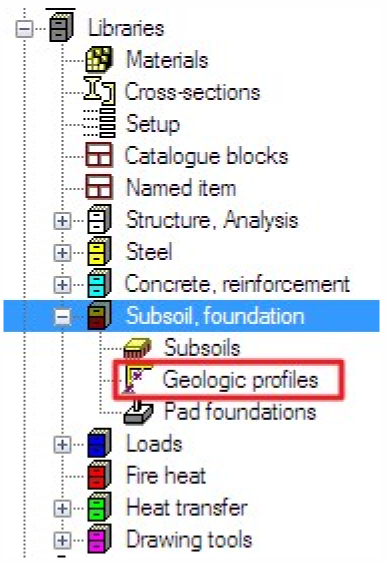
The special property in the inserting dialogue converts standard borehole to the Sand-gravel pile. See more in the separate chapter.

Soil types according to ČSN EN 73 1001

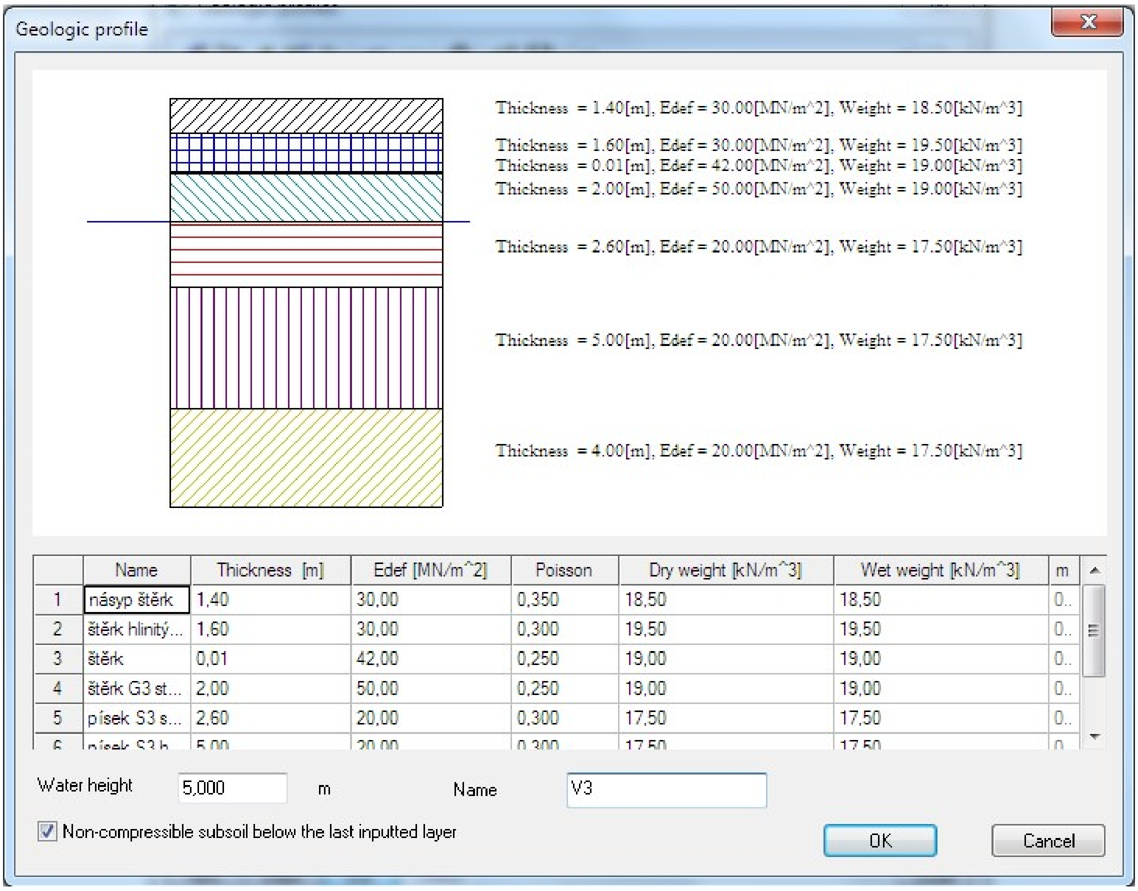
Fine-grained soils		**Sands**		**Gravel**	
F1(MG)	loam, fine-grained	**S1(SW)**	sand, well-grained	**G1(GW)**	gravel well-grained
F2 (CG)	clay, gravelly	**S2(SP)**	sand, poorly-grained	**G2(GP)**	gravel poorly-grained
F3 (MS)	loam, sandy	**S3(S-F)**	sand, with fine-grained soil	**G3(G-F)**	gravel with fine-grained soil
F4 (CS)	clay, sandy	**S4(SM)**	sand, with loam	**G4(GM)**	gravel, with loam
F5 (ML)	loam, small plasticity	**S5(SC)**	sand, with clay	**G5(GS)**	gravel, with clay
F5 (MI)	loam, middle plasticity				
F6 (CL)	clay, small plasticity				
F6 (CI)	clay, middle plasticity				
F7 (MH)	loam, high plasticity				
F7 (MV)	loam, very high plasticity				
F7 (CH)	loam, extremly high plasticity				
F8 (CV)	clay, very high plasticity				
F8 (CE)	clay, extremly high plasticity				

Geologic profile

All profiles are saved to the Geologic profiles library. The geologic profiles can be imported or exported by the DB4 format.



The borehole profile is defined as a simple grid with the preview. Each row represents one layer of soil with the same properties.



Each layer is defined by the soil parameters:

Name:
Specify the name of the layer

Thickness (m):
thickness of the layer

E_{def}:

The module of deformation E_{def} is defined as deformation characteristic of the soil. It is a ratio of the normal stress increment to the increment a linear transformation. For geotechnical categories 1 and 2 the indicative value from e.g. ČSN 73 1001 can be used, for category 3 a survey should be carried out to provide for the value. The value E_m from Eurocode 7 can be used instead of E_{def}.

E_{def} according to ČSN 73 1001:

Class of the subsoil	E_{def} (MPa)
F6-F8 (soft, medium consistency)	1,5-4
F6-F8 (stiff consistency)	6-8
F6-F8 (hard consistency)	10-15
F3-F5 (soft, medium consistency)	3-5
F3-F5 (stiff consistency)	8-10
F3-F4 (hard consistency)	based on survey
F5 (hard consistency)	10-20
F1, F2 (soft, medium consistency)	5-15
F1, F2 (stiff consistency)	12-25
F1, F2 (hard consistency)	based on survey
S4, S5	5-12
S3	12-19
S2	15-35
S1	30-60
G5	40-60
G4	60-80
G3	80-90
G2	100-190
G1	250-390
R6	10-75
R5	20-250
R4	40-750
R3	70-2500
R2	130-7500
R1	250-25000

The E_{def} for R is derived from the number of discontinuous parts in the soil.

Poisson:

Poisson's ratio, coefficient of transverse deformation, an indicative value or experimentally found value can be used, predefined range is 0 – 0.5

Poisson according to ČSN 73 1001:

Class of the subsoil	Poisson v
F8 (soft, medium, stiff consistency)	0,42
F8 (hard consistency)	based on survey
F5-F7 (soft, medium, stiff consistency)	0,40
F5-F7 (hard consistency)	based on survey

Class of the subsoil	Poisson v
F1-F4 (soft, medium, stiff consistency)	0,35
F1-F4 (hard consistency)	based on survey
S5	0,35
S4, S3	0,30
S1, S2	0,28
G4, G5	0,30
G3	0,25
G1, G2	0,20
R6	0,40-0,25
R4, R5	0,30-0,20
R3	0,25-0,15
R1, R2	0,20-0,10

Dry weight:
weight for the dry soil, normally within the range from 18 to 23 kN/m^3, range is 0 – 10000000000 kN/m^3

Wet weight:
weight for the wet (saturated) soil, this value is mostly about 2-3 kN/m^3 higher than the dry weight, range of the values is 10 – 10000000 kN/m^3

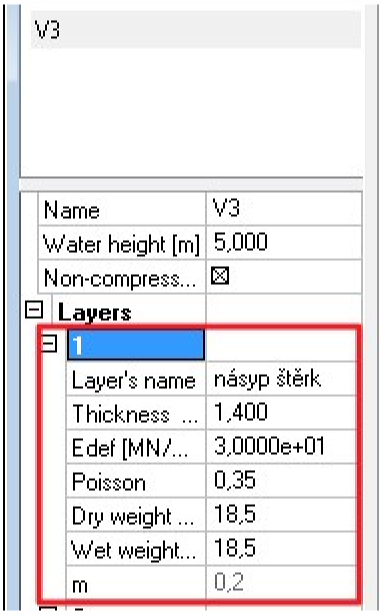
m coefficient:
structural strength coefficient, according to the Eurocode7 is 0,2 (ČSN 73 1001 defines a table). The m coefficient may be modified for the whole Eurocode.

Coefficient m according to ČSN 73 1001:

Class of the subsoil	m
F1-F8 with E_{def}<4MPa, not over consolidated and soft or solid consistency R1, R2 and R4, R5 not affected by erosion	0,1
F1-F8 which don't belong to the first group S1, S2, G1, G2 under the water level R3	0,2
S1, S2, G1, G2 above the water level S3-S5 G3-G5 R4, R5 which don't belong to the first group	0,3
R6	0,4
Loess, loess loam	0,5

Geologic profile must be defined up to such a depth where the bearing pressure is still active, otherwise the program does not have sufficient information.

The defined parameters are displayed in the library as properties.



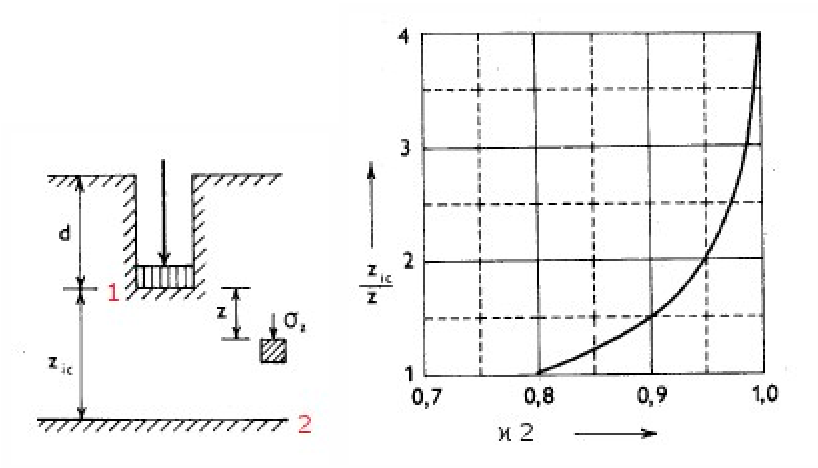
The height of the underground water is defined by the value in the properties. It is a positive value but it represents the depth.

Non-compressible subsoil below the last inputted layer

The checkbox "Non-compressible …" can be used if the soil below the last layer is non-compressible. The system applies coefficient of depth reduction $\varkappa_2$ in this case (calculation of $\varkappa_2$ can be found in ČSN 73 1001, art. 80). This option is recommended when the non-compressible layer is placed in a small depth under the borehole.

Calculation of $\varkappa_2$ according to ČSN 73 1001:

$\varkappa_2$=1-exp((z_{ic}/z) ln0,25 + ln0,8)



1 – foundation base

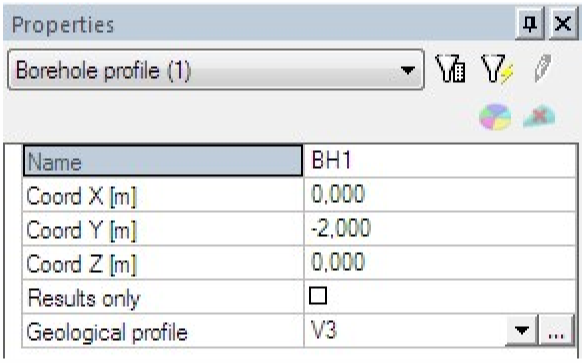
2 – non-compressible layer

z_{ic} – is the depth under the foundation base to the non-compressible layer

z – is the depth from the foundation base to the level where the contact stress σ_z should be calculated

The contact stress σ_z will be calculated by the reduced depth $z_{r2}=\varkappa_2{}^*z$ where z is the depth under the foundation base.

Properties of the borehole profile

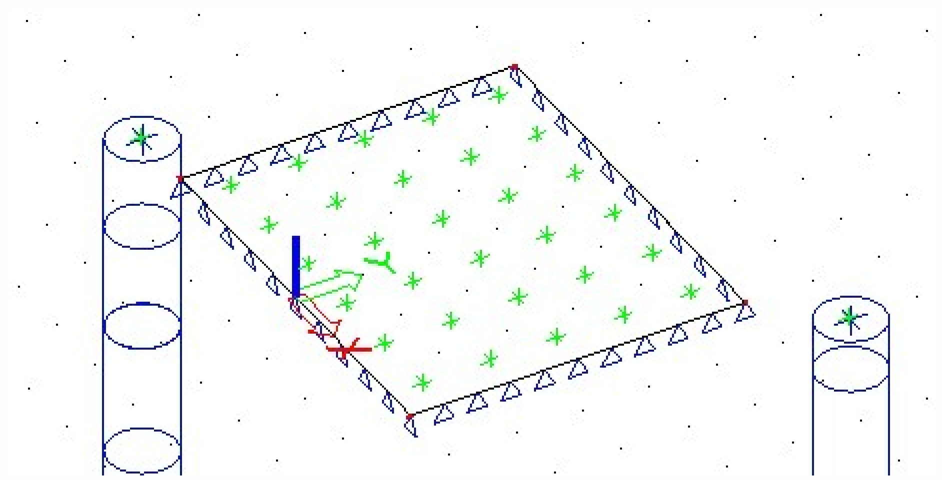


The borehole is defined by the geologic profile and the inserting point in the 3D window. The properties contain only name, its coordinates, the borehole profile and the checkbox Results only.

Settlement input data

Settlement is calculated for each mesh element (in its center of gravity) and for each borehole inserting point. The checkbox Results only exclude a borehole inserting point from the input data. It means that the point is used for the calculation of settlement but the geologic profile is not taken into account for the layers approximation.

The nodes for the settlement calculation (green vertexes):

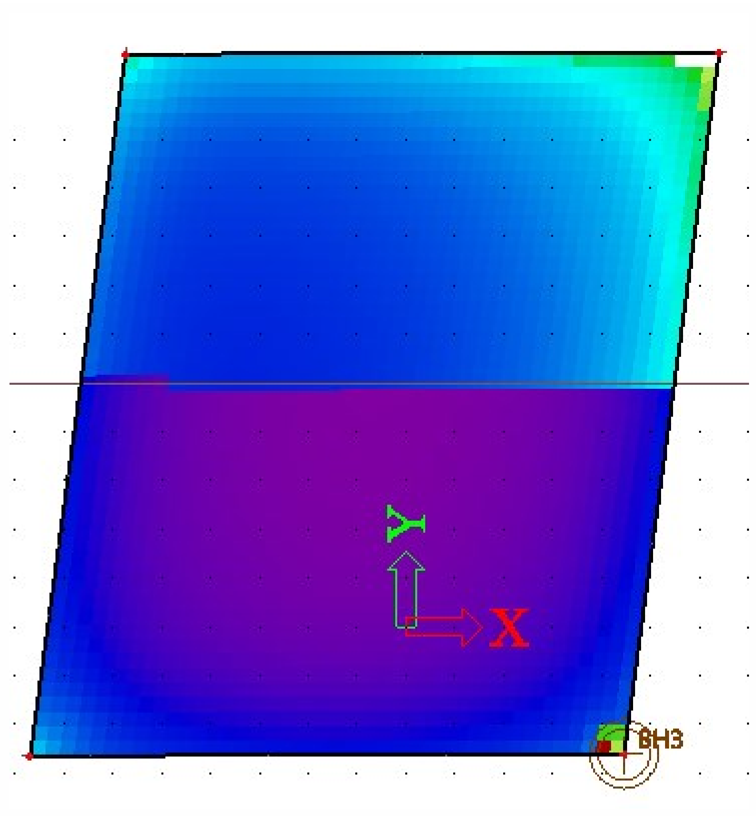


Geological areas

The main geological surface = area is calculated and displayed automatically. It is possible to create more areas in the main one.

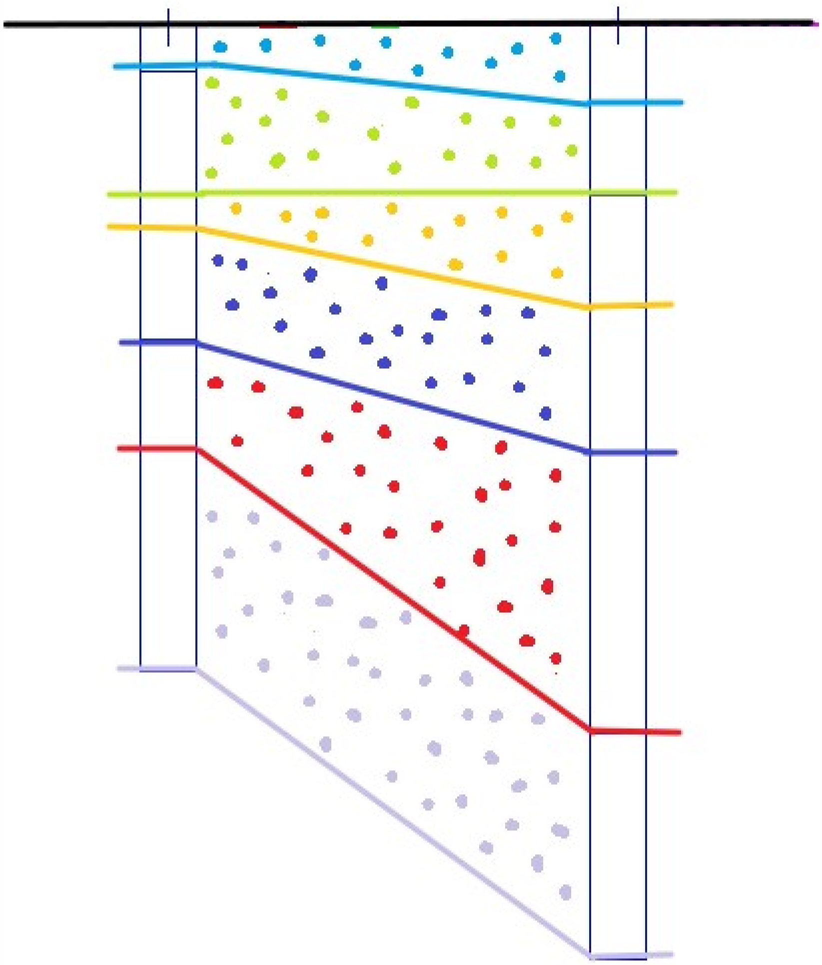
Layers approximation is calculated inside one geological area, independently on neighbouring areas. There is a geological fault on the border between two geological areas.

The Sand-gravel pile is a geological profile which is placed in one geological area.



Layers approximation

When more borehole profiles are used in the project then it must fulfil one important condition – the same number of layers. This is required because of the soil-in approximation.

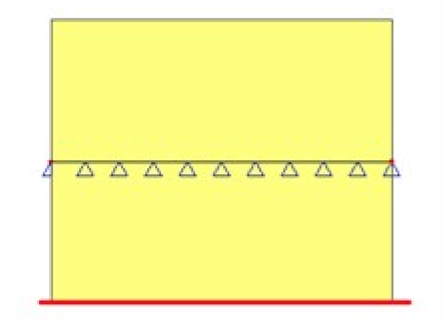


If there is some layer missing in one borehole, then it can be substituted by layer with minimum thickness – e.g. 1mm.o the soil-in has appropriate number of layers for approximation.

Foundation base

The level of the foundation base is considered on the bottom surface of the plate. The eccentricities are also taken into account.

Even the extrem example as this one place the foundation base to the bottom surface.



The red line indicates the foundation base.

Soil surface

Soil surface is a tool for initial approximation of subsoil surface and layers between boreholes.

Surface is calculated automatically according to inserted structure and inserted boreholes.

If it is deleted then it is automatically regenerated before the calculation starts.

Surface has its borders at least 10m outside the structure.

The surface is editable by 2 action buttons:

- Refresh outline: it recalculates the border
- Refresh surface: it recalculate the mesh of the surface

Actions	
Refresh outline	>>>
Refresh surface	>>>

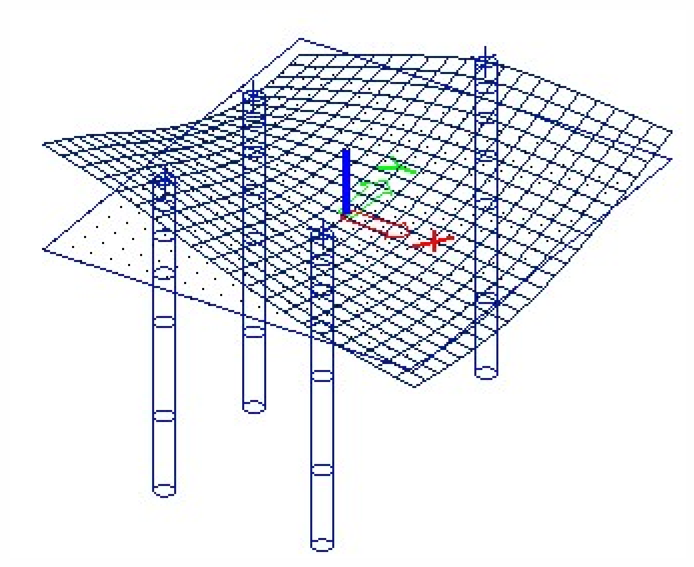
The properties of the surface are simple. Just a name and sizes:

Properties

Soil surface (1)

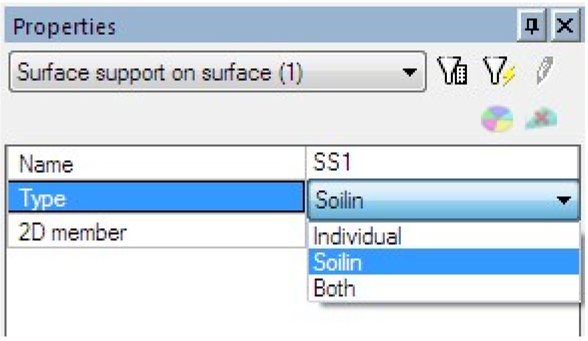
Name	SS1
Max x [m]	10,000
Min x [m]	-10,000
Max y [m]	10,000
Min y [m]	-10,000

It is possible to display deformed subsoil surface. It is created by several boreholes with different Z coordinates. The mesh is used only for displaying of terrain, it is not used for the calculation.



Surface support

The surface support is basic structure object for soil-in. The support type is defined by combobox with 3 items.



Individual:
the C parameters are defined by user in the Subsoil manually (all of them). They are used for calculation. (e.g. contact stresses of the foundation surface)

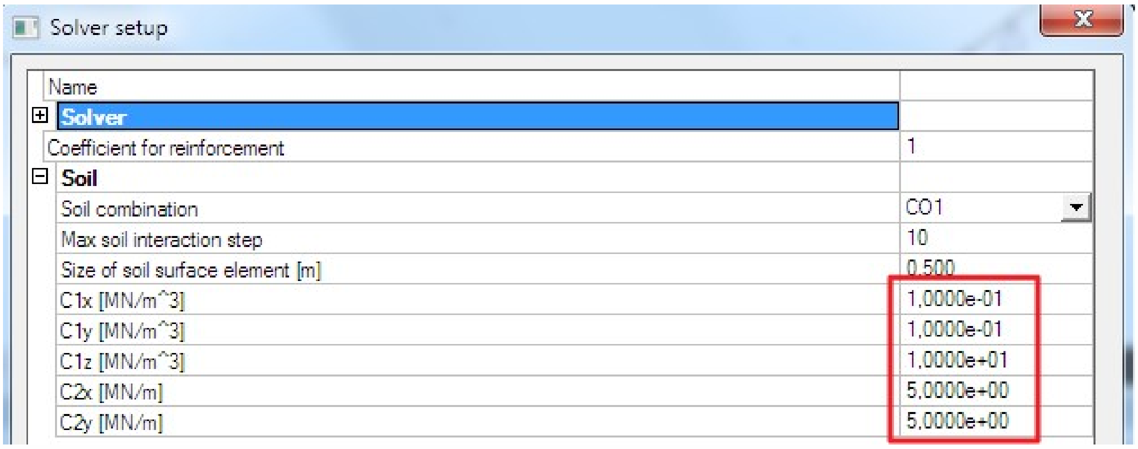
Soilin:
system calculates C parameters (C_{1z}, C_{2x}, C_{2y}) – this type is required for the complete soil-in calculation, C_{1x} and C_{1y} are taken from the solver setup

Both:
system calculates C_{1z}, C_{2x} and C_{2y} if they are set to zero in the Subsoil; the rest is taken from the Subsoil. This item is used rarely only for a very special cases.

Soil-in type

The only type which doesn't use the data from the Subsoil library. All initial values of C parameters are defined by the Solver setup. C_{1x} and C_{1y} are taken from this setup as the results and the rest is calculated by the Soil-in.

The initial values could influence a bit the calculation convergence but their major importance is for setting of non-compressible stiffnesses. These values are 100 times higher than the initial values. That's why a reduction of initial values (e.g. 10 times) can help in a convergence problems (higher depth, small loading, etc.)



Individual type

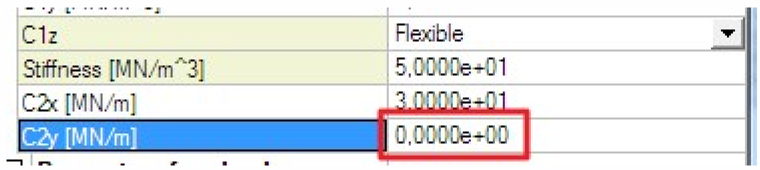
C_{1z}, C_{2x}, C_{2y} parameters are taken from the Subsoil library. It is predefined by the user. The calculation of Soil-in won't start in this case.

Both type

Soil-in calculates C_{1z}, C_{2x} and C_{2y} only when they are set to the zero value by the user.

The parameters with any other value are taken from the library.

Example with type Both:



In this case the C_{2y} parameter is calculated by soil-in. This item could be used only in a case when the soil-in would calculate any extreme values of C_2 parameters. It is a very sporadic case.

The type Both is not too common and it was introduced mainly for two reasons:

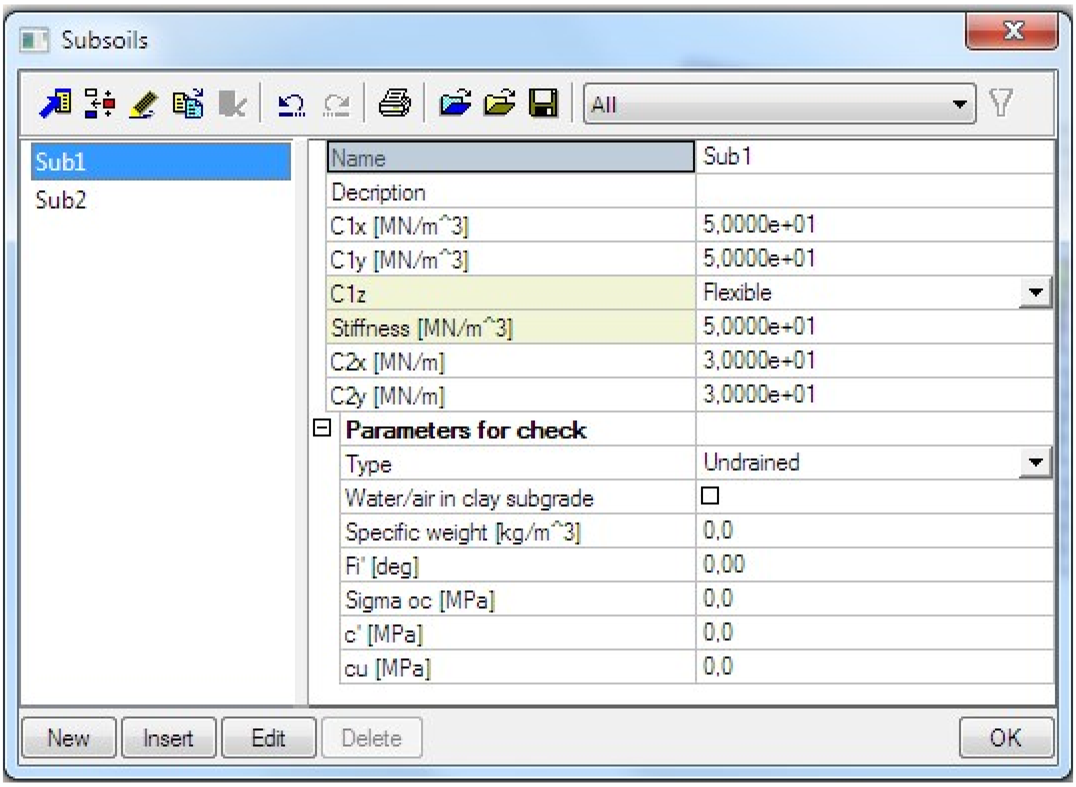
First, I use type Soil-in but I want to have different friction in different parts of the structure. Therefore, the solver setup dialogue is not enough for me, because is just one value can be adjusted there for the friction. Therefore, I can use type Both and thus I am able to define several subsoils with non-zero constants C_{1x} and C_{1y} with all other parameters adjusted to zero. When the Soil-in module runs, the non-zero constants C_{1x} and C_{1y} are of higher priority than those determined by the solver and are applied. Other "zero" values indicate that the values determined by the solver are applied.

Second, sometimes it may be necessary to "suppress" higher values of shear (C_{2x}, C_{2y}) calculated by Soil-in module. This may happen e.g. when a new plate is modelled on an old one and the old plate is defined as the first layer of the subsoil. It is a correct and proper solution, but as E modules of soil and concrete are dramatically different, the Soil-in module calculates high C_2 parameters. Consequently, the stiffness of the foundation slab in the model is bigger than if the two slabs were "joined" together and input as a homogenous monolith. Therefore, C_2 parameters may be reduced artificially. This can be achieved in type Both. I define the subsoil with zero C_{1z} (it will be determined by the Soil-in module) and other non-zero parameters (C_2 and friction). Thus the Soil-in module will provide only for C_{1z} parameter.

Subsoil library

The subsoil contains parameters which can be defined by the user or calculated by soil-in.

Parameters C_{1x} and C_{1y} are always defined by the user.

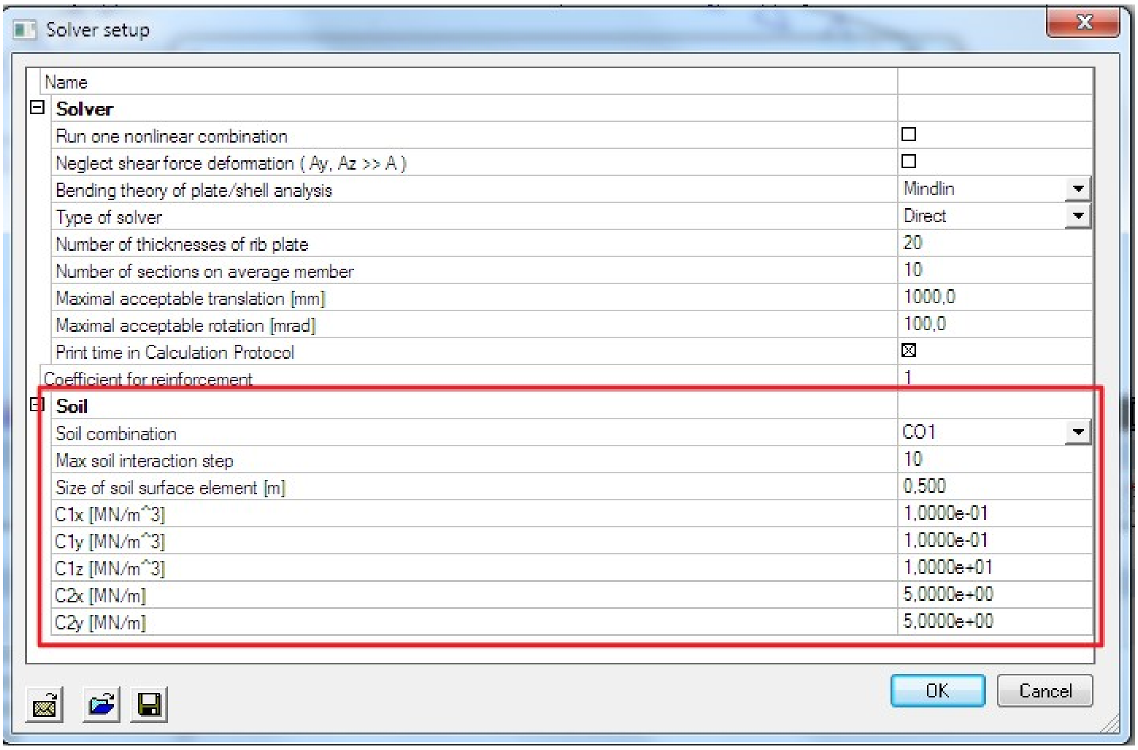


Required parameters for Soil-in calculation

What all must be defined:

- Project with at least one borehole with predefined geologic profile
- Structure with surface support type Soilin or Both
- Load
- Combination type Linear (ULS or SLS)

Soil-in settings in the Solver setup



Soil combination:
linear combination which is used for the soil-in calculation.
Even though it is not an exact solution, for practical reasons the C parameters are not calculated separately for each load case or each load case combination. The user must specify one particular reference combination that is used to calculate the C parameters. The calculated C parameters are then applied in all remaining defined load cases and combinations.

Max soil interaction step:
number of iteration cycles (when the program stops iterations if there are still no proper C parameter calculated, in case that results diverge), the max. limit is 99 steps

Size of soil surface element:
define size of element of surface mesh. It is used for displaying of terrain.

C_{1x}:
the parameter defined by the user

C_{1y}:
the parameter defined by the user

C_{1z}:
initial value for soil-in (if the support type is Soilin)

C_{2x}:
initial value for soil-in (if the support type is Soilin)

C_{2y}:
initial value for soil-in (if the support type is Soilin)

The source of not calculated parameters depends on the support type. It is described in the previous chapter.

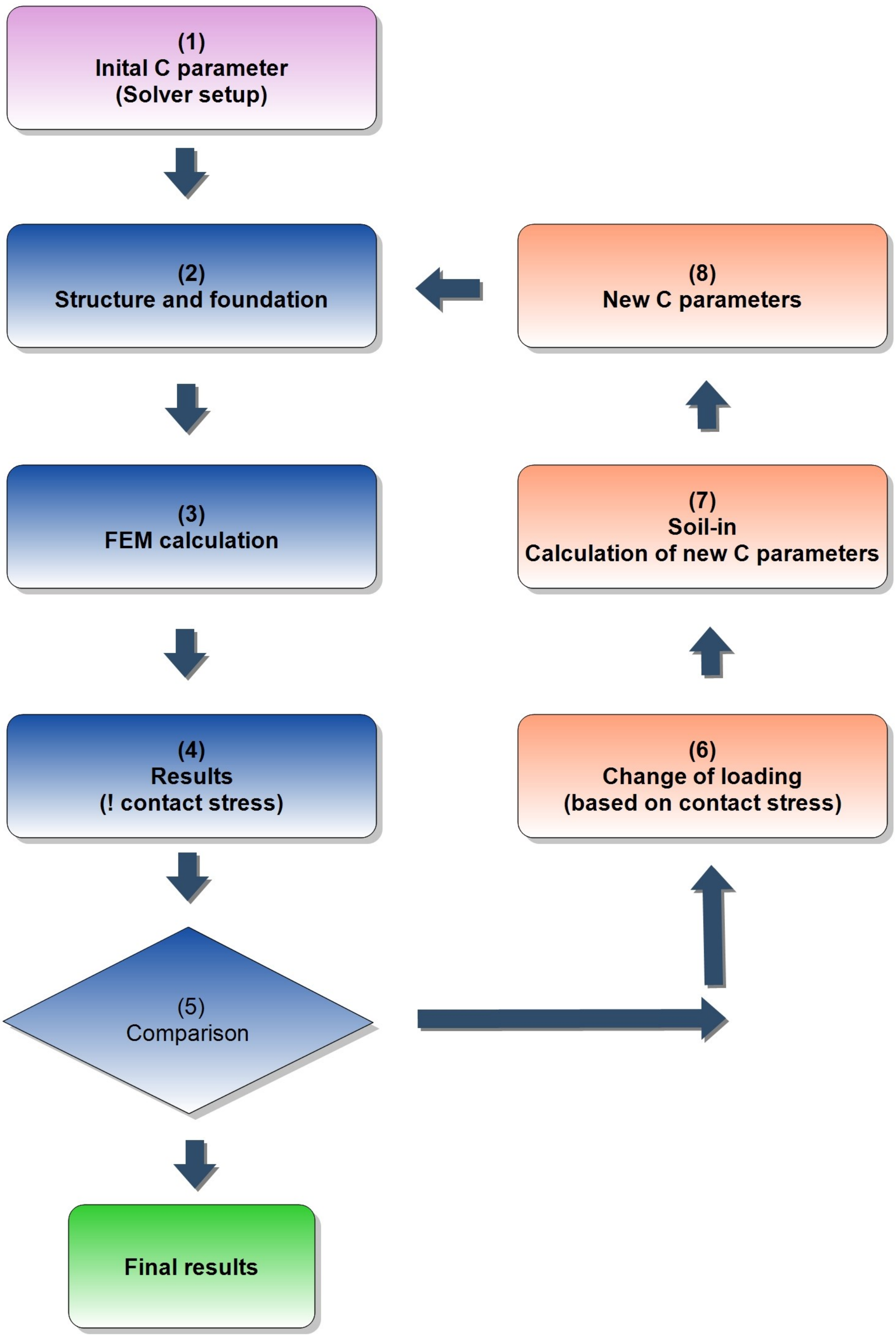
Soil-in calculation

Soil-in iterative cycle

The values from the top structure and the foundation are calculated by FEM. The values are used as the source data for the soil-in.

The iterative process is finished when the contact stress σ_z and displacement u_z does not change significantly in the two subsequent iterations. The special quadratic norms are evaluated in the each iteration cycle to find out if this condition is fulfilled.

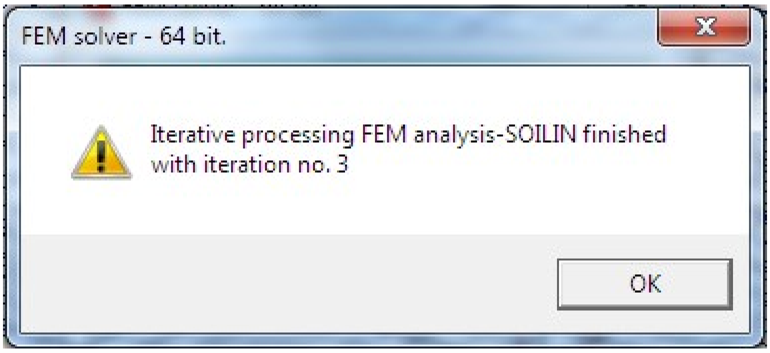
Diagram of the iterative cycle:



1. The values are taken from the solver setup, predefined by the user.
2. Data from the structure and its foundation.
3. FEM calculation – important results for soil-in contact stress σ_z and displacement u_z.
4. The results of i iteration.

5. Comparison of the in contact stress σ_z and u_z – it is based on the quadratic norms, when it does not change significantly, then the calculation is done and Scia Engineer displays results.
6. 1st step of soil-in – the contact stress is recalculated to the new loading.
7. 2nd step of soil-in – the C parameters are recalculated, new loading is taken from the previous step.
8. 3rd step of soil-in – final C parameters from soil-in - the new input data.
9. New C parameters are used for the next FEM calculation.

There is a message when the last iteration is done.



Quadratic norm to compare the results from the last and the previous iteration

The calculation of the settlement of the subsoil and subsequent determination of the C parameters is performed in a standard way using an iterative process. The result of this process is the state in which the contact stress or displacement u_z in two subsequent iterations does not change significantly. For that reason, the following quadratic norms are evaluated in every j-th iteration:

$$\varepsilon_\sigma = \frac{\sum_{i=1}^{n} \left(\sigma_{z,i,j} - \sigma_{z,i,j-1}\right)^2 A_i}{\sum_{i=1}^{n} \left|\sigma_{z,i,j} \cdot \sigma_{z,i,j-1}\right| A_i}$$

$$\varepsilon_u = \frac{\sum_{i=1}^{n} \left(u_{z,i,j} - u_{z,i,j-1}\right)^2 A_i}{\sum_{i=1}^{n} \left|u_{z,i,j} \cdot u_{z,i,j-1}\right| A_i}$$

Where:

n number of nodes

$\sigma_{z,i}$ contact stress in node i

A_i area corresponding to node i

$u_{z,i}$ global displacement of node i in the z-direction

The iterative calculation is stopped if ε_σ<0,001 or ε_u<0,001

Theory about the derivation process

In this text we limit ourselves to a brief derivation for the purpose of the explanation that will follow:

1. The formula for the potential energy of internal forces of the 3D model has the following form:

$$\Pi_{3D}^{i} = \frac{1}{2}\int_{V} \underline{\boldsymbol{\sigma}}^{T}\,\underline{\boldsymbol{\varepsilon}}\,dV = \frac{1}{2}\int_{V} \underline{\boldsymbol{\varepsilon}}^{T}\,\underline{\mathbf{D}}\underline{\boldsymbol{\varepsilon}}\,dV$$

2. Neglecting the effect of horizontal components of deformation, we get the following vectors:

$$\boldsymbol{\sigma} = \left[\sigma_z, \tau_{zx}, \tau_{yz}\right]^{T} = \mathbf{D}\boldsymbol{\varepsilon}$$

$$\boldsymbol{\varepsilon} = \left[\varepsilon_z, \gamma_{zx}, \gamma_{yz}\right]^{T} = \left[\frac{\partial w}{\partial z}, \frac{\partial w}{\partial x}, \frac{\partial w}{\partial y}\right]^{T}$$

3. This means the corresponding simplification of the matrix of physical constants D.

$$\mathbf{D} = \begin{bmatrix} E_z & 0 & 0 \\ 0 & G & 0 \\ 0 & 0 & G \end{bmatrix}$$

4. In order to be able to reduce the problem from 3D to 2D, it is necessary to integrate formula 1) over the z-axis. For this reason, a certain “damping function” f_z is introduced and it is defined by the ratio of the settlement in the given depth to the settlement of the surface $w_0(x,y)$.

$$f(z) = \frac{w(x,y,z)}{w_0(x,y)}$$

5. Modifying formulas from step 2) we get:

$$\boldsymbol{\varepsilon} = \left[w_0(x,y)\frac{\partial f(z)}{\partial z}, \frac{\partial w_0(x,y)}{\partial x} f(z), \frac{\partial w_0(x,y)}{\partial y} f(z)\right]^{T}$$

6. Substituting formula from step 5) into the formula for the potential energy of body V=ΩH, where Ω is the extent of the 2D model and H is the depth of the deformed zone of the 3D model, we obtain the following formula:

$$\Pi_{2D}^{i} = \Pi_{3D}^{i} = \frac{1}{2}\int_V \left[\sigma_z\varepsilon_z + \tau_{zx}\gamma_{zx} + \tau_{yz}\gamma_{yz}\right] dV =$$

$$= \frac{1}{2}\int_V \left[\varepsilon_z^2 E_z + (\gamma_{zx}^2 + \gamma_{yz}^2)G\right] dV =$$

$$= \frac{1}{2}\int_\Omega \left[w_0^2 \int_0^H E_z \left(\frac{\partial f}{\partial z}\right)^2 dz + \left(\frac{\partial w_0}{\partial x}\right)^2 \int_0^H f^2 G dz + \left(\frac{\partial w_0}{\partial y}\right)^2 \int_0^H f^2 G dz \right] d\Omega$$

7. Integrating over z , we get the formula for the potential energy of internal forces of the 2D model with two parameters C_1^S and C_2^S:

$$\Pi_{2D}^{i} = \frac{1}{2}\iint_\Omega \left[C_{1z}^S w_0^2(x,y) + C_{2x}^S \left(\frac{\partial w_0(x,y)}{\partial x}\right)^2 + C_{2y}^S \left(\frac{\partial w_0(x,y)}{\partial y}\right)^2 \right] d\Omega$$

8. Comparing formulas from step 6) and 7), we can define the relation between the parameters of the general (3D) and surface (2D) model:

$$C_{1z}^S = \int_0^H E_z \left(\frac{\partial f(z)}{\partial z}\right)^2 dz \qquad C_{2x}^S = C_{2y}^S = \int_0^H G f^2(z) dz$$

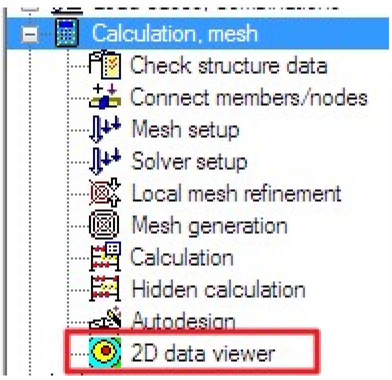
Conclusion:

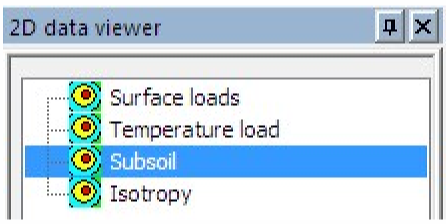
It is also possible to eliminate the automatic calculation of some C parameters and define them manually. This can be achieved by special adjustment of the subsoil parameters and set the type to Both (!).

The results of soil-in

2D data viewer

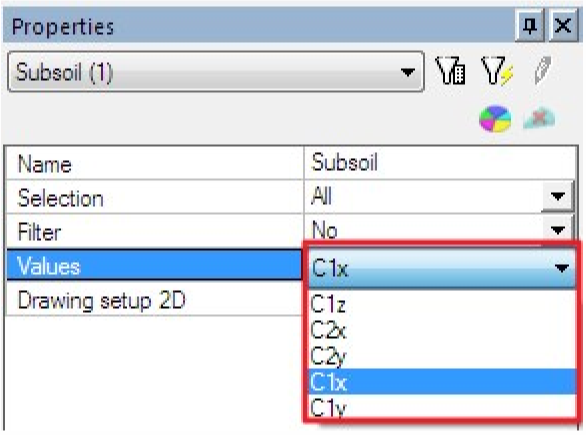
The soil-in results are available in two different services. In the “Calculation, mesh” service is 2D data viewer. There are results for Subsoil.



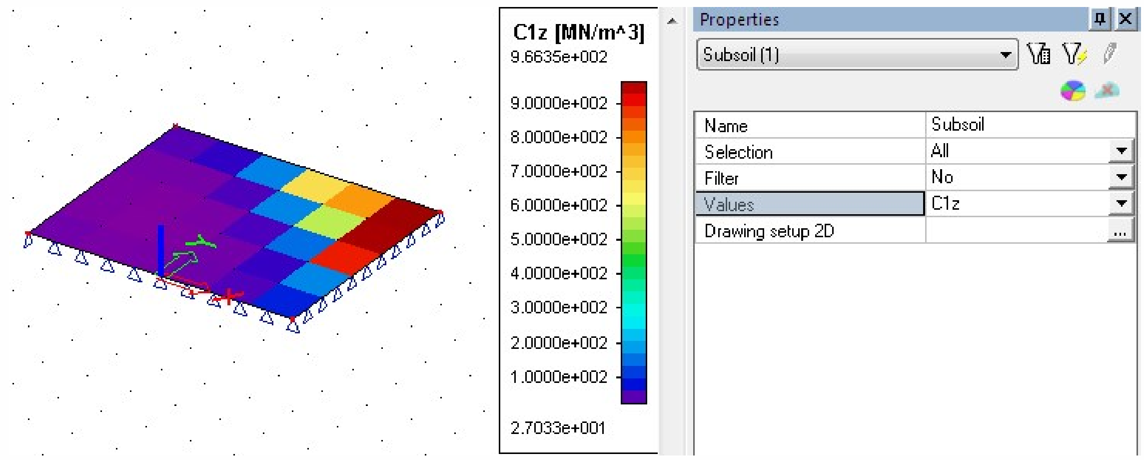


The C parameters are calculated for the mesh on the 2D member. It is displayed by the colour planes.

The results can be displayed for each of C parameters.



The example of calculated C1z:

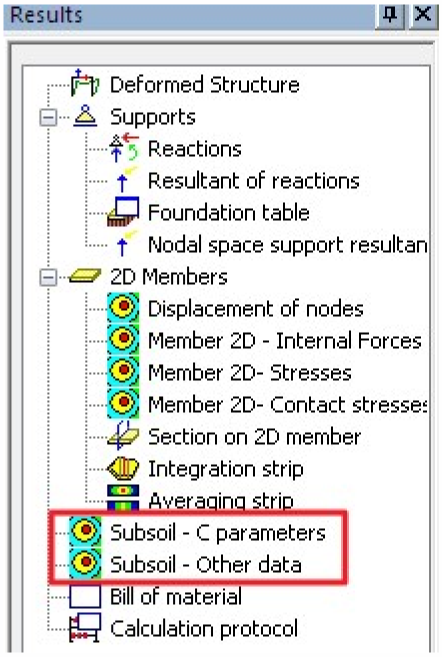


The preview with C parameters in the table can be also displayed in the 2D data viewer.

Results

The service results contain two result previews:

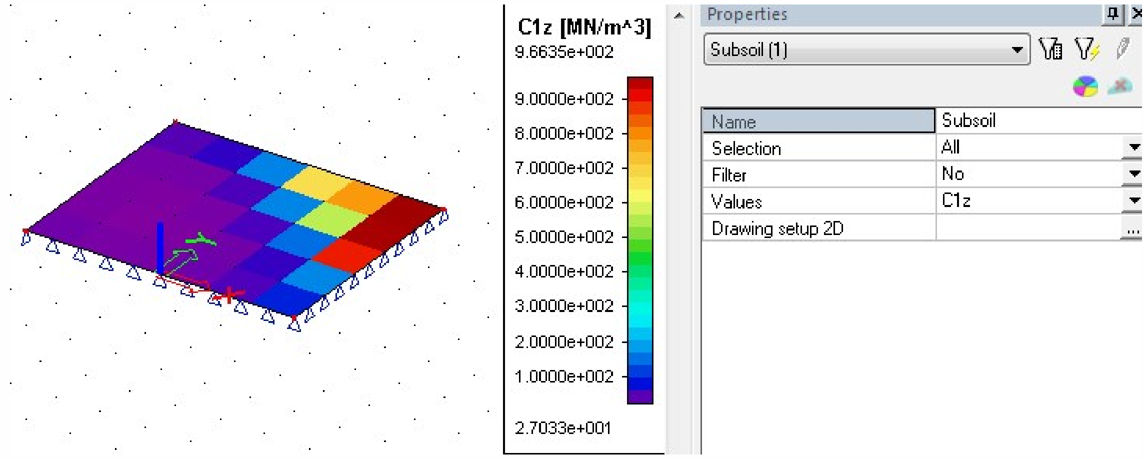
- Subsoil – C parameters
- Subsoil – Other data – this displays settlement (table and diagram for each node)

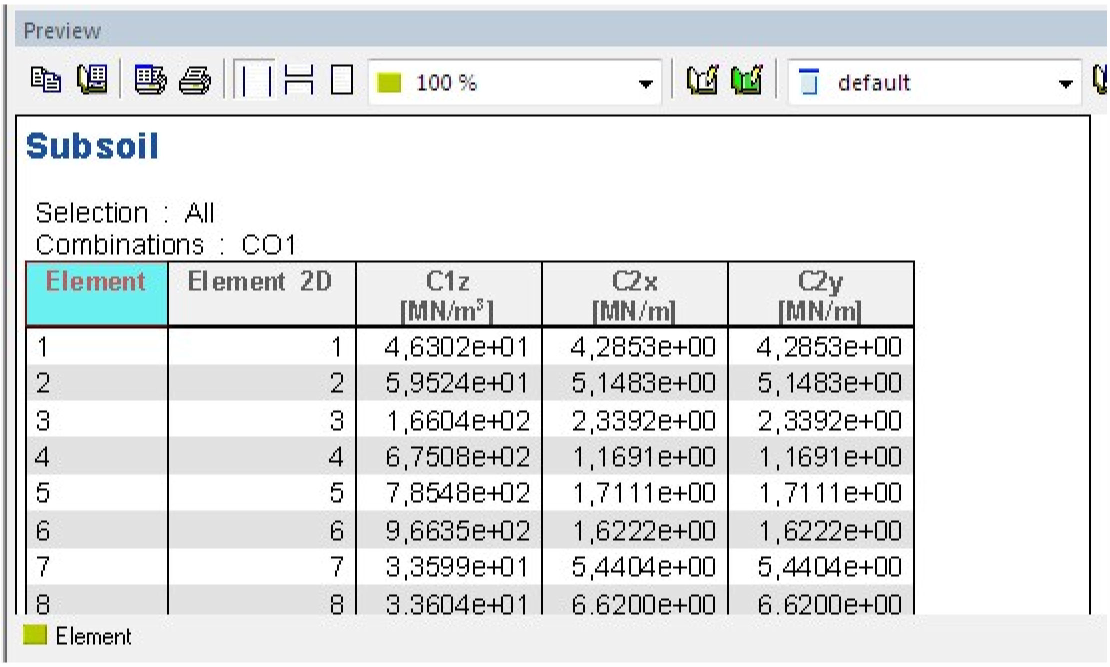


C parameter results

When the Soilin type of the support is used then the preview Subsoil – C parameters displays the same results as 2D data viewer.

When the Both type of the support is used then the preview Subsoil – C parameters displays results of the soilin calculation and the 2D data viewer display data from the Subsoil library.




Preview

100 %

default

Subsoil

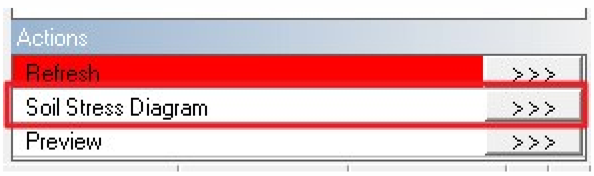
Selection : All
Combinations : CO1

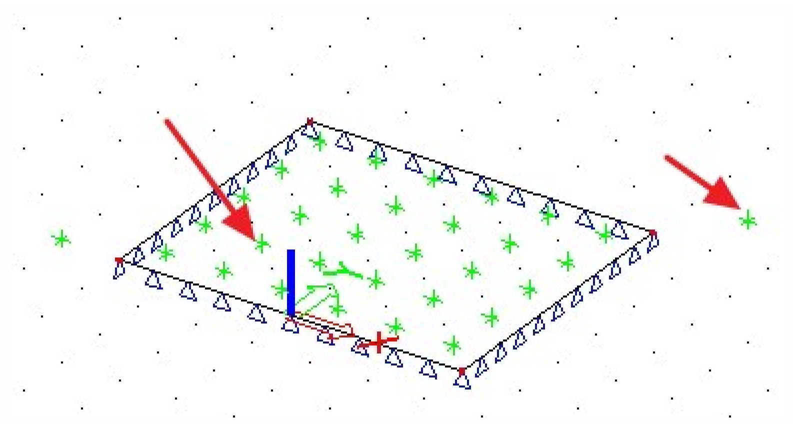
Element	Element 2D	C1z [MN/m^3]	C2x [MN/m]	C2y [MN/m]
1	1	4,6302e+01	4,2853e+00	4,2853e+00
2	2	5,9524e+01	5,1483e+00	5,1483e+00
3	3	1,6604e+02	2,3392e+00	2,3392e+00
4	4	6,7508e+02	1,1691e+00	1,1691e+00
5	5	7,8548e+02	1,7111e+00	1,7111e+00
6	6	9,6635e+02	1,6222e+00	1,6222e+00
7	7	3,3599e+01	5,4404e+00	5,4404e+00
8	8	3,3604e+01	6,6200e+00	6,6200e+00

Element

Soil stress diagram

The “Subsoil – Other data” allow to display Soil structure strength diagram for calculated points. The points are displayed by the action button “Soil stress diagram”.



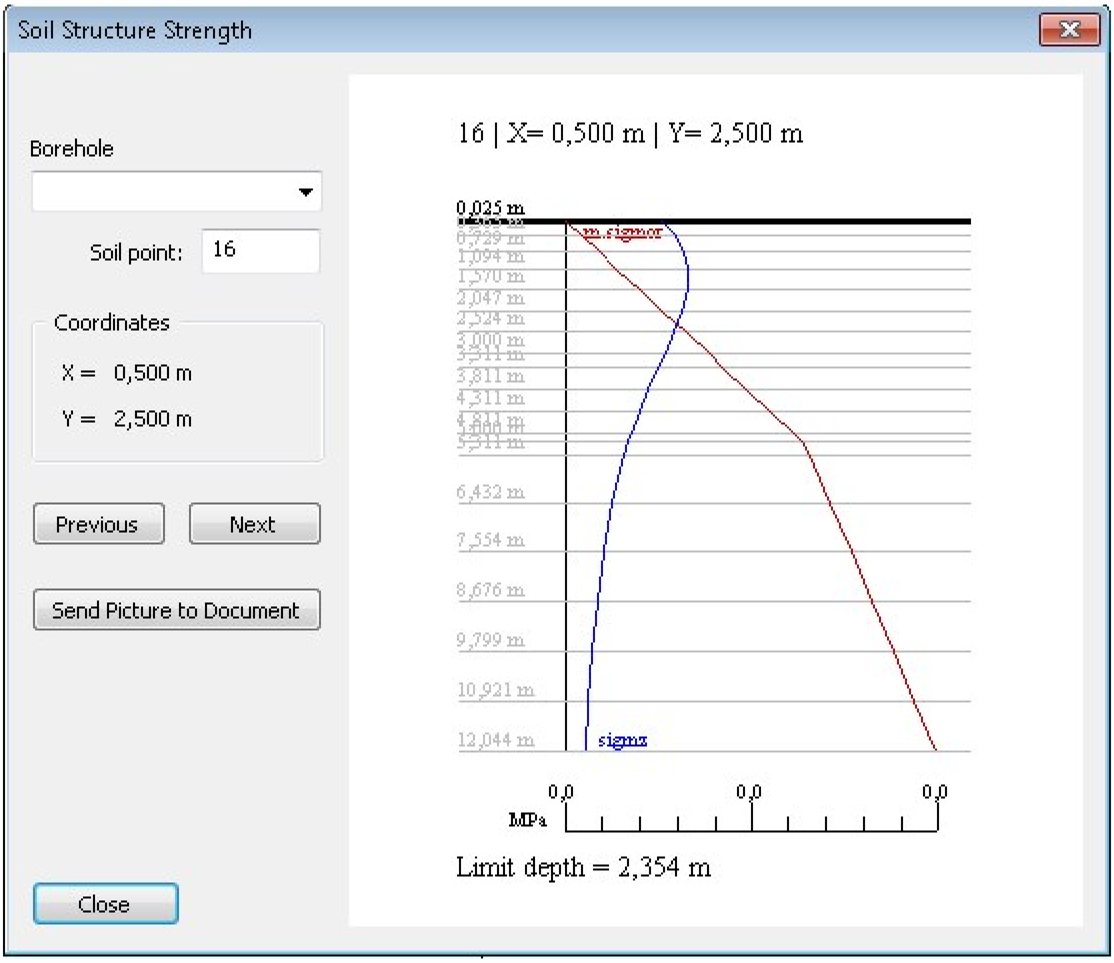


Green vertexes on the plate are centres of elements from 2D mesh. Two green vertexes outside the plate are inserting points from boreholes.

Points are displayed as a green vertex. The vertical axial components of stress and the structure strength (consequently the depth of the deformed subsoil zone) can be displayed for all points from the 2D mesh and for the inserting points of the bore-holes. User just selects the point and the diagram is displayed.

If the borehole is defined as “Results only”, then the point is available for displaying the diagram.

Example of the dialogue Soil Structure Strength:



Previous:
Display the Soil Structure Strength for the previous node

Next:
Display the Soil Structure Strength for the next node

Borehole:
Display the Soil Structure Strength for the selected borehole inserting point

Soil point:
Node number

m*Sigma,or:
The original soil stress

Sigma,z:
The overstress

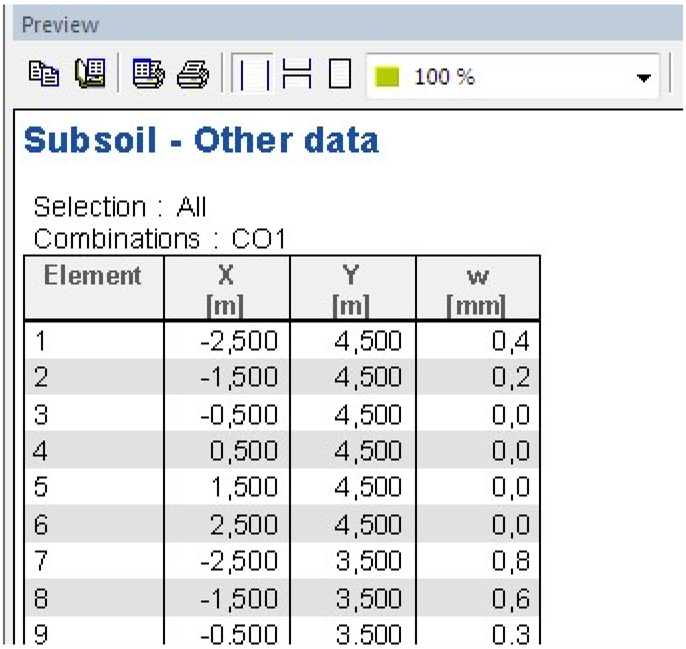
See more about Soil Structure Strength diagram here.

There are two lines - m*Sigma,or and Sigma,z. According to theory, settlement will occur if Sigma,z > m * Sigma,or.

Settlement table

The table is displayed in the Subsoil – other data results. The preview table contain values w for each node.

The settlement w is different from displacement u_z of the foundation plate because w is calculated without stiffness of structure and from the penultimate iteration. Therefore it is useful to watch values w only outside the foundation.


Preview

100 %

Subsoil - Other data

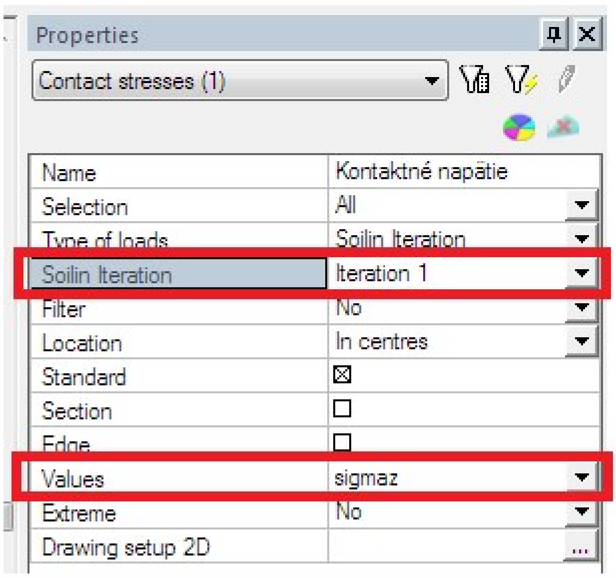
Selection : All
Combinations : CO1

Element	X [m]	Y [m]	w [mm]
1	-2,500	4,500	0,4
2	-1,500	4,500	0,2
3	-0,500	4,500	0,0
4	0,500	4,500	0,0
5	1,500	4,500	0,0
6	2,500	4,500	0,0
7	-2,500	3,500	0,8
8	-1,500	3,500	0,6
9	-0.500	3.500	0.3

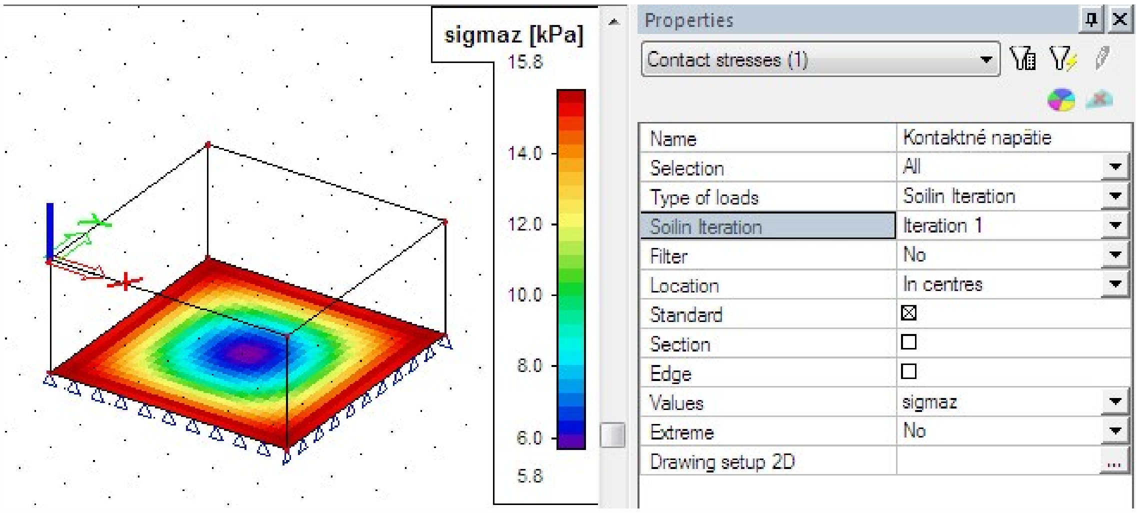
Results for each iteration cycle

When the soil-in won't finish its iteration process in a standard way, the calculation ends after the predefined number of cycles (the solver setup). User can display the contact stresses on the plate for each cycle separately so he is able to find the problem.

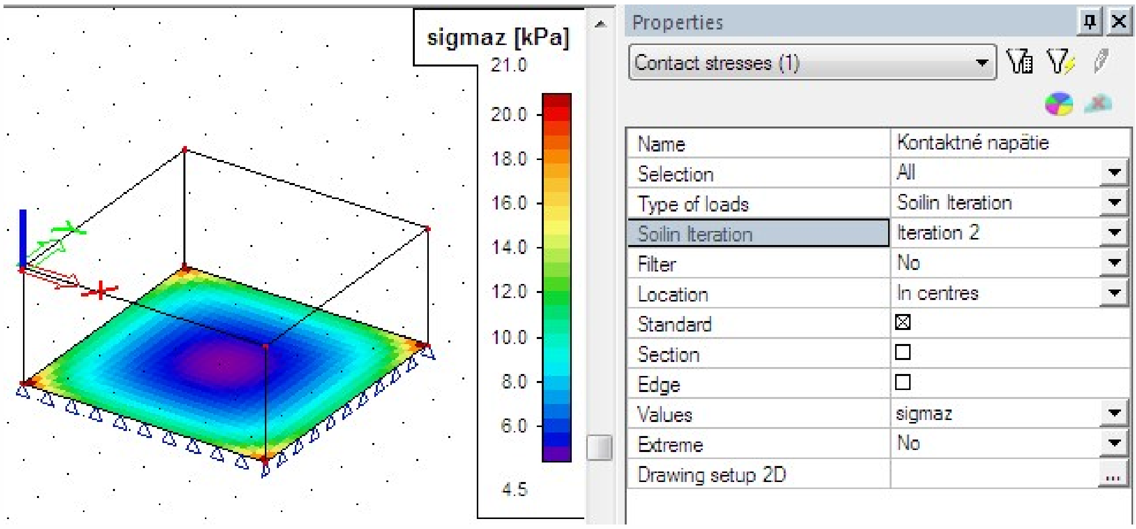
The calculated contact stresses for each iteration cycle can be found in the results.



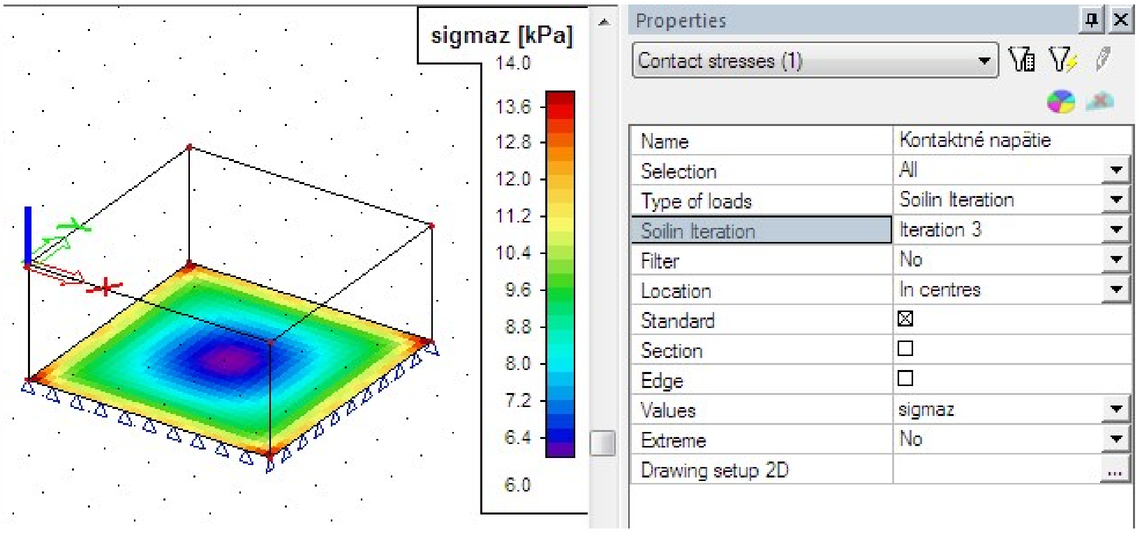
The first iteration cycle



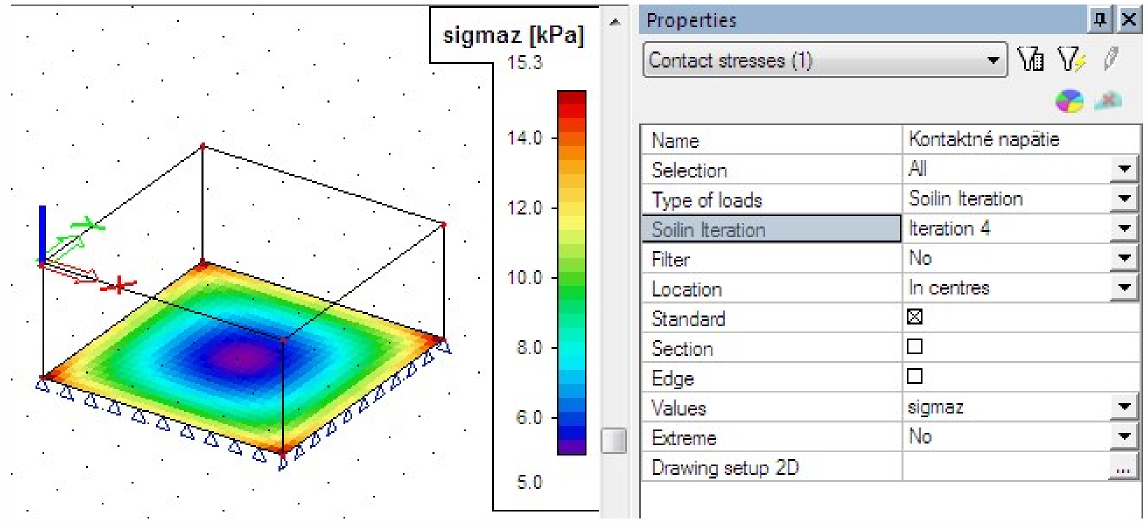
The second iteration cycle



The third iteration cycle



The fourth iteration cycle

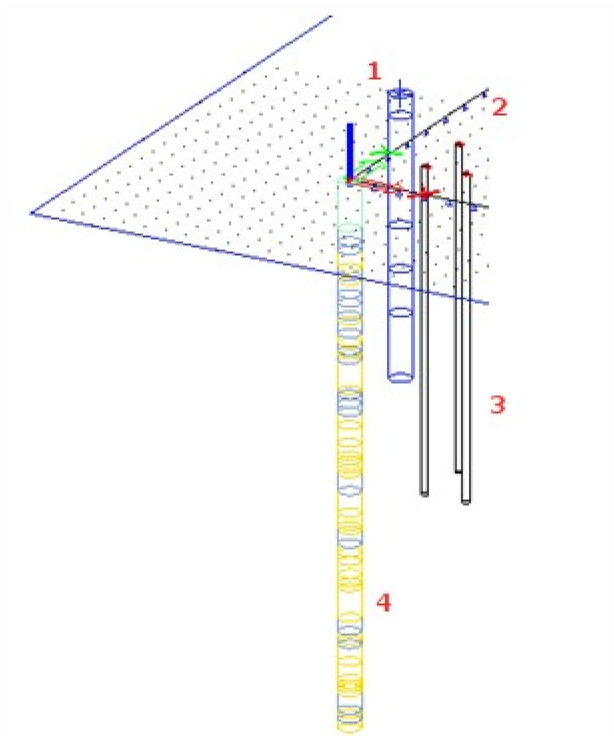


The additional springs are automatically added on the edges if soilin calculation doesn't recognize additional plates around the support. See chapter Advanced tips.

Soil-in and Pile design

Soil-in is a tool for calculation stiffness of the subsoil half-space. The pile is a type of support. Soil- in and piles can be used in one project and system will calculate it together.

Soil-in and Piles are using two different types of boreholes. Piles are based on the CPT profiles; soil-in boreholes are user-defined by layers. Both types of boreholes must be inserted in the project if the user wants to calculate soil-in and pile design.

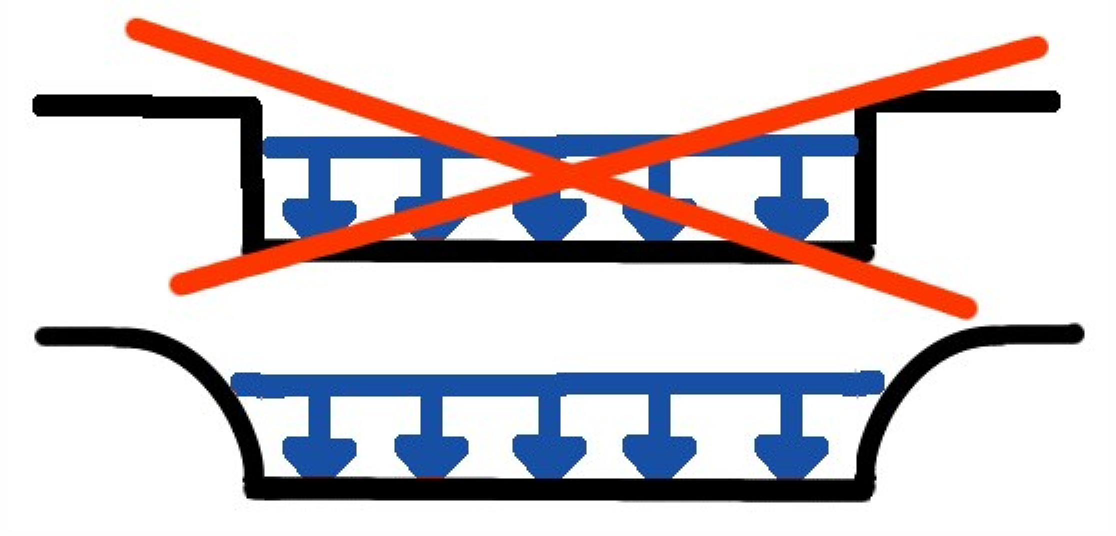


1. Borehole profile for soil-in
2. Surface support for soil-in
3. Piles for Pile design
4. CPT profile for Pile design

Advanced tips

The effect of the subsoil outside the structure

The nearest subsoil around the loaded structure is also affected by its settlement. The better realistic picture how it works in the reality is displayed below.

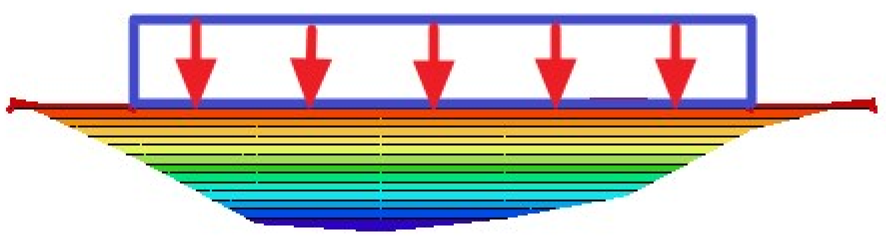


Calculation of the nearest surrounding of the structure is a specific use case. It is recommended to add one more plate to the structure for this purpose – additional subsoil element.

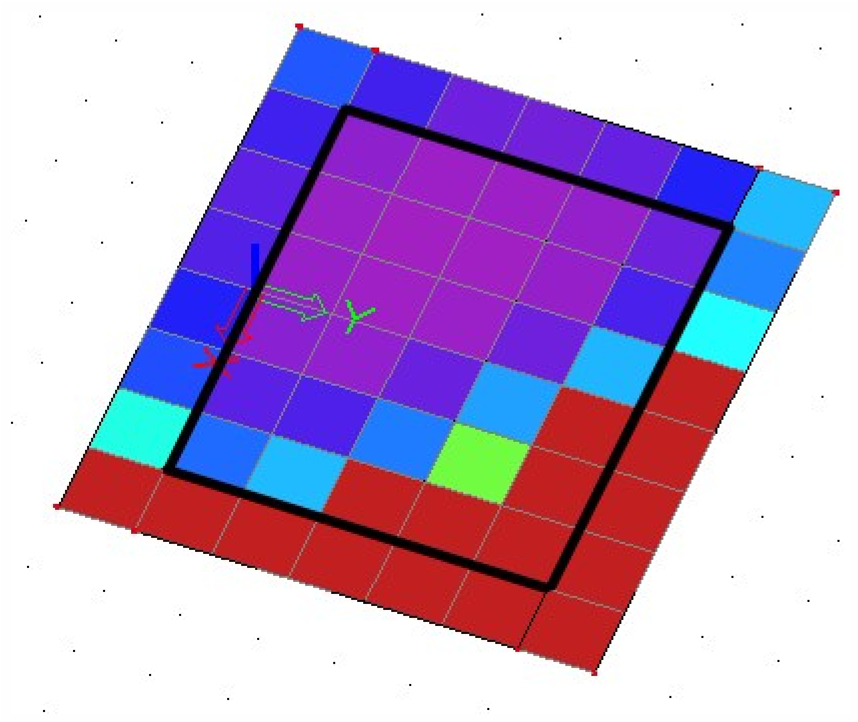
The new plate should be inserted with the minimum thickness (e.g. 0,01mm) and placed next to the foundation.

The C parameters for this affected subsoil around the structure are calculated this way also.

The deformed subsoil calculated by the Scia Engineer:



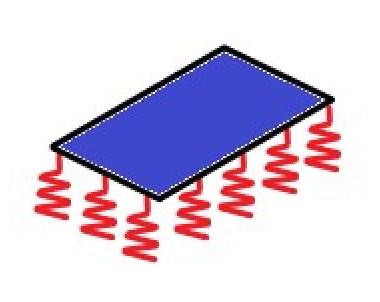
Calculated C parameters:



The structure is marked by the black rectangle and around this is one more plate - surrounding plate – with thickness 0,001mm.

Automatic calculation of the edge supports

When user don't use any subsoil elements then the program will eliminate the neglect of the subsoil on edges by an automatic inserting of vertical supports on the foundation edges.



The calculation of those supports is based on already known C parameters. The program try to support the plate in the same way as it should be supported by the subsoil itself. This leads to approximate model where the sum of reaction is contact stress with reactions in those nodes.

This solution can be sometimes undesirable – e.g. if there is a second foundation near by the calculated one or there is some other support under or near the foundation edge.

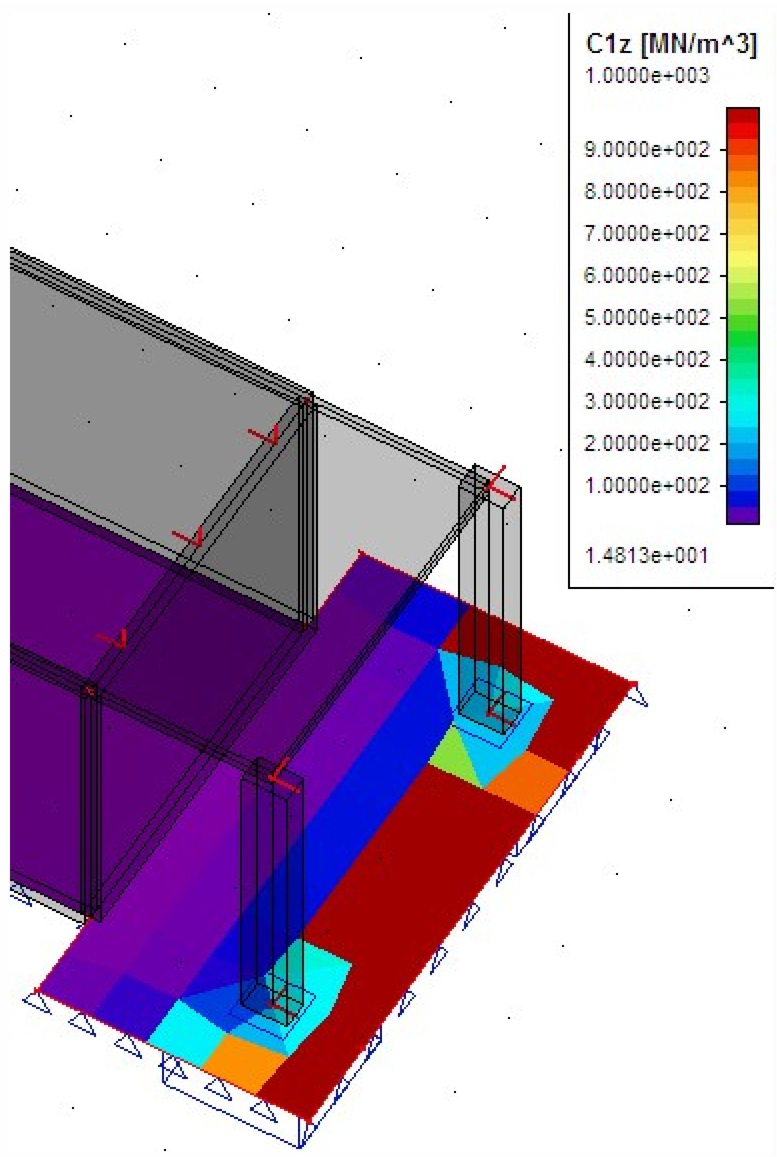
This automatic input can be avoided manually. User can insert a spring with a small stiffness on the plate edges and then the system won't use automatic input of vertical supports. This could be the additional subsoil elements.

Pad foundation and soil-in

The pad foundation is not connected with the soil-in calculation.

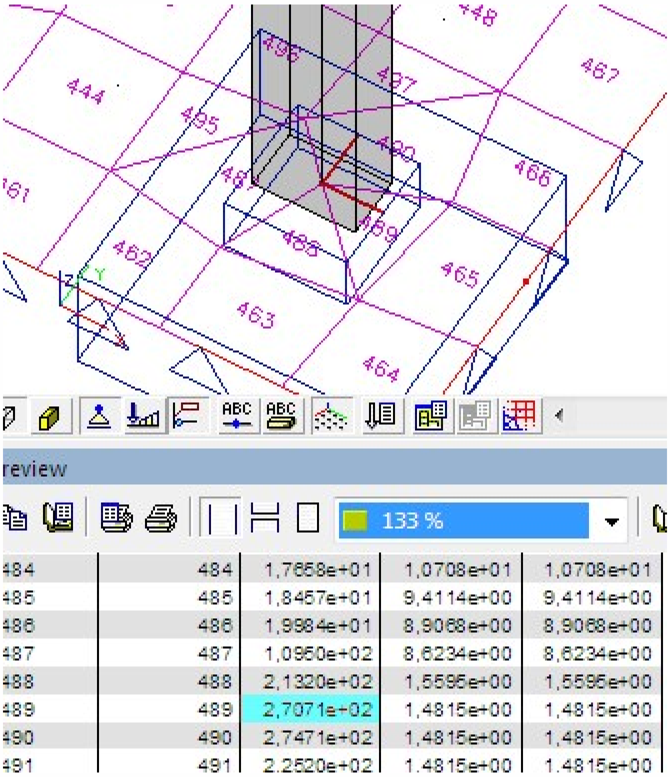
How to use soil-in for the pad foundation check:

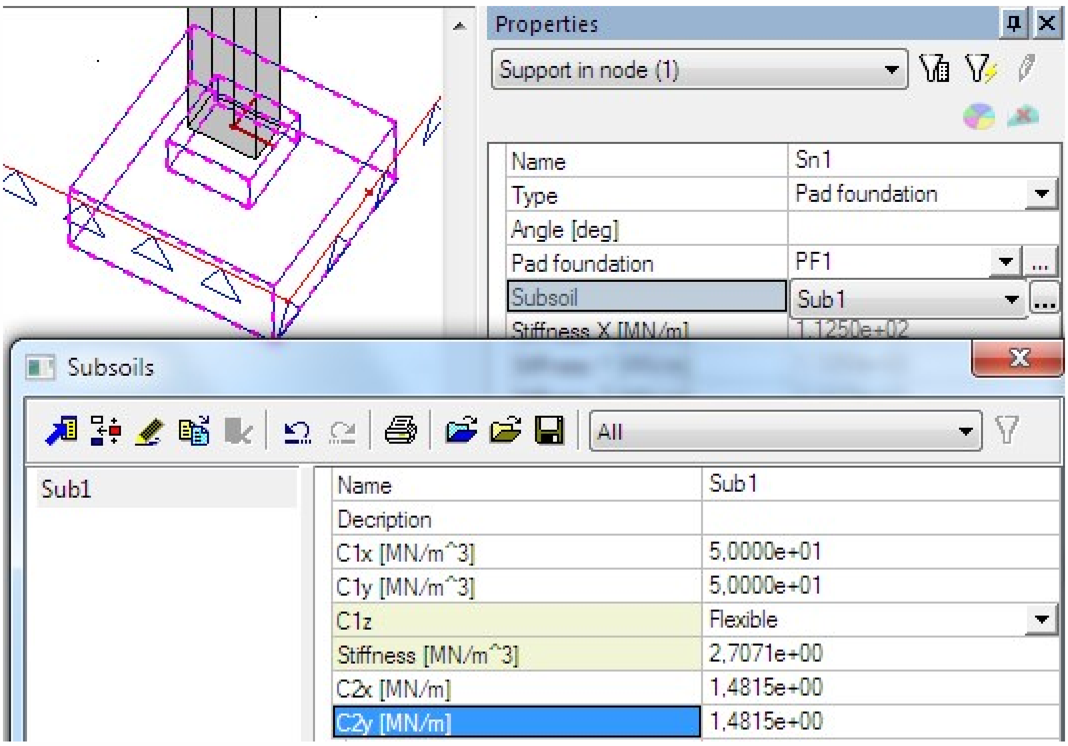
1. Create additional structure to calculate the C parameters in the nearest surrounding (it is described in the previous tip)



Calculated C parameters on the surrounding plate –> C parameters for the pad foundation

2. Calculated C parameters can be used in the Subsoil library. Put the values from the table to the Subsoil library.





3. Run the linear calculation again.
4. Check the pad foundation in a standard way.

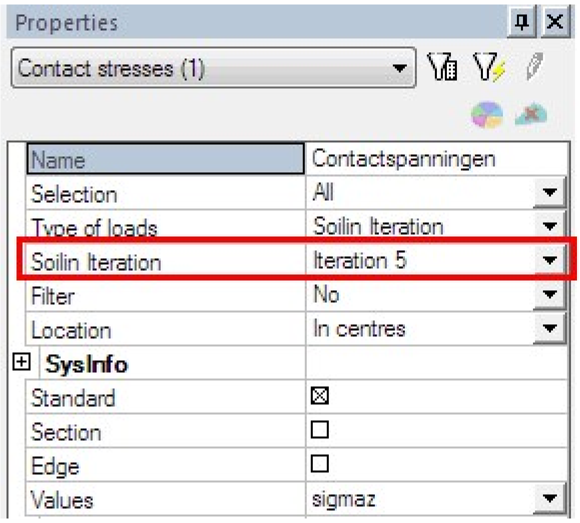
What if the model is correct but the iteration is not finished

Sometimes the model is correct but some circumstances may cause unfinished iterative process. The results in cycles don't lead to one set of C parameters but on the contrary, the results are more and more different.

This can be caused by some tensions in the foundation plate, specific foundation members and similar problems.

How to solve those problems:

1. It is necessary to check the model. It must be correct – the mesh elements are not triangular, the element´s Z axis is upward, the foundation plate must be under the soil surface and so on.
2. Check the iteration cycles in results – contact stresses, type of loads – soilin iteration.
   First few iteration cycles will be probably quite OK and after some time the results become messy.
   Find one cycle (between those correct ones) where the results seem to be close to the reality – e.g. 5thcycle. Use this value in the solver setup for number of iteration cycles.

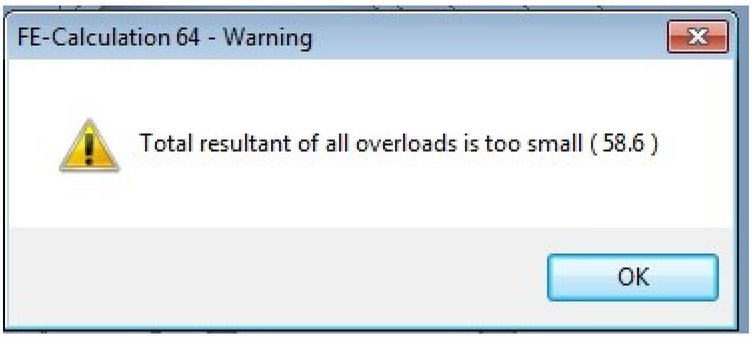


3. Start the linear calculation again, it will be finished after the 5th iteration cycle with results most closest to the reality. The correct cycle is between 2nd and 5th cycle in the most cases.

What it the load is wrongly inserted?

When the plate is not in compression, then soilin cannot be calculated properly.

There could be a message about wrong total resultant:



This may happen when loads are from the bottom to the top, or when there is some change in local LCS of the plate.

What if the symmetrical structure gives non-symmetrical results?

This may happen when additional subsoil elements are not added around the structure.

Also when the soilin didn't find the correct result and calculation is stopped too soon. (For example when solver setup defines only few soilin cycles.)

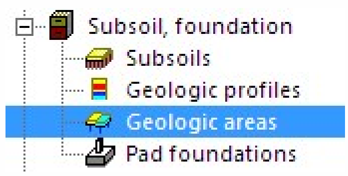
What if geological fault in the subsoil is needed?

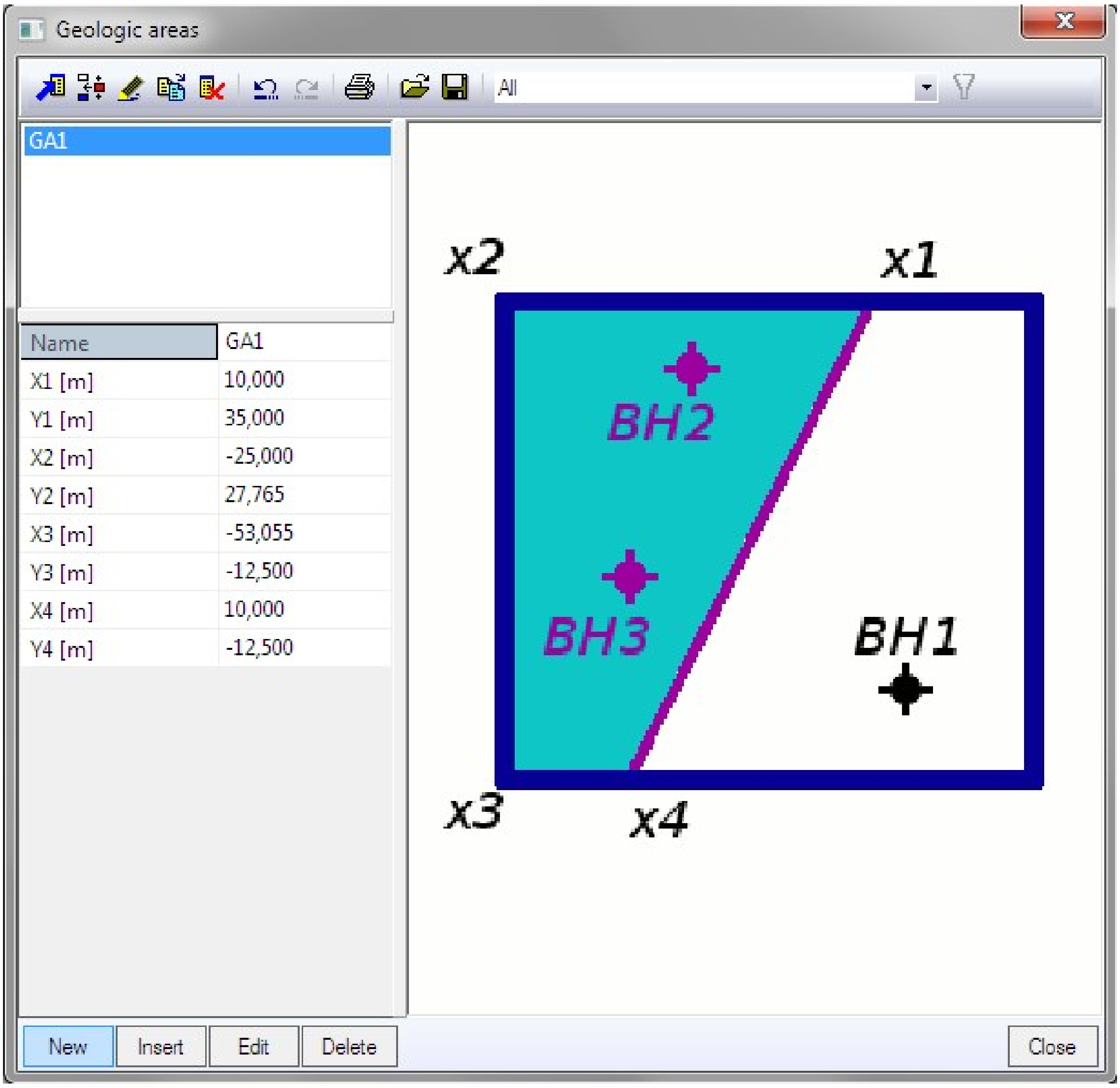
The main subsoil surface is created automatically when a structure and a borehole is inserted to the project. The size of the subsoil outline is calculated from the structure size, positions of boreholes and some offset around.

The geological fault is created when the second area is inserted to the main one.

1. Create structure and insert boreholes.
2. The main subsoil surface is created.

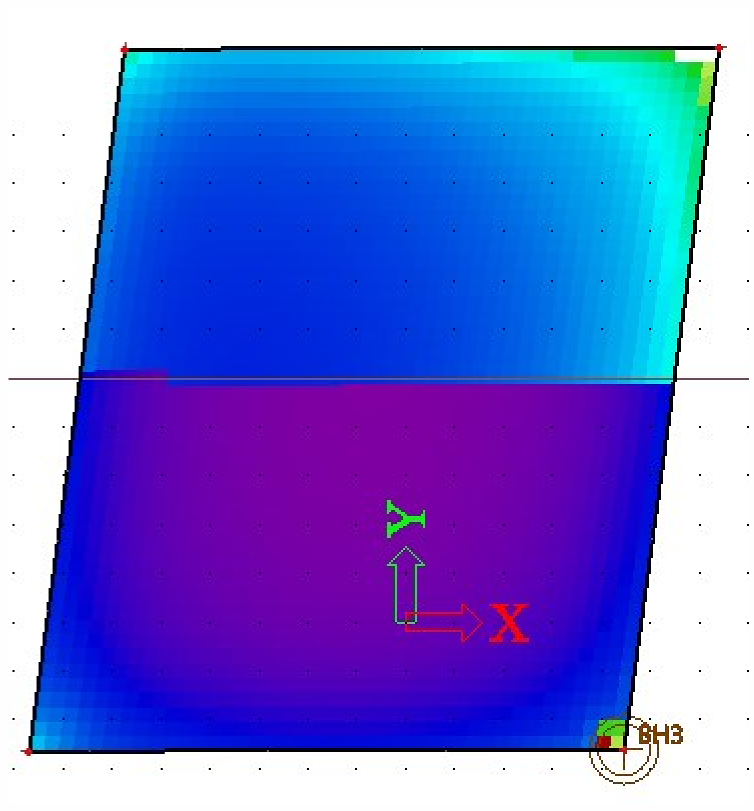
3. Go to Libraries / Subsoil, Foundation / Geological areas, create a new area by its coordinates.





4. Run standard Soilin calculation.

5. Check results where geological fault is visible as a border between two gradients.

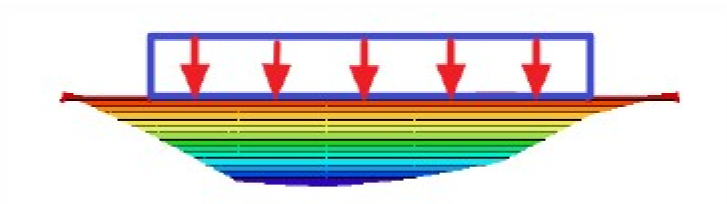


How to use additional plates

Soilin is a tool which calculates C parameters of the subsoil under the surface support. Using the additional plates around the support provides more realistic results.

About C parameters:

1. C parameters are parameters of interaction, so their value depends on the structure, load, stiffness and subsoil. Change in any of those parts causes different C parameters.
2. The whole plate is supported vertically by the soil stiffness – parameter C_1 (winkler) and also in the shear direction – parameter C_2 (pasternak).
3. The plate edges are more supported by the C_2 parameters because it is affected by neglecting.
4. The area around the support is affected by the shear stiffness of the soil and the degrease basin is created.

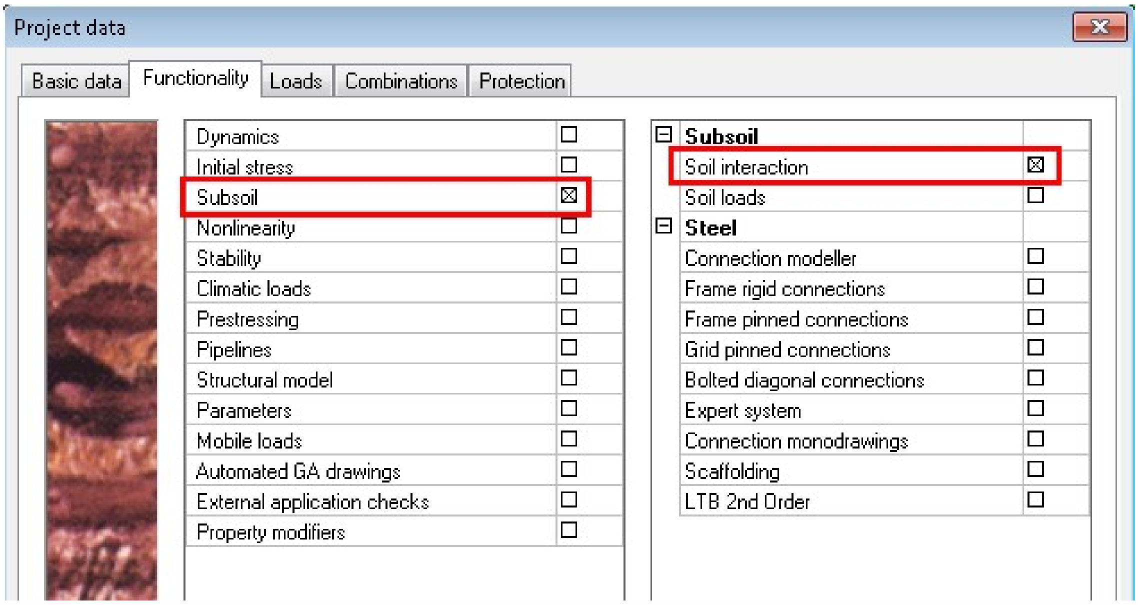


5. The degrease basin can be substituted by spring supports around the plate – this is done automatically in Scia Engineer when user don't add plates around.
6. When user uses the plates around the support, the springs are not added and the C parameters are calculated for the whole area.

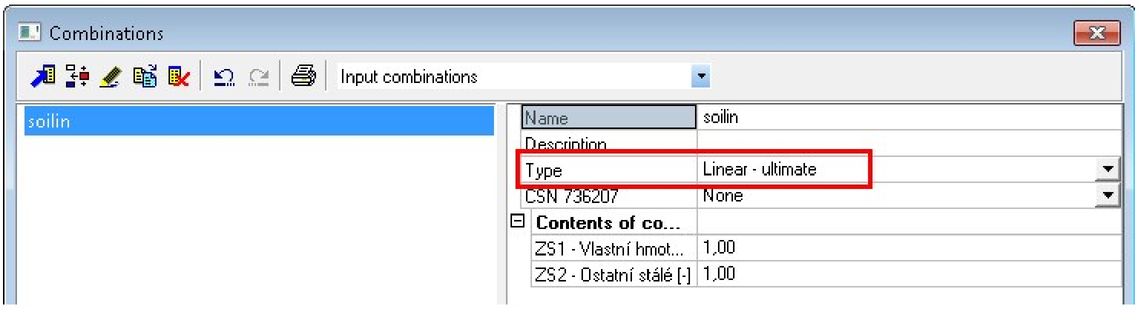
-> The next text describes how to create plates around the support - additional plates.

Settings for soilin calculation

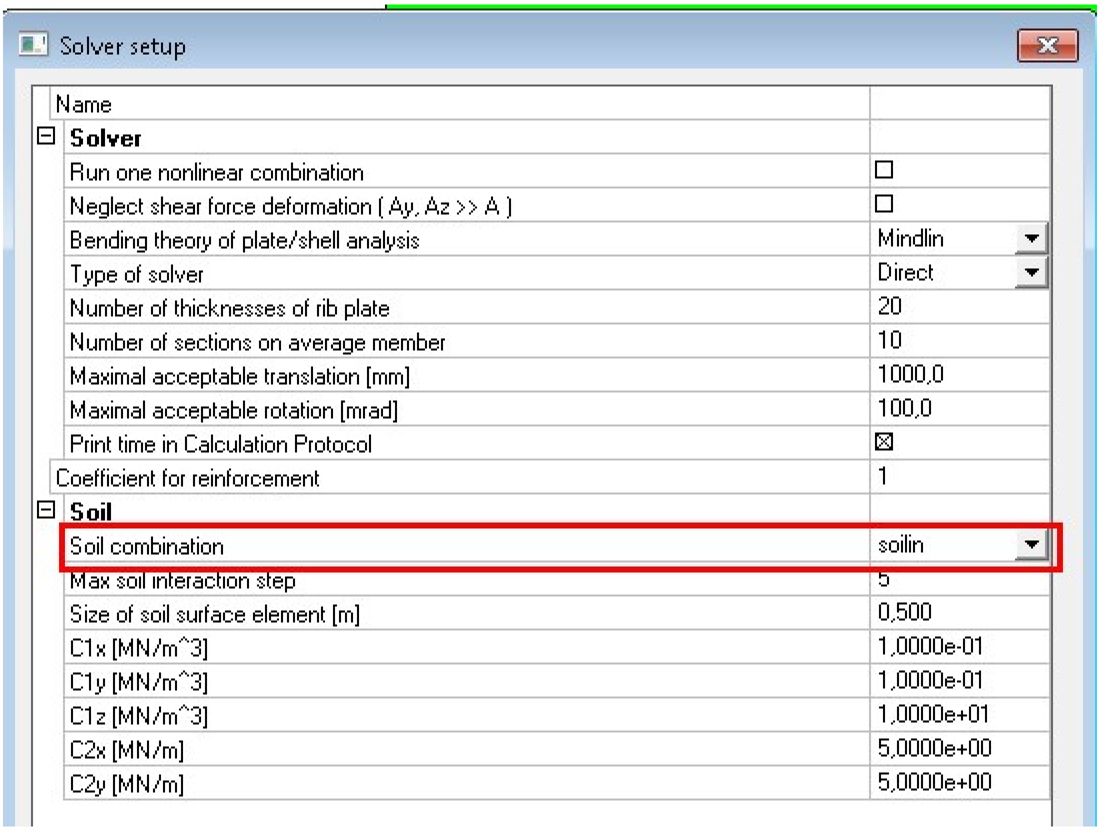
1. The functionality Subsoil and Soil iteration must be checked.



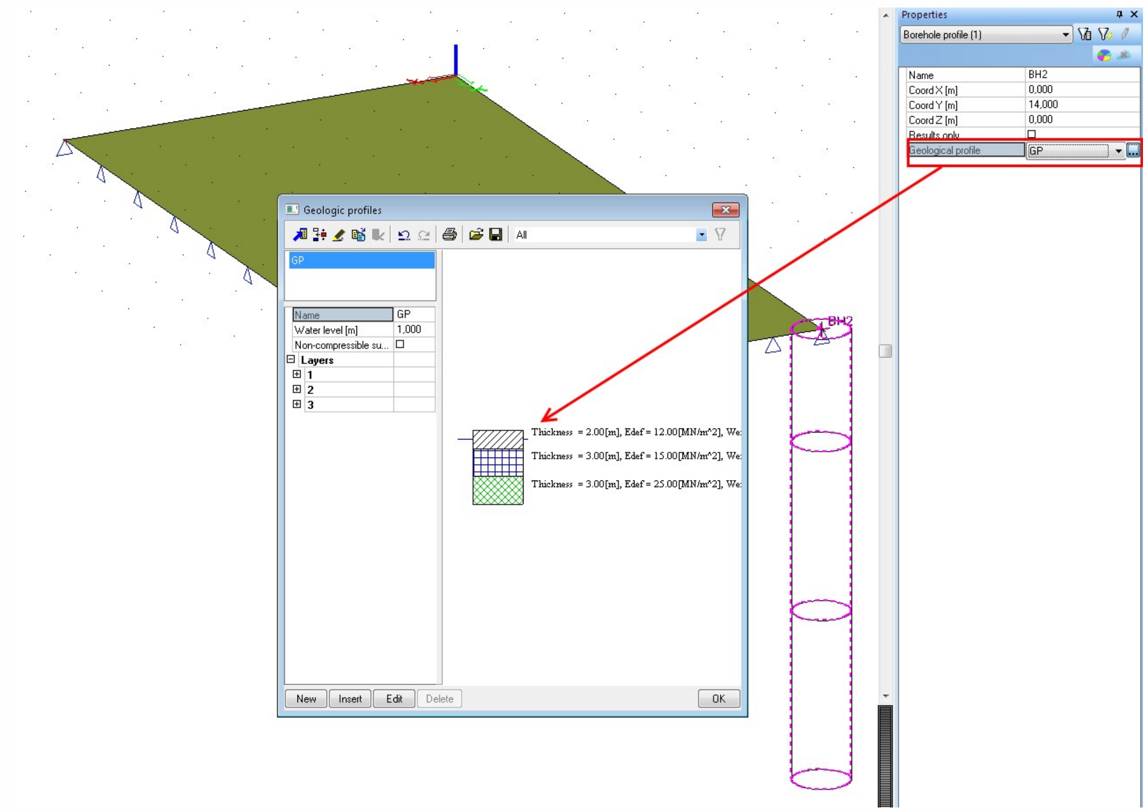
2. One combination must be linear - this combination is used for soilin calculation.



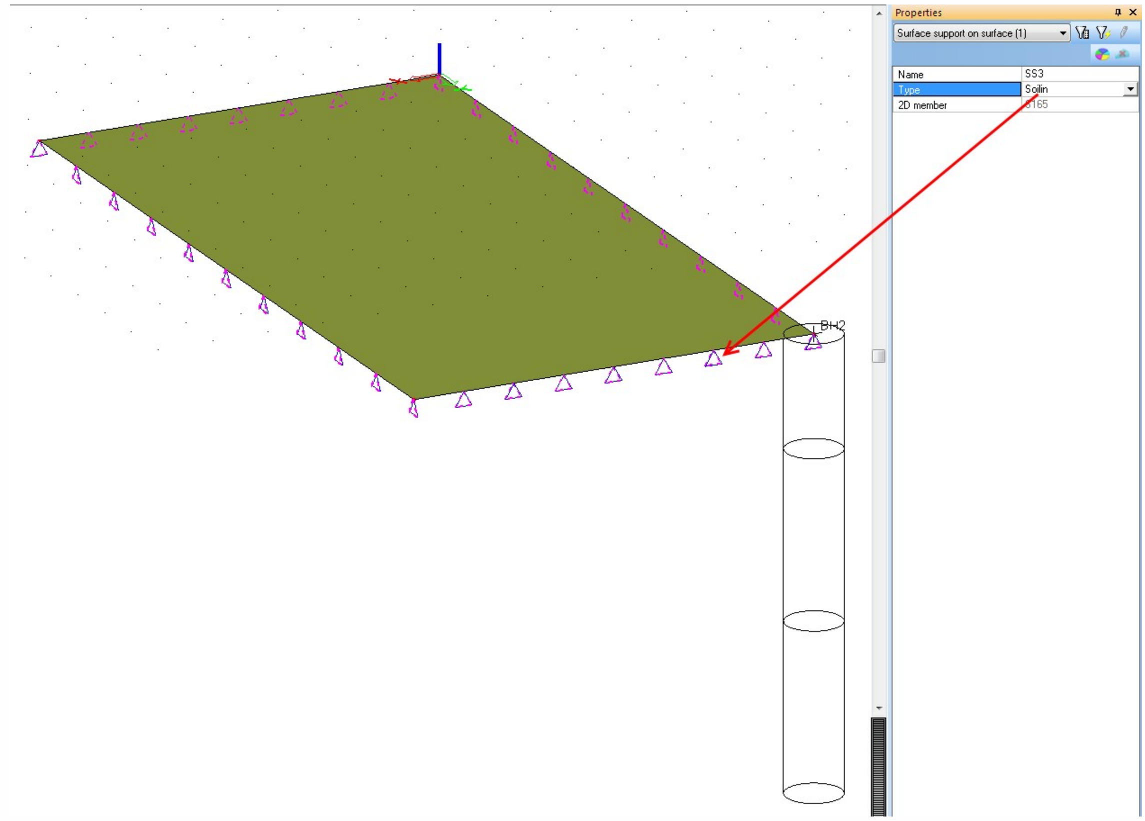
3. This linear combination must be selected in Solver setup to run soilin with it.



4. The project must contain borehole with geologic profile.



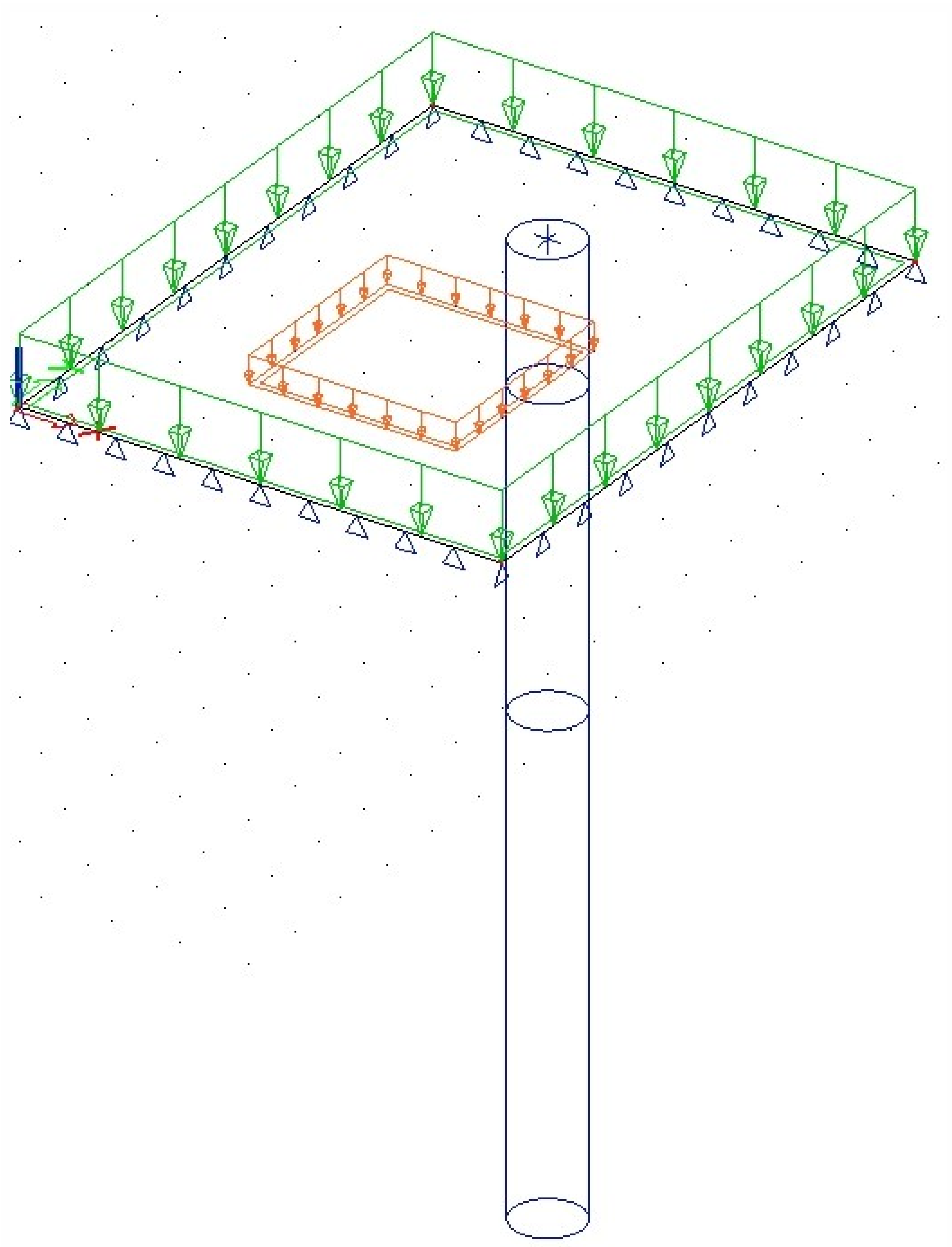
5. The project must contain surface support type soilin.



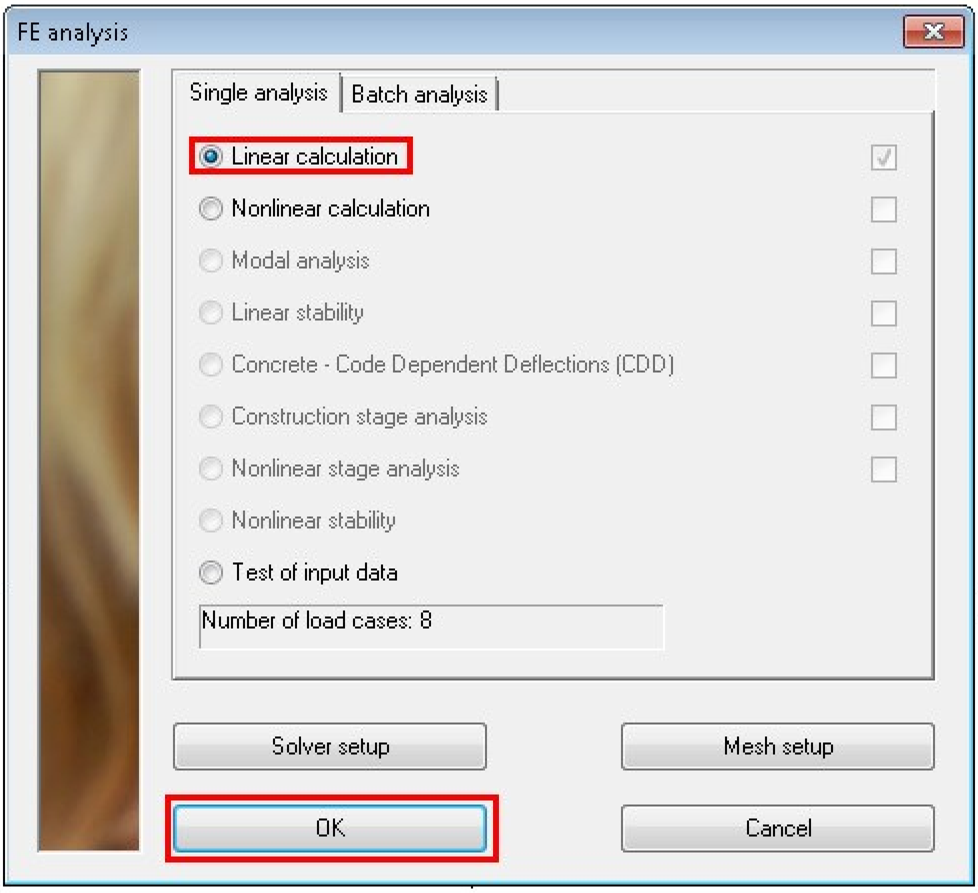
How to calculate the plate without soilin

1. Open the project "soilin_start.esa".

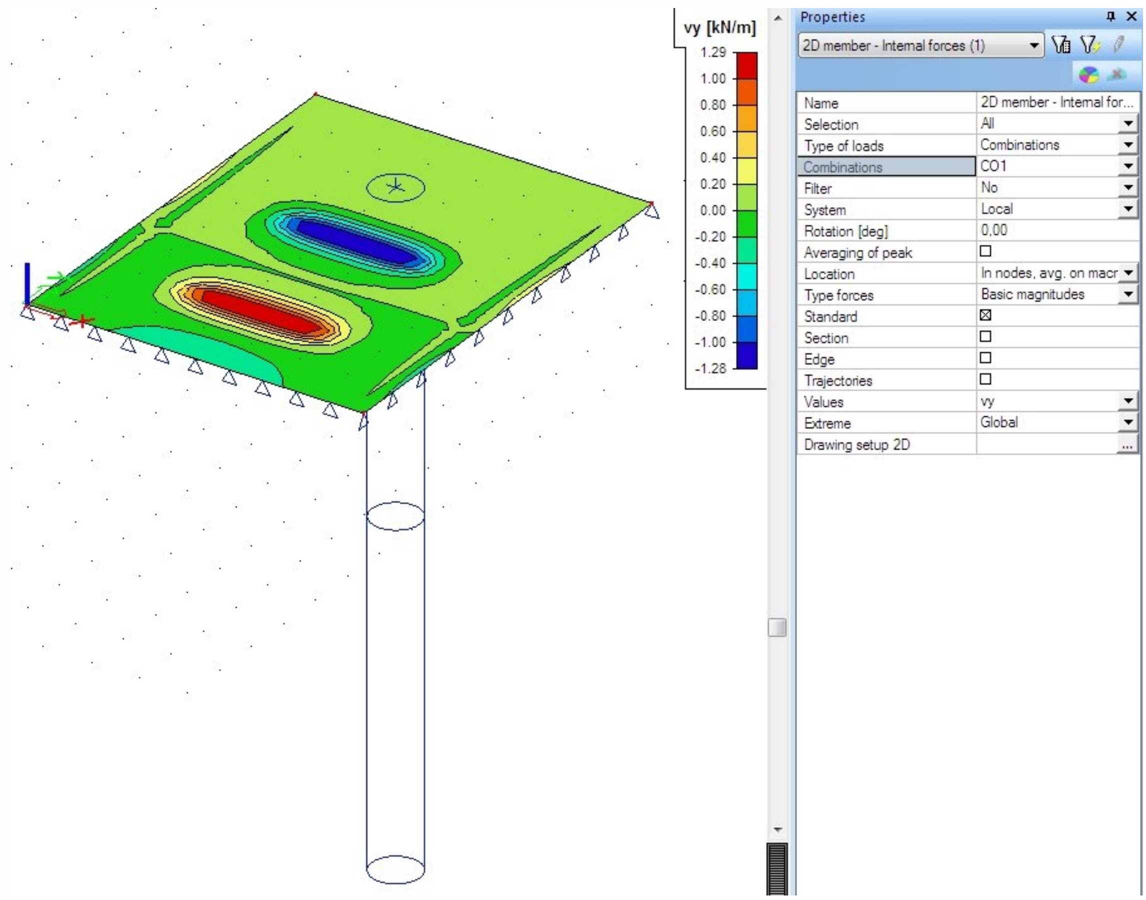
2. There is one plate with the surface support type Individual. This type of the support has a constant parameters C_1 and C_2.



3. Run the linear calculation with the default settings.

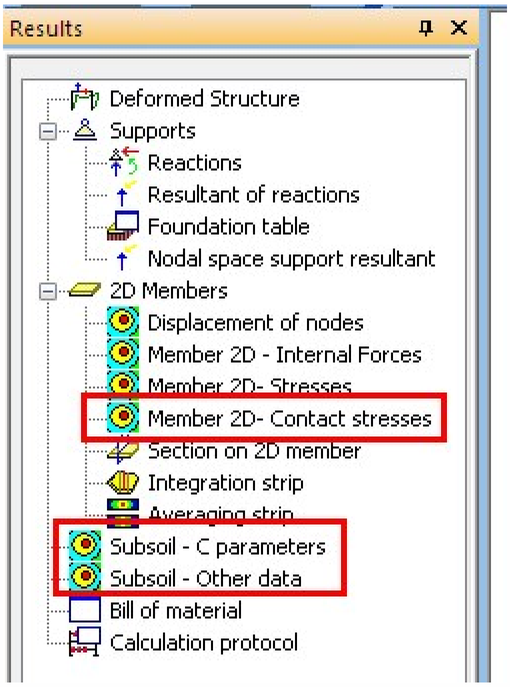


4. Go to the service Results. Display the results for internal forces. There are no results for C parameters.
5. Internal forces - for example vy:

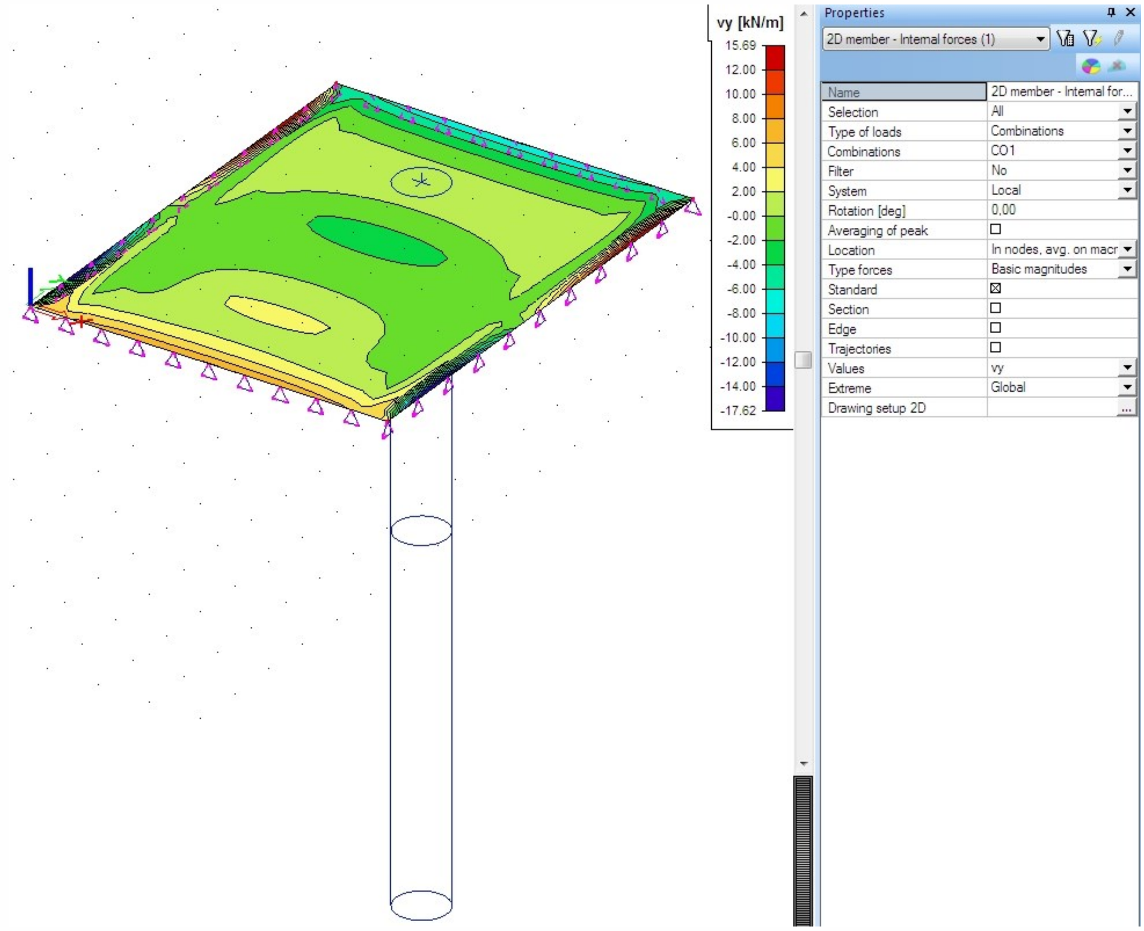


How to calculate the plate with soilin.

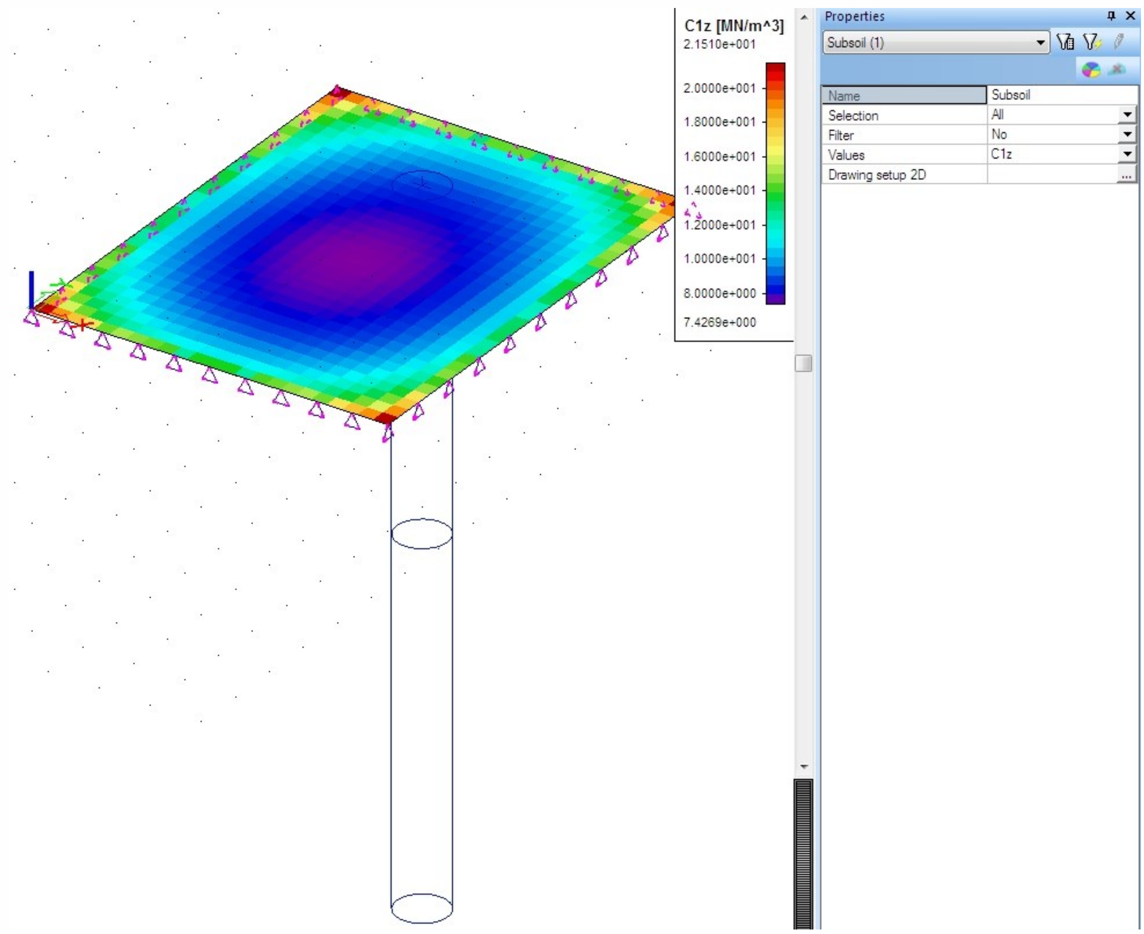
1. Change the support type to soilin.
2. Run the linear calculation again.
3. Go to the service Results. Display the results for internal forces and soilin for combination C01.



4. Internal forces - vy:



5. Subsoil - C parameters - parameter C1z:



6. Subsoil - Other data (see the preview with the table for the settlement):

Preview

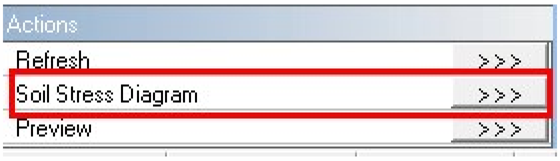
120 %

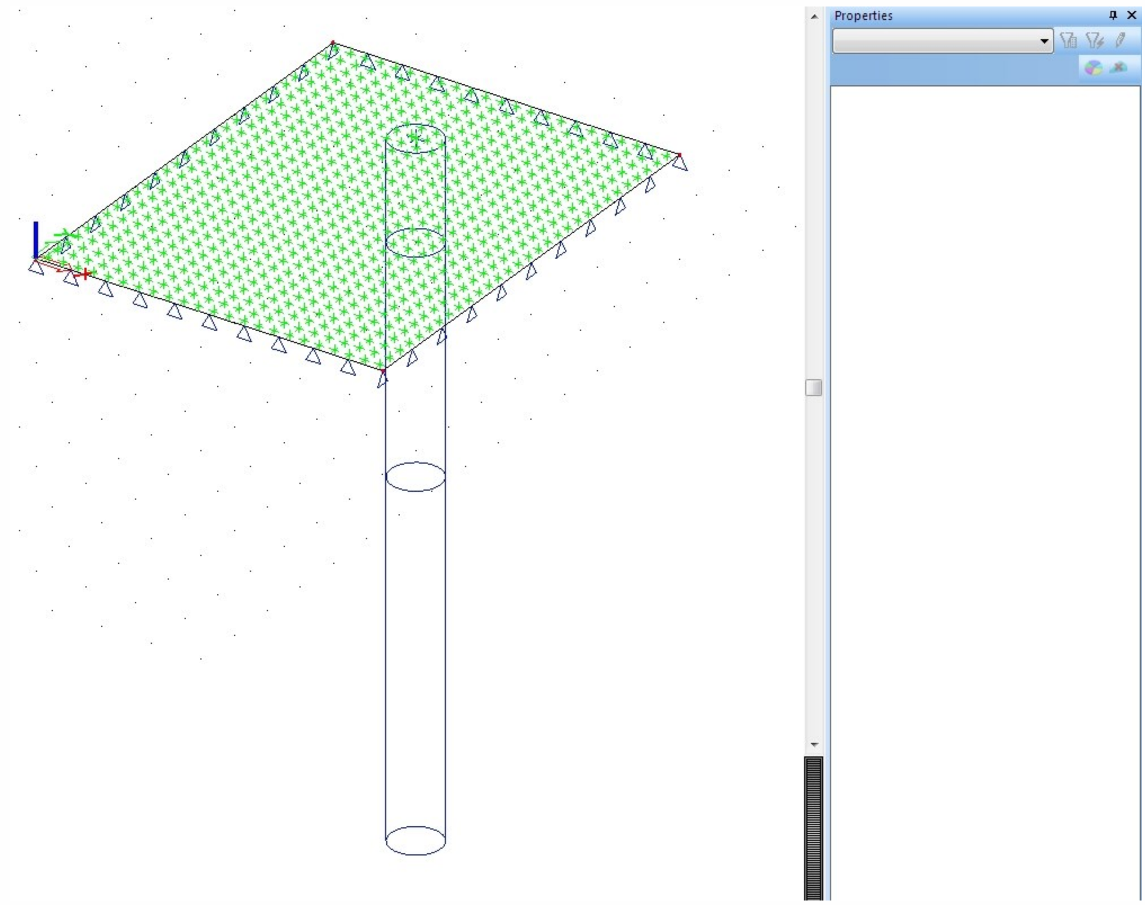
Subsoil - Other data

Selection : All
Combinations : CO1

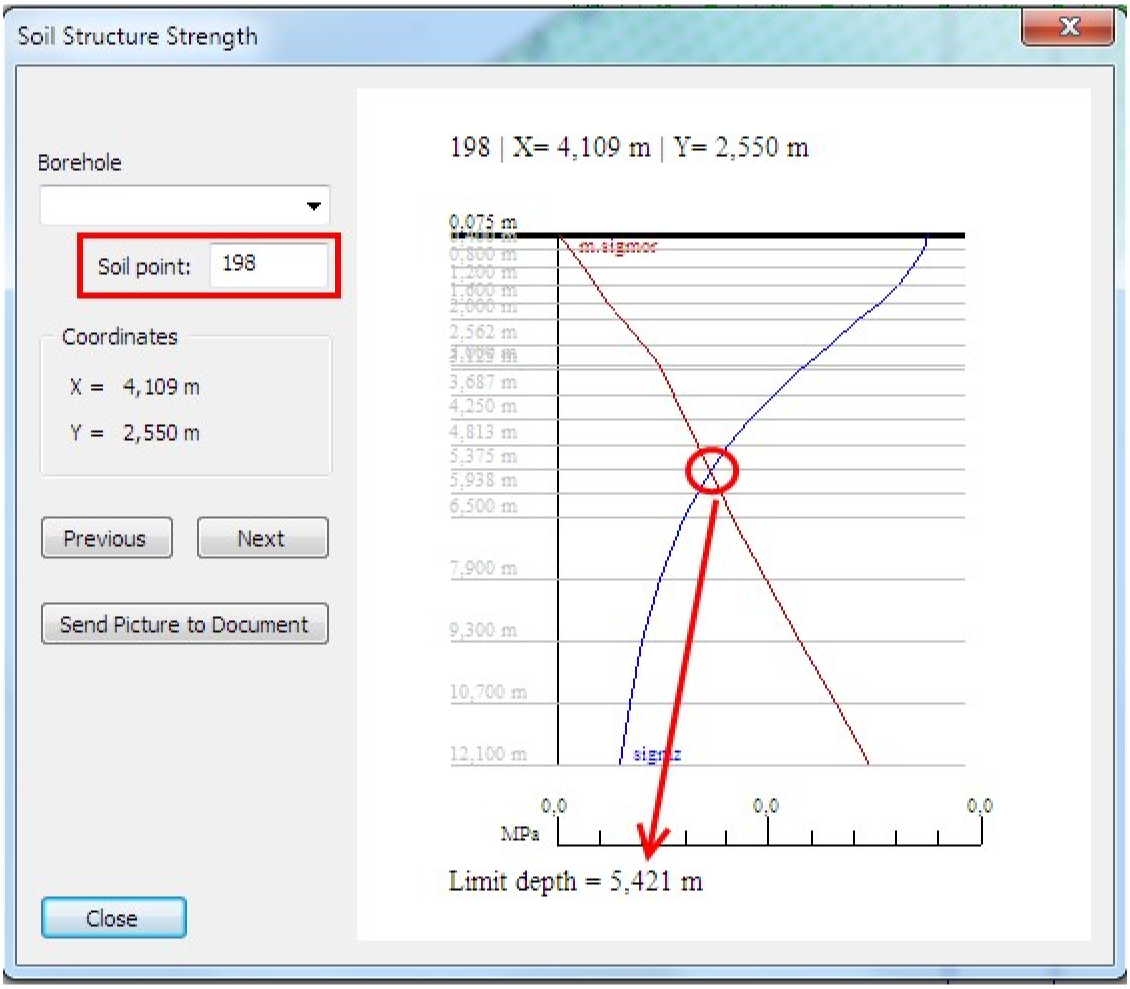
Element	X [m]	Y [m]	w [mm]
1	0,152	0,150	0,7
2	0,457	0,150	1,1
3	0,761	0,150	1,5
4	1,065	0,150	1,6
5	1,370	0,150	1,4
6	1,674	0,150	2,0
7	1,978	0,150	2,2
8	2,283	0,150	2,4
9	2,587	0,150	2,3
10	2,891	0,150	1,8
11	3,196	0,150	2,4
12	3,500	0,150	2,4
13	3,804	0,150	2,4
14	4,109	0,150	1,8
15	4,413	0,150	2,4
16	4,717	0,150	2,3
17	5,022	0,150	2,2
18	5,326	0,150	1,9

7. Subsoil - Other data - use the action button "Soil Stress Diagram" and select one green vertex:





8. A new dialogue appears - there is a stress diagram for the selected mesh element:

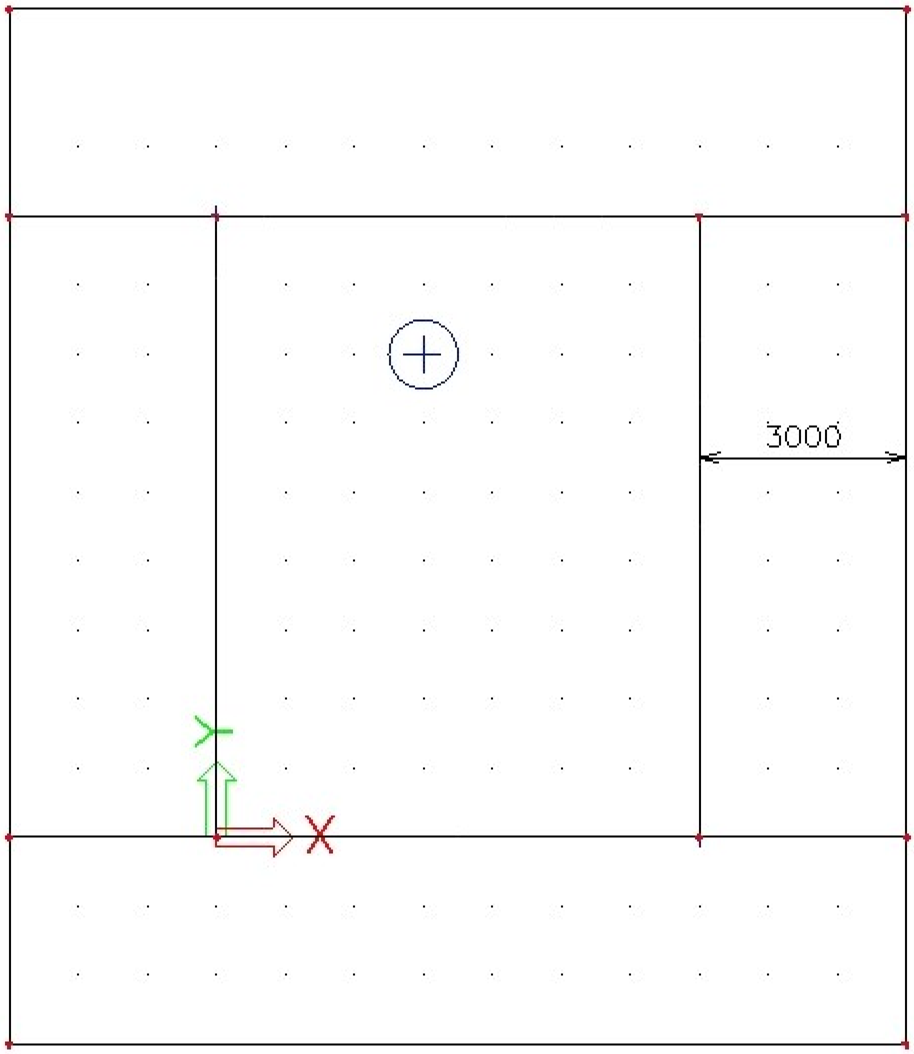


9. Close the dialogue.
10. Use ESC to finish the action.

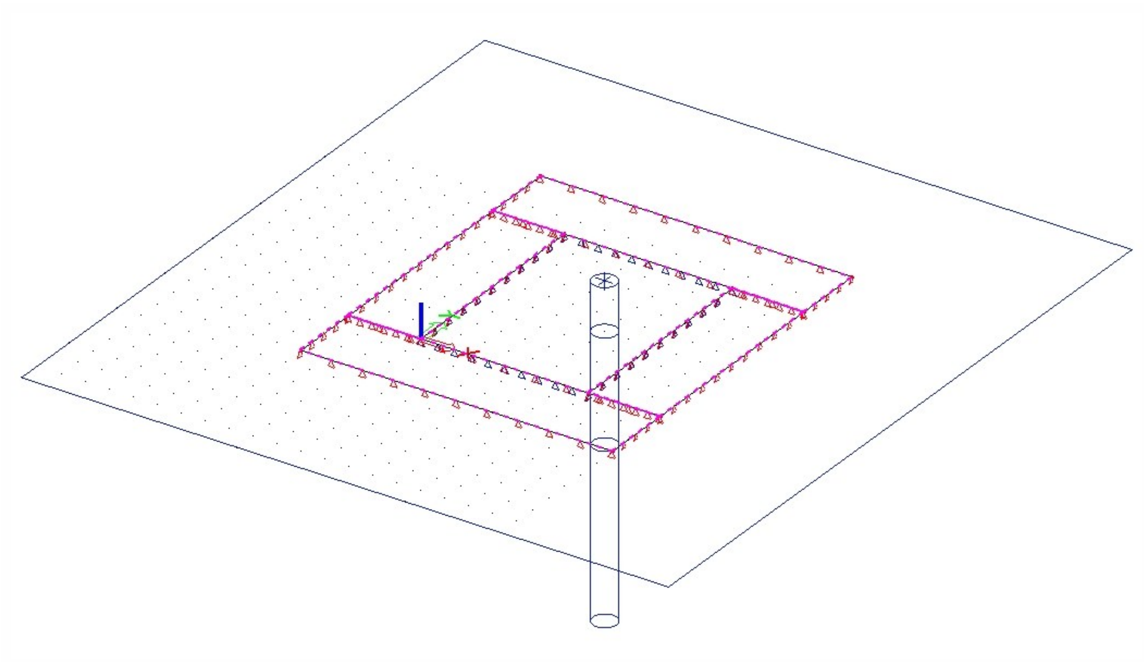
The edges of the plate are supported by springs automatically.

How to create the additional plates around

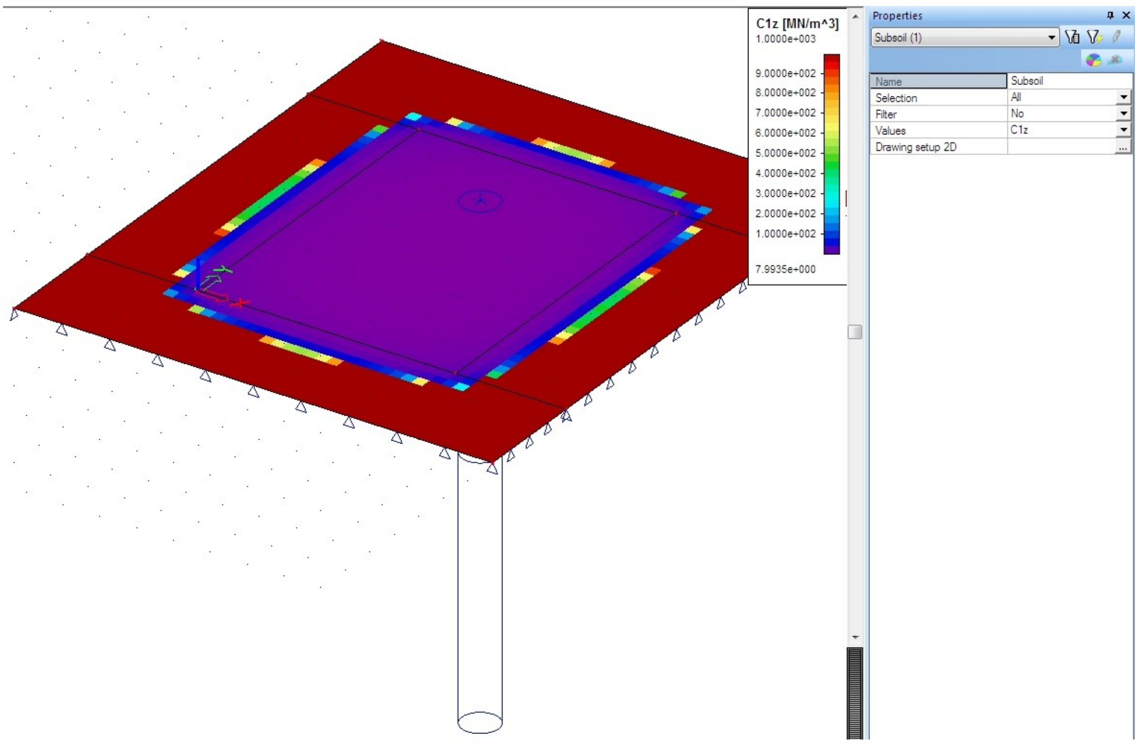
1. Use the same project.
2. Open the Structure service and start the command for inserting a new plate.
3. Set the thickness of the plate to 1mm.
4. Create 4 plates around the surface support according to the picture. The width from the original plate is 3m.



5. Add the surface support type soilin on those plates.



6. Run the linear calculation with the same settings again.
7. Go to the service Results. Display the results for soilin.
8. Subsoil - C parameters - parameter C1z:



9. Subsoil - Other data (see the preview with the table for settlement):

Preview

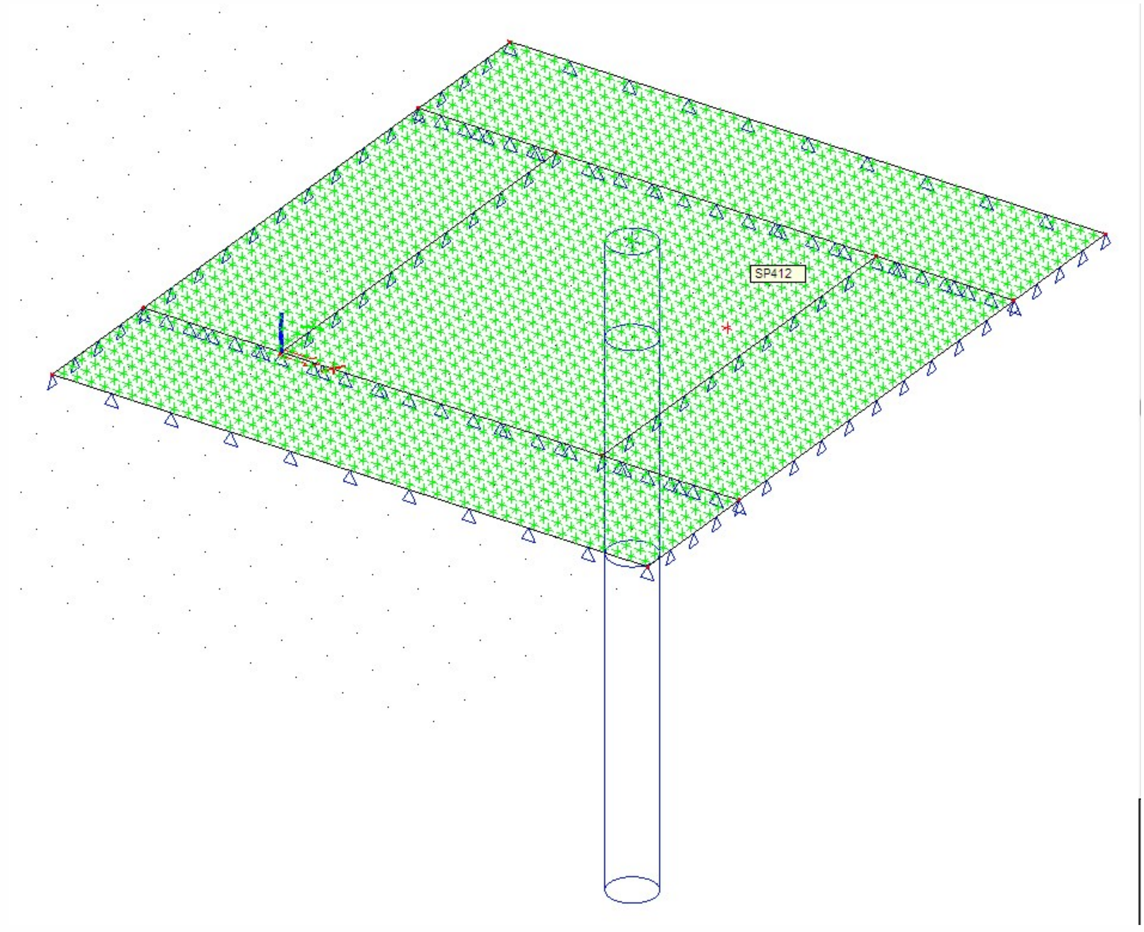
120 %

Subsoil - Other data

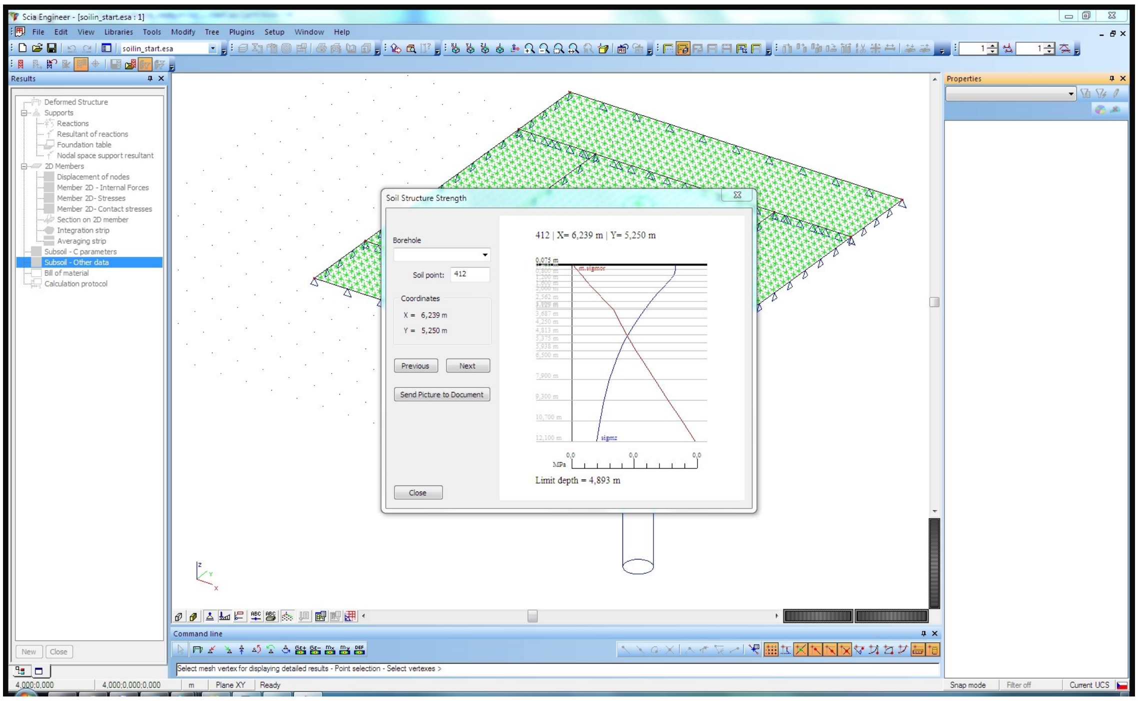
Selection : All
Combinations : CO1

Element	X [m]	Y [m]	w [mm]
1	0,152	0,150	0,6
2	0,457	0,150	0,8
3	0,761	0,150	1,1
4	1,065	0,150	1,3
5	1,370	0,150	1,4
6	1,674	0,150	1,6
7	1,978	0,150	1,7
8	2,283	0,150	1,8
9	2,587	0,150	1,8
10	2,891	0,150	1,9
11	3,196	0,150	1,9
12	3,500	0,150	1,9
13	3,804	0,150	1,9
14	4,109	0,150	1,9
15	4,413	0,150	1,9
16	4,717	0,150	1,8
17	5,022	0,150	1,7
18	5,326	0,150	1,6

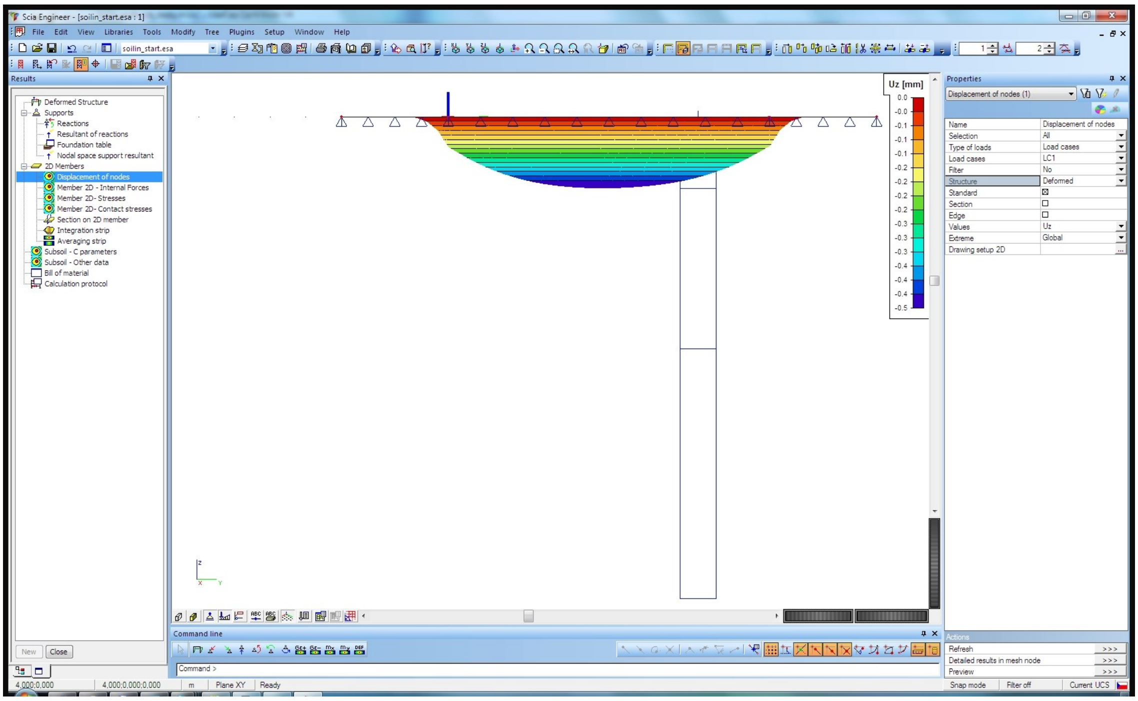
10. Subsoil - Other data - use the action button "Soil Stress Diagram" and select one green vertex:



11. Stress diagram for selected mesh element:



12. Close the dialogue.
13. Use ESC to finish the action.
14. The interesting results are deformations.
15. See the result "Displacement of nodes", value Uz on Deformed structure:



The deformed structure shows the degrease basin.

16. The result is in project "soilin_finished.esa".

Design of piles

Introduction

Pile design functionality is a tool in Scia Engineer developed in co-operation with Deltares. It enables the user to perform the design/verification of bearing piles in accordance with the NEN code (NEN 6740 and NEN 6743) and NEN 9997-1:2009 (NEN EN 1997-1:2005, NEN EN 1997-1:2005/ NB:2008 and NEN 9097-1). This functionality is available only if the national code is NEN or EC-EN.

At the moment, this functionality is available if the user has selected the NEN code or the Eurocode. In the case of the Eurocode, this tool is valid when the user chooses the National Annex for the Netherlands.

The pile design option will facilitate the user to determine the required pile tip level and the bearing capacity at that tip level. The verification option will result in load- settlement curves from ULS and SLS and calculate the pile settlement. The Load settlement curves describes the deformation of a pile as function of the load

The Piles are defined as a special type of support in Scia Engineer. The piles are defined in Pile plans and all the piles in the Pile plan will have the same properties. Piles are fully integrated with the 3D model and the soil profiles. The design and verification of the results depend on the soil profile, properties of pile and the reactions from the structure.

The soil profiles are generated from the Cone Penetration Test data (.GEF – Geotechnical exchange format file - ASCII). The soil profiles from CPT are linked with the pile plan and are used for design and verification. A link has also been provided to import the CPT data from the net (DINO by TNO). This facilitates the user to select the CPT data (.GEF file) available in the specified location.

An automatic interpretation tool is used for generating the soil profile. It uses the stress dependent NEN rule for the interpretation. The predefined soils are used by this interpretation tool.

The definition of the pile shape and type is stored in the pile plan library. All the relevant parameters required by the pile are defined in the Dutch standards.

The piles are represented in the 3D model and their display is controlled by view parameter settings.

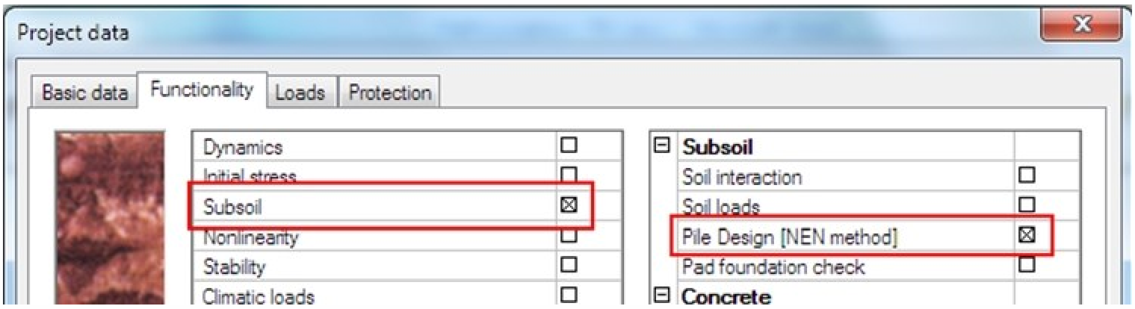
The program enables the user to generate Non-linear functions from load-settlement curves and the generated functions can be associated to the supports (type Z).

Recalculating the entire structure using these non-linear functions will improve the overall results. It leads to ‘new’ loads on the piles. With these loads, the process of pile-design, verification and calculation of the entire structure can be repeated to optimize the total design.

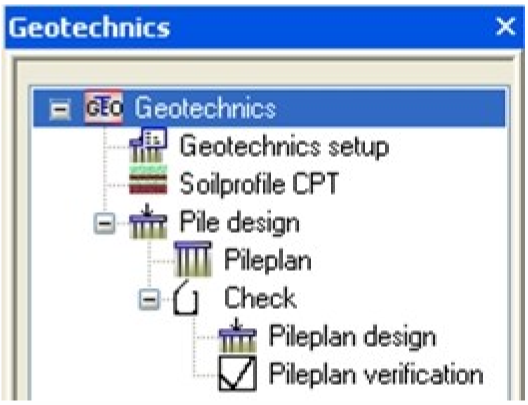
There is a possibility to store the input and output xml files which are used for Pile design and verification. It is possible to use XML in the DFoundation and enhance the pile design there.

Geotechnics service

Pile design (NEN Method) functionality is added as a new type under the subsoil category. This functionality will be available only for the National codes “EC-EN” and “NEN” and for the structure types “General XYZ, Frame XYZ, Frame XZ, Grid XY and Plate XY”.

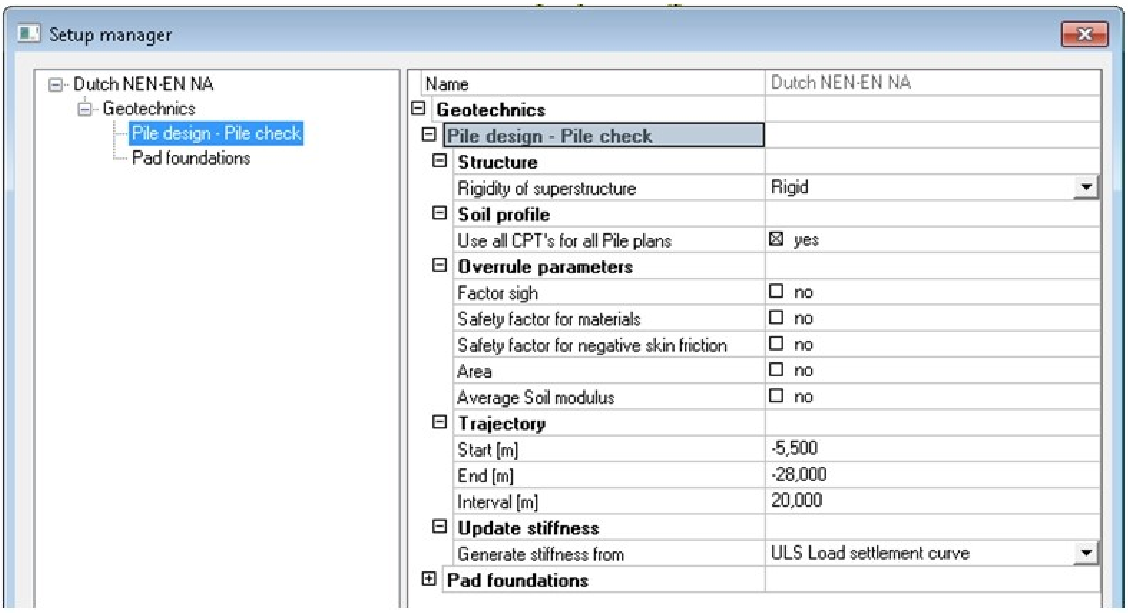


This service is common for Pile design and Pad Foundation. This service is available only if the Pile design or Pad foundation functionality is selected.



Geotechnics setup

This setup is common for the Pile design and Pad Foundation.



Rigidity of Superstructure

The superstructure can be specified either Rigid or Non-rigid. This parameter influences the calculation. The ξ (sigh factor) also depends on the rigidity of the structure.

Use all CPTs for all pile plans

If this option is selected all the soil profiles will be automatically associated to all the Pile plans. If the user wishes to manually associate the CPTs to pile plans this option must be unchecked.

Overrule parameters

The parameters listed under this can be overruled. When it is set to NO then the parameters are set according to the standard or th

ey are calculated. The user has to make sure that the overruling of parameters is allowable.

Factor sigh[ξ]
This factor depends on the number of CPTs and the number of piles under rigid super structure.

Table 1, NEN 6743:

M	N						
	1	2	3	4	5	7	≥10
1 or 2 1)	0,72	0,76	0,77	0,78	0,78	0,79	0,80
3 - 6	0,76	0,80	0,82	0,83	0,83	0,84	0,85
7 - 9	0,78	0,84	0,86	0,87	0,87	0,89	0,90
≥10	0,79	0,85	0,87	0,88	0,88	0,89	0,91
1) Also for 2 piles the system is controlled by one pile.							

N - For structures which belong to GC2 the building place should be researched by means of the terrain tests and laboratory tests. The number of load tests, field tests or borings with undisturbed soil samples on the part of the field for which the calculated value of the maximum capacity of piles is determinate.

M - The number of piles under the considered part of the structure

Safety factor for materials (γ_{mb})
Derived from table 3 in NEN 6740.

Table 3, NEN 6740:

Table 3, NEN 6740:

Type of the geotechnics construction	Partial material factor	Boundary conditions				
		extreme			usefulness	
		1A – favourable 1)	1B – favourable 1)	1A - unfavourable 2)	1B - unfavourable 2)	2
all	γm;g density of the soil	1,1	1,1	1	1	1
Foundation for buildings:						
On piles (pressure loads)	γm;b1without research 3)	1,4	1,4	1	1	1
	γm;b2 with test load	1,25	1,25	1	1	1
	γm;b3 for piles (loaded by test) and anchorages	1,15	1,15	1	1	1
	γm;b4 from CPT	1,25	1,25	1	1	1
On steel	γm;ϕ' tangent of the angle of internal friction	1,15	1,15	1	1	1
	γm;cl cohesive (bearing capacity of the foundations)	1,6	1,6	1	1	1
	γm;fundr undrained shear strength	1,35	1,35	1	1	1
Other constructions piles/anchors (tension/horizontal loading)	γm; b1without research	1,4	1,4	1	1	1
	γm;b2 with test load	1,4	1,1	1	1	1
	γm;b3 for piles (loaded by test) and anchorages	1,25	1,25	1	1	1
	γm;b4 from CPT	1,4	1,4	1	1	1
	γm;ϕ' tangent of the angle of internal friction	1,2	1,2	1	1	1
	γm;c2 cohesive (soil pressures equilibrium slopes)	1,5	1,5	1	1	1
	γm;fundr undrained shear strength	1,5	1,5	1	1	1
	γm;Cc, γm;Cα, γm;Csw	1	1	0,8	0,8	1
	γm;Cp, γm;Cs	1,3	1,3	1	1	1
	γm;E	1,3	1,3	1	1	1

1) 'Favorable': if an increase of the value of the relevant parameter leads to a more favorable result.

2) 'Unfavorable': if the increase of the value of the relevant parameter leads to a more unfavorable result

3) Here should be used the stroke diagrams which are included in the driving piles into the ground, or else the following holds, for states 1A and 1B (favorable): g_m;b1=1,8

Safety factor for negative skin friction ($\gamma_{f;nk}$)
Derived from NEN 6740 11.5.1

ULS: $\gamma_{f;nk}$ = 1,4

SLS: $\gamma_{f;nk}$ = 1,0

Area
The influence area per pile is used in the calculation of negative skin friction for pile groups. If this option is not overruled, the program calculates the influence area. This is done by calculating the average pile distance within the pile group (D_{avg}). Area = D_{avg}*D_{avg}.

Average soil modulus
It is the mean modulus of the soil under 4D-level under the pile tip derived from measurements of the settlement of building constructions in the environment (kN/m^2). (NEN 6743-1:2006, art. 6.3.2)

Trajectory

The pile tip level is located in a user defined pile tip trajectory. The trajectory is defined by its top (Start value), its bottom (End value) and the interval. The interval determines the number of calculations to be performed.

The friction zone levels should not be taken into account, this is automatically adjusted to each interval if it is needed.

Start

Start of the trajectory (which should be calculated)

End
End of the trajectory

Interval
Determines the number of calculations to be performed (max value of intervals in the trajectory is 151)

Requirements for the start and end:

1. The start trajectory must be at least 5*d_{min} below the lowest surface level, excavation level and pile head level. (d_{min} - smallest cross section dimension of the pile)
2. The end trajectory must be at least 4*D_{eq} above the deepest level of the shallow CPT. (D_{eq} – Equivalent diameter)
3. The interval has to be chosen in such a way that the maximum number calculations should not exceed 151.

The example:
CPT borehole top= -1,0m
CPT borehole bottom= -30m
The pile diameter= 0,4m
The surface level thickness= 2,6m

The start must be a least 5x0,4=2m below the surface level -> Start= -5,6m
The end must be at least 4x0,4=1,6m above the borehole bottom -> End= -28,4m
The maximum number of calculation is 151 -> -28,4 - 5,6= 22,8
22,8/151=0,15

The check on these three values is performed.

1.0 < abs ((End - Start) / Interval) < 151.0

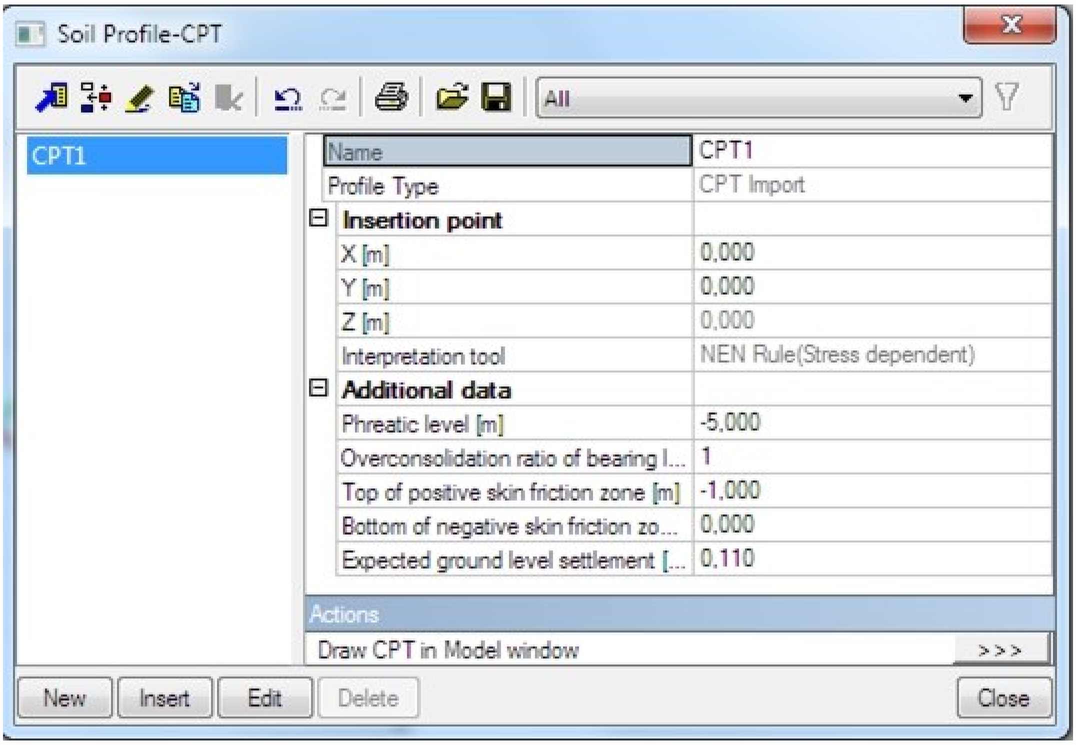
- This means that the absolute value must lie between 1,0 and 151,0
- If this condition is not fulfilled, then interval is not taken from the dialogue. It is calculated according to which part is not fulfilled:

  a. abs ((End - Start) / 1.0)
  b. abs ((End - Start) / 151.0)

This condition is calculated also in the check and when it is fulfilled the data are sent to the calculation.

Soil Profile CPT

The soil profile CPT is a new library in Scia Engineer. It enables the user to generate the soil profiles from CPT data. The generated soil profiles are used in the Pile plan design and verification.



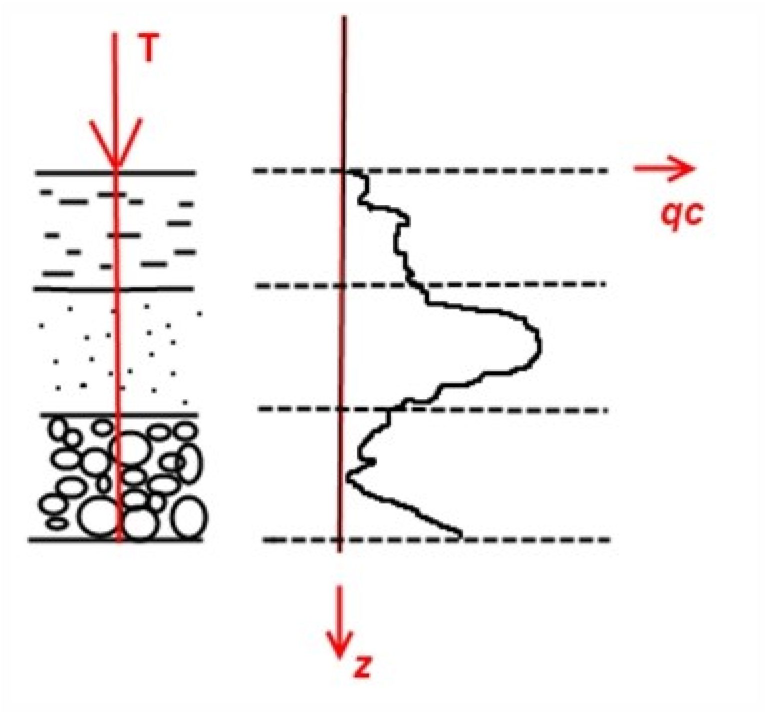
CPT Data

The soil profile can be loaded to the library by the GEF file or by the DB4 format which is based on the library already filled by the GEF file information.

GEF file

The GEF file contains the relevant CPT data - the Level, cone penetration resistance - qc, friction, water pressure and friction number. The program identifies the input data and generates the soil profile based on the input data and the interpretation rule.

The soil layers and the interpreted geometry:



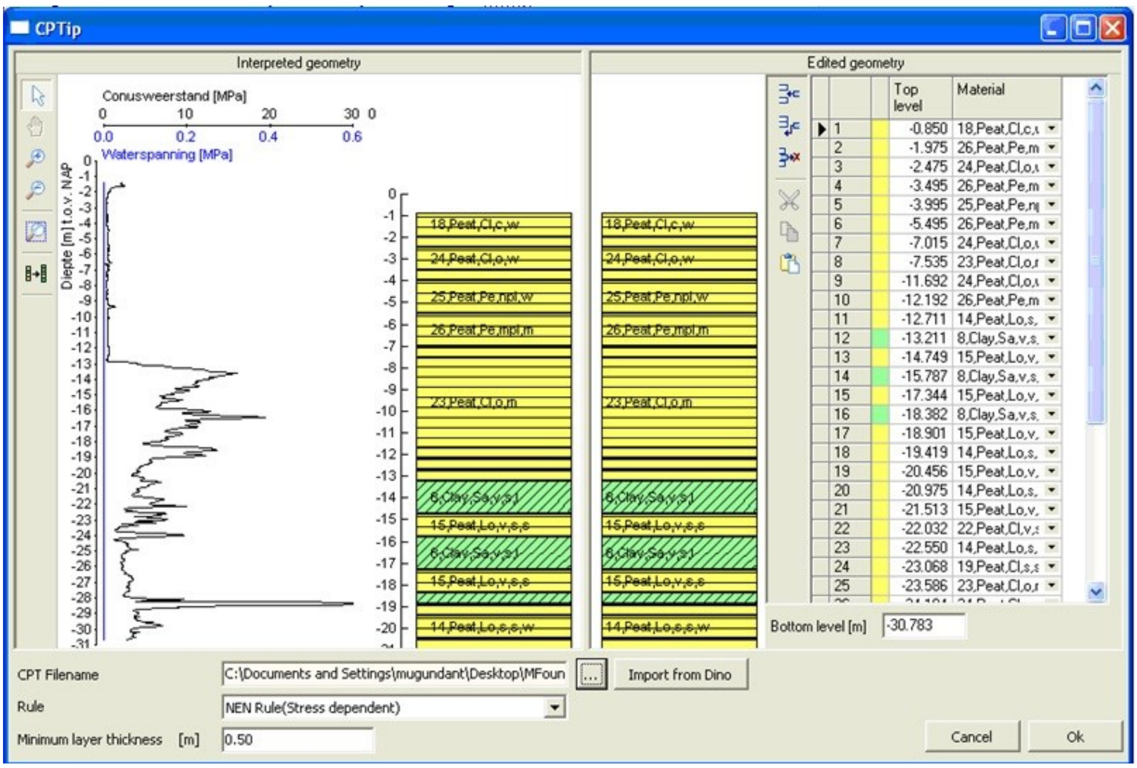
T – penetration test

q_c – cone penetration resistance

z – depth

Inserting the new CPT profile

Adding the new item to the library is done by the dialogue CPTip.



The dialogue has three parts.

1. The part for inserting predefined CPT profile (from DINO) or saved .GEF file.

The user has to select the GEF file using the button or use the button "Import from DINO" and get the GEF file from the map.

This option allows the user to import the GEF files from the map online. The available CPT's in the region is displayed as green flags. User selects one flag by the mouse click and the CPT profile is automatically loaded to the CPTip dialogue.

The interpretation is based on the NEN rule (Stress Dependent) which is based on Table 1 – NEN 6740 (see the Table 1 on the next page). The default min layer thickness is 0.5m.

Table 1 – NEN 6740 (page 18):

Table 1 – NEN 6740 (page 18)

Type of soil			Representative mean value for the soil properties											
Name	Admixture	Consistency 1)	γ (kN/m3) 2)	γsat (kN/m3)	qc (Mpa) 3)6)	C´p	C´s	Cc	Ca 5)	Csw	Ey (Mpa) 6)	ϕ´ (°)	c´ (kPa)	f undr (kPa)
Gravel	weak silty	loose	17	18	15	500	-	0,008	0	0,003	75	32,5	-	-
		moderate	18	20	25	1000	-	0,004	0	0,002	125	35	-	-
		fixed/solid	19 or 20	20 or 21	30	1200 or 1400	-	0,003 or 0,002	0	0,001 or 0	150 or 200	37,5 or 40	-	-
	strong silty	loose	18	20	10	400	-	0,009	0	0,003	50	30	-	-
		moderate	19	21	15	600	-	0,006	0	0,002	75	32,5	-	-
		fixed/solid	20 or 21	22 or 22.5	25	100 or 1500	-	0,003 or 0,002	0	0,001 or 0	125 or 150	35 or 40	-	-
Sand	clean	loose	17	19	5	200	-	0,021	0	0,007	25	30	-	-
		moderate	18	20	15	600	-	0,006	0	0,003	75	32,5	-	-
		fixed/solid	19 or 20	21 or 22	25	1000 or 1500	-	0,003 or 0,002	0	0,001 or 0	125 or 150	35 or 40	-	-
	weak silty clayey		18 or 19	20 or 21	5 or 20	450 or 650	-	0,008 or 0,005	0	0,003 or 0,001	25 or 35	27 or 32,5	-	-
	strong silty clayey		18 or 19	20 or 21	2 or 15	200 or 400	-	0,019 or 0,009	0	0,006 or 0,001	20 or 30	25 or 30	-	-
Loam 4)	weak sandy	weak	-	19	1	25	650	0,168	0,004	0,056	2	27,5 or 30	0	50
		moderate	-	20	2	45	1300	0,084	0,002	0,028	5	27,5 or 32,5	2	100
		fixed/solid	-	21 or 22	3	70 or 100	1900 or 2500	0,049 or 0,030	0,001	0,017 or 0,005	10 or 20	27,5 or 35	5 or 7,5	200 or 300
	strong sandy		-	19 or 20	2	45 or 70	1300 or 2000	0,092 or 0,055	0,002	0,031 or 0,005	5 or 10	27,5 or 35	0 or 2	50 or 100
Clay	clean	weak	-	14	0,5	7	80	1,357	0,013	0,452	1	17,5	0	25
		moderate	-	17	1	15	160	0,362	0,006	0,121	2	17,5	10	50
		fixed/solid	-	19 or 20	2	25 or 30	320 or 500	0,168 or 0,126	0,004	0,056 or 0,042	4 or 10	17,5 or 25	25 or 30	100 or 200
	weak sandy	weak	-	15	0,7	10	110	0,759	0,009	0,253	1,5	22,5	0	40
		moderate	-	18	1,5	20	240	0,237	0,005	0,079	3	22,5	10	80
		fixed/solid	-	20 or 21	2,5	30 or 50	400 or 600	0,126 or 0,069	0,003	0,042 or 0,014	5 or 10	22,5 or 27,5	25 or 30	120 or 170
	strong sandy		-	18 or 20	1	25 or 140	320 or 1680	0,190 or 0,027	0,004	0,063 or 0,025	2 or 5	27,5 or 32,5	0 or 2	0 or 10
	organic	weak	-	13	0,2	7,5	30	1,69	0,015	0,55	0,5	15	0 or 2	10
		moderate	-	15 or 16	0,5	10 or 15	40 or 60	0,760 or 0,420	0,012	0,25 or 0,14	1 or 2	15	0 or 2	25 or 30
Peat	not preloaded	weak	-	10 or 12	0,1 or 0,2	5 or 7,5	20 or 30	7,59 or 1,81	0,023	2,530 or 0,6	0,2 or 0,5	15	2 or 5	10 or 20
	moderate loaded	moderate	-	12 or 13	0,2	7,5 or 10	30 or 40	1,81 or 0,900	0,016	0,6 or 0,3	0,5 or 1	15	5 or 10	20 or 30
Variation coeficient			0,05		-	0,25						0,1	0,2	

Remarks to the table:

The table gives the low representative value of the mean values of the concerning soil type. Within a range, defined by the row of the admixture and the column of the parameter (a 'cell'), following holds:

- for γ, γ_{sat}, C_p, C_s, E, ϕ´, c´ and f_{undr}: if an increase of the value leads to a unfavorable situation (higher dimensions foundation), than the right value on the same row are used, or, if there is no right value mentioned, the value of the row under it;

- for C_c, C_a a C_{sw} if a reduction of the value leads to an unfavorable situation, then the right value on the same row should be used, or if there is no right value, then the value on the row under it.

1) Loose $0<R_n<0{,}33$

Moderate $0{,}33 \leq R_n \leq 0{,}67$

Solid $0{,}6<R_n<1$

2) at natural humidity level

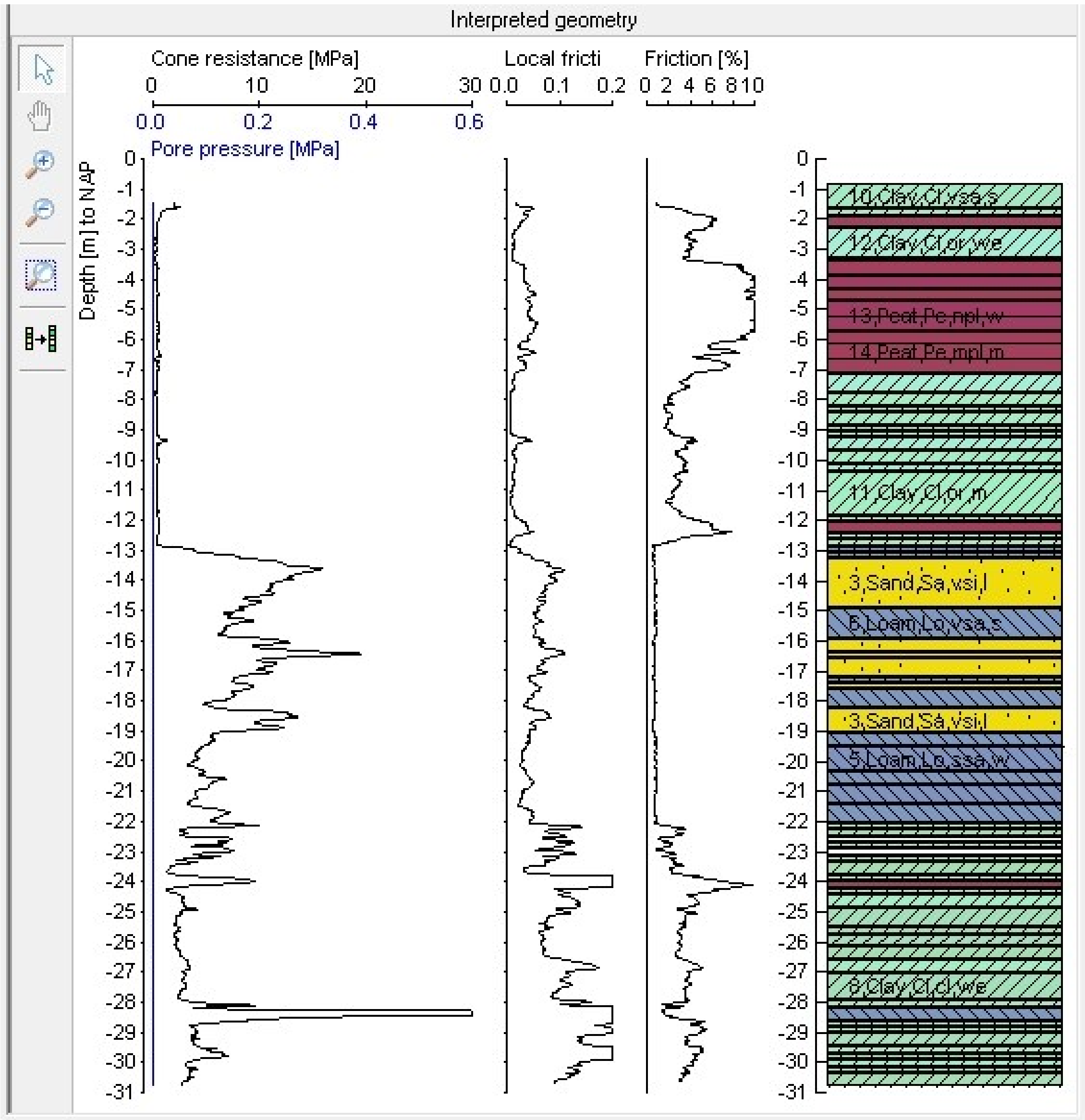
3) here given q_e-values (cone resistance) serve as entry in the table, and may not be used in the calculations

4) calculated is the saturated loam

5) C_a-values are valid for a stress increasing trajectory of max 100%

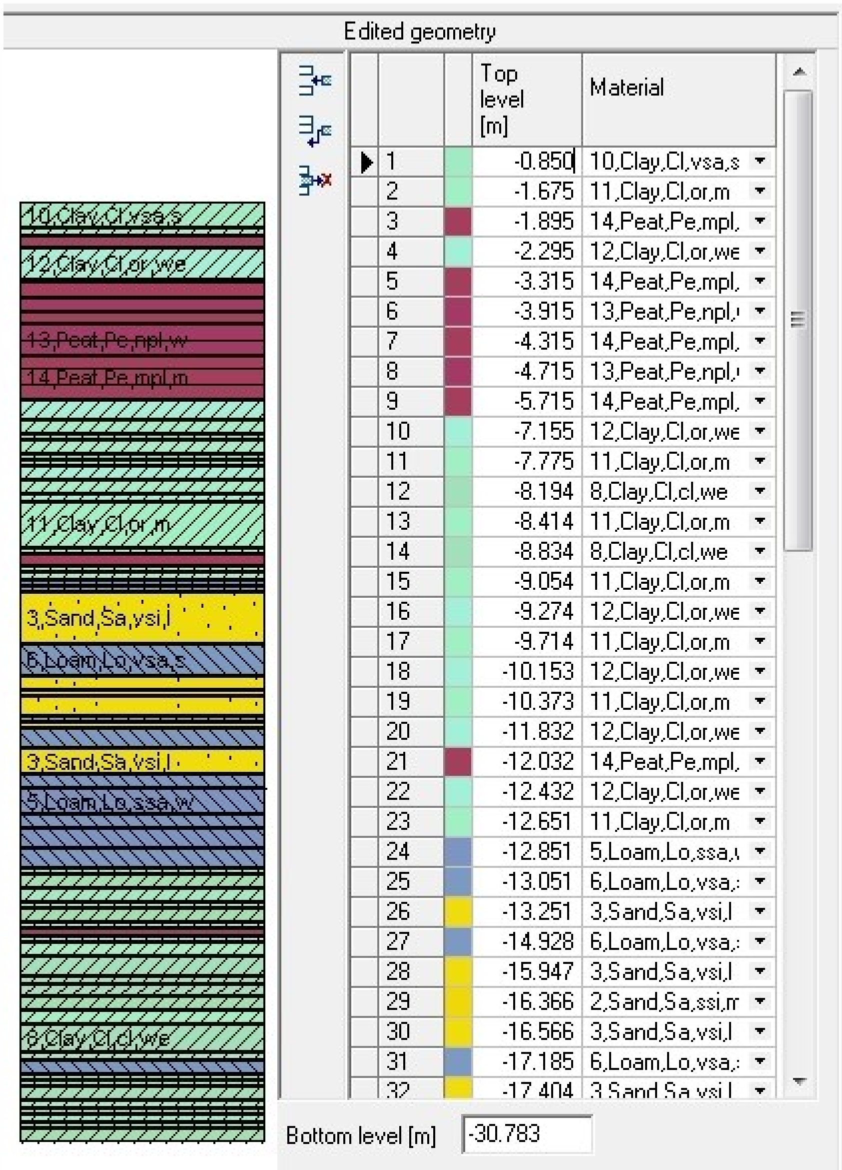
6) q_c and E are standardized on an effective vertical stress of 100 kPa

2. The part with the graphical representation of the imported CPT file – Interpreted geometry.



Data from GEF file are displayed here – q_c, water level, friction zone, layers, thickness and all data for soil.

3. And the second part contains the profile which is used in the Scia Engineer. It can be edited by the grid – changing the layer level, thickness or the material.



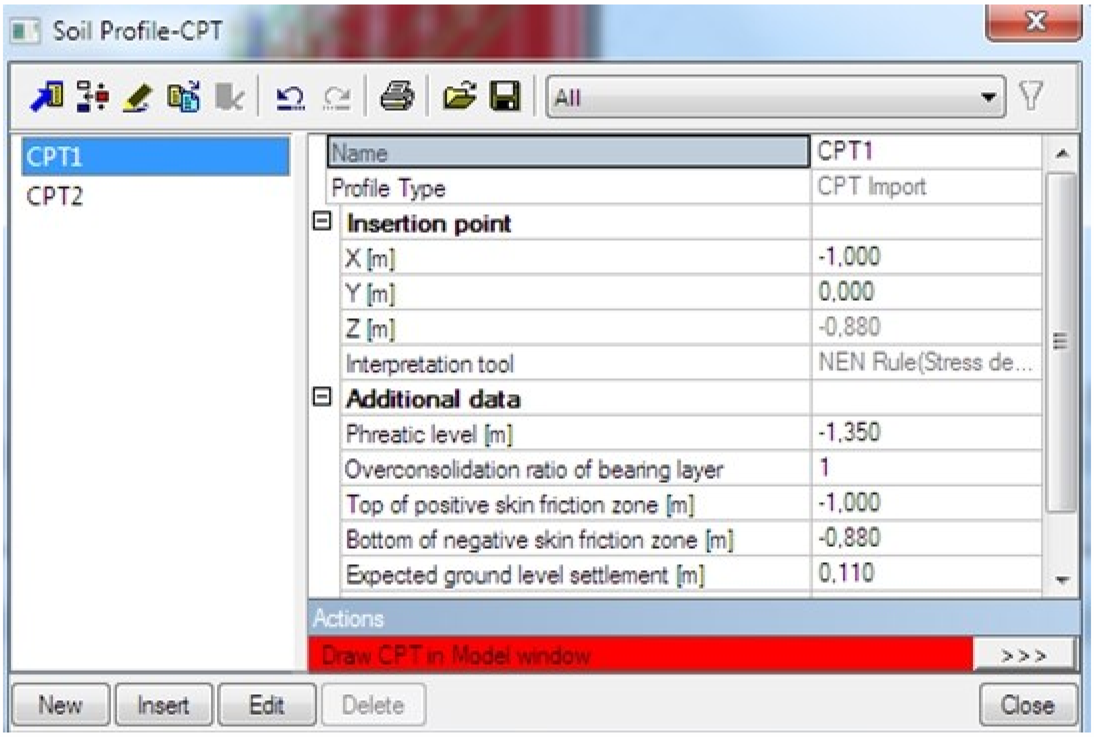
	Top level [m]	Material
1	-0.850	10,Clay,Cl,vsa,s
2	-1.675	11,Clay,Cl,or,m
3	-1.895	14,Peat,Pe,mpl,
4	-2.295	12,Clay,Cl,or,we
5	-3.315	14,Peat,Pe,mpl,
6	-3.915	13,Peat,Pe,npl,
7	-4.315	14,Peat,Pe,mpl,
8	-4.715	13,Peat,Pe,npl,
9	-5.715	14,Peat,Pe,mpl,
10	-7.155	12,Clay,Cl,or,we
11	-7.775	11,Clay,Cl,or,m
12	-8.194	8,Clay,Cl,cl,we
13	-8.414	11,Clay,Cl,or,m
14	-8.834	8,Clay,Cl,cl,we
15	-9.054	11,Clay,Cl,or,m
16	-9.274	12,Clay,Cl,or,we
17	-9.714	11,Clay,Cl,or,m
18	-10.153	12,Clay,Cl,or,we
19	-10.373	11,Clay,Cl,or,m
20	-11.832	12,Clay,Cl,or,we
21	-12.032	14,Peat,Pe,mpl,
22	-12.432	12,Clay,Cl,or,we
23	-12.651	11,Clay,Cl,or,m
24	-12.851	5,Loam,Lo,ssa,
25	-13.051	6,Loam,Lo,vsa,
26	-13.251	3,Sand,Sa,vsi,l
27	-14.928	6,Loam,Lo,vsa,
28	-15.947	3,Sand,Sa,vsi,l
29	-16.366	2,Sand,Sa,ssi,m
30	-16.566	3,Sand,Sa,vsi,l
31	-17.185	6,Loam,Lo,vsa,
32	-17.404	3,Sand,Sa,vsi,l

The soils are defined by the NEN model. They are saved to the soil library database and the user is able to define the new soils there.

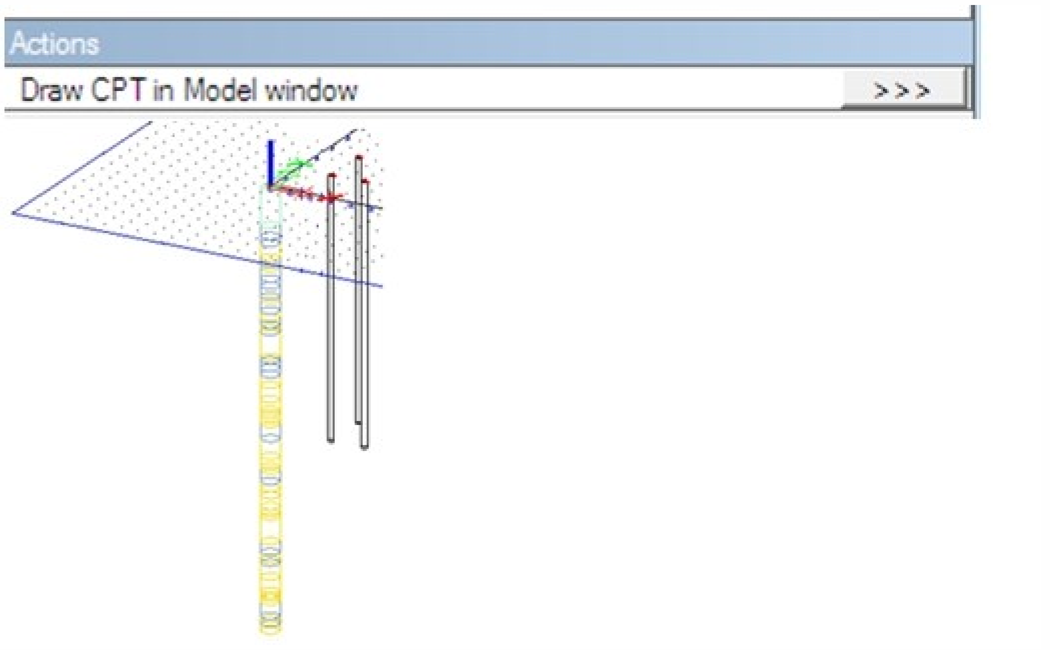
The interpreted geometry can be retrieved by the option "Copy interpreted geometry to edited geometry".

The edited geometry will be used as an input for the pile plan design and verification. The CPT object is represented graphically in the 3D model as a borehole object in Scia Engineer.

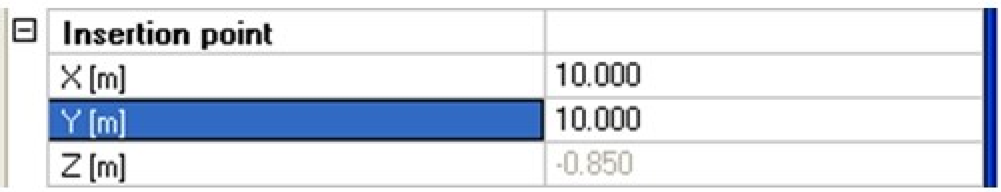
The CPT profile properties:



The user has to use the “Draw CPT in Model window” action button to draw the CPT in the model.



The XY coordinates of the CPT borehole are defined by the user and the top level is predefined by the CPT test. Each soil profile has to have a unique coordinates (XY).



The user has to specify the additional data of the soil profile which are required during the design and verification. Those data are filled automatically to the properties (default values) from the CPT test.

Additional data	
Phreatic level [m]	-5,000
Overconsolidation ratio of bearing l...	1
Top of positive skin friction zone [m]	-1,000
Bottom of negative skin friction zo...	0,000
Expected ground level settlement [...	0,110

Phreatic level (water level)

It is the level between the dry and the wet soil. The number below zero should be defined.

Overconsolidation ratio (ORC)

The value of Overconsolidation ratio of bearing layer determines whether the maximum pile tip resistance has to be reduced due to Overconsolidation or not. Overconsolidation is normally caused by loads which were applied to the bearing layer over a long period of time.

The range and effect of OCR on Pile tip resistance are:

OCR<=2 – No effect on Pile tip resistance

2< OCR < 4 – Maximum Pile tip resistance is reduced by 33%

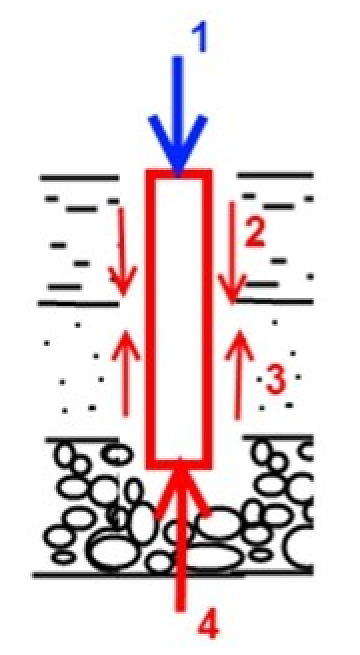
OCR > 4 – Maximum Pile tip resistance is reduced by 50%

Friction zone

The negative skin friction is caused by action of the soil settlement around the pile. The soil formed around the pile is pulling the pile down and reduces its bearing capacity. It decreases the shaft friction and in extreme cases it may annul it. The pile is then supported only by subsoil below the pile tip.

The influence of the friction zone can be eliminated in settlement by the option expected ground level settlement.

The pile shaft can be also protected by providing a protective sleeve or a coating for the section which is surrounded by the settling soil.



1 – load

2 – negative shaft(skin) friction (soft soil layer)

3 – positive shaft friction (firm soil layer)

4 – pile tip bearing capacity

The essential requirements to calculate the positive and negative shaft friction resistance are specified in NEN 6743.

The values for Top of positive skin friction zone and bottom of negative skin friction zone are user-defined.

Positive skin friction zone:

1) pile without widened base ► the bottom is the pile tip level

2) the prefabricated pile with a widened base ► the top of that zone is above the widening (NEN 6743 art. 5.4)

Negative skin friction zone:

► the top of this zone is ground level or excavation level

The rules in the application are fulfilled this way:

1) The bottom of the positive skin friction zone is automatically set to the pile tip level. No user definition is needed.

2) The top of the positive skin friction zone is specified by the user as a level relative to the reference level.

3) The top of the negative skin friction zone is automatically set as the ground level or excavation level. No user definition is needed.

4) The bottom of the negative skin friction zone is specified by the user as a level relative to the reference level.

Expected ground level settlement

The expected ground level settlement determines how the negative skin friction has to be incorporated in the calculations.

1) the value is 0.02m -> negative skin friction is negligible and won't be considered

2) values from 0.02m to 0.10m -> the effect of negative skin friction is directly incorporated into the calculated pile settlement by adding half of the expected ground level settlement to the total pile settlement.

3) values >0.10m -> the maximum forces due to negative skin friction are calculated. These forces are then used to determine the negative skin friction on the pile settlement.

NEN Rule (Stress dependent)

The NEN rule is considered to be a more common and is used for interpretation of the soil profile.

This rule uses 14 different areas and is based on the Dutch standard NEN 6740 (see the table on the page 5). Each area describes certain soil types by defining the relationship between CPT resistance and friction ratio. The friction ratio is defined as the shear resistance in percentage of the cone resistance.

The soils (by the NEN rule) and its properties are defined in a soil library.

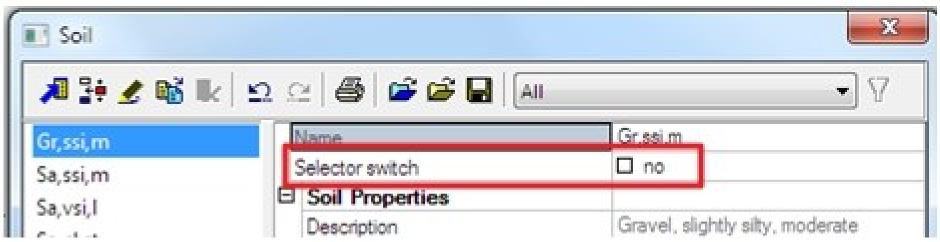
The soil types are:

Gravel, slightly silty, moderate
Sand, slightly silty, moderate
Loam, very sandy, stiff
Clay, very sandy, stiff
Clay, clean, stiff
Clay, organ, moderate
Peat, moderately preloaded, moderate
Sand, clean, stiff
Sand, very silty, loose
Loam, slightly sandy, weak
Clay, slightly sandy, moderate
Clay, clean, weak
Clay, organ, weak
Peat, not preloaded, weak

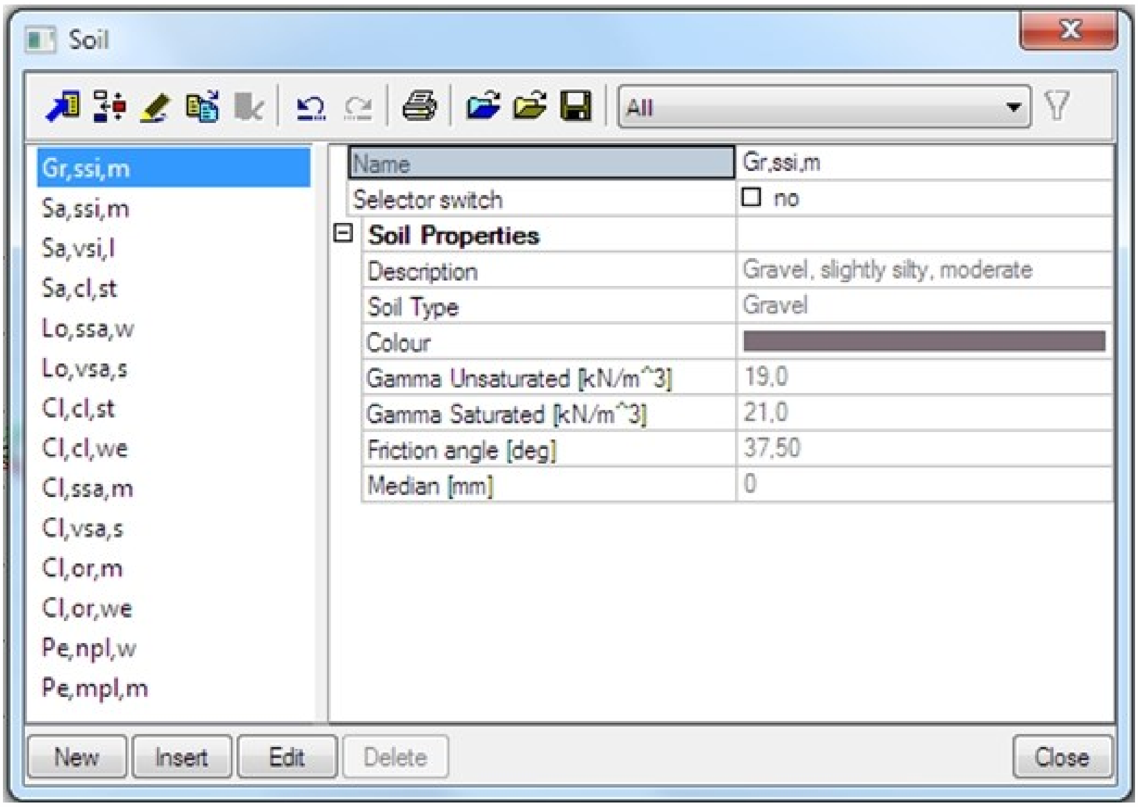
Soil library

The soil library is a standard library in the Scia Engineer. This library is added under the subsoil functionality. The database of soils used by the NEN model is created in a DB4 format and it is loaded automatically.

The soil material and its properties are specified here. The soil properties are non-editable by default. The checkbox “Selector switch” enables editing them.



The program allows the user to define the new soils. The user defined soils can be used in the interpreted geometry of the soil profile.



Description
Description of the Soil

Soil type
It is Gravel, Sand, Loam, Clay or Peat.

This type must be selected in order to perform a correct calculation.

Gamma unsaturated
Dry unit weight of the soil

Gamma saturated
Saturated unit weight of the soil

Friction angle
Angle of internal friction for the soil – The value must be between 0 - 90 degrees

Median
This property applies to soil type's sand and gravel. The size of median will influence the value of a_S which is used to determine the positive skin friction. For sand with median >0.6mm, the values of a_S will be reduced by 25% and for gravel with median >2mm a_S will be reduced by 50%.

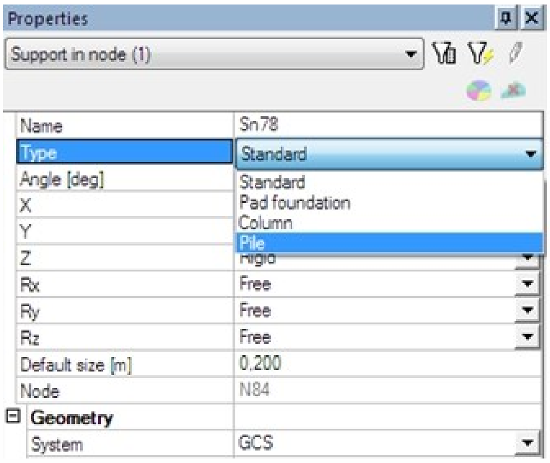
Definition of a new pile

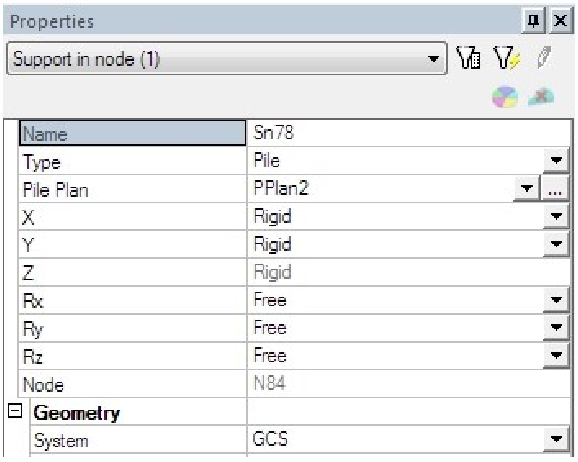
Nodal support - Pile

A nodal support type "Pile" is available when the functionality Subsoil and Pile design is checked. It can be added as a standard point support on node.

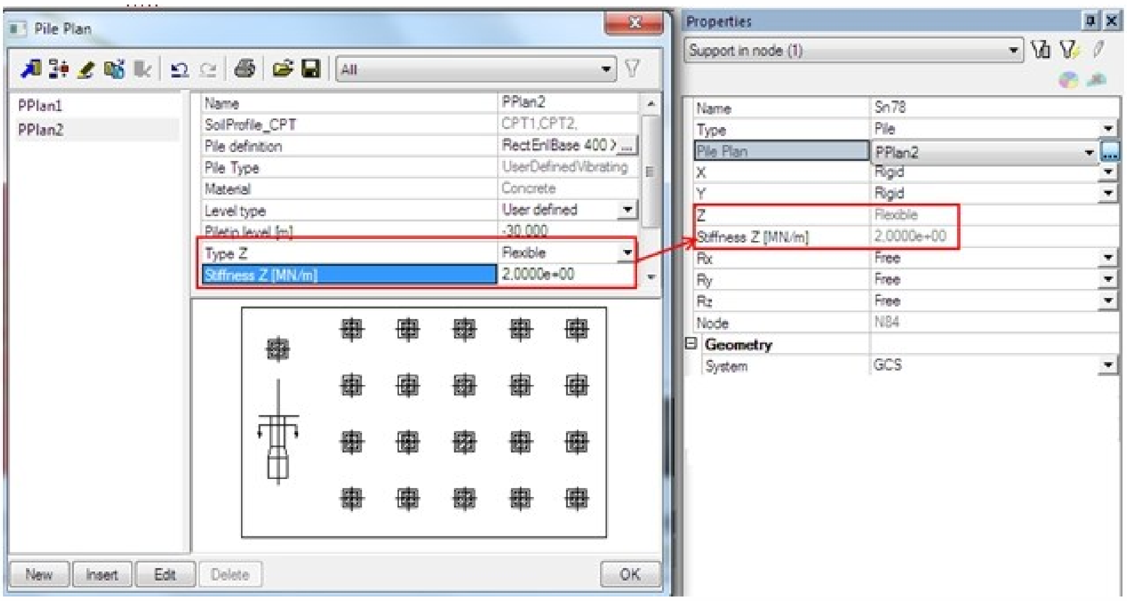
The pile plan library defines properties for all piles in the project. The pile plan is displayed in the properties of the pile. There can be more pile planes specified in the project but each pile must be defined by one pile plan.

The rest of the properties of the pile are standard properties of the support. The support is a hinge i.e. Rx, Ry and Rz are fixed, Mx, My and Mz are free by default.



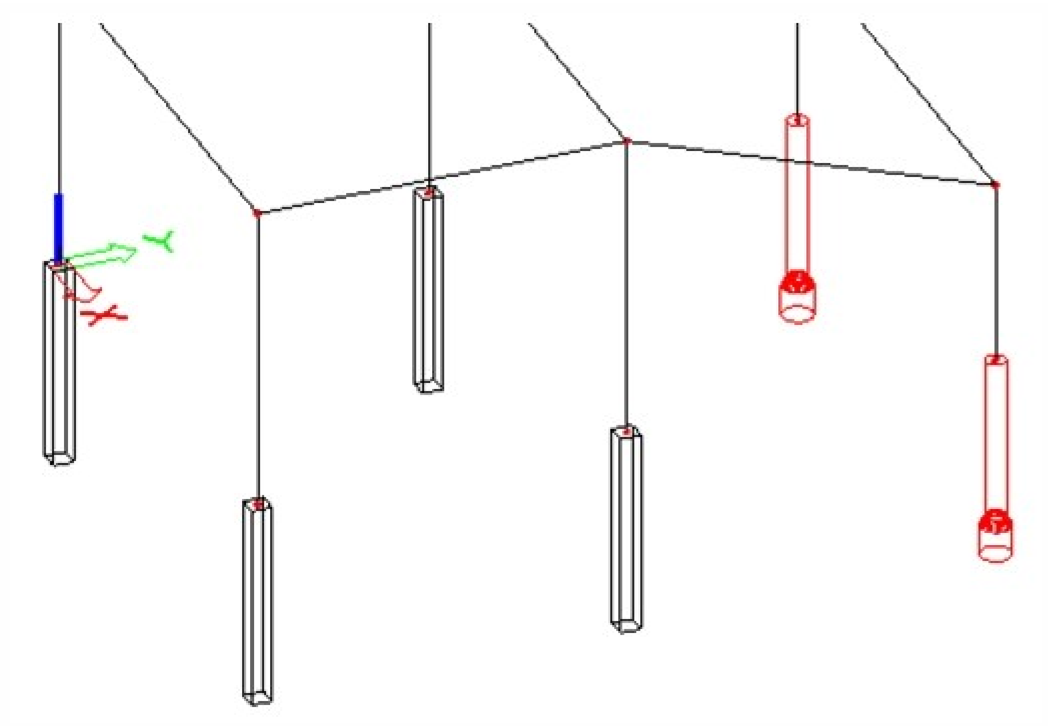


The type Rz is editable only in the Pile plan library and it is directly loaded to the pile properties.



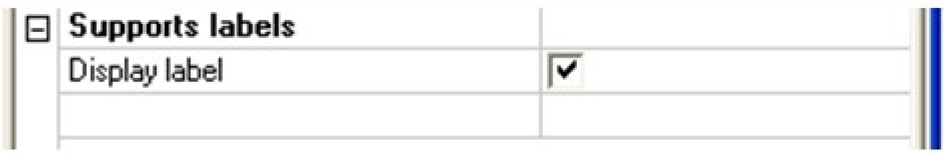
In case Rz is negative (the pile is in tension), no check is executed for the pile and a warning is displayed.

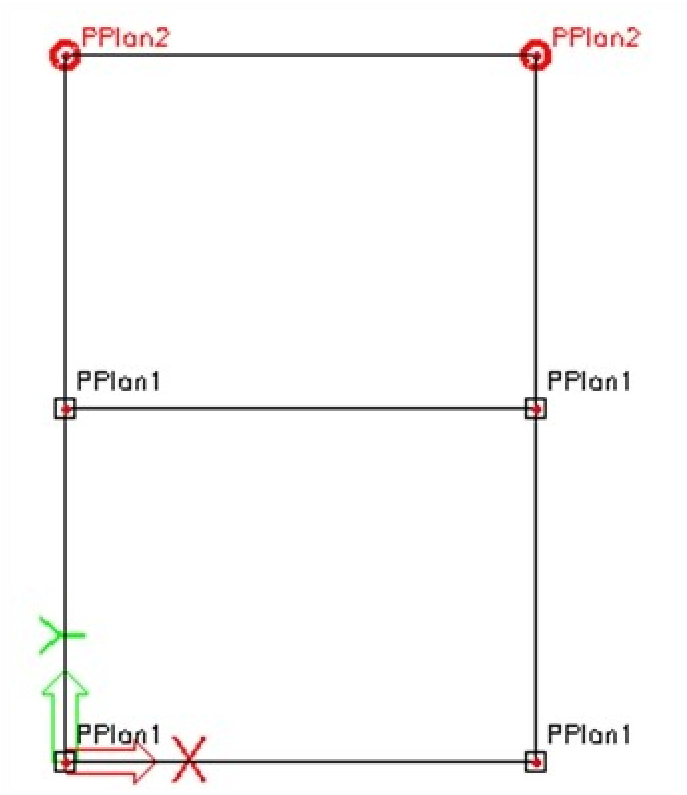
The supports are represented in the Model as shown below. The example contain two different pile plan marked by a different colour.



The colour of piles is controlled by the view flag parameters under the tab “Model” and it is defined in the Pile plan library. The piles are displayed by the support colour when the option is unchecked.

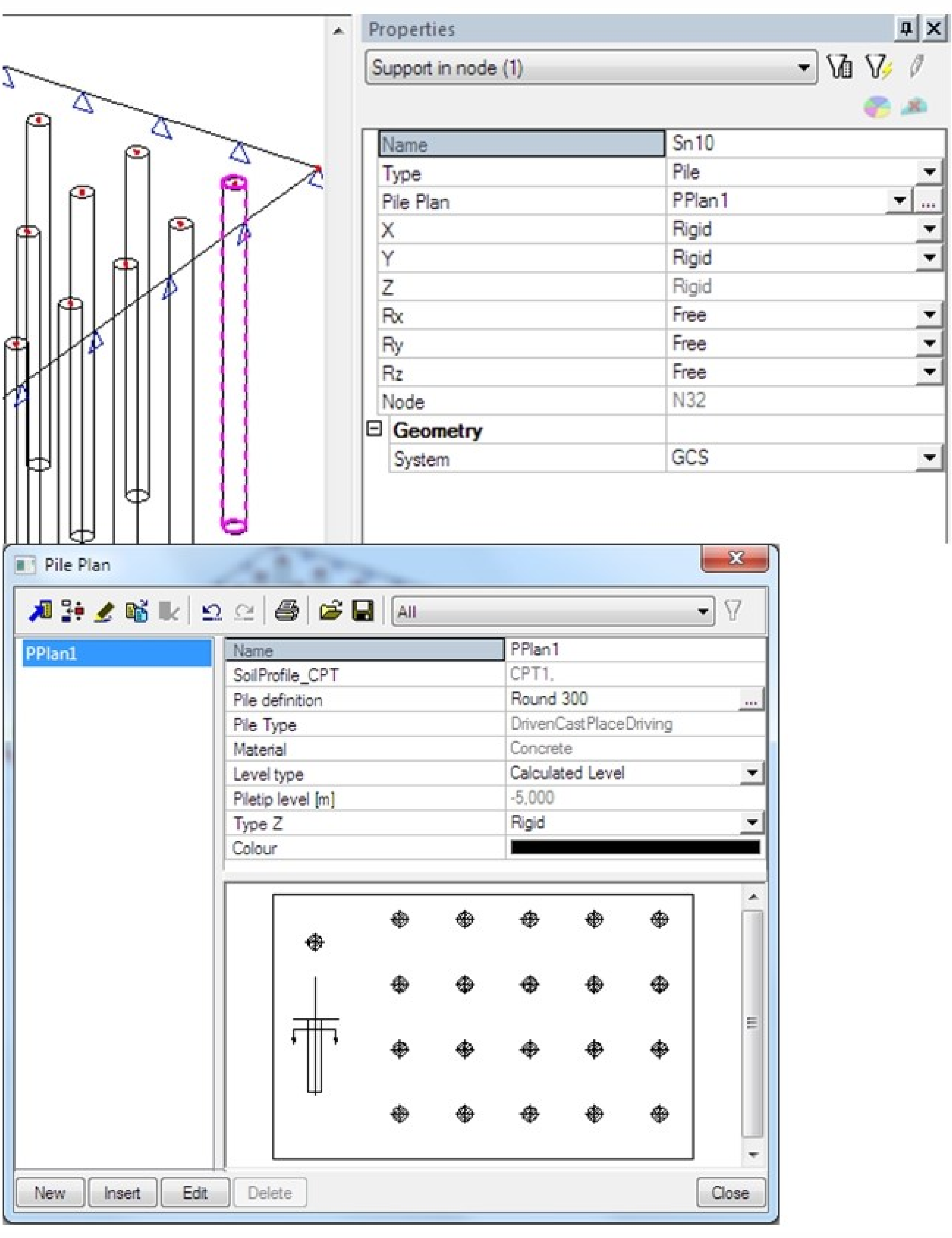
The display of pile labels for the supports is also controlled by the view parameters. The flag contains the pile plan name.





Pile Plan

The piles are defined in the Pile Plan library. They are created as a nodal support type Pile.

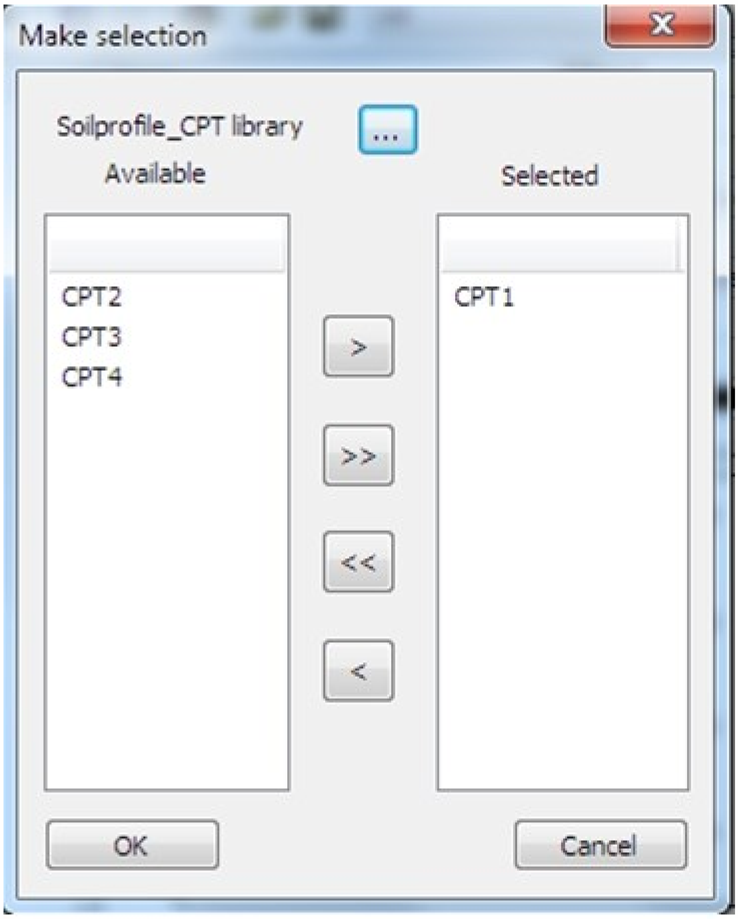


Soil profile CPT

The design and verification of piles is based on the soil profile. The soil profile generated from the CPT data must be associated with the Pile plan. At least one soil profile has to be associated with each pile plan.

If the option “Use all the CPTs for all Pile plans” in the Geotechnics setup is checked, all the soil profiles will be directly associated with all the pile plans.

If this option is unchecked then the selected CPTs are associated to the pile plan. The CPTs can be selected by a standard selection tool. There is also a link button to the soil profile CPT library in the selection dialog.

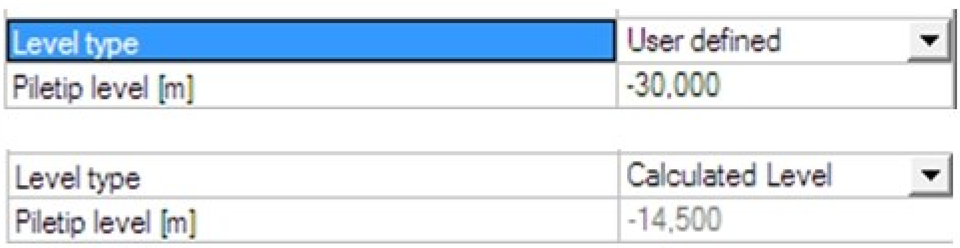


Pile tip level

When the pile plan is defined, the level type is set to “User defined” and the user can specify the pile tip level. This is the initial value of the pile length.

When the calculation and pile design is finished the Level type is changed to “Calculated level” and the Pile tip level is change to the calculated value. This value is non-editable.

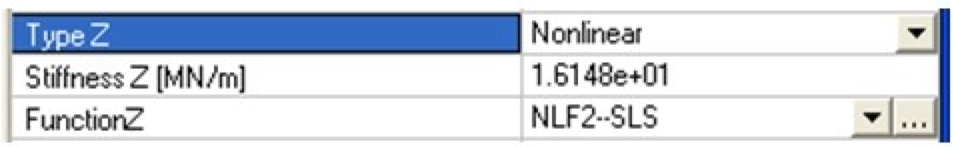
Please note that if the user makes any change in parameters for Pile design (Soil properties, Soil profile and Pile definition), then the pile tip level will be automatically reset to “User defined” and the results are lost. User has to start the Pile design again.



Type Z

The stiffness type for the supports in the Z direction has to be specified. Type rigid or flexible is required for the linear analysis.

Nonlinear is displayed when Non- linear functionality is selected. The Non-linear function library will be associated with this type of the support.



This stiffness can be generated automatically by the action button 'Update stiffness' under the Pileplan verification.

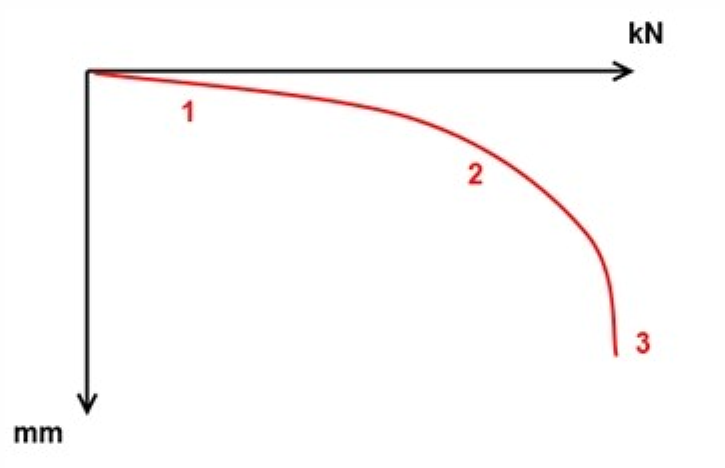
Load settlement curves

It displays the load (kN) and the settlement (mm). It consists from the 3 main parts:

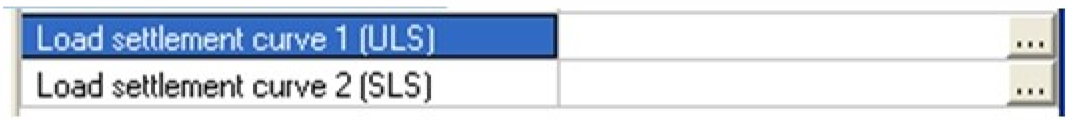
1) Relatively elastic vertical compression – quite vertical part of the graph

2) Local shear failure – the biggest change it the vertical direction

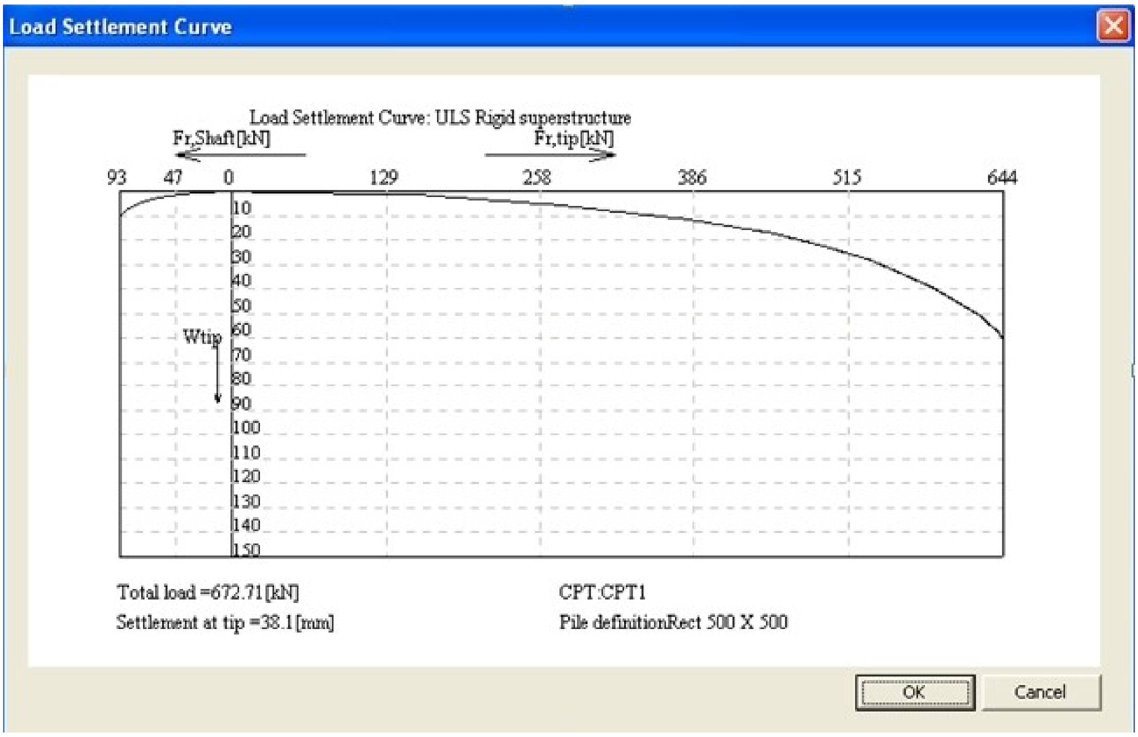
3) General shear failure – position of ultimate load

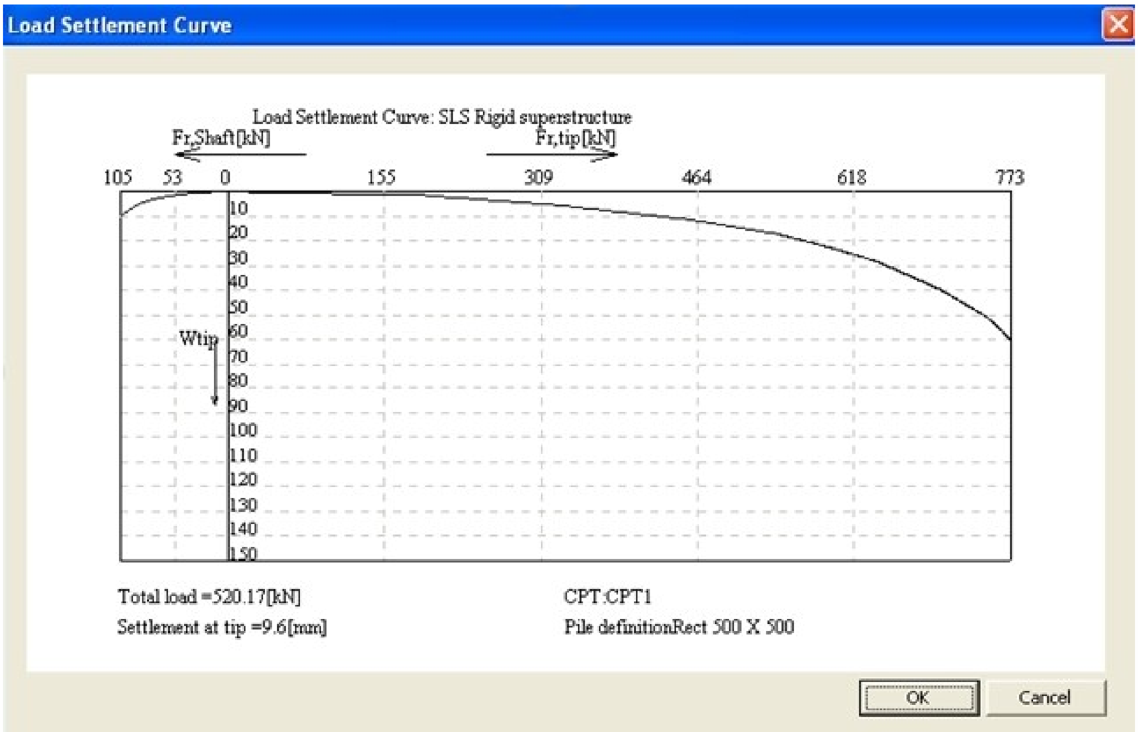


The Load settlement curves are added to the Pile plan library after the Pile verification. It can be displayed by the link button.



Two Load settlement curves are calculated – one for the ultimate state and on one for the serviceability state.





Pile type

The NEN 6743 defines the max pressure on the pile tip:

$$p_{max,tip} = 0,5 \times \alpha_p \times \beta \times s\left(\frac{q_{c,I,m} + q_{c,II,m}}{2} + q_{c,II,m}\right)$$

$q_{c,I,m}$ - mean of the cone tip resistance $q_{c,I}$ (is the mean of the $q_{c;I}$ values over the depth running from the pile base level to a level which is at least 0,7 times and at most 4 times the equivalent pile base diameter D_{eq} deeper)

$q_{c,II,m}$ - mean of the minimum cone tip resistance $q_{c,II}$ (the mean value of the lowest $q_{c;I}$I values over the depth going upwards from the critical depth to the pile base)

$q_{c,II,lm}$ - mean of the cone tip resistance $q_{c,III}$ (the mean value of the $q_{c;III}$ values over a depth interval running from pile base level to a level of 8 times the pile base diameter above the pile base)

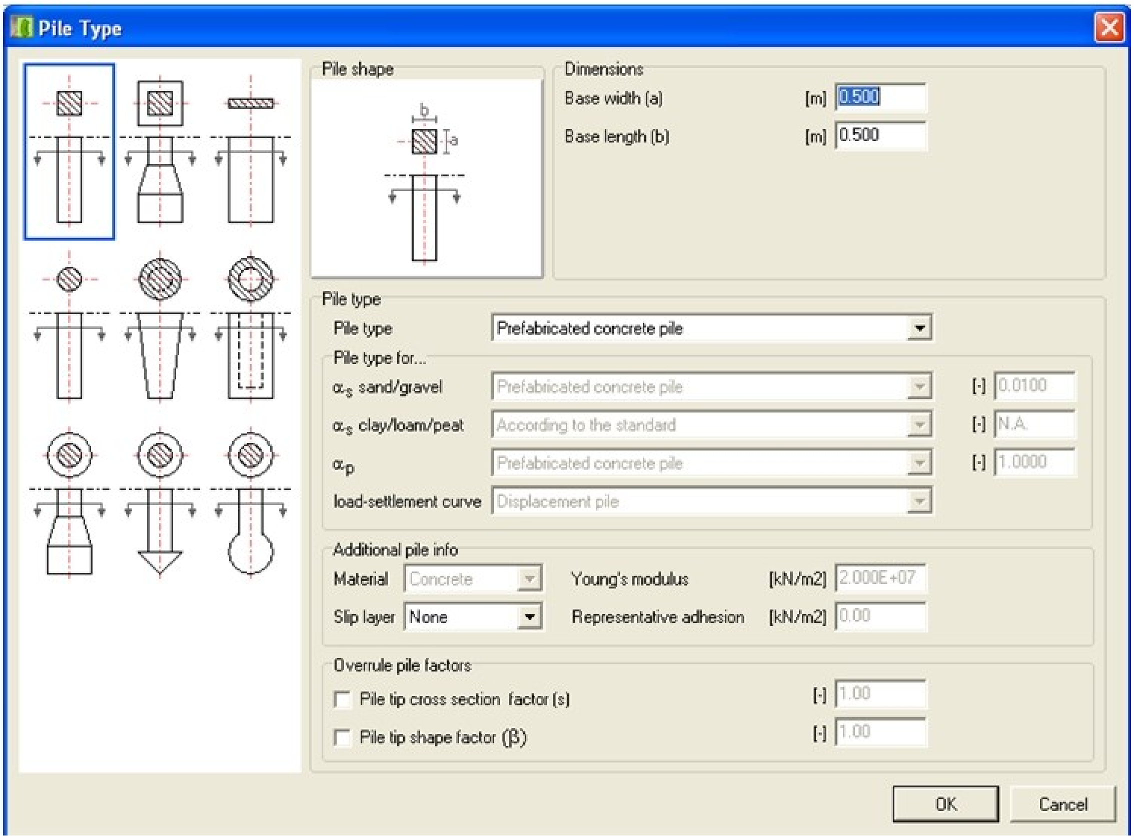
α_p - pile tip coefficient of capacity reduction (is defined by the selected item from the combobox or it can be defined manually)

s – pile tip cross section factor is important mainly for rectangular shape of the pile when the dimensions are not equal, this value can be overruled by the dialogue value (see the specific chapter)

ß – pile tip shape factor represents the influence of an expanded pile tip, its values depends on the pile tip dimensions, this value can be overruled by the dialogue value (see the specific chapter)

The Pile Type dialogue must be filled to define those values.

This dialog is only in English and it is independent on the units settings.



Pile shape
The user can specify the shape of the pile by selecting it on the left part of the dialog. The selected shape will be displayed in the Pile shape dialog.

Dimensions
Based on the shape of the pile, the user has to specify the dimensions for the Pile.

The dimensions have to be specified in “m”, “m/m’ “ or “mm”.

Rectangular Pile => Base width and base length of the Pile

Rectangular Pile (different base) => Width, length and height of base & width and length of shaft

Steel Section => Base width and base length of the Pile

Round pile => Diameter of Pile

Round tapered pile => Diameter at tip and increase in diameter

Round hollow pile => Internal and external diameter of pile

Round enlarged base => Diameter and height of base and Pile diameter

Round lost tip => Diameter and height of base and Pile diameter

Round driven base => Diameter and height of base and Pile diameter

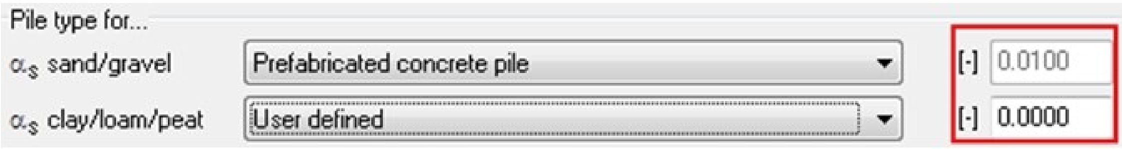
Pile type

Pile type:
There is a combobox with the list of the predefined pile types. User defined value can be defined manually, all other types fill the data automatically and it is non-editable.

Pile type for ...:
This part contains factors for the pile shaft and pile tip.

α_s

It is the pile factor for the shaft friction which reduces the pile tip bearing capacity.



- Cohesive soil (clay, loam, peat) - the value is defined according to NEN 6743-1:2006 and it depends on the soil material.
- Non-cohesive soils (sand, gravel) - α_s depends on the pile type.

Cohesive soils

For cohesive soils (clay, peat, loam) the factor according to the standard is depth-dependent and thus has no single value. The current value box displays N.A. (Not Applicable) as the value cannot be shown as a result.

If “User defined” is selected as the subtype, only the parameter value is entered. That value will be displayed as a current value.

Table 3, NEN 6743:

Soil type	$Q_{C;GEM}$ (MPa)	α_s
Clay	>3	<0,030
Clay	<3	<0,020
Strong sandy loam		Friction Number with a maximum of 0,025
Weak sandy loam		0,025
Peat		0

The friction number should be determined by electric friction cone.

The values for clay have been found in experiments on tubes in Netherlands. It is valid for all clays.

Non-cohesive soils

It can be specified by selecting one of the predefined pile types from the dropdown box. The actual value for α_s will be displayed in the current value box as a result.

If the α_s is set as User defined then the relation between subtype and the pile type doesn't exist. The value is entered by user.

The predefined value for α_s:

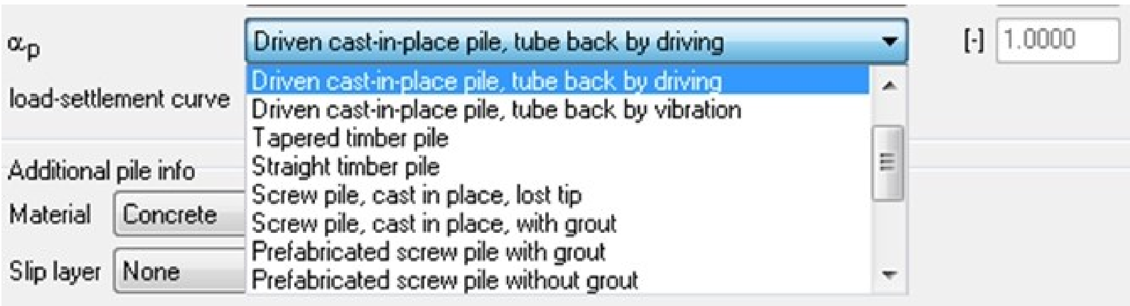
Pile	Pile factor α_p
Prefabricated concrete pile	0,01
Closed-ended steel pipe pile	0,01
Driven cast-in-place pile, tube back by driving	0,014
Driven cast-in-place pile, tube back by vibration	0,012
Tapered timber pile	0,012
Straight timber pile	0,01
Screw pile, cast-in-place, lost tip	0,009
Screw pile, cast-in-place, with grout	0,009
Prefabricated screw pile with grout	0,009
Prefabricated screw pile without grout	0,006
Steel section	0,006
Continuous Flight Auger (CFA) pile	0,006
Bored piles (drilling mud, uncased borehole)	0,006
Bored piles (shelling tech., permanent casing)	0,005

The user defined value has the following consequences on the calculation:

1. The value α_s valid for sand and gravel layers will not be adjusted for any instance of coarse grain (see the Table 3, NEN 6743)
2. The exception for the determination of the pile tip shape factor ß cannot be met because it is impossible to determine that a cast-in place pile with a regained steel driving tube is applied (see the chapter about the pile tip factor ß).
3. The check on length of positive skin friction zone cannot be performed when a weighted tip is applied. It cannot be determined that a pre-fabricated pile is used.

α_p

α_p is the pile factor for the pile tip. α_p value depends on the pile type. Therefore it can be specified by selecting one of the standard pile types from the combo box.

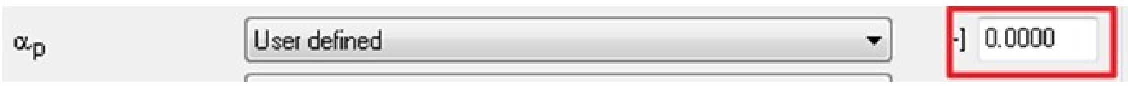


The actual value for α_p is displayed in the current value box.

The predefined value for α_p:

Pile	Pile factor α_p
Prefabricated concrete pile	1
Closed-ended steel pipe pile	1
Driven cast-in-place pile, tube back by driving	1
Driven cast-in-place pile, tube back by vibration	1
Tapered timber pile	1
Straight timber pile	1
Screw pile, cast-in-place, lost tip	0,9
Screw pile, cast-in-place, with grout	0,9
Prefabricated screw pile with grout	0,8
Prefabricated screw pile without grout	0,8
Steel section	1
Continuous Flight Auger (CFA) pile	0,8
Bored piles (drilling mud, uncased borehole)	0,5
Bored piles (shelling tech., permanent casing)	0,5

The value can be set as "User defined" type. In this case the exception for "Continuous flight auger" piles cannot be taken into account for the reduction of q_c-values when determining $q_{c;III;mean}$.



The reason for this is that it cannot be determined when a continuous flight auger pile is used.

Maximum values of α_p and α_s:

Pile class or type	α_p	α_s (fine to coarse sand)
Soil displacement type, diameter >150mm		
Driven fabricated piles	1	0,010
Cast in place (driving steel tube)	1	0,012
Soil replacement type, diameter >150mm		
Flight auger piles	0,8	0,006
Bored piles (drilling mud)	0,6	0,005

For very coarse sand a reduction factor 0,75 is needed.

For gravel a reduction factor 0,5 is needed.

The value α_s=0,006 may be raised to 0,01 when CPTs are used in vicinity of the flight auger piles.

Load-Settlement curves:

Load-Settlement curves contain only lines for three subtypes (according to the code):

1. Displacement pile
2. Continuous flight auger pile
3. Bored pile

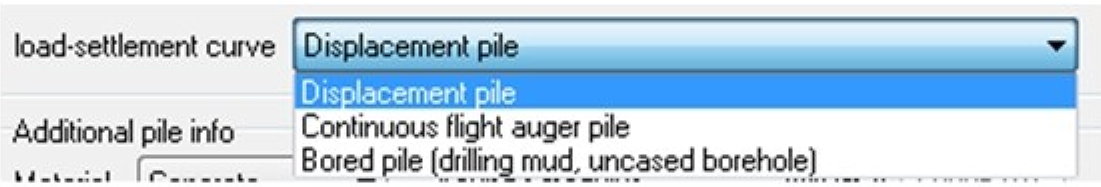


Figure 6, NEN 6743 - Relationship between the force on the pile tip ($F_{r,punt,i}$) in% of maximum force on the pile tip ($F_{r,max,punt,i}$) the ultimate limit state or serviceability limit state and the subsidence the point ($w_{punt,i}$), in% of D_{eq}:

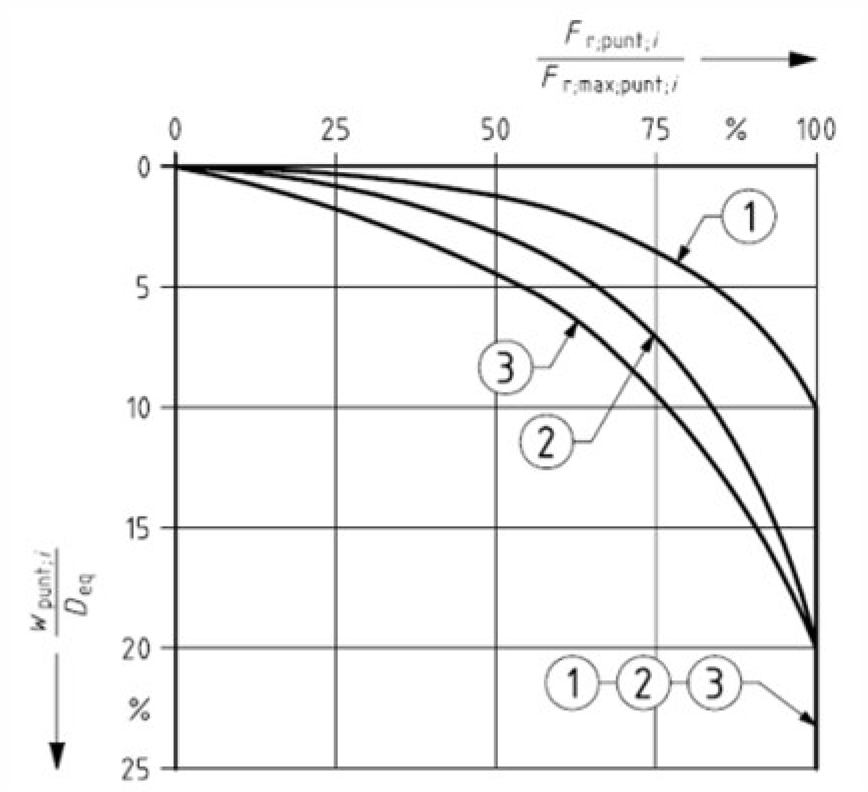
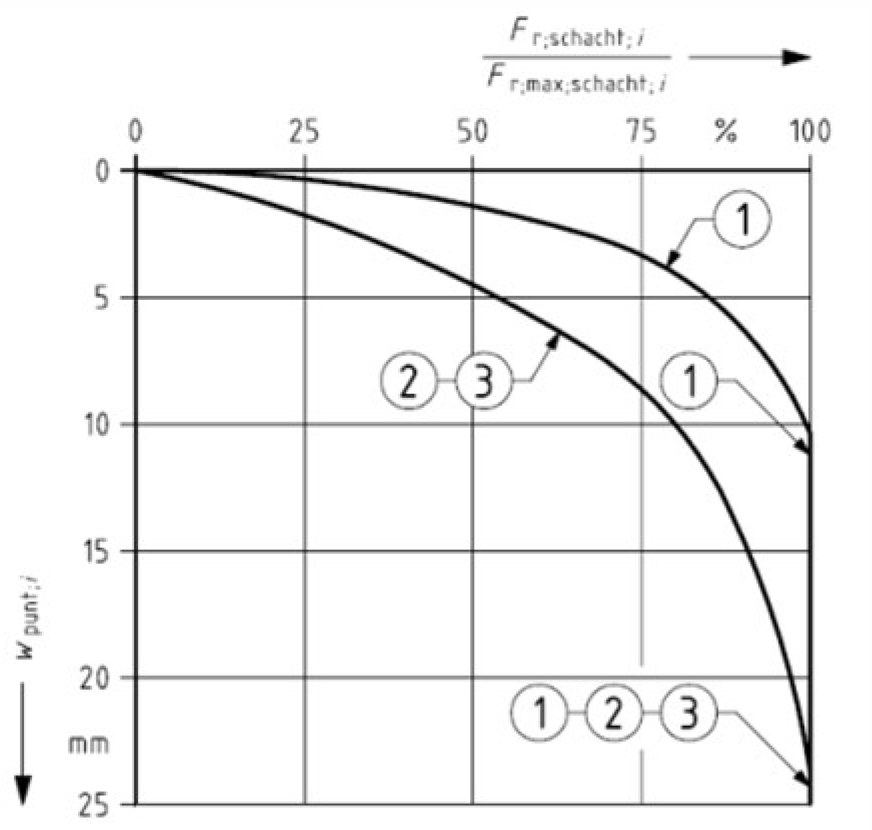


Figure 7, NEN 6743 - Relationship between the shear force on the pile shaft ($F_{r,schacht,i}$) as % of maximum shear force on the pile shaft ($F_{r,max;schacht\ i}$) the ultimate limit state or the serviceability limit state and the subsidence of the point ($w_{punt,i}$) in mm:



Additional pile info:

Material
The material of the user defined pile is selected here.

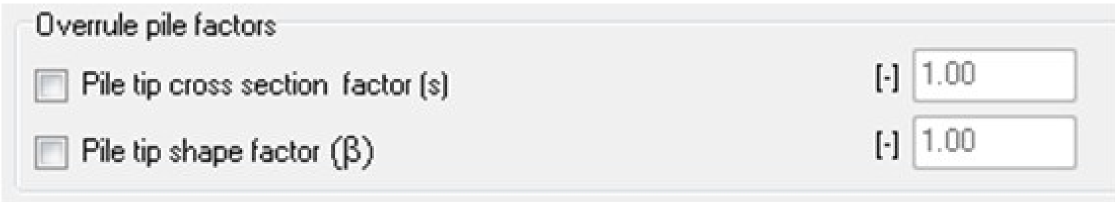
Young's modulus
The corresponding elasticity modulus is provided automatically for concrete, steel and timber and cannot be edited. The Young's modulus must be specified if the material is User defined.

Slip layer
The slip layer for the pile has been specified

Representative adhesion
The corresponding representative adhesion is provided. It can be edited when the User defined option is selected.

Overrule pile factors:

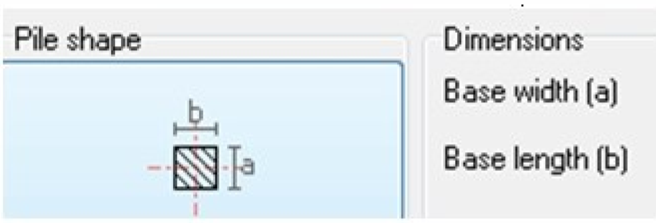
The cross section factor and pile tip shape factor can be overruled by user defined values. The values are editable when the option is checked.



Pile tip cross section factor s

Coefficient of an influence of a rectangular pile

The values "a" and "b" come from pile sizes.



Pile tip shape factor ß

Coefficient derived from the pile tip sizes, it is used if the pile tip is expanded.

H – length of the pile tip

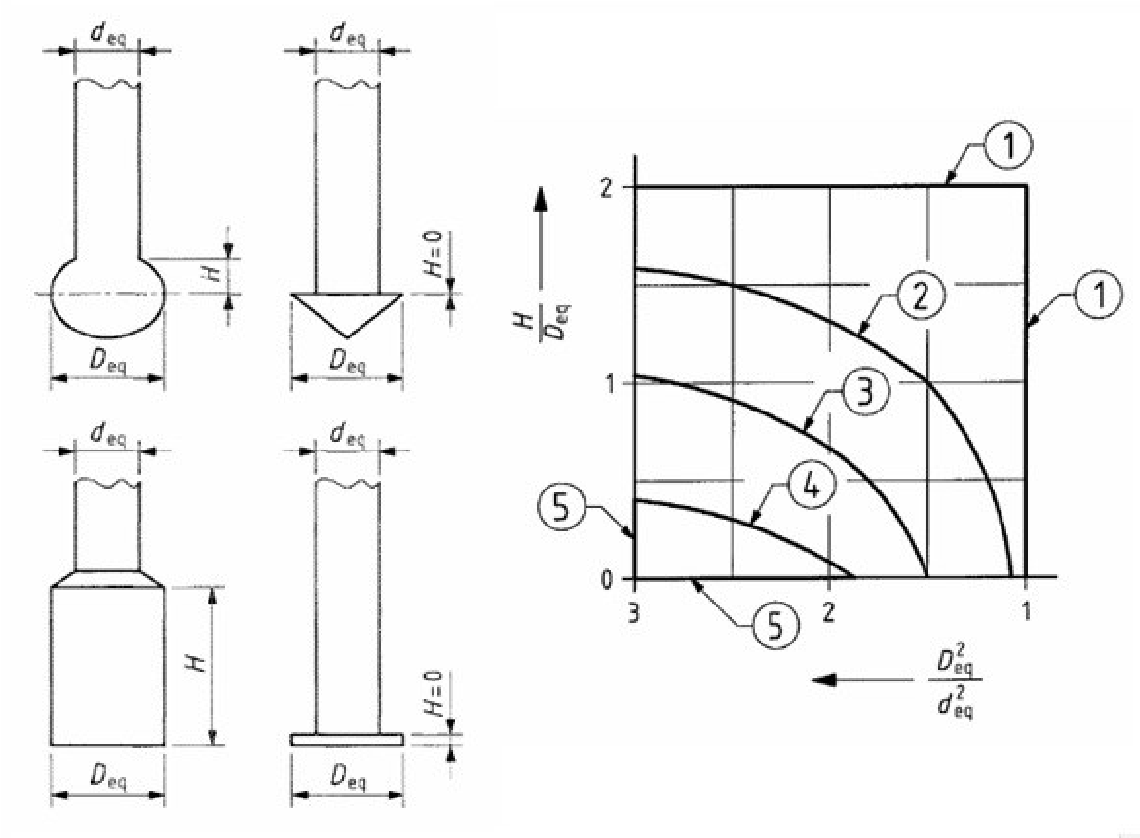
D_{eq} – width of the pile tip

d_{eq} – width of the pile shaft

ß=L/B

It is a shape factor for non-circular or non square pile base cross section.

ß = 1,0 for cylindrical piles



Borderline 1: ß=1,0

Borderline 2: ß=0,9

Borderline 3: ß=0,8

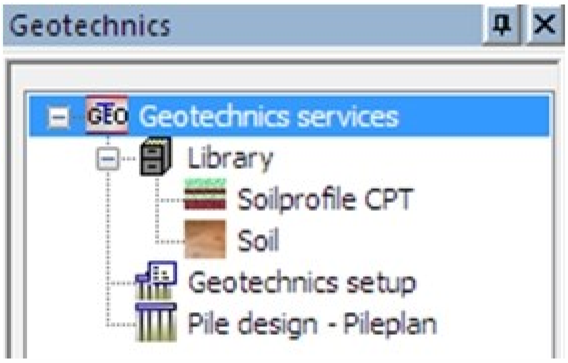
Borderline 4: ß=0,7

Borderline 5: ß=0,6

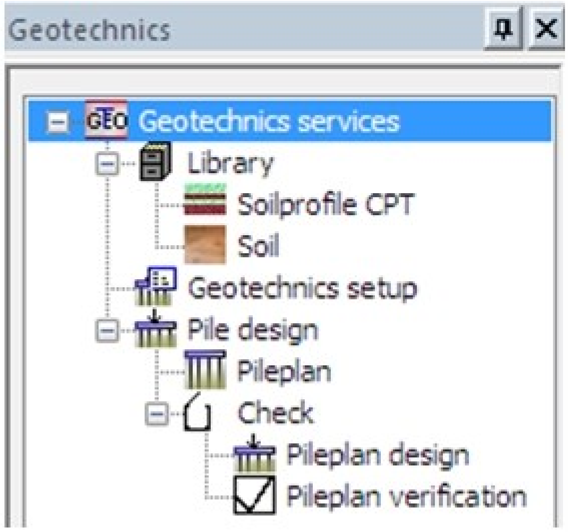
Check – design, verification

After performing the analysis, the pile plan verification and pile plan design command will be enabled in the "Geotechnics" service.

Before the calculation:



After the calculation



Conditions for design

The option is available for NEN and EC-EN nation code.

This option enables the user to calculate the pile tip level. The design and verification of bearing piles are based on the guidelines given by NEN 6740,NEN 6743 and NEN 9997-1:2009 (EC7).

1. The design is performed only for bearing piles which are subjected to static or quasi static loads which cause compressive forces in the piles.
2. The calculation of pile forces and pile displacements are based on cone penetration test (CPT). The pile group effect is not considered.
3. Tension in piles and horizontal displacements of piles are not taken into account. These limitations are explicitly mentioned in the results and the document tables.
4. The program evaluates the pile tip level where the net bearing capacity of the pile is equal to or more than the maximum load on the pile.

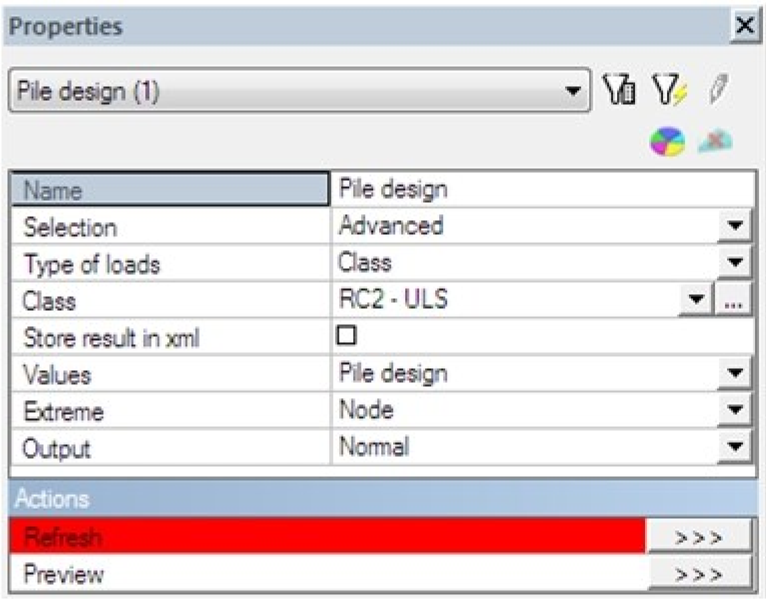
5. The pile design can be evaluated when the linear analysis is done (there are proper definition of the analysis model, sufficient number of piles, correct definition of boreholes and calculated reaction of the supports).

Conditions for verification

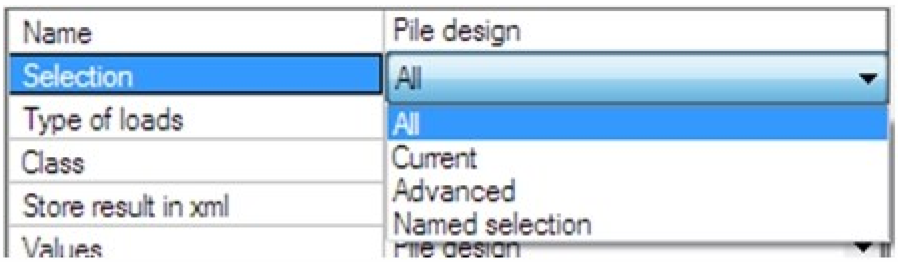
The verification of bearing piles is performed based on the guidelines given by NEN 6740/ NEN 6743 and NEN 9997-1:2009 (EC7). This option is limited to NEN and EC-EN standards.

1. Only bearing piles which are subjected to static or quasi static loads that cause compressive forces in the piles can be verified. Any possibility of tension in piles and horizontal displacements of piles are not taken into account. The pile tip must be calculated.
2. The calculation of pile forces and pile displacements are based on cone penetration test.
3. The entire pile plan is considered during verification and the group effects are considered.
4. The calculation of bearing capacity, settlement and negative skin friction are performed and the result of the verification is a Load settlement curve from ULS and SLS.

The properties of the Pile design



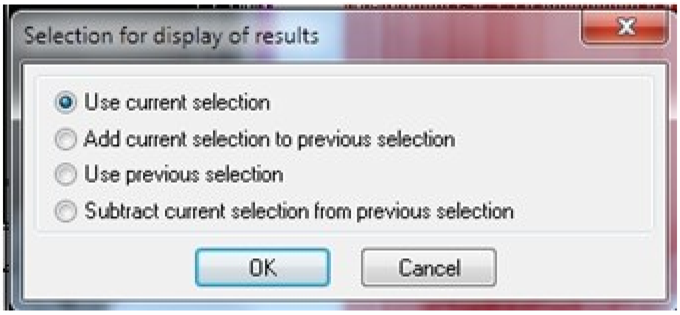
Selection
It is similar to displaying results. There is a possibility to select all pile plans in the project or only some of them. The pile with the maximum vertical reaction (among the selected class) is used for the design which affects all selected piles. The calculated level is applied to all piles in with the same pile plan.



All – all pile plans in the project are considered for the Pile design

Current – user can select only one pile and all piles from its pile plan is also considered for the Pile design

Advanced – it is similar to Current, but it has more options



Named selection – Pile design is performed for defined named selection

Type of loads
Pile plan design will be performed only for the result classes.

Class
Pile plan design will be performed only for the result classes which have the ultimate combinations defined. The filtering is done automatically. If there are only combinations defined in the combination library, then the program will automatically create result class with ULS combination.

Store result
This check box will allow the user to store the input and output xml files which are used for Pile design and verification. This will enable the user to enhance the pile design in DFoundation (creating the project using this xml file is possible).

Path
The path where the XML files is saved. The name of the file is set automatically – it consists from the name of Pile plan and the result class.

Example:
Pile plan = Pplan1
Result class = ULS

- input file name = pplan1ulsin.xml
- output file name =pplan1ulsout.xml

The action button for the Pile design – Refresh:

Pile plan design calculation

The required pile tip level is calculated in a user-defined pile tip trajectory (in the Geotechnics setup). The levels of positive and negative skin friction are automatically adjusted for each calculation step.

The pile design calculates the pile tip level from Rz value. The horizontal forces or any possible rotations are not considered. The maximum load on the support is determined from all the piles of selected pile plan.

Calculation is stopped when the net bearing capacity is equal to more than the maximum load.

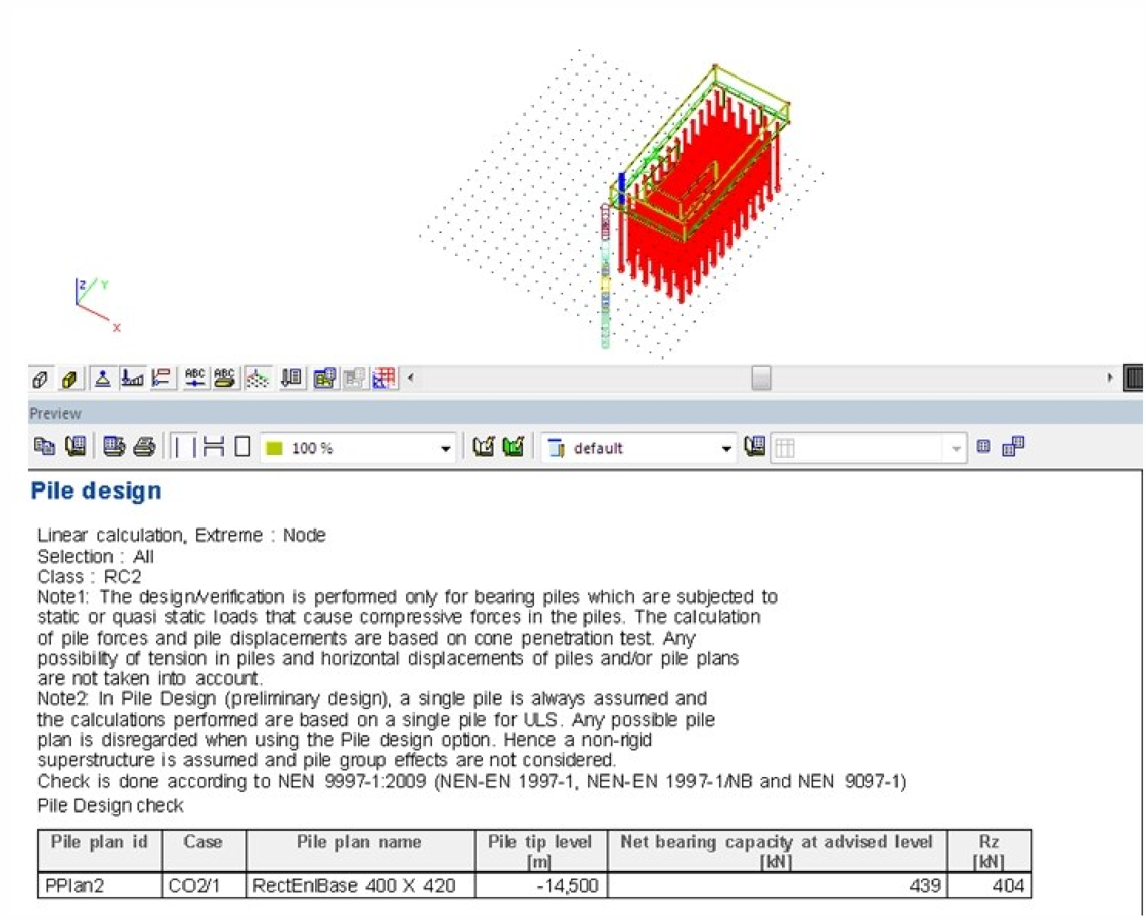
The result is displayed in the preview window.

The maximum support reaction is calculated for each support based on the selected result class (for ULS).

The combination with negative reaction is not considered and the warning message is displayed. The pile is marked as a tension pile.

The value for Pile design is found from the maximum of the maximum support reactions.

If support reactions are negative, the pile design will not be performed and an error will be displayed. The supports with the negative reactions (i.e tension piles) are ignored and the warning is displayed. The design will be performed by considering the remaining supports.



Pile design

Linear calculation, Extreme : Node
Selection : All
Class : RC2
Note1: The design/verification is performed only for bearing piles which are subjected to static or quasi static loads that cause compressive forces in the piles. The calculation of pile forces and pile displacements are based on cone penetration test. Any possibility of tension in piles and horizontal displacements of piles and/or pile plans are not taken into account.
Note2: In Pile Design (preliminary design), a single pile is always assumed and the calculations performed are based on a single pile for ULS. Any possible pile plan is disregarded when using the Pile design option. Hence a non-rigid superstructure is assumed and pile group effects are not considered.
Check is done according to NEN 9997-1:2009 (NEN-EN 1997-1, NEN-EN 1997-1/NB and NEN 9097-1)
Pile Design check

Pile plan id	Case	Pile plan name	Pile tip level [m]	Net bearing capacity at advised level [kN]	Rz [kN]
PPlan2	CO2/1	RectEnlBase 400 X 420	-14,500	439	404

There could be several messages displayed (it can be found also in a special chapter):

1) The required bearing capacity is not met within the specified trajectory for the given cross section of the Pile, Hence the level cannot be calculated

► there is no level found within the trajectory, the net bearing capacity meets the maximum load and the pile tip level is set as zero

2) The start trajectory value should be at least 5 times the pile diameter lower than the lowest ground level and the lowest pile top level

► the start value of the trajectory in the Geotechnics setup is too high

3) The end trajectory value should be at least 4 times the pile diameter above the least deepest CPT.

► the end of the trajectory in the Geotechnics setup is too deep

4) All the piles in the pile group are subjected to tension. The design could not be performed for tension piles

► all the piles in the pile plan are subjected to tension, the pile tip level stays Used defined

5) There are one or more piles in the pile plan subjected to tension

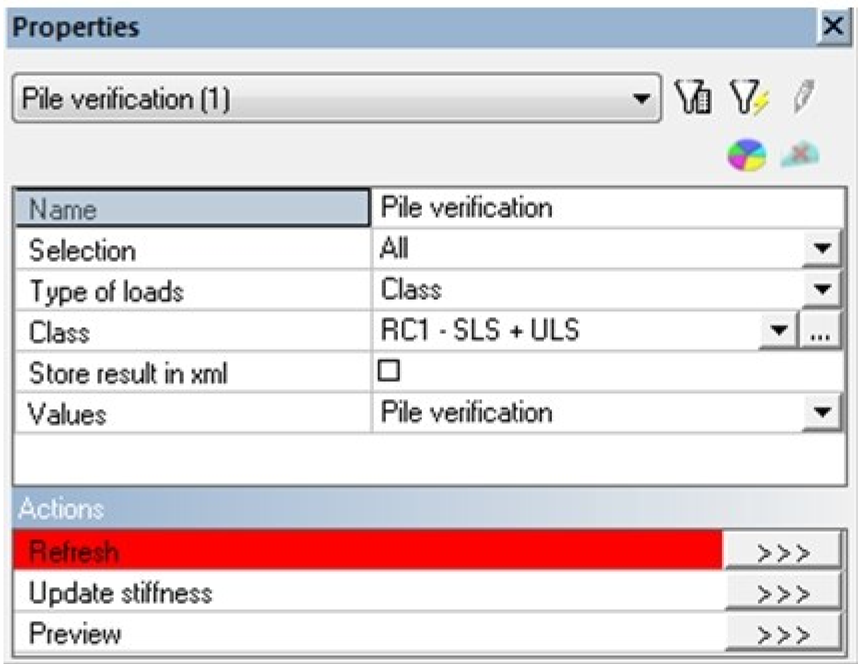
► one or more piles in the pile plan are subjected to tension, the maximum load will be determined from the other piles, the design has been performed for this maximum

The pile tip level calculated by the program is updated in the pile plan and the pile length of appropriate piles is updated.

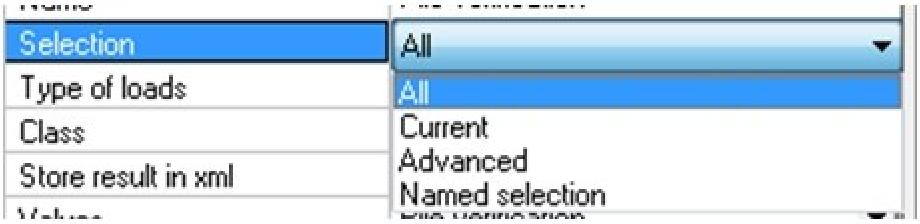
Changes which cause invalid results (design or analysis)

1. Changes made in the Geotechnics setup, soil profile or soil library will not affect the analysis results but affects the pile design results.
   In such case, the program will automatically set the Pile tip level back to user defined. The calculated level is lost. The user has to perform the pile plan design again in order to get the new results.
2. If there is any change in the model, the analysis results and the pile design results become invalid.
   In such case, the pile tip is automatically set back to user defined. The user has to perform the analysis and the design in order to get the new results.

The properties of the verification



Selection
It is similar to displaying results. There is a possibility to select all pile plans in the project or only some of them. The pile with the maximum vertical reaction (among the selected class) is used for the design which affects all selected piles. The calculated level is applied to all piles in with the same pile plan.



Type of loads
Pile plan verification will be performed only for the result classes.

Class
Pile plan verification will be performed only for the result classes which have the ultimate and serviceability combinations defined. The filtering is done automatically. If there are only combinations defined in the combination library, then the program automatically creates a result class with ULS + SLS combination.

Store result in xml
This check box will allow the user to store the input and output xml files which are used for Pile design and verification. This will enable the user to enhance the pile design in DFoundation (creating the project using this xml file is possible).

Path
The path where the XML files is saved. The name of the file is set automatically – it consists from the name of Pile plan and the result class.

Example:
Pile plan = Pplan2
Result class = RC1 – ULS&SLS

- input file name = PPlan2RC1in.xml
- output file name = PPlan2RC1out.xml

The action button for the Pile verification – Refresh:

Pile plan verification - calculation

The verification can run after the pile design is performed. Otherwise some data is missing. The settlement is set to zero and the table displays warning: The input data are not valid.

Type Name	Pile plan Id	Case	CPT	Total load [kN]	Pile plan name	Pile tip level [m]	Settlement at tip-wtip [mm]	Warning
Pile verification	PPlan2	CO2/1	NONAME	0	RectEnlBase 400 X 420	-30,000	0,0	The input data are not valid.

Calculation steps – bearing capacity for a pile

1. The maximum bearing capacity for a single pile is determined as the sum of the maximum bearing capacity of the pile tip and the maximum shaft friction force. This is calculated for every CPT. Those forces are calculated for each interval defined by the Geotechnics setup.

2. Execution factor is defined. It is not a fixed value here and is dependent on the soil type and the depth of the relevant layer. Program defines the correct value of α_s for each layer and then calculates the generated pile shaft friction for that layer. The final value of the pile shaft friction is an aggregation of the pile shaft frictions calculated per layer.
3. If the shape of the pile is constant then the circumference is used for calculation of the maximum shaft friction for the relevant segment.
4. For the piles with a non-constant circumference the program calculates the mean circumference of the relevant pile segment. Those are e.g. tapered wooden piles, piles with a reinforced tip.

Calculation steps – bearing capacity for the foundation:

The program determines the maximum bearing capacity of the foundation.

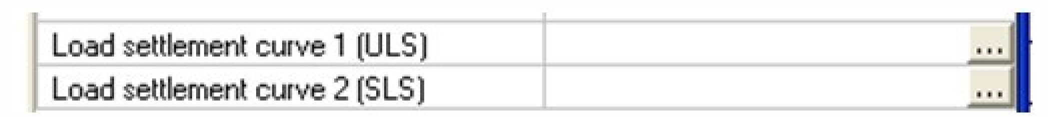
The needed data are:

- the number of piles
- the number of CPTs and whether the structure may be considered as rigid or not (NEN 6743, art. 5.2.2).

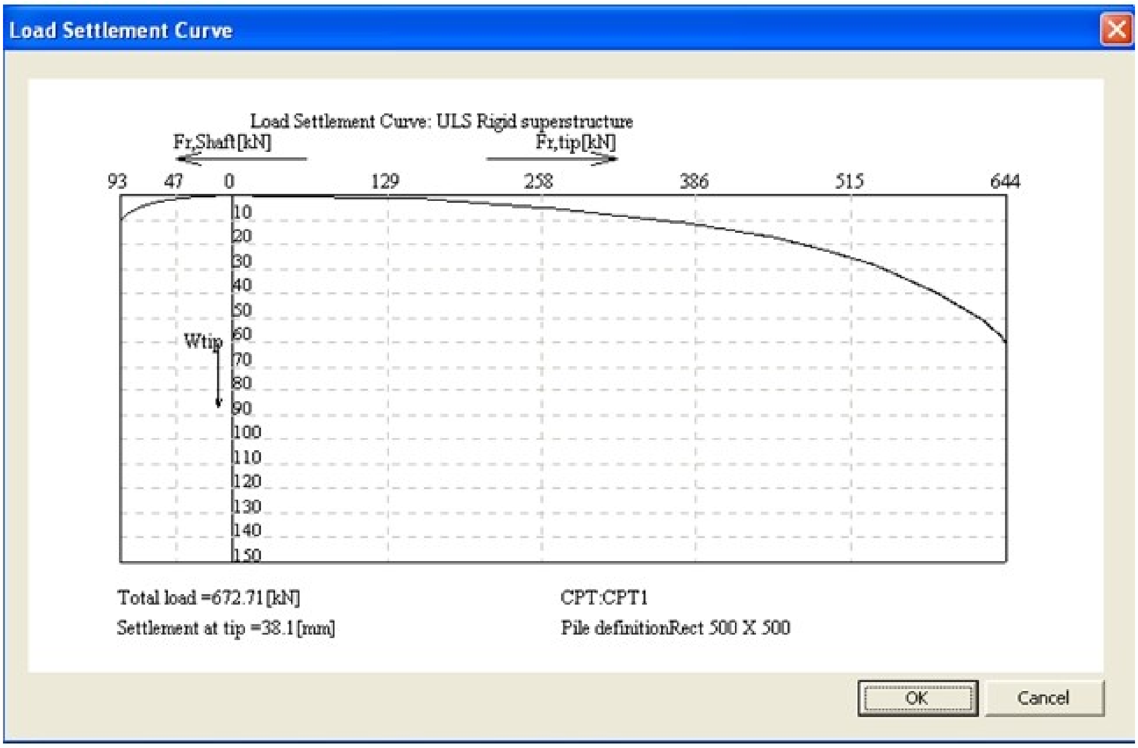
1. The structure is rigid - the program calculates the maximum bearing capacity of the foundation based on the average bearing capacity of a single pile, multiplied by the total number of piles.
2. The structure is non-rigid:

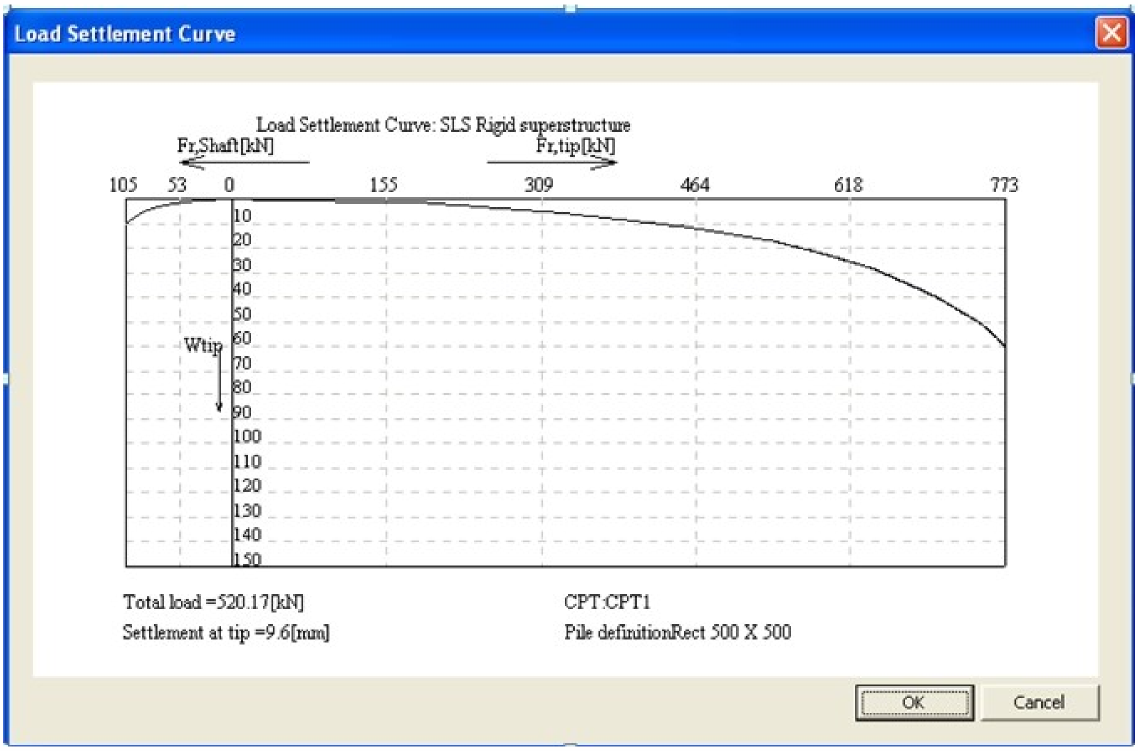
    a. More than 3 CPTs - the definition is again based on the average bearing capacity of a single pile.
    b. 3 or less CPTs - the minimum bearing capacity of a single pile is used. In this case, the bearing capacity of a single pile is not multiplied by the total number of piles because the foundation element consists of a single pile.

The results can be viewed in the preview window and the document. The load –settlement curves are updated in the pile plan library.



If all requirements are fulfilled then both the load –settlement curves are generated. If not the curves may not be generated and warning is displayed.





The maximum support reaction is calculated for each support based on the selected result class (ULS and SLS).

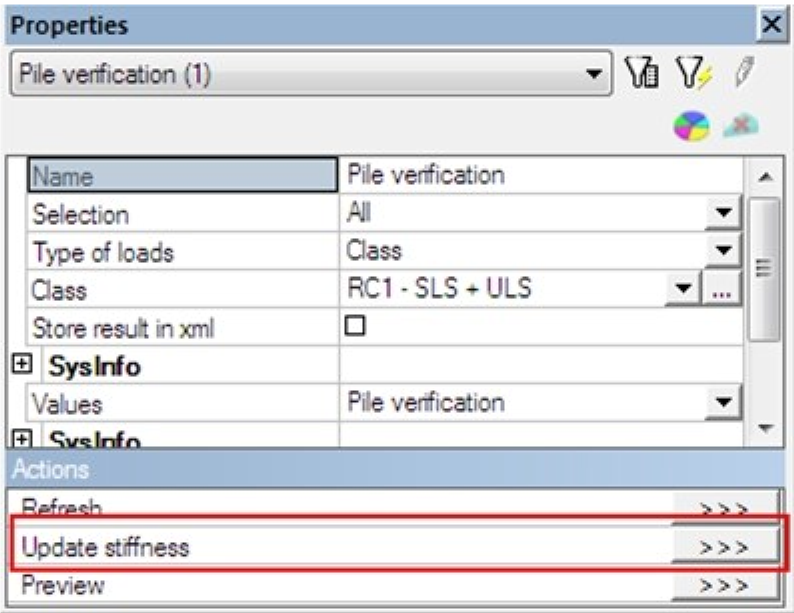
The combination with negative reaction is not considered and the warning message is displayed. The pile is marked as a tension pile.

If support reactions are negative, the pile verification will not be performed and an error will be displayed. The supports with the negative reactions (i.e tension piles) are ignored and the warning is displayed. The verification will be performed by considering the remaining supports.

Update stiffness

There is an option to generate automatically the stiffness from the load-settlement curves. Action button for update is available under the Pile plan verification (it can be used after pile verification). The user can choose between the ULS or the SLS load settlement curve in the geotechnics setup.

The action button:



This method is described by the code. ULS stiffness and SLS stiffness should be taken into account for the further verification.

This button generates automatically stiffness from the load-settlement curves to the properties of the supports.

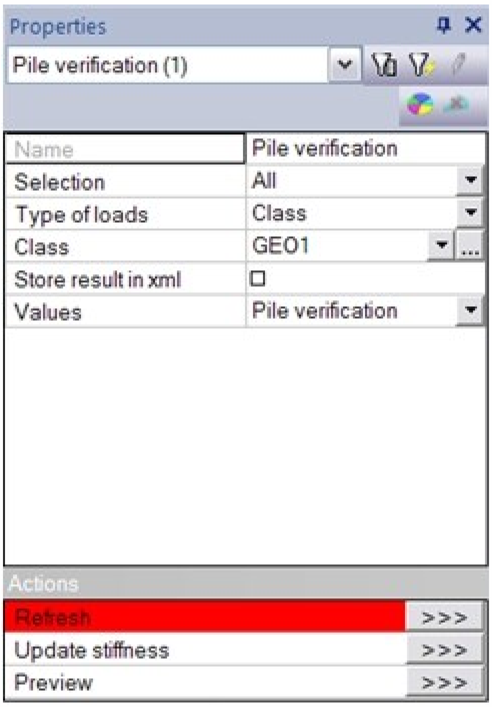
After the linear calculation:

- Z-type becomes flexible

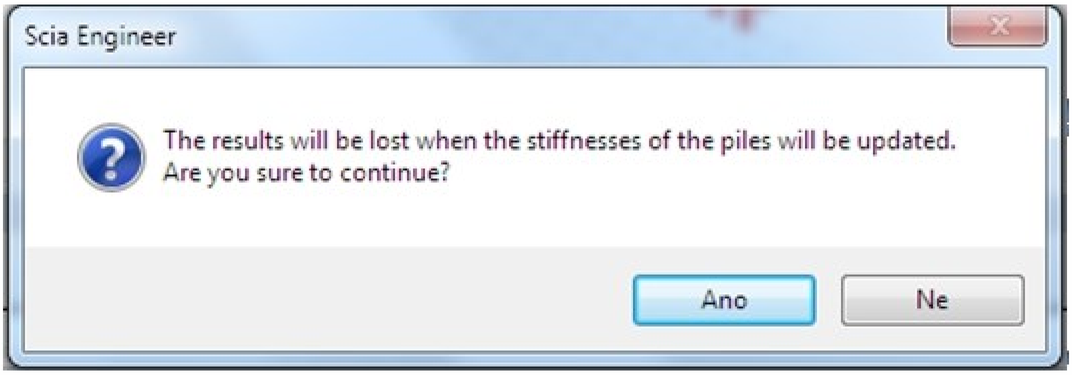
After the non-linear calculation (function Support nonlinearity has to be checked):

- Z-type becomes nonlinear.

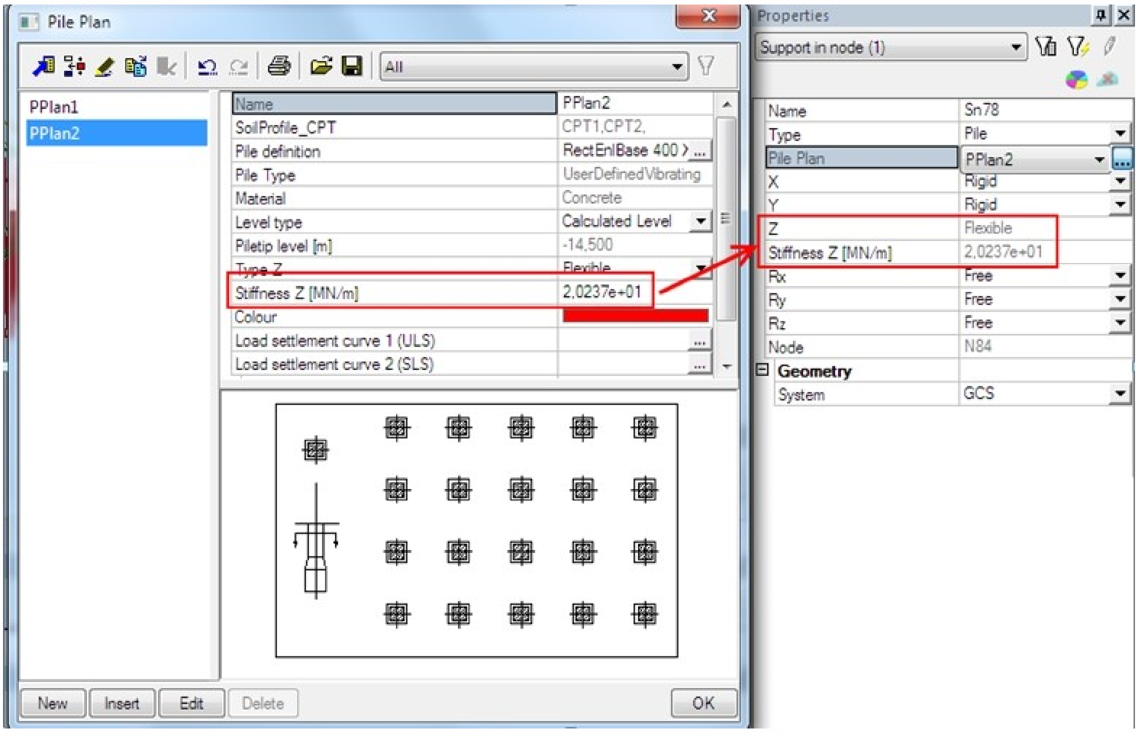
In both cases the stiffness of the pile is taken as the secant defined by zero point and the point in the middle of the curve. The results will be lost after activating this button and the message is displayed.



The message appears. The results will be lost because the model will be changed.

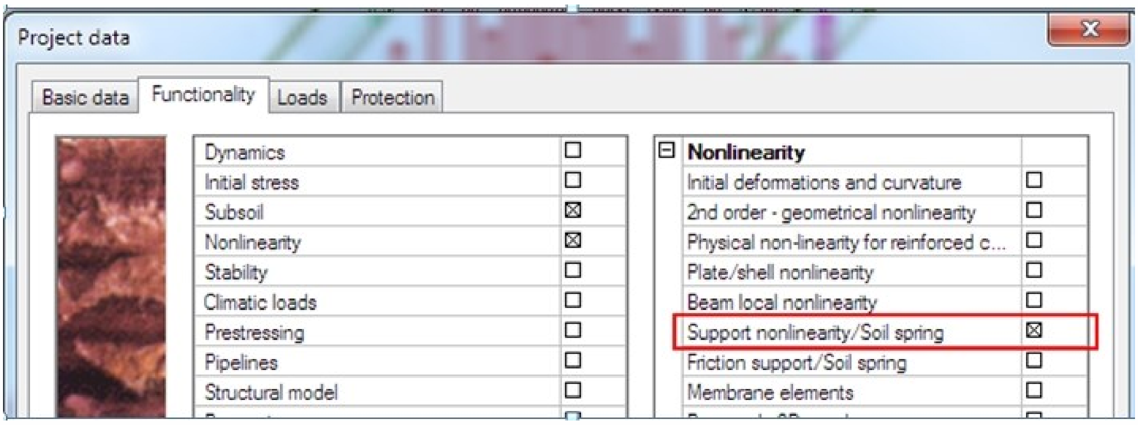


The stiffness is updated in the pile plan and in the properties of the pile.



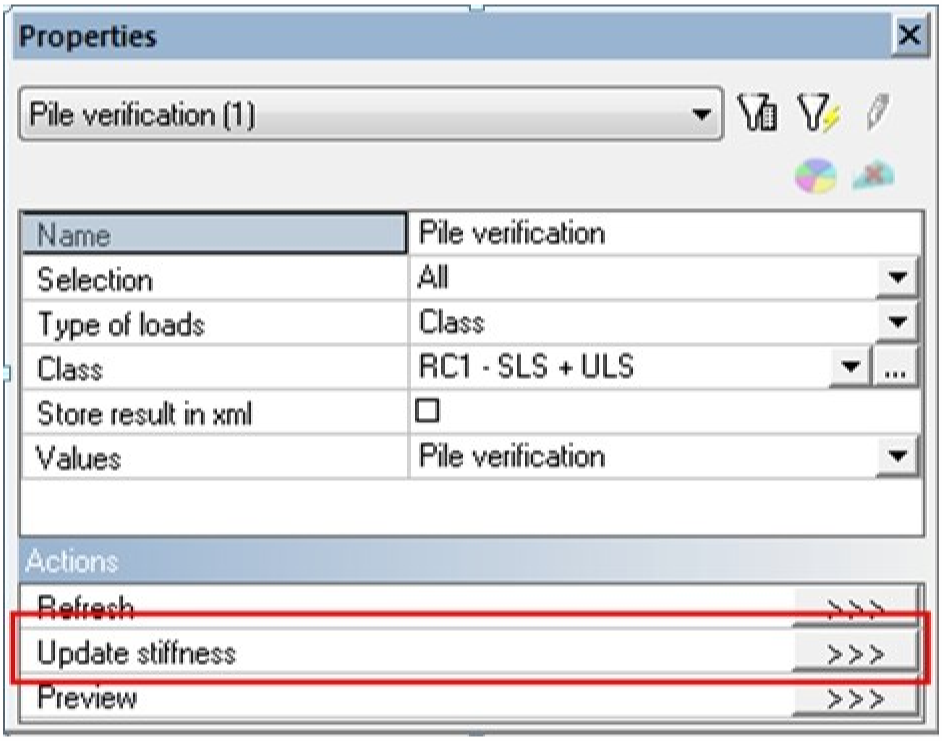
Non-linear Functions

If Support non-linearity/Soil spring functionality is selected, non- linear functions can be automatically generated from the load-settlement curves.

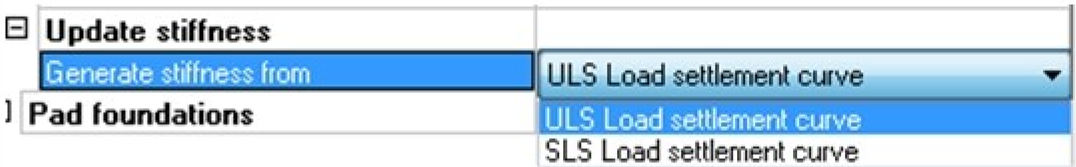


The first step is to run the linear calculation, Pile design, Pile verification and Update stiffness.

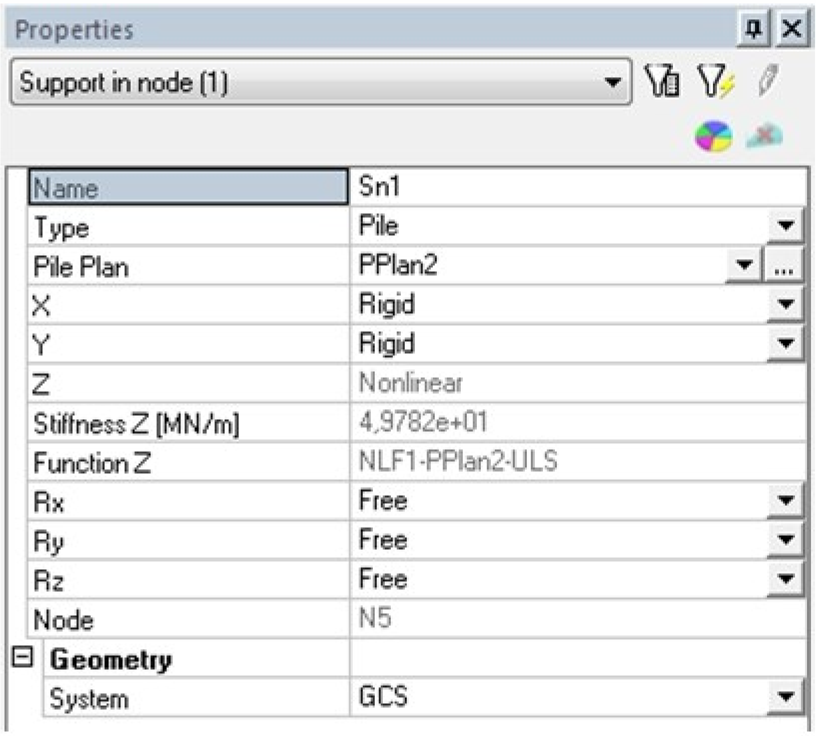
The action button Update Stiffness is available after the pile verification. It will generate automatically non-linear functions from the load settlement.



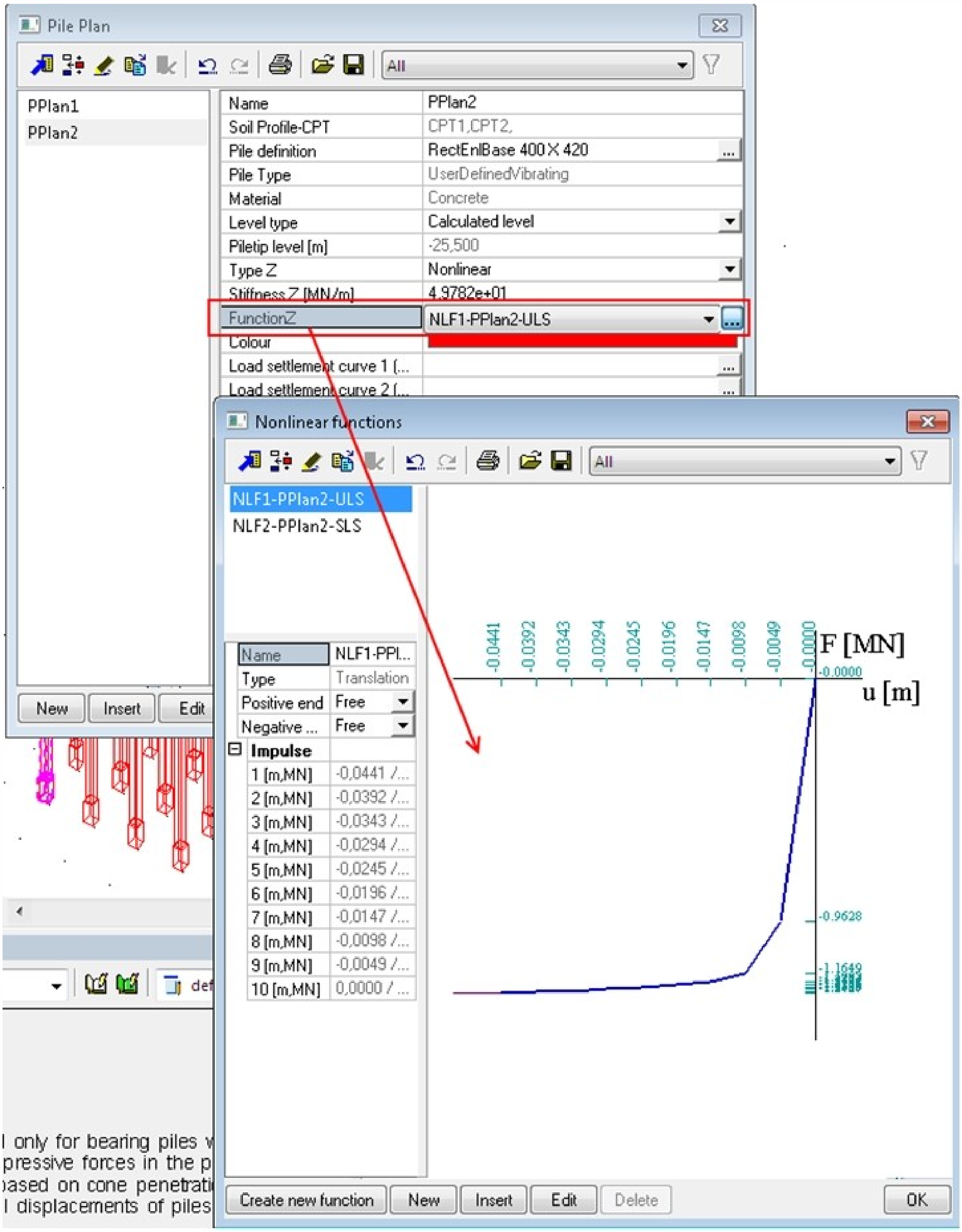
The setup settings determine which settlement curve will be used.



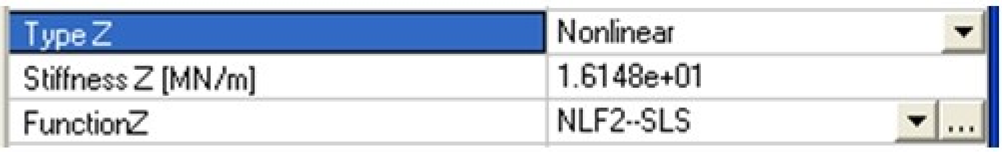
The support property Z is automatically changed to type Nonlinear and the calculated stiffness is added as a value. The non-linear Function is calculated and it is also added to the properties.



The nonlinear function can be viewed in the library:



The stiffness of the pile is taken as the secant defined by zero point and the point in the middle of the curve.

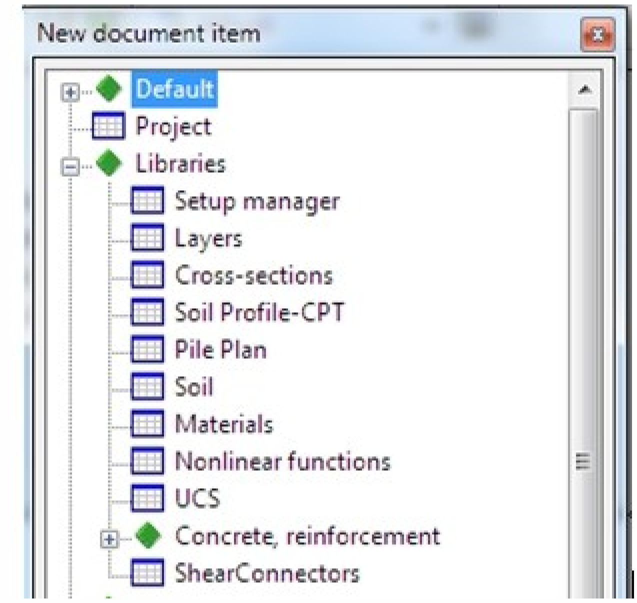


Now it is possible to recalculate the entire structure using these non-linear functions and improve the overall results. This calculation leads to new results and also the new load to the pile head.

The calculation with the new loads can optimize the total design.

Results – document tables

All the output tables of Pile design are available in the standard document.

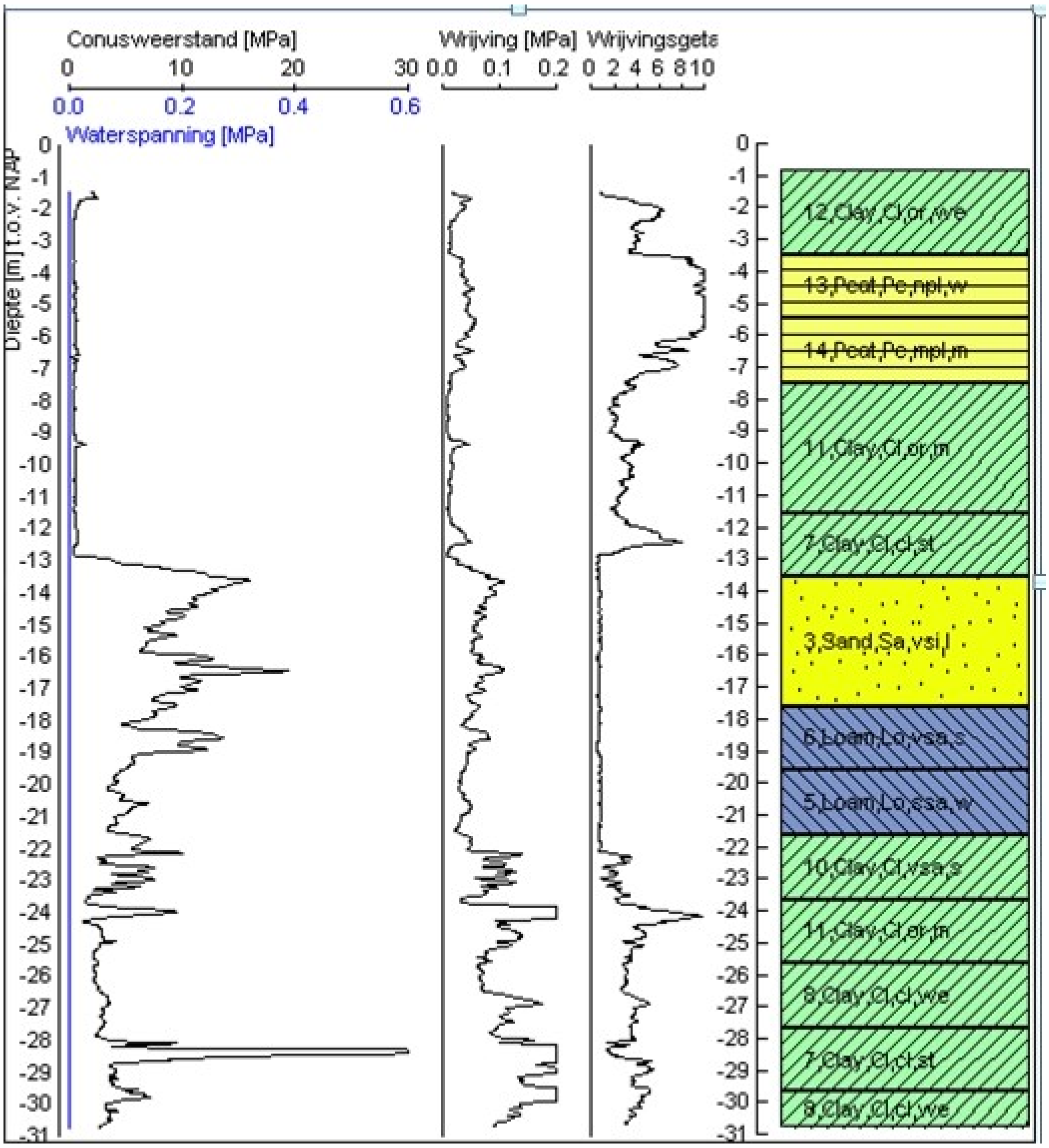


Libraries output tables (Soil profile, Pile plan, Soil):

1. Soil Profile-CPT

CPTData	Name	CPT1
	X [m]	-1,000
	Y [m]	0,000
	Z [m]	-0,880
	Interpretation tool	NEN Rule(Stress dependent)
	Minimum layer thickness [m]	0,200
AddlData	Phreatic level [m]	0,000
	Overconsolidation ratio of bearing layer	1
	Top of positive skin friction zone [m]	-1,000
	Bottom of negative skin friction zone [m]	-0,880
	Expected ground level settlement [m]	0,110

ProfileData	Top level [m]	Soil name	Description
	-0,880	Pe,mpl,m	Peat, mod preloaded, moderate
	-3,005	Cl,or,we	Clay, organ, weak
	-3,205	Pe,mpl,m	Peat, mod preloaded, moderate
	-4,405	Pe,npl,w	Peat, not preloaded, weak
	-5,005	Pe,mpl,m	Peat, mod preloaded, moderate
	-5,205	Pe,npl,w	Peat, not preloaded, weak
	-5,825	Pe,mpl,m	Peat, mod preloaded, moderate
	-7,445	Cl,or,we	Clay, organ, weak
	-7,645	Cl,or,m	Clay, organ, moderate
	-7,845	Cl,or,we	Clay, organ, weak
	-10,524	Cl,or,m	Clay, organ, moderate
	-11,944	Cl,or,we	Clay, organ, weak
	-12,344	Cl,or,m	Clay, organ, moderate
	-12,544	Cl,ssa,m	Clay, slightly sandy, moderate
	-12,744	Lo,ssa,w	Loam,slightly sandy weak
	-12,944	Sa,vsi,l	Sand, very silty, loose
	-13,344	Lo,vsa,s	Loam, very sandy, stiff
	-14,744	Lo,ssa,w	Loam,slightly sandy weak
	-14,963	Lo,vsa,s	Loam, very sandy, stiff
	-15,203	Sa,vsi,l	Sand, very silty, loose



The Soil profile – CPT table contains input CPT data, Additional data, Profile data and graph of the Profile data (the interpreted geometry).

2. Pile Plan

Name	Pileplandata					
PPlan1	Piledata	Soil Profile-CPT	Pile definition	Pile Type	Material	Piletip level [m]
		CPT1,CPT2,	Rect 250 X 250	PrefabConcrete	Concrete	-30,000
PPlan2	Piledata	Soil Profile-CPT	Pile definition	Pile Type	Material	Piletip level [m]
		CPT1,CPT2,	RectEnlBase 400 X 420	UserDefinedVibrating	Concrete	-30,000

The Pile plan table contains input data for each Pile plan in the library.

3. Soil

Name	Description	Soil Type	Gamma unsaturated [kN/m³]	Gamma saturated [kN/m³]	Friction angle [deg]
Gr,ssi,m	Gravel, slightly silty, moderate	Gravel	19,0	21,0	37,50
Sa,ssi,m	Sand, slightly silty, moderate	Sand	19,0	21,0	32,50
Sa,vsi,l	Sand, very silty, loose	Sand	19,0	21,0	30,00
Sa,cl,st	Sand, clean, stiff	Sand	20,0	22,0	40,00
Lo,ssa,w	Loam,slightly sandy weak	Loam	20,0	20,0	30,00

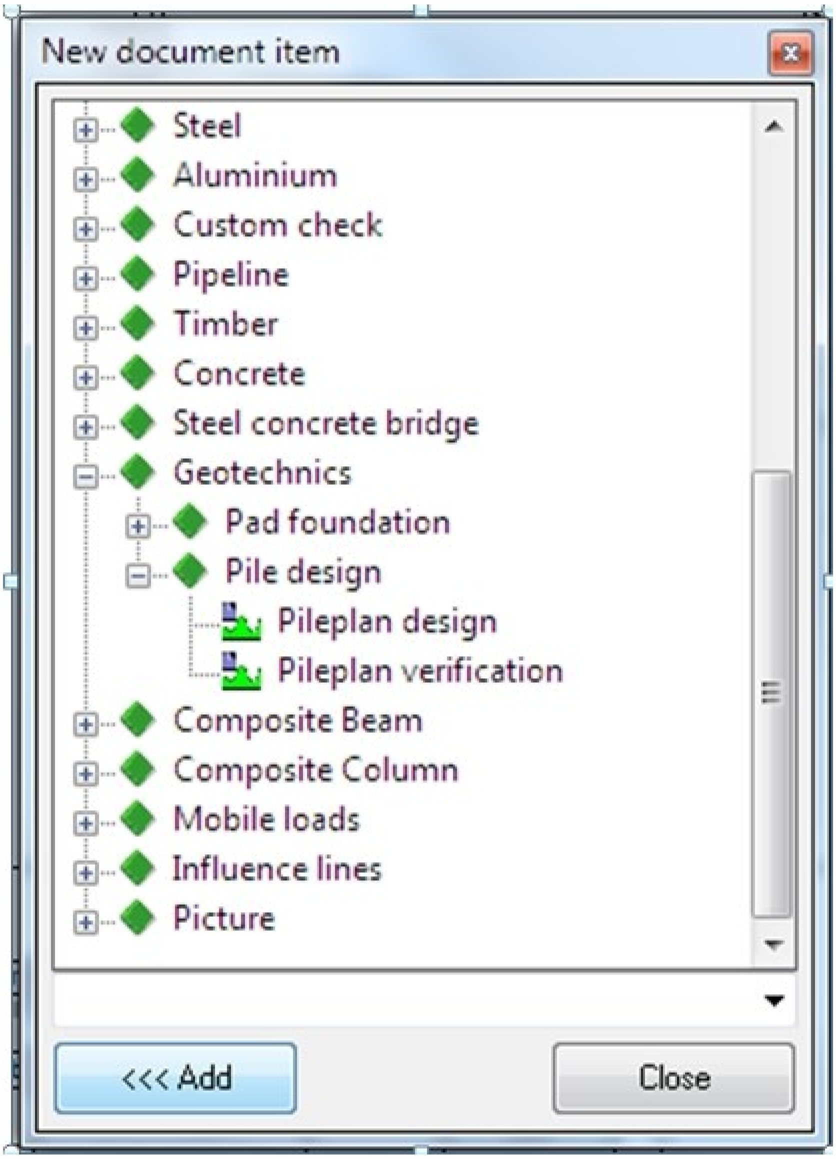
Soil library table contains data for the soil definition.

If the nonlinearity is calculated, then also table for the nonlinear function can be displayed.

4. Nonlinear functions

Name	Type	u / F [m,MN]	Positive end	Negative end
NLF1-PPlan2-ULS	Translation	-4,4100e-02/-1,2427e+00	Free	Free
		-3,9200e-02/-1,2383e+00		
		-3,4300e-02/-1,2330e+00		
		-2,9400e-02/-1,2267e+00		
		-2,4500e-02/-1,2197e+00		
		-1,9600e-02/-1,2101e+00		
		-1,4700e-02/-1,1982e+00		
		-9,8000e-03/-1,1649e+00		
		-4,9000e-03/-9,6284e-01		
		0,0000e+00/0,0000e+00		
NLF2-PPlan2-SLS	Translation	-4,4100e-02/-1,4912e+00	Free	Free
		-3,9200e-02/-1,4860e+00		
		-3,4300e-02/-1,4796e+00		
		-2,9400e-02/-1,4720e+00		
		-2,4500e-02/-1,4636e+00		
		-1,9600e-02/-1,4521e+00		
		-1,4700e-02/-1,4378e+00		
		-9,8000e-03/-1,3979e+00		
		-4,9000e-03/-1,1554e+00		
		0,0000e+00/0,0000e+00		

Geotechnics result tables:



Pile design

Linear calculation, Extreme : Node
Selection : All
Class : RC2
Note1: The design/verification is performed only for bearing piles which are subjected to static or quasi static loads that cause compressive forces in the piles. The calculation of pile forces and pile displacements are based on cone penetration test. Any possibility of tension in piles and horizontal displacements of piles and/or pile plans are not taken into account.
Note2: In Pile Design (preliminary design), a single pile is always assumed and the calculations performed are based on a single pile for ULS. Any possible pile plan is disregarded when using the Pile design option. Hence a non-rigid superstructure is assumed and pile group effects are not considered.
Check is done according to NEN 9997-1:2009 (NEN-EN 1997-1, NEN-EN 1997-1/NB and NEN 9097-1)
Pile Design check

Pile plan id	Case	Pile plan name	Pile tip level [m]	Net bearing capacity at advised level [kN]	Rz [kN]
PPlan2	CO2/1	RectEnlBase 400 X 420	-15.000	492	482

Pile verification

Linear calculation
Selection : All
Class : RC1
Note1: The design/verification is performed only for bearing piles which are subjected to static or quasi static loads that cause compressive forces in the piles. The calculation of pile forces and pile displacements are based on cone penetration test. Any possibility of tension in piles and horizontal displacements of piles and/or pile plans are not taken into account.
Check is done according to NEN 9997-1:2009 (NEN-EN 1997-1, NEN-EN 1997-1/NB and NEN 9097-1)

ULS

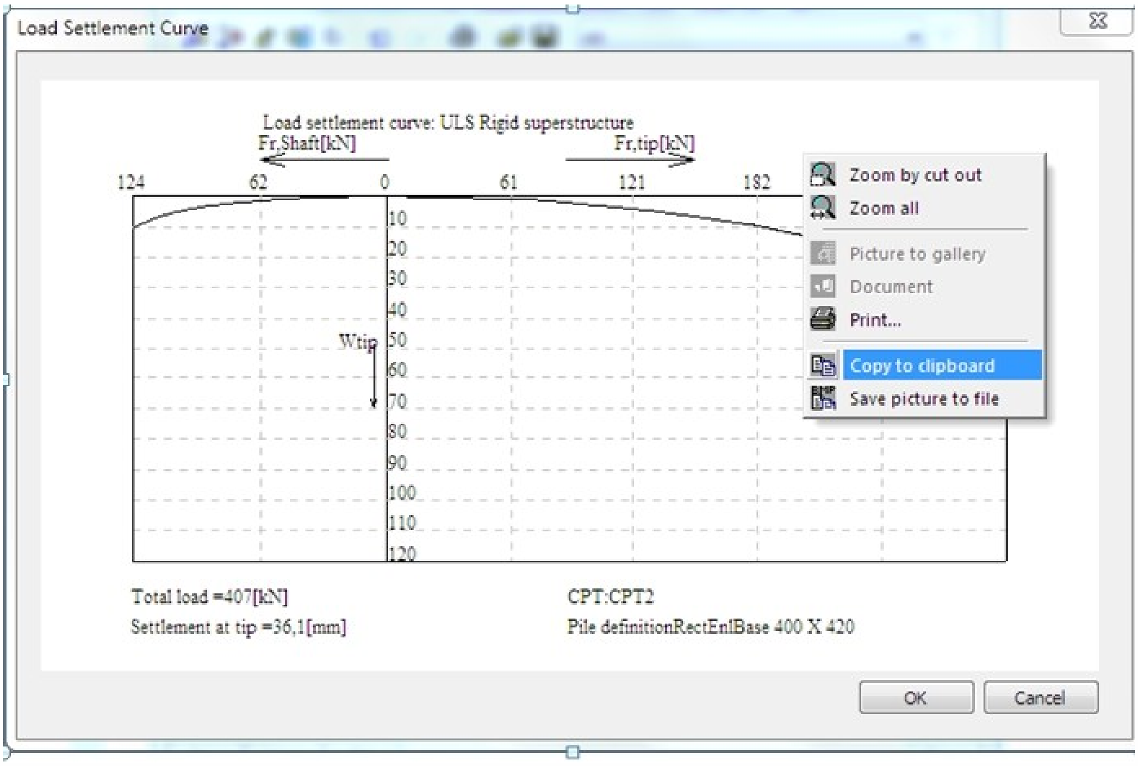
Pile plan Id	Case	CPT	Total load [kN]	Pile plan name	Pile tip level [m]	Settlement at tip-Wtip [mm]
PPlan2	CO2/1	CPT2	482	RectEnlBase 400 X 420	-15,000	43,9

SLS

Type Name	Pile plan Id	Case	CPT	Total load [kN]	Pile plan name	Pile tip level [m]	Settlement at tip-Wtip [mm]
Pile verification	PPlan2	CO2/1	CPT2	362	RectEnlBase 400 X 420	-15,000	8,6

Load settlement curves – picture:

It is possible to save the load settlement picture to the clipboard management. The tool is in the context menu.



Limitations

There are some limitations in the Pile design functionality:

Functionality

This functionality is based on the guidelines given by Dutch standards NEN 6740 and NEN 6743. This functionality is made available only for the EC-EN and NEN National codes.

Soil profile

The soil profiles could be generated only using a valid CPT data and the CPT data should only be in GEF (Geotechnical exchange format) file. No other format is supported.

The user interface for the soil profile dialog does support only English. Other languages are not supported in this special dialog.

The position of CPT (soil profiles) can be defined only in the user coordinates systems (UCS). The GPS coordinates of the project cannot be defined and hence the GPS coordinates are not used.

Interpretation of the soil profile – interpreted geometry

The interpretation tool for generating the soil profile is “NEN rule (Stress Dependent)”. Any tool other than NEN cannot be defined or used.

The automatic interpretation tool uses only the pre-defined soil types. User defined soils cannot be used in the interpreted geometry. The user defined soil can be used only for edited geometry.

Pile plan

The special dialog for the definition of the Pile plan does support only English. All the parameters required by this dialog are based on NEN.

Only the predefined shapes can be used and not possible to define any user defined shapes.

The units specified in this special dialog are not linked with the units setup and hence the user has to specify the values in the respective units.

Design and verification

The design and verification is performed only for bearing piles which are subjected to static or quasi static loads that cause compressive forces in the piles.

The calculation of pile forces and pile displacements are based on the cone penetration test. Any possibility of tension in piles and horizontal displacements of piles are not taken into account.

A single pile is always assumed for the pile design and the calculations performed are based on a single pile for ULS. Any possible pile plan is disregarded when using the Pile design option. Hence a non- rigid superstructure is assumed for pile design and pile group effects are not considered.

Errors/ Warnings

The program validates the input data while performing the Pile design or verification. If the input data contains invalid data, the program will not perform the design or verification and will display the warning.

Also if the input data or the design result doesn’t meet the requirements specified in NEN, the program displays the warning.

The warnings are displayed in the results of design and the verification – preview or the document tables.

Errors:

CPT Level:
The levels of Cone penetration test data must be strictly increasing below the ground level. If not, the program will not perform the design and verification

- The CPT levels must be strictly increasing below ground level. Please check the CPT data or the GEF file.

Tension Piles:
If all Piles in the Pile plan are subjected to tension, the program will not perform the design and verification for tension piles and hence the error will be displayed in the result.

- All piles in the Pile plan are subjected to tension and the current version supports only piles that are subjected to compression.

Reactions:
If the maximum support reactions for all the Piles in the Pile plan are less than 500N, the design and verification could not be performed. The program checks this condition and display the proper error message.

- The loads for limit states ULS or SLS (design values) are very small to perform pile design and verification.

Surface level:
If the Top of positive skin friction zone lies above the surface level or the excavation level, the design and verification will not be performed.

- The top of positive skin friction zone is situated above surface level. Please check the additional data of the Soil profile - CPT.

Skin friction zones:
If the skin friction zones are intersecting (i.e.) if the Top of positive skin friction zone lies above the bottom of negative skin friction zone then the design and verification won't be performed.

- The positive and negative skin friction zones are intersecting. Please check the additional data of the Soil profile - CPT.

Trajectory definition:
If the number of calculation steps is less than one, the program cannot perform the design and verification. The number of calculation steps depends upon the Start and End Trajectory and the interval value.

- The calculation step value should be greater than or equal to one. Please change the trajectory definition.

Interval:
If the interval is more than the difference between the Start and End trajectory then the calculation cannot be performed.

- The interval is too large for the defined trajectory. Please change the trajectory definition.

Top of positive skin friction zone:
If the top of positive skin friction zone lies below the Pile tip level, the verification will not be performed. The user has to adjust any of these levels.

- The top of positive skin friction zone is situated below the pile tip level. Please check the additional data of the Soil profile - CPT.

ULS and SLS:
If the serviceability reactions are higher than ultimate reaction, the pile verification could not be performed and hence the error will be issued.

- The loads for ULS are smaller than the loads for SLS.

Pile tip level:
The pile length should extend at least 5 * D_{min} beneath the surface (or excavation) level. D_{min} stands for the minimum size (in cross section) of the pile base. If not, the program will issue a warning that "The pile length does not meet the requirements set by "NEN 6743 art. 3.1".

- The pile length does not meet the requirements set by NEN 6743. The pile tip level should extend at least 5* D_{min} beneath the surface (or excavation) level.

Warnings:

Trajectory definition:
The value of Start trajectory should be at least 5 times the pile diameter lower than the lowest ground level and the lowest pile top level. If this condition is not met, the program will issue a warning in the results.

- The Start trajectory value should be at least 5 times the pile diameter lower than the lowest ground level and lowest pile top level.

The value of End trajectory should be at least 4 times the pile diameter above the least deepest CPT.

- The End trajectory value must be at least 4 times the pile diameter above the least deepest CPT.

Reactions:
If the net bearing capacity doesn't meets the maximum load within the trajectory then the program returns the pile tip level as "zero" and a warning will be displayed.

- The required bearing capacity is not met within the specified trajectory for the given cross section of the Pile. Hence the level cannot be calculated.

Tension:
If a Pile is subjected to both compression (Eg: CO1 – Compression) and tension (Eg CO2- Tension), the combinations (CO2) which has tension will not be considered. The program will find the max reaction based on the other combinations and the design will be performed. The warning message appears.

- One or more piles in the Pile plan are subjected to both compression and tension. The combinations which yield the tension reactions are not considered. The max reaction is calculated only based on other combinations and the design is performed based on the max reaction.

Tension Piles:
If one or more Piles in the Pile plan are subjected to tension, the program will not consider the tension piles for design and verification and displays the warning. If all the piles are in tension the program will not perform the design and verification and returns error.

- A few piles in the Pile plan are subjected to tension and tension piles are not considered in design. The current version supports only piles that are subjected to compression.

Pile Distance:
The distance between two piles must be at least greater than 3 times the equivalent pile diameter. If this condition doesn't meet, the program will display a warning.

- The Pile plan has a pile to pile distance less than 3*equivalent diameter.

Top of PSFZ:
If the top of positive skin friction zone lies below the top of enlarged base of the pile, the program will set the top of positive skin friction equal to the top of enlarged base. This will be intimated to the user as a warning.

- The top of the positive friction zone is situated above the enlarged base of the pile. The top of the positive friction zone is therefore reset to the top of the enlarged base.