SCIA ENGINEER

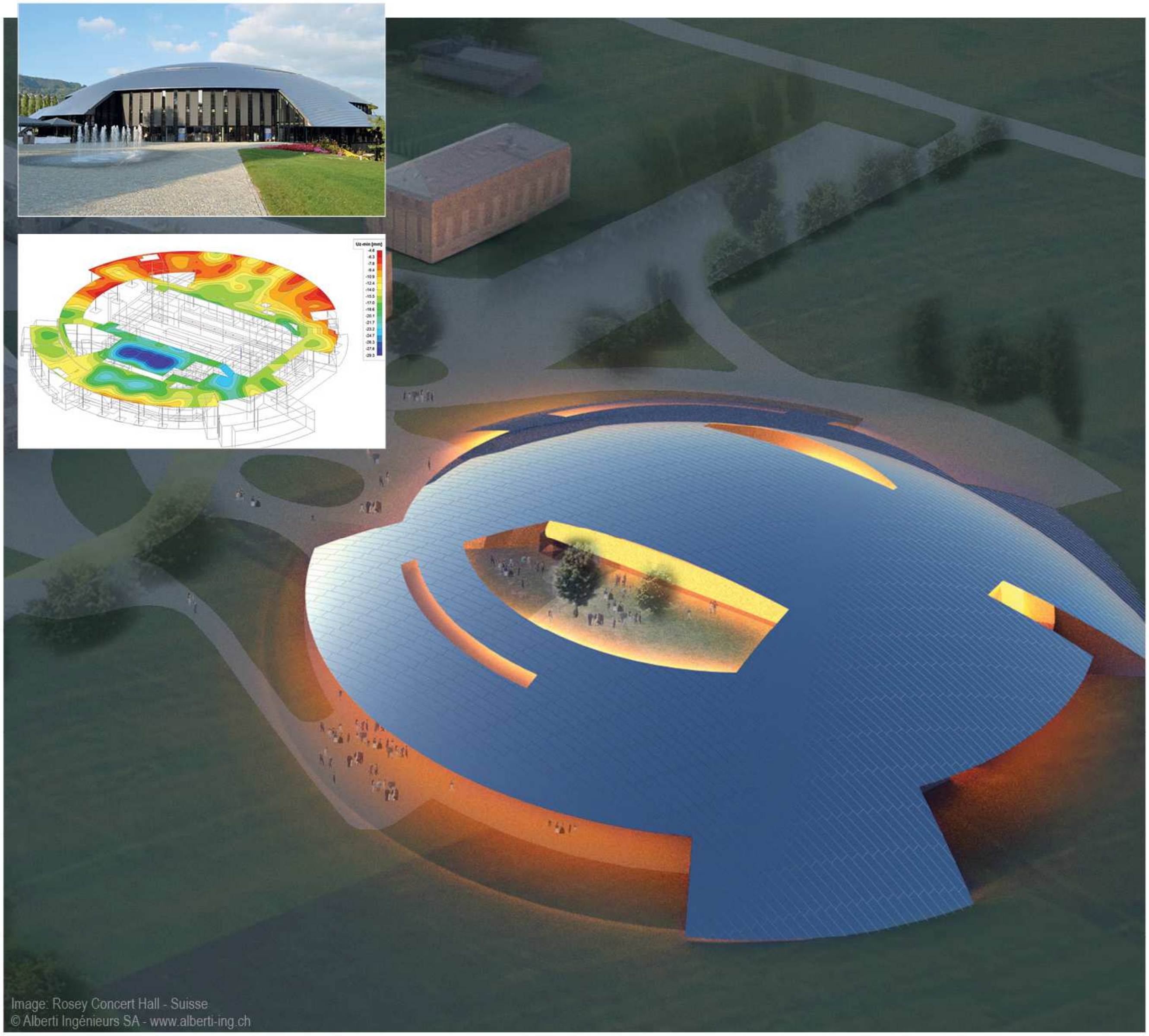


Image: Rosey Concert Hall - Suisse
© Alberti Ingénieurs SA - www.alberti-ing.ch

Topic Training

Foundations

Table of contents

Introduction

This course will explain the principles of the use of Foundations and Subsoil in SCIA Engineer. Most of the modules necessary for these calculations are included in the **Concept Edition**.

For some options a Concept Edition is not sufficient. These specific required modules are included in an **Expert Edition** or even some **extra modules** are necessary.

The methods discussed in this manual are based on Eurocode 7. **EN 1997-1** is intended to be applied to the geotechnical aspects of the design of buildings and civil engineering works. It is concerned with the requirements for strength, stability, serviceability and durability of structures.

List of necessary modules:

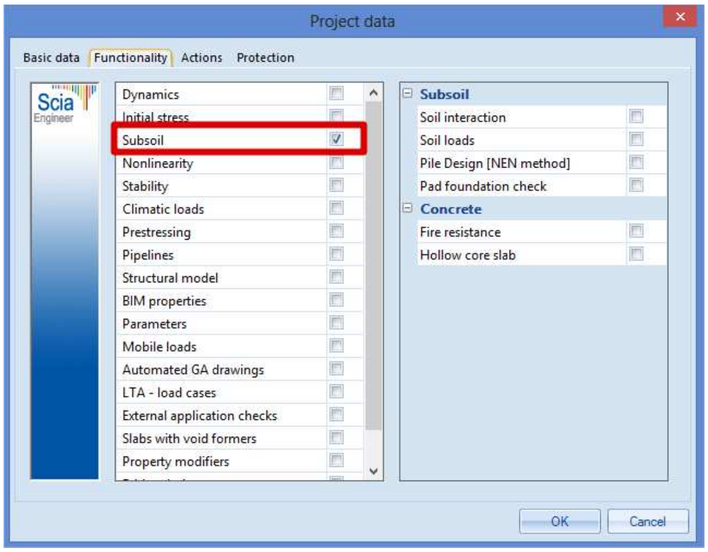
- esafd.02.01 Pad Foundations EC
- esas.06 Soil interaction (Part of Expert Edition)
- esas.08 Soil (part of Concept Edition)

Subsoil

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

- tree menu function Library > Subsoils.
- menu function Libraries > Subsoils.

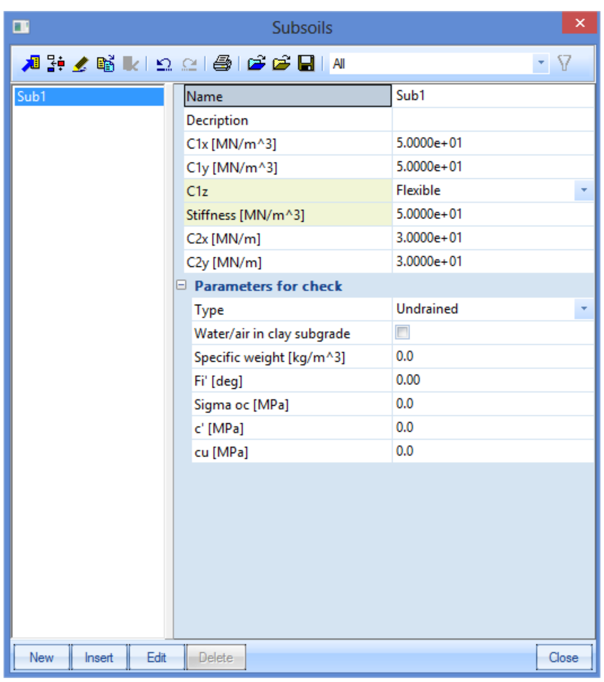
These functions are available when the Functionality “Subsoil” is activated.



Subsoil parameters

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

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



The constants C1 and C2 for directions X, Y, Z are parameters representing the subsoil properties.

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

Parameters for check

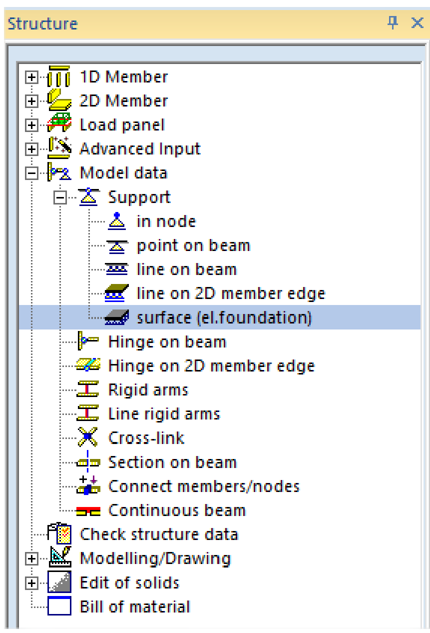
These data are used only for the stability check of a foundation block.

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.

Elastic foundation

In SCIA Engineer it is possible to define a slab on an elastic foundation by means of a subsoil. The principle of an elastic foundation is explained in Annex 1.

When the functionality is activated and the slab is drawn, you can go to: Structure > Model data > Support > Surface (elas. Foundation).



Note: It is not recommended to use parameters C2x and C2y because reliable experimental data are not available for these parameters.

To see the results of the elastic foundation, you go to Results > 2D Members > Contact stresses.

Remark: Convention for the soil stresses is: - positive value = compressive stress
- negative value = tensile stress

To eliminate the tension in the subsoil you have to calculate non linear (module esas.44 (P+E edition)) Functionality: Nonlinearity + Support nonlinearity/Soil spring

Pad Foundations

In this chapter the different steps of the Pad Foundation Checks are specified.

First of all, the required safety and resistance factors need to be determined depending on the chosen Design Approach.

Using these safety factors, the vertical design loading V_d, horizontal design loading H_d and effective geometry of the pad are determined.

Based on this effective geometry, the different checks are executed.

The above steps are explained in detail in the following paragraphs.

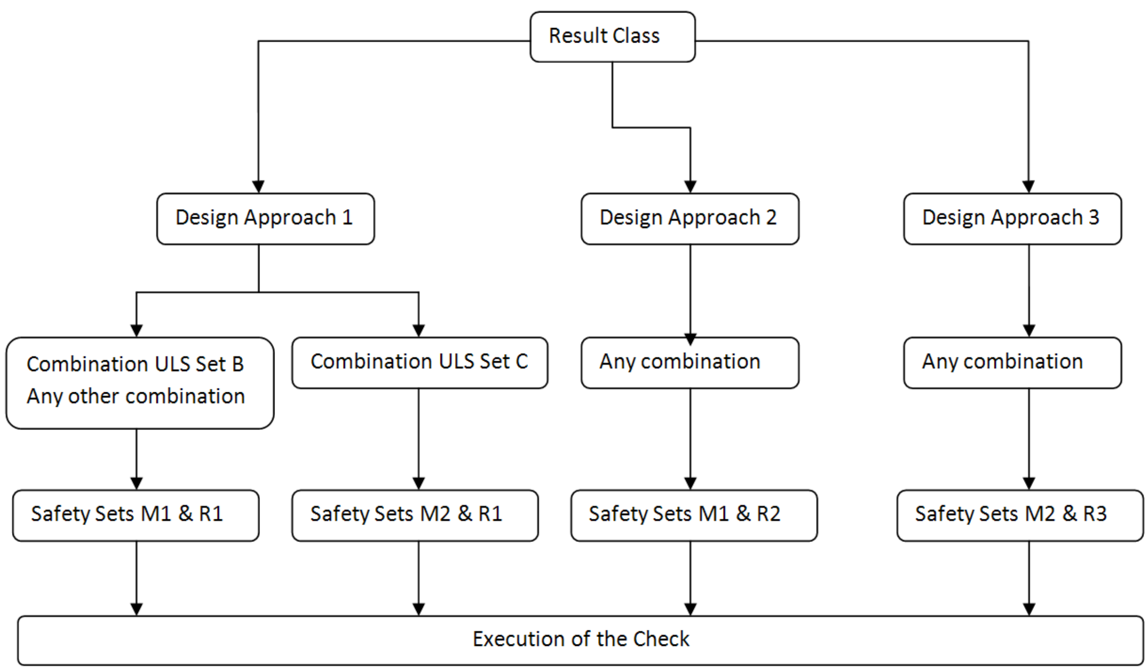
Design Approaches

The Pad Foundation check is executed for a Result Class.

The manner in which the design effects of actions and resistances are applied shall be determined using one of three Design Approaches.

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

Design Approach 1	Combination 1: A1 “+” M1 “+” R1 Combination 2: A2 “+” M2 “+” R1
Design Approach 2	Combination: A1 “+” M1 “+” R2
Design Approach 3	Combination: (A1* or A2**) “+” M2 “+” R3 * On structural actions ** On geotechnical actions



- 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.

The safety factors corresponding with a certain design approach can be found in the Annex A of EN 1997-1:

1) Partial factors on actions or the effects of actions (Set A1-A2)

γ_G: on permanent unfavourable or favourable actions
γ_Q: on variable unfavourable or favourable actions

Table A.3 - Partial factors on actions (γ_F) or the effects of actions (γ_E)

Action		Symbol	Set	
			A1	*A2*
Permanent	Unfavourable	γ_G	1,35	1,0
	Favourable		1,0	1,0
Variable	Unfavourable	γ_Q	1,5	1,3
	Favourable		0	0

2) Partial factors for soil parameters (Set M1-M2)

$\gamma_{\phi'}$: on the tangent of the angle of shearing resistance
$\gamma_{c'}$: on effective cohesion
γ_{cu}: on undrained shear strength
γ_{qu}: on unconfined strength
γ_{γ}: on weight density

Table A.4 - Partial factors for soil parameters(γ_M)

Soil parameter	Symbol	Set	
		M1	*M2*
Angle of shearing resistance[a]	$\gamma_{\varphi'}$	1,0	1,25
Effective cohesion	$\gamma_{c'}$	1,0	1,25
Undrained shear strength	γ_{cu}	1,0	1,4
Unconfined strength	γ_{qu}	1,0	1,4
Weight density	γ_{γ}	1,0	1,0
[a] This factor is applied to tan φ'			

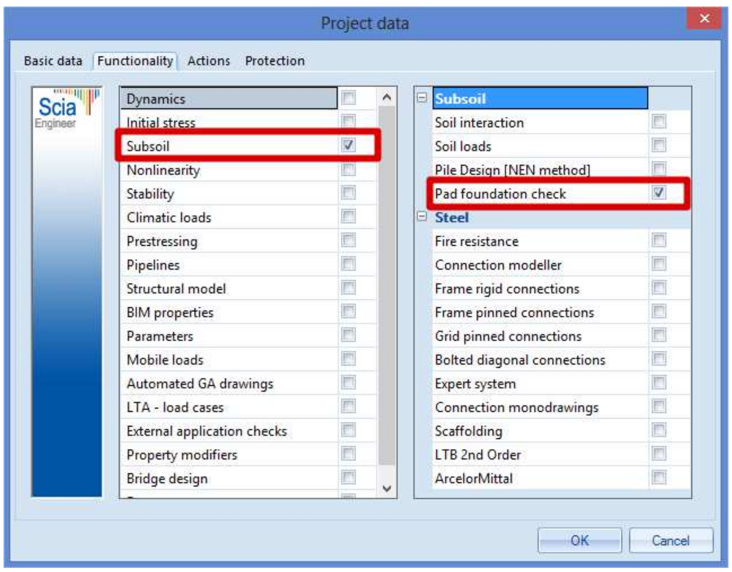
3) <u>Partial resistance factors for pad foundations (Set R1-R3)</u>

$\gamma_{R;v}$: on bearing resistance
$\gamma_{R;h}$: on sliding resistance

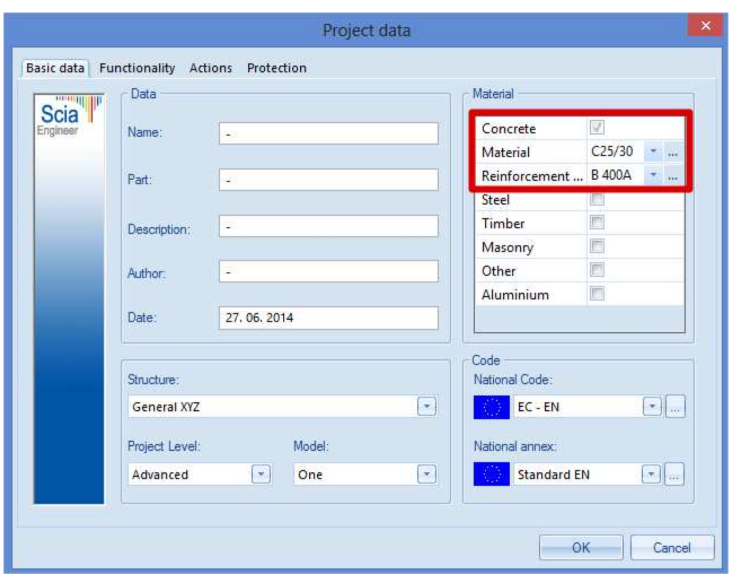
Table A.5 - Partial resistance factors (γ_R) for spread foundations

Resistance	Symbol	Set		
		R1	*R2*	*R3*
Bearing	$\gamma_{R;v}$	1,0	1,4	1,0
Sliding	$\gamma_{R;h}$	1,0	1,1	1,0

To design the pad foundation in SCIA Engineer, the functionalities “Subsoil” and “Pad foundation check” should be activated in the Project data dialogue:

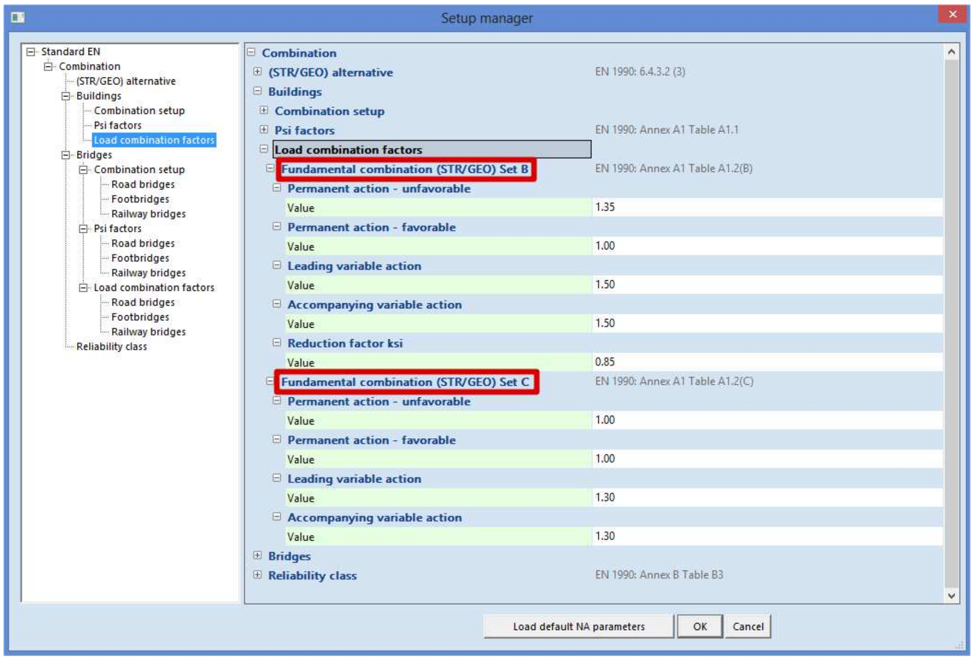


Remark: Make sure that also the material “Concrete” has been activated; otherwise it is not possible to input a pad foundation!

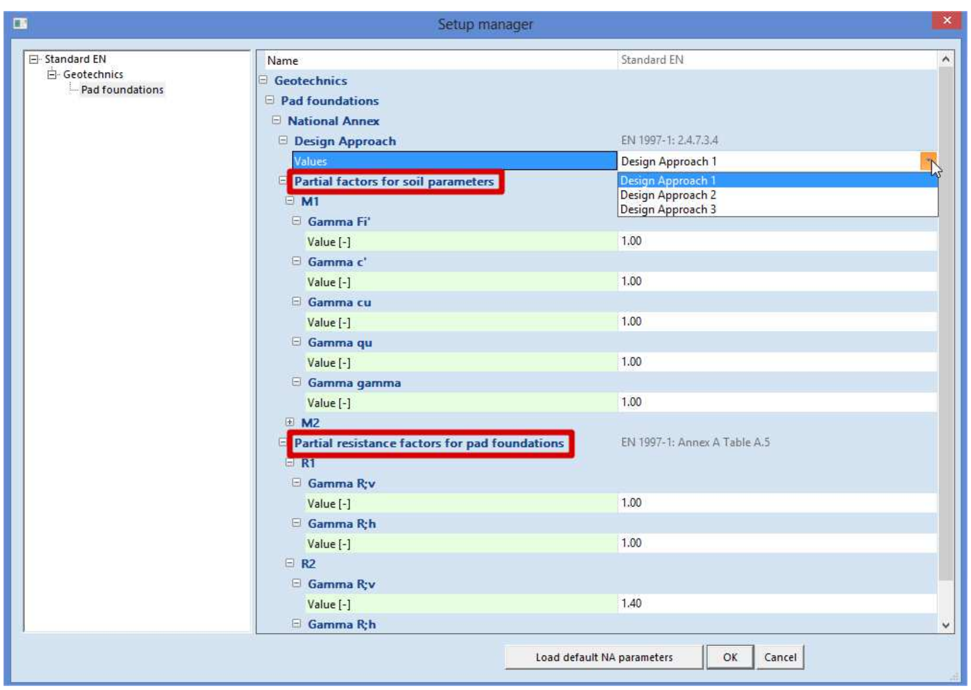


The partial safety factors for the combination are defined in the Manager for National Annexes. It can be opened from the Basic project data dialogue.

Available are factors for Set B of the EN-ULS (STR/GEO) combination defined in EN 1990. In addition, for Geotechnical analysis, also Set C needs to be supported. Also these factors are available in the Combination Setup:



Also the partial factors for soil parameters and the partial resistance factors for pad foundations which are defined in EN 1997 are implemented in the manager for National Annexes.



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

In order to perform a Pad Foundation check, you have to define 2 types of combinations:

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

After the calculation, a new class **GEO** will be generated automatically which contains all combinations of these 2 types.

Remark: The class is only generated in case the functionality **'Subsoil'** is activated.

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

Ground properties

With the correct Design Approach, the design values for the soil properties are determined:

φ'_d $= \text{atan}\,[\frac{\tan(\varphi')}{\gamma_{\varphi'}}]$

With: φ' read from Subsoil Library

$\gamma_{\varphi'}$ read from National Annex Setup

c'_d $= \frac{c'}{\gamma_{c'}}$

With: c' read from Subsoil Library

$\gamma_{c'}$ read from National Annex Setup

c_{ud} $= \frac{c_u}{\gamma_{cu}}$

With: c_u read from Subsoil Library

γ_{cu} read from National Annex Setup

γ'_d $= \frac{\gamma'}{\gamma_\gamma}$

With: γ' specific weight read from Library

γ_γ read from National Annex Setup

$\gamma_{Backfill,d}$ $= \frac{\gamma_{Backfill}}{\gamma_\gamma}$

With: $\gamma_{Backfill}$ weight read from Pad foundation input Data

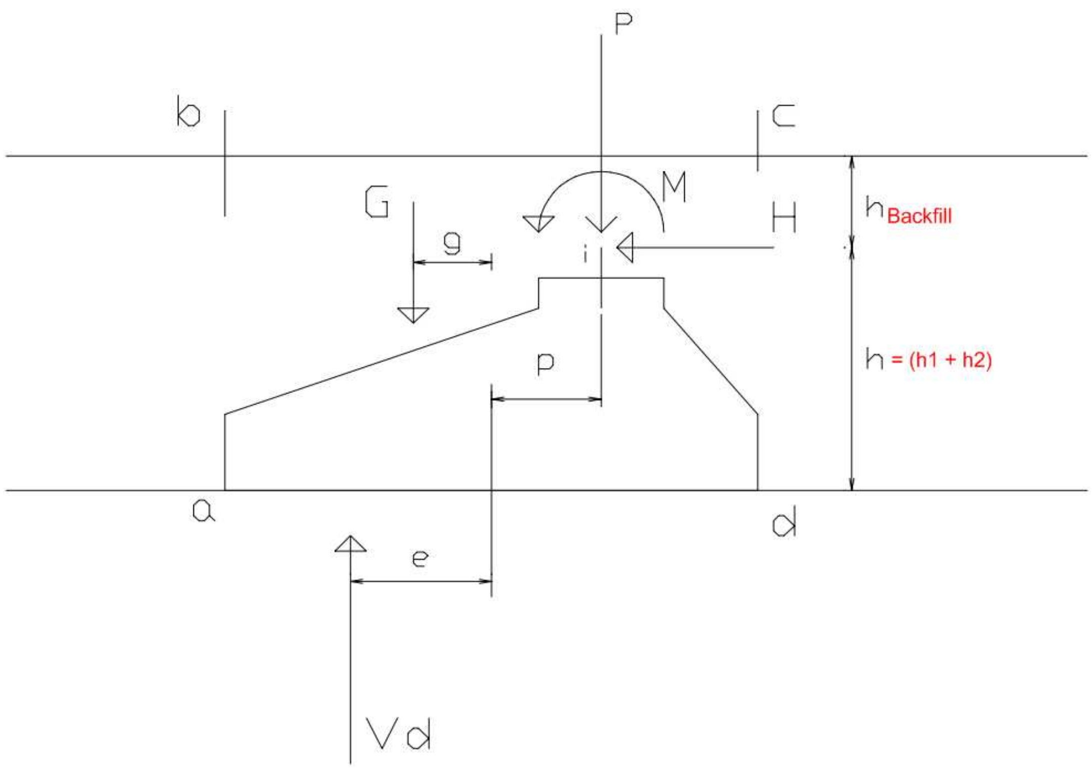
γ_γ read from National Annex Setup

γ_G 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 the first permanent load case for the combination under consideration i.e. γ_G. In case a combination does not have a permanent load case, γ_G is taken as 1,00.

Properties of the Pad Foundation

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:

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

Eccentricity e

The eccentricity **e** is 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) <u>The weight of the foundation block, $\mathbf{G_{Block}}$</u>

- This depends on the shape of the block (prismatic or pyramidal), dimensions and also the density γ_{Block} of the block material.
- The density of the block depends on the **Water table level**.

no influence	γ_{Block}
at foundation base	γ_{Block}
at ground level	$(\gamma_{Block} - \gamma_W)$

- The Water Density γ_W is taken as **9,81 kN/m³**

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

- This depends on the shape of the block (prismatic or pyramidal), dimensions and also the density of the backfill material.
- The backfill density $\gamma_{Backfill,d}$ is specified in Ground properties
- The density of the backfill depends on the **Water table level**.

no influence	$\gamma_{Backfill,d}$
at foundation base	$\gamma_{Backfill,d}$
at ground level	$(\gamma_{Backfill,d} - \gamma_W)$

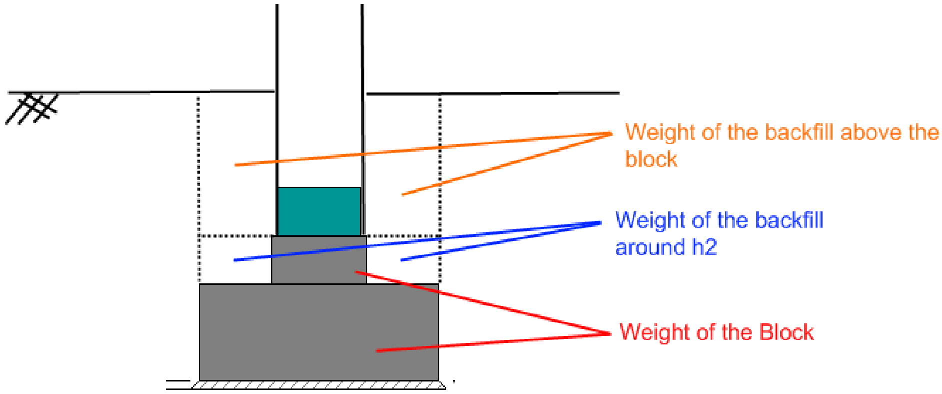
- The Water Density γ_W is taken as **9,81 kN/m³**

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

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

*In SCIA Engineer it is also possible to input a **negative height for the backfill material**. A negative value is used 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:

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

With γ_G the safety factor of the permanent loading for the combination under consideration, as defined in “Ground properties”.

Distances g_x & g_y

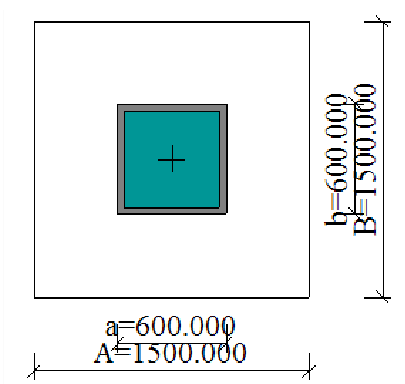
Using the weight and the volume, the center of gravity of the block and backfill are determined. The distances g_x and g_y are then calculated from this centroid to the center point of the foundation base.

Effective geometry

As a final step, using the eccentricities e_x and e_y, the effective geometry of the foundation base is calculated as follows:

$L_1 = A - 2 * |e_x|$
$L_2 = B - 2 * |e_y|$

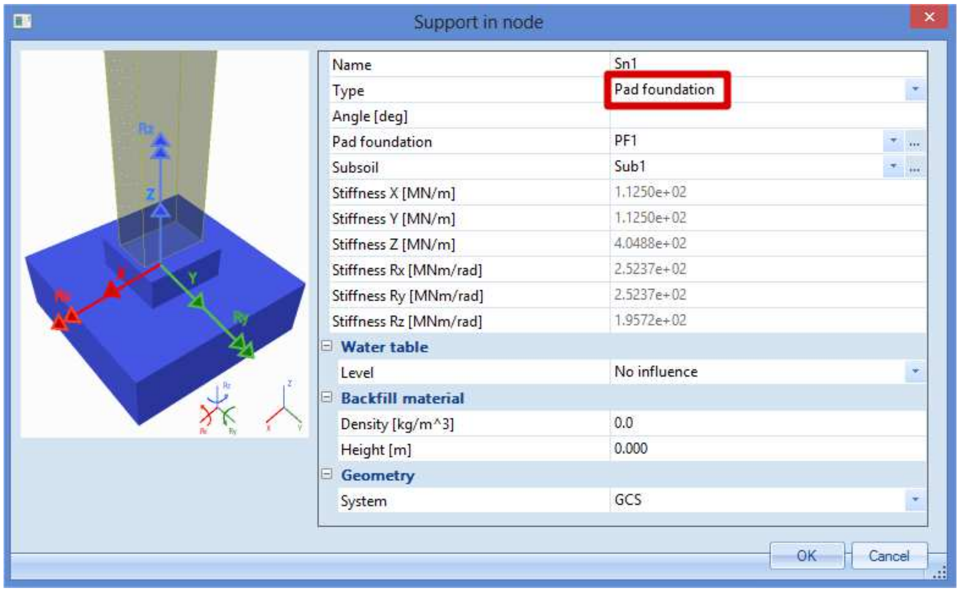
With A and B read from the Pad Foundation library:



$B' = \min (L_1 ; L_2)$
$L' = \max (L_1 ; L_2)$
$A' = B' * L'$

Remark: In case SCIA Engineer will find a value B’< 0 or L’< 0, the geometry is incorrect so the check is not executed and a warning is given on the output.

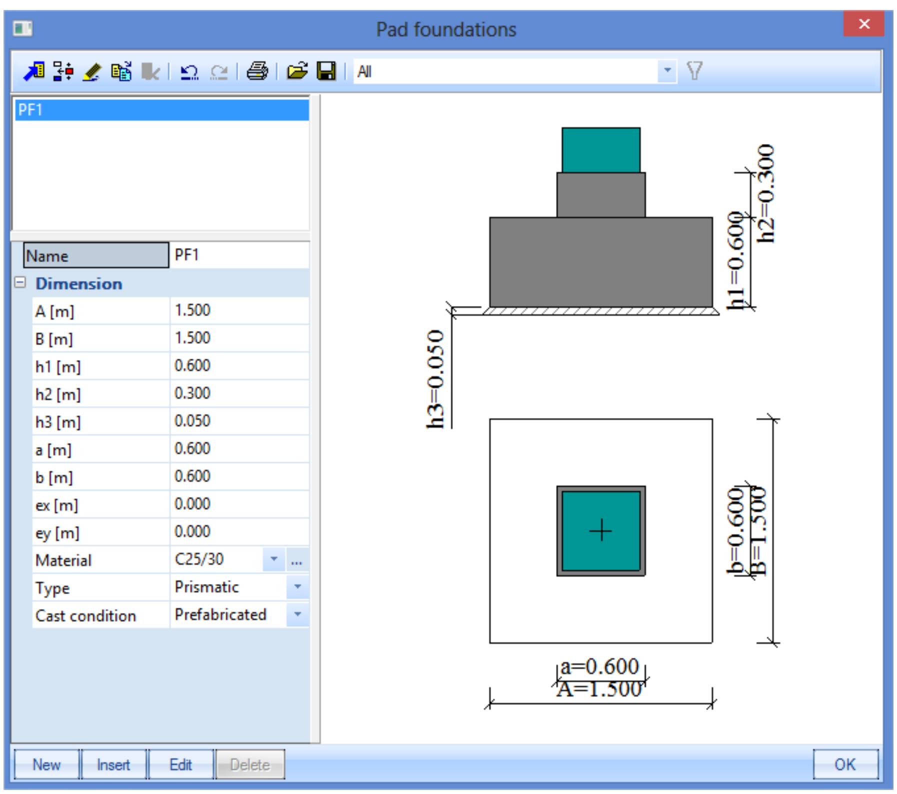
The option “Pad foundation” can be chosen in the Properties window of the supports:



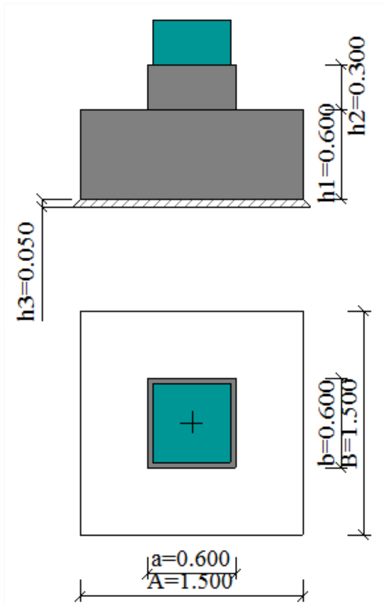
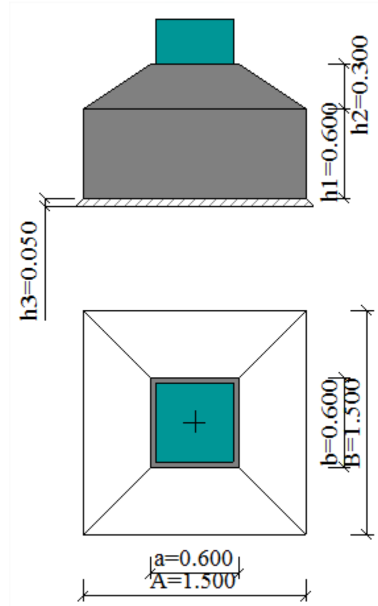
Here it is possible to input the influence of the water table and the properties of the backfill material. Also the type of the subsoil can be chosen under “Subsoil”.

The given stiffnesses of the Pad Foundation are automatically calculated by the program by formulas (see Annex 2).

The dimensions of the pad foundations can be inputted at the option “Pad foundation” (or you can open the Pad foundation library via Libraries > Subsoil, foundation > Pad foundations):

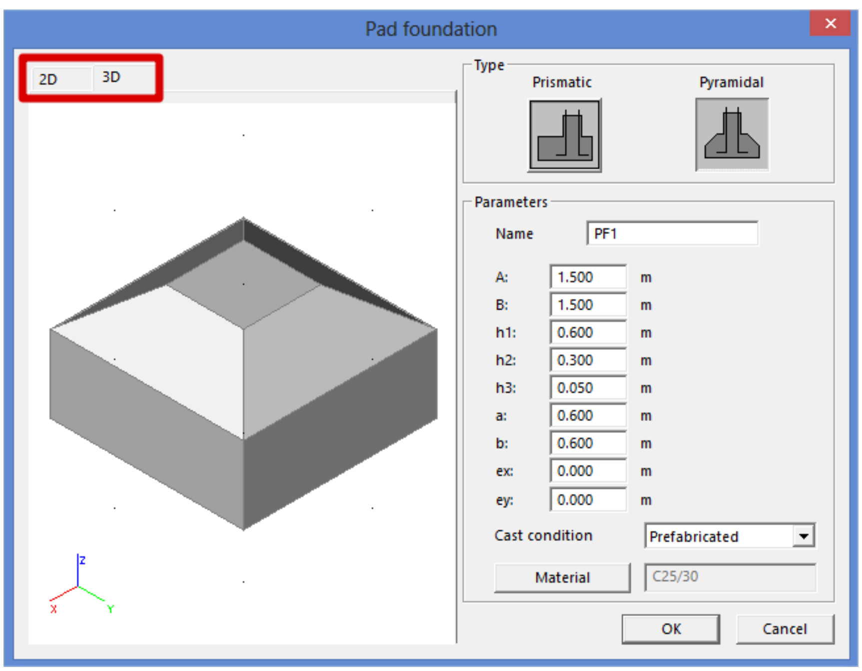


The user can choose from two variant shapes of pad foundations:

Prismatic	Pyramidal
	

When clicking ‘Edit’ the pad foundation can be displayed in 2D or 3D mode:

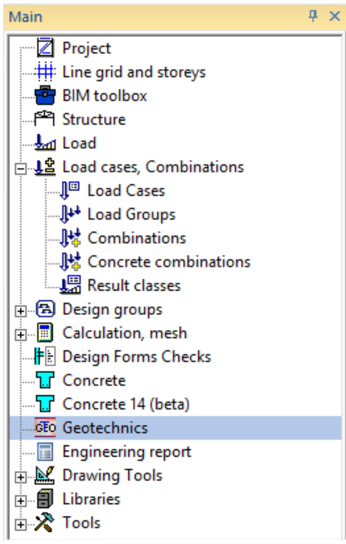
- The 2D mode shows side view, plan view and dimension lines for all input values.
- The 3D mode enables the user to make a good visualisation of the defined foundation block



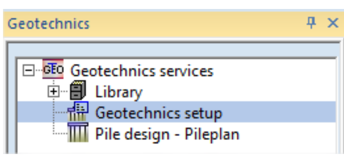
The input of dimensions can be performed with the interactive drawing of the block. That means that the user may click on a dimension line in the drawing and the corresponding item of the dialogue gets the focus. Therefore, it is very simple to specify the dimensions of the foundation block.

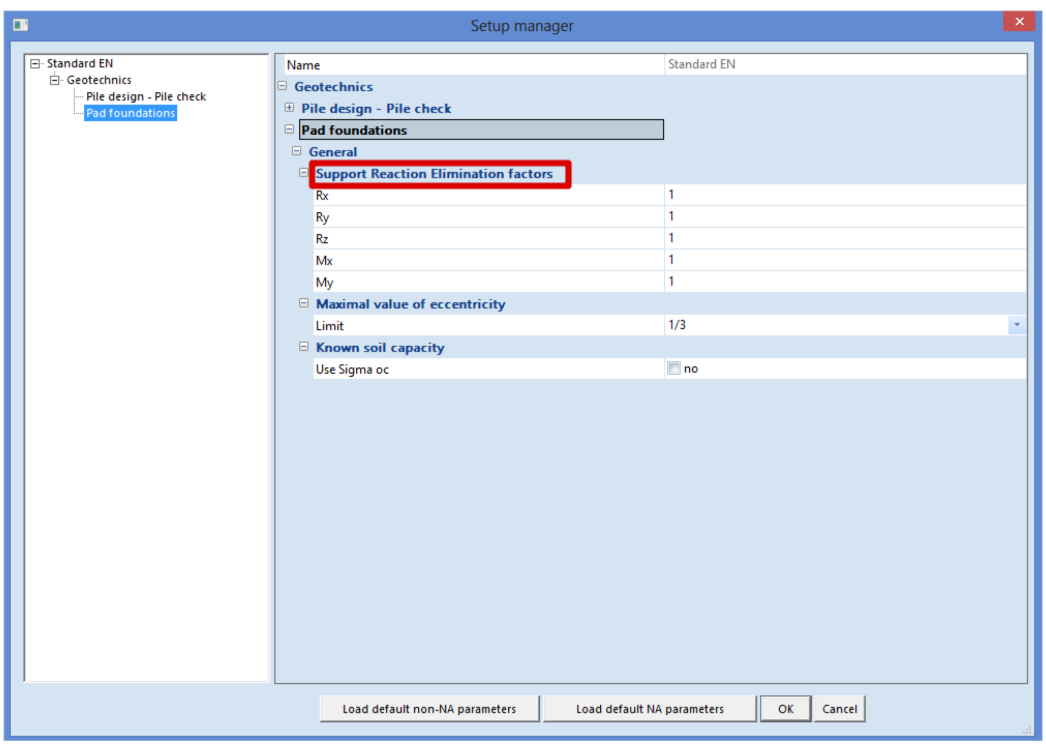
In SCIA Engineer **Support Reaction Elimination factors** can be defined in the Geotechnics setup.

Go to the menu “Geotechnics”:



And open the “Geotechnics setup”:





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 for example only 50% of a reaction should be used to design the pad foundation since the other 50% goes into the ring beam.

Pad Foundation checks

In general three separate checks are executed:

- Bearing check
- Sliding check
- Eccentricity check

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

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 is shown instead.

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

Bearing check

The Bearing check is executed according to **EN 1997-1 art. 6.5.2** and **Annex D.**

$$V_d \leq R_d$$

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

In case the user 'knows' the soil capacity, for example from a geotechnical report, $\mathbf{R_d}$ can be read directly from the input data instead of calculated.

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:

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

c_{ud} As specified in the National Annex Setup

b_c Inclination of the foundation base

In SCIA Engineer, the foundation base is always horizontal, thus: $b_c = 1{,}00$

s_c Shape of the foundation

In SCIA Engineer the foundation block has a rectangular shape, $s_c = 1 + 0{,}2 * \frac{B'}{L'}$

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 $\mathbf{i_c}$ is set to **0,5**

H_d Resulting horizontal load

$= \sqrt{H_x^2 + H_y^2}$

H_x Horizontal support reaction Rx as defined in “General”

H_y Horizontal support reaction Ry as defined in “General”

B’ Effective width as defined in “General”

L’ Effective length as defined in “General”

A’ Effective area as defined in “General”

q Overburden at the foundation base

= (h1 + h2 + $h_{backfill}$)* $\gamma_{Backfill,d}$

With:

h1 & h2 read from the Pad Foundation Library

$h_{backfill}$ read from the Pad Foundation input

$\gamma_{Backfill,d}$ as defined in ground properties

$\gamma_{R,v}$ Resistance factor read from the National Annex 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:

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

c_d’ As specified in the National Annex Setup

N_c Bearing resistance factor

$= (N_q - 1) * \cot(\varphi'_d)$

N_q Bearing resistance factor

$= e^{\pi * \tan(\varphi'_d)} * \tan^2(45 + \frac{\varphi'_d}{2})$

N_γ Bearing resistance factor

$= 2 * (N_q - 1) * \tan(\varphi'_d)$

b_c Inclination of the foundation base

In SCIA Engineer, the foundation base is always horizontal, thus: b_c = 1,00

b_q Inclination of the foundation base

In SCIA Engineer, the foundation base is always horizontal, thus: b_q= 1,00

b_γ Inclination of the foundation base

In SCIA Engineer, the foundation base is always horizontal, thus: b_γ = 1,00

s_c Shape of the foundation

In SCIA Engineer the foundation block has a rectangular shape,

$s_c = \frac{s_q * N_q - 1}{N_q - 1}$

s_q Shape of the foundation

In SCIA Engineer the foundation block has a rectangular shape,

$s_q = 1 + \left(\frac{B'}{L'}\right) * \sin(\varphi'_d)$

s_γ Shape of the foundation

In SCIA Engineer the foundation block has a rectangular shape, $s_\gamma = 1 - 0{,}3 * \frac{B'}{L'}$

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

$$= \left[1 - \frac{H_d}{V_d + A' * c'_d * cot(\varphi'_d)}\right]^m$$

i_γ 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 $$= \frac{\left[2 + \left(\frac{B'}{L'}\right)\right]}{\left[1 + \left(\frac{B'}{L'}\right)\right]}$$

θ Angle of the horizontal load H_d with the direction L'

φ'_d As specified in the Ground properties

B' Effective width as defined in general

L' Effective length as defined in general

A' Effective area as defined in General

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 Vertical reaction as specified in “General”

q'_d Effective overburden at the foundation base

$=(h1 + h2 + h_{backfill}) * \gamma'_t$

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:

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

$\gamma_{Backfill,d}$ as defined in Ground properties

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:

No influence	γ'_d
at foundation base	$(\gamma'_d - \gamma_W)$
at ground level	$(\gamma'_d - \gamma_W)$

γ'_d as defined in Ground properties

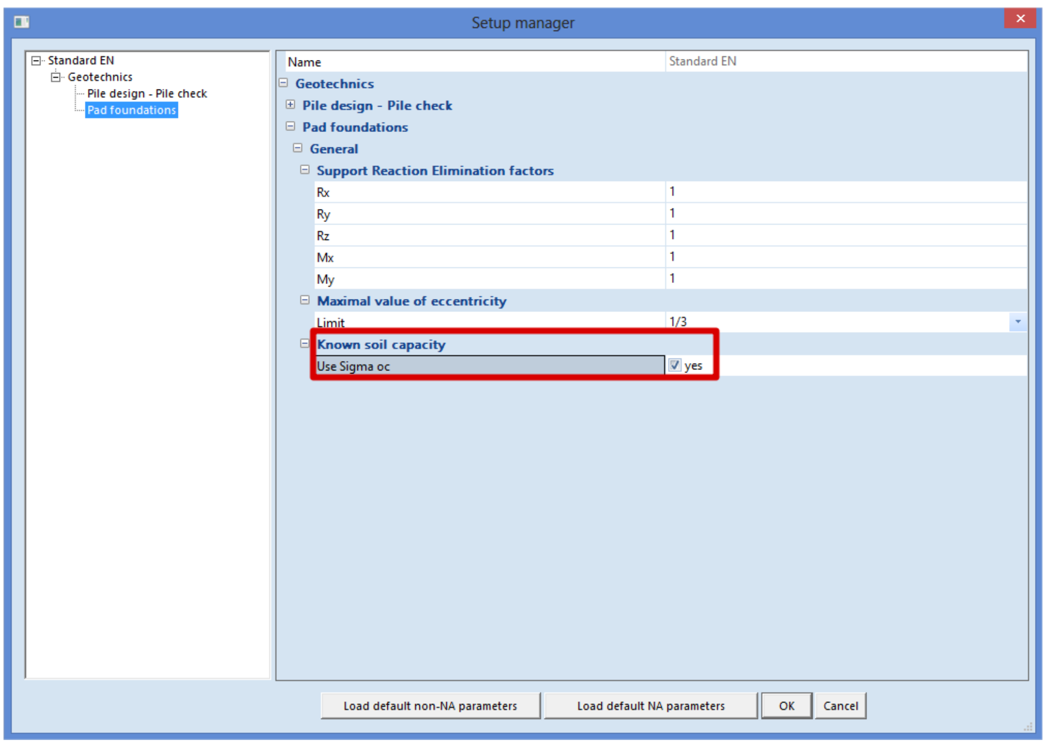
γ_W is taken as **9,81 kN/m³**

$\gamma_{R,v}$ Resistance factor read from the National Annex Setup

Known Soil Capacity Bearing Resistance

In case the Soil capacity is known, this value can be used directly instead of using the EN 1997-1 bearing resistance calculation outlined above.

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:

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

A' Effective area as defined in "general

σ_{od} Design value of the admissible soil capacity, taken as $\boldsymbol{\sigma_{oc}}$

σ_{oc} Read from the Subsoil Library

Sliding Check

The Sliding check is executed according to **EN 1997-1 art. 6.5.3** [Ref.1]

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

The Sliding resistance $\mathbf{R_d}$ depends on the fact if 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 in SCIA Engineer.

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**.

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

c_{ud}	As defined in Ground properties
A’	Effective area as defined in “General”
$\gamma_{R,h}$	Resistance factor read from the National Annex Setup

- In case the checkbox **Water/air in clay subgrade** in the Subsoil Library is activated, it means that it is possible for water or air to reach the interface between a foundation and an undrained clay subgrade. Following EN 1997-1 § 6.5.3(12), the value of $\mathbf{R_d}$ is limited as follows:

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

V_d	Vertical reaction as defined in “General”

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

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

V_d	Vertical reaction as defined in “General”
δ_d	Design friction angle at the foundation base
	Dependent on the **Cast condition** specified in the Pad Foundation Library:

Prefabricated	$\frac{2}{3} * \varphi'_d$
In situ	φ'_d

φ'_d	As specified in Ground properties
$\gamma_{R,h}$	Resistance factor read from the National Annex Setup

Eccentricity check

EN 1997-1 art. 6.5.4 specifies that special precautions are required for loads with large eccentricities:

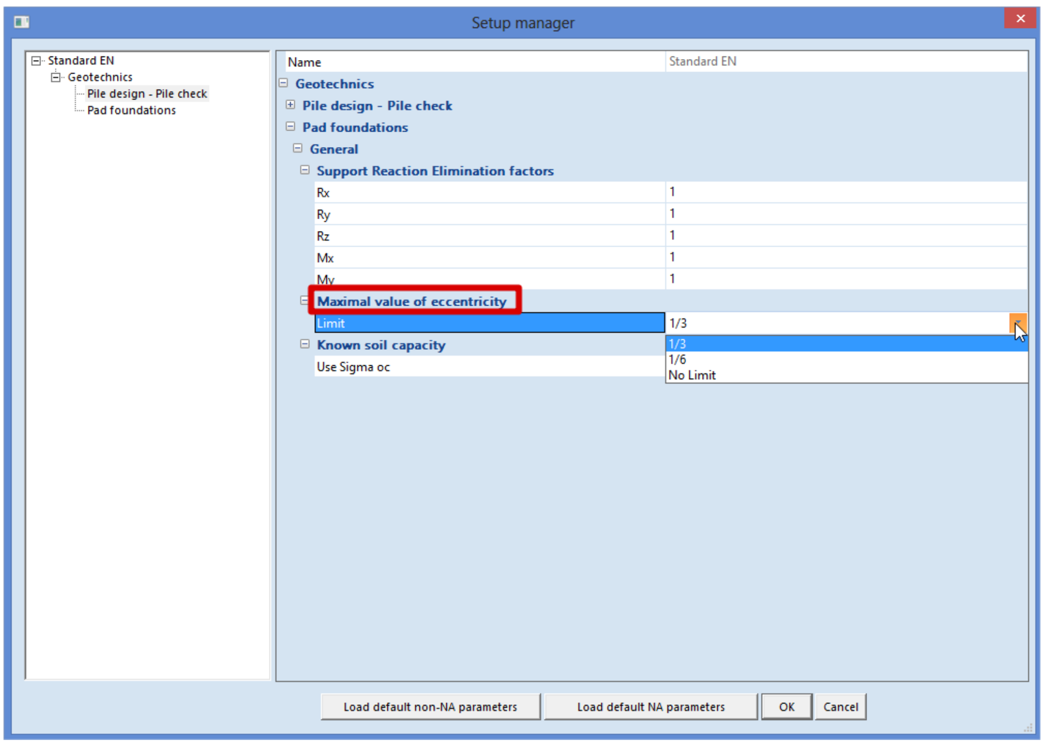
Special precautions shall be taken where the eccentricity of loading exceeds 1/3 of the width of a rectangular footing or 0,6 of the radius of a circular footing.
Such precautions include:

- *careful review of the design values of actions in accordance with 2.4.2*
- *designing the location of the foundation edge by taking into account the magnitude of construction tolerances.*

It is common practice (**although not required by EN 1997-1**) to put some limit on the eccentricity under characteristic values of actions.

This can done by checking if the design load is within a critical ellipse or critical diamond.
More specifically the eccentricity of the load should not exceed **1/3** or **1/6** of the width.

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



Based on the maximal value an eccentricity check is executed as follows:

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

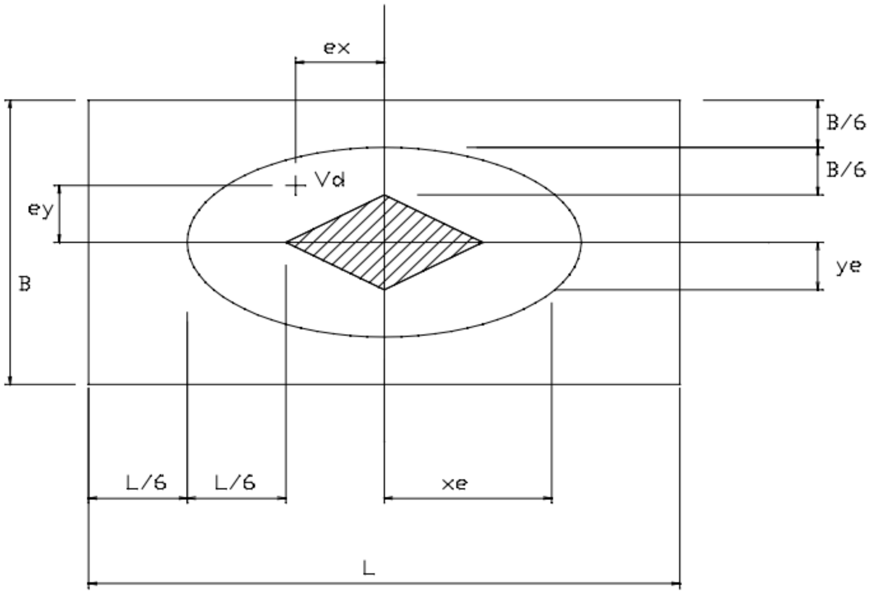
$$\left(\frac{e_x}{A}\right)^2 + \left(\frac{e_y}{B}\right)^2 \leq \frac{1}{9}$$

The eccentricity check of **1/3** takes into account that the pad foundation will not lose contact with the ground over more than half its width under the service loads.

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

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

e_x	As specified in “General”
e_y	As specified in “General”
A	Read from Pad Foundation Library
B	Read from Pad Foundation Library



The eccentricity check of **1/6** takes into account that the whole pad foundation is under pressure. The foundation will not lose contact with the ground over the whole area.

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 then set to **0,00**.

Following EN 1997-1 it is not required to put limits on the eccentricity calculation

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 is written out as follows and is **executed instead of the Bearing, Sliding and Eccentricity checks**:

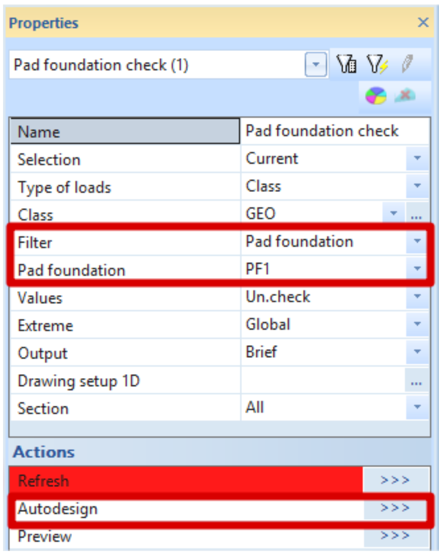
$$|P| \leq G_d$$

P:	The vertical **Rz** reaction as specified in “General”
G_d	The weight of the foundation and any backfill as specified in “General”

Pad foundation Autodesign

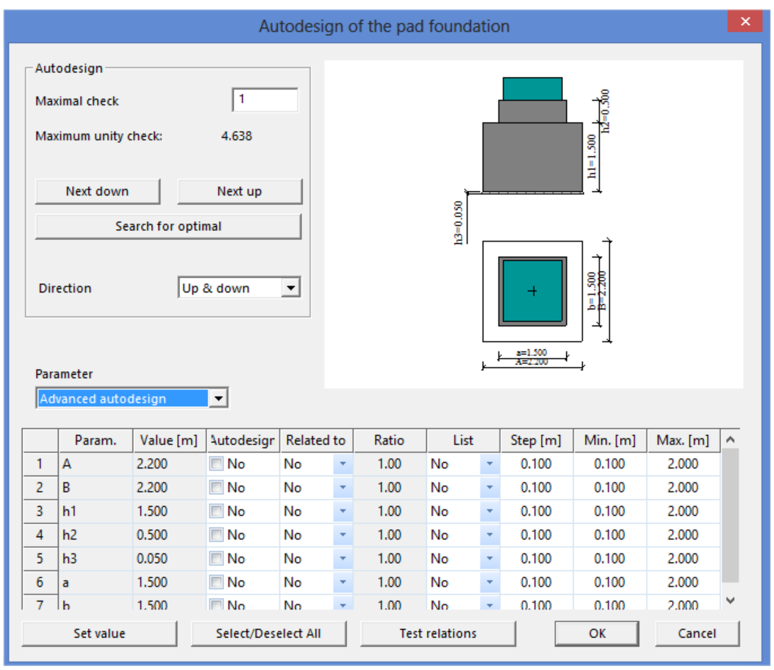
Autodesign of a concrete pad foundation is located in the actions window of Geotechnics > Geotechnics services > Pad foundation – stability check.

The autodesign can run after the calculation.



The filter will be automatically switched to Pad foundation when choosing Autodesign.

When starting the autodesign, the following dialogue is opened:



You can choose which parameter has to be considered in the autodesign. When you select the 'Advanced autodesign' multiple dimensions can be selected to be autodesigned.

Next step is to click 'Search for optimal' to find the optimal dimensions of the selected pad foundation. This means that the maximum unity check has to be smaller than 1.

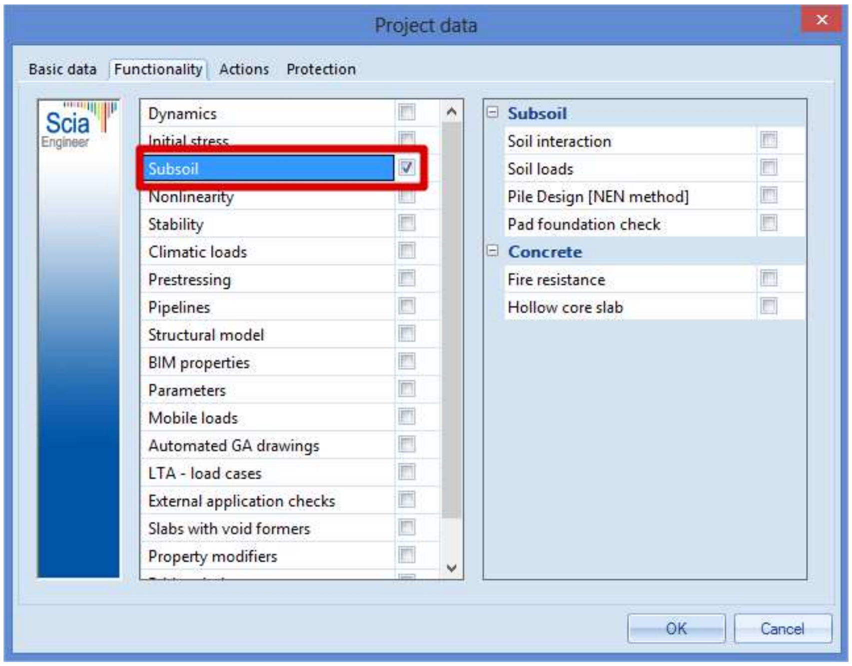
After clicking 'OK', the pad foundation is automatically replaced with the new designed one.

Foundation strips

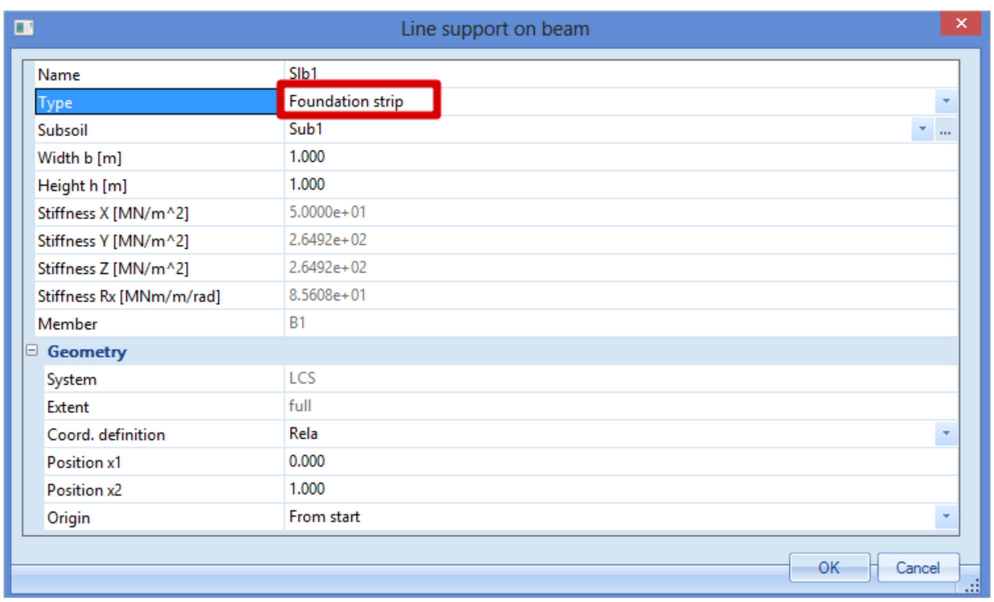
A linear support may be defined in the form of a foundation strip. The supporting is then specified by the properties and dimensions of the strip together with the properties of the soil below the footing surface.

Definition

Before you can define a foundation strip, you have to activate the functionality Subsoil.



Next you can insert a line support on beam and choose as Type the Foundation strip.



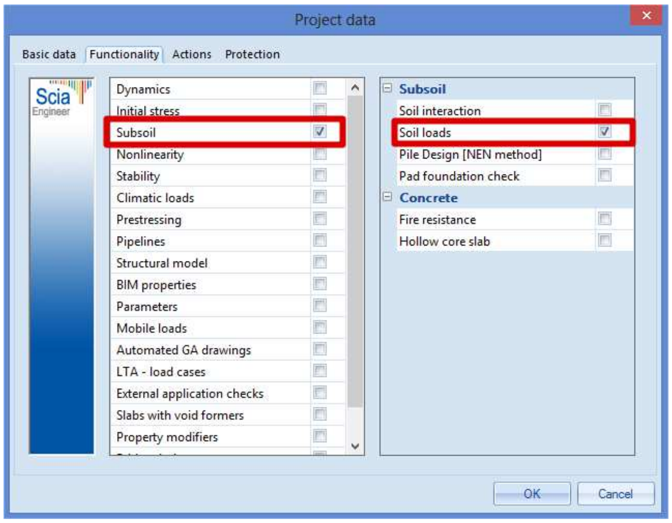
The stiffness of the foundation strip is defined by its width, height and the subsoil.

Geologic profiles, Geologic areas and Boreholes

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 extrapolated.

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

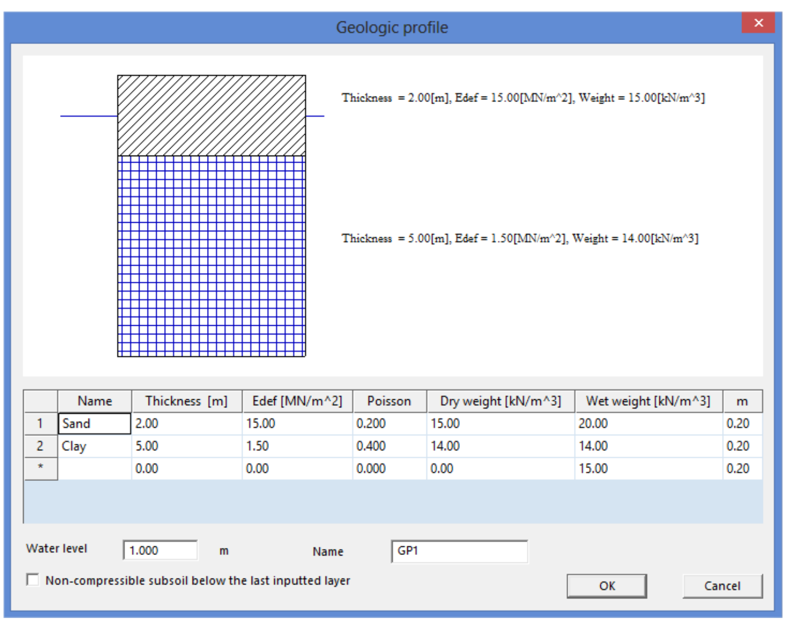
To insert geologic profiles, geologic areas and boreholes in SCIA Engineer, you have to check the functionality ‘Subsoil’ and ‘Soil loads’.



Geologic profile

You can define a new geologic profile in the Geologic profile manager via:

- Tree menu Libraries > Subsoil, foundation > Geologic profiles
- Menu Libraries > Subsoil, foundation > Geologic profiles



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	If 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 inaccuracies.

Layer-related parameters

Name	Name of the layer.
Thickness	Thickness of the layer.
Edef	Module of deformation (see Annex 3). 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 the value.
Poisson	Coefficient of transverse deformation (range: 0 – 0.5). An indicative value or experimentally found value can be used.
Dry weight	Specific soil weight for dry soil, normally within the range from 18 to 23 kN/m^3.
Wet 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.

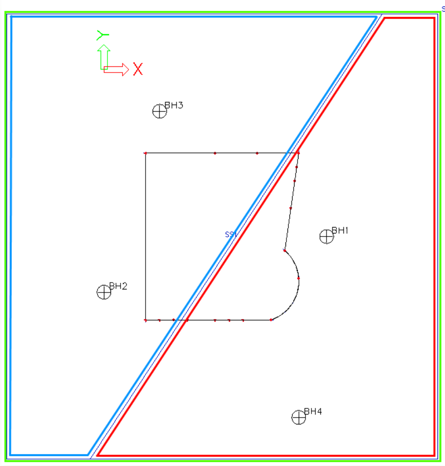
To edit the content of the table, it is possible to copy and paste the content from the clipboard.

Note: The 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.

Geologic area

The basic surface polygon has been divided to the separate areas which are inter- and extrapolated, but the first area does not 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 2 geologic areas is a geologic fault.



Green: basic outline of the subsoil surface

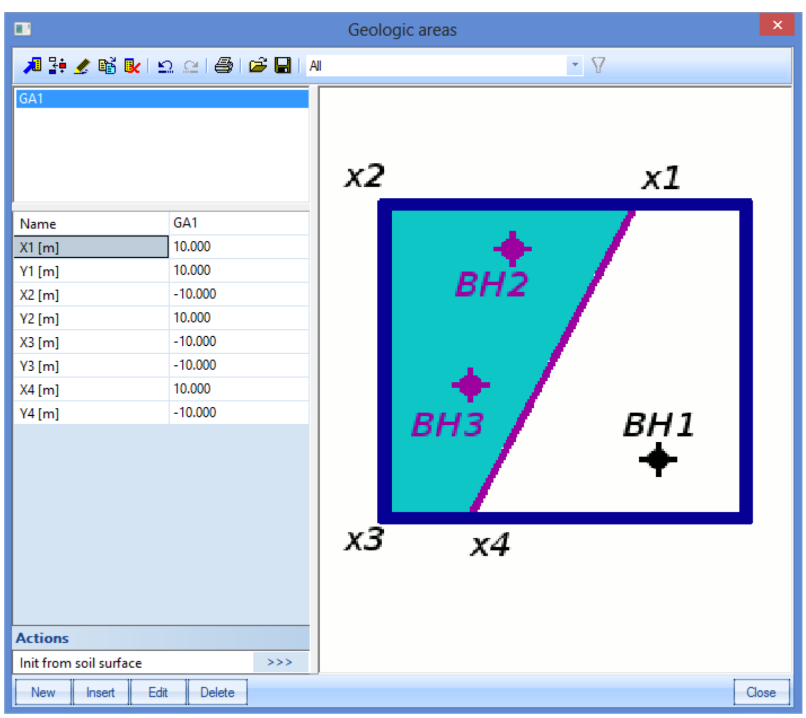
Red (right side): geologic area 1

Blue (left side): geologic area 2

Red-Blue line: geologic fault

A new geologic area can be defined in the Geologic area library which contains the geometry (4 points) and can be opened via:

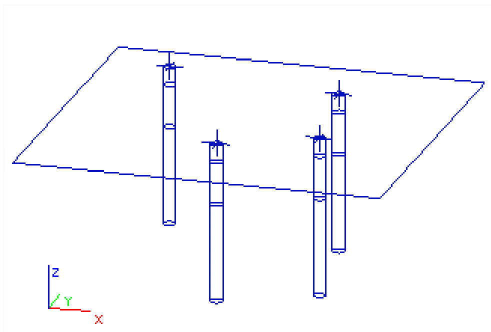
- Tree menu Libraries > Subsoil, foundation > Geologic areas
- Menu Libraries > Subsoil, foundation > Geologic areas



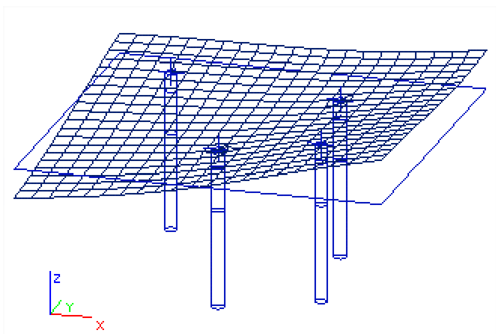
Boreholes

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 surroundings. This surface can be used for impressive presentations of projects. The surface itself is not taken into account during the calculation.

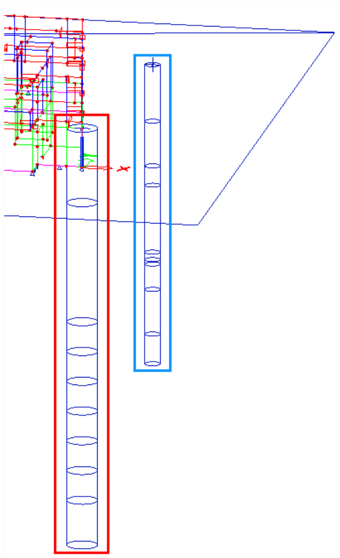
The following picture shows an example of defined boreholes. The rectangle represents the patch of land over which the soil properties can be inter- and extrapolated.



Next picture shows the calculated surface.



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

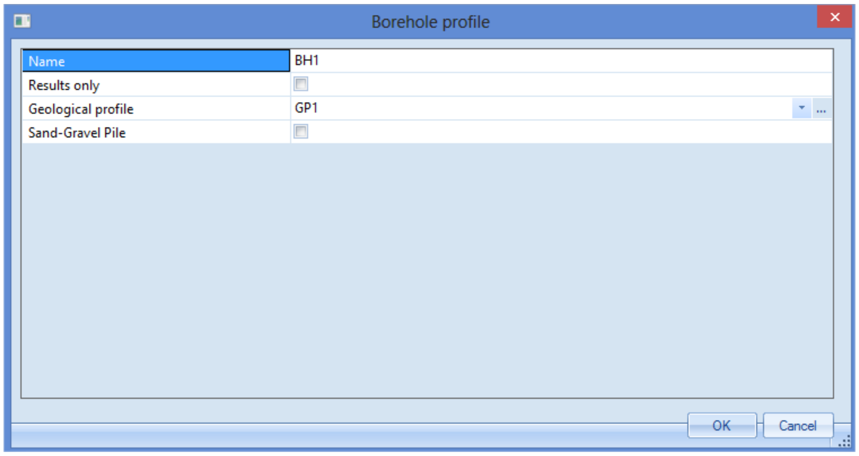


Red (left side): sand-gravel pile with diameter 1m

Blue (right side): standard borehole

Both are displayed inside the subsoil surface outline

A new borehole can be defined via menu Structure > Model data > Borehole profile.



Borehole parameters

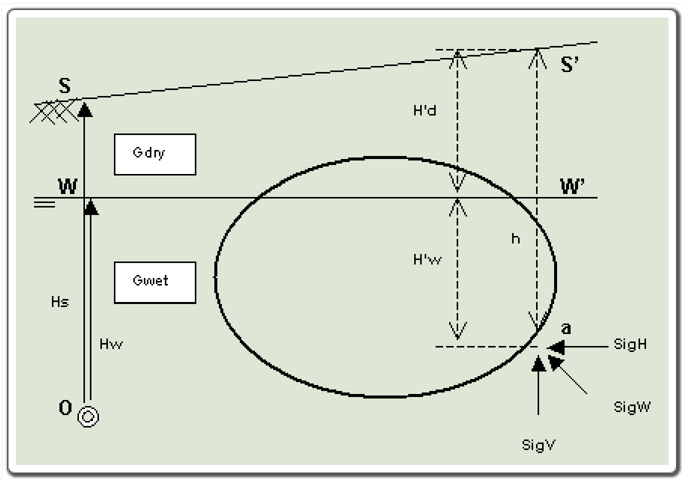
Name	Identifies the borehole profile.
Coord X, Y, Z	Coordinates of the inserting point of the borehole.
Results only	When the calculation is performed, you can obtain a table of settlement. The values of settlement are calculated in places where boreholes are located. The borehole itself (the corresponding 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.

Note: After some modification (especially modification of the position) of the borehole, it may be necessary to refresh the surface.

Soil pressure and water pressure

Several types of load (point force, line load and surface load) can be defined as what is called "soil pressure" or "water pressure ". Both loads are quite related and will be explained together.

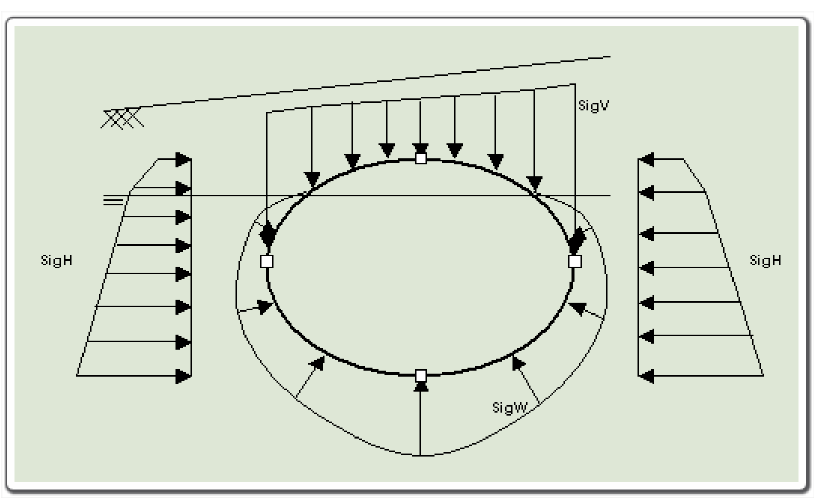
Both load types appear only if a structure is located underground. Depending on the surrounding soil, level of underground water and depth below the surface, the program automatically calculates the soil pressure and water pressure.



In depth h (point a), the intensities of the generated loads are:

SigV,a	If a is located above water level (h <= H'd), then (h * Gdry) If a is located below water level (h > H'd), then (H'd * Gdry + H'w * Gwet) It works ONLY in the negative direction of global Z-axis!
SigH,a	SigH,a = SigV,a * k0
SigW,a	If a is located above water level (h <= H'd), then (0) If a is located below water level (h > H'd), then (H'w * Gwater)

This would lead to a distributed load as in the image below:



Water and soil loads can be inputted for the following load cases:

- action type = "permanent" and load type = "standard",
- action type = "variable" and load type = "static".

The procedure to input soil / water pressure:

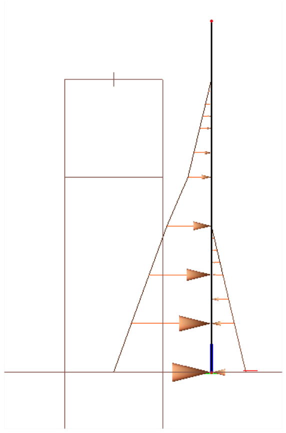
1. Open service Load.
2. Start function the required load type (point, line, surface).
3. Adjust the parameters - see below.
4. Confirm with [OK].
5. Apply the load on required entities.

Soil / water load parameters

In addition to common parameters for point, line and slab load, this load type requires the input of the following data:

Type	Must be set to Soil pressure or Water pressure.
Distribution	Only for line load. The line load may be uniform or trapezoidal.
Acting area	Only for point load. Defines the acting area for the load.
Acting width	Only for line load. Defines the acting width for the load.
Coefficient	Only for soil pressure. This coefficient must be defined for horizontal soil pressure. It specifies the ration between vertical and horizontal soil pressure. (I.e. for vertical pressure it should be equal to 1).
Borehole profile	Specifies the borehole that is used for the generation of the pressure.

The soil / water pressure is displayed as shown in the picture below.



Both are generated (orange) loads. The generated soil pressure (left part) reaches just to the top of the borehole (that was used as the reference borehole). The generated water pressure (right part) is defined only below the level of underground water. So if the whole model is above the water level, no pressure is generated at all.

The calculation considers these generated loads.

Soil-In

The analysis of foundation structures is challenged by the problem of modeling of the part of the foundation that is in contact with subsoil. The best solution is to use a 2D model of the subsoil that properly represents the deformation properties of the whole under-foundation massif by means of a 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.

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 has impact on the settlement of the footing surface and thus the C parameters, it is necessary to use an iterative solution.

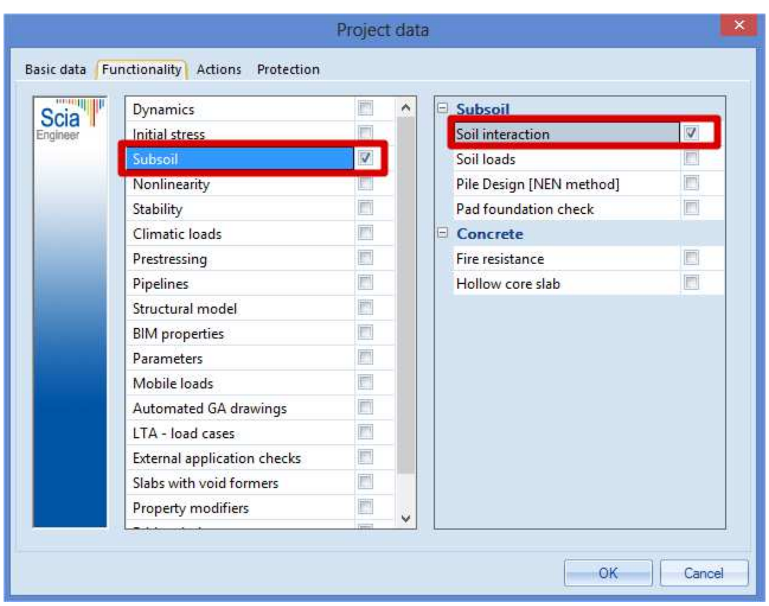
The results from the soil-in iteration are the C-parameters C1z, C2x and C2y. The parameters C1x and C1y are always defined by the user.

- 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]

Note: 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.

Soil-in calculation

The soil-in calculation is available when the functionalities Subsoil and Soil interaction are active.



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

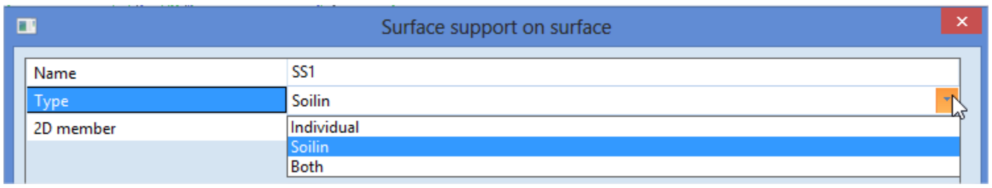
Subsoil in the 3D model

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

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

Surface support

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



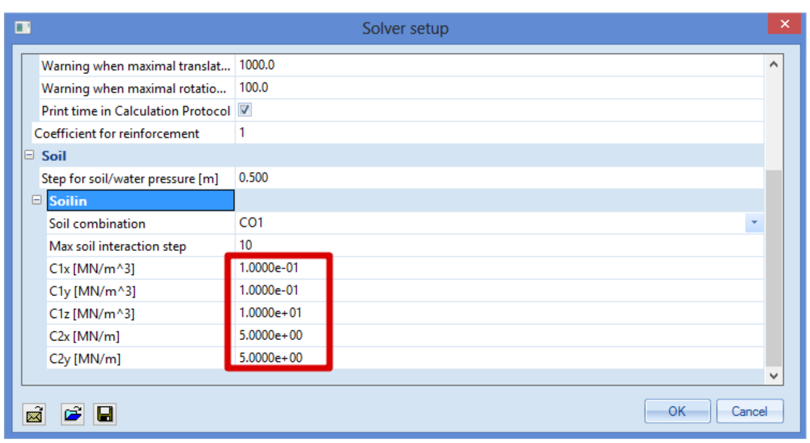
Name: Specifies the name of the support.
Type: Defines the type of support – see below.
Subsoil: If necessary for the selected type, this item specifies the subsoil parameters.

- **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 (e.g contact stress of the foundation surface)

- **Soil-in:**

  For such a support, the interaction of the structure with the foundation subsoil is carried out by means of the SOIL-IN module.
  All initial values of C parameters are defined in the Solver setup.



  Parameters C1z, C2x, C2y are calculated by SOIL-IN module, C1x and C1y are taken from the solver setup.

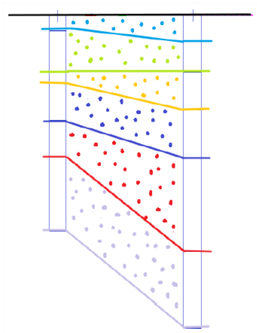
- **Both:**

  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. The parameters C1z, C2x and/or C2y that are set in the subsoil-property

dialogue as zero will be calculated by the SOIL-IN module. Nonzero parameters will be taken as they are inputted in the Subsoil. Parameters C1x and C1y are always defined by the user.

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.



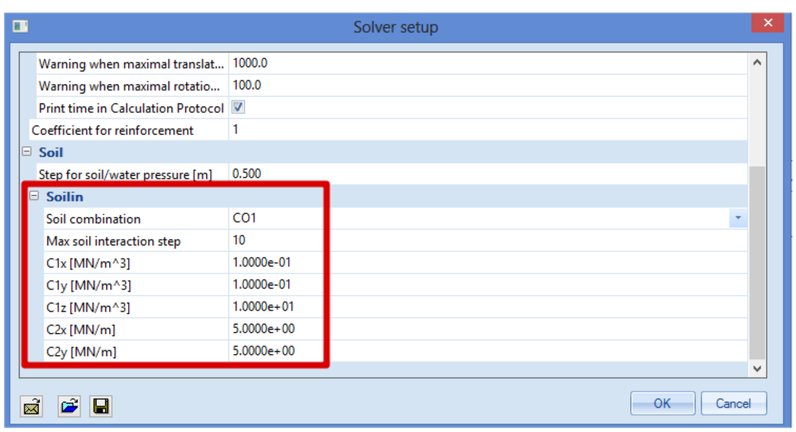
If there is some layer missing in one borehole, then it can be substituted by a layer with minimum thickness – e.g. 1mm so the soil-in has appropriate number of layers for approximation

Settings

There are some parameters that are required in a project in order to do a Soilin-in calculation:

- 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)

There are also several settings for Soilin 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 parameters calculated, in case those results diverge), the max. limit is 99 steps.

C1x, C1y: parameters defined by the user.

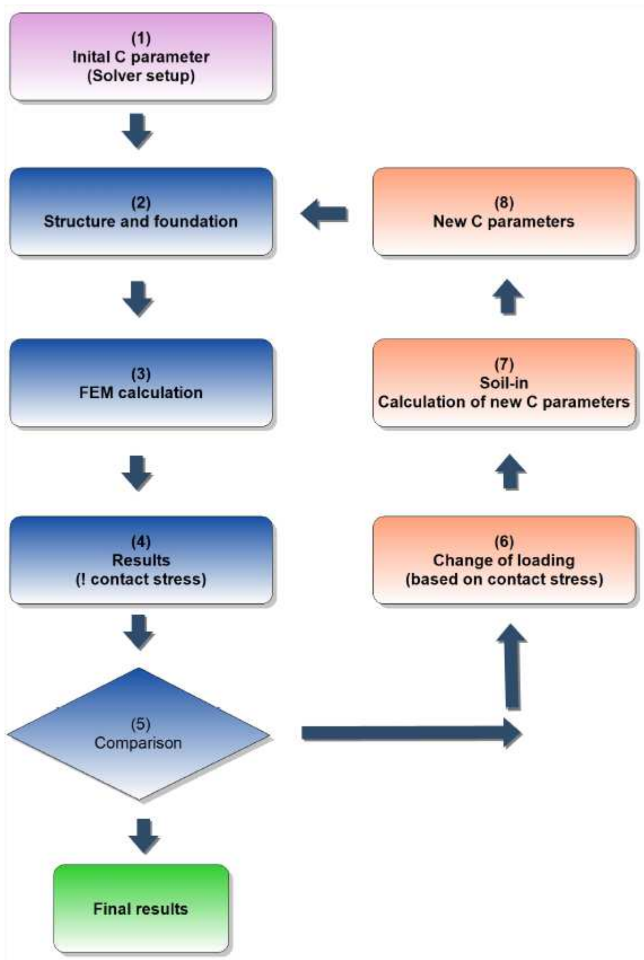
C1z, C2x, C2y: initial values for soil-in (if the support type is Soilin).

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.

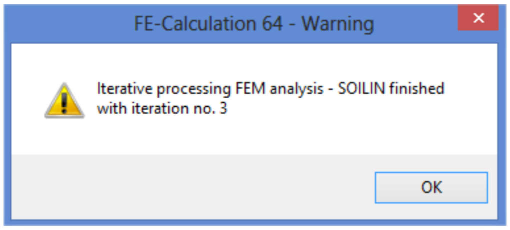
Diagram of the iterative cycle:



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 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.

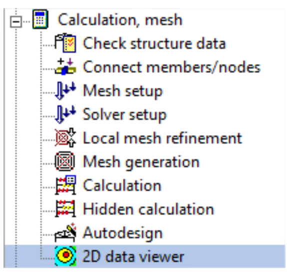
There is a message when the last iteration is done.

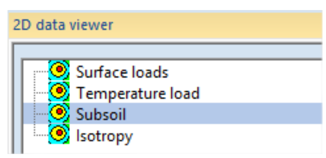


Results of soil-in

2D data viewer

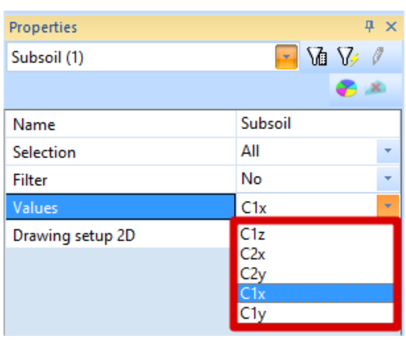
In the “Calculation, mesh” service is the 2D data viewer available to see some results for subsoil.



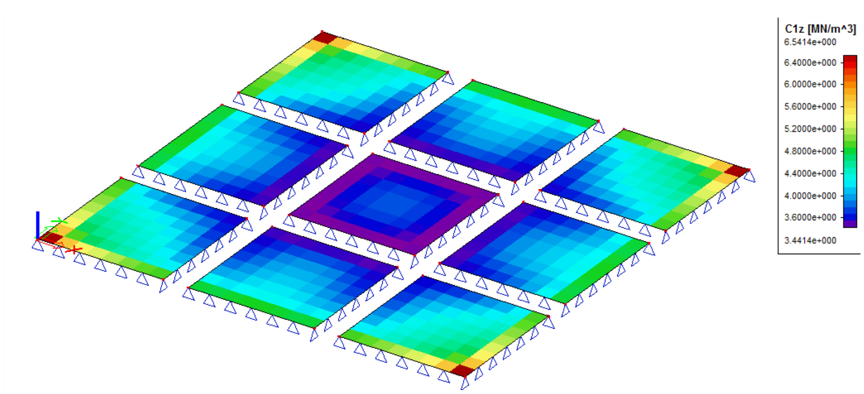


The C parameters are calculated for the mesh on the 2D member. It is displayed by the colour planes.

The results can be displayed for each C parameter.



Following is an example of the calculated C1z:

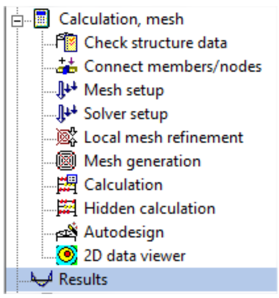


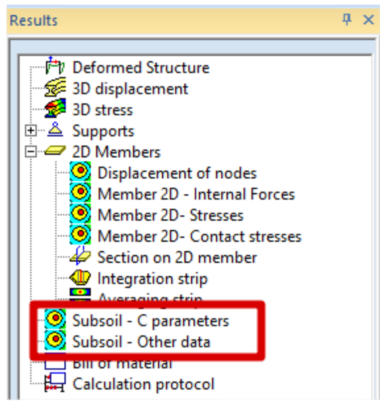
Also the preview with C parameters in the table can be displayed in the 2D data viewer.

Results menu

The service results contains two result previews:

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

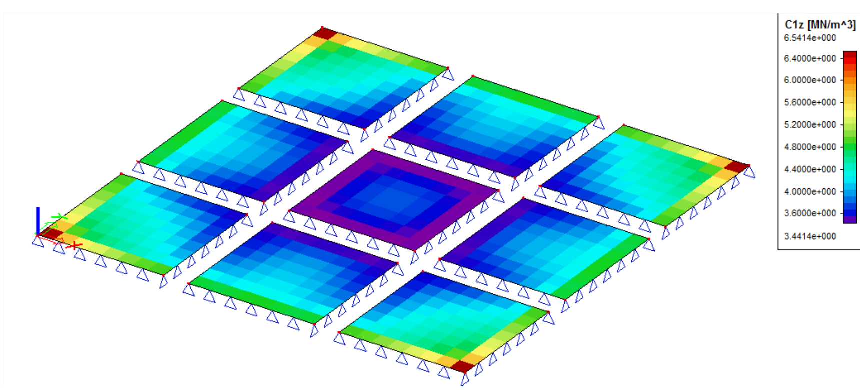


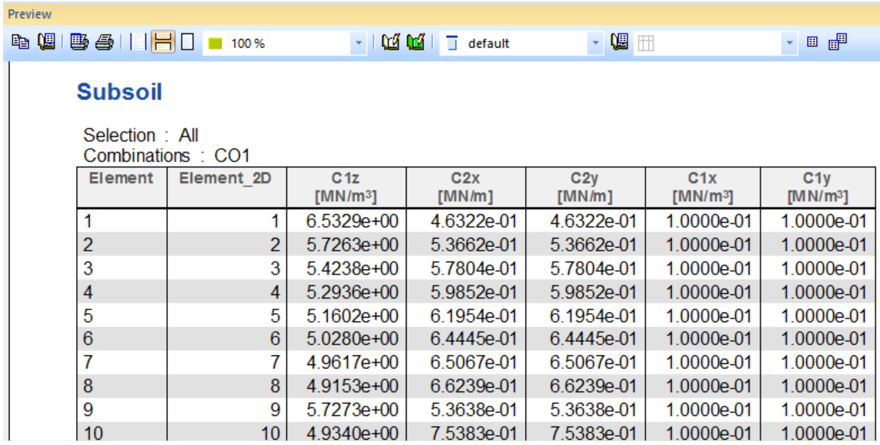


C parameter results

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





Subsoil

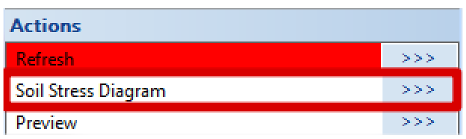
Selection : All
Combinations : CO1

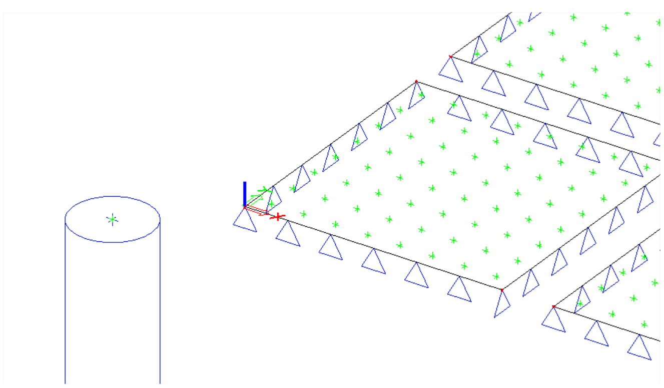
Element	Element_2D	C1z [MN/m³]	C2x [MN/m]	C2y [MN/m]	C1x [MN/m³]	C1y [MN/m³]
1	1	6.5329e+00	4.6322e-01	4.6322e-01	1.0000e-01	1.0000e-01
2	2	5.7263e+00	5.3662e-01	5.3662e-01	1.0000e-01	1.0000e-01
3	3	5.4238e+00	5.7804e-01	5.7804e-01	1.0000e-01	1.0000e-01
4	4	5.2936e+00	5.9852e-01	5.9852e-01	1.0000e-01	1.0000e-01
5	5	5.1602e+00	6.1954e-01	6.1954e-01	1.0000e-01	1.0000e-01
6	6	5.0280e+00	6.4445e-01	6.4445e-01	1.0000e-01	1.0000e-01
7	7	4.9617e+00	6.5067e-01	6.5067e-01	1.0000e-01	1.0000e-01
8	8	4.9153e+00	6.6239e-01	6.6239e-01	1.0000e-01	1.0000e-01
9	9	5.7273e+00	5.3638e-01	5.3638e-01	1.0000e-01	1.0000e-01
10	10	4.9340e+00	7.5383e-01	7.5383e-01	1.0000e-01	1.0000e-01

Soil stress diagram

The settlement is calculated for each mesh element (in its centre 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.

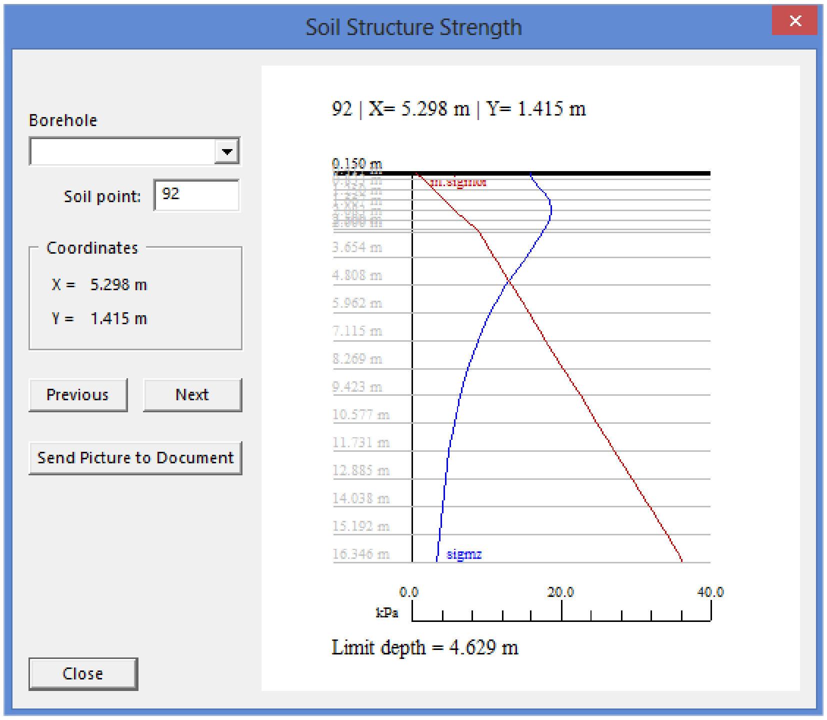
The points for the settlement calculation are shown when selecting Soil Stress Diagram in the actions window of Results > Subsoil – Other data.





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

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 boreholes. User just selects the point and the diagram is displayed.



- **Previous:** displays the Soil Structure Strength for the previous node
- **Next:** displays the Soil Structure Strength for the next node
- **Borehole:** displays 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

The Soilin module calculates two stresses: the overstress Sigma,z and the original soil stress Sigma,or. According to theory, settlement will occur if Sigma,z > m * Sigma,or.

The m-value is code dependent: (i) for the CSN code it can vary, for EC & DIN it is fixed at 0,2. It practically means that settlement occurs in case the overstress is bigger than 20% of the original soil stress.

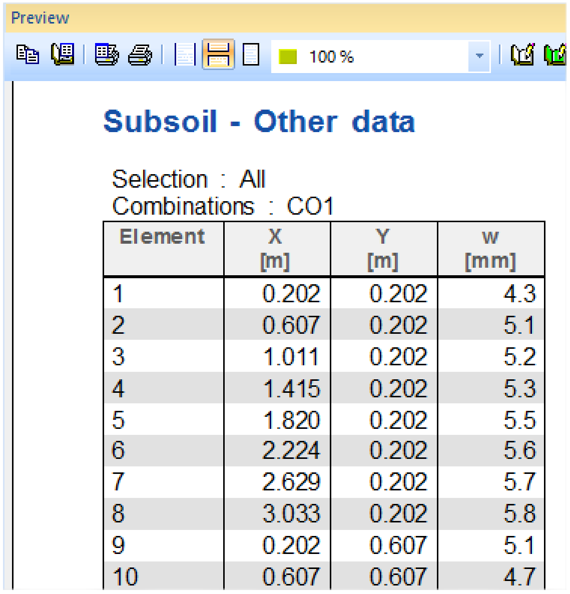
The picture shows these two lines: Sigma,z in blue and m * Sigma,or in red. The program is looking for the intersection of the two lines: all layers above have Sigma,z > m * Sigma,or and settlement occurs in them and all the layers below have Sigma,z < m * Sigma,or, which means that no settlement is there. The depth at which the lines intersect is called the "limit depth"

In case the user has not input a sufficient geological profile i.e. not deep enough, the intersection point cannot be determined. It means that the calculated settlement will be too small since there are still deeper layers which will also be compressed and will thus settle. Therefore, the program gives a warning that the geology is "Insufficient".

Settlement table

The table is displayed in the Subsoil – other data results. The preview table contains 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 the structure and from the penultimate iteration. Therefore it is useful to watch values w only outside the foundation (see chapter additional plates to check the settlement around the surface support).



Preview

Subsoil - Other data

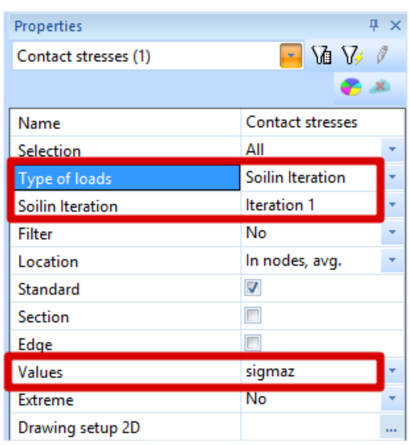
Selection : All
Combinations : CO1

Element	X [m]	Y [m]	w [mm]
1	0.202	0.202	4.3
2	0.607	0.202	5.1
3	1.011	0.202	5.2
4	1.415	0.202	5.3
5	1.820	0.202	5.5
6	2.224	0.202	5.6
7	2.629	0.202	5.7
8	3.033	0.202	5.8
9	0.202	0.607	5.1
10	0.607	0.607	4.7

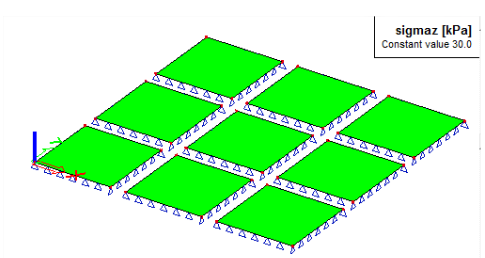
Results for each iteration cycle

When the soil-in does not 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.

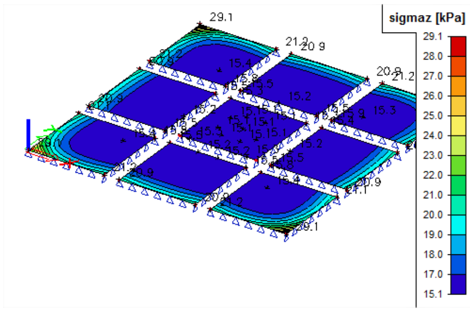
The calculated contact stresses for each iteration cycle can be found in the results.



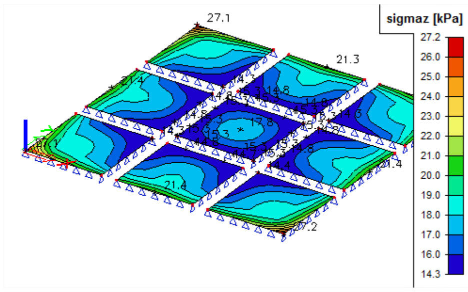
First iteration cycle:



Second iteration cycle:



Third iteration cycle:

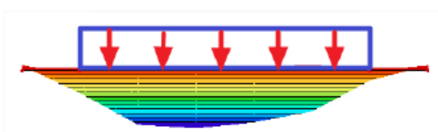


Additional plates

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

<u>About C parameters:</u>

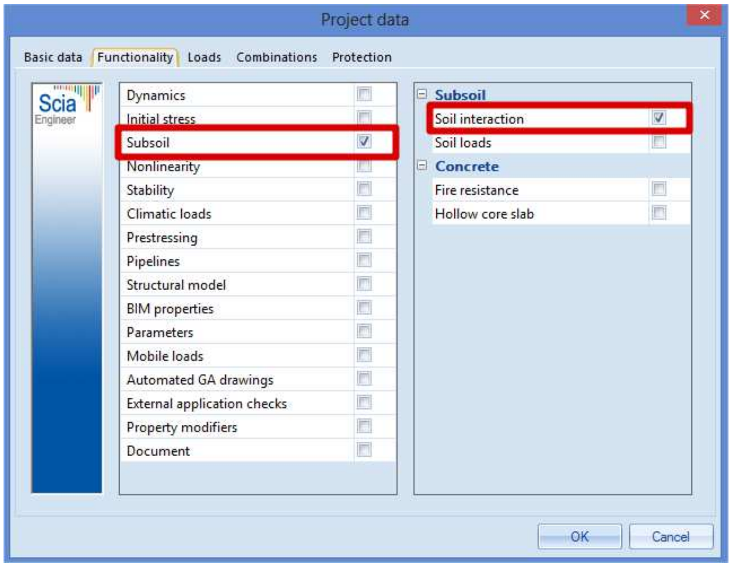
1. C parameters are parameters of interaction, so their value depends on the structure, load, stiffness and subsoil. Change in any of those parts causes different C parameters.
2. The whole plate is supported vertically by the soil stiffness – parameter C_1 (winkler) and also in the shear direction – 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.



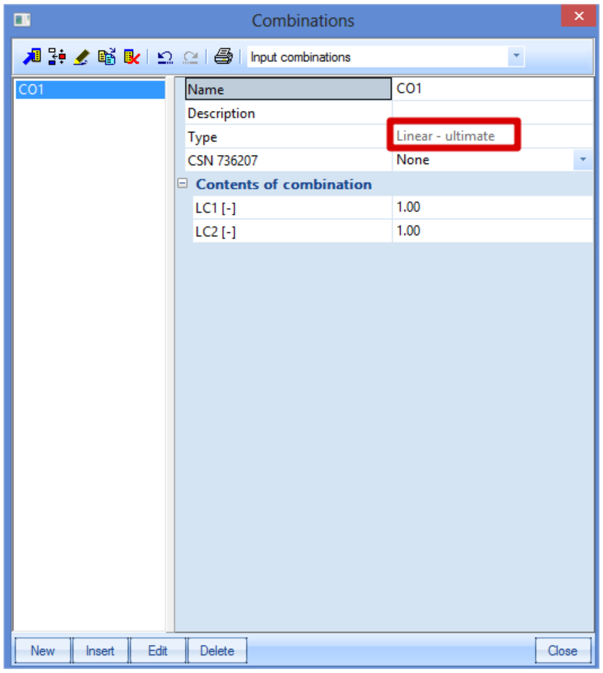
5. The degrease basin can be substituted by spring supports around the plate – this is done automatically in SCIA Engineer when user does not add plates around.
6. When user uses the plates around the support, the springs are not added and the C parameters are calculated for the whole area.

Settings for soilin calculation

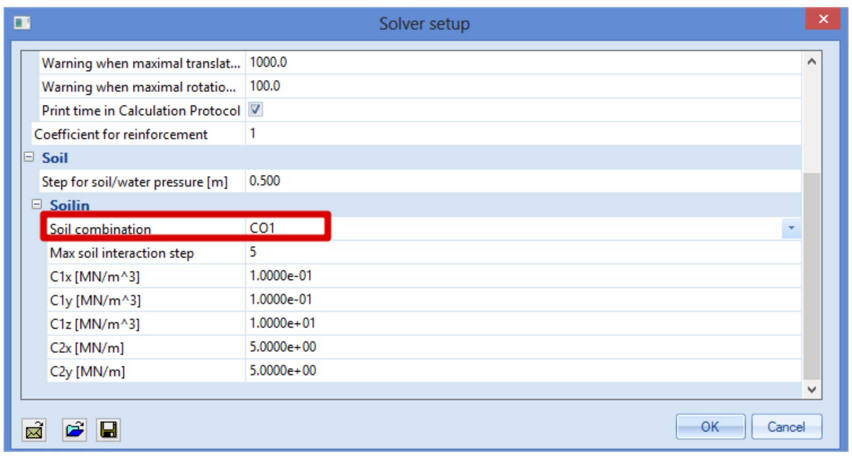
1. The functionality Subsoil and Soil iteration must be checked.


Project data
Basic data
Functionality
Loads
Combinations
Protection
Scia
Engineer
Dynamics
Initial stress
Subsoil
Nonlinearity
Stability
Climatic loads
Prestressing
Pipelines
Structural model
BIM properties
Parameters
Mobile loads
Automated GA drawings
External application checks
Property modifiers
Document
Subsoil
Soil interaction
Soil loads
Concrete
Fire resistance
Hollow core slab
OK
Cancel

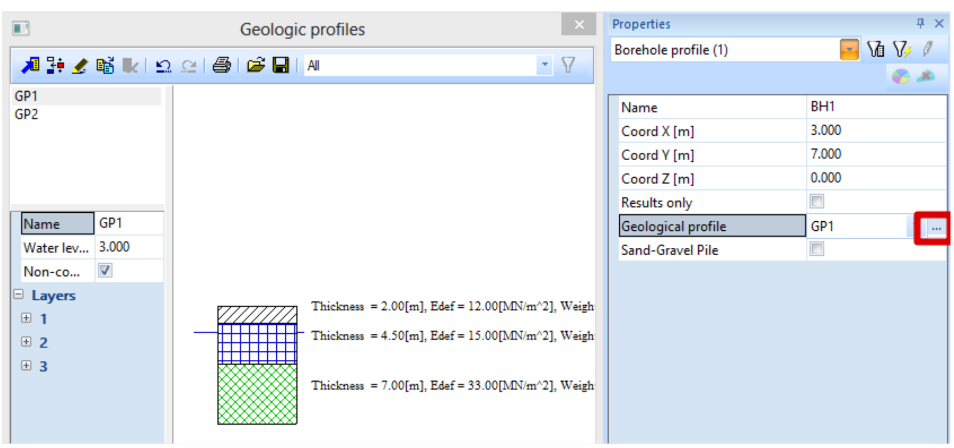
2. One combination must be linear - this combination is used for soilin calculation.



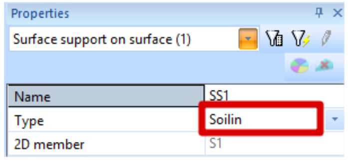
3. This linear combination must be selected in Solver setup to run soilin with it.



4. The project must contain a borehole with geologic profile.

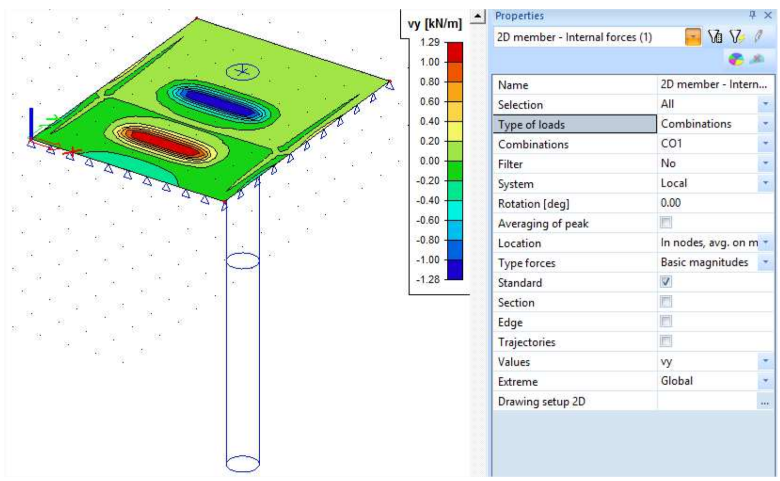


5. The project must contain a surface support type soilin.



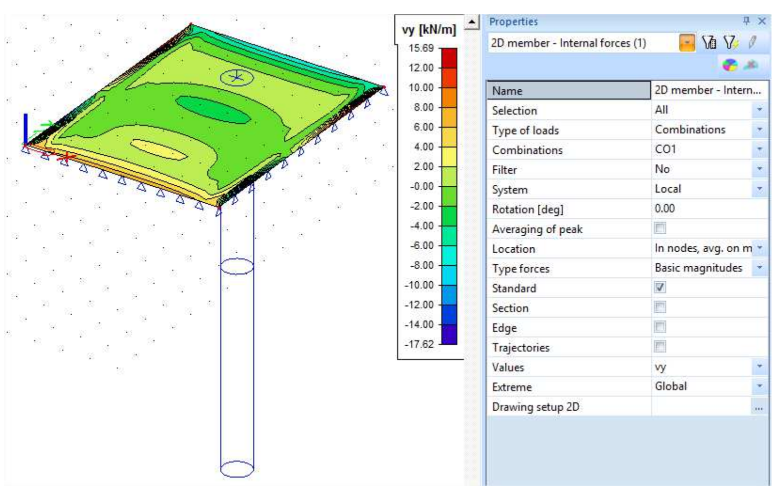
How to calculate the plate without soilin

1. Open the project “soilin_start.esa”.
2. There is one plate with the surface support type Individual. This type of the support has constant parameters C1 and C2.
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:

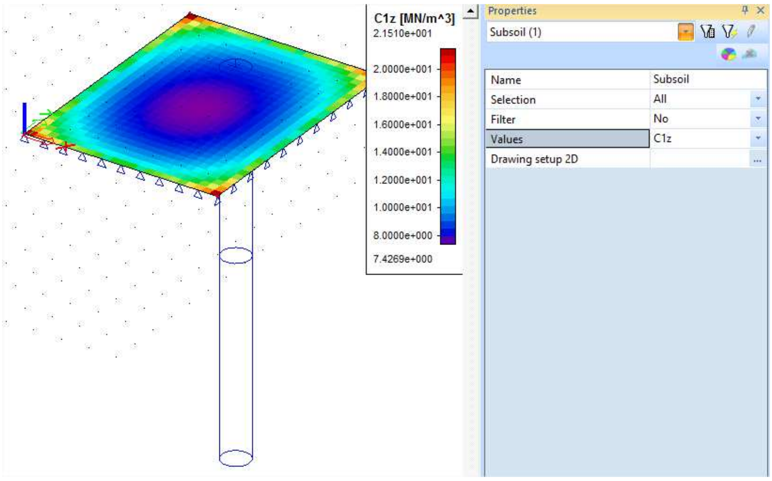


How to calculate the plate with soilin

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:



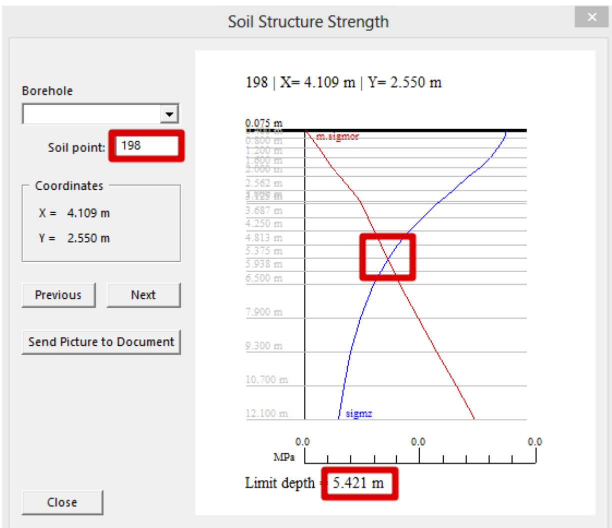
6. Subsoil - Other data (see the preview with the table for the settlement):

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

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:

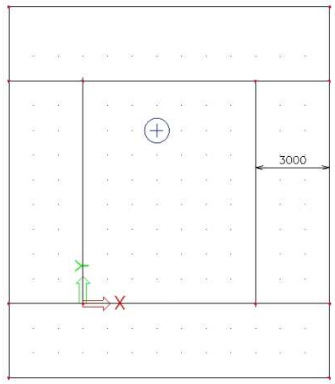


9. Close the dialogue.
10. Use ESC to finish the action.

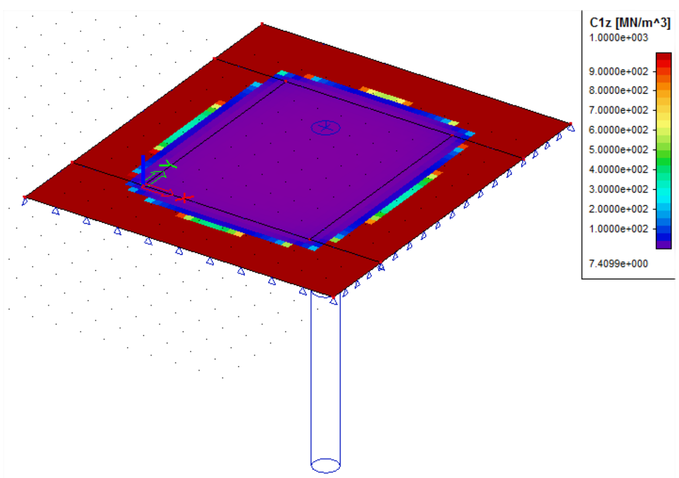
The edges of the plate are supported by springs automatically.

How to create the additional plates

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:



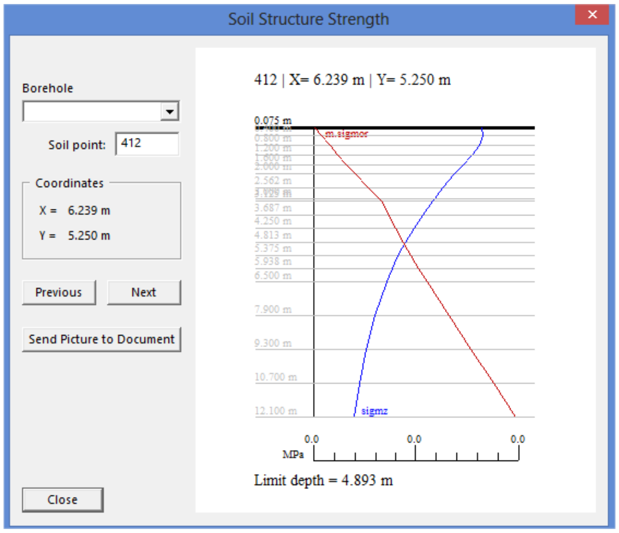
9. Subsoil - Other data (see the preview with the table for settlement):

Subsoil - Other data

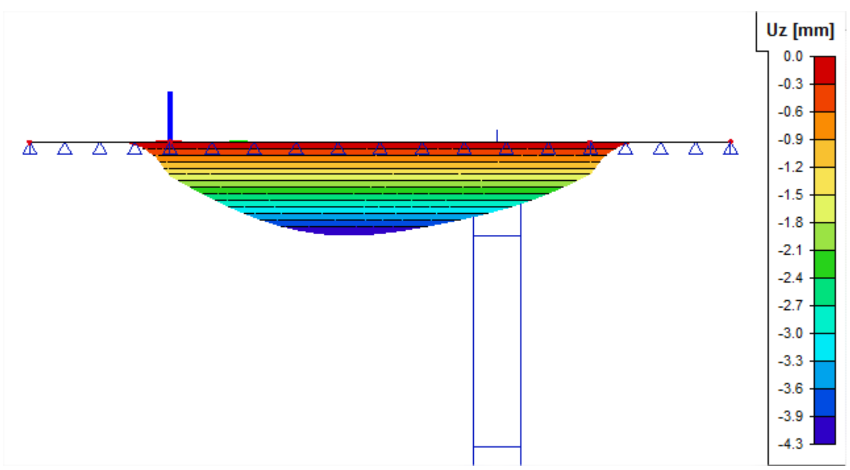
Selection : All
Combinations : CO1

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

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:

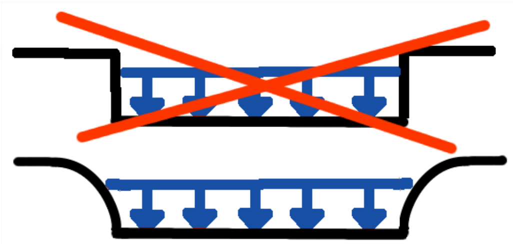


The deformed structure shows the degrease basin.

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.

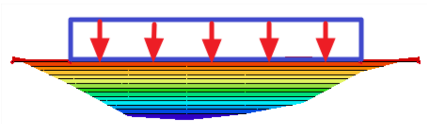


Calculation of the nearest surrounding of the structure is a specific use case. It is recommended to add one more plate to the structure for this purpose – additional subsoil element.

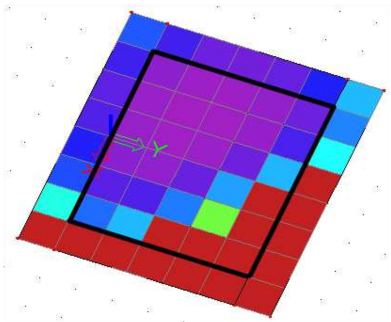
The new plate should be inserted with the minimum thickness (e.g. 0,01mm) and placed next to the foundation.

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

The deformed subsoil calculated by the SCIA Engineer:



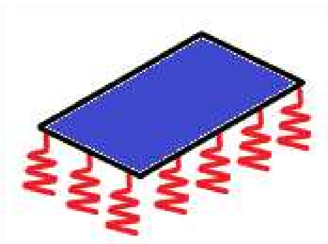
Calculated C parameters:



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 the user does not 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 tries 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.

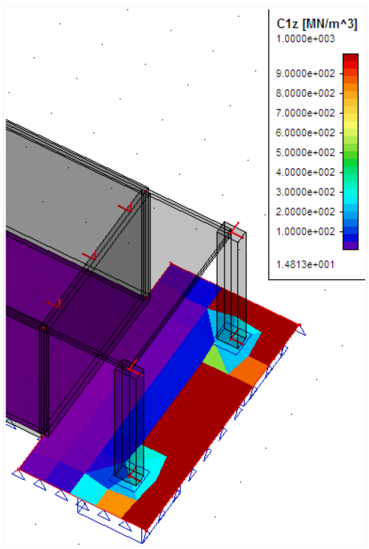
This automatic input can be avoided manually. User can insert a spring with a small stiffness on the plate edges and then the system won't use automatic input of vertical supports. This could be the additional subsoil elements.

Pad foundation and soil-in

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

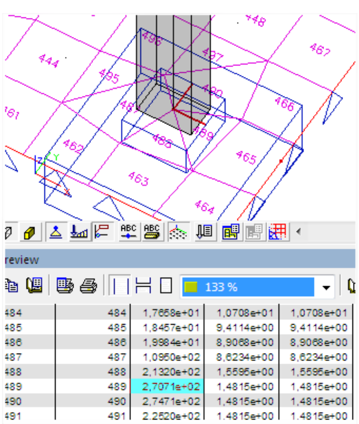
How to use soil-in for the pad foundation check:

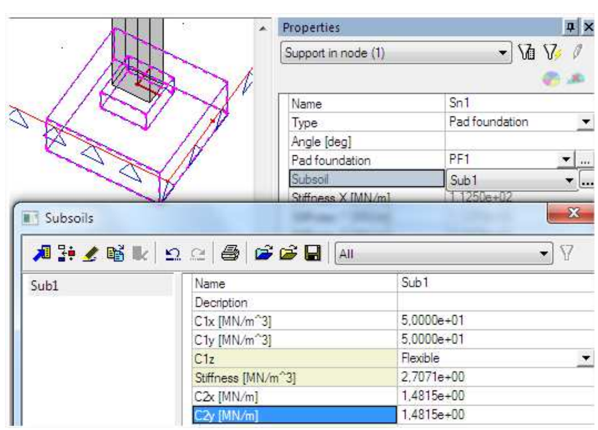
1. Create an additional structure to calculate the C parameters in the nearest surrounding:



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.

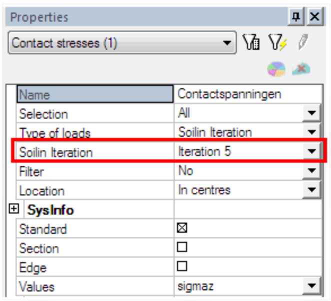
What if the model is correct but the iteration is not finished

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.

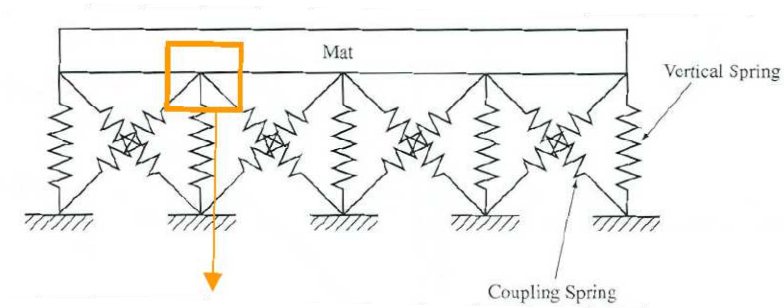
What if the symmetrical structure gives non-symmetrical results?

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.)

Annex 1: Elastic Foundation

In SCIA Engineer the soil can be modelled as an elastic foundation where the soil under a plate is represented by springs.

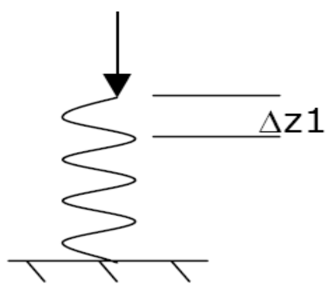


The subsoil parameters C1 and C2 represent the stiffnesses of these springs.

<u>Winkler Model</u>

The Winkler method is the most common and simple method. This model is based on a uniform settlement of the plate. A load F_{1z} will give a certain deformation Δz_1 so the subsoil parameter C_{1z} can be determined.

The parameters C_{1X}, C_{1y} and C_{1z} will represent a linear stiffness.

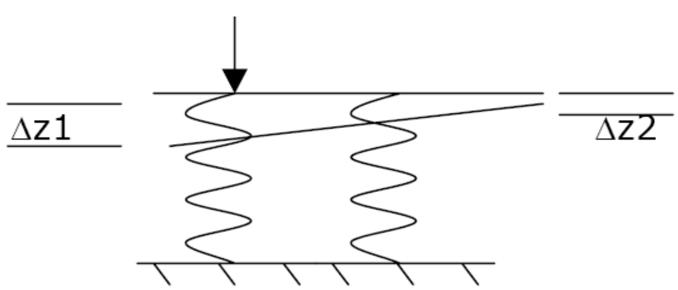


$$F_{1z} = C_{1z} \cdot \Delta z_1$$

The horizontal subsoil parameters C_{1x} and C_{1y} indicate the friction between the plate and the ground. In literature more information can be retrieved for the calculation of these parameters. For normal soils (no rock, peat ...) a guide value of 10% of the vertical stiffness C_{1z} may be taken.

<u>Pasternak Model</u>

The Winkler model can be extended with the Pasternak model (C_2 constants). The springs between the points of the soil are now connected with this value. So a point load in a certain point, will also give a deformation a bit further in the X and Y direction.

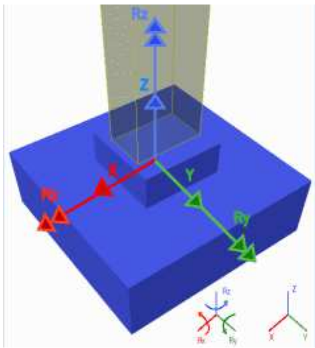


The calculation of those parameters is not easy, but it could be done by the module Soilin of SCIA Engineer. If Soilin is not being used in SCIA Engineer, it is recommended to have zero values for these C_2 parameters.

Annex 2: Pad Foundation Stiffness

This annex specifies the calculation of the stiffness coefficients of a pad foundation.

In the stiffness calculation has been assumed that C2x = C2y.



Pad foundation	PF1
Subsoil	Sub1
Stiffness X [MN/m]	1.1250e+02
Stiffness Y [MN/m]	1.1250e+02
Stiffness Z [MN/m]	4.0488e+02
Stiffness Rx [MNm/rad]	2.5237e+02
Stiffness Ry [MNm/rad]	2.5237e+02
Stiffness Rz [MNm/rad]	1.9572e+02
Water table	
Level	No influence
Backfill material	
Density [kg/m^3]	0.0

Stiffness	Formula
Stiffness X	$A \cdot B \cdot C1x$
Stiffness Y	$A \cdot B \cdot C1y$
Stiffness Z	$A \cdot B \cdot C1z + 2 \cdot (A+B) \cdot \sqrt{C1z \cdot C2x} + 2C2x$
Stiffness Rx	$B^3 \cdot \frac{A \cdot C1z + 2 \cdot \sqrt{C1z \cdot C2x}}{6} + \frac{A \cdot B^2 \cdot \sqrt{C1z \cdot C2x}}{2} + \frac{B^2 \cdot C2x}{2} + A \cdot B \cdot C2x$
Stiffness Ry	$A^3 \cdot \frac{B \cdot C1z + 2 \cdot \sqrt{C1z \cdot C2x}}{6} + \frac{B \cdot A^2 \cdot \sqrt{C1z \cdot C2x}}{2} + \frac{A^2 \cdot C2x}{2} + B \cdot A \cdot C2x$
Stiffness Rz	$C1y \cdot Ix + C1x \cdot Iy + \frac{h1 \cdot A^3 \cdot C1z}{6} + \frac{h1 \cdot B^3 \cdot C1z}{6} + \frac{2 \cdot \sqrt{C1z \cdot C2x} \cdot A^2 \cdot h1}{4} + \frac{2 \cdot \sqrt{C1z \cdot C2x} \cdot B^2 \cdot h1}{4} + \frac{C2x \cdot A^2}{2} + \frac{C2x \cdot B^2}{2}$

Parameters	
A	Dimension read from Pad Foundation library
B	Dimension read from Pad Foundation library
C1x	Soil stiffness read from Subsoil library
C1y	Soil stiffness read from Subsoil library
C1z	Soil stiffness read from Subsoil library
C2x	Soil stiffness read from Subsoil library
Ix	$\frac{A \cdot B^3}{12}$
Iy	$\frac{B \cdot A^3}{12}$

Annex 3: Recommended geotechnical data

All geological layers of a subsoil are represented by their 3D geotechnical properties defined according National Standards. The exactness of these input data depends firstly on the geotechnical category of foundation problem, defined in EC7. Shortly: the 1st and 2nd category pertains to common buildings founded on common subsoil, previous as well as definitive design, without extraordinary complications. The 3rd category includes very important buildings in complicate foundation conditions whose geotechnical properties must be investigated in situ in any case separately with sufficient number of deep test pits or other secure methods. Nonlinear and time dependent behaviour must be taken into account which means an iterative Soilin procedure respecting the increase and decrease of overload.

Such an exacting analysis presents only a few percent of the common design practice. Therefore, a recommendation of certain mean European values for the first calculations using Soilin can be useful.

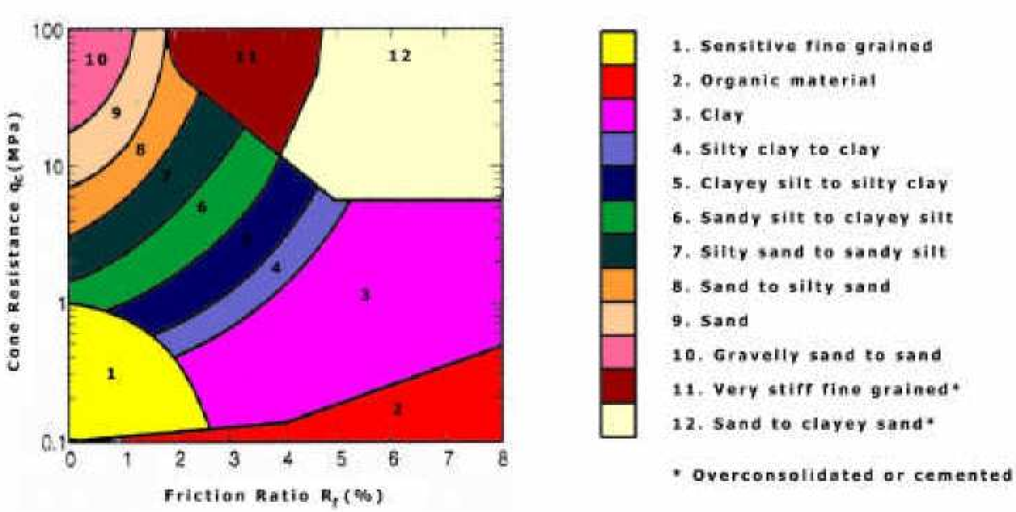
Robertson

Where a building will be established, we need to know the soil profile. A deep knowledge of the ground under the base of the foundation is important as the layers below the base determine the bearing capacity. In order to gain an insight into the ground profile, many properties such as the thickness and composition must be known. We can derive these data from a geotechnical atlas or experiences, but we will mainly derive it from in situ soil research or laboratory tests.

In order to be able to identify a ground, the existence of a ground classification is necessary. For the interpretation of the CPT data, there exist several methods of identification. For example the method according to Robertson is a well known method for electric CPT's and it appears to give the most reliable results in Belgium.

In the following diagram, the cone resistance and friction ratio are used to determine a soil type, wherein the friction ratio is the ratio between the frictional resistance and the cone resistance.

$$R_f = \frac{f_s}{q_c}$$



An identification of the ground means that we now know the soil characteristics at any depth. In tables, you can read these characteristics (angle of friction, cohesion, E-modulus ...) by soil type. Finding of this E-modulus is necessary for the determination of the constant C. We attempt rather to distract the E-modulus from CPT results via soil identification since an additional ground investigation involves an additional cost.

In SCIA Engineer it is necessary to insert the parameter Edef. As said before, it is best to enter a value which is defined directly by a geologist from a real geologic profile. If this is not available, you must use standard values (each country has its own standards for classification of soils). For every soil, there is a range for value Edef (the smaller values are on the safe side).

Table 1. Recommended values for Fine – Grained Soil

No.	Short Soil Name	Quantity	Consistency			
			soft	stiff	rigid	hard
F1	gravel clay $\gamma = 19$ kNm^{-3} $\nu = 0.35$	E_0 (MPa) c_u (kPa) φ_u (deg)	5-15 40 0	10-20 70 0	12-30 70-80 10-15	needs to be invest-igated seperat-ely
F2	gravel clay loam $\gamma = 19.5$ kNm^{-3} $\nu = 0.35$	E_0 (MPa) c_u (kPa) φ_u (deg)	4-8 30 0	7-15 60 0	10-25 60-70 10-15	
F3	sand clay $\gamma = 18$ kNm^{-3} $\nu = 0.35$	E_0 (MPa) c_u (kPa) φ_u (deg)	3-6 30 0	5-8 60 0	8-15 60-70 10-15	
F4	sand clay loam $\gamma = 18.5$ kNm^{-3} $\nu = 0.35$	E_0 (MPa) c_u (kPa) φ_u (deg)	2.5-4 30 0	4-6 50 0	5-12 70-80 5-14	
F5	clay $\gamma = 20$ kNm^{-3} $\nu = 0.40$	E_0 (MPa) c_u (kPa) φ_u (deg)	1.5-3 30 0	3-5 60 0	5-10 70-80 5-14	10-20 80-200 0-20
F6	clay loam $\gamma = 21$ kNm^{-3} $\nu = 0.40$	E_0 (MPa) c_u (kPa) φ_u (deg)	1-3 25 0	3-5 50 0	5-10 80-90 0-12	10-20 80-170 0-18
F7	plastic clay $\gamma = 21$ kNm^{-3} $\nu = 0.40$	E_0 (MPa) c_u (kPa) φ_u (deg)	1.5-3 25 0	3-6 50 0	6-12 80-90 0-12	10-20 80-170 0-18
F8	high plastic $\gamma = 19.5$ kNm^{-3} $\nu = 0.35$	E_0 (MPa) c_u (kPa) φ_u (deg)	1-2 20 0	2-4 40 0	4-8 80-90 0-10	8-15 80-150 0-16

c_u, φ_u .=. undrained cohesion and internal friction angle for the 1st limit state (soil collapse)

$E_0 = E_{def}$ = average secante deformation modulus at common pressure level

m (soil structure strength factor):

F1 – F8 at E_0 = 1 – 4: m = 0.1
F1 – F8 at E_0 = 4 – 30:d m = 0.2, except of:
F5 and F7 at E_0 = 1 – 10: m = 0.5 (maximum)

(γ - 10) kNm^{-3} specific eigenweight below water level

Table 1a. Recommended values for Fine – Grained Soil

The first estimation (without any geotechnical investigation) of the result order may be done at the following mean values:

Fine-Grained Soil No.	E_0 [MPa]	ν [1]	γ[kNm^{-3}]	m [1]
F1 – 4 soft/stiff	3 – 20	0.35	18 – 20	0.2
average	10	0.35	19	0.2
F1 – 4 rigid	5 – 30	0.35	18 - 20	0.2
average	15	0.35	19	0.2
F5 – 8 soft/stiff *)	1 – 6	0.4	20 – 21	0.1
average	3	0.4	20.5	0.1
F5 – 8 rigid	4 – 12	0.4	20 – 21	– 0.2
average	8	0.4	20.5	0.15
F5 – 8 hard	8 – 20	0.4	20 – 21	0.2
average	15	0.4	20.5	0.2

*) Mould types ML, MI, MH, MV over the water level if the water never will rise and the soil remain dry: m = 0.5.

Table 2 Recommended Values for Sands (S1 to S5) and Gravels (G1 to G5)

No.	Short Soil Name	γ (kNm^{-3})	ν (1)	E_0 (MPa)	c_{ef} (kPa)	φ_{ef} (deg)
S1	course sand	20.0	0.28	30-100	0	34-42
S2	sand	18.5	0.28	15-50	0	32-37
S3	fine sand	17.5	0.30	15-25	0	28-33
S4	clayer sand	18.0	0.30	5-15	0-10	28-30
S5	loam mouldign sand	18.5	0.35	4-12	4-12	26-28

G1	well granulated gravel	21.0	0.20	250-500	0	36-44
G2	poorly granulated gravel	20.0	0.20	100-250	0	33-41
G3	gravel with fine soil	19.0	0.25	80-100	0	30-38
G4	clayey gravel	19.0	0.30	60-80	0-8	30-35
G5	loam moulding gravel	19.5	0.30	40-60	2-10	28-32

c_{ef}, φ_{ef} ... effective (drained) cohesion and internal friction angle for the 1st st limit state (soil collapse)

m (soil structure strength factor):

S1, S2, G1, G2 ..... m = 0.2 to 0.3
S3 – 5, G3 – 5 ..... m = 0.3

Table 2a Mean values for the first estimation on a common subsoil

Subsoil	**E_0 (MPa)**	**ν (1)**	**γ (kNm⁻³)**	**m (1)**
Sand	5 – 40	0.3	17.5 – 20.0	– 0.3
average	20	0.3	18.5	0.25
Gravel	50 - 200	0.2– 0.3	19.0 – 21.0	0.3
average	100	0.25	20.0	0.3

Table 3 Recommended values for some rocks in subsoil

Rock	**E_0 (MPa)**	**ν (1)**	**γ (kNm^{-3})**	**m (1)**
R1, 2, 4, 5 (II.)	20-25000	0.1-0.3	18-31	0.1
R3	15-10000	0.1-0.35	18.5-26	0.2
R4, 5 (IX.)	20-3000	0.2-0.3	18-31	0.3
R6	10-300	0.25-0.4	21-26	0.4

Remark: Withered rocks as subsoil layers are of small efficient E_0 values up to 100 – 300 MPa. Sound rock is usually assumed as an undeformable deepest layer with E_0 substantially larger than the other layers without influence on surface settlement. See also Tab. 4.

Table 4. Subsoils ordered by their soil structure strength factor m

No.	EC7 sort	E_0 (MPa)	ν (1)	γ (kNm^{-3})	m (1)
I. II.	F1-F8 R1, 2, 4, 5	1-4 20-25000	0.35-0.42 0.10-0.30	18-21 18-31	0.1 0.1
III. IV.	F1-F8 S1, 2; G1, 2, R3	4-30 15-10000	0.35-0.42 0.10-0.35	18-21 18.5-26	0.2 0.2
V. VI. VII. VIII. IX.	S1, 2 G1, 2 S3, 4, 5 G3, 4, 5 R4, 5	15-100 100-500 4-25 40-100 20-3000	0.28 0.20 0.30-0.35 0.25-0.30 0.20-0.30	18.5-20 20-21 17.5-18.5 19-19.5 18-31	0.3 0.3 0.3 0.3 0.3
X.	R6	10-300	0.25-0.40	21-26	0.4
XI.	F5 (ML, MI) F7 (MH, MV)	1-10	0.40	20-21	0.5
I.–XI.	Below the water level			γ - 10	

I. Fine – grained soils F1 – F8 easily compressible without previous compactness improvement, soft to stiff, $E_0 = 1 - 4$ MPa. Subsoil till this time not overloaded, loose scattering, fillers.

II. Rocks R1 – R2. Unchanged sediments R4 from the 2nd and 3rd geological epoch, most of R5.

III. All fine – grained soils with exception of I., X., XI.

IV. Sand and gravel S1 – 2, G1 – 2 below the water level and rocks R3.

V. Sand S1 – 2 over the water level.

VI. Gravel G1 – 2 over the water level.

VII. Fine, clayer and loam moulding sands S3 – S5 with substantial clay content.

VIII. Clayey and loam moulding gravel with substantial fine soil content G3 – G5.

IX. Rocks R4 – R5 with exception of II.

X. Rocks R6 (eluvium).

XI. Dry clay mould F5 (ML, MI), F7 (MH, MV) permanently over the water level.