SCIA ENGINEER

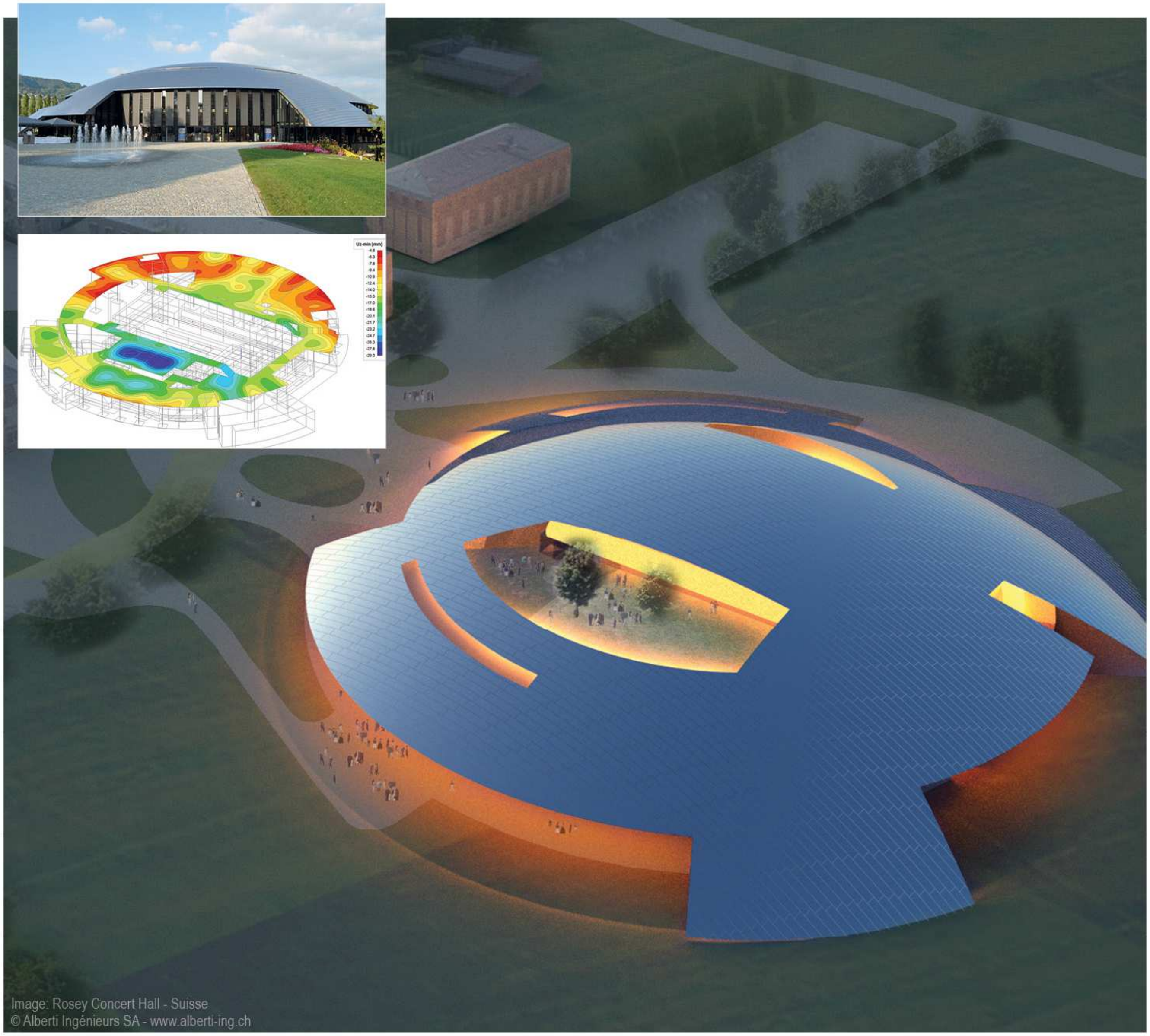


Image: Rosey Concert Hall - Suisse
© Alberti Ingénieurs SA - www.alberti-ing.ch

Advanced Training

Parameters

All information in this document is subject to modification without prior notice. No part of this manual may be reproduced, stored in a database or retrieval system or published, in any form or in any way, electronically, mechanically, by print, photo print, microfilm or any other means without prior written permission from the publisher. SCIA is not responsible for any direct or indirect damage because of imperfections in the documentation and/or the software.

Table of contents

Preface

This workshop contains an assortment of examples on parameters.

What is the purpose of this functionality?
Suppose that a SCIA Engineer user handles nearly every day with the same type of structure.
Only the dimensions, cross-sections, height, number of spans, … differs in the different projects. For this, the option '**Parameters**' can be used.
Every type -as mentioned above- can be parameterized.
The parameters are fully editable and when changed they may lead to a very straightforward modification of the calculated model.
What's more, a model defined by means of parameters can be saved as a template. When opened, the user is first asked to fill in the table with all the parameters present in the model. This may be effectively used for creation of simple "programs" for e.g. calculation of continuous beam, simple frame, etc.

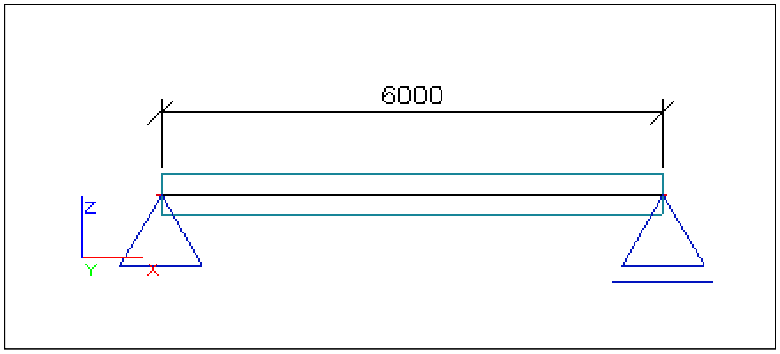
The user has to create the structure only once. Then he/she has to define the parameters and save the structure as template. In the future, he/she just fills in the table with a few parameters and can immediately proceed to calculation and evaluation of results.
This leads to a huge reduction of the working hours and will be more effective.

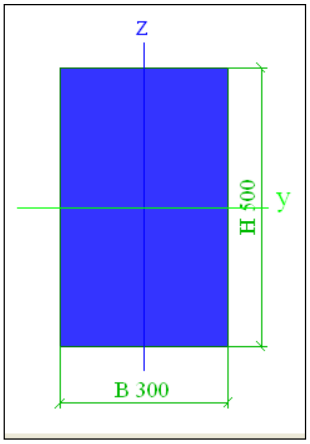
Above this, this workshop contains also some extra's which can help the user to become more effective in working with SCIA Engineer:

- **XML**: SCIA Engineer supports exporting and importing to/of this format.
- **ODA**: One dialog application
- **Batch optimizer**: The user can give a range of values to a parameter and the solution is also a range of results.
- **User blocks**: A project can be saved as a user block. Afterwards, this block can be imported in another project.
- **Project templates**: As mentioned before, the user can make a standard project which can be used for all the other projects.

Beam on two supports

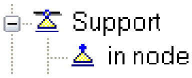
A beam manufactured of C30/37 according to the EC-EN is supported on two ends.
We will input parameters for the length and the cross-section dimensions.





- Project data: - Construction type: Frame XZ
  - Project level advanced

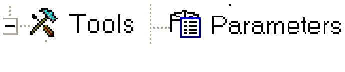
- Input support:



Support
in node

Method

- After activating the functionality parameters, you will see the function

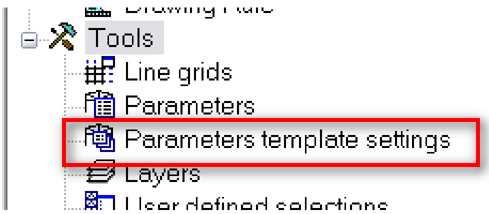


Tools
Parameters

- The parameters get following properties with the values:

- Length L: 10 m
- Width B: 300 mm
- Height H: 500 mm

- Next, go to:



Drawing Plate

Tools

Line grids

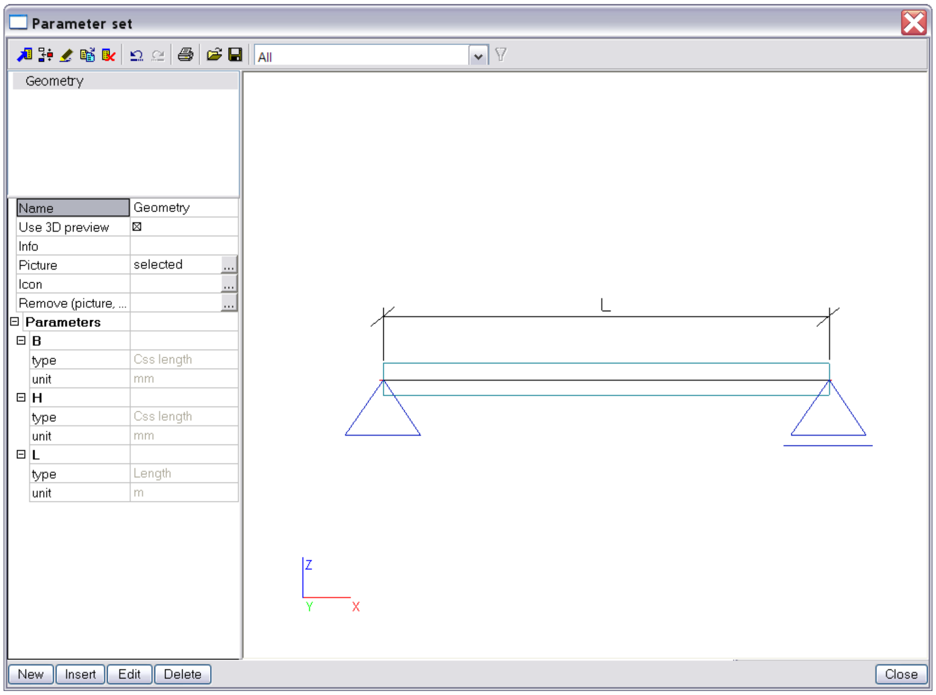
Parameters

Parameters template settings

Layers

User defined selections

Here the template tabs are made



Parameter set

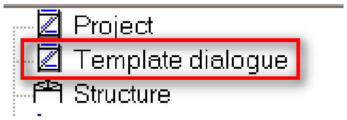
All

Geometry

Name	Geometry
Use 3D preview	☒
Info	
Picture	selected ...
Icon	...
Remove (picture, ...	...
⊟ Parameters	
⊟ B	
type	Css length
unit	mm
⊟ H	
type	Css length
unit	mm
⊟ L	
type	Length
unit	m

Remark: the introduction of the picture will be explained later.

- Subsequently you can use a new function:

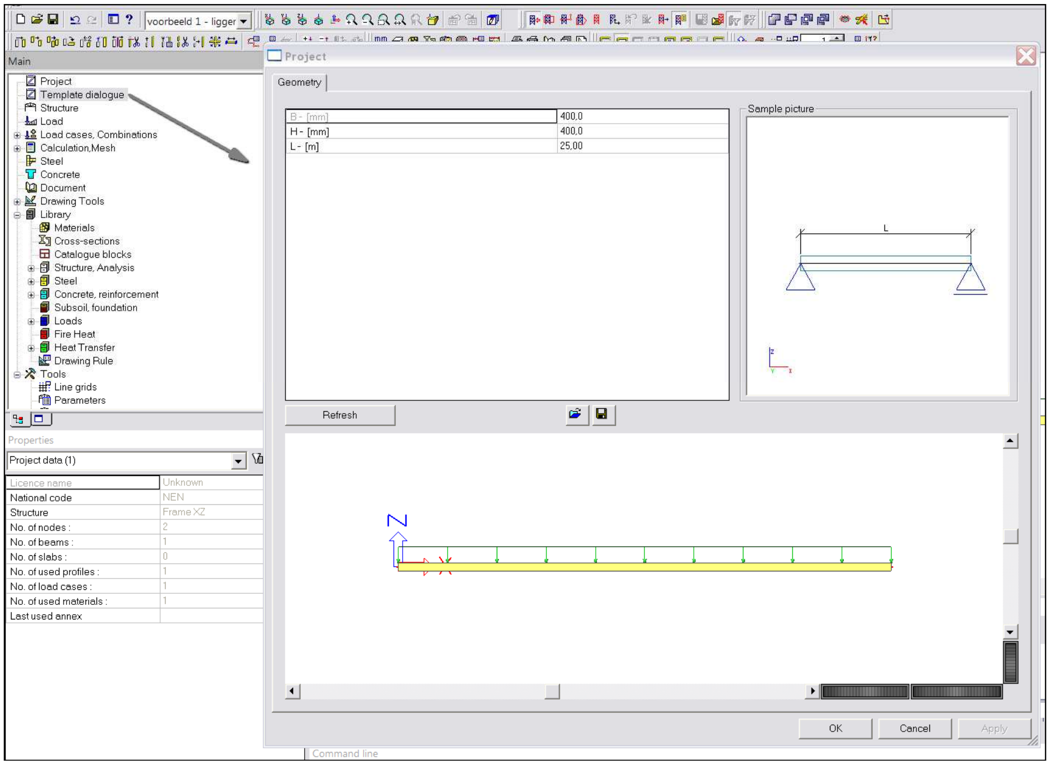


Project

Template dialogue

Structure

Here the values of the various parameters can be adapted.



Project data (1)

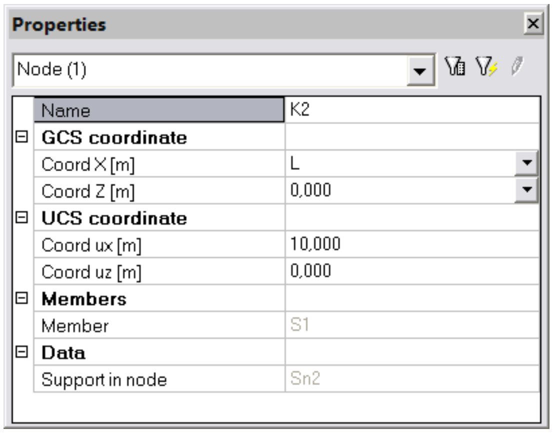
Licence name	Unknown
National code	NEN
Structure	Frame XZ
No. of nodes :	2
No. of beams :	1
No. of slabs :	0
No. of used profiles :	1
No. of load cases :	1
No. of used materials :	1
Last used annex	

Geometry

B - [mm]	400,0
H - [mm]	400,0
L - [m]	25,00

- The various parameters has to be attributed now to the cross-section and the length of the beam.

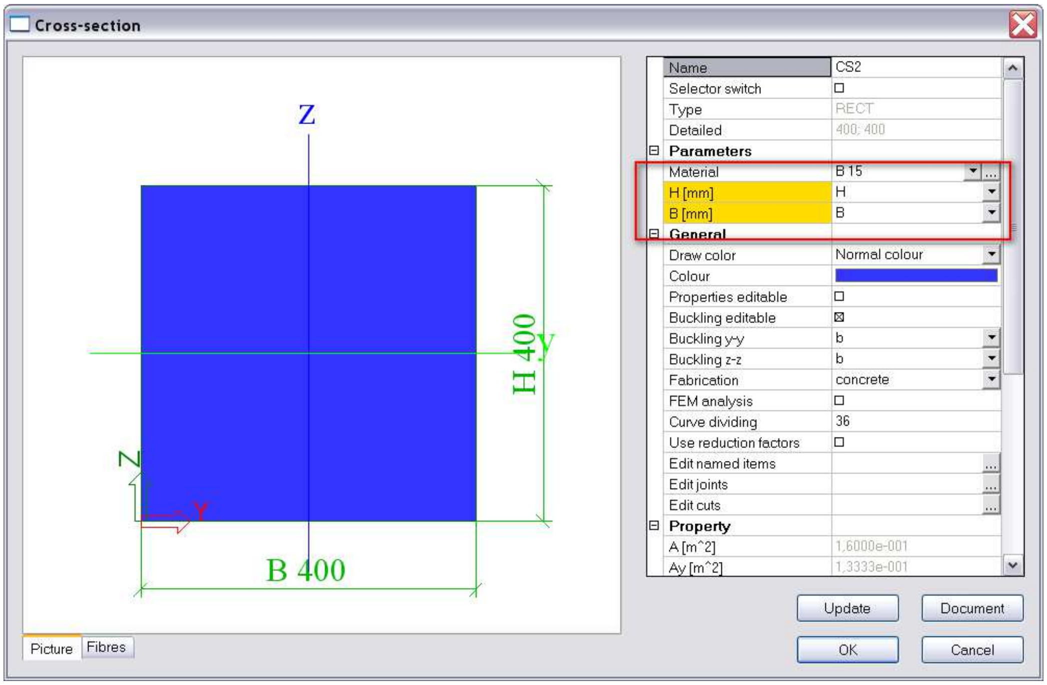
→ Length beam: X-coordinate of node K2:



Node (1)

Name	K2
GCS coordinate	
Coord X [m]	L
Coord Z [m]	0,000
UCS coordinate	
Coord ux [m]	10,000
Coord uz [m]	0,000
Members	
Member	S1
Data	
Support in node	Sn2

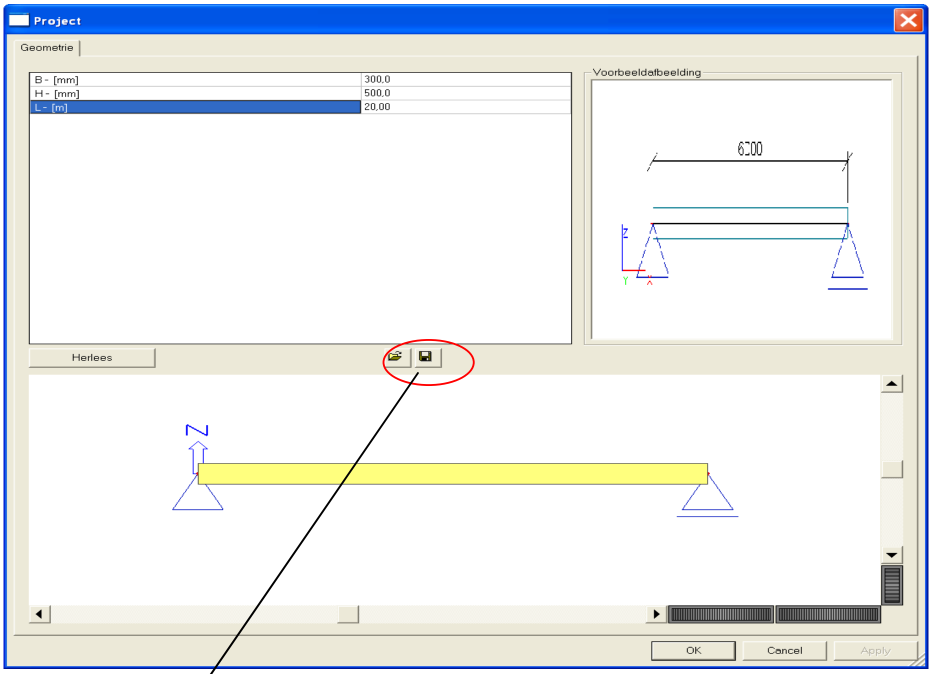
→ Width & height profile:



Name	CS2
Selector switch	□
Type	RECT
Detailed	400; 400
⊟ **Parameters**	
Material	B 15
H [mm]	H
B [mm]	B
⊟ **General**	
Draw color	Normal colour
Colour	🟦
Properties editable	□
Buckling editable	☒
Buckling y-y	b
Buckling z-z	b
Fabrication	concrete
FEM analysis	□
Curve dividing	36
Use reduction factors	□
Edit named items	...
Edit joints	...
Edit cuts	...
⊟ **Property**	
A [m^2]	1,6000e-001
Ay [m^2]	1,3333e-001

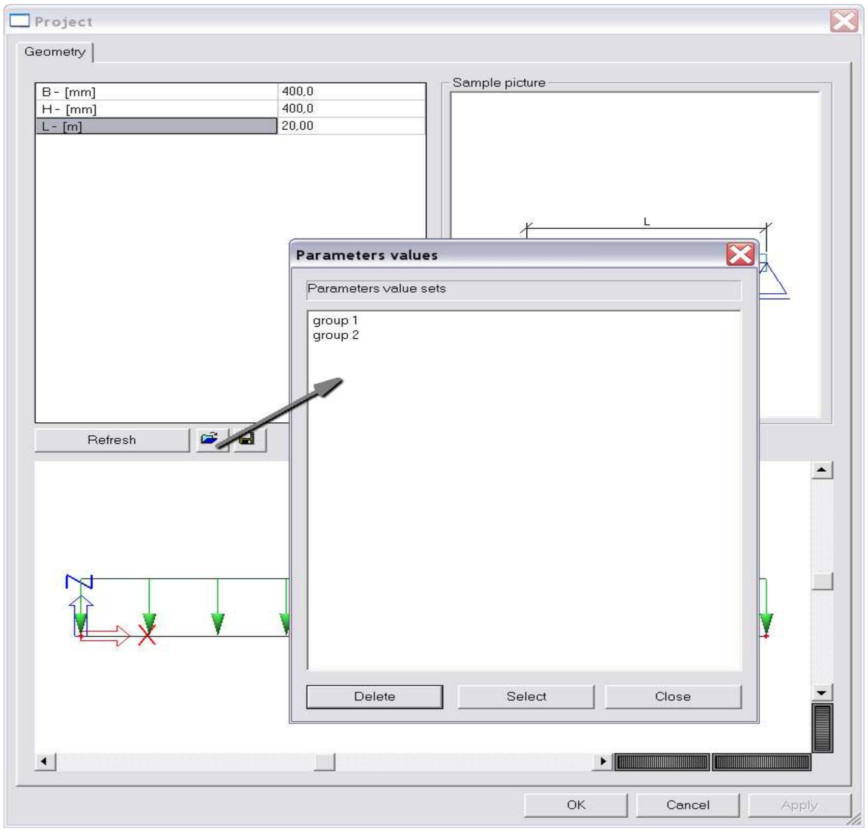
- Changing a parameter:

The value of the length L is adjusted of 10m to 20m.



B - [mm]	300,0
H - [mm]	500,0
L - [m]	20,00

If you use ‘Store users- default settings’, you can make various compositions which you can load afterwards:



Project

Geometry

B - [mm]	400,0
H - [mm]	400,0
L - [m]	20,00

Parameters values

Parameters value sets

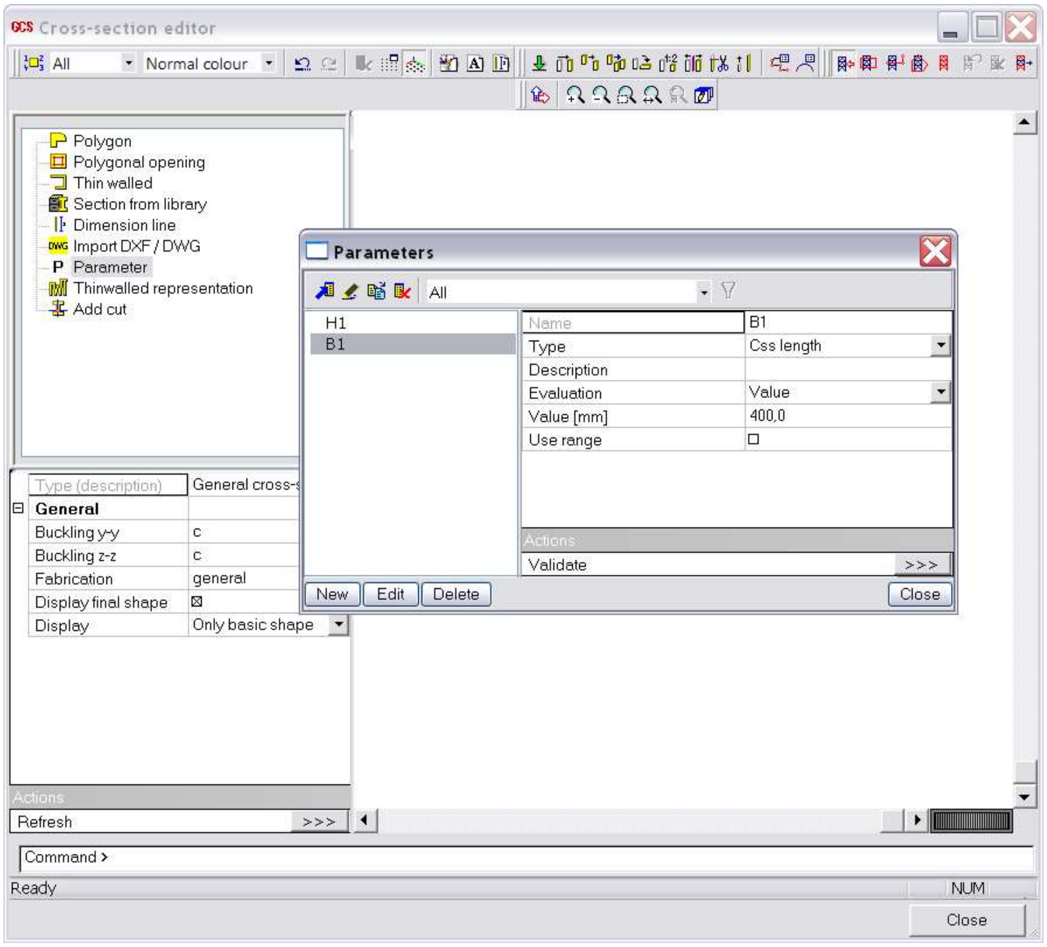
group 1
group 2

Cross-sections

The same example will be used to explain the type of parameters for cross-sections. For this ‘Cross-sections.esa’ can be used.
First of all the parameters for graphical cross-sections are explained. Next, the parameter for the type of cross-section will be shown.

General cross-section

- If the user decides to work with a graphical cross-section, the method of parameterizing is different from that of standard cross-sections.
- First of all parameters has to be created for the height en width of the cross-section.

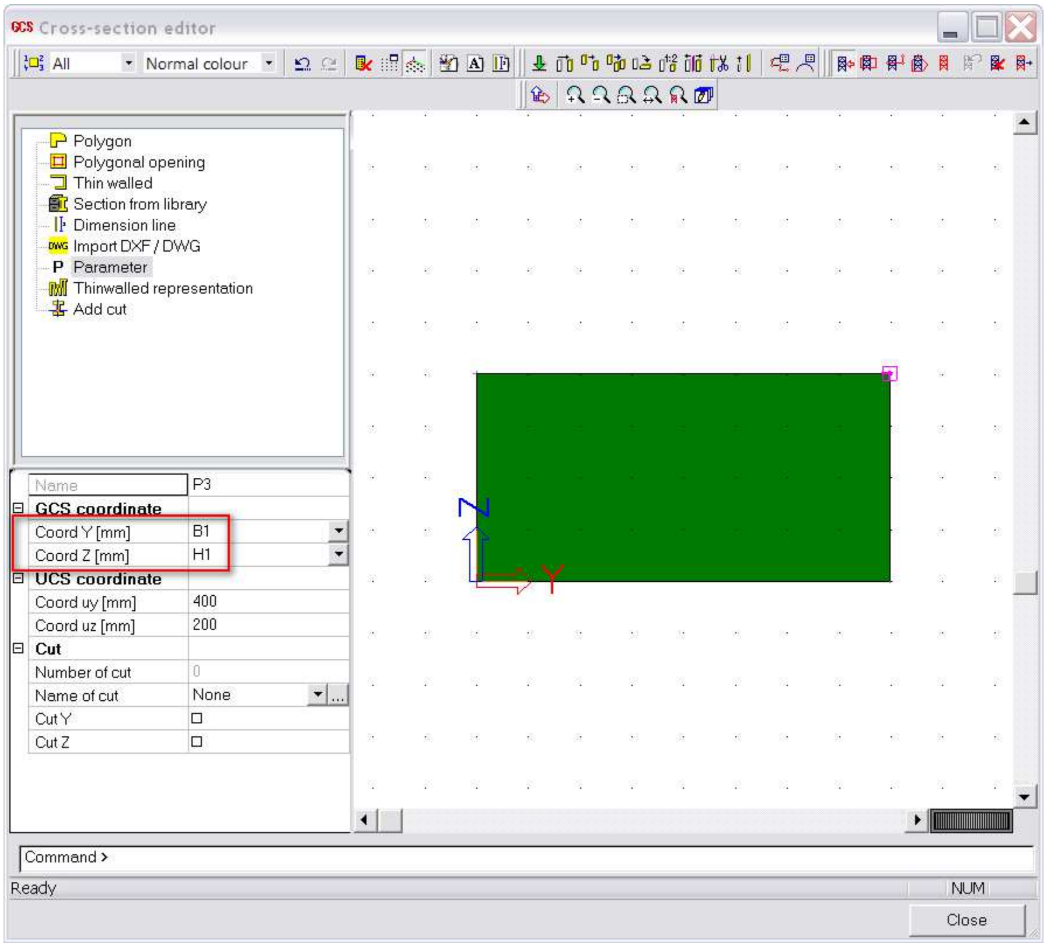


Type (description)	General cross-s
General	
Buckling y-y	c
Buckling z-z	c
Fabrication	general
Display final shape	☒
Display	Only basic shape

H1
B1

Name	B1
Type	Css length
Description	
Evaluation	Value
Value [mm]	400,0
Use range	□

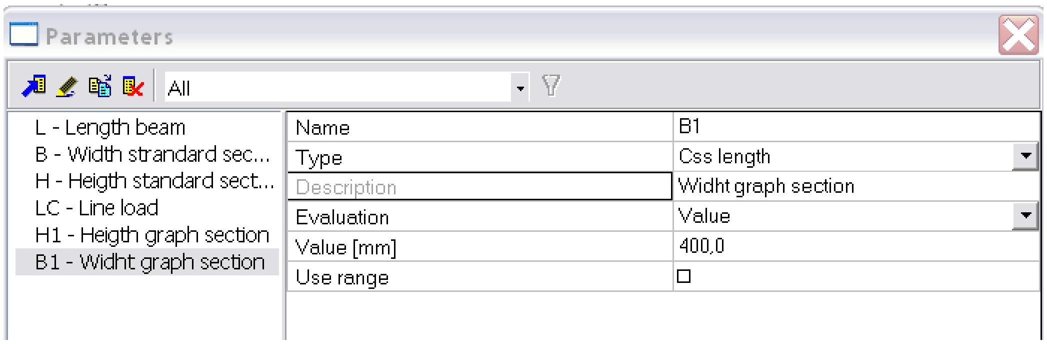
- Next, the polygon can be introduced and after this, the parameters can be attributed to the Y and Z coordinates:



Name	P3
GCS coordinate	
Coord Y [mm]	B1
Coord Z [mm]	H1
UCS coordinate	
Coord uy [mm]	400
Coord uz [mm]	200
Cut	
Number of cut	0
Name of cut	None
Cut Y	□
Cut Z	□

The difference with the parameters of a profile from a library is that here the parameters can be made in the cross-section editor itself.

- Above this, if the user wants to adapt these parameters, two parameters has to be created in 'Tools > Parameters'.
  Here, we can create the same parameters H1 and B1:



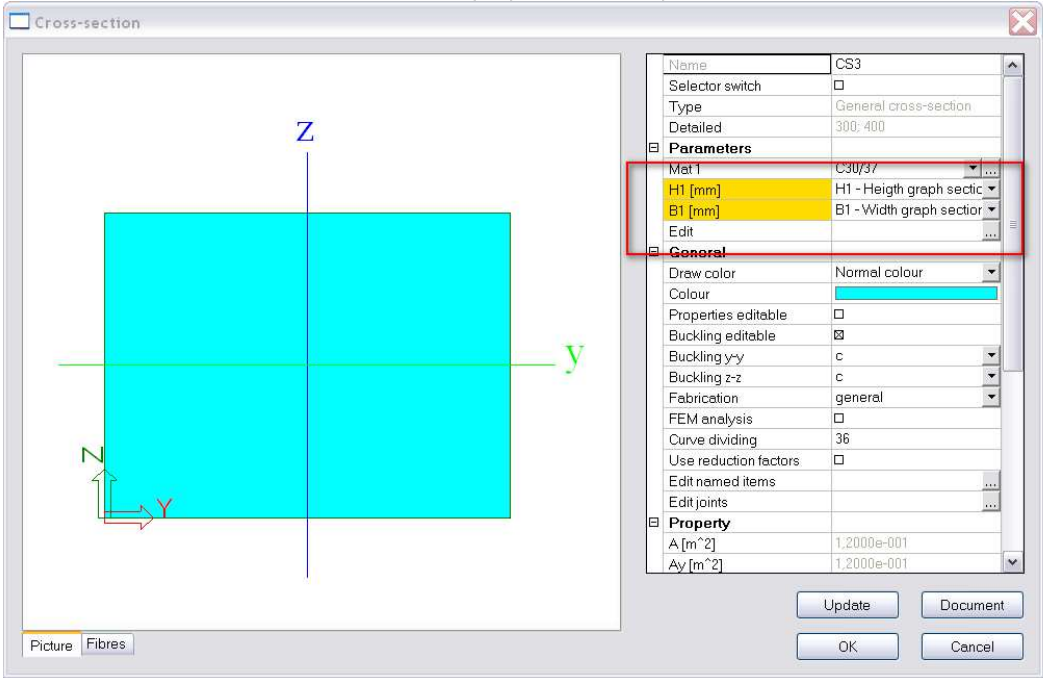
L - Length beam
B - Width strandard sec...
H - Heigth standard sect...
LC - Line load
H1 - Heigth graph section
B1 - Widht graph section

Name	B1
Type	Css length
Description	Widht graph section
Evaluation	Value
Value [mm]	400,0
Use range	□

Note:

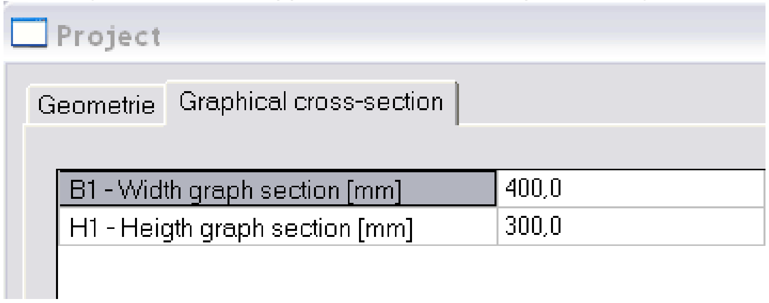
It's very important that you give a good description to your parameters. On that way, it simplifies afterwards the input and the adaptation of the parameters.

- These values can now be attributed to the properties of the profile:



Name	CS3
Selector switch	□
Type	General cross-section
Detailed	300; 400
⊟ **Parameters**	
Mat 1	C30/37
H1 [mm]	H1 - Heigth graph sectic
B1 [mm]	B1 - Width graph sectior
Edit	...
⊟ **General**	
Draw color	Normal colour
Colour	
Properties editable	□
Buckling editable	☒
Buckling y-y	c
Buckling z-z	c
Fabrication	general
FEM analysis	□
Curve dividing	36
Use reduction factors	□
Edit named items	...
Edit joints	...
⊟ **Property**	
A [m^2]	1,2000e-001
Ay [m^2]	1,2000e-001

- To change these values, we can make a new tab in the 'Parameters template settings' and this will appear now in the 'Template dialogue':



B1 - Width graph section [mm]	400,0
H1 - Heigth graph section [mm]	300,0

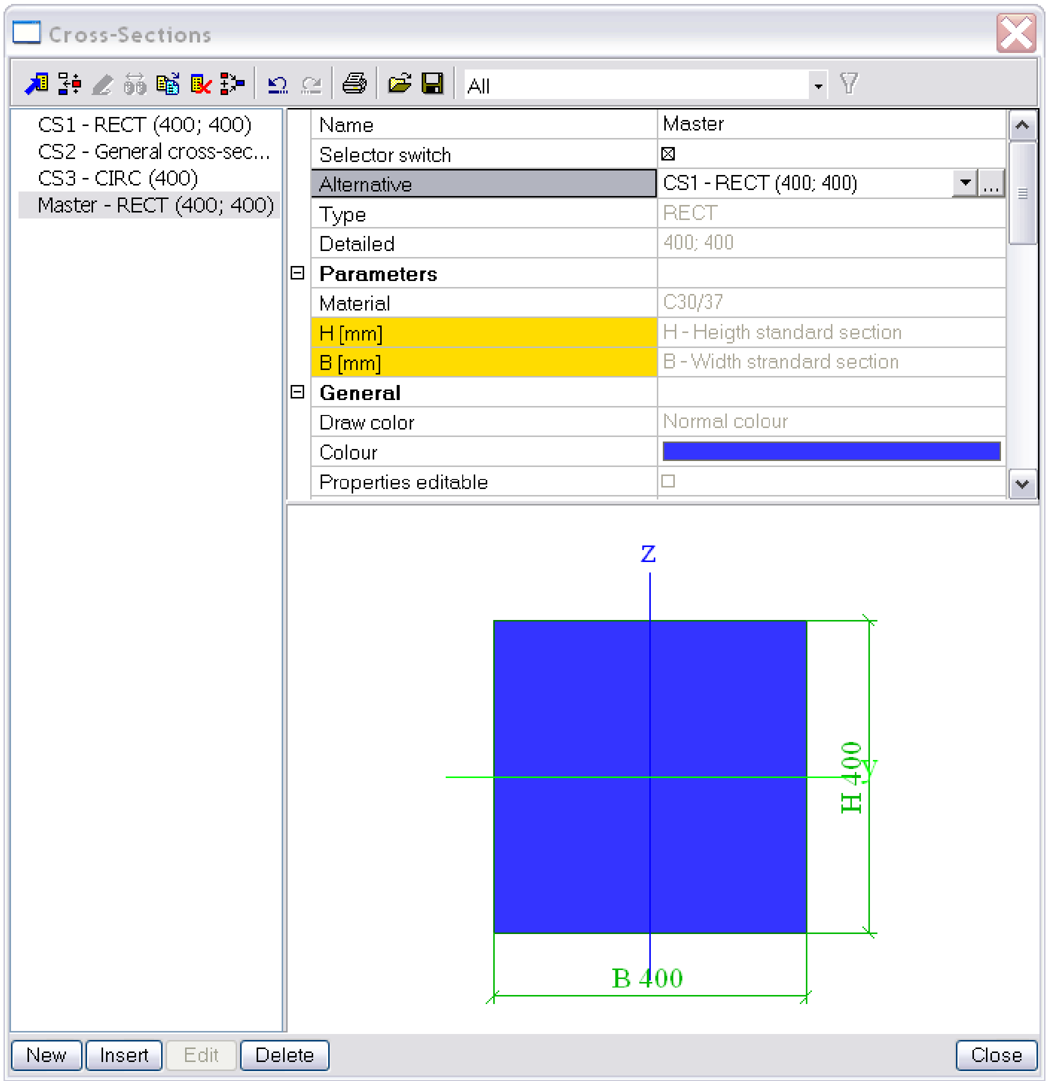
Type of Cross-sections

In SCIA Engineer it's also possible to parameterize libraries. Examples here are: Materials, cross-sections, reinforcement, subsoils,…
In this topic, we will explain it for the type of cross-sections.
The other library types can be handled with the same principle.

Again, this will be explained by means of example 'Cross-sections.esa'.

- Three cross-sections are inserted in the project:
  CS1: RECT(400;400)
  CS2: Graphical cross-section
  CS3: CIRC (400)

- We define a 'master' cross-section. This cross-section is fictive and will be the parameter. For this, we copy arbitrarily CS1 and we call it 'Master'.
- For this master cross-section, the option 'Selector Switch' has to be checked and an Alternative profile can be choosen now:
- Creation of the parameter:



Cross-Sections

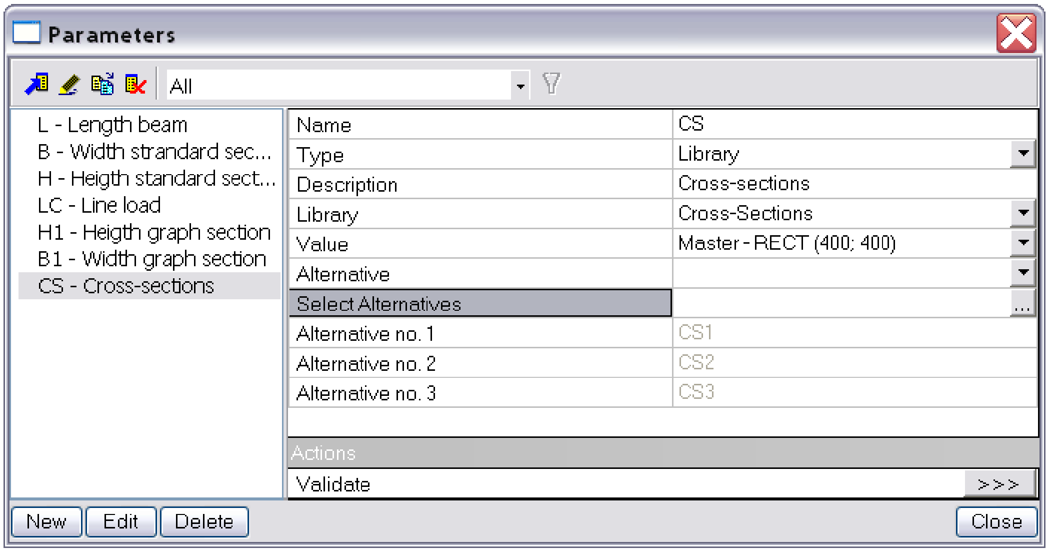
All

CS1 - RECT (400; 400)
CS2 - General cross-sec…
CS3 - CIRC (400)
Master - RECT (400; 400)

Name	Master
Selector switch	☒
Alternative	CS1 - RECT (400; 400)
Type	RECT
Detailed	400; 400
Parameters	
Material	C30/37
H [mm]	H - Heigth standard section
B [mm]	B - Width standard section
General	
Draw color	Normal colour
Colour	🟦
Properties editable	□

Without this option ‘selector switch’, we cannot use the parameter for the cross-section type.

- Go to ‘Tools > Parameters’ and the following parameter can be created:



Parameters

All

- L - Length beam
- B - Width strandard sec...
- H - Heigth standard sect...
- LC - Line load
- H1 - Heigth graph section
- B1 - Width graph section
- CS - Cross-sections

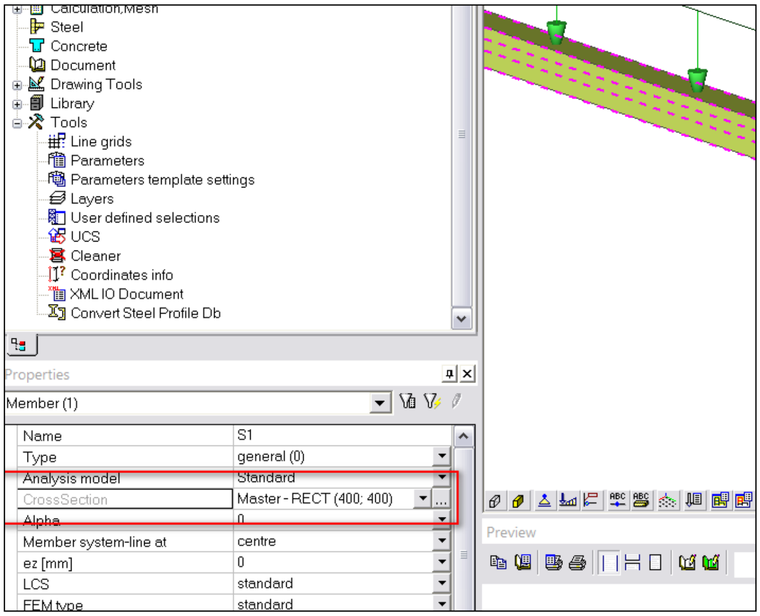
Name	CS
Type	Library
Description	Cross-sections
Library	Cross-Sections
Value	Master - RECT (400; 400)
Alternative	
Select Alternatives	...
Alternative no. 1	CS1
Alternative no. 2	CS2
Alternative no. 3	CS3

Actions

Validate >>>

Type: Library
Library: Cross-sections
Value: Here the fictive ‘master’ cross-section can be input.
Alternative: One of the 3 alternatives can be choosen. This alternative will be the default profile.
Select Alternatives: The 3 alternatives can be inserted here

- Attribute this parameter to the cross-section type of the beam. Pay attention: The master profile becomes the parameter. But the content of this master will change after choosing another alternative:

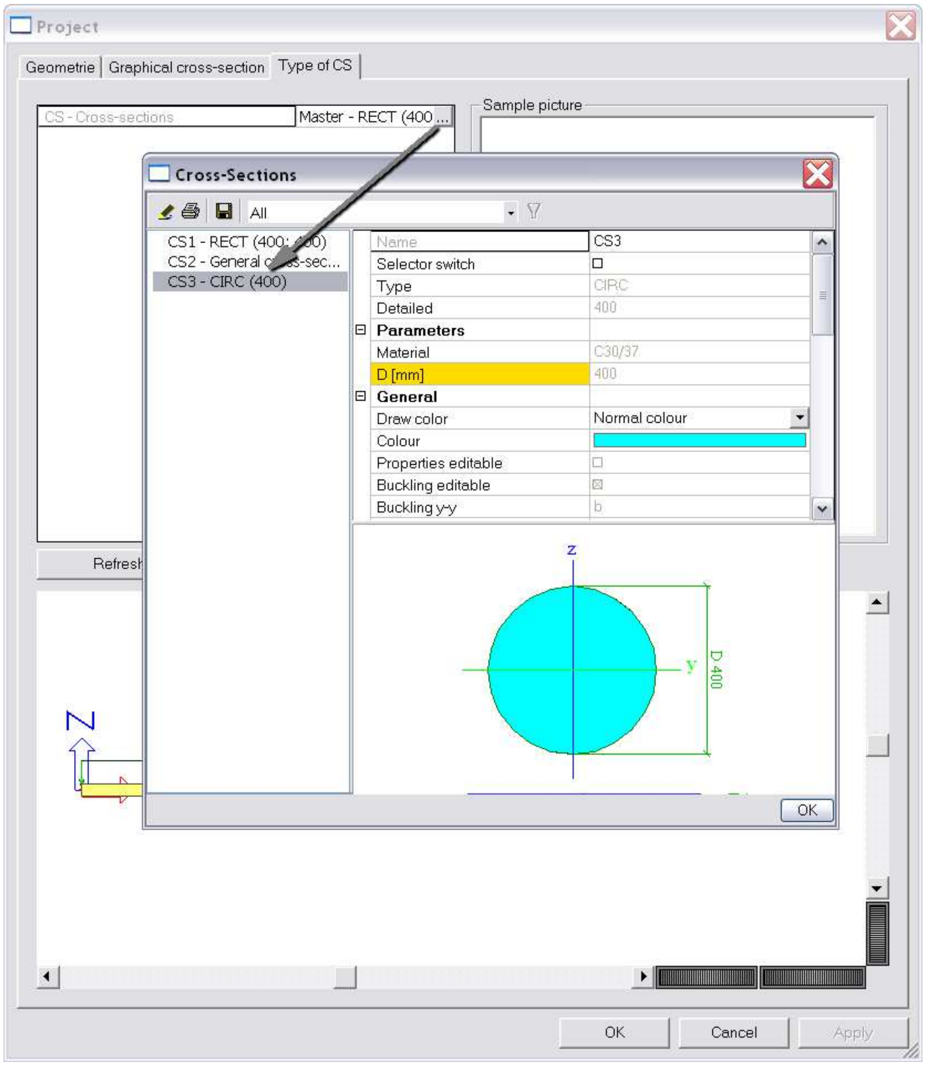


Properties

Member (1)

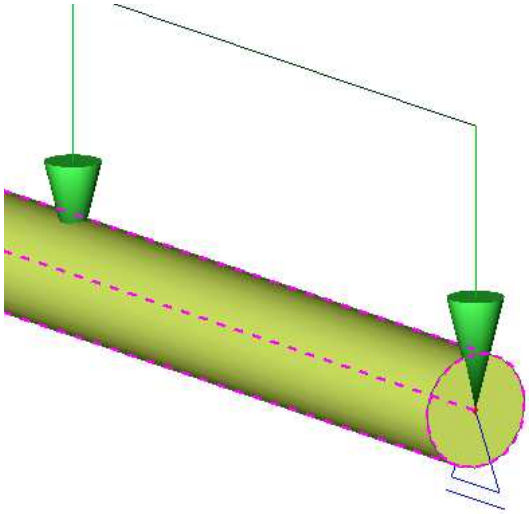
Name	S1
Type	general (0)
Analysis model	Standard
CrossSection	Master - RECT (400; 400)
Alpha	0
Member system-line at	centre
ez [mm]	0
LCS	standard
FEM type	standard

- After creating a new tab in ‘Tools > Parameters Template settings’, it appeared in the dialogue box of the ‘Template Dialogue’. And again, this box can be used to change the cross-section type:



Name	CS3
Selector switch	□
Type	CIRC
Detailed	400
Parameters	
Material	C30/37
D [mm]	400
General	
Draw color	Normal colour
Colour	🟦
Properties editable	□
Buckling editable	☒
Buckling y-y	b

And this results into:

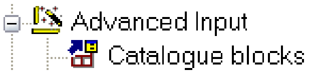


Catalogue block

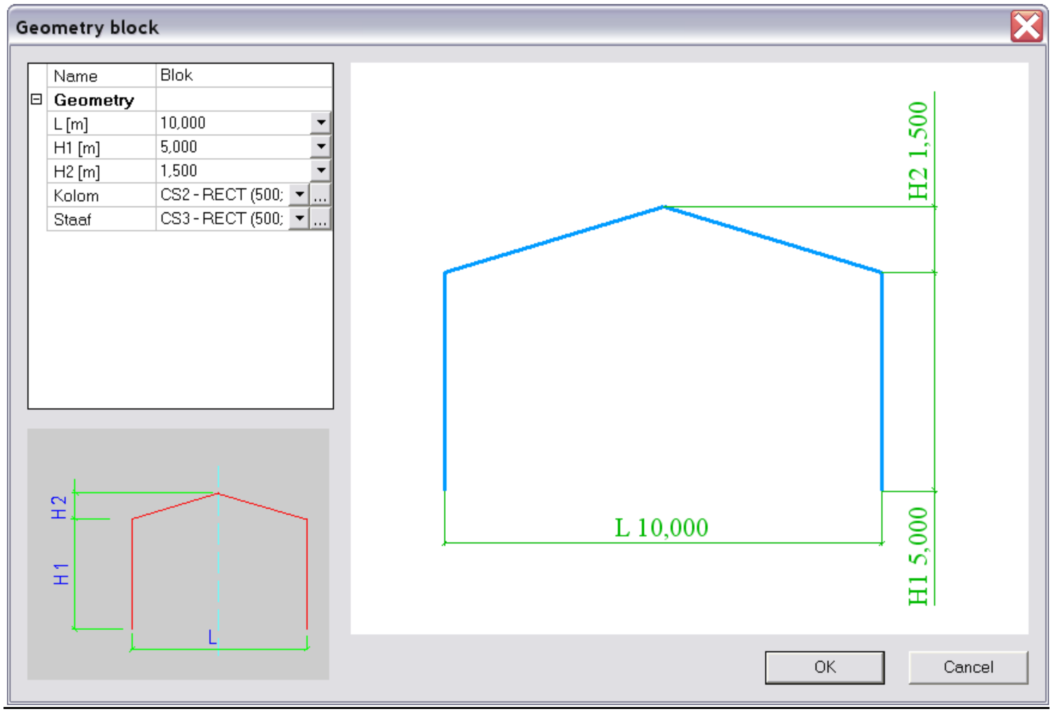
In this example a catalogue block will be inserted. After the introduction of it, it will be parametrized.

- Project data: - Construction type: Frame XYZ
  - Project level : advanced

- Go to ‘Structure > Advanced input > Catalogue blocks’.

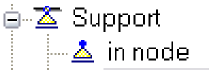

Advanced Input
Catalogue blocks

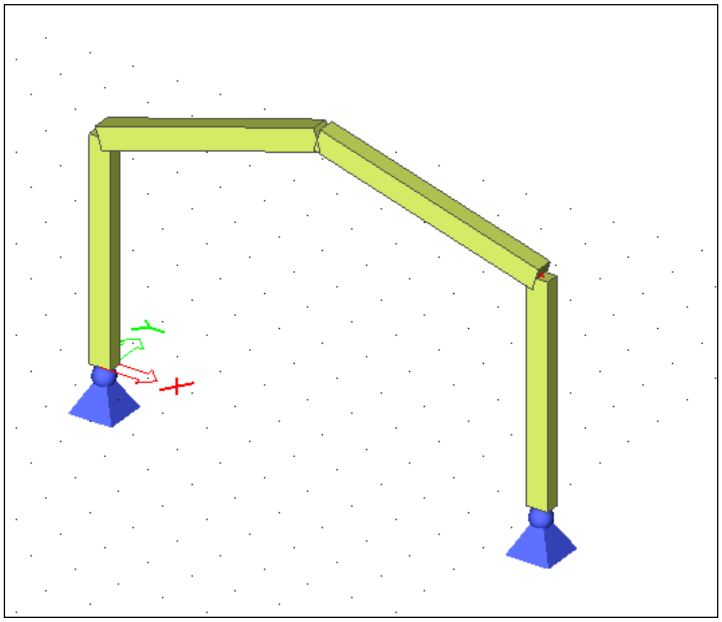
The first block in the menu ‘Frame 2D’ is choosen. We keep the standard configuration:


Geometry block

Name	Blok
Geometry	
L [m]	10,000
H1 [m]	5,000
H2 [m]	1,500
Kolom	CS2 - RECT (500;
Staaf	CS3 - RECT (500;

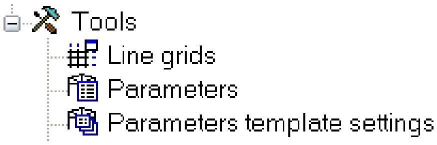
- Input support:


Support
in node



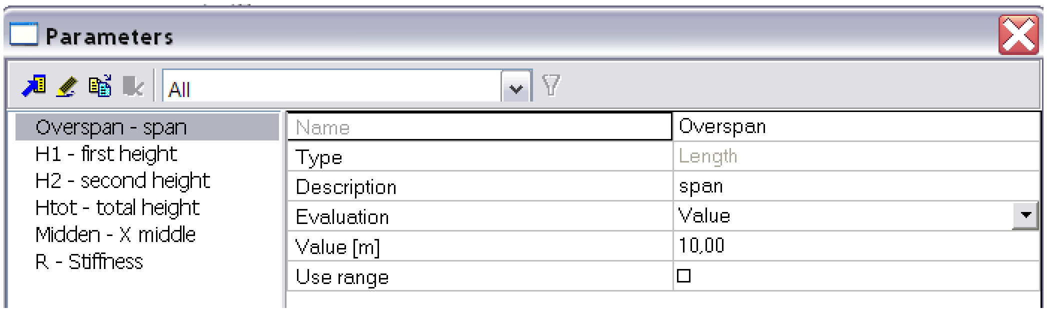
Method:

- First of all, the parameters will be created. Go to ‘Tools > Parameters’:



Tools
- Line grids
- Parameters
- Parameters template settings

- The following properties are parameterized with a begin value of:
  1. Span L: 10 m
  2. Height H1: 5 m
  3. Height H2: 1,5 m
  4. Htot: H1 + H2
  5. Middle: L/2
  6. Rigidity support: $5*10^{-6}$

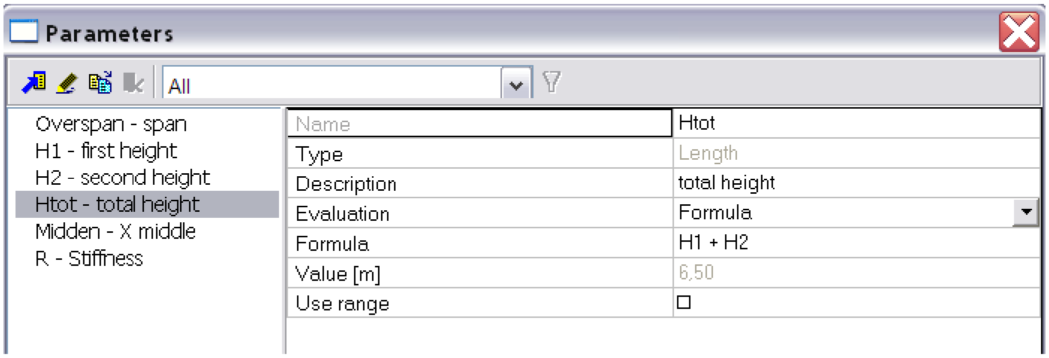


Parameters

All

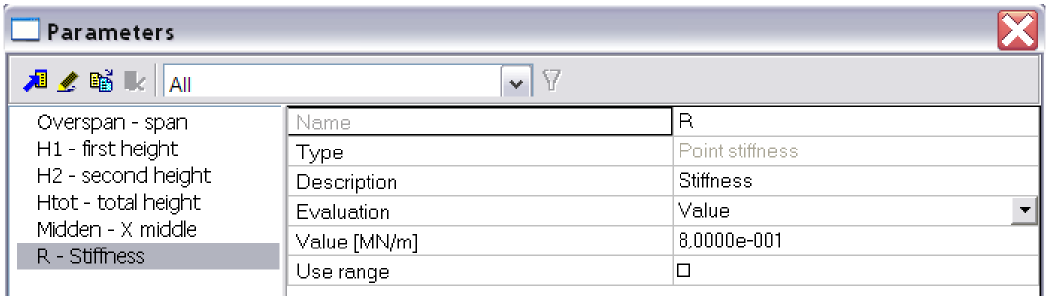
- Overspan - span
- H1 - first height
- H2 - second height
- Htot - total height
- Midden - X middle
- R - Stiffness

Name	Overspan
Type	Length
Description	span
Evaluation	Value
Value [m]	10,00
Use range	□



Overspan - span
H1 - first height
H2 - second height
Htot - total height
Midden - X middle
R - Stiffness

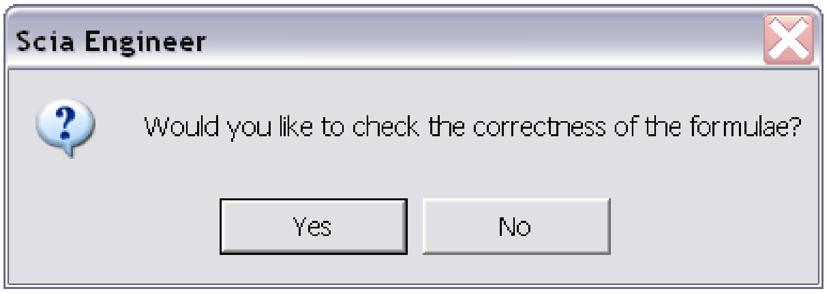
Name	Htot
Type	Length
Description	total height
Evaluation	Formula
Formula	H1 + H2
Value [m]	6,50
Use range	□



Overspan - span
H1 - first height
H2 - second height
Htot - total height
Midden - X middle
R - Stiffness

Name	R
Type	Point stiffness
Description	Stiffness
Evaluation	Value
Value [MN/m]	8,0000e-001
Use range	□

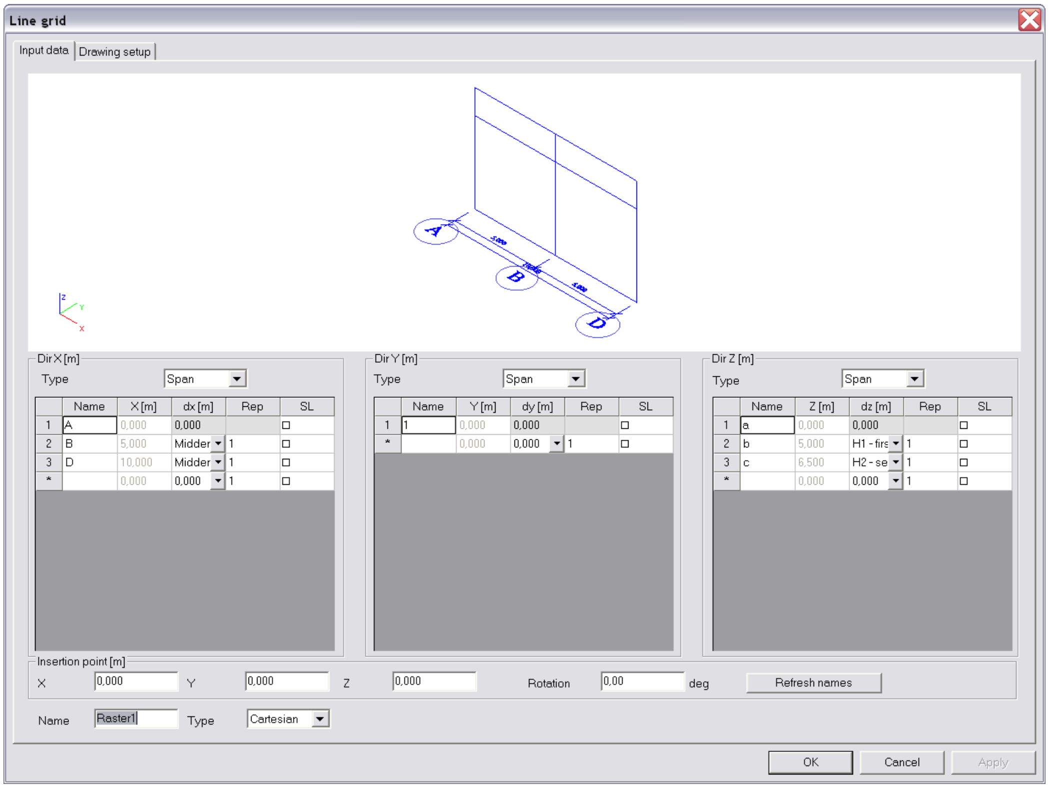
- When closing this window, the following question appears:



Would you like to check the correctness of the formulae?

Click ‘Yes’ if formulae were used so they can be checked.

- Now the parameters are attributed to the various geometry-properties.
- Subsequently a line grid can be made:



Dir X [m]

Type: Span

	Name	X [m]	dx [m]	Rep	SL
1	A	0,000	0,000		□
2	B	5,000	Midder	1	□
3	D	10,000	Midder	1	□
*		0,000	0,000	1	□

Dir Y [m]

Type: Span

	Name	Y [m]	dy [m]	Rep	SL
1	1	0,000	0,000		□
*		0,000	0,000	1	□

Dir Z [m]

Type: Span

	Name	Z [m]	dz [m]	Rep	SL
1	a	0,000	0,000		□
2	b	5,000	H1 - firs	1	□
3	c	6,500	H2 - se	1	□
*		0,000	0,000	1	□

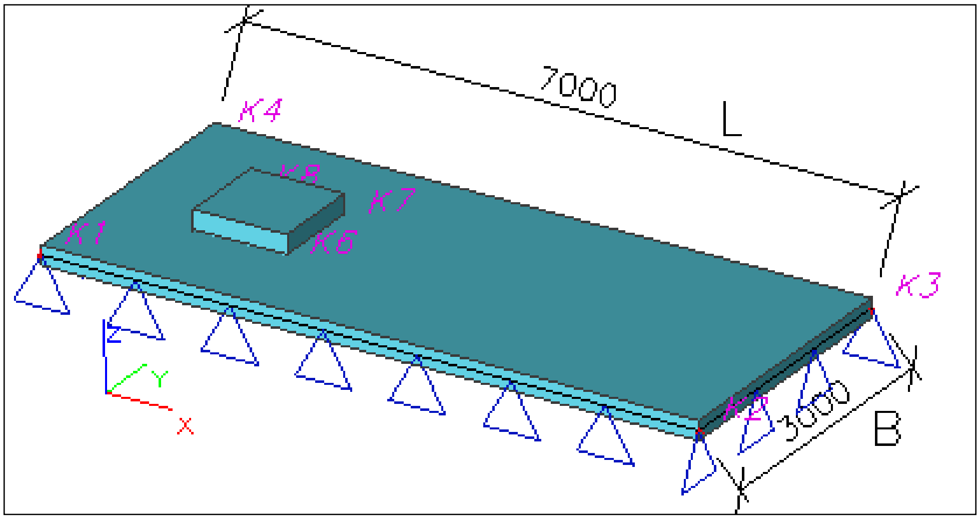
Insertion point [m]

X 0,000 Y 0,000 Z 0,000 Rotation 0,00 deg Refresh names

Name Raster1 Type Cartesian

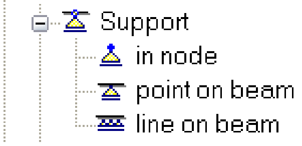
- The line grid can also obtain a parameter to adjust the view. This is possible through a binary parameter with a Boolean value (yes/no).

Plate on subsoil



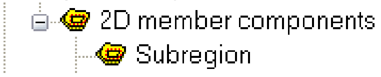
- Project data: - Construction type: General XYZ
  - Project level advanced

- Input support:



Support
- in node
- point on beam
- line on beam

- Input subregion:



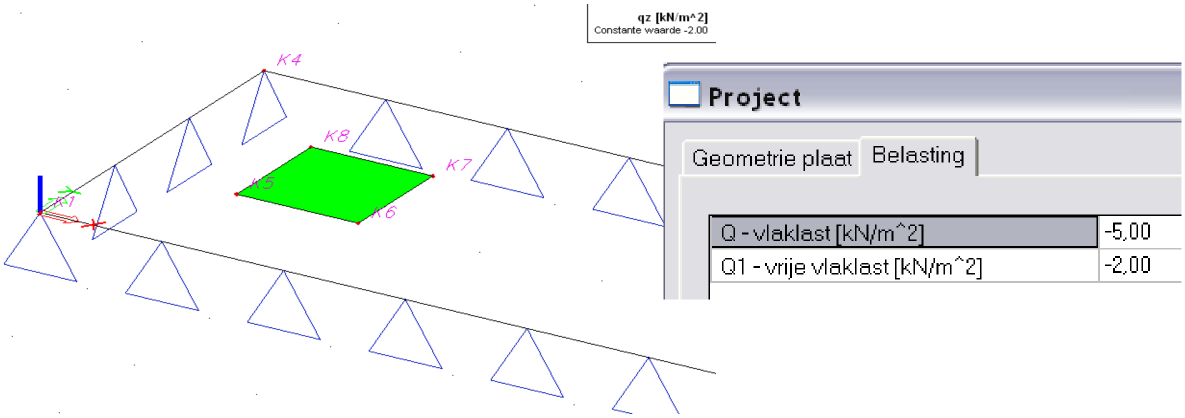
2D member components
- Subregion

- Load cases:
- LC1: Selfweight
- LC2: Permanent plane load q [kN/m^2]
- LC3: Free surface load on projected area: q [kN/m^2]

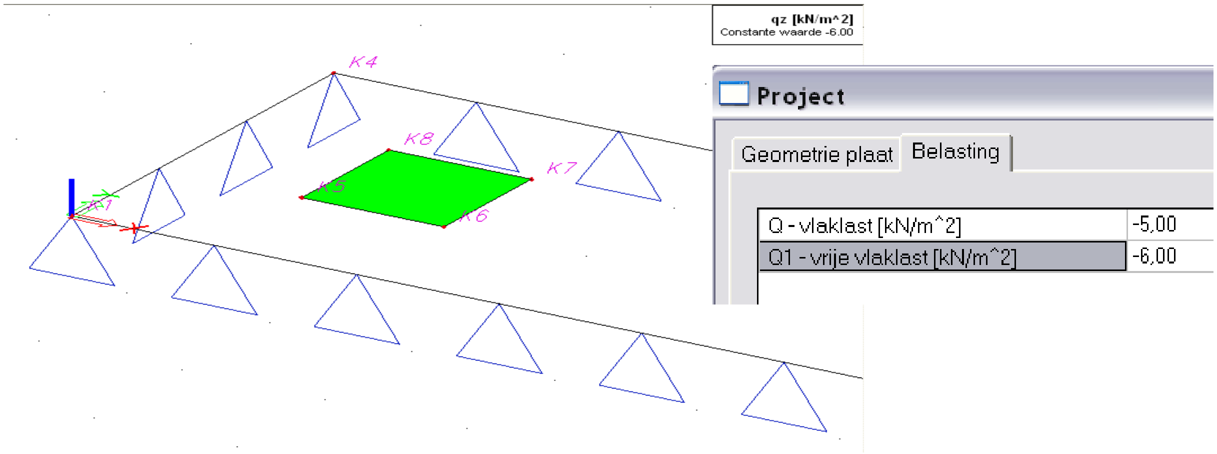
Method

- The following parameters are applied:
- Length L: 7 m
- Width B: 3 m
- Permanent plane load: 5 kN/m^2
- Free plane load : 2 kN/m^2
- Thickness plate d: 200 mm
- Thickness subregion d1: 400 mm

- Adjusting the value of the free plane load:

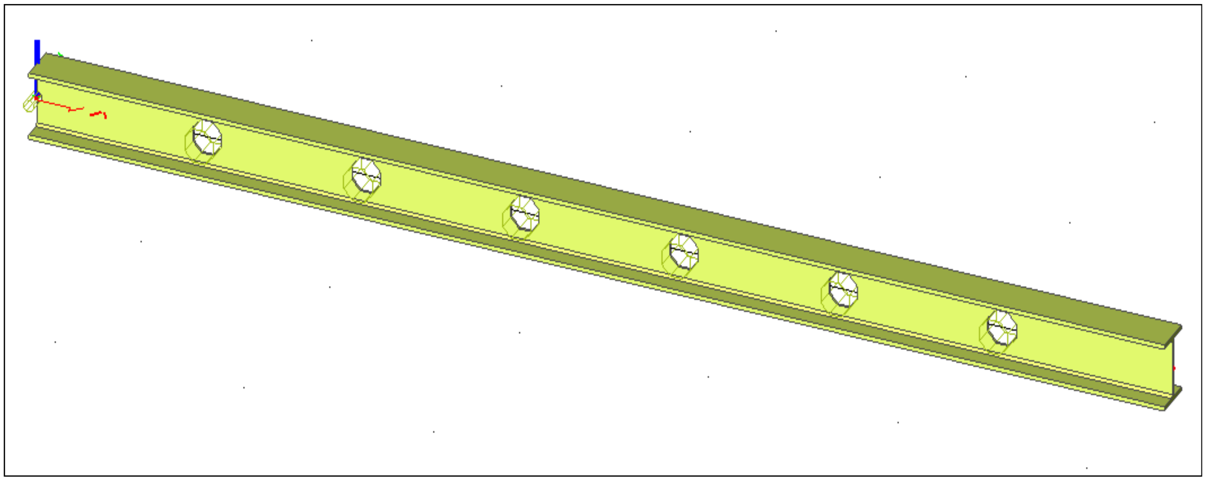


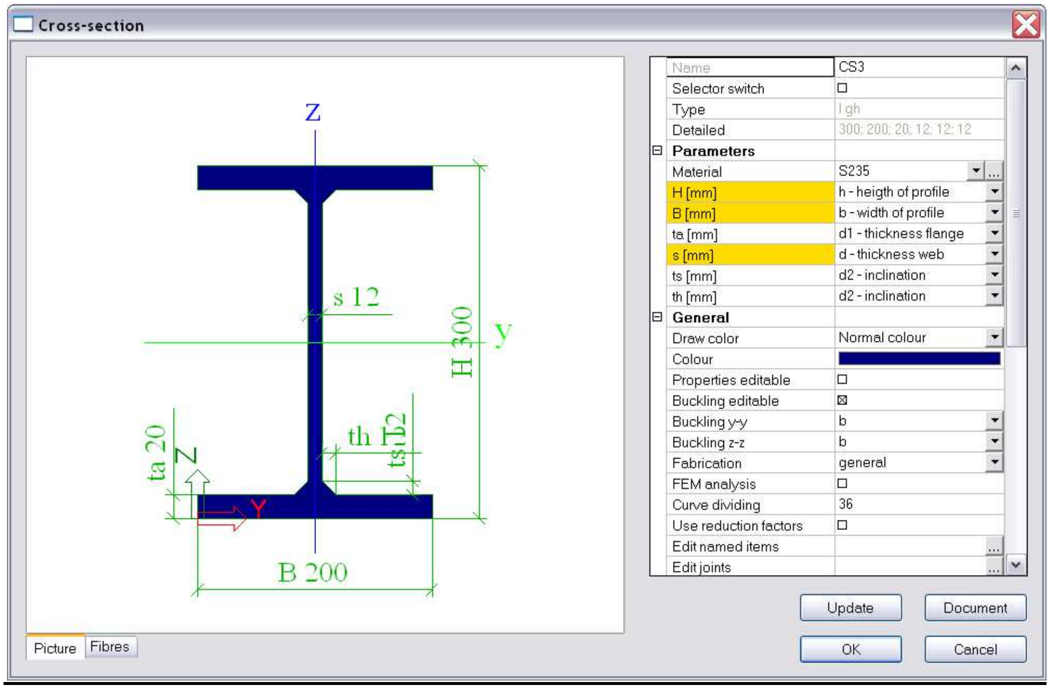
Q - vlaklast [kN/m^2]	-5,00
Q1 - vrije vlaklast [kN/m^2]	-2,00



Q - vlaklast [kN/m^2]	-5,00
Q1 - vrije vlaklast [kN/m^2]	-6,00

Cellular beam



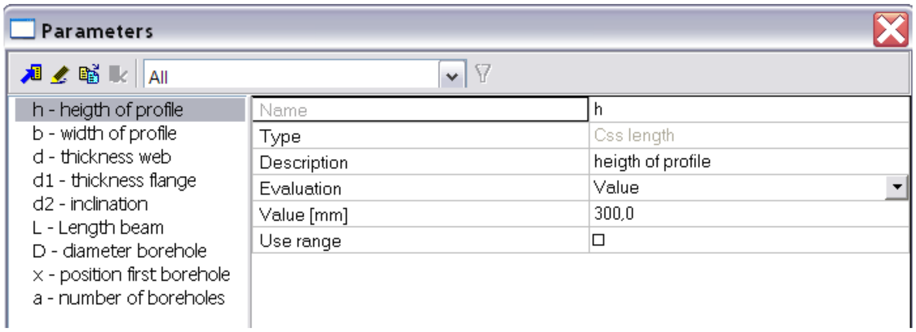


Name	CS3
Selector switch	□
Type	I gh
Detailed	300; 200; 20; 12; 12; 12
⊟ **Parameters**	
Material	S235
H [mm]	h - heigth of profile
B [mm]	b - width of profile
ta [mm]	d1 - thickness flange
s [mm]	d - thickness web
ts [mm]	d2 - inclination
th [mm]	d2 - inclination
⊟ **General**	
Draw color	Normal colour
Colour	■
Properties editable	□
Buckling editable	☒
Buckling y-y	b
Buckling z-z	b
Fabrication	general
FEM analysis	□
Curve dividing	36
Use reduction factors	□
Edit named items	...
Edit joints	...

- Project data: - Construction type: General XYZ
  - Project level advanced
- Input opening through: ‘Structure > 1D member > Modificator > Opening’.

Method

- The following parameters are attributed to the member:

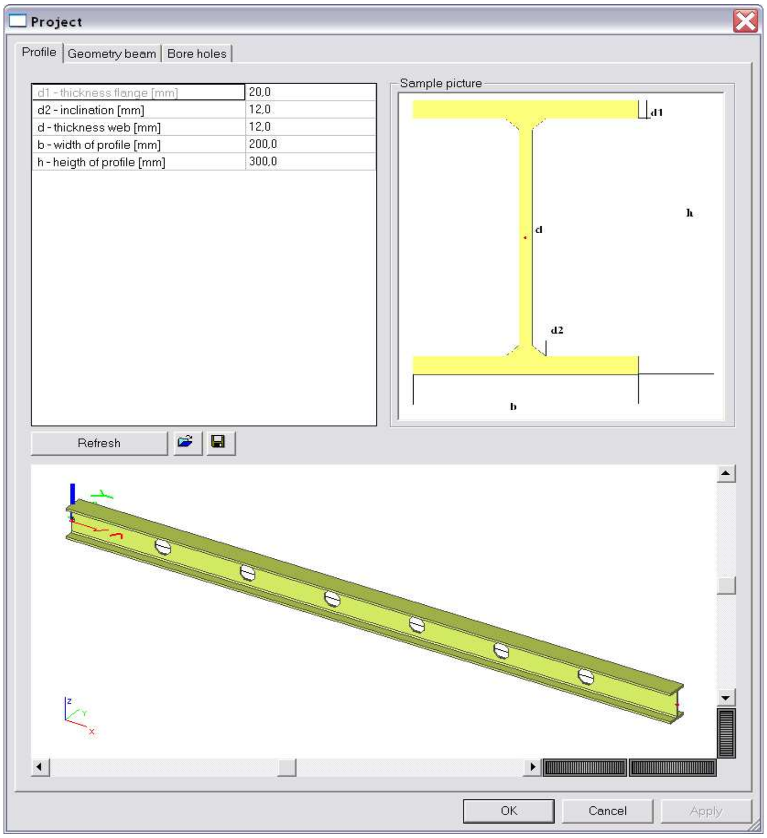


h - heigth of profile
b - width of profile
d - thickness web
d1 - thickness flange
d2 - inclination
L - Length beam
D - diameter borehole
x - position first borehole
a - number of boreholes

Name	h
Type	Css length
Description	heigth of profile
Evaluation	Value
Value [mm]	300,0
Use range	□

- Height profile h: 300 mm
- Width profile b: 200 mm
- Thickness web d: 12 mm
- Thickness flange d1: 20 mm
- fillet d2: 12 mm
- Length member L: 6 m
- Diameter bore hole D: 150 mm
- Position first bore hole x: 0,15
- Number of bore holes a: 6

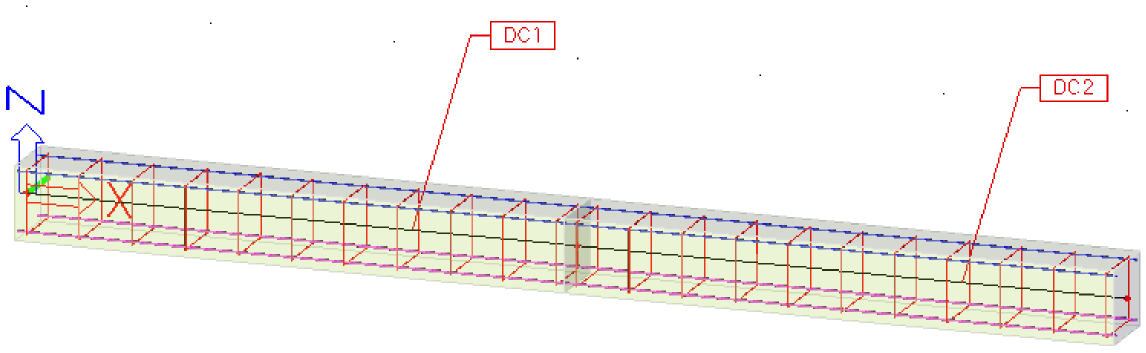
- Next, these items are attributed to the structure.
- Subsequently the template settings and template dialog is made.



d1 - thickness flange [mm]	20,0
d2 - inclination [mm]	12,0
d - thickness web [mm]	12,0
b - width of profile [mm]	200,0
h - heigth of profile [mm]	300,0

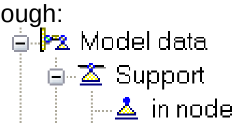
Beam with practical reinforcement

It's also possible to parameterize the reinforcement. This will be shown in following example.



- Project data:
  - Construction type: Frame XZ
  - Project level advanced

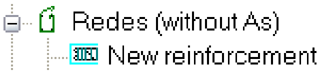
- Input through:



Model data
Support
in node

- Load cases:
  - LC1: Selfweight
  - LC2: Variable load: q [kN/m]

- Input practical reinforcement:



Redes (without As)
New reinforcement

Method

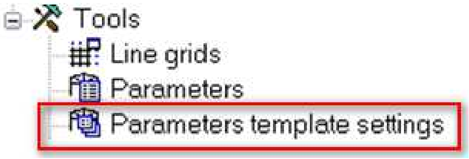
- Input of the following parameters with begin value:

- Height profile H: 400 mm
- Width profile B: 400 mm
- Span L1: 3m
- Span L2: 3m
- Total span Ltot: L1 + L2
- Variable line load Q: 15 kN/m
- Reinforcement diameter above db: 12 mm
- Reinforcement diameter below do: 16 mm
- Reinforcement diameter stirrups dstirrup: 8 mm

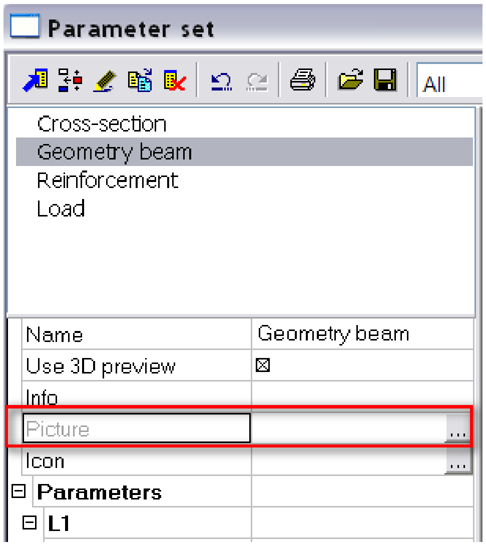
Remark: the reinforcement diameters can only be parameterized when using the practical reinforcement.

- Stirrup distance a: 0,3 m

- Group parameters through:



In the dialogue box of the parameter template settings, pictures can be attributed from the gallery:

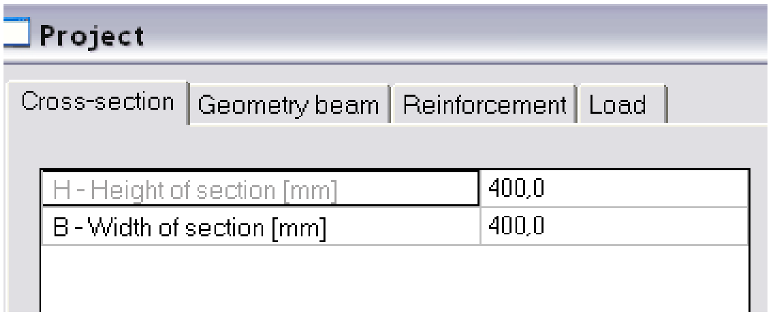


Name	Geometry beam
Use 3D preview	☒
Info	
Picture	...
Icon	...
⊟ **Parameters**	
⊟ **L1**	

You get the best resolution if you choose the following properties during saving:

- .wmf
- 550 x 550

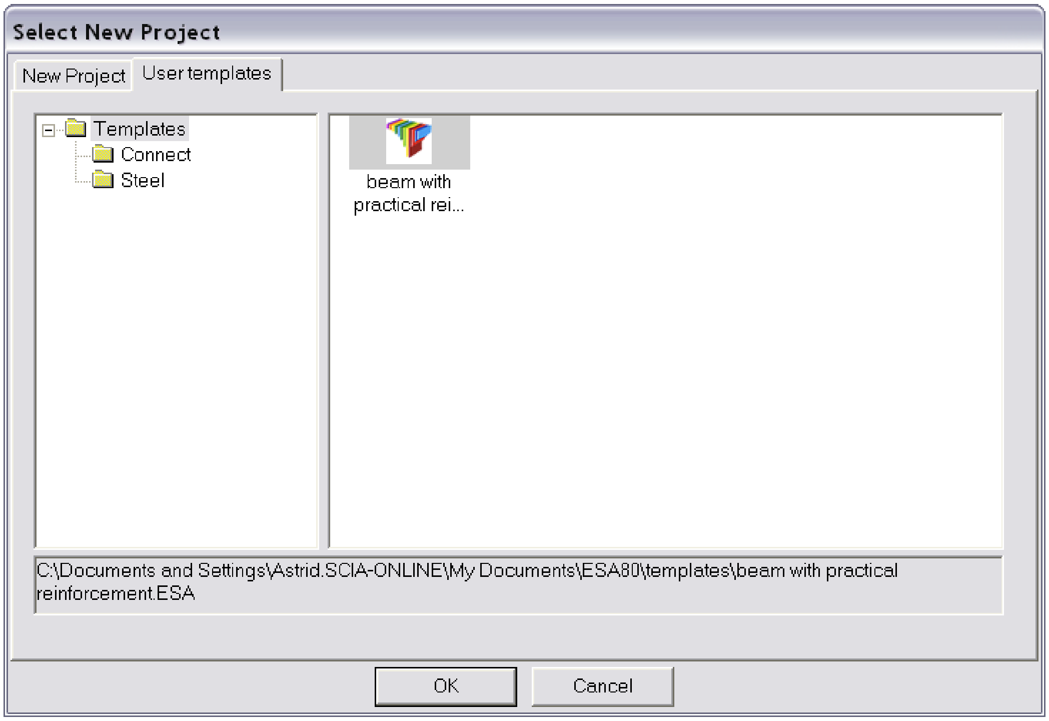
- Check through Template dialogue



H - Height of section [mm]	400,0
B - Width of section [mm]	400,0

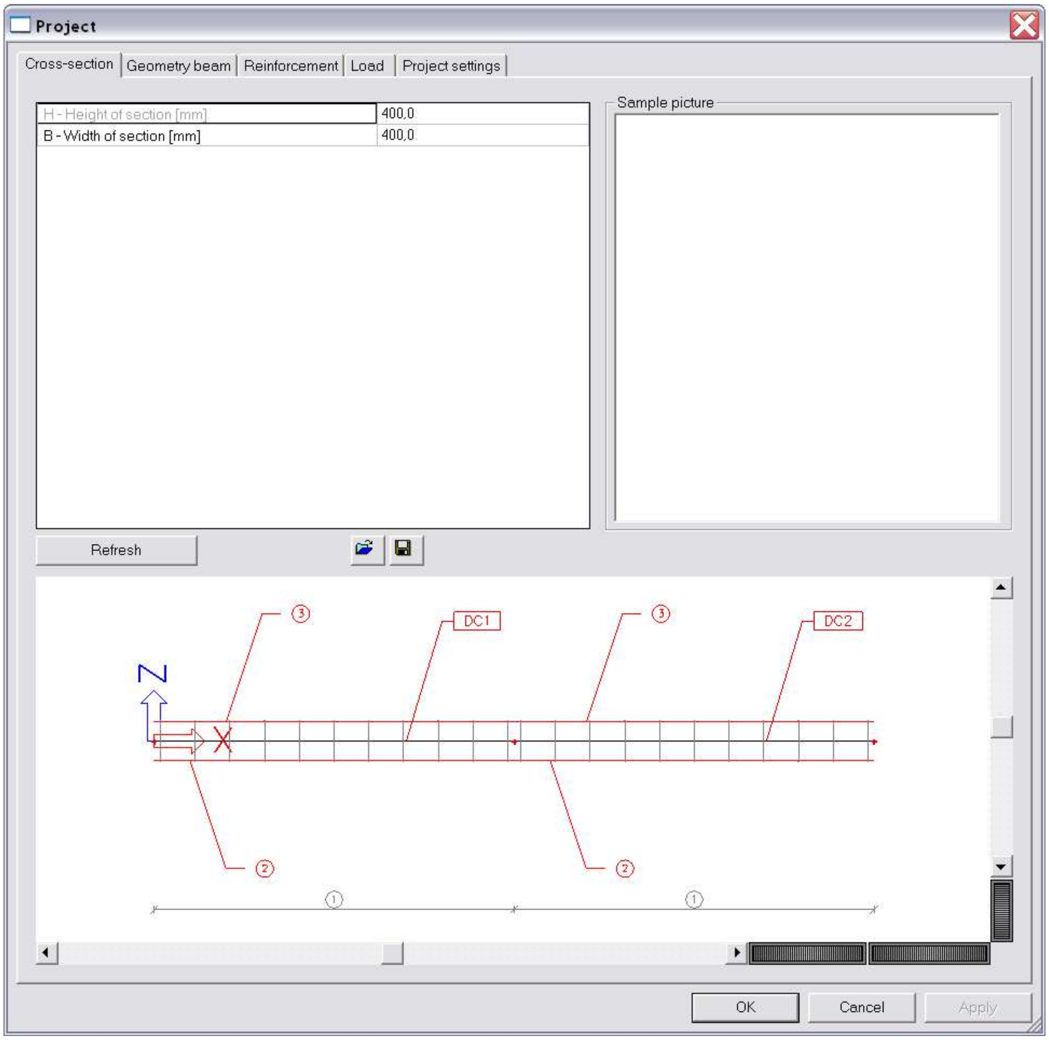
- Afterwards, the entire project can be saved as a template.
  This by doing the following: save in the directory ‘Esa xx > Templates’

- The project can now be opened as a template and adjusted:



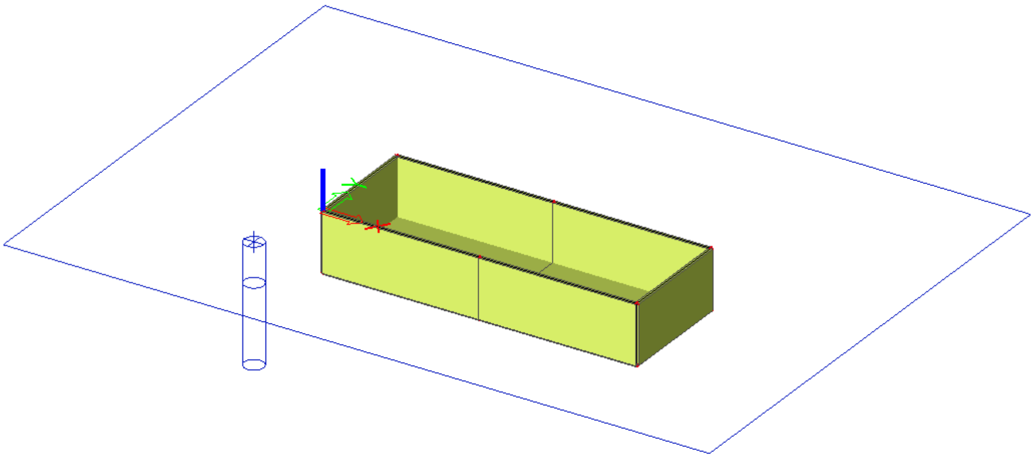
C:\Documents and Settings\Astrid.SCIA-ONLINE\My Documents\ESA80\templates\beam with practical reinforcement.ESA

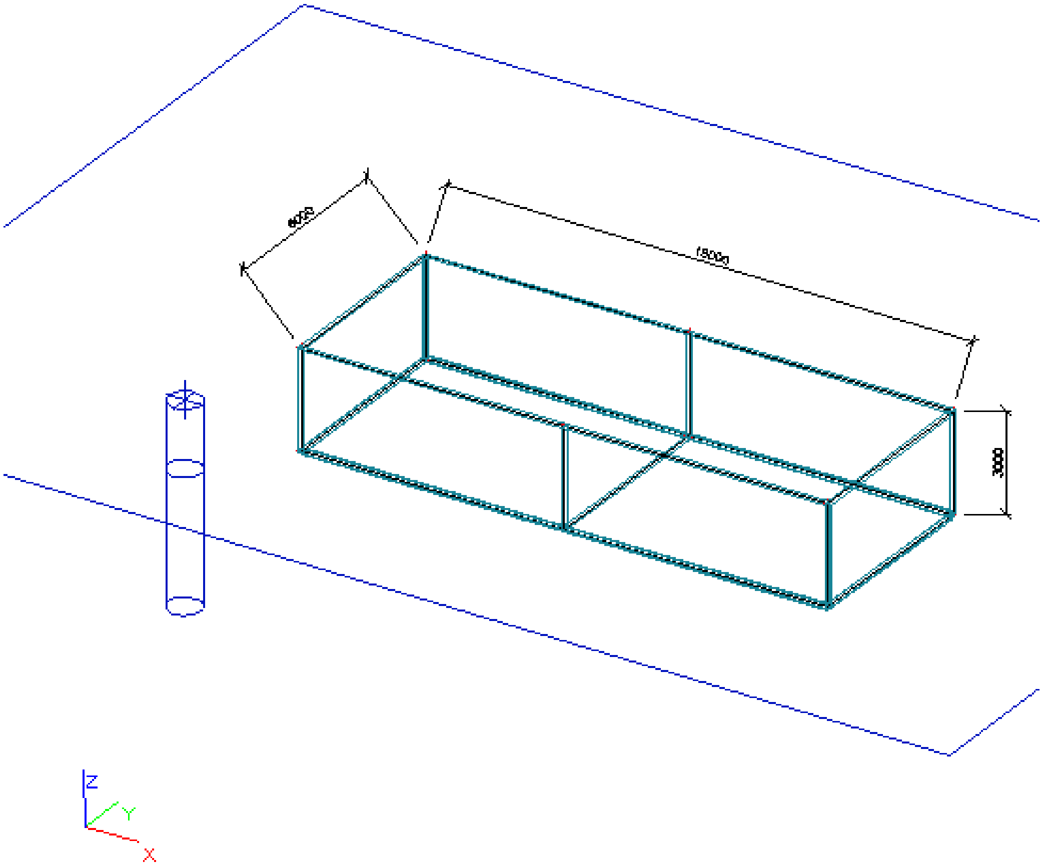
- After opening this template, a dialog box with the entered parameters appears. If desired, particular properties can be adjusted here.



H - Height of section [mm]	400,0
B - Width of section [mm]	400,0

Ground-level

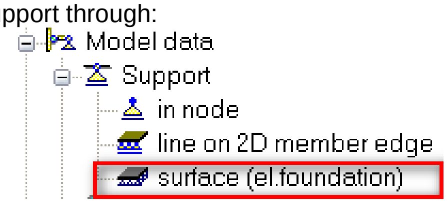




- Project data:
  - Construction type: General XYZ
  - Project level advanced

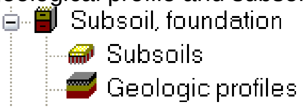
- Functionalities :
  - Subsoil
  - Soil loads
  - Parameters

- Input support through:

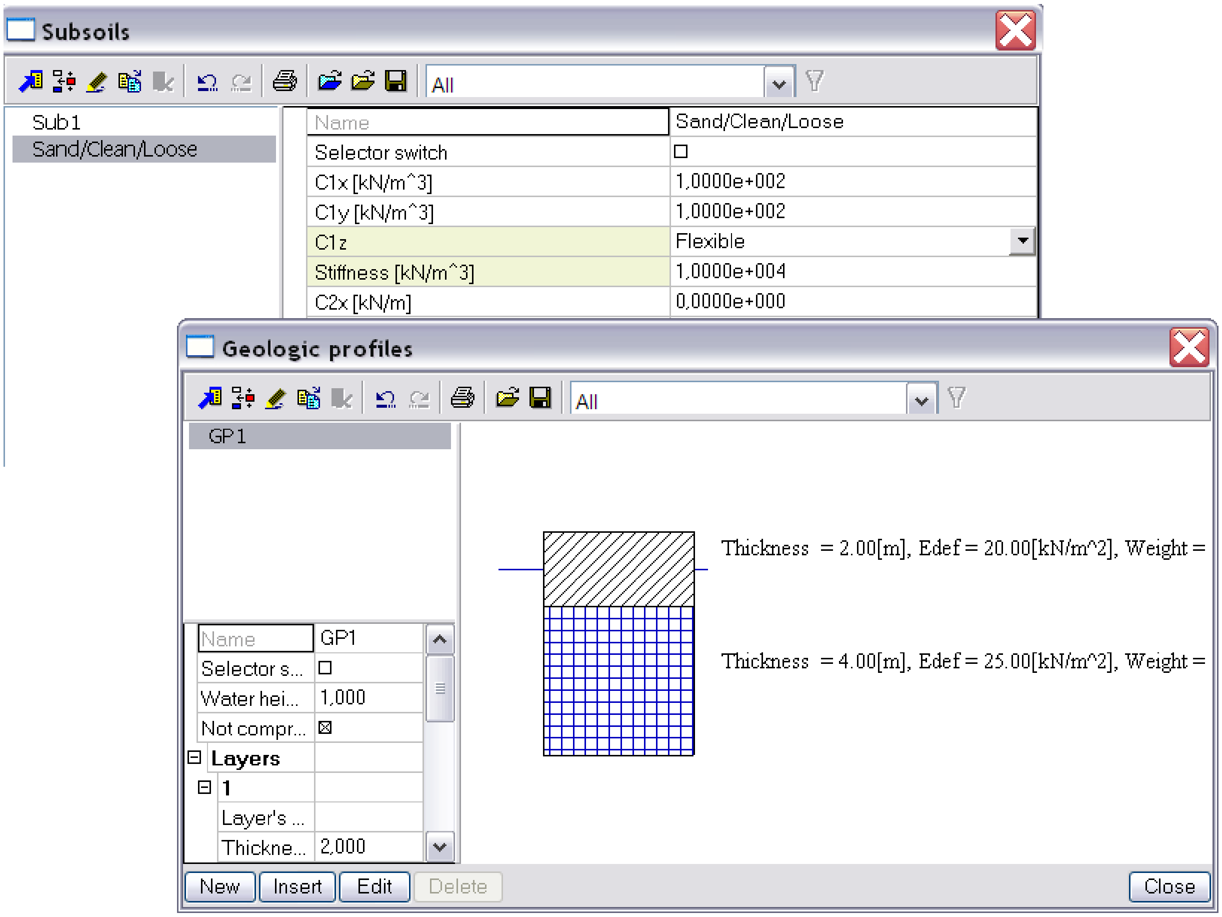


Model data
- Support
  - in node
  - line on 2D member edge
  - surface (el.foundation)

- Input geological profile and subsoil through:



Subsoil, foundation
- Subsoils
- Geologic profiles



Sub1
Sand/Clean/Loose

Name	Sand/Clean/Loose
Selector switch	□
C1x [kN/m^3]	1,0000e+002
C1y [kN/m^3]	1,0000e+002
C1z	Flexible
Stiffness [kN/m^3]	1,0000e+004
C2x [kN/m]	0,0000e+000

GP1

Name	GP1
Selector s...	□
Water hei...	1,000
Not compr...	☒
⊟ Layers	
⊟ 1	
Layer's ...	
Thickne...	2,000

Thickness = 2.00[m], Edef = 20.00[kN/m^2], Weight =

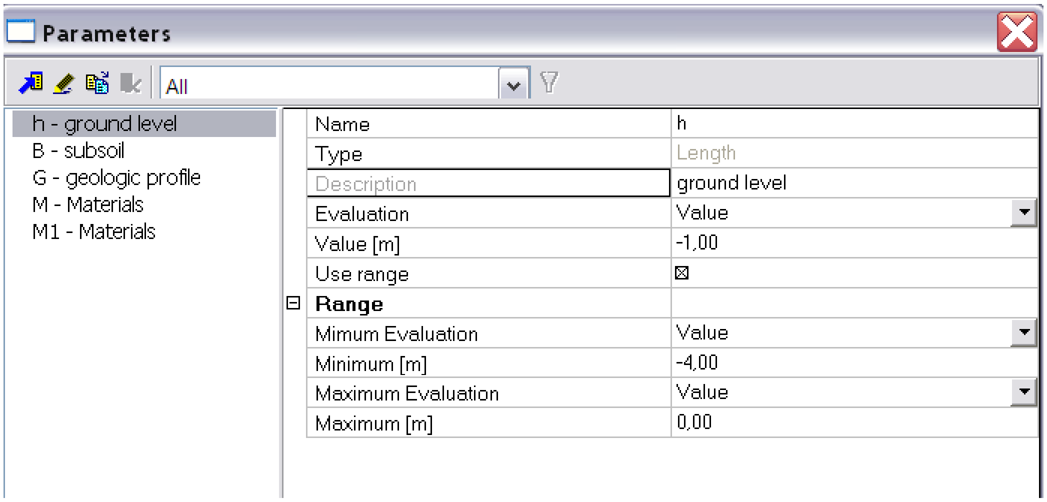
Thickness = 4.00[m], Edef = 25.00[kN/m^2], Weight =

- Load cases: - Selfweight
  - Water pressure
  - Soil pressure

Method

- Parameter:

- Height bore hole profile h: -1 m, restriction between 0 m and -4 m



Parameters

All

h - ground level
B - subsoil
G - geologic profile
M - Materials
M1 - Materials

Name	h
Type	Length
Description	ground level
Evaluation	Value
Value [m]	-1,00
Use range	☒
⊟ **Range**	
Mimum Evaluation	Value
Minimum [m]	-4,00
Maximum Evaluation	Value
Maximum [m]	0,00

- Test of input data through:

2D data viewer

Surface loads

Layers

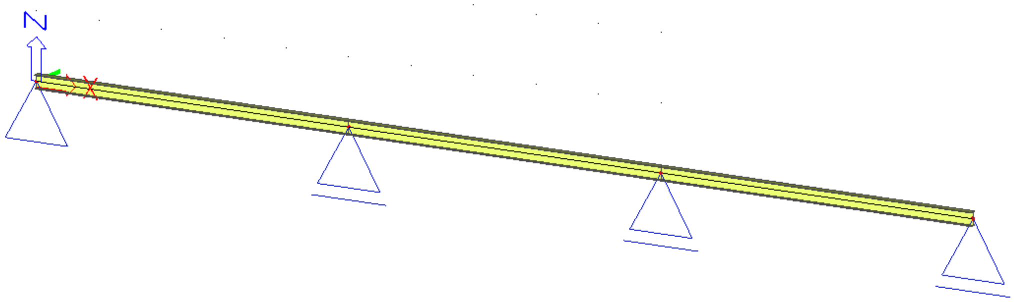
In this example the number of spans will be varied. This can be done by means of a truc. The basic principle is: every beam belongs to a layer.

Above this a layer has two properties: Activity and Structural type (also Drawing model).

The principle is as follows:
A layer that is not active will not be displayed on the screen.
A layer that has the structural type will not be calculated.

This means that if you put a beam into a layer that is both non-active as in the structural model, it will not be visible and above this, it will not be calculated.

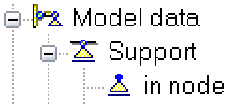
On that way we can make parameters for the properties of the layers.



- Project data:
- Construction type: Frame XZ
- Project level advanced

- Functionalities - Parameters

- Input support through:



Method

- The number of members is varied. This is done by means of Parameterizing the activity and the type of model (analysis/structural)

- Input parameters:

1. Number of members N: 3
   Remark:
   The maximal number of beams is the number of beams that had been introduced in the project.

2. Activity of the layers : → Boolean
   - L1: formula N>=1
   - L2: formula N>=2
   - L3: formula N>=3

3. Model type of the layers: →Boolean
   - C1: formula not (L1)
   - C2: formula not (L2)
   - C3: formula not (L3)

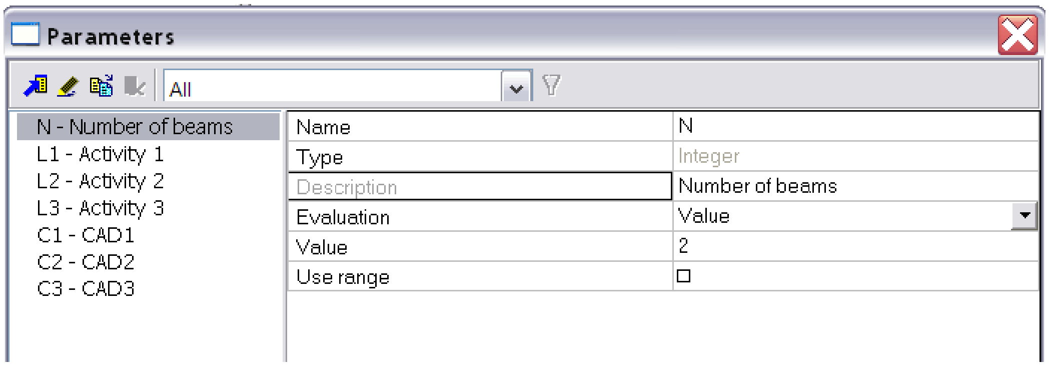
Example:

N=2:

➔ L1 and L2: true, L3 is false

➔ C1 and C2: false, C3 is true

This means that Layer 1 and 2 are visible and not in the structural model. Layer 3 will not be displayed and will be in the drawing model (not in the analysis model).



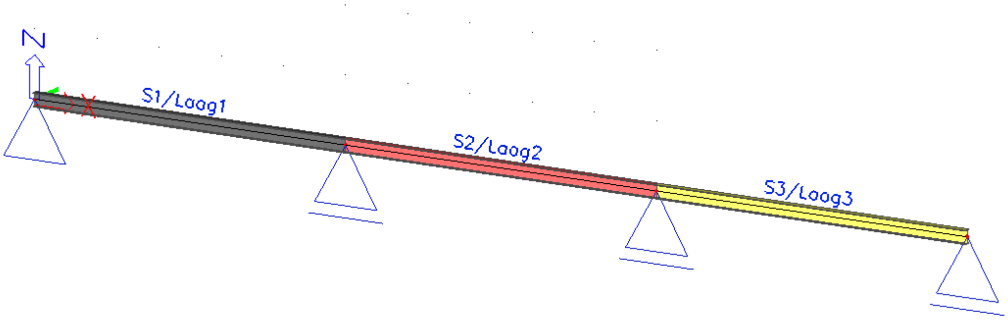
Parameters

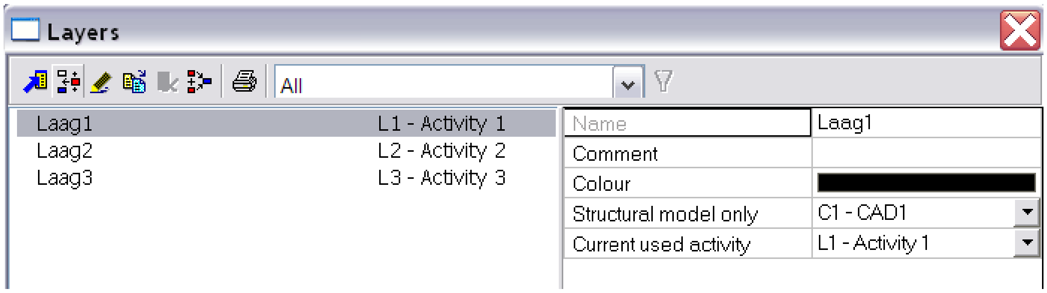
All

- N - Number of beams
- L1 - Activity 1
- L2 - Activity 2
- L3 - Activity 3
- C1 - CAD1
- C2 - CAD2
- C3 - CAD3

Name	N
Type	Integer
Description	Number of beams
Evaluation	Value
Value	2
Use range	□

- Grouping the parameters through: Tools > Parameters template settings
- Subsequently a parameter is attributed to the various layers:



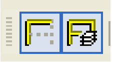


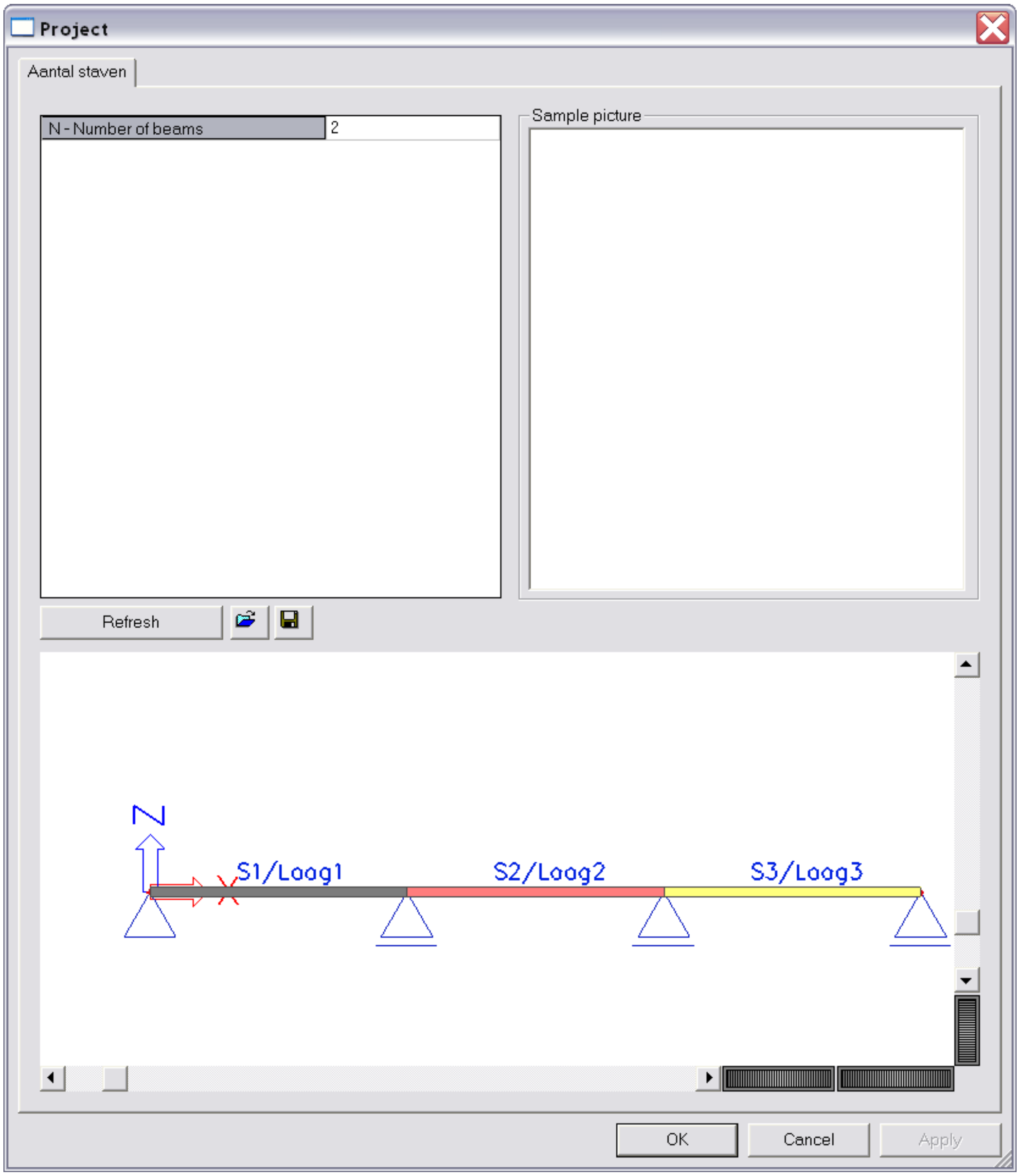
Layers

All

Laag1	L1 - Activity 1
Laag2	L2 - Activity 2
Laag3	L3 - Activity 3

Name	Laag1
Comment	
Colour	■
Structural model only	C1 - CAD1
Current used activity	L1 - Activity 1

- Variation number of member through the template dialogue
- Use the activity toggle
    - 

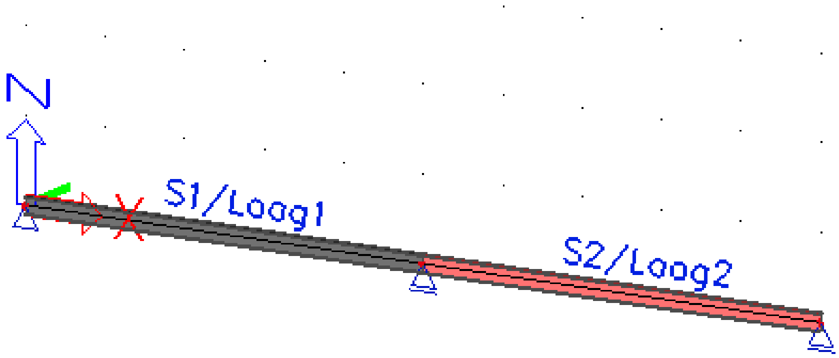


Project

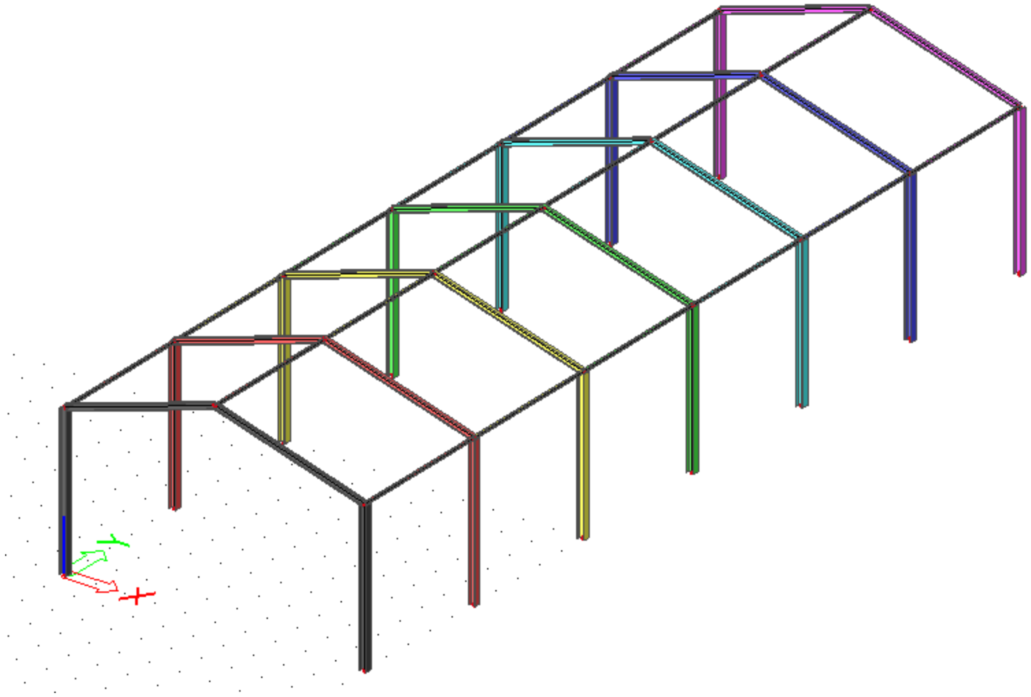
Aantal staven

N - Number of beams	2

This gives as result:



Steel hall



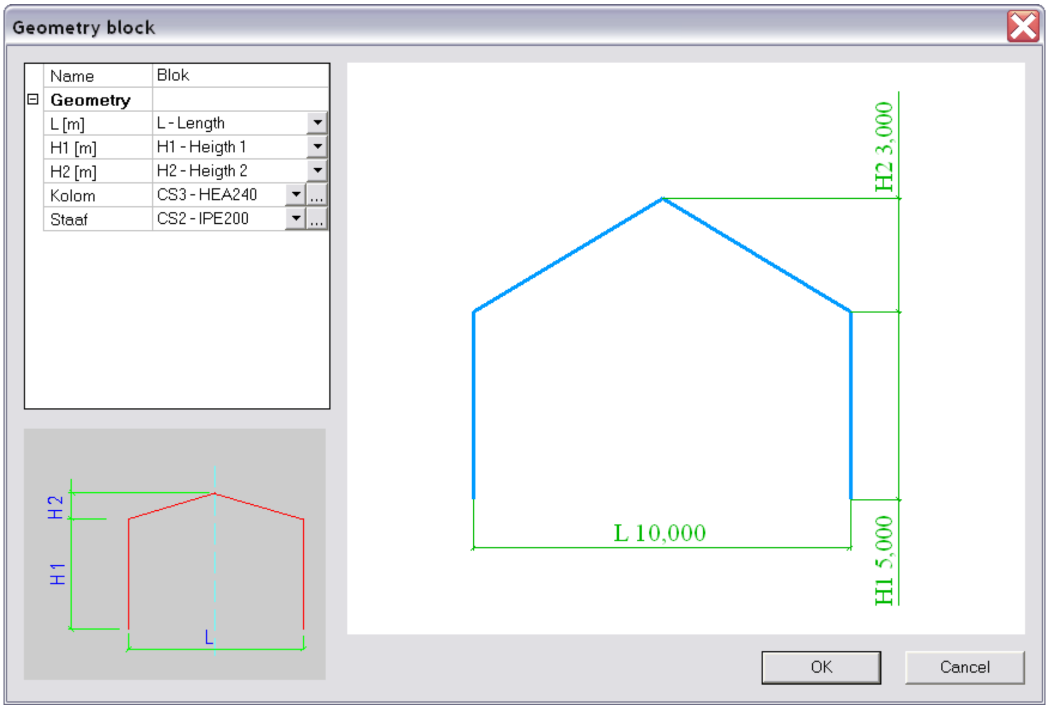
In this example, the number of spans will be parameterized. The same principle with the layers as in the previous example will be used.

- Project data:
- Construction type: Frame XYZ
- Project level advanced

- Functionalities - Parameters

- Input support through: Support > nodal support.

- Input catalogue block with parameters H1, H2 and L



Geometry block

Name	Blok
⊟ **Geometry**	
L [m]	L - Length
H1 [m]	H1 - Heigth 1
H2 [m]	H2 - Heigth 2
Kolom	CS3 - HEA240
Staaf	CS2 - IPE200

Method

- The number of frames is varied, ranging from 1 up to and including 7, through the binary parameters for the model type and the activity.

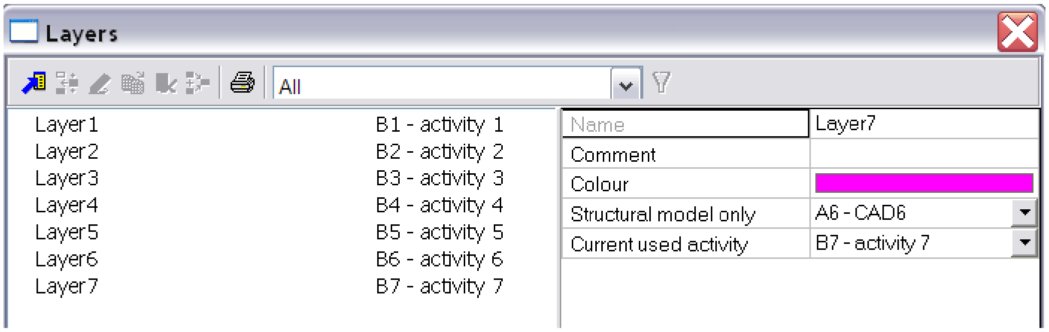
- Applying the following parameters:
    - Height H1: 5 m
    - Height H2: 1,5 m
    - Total Height Htot: H1 + H2
    - Length L: 10 m
    - Number of frames NS: 7
    - Frame distance S: 5 m
    - Frame distance 1 t/m 5 (S1/5): S*1 ... S*5
    - Total frame distance S6: S*(NS-1)
    - Activity layers B1 t/m B7: NS >=1 .. NS>=7
    - Construction model layers A1 t/m A7: not (B1) … not (B7)

  At which the following parameter types are used:

    - Height, length and frame distance : length
    - Number of frames: integer
    - Activity and type of model layers: boolean

- Now the parameters can be attributed to the properties of the model:

- Layers



Layer1	B1 - activity 1
Layer2	B2 - activity 2
Layer3	B3 - activity 3
Layer4	B4 - activity 4
Layer5	B5 - activity 5
Layer6	B6 - activity 6
Layer7	B7 - activity 7

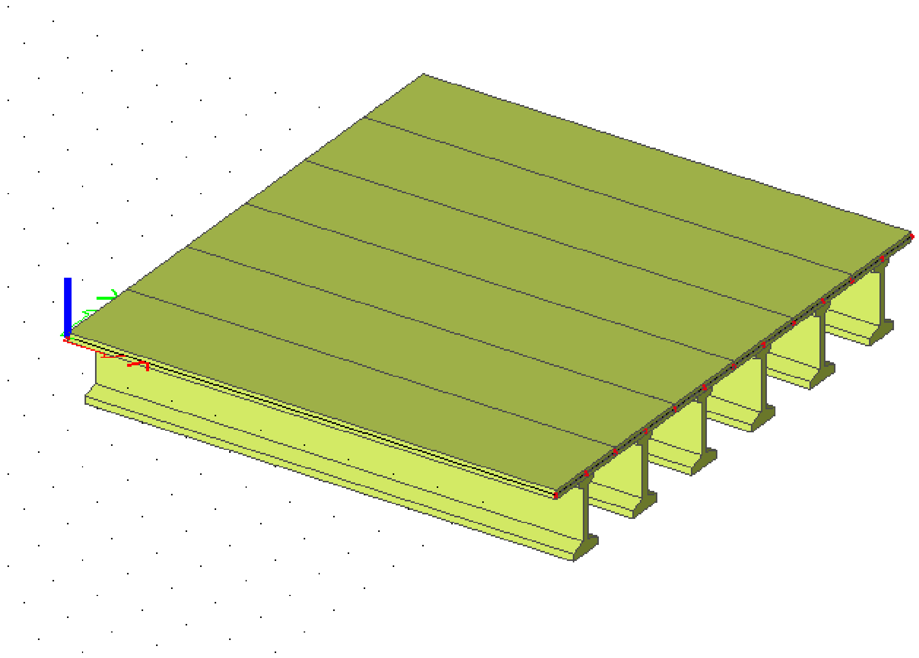
Name	Layer7
Comment	
Colour	🟪
Structural model only	A6 - CAD6
Current used activity	B7 - activity 7

- Geometry properties of the catalogue block
- Coordinate Y of the nodes of the model: parameter frame distance

Plate with ribs

Two types of parameterized plate with ribs are shown here. The previous principle of the properties of the layers is used here.

Plate with ribs_1.esa



- In the first case the number of ribs is dependent on the width of the plate. The offset of the ribs and the distance between several ribs remains the same in each case.

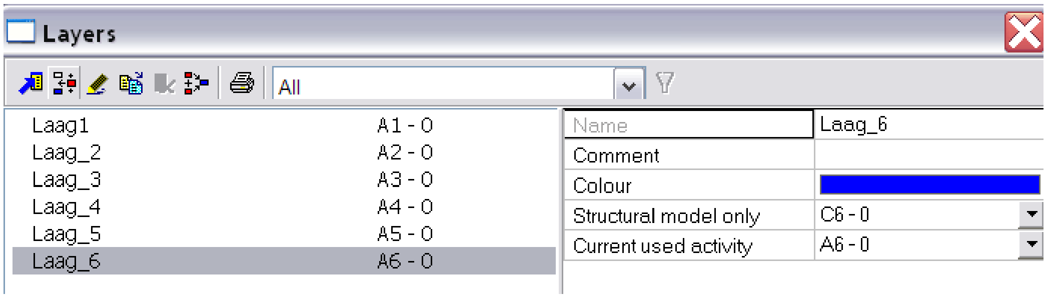
- To obtain this, we divide the plate into several parts. The number of the parts is equal to the maximal number of ribs.

- In this case we the maximal number of ribs is 6.
  So, a parameter N with type 'integer' is made with standard value 6.

- Other parameters are:
  A1...A6: Boolean
  Formula: N>=1…6

  C1…C6: Boolean
  Formula: not(A1…A6)

- Afterwards, these parameters can be attributed to the properties of the 6 created layers:



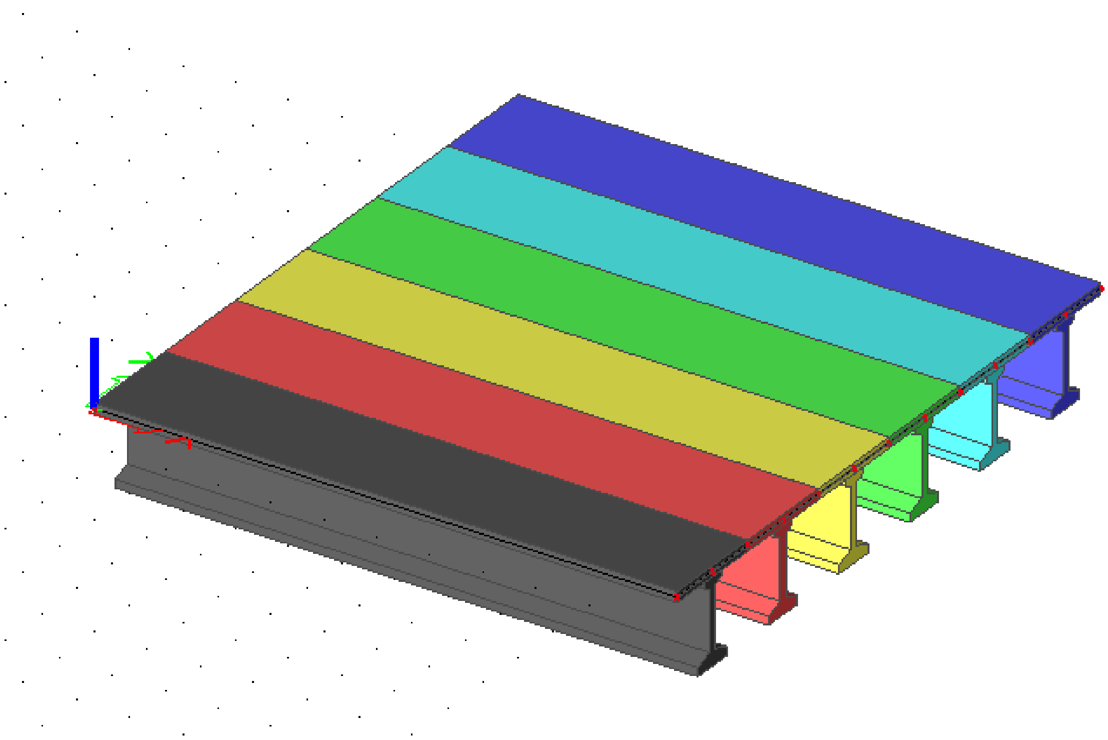
Layers

All

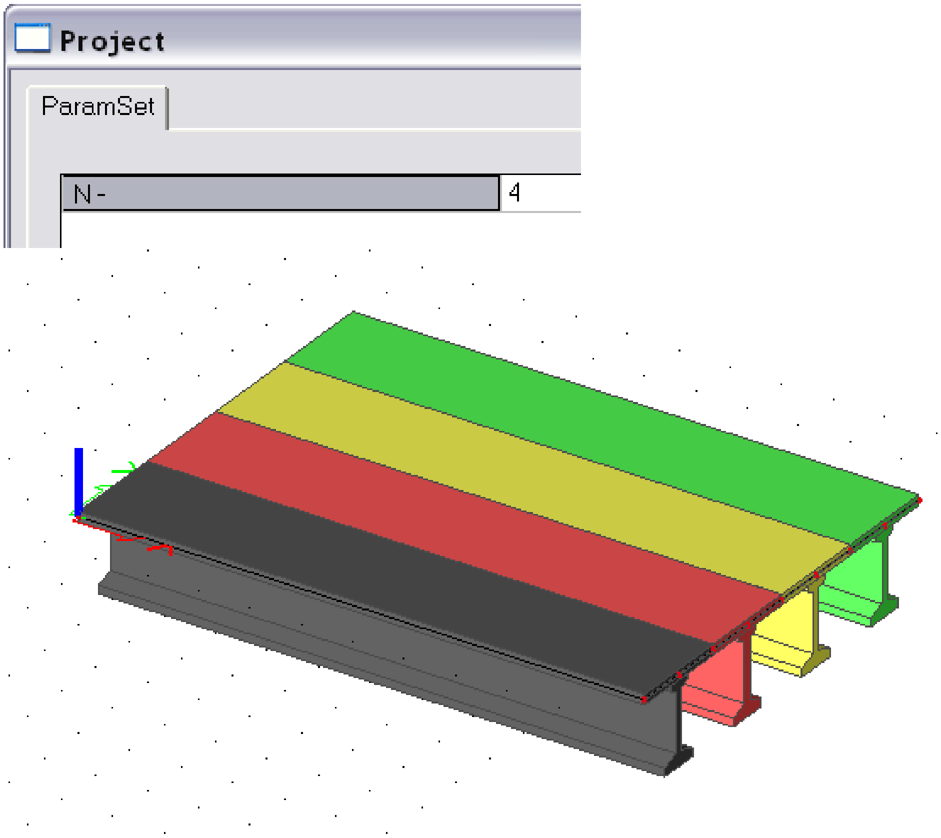
Laag1	A1 - 0
Laag_2	A2 - 0
Laag_3	A3 - 0
Laag_4	A4 - 0
Laag_5	A5 - 0
Laag_6	A6 - 0

Name	Laag_6
Comment	
Colour	🟦
Structural model only	C6 - 0
Current used activity	A6 - 0

The plate with the connected rib can now be input in the respective layer.
In the ‘view parameters for all’ , the colour can be set on ‘colour by layer’.
This is practical to verify if each rib is in the correct layer:



- After creating a group in the ‘parameters template settings’, the number of ribs can be adapted in the Template dialogue:



N - 4

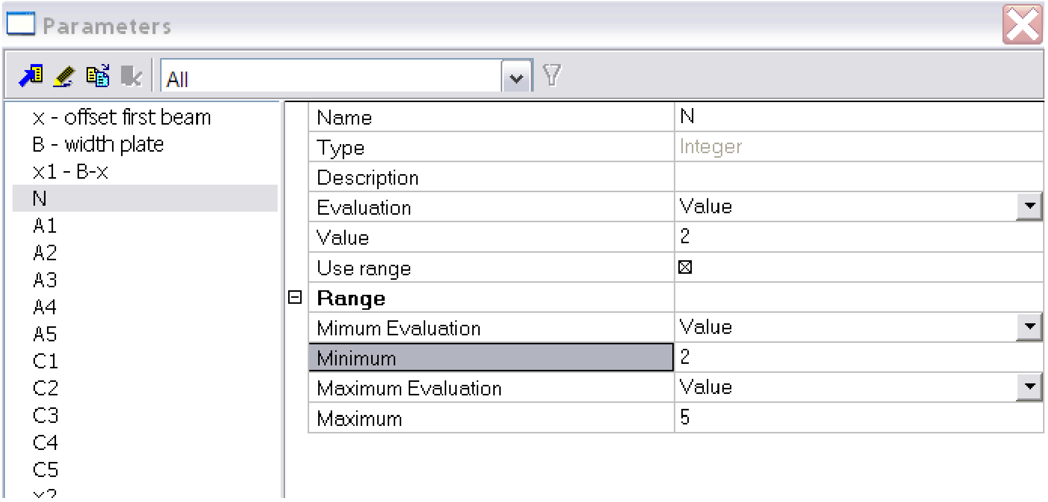
Plate with ribs_2.esa

In this case, the offset between the outer ribs and the end of the plate remain constant. The distance between the inner ribs is always regular.

This is shown in the understanding example.

Originally, a plate with 5 ribs is created. Afterwards, the ribs are input in layers and the x-coordinates are parameterized.

- First of al the parameter for the number of ribs is created:
  This can be an integer value between the range of 2 to 5.



x - offset first beam
B - width plate
x1 - B-x
N
A1
A2
A3
A4
A5
C1
C2
C3
C4
C5
x2

Name	N
Type	Integer
Description	
Evaluation	Value
Value	2
Use range	☒
⊟ **Range**	
Mimum Evaluation	Value
Minimum	2
Maximum Evaluation	Value
Maximum	5

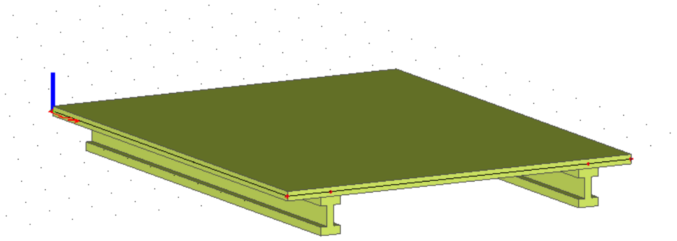
- The offset coordinates of the outer ribs are:
- x for the first ribs
- B- x for the last rib, with B the parameter for the width of the plate

- The 3 inner ribs get the following coordinates:
- x2 = x+((B-2x)/(N-1))
- x3 = x2+((B-2x)/(N-1))
- x4 = x3+((B-2x)/(N-1))

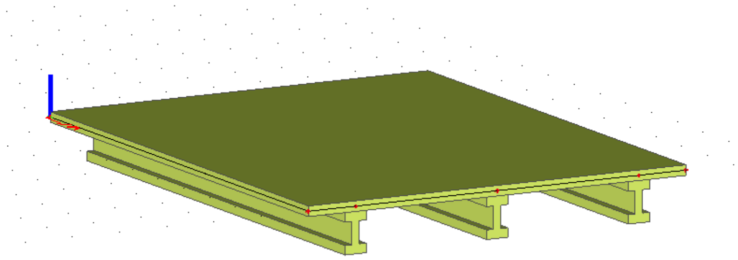
- As in the previous examples, parameters A1..A5 and C1..C5 are created for the properties of the layers
- To finish this, the ribs are attributed to the correct layers.

- When changing the number N in the template dialogue, we get the following results:

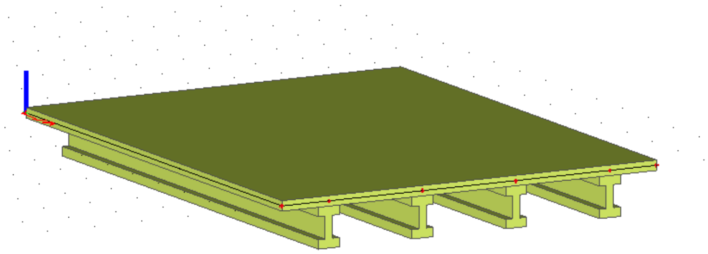
  - N = 2



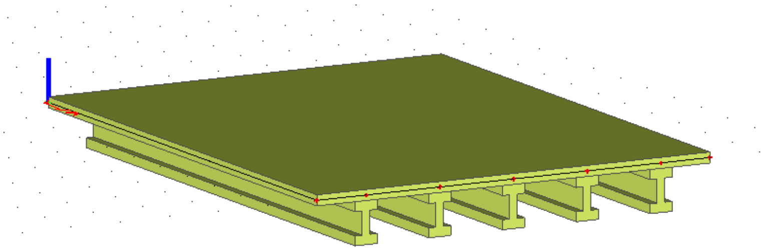
  - N = 3



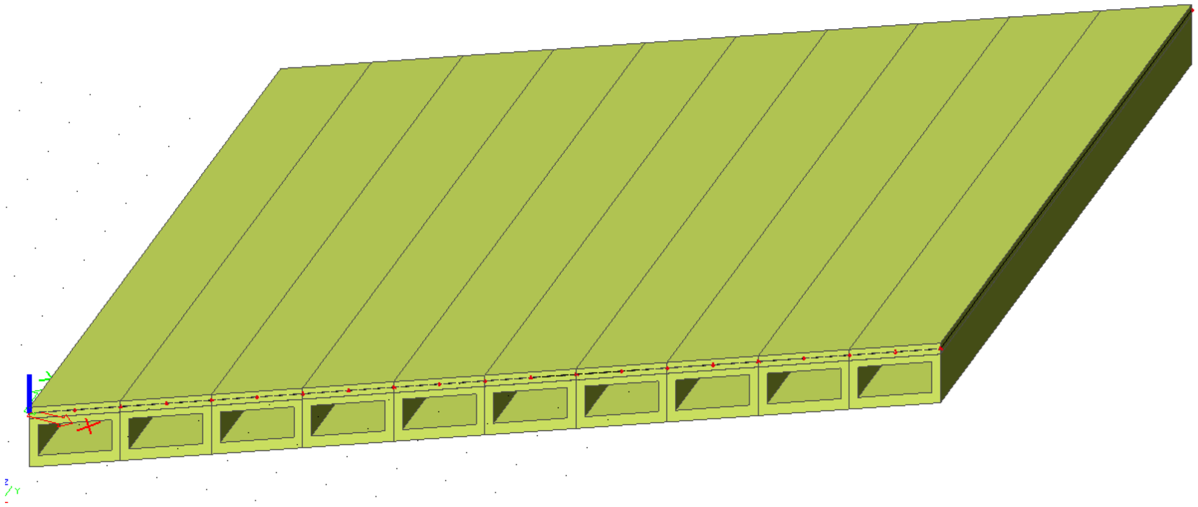
  - N = 4



  - N = 5



Tubular plates



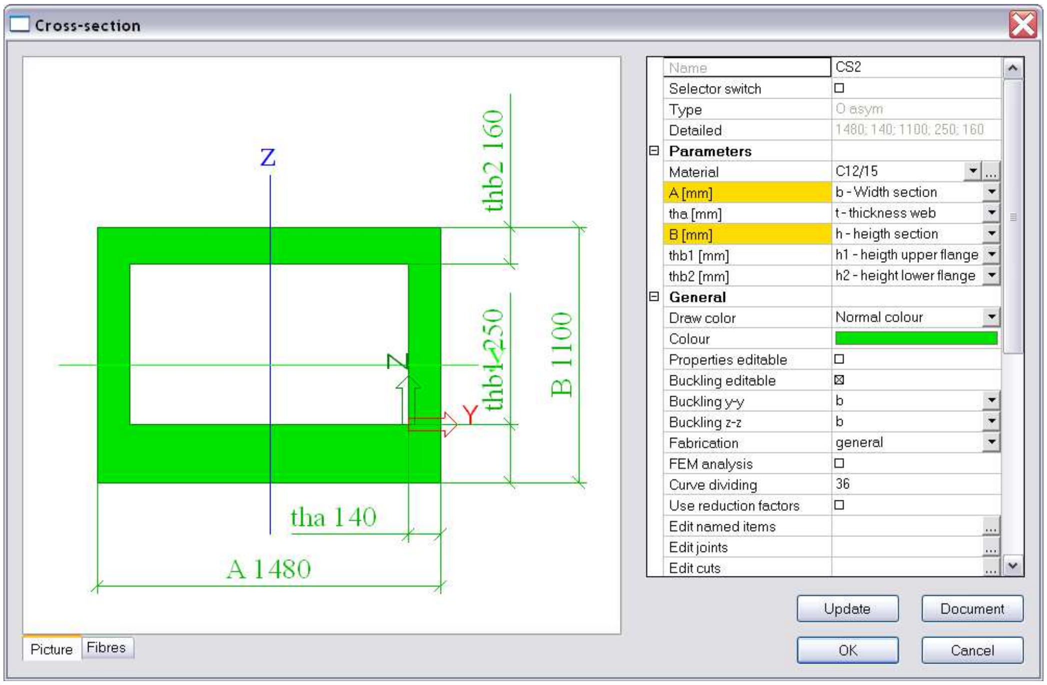
- Project data:
- Construction type: General XYZ
- Project level advanced

- Functionalities – Parameters
- Input channels as ribs: Plate rib

Method

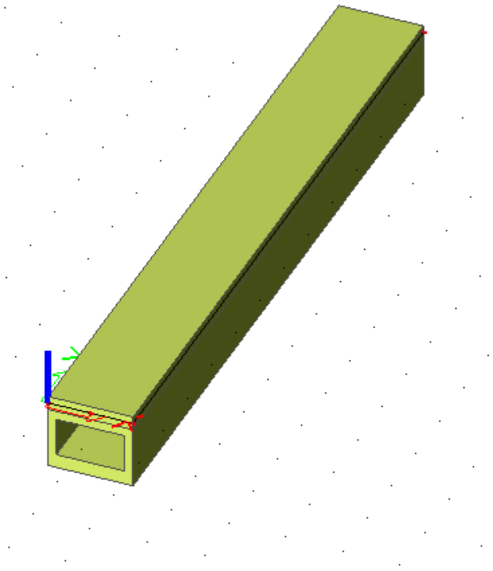
- The principle is as follows:
  The plate is divided into different subplates. The reason for this is if the plate will shorten, also a rib will disappear. By making different little plates, this can be solved.
- Making the parameters that determine the geometry properties of the plate and rib:
  - width profile b: 1480 mm
  - height profile h: 1100 mm
  - height upper flange h1: 250 mm
  - height lower flange h2: 160 mm
  - thickness web t: 140 mm
  - thickness plate d: =h1
  - length partial panel: =b
  - angle of fillet plate alpha: 30°
  - total length plate Ltot: 12 m

- These parameters are attributed to the channel



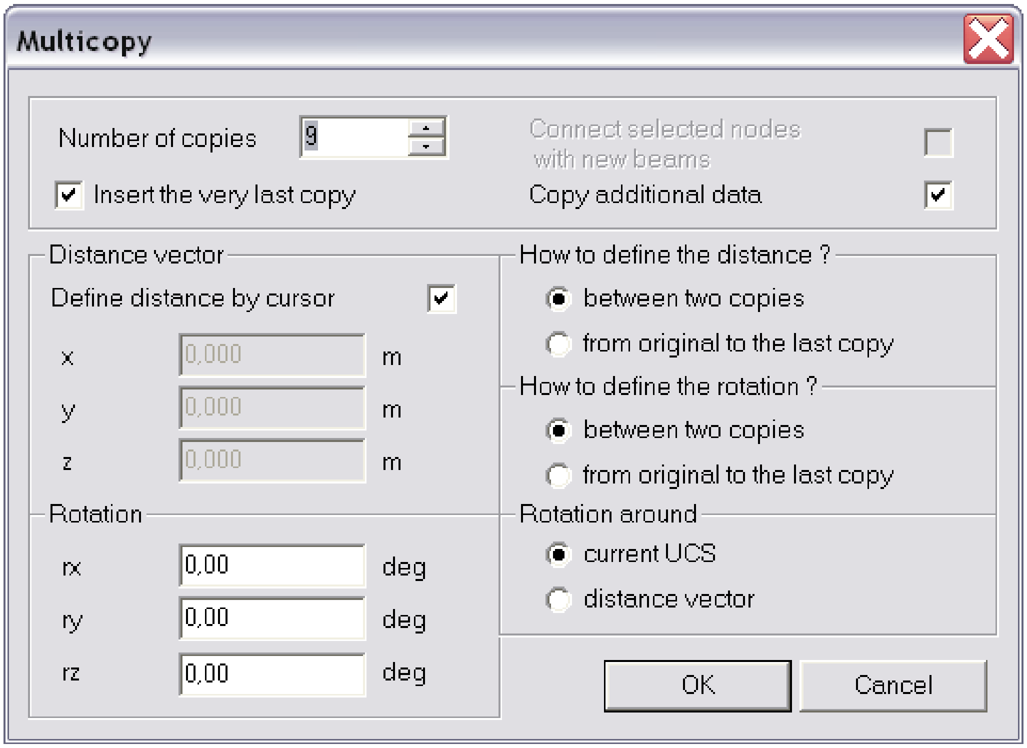
Name	CS2
Selector switch	□
Type	O asym
Detailed	1480; 140; 1100; 250; 160
⊟ **Parameters**	
Material	C12/15
A [mm]	b - Width section
tha [mm]	t - thickness web
B [mm]	h - heigth section
thb1 [mm]	h1 - heigth upper flange
thb2 [mm]	h2 - height lower flange
⊟ **General**	
Draw color	Normal colour
Colour	🟩
Properties editable	□
Buckling editable	☒
Buckling y-y	b
Buckling z-z	b
Fabrication	general
FEM analysis	□
Curve dividing	36
Use reduction factors	□
Edit named items	...
Edit joints	...
Edit cuts	...

- Input of 1 plate (as an element of the whole plate)
- Input of the plate rib with effective width b



- Use of dummy members, e.g. circular profile with diameter 5mm so the rib and member can be connected by a common node.
  Without these dummy beams, the ribs will not follow the plate after changing the dimensions of the plate.
- Connect nodes/edges to members!

- Input other plates through “more copies”:



Multicopy

Number of copies: 9

☑ Insert the very last copy

Connect selected nodes with new beams □

Copy additional data ☑

Distance vector

Define distance by cursor ☑

x	0,000	m
y	0,000	m
z	0,000	m

Rotation

rx	0,00	deg
ry	0,00	deg
rz	0,00	deg

How to define the distance ?

● between two copies

○ from original to the last copy

How to define the rotation ?

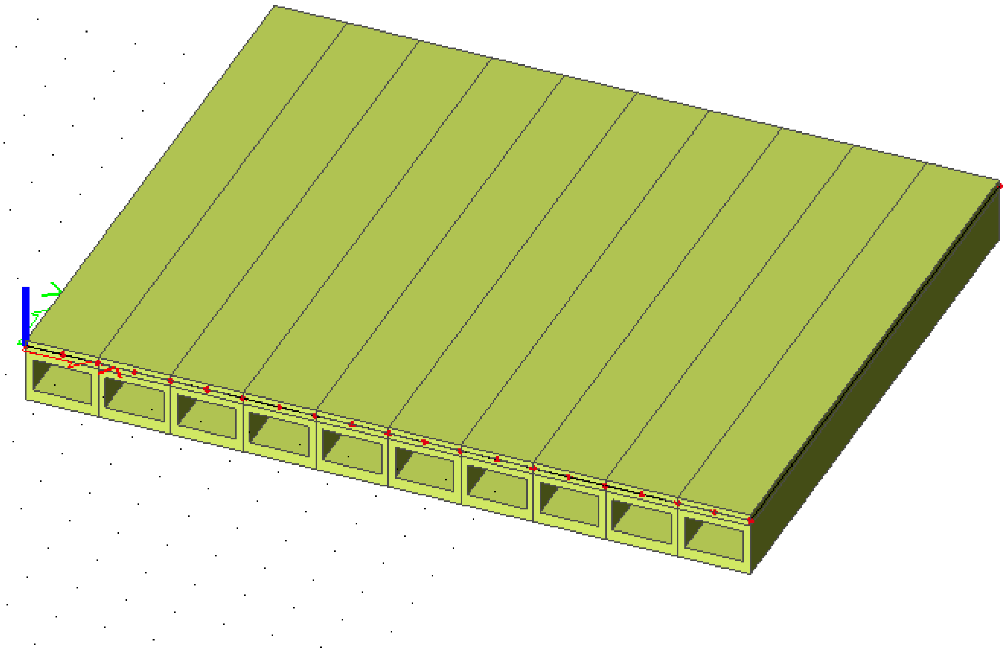
● between two copies

○ from original to the last copy

Rotation around

● current UCS

○ distance vector

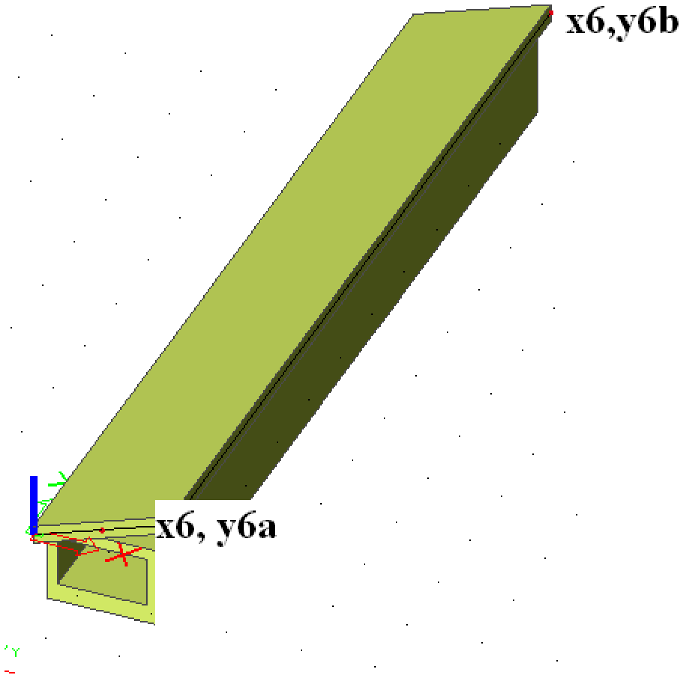


- An angle alpha of 30° is entered as parameter. With this, the x- and y-coordinates of the vertices of the partial plates can be entered:

Example:

Partial plate 6:

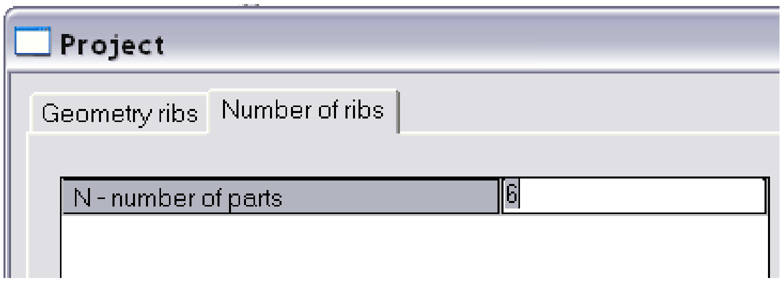
- x-coordinate x6: = b*6
- y-coordinate y6a: tg (alpha) * x6
- y-coordinate y6b: tg(alpha)*x6+ Ltot



This way, an inclined plate is obtained.

- Adjusting the number of partial plates/ribs:
  - Parameter for the number of parts: N (< = 10)
  - Parameter L1..10: They determine the activity of the various layers
    Vb. L6: boolean with formula N>=6
  - Parameter C1..10: they determine the type of model of the various layers
    Vb. C6: boolean with formula not(L6)

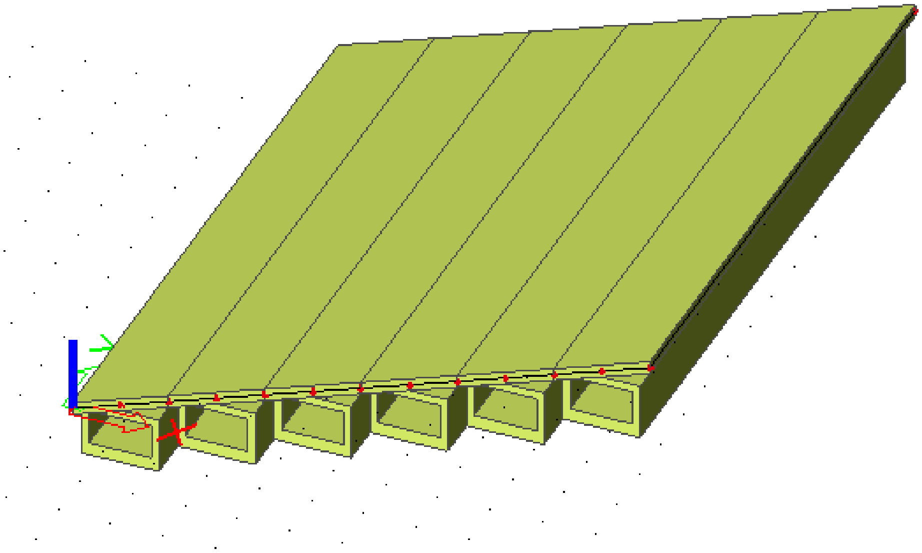
- Making the different parameter template and adjusting the number of parts to e.g. 6 through the template dialog:



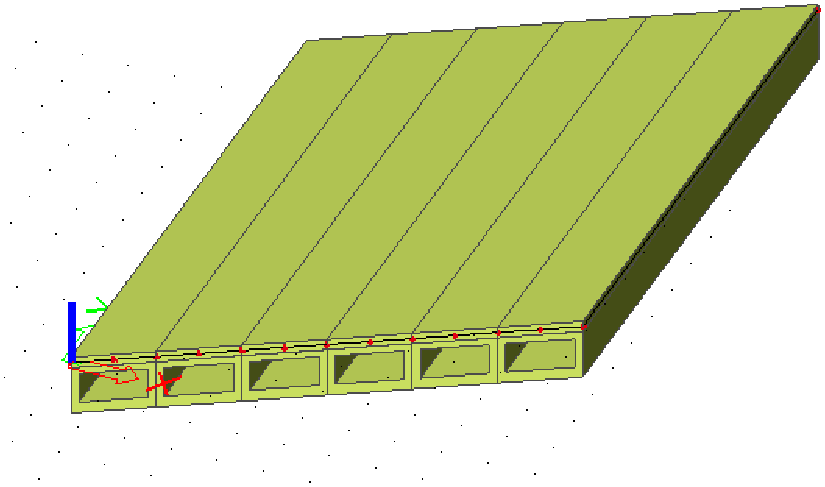
Project

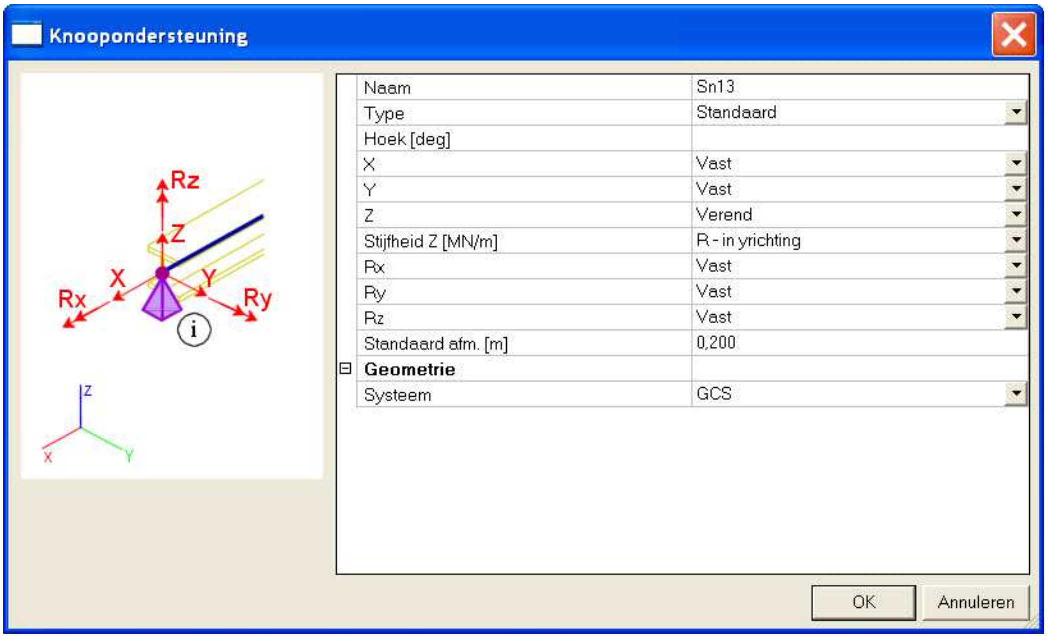
N - number of parts 6

→ Analysis model:

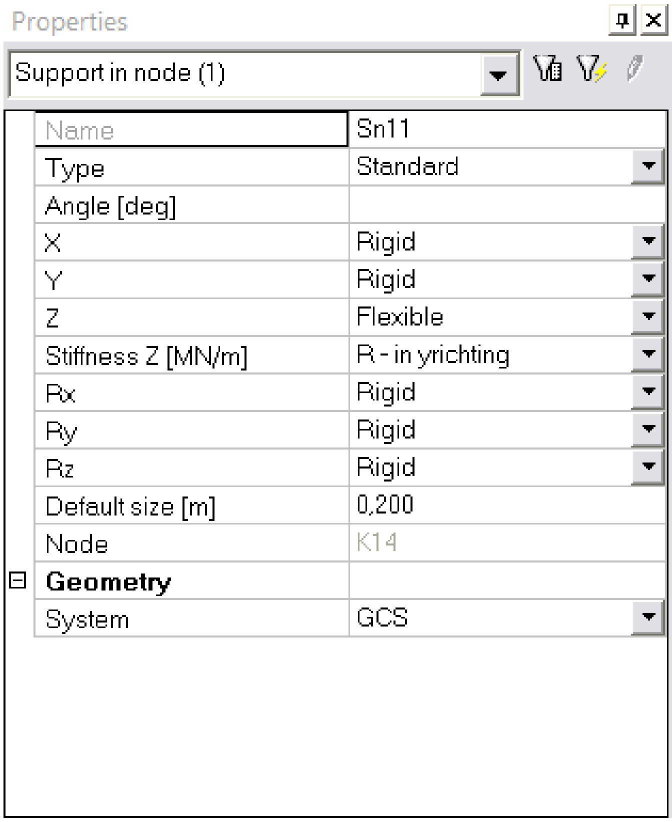


→ Structural model





Naam	Sn13
Type	Standaard
Hoek [deg]	
X	Vast
Y	Vast
Z	Verend
Stijfheid Z [MN/m]	R - in yrichting
Rx	Vast
Ry	Vast
Rz	Vast
Standaard afm. [m]	0,200
⊟ **Geometrie**	
Systeem	GCS

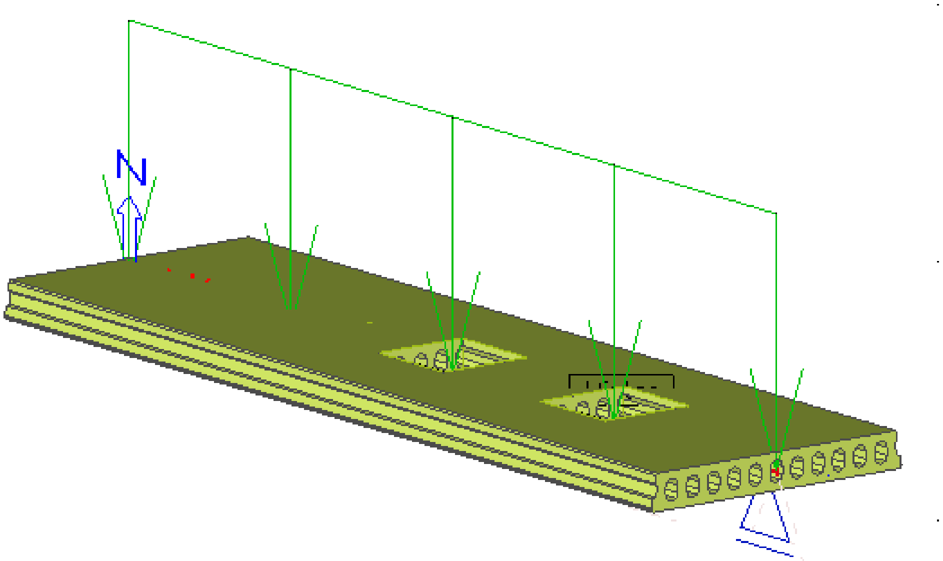


Name	Sn11
Type	Standard
Angle [deg]	
X	Rigid
Y	Rigid
Z	Flexible
Stiffness Z [MN/m]	R - in yrichting
Rx	Rigid
Ry	Rigid
Rz	Rigid
Default size [m]	0,200
Node	K14
⊟ **Geometry**	
System	GCS

Hollow core slab

A practical use of parameters is in the case of hollow core slabs.
In this example several functionalities are applied:

- prestress
- building stages
- TDA
- Parameters



- Project data:
- Construction type: Frame XZ
- Project level advanced

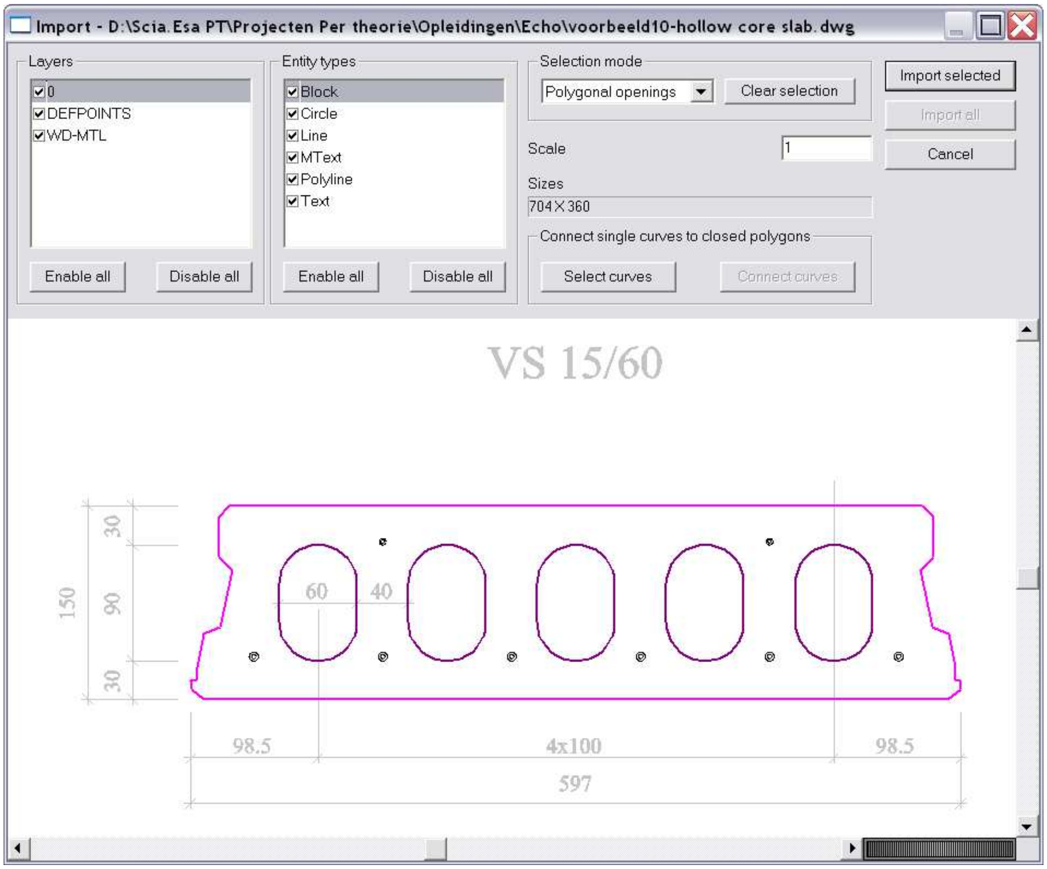
- Functionalities
  - Parameters
  - Concrete: fire resistance
  - Prestress:
    Advanced
    Hollow core slab
- Model: Construction stages
- Span: 6.2 m
- Material type: concrete C25/30

- The hollow core slab is entered by means of a dwg

Method

1. Entering a cross-section

The outline and the opening are selected alternately, after which they can be imported through 'import selection'.



Import - D:\Scia.Esa PT\Projecten Per theorie\Opleidingen\Echo\voorbeeld10-hollow core slab.dwg

Layers
☑ 0
☑ DEFPOINTS
☑ WD-MTL

Entity types
☑ Block
☑ Circle
☑ Line
☑ MText
☑ Polyline
☑ Text

Selection mode: Polygonal openings

Scale: 1

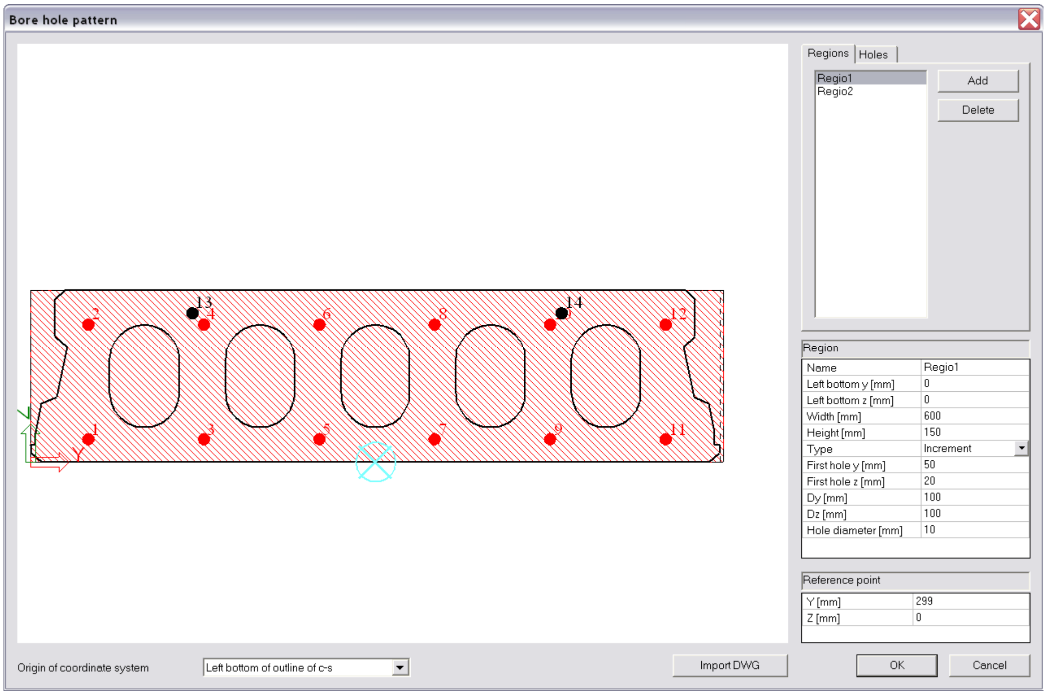
Sizes: 704 X 360

2. Inserting the prestressed tendons

Subsequently the prestressed tendons are entered:

Go to the concrete menu > Prestress – strand pattern:

2 regions are now made for the bore hole pattern.



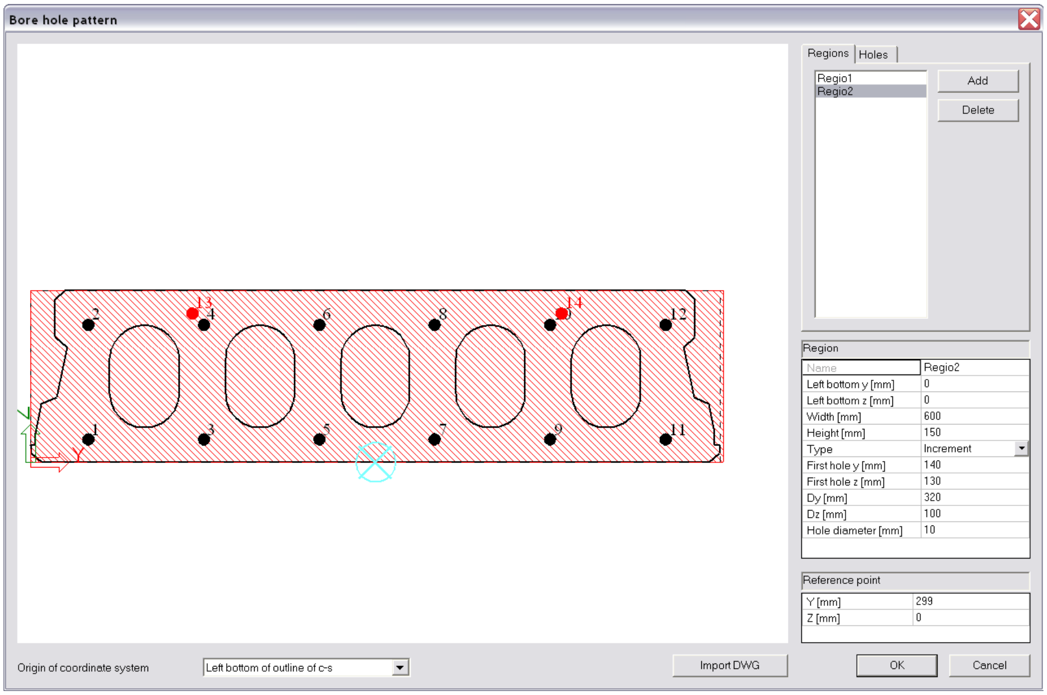
Region

Name	Regio1
Left bottom y [mm]	0
Left bottom z [mm]	0
Width [mm]	600
Height [mm]	150
Type	Increment
First hole y [mm]	50
First hole z [mm]	20
Dy [mm]	100
Dz [mm]	100
Hole diameter [mm]	10

Reference point

Y [mm]	299
Z [mm]	0

Origin of coordinate system: Left bottom of outline of c-s



Region

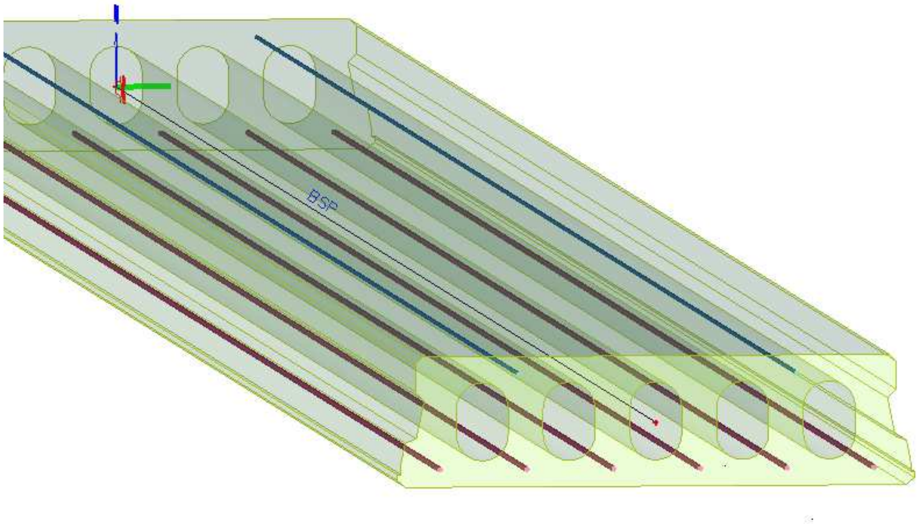
Name	Regio2
Left bottom y [mm]	0
Left bottom z [mm]	0
Width [mm]	600
Height [mm]	150
Type	Increment
First hole y [mm]	140
First hole z [mm]	130
Dy [mm]	320
Dz [mm]	100
Hole diameter [mm]	10

Reference point

Y [mm]	299
Z [mm]	0

Origin of coordinate system: Left bottom of outline of c-s

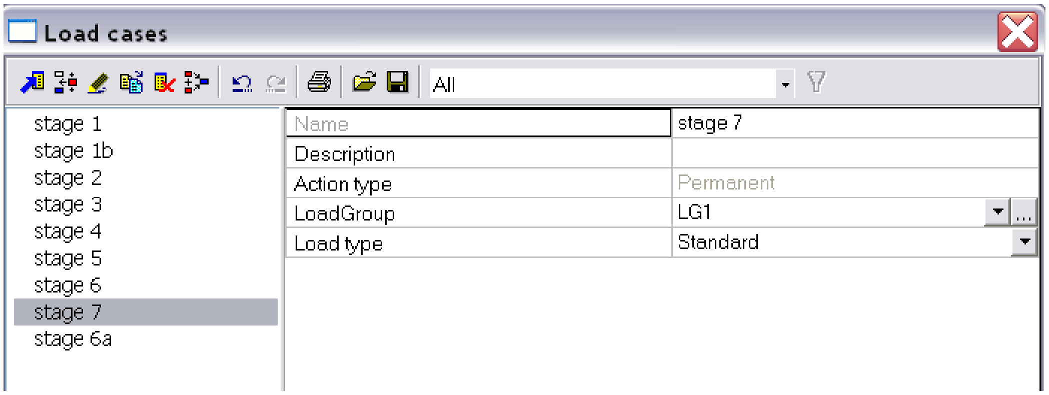
The initial stress of the prestressed tendons is 1250 Mpa.



3. Inserting the supports

- point load on node, at begin and end node of the member
- point load on member: on0,07 and 6,13 m
- point load on member: on 0,8 and 5,4 m

4. Making the load cases


Load cases

All

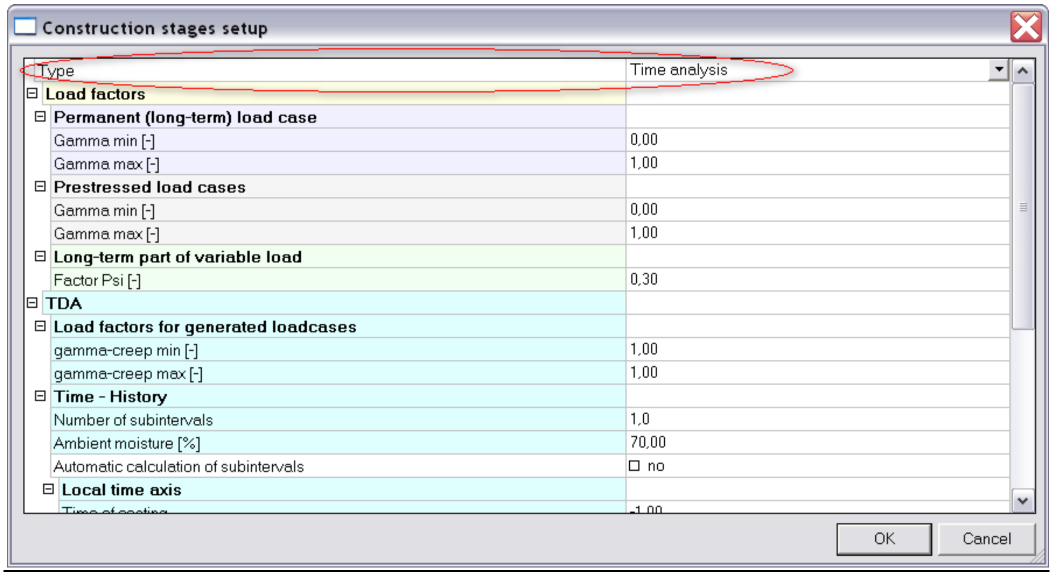
stage 1
stage 1b
stage 2
stage 3
stage 4
stage 5
stage 6
stage 7
stage 6a

Name	stage 7
Description	
Action type	Permanent
LoadGroup	LG1
Load type	Standard

- Selfweight member
- Prestress 1250 Mpa
- Stage 2: empty (permanent)
- Stage 3: empty (permanent)
- Stage 4: line load on member 0,6 kN/m (permanent)

- Stage 5: line load on member 1,2 kN/m (permanent)
- Stage 6: line load on member 1,2 kN/m (variable – long term)
- Stage 7: empty (permanent)

5. Inserting the construction stages



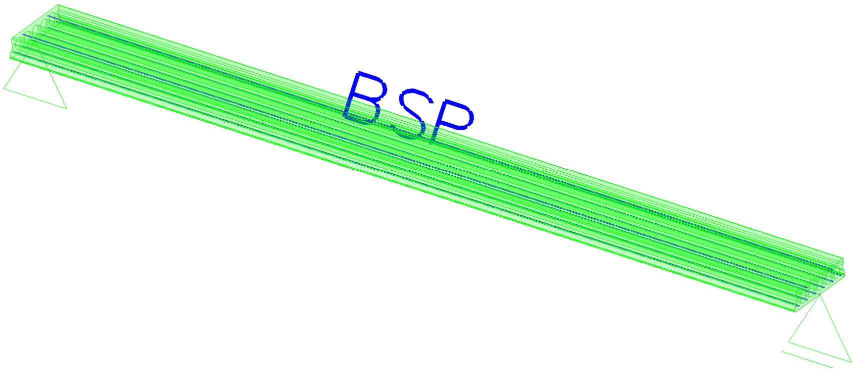
Construction stages setup

Type	Time analysis
⊟ **Load factors**	
⊟ **Permanent (long-term) load case**	
Gamma min [-]	0,00
Gamma max [-]	1,00
⊟ **Prestressed load cases**	
Gamma min [-]	0,00
Gamma max [-]	1,00
⊟ **Long-term part of variable load**	
Factor Psi [-]	0,30
⊟ **TDA**	
⊟ **Load factors for generated loadcases**	
gamma-creep min [-]	1,00
gamma-creep max [-]	1,00
⊟ **Time - History**	
Number of subintervals	1,0
Ambient moisture [%]	70,00
Automatic calculation of subintervals	□ no
⊟ **Local time axis**	
Time of casting	-1,00

Stage 1

Prestress and selfweight of the member

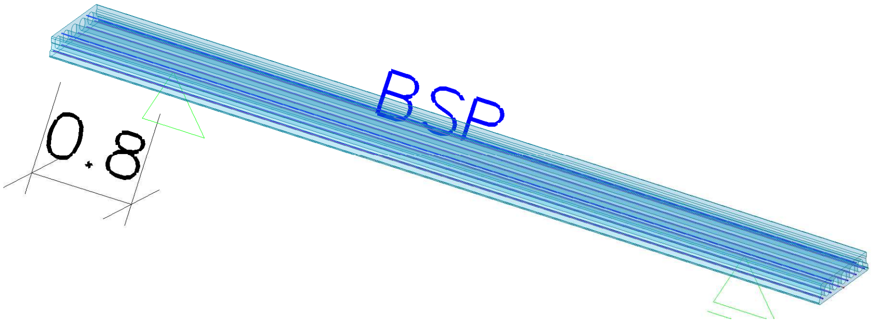
Load cases: presstress + selfweight
Time: day 1
Relative humidity: 100%



Stage 2

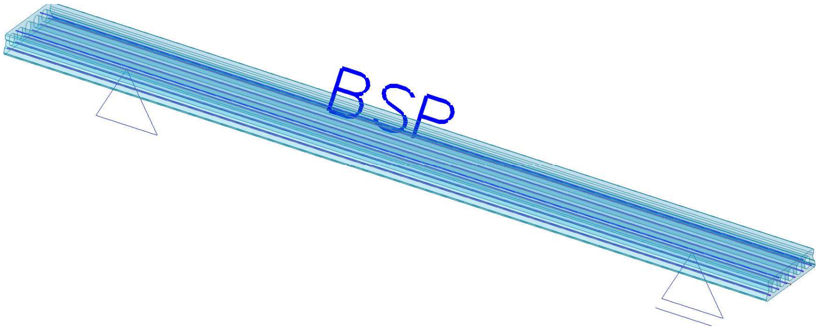
Transport

Load cases: empty
Time: day 1,1
Relative humidity: 100%



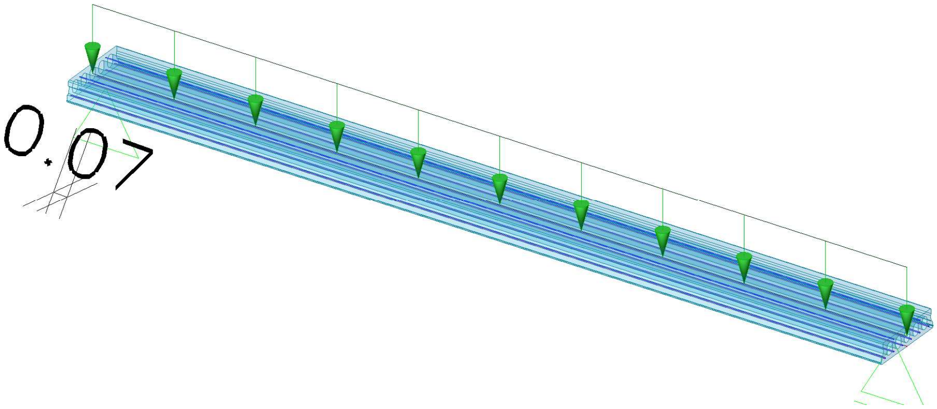
<u>Stage 3</u>
Storage

Load cases: empty 2
Time: day 1,2
Relative humidity: 70%



<u>Stage 4</u>
In situ, placing the plate

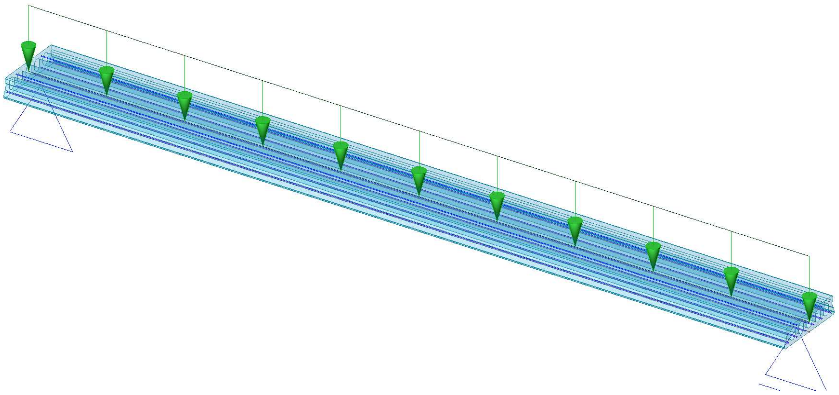
Load cases: q wall 0,6 kN/m
Time: day 28
Relative humidity: 70%

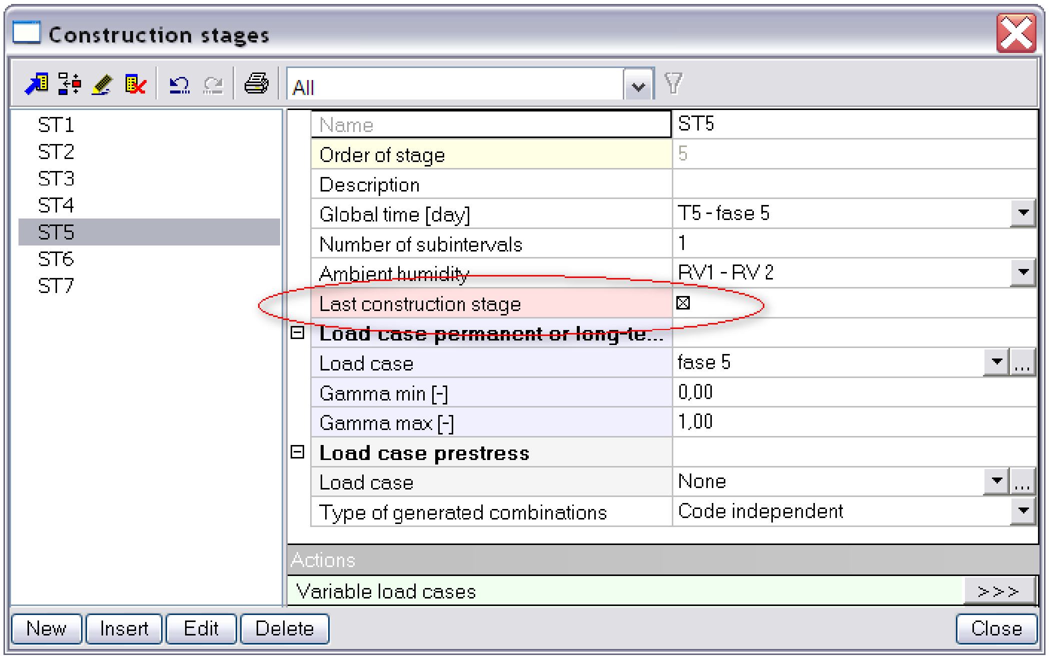


<u>Stage 5</u>

Casting of the coating

Load cases: 1.2 kN/m
Time: day 50
Relative humidity: 70%





ST1
ST2
ST3
ST4
ST5
ST6
ST7

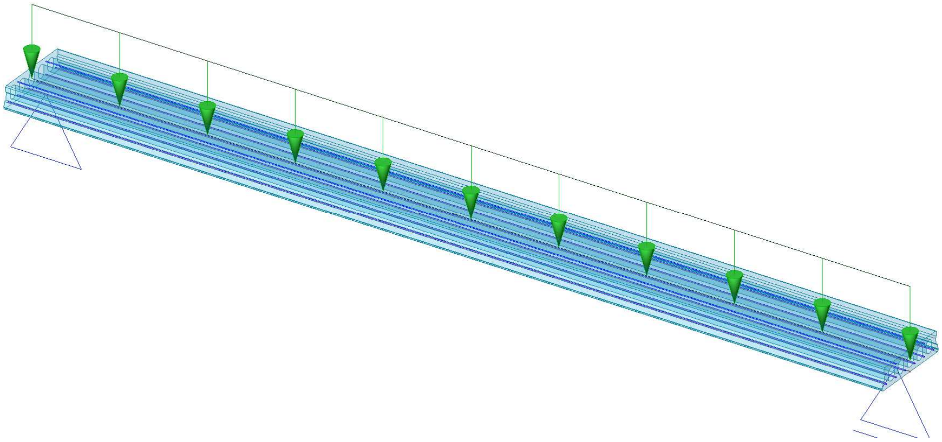
Name	ST5
Order of stage	5
Description	
Global time [day]	T5 - fase 5
Number of subintervals	1
Ambient humidity	RV1 - RV 2
Last construction stage	☒
⊟ **Load case permanent or long-te...**	
Load case	fase 5
Gamma min [-]	0,00
Gamma max [-]	1,00
⊟ **Load case prestress**	
Load case	None
Type of generated combinations	Code independent

Actions
Variable load cases >>>

<u>*Stage 6*</u>
Service

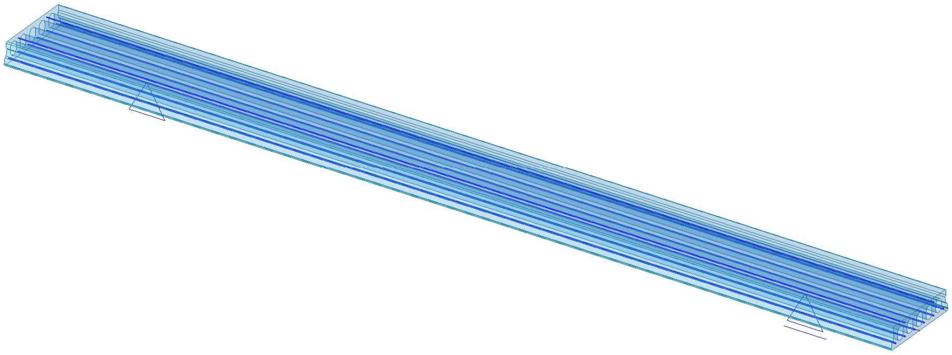
Load cases: long termVar 1.2 kN/m → will be used as long term load. Here it is important to pay attention to the duration of the load case, sc 'long'.

Time: day 100
Relative humidity: 70%



Stage 7
After 50 years

Loads
Load cases: empty3
Time: day 18000
Relative humidity: 70%



6. Inserting the parameters

- ***Length hollow core slab:***

Type: length

Evaluation: value: 5,25 m

- ***Height hollow core slab:***

Type: cross-section length

Evaluation: value: 150 mm

- ***Location support on member:***

Type: length

Evaluation: formula: resp. Length member – 0,07 and length member – 0,8

- ***Parameters general cross-section:***

Type: cross-section length

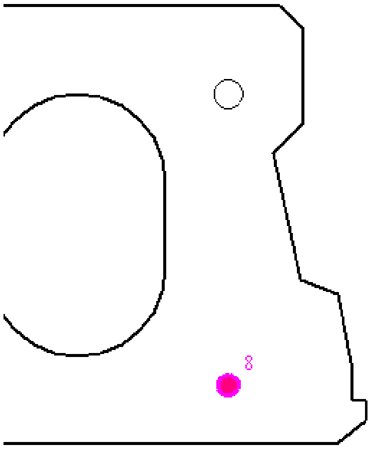
Evaluation: value

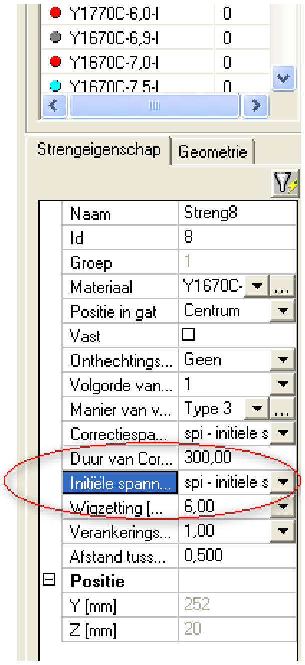
(examples: width to first opening, width opening, width between the openings,...)

- ***Initial stress:***

Type: stress

Evaluation: 1200 Mpa





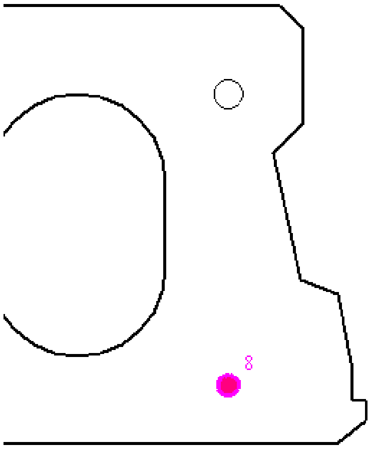
● Y1770C-6,0-I	0
● Y1670C-6,9-I	0
● Y1670C-7,0-I	0
● Y1670C-7,5-I	0

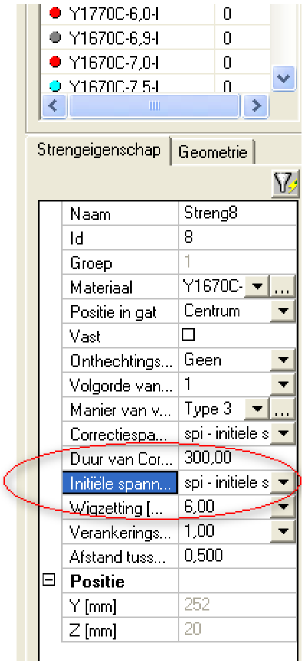
Naam	Streng8
Id	8
Groep	1
Materiaal	Y1670C-
Positie in gat	Centrum
Vast	□
Onthechtings...	Geen
Volgorde van...	1
Manier van v...	Type 3
Correctiespa...	spi - initiele s
Duur van Cor...	300,00
Initiële spann...	spi - initiele s
Wigzetting [...	6,00
Verankerings...	1,00
Afstand tuss...	0,500
Positie	
Y [mm]	252
Z [mm]	20

- ***Anchorage length:***

Type: length

Value: 1m





● Y1770C-6,0-I	0
● Y1670C-6,9-I	0
● Y1670C-7,0-I	0
● Y1670C-7,5-I	0

Strengeigenschap | Geometrie

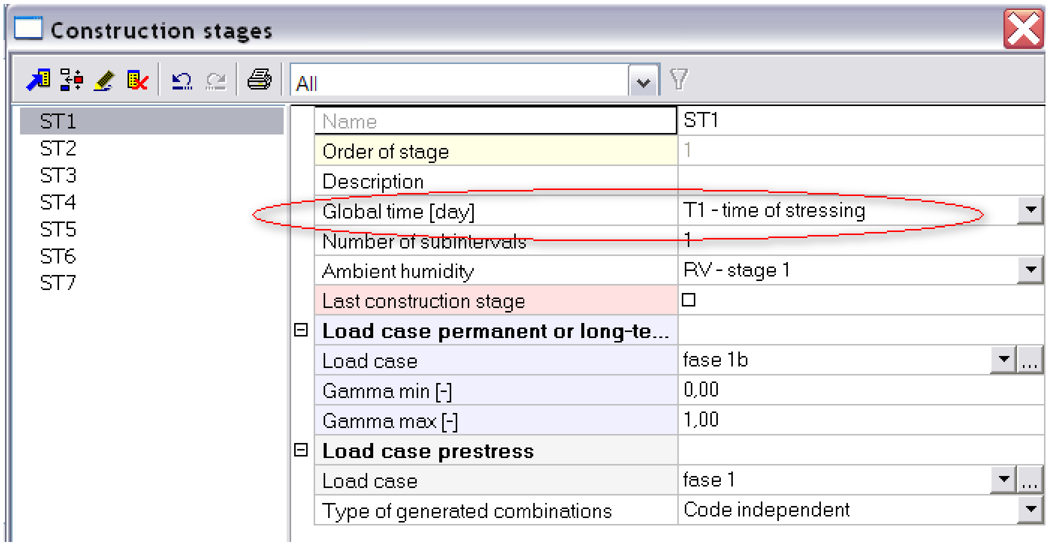
Naam	Streng8
Id	8
Groep	1
Materiaal	Y1670C-
Positie in gat	Centrum
Vast	□
Onthechtings...	Geen
Volgorde van...	1
Manier van v...	Type 3
Correctiespa...	spi - initiele s
Duur van Cor...	300,00
Initiële spann...	spi - initiele s
Wigzetting [...	6,00
Verankerings...	1,00
Afstand tuss...	0,500
Positie	
Y [mm]	252
Z [mm]	20

- ***Line loads***:

Type: line load
Value:
- wall: -0,5 kN/m
- casting coating: -1,75 kN/m
- long term var: -1,75 kN/m

- ***Time of prestressing***

Type: time (history)
Value: 1 day



Construction stages

All

ST1
ST2
ST3
ST4
ST5
ST6
ST7

Name	ST1
Order of stage	1
Description	
Global time [day]	T1 - time of stressing
Number of subintervals	1
Ambient humidity	RV - stage 1
Last construction stage	□
Load case permanent or long-te...	
Load case	fase 1b
Gamma min [-]	0,00
Gamma max [-]	1,00
Load case prestress	
Load case	fase 1
Type of generated combinations	Code independent

- ***T2 up to and including T7:***

Type: time (history)
Value: 1,1; 1,2; 28; 50; 100; 18000

- ***Relative humidity***

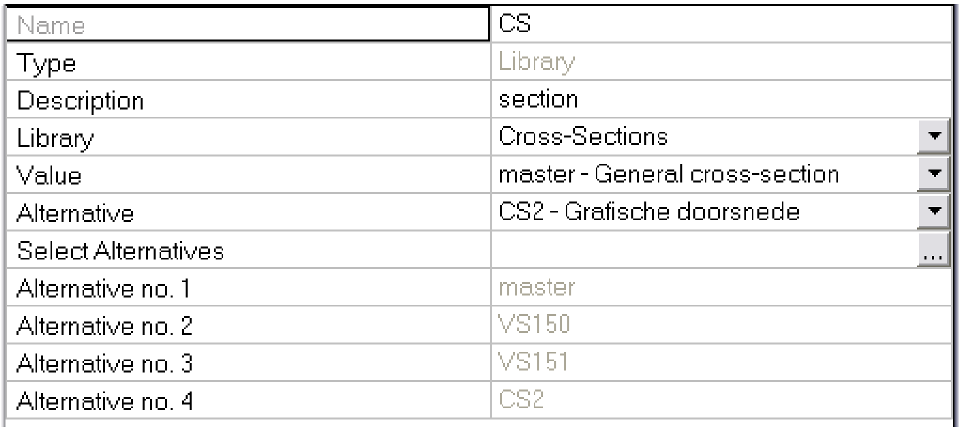
Type: relative humidity:
Value: 70 and 100%

- ***Cross-section type***

Type: library

Method: make a ‘master’ cross-section (copy of the original cross-section) and use the option ‘possible alternative’.

Subsequently it can be used with parameters as follows:

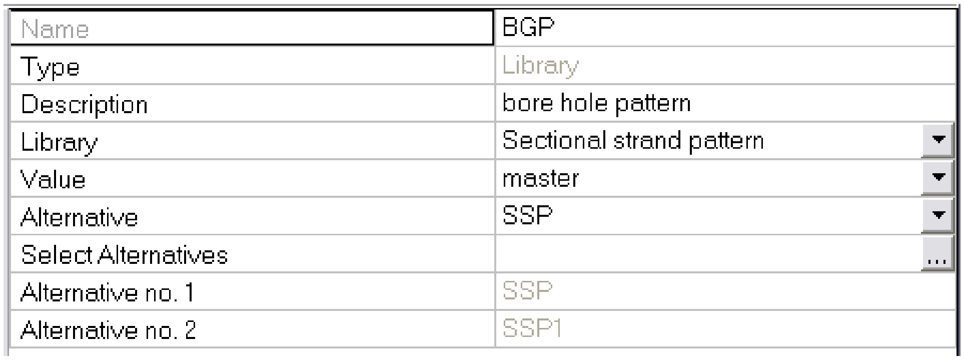


Name	CS
Type	Library
Description	section
Library	Cross-Sections
Value	master - General cross-section
Alternative	CS2 - Grafische doorsnede
Select Alternatives	...
Alternative no. 1	master
Alternative no. 2	VS150
Alternative no. 3	VS151
Alternative no. 4	CS2

- ***Strand pattern***

Type: library

Method: idem as the cross-section type. Make various strand patterns and copy a type as a master case.



Name	BGP
Type	Library
Description	bore hole pattern
Library	Sectional strand pattern
Value	master
Alternative	SSP
Select Alternatives	...
Alternative no. 1	SSP
Alternative no. 2	SSP1

Analogous possibilities for the type of library are: materials, stressing beds, bore hole patterns, …

Other possibilities to parameterize:
Moment of casting, Moment of releasing the casting

7. Making openings in de hollow core slab

Two openings are made, of which a number of values are parameterized, sc:

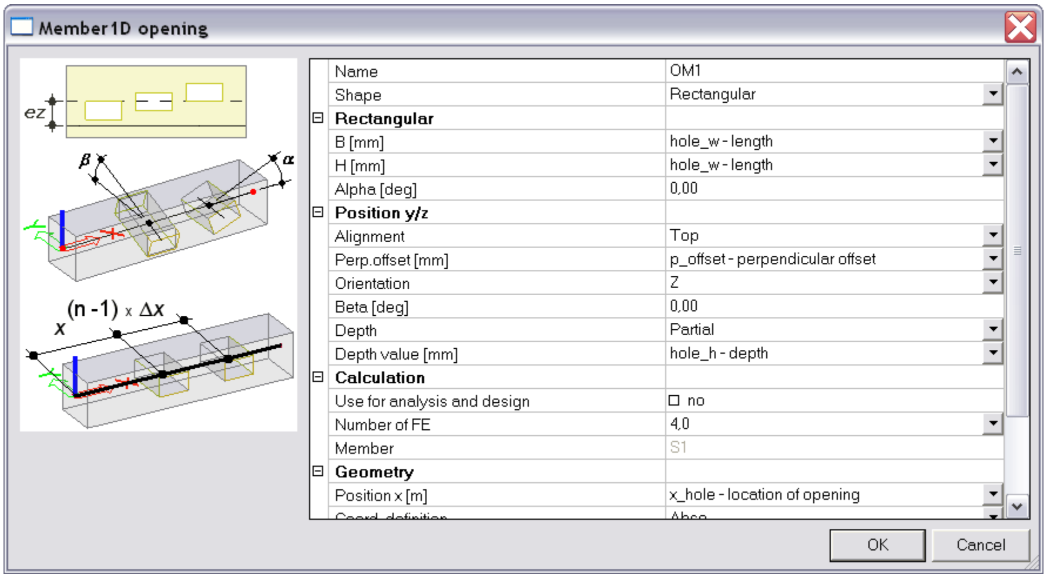
- ***delta x:***
Type: length
Value: 1m

- ***Width, length and depth opening:***
Type: cross-section length
Value: 500 and 150 mm

- ***x_offset:***
Type: cross-section length
Formula: -width/2

- ***x_hole:***
Type: length
Value: 2,1 m

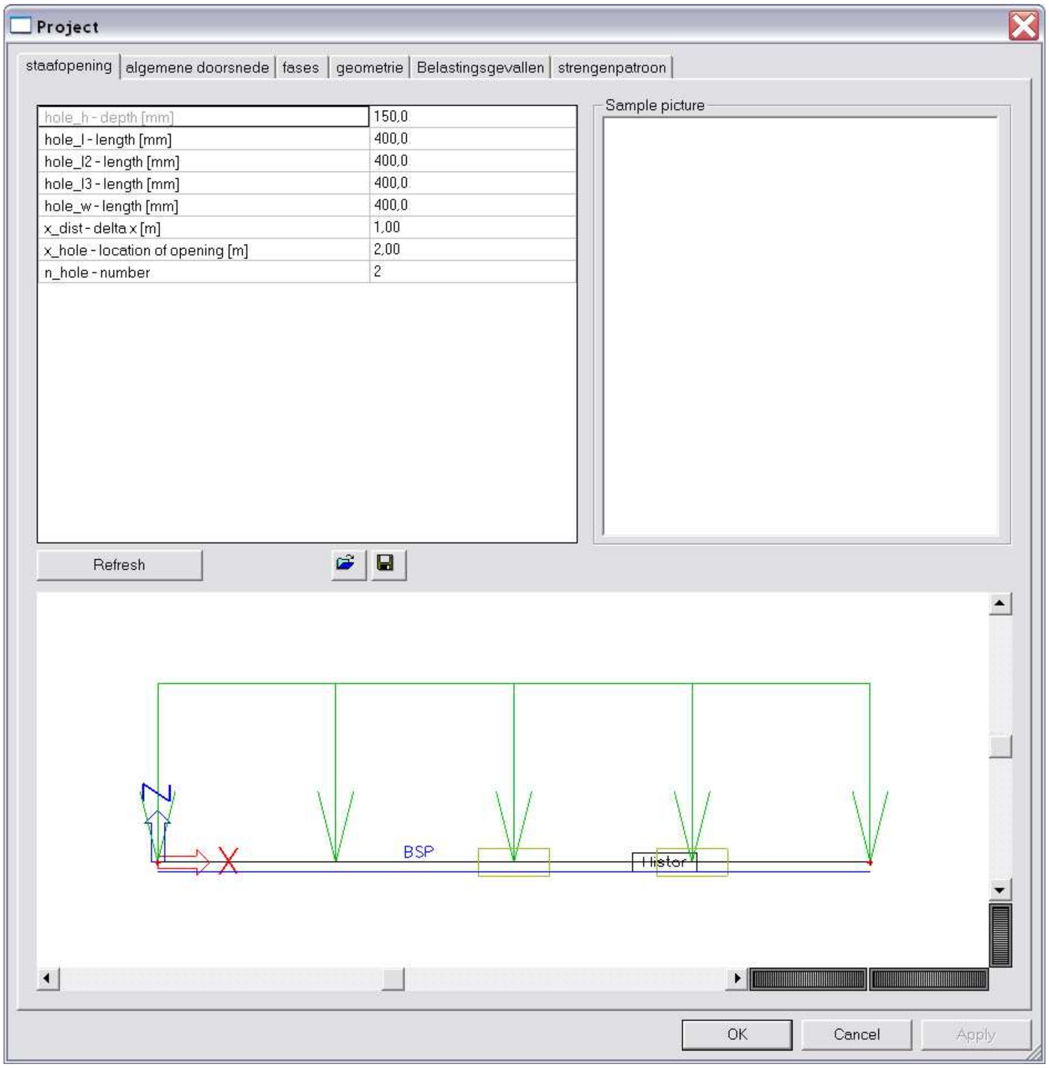
- ***n_hole:***
Type: integer
Value: 2



Name	OM1
Shape	Rectangular
⊟ **Rectangular**	
B [mm]	hole_w - length
H [mm]	hole_w - length
Alpha [deg]	0,00
⊟ **Position y/z**	
Alignment	Top
Perp.offset [mm]	p_offset - perpendicular offset
Orientation	Z
Beta [deg]	0,00
Depth	Partial
Depth value [mm]	hole_h - depth
⊟ **Calculation**	
Use for analysis and design	□ no
Number of FE	4,0
Member	S1
⊟ **Geometry**	
Position x [m]	x_hole - location of opening
Coord. definition	Abso

8. Using the template dialog

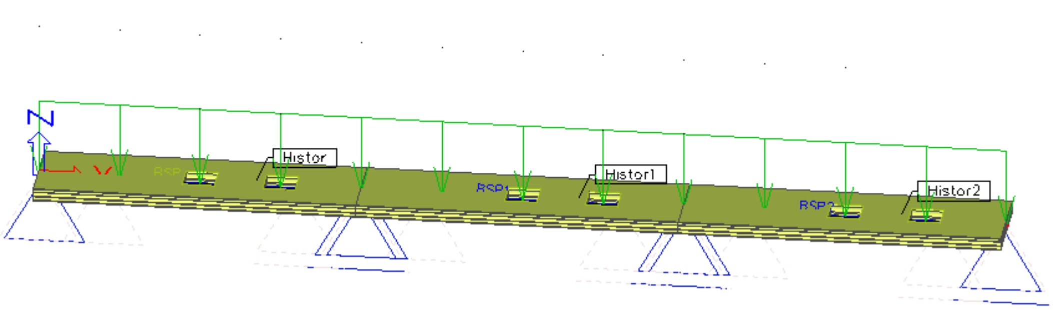
Subsequently these parameters can be divided and the template dialog can be used:



hole_h - depth [mm]	150,0
hole_l - length [mm]	400,0
hole_l2 - length [mm]	400,0
hole_l3 - length [mm]	400,0
hole_w - length [mm]	400,0
x_dist - delta x [m]	1,00
x_hole - location of opening [m]	2,00
n_hole - number	2

9. Different number of spans

Now the number of spans can be adjusted by means of copying the hollow core slab and to divide it into layers. For example: 3 elements.



Subsequently the following parameters can be used:

- N: number of spans

Type: integer
Value: 3

- Activities of the various layers:

Type: boolean
Evaluation: formula
A1: N>=1
A2: N>=2
A3: N>=3

- Construction type of the various layers:

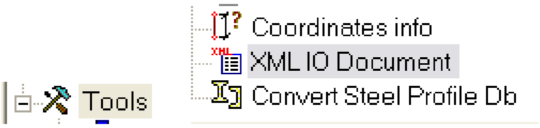
Type: boolean
Evaluation: formula
C1: not(A1)
C2: not(A2)
C3: not(A3)

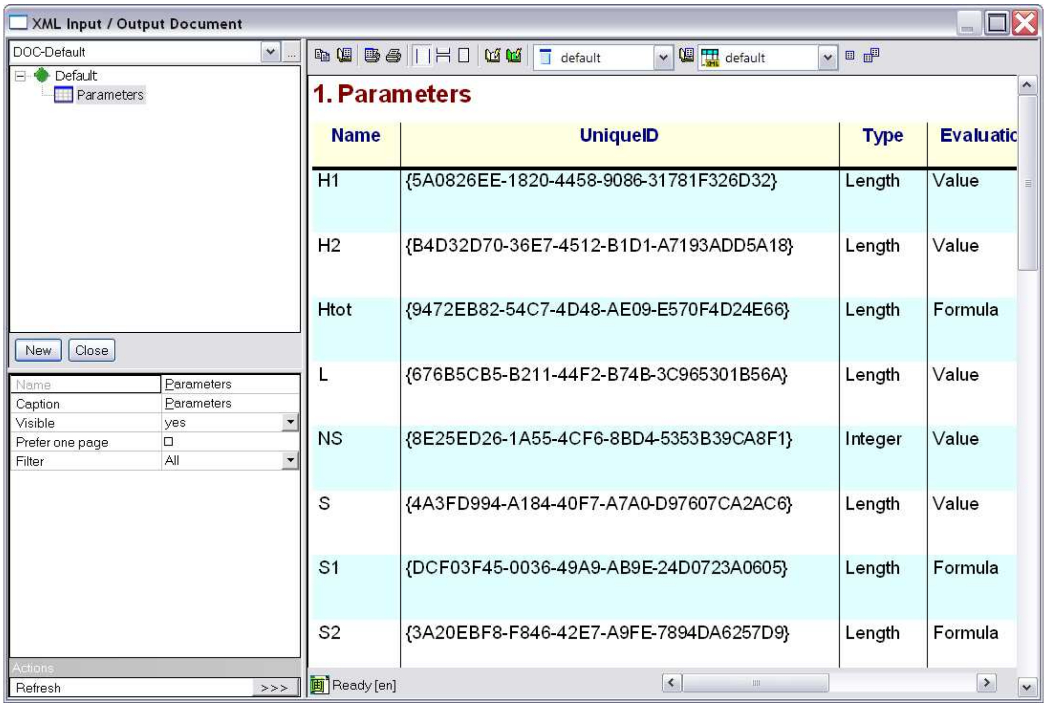
XML

SCIA Engineer allows for the exchange of data with third-party applications through the popular and powerful XML format. Moreover, XML format can be used to develop tailor-made applications that use SCIA Engineer as a "hidden" engine working on the background and performing calculations of company-specific problems.

The XML file editor is very similar to the Document of SCIA Engineer. The principle is that the user defines tables describing individual entities of the SCIA Engineer projects and there order. This table-form can be easily previewed (it is in fact identical to the standard SCIA Engineer document). When ready, the final XML file contents can be transformed into the real XML format through the export function.

- Take example 8 again: steel frame.esa

Through 

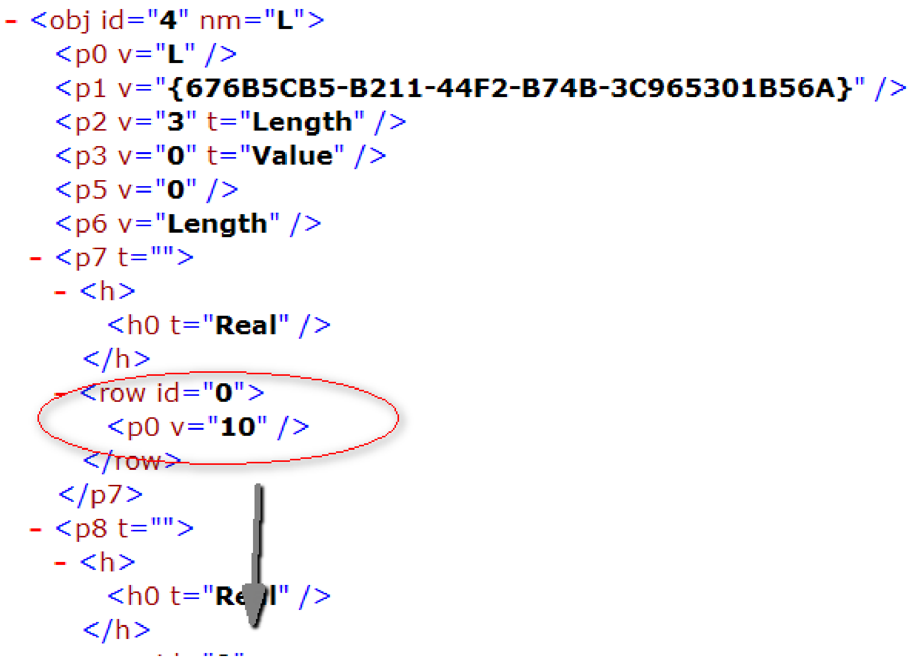


Name	Parameters
Caption	Parameters
Visible	yes
Prefer one page	□
Filter	All

1. Parameters

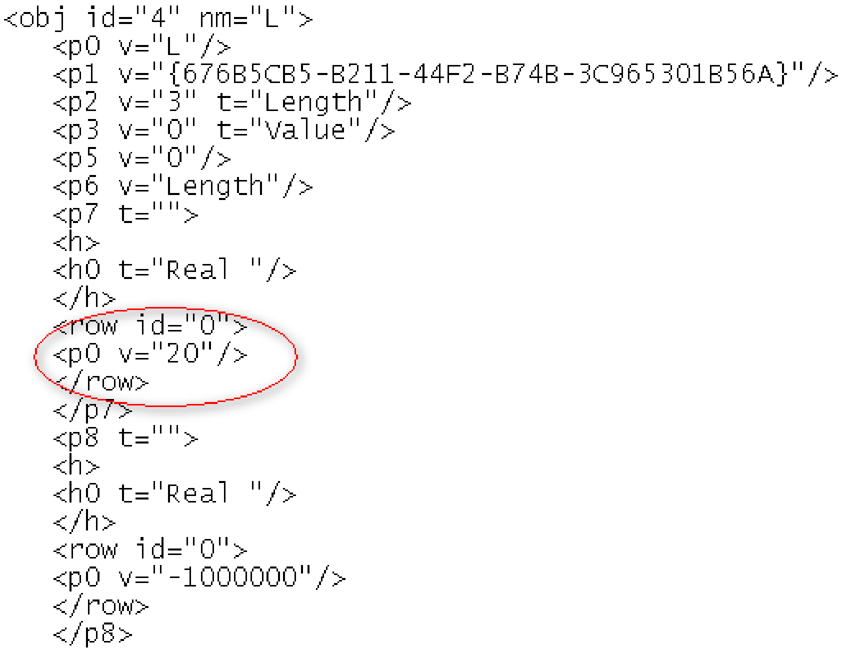
Name	UniqueID	Type	Evaluatic
H1	{5A0826EE-1820-4458-9086-31781F326D32}	Length	Value
H2	{B4D32D70-36E7-4512-B1D1-A7193ADD5A18}	Length	Value
Htot	{9472EB82-54C7-4D48-AE09-E570F4D24E66}	Length	Formula
L	{676B5CB5-B211-44F2-B74B-3C965301B56A}	Length	Value
NS	{8E25ED26-1A55-4CF6-8BD4-5353B39CA8F1}	Integer	Value
S	{4A3FD994-A184-40F7-A7A0-D97607CA2AC6}	Length	Value
S1	{DCF03F45-0036-49A9-AB9E-24D0723A0605}	Length	Formula
S2	{3A20EBF8-F846-42E7-A9FE-7894DA6257D9}	Length	Formula

- Export to XML:



```
- <obj id="4" nm="L">
    <p0 v="L" />
    <p1 v="{676B5CB5-B211-44F2-B74B-3C965301B56A}" />
    <p2 v="3" t="Length" />
    <p3 v="0" t="Value" />
    <p5 v="0" />
    <p6 v="Length" />
  - <p7 t="">
    - <h>
        <h0 t="Real" />
      </h>
    - <row id="0">
        <p0 v="10" />
      </row>
    </p7>
  - <p8 t="">
    - <h>
        <h0 t="Real" />
      </h>
```

- Adjusting the parameter L: 10 → 20
  Remark:
  For this purpose, it's necessary that the XML-document is opened in a text-file, for example: notepad.



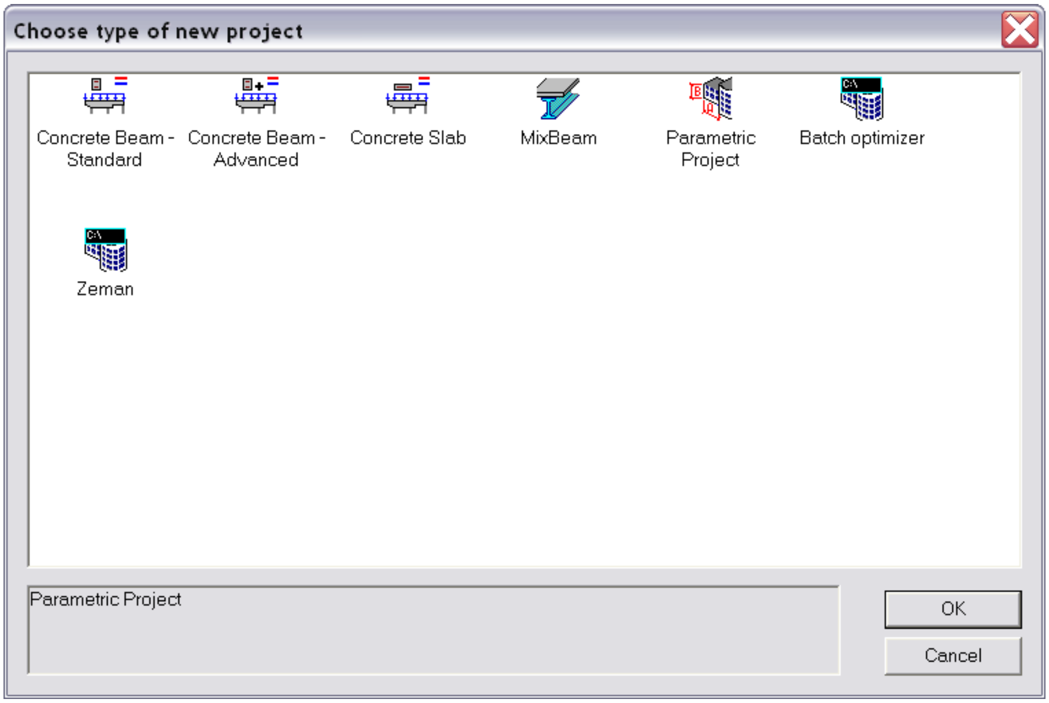
```
<obj id="4" nm="L">
    <p0 v="L"/>
    <p1 v="{676B5CB5-B211-44F2-B74B-3C965301B56A}"/>
    <p2 v="3" t="Length"/>
    <p3 v="0" t="Value"/>
    <p5 v="0"/>
    <p6 v="Length"/>
    <p7 t="">
    <h>
    <h0 t="Real "/>
    </h>
    <row id="0">
    <p0 v="20"/>
    </row>
    </p7>
    <p8 t="">
    <h>
    <h0 t="Real "/>
    </h>
    <row id="0">
    <p0 v="-1000000"/>
    </row>
    </p8>
```

- Regenerate the project through 'File > Update > XML file'. You will see that the structure has been changed now.

ODA

Oda is the abbreviation of ‘One dialog application’. It’s also called the SCIA engineer Template.

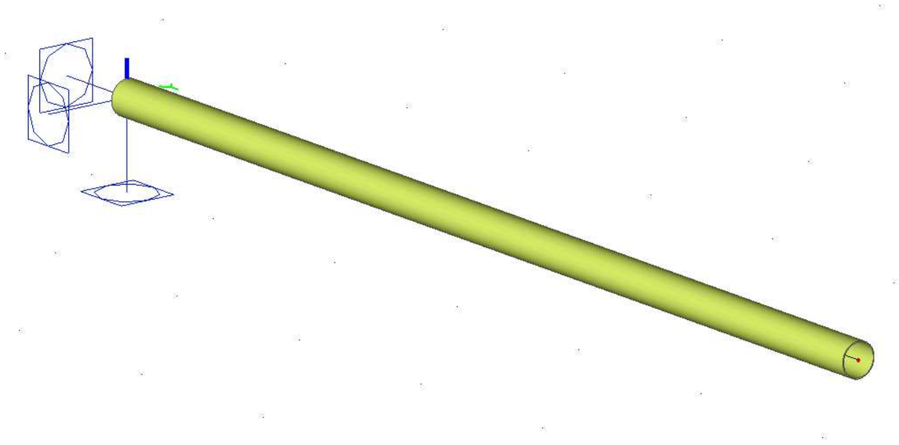
It can be used for following type of projects:



In this workshop the options ‘Parametric project’ and ‘Batch optimizer’ will be used.
The Batch optimizer will be explained in the next example.

The main principle of ODA is as follows:

- In SCIA Engineer a project is made, eventually with parameters and with document, and saved as a template.
- After this, SCIA Engineer has to be closed. It’s not possible to open the same project in ODA and in SCIA Engineer at the same time.
- Now, ODA can be opened and the user has to choose the type of project.
- In the most used case ‘parametric project’, the user can change the value of the parameters and the result is a document with all the added tables.
- This is shown in the following model.



- Project data:
  Construction type: General XYZ
  Project level advanced
  Material: concrete C20/25

- Functionalities – Parameters

- Load cases:
  - Selfweight
  - Variable line load: 1 kN/m

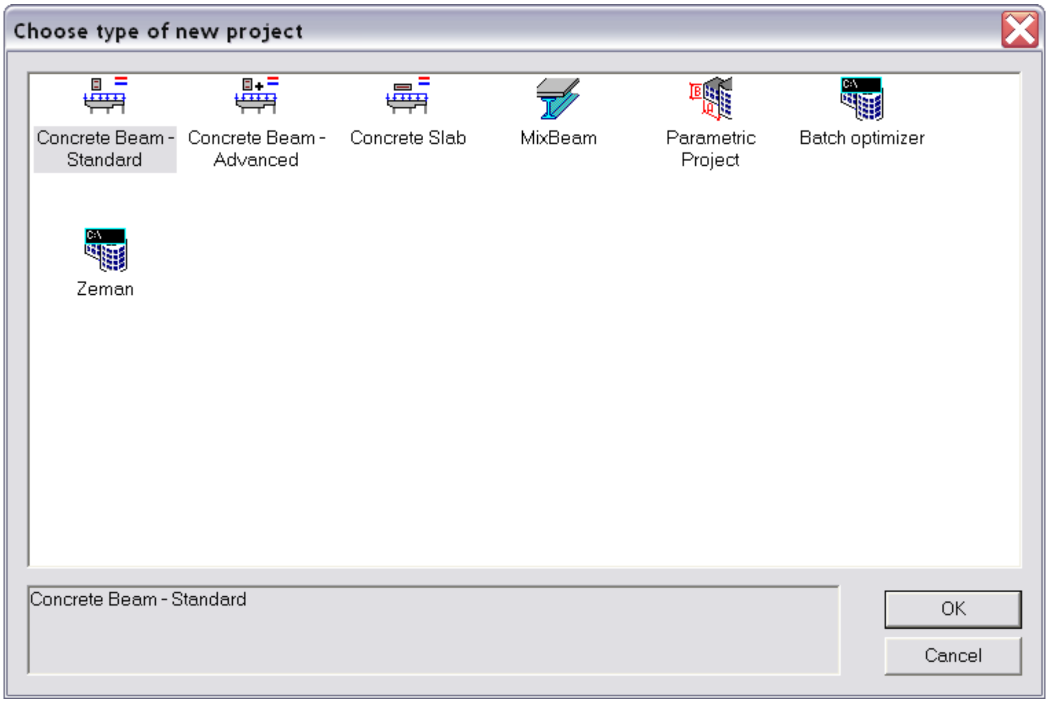
- Combinations: UGT and BGT
- Document input:
  - Nodes
  - Load cases
  - Combinations
  - Internal forces: My
  - Deformations: uz

- Parameters:
  - Diameter cross-section: 200 mm
  - Thickness cross-section: 5mm

Method

- Save this project, e.g. Oda.Esa, in the folder 'Templates' and close (if this is not performed, a conflict will arise when opening ODA).
- then open the program SCIA ODA:

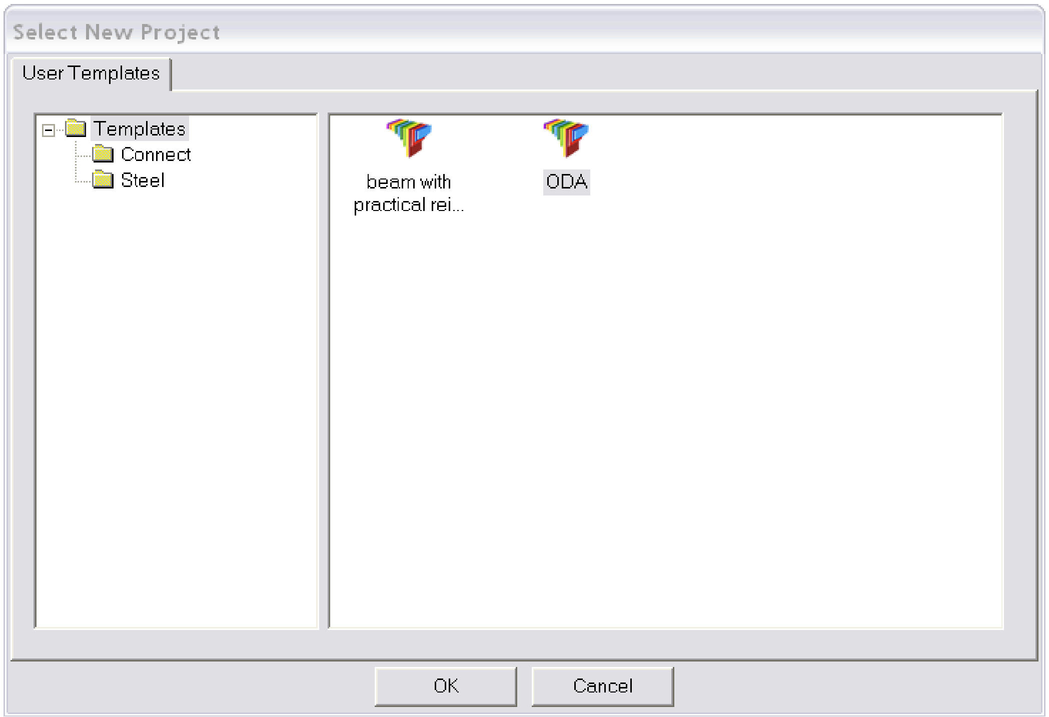
- The following dialog box appears:



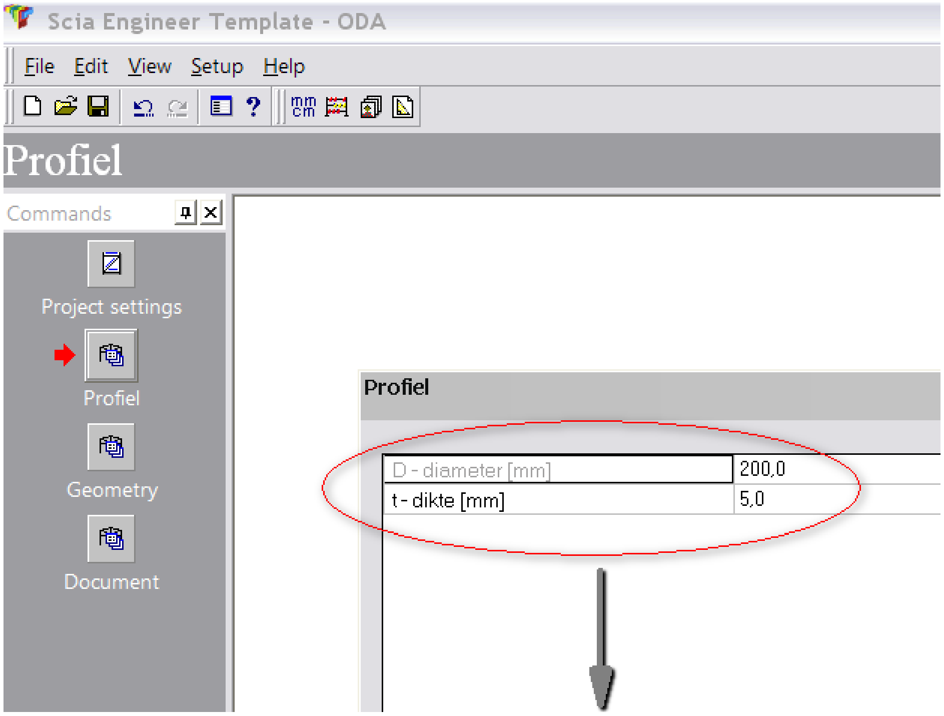
Concrete Beam - Standard

Choose the option ‘Parametric project’.

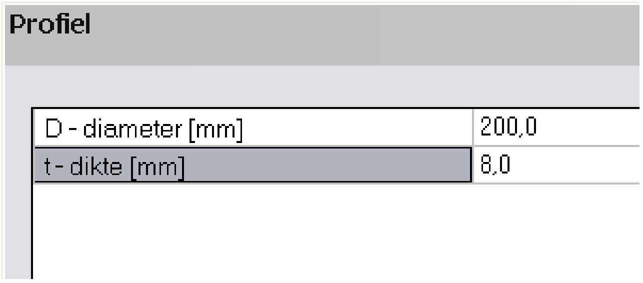
- The following dialogue box appears:



- Now the various steps can be re-run and possibly adjusted:

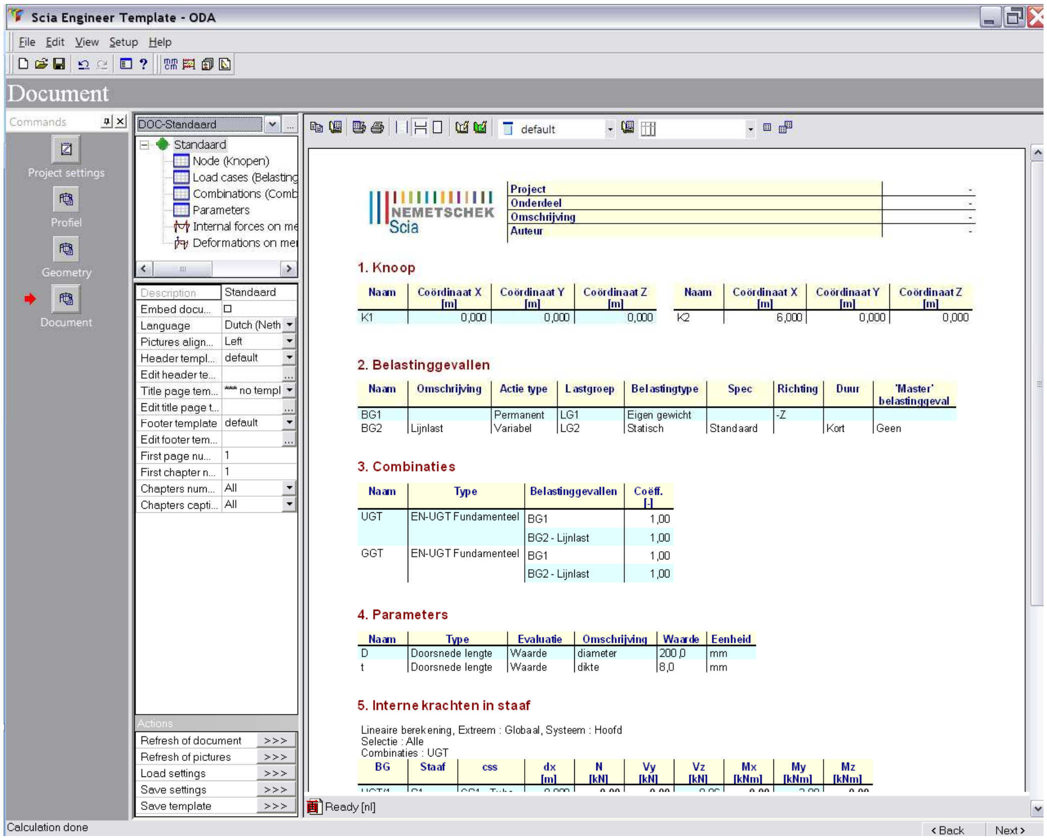


D - diameter [mm]	200,0
t - dikte [mm]	5,0



D - diameter [mm]	200,0
t - dikte [mm]	8,0

- Finally the following document is obtained in ODA after an automatic calculation:



Description	Standaard
Embed docu...	□
Language	Dutch (Neth
Pictures align...	Left
Header templ...	default
Edit header te...	...
Title page tem...	*** no templ
Edit title page t...	...
Footer template	default
Edit footer tem...	...
First page nu...	1
First chapter n...	1
Chapters num...	All
Chapters capti...	All

Project	-
Onderdeel	-
Omschrijving	-
Auteur	-

1. Knoop

Naam	Coördinaat X [m]	Coördinaat Y [m]	Coördinaat Z [m]	Naam	Coördinaat X [m]	Coördinaat Y [m]	Coördinaat Z [m]
K1	0,000	0,000	0,000	K2	6,000	0,000	0,000

2. Belastinggevallen

Naam	Omschrijving	Actie type	Lastgroep	Belastingtype	Spec	Richting	Duur	'Master' belastinggeval
BG1		Permanent	LG1	Eigen gewicht		-Z		
BG2	Lijnlast	Variabel	LG2	Statisch	Standaard		Kort	Geen

3. Combinaties

Naam	Type	Belastinggevallen	Coëff. [-]
UGT	EN-UGT Fundamenteel	BG1	1,00
		BG2 - Lijnlast	1,00
GGT	EN-UGT Fundamenteel	BG1	1,00
		BG2 - Lijnlast	1,00

4. Parameters

Naam	Type	Evaluatie	Omschrijving	Waarde	Eenheid
D	Doorsnede lengte	Waarde	diameter	200,0	mm
t	Doorsnede lengte	Waarde	dikte	8,0	mm

5. Interne krachten in staaf

Lineaire berekening, Extreem : Globaal, Systeem : Hoofd
Selectie : Alle
Combinaties : UGT

BG	Staaf	css	dx [m]	N [kN]	Vy [kN]	Vz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]

Note:

So, it's not possible to adapt the geometry on the graphical screen like we do in SCIA Engineer.
In ODA, only the inserted parameters can be adapted.
It's also not possible to add new tables in the document. ODA uses the document from the template.

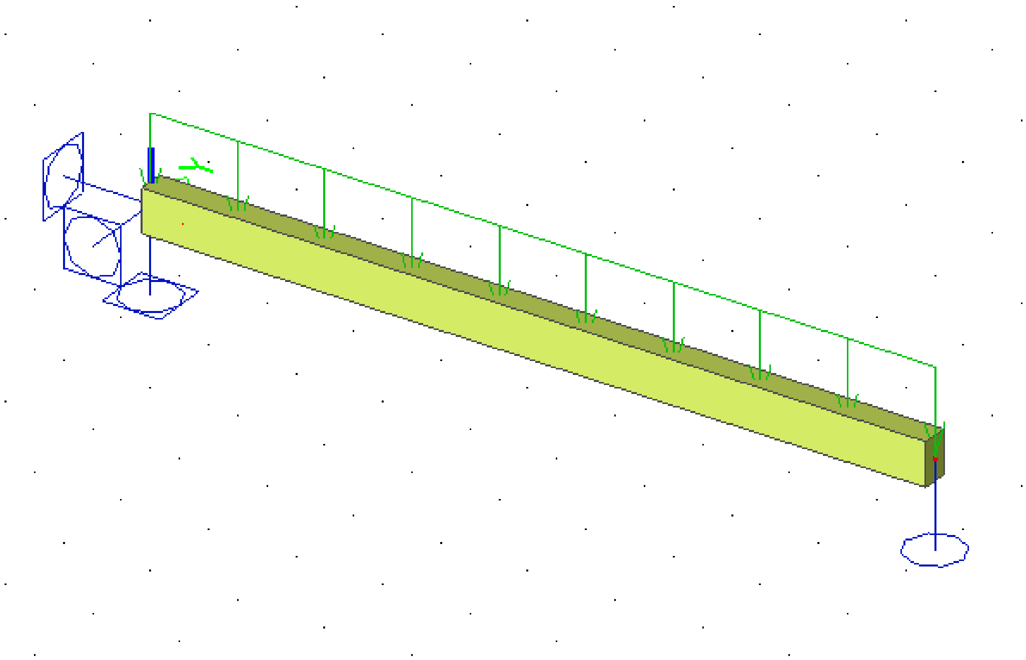
Batch optimizer

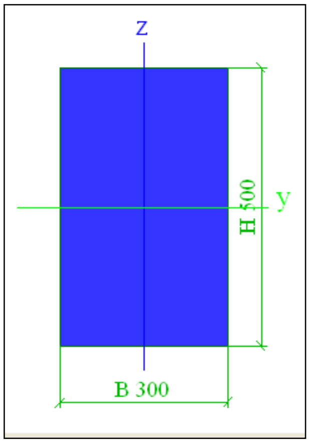
The batch optimizer is a special function of ODA. First of all a parametric project is made in SCIA Engineer.

Then an input and output file has to be made in XML. Subsequently they can be read in ODA.
A begin and end value of the parameter can be filled in and an iteration step to be performed. The results are the values of the desired output parameter.

Example:

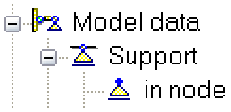
A beam on two supports is created in SCIA Engineer. The line load is parameterized. In ODA the moment My will be asked for a certain range of this parameter.





- Project data: - Construction type: Frame XYZ
  - Project level advanced

- Input support:

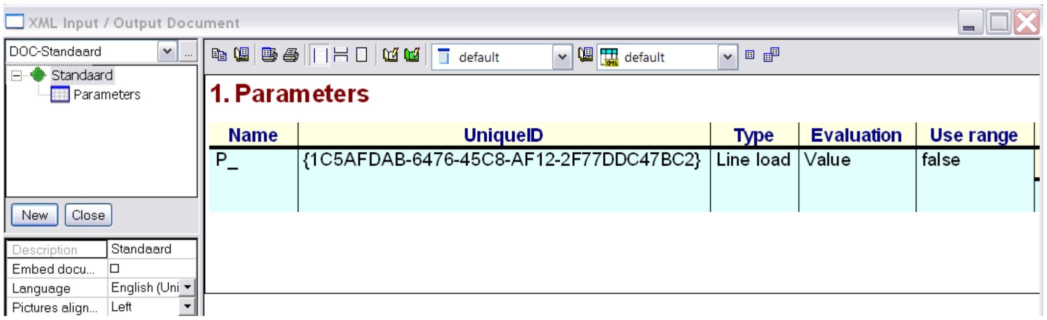


Method

- The line load is parameterized with a begin value of –5 kN/m

- Subsequently go to ‘Tools > XML IO document’ and export an input and output file:

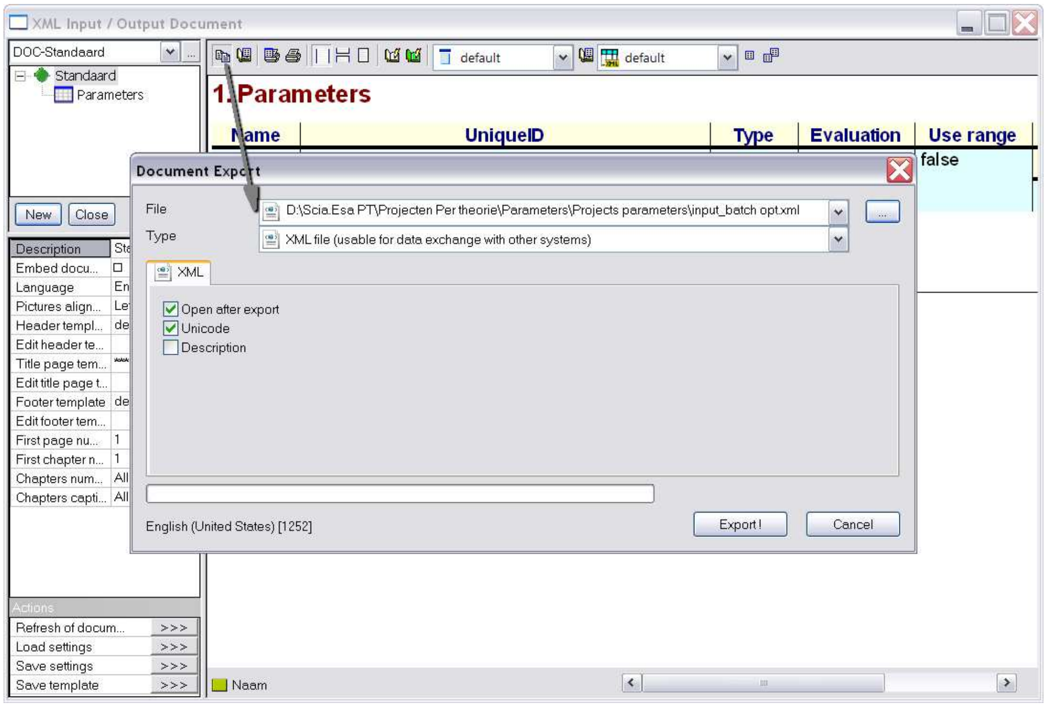
Input file:



1. Parameters

Name	UniqueID	Type	Evaluation	Use range
P_	{1C5AFDAB-6476-45C8-AF12-2F77DDC47BC2}	Line load	Value	false

Description	Standaard
Embed docu...	□
Language	English (Uni
Pictures align...	Left



Document Export

File D:\Scia.Esa PT\Projecten Per theorie\Parameters\Projects parameters\input_batch opt.xml

Type XML file (usable for data exchange with other systems)

XML

☑ Open after export

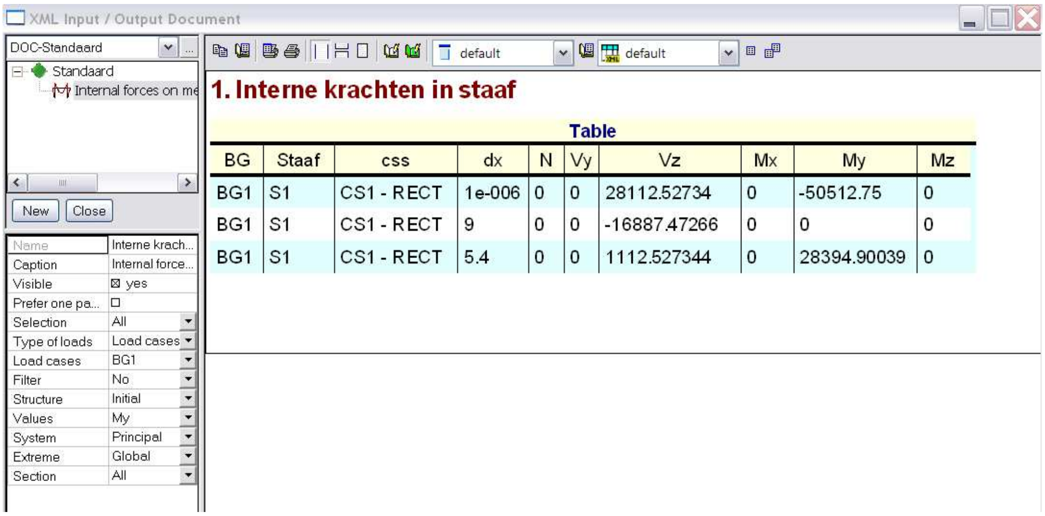
☑ Unicode

□ Description

English (United States) [1252]

Export! Cancel

Output file:

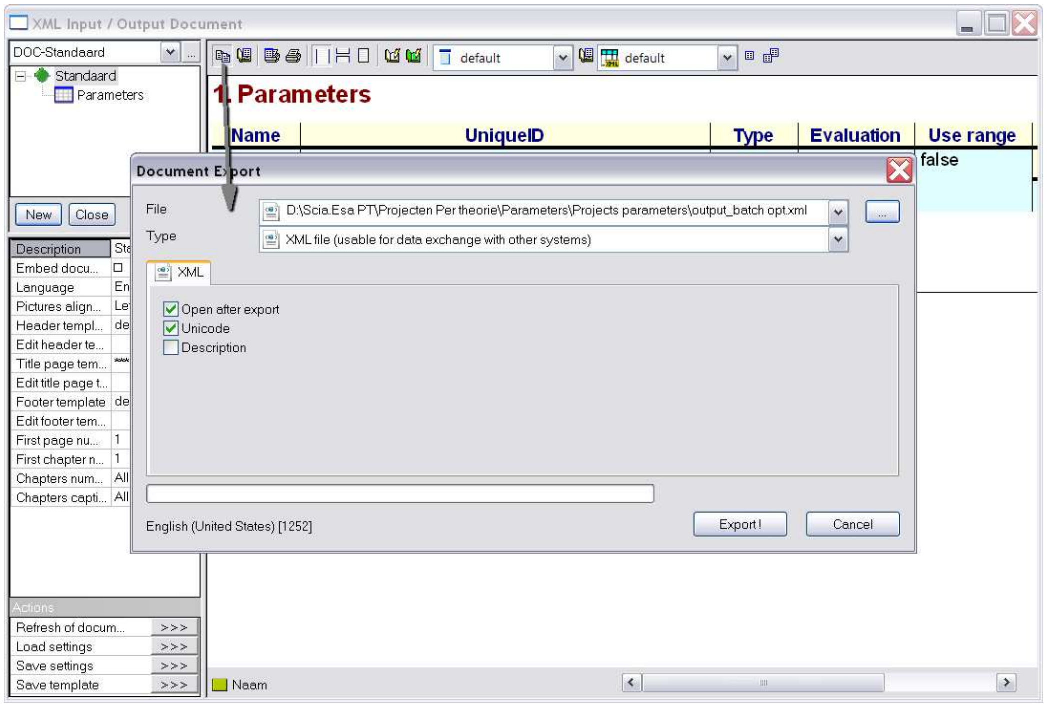


1. Interne krachten in staaf

Table

BG	Staaf	css	dx	N	Vy	Vz	Mx	My	Mz
BG1	S1	CS1 - RECT	1e-006	0	0	28112.52734	0	-50512.75	0
BG1	S1	CS1 - RECT	9	0	0	-16887.47266	0	0	0
BG1	S1	CS1 - RECT	5.4	0	0	1112.527344	0	28394.90039	0

Name	Interne krach...
Caption	Internal force...
Visible	☒ yes
Prefer one pa...	□
Selection	All
Type of loads	Load cases
Load cases	BG1
Filter	No
Structure	Initial
Values	My
System	Principal
Extreme	Global
Section	All



1. Parameters

Name	UniqueID	Type	Evaluation	Use range
				false

Description	Ste
Embed docu...	□
Language	En
Pictures align...	Le
Header templ...	de
Edit header te...	
Title page tem...	***
Edit title page t...	
Footer template	de
Edit footer tem...	
First page nu...	1
First chapter n...	1
Chapters num...	All
Chapters capti...	All

Refresh of docum...	>>>
Load settings	>>>
Save settings	>>>
Save template	>>>

Document Export

File: D:\Scia.Esa PT\Projecten Per theorie\Parameters\Projects parameters\output_batch opt.xml

Type: XML file (usable for data exchange with other systems)

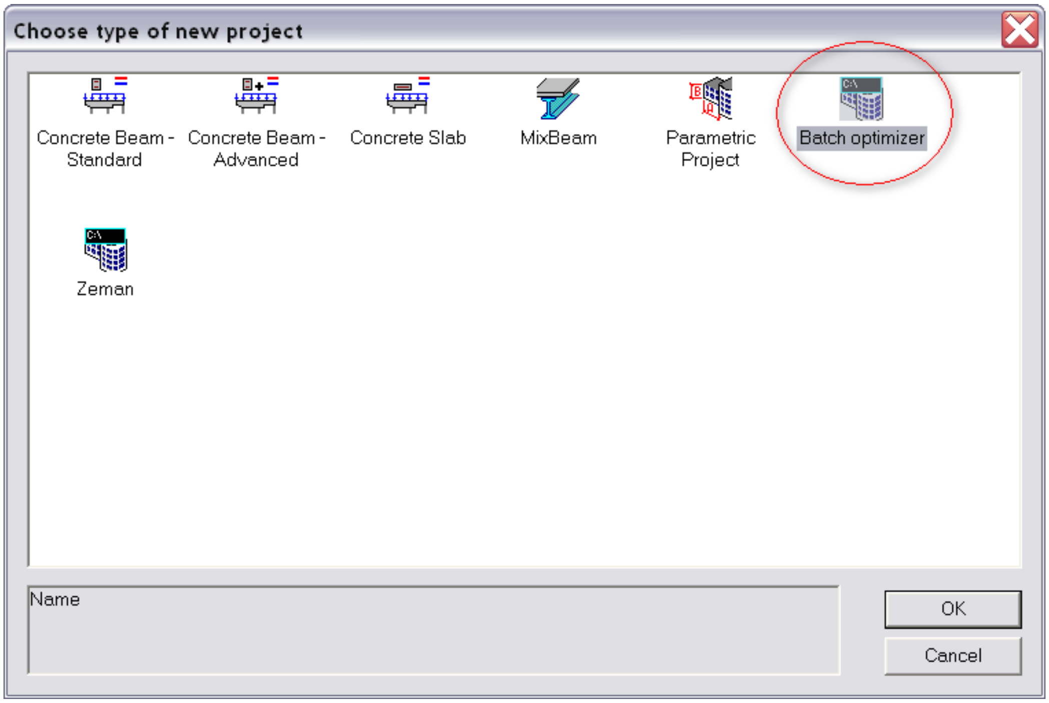
XML

☑ Open after export
☑ Unicode
□ Description

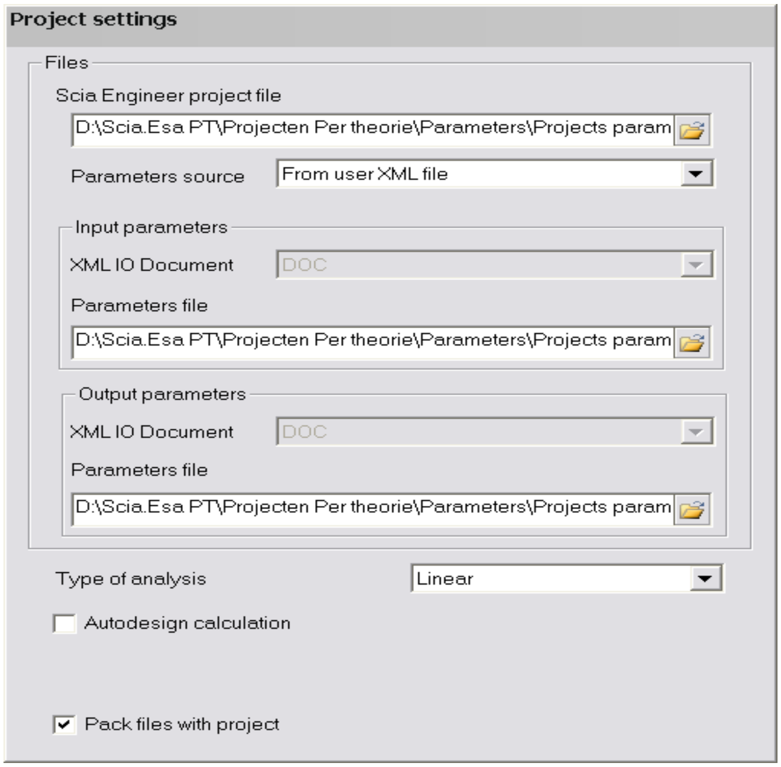
English (United States) [1252]

Export! Cancel

- This project can now be opened in ODA. For this, you choose the option ‘Batch optimizer’.



- Subsequently a dialog box appears in which the Esa project and both XML files have to be collected:


Project settings

Files

Scia Engineer project file: D:\Scia.Esa PT\Projecten Per theorie\Parameters\Projects param

Parameters source: From user XML file

Input parameters

XML IO Document: DOC

Parameters file: D:\Scia.Esa PT\Projecten Per theorie\Parameters\Projects param

Output parameters

XML IO Document: DOC

Parameters file: D:\Scia.Esa PT\Projecten Per theorie\Parameters\Projects param

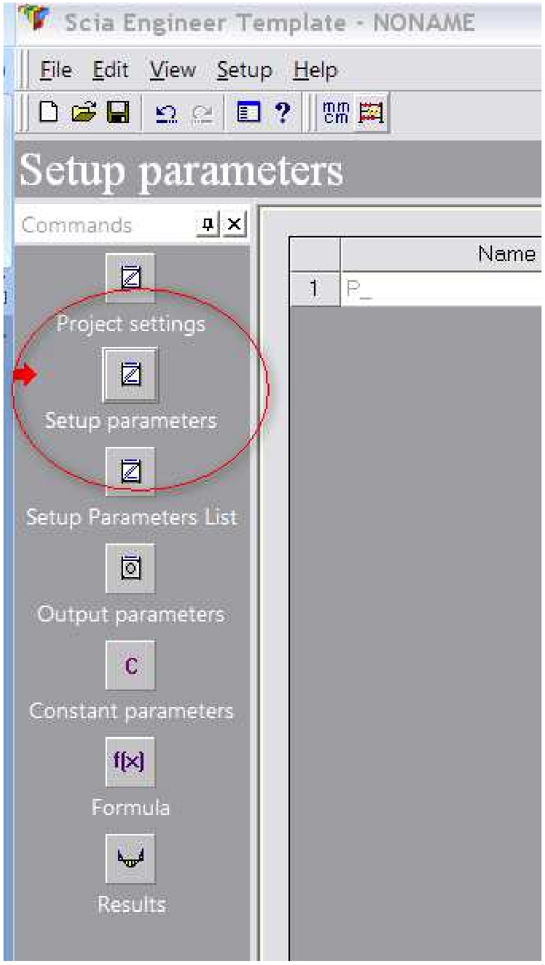
Type of analysis: Linear

☐ Autodesign calculation

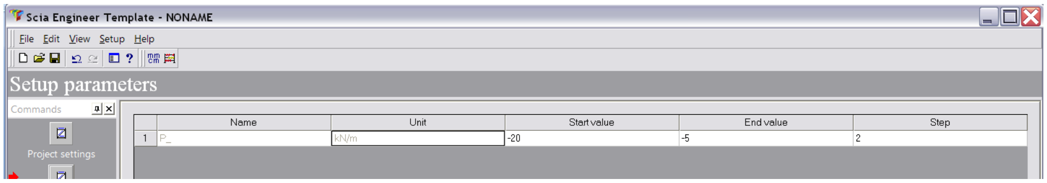
☑ Pack files with project

- If an optimisation was used in the Esa project, it can be taken into account here by selecting the option ‘optimisation calculation’.

You can ask for the results for a number of values of the parameter. Here the parameter ‘line load on member’ can be used. A begin value of -5 and an end value of -10 can be taken with an iteration step 1.

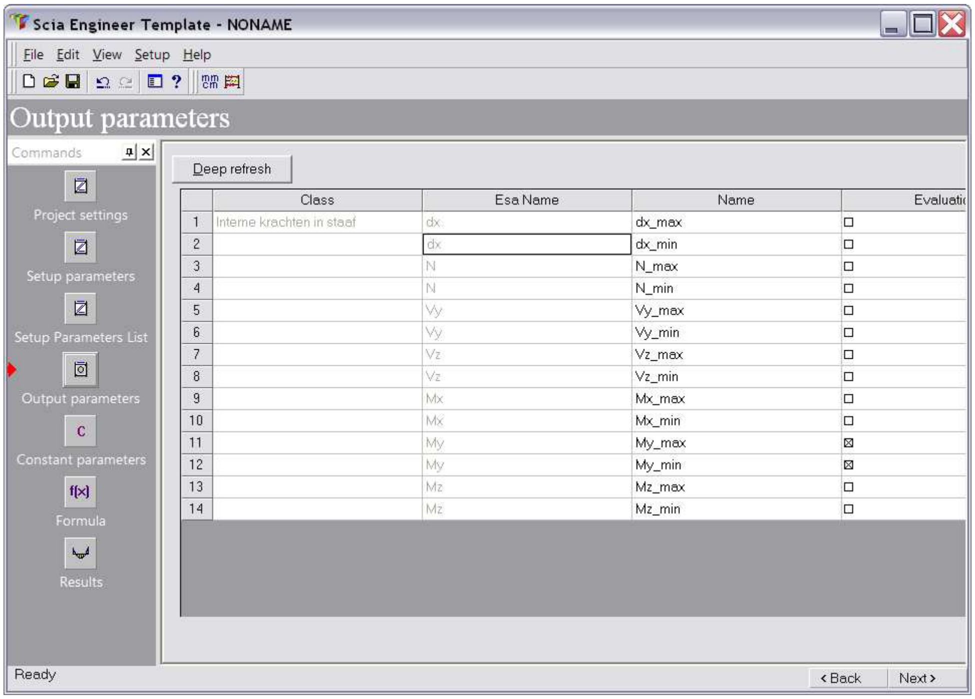


	Name
1	P_



	Name	Unit	Start value	End value	Step
1	P_	kN/m	-20	-5	2

The values of the output XML file that have to be evaluated, can be marked. The values for My_max and My_min are checked.

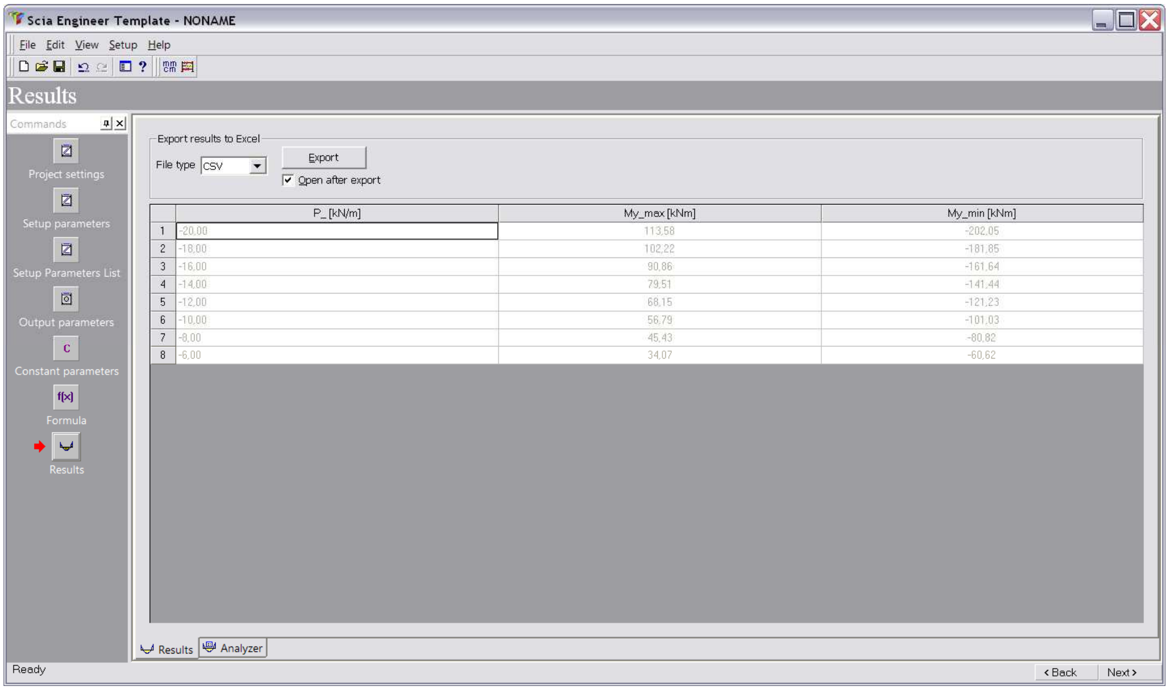


	Class	Esa Name	Name	Evaluati
1	Interne krachten in staaf	dx	dx_max	□
2		dx	dx_min	□
3		N	N_max	□
4		N	N_min	□
5		Vy	Vy_max	□
6		Vy	Vy_min	□
7		Vz	Vz_max	□
8		Vz	Vz_min	□
9		Mx	Mx_max	□
10		Mx	Mx_min	□
11		My	My_max	☒
12		My	My_min	☒
13		Mz	Mz_max	□
14		Mz	Mz_min	□

Possibly constant values and formulae can be inserted. They will be taken into account in the result table.

Finally you can go to the results, where a calculation is performed for the various line loads. This is performed in various iteration steps, which are executed independently from each other.

The result is as follows:



File type CSV

☑ Open after export

	P_ [kN/m]	My_max [kNm]	My_min [kNm]
1	-20,00	113,58	-202,05
2	-18,00	102,22	-181,85
3	-16,00	90,86	-161,64
4	-14,00	79,51	-141,44
5	-12,00	68,15	-121,23
6	-10,00	56,79	-101,03
7	-8,00	45,43	-80,82
8	-6,00	34,07	-60,62

Example 14: User blocks & ProjectTemplates

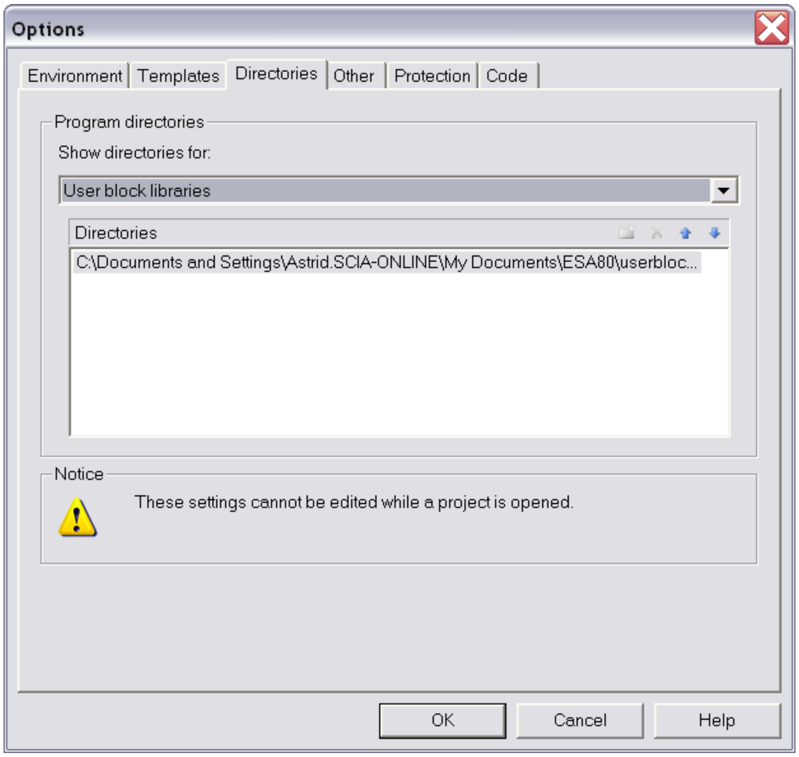
User blocks

SCIA Engineer enables the user to make a library of his/her projects that are used over and over again. These projects may be at any time included into a newly created project or appended to an earlier created and currently edited project.
The projects in this user-created library are called **User blocks** and the library is called **User block library**.

We take example 8 again: ***Steel frame.esa***

Save as user block:
'File > Save as ' and then save in the file userblocks.

You can find it in:



Options

Environment | Templates | Directories | Other | Protection | Code

Program directories

Show directories for: User block libraries

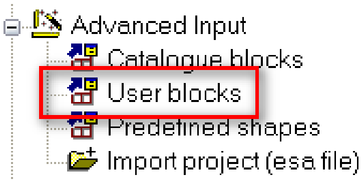
Directories

C:\Documents and Settings\Astrid.SCIA-ONLINE\My Documents\ESA80\userbloc...

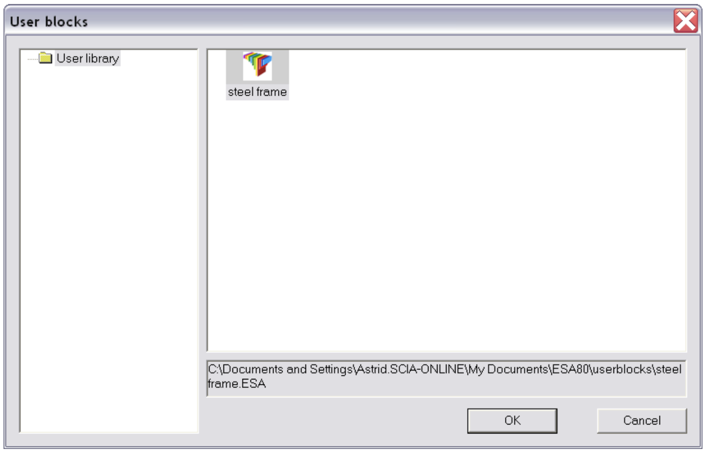
Notice

These settings cannot be edited while a project is opened.

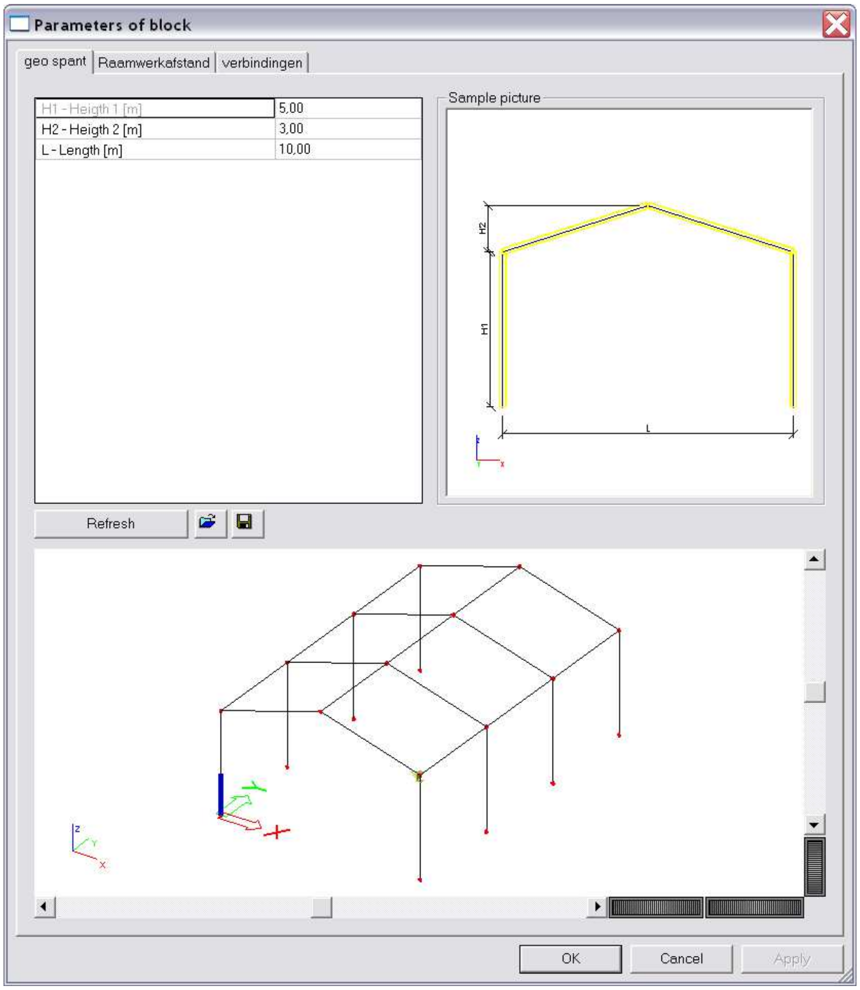
- Make a new project, e.g. in general XYZ
- Open the Structure menu :



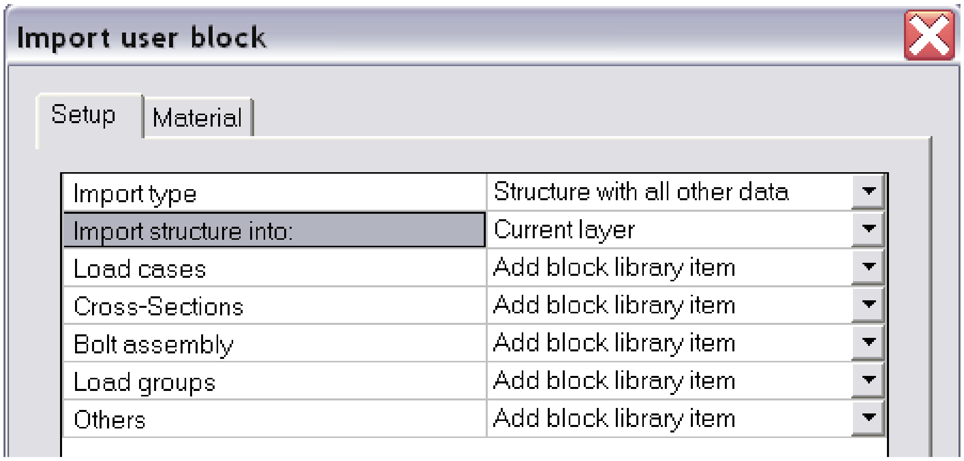
The following dialog boxes appear:



C:\Documents and Settings\Astrid.SCIA-ONLINE\My Documents\ESA80\userblocks\steel frame.ESA



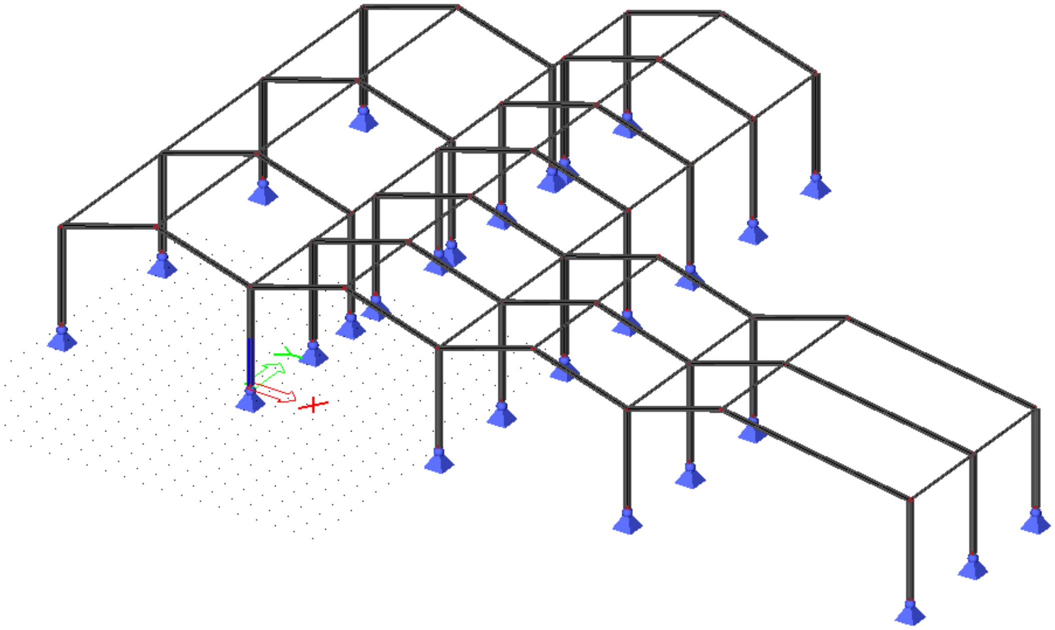
H1 - Heigth 1 [m]	5,00
H2 - Heigth 2 [m]	3,00
L - Length [m]	10,00



Import type	Structure with all other data
Import structure into:	Current layer
Load cases	Add block library item
Cross-Sections	Add block library item
Bolt assembly	Add block library item
Load groups	Add block library item
Others	Add block library item

- Insert another user block
  - with NS = 3
  - frame distance = 8 and NS = 4
  - length L = 15 and NS = 3

- Here you can get the following example:



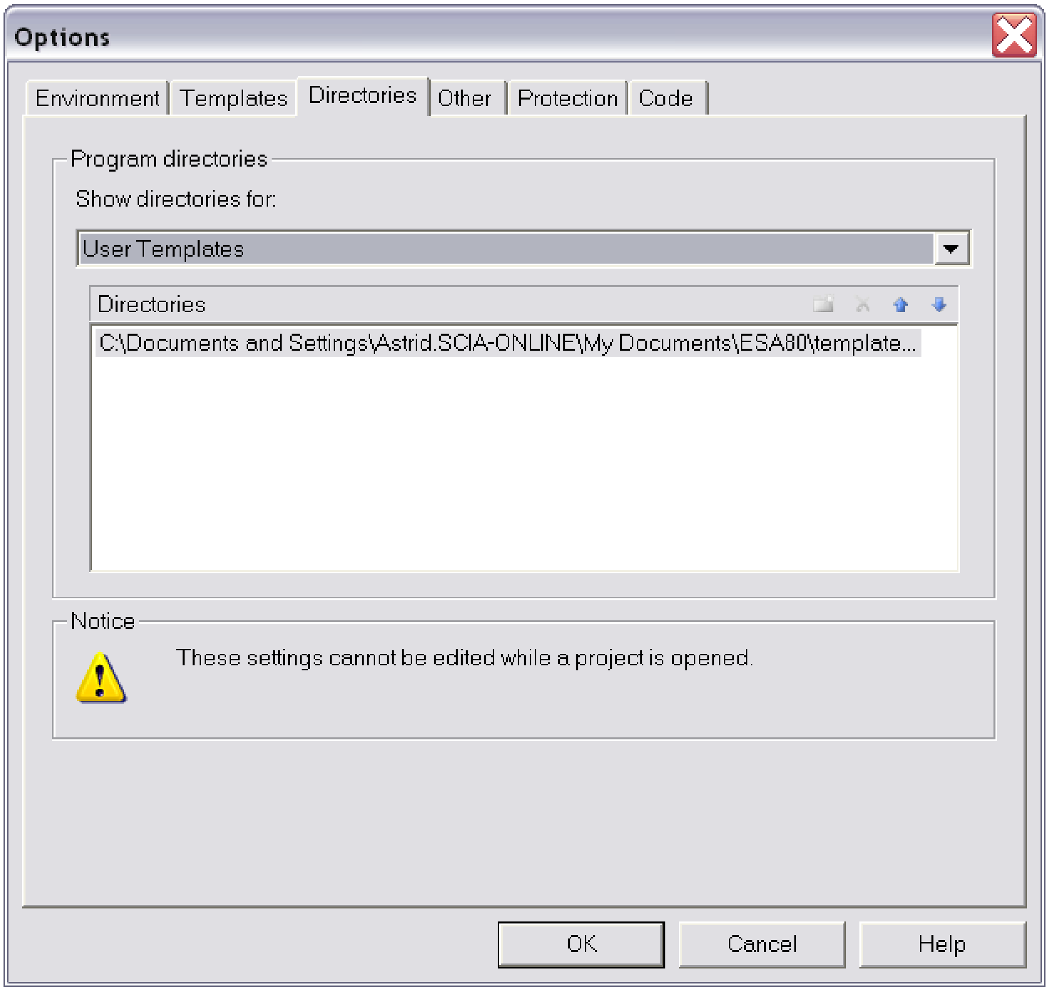
Project templates

In practice it may quite often happen that some elements are used in every project. For example, material types, cross-sections, predefined loads, and even parts of a structure may be the same in various projects. Therefore, it would be efficient, if the user could store the repetitious elements aside and load them quickly into every new project.

In SCIA Engineer this may be achieved via templates. Generally speaking, a template is an ordinary project that holds required information and is saved in a special way.

Let’s take example 8 again: Steel frame.esa

- Save as template: ‘File > Save as' in the folder:

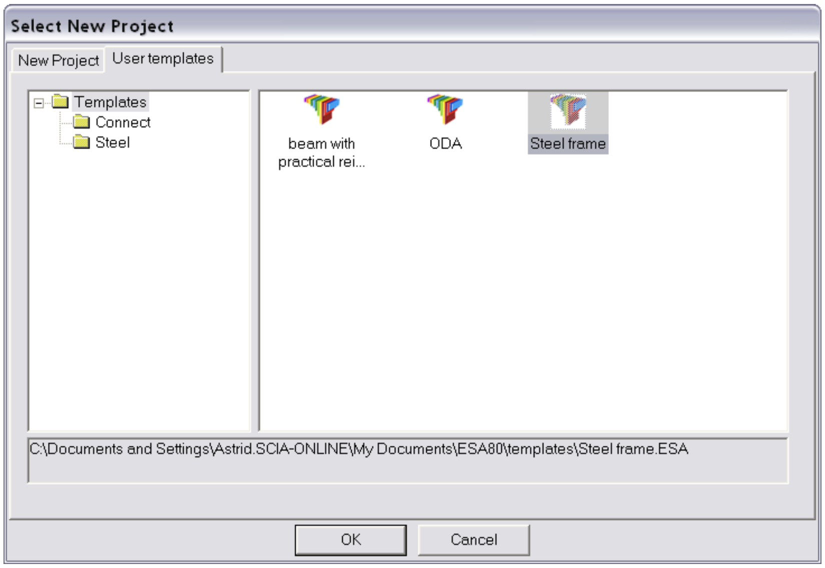


Show directories for: User Templates

C:\Documents and Settings\Astrid.SCIA-ONLINE\My Documents\ESA80\template...

These settings cannot be edited while a project is opened.

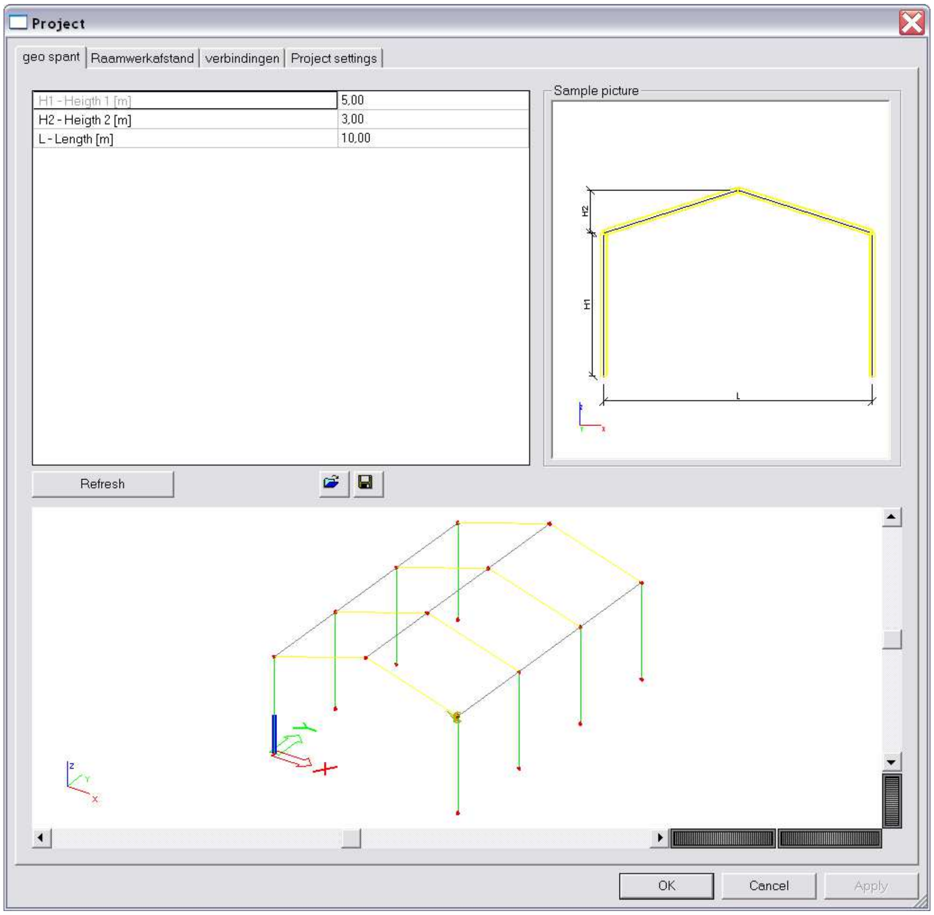
- Close example 8 and open a new project. Go to the tab 'User templates'.



Select New Project

C:\Documents and Settings\Astrid.SCIA-ONLINE\My Documents\ESA80\templates\Steel frame.ESA

- When opening this template, also here a dialog box appears with possible parameters that can be adjusted:



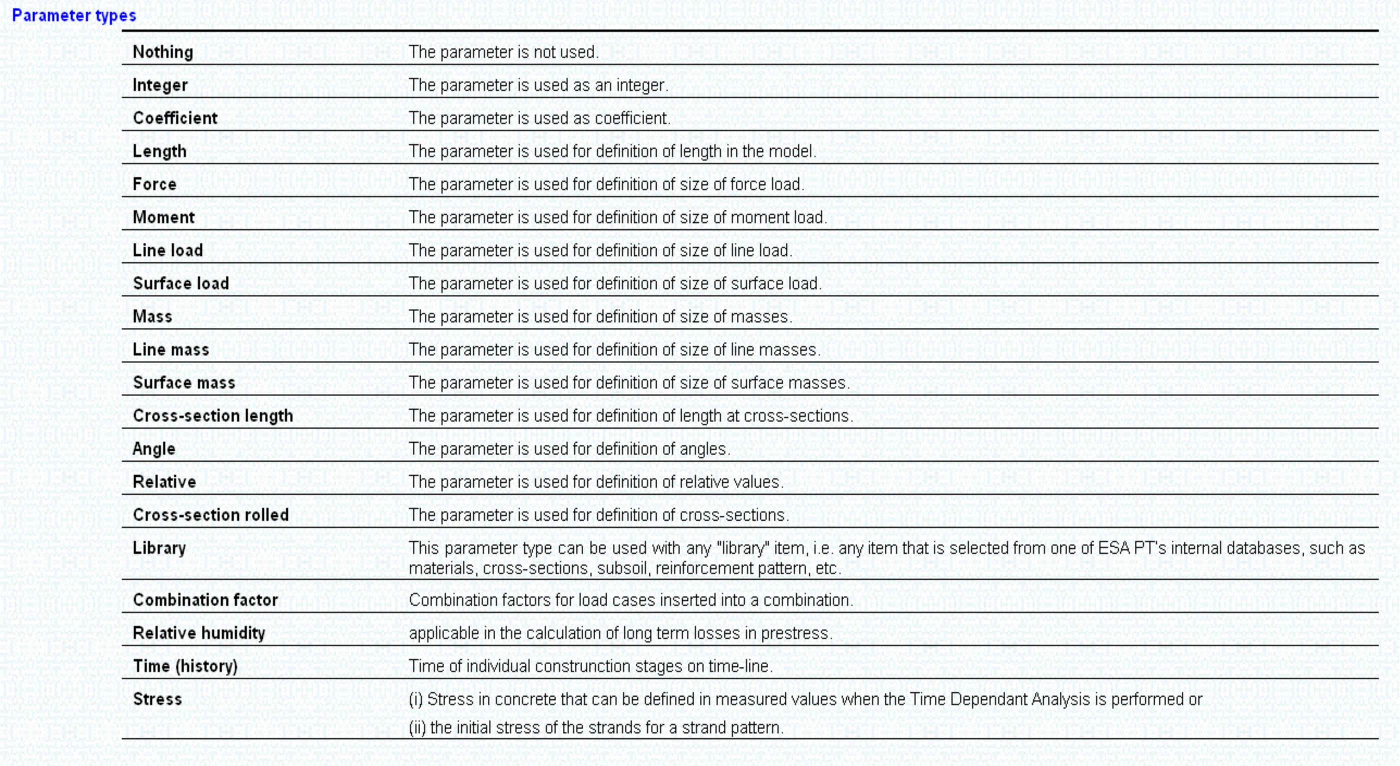
Project

H1 - Heigth 1 [m]	5,00
H2 - Heigth 2 [m]	3,00
L - Length [m]	10,00

General parameterizing

If you doubt if a certain property can be parameterized or not, it is advisable to create as many types of parameters as possible that can possibly be the required parameter.

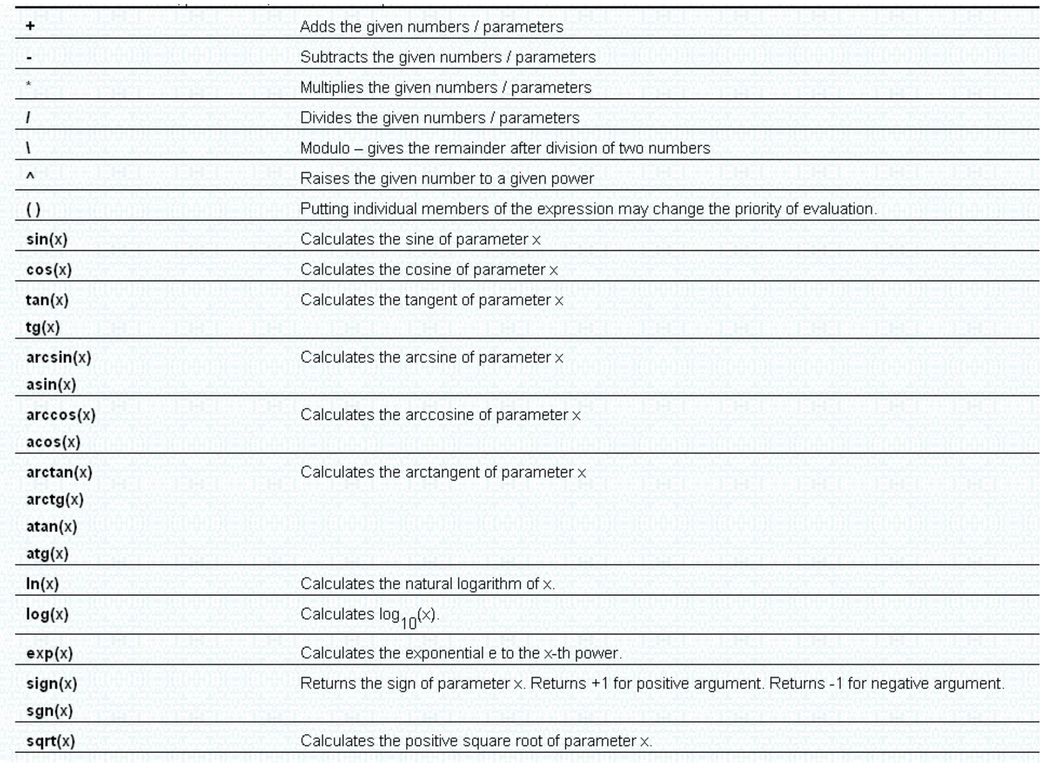
Type of parameters:



Parameter types

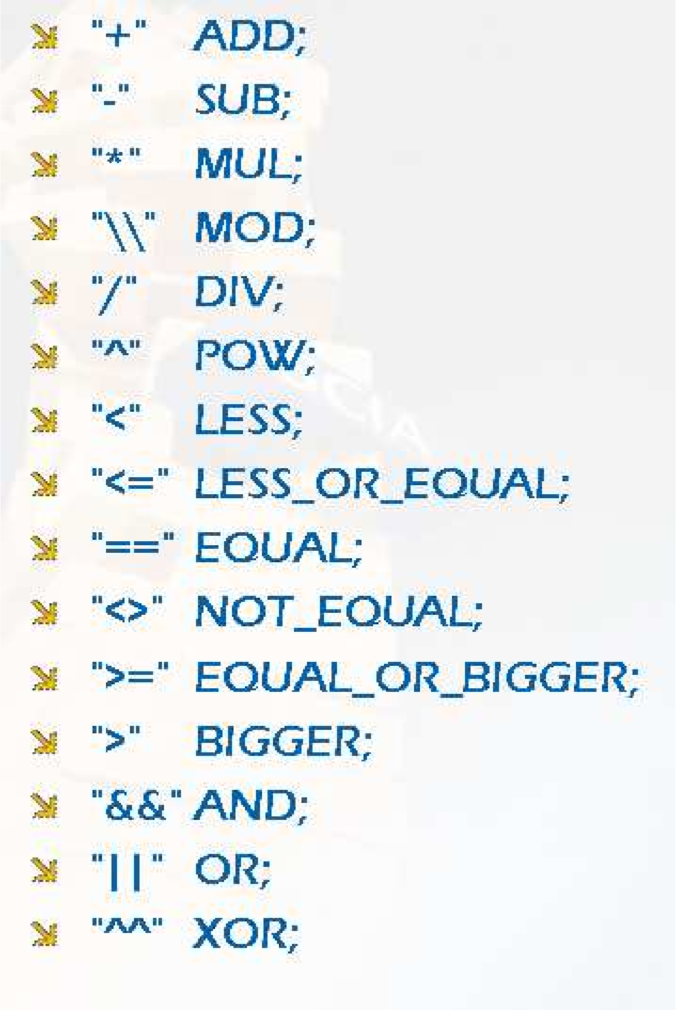
Nothing	The parameter is not used.
Integer	The parameter is used as an integer.
Coefficient	The parameter is used as coefficient.
Length	The parameter is used for definition of length in the model.
Force	The parameter is used for definition of size of force load.
Moment	The parameter is used for definition of size of moment load.
Line load	The parameter is used for definition of size of line load.
Surface load	The parameter is used for definition of size of surface load.
Mass	The parameter is used for definition of size of masses.
Line mass	The parameter is used for definition of size of line masses.
Surface mass	The parameter is used for definition of size of surface masses.
Cross-section length	The parameter is used for definition of length at cross-sections.
Angle	The parameter is used for definition of angles.
Relative	The parameter is used for definition of relative values.
Cross-section rolled	The parameter is used for definition of cross-sections.
Library	This parameter type can be used with any "library" item, i.e. any item that is selected from one of ESA PT's internal databases, such as materials, cross-sections, subsoil, reinforcement pattern, etc.
Combination factor	Combination factors for load cases inserted into a combination.
Relative humidity	applicable in the calculation of long term losses in prestress.
Time (history)	Time of individual construnction stages on time-line.
Stress	(i) Stress in concrete that can be defined in measured values when the Time Dependant Analysis is performed or (ii) the initial stress of the strands for a strand pattern.

Possible formulae:



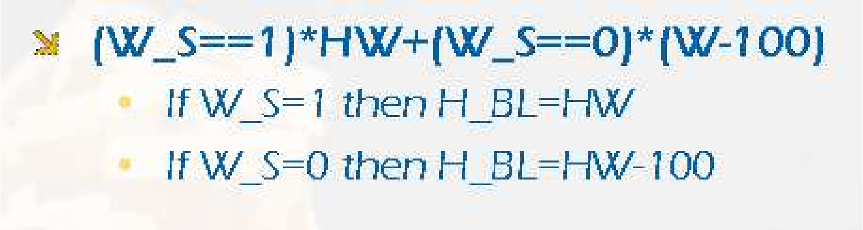
+	Adds the given numbers / parameters
-	Subtracts the given numbers / parameters
*	Multiplies the given numbers / parameters
/	Divides the given numbers / parameters
\	Modulo – gives the remainder after division of two numbers
^	Raises the given number to a given power
( )	Putting individual members of the expression may change the priority of evaluation.
sin(x)	Calculates the sine of parameter x
cos(x)	Calculates the cosine of parameter x
tan(x) **tg**(x)	Calculates the tangent of parameter x
arcsin(x) **asin**(x)	Calculates the arcsine of parameter x
arccos(x) **acos**(x)	Calculates the arccosine of parameter x
arctan(x) **arctg**(x) **atan**(x) **atg**(x)	Calculates the arctangent of parameter x
ln(x)	Calculates the natural logarithm of x.
log(x)	Calculates $\log_{10}$(x).
exp(x)	Calculates the exponential e to the x-th power.
sign(x) **sgn**(x)	Returns the sign of parameter x. Returns +1 for positive argument. Returns -1 for negative argument.
sqrt(x)	Calculates the positive square root of parameter x.

Possible operators in SCIA Engineer:

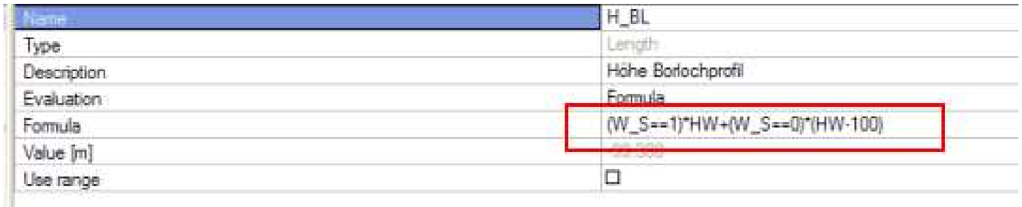


- "+" ADD;
- "-" SUB;
- "*" MUL;
- "\\" MOD;
- "/" DIV;
- "^" POW;
- "<" LESS;
- "<=" LESS_OR_EQUAL;
- "==" EQUAL;
- "<>" NOT_EQUAL;
- ">=" EQUAL_OR_BIGGER;
- ">" BIGGER;
- "&&" AND;
- "||" OR;
- "^^" XOR;

Example of a formula:



- (W_S==1)*HW+(W_S==0)*(W-100)
  - If W_S=1 then H_BL=HW
  - If W_S=0 then H_BL=HW-100



Name	H_BL
Type	Length
Description	Höhe Borlochprofil
Evaluation	Formula
Formula	(W_S==1)*HW+(W_S==0)*(HW-100)
Value [m]	-99.300
Use range	□