Parameters



Advanced Training Parameters



1

All information in this document is subject to modification without prior notice. No part or 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. Nemetschek Scia is not responsible for any direct or indirect damage because of imperfections in the documentation and/or the software.

© Copyright 2013 Nemetschek Scia. All rights reserved.

Preface	4
Beam on two supports	5
Cross-sections	10
General cross-section	10
Type of Cross-sections	13
Catalogue block	16
Plate on subsoil	20
Cellular beam	22
Beam with practical reinforcement	24
Ground-level	28
Layers	31
Steel hall	
Plate with ribs	
Plate with ribs_1.esa	38
Plate with ribs_2.esa	41
Tubular plates	43
Hollow core slab	49
XML	63
ODA	65
Batch optimizer	70
Example 14: User blocks & ProjectTemplates	76
User blocks	76
Project templates	79
General parameterizing	81
Type of parameters:	31
Possible formulae:	32
Possible operators in Scia Engineer:	33

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:

<u>Method</u>

• 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:



Here the template tabs are made

	Parameter se	et		
,) 🤮 🗶 📸 🗽	🖸 🗠 🖨 🕞 🖬	All V	
6	Geometry			
Na	ame	Geometry		
Us	se 3D preview			
Inf	fo			
Pi	cture	selected		
lc	on			
Re	emove (picture,			
EF	Parameters			
	В			
	type	Css length		
	unit	mm		
	H	Castanath		
	type	Cssiength		
	unit	mm		
	L type	Length		
	unit	m		
L L	anne			
			Z	
			Y X	
Ne	ew Insert E	Edit Delete		Close

🛛 Project

Pl Structure

🔟 Template dialogue

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

• Subsequently you can use a new function:

Here the values of the various parameters can be adapted.

🗅 😂 🖬 💁 😑 🗉 📍 voorbeeld 1 - ligger 💌	▝▝▝▝▝▝▌▌▌▌▌▌▌▌▌▌▌▌▌▌▌▌▌▌▌▌▌▌▌▌▌▌▌▌▌▌▌▌▌
이 여 역 여 여 여 여 대 제 대 위 응 부 주 🖷	
Main	Project 🗙
Main Project Template dialogue Structure Load cases. Combinations Colourient Colourient Concrete Concrete Concrete Concrete Subcay Consesections Coloure. Analysis Social Concrete. reinforcement Subcay Consesections Concrete. reinforcement Subcay For Heat Social fundation For Heat Social fundation For Heat For H	Project Geomety B-(mm) H+(mm) 400.0 L-(m) 25.00
Parameters	
Properties	
Project data (1) 🗸 Vē	
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 :	
No. of used materials : 1	
Last used annex	
	OK Cencel Apply

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

Pr	operties	×
N	ode (1)	▼ Va V/ Ø
	Name	K2
	GCS coordinate	
	Coord×[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

 \rightarrow Length beam: X-coordinate of node K2:

→ Width & height profile:



• Changing a parameter:

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





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

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.

GC	S Cross-section ed	itor					
Π	¹ □ ² All ▼ Norm	al colour 🔹	ລ 🖂 💺 🖩 📥	🖌 🖌 🕑	🕹 🙃 🕫 🔂 🛋 diế điể tử t		8 F 8 8+
					ା <u>ଳ</u> ର ର ର ର ଲ 🧰 🗌		
	Polygon Polygonal open Thin walled Section from libr II- Dimension line II- Dimension line P Parameter M Thinwalled repro-	ing ary /G esentation	Parameter	's All	• 7		<u> </u>
			H1		Name	B1	
			B1		Туре	Css length	-
					Description		
					E∨aluation	Value	-
					Value [mm]	400,0	
					Use range		
	Type (description) General Buckling y-y Buckling z-z	General cross-: c			Actions		
	Fabrication	aeneral			Validate	>>>	
	Display final shape	Ø	New Edit	Delete		Close	
	Display	Only basic sha	oe 💌				_
A F	ctions Refresh Command >		>>> (•
	eadv						NUM
-							Class
							Ciuse

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

6 C	S Cross-section	editor															
	⊐² All ▼ No	ormal colour	• <u>5</u> 0	😺 ::🖥 g	<u>s</u> 8	A II	Ŧ	0°0 0°0	0 ja 0 👌	otë čl i t	% † [42.0	₿ ⊅ ₿	# 🚯) 隆	₿+
-11							<u>(</u>	Ω	ົລຸດ	Ω 📶							_
Г				- f ·		•					•	•	•		•	•	
				1													
	🗖 Polygonal op	pening															
	Thin walled	Denen i															
	Burnersion li	ne ne															
	ws Import DXF /	DWG															
	P Parameter																
	- 📶 Thinwalled re	epresentation															
						-											
H				- · ·													
L	Name	P3		_													
屵	GCS coordinate																
	Coord Y [mm]			-		1											
F	UCS coordinate			9													
Γ	Coord uv [mm]	, 400		-1 `		-								-			
	Coord uz [mm]	200		-													
	Cut															·	
	Number of cut	0		1													
	Name of cut	None	▼		•		•								•	·	
	CutY																
	Cut Z			_ ·			•				·					·	
				•			•	•			•				•	·	•
				•													
F	Command >																-
, Re	eady														NU	И	
															Clea		T
															Cids	e	

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:

Parameters		
🔎 🖋 👪 🗽 All	- 7	
L - Length beam	Name	B1
B - Width strandard sec	Туре	Css length 🔹
H - Heigth standard sect	Description	Widht graph section
LC - Line load	E∨aluation	Value 💌
H1 - Heigth graph section	Value [mm]	400,0
B1 - Widhi, graph secuori	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:

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

Geometrie	Graphical cross-section	
		Los o
	th aronh costion [mm]	1400.0
BI-Wid	an graph section [mm]	

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:



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	• 7		
L - Length beam B - Width strandard sec H - Heigth standard sect LC - Line load H1 - Heigth graph section B1 - Width graph section	Name	CS	
	Туре	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		>>>
New Edit Delete			Close

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:

Calculation, Mesn Steel Concrete			
Document			
Crawing Tools			
HIP Line grids		=	
Parameters			
🔞 Parameters template sett	inas		
lavers			
User defined selections			
- 🔀 UCS			
📲 Cleaner			
- []? Coordinates info			
- 1 XML IO Document			
Convert Steel Profile Db		~	
n_			1
<u>*</u>			
Properties		Ψ×	
Member (1)	•	通 🌾 🧷	
Name	S1	~	
Туре	general (0)	-	
Analysis model	Standard	-	
CrossSection	Master - RECT (400; 400)	▼	00 🕖 🔺 🛵 🚝 📽 👺 🤝 🚚 📑
Alpha	n	-	
Member system-line at	centre	•	Preview
ez [mm]	0	▼ =	Pa 😃 👺 姜 H □ 🗹 🔟
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:



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'.



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







Method:

First of all, the parameters will be created. Go to 'Tools > Parameters': •



- The following properties are parameterized with a begin value of:

 - Span L: 10 m
 Height H1: 5 m
 Height H2: 1,5 m

 - 4. Htot: H1 + H2
 - 5. Middle: L/2
 - 6. Rigidity support: $5*10^{-6}$

🔜 Parameters			×
🥒 🗶 📸 🗽 🛛 🗛		• 7	
Overspan - span	Name		Overspan
H1 - first height	Туре		Length
H2 - second height	Description		span
Htot - total height	E∨aluation		Value 💌
Miaden - X miadie	Value [m]		10,00
R - Stittness	Use range		

Parameters				
🔎 🗶 📸 🗽 📶		• 7		
Overspan - span	Name		Htot	
H1 - first height	Туре		Length	
H2 - second height Htot - total height	Description		total height	
	Evaluation		Formula	•
Midden - X middle	Formula		H1 + H2	
R - Sunness	Value [m]		6,50	
	Use range			

Parameters			
🔎 🗶 📸 🗽 🛛 All		▼ 7	
Overspan - span H1 - first height H2 - second height Htot - total height Midden - X middle R - Stiffness	Name	R	
	Туре	Point stiffness	
	Description	Stiffness	
	Evaluation	Value	-
	Value [MN/m]	8,0000e-001	
	Use range		

• When closing this window, the following question appears:

Scia En	gineer 🔀
?	Would you like to check the correctness of the formulae?
	Yes No

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:



• 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:



• 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:



Cellular beam





- 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:

Parameters		
🔎 🗶 🛍 🎚 🗛		 ▼
h - heigth of profile	Name	h
b - width of profile	Туре	Css length
d - thickness web	Description	heigth of profile
d1 - thickness flange d2 - inclination L - Length beam D - diameter borebole	E∨aluation	Value
	Value [mm]	300,0
	Use range	
x - position first borehole a - number of boreholes		

- 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.



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:



- · Load cases:
 - LC1: Selfweight
 - LC2: Variable load: q [kN/m]
 - 🖕 🖞 Redes (without As)

🔤 🔤 New reinforcement

• Input practical 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:

🔜 Parameter set				
🏓 🤮 🗶 🛍 🖳 🤉	2 🎒 🗳 🖬 📶			
Cross-section Geometry beam Reinforcement Load				
Name	Geometry beam			
Use 3D preview				
Info				
Picture				
lcon				
Parameters				
□ L1				

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

- .wmf
- 550 x 550
- Check through Template dialogue

Project	
Cross-section Geometry beam F	Reinforcement Load
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:

Select New Project	
New Project User templates	
C:\Documents and Settings\Astrid.t	beam with practical rei SCIA-ONLINE\My Documents\ESA80\templates\beam with practical
	OK Cancel

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

Parameters





- 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

🔜 Subsoils					
🔎 🤮 🗶 📑 🕏	<u>n</u> 🖂 🖨	🗳 🚅 🖬 🛛 🗛		▼ 7	
Sub1		Name		Sand/Clean/Loose	
Sand/Clean/Loose	9	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	
t	Geolog	ic profiles			×
	,11 💱 🏒 🛙	i k 🗅 🗠 🖨 🛛	🖻 🖬 🛛 🗛	•	7
-	GP1				
	Name Selector s Water hei Not compr Layers 1 Layer's Thickne	GP1 ▲ □ 1,000 ■ 2,000 ▼		Thickness = 2.00[m], 1 Thickness = 4.00[m], 1	Edef = 20.00[kN/m^2], Weight = Edef = 25.00[kN/m^2], Weight =
	New Inse	ert Edit Delete			Close

- Load cases:
- SelfweightWater pressure
- Soil pressure

Method

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

Parameters 🔀				
🥕 🗶 📸 🗽 🛛 All	🚚 🧷 📸 👞 🗛			
h - ground level		Name	h	
B - subsoil		Туре	Length	
G - geologic profile		Description	ground level	
M - Materials		E∨aluation	Value	-
M1 - Materiais		Value [m]	-1,00	
		Use range		
	Ξ	Range		
		Mimum Evaluation	Value	-
		Minimum [m]	-4,00	
		Maximum Evaluation	Value	-
		Maximum [m]	0,00	

• Test of input data through:



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:
 - 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 : \rightarrow Boolean
 - L1: formula N>=1
 - L2: formula N>=2
 - L3: formula N>=3

- 3. Model type of the layers: \rightarrow 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 🔀				
🥕 🧶 👪 🗽 🗛		▼ 7		
N - Number of beams	Name	N		
L1 - Activity 1	Туре	Integer		
L2 - Activity 2 L3 - Activity 3 C1 - CAD1 C2 - CAD2 C3 - CAD3	Description	Number of beams		
	Evaluation	Value 💌		
	Value	2		
	Use range			
1	1			

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



Layers				
🔎 🤮 🗶 🛍 🔛 🎒 🗛		▼ 7		
Laag1	L1 - Activity 1	Name	Laag1	
Laag2	L2 - Activity 2	Comment		
Laag3	L3 - Activity 3	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	
	Sample picture
IN - Number of beams 2	
Befrech 🔗 📮	
N	
	2/10002 \$3/10003
	<u>_</u>
	OK Cancel Apply

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



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 -

Layers				
1 1 2 📽 🕸 🎒 🗛		▼ 7		
Layer1	B1 - activity 1	Name	Layer7	
Layer2	B2 - activity 2	Comment		
Layer3	B3 - activity 3	Colour		
Layer4	B4 - activity 4	Structural model only	A6 - CAD6	-
Layer5	B5 - activity 5	Current used activity	B7 - activity 7	•
Layer6	B6 - activity 6	Current used deavily	Di Golinyi	
Layer7	B7 - activity 7			
1				

- -
- 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				X
🎜 😳 🖉 🖬 🔛 🧉	All	v 7		
Laag1	A1-0	Name	Laag_6	
Laag_2	A2 - 0	Comment		
Laag_3	A3 - 0	Colour		
Laag_4	A4 - 0	Structural model only	C6-0	-
Laag_5	A5-0	Current used activity	A6-0	•
Laag_6	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:



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.

Parameters			\mathbf{X}
🥒 🗶 📸 🔜 🗛		▼ 7	
× - offset first beam		Name	N
B - width plate		Туре	Integer
×1 - B-×		Description	
N		E∨aluation	Value 🔹
A1		Value	2
A2		Use range	⊠
84		Range	
A5		Mimum Evaluation	Value 🔹
C1		Minimum	2
C2		Maximum Evaluation	Value 🔹
СЗ		Maximum	5
C4			
C5			
I ∨2			

- 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:
- $x^2 = x + ((B 2x)/(N 1))$
- 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:



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

- 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	•	Connect selected nodes with new beams Copy additional data
Distance	vector		How to define the distance ?
Define dis	stance by cursor		between two copies
x	0,000	m	\bigcirc from original to the last copy
У	0,000	m	 How to define the rotation ? between two copies
z	0,000	m	from original to the last copy
-Rotation-			-Rotation around
rx	0,00	deg	current UCS
ry	0,00	deg	O distance vector
rz	0,00	deg	OK Cancel



• 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		
Geometry ribs Nun	nber of ribs	
N-number of par	ts E	<u>}</u>

→ Analysis model:



→ Structural model



Knoopondersteuning			×
Rx Rx i ry	Naam Type Hoek [deg] X Y Z Stijfheid Z [MN/m] Rx Py Rz Standaard afm. [m] Geometrie Systeem	Sn13 Standaard Vast Vast Verend R - in yrichting Vast Vast Vast Qast GCS	
		ОК	Annuleren

Pr	Properties 4 ×				
s	upport in node (1)	▼ Va V/ Ø			
	Name	Sn11			
	Туре	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			
Ξ	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'.



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.

Parameters



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 🛛 🔀				
🔎 🤮 🗶 🖬 K 🕽	• 🗠 🖂 🎒 📽 🖬 🗛	• 🖗		
stage 1	Name	stage 7		
stage 1b	Description			
stage 2	Action type	Permanent		
stage 3	LoadGroup	LG1	▼	
stage 4	Load type	Standard	•	
stage 5				
stage 6				
stage /				
stage 6a				

- 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			X
¢		Time analysis	•	^
	Permanent (long-term) load case	0.00		
	Gamma min [-]	1.00		
	Gamma max [-]	1,00		
	Prestressed load cases			
	Gamma min [-]	0,00		=
	Gamma max [-]	1,00		
E	Long-term part of variable load			
	Factor Psi [-]	0,30		
	TDA			
E	Load factors for generated loadcases			
	gamma-creep min [-]	1,00		
	gamma-creep max [-]	1,00		
E E	Time - History			
	Number of subintervals	1,0		
	Ambient moisture [%]	70,00		
	Automatic calculation of subintervals	🗆 no		
	🗆 Local time axis			
	Time of costing	-1.00		<u> </u>
		OK	Cancel	

<u>Stage 1</u> Prestress and selfweight of the member

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





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



<u>Stage 5</u> Casting of the coating

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



Construction stages			
🧯 🕃 🗶 😒 😂 😂	A	I 🗸	7
ST1		Name	ST5
ST2		Order of stage	5
ST3		Description	
ST4		Global time [day]	T5-fase 5 🔹
SIS		Number of subintervals	1
		Ambient humidity	RV1-RV2
		Last construction stage	
	Ξ	Load case permanent or long-te	
		Load case	fase 5 💌 💌
		Gamma min [-]	0,00
		Gamma max [-]	1,00
	Ð	Load case prestress	
		Load case	None 💌
		Type of generated combinations	Code independent 🔹
		- 1 ¹ - 11 - 11 - 11 - 11 - 11 - 11 - 1	
	Actions		
Variable load cases			
New Insert Edit Delete Close			

<u>Stage 6</u> Service

Load cases: long termVar 1.2 kN/m \rightarrow 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%



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-L	η 💌
	<		>
	<u>.</u> .		- 1
	Stre	ngeigenschap	Geometrie
			<u>V</u> ;
		Naam	Streng8
		ld	8
		Groep	1
		Materiaal	Y1670C- 🔻
		Positie in gat	Centrum 📃 💌
		Vast	
		Onthechtings	Geen 💌
		Volgorde van	1 🗾
		Manier van v	Туре 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
	Ð	Positie	
		Y [mm]	252
		Z [mm]	20
	I		

- Anchorage length:

Type: length Value: 1m



	Y1770C-6,0-I Y1670C-6,9-I Y1670C-7,0-I Y1670C-7,5-I					
Stre	engeigenschap	Geometrie				
	Naam	Streng8				
	Id	8				
	Groep	1				
	Materiaal	Y1670C 🔽				
	Positie in gat	Centrum 💌				
	Vast					
	Onthechtings	Geen 💌				
	Volgorde van	1 💌				
	Manier van v	Туре 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				
Ξ	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			
🏓 🤮 🗶 🖭 😂 🎒	AI		7
ST1	Γ	Name	ST1
ST2		Order of stage	1
ST3		Description	
		Global time [day]	T1 - time of stressing
515 CT2		Number of subintervals	1
ST7		Ambient humidity	RV-stage 1
317		Last construction stage	
		Load case permanent or long-te	
		Load case	fase 1b 💌 💷
		Gamma min [-]	0,00
		Gamma max [-]	1,00
	Ξ	Load case prestress	
		Load case	fase 1 💽 💌
		Type of generated combinations	Code independent
1			

- 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
Туре	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
Туре	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

C Member1D opening					X
	Γ	Name	OM1		^
		Shape	Rectangular	-	
ez		Rectangular			
		B [mm]	hole_w-length	•	
^β X X ^a		H [mm]	hole_w-length	•	
		Alpha [deg]	0,00		
		Position y/z			
The state of the s		Alignment	Тор	•	
		Perp.offset [mm]	p_offset - perpendicular offset	•	=
		Orientation	Z	•	
(n -1) x ∆x 🚬 💦		Beta [deg]	0,00		
x		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	Abco	-	Υ.
			OK	Cancel	

8. Using the template dialog

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



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 tailormade 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



Coordinates info

XML Input / Output Document	🗆 XML Input / Output Document 📃 🔲 🔀							
DOC-Default	Þ W 5 4	🛛 📙 🕂 🖬 🖬 📑 default 🛛 🖌 🖉 🎇 default	✓ □ □□					
Pefault Parameters	1. Parar	1. Parameters						
	Name	UniquelD	Туре	Evaluatic				
	H1	{5A0826EE-1820-4458-9086-31781F326D32}	Length	Value ■				
	H2	{B4D32D70-36E7-4512-B1D1-A7193ADD5A18}	Length	Value				
New Close	Htot	{9472EB82-54C7-4D48-AE09-E570F4D24E66}	Length	Formula				
Name Parameters Caption Parameters Visible yes V	L	{676B5CB5-B211-44F2-B74B-3C965301B56A}	Length	Value				
Frefer one page	NS	{8E25ED26-1A55-4CF6-8BD4-5353B39CA8F1}	Integer	Value				
	S	{4A3FD994-A184-40F7-A7A0-D97607CA2AC6}	Length	Value				
	S1	{DCF03F45-0036-49A9-AB9E-24D0723A0605}	Length	Formula				
Actions	S2	{3A20EBF8-F846-42E7-A9FE-7894DA6257D9}	Length	Formula				
Refresh >>>	📕 Ready [en]	×		> •				

• 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="R@]|"
                   />
     </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">
  <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 is the abbreviation of 'One dialog application'. It's also called the Scia engineer Template.

It can be used for following type of projects:

c	hoose type of i	new project					
	Concrete Beam - Standard	Concrete Beam - Advanced	Concrete Slab	MixBeam	Parametric Project	Batch optimizer	
	Parametric Project	l				OK	el

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:

c	hoose type of r	new project					X
		□+ = ₩₩		7			
	Concrete Beam - Standard	Concrete Beam - Advanced	Concrete Slab	MixBeam	Parametric Project	Batch optimizer	
	Zomen						
	Zeman						
	Concrete Beam - S	Standard				ОК	
						Cancel	

Choose the option 'Parametric project'.

• The following dialogue box appears:

Select New Project			
User Templates			
Templates	beam with practical rei	VDA	
	ОК	Cancel	

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



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

🌾 Scia Engineer Te	emplate - ODA										
<u>Eile Edit View Setu</u>	p <u>H</u> elp										
🗍 🗅 🚅 🖶 😖 🖂 🗉	? 🕅 🎮 🗊 🗋										
Document											
Commands 👖 🗙	DOC-Standaard 🗸 🛁	nu 55		📕 default		VII 🔢	•				
Commands Project settings Profiel R Geometry R Document	DOC-Standaard m Standaard Node (knopen) Load cases (Belasting Combinations (Comb Parameters Mode (knopen) Combinations on me Parameters Mode of the combinations on me parameters Mode (knopen) Combinations on me Parameters Mode of the combinations on me Parameters Testers and the combinations on me Pooter templete default Edit header term. Combinations on templete default Edit the page true. First chapter m. Chapters num. All Chapters capt. All Editone Editone Combinational combinations on the combination of the combinatin the combination of the combination of the		IIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIII	default Project Omschrijk Omschrijk Auteur Orderde Omschrijk Auteur Orderde Omschrijk Auteur Orderde Onschrijk Auteur Orderde Onschrijk Auteur Orderde Onschrijk Orderde Onschrijk Orderde Onschrijk Orderde Onschrijk Orderde Onschrijk Orderde Onschrijk Onschri Onschrijk Onschrijk Onschri		aat Z 0,000 F Belasting Eigen gewin Statisch 1,00 1,00 1,00 1,00 1,00 1,00	Naam Coord 2 2 type Spec cht Standaard arde Eenheid 0 mm	Imaat X Minaat X 6,000 Richting -Z	Coördinaat Imi 0,0	Y Coordinaat Z Ini 000 0,000 Master' belastingeval Geen	
	Refresh of document >>>	Selec	tie : Alle								
	Ketresh of pictures >>> Load settings >>>	BG	i Staaf cs	s dx	N	Vy	Vz Mx	My	Mz		
	Save settings >>>	LICT	01 001	Tube O		KN 0.00			[kNm]		
	Save template >>>	菌 Ready [nl]									~
Calculation done	·										h lau d h

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

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.





70

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

	😑 🚧 Model data
	🖨 🛣 Support
Input support:	🛁 📥 in node

<u>Method</u>

٠

- 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:

🗆 XML Input / Output Document									
/ 🖪 🕨		📔 🕂 🖸 🚾 📑 default 🔷 🖳 default	✓ □ □□□						
1. Parameters									
Na	ame	UniqueID	Туре	Evaluation	Use range				
P_		{1C5AFDAB-6476-45C8-AF12-2F77DDC47BC2}	Line load	Value	false				
4									
	Document	Document I. Paran Name P_	Document	Document Image: Second seco	Document Image: December 1 Image: December 2 Image: December 2 Image: December 2 Image: December 2<				

🔲 XML Input / Ou	tput Document			
DOC-Standaard	💌 📭 💷 👺 👙 📋 🕂 🔲 🔟 🔟 🔟 🚺 📑 default 🛛 🗸 🖳 default	✓ □ □□		
E 🔶 Standaard	1 Parameters			
	Name UniqueID	Туре	Evaluation	Use range
Í	Document Export		X	false
New Close Description Site Embed docu D Language Em Pictures align Le Header templ de Edit header te Title page tem *** Edit tile page tem *** Edit tile page tem First page nu 1 First page nu 1	File D\Scia.Esa PT\Projecten Per theorie\Parameters\Projects parameter Type XML file (usable for data exchange with other systems) Image: XML Open after export Volicode Description	rrs\input_batch opt.xm		
Actions Refresh of docum Load settings	English (United States) [1252]	Export	Cancel	
Save settings Save template	>>> Naam	1111		>

Output file:

DOC-Standaard ✓ Image: Constraint of the standaard Standaard Image: Constraint of the standaard Standaard Image: Constraint of the standaard Mame Internel forces on me Name Interne kracht. Name Interne kracht. Visible Image: Caption Selection All Load cases BG1 Structure Initial Values My	en in staa	a	💹 🛄 default								
Standaard Minternal forces on me BG Staaf cs BG1 S1 CS1 - R BG1 CS1 CS1 - R BG1 CS1 CS1 - R BG1 CS1 CS1 CS1 - R BG1 CS	en in staa			~	□ 📲						
New Close Neme Interne krach Caption Interne frach Visible Byes Prefer one pa D Selection All Load cases BG1 Filter No<		Standaard M Internal forces on me 1. Interne krachten in staaf									
New Close BG Staaf Cs Name Interne krach BG1 S1 CS1 - R Caption Interne frach BG1 S1 CS1 - R Visible Ø yes BG1 S1 CS1 - R Prefer one pa D Selection All Type of loads Load cases G1 Filter No Values My	Table										
New Close BG1 S1 CS1 - R Name Internet krach BG1 S1 CS1 - R Caption Internet force BG1 S1 CS1 - R Visible Ø yes Prefer one pa D Selection All Selection All Load cases BG1 Visues My	s dx		Vy Vz		М×	My	Mz				
New Close Name Interne krach Caption Internal force Visible ⊠ yes Prefer one pa □ Selection All Type of loads Load cases Load cases BG1 Filter No<	RECT 1e-0	2	28112.527	734	0	-50512.75	0				
Name Interne krach Caption Internal force Visible Ø yes Prefer one pa □ Selection All Type of loads Load cases = Load cases BG1 Filter No Structure Initial	RECT 9	8	-16887.47	7266	0	0	0				
Visible Ø yes Prefer one pa Selection All • Type of loads Load cases • Load cases BG1 • Filter No • Structure Initial •	RECT 5.4	2.) 1112.5273	344	0	28394.90039	0				
Preferone pa Selection All Type of loads Load cases Load cases Filter No Structure Initial Values My				I							
Selection All Type of loads Load cases Load cases BG1 Filter No Structure Initial Values My											
Type of loads Load cases Load cases BG1 Filter No Structure Initial Values My											
Load cases BG1 Filter No Structure Initial Values My											
Filter No Structure Initial Values My											
Structure Initial Values My											
Values My 🔹											
System Principal 💌											
Extreme Global 💌											
Section All 🔹											

🔲 XML Input / Ou	itput Docume	ent						
DOC-Standaard	·	6 W B6	🗿 🔲 🕂 🔲 🔟 🛍 🧻 default	🗸 🚇 🏪 default	✓ □ □□			
Standaard Parameters 1. Parameters								
		Name	UniquelD		Туре	Evaluation	Use range	
	Document E	port				X	false	
New Close Description Ste Embed docu □ Language En Pictures align Le	File Type SML Open a	Ifter export)\Scia.Esa PT\Projecten Per theorie\Parame	iters\Projects parameters\ou ;ystems)	tput_batch opt xi	nl 🗸		
Header templ de Edit header te Title page tem Edit title page t. Footer template de Edit footer tem First page nu Chapters num All Chapters capti All	Unicod	e otion						
Actions Refresh of docum Load settings Save settings	English (Unite	d States) [12	252]		Export!	Cancel		
Save template	>>>	Naam		<	ш		>	
• This project can now be opened in ODA. For this, you choose the option 'Batch optimizer'.

с	Choose type of new project					
	Concrete Beam - Standard	Concrete Beam - Advanced	Concrete Slab	MixBeam	Parametric Project	pptimizer
	Name					OK Cancel

• 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					
Parameters file					
D:\Scia.Esa PT\Projecten Per theorie\Parameters\Projects param					
Output parameters					
XMLIO Document					
Parameters file					
D:\Scia.Esa PT\Projecten Per theorie\Parameters\Projects param					
Type of analysis					
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.



🌾 Scia Engineer Tem	🕼 Scia Engineer Template - NONAME					
Eile Edit View Setup	Help					
🛛 🗅 🖨 🖬 🗠 🗠 🔳 ។	? 🛛 🕅 🎮					
Setup parame	Setup parameters					
Commands a ×						
	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.

🐨 Scia Engineer Template - NONAME 📃 🔲 🔀					
Eile Edit View Setup	<u>H</u> elp)			
🗍 🗅 🚅 🖬 🖕 🗠 🔲	? m	m 🕅			
Output param	iete	rs			
Commands # ×	De	ep refresh			
		Class	Esa Name	Name	E∨aluatio
Project settings	1	Interne krachten in staaf	dx	dx_max	
	2		dx	dx_min	
Setup parameters	3		N	N_max	
Setup parameters	4		N	N_min	
	5		Vy	Vy_max	
Setup Parameters List	6		Vy	Vy_min	
	7		√z	Vz_max	
	8		√z	Vz_min	
Output parameters	9		Mx	Mx_max	
C	10		Mx	Mx_min	
	11		My	My_max	
Constant parameters	12		My	My_min	
f(×)	13		Mz	Mz_max	
Formula	14		Mz	Mz_min	
Results					
Ready				< [Back Next >

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:

🕫 Scia Engineer Template - NONAME					
Elle Edit View Setup Help					
🗍 🗅 🚅 🖬 🗠 😂 🔳	? NR #				
Results					
Commands • X Project settings	Export results to Excel				
Colore and the second s	P_[kN/m]	My_max [kNm]	My_min [kNm]		
Setup parameters	1 -20,00	113,58	-202,05		
	2 -18,00	102,22	-181,85		
Setup Parameters List	3 -16,00	90,86	-161,64		
	4 -14,00	79,51	-141,44		
<u></u>	5 -12,00	68,15	-121,23		
Output parameters	6 -10,00	56,79	-101,03		
C	7 -8,00	45,43	-80,82		
Constant parameters					
Ready			< Back Next>		

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				
These settings cannot be edited while a project is opened.				
OK Cancel Help				

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

🖮 塔 Advanced Input



The following dialog boxes appear:

User blocks 🔀				
User library	Steel frame C\Documents and Settings\Astrid.SCIA-ONLINE\My Documents\ESA80\userblocks\steel frame.ESA OK Cancel			



Im	mport user block 🛛 🔀					
	Setup Material					
	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:

Options				
Environment Templates Directories Other Protection Code				
Program directories				
User Templates				
Directories 🔤 🗙 🛊 🗣				
C:\Documents and Settings\Astrid.SCIA-ONLINE\My Documents\ESA80\template				
Notice				
These settings cannot be edited while a project is opened.				
OK Cancel Help				

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

Select New Project				
New Project User templates				
C\Documents and Settings\Astrid.	beam with practical rei	ODA oda	Steel frame	SA
,				
	ОК	Cancel		

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



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:

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	
化中间层间中间层间中间	Subtracts the given numbers / parameters	
*	Multiplies the given numbers / parameters	
1	Divides the given numbers / parameters	
	Modulo – gives the remainder after division of two numbers	
Λ	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 $ imes$	
tan(x)	Calculates the tangent of parameter $ imes$	
tg(x)		
arcsin(x)	Calculates the arcsine of parameter ×	
asin(x)		
arccos(x)	Calculates the arccosine of parameter x	
acos(x)		
arctan(x)	Calculates the arctangent of parameter \times	
arctg(x)		
atan(x)		
atg(x)		
ln(x)	Calculates the natural logarithm of ×.	
log(x)	Calculates log ₁₀ (x).	
exp(x)	Calculates the exponential e to the x-th power.	
sign(x)	Returns the sign of parameter x. Returns +1 for positive argument. Returns -1 for negative argument.	
sgn(x)		
sqrt(x)	Calculates the positive square root of parameter x.	

Possible operators in Scia Engineer:



Example of a formula:



Name	H_BL
Type	Length
Description	Höhe Borlochprofil
Evaluation	Formula
Fomula	(W_S==1)*HW+(W_S==0)*(HW-100)
Value [m]	-00,000
Use range	