# NEMETSCHEK Scia



**Tutorial** Parametric input Scia Engineer 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. Scia is not responsible for any direct or indirect damage because of imperfections in the documentation and/or the software.

© Copyright 2008 Scia. All rights reserved.

## **Table of Contents**

General Information	5
Welcome	5
Scia Engineer Support	5
Website	6
Project management	7
Save, Save as, Close and open	7
Introduction	8
Parametric project templates	8
Requirements for parametric templates	8
Storage place of project templates	9
Using of project templates	9
Example #1: 3-span beam	11
Definition of the structure	11
Definition of parameters	12
Assigning parameters to the structure	14
Template dialogue	15
Saving the project as a template	18
Creating of a new project based on the parametric Project template	19
Example #2: L-shaped slab	21
Definition of the structure	
Definition of parameters	
Assigning parameters to the structure	
Template dialogue	25

Tutorial Parametric Input

# **General Information**

## Welcome

Welcome to the Scia Engineer Tutorial Shell. Scia Engineer is a design program under Windows with a broad application field: from checking simple frames to the advanced design of complex projects in steel, concrete, timber,...

The program treats the calculation of 2D/3D frameworks, profile check and check of connections for steel structures included. Besides frames, it is also possible to dimension plate structures, inclusive of advanced concrete calculations.

The complete process of calculation and design has been integrated in one program: input of the geometry, input of the calculation model (loads, supports, ...), linear and non-linear calculation, output of results, member check and optimization according to various codes, generating the calculation note, ...

Scia Engineer is available in three different editions:

#### License version

The license version of Scia Engineer is secured with a 'dongle', a code lock, which you apply to the parallel or USB gate of your computer or a softwarematic license on your network.

Scia Engineer is modular and consists of various modules. The user chooses from the available modules and composes a custom design program, perfectly tuned to his needs.

In the general product overview of Scia Engineer you will find an overview of the different modules that are available.

#### Demo version

If the program doesn't find a protection, it will automatically start the demo version. The properties of the demo version are: All projects can be inserted;

The calculation is restricted to projects with 25 elements, 3 plates/shells and two load cases;

The output contains a watermark "Unlicensed software";

The projects that are stored in the demo version cannot be opened in a license version.

#### Student version

The student version has the same possibilities as the license version for all modules. This version is also secured by a 'dongle' or a softwarematic protection.

The output contains a watermark "Student version".

Projects that are stored in the student version cannot be opened in the license version.

### **Scia Engineer Support**

You can contact the Scia Engineer support service

#### By e-mail

Send an e-mail to support@Scia.be with a description of the problem and the concerning \*.esa file, and mention the number of the version you are currently working with.

By telephone

From Belgium : +32 13 350310

From the Netherlands: +31 26 3201230

Via the Scia Support website

http://www.Scia-online.com/en/online-support.html

## Website

### Link to Tutorials

http://www.Scia-online.com > Support & Downloads > Free Downloads > input e-mail address > Scia Engineer > Scia Engineer Manuals & Tutorials

#### Link to eLearning

http://www.Scia-online.com > Support & Downloads > eLearning

Link to Demo version

http://www.Scia-online.com > Support & Downloads > Secured Downloads > input username and password > Service Packs > Scia Engineer > Setup - Scia Engineer

#### Start writing on an odd page.

End each chapter using Insert > Section break – next page and type the new chapter name to the odd-page heading. Keep separate headings for each section.

# Project management

## Save, Save as, Close and open

Before entering the construction, we first discuss how to save a project, how to open an existing project and how to close a project. When running a project of this Tutorial, the project can be saved at any time. That way you can leave the program at any time and resume the project from there afterwards.

### Saving a project

Click on in the toolbar.

If a project has not yet been saved, the dialog box Save as appears. Click on the arrow in the list Save to choose the drive you want to save your project in. Select the file in which you want to put the project and click on [Open]. Select the subfolders. Enter the file name in File name and click on [Save] to save the project.



twice, the project is automatically stored with the same name. If you choose File > Save as in the main If you press menu, you can enter a new/other drive, folder and name for the project file.

### **Closing a project**

To close a project, choose File > Close in the main menu.

A dialog box appears asking if you want to save the project. Depending on your choice, the project is saved and the active dialog is closed.

### **Opening a project**

Click on

to open an existing project.

A list with projects appears. Select the desired project and click [OK] (or double-click on the project to open it).

# Introduction

Example of this Tutorial can be designed with the Licensed or Student versions.

Before you proceed, you must be familiar with your operating system: for instance working with dialogues, menu bars, toolbars, status bars, handling the mouse, etc. Basic knowledge of input and editing of the structure in Scia Engineer is needed too. You should know how to input beams and slabs, loads and usage of the property window in Scia Engineer.

This Tutorial describes the procedure how to parameterize Scia Engineer projects and make parametric user blocks usable directly in Scia Engineer or in Scia ODA user-environment.

First, we will explain how to parameterize continuous beam including loads and how to use this parametric user block. Second sample project is focused to a structure including slabs.

## Parametric project templates

The program Scia Engineer is based on templates. It defines more types of templates:

- · document templates: a list of tables and pictures included to the document
- document table templates: for each type of the table in the document can be defined
- · one or more templates for further usage in other documents and projects,
- document page style templates
- print templates
- project templates: whole project including settings of design codes,
- parametric project templates: enhancement of standard project templates by parameters.

## **Requirements for parametric templates**

If the user wants to create parametric templates or user blocks then he or she has to buy module **ESA.11 Parametric Modelling**.

Then there will be an item called "**Parameters**" in the Project data dialogue on the tab Functionality. This functionality must be switched on if we want to parameterize the structure.

It is possible to load the non-parametric structure (standard Scia Engineer project) and add parameters later.

Dynamics	Steel	
Initial stress	Fire resistance	0
Subsoil	Connection modeller	C
Nonlinearity	Frame rigid connections	E
Stability	Frame pinned connections	[
Climatic loads	Grid pinned connections	E
Prestressing	Bolted diagonal connections	C
Pipelines	Expert system	E
Structural model	Connection monodrawings	1
Parameters	Scaffolding	1
Mobile loads	LTB 2nd Order	[
Overview drawings	ArcelorMittal	E
LTA - load cases		
External application checks		

## Storage place of project templates

All user defined project templates are stored in special folders. They can define one or more folders on local or network harddrives. It allows sharing of project templates among more users in a company.

Environment   Templates Directories   Other   Protection   Code   Program directories	
Show directories for: User Templates Directories	 <b>→</b>
C:\Documents and Settings\Frederik.SCIA-ONLINE\ESA90\Templates\	
Notice	
Notice These settings cannot be edited while a project is opened.	

## Using of project templates

Standard and parametric project templates can be used in Scia Engineer or in Scia ODA user environment:

- Scia Engineer allows to input, edit and analyze structures created on basis of the template.
- Scia ODA user environment is designed for inputting of parametric project templates and analyzing those templates. It means that general editing of a model is not allowed. On the other hand templates designed for Scia ODA must define document and/or pictures in the picture gallery.

This tutorial is focused on Scia Engineer.

- 1. Run Scia Engineer
- 2. Select New project from Scia Engineer File menu, the standard New project dialogue is displayed

elect New P	2 - 2 - 2 - 3 - 3	ates		
Structure	LTA	Kree Form Modeller	MWell	Modeller
Empty Scia E	ngineer pr	oject.		
			OK	< Cancel

- 3. If some project templates are stored in corresponding folder(s) the User templates tab appears. Switch to this tab.
- 4. Select folder Sample Projects

New Project User templates	3 span beam L-shaped slab	-
C.\Documents and Settings\Free	derik.SCIA-ONLINE\ESA90\Templates\sample project\3 span beam.ESA	_
<u></u>	OK Cancel	

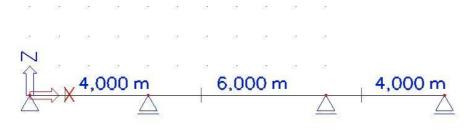
- 5. Select a template and press button <OK>
- 6. Template dialogue with parameters is displayed
- 7. Fill in parameters and press button <OK>
- 8. A standard project based on the selected template is created. The user can edit the project by the **Template dialogue** (in the **Main menu**) or by standard Scia Engineer functions.

## Example #1: 3-span beam

The goal of this sample project is to create a project template for parametric input of a three span concrete beam loaded by self weight and standard permanent load. The project template can be use for easy input of parameterized structure directly in Scia Engineer or in Scia ODA user environment.

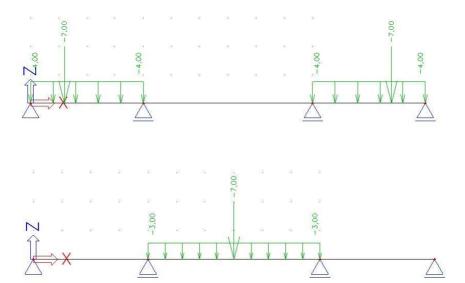
## **Definition of the structure**

1. Create a simple 3 span concrete beam by standard Scia Engineer functions in a **Frame XY**. For parametric input, it is much easier to place the first node in the origin of coordinate system. For the purpose of this Tutorial, the material quality and cross-section used are not important. The geometry is shown in the picture below.



- 2. The next step is to define loads in 3 load cases:
  - LC 1: self weight all beams are automatically loaded by self weight
  - LC 2: permanent, standard load: line and point load on 1st and 3rd spans. The line load has a value of -4kN/m. The point loads have a value of -7kN and are placed relatively on the outside beams respectively on 30% and 70%.
  - LC 3: permanent, standard load: line and point load on 2nd span. The line load has a value of -3kN/m. The point load has value of -7kN and is placed in the middle of the middle beam.





LC2:

LC3:

# **Definition of parameters**

We can continue by defining the required parameters. The **Parameter Menu** can be accessed by clicking on **Main Menu** → **Tools** → **Parameters** in **Tree window**.

### Geometry:

Name	Туре	Description	Evaluation	Formula	Value
length_1	length	Length of 1 <sup>st</sup> span	value		4 m
length_2	length	Length of 2 <sup>nd</sup> span	value		6 m
length_3	length	Length of 3 <sup>rd</sup> span	value		4 m
pos_2	length	position of 2 <sup>nd</sup> span	formula	length_1+length_2	
pos_3	length	position of 3rd span	formula	pos_2+length_3	

### **Cross-section:**

Name	Туре	Description	Evaluation	Value
width_1	css length	width of 1 <sup>st</sup> cross-section	value	500 mm
width_2	css length	width of 2 <sup>nd</sup> cross-section	value	500 mm
width_3	css length	width of 3 <sup>rd</sup> cross-section	value	500 mm
heigth_1	height	height of 1 <sup>st</sup> cross-section	value	500 mm
height_2	height	height of 1 <sup>st</sup> cross-section	value	500 mm
height_3	height	height of 1 <sup>st</sup> cross-section	value	500 mm

#### Loads:

Name	Туре	Description	Evaluation	Value
I_load_1	line load	line load on 1 <sup>st</sup> span	value	-4 kN/m
I_load_2	line load	line load on 2 <sup>nd</sup> span	value	-3 kN/m
I_load_3	line load	line load on 3 <sup>rd</sup> span	value	-4 kN/m
f_load_1	force	force load on outer beam	value	-7 kN
f_load_2	force	force load on inner beam	value	-7 kN
f_pos_1	relative	position of force on outer beams	value	0.3
f_pos_2	relative	position of force on inner beam	value	0.5

Parameters			D	<
🔎 🛃 🖬 🗽 All		• 7		
length_1 - Length of 1st s	Name	pos_3		_
length_2 - Length of 2nd	Туре	Length		•
length_3 - Length of 3rd	Description	positiond of	3rd span	
pos_2 - positiond of 2nd s	Evaluation	Formula		•
pos_3 - positiond of 3rd s width 1 - width of 1st cro	Formula	pos_2 + leng	gth_3	
width 2 - width of 1st cro	Value [m]	14,00		
width 3 - width of 3rd cro	Use range			
Lload_3 - line load on 3rd f_load_1 - force load on o f_load_2 - force load on in f_pos_1 - position of forc f_pos_2 - position of forc	Actions			
	Validate		>>>	
New Edit Delete	- Vanadio		Close	

Note: Formula type parameters are calculated from other parameters and are for internal use (they are not usable in parametric input dialog). For example in this case formulas are used for definition of end node position of second and third beams.

After defining all necessary parameters, click <CLOSE>.

Scia Engineer now checks the correctness of the use formulas. Click  ${<\!{\rm YES}\!>}.$ 

Parameters			×
🔊 🗶 🖬 🗽 All		• 🕅	
length_1 - Length of 1st s	Name	pos_3	
length_2 - Length of 2nd	Туре	Length	-
length_3 - Length of 3rd	Description	positiond of 3rd span	
pos_2 - positiond of 2nd s	Evaluation	Formula	-
pos_3 - positiond of 3rd s width 1 - width of 1st cro	Formula	pos_2 + length_3	
width 2 - width of 1st cro	Value [m]	14,00	
width 3 - width of 3rd cro	Use range		
height_1 - height of 1st cr			
height_2 - height of 1st cr			
height_3 - height of 3rd c			
l_load_1 - line load on 1st l load_2 - line load on 2nd			
I load 3 - line load on 3rd			
Elevel & Connection		100	
f_load_2 - force load	ngineer		
f_pos_1 - position of			
f_pos_2 - position of 📿	Would you like to	check the correctness of the formulae?	
	Yes	No	
	103		
3	Actions		
	Validate		>>>
New Edit Delete			Close
			Ciose

## Assigning parameters to the structure

1. Select the end node of the first span

æ	84	Ω.	82	æ	23	œ	81	æ	82	15		
N	2	8	2	N	ode N2 (4,00	0:0,000)	2	8	2	55		
Ł	X										 	

- Select item GCS coordinates > Coord X [m] in the Property window. If appropriate parameters are defined then the edit box is replaced by a combo box. This means, you can now input not only a real value, but also a defined parameter.
- 3. Change the value to "length\_1" parameter (using the little down-arrow button).

N	ode (1)	• Va V /			
	Name	N2			
Ξ	GCS coordinate				
	Coord X [m]	length_1 - Length of 1st sp: 👻			
	Coord Z [m]	Insert value			
Ξ	UCS coordinate	length_1 - Length of 1st span			
	Coord ux [m]	length_2 - Length of 2nd spar length_3 - Length of 3rd span			
	Coord uz [m]	pos_2 - positiond of 2nd span			
Ξ	Members	pos 3 - positiond of 3rd spa			
	Member	81			
	Member	B2			
Ð	Data				
	Support in node	Sn1			

- 4. Change the X co-ordinates of end nodes for the second and third span in the same way (node N3 and N4).
- 5. Change their values to "pos\_2" and "pos\_3" respectively.

Properties # ×			Properties					
N	ode (1)	✓ \u0 \u2	N	lode (1)	▼ ¼ V Ø			
8	Name GCS coordinate	N3		Name GCS coordinate	N4			
	Coord X [m] Coord Z [m]	pos_2 - positiond of 2nd sr 💙		Coord X [m] Coord Z [m]	pos_3 - positiond of 3rd sp 👻			
Ξ	UCS coordinate	length_1 - Length of 1st span	Ξ	UCS coordinate	length_1 - Length of 1st span			
	Coord ux [m]	length_2 - Length of 2nd span length 3 - Length of 3rd span		Coord ux [m]	length_2 - Length of 2nd span length_3 - Length of 3rd span			
	Coord uz [m]	pos 2 - positiond of 2nd span		Coord uz [m]	pos_2 - positiond of 2nd span			
Ξ	Members	pos_3 - positiond of 3rd span		Members	pos_3 - positiond of 3rd span			
	Member	82		Member	B3			
	Member	83	Ξ	Data				
Ð	Data			Support in node	Sn3			
	Support in node	Sn2						

- 6. The next step is to assign parameters to loads.
  - Adjust the second load case to be displayed in the window.
  - Select the first load and let's focus on the Property window again.

95 1	Ħ	Ħ	95	*	Line for	ce on be	am LF1 i	Force : -	4,00 / -4	.00 [kN/m]	
N8	7,00	×	×	8	1	ж	×	ж	96	. 00	7,00
1	*	Ħ	Ħ	Ť		×	96	96	×	-4,0	-4,0
<b>F</b>	D)	(11	TT	Ţ						1 + + +	+++++++

Change the load value to parameter "I\_load\_1".

	Name	LF1
	Direction	Z
	Туре	Force
	Angle [deg]	
	Distribution	Uniform
	Value - P [kN/m]	l_load_1 - line load on 1st span 🚿
	Load above joint	Insert value
	Member	I_load_1 - line load on 1st span
	Load case	I_load_2 - line load on 2nd span
Ŧ	Geometry	l_load_3 - line load on 3rd span
Ŧ	Eccentricity	

- Adjust value of the forces and other line load one by one similarly.
- 7. Do the same change with the load in the last load case.

## **Template dialogue**

The user can define a template dialogue for easier editing of input parameters. Defined parameters can be arranged to sets (represented by tabs in the dialogue) and to groups (represented by trees).

- 1. Open the **Parameter template settings** in the **Tools menu**. Let's define three parameter sets according to the pictures below.
- 2. Select parameters in the **Available parameters** window and move them to the **Selected parameters** window by pressing **Add selected** button.

Edit parameter set						
Parameter set name: Set description:	Beams Input length of beams	Use 3D pro	eview			
Available para	meters		s	elected	parameters	
Name unit typ			Name	unit	type	
f_load_1 f_load_2		Add selected	length_1 length_2			
f_pos_1 f_pos_2		Add all	length_3			
height_1 height_2		Remove all				
height_3 I_load_1		Remove				
Load_2		Insert group				
width_1 width_2 width_3		Remove group				
widu_3		Rename group				
				OK	Ca	ncel

• First tab:

Click <OK> to save this set.

Click <NEW> to add a second list (see below).

• Second tab:

Parame	ter set na	me: Cross section	Use 3D p	review			
Se	t descript	ion: Input width and	height of the cross-section				
	wailable	parameters	_		Selected	parameters	
ame	unit	type		Name	unit	type	
oad_1 oad_2			Add selected	height_1 height_2			
oos_1 oos_2			Add all	height_3 width_1			
oad_1 oad_2			Remove all	width_2 width_3			
oad_3			Remove				
			Insert group				
			Remove group				
			Rename group				

• Third tab:

i aramei	er set nam	ne: Loads	Use 3D p	review			
Set	descriptio	on: Input loads, for	ces and positions of loads				
A	wailable p	parameters		S	Selected	parameters	
Vame	unit	type		Name	unit	type	
			Add selected	I_load_1 I_load_2			
			Add all	l_load_3 f_load_1			
			Remove all	f_load_2 f_pos_1			
			Remove	f_pos_2			
			Insert group				
			Remove group	1			
			Rename group				

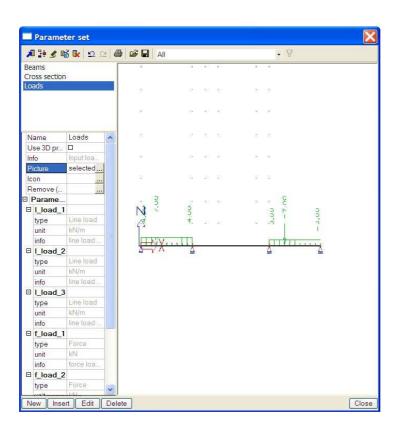
3. The Parameter set window gives the 3 parameter sets: Beams, Cross section and Loads

B			
B	0 🤮 🗶 📸 🗽 🕮	8 6 8	A
	eams		
10	ross section pads		
	Ddus		
N	Vame	Cro	^
	Jse 3D preview		
1.0	nfo	Inpu	
F	Picture		
وتنا	con		
	Parameters		
E	height_1		
	type	Css	
	unit	กากา	
	info	hei	
E	height_2		
	type	Css	
	unit	mm	
	info	hei	
E	height_3		
	type	Css	
	unit	mm	
	info	hei	
E	width_1	Cor	v
	lew Insert Edit	Delete	-

- 4. The next step is to prepare a picture to accompany the first set.
  - Close the manager and adjust
  - Set a standard "View Y" view in the graphical window.
  - Use function File > Print picture > Save picture to file to save the drawing into an external BMP file.

Note: In this case we made just one picture to accompany with all three sets. Character of the structure is rather longitudinal so all three pictures in one will better fit window.

- 5. Alternatively pictures may be easily composed in Picture gallery (see User's guide for further information), and/or imported from any external picture editor.
- 6. Open the **Parameter sets manager** (**Tools > Parameters template settings**) and **edit** the Geometry set (or Loads set or Cross section set).
- 7. Assign the picture to the tab using button [Picture].
- 8. At this moment the project is prepared to be saved as parameterized template.



### Saving the project as a template

All Scia Engineer projects can be used as templates. When the user wants to use a standard or parameterized project as a template then he has to copy it to a folder which is defined in the **Options** dialogue on the tab **Directories** (**Setup > Options > Directories**):

1. Save the project to standard Engineer file to a standard folder defined for Engineer files

Options	
Environment Templates Directories Other Protection Code	1
Program directories Show directories for:	
User Templates	-
Directories	
C:\Documents and Settings\Frederik.SCIA-ONLINE\ESA90\Templates\	
Notice These settings cannot be edited while a project is opened.	
OK Cancel He	lp

2. Copy the project to the folder defined for User Templates.

- 3. Restart Scia Engineer
- 4. Alternatively it is possible to save the file directly to the folder defined for **User Templates**.

### Creating of a new project based on the parametric Project template

Using of parametric templates is very easy.

1. Open a new project:

٠

- Call function **File > New**,
  - select a tab User templates,

Select New Project							
New Project User templates							
Connect	3 span beam						
C:\Documents and Settings\Frederik.SCIA-ONLINE\ESA90\Templates\sample project\3 span beam.ESA							
	OK Cancel						

- select a project,
- press OK button.
- 2. Go through individual tabs and fill in the parameters

ams Cross section Loads Project settings		
Lload_1 - line load on 1st span [kN/4.00	Sample picture	
I_load_2 - line load on 2nd span [kN3.00		
Lload_3 - line load on 3rd span [kN/4.00		
f_load_1 - force load on outer beam		
f_load_2 - force load on oner beam7.00		
f pos_1 - position of force on outer b 0.3		
f_pos_2 - position of force on inner b 0,5		
-hos_s, hosmon on once of title p  0,5		
	NI STATE	
	Terstand Terstand	
	- Description -	
	Description Input loads, forces and positions of loads	
2   <b>1</b>		
<b>*</b>		

Once the parameters have been defined (notice that only few numbers had to be inserted), the project is opened and a new 3-span continuous beam is automatically created in front of you.

The project template can pre-define also document, pictures in the picture gallery and in the paper space gallery and default values for design on members.

### Editing of a parametric project

If the template dialogue for the parametric project is created then a user can edit parameters by it. It means that all parameters are kept in the project and they are not replaced by actual values. This feature brings possibility to edit parameters by the template dialogue anytime.

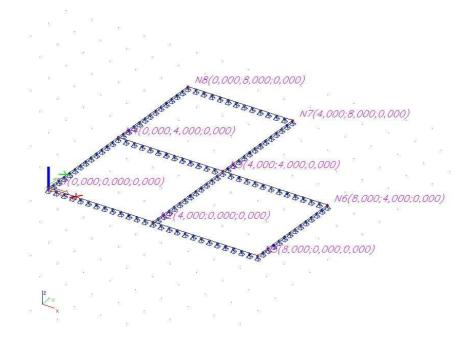
# Example #2: L-shaped slab

The goal of this sample project is to create project template for parametric input of L-shaped slab. The slab is loaded by self weight and surface load. Also the surface load is parameterized.

Storing, opening and editing of the parametric template is the same as in the 1<sup>st</sup> example.

## **Definition of the structure**

1. Create a simple L shaped slab. Coordinates of the nodal points are shown in the picture below.

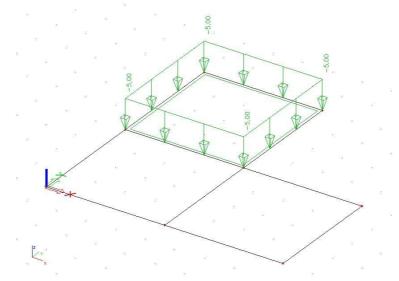


Note: As in the previous example we place the first node to the origin of coordinate system.

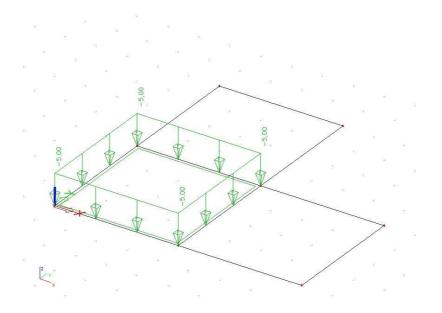
2. Define loads in 4 load cases:

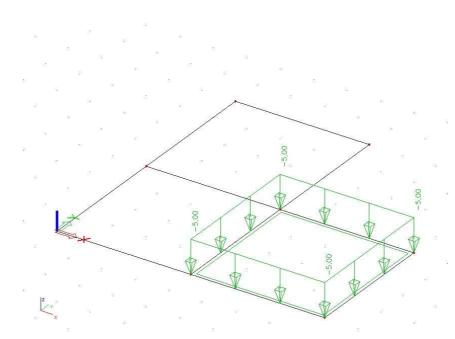
- LC 1: self weight
- LC 2: variable, standard
- LC 3: variable, standard
- LC 4: variable, standard











## **Definition of parameters**

Define parameters: use function  $\ensuremath{\text{Tools}}\xspace \ensuremath{\text{->}}\xspace \ensuremath{\text{Parameters}}\xspace$  in the  $\ensuremath{\text{Main menu:}}\xspace$ 

### **Geometry parameters:**

Name	Туре	Description	Evaluation	Formula	Value
x_len_1	length	X Length of 1 <sup>st</sup> span	value		4 m
x_len_2	length	X Length of 2 <sup>nd</sup> span	value		4 m
y_len_1	length	Y Length of 1 <sup>st</sup> span	value		4 m
y_len_2	length	Y Length of 2 <sup>nd</sup> span	value		4 m
x_pos_2	length	X end position of 2 <sup>nd</sup> span	formula	x_len_1 + x_len_2	
y_pos_2	length	Y end position of 2 <sup>nd</sup> span	formula	y_len_1 + y_len_2	
thick_1	css length	thickness of 1 <sup>st</sup> slab	value		200 mm
thick_2	css length	thickness of 2 <sup>nd</sup> slab	value		200 mm
thick_3	css length	thickness of 3rd slab	value		200 mm

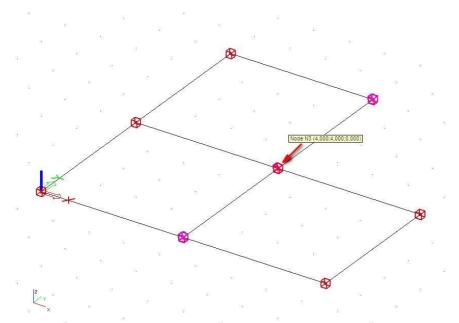
### Loads:

Name	Туре	Type Description Evaluation		Formula	Value
load_1	surface load	surface load on 1 <sup>st</sup> slab	value		-5 kN/m²
load_2	surface load	surface load on 2 <sup>nd</sup> slab	value		-5 kN/m²
load_3	surface load	surface load on 3 <sup>rd</sup> slab	value		-5 kN/m²

All 🖌 🖬 🗽		• 17	
x_len_1 - x length of 1st	Name	x_pos_2	
x_len_2 - x length of 2nd	Туре	Length	
y_len_1 - y length of 1st	Description	x end of 2nd span	
y_len_2 - y length of 2nd		Formula	
x_pos_2 - x end of 2nd s	Formula	x_len_1 + x_len_2	
y_pos_2 - y end of 2nd s thick 1 - thickness of 1st	Value [m]	8:00	
thick 2 - thickness of 2nd	Use range		
thick_3 - thickness of 3rd load_1 - surface load on 1 load_2 - surface load on 2 load_3 - surface load on 3		U.	
	Actions		
	Validate		>>>

## Assigning parameters to the structure

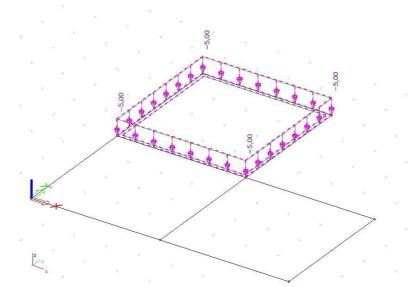
1. Select nodes which have the same X coordinates



2. Select item GCS coordinates > Coord X [m] and change its content to x\_len\_1, in the Property window (using the little down-arrow button)

N	ode (3)	
	GCS coordinate	
	Coord X [m]	x_len_1 - x length of 1st spa 🔻
	Coord Y [m]	Insert value
Ξ	Coord Z [m]	x_len_1 - x length of 1st span
	UCS coordinate	x_len_2 - x length of 2nd span
	Coord ux [m]	y_len_1 - y length of 1st span y len 2 - y length of 2nd spar
	Coord uy [m]	x_pos_2 - x end of 2nd span
	Coord uz [m]	y_pos_2 - y end of 2nd span
Ξ	2D members	

- 3. Change the X co-ordinates of nodes on the right side of the picture for coordinate x\_pos2
- 4. Do the same changes for Y direction and use parameters **y\_len\_1** and **y\_pos2**
- 5. Adjust the second load case to be displayed in the window. Select the surface load and change the load value to parameter load\_1



1
st sla 🕚
it slab
id slab
d slab

6. Make the same changes with the load in the next load cases

## **Template dialogue**

The user can define a template dialogue for easier editing of input parameters. Defined parameters can be arranged to sets (represented by tabs in the dialogue) and to groups (represented by trees).

- 1. Open the Parameter set manager (Main menu > Tools > Parameters template settings). Let's define three parameter sets according to the pictures below.
- 2. Select parameters in the Available parameters window and move them to the Selected parameters window by pressing the Add selected button.

#### First set:

•

Edit param	eter set	ł							
Parame	eter set na	me:	Geometry		Use 3D p	re∨iew			
Se	et descript	ion:	Input of geome	try parameters					
	Available	para	meters				Selected	parameters	
Name	unit	typ	e			Name	unit	type	
load_1 load_2				Add	Iselected	x_len_1 x_len_2			
load_3 thick_1					Add all	y_len_1 y_len_2			
thick_2 thick_3				Re	move all				
				P	emove				
				Ins	ert group				
				Rem	iove group				
				Ren	ame group				
1									
							OK	c	ancel

Second set:

٠

Edit parameter set						X
Parameter set name:	Thickness	Use 3D p	review			
Set description:	Input thickness of the	slabs				
Available para	ameters		S	Selected	paramete	rs
Name unit typ	pe		Name	unit	type	
load_1 load_2		Add selected	thick_1 thick_2			
load_3		Add all	thick_3			
	1	Remove all				
		Remove				
		Insert group				
		Remove group				
	I	Rename group				
				OK		Cancel

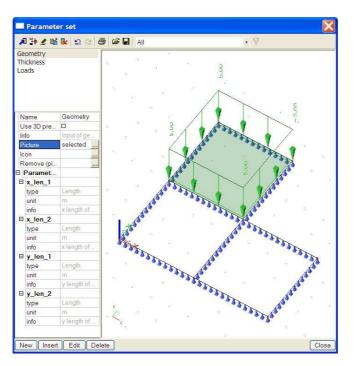
#### Third set:

•

Parameter set name: Loads	Use 3D	preview			
Set description: Input of sur	ace loads				
Available parameters			Selected	paramete	ire
Jame unit type		Name	unit	type	
	Add selected	load_1 load_2			
	Add all	load_3			
	Remove all				
	Remove				
	Insert group				
	Remove group				
	Rename group				
		l.			

- 3. The next step is to prepare a picture to accompany the first set.
  - Close the Parameter template settings
  - Set a standard AXO view in the graphical window
  - Use function File > Print picture > Save picture to file to save the drawing into an external BMP file
  - Alternatively pictures can be easily composed in Picture gallery (see User's guide for further information), and/or imported from any external picture editor.
- 4. Open the Parameter template settings
- 5. Select the Geometry
- 6. Assign the picture to the tab using button [Picture]





At this moment the project is prepared to be saved as parameterized template