

**Tutorial** Plate Concrete 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	1
Welcome	1
Scia Engineer Support	1
Website	2
Introduction	3
Getting started	4
Starting a project	4
Project management	7
Save, Save as, Close and open	7
Saving a project	7
Closing a project	7
Opening a project	
Geometry input	8
Input of the geometry	8
Geometry	8
Supports	13
Check Structure data	15
Checking the structure	
Connecting entities	15
Graphic representation of the structure	19
Input of the Calculation Data	23
Load Cases and Load Groups	23
Defining a Permanent Load Case	23
Defining a Variable Load Case	
Loads	25
Combinations	32
Calculation and Mesh generation	34
Mesh generation	34
Linear Calculation	36
Results	
Viewing results	37
Reinforcement design	
Changing the diameter of the bars	42
Required areas	43
Input user reinforcement	48
Document	52
Formatting the Document	52
Displaying results in the document	53
Adding an image to the document	54
Epilogue	57

# **General Information**

## Welcome

Welcome to the Scia Engineer Tutorial Plate Concrete. 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.

# Introduction

The 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.

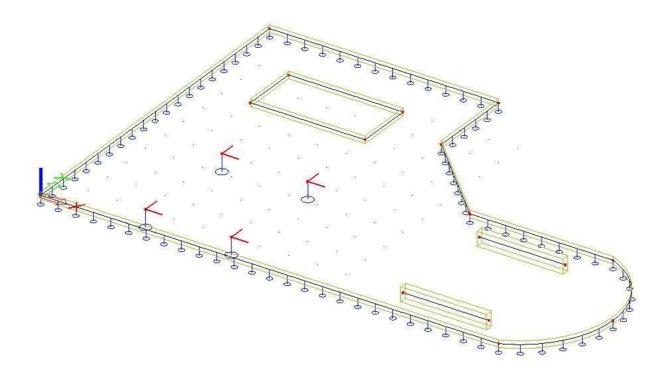
This Tutorial describes the main functions of Scia Engineer for the input and calculation of a plate.

First, we will explain how to create a new project and the set-up of the structure. After the geometry and load input, the structure will be calculated and the results can be viewed.

Next, we will discuss the input of the buckling parameters and we will perform the steel check, the profile optimization and calculate a connection.

The Tutorial ends with a brief introduction to the calculation note.

The figure below shows the calculation model of the structure to be designed:



# **Getting started**

# Starting a project

### Starting the program

Before you can start a project, you need to start the program first.

1. Double-click on the Scia Engineer shortcut in the Windows Desktop.

Or :

2. If the shortcut is not installed, click [Start] and choose Programs > Scia Engineer 2009.0 > Scia Engineer 2009.0.

If the program does not find any protection, you will obtain a dialogue indicating that no protection was found. A second dialogue will list the restrictions of the demo version. Click **[OK]** in both windows.

For this Tutorial, you must start a new project.

### Starting a new project

- 1. If the dialogue Open appears, click [Cancel].
- 2. Click the **New** icon in the toolbar.

elect New P New Project S		mplates		
		<i>*</i>	Ð	
Structure	LTA	Free Form Modeller	MWell	Modeller
, Empty Scia E	ngineer pr	oject.		
			ОК	Cancel

In the **Select New Project** dialogue, choose for the **Structure** environment by clicking on the corresponding icon. Confirm your choice by clicking 'OK'.

Now, the Project data dialogue is opened. Here, you can enter general data about the project.

	Data				Structure :	
C.W.S.					Plate XY	<u> </u>
100					Material:	
Reference in	Name	Plate			Concrete	⊠
-110	Part	-			Material	C30/37
	Pan	1-			Reinforcement mat	
	Description	Tutorial Plate	Concrete		Steel	
80.5		Tratenari			Timber	
Net I	Author	ND			Other	
	Date	06.08.2009			Aluminium	
Car						
	Project Level :		Model:			
	Advanced	•	One	•		
	National Code :		National annex:			
	EC-EN		EC-EN	*		

- 3. In the **Data** group, enter your preferred data. These data can be mentioned on the output, e.g. in the document and on the drawings.
- 4. Choose the Project level: Advanced and Model: One.
- 5. Click on the rectangular button below **National Code** to choose the default code for the project. This code will determine the available materials, combination rules and code checks. For the project of this Tutorial, choose EC-EN. The window **Codes in project** is opened.
  - Click [Add].
  - The dialogue Available national codes is opened.

vailable na	tional cod	es		•••				Σ
AISC ASD	America (LRFD)	BS	CSN	DIN	EC-ENV	France	NEN	ONORM
SIA 26x	* Slovakia	) India	spain	IBC				
AISCASD								OK Cancel

- Select the European flag and click [OK].
- You will return to the Codes in project dialogue and EC-EN is added.
- Select the flag named EC-EN.
- Select the Active code option and click [Close].
- You will return to the Project data window and EC-EN is the active code.
- 6. If you have chosen for the EC-EN, you will have the possibility to choose for a National Annex for an European country.
- Select Plate XY in the Structure field.
   The structure type (Frame XZ, Frame XYZ, Plate XY, General XYZ,...) will restrict the input possibilities during the calculation.
- In the Material group, select Concrete.
   Below the item Concrete, a new item Material will appear.
- 9. Choose C30/37 from the menu.
- 10. Confirm your input with [OK].

#### Notes:

On the **Basic data** tab, you can set a project level. If you choose "standard", the program will only show the most frequently used basic functions. If you choose "advanced", all basic functions will be shown.

On the **Functionality** tab, you choose the options you need. The non-selected functionalities will be filtered from the menus, thus simplifying the program.

On the **Combinations** tab, you will find the values for the partial safety factors. In this Tutorial, we will use the default settings.

# 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.



If you press twice, the project is automatically stored with the same name. If you choose **File > Save as** in the main 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).

# **Geometry input**

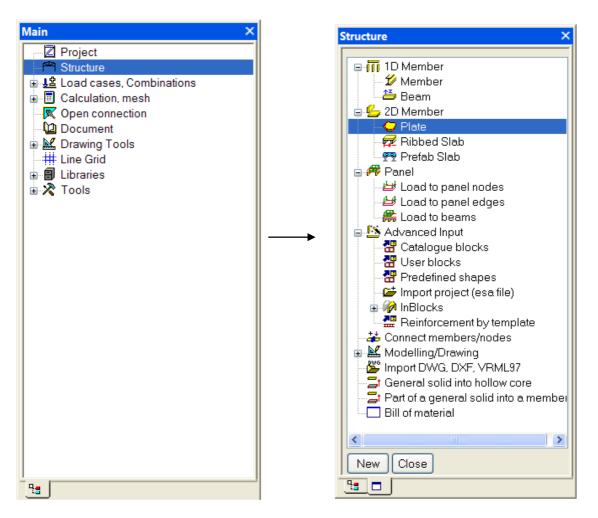
## Input of the geometry

If you start a new project, the geometry of the structure must be entered. The structure can be entered directly, but you can also use for instance templates with parametric blocks, DXF files, DWG files and other formats.

### Geometry

#### Structure menu

1. When a new project is started, the **Structure menu** is automatically opened in the **Main window**. If you want to modify the structure at a later time, you must double-click on **Structure** in the **Main window**.



2. In the **Structure menu** different branches will appear, in accordance with the already input items, i.e. support branch will appear if a structure is physically available.

We will input the structure as a plane 2D member. We will use the advanced input options, like definition of an opening in the slab or drawing of a plate rib.

### Input of a plane 2D member

- 1. In the structure menu one should double click on the Plate in the 2D member branch
- 2. The window **2D member** will be opened.

2D member			
	Name	S1	
	Туре	plate (90)	
	Analysis model	Standard	
	Material	C30/37	•
Ja ja	FEM model	Isotropic	12
7	T Thickness type	constant	
	Thickness [mm]	250	
	LCS Type	Standard	
1 1	LCS Angle [deg]	0.00	
	Layer	Layer1	-
x			
	2 <del>1</del>	OK	

Secondly the following properties can be defined: name=Slab, type=plate (90), Material=C30/37, Thickness = 250mm.

- 3. After accepting with [OK] the program asks in the command line for the starting point of the polyline.
- 4. The buttons in the **command line** allow one to built up the polygonal edges using different line types, or to choose directly for a circular or rectangular surface.

Command line	φ ×
R COLD/CJRS7 X	「マンダメートをなく」として、シントメダント
New polygon - Start point >	

5. The geometry can be input with help of a dot or line grid or with use of the mouse or direct input of coordinates in the command line :

Starting point:

0;0 <enter> 16;0 <enter>

New polygon – Circular Arc – Intermediate point > : @2;3 <enter> @-2;3 <enter> @-5;0 <enter> @-3;3 <enter> @0;3 <enter> @-8;0 <enter>

right mouse click in order to select the command End and close the input of the polygon.

The program proposes to draw a New polygon. The right mouse click will end the drawing .

The following picutre is now depicted in the screen:

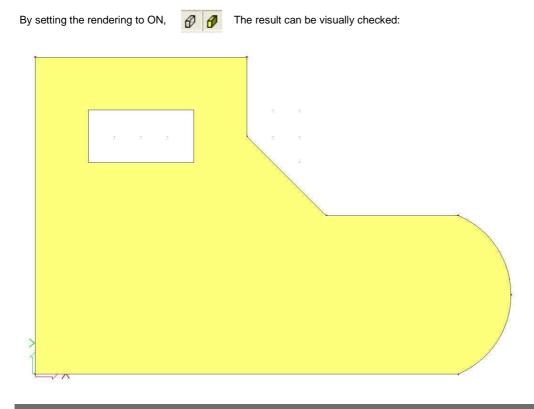
T		 			 	-				
		8	ĸ	ĸ						
		ĸ	12			1				
		s:		*		-2	1			
		8	8	8				1		
									\	 ~
										1
+										
1	⇒×	 			 					 -

### Definition of an opening

- 1. In the Structure menu under 2D member components we will create an Opening with the name Stair.
- 2. In the command line quick access buttons are available for quick definition of geometrical outlines like circular or rectangular slabs. By default definition by drawing of a closed polygon is started, in this example we will define a rectangular opening.
- 3. The two nodes of its diagonal define a rectangle. This is also depicted by the two red dots on the icon.

New rectangle Starting Point: 2;8 Endpoint : 6;10

The program displays the proposed rectangle. Accept the New Rectangle by clicking the right mouse button.



With the option Sub region a different thickness can be set using exactly the same principle.

### Input of internal nodes

Note

- 1. In the Structure menu we choose under 2D element components to input internal nodes.
- 2. We wil add four new internal nodes:
  - 3;1 <enter> 3;5 <enter> 6;5 <enter> 6;1 <enter>

right mouse click in order to end the input session.

### Input of plate ribs

- 1. In the Structure menu, under 2D member components we choose Rib.
- 2. If no cross-section was chosen in the project, the dialogue New cross-section will pop up.

Here we will be able to select and define from the **Concrete** a **rectangular crosssection** as the new rib.

New cross-section Available groups Concrete Geometric shapes Numerical General Precast Bridge	Available items of this group	Etems in project
Rectangle	Profile Library filter	Add Close

3. Click **[Add]**. This will take us to a new dialogue **Cross-section**. For this Tutorial we will take a rectangular concrete crosssection with **height 500 mm** and **width 250 mm** and a concrete grade of C30/37.

Cross-section		×
	Name CS1	
	Type Rectangle	
Z	Detailed 500; 250	
ĩ	Parameters	
	Material C30/37	·
	H [mm] 500	
	B [mm] 250	
	General	
	Draw color Normal colour	-
	Colour	
	Properties editable	
	Buckling editable	
<u>8</u> _y	Buckling y-y b	•
	Buckling z-z b	-
<b>T</b>	Fabrication concrete	-
	FEM analysis	
	Concrete	
	Curve dividing 36	
	Edit joints	
	Edit cuts	
	Edit named items	
-XY	Autodesign constraints	
	Property reduction fa	
	Use reduction factors	
B 250	Update Docu	
Picture	OK Car	cel

- 4. We will accept the cross-section by pressing the [OK] button. After this we will [Close] both dialogues.
- 5. In window Plate rib we can define the parameters of the ribs:

B1
plate rib (92)
Standard
CS1 - Rectangle (500; 250)
T symmetric
width
1000
1000
standard
Default
Layer1

6. After **<OK>** we will have to define the starting and end points of the ribs.

1st rib : Starting point : 12;5 <enter> End point : 15;5 <enter>

2nd rib : Starting point : 12;1 <enter> End point : 15;1 <enter>

right mouse click to end the input session.

### Note

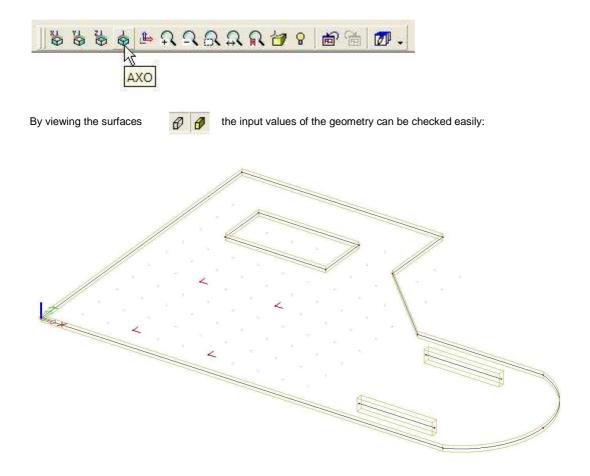
Properties	<b>I</b>
Member (1)	🗾 Va V/ 🥖
Name	B1
Type rib	plate rib (92)
Analysis model	Standard
CrossSection	CS1 - Rectangle (500; 250)
Shape of rib	T symmetric
Effective width	width 🔀
for int. forces [mm]	default
for check [mm]	number of thickness
FEM type	width
Buckling and relative le	Default 🔄 🛃
Layer	Layer1 -
2D member	S1
Geometry	
	0.000

width : The user can input the width for the internal forces (FE analysis) or the checks (Design As) by hand.

number of thickness : The width of the slab for the rib is defined as a factor times the plate thickness. The user enters the factor by hand.

default : The width of the slab for the rib is defined as a factor times the plate thickness. The factor is set in Setup > Solver > Number of thicknesses of rib plate

#### One can ask for a 3D view on the slab by the button [view in direction AXO]



By pressing **<Esc>** one can easily cancel the selection.

### **Supports**

The input of the geometry can be finalized by definition of the support conditions. We assume that the whole edge is supported in global z-direction. Thus we simulate that a masonry wall supports i.e. the slab.

### Definition of a support on an edge

- 1. Select in the Structure menu Model data -> Support > line on 2D member edge
- 2. The window Line support on 2D member edge will pop up.

	Name	Sle1	
	Z	Rigid	
	Rx	Free	
	Ry	Free	
<b>AR</b> z	Geometry		
7	System	GCS	
	Position x1	0,000	
X	Position x2	1,000	
Ry	Coord. definition	Rela	
x1 x2	Origin	From start	
Z			
x			

- 3. We will support the edge on in z-direction.
- 4. Finally we will select the edges around the slab one by one; edge1, edge2, edge3, edge4, edge5, edge6, edge7.
- 5. Press **<ESC>** to cancel the input command

### Input of Nodal supports

1. In order to input the nodal supports for the four internal nodes, we will use the option **Model data -> Support > in Node** in the **Structure menu.** 

Support in node			
	Name	Sn1	
	Туре	Standard	•
	Z	Rigid	-
	Rx	Free	•
	Ry	Free	•
	Default size [m]	0,200	
AZ /	Geometry		
Rx Ry	System	GCS	•
	3 <del>.</del>	OK	Cancel

- 2. We will support the nodes only in z-direction. This supports can for instance by columns under the slab.
- 3. We apply the nodal supports for the internal nodes N13, N14, N15 and N16

### Note

- If required a flexible support can be defined in order to model the behavior of the columns more adequately. Also only can model the supports as "column", then the stiffness is directlyderived for the entered column data.
- A set of shortcuts of supports is defined in the **Command line**. In this project the button **Hinged Support** could have been used.

## **Check Structure data**

After input of the geometry, the input can be checked for errors by means of the option **Check Structure data**. With this tool, the geometry is checked for duplicate nodes, null- bars, duplicate bars...

### **Checking the structure**

- 1. Press on **<Esc>** or click on the button **Cancel Selection** in order to take care that no entities are selected anymore.
- 2. Double-click on the Check Structure data option in the Structure Menu or click on the <sup>1</sup>/<sub>1</sub> icon in the toolbar.
- 3. The window Check of structure data will pop up for which a different set of checks is depicted.

Check of nodes	
Search duplicate nodes	Ignore parameters
Check of members	
<ul> <li>Check members</li> <li>Bearch null members</li> </ul>	Null members: 0
Search duplicate members	Duplicate members: 0 Delete duplicate members
	Invalid parts: 0 Invalid parts
Check of additional data Check additional data position	Invalid position
Check of steel connections Check steel connections	Invalid connections

- 4. Click [Check] to perform the checks.
- 5. The message box Data Check Report pops up, indicating that no problems were found.

Data checl	( report	X
Data chec	k finished. No proble	ems found.
	OK	

6. Close the check by clicking [OK].

7.

## **Connecting entities**

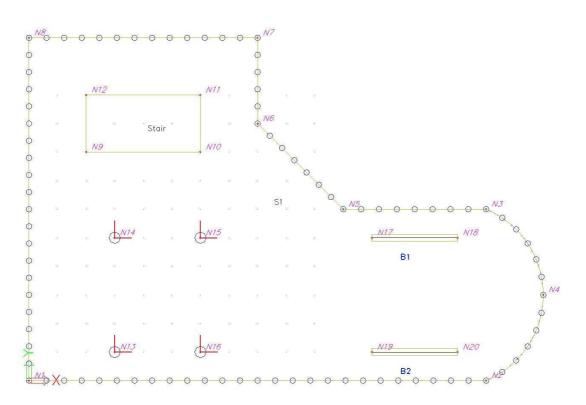
The plate ribs have to be connected to the slab. A node that is not connected to the slab, is depicted as a red dot. A node that is connected to a slab is depicted as a red dot with two straight lines :

In order to display the names of the entered bars and nodes or support symbols the labels of each item can be turned ON / OFF by the shortcut button in the lower left corner of the graphical screen  $\mathcal{O} \mathcal{O} \mathcal{A}$  and  $\mathcal{O} \mathcal{O} \mathcal{A}$  and  $\mathcal{O} \mathcal{O} \mathcal{O} \mathcal{O} \mathcal{O}$  above the **Command line**.

The 3rd button can visualize supports.

Labels of nodes can be activated by the button 💾 located above the **Command line** 

Labels of bars can be activated by the button 🕮 located above the **Command line**.



If the slab is selected by single clicking with the left mouse button on the 2D member edge, the properties of the slab can be reviewed in the **Properties window**:

20	2D member (1)		Y aV ≥
	Name	S1	
	Туре	plate (90)	¥
	Analysis model	Standard	-
	Shape	Flat	
	Material	C30/37	·
	FEM model	Isotropic	*
	Thickness type	constant	
	Thickness [mm]	250	
	LCS Type	Standard	
	LCS Angle [de	0,00	
	Layer	Layer1	·
Ξ	Nodes		
	N1	abso	
	N2	abso	
	N3	abso	
	N4	abso	
	N5	abso	
	N6	abso	
	N7	abso	
	N8	abso	

The properties contain for instance also the nodes on the outline of the slab. Additional data, like predefined line supports, internal nodes, openings and ribs will also be depicted. However it is required that the elements have to be connected using the option **Connect members/nodes**.

### **Connecting entities**

- 1. Press **<ESC>** or click the **Cancel selection** kicon to deactivate any selection of entities.
- 2. Double-click on the Model data > Connect members/nodes option in the Structure menu or click the icon in the toolbar.
- 3. A dialogue asks if all nodes must be connected to bars:

Scia Engineer		X
Do you	I want to proceed	d with all entities?
Yes	No	Cancel

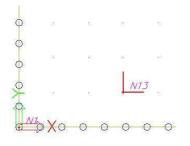
- 4. Click <Yes>.
- 5. The Setup for connection of structural entities dialogue box now appears.

E	Align structural entities to planes (moving nodes)	
	Align	
E	Geometrical tolerance	
	Min. distance of two nodes, node to curve [m]	0,001
	Max. distance of node to 2D member plane [m]	0,000
E	Connect (generate linked nodes, intersections,	
	Connect	
	Check structure data	
- Constal	Check (merge duplicate nodes, erase invalid entities)	

- 6. Confirm the settings by clicking **<OK>**.
- 7. A window appears to indicate the number of connected nodes:

Scia En	gineer 🔀
į)	4 nodes have been successfully connected to selected members.

8. Connected nodes are represented in the graphical screen by means of a red dot with two straight lines:



### Note

One can also connect the entities for a **selection** of members.

9. Click on [Close] in the bottom of the Structure menu

# **Graphic representation of the structure**

### **Edit view**

Within Scia Engineer there are several possibilities to edit the graphic representation of the construction. Below you will find the most important options:

- Edit the view point on the Construction
- Set a view direction
- Use the magnifier
- Edit view parameters through the menu View parameters

### Editing the view point on the construction

Set view point through the wheels. Bottom right of the graphic window there are three wheels; two are horizontal and one is vertical. With these **wheels** you can **zoom in** on the construction or **turn** it.

1. To be able to zoom in on the construction or to turn the structure, click on the wheel (the cursor will change into a hand), keep the left mouse button pressed and move the wheel

OR

Set the view point by combining the buttons and mouse.

- 2. Press CTRL + right mouse button at the same time and move the mouse to turn the construction.
- 3. Press SHIFT + right mouse button at the same time and move the mouse move the construction.
- 4. Press CTRL + SHIFT + right mouse button at the same time and move the mouse to **zoom in or out** on the construction.

Not

If the structure is being turned while a node is selected, the structure will turn around the selected node.

### Setting a view direction with regard to the global coordinate system

- 1. Click on the button **View in direction- X** for a view in the X-direction.
- 2. Click on the button View in direction- Y



Q

3. Click on the button View in direction- Z

for a view in the Z-direction.

for a view in the Y-direction.

### The magnifier

- Use to enlarge.
- Use Sto decrease.
- Use to zoom in on a window.
- Use to view the whole structure.
- Use R to zoom in on the selection.

### Editing view parameters through the menu View parameters

- 1. Click in the graphic window on the right mouse button. The following shortcut menu appears:
  - 2oom all
    Zoom by cut out
    Set view parameters for all
    Cursor snap setting
    Cursor snap setting
    Print/ Preview table
    Table to document
    Print picture
    Picture to document
    Picture to gallery
    Save picture to file
    Copy picture to clipboard
    Wired model in view manipulations
    Advanced graphic setup ...
    Coordinates info

### 🖹 Note

If an element was selected previously, you can define a setting that only applies to the selected elements. (An adapted shortcut menu appears).

2. Choose the option **Set view parameters for all**. The window **View parameter setting** appears. The menu consists of various tabs. You can set the view parameters for all entities or just for the selected entities.

### **View parameters – Entities**

Using the tab page **Structure** graphical representation of various entities can be adjusted. From this tab page the following items are off importance: **Style and Colour:** One can depict the colours by layer, by material, by cross-section, or by structural type.

Draw Cross-section: With this tick box a graphical representation of the cross-section is depicted in the reference line of a bar.

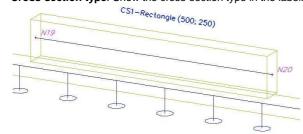
Local Axes: Using this tool the local axes can be set for nodes, 1D and 2D members..

Check / Uncheck group	Lock position
Modelling/Drawing	💯 Misc.   🔍 View 🕮 Labels   🖾 Model
Check/Uncheck all	
Service	
Display on opening the serv	ie 🔽
Structure	
Style + colour	normal
Draw member system line	
Member system line style	system line 🔄
Model type	analysis model
Display both models	
Member surface	
Rendering	wired
Draw cross-section	
Cross-section style	section
- Panel	
Member surface	
Rendering	wired
Structure nodes	
Display	
Mark style	Dot
Member parameters	20.
System lengths	
Member nonlinearities	
FEM type	
🛛 Local axes	
Nodes	
Members 1D	
Members 2D	
ОК	Cancel

### View parameters – Labels and description

Through the tab **Labels**, the labels of different entities can be displayed. In the group **beam labels** the following items can be displayed in the label:

- Cross-section Name: The name of the cross-section is plotted in the label.
- Cross-section type: Show the cross-section type in the label.



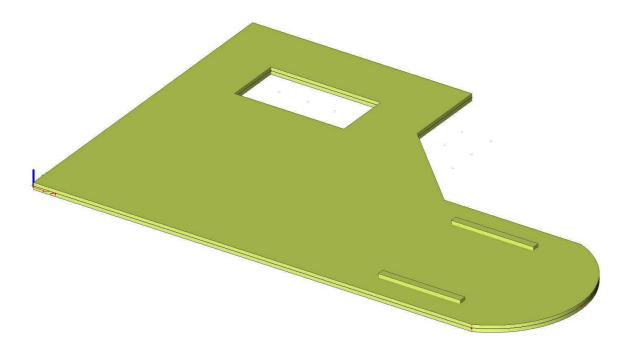
- Length: show the length of the beam in the label.
- Display labels: Only when this tick box is turned ON, the labels will be depicted on the graphical screen.

### View parameters – shortcuts

In the tool bar above the Command line, several frequently used options are grouped among which:

- Show/hide surfaces 🖉 to show the surfaces of the elements.
- Render geometry *I* to view the rendered members.
- Show/hide supports
   to show supports and hinges.
- Show/hide load <sup>1</sup> to show the load case.
- Show/hide node labels
   ABC
   to view the label of the nodes.
- Show/hide member labels
   to view the label of members.
- Set load case for view to edit the active load case.
- Fast adjustment of view parameters on the whole construction it to quickly access to the options from the menu View parameters.

After rendering the following structure is obtained:



# Input of the Calculation Data

# Load Cases and Load Groups

Each load is attributed to a load case. A load case can contain different load types.

To each load case, properties are attributed, which are determinant for the generation of combinations. The action type of a load case can be permanent or variable.

Each variable load case is associated with a load group. The group contains information about the category of the load (service load, wind, snow...) and its appearance (default, together, exclusive). In an exclusive group, the different loads attributed to the group cannot act together in a norm combination. For default combinations on the other hand, the combination generator allows the simultaneous action of the loads of a same group.

The way, in which load cases are defined, is decisive for the load combinations created by the generator. We recommend that you thoroughly read the chapter about loads and combinations in the reference manual.

In this project, two load cases are entered:

- LC1: Dead load
- LC2: Life Load Case

### **Defining a Permanent Load Case**

- 1. Double-click on <sup>Load</sup> in the **Main window**.
- 2. Before you can define loads, you first must enter load cases. Since this project does not contain any load cases yet, the Load Cases Manager will automatically appear.
- 3. By default, the load case LC1 is created. This load is a permanent load of the Self Weight load type. The self weight of the structure is automatically calculated by means of this type.
- 4. Since you will also manually enter loads in the first load case of this project (Roof Weight), you must change the Load Type to **Standard**.
- 5. In the Description field, you can describe the content of this load case. For this project, enter the description "Self Weight Structure".

Load cases			×
🍬 រិះ 🏹 📑 💌 រភ	으 🚭 📽 🖬 🛛 Al	• 7	
DL - Dead Load	Name	DL	
	Description	Dead Load	
	Action type	Permanent	
	LuadGruup	LG1	-
	Lcad type	Standard	
New Insert Edit I	Delete		Close

### **Defining a Variable Load Case**

1. Click New or to create a second load case.

- 2. Enter the description "Live load".
- 3. As this is a variable load, change the Action type to Variable.
- 4. The Load Group LG2 is automatically created. Click ... to display the properties of the Load Group.

Load groups		X	
🔎 🦆 🗶 📸 🕷	n 🗠 🖨 🖻		
LG2	Name	LG2	~
	Relation	Standard	-
	Load	Variable	
	EC1 - load type	Cat A : Domestic	
New Insert Ed	dit Delete		OK

The EC1 - load type determines the composition factor that are attributed to the load cases in this load group. In this project **Cat A: Domestic** is chosen.

5. Click [OK] to close the Load group manager and to return to the Load cases manager.

Load cases				
🔎 🦆 🖋 🖬 🔽 📴	💁 🗠 🚑 🎏 🖬 Ali	- V		
DL - Dead Load	Name	LL		
LL - Live Load	Description	Live Load		
	Action type	Variable 🔹		
	LoadGroup	LG2 🔽		
	Load type	Static 💌		
	Specification	Standard 🔹		
	Duration	Short 🗸		
	Master load case	None 🗸		
New Insert Edit Delete Clo				

6. Click [Close] to close the Load cases manager.

#### Note: load groups

Each load is classified in a group. These groups influence the combinations that are generated as well as the standarddependant factors to be applied. The following logic is adopted.

Variable load cases that are independent from each other are associated to different variable groups. For each group, you set the load category (see EC1). The combination factors from the Eurocode are generated from the available load groups. When a generated combination contains two load cases belonging to different groups, reduction factors will be applied for the transient loads.

If the load is divisible, its different components are entered as individual load cases. As long as the load combination does not contain any variable load belonging to another group, no reduction factors may be applied. The different load cases of a divisible load are therefore associated to one variable group.

Load cases of the same type that may not act together, are put into one group, which is made exclusive, e.g. "Wind X" and "Wind -X" are associated to one exclusive group "Wind".

## Loads

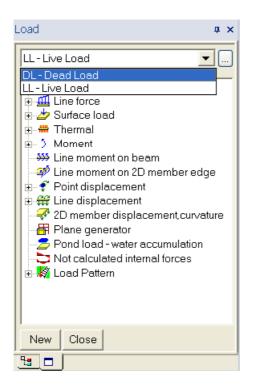
After input of the Load cases, the Loads menu will automatically appear:

The first load case includes two loads:

- Dead load
- Self weight

### Switching between load cases

Activate LC1 by selecting this load case with the mouse pointer in the list box:



### Entering the self weight of ribs as line loads

- 1. Cancel any possibly active selection by pressing **<ESC>**.
- 2. Click on Line Force on beam in the Loads Menu. The dialogue Line Force on beam appears.

4-P2	Name	LF1
	Direction	Z
	Туре	Selfweight
-P1	Gravity coef.	-1
ey	Distribution	Uniform
	Load above joint	🗆 no
x1 x2	Geometry	
	Eccentricity	
$\sim$ $\prec$	Eccentricity ey [m]	0,000
$Q = \frac{q_{max} \cdot q_{min}}{q_{max} + q_{min}}$		
*max **min		

- 3. In the field **Type**, choose **Self Weight**. The Direction is the global Z-direction and the Gravity coefficient is set to –1, so that the load is acting vertically downwards.
- 4. Confirm your input with **[OK]**.
- 5. Select all the bars by means of the **Select all** icon in the toolbar.
- 6. Press **<ESC>** to finish the input.
- 7. Press **<ESC>** once more to finish the selection.

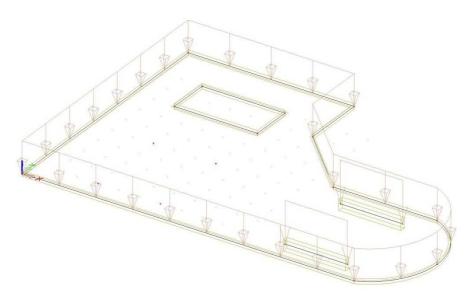
### Input of the selfweight of the slab as surface load

- 1. Cancel any possibly active selection by pressing **<Esc>**
- 2. Click on Surface load on 2D member in the Loadsmenu. The dialogue window Surface force will pop up

Surface force					
	Г	Name	SF1		
		Direction	Z		-
		Туре	Selfweight		•
		Gravity coef.	-1		
-P	Ξ	Geometry			
1 strate		System	GCS		
x y					
			 0	K	Cancel

- 3. In the field **Type**, choose **Self Weight**. The Direction is the global Z-direction and the Gravity coefficient is set to -1, so that the load is acting vertically downwards.
- 4. Confirm your input with [OK].
- 5. If there is only one slab in the project, the load will be automatically put on the slab.

The self-weight is depicted by a brown colour:



The entered loads are so-called self weight loads. The load caused by the non-structural topping will be added to the dead load load case. So that dead loads are combined into one load case.

The live load is input as free loads on a part of the slab. For the live load a different load case will be used.

### Input of dead surface loads

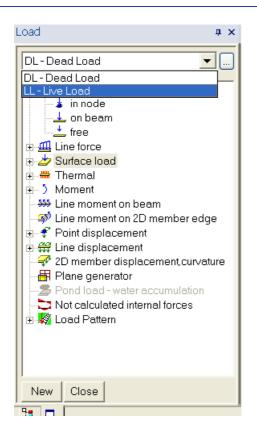
1. Click on Surface load – on 2D element in the Loads menu. The dialogue window Surface force will pop up.

Surface force			X
	Name         Direction         Type         Value [kN/m^2]         Geometry         System	SF2 Z Force -2,00 GCS	• • •
		ОК	Cancel

- 2. The Type of the Surface load on 2D element will be set to Force.
- 3. The **Direction** of the load is **Z** and the **System** is the global coordinate system **GCS**. This causes the fact that all loads in zdirection have a negative value.
- 4. The Value of the surface load will be set to -2 kN/m<sup>2</sup>.
- 5. Confirm your input with **[OK]**.
- 6. If there is only one slab in the project, the load will be automatically put on the slab.

### Switching between load cases

Activate load case 2 "Live load" by selecting this load case with the mouse pointer in the list box:



### Input of live surface loads

- 1. Cancel any possibly active selection by pressing **<Esc>**.
- 2. Click on Surface load free in the Loads menu. The dialogue window Surface force free will pop up.

Surface force free			×
р	Name	FF2	
-P	Direction	Z	-
A MAN	Туре	Force	•
ALL AN	Distribution	Uniform	•
The fire	q [kN/m^2]	-5,00	
and the	Validity	All	
	Select	Auto	•
	Geometry		
	System	GCS	-
	Location	Length	•
X Y Y Y Y Y Y Y Y Y Y Y Y Y Y Y Y Y Y Y	Actions		
	Generate loads		>>>
		ОК	Cancel

- For the field Type the Force is chosen. The Direction is the Z-direction in the coordinate system that you have chosen in System. We will choose global coordinate system for this exercise. The force is -5 kN/m<sup>2</sup> and equally distributed over the surface.
- 4. Confirm your input with **[OK]**.
- 5. The program will asks us to define the outline polygon of the free surface load

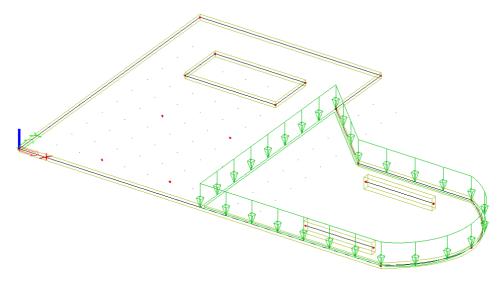
Starting point:

8;9 <enter>

8;0 <enter> 16;0 <enter> New circle arc – intermediate point : 18;3 <enter>

> 16;6 <enter> 11;6 <enter> 8;9 <enter>

#### Right mouse click on New Polygon to close the input:



### Input of variable line load

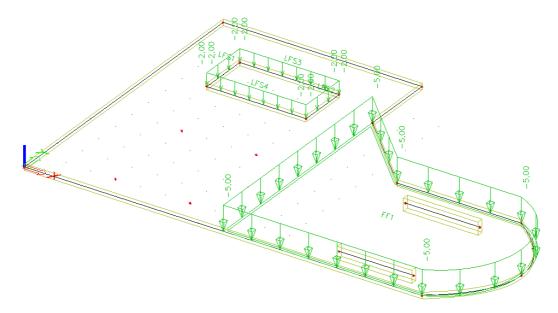
- 1. Cancel any possibly active selection by pressing **<Esc>**.
- 2. Click on Line Force on 2D member edge in the Loads menu. The dialogue window Line force on 2D member edge will pop up.

Line force on 2D member edg	9 9		×
	Name	LFS1	
	Direction	Z	
	Туре	Force	-
-P2	Distribution	Uniform	•
	Value - P [kN/m]	-2,00	28
-P1	Geometry		
	System	LCS	-
- Andrew	Location	Length	
	Position x1	0,000	
	Position x2	1,000	
x1 x2	Coord, definition	Rela	-
1	Origin	From start	•
XYY			
	<u>.</u>	OK	Cancel

3. For the field Type the Force is chosen. The Direction is the global Z-direction. The input value is –2.00 kN/m

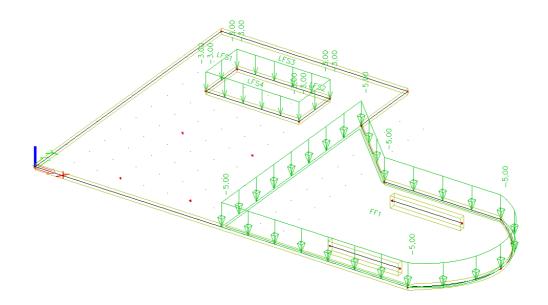
- 4. Confirm your input with **[OK]**.
- 5. Select the four edges around the hole of the staircases.
- 6. Click on the right mouse button to finish the input.
- 7. Click on <ESC> to cancel the selection.

Use the Fast adjustment of view flags on whole model icon on top of the Command line to activate the Labels of Loads option in the Loads/Masses group. A normal load is displayed in green.



### Adapting a load

- 1. Select the variable line loads around the opening by clicking with the left mouse button on these loads.
- 2. The common properties of the 4 series are displayed in the **Properties window**.
- 3. Change the Value from -2,0 kN to -3,0 kN in the Properties window.
- 4. Confirm the modification with **<ENTER>**.
- 5. Press **<ESC>** to finish the selection.



### Input of a free line load

- 1. Cancel any possibly active selection by pressing **<Esc>**.
- 2. Click on Line Force free in the Loads menu. The dialogue window Free line load will pop up.

-P2	Name	FL1	
-Platest	Direction	Z	27) 27)
	Туре	Force	
	Distribution	Uniform	1
	Value - P [kN/m]	-2,00	8
	Validity	All	33
all the second s	Select	Auto	
	Geometry		
	System	GCS	-
	Location	Length	
Y THE			
A	ctions		

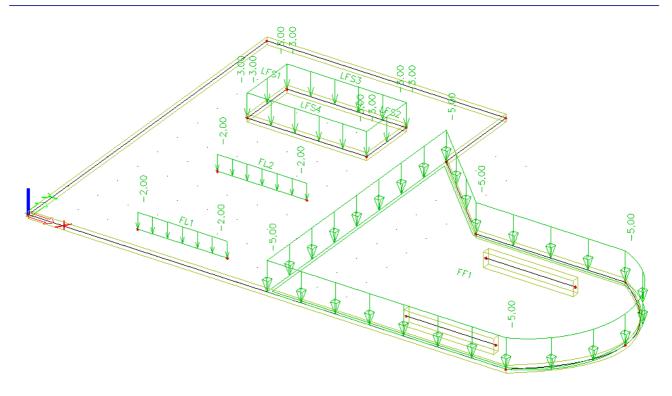
- 3. For the field **Type** the **Force** is chosen. We will enter a value of -2kN/m. The **Direction** is the global Z-direction.
- 4. Confirm your input with **[OK]**.
- 5. The dialogue window will disappear and the coordinates of the new free line load have to be entered.

1st free line load

Starting point:3;1 <enter>End point :6;1 <enter>Right mouse click to end command Polyline .

2n free line load

Starting point:3;5 <enter>End point :6;5 <enter>Right mouse click to end command Polyline .



Click [Close] to quit the Loads menu and to return to the Main window.

#### Note:

The **Command line** includes a number of predefined loads:

# Combinations

After input of the load cases, the latter can be grouped in combinations. In this project, two combinations are created, one for the Ultimate Limit State and one for the Ultimate Serviceability State.

### **Defining Combinations**

- 1. Double-click on 🍀 Combinations below 🔡 Load cases, Combinations in the Main window.
- 2. Since no combination has been entered yet, the window to create a new combination will automatically appear.

Combination	- CO1		X
Contents of con	nbination	List of load cases	
	se Dead Load Live Load	<ul> <li>Load case</li> <li>DL - Dead Load</li> <li>LL - Live Load</li> </ul>	
Name :	C01	Delete	Add
Coeff:	1 Correct	Delete All	Add All
Type : Description :	EN-ULS (STR/GEO) Set B		
Nonlinear combination :		OK	Cancel

- 3. The Type of the combination is changed to **EN ULS (STR/GEO) Set B**. With this combination type, Scia Engineer will automatically generate combinations in accordance with the complex composition rules of the Eurocode.
- 4. By means of the button [Add all], all load cases can be added to the combination.
- 5. Confirm your input with [OK]. The Combination Manager is opened.
- 6. Click New or 🗾 to create a second combination.
- 7. Change the Type of the combination to EN-SLS Char.
- 8. Confirm your input with [OK].
- 9. Click [Close] to close the Combination manager.

Combinations				
🎜 🤮 🇶 😫 🕵	2	🕾 🛛 🚭 📔 Input combinations		•
CO1		Name	CO2	
CO2		Description		
		Туре	EN-SLS Char.	<b>T</b>
		Nonlinear combination		-
		Contents of combination		
		DL - Dead Load	1,00	
		LL - Live Load	1,00	
	Ac	tions		
	E	xplode to envelopes		>>>
	Б	xplode to linear		>>>
New Insert Edit		Delete		Close

# **Calculation and Mesh generation**

The calculation of the slab will be done according to the finite element method. According to the calculation method a mesh will be generated on the slab and the results will be calculated in the internal nodes of each element. The result in the middle of a finite element is determined as the average value of the results in the four internal nodes of the element.

### **Mesh generation**

#### Mesh setup

- 1. In order to see the mesh setup, double click on under
- 2. The dialogue box Mesh setup pop up.

Name	
I Mesh	
Minimal distance between two points [m]	0.001
Average number of tiles of 1D element	1
Average size of 2D element/curved element [m]	0,500
□ 1D elements	
Minimal length of beam element [m]	0,100
Maximal length of beam element [m]	100,000
Average size of cables, tendons, elements on subsoil, nonlinear soil s	spring [m] 1,000
Generation of nodes in connections of beam elements	
Generation of nodes under concentrated loads on beam elements	
Generation of eccentric elements on members with variable height	
No. of FE per haunch	5
Apply the nodal refinement	No members 💌
□ 2D elements	
To generate predefined mesh	
To smooth the border of predefined mesh	
Maximal out of plane angle of a quadrilateral [mrad]	30,0
Predefined mesh ratio	1,5
Hanging nodes for prestressing	

3. The Average size of 2D element/curved element/nonlinear soil spring [m] will be used for the mesh generation if no local mesh refinements have been defined. Change this value in 0,500m.

#### Generation of the mesh

- 4. In order to start the mesh generation you can start
- 5. The program informs you that the mesh is generated and states the number of nodes and 1D and 2D elements have be generated.

#### Note:

In the calculation menu you can adjust the local mesh by clicking on **local mesh refinement**. The program gives you three possibilities.

- Node mesh refinement; refines the mesh around a node.
- 2D member edge mesh refinement; refines the mesh along the edge or internal line of a plate .
- Surface mesh refinement; For the whole surface a denser mesh will be applied.

#### **Display of the mesh**

- The mesh can be displayed using the shortcut button located at the bottom of the graphical screen E > Structure > net.
- 2. The precise settings can be adjusted using the menu item "set view parameters for all" located in the right mouse button menu.



3. On the tab page "Structure" the display of the mesh can be toggled ON / OFF.

View parameters setting	
🦳 Check / Uncheck group	Lock position
🔣 Modelling/Drawing 📔 🎯 Attribute	s 📔 🌌 Misc. 📔 🔍 View 🛛
🕾 Structure 🛛 🖴 Labels 🛛 👗 Mi	odel 🛛 🛃 Loads/masses
🔽 Check / Uncheck all	
E Service	
Structure	
🗄 Panel	
E Structure nodes	
Hember parameters	
🛛 Mesh	
Draw mesh	
Free edges	L <sub>r</sub> e
Display mode	wired 🗾
🕂 Local axes	
ОК	Cancel

4. On the tab page "Labels" different labels for the mesh can be toggled ON / OFF.

View parameters setting	
Check / Uncheck group	Lock position 🗾
🛛 🔛 Modelling/Drawing 🗍 🧐 Attribu	tes 📔 🌌 Misc. 📔 🔍 View 🗎
🖻 Structure 斗 Labels 🔼	Model 🛛 🛃 Loads/masses 🗎
Check / Uncheck all	
Service	
🗄 Beam labels	
H Nodes labels     Second Se	
H Slab members     Slab members	
🛛 Mesh	
Display label	
Nodes	
Elements 1D	
Elements 2D	✓
System lengths	
Nonlinearities	
+ Labels of local axes	
General structural shape	
Display vertex label	
ОК	Cancel

After the adjustment of the mesh and final generation of the mesh, the linear calculation can be started. A dense mesh will in many cases result in more adequate result, yet leading to more calculation time.

If the mesh is not generated before the start of the calculation, the programme will automatically generate the mesh before starting the calculation

### **Linear Calculation**

As the calculation model is completely ready, you now can start the calculation.

#### **Solver settings**

- 1. Double-click on below in the Main window.
- 2. The FE Analysis window appears. Click [OK] to start the calculation.

FE analysis			X
	Single analysis Batch analysis		
1.00	Linear calculation	Г	
11	C Nonlinear calculation	Г	
	C Modal analysis	Г	
1.15	C Linear stability	Г	
1 1	C Concrete - Code Dependent Deflections (C	Г	
	$m{c}$ Construction stage analysis	Г	
	C Nonlinear stage analysis	Г	
	C Nonlinear stability		
	C Test of input data		
	Number of load cases: 2		
10.3			
1	Solver setup	Mesh setup	
	ОК	Cancel	

3. After the calculation, a window announces that the calculation is finished. Click **[OK]** to close this window.

# Results

### **Viewing results**

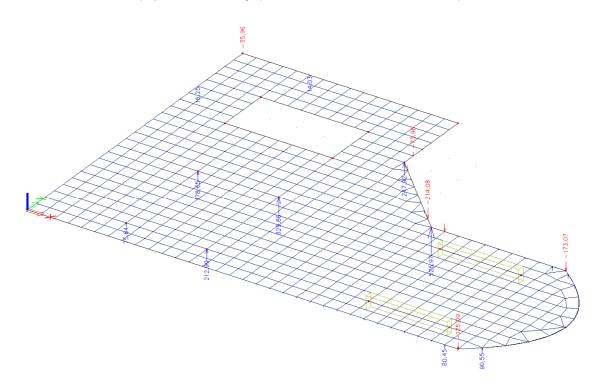
After the calculation is executed, the results can be viewed.

#### **Viewing the Reaction Forces**

- 1. Double-click on Versults in the Main window. The Results menu appears.
- 2. Below Supports, click Reactions.
- 3. The options in the **Property Window** are configured in the following way:
  - The Selection field is set to All.
  - The Load type is set to **Combinations** and the Combination to **CO1**.
  - The Values are wanted for Rz.
  - The Extreme field is changed to Node.

		μх
Reactions (1)	✓ \a \	7 0
Name	Reactions	
Selection	All	*
Type of loads	Combinations	*
Combinations	CO1	•
Filter	No	-
Values	Rz	-
Extreme	Node	*
Drawing setup		
Rotated supports		

4. The action **Refresh** has a red background, i.e. the graphical screen must be refreshed. Click on the behind **Refresh** to display the results in the graphical screen in accordance with the set options.



5. To display these results in a table, the **Preview** action is used. Click on the >>> behind **Preview** to open the Preview.

eaction	S				
	lation, Extreme	: Node			
Selection : . Combination					
Support	Case	dx [m]	Rz [kN]	Mx [kNm]	My [kNm]
Sn1/N13	CO1/1		44,54	0,00	0,00
Sn1/N13	CO1/2		75,64	0,00	0,00
Sn2/N14	CO1/3		128,34	0,00	0,00
Sn2/N14	CO1/4		178,65	0,00	0,00
Sn2/N14	CO1/2		173,26	0,00	0,00
Sn3/N15	CO1/3		193,92	0,00	0,00
Sn3/N15	CO1/4		329,88	0,00	0,00
Sn3/N15	CO1/2		261,79	0,00	0,00
Sn4/N16	CO1/3		123,31	0,00	0,00
Sn4/N16	CO1/4		212,99	0,00	0,00
Sn4/N16	CO1/2		166,47	0,00	0,00
Sle1/S1	CO1/4	16,000	-125,09	0,00	0,00
Sle1/S1	CO1/4	15,500	80,45	0,00	0,00
Sle1/S1	CO1/2	0,000	-5,95	0,00	0,00
Sle2/S1	CO1/4	7,472	-173,07	0,00	0,00
Sle2/S1	CO1/4	0,747	90,55	0,00	0,00
Sle2/S1	CO1/2	0,000	-76,13	0,00	0,00
Sle3/S1	CO1/4	0,000	-173,07	0,00	0,00
21-2/21	001/4	5 000	526.07	0.00	0.00

Note:

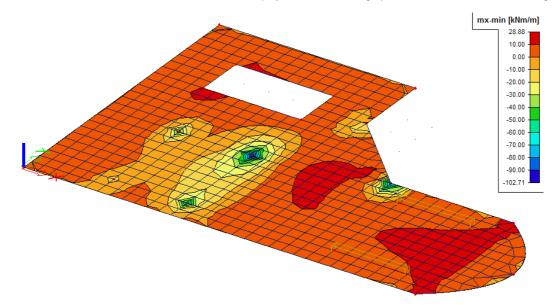
The Preview appears between the Graphical Screen and the Command line. This screen can be maximised to display more data at once.

#### Viewing internal forces on 2D elements

- 1. Click on 2D element > 2D element Internal forces
- 2. The options in the **Property Window** are configured in the following way:
  - The Selection field is set to All
  - The Load type is set to Combinations and Combination to CO1
  - The Values will be reviewed for mx.
  - The Extreme field is changed to Global.

Properties	ά Χ
2D member - Internal forces (1)	🖸 🖬 🎶 🦉
Name	2D member - Internal forces
Selection	All
Type of loads	Combinations 🔹
Combinations	C01 💌
Filter	No
System	Local
Rotation [deg]	0.00
Averaging of peak	
Location	In nodes, avg. on macro 💌
Type forces	Basic magnitudes 🔹
Envelope	Minimum 💌
Standard	
Section	
Edge	
Trajectories	
Rib / Integration strip	
Values	mx 💌
Extreme	Global
Drawing setup	
Actions	
Refresh	>>>
Detailed results in mesh node	>>>
Preview	>>>

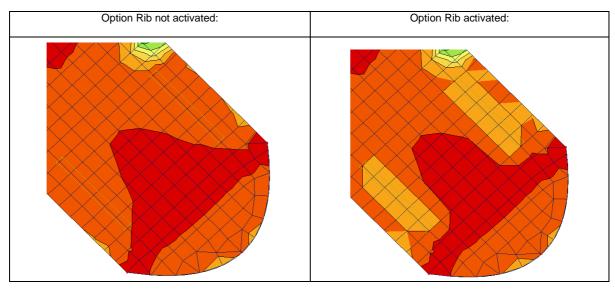
3. Click on the >>> button behind **Refresh** to display the results in the graphical screen for the chosen settings.



For changing the display of the results the settings of the Graphical Screen have to be adapted.

#### **Results for (individual) ribs**

1. By clicking on the tick box '**Rib**' in the Properties window, the results will be adjusted in order to take into account the stiffness of the rib



2. Note the difference for the two ribs that are modelled. It is clear that the forces in the plate are reduced, because the joint stiffness of the slab and the ribs is now considered.

#### **Configuring the Graphical Screen**

1. In the **Properties window**, click the ... icon behind **Drawing Setup** the different options for the graphical screen appear.

2D results display	X
Display Isobands	Minimum and maximum settings
☐ Display mesh ☐ Lighting	Ground value
Advanced settings	Local extrems
Automatic palette values - rounded	None  Style  Text with cross  Description colour
OK Cancel	Help

- 2. Four the group **Display** the option in the combo box '**Isobands**' will be chosen.
- 3. The button Advanced settings... allows setting the legend for the graphical screen.

- 4. Click [OK] to accept the settings or [Cancel] to ignore the selected settings.
- 5. Click in the **Property Window**, on the button **>>>** behind **Refresh** in order to display the results in the graphical screen in accordance with the set options.
- 6. Click [Close] to leave the Results Menu.

Note:

To change the font size of the displayed results, you can use the **Setup > Fonts** menu. In this menu, the different sizes of the displayed labels can be changed.

# **Reinforcement design**

The reinforcement design can be viewed in the concrete menu.

crete	τ×
-I 🖳 Design defaults	
- TD member	
2D member	
- Member data	
🖃 💽 Member design	
Internal forces ULS	
📀 Member design ULS	
Member design ULS+SLS	
🚽 🌽 Section on 2D member	
- 🊟 Reinforcement 2D	
🔤 📇 Averaging strip	
🗠 🔁 New free bars - New free bar	

Double-click Concrete in the Main Window. The Concrete menu appears.

## Changing the diameter of the bars

- 1. Double-click on T<sup>ID</sup> Design defaults (first option in the Concrete menu) to change the diameter of the bars.
- 2. In the menu that appears, choose for "EC-EN -> Concrete -> Design defaults -> 2D structures and beam slabs" and change the diameter of the upper and lower reinforcement in 8mm:

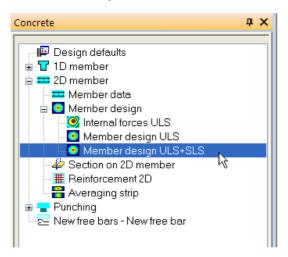
Ĩ.,					
E	Type of members		EC-EN	Name	EC-EN
	1D		i⊟- Concrete	Concrete	
	2D			E Design defaults	
E	Type of values		- Concrete cover	E Concrete cover	
	Design default		Columns	Columns	
	Drawing settings		- Beams - 2D structures and beam slabs	Beams	
			- Punching	2D structures and beam slabs	
			Default sway type (for columns and beams	Use min concrete cover	
			Reinforcement and reinforcement design	Upper	
			-Input of reinforcement	Concrete cover [mm]	30,0
			🗄 Hooks	Diameter [mm]	8,0
			<ul> <li>Anchorage of stirrups</li> </ul>	Angle [deg]	0,00
			- Anchorage of longitudinal reinforcement	Lower	
			Warnings and errors	Concrete cover [mm]	30,0
				Diameter [mm]	8,0
				Angle [deg]	0,00
				Punching	
				Default sway type (for columns and b	
				Reinforcement and reinforcement desi	
				Reference: EN 1992-1-1 Description: Diameter for flexural reinforcement in Application: Design reinforcement to 2D structure	lower layer of 2D structures and slabs
				Application: Design reinforcement to 2D structures	
_	Select all Unselect all	Refresh			OK Cancel

3. And click "OK" to close the concrete setup window.

The reinforcement will be calculated with bars with 8,0 mm diameter.

### **Required areas**

1. Select "Member design ULS+SLS" in the Concrete menu:



- 2. The option in the **Property Window** are configured in the following way:
  - The Selection field is set to All.
  - The Load type is set to Class
- 3. And click now on the three points behind Class:

|--|

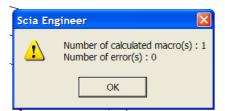
- Double-click on CO1 and CO2
- Change the name in CO1+CO2
- Click two times on **OK**
- 4. The option in the **Property Window** are configured in the following way:
  - The Selection field is set to All.
  - The Load type is set to Class
  - The Class is changed to CO1+CO2
  - The Filter is set to No.
  - The options Show errors and Show warnings are activated.
  - The Location is set to In nodes, avg.
  - The Type values are wanted for Required areas.
  - The Reinforcement is put on Required reinforcement.
  - The option Standard is activated.
  - The option Values is changed to Ar2-
  - The Extreme field is changed to Member.

Member 2D - check cracks - required areas (1) 💽 🚺 🌾 🧳						
Name Member 2D - check cracks - required area						
Selection	All					
Type of loads	Class					
Class	C01+C02					
Filter	No					
System	Local					
Output	Brief					
Show errors						
Show warnings						
Print explanation of errors and warnings						
Use user scale isolinies						
Averaging of peak						
Location	In nodes, avg.					
Type values	Required areas					
Reinforcement	Required reinforcement					
Standard	$\boxtimes$					
Section						
Edge						
Trajectories						
Rib / Integration strip						
Values	Ar2-					
Extreme	Member					
Drawing setup						

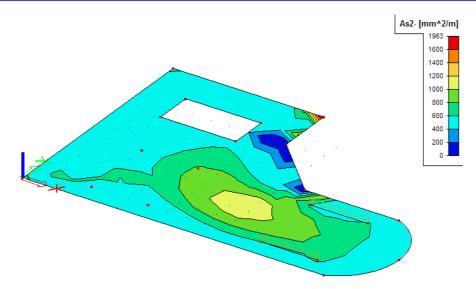
- 5. With the option **values** the reinforcement can be viewed in 2 directions, and at the upper (+) and lower (-) side of the plate. In this example the reinforcement is calculated in the direction 2 (y-direction) at the lower side (-) of the plate: Ar2-.
- 6. Click on Refresh to see the results



7. Scia Engineer will give a message that 1 plate is calculated and no errors were found:



8. Click **OK** to see the results:



The high values in the corner are due to singularities in this model.

9. Click on **Preview** to see the numerical results:

Member 2D - check cracks - required areas								
inger calculation. Extreme : Momber								
All	LAUCINC . MC							
01+CO2								
ary are	ea for sele	ected 2D m	ember					
Node	Case	A <sub>r1-</sub> [mm <sup>4/</sup> m]	A <sub>r2.</sub> [mm <sup>2</sup> /m]	A <sub>r1+</sub> [mm²/m]	A <sub>r2+</sub> [mm²/m]	A <sub>sw</sub> [mm²/m²]		
127	CO1+CO2	1869	983	2203	441	2514		
N7	CO1+CO2	998	1963	0	571	1686		
N1	CO1+CO2	688	714	407	407	733		
			1055	1170	1787	1721		
	ulation, All D1+CO2 einforcer ary are Node 127 N7	ulation, Extreme : Me All D1+CO2 einforcement ary area for sele Node Case 127 CO1+CO2 N7 CO1+CO2	ulation, Extreme : Member All D1+CO2 einforcement <b>ary area for selected 2D m</b> Node Case A <sub>rj.</sub> [mm <sup>3</sup> m] 127 CO1+CO2 1869 N7 CO1+CO2 998	Waltion, Extreme : Member All D1+CO2 einforcement ary area for selected 2D member       Node     Case     A <sub>r1</sub> [mm <sup>2</sup> /m]     A <sub>r2</sub> [mm <sup>2</sup> /m]       127     C01+C02     1869     983       N7     C01+C02     998     1963	Valation, Extreme : Member All D1+CO2 einforcement ary area for selected 2D member           Node         Case         A <sub>r1</sub> . [mm²/m]         A <sub>r2</sub> . [mm²/m]         A <sub>r1+</sub> . [mm²/m]           127         C01+C02         1869         983         2203           N7         C01+C02         998         1963         0	Valation, Extreme : Member All D1+CO2 einforcement ary area for selected 2D member           Node         Case         A <sub>r1</sub> , [mm²/m]         A <sub>r2</sub> , [mm²/m]         A <sub>r1+</sub> [mm²/m]         A <sub>r2+</sub> [mm²/m]           127         C01+C02         1869         983         2203         441           N7         C01+C02         998         1963         0         571	Valation, Extreme : Member All D1+CO2 einforcement ary area for selected 2D member           Node         Case [mm <sup>2</sup> /m]         A <sub>r1</sub> . [mm <sup>2</sup> /m]         A <sub>r2</sub> . [mm <sup>2</sup> /m]         A <sub>r1+</sub> [mm <sup>2</sup> /m]         A <sub>r2+</sub> [mm <sup>2</sup> /m]         A <sub>sw</sub> [mm <sup>2</sup> /m <sup>2</sup> ]           127         C01+C02         1869         983         2203         441         2514           N7         C01+C02         998         1963         0         571         1686	

- 10. To use user scale isolines, activate this option: Use user scale isolinies
- 11. The "User scale isolines"-window will open Click on **New** to define a new User scale isoline
- 12. The following scale options are set
  - The Name is changed to User
  - The Diameter is set to 8.0 mm.
  - The **Distance** is changed in **100** mm.
- 13. Click on Copy to legend to save this setting in the legend.

Scale of isoline	=n 25		1		×
Name	User				
New level Diameter Distance Amount	8.0 100 503	0,0	0,0 0 0	Mean 8,0 100 503	mm mm mm^2
Legend		Copy to I	egend	Clear lev	el
User default reinforcement 8.0-100.0		Mean 8.0-100	As [mm^2 .0 502.65	2]	
		Delete a	ct.level	Delete a	11
			40		Cancel

14. Change the **Distance** in **150** mm. And click on **Copy to legend** again.

Scale of isolir	ies				
Name	User				
_New level —				Mean	
Diameter	8,0	0,0	0,0	8,0	mm
Distance	150	0	0	150	mm
Amount	335	0	0	335	mm^2
Legend		Copy to	legend	Clear le	vel
	reinforcement	Mean	As [mi	m^2]	
8.0-100.0 8.0-150.0		8.0-100 8.0-150	0.0 502.65	i	
		Delete a	ct.level	Delete	all
				ОК	Cancel

- 15. The legend is shown below. 8.0 100.0 means a diameter of 8mm with a distance of 100mm. Input the following diameter option and click always on Copy to legend:
  - Diameter = 8.0mm; Distance = 200mm
  - Diameter = 8.0mm; Distance = 250mm
  - Diameter = 10.0mm; Distance = 100mm
  - Diameter = 10.0mm; Distance = 150mm
  - Diameter = 10.0mm; Distance = 200mm

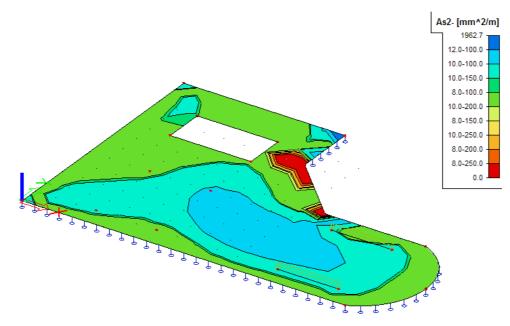
- Diameter = 10.0mm; Distance = 250mm
- Diameter = 12.0mm; Distance = 100mm

•

s	cale of isolines									X
1	Name									
Ι,	New level							Mean		_
	Diameter	14,0	0,0		0,0		-	14,0	mm	
	Distance	100	0		0		-	100	mm	
	Amount	1539	0		0			1539	 mm^2	
1	Legend		(	Copy to le	egen	d		Clearle	evel	
	User default reir	nforcement		Mean		As [mr	n^21			
	12.0-100.0		12.0-100.0		1.0	1130.9				
	10.0-100.0					785.40				
	10.0-150.0		10.0-150.0 523.60							
	8.0-100.0			8.0-100.0		502.65 392.70				
	10.0-200.0 8.0-150.0			8.0-150.0		335.10				
	10.0-250.0			10.0-250		314.16				
	8.0-200.0			8.0-200.0		251.33				
	8.0-250.0			8.0-250.0	D	201.06				
	,									
				Delete ac	tleve	91		Delete	all	
							OK		Cancel	

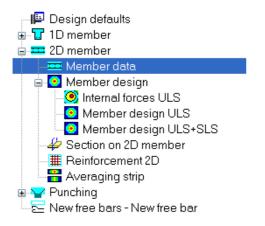
16. Click OK to save this legend

- 17. Click **OK** to use this User scale
- 18. Click on Refresh to see the required reinforcement with the user scale



## Input user reinforcement

1. Select in the Concrete menu:



- 2. Select the plate
- 3. The option in the **Data slab concrete** Window are configured in the following way:
  - The **Type** field is set to **Plate**.
  - The option Advanced mode is activated.
  - The Type of reinforcement geometry is set to Orthogonal.
  - The Type of cover is set to use minimal cover.
  - Activate the option **User reinforcement**.
  - The Main reinforcement steel and Shear reinforcement steel are set to B 400A
  - The First direction angle [deg] is left on 0.0 degrees
  - Under the title "1" the Diameter is set to 8.0 mm and the Layer angle to 0.000.
  - Under the title "2" the Diameter is set to 8.0 mm and the Layer angle to layer on previous layer.
  - The Exposure class is changed to XC2.
  - The Abrasion class is set to None.
  - The situation of Delta;cdev is set to In-situ concrete.

Name	DSC1		
du 2D member	S1 Plate		
Tuno			
Advanced mode			
Basic data			
Type of reinforcement geometry	Orthogonal		
Type of cover	use minimal cover		
Different layers per side	0		
CI User reinforcement			
User input thickness			
dl Main reinforcement steel	B 400A 🔹		
Shear reinforcement steel	B 400A		
Longitudinal			
de First direction angle [deg]	0,00		
dp Number of reinforcement layers	2		
······································			
Diameter (du,dl) [mm]	8,0		
Layer angle	0.000		
Concrete cover (cu,cl) [mm]	30		
Basic distance [mm]	200		
□ 2	1.24 2002 20		
Diameter (du,dl) [mm]	8,0		
Layer angle	90.000		
Type of cover	layer on previous layer		
Concrete cover [mm]	38		
Basic distance [mm]	200		
Minimal concrete cover			
Input for sides			
Structural class	S3		
Exposure class	XC2		
Abrasion class	None		
Situation of Delta;cdev	In-situ concrete		
Concrete	C30/37		
Stone diameter [mm]	32		
Special quality control			
cmin.dur [mm]	20		
Delta;cdur [mm]	0		
Position of view flag			

- 4. And click now on **OK**
- 5. Click on **Esc** to end this function.
- 6. Now select again the option **Member design ULS+SLS**.
- 7. The option in the **Property Window** are configured in the following way:
  - The Selection field is set to All.
  - The Load type is set to Class
  - The **Class** is changed to **CO1+CO2**
  - The **Filter** is set to **No**.
  - The options Show errors and Show warnings are activated.
  - The Location is set to In nodes, avg.
  - The **Type values** are wanted for **Required areas**.
  - The Reinforcement is put on User reinforcement.
  - The option **Standard** is activated.

- The option Values is changed to Ar2-
- The Extreme field is changed to Member.

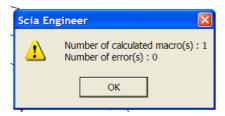
us (1) 💽 🚺 🏹 🦉
Member 2D - check cracks - required areas
All
Class 🔹
C01+C02
No
Local
Brief 🔹
User 💌
In nodes, avg.
Required areas
User reinforcement
⊠
Ar2-
Member 🔹

8. With the option **User reinforcement**, the reinforcement inputted by the user will be shown.

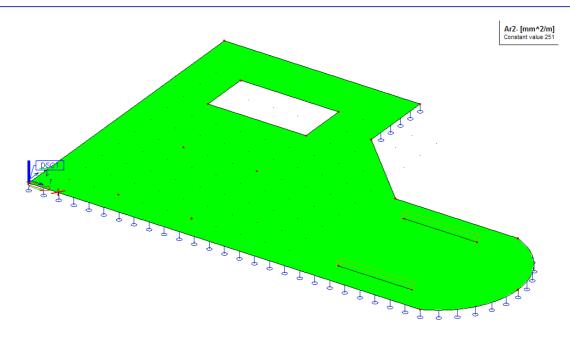
Click on Refresh to see the results



9. Scia Engineer will give a message that 1 plate is calculated and no errors were found:



10. Click **OK** to see the results:



This reinforcement has a constant value of 251 mm<sup>2</sup>/m.

## Document

In this final part of the tutorial, we will explain how a calculation note can be drawn up.

## Formatting the Document

1. Double-click Document in the Main Window or click Document appears.

The Project data are automatically displayed in the header of the document.

	Project	Plate
NEMETSCHEK Scia	Part Description Author	- Tutorial Plate Concrete ND

2. Click the [New] button below the Document Menu. The window New Document item appears.

Default     Project     Ubraries	
🗄 🔺 Librarian	
🕀 🗣 Libraries	
🗉 🔶 Sets	
Solver and Mesh	
🗉 🔶 Structure	
🗄 🔶 Load	
Construction stages	
🗉 🔶 Results	
🗉 🔶 Solver Files	
🗉 🔶 Steel	
🗄 🔶 Aluminium	
🗉 🔶 Custom check	
🕀 🔶 Pipeline	
🗉 🔶 Timber	
🗉 🔶 Concrete	
🗉 🔶 Steel concrete bridge	
🕀 🔶 Geotechnics	
🕀 🔶 Composite Beam	
🗉 🔶 Composite Column	
🗉 🔶 Mobile loads	
🗉 🔶 Influence lines	
🗉 🔶 Picture	
🗄 🔶 Special	
	•
< Add 0	Close

- 3. By means of this window, several data can be added to the document.
  - Open the Libraries group and click on Materials. Click [<<< Add] to add this item to the document.

🗄 🔶 Libr	raries
	Setup manager
	Layers
	Cross-sections
	Materials
	UCS

• Click Cross-Sections. Click [<<< Add] to add this item to the document.

Open the Structure group and click on 2D Members. Click [<<< Add] to add this item to the document.



- Open the Results group and click Reactions. Click [<<< Add] to add this item to the document.
  - Results ₩ Internal forces on beam ✤ Deformation on beam by Displacement of nodes 🐐 Reactions Resultant of reactions 🕫 Nodal space support resultant 🛱 Intensity 🏪 Member Stress 払 Shear in joint Relative deformation 🖽 Bill of material Connection Forces Foundation table Displacement of nodes - mesh 🧿 Member2D - Internal Forces
- 4. Click [Close] to close the New document item window and to return to the document.

The items that were added to the document are displayed in the **Document Menu**. Dragging the items with the mouse can change their order. At the right-hand side of the screen, the Preview of the document is displayed.

### Displaying results in the document

- 1. In the **Document menu**, click **Reactions**. In the **Properties window** the properties of this table are displayed. The parameters for displaying the results in the **Document** are configured in the same way as the parameters for viewing the results in the **Results Menu**.
  - The option Visible is activated.
  - The Selection field is set to All.
  - The Load type is set to Combinations and the Combination to CO1.
  - The Filter is set to No.
  - The Values are wanted for Rz.
  - The Extreme field is changed to Global.
- 2. Click the button behind **Refresh** to display the table in accordance with the set options.

	[ProjectIn2010 : 2] Setup Window Help												
	Cel 💽 ?   Projectin2010	🔹 💂 : DM 🛤	l 🗗 🗋									- 6	57
ocument		+ × 🗈 ()			120 %	-   ()	3 ( <b>1</b>   -	default	- 🕼 🖽		- 11		
DOC-Default			a Type		Reda	Participation of the second se	a Ay	and the second	1,041/e-01	1,041/e-01			^
			Detailer	1	500;	250	— I y,	z [m <sup>4</sup> ]	2,6042e-03	6,5104e-04			
🗉 🔶 Default			Item m	223 X X X X X	C	0/37		[m <sup>6</sup> ], t [m <sup>4</sup> ]	0,0000e+00	1,7867e-03			
- Materials			Fabrica		con	arete	Wel	y, z [m <sup>3</sup> ]	1,0417e-02	5,2083e-03			
Cross-secti			Bucklin	g y-y, z-z		b b	Wpl	y, z [m <sup>3</sup> ]	1,5625e-02	7,8125e-03			
	D (2D members)		FEM a	nalysis		×	dy,	z [mm]	0	0			
* Reactions			a Picture				c YI	CS, ZLCS [mm	125	250			
			at Picture				alph	a [deg]	0,00				
							1007023	m²/m]	1,5000e+00				
esults (1)	Reactions Reactions	<b>₽</b> × 1√ Ø	3. Memb Name S1 4. React	Material C30/37	Th. [mm] 250	Thickness typ constant	pe Type plate (90)	Layer Layer1					
lisible	🖾 yes		4. 110401										
Prefer one page				ulation, Extrem	e : Global								
Rotate			Selection :										
	All	<u> </u>	Combination Support	Case	dx	Rz	Mx	Му					
	Combinations	-	support	Case	ax [m]	[kN]	[kNm]	[kNm]					
ype of loads	001		and a subscription of the	0044	4.000	-317,11	0.00	0.00					
ype of loads combinations	C01	-	Sle6/S1										
ype of loads combinations ilter	No	-	Sle6/S1 Sle6/S1	C01/1		715 29	0.00	11111					
Гуре of loads Combinations Filter /alues	No Rz	-	Sle6/S1	C01/1	5,000	715,29 -55.44	0,00	0,00					
Selection Type of loads Combinations Filter Values Extreme	No Rz	-	1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1			715,29 -55,44	0,00 <b>0,00</b>	0,00					
Type of loads Combinations Filter	No		Sle6/S1	C01/1	5,000								
ype of loads combinations ilter 'alues xtreme	No Rz Global	-	Sle6/S1 Sle1/S1	C01/1	5,000				10/6			2	~
ype of loads combinations ilter 'alues ixtreme	No Rz Global	•	Sle6/S1 Sle1/S1	C01/1	5,000		0,00	0,00				ą	×

3. Click the [Close] button below the Document Menu to close the document and to return to the structure.

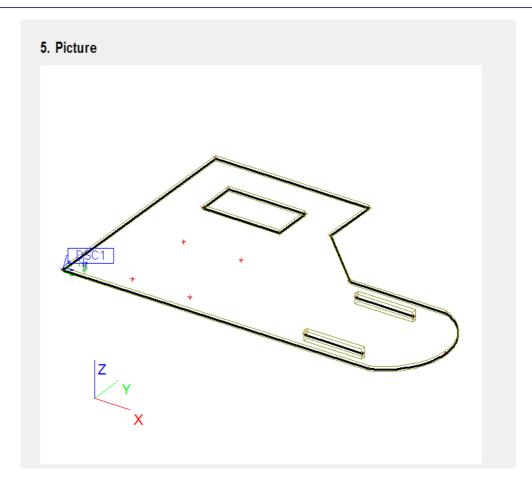
## Adding an image to the document

- 1. Click on the **Print Picture** icon in the button bar.
- 2. Choose the **Picture to document** option in the list box to send the image, which is currently displayed in the graphical screen, to the document.

The window Insert item(s) into document appears.

	Name	Picture				
	Caption	Picture				
	Visible	⊠ yes				
	Prefer one page					
	Rotate	-				
	Size definition	Percent of page height 50				
	Percentage of page height Fit	Fit to window				
	Rotate					
	3D image in PDF					
	Results					
Ξ	Data picture	<u></u>				
	Display mode	Wired				
	Load units in regen.					
	Load activity in regen.					
	Text scale factor	1				
	Charset of texts	ANSI (USA, UK, Europe)				
	Line pattern length	3				
Ξ	Display GCS icon	To picture.corner				
	Performance					
	Settings	>>>				

- 3. The Caption field is changed to Picture as the title of this picture.
- 4. The Percentage of page height field is changed to 50 so that the image covers 50% of a page, i.e. half a page.
- 5. Confirm your input with **[OK]** so that the image is sent to the document.
- 6. Click in the button bar to open the **Document**.
- 7. In the **Document Menu**, click **Picture**. The image is displayed in the Preview of the **Document**.



8. Click [Close] below the Document Menu to close the document and to return to the structure.

# Epilogue

In this syllabus, the basic functionalities of Scia Engineer for the input of a concrete plate, including the calculation of the reinforcement, were introduced by means of an example.

After reading the text and executing the example, the user should be able to model and calculate a simple concrete plate.

For more detailed information about steel calculations, we refer to the Advanced Training Concrete.