

TUTORIAL CONCRETE PLATE

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

© Copyright 2018 SCIA nv. All rights reserved.

ntents	
General Information	
Welcome SCIA Engineer Support Websites Introduction	
Getting started	
Starting a project Project management	
Save, Save as, Close and open	
Saving a project	
Closing a project	
Opening a project	
Start project manger	
Geometry input	
Input of the geometry	
Geometry	
Supports Check Structure data	
Checking the structure	
Connected entities	
Graphic representation of the structure	
Load cases and Combinations	
Load Cases and Load Groups	
Defining a Permanent Load Case	
Defining a Variable Load Case	
Loads Combinations	
Calculation and Mesh generation	
Mesh generation	
Linear Calculation	
Results	
Viewing results	
Reinforcement design	
Viewing design internal forces	
Reinforcement design ULS Input of practical reinforcement	
Document	
Engineering Report	
Epilogue	

Welcome

Welcome to the SCIA Engineer Tutorial Frame Concrete. SCIA Engineer is an integrated, multi-material structural analysis and design software for all kinds of structures. Its wide range of functionality makes it deployable for any construction type: design office buildings, industrial plants, bridges or any other project, all within the same easy-to-use environment.

The program treats the calculation of 2D/3D frameworks, design and check of reinforcement 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, reinforcement design and checks according to various codes, generating the calculation report, etc.

SCIA Engineer is available in three different editions:

License version

The license version of SCIA Engineer is secured with a 'dongle', a hardlock, which you apply to the USB gate of your computer or a softwarematic license in 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 or module editions that are available.

Viewer mode

If the program doesn't find a licence it can be used as a viewer only. That means that any project can be opened, properties of entities can be checked, if the calculation has been done also results can be seen and report can be printed.

However, no change of the model is possible, no calculation can be run, no new output can be created.

Student version

The student version has the same possibilities as the license version for all of modules. This version is also secured by 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.net 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

For various phone numbers to different offices visit our page https://www.scia.net/en/contact/offices Via the SCIA Customer Portal website

http://www.scia.net/en/portal

Websites

Link to Manuals and Tutorials https://www.scia.net/en/support/downloads/scia-engineer-manuals-tutorials Link to eLearning http://elearning.scia.net/ Link to Web help http://help.scia.net/

Introduction

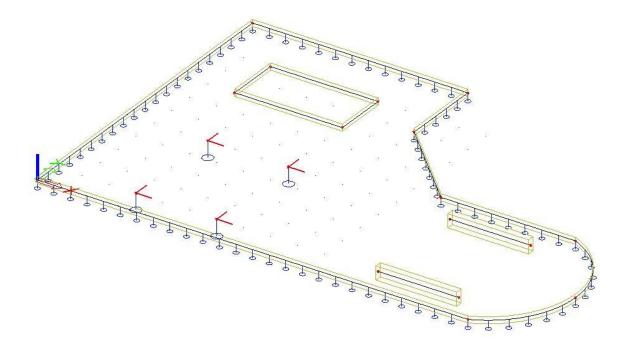
This Tutorial describes the basic functions of SCIA Engineer, the input, analysis and design of a concrete slab.

Before you start, you must be familiar with your operating system: for instance working with dialogues, menu bars, toolbars, status bars, handling the mouse, etc.

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

The major part is related to design of reinforcement and check according to design code. The Tutorial ends with a brief introduction to the calculation report.

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



Starting a project

Starting the program

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

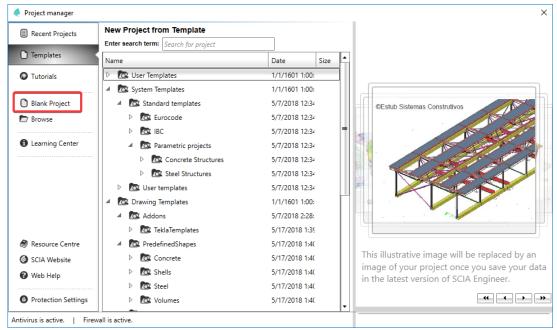
- 1. Double-click on the SCIA Engineer shortcut in the Windows Desktop, or
- If the shortcut is not installed, click [Start] and choose All apps > SCIA Engineer 18.0 > SCIA Engineer 18.0.

If the program does not find any protection, you will see a dialogue indicating that no protection was found. You are offered to run Protection setup and select appropriate protection type (e.g. try-out), or run the program in Viewer mode.

For this Tutorial, you must start a new project with standard licence.

Starting a new project

1. When the **Project manager** dialogue appears, click **Blank project** and double-click button **Analysis**.



2. You can also start new project with an icon \Box in the toolbar or with a key combination **Ctrl+N**.

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

Project data				×
Basic data Fur	nctionality Actio	ons Unit Set Protection		
	Data		Material	
	Name:		Concrete	✓
ARR A	Hamo.	-	Material	C25/30
HAR	Part:		Reinforcement	B 500B 👻
THE			Steel	
THE	Description:	•	Masonry	
1122			Aluminium	
	Author:	SCIA	Timber	
			Steel fibre conc	
NINE COMP	Date:	17. 05. 2018	Other	
	Structure:	Post processing	National Code:	
		environment	EC - EN	·
	😭 Plate XY	🝷 🍭 v17 🔹		
	Model:		National annex:	
	🕅 One	w	Standard EN	·
	'Thick-walled' Cor	ncrete cross-sections: the advanced 2D FEI	M method is off!	
			ОК	Cancel

- 3. In the **Basic data** group, enter your preferred data. These data can be mentioned on the output, e.g. in the report and on the drawings.
- 4. Choose the **Structure: Plate XY** (to limit input possibilities to 2D members in one plane only + 1D members as plate ribs for example) and **Model: One**.
- 5. In the Material group, tick Concrete checkbox.

Material is the only required setting to proceed.

Choose $\mbox{C30/37}$ from the combo-box Material and choose also steel grade B500A for Reinforcement.

- 6. In the Code frame select National Code EC-EN and National annex: Standard EN
- 7. Confirm your input with [OK] button.

Notes:

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

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

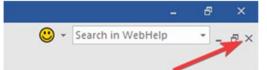
Click on **I** in the toolbar or press **Ctrl+S**.

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 choose **File > Save as** in the main menu, you can enter a new/other drive, folder and name for the project file.

Note: Autosave function creates a backup file every 15 minutes by default. These backup projects can be found in folder c:\Users*username*\Documents\ESA16.0\Autosave\

Closing a project



To close a project, choose **File > Close** in the main menu or click the smaller X button on top right corner of the application.

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

Opening a project

Click on For 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).

Start project manger

Click on to open project manager. Here the recently closed project can be found, as well as sample projects.

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 **Main** tree is visible on the left hand side. If you want to enter a structure you must double-click on **Structure** in the **Main** tree.

Tree		▼ ₽ ×	Tree	▼ ₽ ×	
te Mair	n		Te Main 🛥 Structure 🗙		
	Project Line grid and storeys BIM toolbox Structure Load cases, Combinations Calculation, mesh Engineering report Drawing Tools Libraries Tools		 ID Members 2D Members - Plates 2D Members - Plates Plate Ribbed Slab Prefab Slab Prefab Slab Advanced Input Model data Model ling/Drawing		

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 already available.

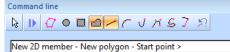
We will input the structure as a plane 2D member. We will use the advanced input options, like definition of an opening in the slab or drawing of a plate rib.

Input of a plane 2D member

- 1. In the structure menu double-click on the Plate command in the 2D member chapter.
- 2. The window with 2D member parameters will be opened.

S1	
Standard	
Standard FEM	
plate (90)	
C30/37	Ψ.
Isotropic	
none	
constant	
250	
Standard	
0,00	
Layer1	Ψ.
	OK

- 3. Default concrete grade C30/37, as specified in the Project data previously, is selected in **Material**. Change **Thickness** to 250 mm.
- 4. After accepting by **[OK]** button the program asks (in the command line) for the starting point of new polyline which defines the plate.



5. The buttons in the **Command line** allow to built up polygonal edges using different line types, or to choose directly for a circular or rectangular surface. The geometry can be input with a help of a dot grid or line grid or by direct input of coordinates in the command line. Proceed with typing the following coordinates, confirmed by Enter key each time.

Starting point: 0;0 <enter> 16;0 <enter>

Click on icon **New circular Arc** and define the following points (intermediate and end point of an arc): @2;3 <enter>

@-2;3 <enter> Continue with polylines:

@-5;0 <enter> @-3;3 <enter> @0;3 <enter> @-8;0 <enter>

and finish the input with **<ESC>** key.

The following picture is now depicted in the screen:



Note:

Coordinates are defined with either semicolon or space between X and Y coordinates. Don't use comma which defines decimal mark.

@ symbol defines relative coordinates, instead of global coordinates.

Definition of an opening

1. In the **Structure** menu under **2D member components** we will create an **Opening** with the name *Stairs*.

Structure P ×	Opening/Panel	×
Structure + ×	Opening/Panel Name 2D member Panel Cut 1D member Convert into 1D member Image: Panel Nodes	Stairs S1 no no no no
		OK Cancel

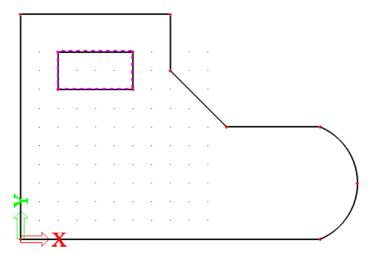
- 2. Type the naming Stairs in the first line of Opening/Panel properties and confirm by [OK] button
- 3. Click on icon New rectangle in Command line.



4. The two nodes of its diagonal define a rectangle opening. This is also depicted by the two red dots on the icon. Type the following point into command line

New rectangle – Begin Point:	2;8 <enter></enter>
End Point:	6;10 <enter></enter>

and finish the input with **<ESC>** key.

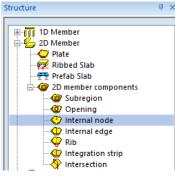


You can also activate rendering by the icon to see the real "hole" in the slab.

00	<	

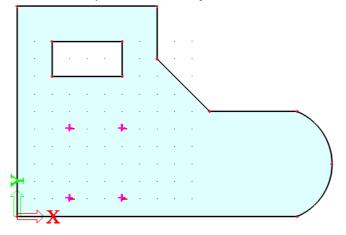
Input of internal nodes

1. In the Structure menu choose under 2D element components command Internal node.



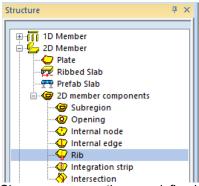
- 2. We will add four new internal nodes by typing its coordinates into command line:
 - 3;1 <enter> 3;5 <enter> 6;5 <enter> 6;1 <enter>

and finish the input with **<ESC>** key.



Input of plate ribs

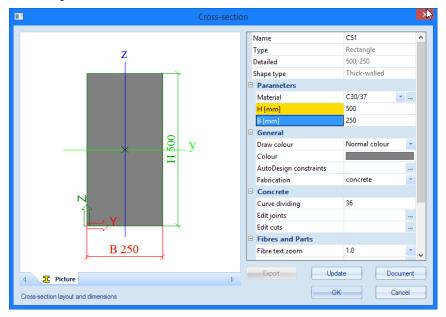
1. In the Structure menu, under 2D member components choose command Rib.



2. Since no cross-section was defined in the project previously, the dialogue **New cross-section** pops up. Here we will be able to select and define from the **Concrete** group a **Rectangle** shape.

	New cross-section	
Available groups	Available items of this group	Items in project
	Profile Library filter	Add Close

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



4. We will accept the cross-section by pressing the **[OK]** button. After this we will **[Close]** the two following dialogues.

	Plate rib		
	Name		
	Type rib	plate rib (92)	
╞═══╌═╴┲╴╧╧╧╧	Analysis model	Standard	Ψ.
	CrossSection	CS1 - Rectangle (500; 250)	×
	Shape of rib	T symmetric	-
	Effective width	width	-
	for int. forces [mm]	1000	
	for check [mm]	1000	
	FEM type	standard	-
	Buckling and relative lengths	Default	
	Layer	Layer1	×
		ОК	Cancel

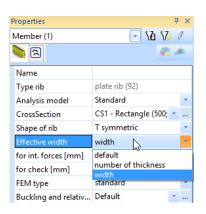
5. Plate rib parameters dialogue appears. No changes are applied here.

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

First rib	Start point: End point:	12;5 <enter> 15;5 <enter></enter></enter>
Second rib	Start point: End point:	12;1 <enter> 15;1 <enter></enter></enter>

and finish the input with **<ESC>** key.

Note to rib property – Effective width

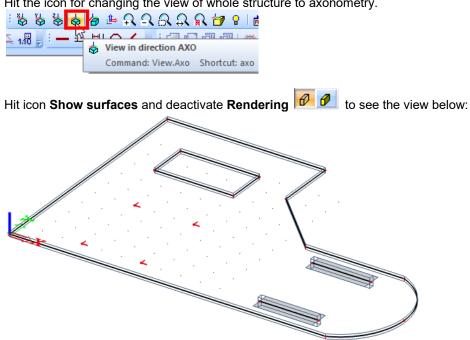


width : The user can input the effective width of the substitute T-section for the internal forces (FE analysis) or the checks (Design As) manually.

number of thickness : The width of the slab for the T-section is defined as a factor which multiplies the plate thickness. User enters the factor manually.

default : The width of T-section is defined as a factor which multiplies the plate thickness. The factor is defined in Setup > Solver > Number of thicknesses of rib plate. Default is 20.

Effective width is graphically represented by a thin rectangular line around the rib.



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

Supports

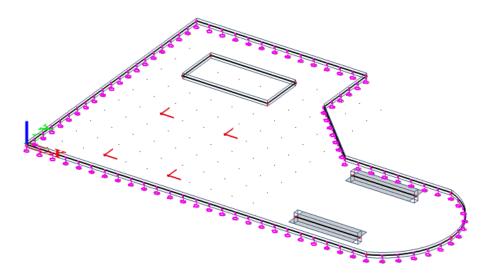
The input of the geometry can be finalized by definition of support conditions. We assume that the whole plate edge is supported in global z-direction. Thus we simulate that e.g. masonry wall supports the slab.

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 appears.

Structure # ×		Line support on 2D member edge		
D Member 20 Member 20 Member 20 Member Plate Prefab Slab Prefab Slab Prefab Slab Do member components Advanced Input Model dat Do member edge Ininge on 20 member edge Rigid arms Conset members/nodes Connect members/nodes Connect members/nodes Continuous beam Check structure data Check structure data Dember edge Difference Section on beam Dembers/nodes Connect members/nodes Context members/nodes Dember edge Difference Difference	Rz Rz x1 x2 x2 x2 x2 x2 x2 x2 x2 x2 x2 x2 x2 x2	Name Constraint Z Rx Ry G Geometry System Position x1 Position x2 Coord. definition Origin	Sle1 Hinged Rigid Free Free GCS 0,000 1,000 Rela From start	V V V V
			OK Cance	el:

- 3. Change the **Constraint** to **Hinged** so that Z axis deformation is prohibited but rotations around X and Y axes are free.
- 4. Select the edges around the slab one by one with mouse cursor edge1, edge2, edge3, edge4, edge5, edge6, edge7.
- 5. Press **<ESC>** to cancel the input command.



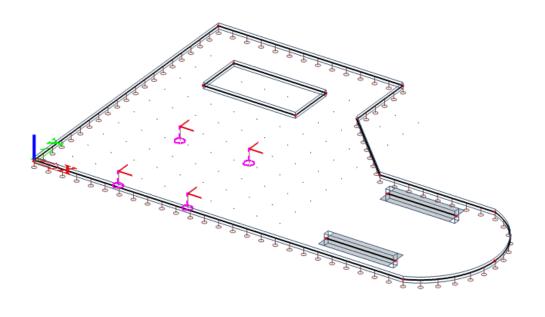
Input of Nodal supports

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

 Structure
 * Support in node

Structure $\Psi \times$		Support in node		
Bructure Avanced Input D Member Plate Plate Plate Plate Ribbed Slab Perefab Slab Perefab Slab Portefab Slab	Rx Ry (j)	Support in node Name Type Constraint Z Rx Ry Default size [m] Geometry System	Sn1 Standard Hinged Rigid Free 0,200 GCS	*
			ОК Са	ncel

- 2. We will support the nodes in Z-direction only again. Therefore change the **Constraint** to **Hinged** again.
- 3. We apply the nodal supports for the internal nodes N13, N14, N15 and N16 by mouse cursor.



Note

- If required a flexible support can be defined in order to model the behavior of the columns more adequately. Also, we can model the supports as "column", then the stiffness is directly derived 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, zero beams, duplicate members, wrong references of hinges or supports etc. However, this tool does not check if the structure is correctly supported or if it is a mechanism.

Checking the structure

1. Double-click on the Check structure data option in the Structure service

Check structure data	or click on the	- <i>5</i> Ijfi 🞯 🛱	8 A 🗐 🗊	icon in the
toolbar.				

2. The Structure data check window appears, listing the different available checks.

Check of	structure data	
Check of nodes		
Search nodes		
Search duplicate nodes	Ignore parame	eters
Check of members		
Check members Search null members	Null members:	0 embers
Search duplicate members	Duplicate	0
	Invalid parts:	0
Check of data references		
Check data references	O Memory efficie	nt method
	Fast method	
Check of additional data		
Check additional data position	Invalid position	0
	Correct position	on
Check free load distribution points	Invalid Ioads	0
Check of steel connections		
Check steel connections	Invalid ✓ Delete invalid	0 connections
Check load panels Check cro Check additional data Check duplic	oss-links	Check Cancel
circle dollar data circle dapie	.,	L

- 3. Click [Check] to perform the checks.
- 4. The Data Check Report window appears, indicating that no problems were found.

	Check of structure data	×
Check of nodes ✓ Search nodes		
Search duplicate nodes	🗌 Ignore pa	arameters
Check of members Check members Search null members	Null member ✓ Delete nu	
Search duplicate members	Duplicate Delete du Data check report	0 uplicate members 0 id parts
Check of data references Check data references	Data check finished.	cient method
Check of additional data Check additional data position		n 0
✓ Check free load distribution point		
Check of steel connections Check steel connections	invalid V Delete inv	0 valid connections
Check load panels	Check cross-links	Continue Cancel

- 5. Close the check by clicking [OK].
- 6. In case of any problem SCIA Engineer can automatically correct the structure data (delete duplicated entities, correct wrong reference, etc.)

Connected entities

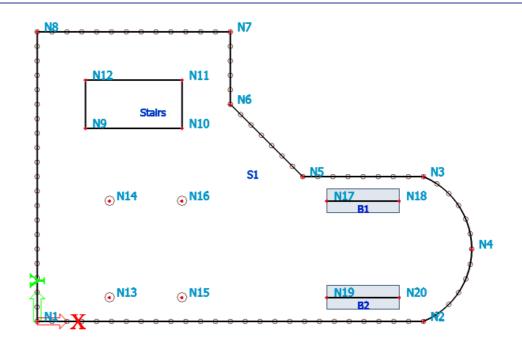
A node that is not connected to the slab is depicted as a red dot. A node that is connected to a slab is depicted as a red dot with two straight lines \mathbf{L} .

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 above the **Command line**.



The third button can visualize supports. Labels of nodes can be activated by the sixth button. Labels of bars can be activated by the seventh button.

A view in direction Z 🗄 🗞 🏟 💩 🚖 ⊨ 🔍 🔍 🔍 🔍 🖓 😵 📾 📾 🗇 🗾 shows the following:



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

Properties		1	ł x				
2D member (1)	🗖 Va	7/	0		N6	abso	
			×		N7	abso	
		<u> </u>			N8	abso	
Name	S1		^		Data		
Туре	plate (90)	-					
Analysis model	Standard	-			Opening/Panel	Stairs	
Shape	Flat				Node	N13	
Material	C30/37	*			Node	N14	
FEM model	Isotropic	-			Node	N15	
FEM nonlinear model	none	-			Node	N16	
Thickness type	constant	-			Member	B1	
Thickness [mm]	250				Member	B2	
LCS type	Standard	-			Line support on 2	Sle1	
LCS angle [deg]	0,00				Line support on 2	Sle2	
Layer	Layer1	×			Line support on 2	Sle3	
Nodes					Line support on 2	Sle4	
N1	abso				Line support on 2	Sle5	
N2	abso				Line support on 2	Sle6	
N3	abso				Line support on 2	Sle7	
N4	abso						~
N5	abso			A	ctions		
N6	abso			Т	able edit geometry		>>>

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 listed.

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 model
- Set a view direction
- Use the magnifier
- Edit view parameters through the menu View parameters

Editing the view point on the model

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 model, 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.

Remark:

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

You can also easily **zoom in** and **zoom out** with the mouse wheel. The same mouse wheel can be used to **move** the model in case you press it and hold. Double-click of the wheel zooms the structure so that it can be seen completely (the whole modelling windows is filled by the structure).

Setting a view direction with regard to the global coordinate system

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

Remark:

You can also type the letter X, Y, or Z into command line and click **<Enter>** to activate the view in desired direction.

The magnifier

- Use Lse to enlarge.
- Use <a>Image: Image: Object to decrease.
- Use 🔂 to zoom in on a window.

- Use to view the whole structure.
- Use 🔍 to zoom in on the selection of modeling entities.

Editing view parameters through the menu View parameters

Click in the graphic window on the right mouse button. The following shortcut menu appears:

ĘS	Zoom all
R	Zoom by cut out
84	Set view parameters for all
æ	Cursor snap setting
3	Print/ Preview table
	Table to Engineering report
f	Print picture
1	Picture to gallery
H	Save picture to file
Control	Copy picture to clipboard
[*]	Screenshot into Engineering report
	Live picture into Engineering report
0 ₀	Wired model in view manipulations
<mark>8</mark>	Advanced graphic setup
[]?	Coordinates info
X	Picture wizard

🖹 Note

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

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

View parameters – Entities

Graphical representation of various entities can be adjusted using the tab Structure.

From this tab page the following items are important for this project:

Style + Colour: you can specify colours by layer, by material, by cross-section, or by structural type.

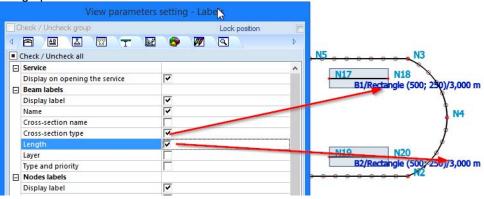
Local Axes: Using this tool the local axes can be set for nodes, 1D and 2D members.

View parameters setting - Structure				
Check / Uncheck group	Lock position			
		Þ		
Check / Uncheck all				
Service		^		
Display on opening the service				
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	-		
Effective width of plate ribs				
Draw effective width	✓			
Rendering	transparent	•		
Panel				
Member surface				
Rendering	wired	-		
Highlight supporting edges/nodes				
Load distribution symbol	✓			
Display linked members				
Structure nodes				
Display	✓			
Mark style	Dot	-		
Member parameters				
System lengths				
Member nonlinearities	v			
FEM type	v			
Joists				
Local axes				
Nodes				
Members 1D				
Members 2D				
Design groups				
Display		×		
Show names in tab	OK Apply Ca	ancel		

View parameters – Labels and description

Through the second tab **Labels** the naming 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 ticket ON, the labels will be displayed on the graphical screen.

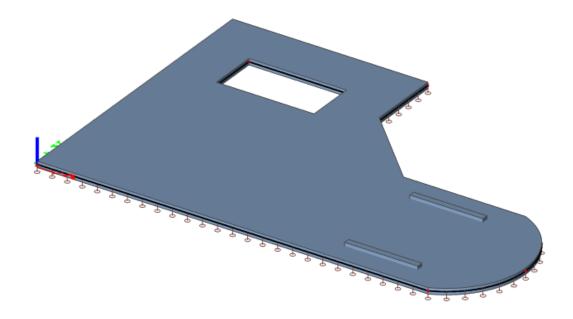


View parameters – shortcuts

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 cross-sections.
- Render geometry 🕖 to view the rendered members.
- Show/hide supports 📥 to show supports and hinges.
- Show/hide load 🌆 to show the load case.
- Show/hide other model data 📙 to show other model data (like hinges, internal nodes, ...).
- Show/hide node labels 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 🗒 to quickly access to the options from the menu View parameters.

After rendering, the following picture of the structure is obtained (Axonometric view):



Load cases and Combinations

Load Cases and Load Groups

Each load is attributed to a **load case**. One 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 single 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 - permanent load case

- LC2: Live load - variable load case

Defining a Permanent Load Case

- 1. Double-click on <u>Load</u> 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 we will also manually enter loads in the first load case of this project (surface load), you must change the Load type to **Standard**.

× 👬 🖌 🖬 🔣	🎦 ର ଜା 🚭 🖷 🖶	Al	- 8
LC1 - Dead load	Name	LC1	
	Description	Dead load	
	Action type	Permanent	
	LoadGroup	LG1	*
	Load type	Standard	
	Load type	Standard	
	Actions	Januara	
			>>>

5. In the Description field, you can describe the specification of this load case. For this project, type the description "**Dead load**".

Defining a Variable Load Case

- 1. Click 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**.

as this is a variable load, change the Action type to variable .					
	Load cases	2			
🏓 🤮 🗶 🛱 💺 🕃	- 🕰 🗠 🎒 😂 🔚 Al	• 7			
LC1 - Dead load	Name	LC2			
LC2 - Live load	Description	Live load			
	Action type	Variable			
	LoadGroup	LG2 👻 .			
	Load type	Static			
	Specification	Standard			
	Duration	Short			
	Master load case	None			
	Actions				
	Delete all loads	>>>			
	Copy all loads to another loadca	ase			
New Insert Ed	lit Delete	Close			

4. New Load group LG2 is automatically created. Click it to display the properties of the Load group.

	Load gro	oups	
A 😳 🖌 🞼	k 🗅 🗠 🖨 🖨 🔒		
LG2	Name	LG2	
	Relation	Standard	-
	Load	Variable	-
	Structure	Building	
	Load type	Cat A : Domestic	-
New Insert	Edit Delete		ОК

The **Load type** determines the partial safety factor that is attributed to the load cases in this load group for Eurocode combination. In this project **Cat A: Domestic** is selected.

- 5. Click **[OK]** to close the **Load group manager** and to return to the **Load cases manager**.
- 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 partial safety 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" to avoid simultaneous action.

Loads

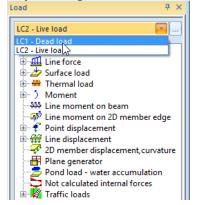
After input of the Load cases, the Loads service will automatically appear:

The first load case (LC1) includes two loads:

- Dead load
- Self weight

Switching between load cases

Activate LC1 by selecting this load case with the mouse pointer in the combo-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.

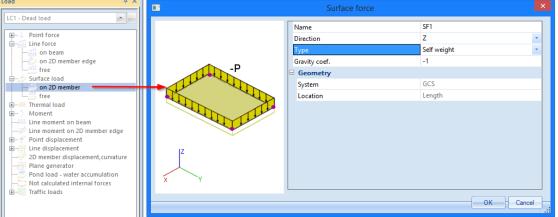
Load 4 ×	•	Line force on beam	
LC1 - Dead load Import Point force On 2D member edge free B→ Surface load On 2D member edge free B→ Surface load B→ Moment Une moment on beam Une moment on beam Une displacement Une displacement Doint displacement Diade generator Pond load - water accumulation Not calculated internal forces D→ Doint displacement Doint displacement Doint displacement Diade generator Pond load - water accumulation Not calculated internal forces D→ Doint displacement Doint displacement Diade displacement	$q_{max} \qquad q_{min} \qquad Q = \frac{q_{max} - q_{min}}{q_{max} + q_{min}}$	Name Direction Type Gravity coef. Distribution Load above joint Geometry System Location Extent Coord. definition Position x1 Position x2 Origin	LF1 Z • Self weight • -1 Uniform • n no GCS Length full Rela 0,000 1,000 From start
		 Eccentricity Eccentricity ey [m] 	0,000

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.

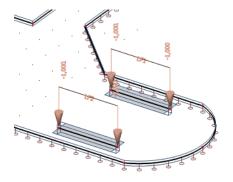
Input of the self weight of the slab as surface load

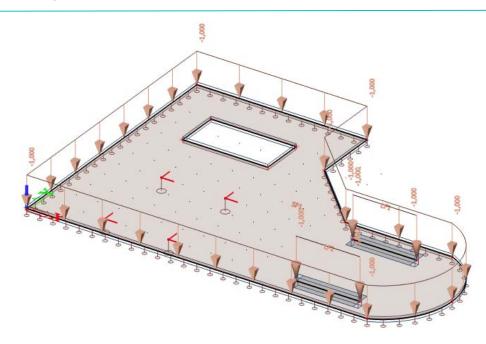
- 1. Cancel any possibly active selection by pressing **<Esc>**.
- 2. Click on **Surface load on 2D member** in the **Load** service. The following dialogue **Surface force** pops up.



- 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. Since there is only one slab in the project load is automatically put on the slab.

The self-weight is depicted by a brown colour:





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

Live load is input as free loads on a part of the slab. For the live load a different load case (LC2) will be used.

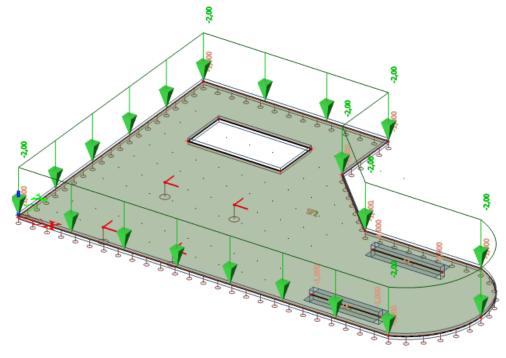
Input of dead surface loads

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

d + ×	Surface for	ie	×
	Name	SF2	
Ð ↓ Point force	Direction	Z	*
E	Туре	Force	*
	Value [kN/m^2]	-2,00	
	-P Geometry		
Surface load	System	GCS	-
free	Location	Length	*
> Moment	The second second		
Point displacement			
Line displacement	7		
	-		
	Y		
Ð Traffic loads			
		0	K 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 Z-direction have a negative value.
- 4. The Value of the surface load will be set to -2 kN/m².
- 5. Confirm your input with **[OK]**.

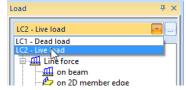
6. Since there is only one slab in the project load will be automatically put on the slab.



The self-weight is depicted by a green colour:

Switching between load cases

Activate the second load case "Live load" by selecting this load case with the mouse cursor in the combo-box:

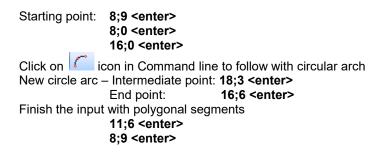


Input of live surface loads

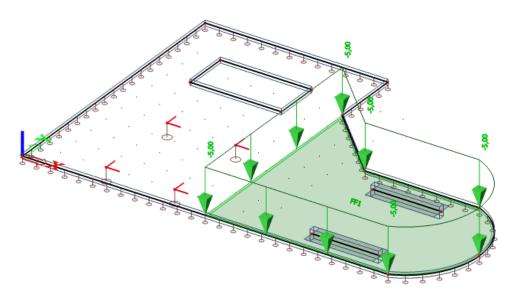
- 1. Cancel any possibly active selection by pressing **<Esc>**.
- 2. Click on **Surface load free** in the **Loads** menu. The dialogue window **Surface force free** pops up.

Load 4 ×		Surface force free		ĸ
LC2 - Live load		Name	FF1	
Point force		Direction	Z 🔹	
i⊡…∰ Line force		Туре	Force	
		Distribution	Uniform	
free free		g [kN/m^2]	-5,00	
🖃 👉 Surface load		Validity	All	
			Auto	
free Thermal load		Select	Auto	
		Geometry		
Line moment on beam		System	GCS	
		Location	Length 🔹	
🕂 🕂 🌮 Point displacement				
⊞∰ Line displacement				
2D member displacement, curvature	X			
Plane generator				
Not calculated internal forces				
		L		
			OK Cancel	

- 3. For the field **Type** the **Force** is chosen. The **Direction** is the Z-direction in the coordinate system that you have chosen in **System** here the global one (GCS). The value is –5 kN/m² and uniformly distributed over the surface.
- 4. Confirm your input with [OK].
- 5. The program asks to define the outline polygon (as free load is not applied to any particular member) of the free surface load. Type the following coordinated into Command line:

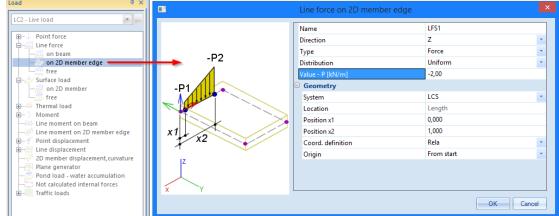


6. Click on **<Esc>** to end this function.

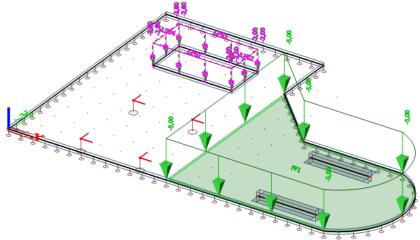


Input of variable line load

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



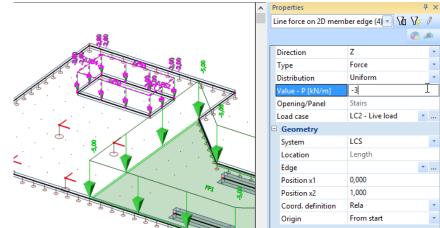
- 3. For the field **Type** the **Force** is chosen. The **Direction** is the global Z-direction. The input value P is –2.00 kN/m
- 4. Confirm your input with [OK].
- 5. Select the four edges around the opening of the staircases.
- 6. Click on **<Esc>** to end this function.



7. Click on <ESC> to cancel the selection.

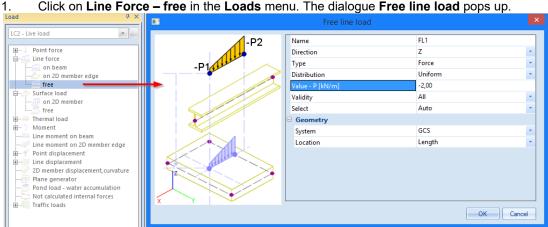
Adapting a load

- 1. Select the variable line loads around the opening by clicking with the left mouse button on these loads. The common properties of the 4 loads are displayed in the **Properties** window.
- 2. Change the Value from -2,0 kN to -3,0 kN in the Properties window.



- 3. Confirm the modification with **<ENTER>**.
- Press <ESC> to cancel the selection. 4.

Input of a free line load



Click on Line Force - free in the Loads menu. The dialogue Free line load pops up.

2. For the field Type the Force is chosen. We will enter a value of -2 kN/m. The Direction is the global Z-direction.

3. Confirm your input with [OK].

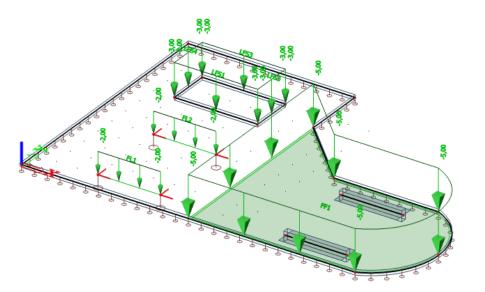
4. The dialogue window disappears and the coordinates of the new free line load have to be entered. Type the following values in Command line

1st free line load

Starting point: 3;1 <enter> End point : 6;1 <enter> Press **<ESC>** to finish the first **Polyline** command. But continue with second line load.

2n free line load

Starting point: 3:5 <enter> End point : 6:5 <enter> Press <ESC> to finish the command and press <ESC> again to finish the input completely.



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

Note:	
The Command line includes a number of predefined loads fast and simple input of some particular loads.	↓ ⊥ ⊥ ⊥ ⊥ m which enable

Combinations

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

Defining Combinations

- 1. Double-click on Combinations in the Main tree.
- 2. Since no combination has been entered yet, the window to create a new combination automatically appears.

	cases, Combinations
Ĵ⊡ L	oad Cases
	oad Groups
	Combinations
Jt‡ C	Concrete combinations
🛄 🛄 R	lesult classes

Combination - CO1				
Contents of co	mbination	List of load cases		
	:ase 1 - Dead load 2 - Live load	Coad case ▲ LC1 - Dead load ▲ LC2 - Live load		
Name :	C01	Delete Add		
Coeff :	1 Correct	Delete All Add All		
Type :	EN-ULS (STR/GEO) Set B]		
Structure:	Building			
Description :	ULS]		
Nonlinear combination :		OK Cancel		

- The Type of the combination is changed to EN ULS (STR/GEO) Set B. With this envelope combination type SCIA Engineer will automatically generate linear combinations in accordance with the complex composition rules of the Eurocode.
- 4. A warning message that controls the content of code combinations with respect to load type may appear. Close it with **[Yes]**

Warnin	ng ×	
A Content of combination may be changed! Do you want to continue?		
	<u>A</u> no <u>N</u> e	

- 5. With the button **[Add all]**, all load cases can be added to the combination. Otherwise you can manually drag&drop load cases from the list of load cases (right frame) to the contents of combinations (left frame).
- 6. Type "ULS" into Description row to distinguish the combination from the second one.
- 7. Confirm your input with [OK]. The Combinations manager is opened.
- 8. Click even or to create a second combination.
- 9. Change the **Type** of the combination to **EN-SLS Characteristic**. Type "SLS" into Description row to distinguish the combination from the first one.

- 10. Confirm your input with **[OK]**.
- 11. Click [Close] to close the Combination manager.

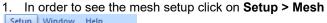
•	Combinations	×
🎜 🤮 🗶 📸	🖳 🖸 🖂 🛛 🖉 🛛 Input combinations	Υ
CO1 - ULS	Name	CO2
CO2 - SLS	Description	SLS
	Туре	EN-SLS Characteristic
	Structure	Building
	Active coefficients	
	Contents of combination	on
	LC1 - Dead load [-]	1,00
	LC2 - Live load [-]	1,00
	Actions	
	Explode to envelopes	>>>
	Explode to linear	>>>
	Show Decomposed EN comb	inations >>>
New Insert	Edit Delete	Close

Calculation and Mesh generation

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

Mesh generation

Mesh setup



	ip minden neip							
	Options							
	Geometry/Graphics							
	Delete							
	Colours/Lines							
	Fonts							
	Beam types (structural)							
	Dimension lines							
mm	Units							
1:10	Scale							
IJ	Cross-sections							
<u></u> ¶∔∔	Mesh							
<u>]</u> ++	Solver							
	Concrete solver							
	Gallery							

2. The dialogue Mesh setup pops up.

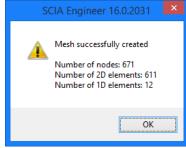
	Mesh setup	×
Name	MeshSetup1	•
General mesh settings		
Minimal distance between two points [m]	0,001	
Average number of tiles of 1d element	1	
Average size of 2d element/curved elem	0,500	
Definition of mesh element size for panels	Manual	*
Average size of panel element [m]	1,000	
Elastic mesh		
Use automatic mesh refinement		
ID elements		
Minimal length of beam element [m]	0,100	
Maximal length of beam element [m]	1000,000	
Average size of cables, tendons, element	1,000	*
Average size of 2d element/curved element		
		OK Cancel

 The Average size of 2D element/curved element [m] will be used for the mesh generation if no local mesh refinements have been defined. Change this value in 0,500 m because the default value 1 m is too coarse.

Generation of the mesh

4. Mesh is generated automatically before each calculation. However, you can create mesh manually in advance by command **Mesh generation** in **Calculation**, mesh group in **Main** tree.

- Calculation, mesh Check structure data Connect members/nodes W Mesh setup Color setup Color setup Color setup Calculation Calculation Autodesign Autodesign
- 5. The program informs you that the mesh is generated and states the number of nodes, 1D and 2D elements that was generated.



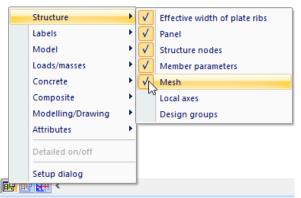
Note:

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

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

Display of the mesh

1. The mesh can be displayed using the shortcut button **Fast adjustment of view parameters on whole model** located at the bottom of the graphical screen



- 2. The precise settings can be adjusted using the menu item "**Setup dialogue**" located at the bottom of the menu on picture above.
- 3. On the tab "Structure" you can tick on/off mesh drawing

	View parameters setting - Structure									
	heck / Uncheck group	Lock position 📃								
4	🖻 😬 👗 😨 🛨	N 😰 🞯 💯 🔍 🔹 🕨								
	Check / Uncheck all									
	Highlight supporting edges/nodes	□ ^								
	Load distribution symbol									
	Display linked members									
	Structure nodes									
	Display	V								
	Mark style	Dot 🗸								
	Member parameters									
	System lengths									
	Member nonlinearities									
	FEM type									
	Joists									
	Mesh									
	Draw mesh									
	Free edges									
	Display mode	wired								
	Local axes									
	Nodes Members 1D									
	Members 2D									
	Members 2D Mesh elements									
	Design groups									
	Display									
	Uspiay									
		×								
	Show names in tab	OK Apply Cancel								

4. On the tab "Labels" different labels for the mesh can be toggled on/off.

	View parameters settin	g - Labels
	Check / Uncheck group	Lock position
4	/ 🕾 🖉 🛃 🔛 🐨 🕆 🔛 🗸	🚱 💹 🔍 ト
	Check / Uncheck all	
	Type and priority	^
	Edges	
	FEM model	
	Material	
	Profiled sheeting	
E	Mesh	
	Display label	
	Nodes	
	Elements 1D	
	Elements 2D	
E	System lengths	
	Display label	
	Name	
	Label	
E	Nonlinearities	
	Display label	
	Labels of local axes	
	Members 1D	
	Members 2D	
	General structural shape	
	Display vertex label	
		· · · · · · · · · · · · · · · · · · ·
	Show names in tab	OK Apply Cancel

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

If the mesh has not generated before the start of the calculation the programme will automatically generate the mesh itself.

Linear Calculation

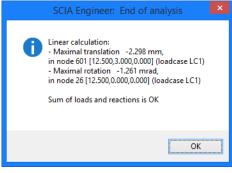
As the calculation model is completely ready you now can start the calculation.

Executing the Linear Calculation

- 1. Double-click on Calculation in the **Main window**, or use identical icon 📕 in toolbars.
 - Calculation, mesh Check structure data Connect members/nodes Wesh setup Colculation Colculation Calculation Hidden calculation Autodesign
- 2. The FE analysis window appears. Click [OK] to start the calculation.

	FE analysis	
	Single analysis Batch analysis	
	Linear calculation	
	O Nonlinear calculation	
	O Modal analysis	
2.	C Linear stability	
11	Concrete - Code Dependent Deflections (CDD)	
133	Construction stage analysis	
	O Nonlinear stage analysis	
	O Nonlinear stability	
	○ Test of input data	
	Number of load cases: 2	
all the second		
	Solver setup Mesh setup	
Barry 1		
	OK Cancel	

3. After the calculation, a window announces that the calculation is finished and the maximum deformation and rotation for the normative load case is shown. Click **[OK]** to close this window.



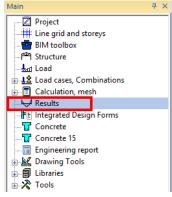
Results

Viewing results

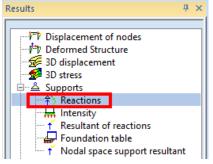
After the calculation is executed, the results can be viewed.

Viewing the Reaction Forces

1. Double-click on Results in the Main window. The Results menu appears.



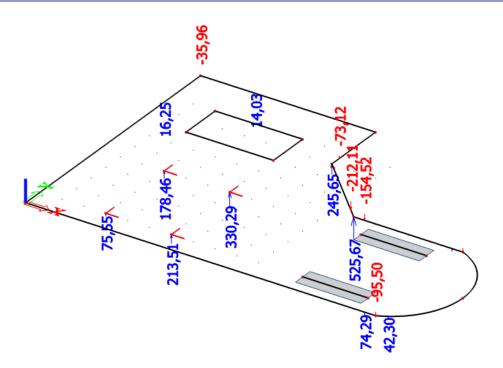
2. In chapter Supports click Reactions.



- 3. The options in the **Properties** window are configured in the following way:
 - Selection field is set to All.
 - Type of load is set to Combinations and the Combination to CO1 ULS.
 - Values are wanted for Rz.
 - **Extreme** field is changed to **Node**.

Properties	4 ×
Reactions (1)	🧧 Va V/ 🖉
	💞 🍂
Name	Reactions
Selection	All
Type of loads	Combinations 🔹
Combinations	CO1 - ULS 🔹
Filter	No
Values	Rz 💌
Extreme	Node 🔹
Drawing setup 1D	
Rotated supports	

4. The action button **Refresh** has a red background, i.e. the graphical screen must be refreshed. Click on the button next to **Refresh** to display the results in the graphical screen in accordance with requested properties.



5. To display these results in a table form, the **Preview** action button is used. Click on the

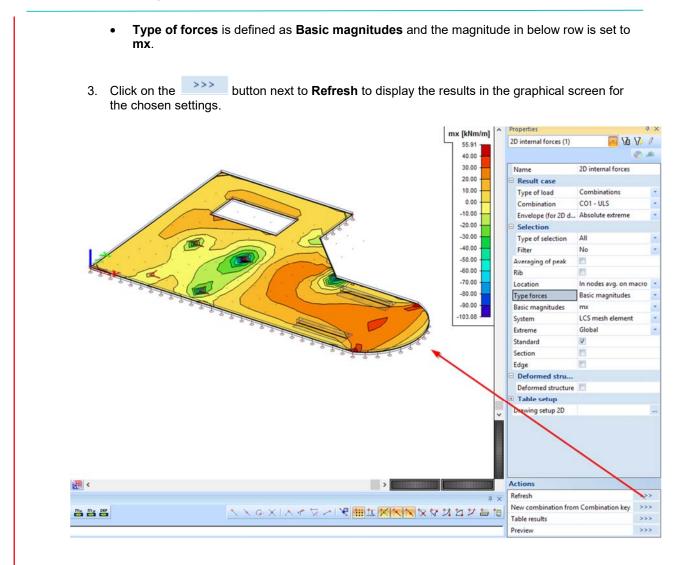
wiew 🗟 💽 Defau	t		- 1 🖽		
Reacti Linear calcu Selection : Combination	alation, Ext	treme : No	ode		
Support	Case	dx [m]	Rz [kN]	Mx [kNm]	My [kNm]
Sn1/N14	C01/1		128,26	0,00	0,00
Sn1/N14	C01/2		178,46	0,00	0,00
	C01/3	[]	173,15	0,00	0,00
Sn1/N14					
Sn1/N14 Sn2/N13	C01/4	i i	44,43	0,00	0,00
	C01/4 C01/3		44,43 75,55	0,00 0,00	0,00
Sn2/N13					0,00
Sn2/N13 Sn2/N13	CO1/3		75,55	0,00	
Sn2/N13 Sn2/N13 Sn3/N15	CO1/3 CO1/1		75,55 123,54	0,00 0,00	0,00
Sn2/N13 Sn2/N13 Sn3/N15 Sn3/N15	C01/3 C01/1 C01/2		75,55 123,54 213,51	0,00 0,00 0,00	0,00 0,00 0,00

Note:

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

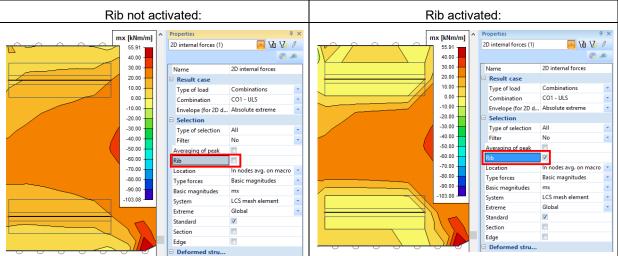
Viewing internal forces on 2D elements

- 1. Click on 2D Members (beta) > 2D internal forces in Results service
- 2. The options in the **Properties** window are configured in the following way:
 - Type of load is set to Combinations and Combination to CO1 ULS
 - Type of selection field is set to All



Results for (individual) ribs

1. By clicking on the check box '**Rib**' in Properties window, the results will be adjusted in order to take into account the stiffness of the complete T-section (rectangular cross-section + slab segment)



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 next to **Drawing setup 2D** and the various options for the graphical representation appear.

	2D results display	×
S4 S3 S2 S1 S2 S2	Display	Minimum and maximum settings Ground value Use value Draw isoline Draw isoline Local extrems None Style Text with cross Style Styl
S ₁	Advanced settings Automatic palette values - rounded	QK Cancel Help

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

Isobands properties							
Number of isobands 16 • 16	Palette colours 55.91 40.00 30.00 20.00						
Isoband contours	10.00 · · · · · · · · · · · · · · · · · ·						
Predefined palette colours	-20.00 -30.00						
Dark rainbow Greyscale	-50.00						
Palette values Normal	-70.00						
Rounded From parameters	-103.08						
	cel <u>H</u> elp						

4. Click [OK] to accept the settings or [Cancel] to ignore the selected settings.

- 5. Click in the **Property Window**, on the button next to **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

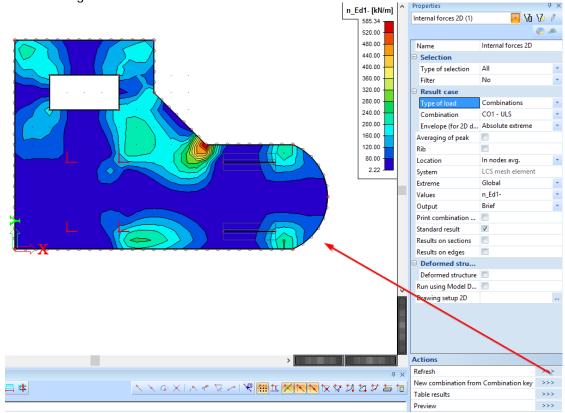
The reinforcement design can be performed in the Concrete 15 menu.

Double-click on Concrete 15 in the Main window. The following menu appears

Viewing design internal forces

- 1. Click on Reinforcement design > 2D Members > Internal forces in Concrete 15 service
- 2. The options in the **Properties** window are configured in the following way:
 - Type of selection field is set to All
 - Type of load is set to Combinations and Combination to CO1 ULS
 - Values is defined as n_Ed1- to see design force in reinforcement parallel to local X axis on bottom edge of the plate.

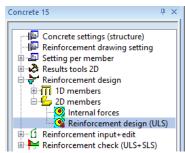
3. Hit the button next to **Refresh** to display the results in the graphical screen for the chosen settings.



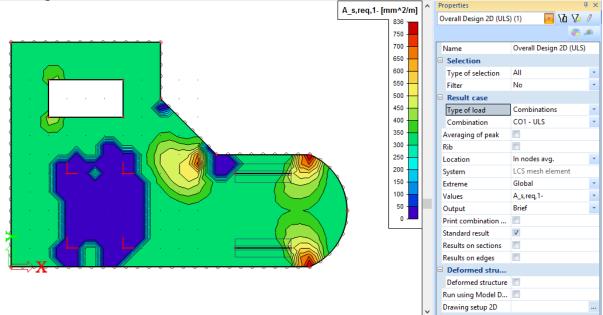
•

Reinforcement design ULS

1. Select Reinforcement design (ULS) in the Concrete 15 menu:



- 2. The options in the Properties window are configured in the following way:
 - Type of selection field is set to All.
 - Type of load is set to Combinations and combination which is selected is CO1 ULS
 - Values is set to **A_s,req,1-** to see amount of reinforcement in bottom layer parallel to local X axis.
- 3. Hit the button next to **Refresh** to display the results in the graphical screen for the chosen settings.



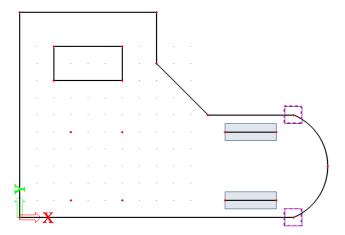
4. Change **Output** option to **Standard** in Properties and click on **Preview** action button to see onepage text output with the most important data.

Membe				Type	Dista	[h-25/	0	1		^	Name	Overall D
						[h=250				6	Selection	
EC EN 1992-1-1				Location	Node 2/1 [)	(=16m, Y=0m	n, Z=0m]				Type of selection	All
	-										Filter	No
Member dat Concrete:		20.440-			_					(Result case	
Concrete:	C30/37 f _{ck} =	= 30 MPa = 20 MPa									Type of load	Combin
		= 20 MPa a = 3.5 %	\leq	+ +		[2+]\$10/25	~				Combination	CO1 - U
Deleterer	ecu3 t: B 500A fyk =		- F	4							Averaging of peak	
Remorcement		= 500 MPa = 434.78 MF	$a \rightarrow x$		\rightarrow						Rib	
	,	= 2.174 %	- I I I I I I I I I I I I I I I I I I I	1		12_1610/220					Location	In node
Cover:	upper: c _{nom} =		\leq	- + = =	=	[1-]φ10/20					System	LCS me
	lower: cnom =					(1-)ψ10/30					Extreme	Global
Desian force											Values	A_s, req
	1,4 kN/m m _{Ed} = 1,4 kN/m m _{Ed} =)1,4 kN/m m _{Ed} =	6,8 kNm/m	[CO1/1]								Standard result Results on sections	
[2-]: n _{Ed} = 20 [2+]: n _{Ed} = 20 [CO1/1]: 1.35	1,4 kN/m m _{Ed} = 01,4 kN/m m _{Ed} = 5*LC1+1.50*LC2	6,8 kNm/m 6,8 kNm/m	[CO1/1]							C	Results on sections Results on edges Deformed stru	
[2-]: n _{Ed} = 20 [2+]: n _{Ed} = 20 [CO1/1]: 1.35	1,4 kN/m m _{Ed} =)1,4 kN/m m _{Ed} =	6,8 kNm/m 6,8 kNm/m	[CO1/1]					1		Q	Results on sections Results on edges Deformed stru Deformed structure	
[2-]: n _{Ed} = 20 [2+]: n _{Ed} = 20 [CO1/1]: 1.35	1,4 kN/m m _{Ed} = 01,4 kN/m m _{Ed} = 5*LC1+1.50*LC2	6,8 kNm/m 6,8 kNm/m	[CO1/1]	A _{sreq} [mm ² /m]	A _{smin} [mm²/m]	A _{s.max} [mm ² /m]	A _{sprov} [mm ² /m]			Q	Results on sections Results on edges Deformed stru	
[2-]: n _{Ed} = 20 [2+]: n _{Ed} = 20 [CO1/1]: 1.35 Longitudina	1.4 kN/m m _{Ed} = 01.4 kN/m m _{Ed} = 5*LC1+1.50*LC2 11 reinforcen	6,8 kNm/m 6,8 kNm/m nent Type	[CO1/1] [CO1/1]							Q	Results on sections Results on edges Deformed structure Run using Model D	
[2–]: n _{Ed} = 20 [2+]: n _{Ed} = 20 [CO1/1]: 1.35 Longitudina Layer	14 kN/m med = 01.4 kN/m med = 3*LC1+1.50*LC2 al reinforcen Direction	6,8 kNm/m 6,8 kNm/m nent Type	[CO1/1] [CO1/1] Case	[mm ² /m]	[mm²/m]	[mm ² /m]	[mm²/m]			Q	Results on sections Results on edges Deformed structure Run using Model D	
[2–]: n _{Ed} = 20 [2+]: n _{Ed} = 20 [CO1/1]: 1.35 Longitudina Layer Upper [2+]	1.4 kN/m m _{Ed} = 01.4 kN/m m _{Ed} = 5*LC1+1.50*LC2 al reinforcen Direction Second [90°]	6,8 kNm/m 6,8 kNm/m nent Type Principal Principal	[CO1/1] [CO1/1] Case CO1/1	[mm ² /m] 309 (134)	[mm ² /m] 309	[mm ² /m] 1671	[mm ² /m] 314 (ф10/250)				Results on sections Results on edges Deformed structure Run using Model D	
[2-]: n _{Ed} = 20 [2+]: n _{Ed} = 20 [CO1/1]: 1.35 Longitudina Layer Upper [2+] Lower [1-] Lower [2-] A _{1.req} -statically	1.4 kN/m m _{Ed} = 1.4 kN/m m _{Ed} = 1.4 kN/m m _{Ed} = 1.4 kN/m content 1.4 kN/m m _{Ed} = 1.4 kN/m m _E	6,8 kNm/m 6,8 kNm/m nent Type Principal Principal Principal ment including	[CO1/1] [CO1/1] [CO1/1] CO1/1 CO1/1 CO1/1 g detailing prov	[mm ² /m] 309 (134) 841 329 visions, A _{s.min} - m	[mm ² /m] 309 324 309 inimal reinforcer	[mm ² /m] 1671 1671 1671	[mm ² /m] 314 (ф10/250) 873 (ф10/90)				Results on sections Results on edges Deformed structure Run using Model D	
[2-]: n _{Ed} = 20 [2+]: n _{Ed} = 20 [CO1/1]: 1.35 Longitudina Layer Upper [2+] Lower [1-] Lower [2-] A _{1.req} -statically	14 kN/m med = 14 kN/m med = 14 kN/m med = *LC1+1.50*LC2 1 reinforcen Direction Second [90°] First [0°] Second [90°] required reinforcer	6,8 kNm/m 6,8 kNm/m nent Type Principal Principal Principal ment including	[CO1/1] [CO1/1] [CO1/1] CO1/1 CO1/1 CO1/1 g detailing prov	[mm ² /m] 309 (134) 841 329 visions, A _{s.min} - m	[mm ² /m] 309 324 309 inimal reinforcer	[mm ² /m] 1671 1671 1671	[mm ² /m] 314 (ф10/250) 873 (ф10/90) 341 (ф10/230)	ge width 🕞 —			Results on sections Results on edges Deformed structure Run using Model D Drawing setup 2D Actions	
[2-]: n _{Ed} = 20 [2+]: n _{Ed} = 20 [CO1/1]: 1.35 Longitudina Layer Upper [2+] Lower [1-] Lower [2-] A _{1.req} -statically	14 kN/m med = 14 kN/m med = 14 kN/m med = *LC1+1.50*LC2 1 reinforcen Direction Second [90°] First [0°] Second [90°] required reinforcer	6,8 kNm/m 6,8 kNm/m nent Type Principal Principal Principal ment including	[CO1/1] [CO1/1] [CO1/1] CO1/1 CO1/1 CO1/1 g detailing prov	[mm ² /m] 309 (134) 841 329 visions, A _{s.min} - m	[mm ² /m] 309 324 309 inimal reinforcer	[mm ² /m] 1671 1671 1671	[mm ² /m] 314 (φ10/250) 873 (φ10/90) 341 (φ10/230) ing provisions, A s.max -	ge width 🕞 —) ×	Results on sections Results on edges Deformed structure Run using Model D Drawing setup 2D Actions Refresh	
[2-]: n _{Ed} = 20 [2+]: n _{Ed} = 20 [CO1/1]: 1.35 Congitudina Layer Upper [2+] Lower [1-] Lower [2-] A _{1:Req} - statically maximal reinfore	14 kN/m med = 14 kN/m med = 14 kN/m med = *LC1+1.50*LC2 1 reinforcen Direction Second [90°] First [0°] Second [90°] required reinforcer	6.8 kNm/m 6.8 kNm/m nent Type Principal Principal Principal ment including ing provisions	[CO1/1] [CO1/1] [CO1/1] CO1/1 CO1/1 CO1/1 CO1/1 Q detailing prov. A sprov = prov	[mm ² /m] 309 (134) 841 329 visions, A _{s.min} - m	[mm ² /m] 309 324 309 inimal reinforcer it	[mm ² /m] 1671 1671 1671 1671 nent from detail	[mm ² /m] 314 (φ10/250) 873 (φ10/90) 341 (φ10/230) ing provisions, A s.max -		0 • •	×	Results on sections Results on edges Deformed structure Run using Model D Drawing setup 2D Actions	

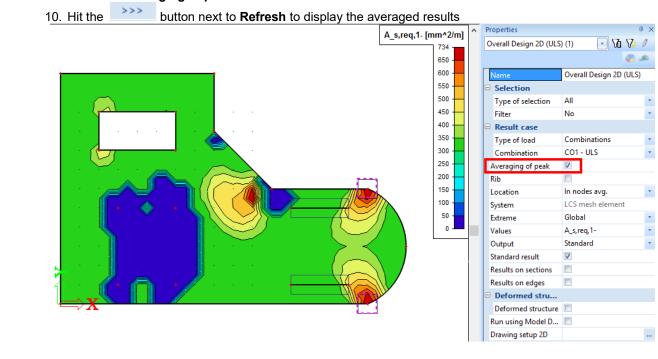
5. FEM analysis gives peak values in places of singularities (e.g. local node support, plate edge, ...) Such extremes reflect in reinforcement design. Input averaging strips that can handle that and lower the unrealistic peak values. Click on **Results tools 2D > Averaging strip**:

ncrete 15 🛛 🕂 🗙	•	RS	
Concrete settings (structure)	Name	RS1	
Setting per member	Туре	Point	
Results tools 2D	Width [m]	1,000	
Averaging strip	Length [m]	1,000	
Reinforcement design	Angle [deg]	0,00	
	Direction	both	
- Control (ULS) Reinforcement input-edit Reinforcement input-edit Reinforcement check (ULS+SLS)			OK Cancel

- 6. Set Type to Point, Width and Length to 1,000 m and Direction change to both.
- 7. Click [OK] and select nodes N2 and N3 with mouse cursor:



- 8. End the command with <ESC> key.
- 9. Select **Reinforcement design (ULS)** in the Concrete 15 menu and in **Properties** windows tick checkbox **Averaging of peak**.



Input of practical reinforcement

1. Go to **Reinforcement input + edit > Reinforcement 2D** in the Concrete 15 menu:

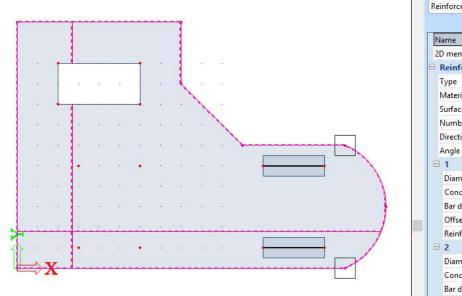
Concrete 15	• ×	Reinforcement 2D						
Concrete settings (structure) Reinforcement drawing setting Results tools 2D Reinforcement design In the Reinforcement design Reinforcement 2D Reinforcement 2D Reinforcement 2D Reinforcement check (ULS+SLS)		Name 2D member 2D member Reinforcement Type Material Surface Number of directions Direction closest to surface Angle of first direction [deg] I Diameter (dl) [mm] Concrete cover (cl, cu) [mm] Actions Load from setup	RR1 S1 Bars B 500A Lower 2 1 0,00 10,0 30 200					
			OK Cancel					

- 2. The parameters in the Reinforcement 2D window are configured in the following way:
 - Type field is set to Bars.
 - You can define lower and upper Surface layer independently. Select Lower here.
 - The Diameter (dl) of layer 1 reinforcement is 10 mm.
 - Concrete cover leave as default 30 mm.
 - Define Bar distance as 150 mm.
 - The Diameter (dl) of layer 2 reinforcement is also 10 mm.
 - Define **Bar distance** as 150 mm for layer 2 too.

- 3. Click **OK** to confirm that setting and define the polygonal reinforcement area.
- 4. Hit the button **Select existing polygon/polyline** in **Command line** and click on the slab polygon with mouse cursor.

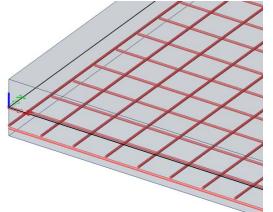


5. Press <ESC> key and two lines, representing two reinforcement directions appear. Properties of the reinforcement can be checked in **Properties** window."



Reinforcement 2D (1)	- Va V/ /
	a, 39
Name	RR1
2D member	S1
Reinforcement	
Туре	Bars
Material	B 500A 🔹
Surface	Lower
Number of directi	2
Direction closest t	1
Angle of first direc	0,00
1	
Diameter (dl) [m	10,0
Concrete cover (30
Bar distance (sl) [150
Offset [mm]	0
Reinf. area [mm	524
E 2	
Diameter (dl) [m	10,0
Concrete cover (40
Bar distance (sl) [150
Offset [mm]	0
Reinf. area [mm	524
Total weight [kg]	1240,6
Geometry	
Geometry defined	Polygon

- 6. Press **<ESC>** to cancel the selection.
- 7. Go to View parameters setting by using the right mouse click and command Set view parameters for all, and in tab Concrete choose the following:
 - **Reinforcement drawing type** is set to **3D**.
 - **Display style** is changed to **Real positions**.
- 8. Press OK and zoom in the structure to see the effect.



	View parameters se	tung - concrete
	heck / Uncheck group	Lock position
4 /		2 🔗 🌌 🔍 🕨
E.	Check / Uncheck all	
	Service	
	Display on opening the service	v
E	Concrete + reinforcement	
	Display	
	Member data	v
	SaT detail data	V
	Drawing directions for design	
	Color of reinforcement	normal 🔹
	Reinforcement drawing type	3D -
	Rounded bends	
	Concrete labels	
	Display label	V
	Name	v
	User defined reinforcement	
	Diameter	
	Material	
	Cover	
	Environment class	
	Reinforcement regions 2D	
	Display Display style	Real positions
	Upper layer	
	Lower layer	
	Display label	
	Name	
	Diameter+distance	·
	Show names in tab	OK Apply Cancel
		Cancer

Document

In this final part of the tutorial, we will explain how to make nice report of the calculation and design.

Engineering Report

1. Double-click Engineering report in the Main Window or click in the toolbar. Because no report was created before, **Report_1** directly appears as a new application. This application is in a certain way independent on the SCIA Engineer application. That is significant also in the Windows main bar



2. Click **Insert** button in the ribbon to start inputting items in the report navigator. Windows with **New items** appears just below the **Insert** icon:



- 3. Using this window, various data can be added to the report.
 - Open the Libraries group and select Materials. Double-click in this item or hit item to the document navigator.
 - Add also Cross-Sections one row above.
 - Open the **Structure** group and double-click **2D members**.
 - Open the Results group and click 2D results > 2D internal forces.
- 4. You can directly see these items in the Navigator and on the paper preview as well:

avigator	4 × 1	lew items	4 ×			
Materials	• •	₩ ≪ 飞船带目前	R			
Cross-sections	₽ ⊙			1. Materials		
2D members	₽ 0 <mark>-</mark>	Supports on 2D men	abox of a	Concrete EC2		
2D internal forces		- Load	inder ec. A	Name Type	ρ E _{mod} [kg/m³] [MPa]	μ α f _{c.k.28} Colour [m/mK] [MPa]
		Construction stages Results		C30/37 Concrete	2500,0 3,2800e+04	
		Deformed Structure		Reinforcement EC2		
		3D displacement		Name Ty		E _{mod} G _{mod} a
		- 3D stress		B 500A Reinforcem	[kg/m ³] ent steel 7850,0	[MPa] [MPa] [m/mK] [2,0000e+05 8,3333e+04 0,00
		Internal forces on be Deformation on bea	cam	b boort preimoreen	1000/0	
		Displacement of noc		2. Cross-secti	ons	
		- Acceleration of node	ar in the second s	C51	0115	
		- Reactions		Туре	Rectangle	
		 Resultant of reaction 		Detailed	500; 250	
		Nodal space suppor	t result	Shape type	Thick-walled	
		- Intensity		Item material	C30/37	
		Member Stress		Fabrication	concrete	
		Shear in joint		Colour		

Drag the items with the mouse to change their order if needed.

Displaying results in the document

- In the Navigator click 2D internal forces. The red exclamation mark both in Navigator and preview indicates that the values presented are not up-to-date. In the Properties window the setting of this table is displayed. Parameters for displaying the results in the Engineering Report are configured in the same way as the parameters for viewing the results in the Results Menu of the SCIA Engineer application.
 - Selection type is set to All.
 - Type of load is set to Combinations and the Combination to CO1 ULS.
 - Extreme field is changed to Global.
 - Table setup let all values (from mx to vy) be ticked



2. Click the selected button in the top ribbon to display the table in accordance with the predefined options. Red exclamation mark disappears.

lavigator	4×	New items 4 ×
Materials	£ 0	计传输管理 🗋
Cross-sections	• •	
2D members	₽ 0	
2D internal forces	0	
		E- Results
		- Deformed Structure
		- 3D displacement
		3D stress
		Internal forces on beam
		Deformation on beam
		Displacement of nodes
		- Acceleration of nodes
		Reactions
		Resultant of reactions
		Nodal space support result
		Intensity
		Member Stress
		Shear in joint Relative deformation
		Bill of material
		- Connection Forces
		- Foundation table
		Displacement of nodes - m
		Member2D - Internal Force:
		Member2D - Stresses
		Member2D - Contact stress
		Subsoil - C parameters
		Subsoil - Other data
		⊕ 1D results (beta)
		⊡ · 2D results (beta)
		2D internal forces
		2D stress/strain
		2D displacement
		2D contact stresses

4. 2D internal forces

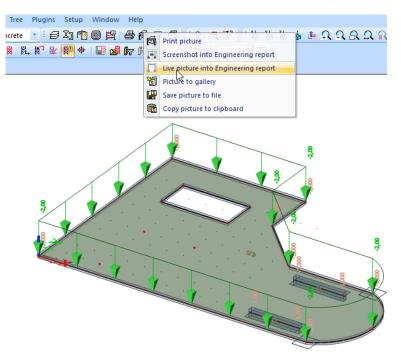
Linear calculation Combination: CO1 Selection: All Location: In nodes avg. on macro. System: LCS mesh element Basic magnitudes Extreme: Global

Name	Mesh	Position [m]	Case	mx [kNm/m] my [kNm/m]	mxy [kNm/m]	vx [kN/m] vy [kN/m]
S1	Element: 3 Node: 2	16,000 0,000 0,000	CO1/1	55,91 3,57	16,74	60,0 -132,9
S1	Element: 541 Node: 20	6,000 5,000 0,000	CO1/1	-93,62 - 105,06	23,59	-276,8 282,7
S1	Element: 519 Node: 601	12,500 3,000 0,000	CO1/1	22,21 62,45	-0,66	2,3 -2,4
S1	Element: 167 Node: 168	7,000 0,500 0,000	CO1/1	-1,11 -2,93	-25,88	61,7 -2,9
S1	Element: 20 Node: 5	11,000 6,000 0,000	CO1/1	-81,57 -54,72	56,50	-127,9 -113,5
S1	Element: 21 Node: 209	10,496 5,920 0,000	CO1/1	-3,37 -52,09	28,03	- 309,8 122,1
S1	Element: 19 Node: 58	11,500 6,000 0,000	CO1/1	-33,04 9,23	2,01	318,8 192,7
S1	Element: 593 Node: 20	6,000 5,000 0,000	CO1/1	- 103,08 -95,64	27,67	290,3 - 282,2
S1	Element: 567 Node: 20	6,000 5,000 0,000	CO1/1	-100,61 -97,93	-13,47	290,1 283,6

Adding an image to the Report

- 1. Any picture from SCIA Engineer application can be set to Engineering Report. Either as printscreen (that is unchanged for ever) or as live picture (that can be regenerated and is always up-to date).
- Prepare any scene in the 3D modelling window, for example the analytic model with loads. You can use the icons above the **Command line** to hide surfaces and rendering and show loads:

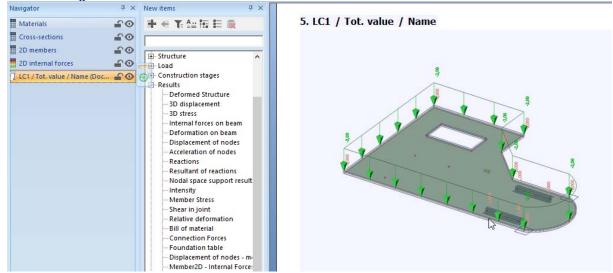
 Image: Ima
- 3. Click on button Print Picture in toolbars and select Live picture into Engineering Report



 Document picture properties dialog is opened. Here you can arrange the picture caption, scale, size etc. Use button Two at page at the top ribbon and click on button Insert & Close into selected report

Insett Close One ad Prove page Provepage Provepage Pr	Document picture - Insert objects	to Engineering report Inbox	2	
Picture size definition Two at page Picture size definition Two at page Automatic scale to fit size Scale 1: 104,596767804259 Stretch mode Dark lines Rendering Standard Antialiasing quality None Rotation None Result legend Right Export to PDF as 3D Position One below another Load units in regen. (related to objects created in picture editor) Draw inactive members as is in the window Settings of activity Text scale factor 1 Charset of texts Western European, UK, USA (Windows-1252) Display GCS icon To picture corner Performance Set as non-editable	Meert Insert Insert Close One at Two at page width	1 2 1 4 1 2 1 Save ▼		
Picture size definition Two at page • Automatic scale to fit size • • Scale 1: 104,596767804259 • Stretch mode Dark lines • Rendering Standard • Antialiasing quality None • Rotation None • Result legend Right • Export to PDF as 3D • • Position One below another • Load units in regen. (related to objects created in picture editor) • • Draw inactive members as is in the window • Settings of activity 1 >>>> Charset of texts Western European, UK, USA (Windows-1252) • Line pattern length 3 • • Display GCS icon To picture corner • >>>>	Cantion	I C1 / Tot. value / Name		
Automatic scale to fit size Scale 1: Scale 1: Stretch mode Dark lines Standard Stand			-	
Scale 1: 104,596767804259 Stretch mode Dark lines • Rendering Standard • Antialiasing quality None • Rotation None • Result legend Right • Export to PDF as 3D • • Position One below another • Load units in regen. (related to objects created in picture editor • • Draw inactive members as is in the window • Settings of activity • • Charse of texts Western European, UK, USA (Windows-1252) • Line pattern length 3 • • Display GCS icon To picture corner • • Set as non-editable ● ● >>>>				
Rendering Standard • Antialising quality None • Result legend Right • Result legend Right • Export to PDF as 3D • • Position One below another • Load units in regen. (related to objects created in picture editor • • Load activity in regen. • • Draw inactive members as is in the window • Settings of activity • • Charset of texts Western European, UK, USA (Windows-1252) • Line pattern length 3 • Display GCS icon To picture corner • Performance >>>> >>>>		104,596767804259		
Antialising quality None Antialising quality None Rotation None Result legend Right Export to PDF as 3D Position One below another Load units in regen. (related to objects created in picture editor Load activity in regen. Load activity in regen. Load activity in regen. Intervention as is in the window Settings of activity Text scale factor Charset of texts Vestern European, UK, USA (Windows-1252) Line pattern length Jacob Performance Performance None None None None None None None None None None None None None	Stretch mode		-	
Antialising quality None Rotation None Result legend Right Export to PDF as 3D Position One below another Coad units in regen. (related to objects created in picture editor Oraw inactive members as is in the window Settings of activity Text scale factor Charset of texts Western European, UK, USA (Windows-1252) Line pattern length 3 Siplay GCS icon To picture corner Performance Setting sol	Rendering	Standard	-	
Rotation None v Result legend Right v Export to PDF as 3D Image: Second Se	-	None	-	
Export to PDF as 3D Image: Constraint of PDF as 3D Position One below another Image: Constraint of PDF as 3D Load units in regen. (related to objects created in picture editor Image: Constraint of PDF as 3D Image: Constraint of PDF as 3D Draw inactive members as is in the window Image: Constraint of PDF as 3D Image: Constraint of PDF as 3D Draw inactive members as is in the window Image: Constraint of PDF as 3D Image: Constraint of PDF as 3D Text scale factor 1 Image: Constraint of PDF as 3D Image: Constraint of PDF as 3D Display GCS icon To picture corner Image: Constraint of PDF as 3D Image: Constraint of PDF as 3D Image: Performance Set as non-editable >>>>		None	+	
Position One below another Coad units in regen. (related to objects created in picture editor Image: Image: Ima	Result legend	Right		
Load units in regen. (related to objects created in picture editor Load activity in regen. Draw inactive members Settings of activity Text scale factor Charset of texts Line pattern length Display GCS icon Performance Set as non-editable Set as non-editable	Export to PDF as 3D			
Load activity in regen. Draw inactive members as is in the window Settings of activity Text scale factor 1 Charset of texts Western European, UK, USA (Windows-1252) Line pattern length 3 Display GCS icon Performance Set as non-editable	Position	One below another	*	
Draw inactive members as is in the window * Settings of activity Image: Setion of texts >>> Charset of texts Western European, UK, USA (Windows-1252) * Line pattern length 3 * Display GCS icon To picture corner * Performance * * Set as non-editable >>>	Load units in regen. (related to objects created in picture editor			
Settings of activity Image: Constraint of the constrai	Load activity in regen.			
Text scale factor 1 Charset of texts Western European, UK, USA (Windows-1252) Line pattern length 3 Display GCS icon To picture corner Performance Set as non-editable	Draw inactive members	as is in the window	Ψ.	
Charset of texts Western European, UK, USA (Windows-1252)	Settings of activity	>>	·>	
Line pattern length 3 Display GCS icon To picture corner Performance Set as non-editable >>>	Text scale factor	1		
Display GCS icon To picture corner Performance Set as non-editable >>>	Charset of texts	Western European, UK, USA (Windows-1252)	Ψ.	
Performance Set as non-editable	Line pattern length	3		
Set as non-editable >>>	Display GCS icon	To picture corner	Ψ.	
	Performance			
Settings >>>	Set as non-editable	>>	·>	
	Settings	>>	·>	

5. Switch to Engineering Report application again and see the picture as the very last item in the navigator:



Printing Engineering Report

Once the report is completed you can print it or export into various formats (e.g. PDF, RTF, HTML) by clicking the top left button of the window.

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 concrete calculations we refer to the Advanced Concrete Training documentation or the dedicated Web help chapters.