

Port House Antwerp - © Mouton cvba

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.

Contents

General Information	1
Welcome	1
SCIA Engineer Support	1
Websites	1
Introduction	2
Getting started	3
Starting a project	3
Project management	5
Save, Save as, Close and open	5
Saving a project.....	5
Closing a project.....	5
Opening a project.....	5
Start project manger.....	5
Geometry input	6
Input of the geometry	6
Geometry.....	6
Supports.....	12
Check Structure data	15
Checking the structure.....	15
Connected entities.....	16
Graphic representation of the structure	18
Load cases and Combinations	22
Load Cases and Load Groups	22
Defining a Permanent Load Case.....	22
Defining a Variable Load Case.....	22
Loads	24
Combinations	31
Calculation and Mesh generation	34
Mesh generation	34
Linear Calculation	37
Results	38
Viewing results	38
Reinforcement design	43
Viewing design internal forces	43
Reinforcement design ULS	44
Input of practical reinforcement	46
Document	48
Engineering Report.....	48
Epilogue	53

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

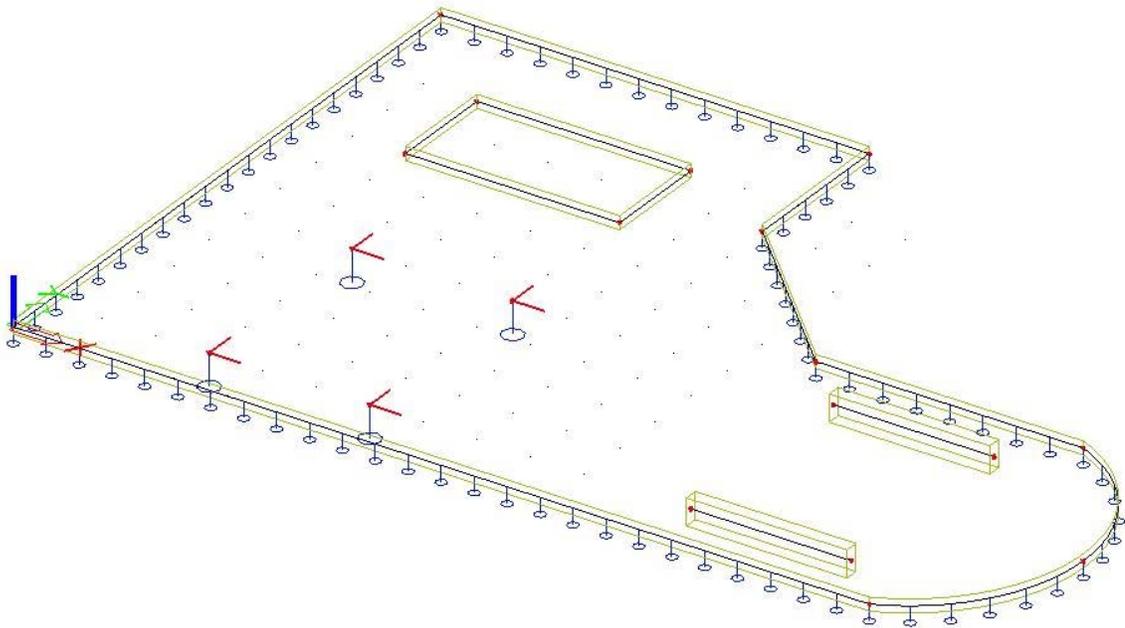
This Tutorial describes the basic functions of SCIA Engineer, the input, analysis and design of a concrete slab.

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.

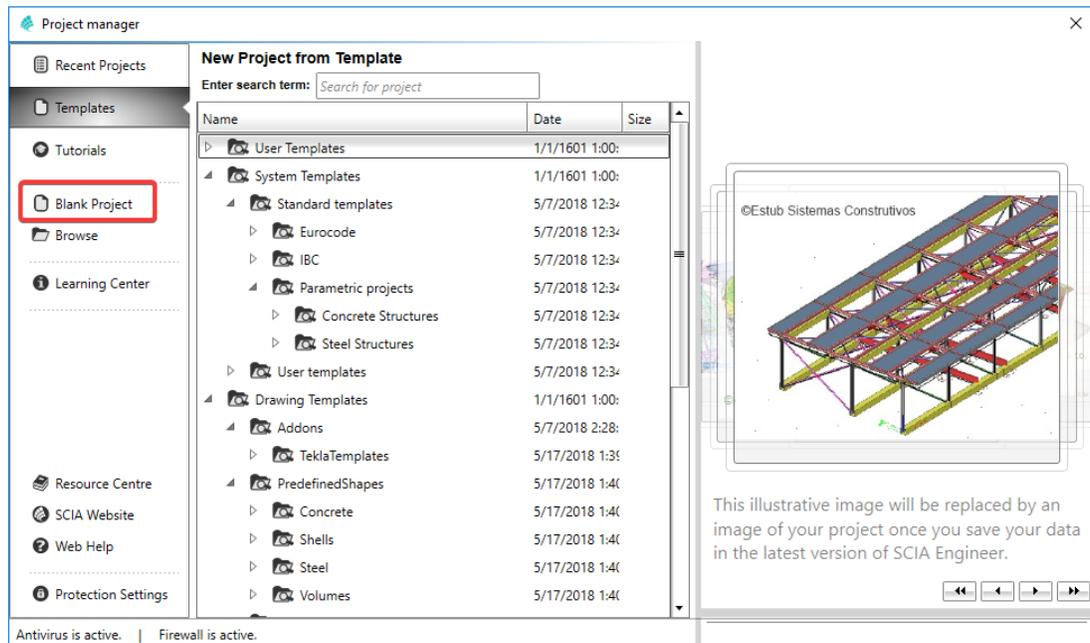
1. Double-click on the SCIA Engineer shortcut in the Windows Desktop, or
2. If the shortcut is not installed, click **[Start]** and choose **All apps > SCIA Engineer 18.0 > SCIA Engineer 18.0**.

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  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 | Functionality | Actions | Unit Set | Protection

Data

Name: -

Part: -

Description: -

Author: SCIA

Date: 17.05.2018

Material

Concrete

Material C25/30

Reinforcement ... B 500B

Steel

Masonry

Aluminium

Timber

Steel fibre conc...

Other

Code

National Code: EC - EN

National annex: Standard EN

Structure: [Post processing environment](#)

Plate XY v17

Model: One

"Thick-walled" Concrete cross-sections: the advanced 2D FEM method is off!

OK 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 **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.*

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

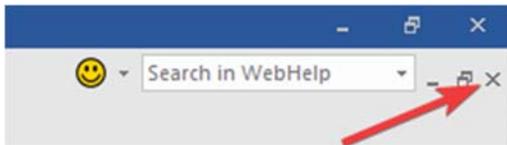
Click on  in the toolbar or press **Ctrl+S**.

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  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

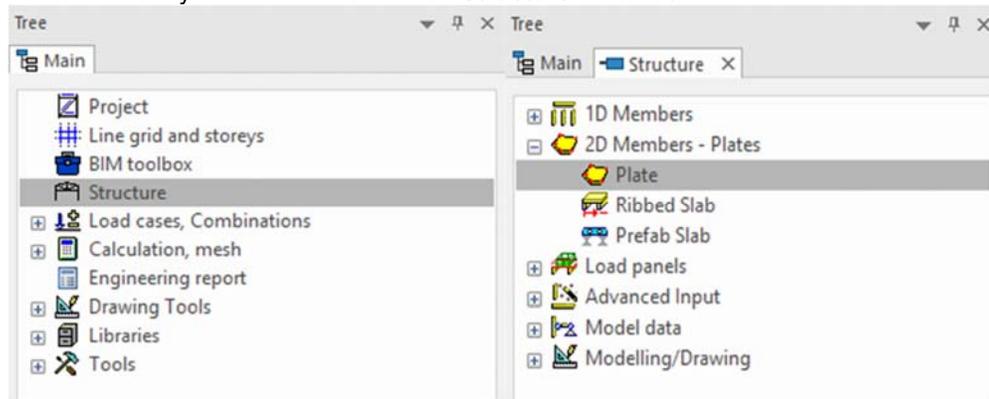
Input of the geometry

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.

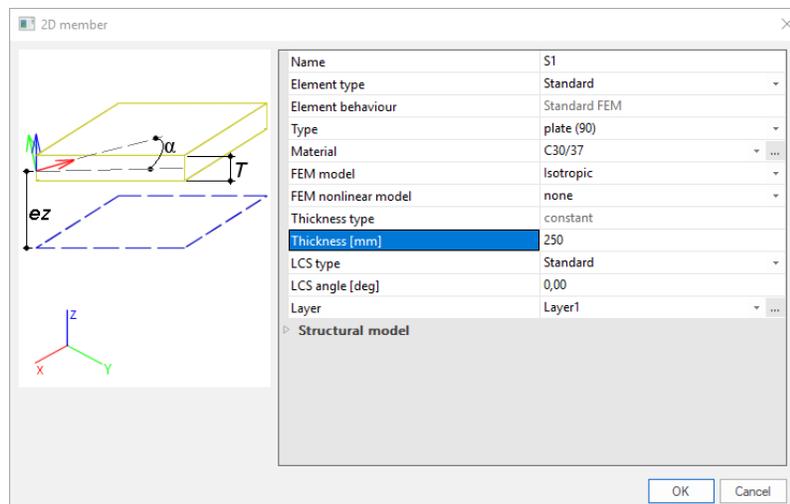


2. In the **Structure menu** different branches will appear, in accordance with the already input items, i.e. support branch will appear if a structure is already available.

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.



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.

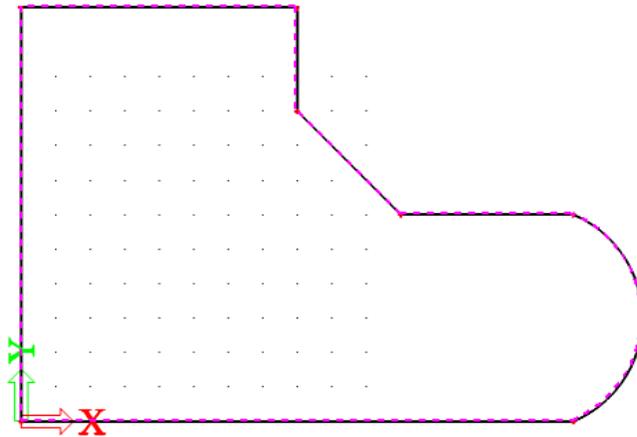
Starting point: 0;0 <enter>
16;0 <enter>

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:

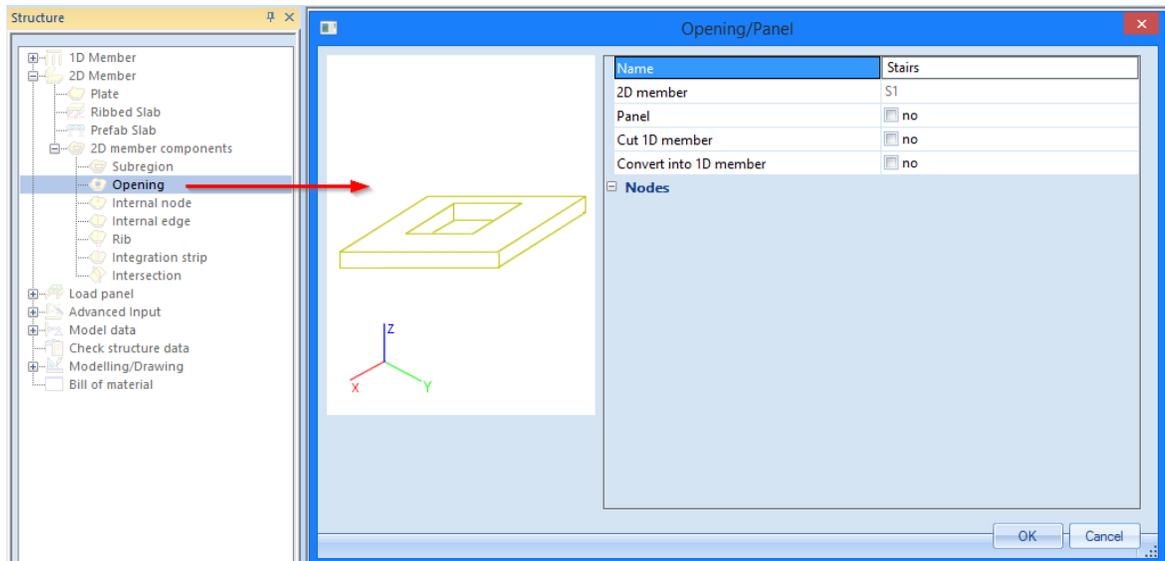


Note:

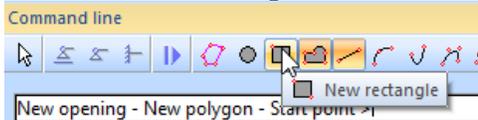
*Coordinates are defined with either semicolon or space between X and Y coordinates. Don't use comma which defines decimal mark.
@ symbol defines relative coordinates, instead of global coordinates.*

Definition of an opening

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



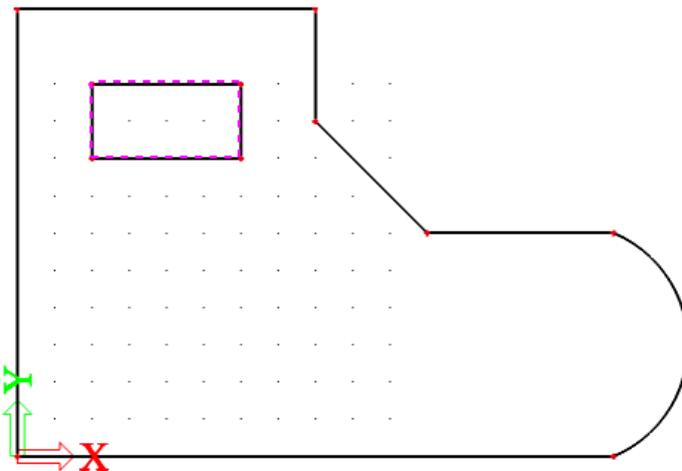
2. Type the naming *Stairs* in the first line of *Opening/Panel* properties and confirm by **[OK]** button
3. Click on icon **New rectangle** in **Command line**.



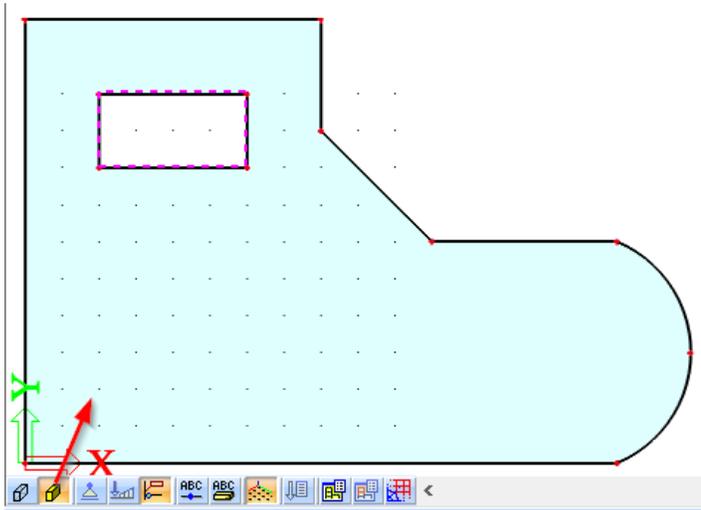
4. The two nodes of its diagonal define a rectangle opening. This is also depicted by the two red dots on the icon. Type the following point into command line

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

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

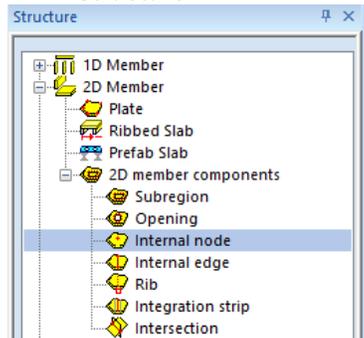


You can also activate rendering by the icon to see the real “hole” in the slab.



Input of internal nodes

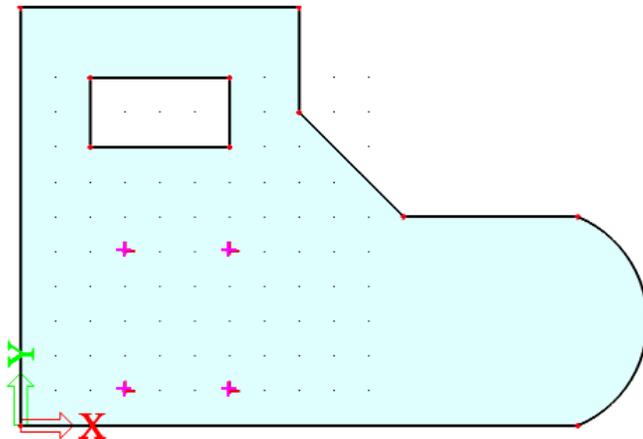
1. In the **Structure** menu choose under **2D element components** command **Internal node**.



2. We will add four new internal nodes by typing its coordinates into command line:

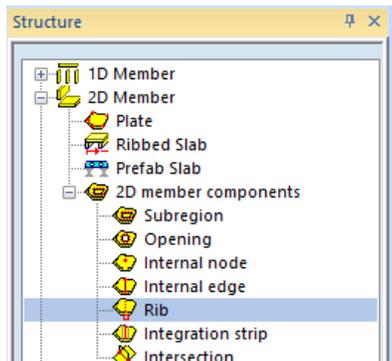
```
3;1 <enter>
3;5 <enter>
6;5 <enter>
6;1 <enter>
```

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

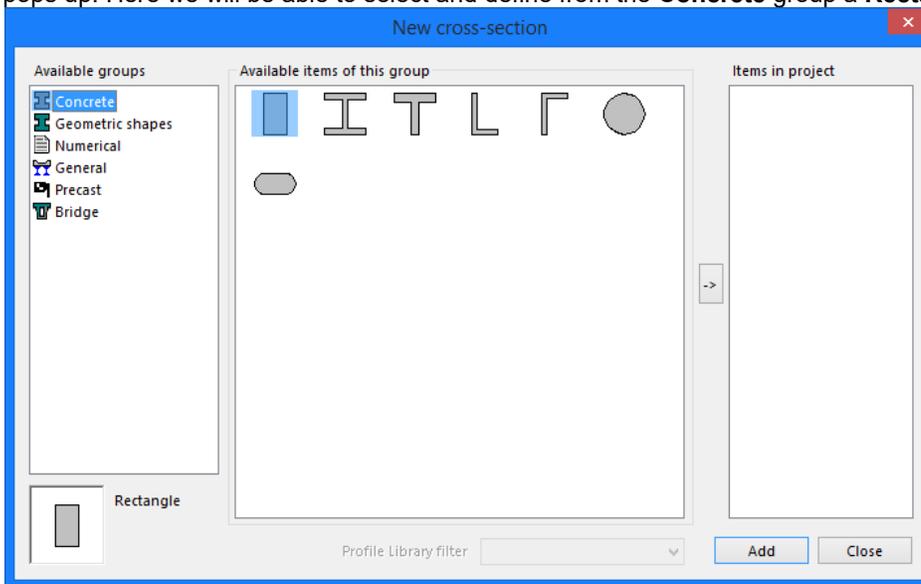


Input of plate ribs

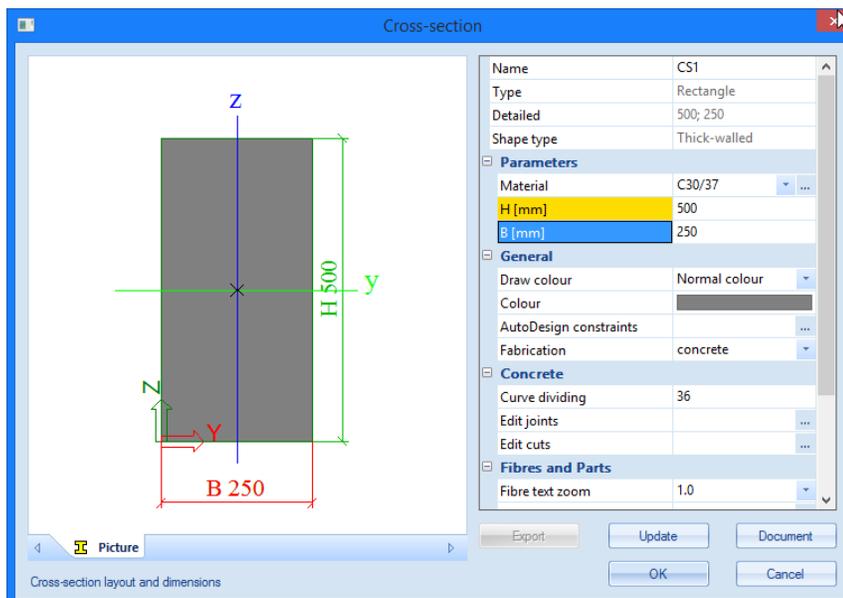
1. In the **Structure** menu, under **2D member components** choose command **Rib**.



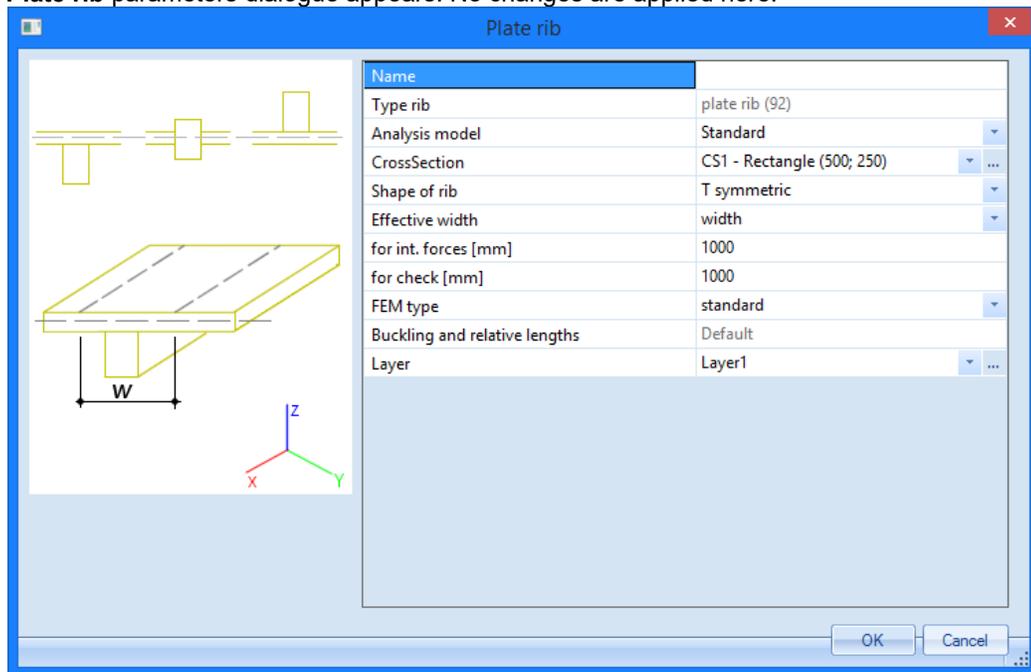
2. Since no cross-section was defined in the project previously, the dialogue **New cross-section** pops up. Here we will be able to select and define from the **Concrete** group a **Rectangle** shape.



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



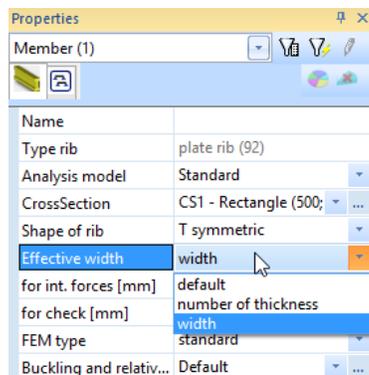
6. When pressing **<OK>** we have to define the starting and end points of the ribs.

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

Second rib Start point: 12;1 <enter>
 End point: 15;1 <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.

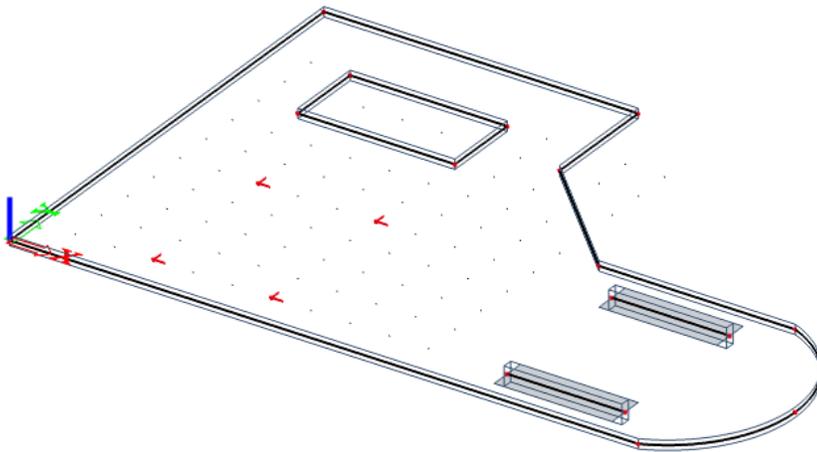
default : The width of T-section is defined as a factor which multiplies the plate thickness. The factor is defined in **Setup > Solver > Number of thicknesses of rib plate**. Default is 20.

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

Hit the icon for changing the view of whole structure to axonometry.



Hit icon **Show surfaces** and deactivate **Rendering**   to see the view below:



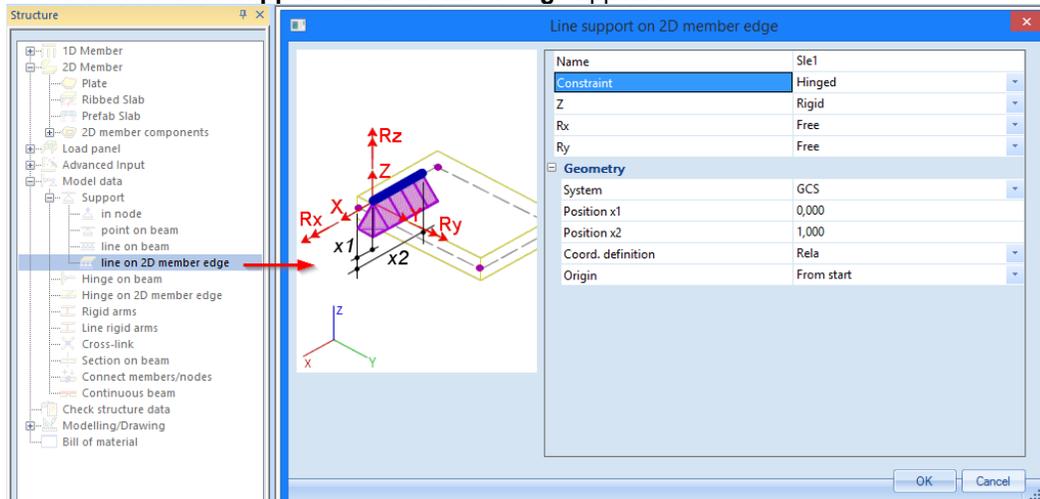
By pressing **<Esc>** one can easily cancel any selection.

Supports

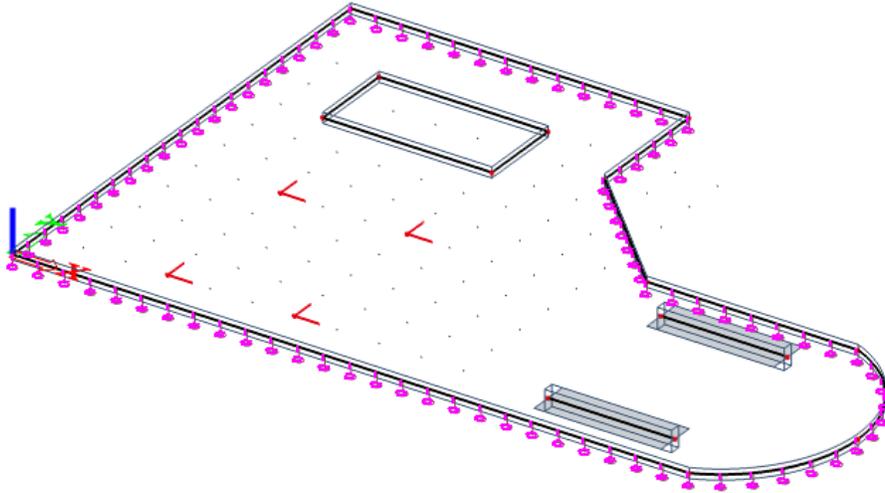
The input of the geometry can be finalized by definition of support conditions. We assume that the whole plate edge is supported in global z-direction. Thus we simulate that e.g. masonry wall supports the slab.

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.

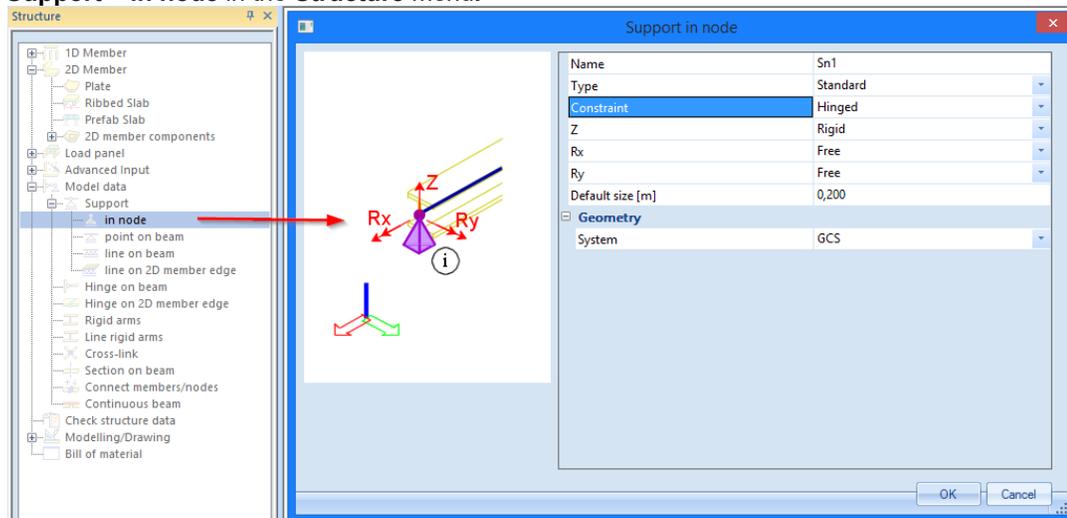


3. Change the **Constraint** to **Hinged** so that Z axis deformation is prohibited but rotations around X and Y axes are free.
4. Select the edges around the slab one by one with mouse cursor - edge1, edge2, edge3, edge4, edge5, edge6, edge7.
5. Press **<ESC>** to cancel the input command.

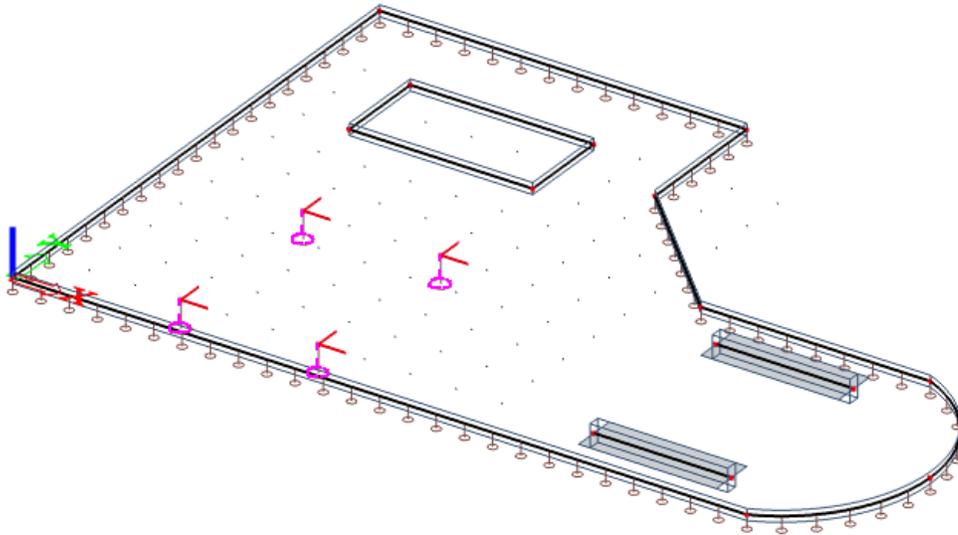


Input of Nodal supports

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.

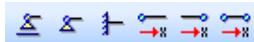


2. We will support the nodes in Z-direction only again. Therefore change the **Constraint** to **Hinged** again.
3. We apply the nodal supports for the internal nodes N13, N14, N15 and N16 by mouse cursor.



 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.

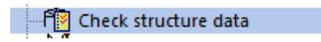
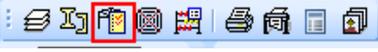


Check Structure data

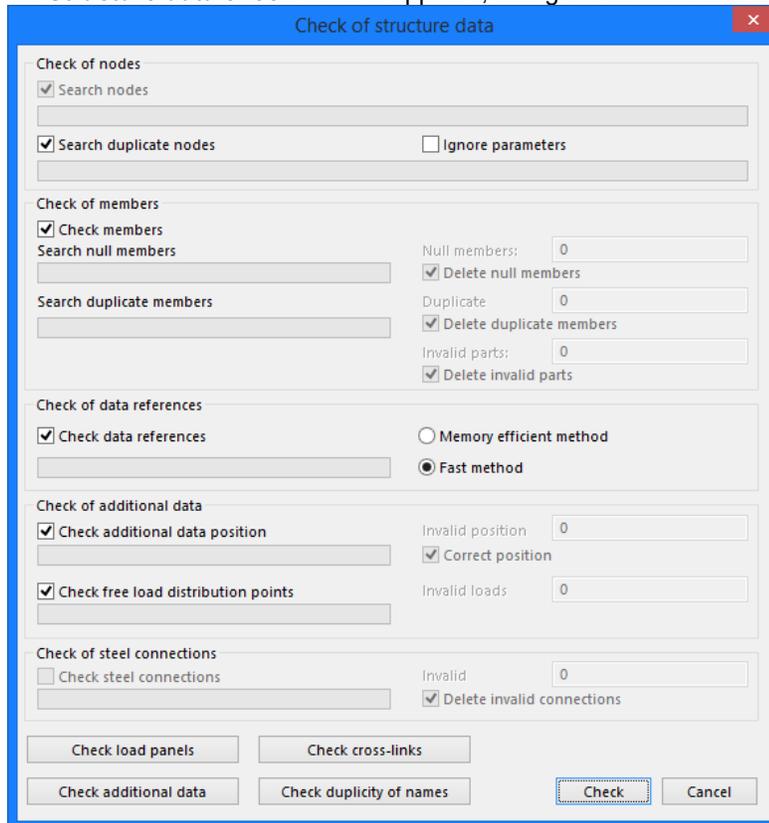
After input of the geometry, the input can be checked for errors by means of the option **Check Structure data**. With this tool, the geometry is checked for duplicate nodes, zero beams, duplicate members, wrong references of hinges or supports etc. However, this tool does not check if the structure 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

 or click on the  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 parameters

Check of members

Check members

Search null members Null members:

Delete null members

Search duplicate members Duplicate:

Delete duplicate members

Invalid parts:

Delete invalid parts

Check of data references

Check data references Memory efficient method

Fast method

Check of additional data

Check additional data position Invalid position:

Correct position

Check free load distribution points Invalid loads:

Check of steel connections

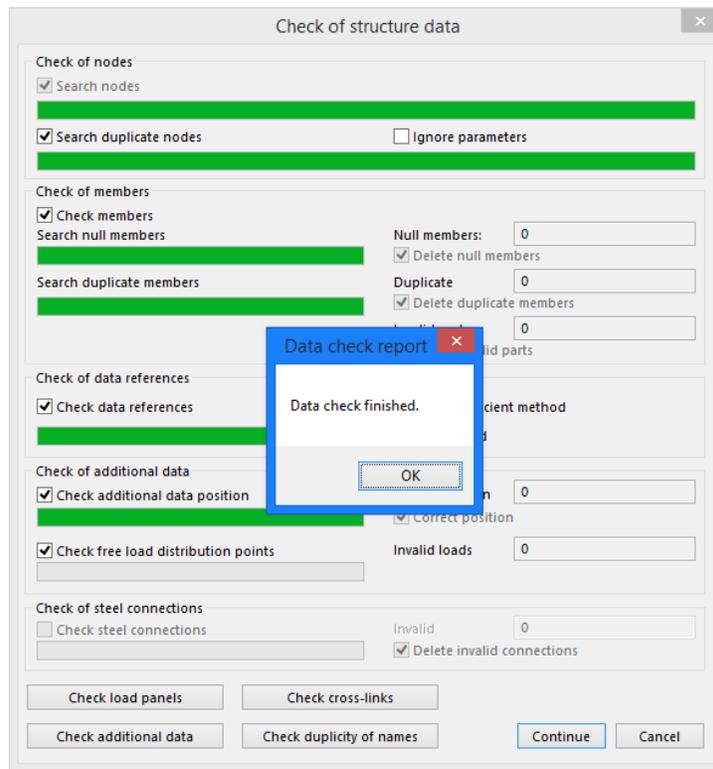
Check steel connections Invalid:

Delete invalid connections

Check load panels Check cross-links

Check additional data Check duplicity of names **Check** Cancel

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



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 .

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**.

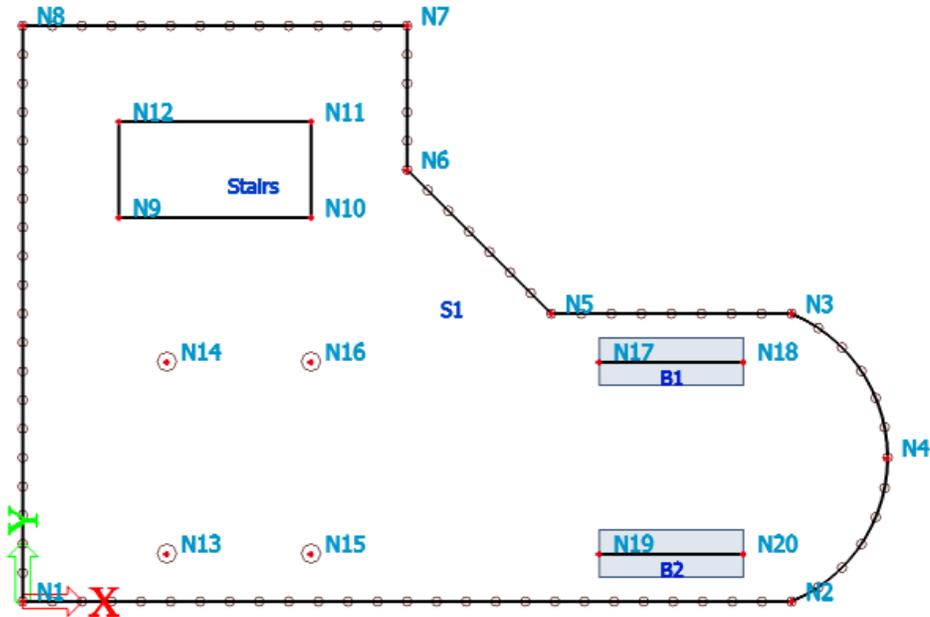


The third button can visualize supports.

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:

The screenshot shows the Properties window for a 2D member (S1). The window is divided into several sections:

- Name:** S1
- Type:** plate (90)
- Analysis model:** Standard
- Shape:** Flat
- Material:** C30/37
- FEM model:** Isotropic
- FEM nonlinear model:** none
- Thickness type:** constant
- Thickness [mm]:** 250
- LCS type:** Standard
- LCS angle [deg]:** 0,00
- Layer:** Layer1
- Nodes:**

N1	abso
N2	abso
N3	abso
N4	abso
N5	abso
N6	abso
- Data:**

Opening/Panel	Stairs
Node	N13
Node	N14
Node	N15
Node	N16
Member	B1
Member	B2
Line support on 2...	Sle1
Line support on 2...	Sle2
Line support on 2...	Sle3
Line support on 2...	Sle4
Line support on 2...	Sle5
Line support on 2...	Sle6
Line support on 2...	Sle7
- Actions:**

Table 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**  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.*

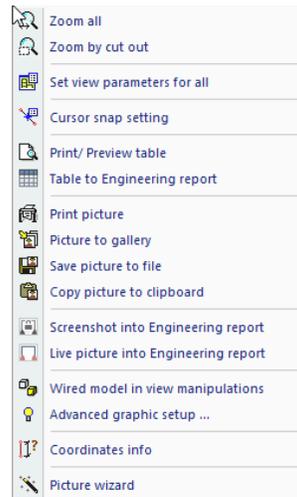
The magnifier

- Use  to enlarge.
- Use  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:



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.

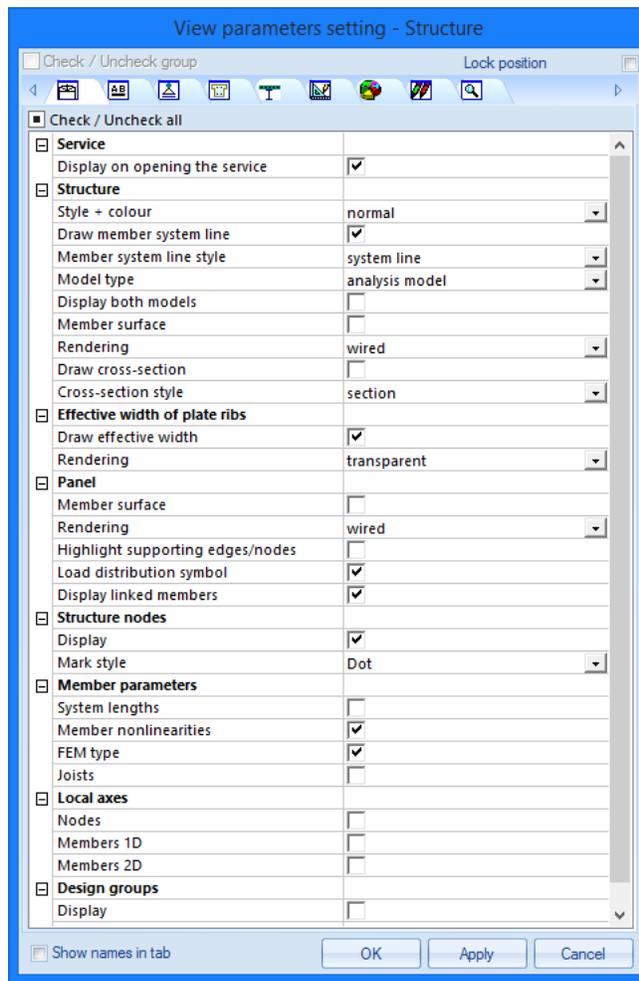
View parameters – Entities

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.

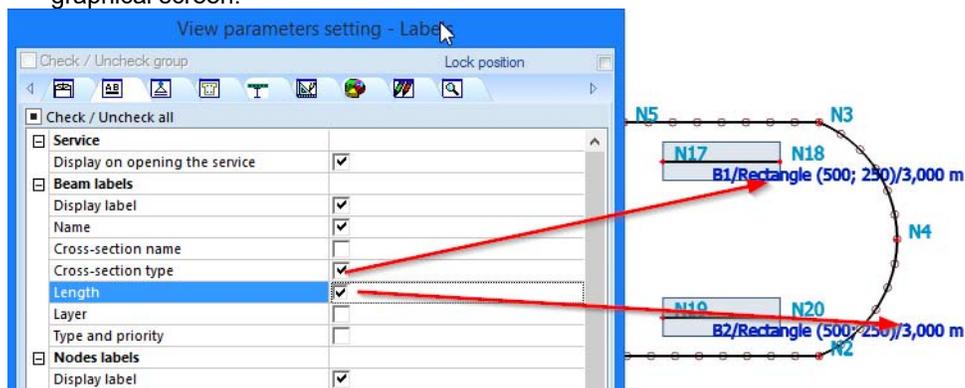
Local Axes: Using this tool the local axes can be set for nodes, 1D and 2D members.



View parameters – 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 ticked ON, the labels will be displayed on the graphical screen.

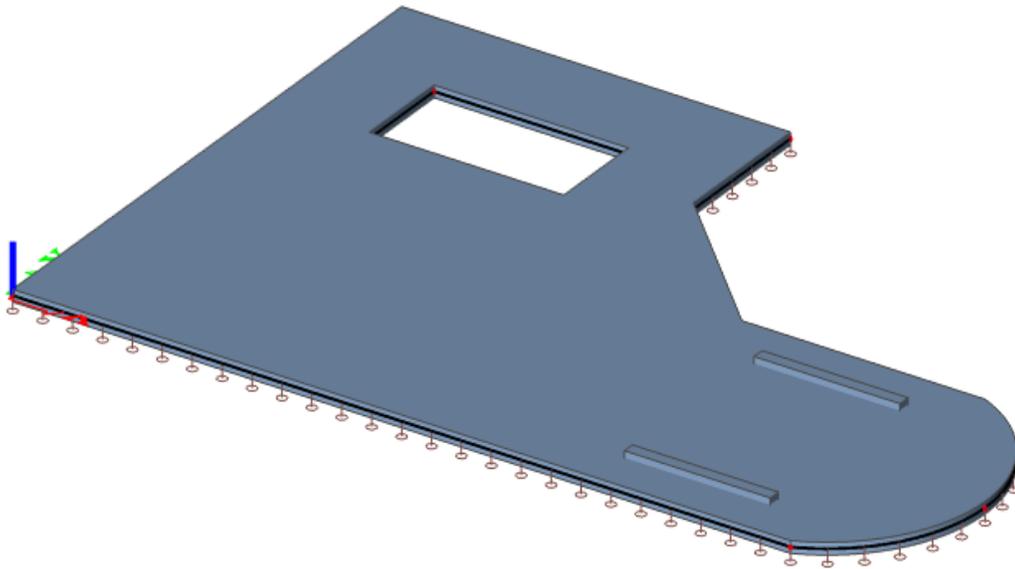


View parameters – shortcuts

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.

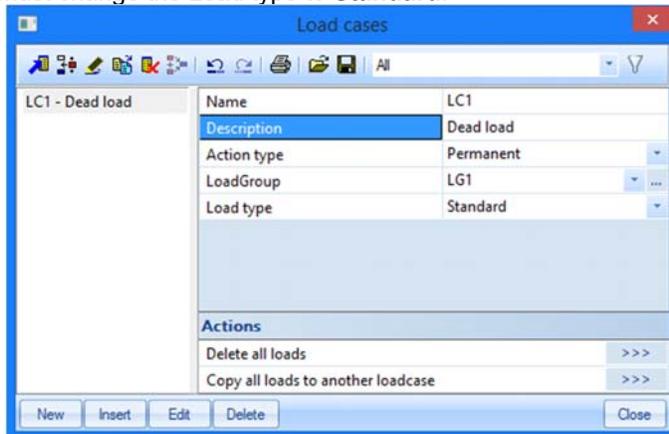
The way in which load cases are defined is decisive for the load combinations created by the generator. We recommend that you thoroughly read the chapter about loads and combinations in the reference manual.

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  **Load** 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**.

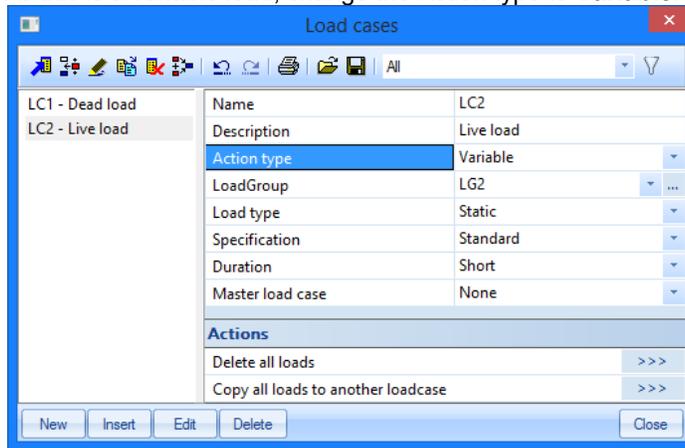


5. In the Description field, you can describe the specification of this load case. For this project, type the description "**Dead load**".

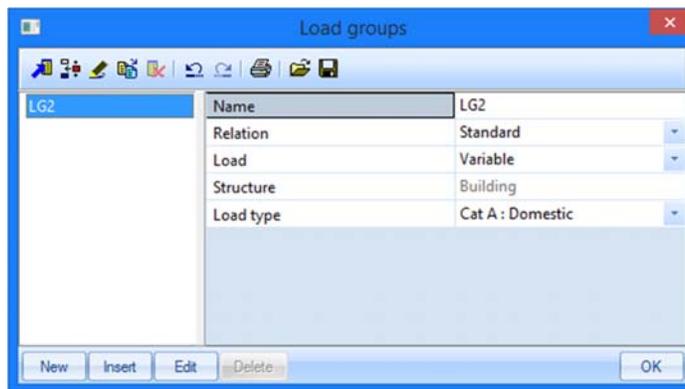
Defining a Variable Load Case

1. Click  or  to create a second load case.
2. Enter the description "**Live load**".

- As this is a variable load, change the Action type to **Variable**.



- New Load group LG2 is automatically created. Click **...** to display the properties of the Load group.



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.

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

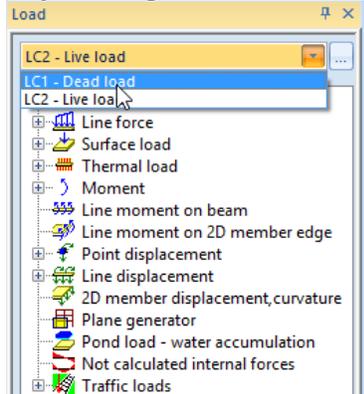
After input of the Load cases, the **Loads service** will automatically appear:

The first load case (LC1) includes two loads:

- Dead load
- Self weight

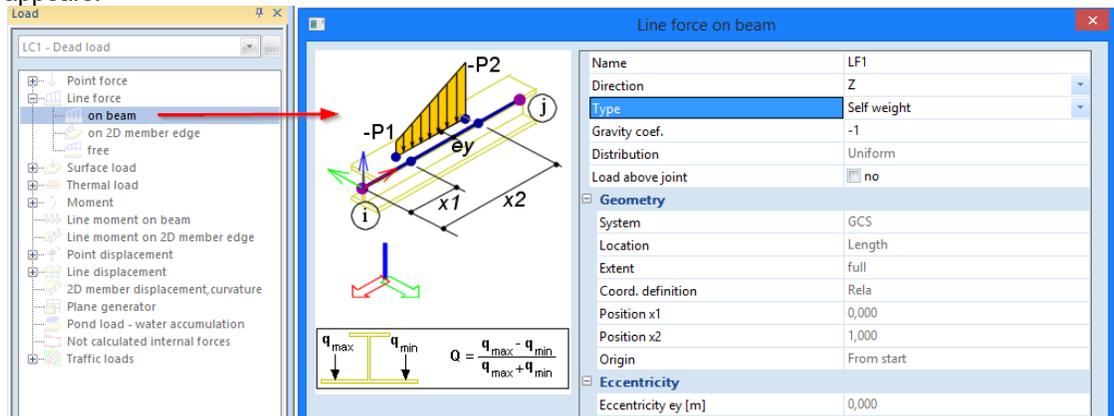
Switching between load cases

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



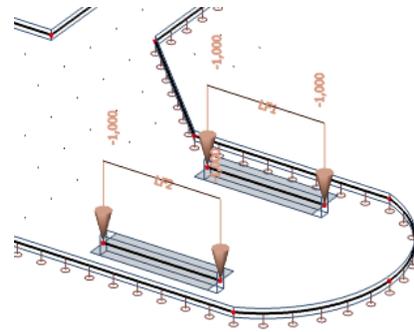
Entering the self weight of ribs as line loads

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.



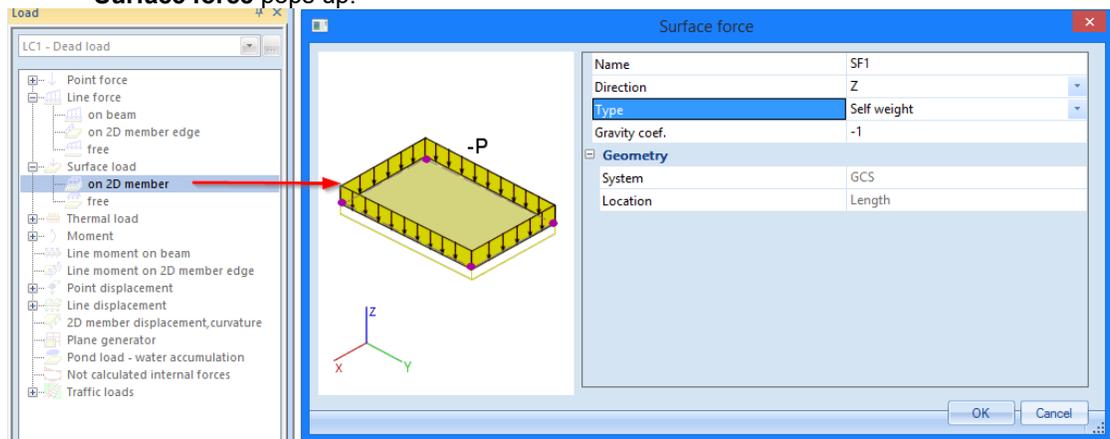
3. In the field **Type** choose **Self Weight**. The Direction is the global Z-direction and the Gravity coefficient is set to -1, so that the load is acting vertically downwards.

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.



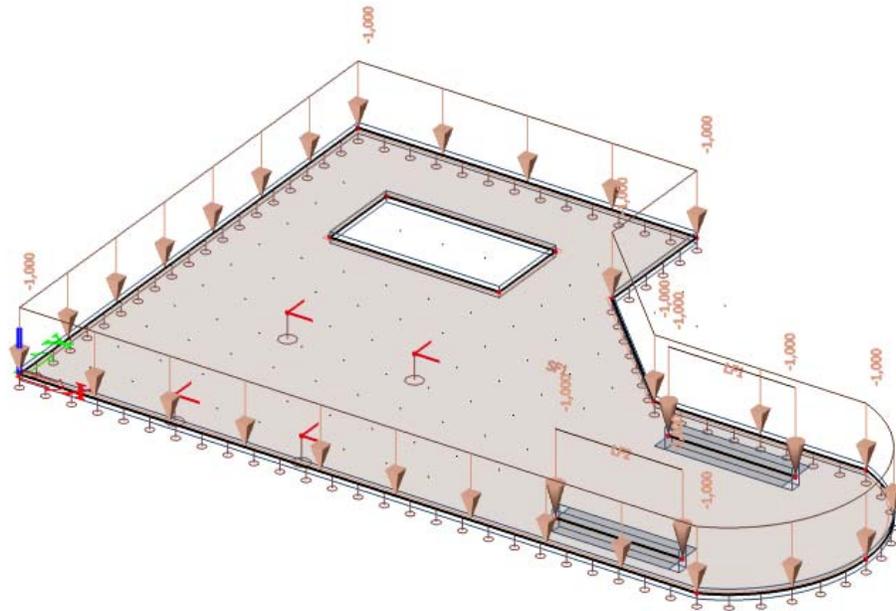
Input of the self weight of the slab as surface load

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:

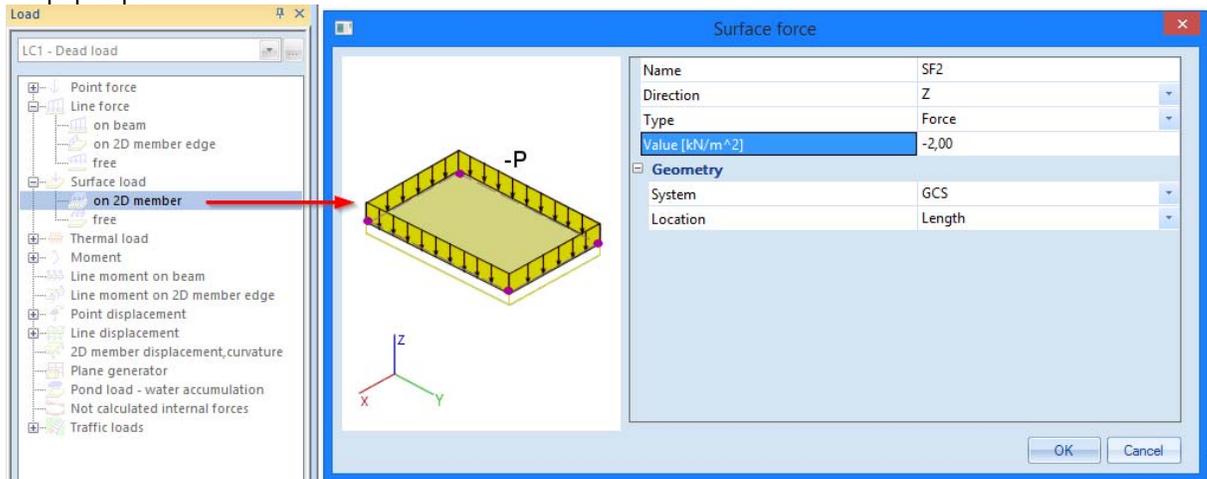


The entered loads are so-called self weight loads. Other dead load caused by the non-structural topping will be added to the same load case. So that permanent loads are combined into one load case.

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

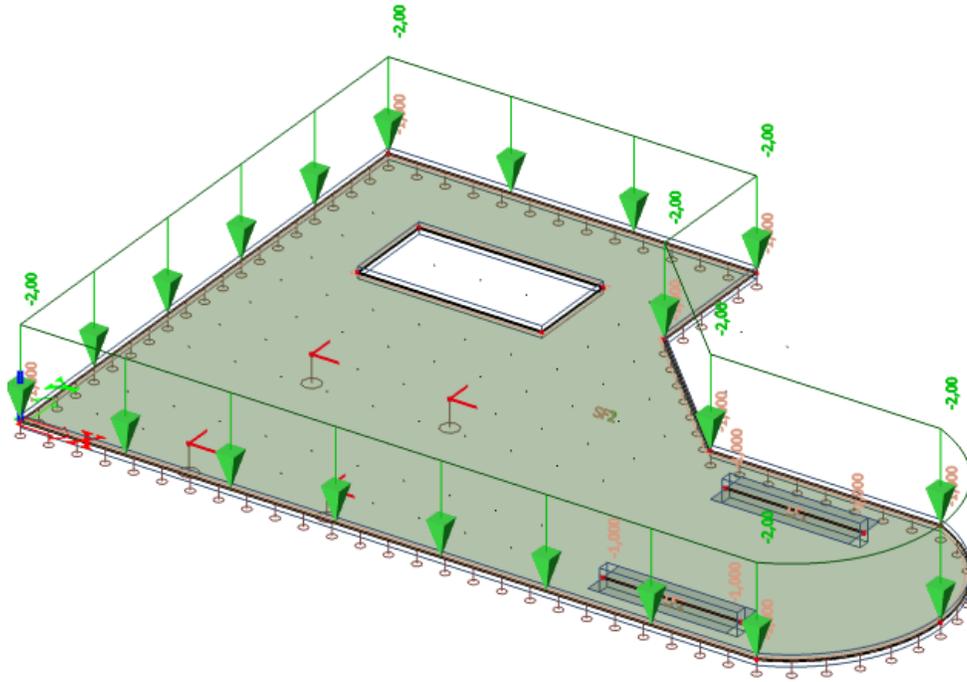
1. Click on **Surface load – on 2D element** in the **Loads** menu. The dialogue window **Surface force** pops up.



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]**.

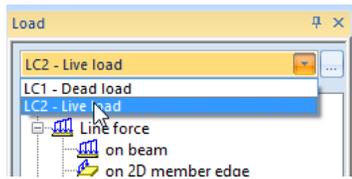
6. Since there is only one slab in the project load will be automatically put on the slab.

The self-weight is depicted by a green colour:



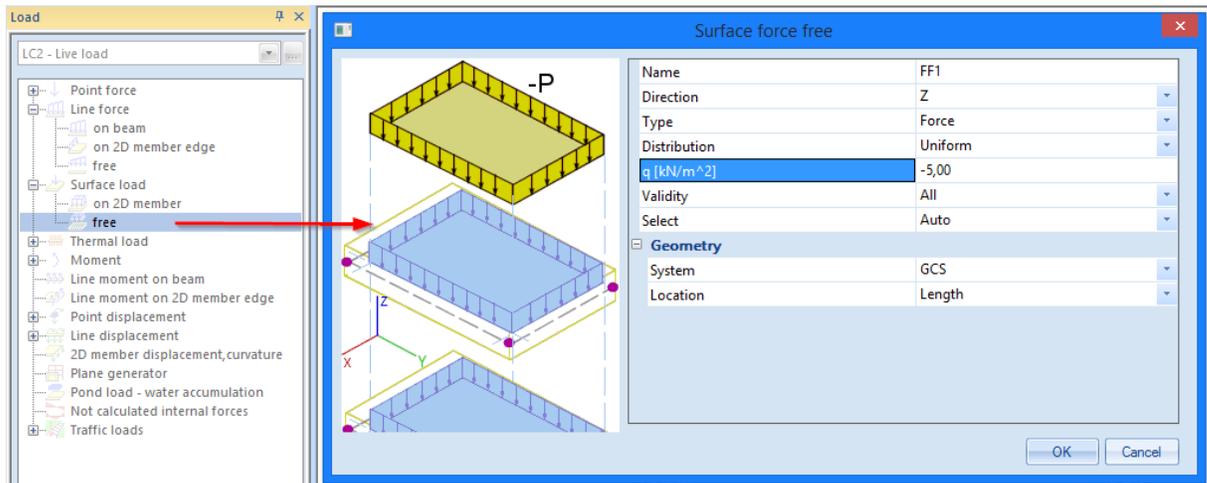
Switching between load cases

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.



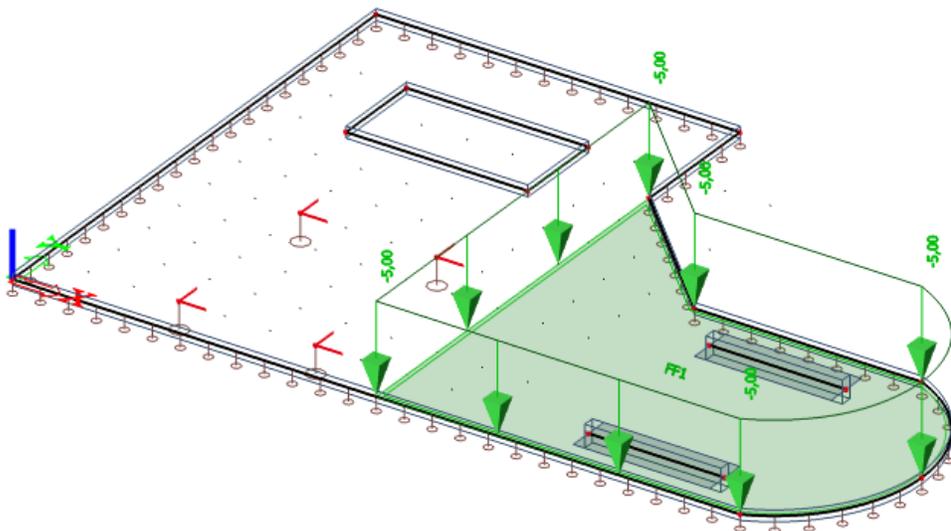
3. For the field **Type** the **Force** is chosen. The **Direction** is the Z-direction in the coordinate system that you have chosen in **System** – here the global one (GCS). The value is -5 kN/m^2 and uniformly distributed over the surface.
4. Confirm your input with **[OK]**.
5. The program asks 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:

Starting point: **8;9 <enter>**
8;0 <enter>
16;0 <enter>

Click on  icon in Command line to follow with circular arch
 New circle arc – Intermediate point: **18;3 <enter>**
 End point: **16;6 <enter>**

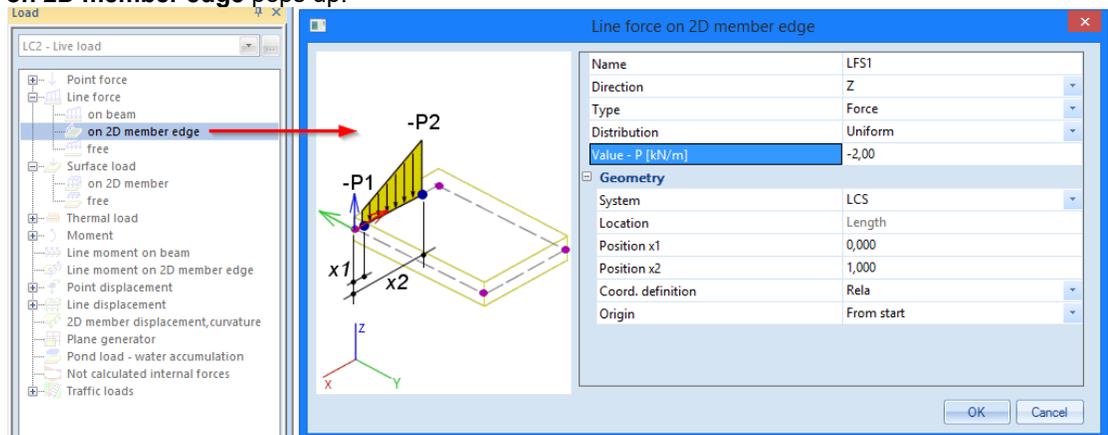
Finish the input with polygonal segments
11;6 <enter>
8;9 <enter>

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

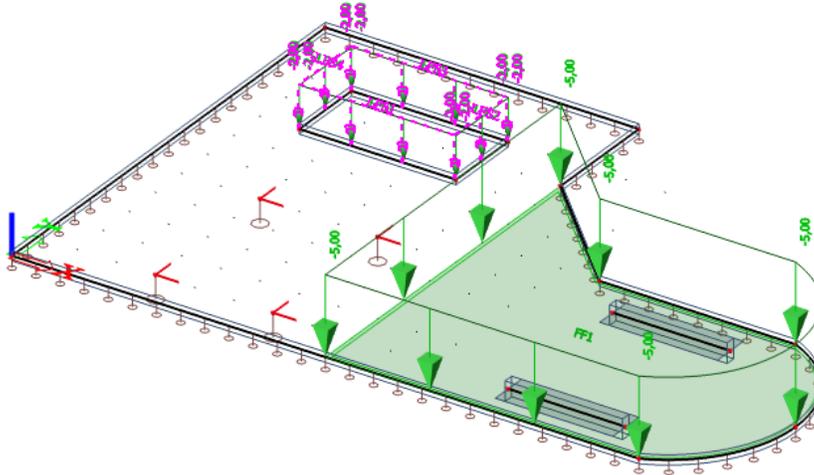


Input of variable line load

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** pops up.



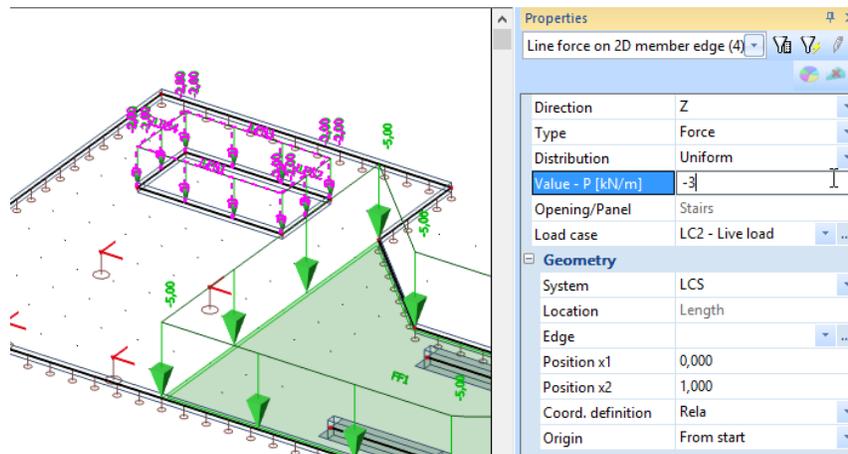
3. For the field **Type** the **Force** is chosen. The **Direction** is the global Z-direction. The input value P is **-2.00 kN/m**
4. Confirm your input with **[OK]**.
5. Select the four edges around the opening of the staircases.
6. Click on **<Esc>** to end this function.



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

Adapting a load

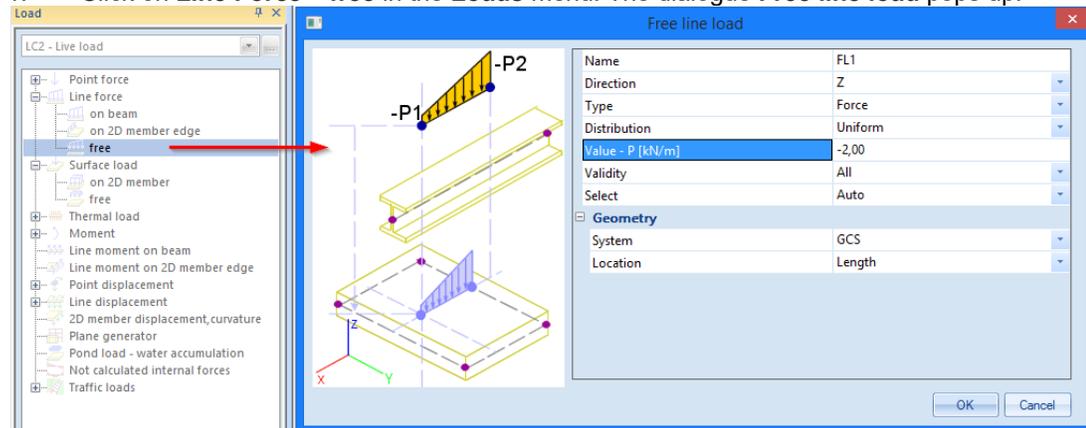
1. Select the variable line loads around the opening by clicking with the left mouse button on these loads. The common properties of the 4 loads are displayed in the **Properties** window.
2. Change the **Value** from **-2,0 kN** to **-3,0 kN** in the Properties window.



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

Input of a free line load

1. 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>**

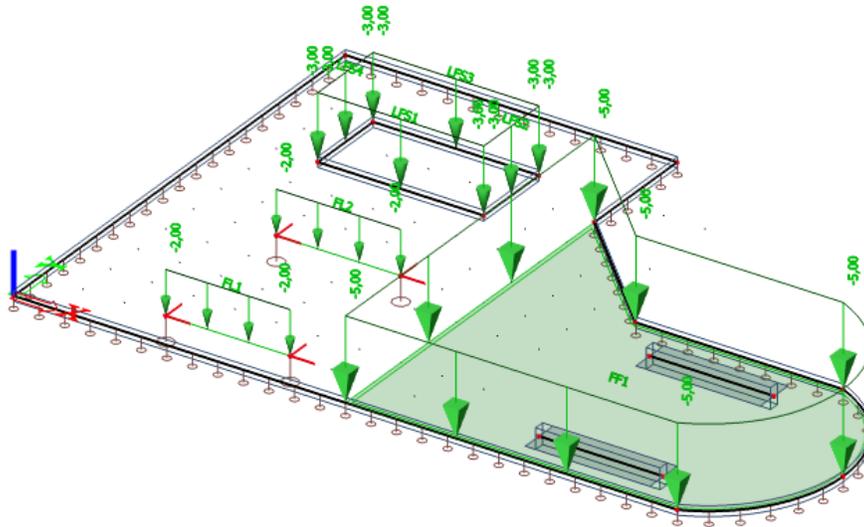
Press **<ESC>** to finish the first **Polyline** command. But continue with second line load.

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



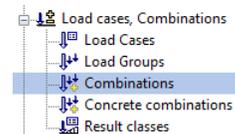
which enable fast and simple input of some particular loads.

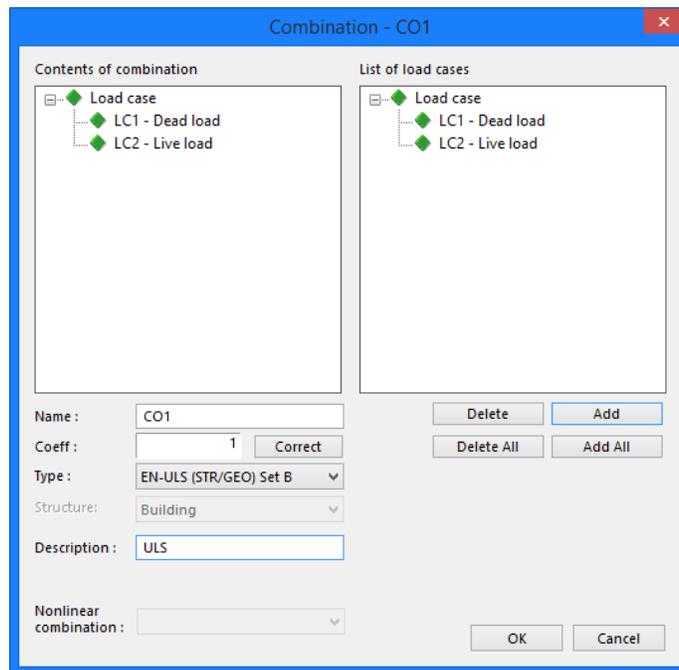
Combinations

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.

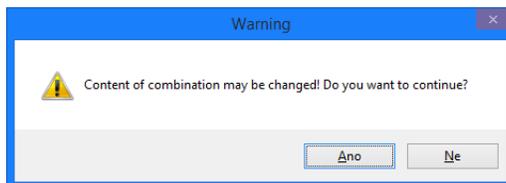
Defining Combinations

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.





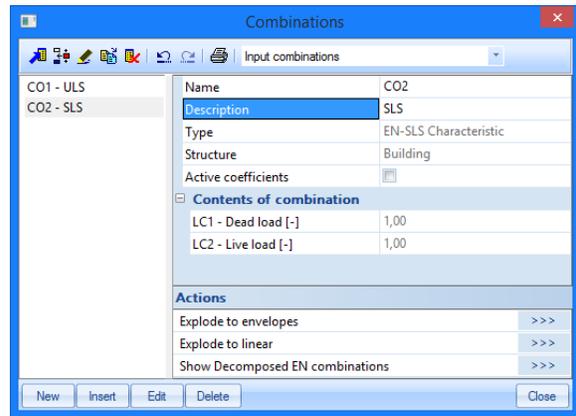
3. The Type of the combination is changed to **EN – ULS (STR/GEO) Set B**. With this envelope combination type SCIA Engineer will automatically generate linear combinations in accordance with the complex composition rules of the Eurocode.
4. A warning message that controls the content of code combinations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



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  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**.



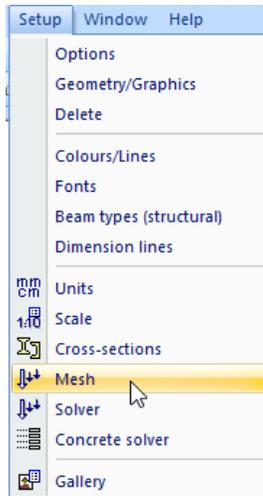
Calculation and Mesh generation

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.

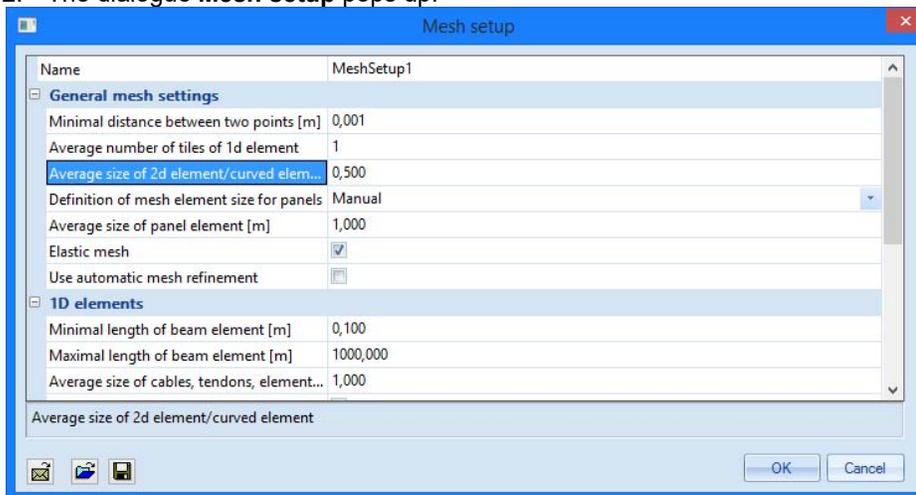
Mesh generation

Mesh setup

1. In order to see the mesh setup click on **Setup > Mesh**



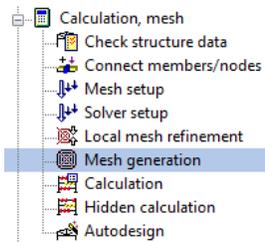
2. The dialogue **Mesh setup** pops up.



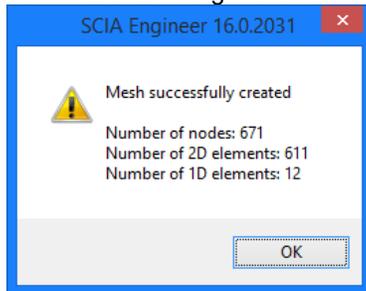
3. The **Average size of 2D element/curved element [m]** will be used for the mesh generation if no local mesh refinements have been defined. Change this value in 0,500 m because the default value 1 m is too coarse.

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.



5. The program informs you that the mesh is generated and states the number of nodes, 1D and 2D elements that was generated.



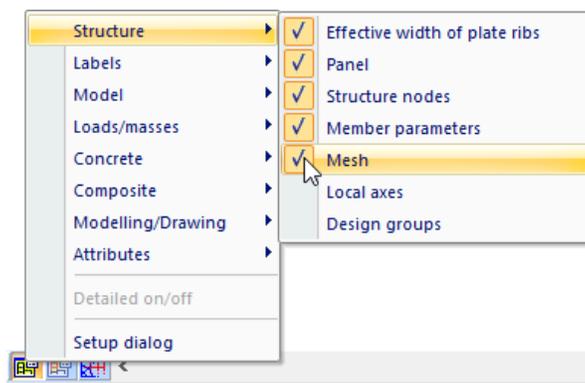
Note:

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

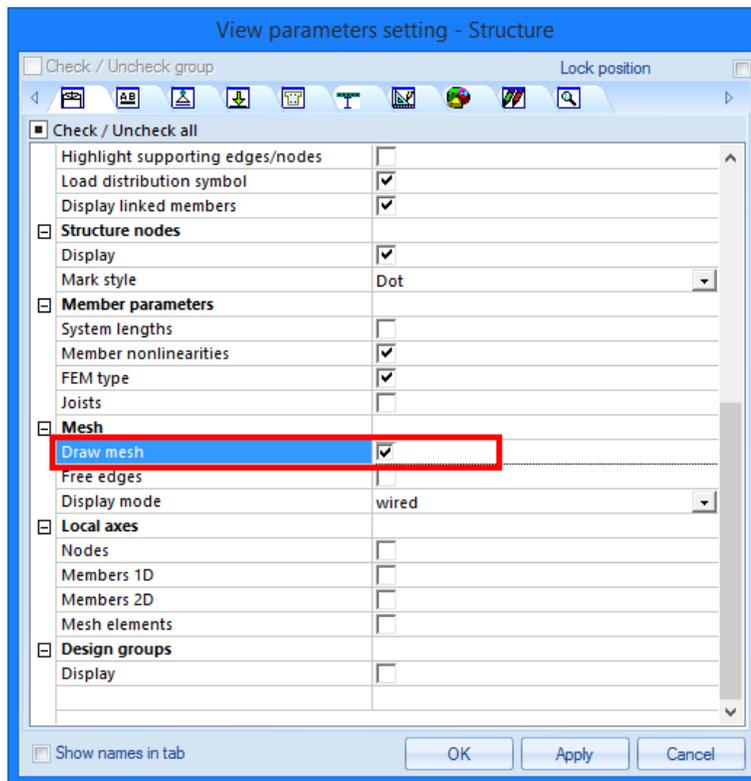
- **Node mesh refinement**; refines the mesh around a single node.
- **2D member edge mesh refinement**; refines the mesh along particular edge or internal line of a plate.
- **Surface mesh refinement**; for the whole surface a denser mesh will be applied.

Display of the mesh

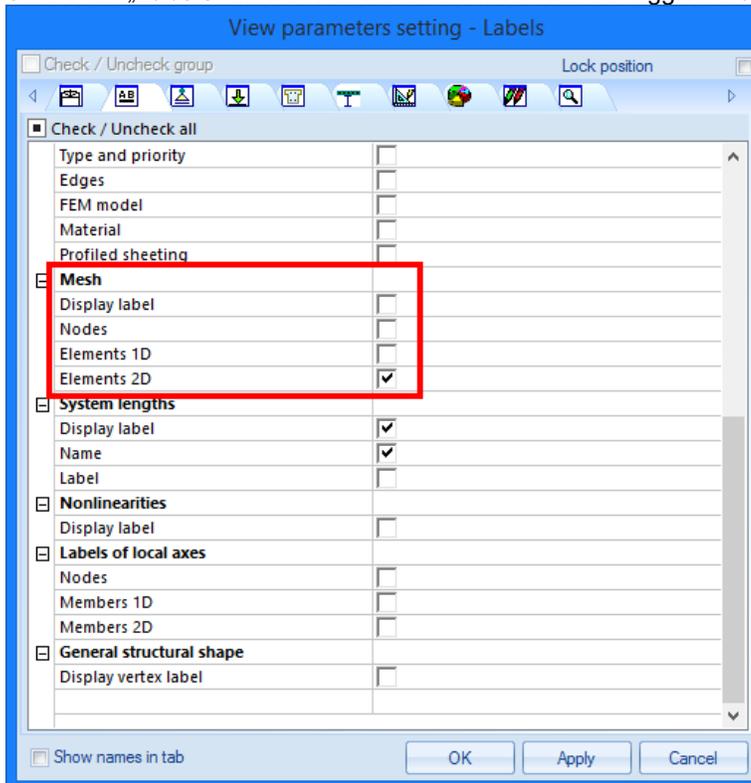
1. The mesh can be displayed using the shortcut button **Fast adjustment of view parameters on whole model** located at the bottom of the graphical screen



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



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



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.

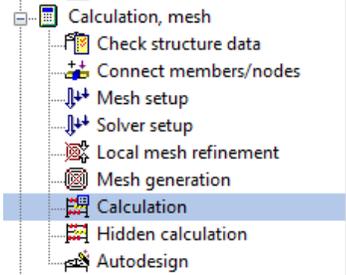
If the mesh has not generated before the start of the calculation the programme will automatically generate the mesh itself.

Linear Calculation

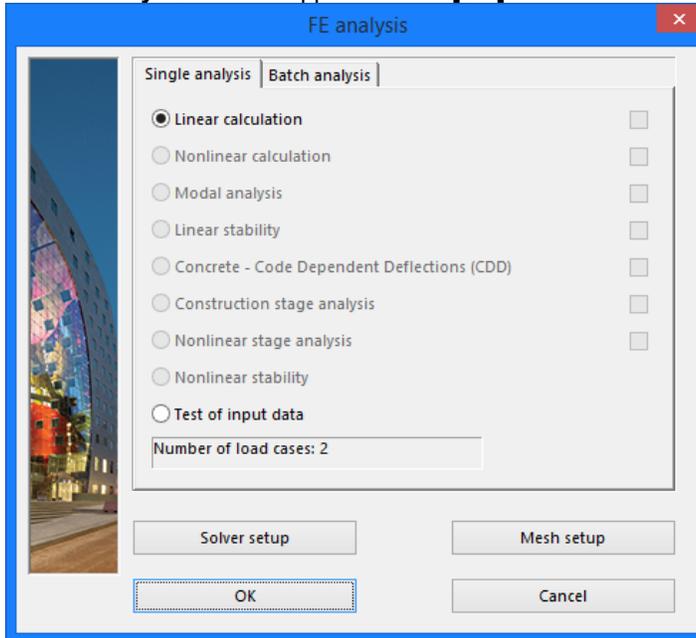
As the calculation model is completely ready you now can start the calculation.

Executing the Linear Calculation

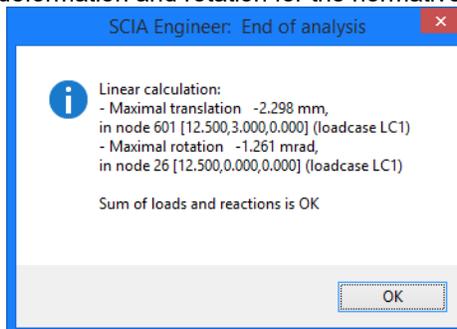
1. Double-click on Calculation in the **Main window**, or use identical icon  in toolbars.



2. The **FE analysis** window appears. Click **[OK]** to start the calculation.



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.



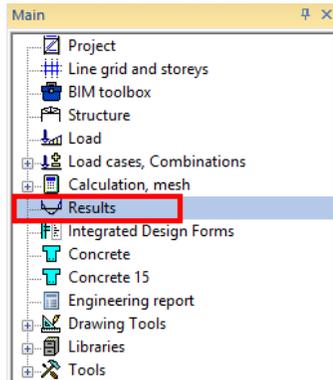
Results

Viewing results

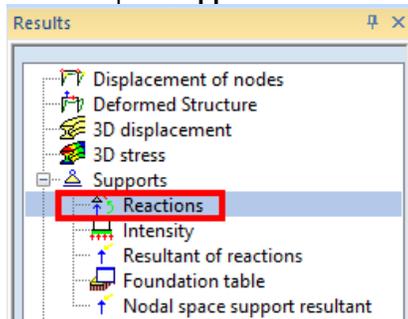
After the calculation is executed, the results can be viewed.

Viewing the Reaction Forces

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

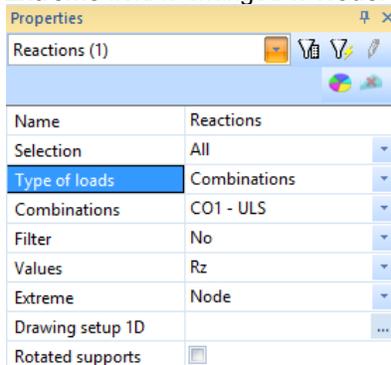


2. In chapter **Supports** click **Reactions**.



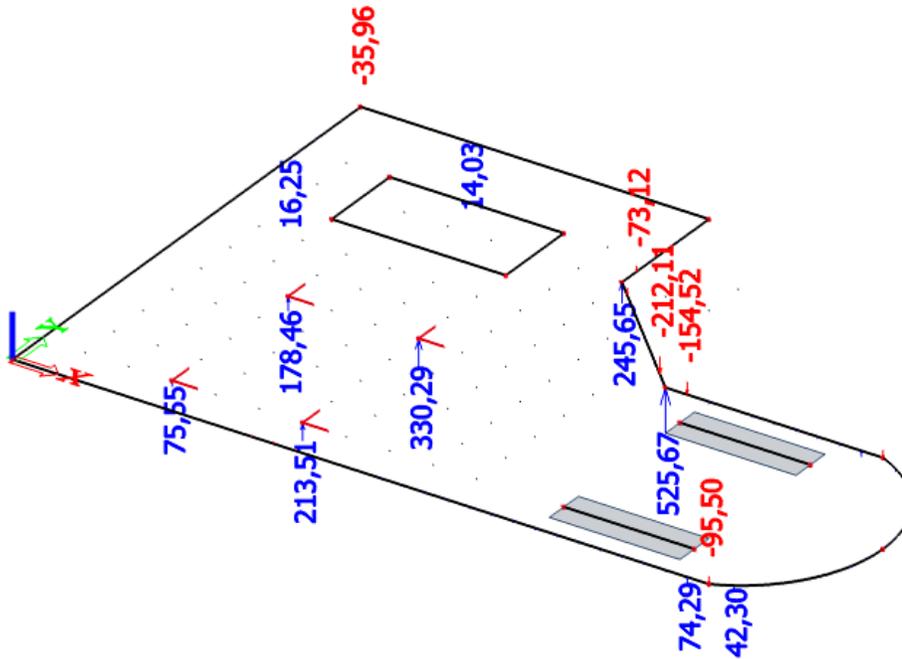
3. The options in the **Properties** window are configured in the following way:

- **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**.



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.



- To display these results in a table form, the **Preview** action button is used. Click on the next to **Preview** to open it.

Report preview

Default

Reactions

Linear calculation, Extreme : Node
 Selection : All
 Combinations : CO1

Support	Case	dx [m]	Rz [kN]	Mx [kNm]	My [kNm]
Sn1/N14	CO1/1		128,26	0,00	0,00
Sn1/N14	CO1/2		178,46	0,00	0,00
Sn1/N14	CO1/3		173,15	0,00	0,00
Sn2/N13	CO1/4		44,43	0,00	0,00
Sn2/N13	CO1/3		75,55	0,00	0,00
Sn3/N15	CO1/1		123,54	0,00	0,00
Sn3/N15	CO1/2		213,51	0,00	0,00
Sn3/N15	CO1/3		166,78	0,00	0,00
Sn4/N16	CO1/1		194,12	0,00	0,00
Sn4/N16	CO1/2		330,29	0,00	0,00

Note:

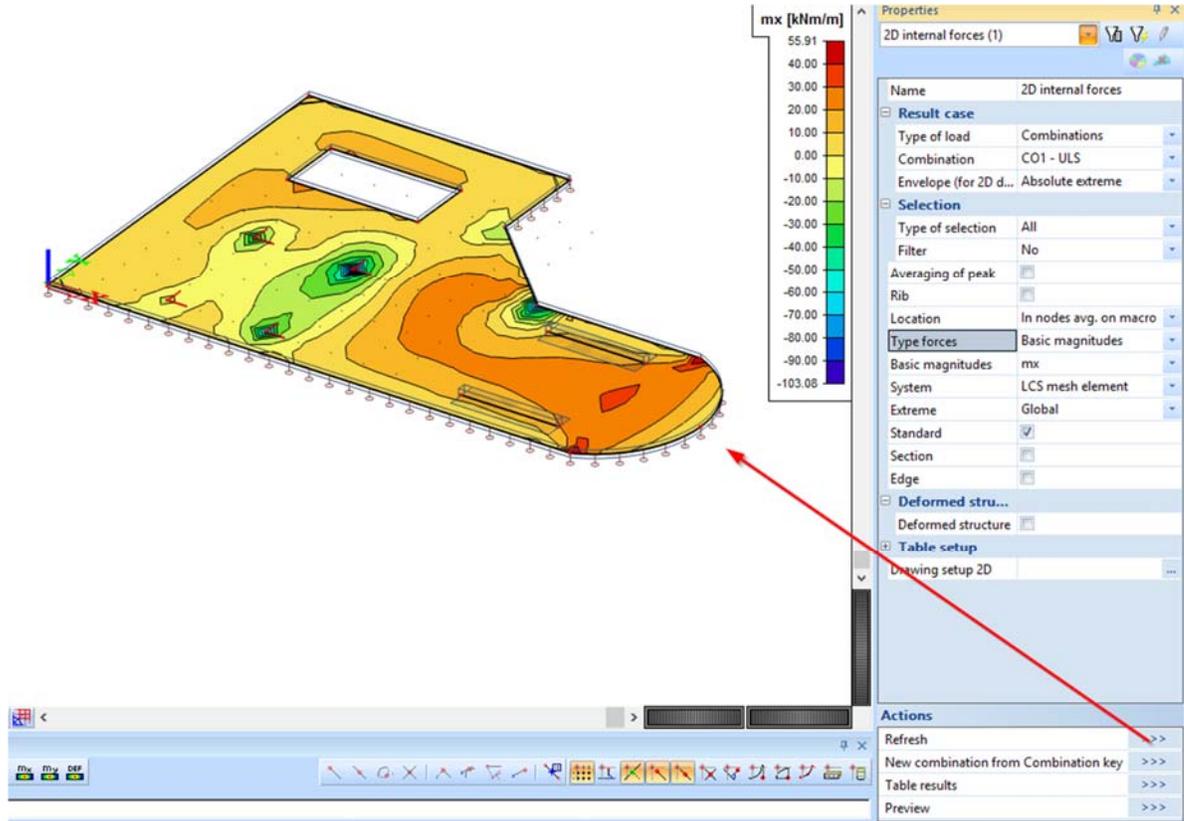
The Report preview appears between the Graphical screen and the Command line. This screen can be maximised to display more data at once.

Viewing internal forces on 2D elements

- Click on **2D Members (beta) > 2D internal forces** in **Results** service
- 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**

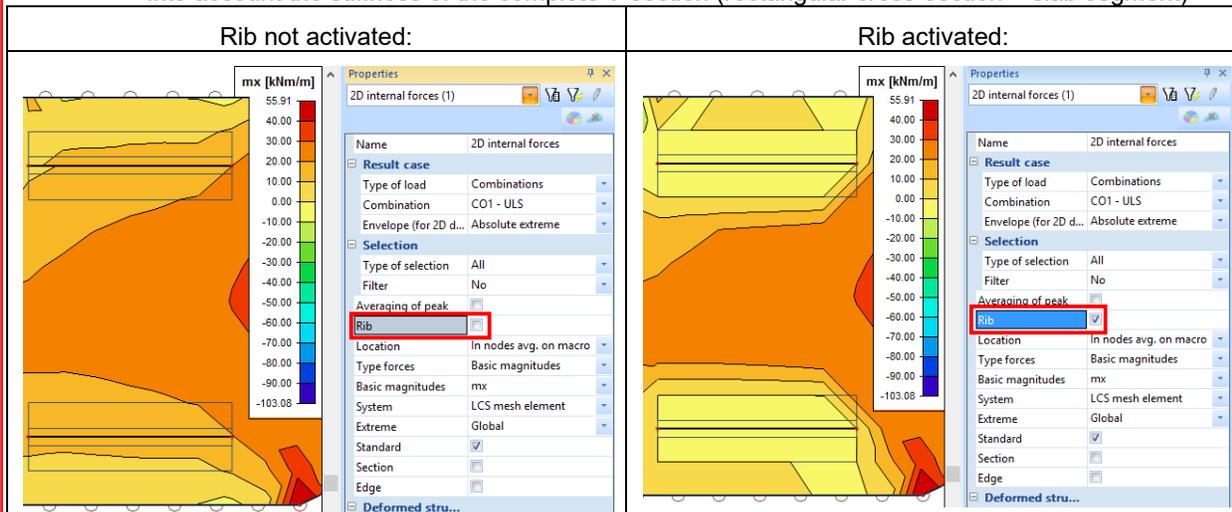
- **Type of forces** is defined as **Basic magnitudes** and the magnitude in below row is set to **mx**.

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



Results for (individual) ribs

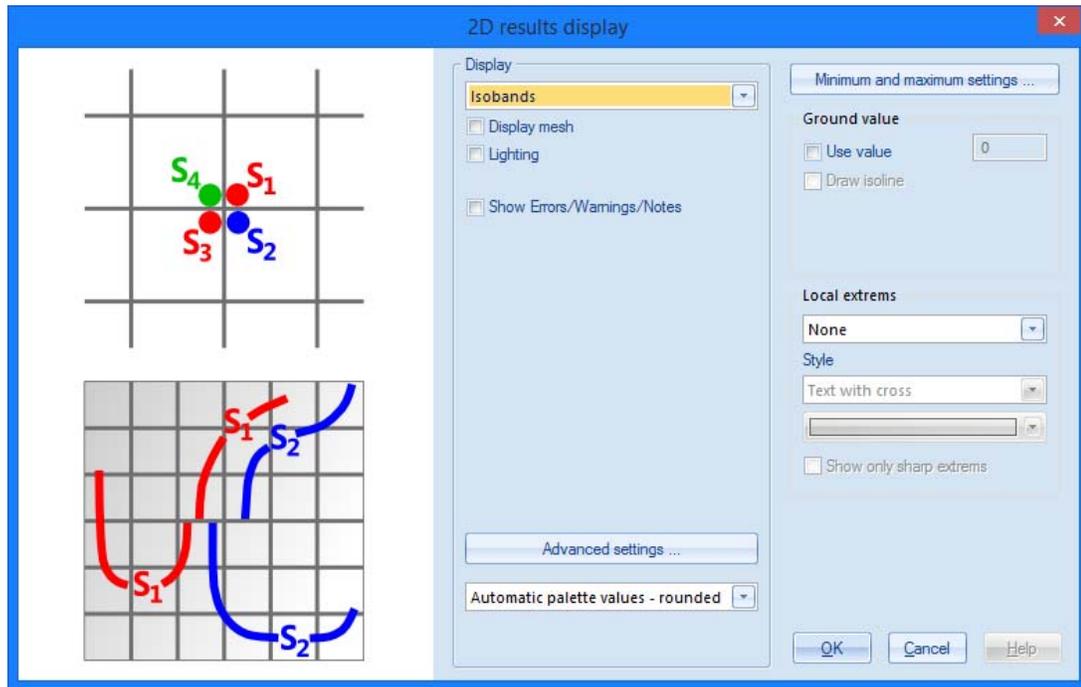
1. By clicking on the check box **'Rib'** in Properties window, the results will be adjusted in order to take into account the stiffness of the complete T-section (rectangular cross-section + slab segment)



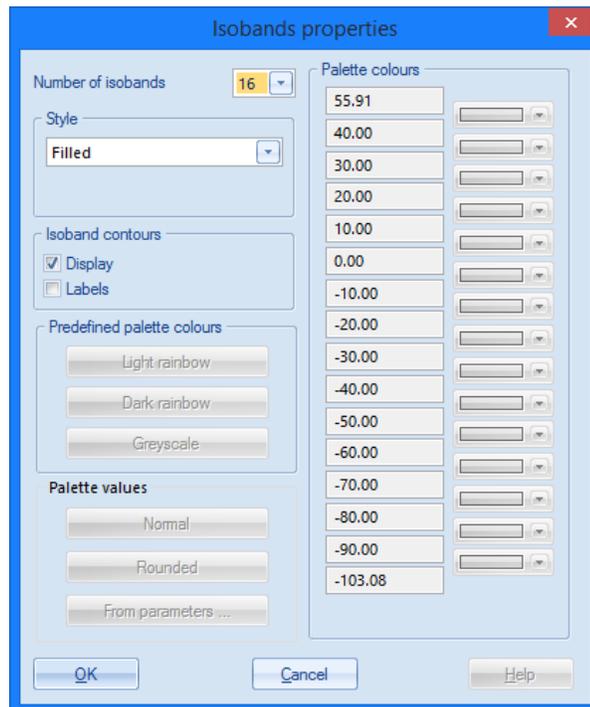
2. Note the difference for the two ribs that are modelled. It is clear that the forces in the plate are reduced, because the joint stiffness of the slab and the ribs is now considered.

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.



2. For 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.



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.

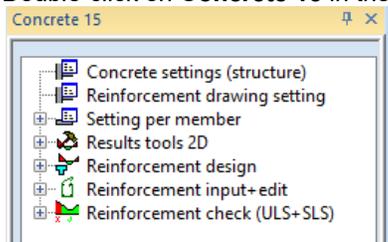
Note:

*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

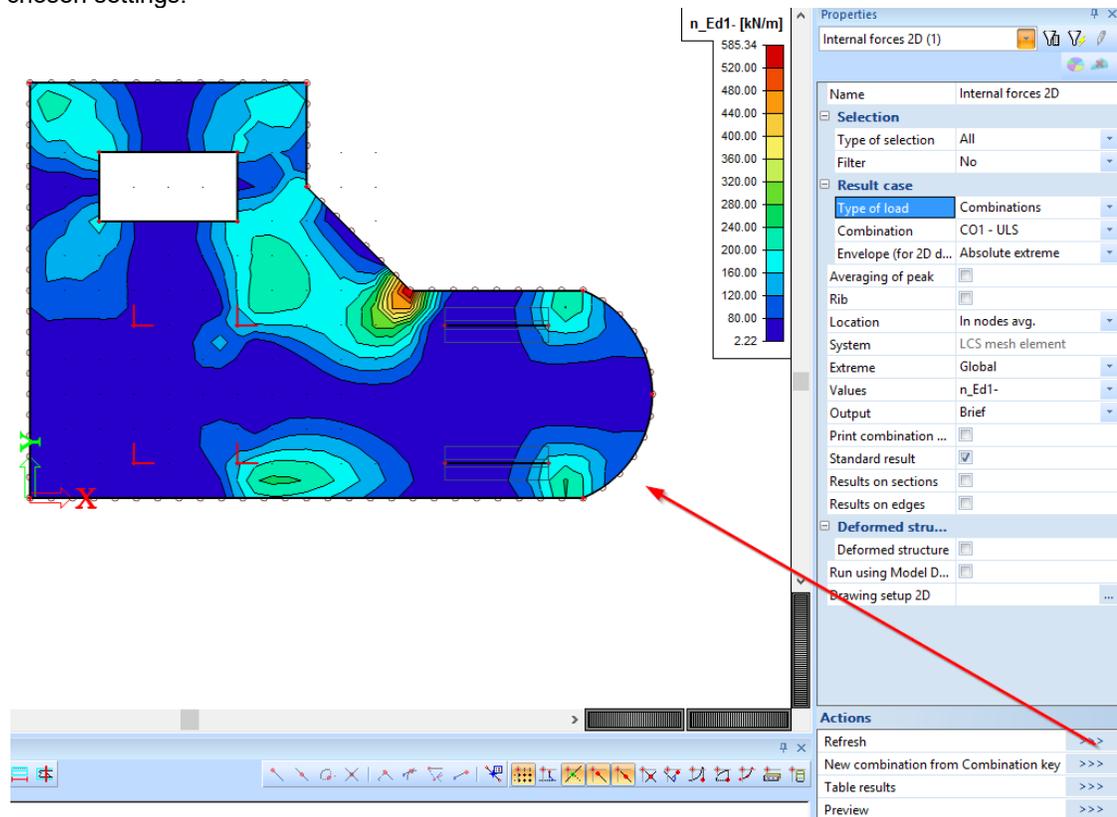
The reinforcement design can be performed in the Concrete 15 menu.

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



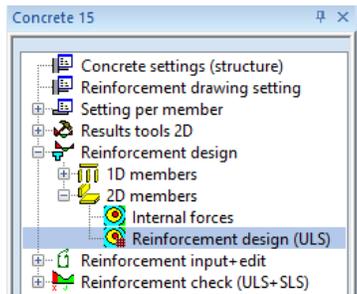
Viewing design internal forces

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.

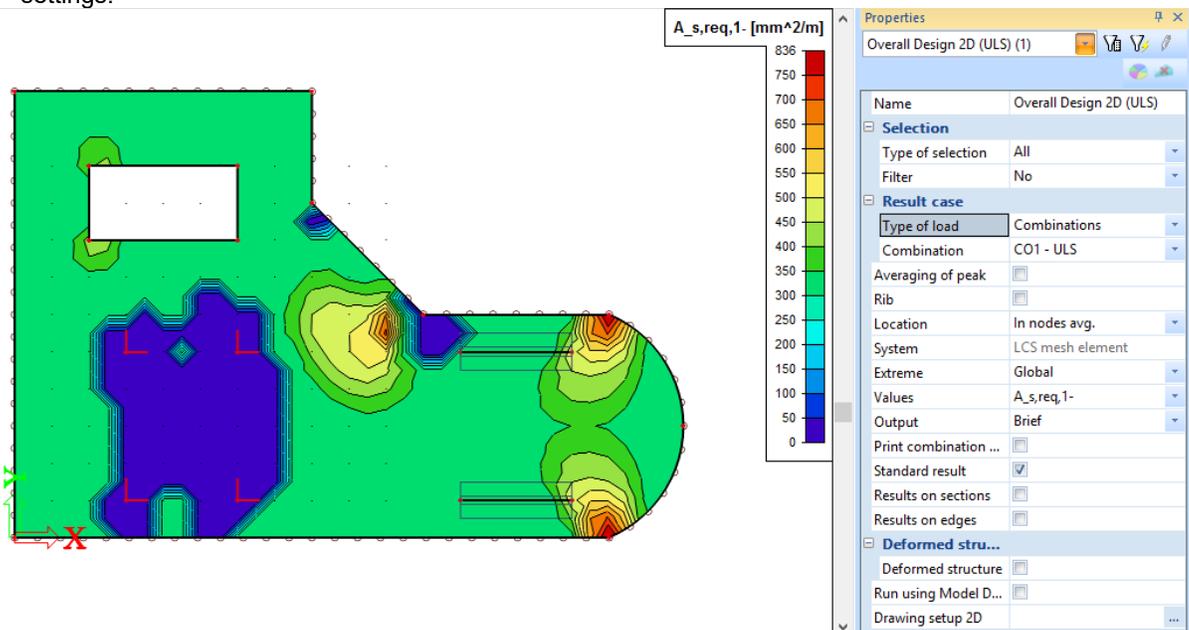


Reinforcement design ULS

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 one-page text output with the most important data.

Report preview

Member S1 Type **Plate [h=250mm]**
 EC EN 1992-1-1 Location Node 2/1 [X=16m, Y=0m, Z=0m]

Member data
 Concrete: **C30/37** $f_{ck} = 30$ MPa
 $f_{cd} = 20$ MPa
 $\epsilon_{cu3} = 3.5$ ‰
 Reinforcement: **B 500A** $f_{yk} = 500$ MPa
 $f_{yd} = 434.78$ MPa
 $\epsilon_{yk} = 2.174$ ‰
 Cover: upper: $c_{nom} = 30$ mm
 lower: $c_{nom} = 30$ mm

Design forces
 [1-]: $n_{Ed} = 201.4$ kN/m $m_{Ed} = 52.7$ kNm/m [CO1/1]
 [2-]: $n_{Ed} = 201.4$ kN/m $m_{Ed} = 6.8$ kNm/m [CO1/1]
 [2+]: $n_{Ed} = 201.4$ kN/m $m_{Ed} = 6.8$ kNm/m [CO1/1]
 [CO1/1]: 1.35*LC1+1.50*LC2

Longitudinal reinforcement

Layer	Direction	Type	Case	$A_{s,req}$ [mm ² /m]	$A_{s,min}$ [mm ² /m]	$A_{s,max}$ [mm ² /m]	$A_{s,prov}$ [mm ² /m]
Upper [2+]	Second [90°]	Principal	CO1/1	309 (134)	309	1671	314 (φ10/250)
Lower [1-]	First [0°]	Principal	CO1/1	841	324	1671	873 (φ10/90)
Lower [2-]	Second [90°]	Principal	CO1/1	329	309	1671	341 (φ10/230)

$A_{s,req}$ - statically required reinforcement including detailing provisions, $A_{s,min}$ - minimal reinforcement from detailing provisions, $A_{s,max}$ - maximal reinforcement from detailing provisions, $A_{s,prov}$ - provided reinforcement

Tasks Page width

Command line

Command >

Overall Design 2D (ULS) (1)

Name Overall Design 2D (ULS)

Selection
 Type of selection All
 Filter No

Result case
 Type of load Combinations
 Combination CO1 - ULS
 Averaging of peak
 Rib
 Location In nodes avg.
 System LCS mesh element
 Extreme Global
 Values $A_{s,req,1}$
 Output **Standard**
 Standard result
 Results on sections
 Results on edges

Deformed stru...
 Deformed structure
 Run using Model D...
 Drawing setup 2D

Actions
 Refresh >>>
 New combination from Combination key >>>
 Table results >>>
 Preview >>>

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**:

Concrete 15

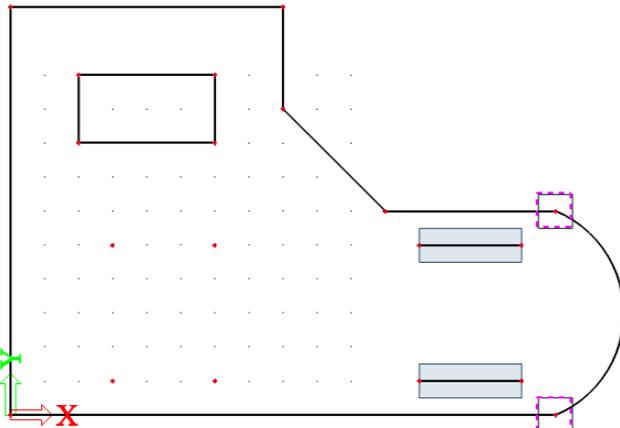
RS

Concrete settings (structure)
 Reinforcement drawing setting
 Setting per member
 Results tools 2D
 Section on 2D member
 Averaging strip
 Reinforcement design
 1D members
 2D members
 Internal forces
 Reinforcement design (ULS)
 Reinforcement input+edit
 Reinforcement check (ULS+SLS)

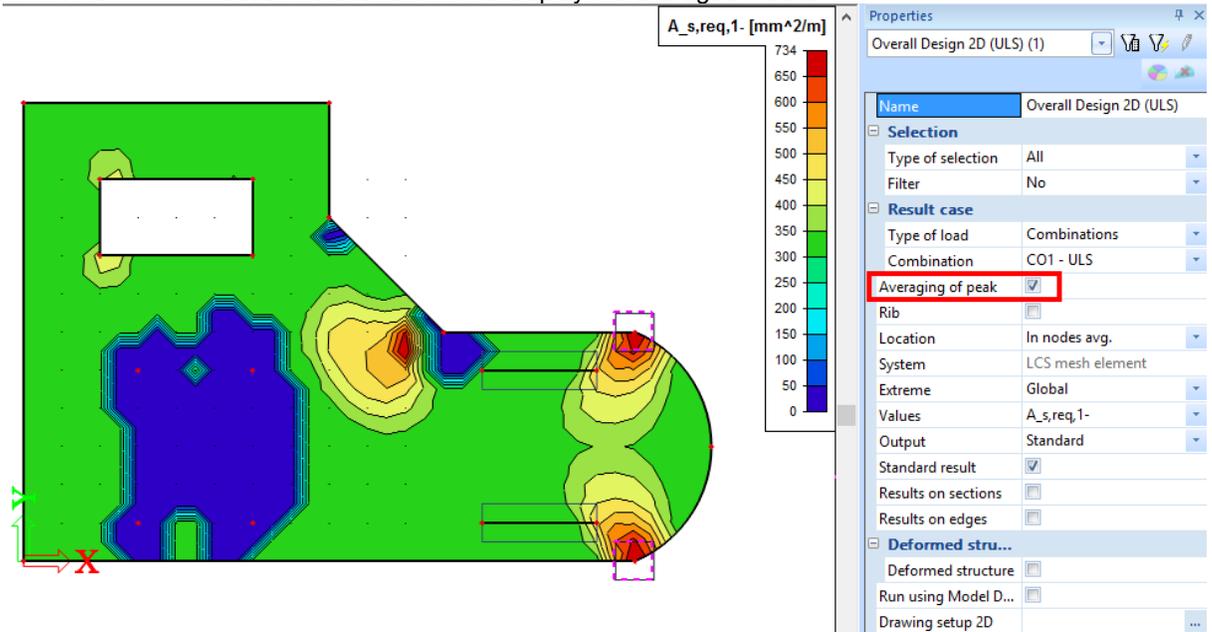
Name RS1
 Type Point
 Width [m] 1,000
 Length [m] 1,000
 Angle [deg] 0,00
 Direction both

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:

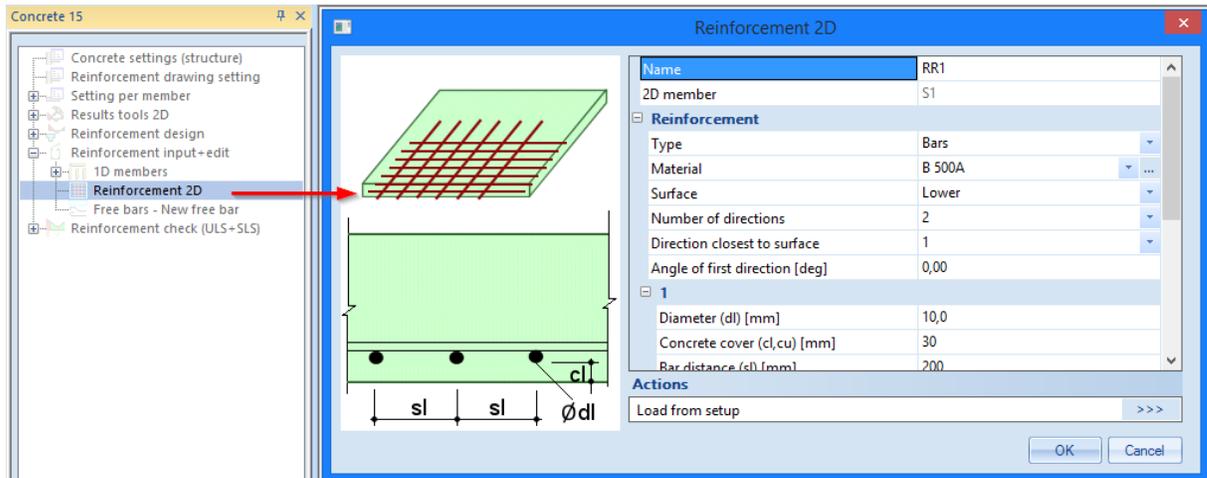


8. End the command with <ESC> key.
9. Select **Reinforcement design (ULS)** in the Concrete 15 menu and in **Properties** windows tick checkbox **Averaging of peak**.
10. Hit the **>>>** button next to **Refresh** to display the averaged results



Input of practical reinforcement

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



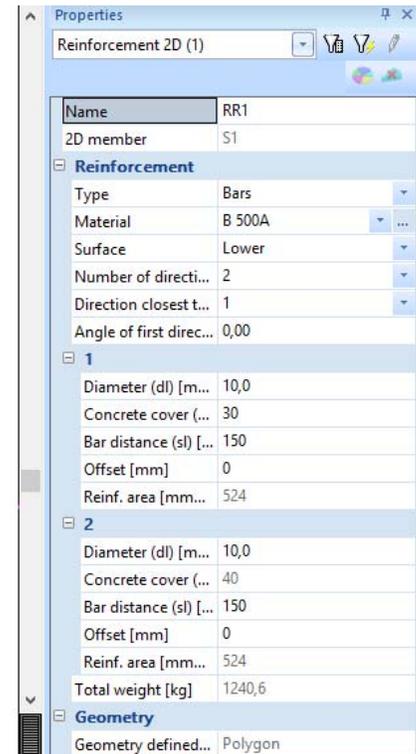
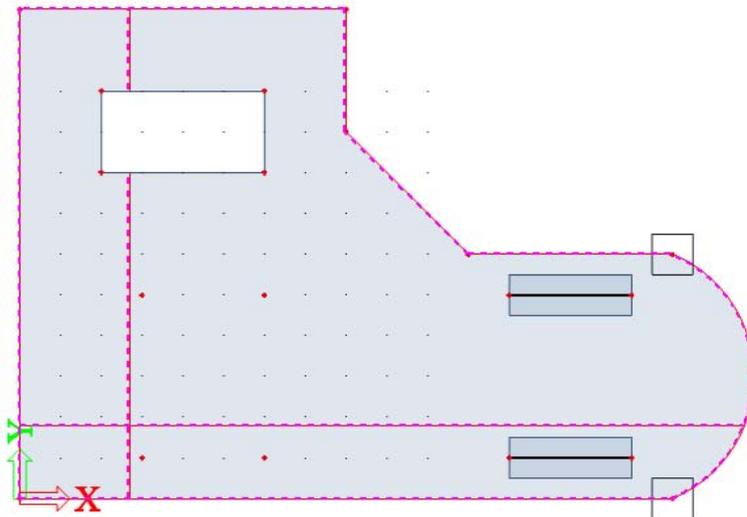
2. The parameters in the **Reinforcement 2D** window are configured in the following way:

- **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.

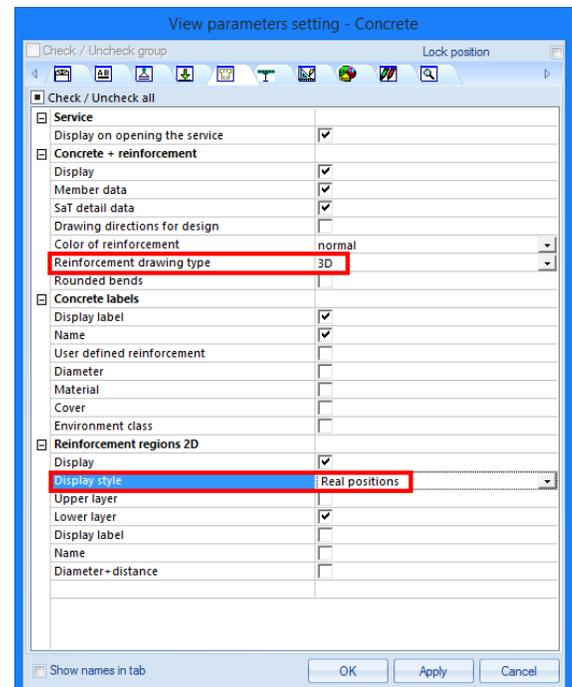
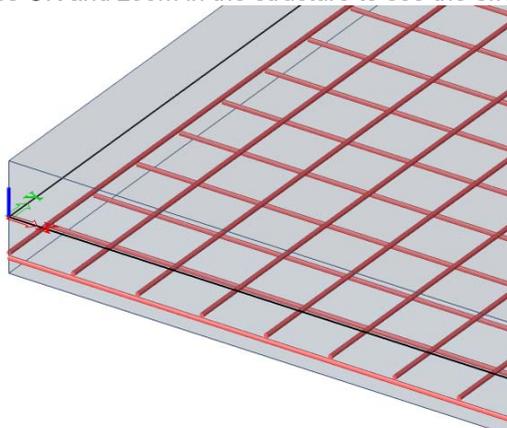
- Click **OK** to confirm that setting and define the polygonal reinforcement area.
- Hit the button **Select existing polygon/polyline** in **Command line** and click on the slab polygon with mouse cursor.



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



- Press **<ESC>** to cancel the selection.
- 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**.
- Press OK and zoom in the structure to see the effect.



Document

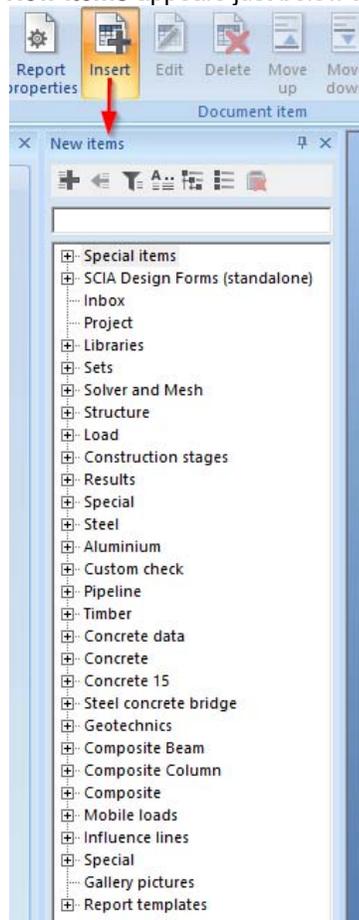
In this final part of the tutorial, we will explain how to make nice report of the calculation and design.

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  button to add this 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:

1. Materials

Name	Type	ρ [kg/m ³]	E_{mod} [MPa]	μ	α [m/mK]	$f_{ct,28}$ [MPa]	Colour
C30/37	Concrete	2500,0	3,2800e+04	0,2	0,00	30,00	

Reinforcement EC2

Name	Type	ρ [kg/m ³]	E_{mod} [MPa]	G_{mod} [MPa]	α [m/mK]	f_{yk} [MPa]
B 500A	Reinforcement steel	7850,0	2,0000e+05	8,3333e+04	0,00	500,0

2. Cross-sections

CS1	Type	
Detailed	Rectangle	
Shape type	500; 250	
Item type	Thick-walled	
Item material	C30/37	
Fabrication	concrete	
Colour		

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

Displaying results in the document

1. 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 **Regenerate selected** button in the top ribbon to display the table in accordance with the predefined options. Red exclamation mark disappears.

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]	mxy [kNm/m]	vx [kN/m]
				my [kNm/m]		vy [kN/m]
S1	Element: 3 Node: 2	16,000 0,000 0,000	CO1/1	55,91 3,57	16,74	60,06 -132,93
S1	Element: 541 Node: 20	6,000 5,000 0,000	CO1/1	-93,62 -105,06	23,59	-276,83 282,73
S1	Element: 519 Node: 601	12,500 3,000 0,000	CO1/1	22,21 62,45	-0,66	2,39 -2,48
S1	Element: 167 Node: 168	7,000 0,500 0,000	CO1/1	-1,11 -2,93	-25,88	61,78 -2,91
S1	Element: 20 Node: 5	11,000 6,000 0,000	CO1/1	-81,57 -54,72	56,50	-127,93 -113,53
S1	Element: 21 Node: 209	10,496 5,920 0,000	CO1/1	-3,37 -52,09	28,03	-309,83 122,12
S1	Element: 19 Node: 58	11,500 6,000 0,000	CO1/1	-33,04 9,23	2,01	318,87 192,74
S1	Element: 593 Node: 20	6,000 5,000 0,000	CO1/1	-103,08 -95,64	27,67	290,34 -282,22
S1	Element: 567 Node: 20	6,000 5,000 0,000	CO1/1	-100,61 -97,93	-13,47	290,13 283,69

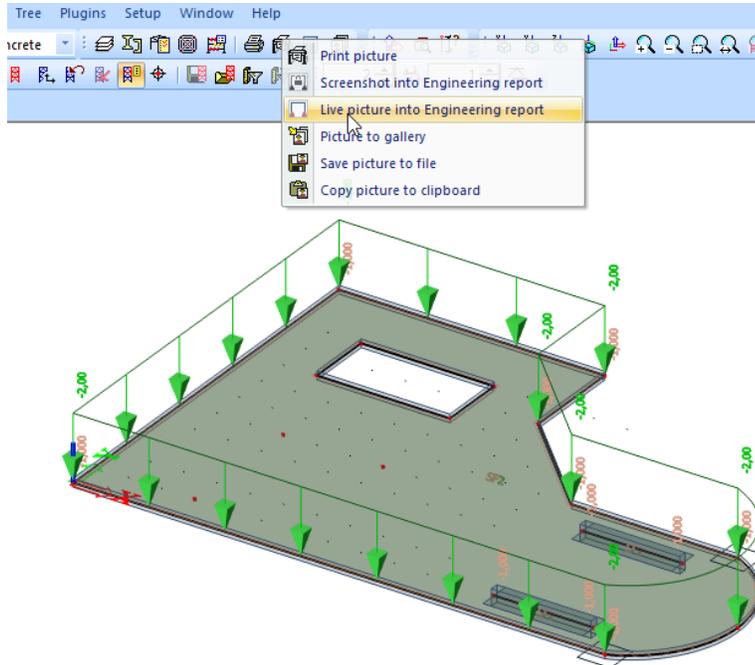
Name	Combination key
CO1/1	1.35*LC1 + 1.50*LC2

Adding an image to the Report

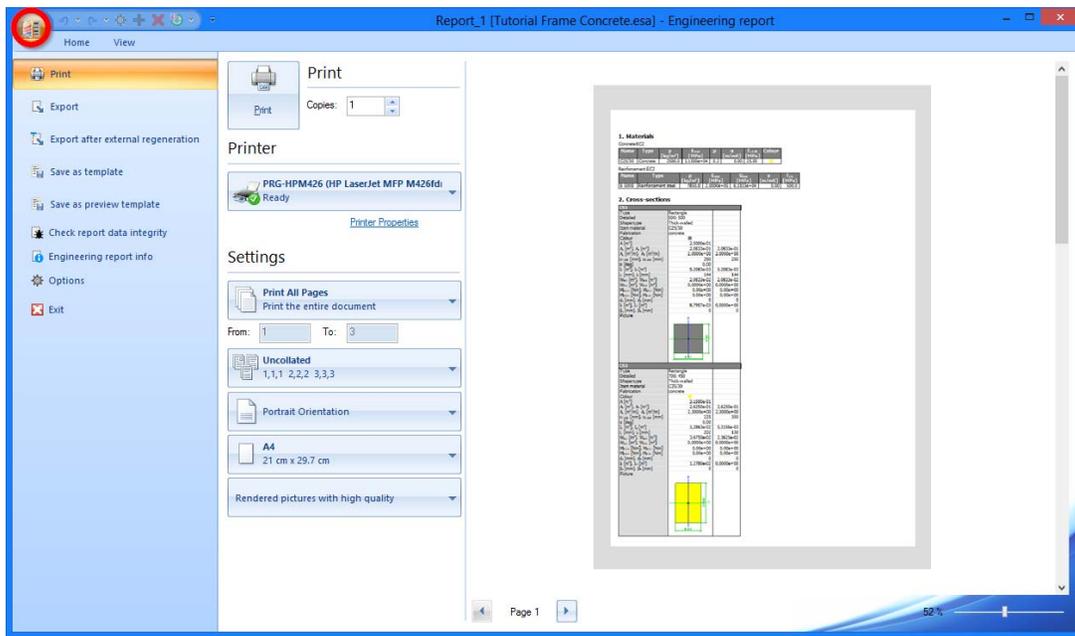
1. Any picture from SCIA Engineer application can be set to Engineering Report. Either as print-screen (that is unchanged for ever) or as live picture (that can be regenerated and is always up-to date).
2. Prepare any scene in the 3D modelling window, for example the analytic model with loads. You can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



3. Click on button **Print Picture in toolbars** and select **Live picture into Engineering Report**



4. 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



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.