NEMETSCHEK
Scia


Stade des Alpes - Grenoble
Etudes et Techniques Internationales (ETI)

Tutorial
Plate Concrete

Scia
Engineer

Table of Contents

General Information

Welcome

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

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

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

Scia Engineer is available in three different editions:

<u>License version</u>

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

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

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

<u>Demo version</u>

If the program doesn't find a protection, it will automatically start the demo version. The properties of the demo version are:

All projects can be inserted;

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

The output contains a watermark "Unlicensed software";

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

<u>Student version</u>

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

The output contains a watermark "Student version".

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

Scia Engineer Support

You can contact the Scia Engineer support service

By e-mail

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

By telephone

From Belgium : +32 13 350310

From the Netherlands : +31 26 3201230

Via the Scia Support website

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

Website

Link to Tutorials

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

Link to eLearning

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

Link to Demo version

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

Start writing on an odd page.

Introduction

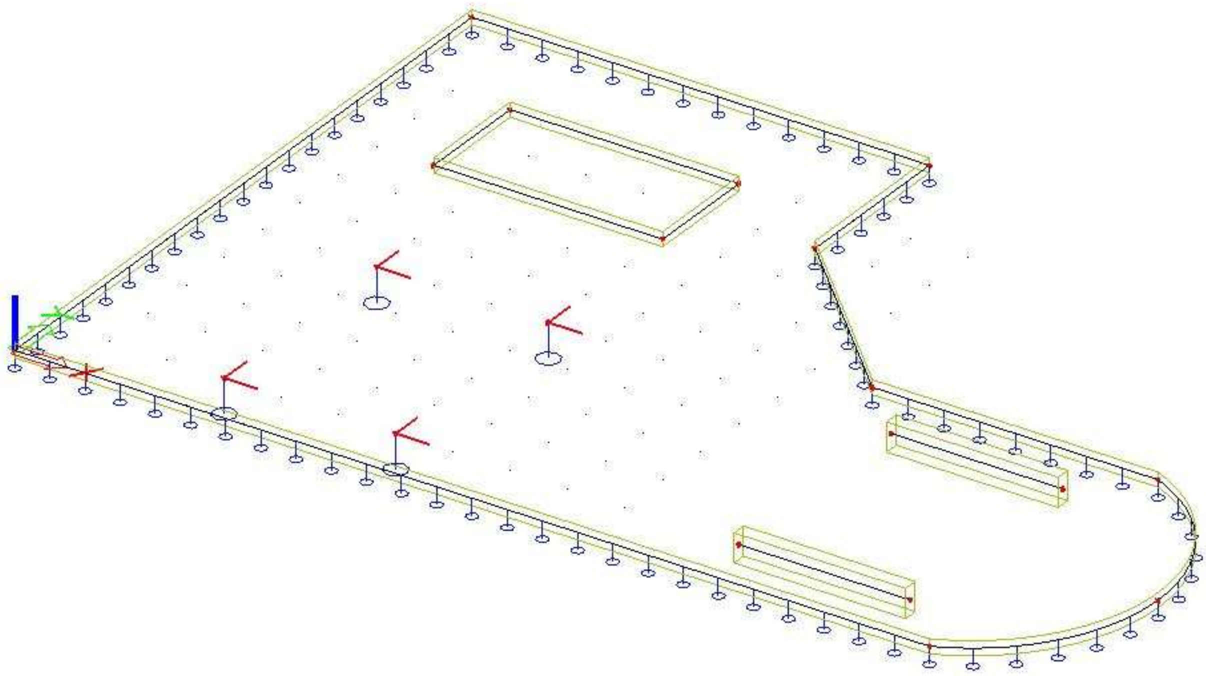
The example of this Tutorial can be designed with the **Licensed** or **Student** Versions. Before you proceed, you must be familiar with your operating system: for instance working with dialogues, menu bars, toolbars, status bars, handling the mouse, etc.

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

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

Next, we will discuss the input of the buckling parameters and we will perform the steel check, the profile optimization and calculate a connection.
The Tutorial ends with a brief introduction to the calculation note.

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



Getting started

Starting a project

Starting the program

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

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

   Or :

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

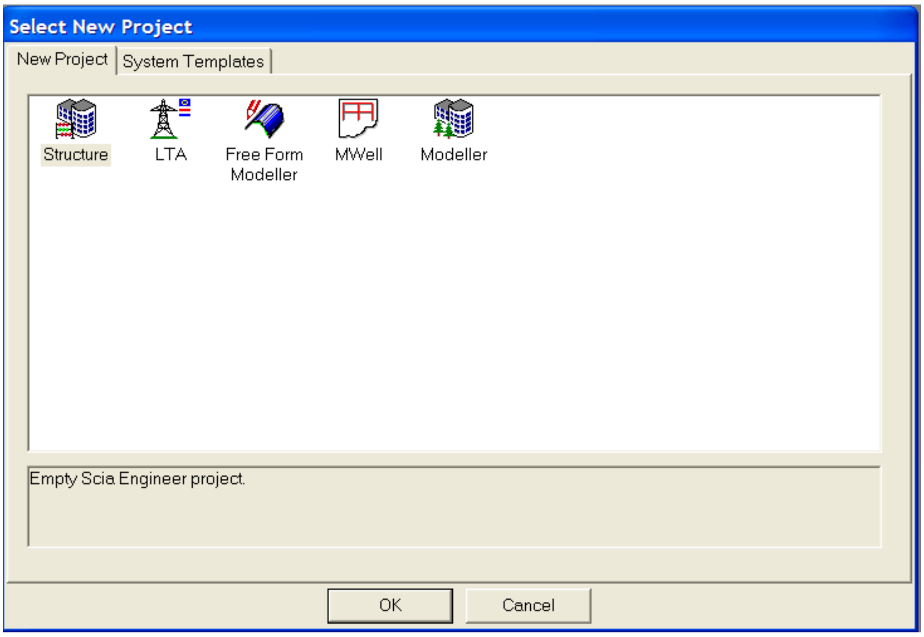
If the program does not find any protection, you will obtain a dialogue indicating that no protection was found. A second dialogue will list the restrictions of the demo version. Click **[OK]** in both windows.

For this Tutorial, you must start a new project.

Starting a new project

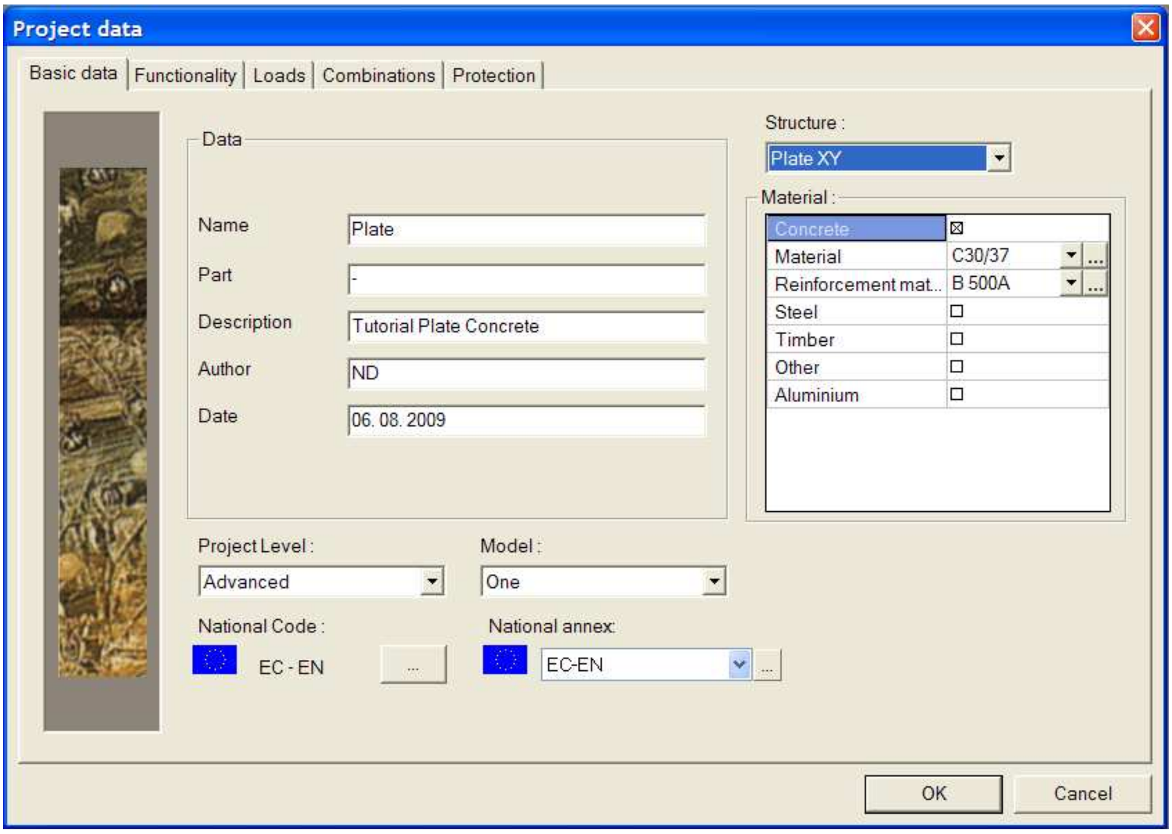
1. If the dialogue **Open** appears, click **[Cancel]**.

2. Click the **New** icon  in the toolbar.



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

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



3. In the **Data** group, enter your preferred data. These data can be mentioned on the output, e.g. in the document and on the drawings.

4. Choose the **Project level: Advanced** and **Model: One**.

5. Click on the rectangular button  below **National Code** to choose the default code for the project. This code will determine the available materials, combination rules and code checks. For the project of this Tutorial, choose EC-EN. The window **Codes in project** is opened.

- Click **[Add]**.
- The dialogue **Available national codes** is opened.

- Select the European flag and click **[OK]**.
- You will return to the **Codes in project** dialogue and **EC-EN** is added.
- Select the flag named **EC-EN**.
- Select the **Active code** option and click **[Close]**.
- You will return to the **Project data** window and **EC-EN** is the active code.

6. If you have chosen for the EC-EN, you will have the possibility to choose for a National Annex for an European country.

7. Select **Plate XY** in the **Structure** field.

   The structure type (Frame XZ, Frame XYZ, Plate XY, General XYZ,...) will restrict the input possibilities during the calculation.

8. In the **Material** group, select **Concrete**.

   Below the item **Concrete**, a new item **Material** will appear.

9. Choose **C30/37** from the menu.

10. Confirm your input with **[OK]**.

Notes:

*On the **Basic data** tab, you can set a project level. If you choose "standard", the program will only show the most frequently used basic functions. If you choose "advanced", all basic functions will be shown.*
*On the **Functionality** tab, you choose the options you need. The non-selected functionalities will be filtered from the menus, thus simplifying the program.*
*On the **Combinations** tab, you will find the values for the partial safety factors. In this Tutorial, we will use the default settings.*

Project management

Save, Save as, Close and open

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

Saving a project

Click on  in the toolbar.

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

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

Closing a project

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

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

Opening a project

Click on  to open an existing project.

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

Geometry input

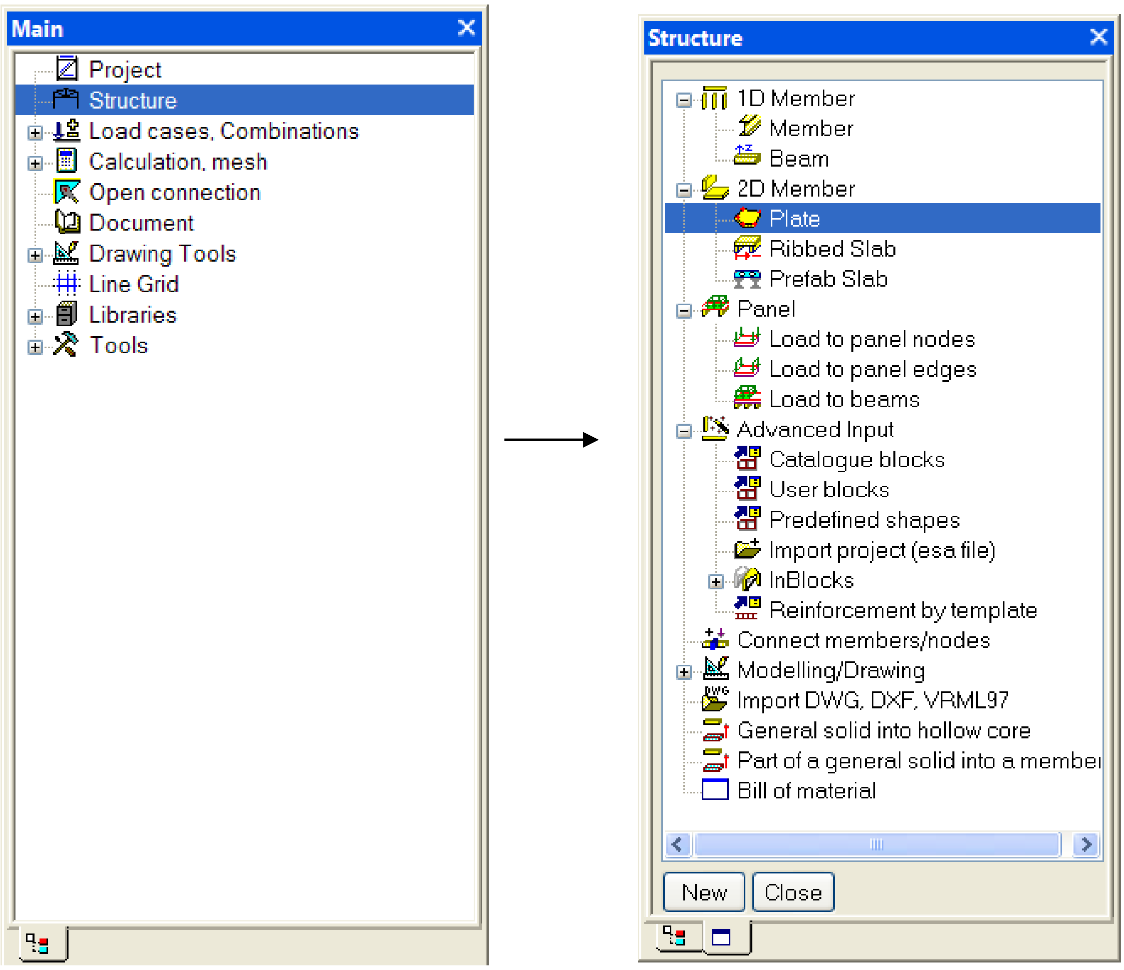
Input of the geometry

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

Geometry

Structure menu

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

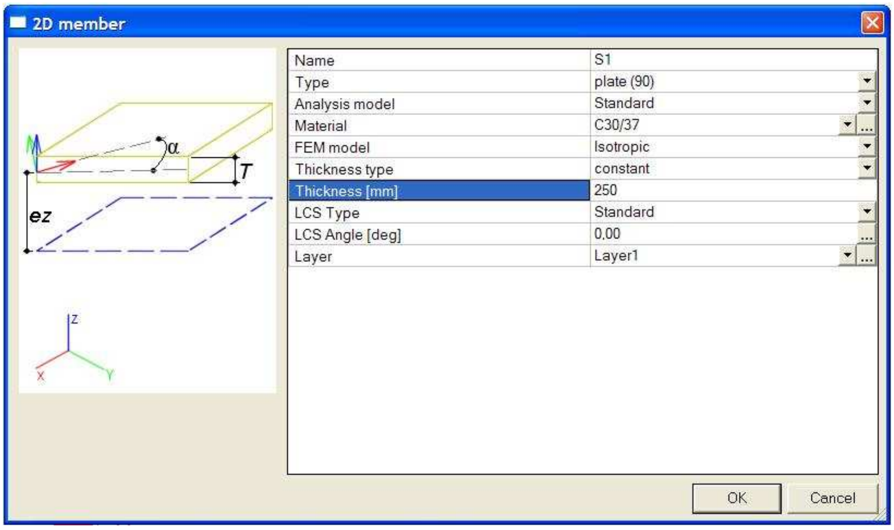


2. In the **Structure menu** different branches will appear, in accordance with the already input items, i.e. support branch will appear if a structure is physically available.

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

Input of a plane 2D member

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



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

3. After accepting with [OK] the program asks in the command line for the starting point of the polyline.

4. The buttons in the **command line** allow one to built up the polygonal edges using different line types, or to choose directly for a circular or rectangular surface.

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

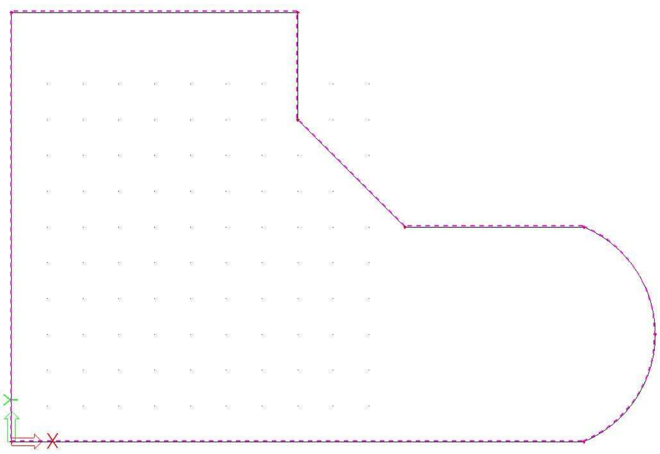
Starting point: 0;0 <enter>
16;0 <enter>

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

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

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

The following picutre is now depicted in the screen:



Definition of an opening

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

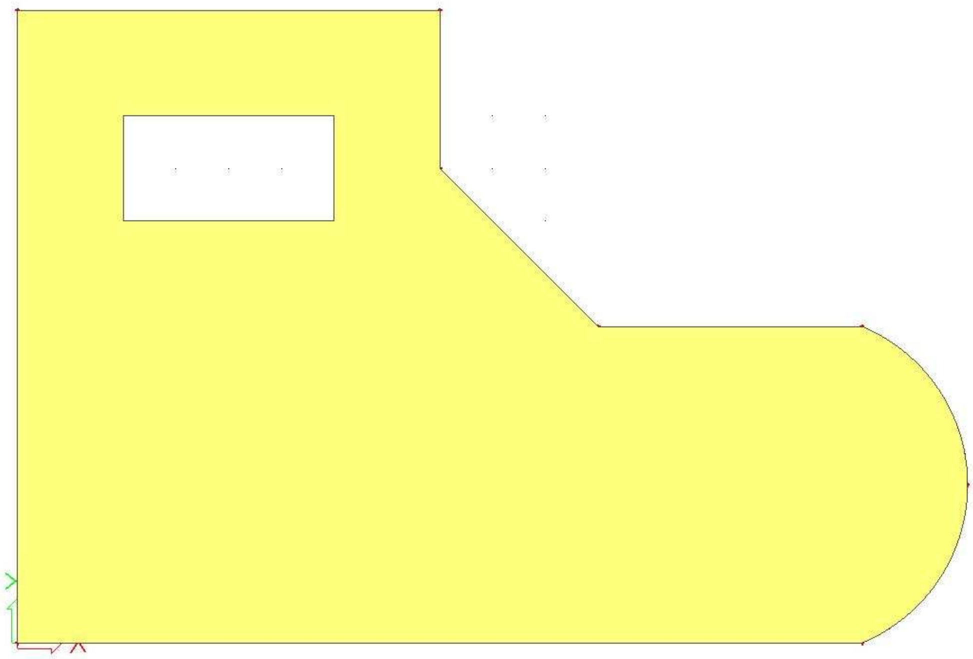
2. In the command line quick access buttons are available for quick definition of geometrical outlines like circular or rectangular slabs. By default definition by drawing of a closed polygon is started, in this example we will define a rectangular opening.

3. The two nodes of its diagonal define a **rectangle**. This is also depicted by the two red dots on the icon.

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

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

By setting the rendering to ON,  The result can be visually checked:



Note

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

Input of internal nodes

1. In the **Structure menu** we choose under **2D element components** to input **internal nodes.**

2. We wil add four new internal nodes:

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

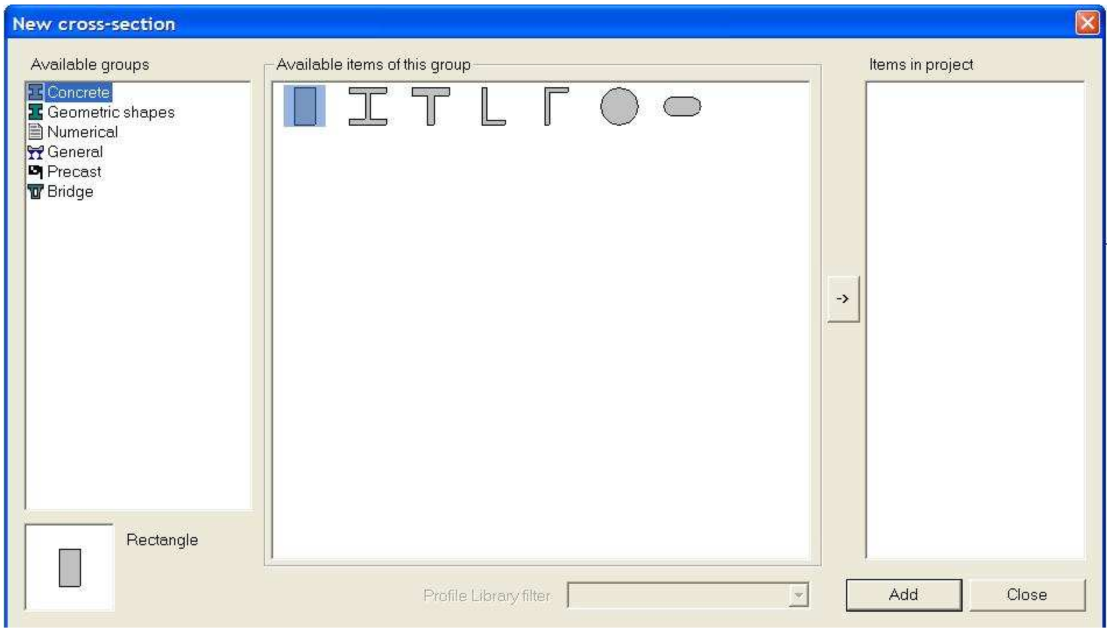
   right mouse click in order to end the input session.

Input of plate ribs

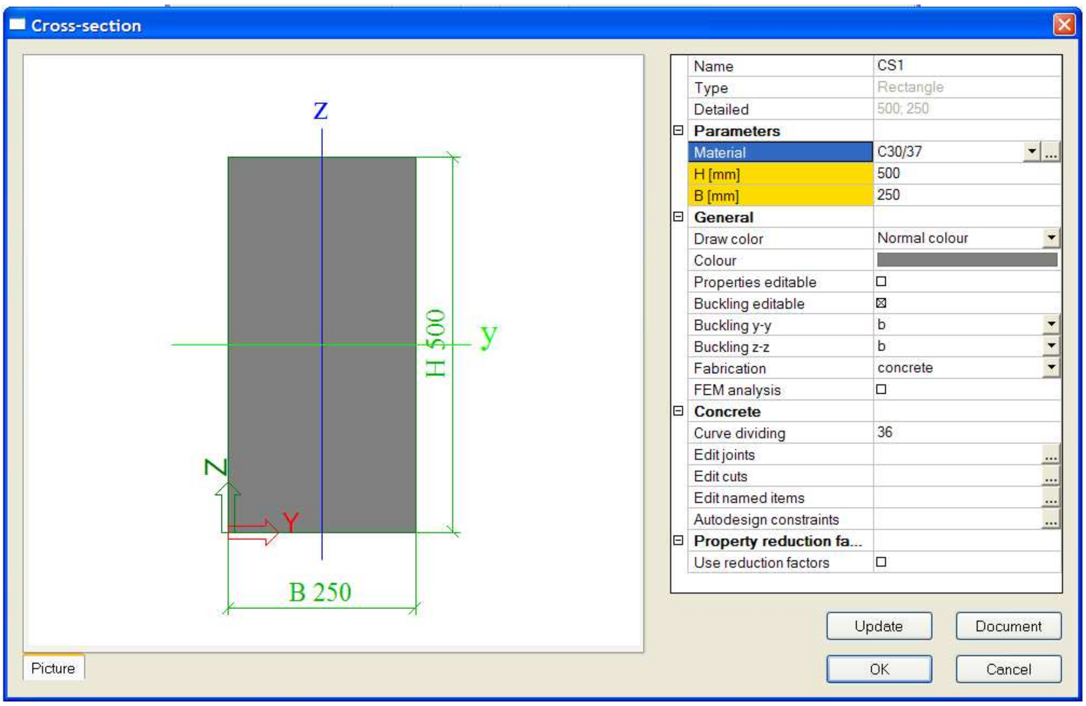
1. In the **Structure menu**, under **2D member components** we choose R**ib.**

2. If no cross-section was chosen in the project, the dialogue **New cross-section** will pop up.

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

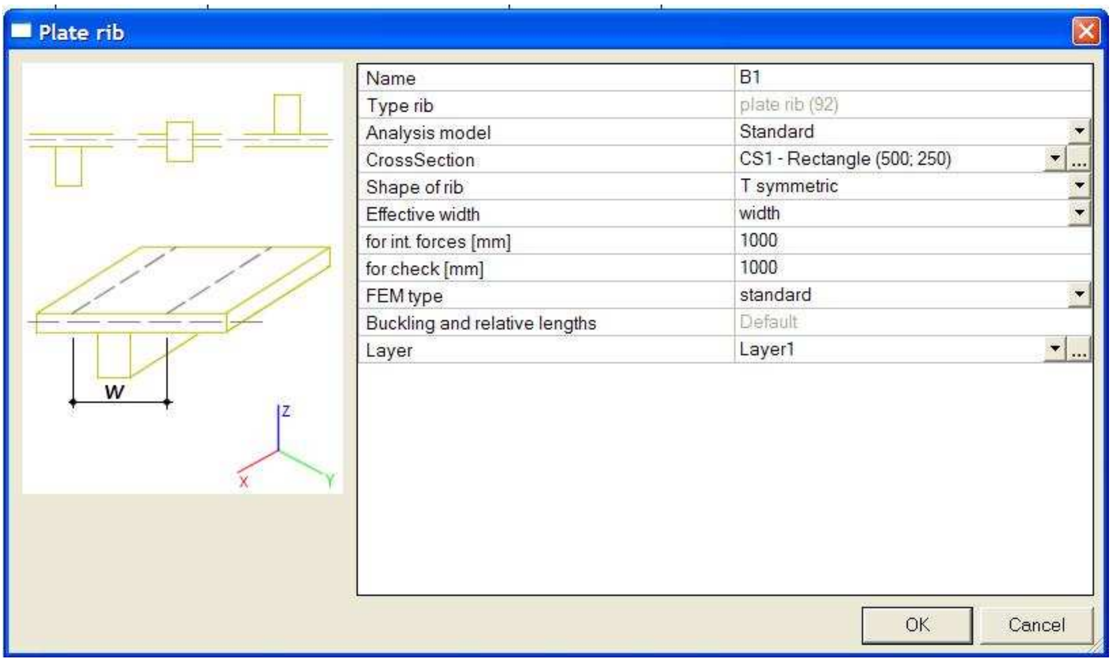


3. Click **[Add]**. This will take us to a new dialogue **Cross-section**. For this Tutorial we will take a rectangular concrete cross-section with **height 500 mm** and **width 250 mm** and a concrete grade of C30/37.



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

5. In window **Plate rib** we can define the parameters of the ribs:



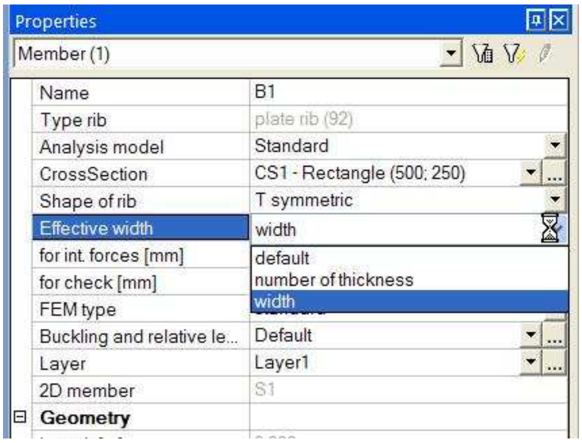
6. After **<OK>** we will have to define the starting and end points of the ribs.

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

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

   right mouse click to end the input session.

Note



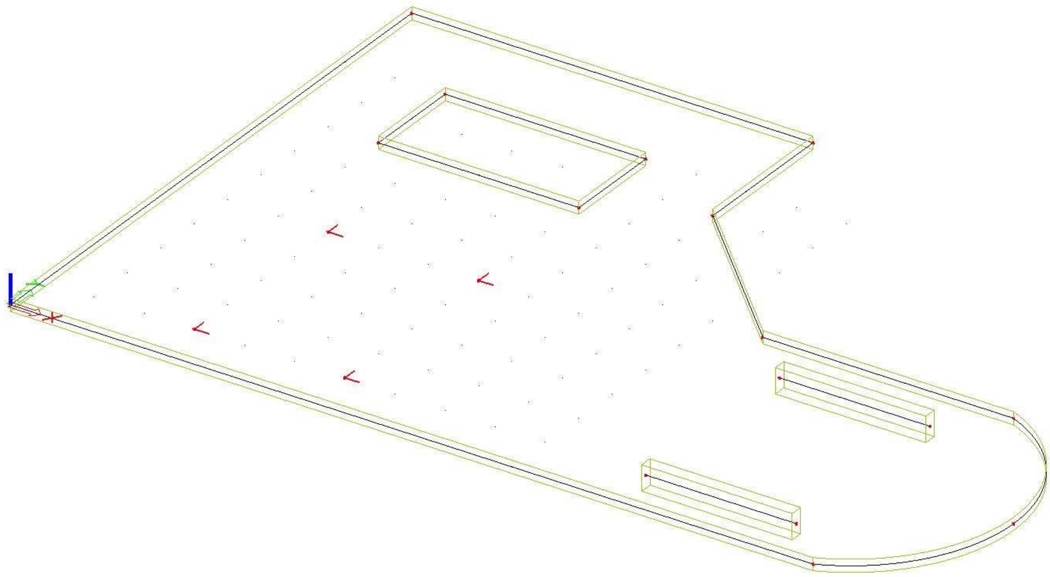
width *: The user can input the width for the internal forces (FE analysis) or the checks (Design As) by hand.*

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

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

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

By viewing the surfaces  the input values of the geometry can be checked easily:



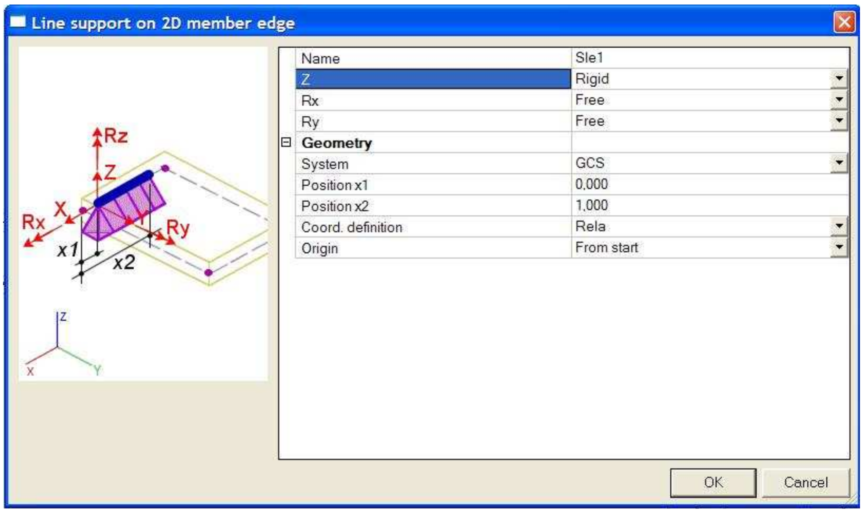
By pressing **<Esc>** one can easily cancel the selection.

Supports

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

Definition of a support on an edge

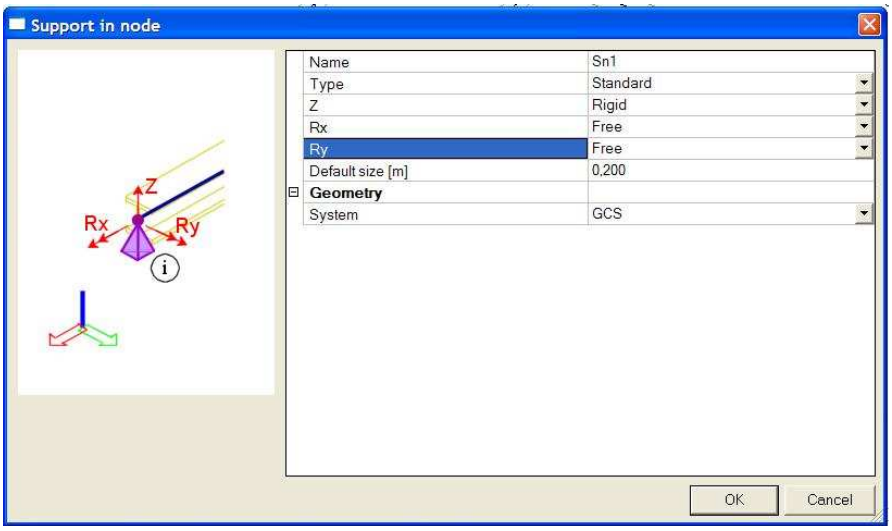
1. Select in the **Structure** menu **Model data -> Support > line on 2D member edge**
2. The window **Line support on 2D member edge** will pop up.



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

Input of Nodal supports

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



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

Note

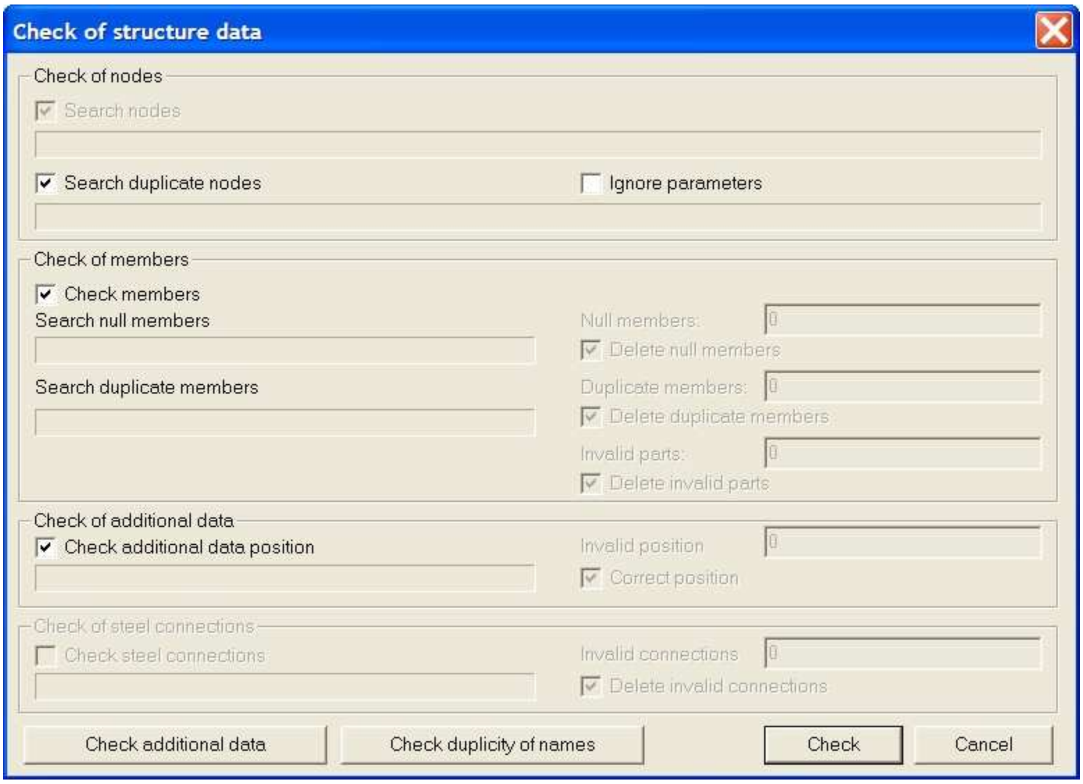
- *If required a flexible support can be defined in order to model the behavior of the columns more adequately. Also only can model the supports as „column ‟, then the stiffness is directlyderived for the entered column data.*
- *A set of shortcuts of supports is defined in the **Command line**. In this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

Check Structure data

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

Checking the structure

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



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

6. Close the check by clicking **[OK]**.
7.

Connecting entities

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

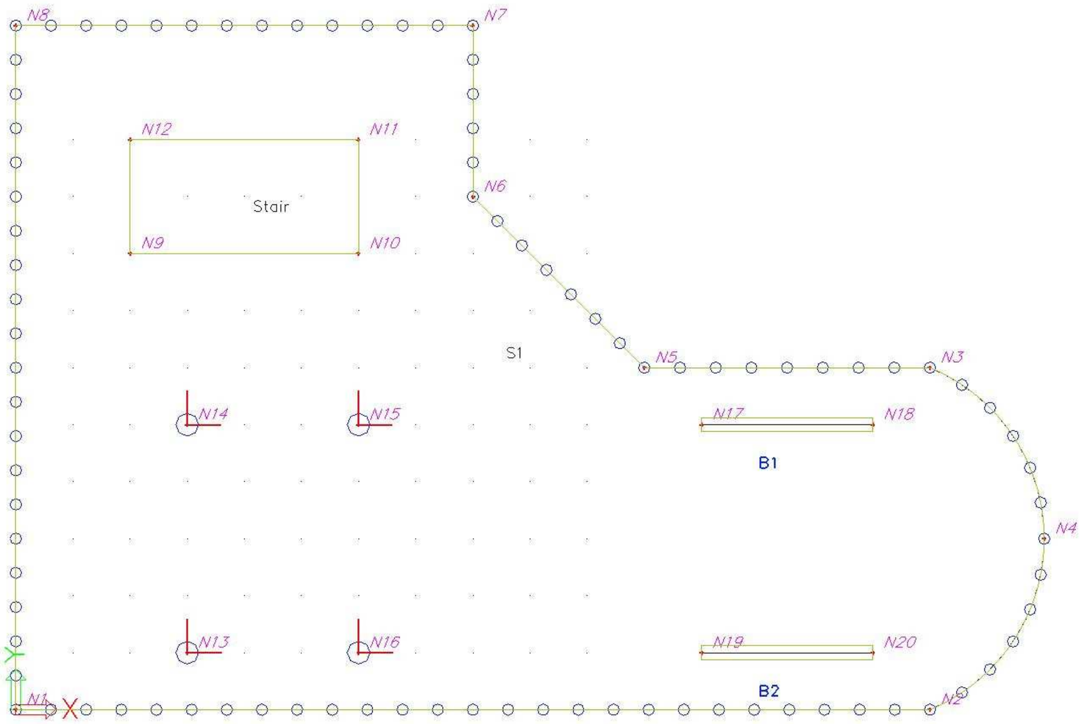
In order to display the names of the entered bars and nodes or support symbols the labels of each item can be turned ON / OFF by the shortcut button in the lower left corner of the graphical screen  above the **Command line**.

The 3rd button can visualize supports. 

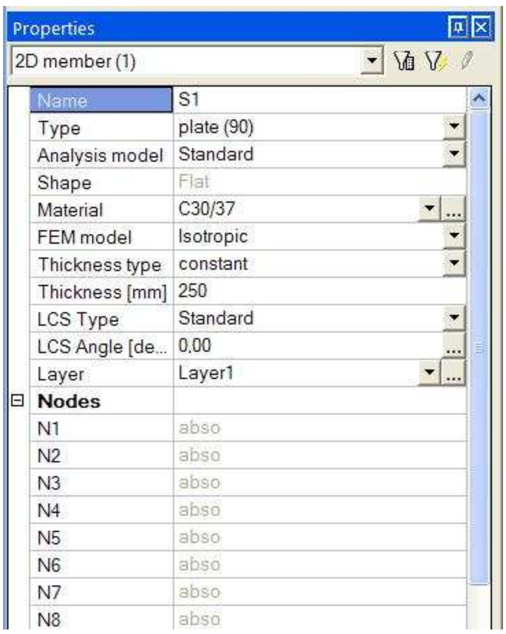
Labels of nodes can be activated by the button  located above the **Command line**

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

A view in direction Z  shows the following:



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



Properties

2D member (1)

Name	S1
Type	plate (90)
Analysis model	Standard
Shape	Flat
Material	C30/37
FEM model	Isotropic
Thickness type	constant
Thickness [mm]	250
LCS Type	Standard
LCS Angle [de...	0.00
Layer	Layer1
Nodes	
N1	abso
N2	abso
N3	abso
N4	abso
N5	abso
N6	abso
N7	abso
N8	abso

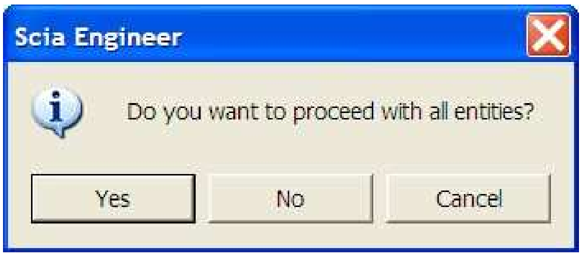
The properties contain for instance also the nodes on the outline of the slab. Additional data, like predefined line supports, internal nodes, openings and ribs will also be depicted. However it is required that the elements have to be connected using the option **Connect members/nodes**.

Connecting entities

1. Press **<ESC>** or click the **Cancel selection** icon to deactivate any selection of entities.

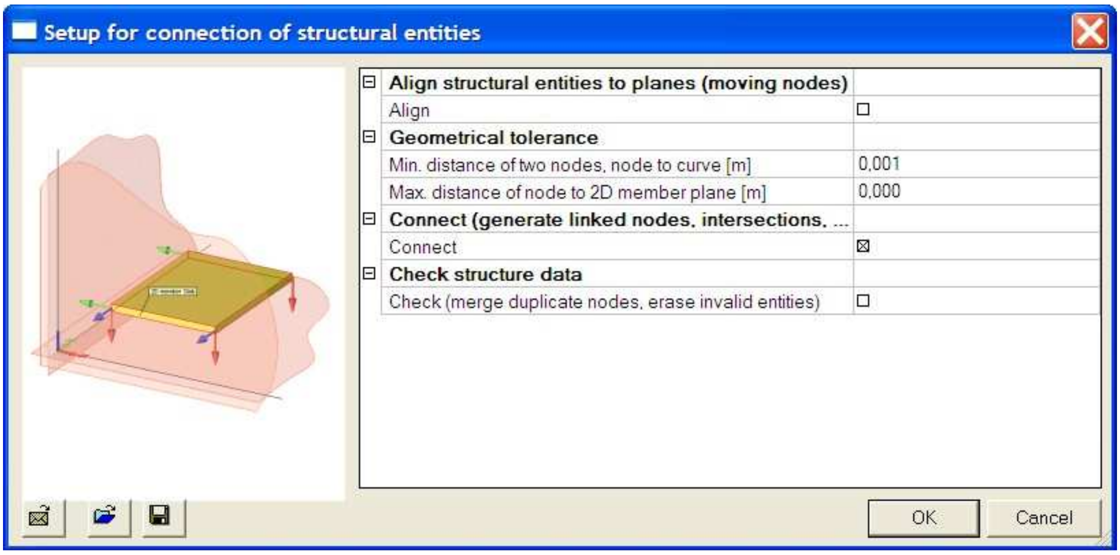
2. Double-click on the **Model data > Connect members/nodes** option in the **Structure menu** or click the icon in the toolbar.

3. A dialogue asks if all nodes must be connected to bars:



4. Click **<Yes>**.

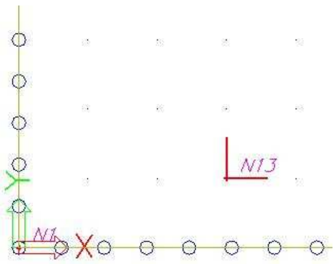
5. The **Setup for connection of structural entities** dialogue box now appears.



6. Confirm the settings by clicking **<OK>**.

7. A window appears to indicate the number of connected nodes:

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



Note

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

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

Graphic representation of the structure

Edit view

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

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

Editing the view point on the construction

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

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

   OR

   Set the view point by combining the buttons and mouse.

2. Press CTRL + right mouse button at the same time and move the mouse to **turn** the construction.

3. Press SHIFT + right mouse button at the same time and move the mouse **move** the construction.

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

Note

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

Setting a view direction with regard to the global coordinate system

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

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

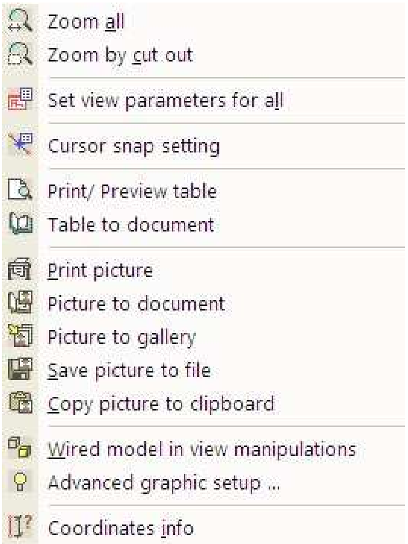
3. Click on the button **View in direction- Z**  for a view in the Z-direction.

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.

Editing view parameters through the menu View parameters

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



Note

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

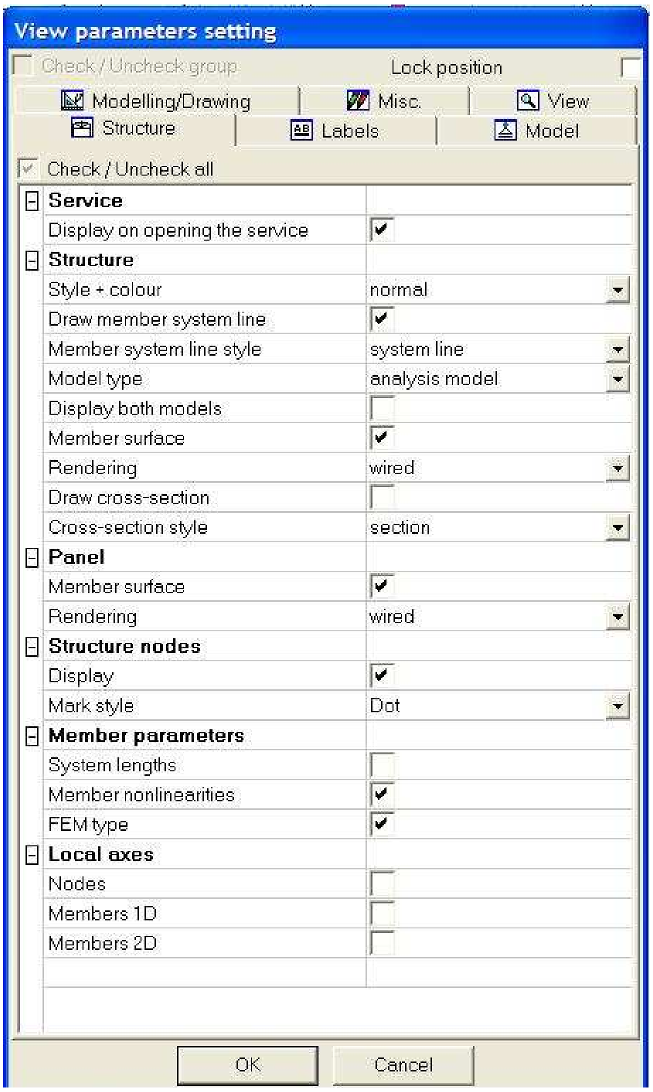
2. Choose the option **Set view parameters for all**. The window **View parameter setting** appears. The menu consists of various tabs. You can set the view parameters for all entities or just for the selected entities.

View parameters – Entities

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

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

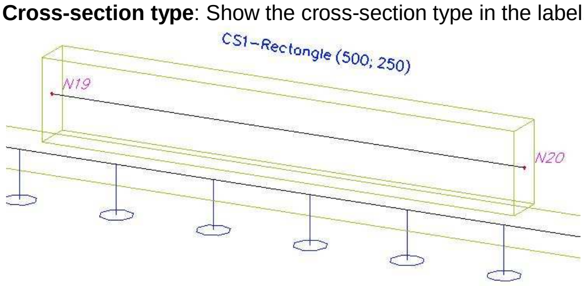
Local Axes: Using this tool the local axes can be set for nodes, 1D and 2D members..



View parameters – Labels and description

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

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



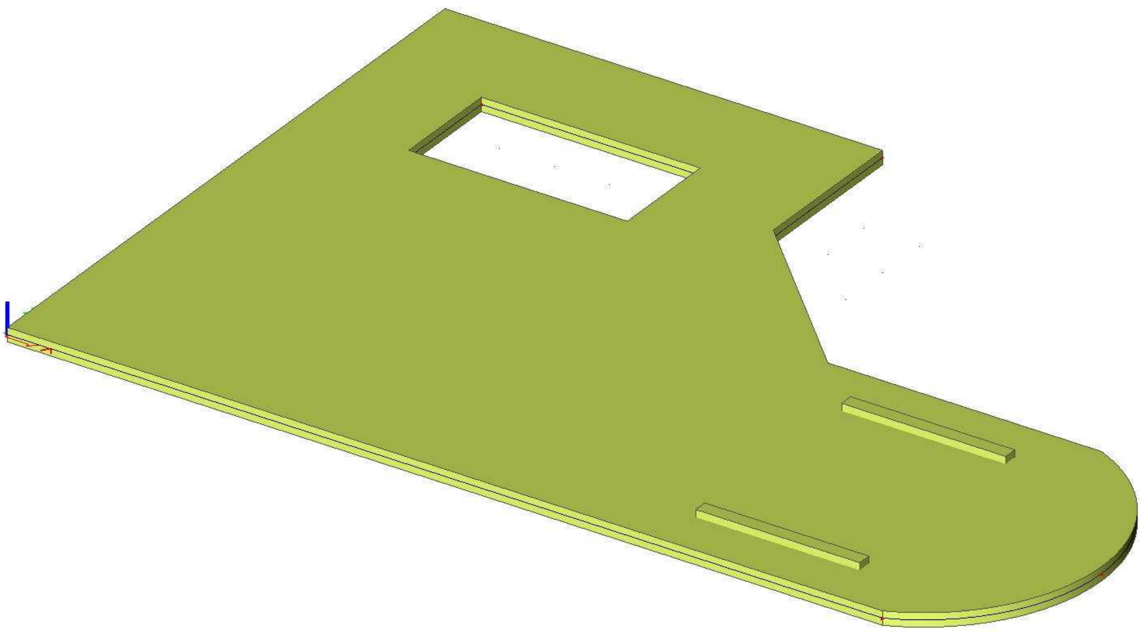
- **Length**: show the length of the beam in the label.
- **Display labels**: Only when this tick box is turned ON, the labels will be depicted on the graphical screen.

View parameters – shortcuts

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

- **Show/hide surfaces** to show the surfaces of the elements.
- **Render geometry** to view the rendered members.
- **Show/hide supports** to show supports and hinges.
- **Show/hide load** to show the load case.
- **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 structure is obtained:



Input of the Calculation Data

Load Cases and Load Groups

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

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

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

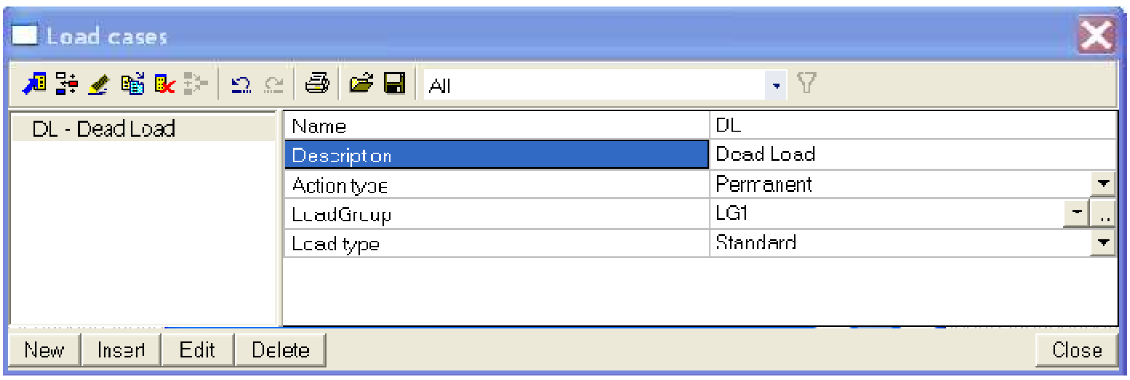
The way, in which load cases are defined, is decisive for the load combinations created by the generator. We recommend that you thoroughly read the chapter about loads and combinations in the reference manual.

In this project, two load cases are entered:

- **LC1**: Dead load
- **LC2**: Life Load Case

Defining a Permanent Load Case

1. Double-click on 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 you will also manually enter loads in the first load case of this project (Roof Weight), you must change the Load Type to **Standard**.
5. In the Description field, you can describe the content of this load case. For this project, enter the description **"Self Weight Structure"**.



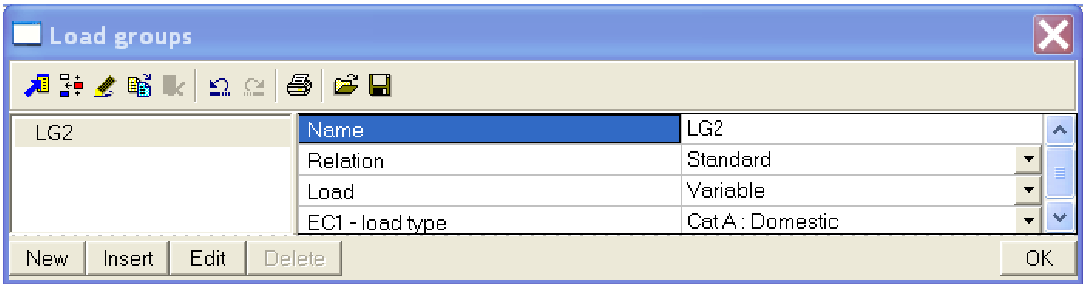
Defining a Variable Load Case

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

2. Enter the description **"Live load"**.

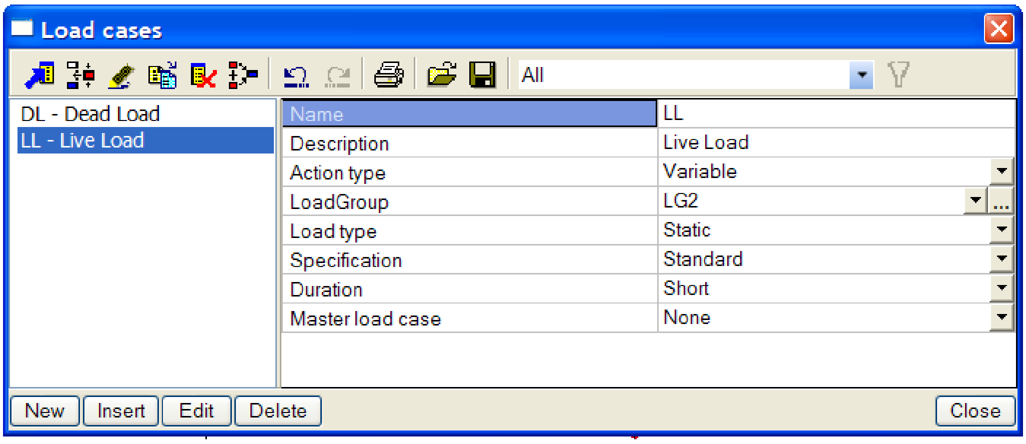
3. As this is a variable load, change the Action type to **Variable**.

4. The Load Group LG2 is automatically created. Click ... to display the properties of the Load Group.



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

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



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 standard-dependant factors to be applied. The following logic is adopted.

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

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

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

Loads

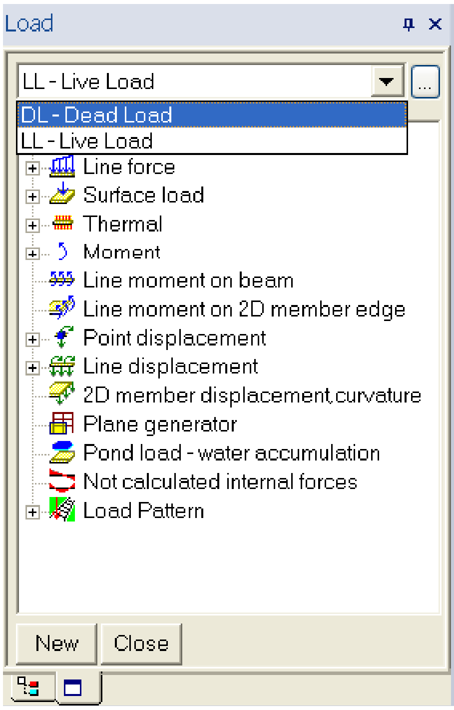
After input of the Load cases, the **Loads menu** will automatically appear:

The first load case includes two loads:

- Dead load
- Self weight

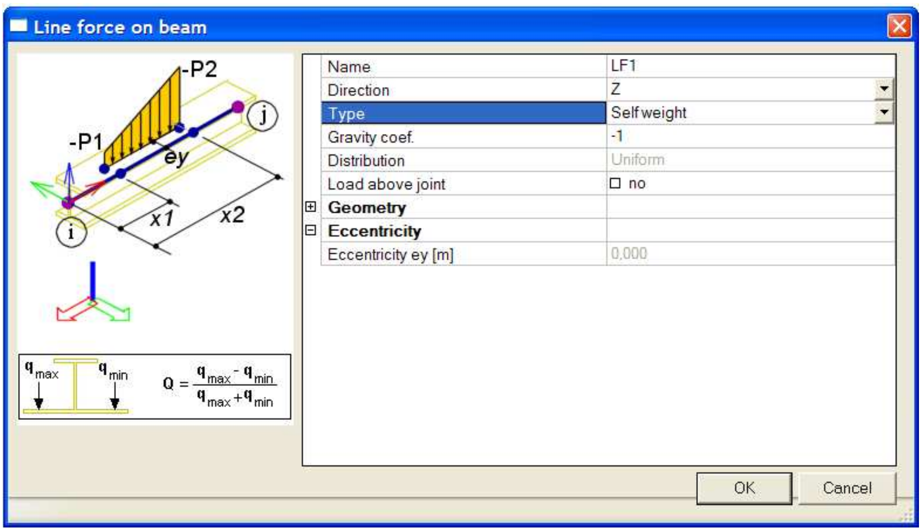
Switching between load cases

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



Entering the self weight of ribs as line loads

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



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

4. Confirm your input with **[OK]**.

5. Select all the bars by means of the **Select all**  icon in the toolbar.

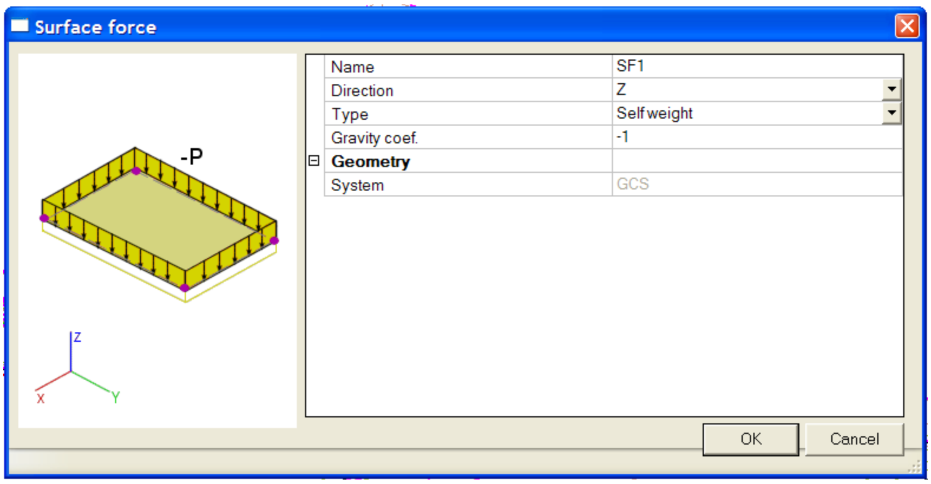
6. Press **<ESC>** to finish the input.

7. Press **<ESC>** once more to finish the selection.

Input of the selfweight of the slab as surface load

1. Cancel any possibly active selection by pressing **<Esc>**

2. Click on **Surface load on 2D member** in the **Loadsmenu**. The dialogue window **Surface force** will pop up

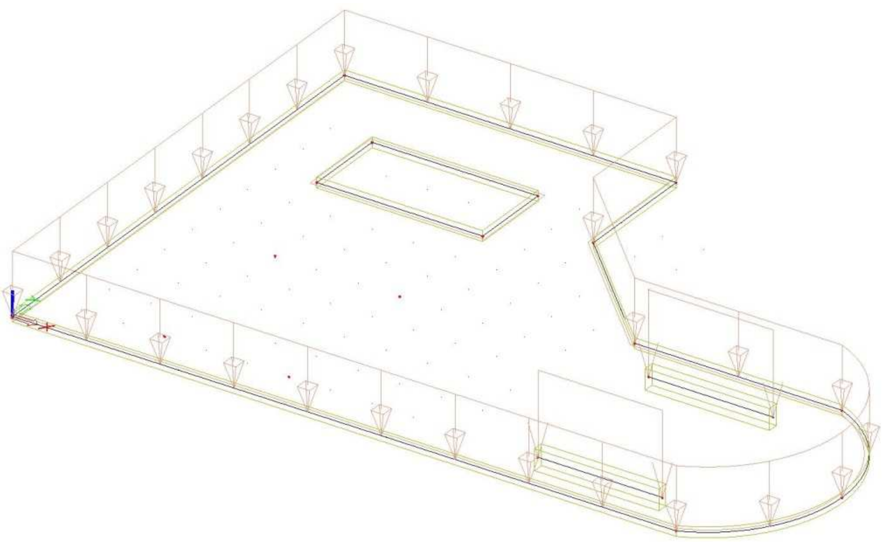


3. In the field **Type**, choose **Self Weight**. The Direction is the global Z-direction and the Gravity coefficient is set to –1, so that the load is acting vertically downwards.

4. Confirm your input with **[OK]**.

5. If there is only one slab in the project, the load will be automatically put on the slab.

The self-weight is depicted by a brown colour:

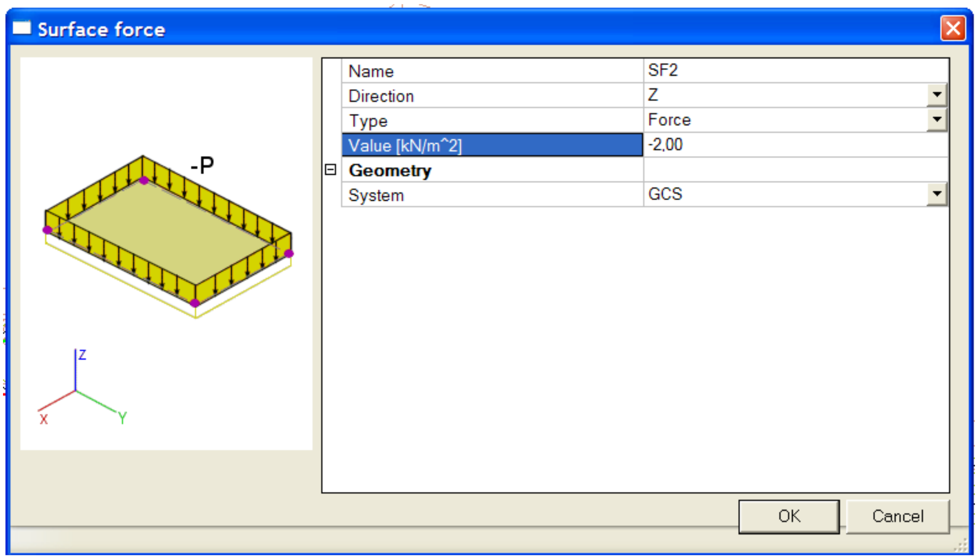


The entered loads are so-called self weight loads. The load caused by the non-structural topping will be added to the dead load load case. So that dead loads are combined into one load case.

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

Input of dead surface loads

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



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.

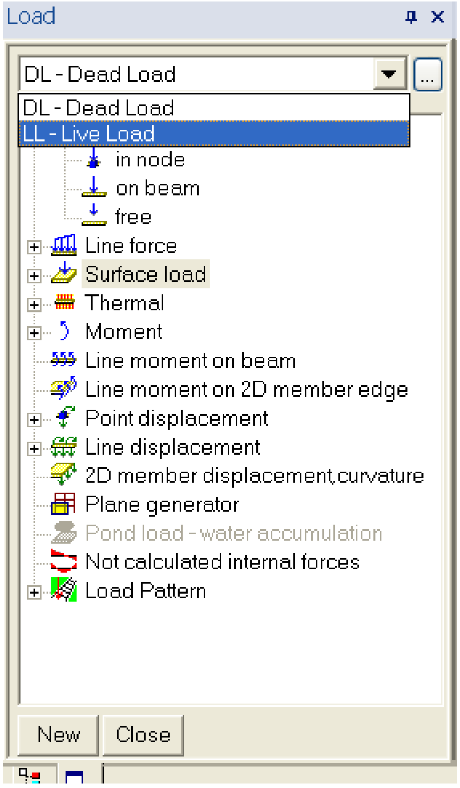
4. The **Value** of the surface load will be set to **–2 kN/m²**.

5. Confirm your input with **[OK]**.

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

Switching between load cases

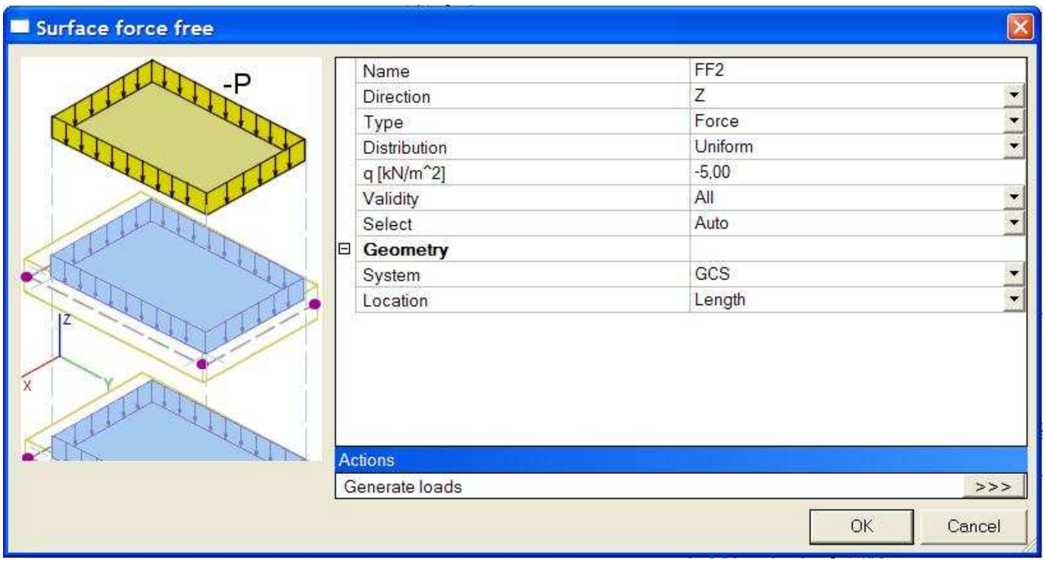
Activate load case 2 “Live load” by selecting this load case with the mouse pointer in the list box:



Input of live surface loads

1. Cancel any possibly active selection by pressing **<Esc>**.

2. Click on **Surface load – free** in the **Loads menu**. The dialogue window **Surface force free** will pop up.



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**. We will choose global coordinate system for this exercise. The force is –5 kN/m² and equally distributed over the surface.

4. Confirm your input with **[OK]**.

5. The program will asks us to define the outline polygon of the free surface load

   Starting point:

   8;9 <enter>

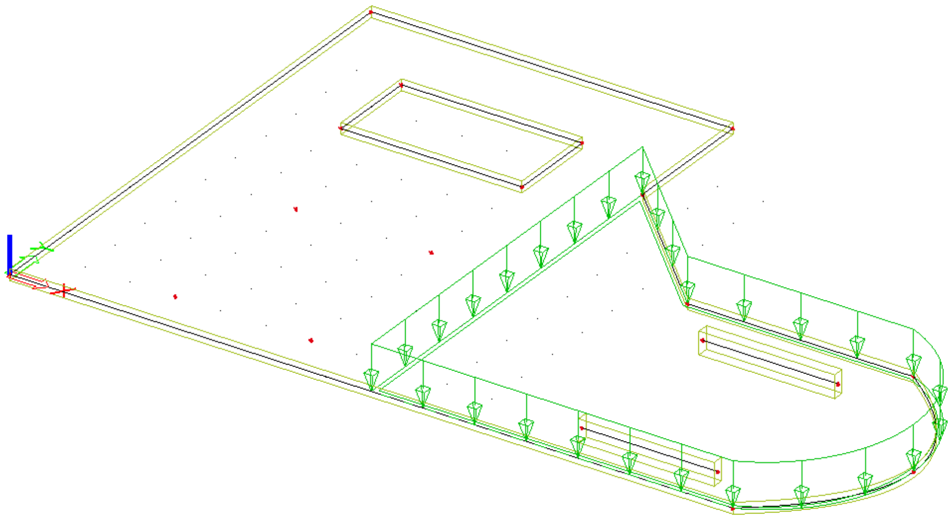
8;0 <enter>
16;0 <enter>

New circle arc – intermediate point :

18;3 <enter>

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

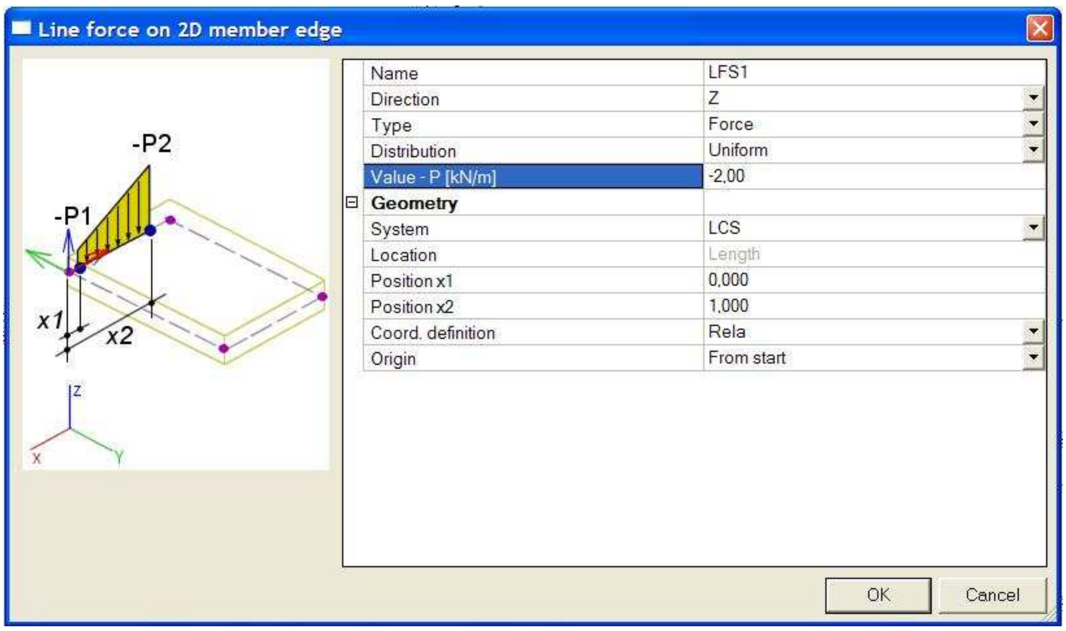
Right mouse click on **New Polygon** to close the input:



Input of variable line load

1. Cancel any possibly active selection by pressing **<Esc>**.

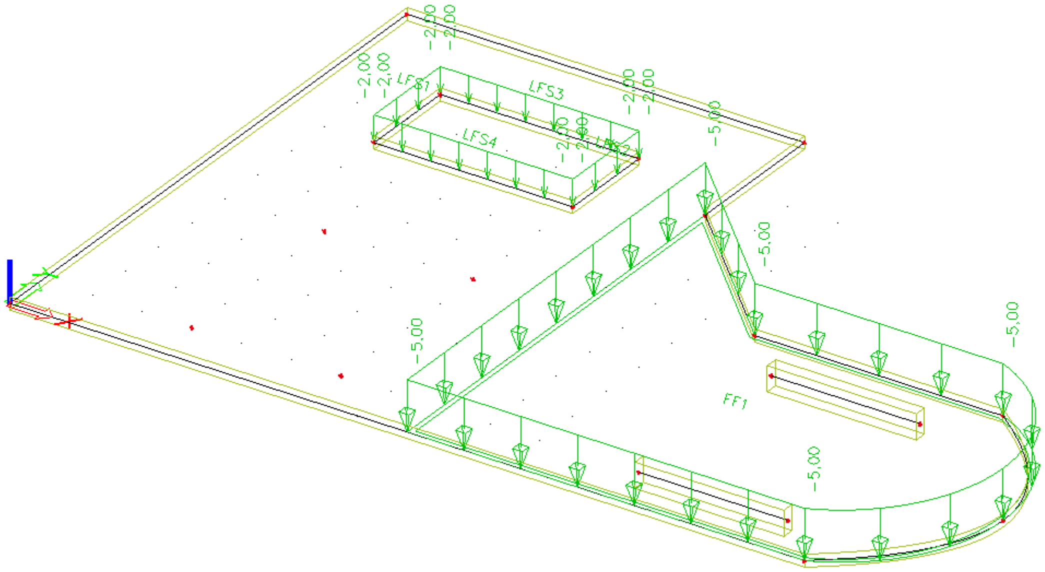
2. Click on **Line Force – on 2D member edge** in the **Loads menu**. The dialogue window **Line force on 2D member edge** will pop up.



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

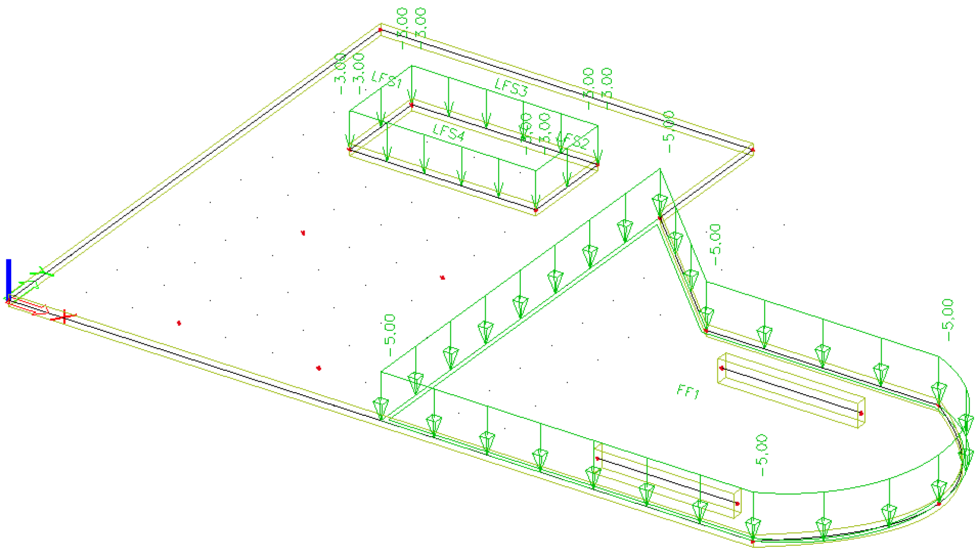
4. Confirm your input with **[OK]**.
5. Select the four edges around the hole of the staircases.
6. Click on the right mouse button to finish the input.
7. Click on <ESC> to cancel the selection.

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



Adapting a load

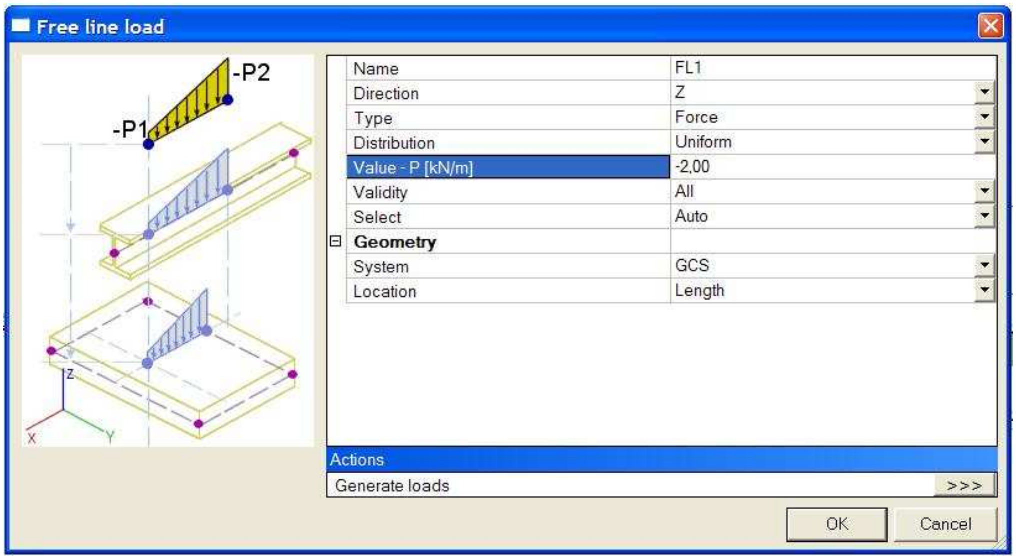
1. Select the variable line loads around the opening by clicking with the left mouse button on these loads.
2. The common properties of the 4 series are displayed in the **Properties window**.
3. Change the **Value** from **–2,0 kN** to **–3,0 kN** in the Properties window.
4. Confirm the modification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



Input of a free line load

1. Cancel any possibly active selection by pressing **<Esc>**.

2. Click on **Line Force – free** in the **Loads menu**. The dialogue window **Free line load** will pop up.



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

4. Confirm your input with **[OK]**.

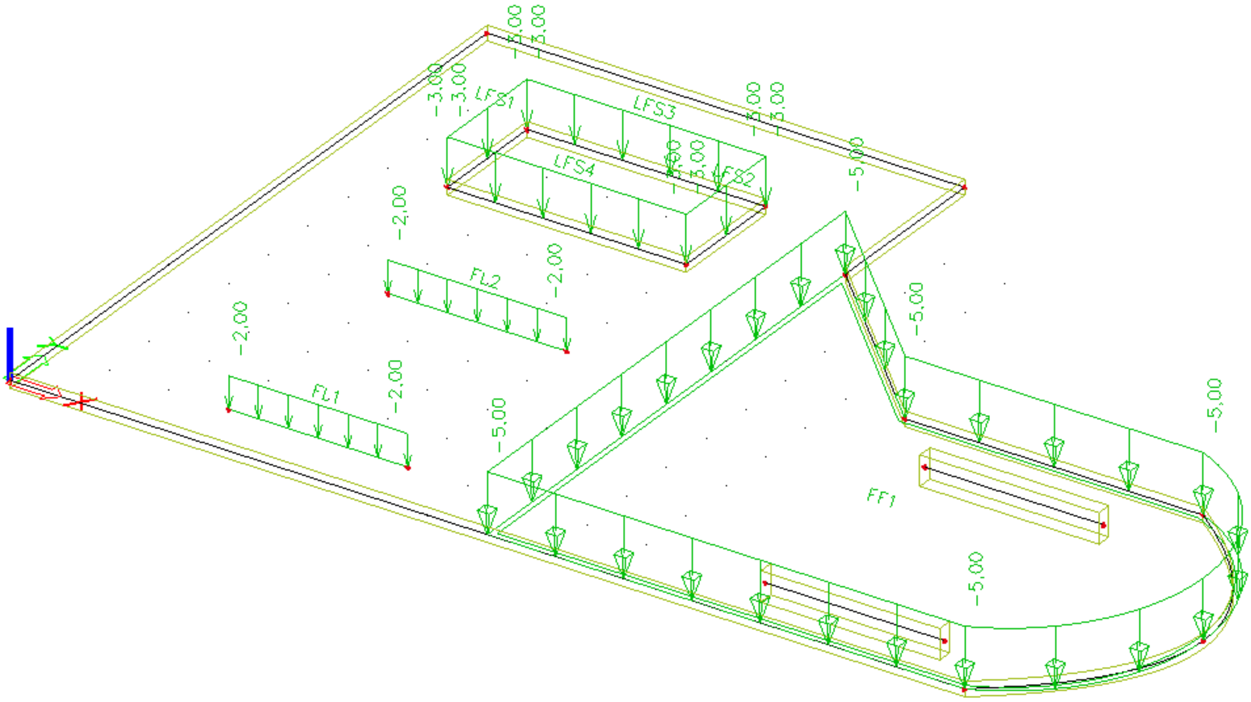
5. The dialogue window will disappear and the coordinates of the new free line load have to be entered.

   1st free line load

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

   2n free line load

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



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

Note:

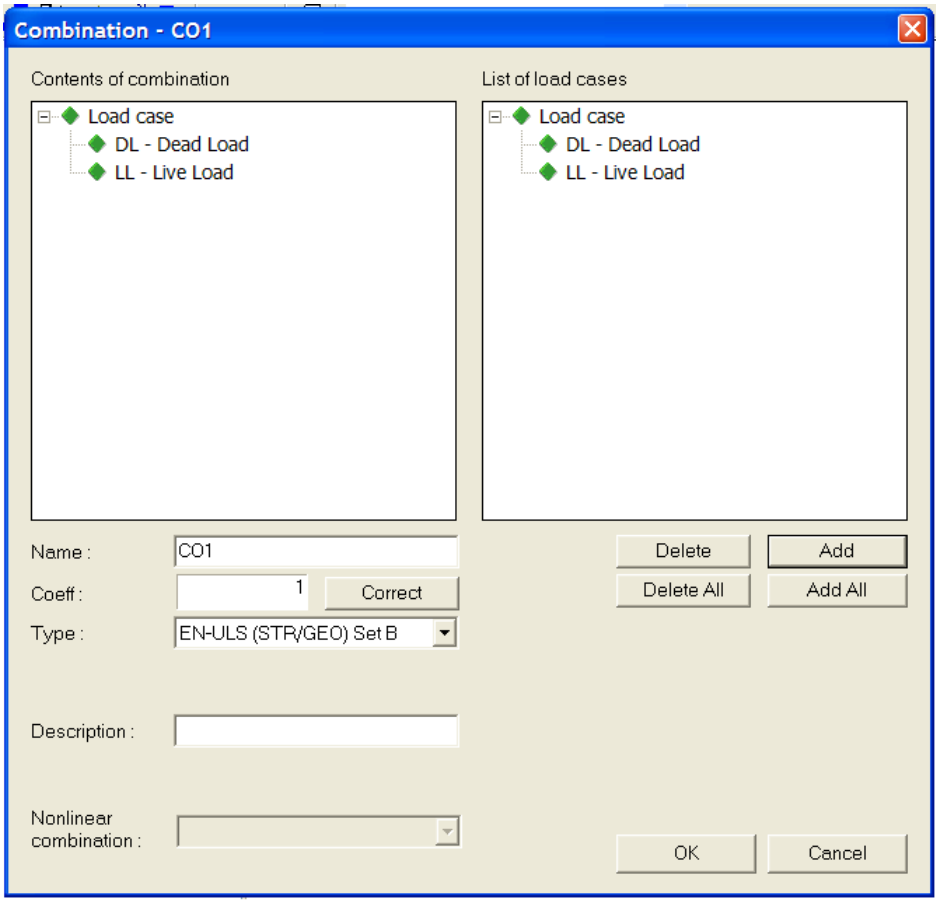
*The **Command line** includes a number of predefined loads:*  *, which enable a fast and simple input of loads.*

Combinations

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

Defining Combinations

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



3. The Type of the combination is changed to **EN – ULS (STR/GEO) Set B**. With this combination type, Scia Engineer will automatically generate combinations in accordance with the complex composition rules of the Eurocode.

4. By means of the button **[Add all]**, all load cases can be added to the combination.

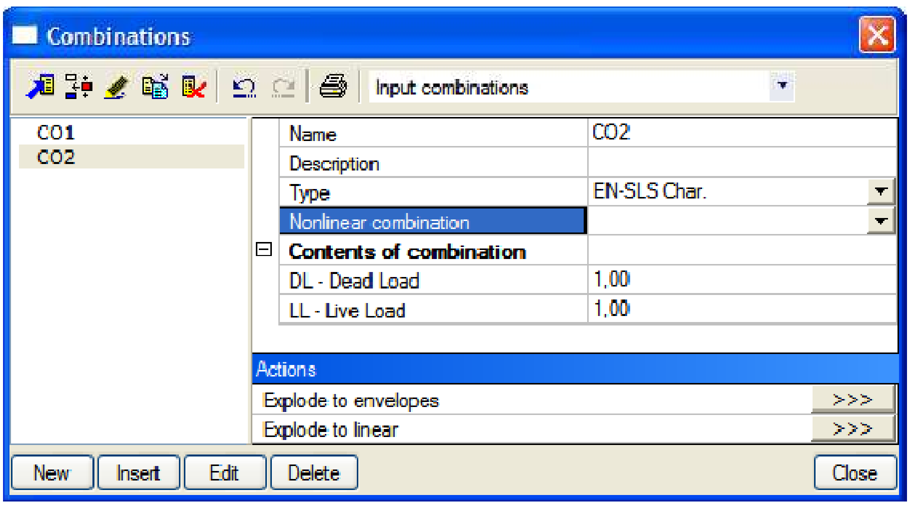
5. Confirm your input with **[OK]**. The **Combination Manager** is opened.

6. Click New or  to create a second combination.

7. Change the **Type** of the combination to **EN-SLS Char**.

8. Confirm your input with **[OK]**.

9. Click **[Close]** to close the **Combination manager**.



Calculation and Mesh generation

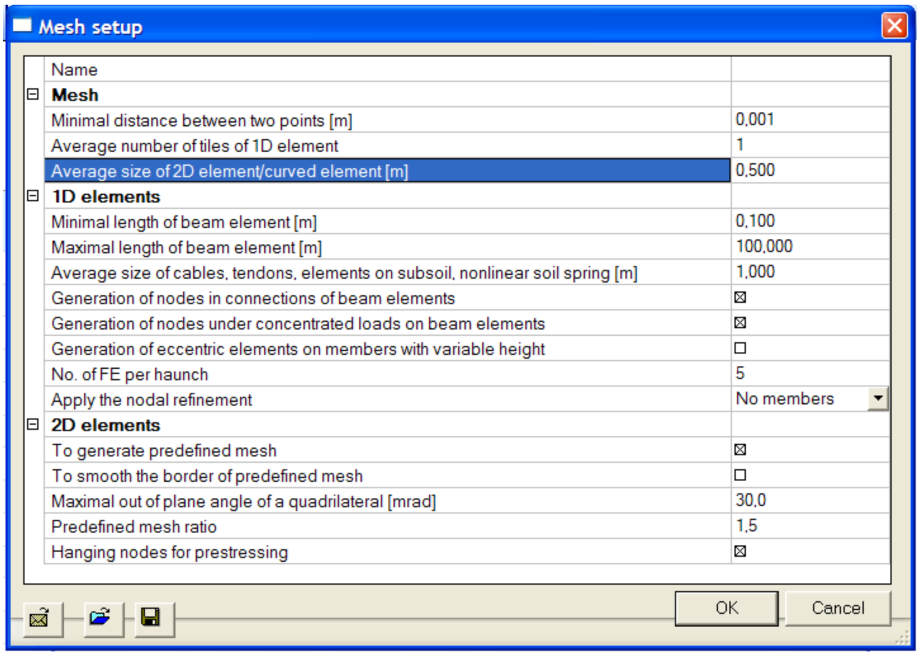
The calculation of the slab will be done according to the finite element method. According to the calculation method a mesh will be generated on the slab and the results will be calculated in the internal nodes of each element. The result in the middle of a finite element is determined as the average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

Mesh generation

Mesh setup

1. In order to see the mesh setup, double click on under

2. The dialogue box **Mesh setup** pop up.



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

Generation of the mesh

4. In order to start the mesh generation you can start

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

Note:

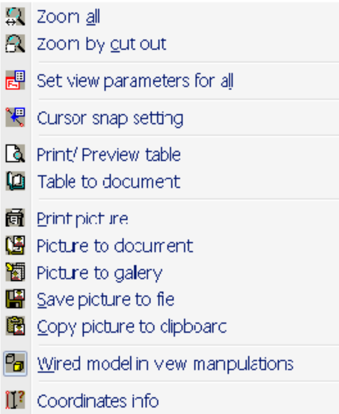
*In the calculation menu you can adjust the local mesh by clicking on **local mesh refinement**. The program gives you three possibilities.*

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

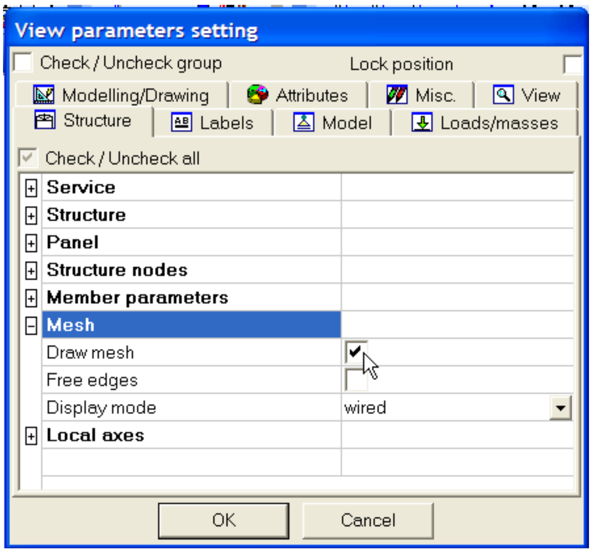
Display of the mesh

1. The mesh can be displayed using the shortcut button located at the bottom of the graphical screen > Structure > net.

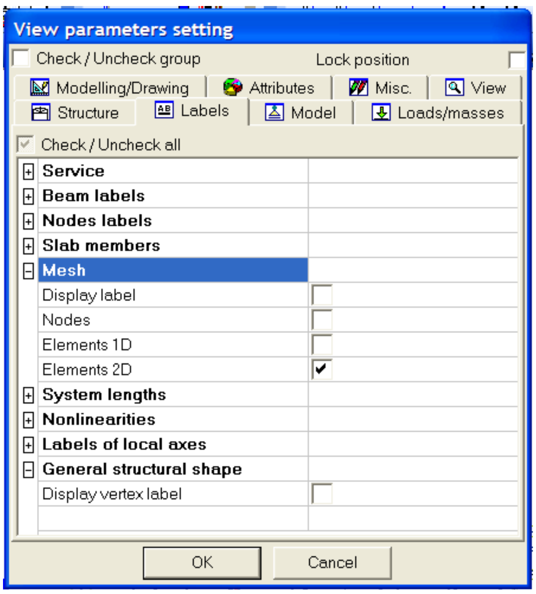
2. The precise settings can be adjusted using the menu item „set view parameters for all“ located in the right mouse button menu.



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



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



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

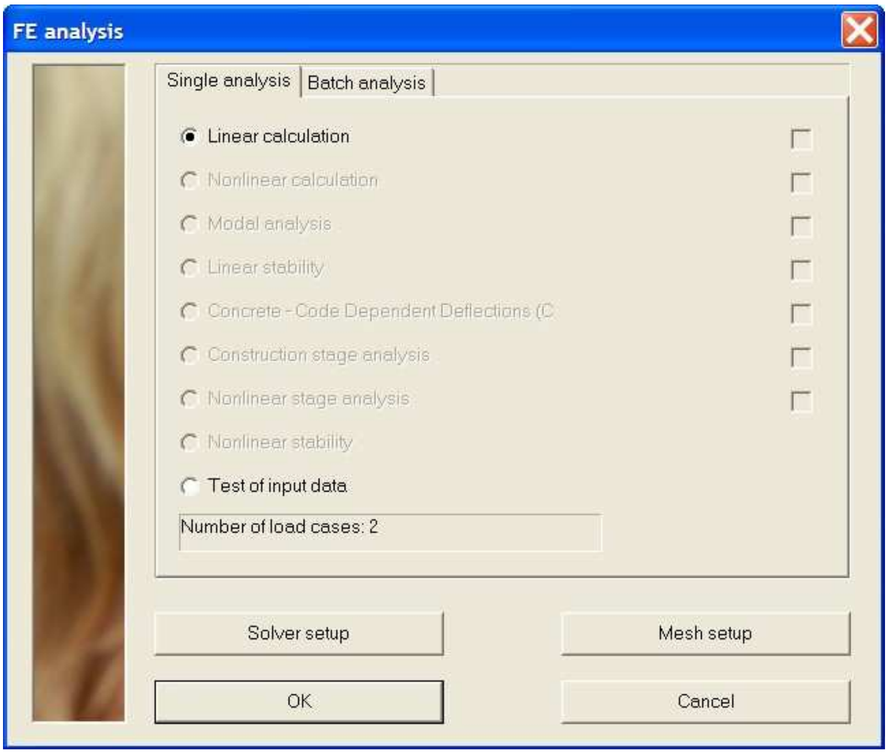
If the mesh is not generated before the start of the calculation, the programme will automatically generate the mesh before starting the calculation

Linear Calculation

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

Solver settings

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



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

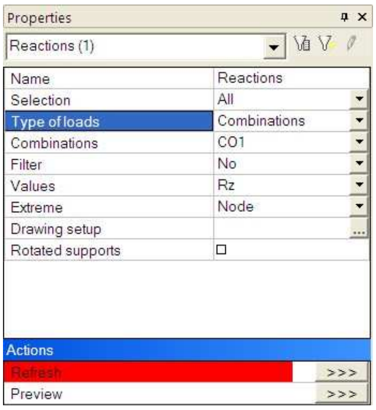
Results

Viewing results

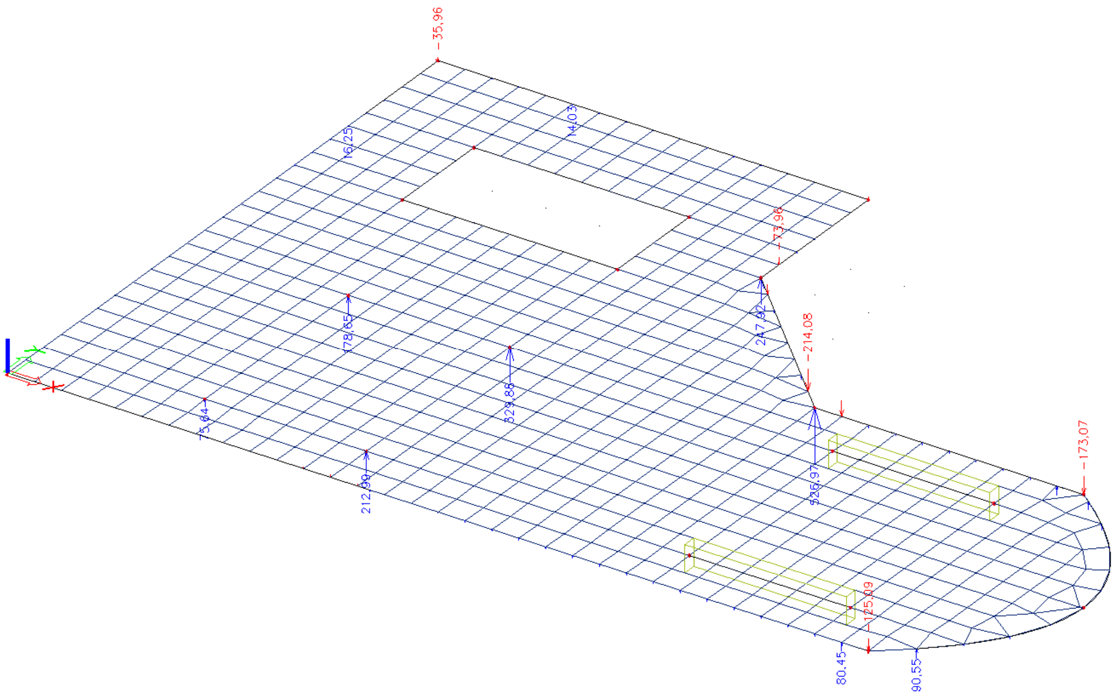
After the calculation is executed, the results can be viewed.

Viewing the Reaction Forces

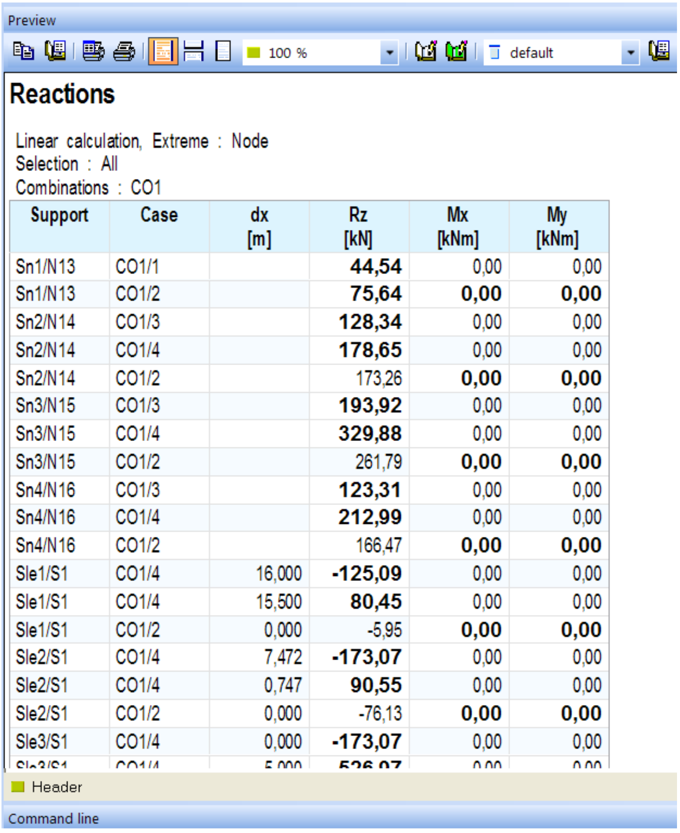
1. Double-click on Results in the **Main window**. The **Results menu** appears.
2. Below **Supports**, click **Reactions**.
3. The options in the **Property Window** are configured in the following way:
   - The Selection field is set to **All**.
   - The Load type is set to **Combinations** and the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



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



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



Reactions

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

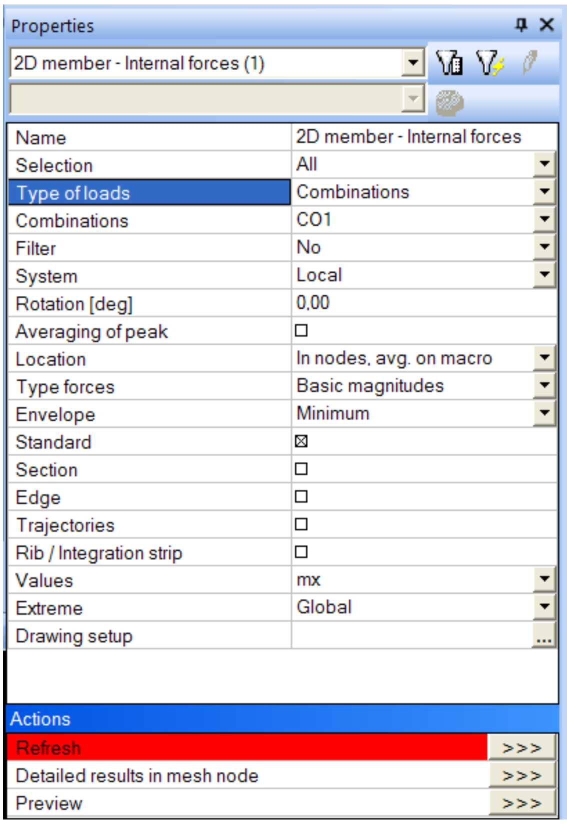
Support	Case	dx [m]	Rz [kN]	Mx [kNm]	My [kNm]
Sn1/N13	CO1/1		**44,54**	0,00	0,00
Sn1/N13	CO1/2		**75,64**	**0,00**	**0,00**
Sn2/N14	CO1/3		**128,34**	0,00	0,00
Sn2/N14	CO1/4		**178,65**	0,00	0,00
Sn2/N14	CO1/2		173,26	**0,00**	**0,00**
Sn3/N15	CO1/3		**193,92**	0,00	0,00
Sn3/N15	CO1/4		**329,88**	0,00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

Note:

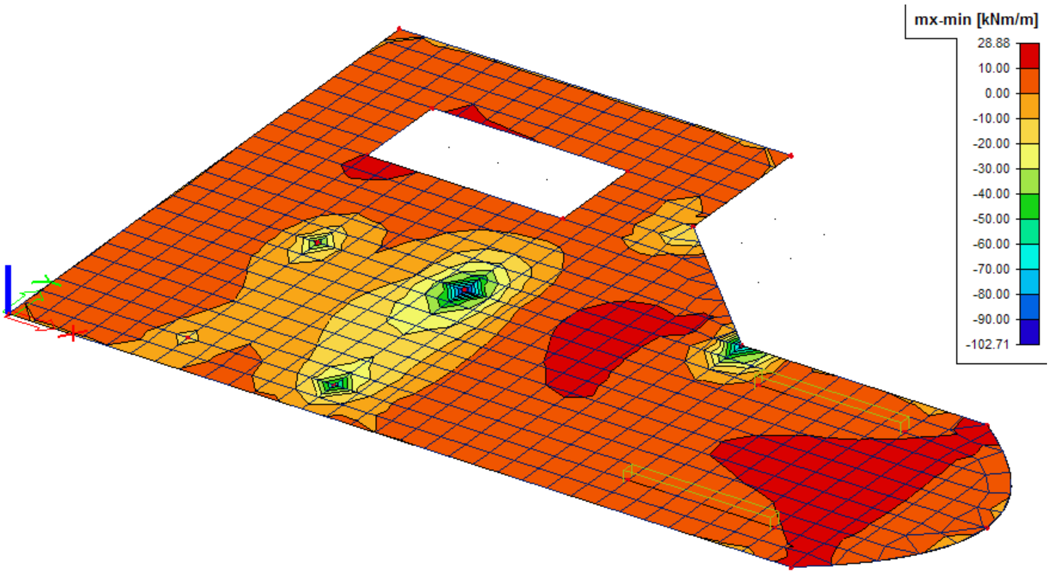
The Preview appears between the Graphical Screen and the Command line. This screen can be maximised to display more data at once.

Viewing internal forces on 2D elements

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



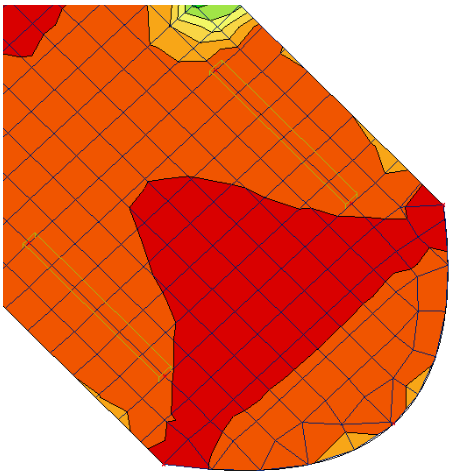
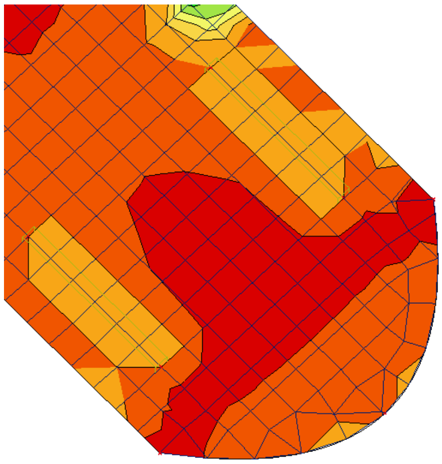
3. Click on the >>> button behind **Refresh** to display the results in the graphical screen for the chosen settings.



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

Results for (individual) ribs

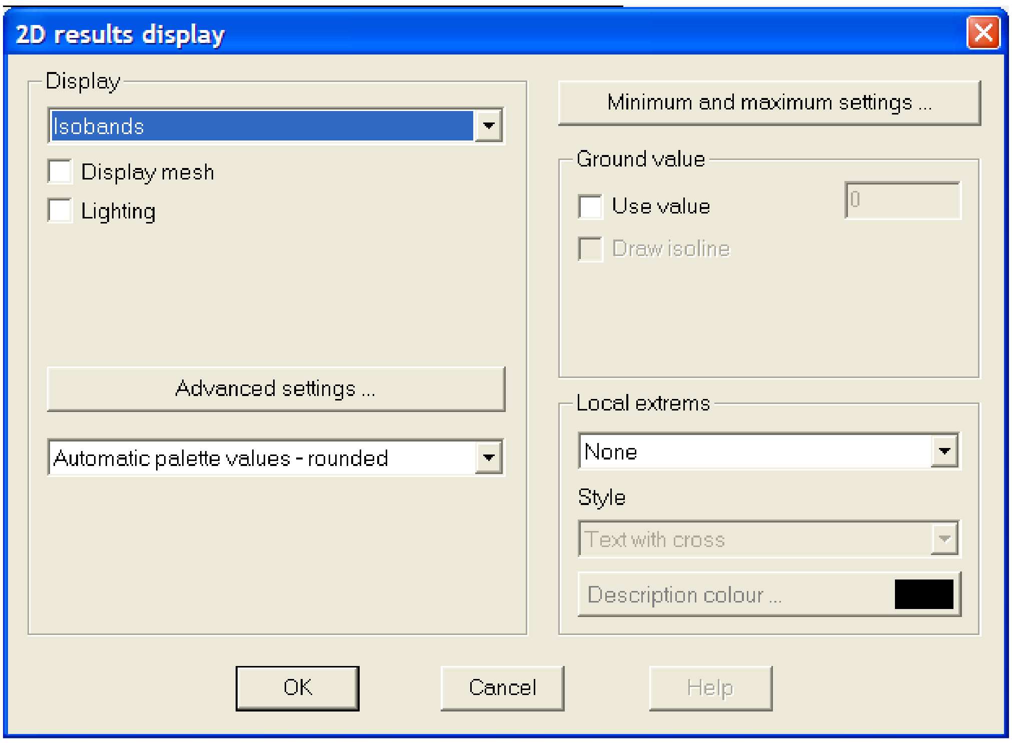
1. By clicking on the tick box '**Rib**' in the Properties window, the results will be adjusted in order to take into account the stiffness of the rib

Option Rib not activated:	Option Rib activated:
	

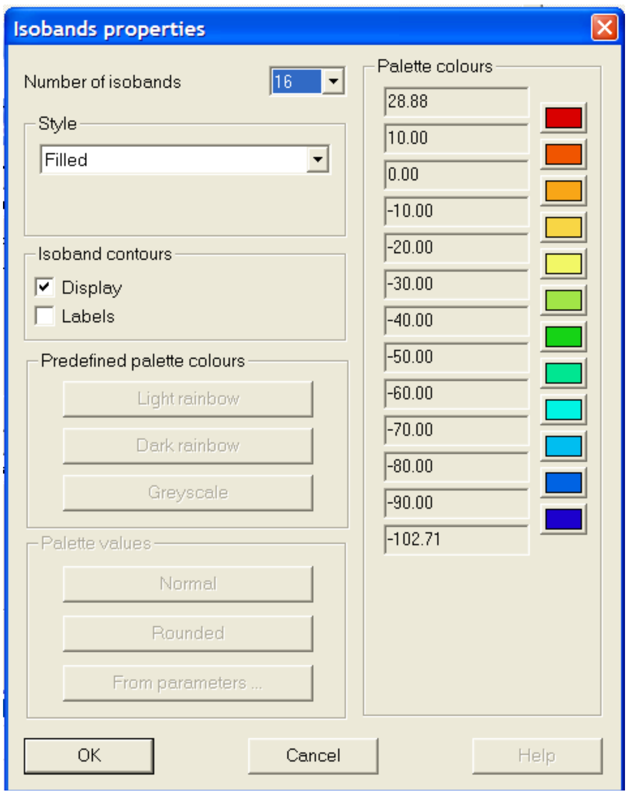
2. Note the difference for the two ribs that are modelled. It is clear that the forces in the plate are reduced, because the joint stiffness of the slab and the ribs is now considered.

Configuring the Graphical Screen

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



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



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

5. Click in the **Property Window**, on the button >>> behind **Refresh** in order to display the results in the graphical screen in accordance with the set options.

6. Click **[Close]** to leave the **Results Menu**.

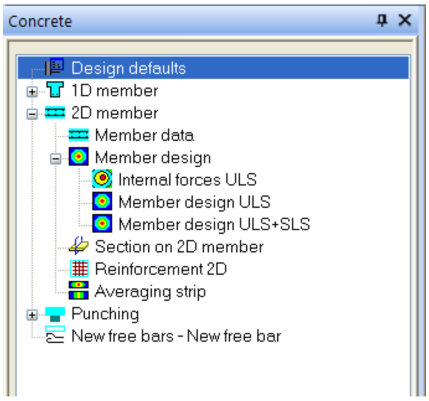
Note:

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

Reinforcement design

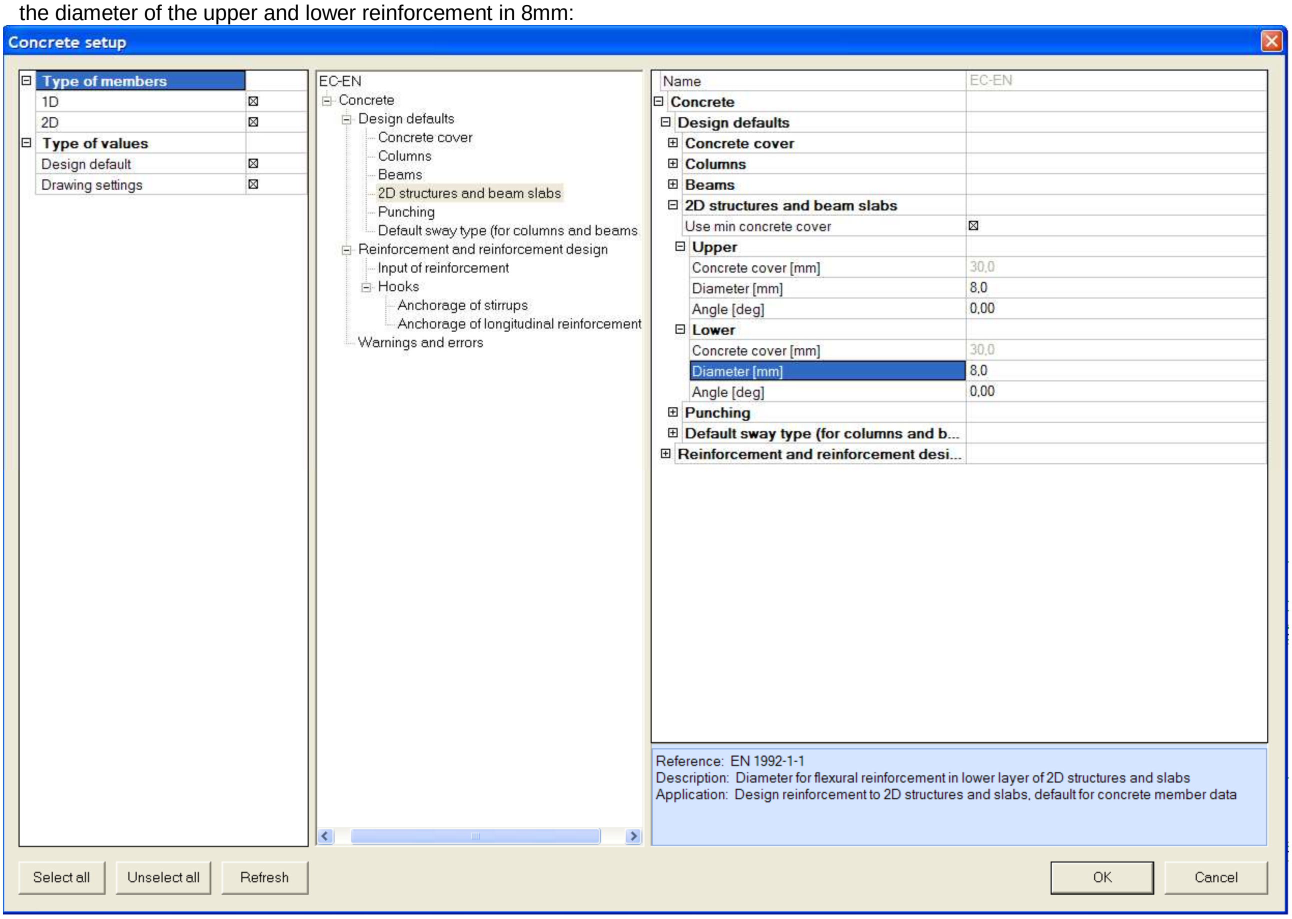
The reinforcement design can be viewed in the concrete menu.

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



Changing the diameter of the bars

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

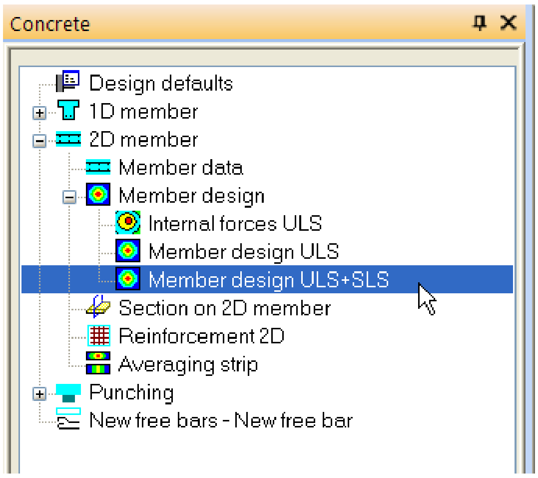


3. And click "OK" to close the concrete setup window.

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

Required areas

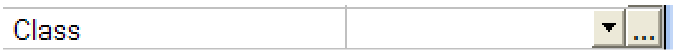
1. Select “Member design ULS+SLS” in the Concrete menu:



2. The option in the **Property Window** are configured in the following way:

   - The **Selection** field is set to **All**.
   - The **Load type** is set to **Class**

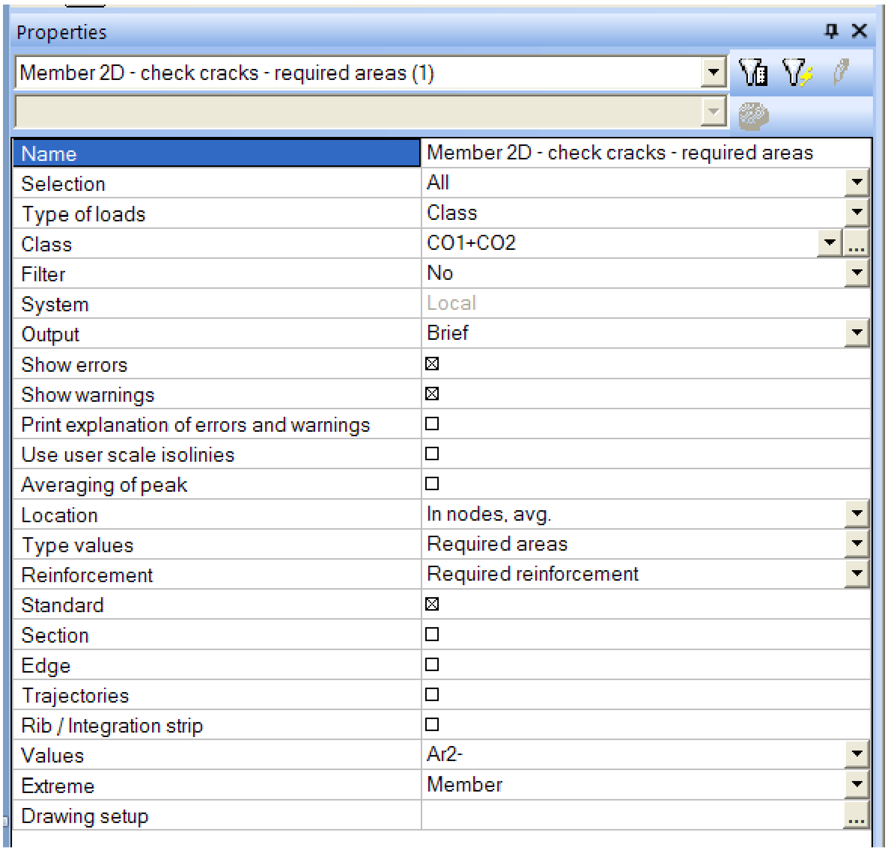
3. And click now on the three points behind **Class**:



   - Double-click on **CO1** and **CO2**
   - Change the name in **CO1+CO2**
   - Click two times on **OK**

4. The option in the **Property Window** are configured in the following way:

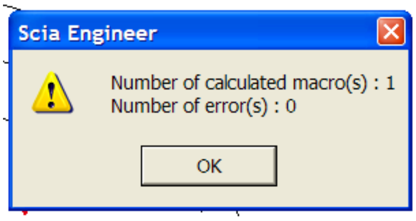
   - The **Selection** field is set to **All**.
   - The **Load type** is set to **Class**
   - The **Class** is changed to **CO1+CO2**
   - The **Filter** is set to **No**.
   - The options **Show errors** and **Show warnings** are activated.
   - The **Location** is set to **In nodes, avg.**
   - The **Type values** are wanted for **Required areas**.
   - The **Reinforcement** is put on **Required reinforcement**.
   - The option **Standard** is activated.
   - The option **Values** is changed to **Ar2-**
   - The **Extreme** field is changed to **Member**.



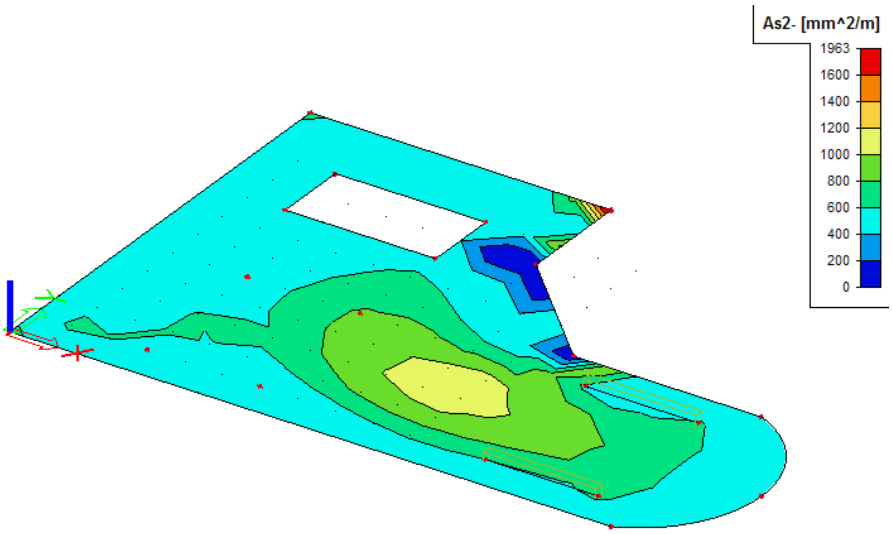
5. With the option **values** the reinforcement can be viewed in 2 directions, and at the upper (+) and lower (-) side of the plate. In this example the reinforcement is calculated in the direction 2 (y-direction) at the lower side (-) of the plate: Ar2-.

6. Click on **Refresh** to see the results

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

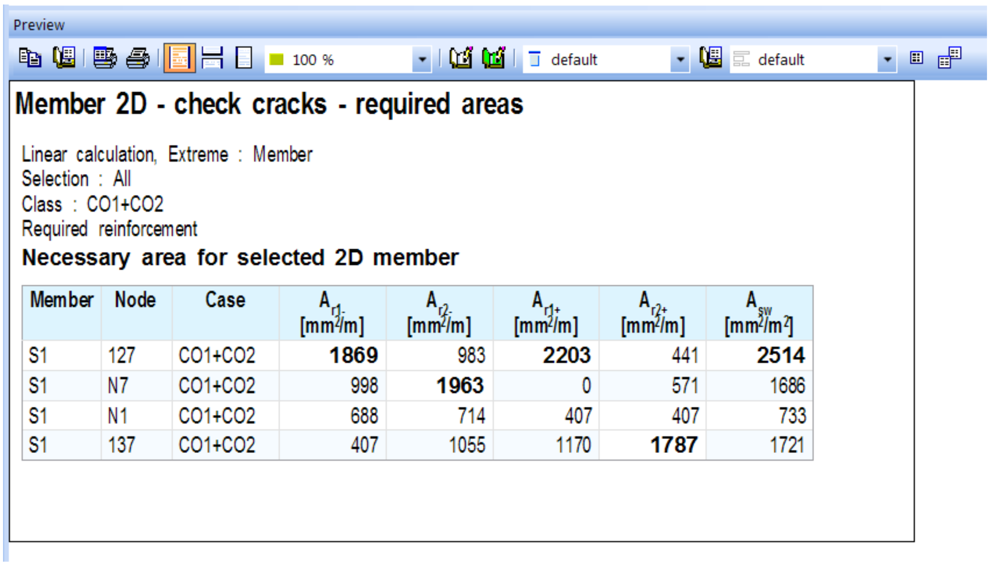


8. Click **OK** to see the results:



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

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



Member 2D - check cracks - required areas

Linear calculation, Extreme : Member
Selection : All
Class : CO1+CO2
Required reinforcement

Necessary area for selected 2D member

Member	Node	Case	A_{r1-} [mm^2/m]	A_{r2-} [mm^2/m]	A_{r1+} [mm^2/m]	A_{r2+} [mm^2/m]	A_{sw} [mm^2/m^2]
S1	127	CO1+CO2	**1869**	983	**2203**	441	**2514**
S1	N7	CO1+CO2	998	**1963**	0	571	1686
S1	N1	CO1+CO2	688	714	407	407	733
S1	137	CO1+CO2	407	1055	1170	**1787**	1721

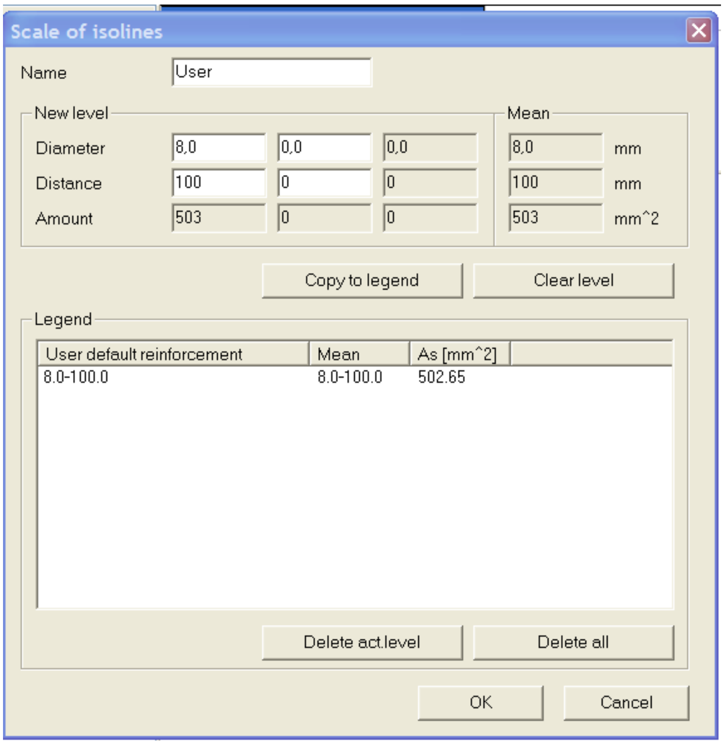
10. To use user scale isolines, activate this option: Use user scale isolinies ☒

11. The “User scale isolines”-window will open
Click on **New** to define a new User scale isoline

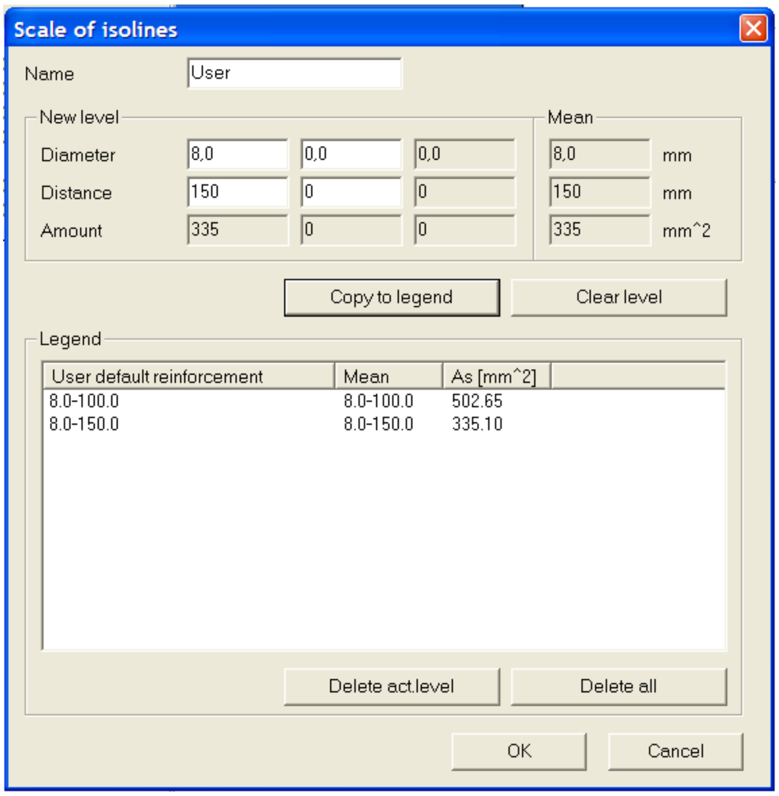
12. The following scale options are set

- The **Name** is changed to **User**
- The **Diameter** is set to **8.0** mm.
- The **Distance** is changed in **100** mm.

13. Click on **Copy to legend** to save this setting in the legend.



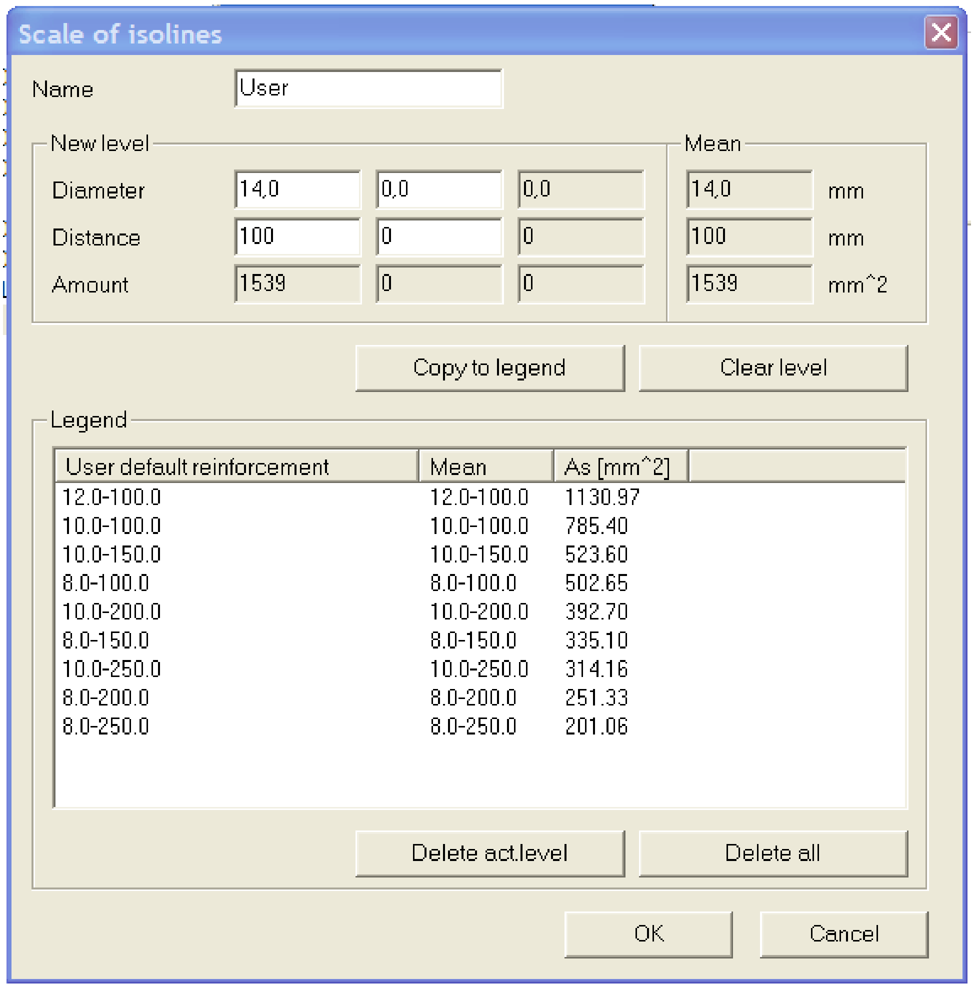
14. Change the **Distance** in **150** mm. And click on **Copy to legend** again.



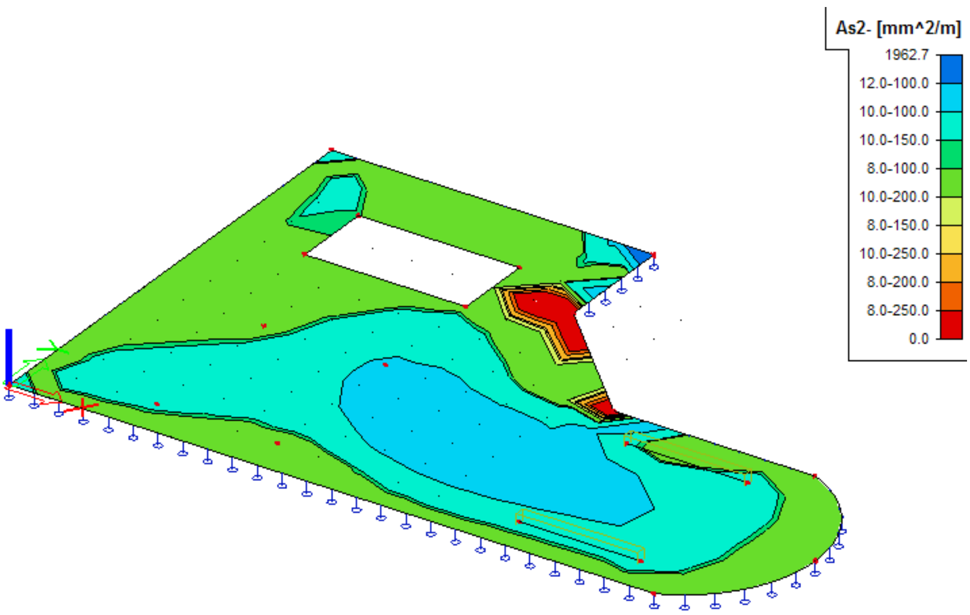
15. The legend is shown below. 8.0 – 100.0 means a diameter of 8mm with a distance of 100mm. Input the following diameter option and click always on **Copy to legend**:

- Diameter = 8.0mm; Distance = 200mm
- Diameter = 8.0mm; Distance = 250mm
- Diameter = 10.0mm; Distance = 100mm
- Diameter = 10.0mm; Distance = 150mm
- Diameter = 10.0mm; Distance = 200mm

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

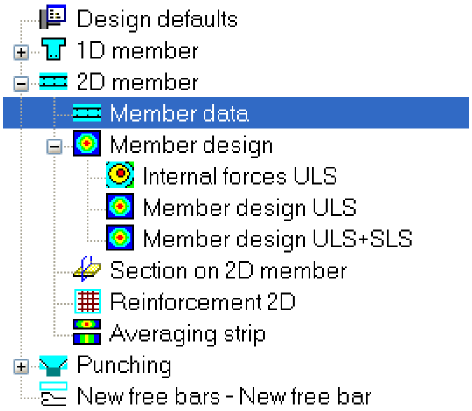


16. Click **OK** to save this legend
17. Click **OK** to use this User scale
18. Click on **Refresh** to see the required reinforcement with the user scale



Input user reinforcement

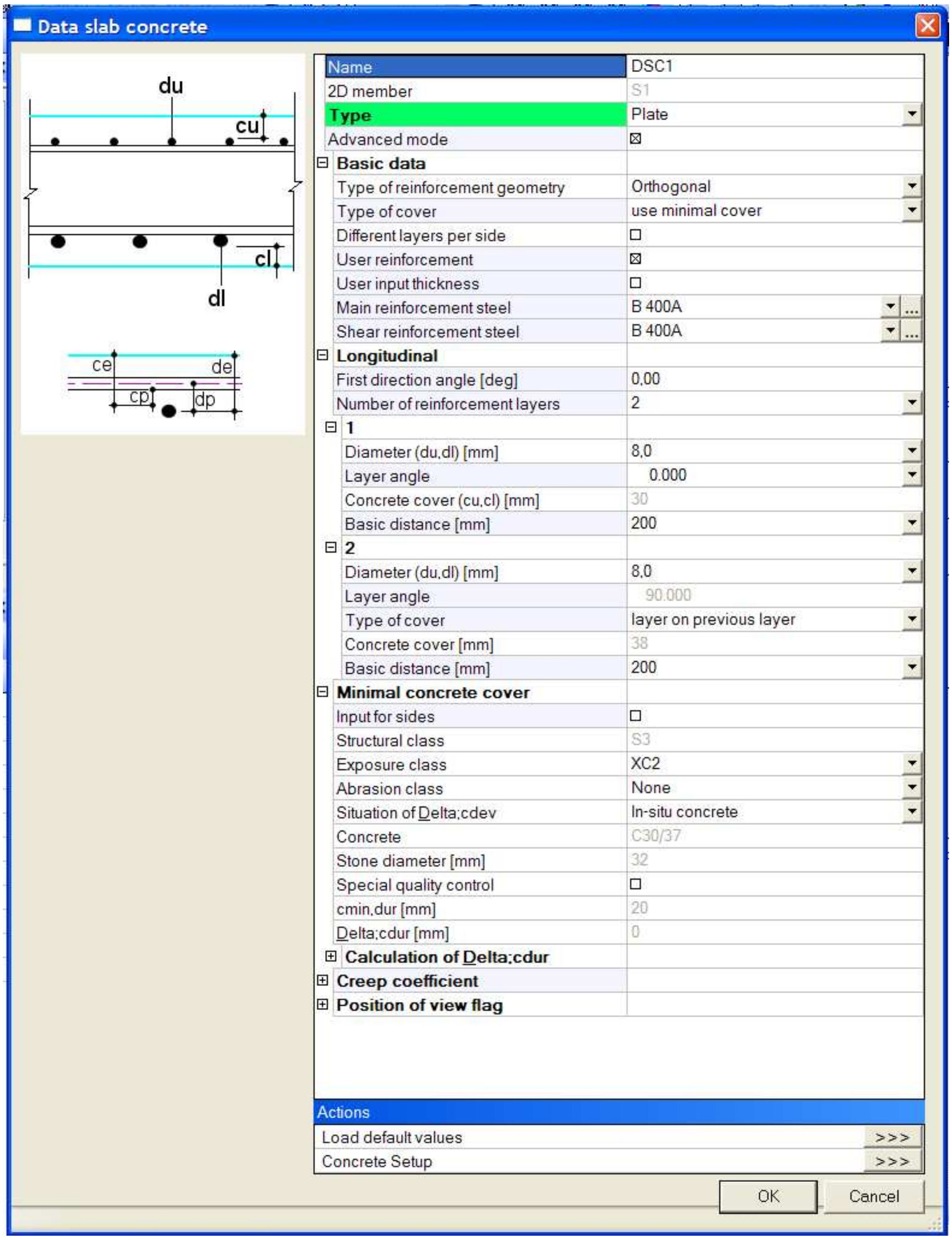
1. Select in the Concrete menu:



2. Select the plate

3. The option in the **Data slab concrete** Window are configured in the following way:

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



4. And click now on **OK**

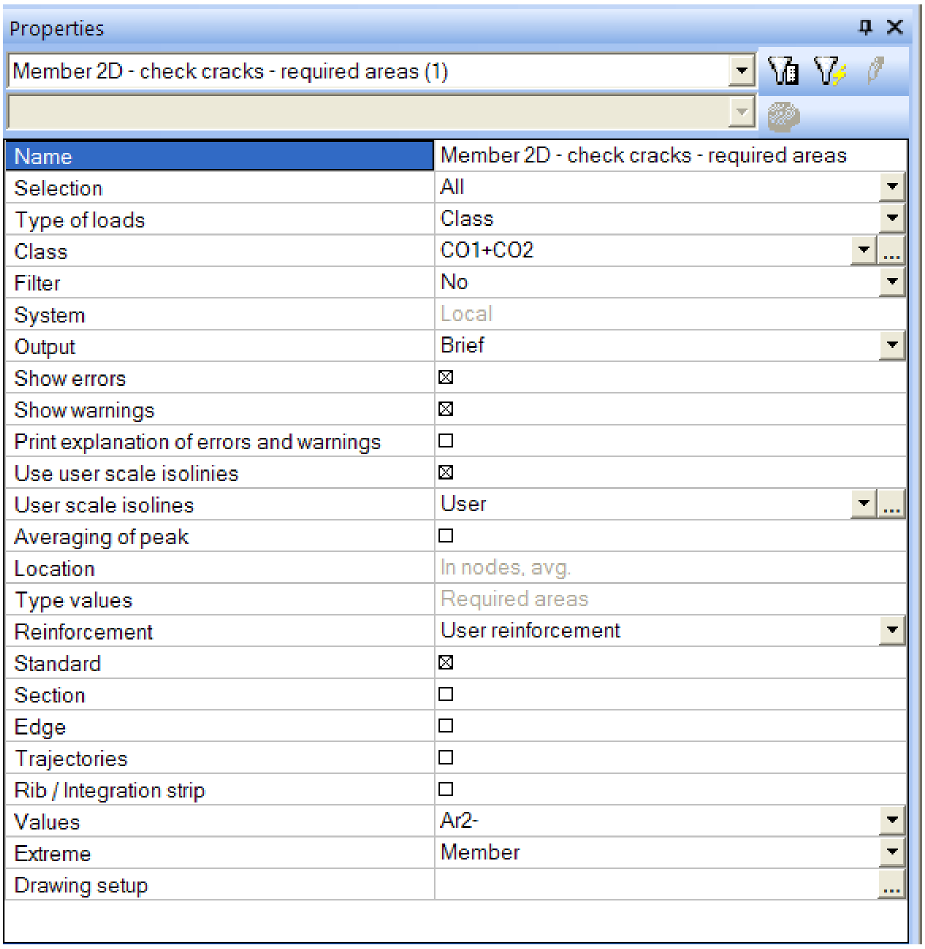
5. Click on **Esc** to end this function.

6. Now select again the option **Member design ULS+SLS**.

7. The option in the **Property Window** are configured in the following way:

    - The **Selection** field is set to **All**.
    - The **Load type** is set to **Class**
    - The **Class** is changed to **CO1+CO2**
    - The **Filter** is set to **No**.
    - The options **Show errors** and **Show warnings** are activated.
    - The **Location** is set to **In nodes, avg.**
    - The **Type values** are wanted for **Required areas**.
    - The **Reinforcement** is put on **User reinforcement**.
    - The option **Standard** is activated.

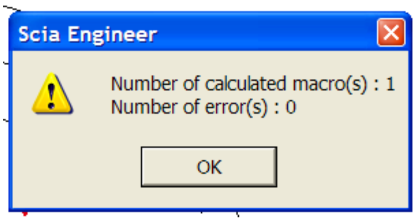
- The option **Values** is changed to **Ar2-**
- The **Extreme** field is changed to **Member**.



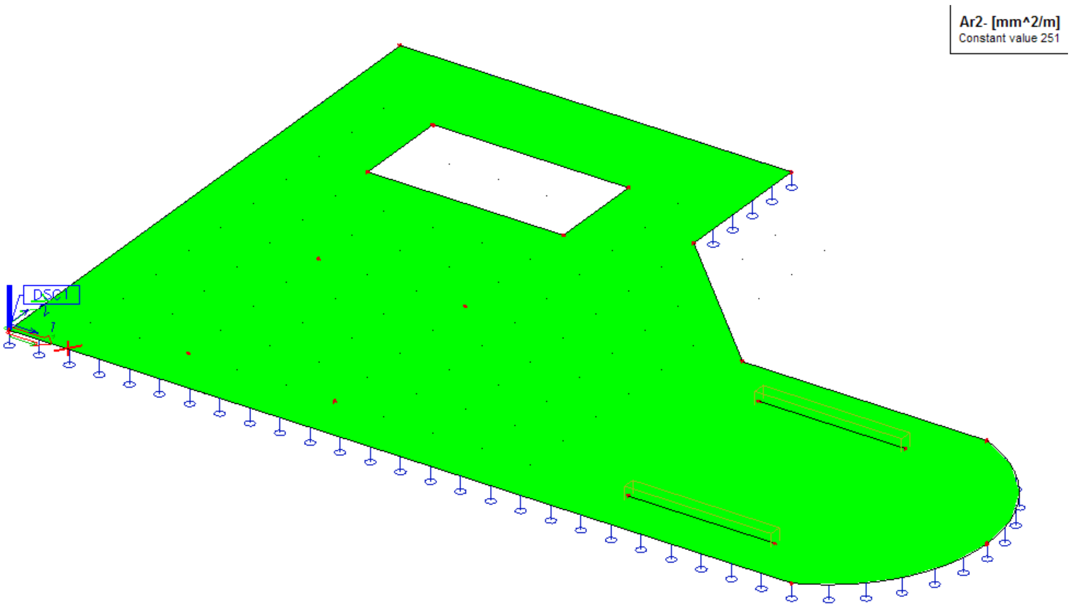
8. With the option **User reinforcement**, the reinforcement inputted by the user will be shown.

   Click on **Refresh** to see the results

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



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



This reinforcement has a constant value of 251 mm²/m.

Document

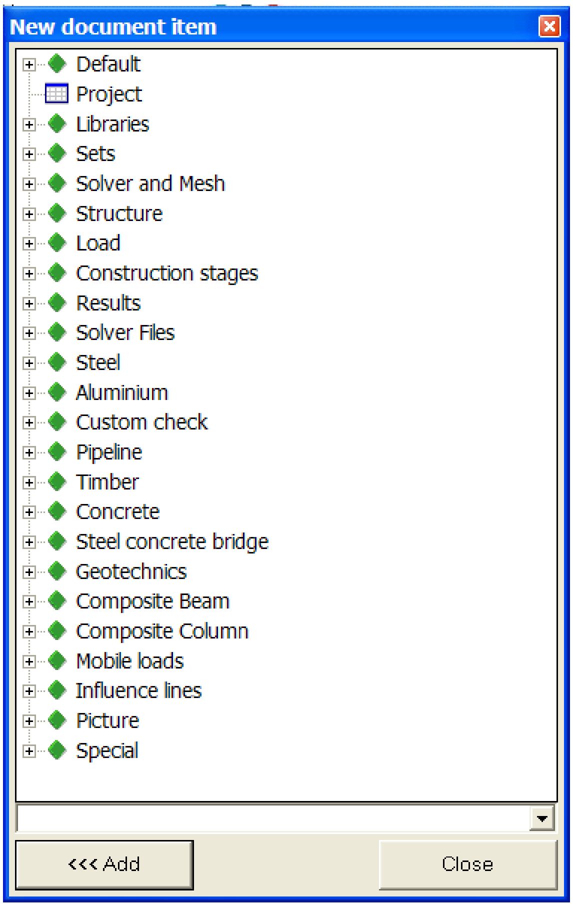
In this final part of the tutorial, we will explain how a calculation note can be drawn up.

Formatting the Document

1. Double-click Document in the **Main Window** or click in the button bar. The **Document** appears.

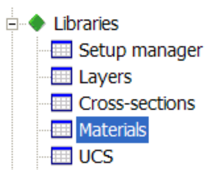
   The Project data are automatically displayed in the header of the document.

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



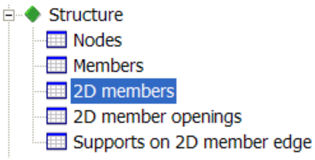
3. By means of this window, several data can be added to the document.

   - Open the **Libraries** group and click on **Materials**. Click **[<<< Add]** to add this item to the document.

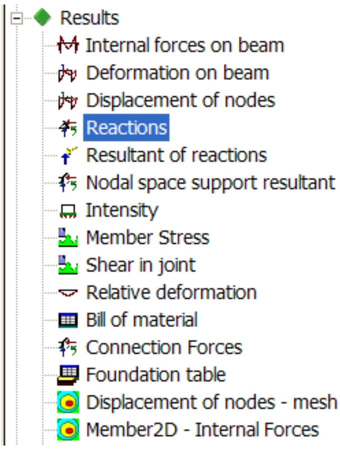


   - Click **Cross-Sections**. Click **[<<< Add]** to add this item to the document.

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



- Open the **Results** group and click **Reactions**. Click **[<<< Add]** to add this item to the document.



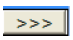
4. Click **[Close]** to close the **New document item** window and to return to the document.

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

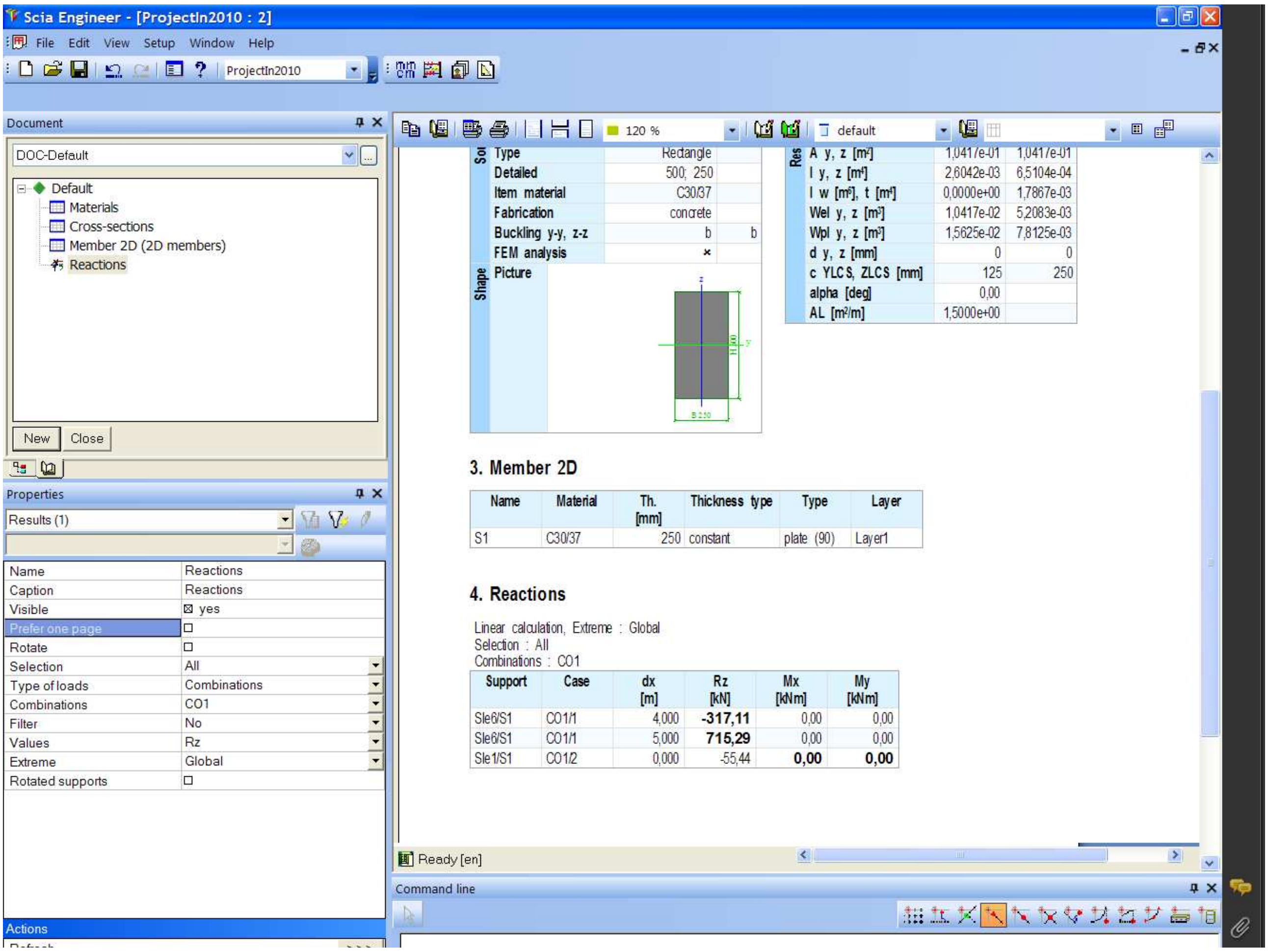
Displaying results in the document

1. In the **Document menu**, click **Reactions**. In the **Properties window** the properties of this table are displayed. The parameters for displaying the results in the **Document** are configured in the same way as the parameters for viewing the results in the **Results Menu**.

   - The option **Visible** is activated.
   - The **Selection** field is set to **All**.
   - The **Load type** is set to **Combinations** and the Combination to **CO1**.
   - The **Filter** is set to **No**.
   - The **Values** are wanted for **Rz**.
   - The **Extreme** field is changed to **Global**.

2. Click the 

button behind **Refresh** to display the table in accordance with the set options.

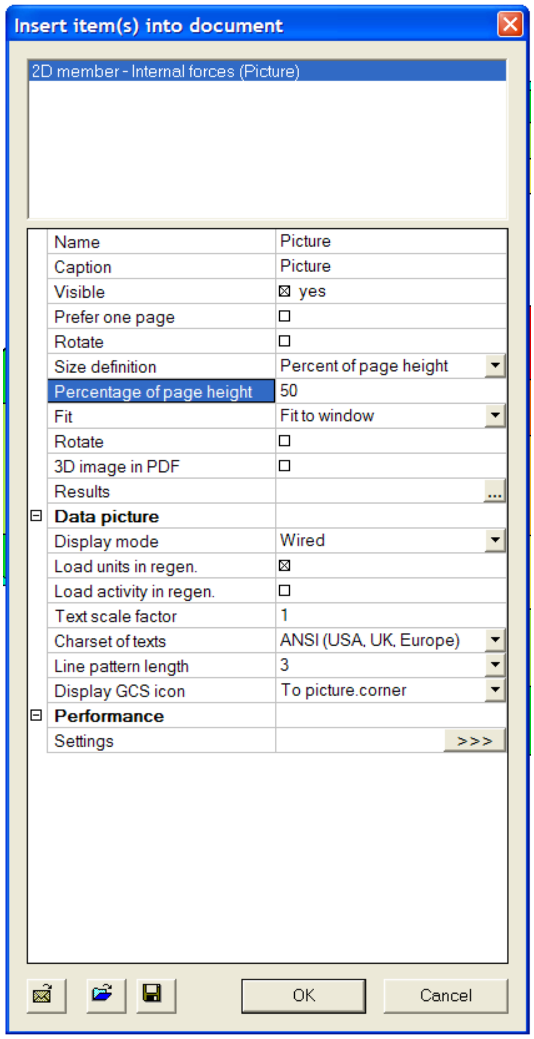


3. Click the **[Close]** button below the **Document Menu** to close the document and to return to the structure.

Adding an image to the document

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

   The window **Insert item(s) into document** appears.



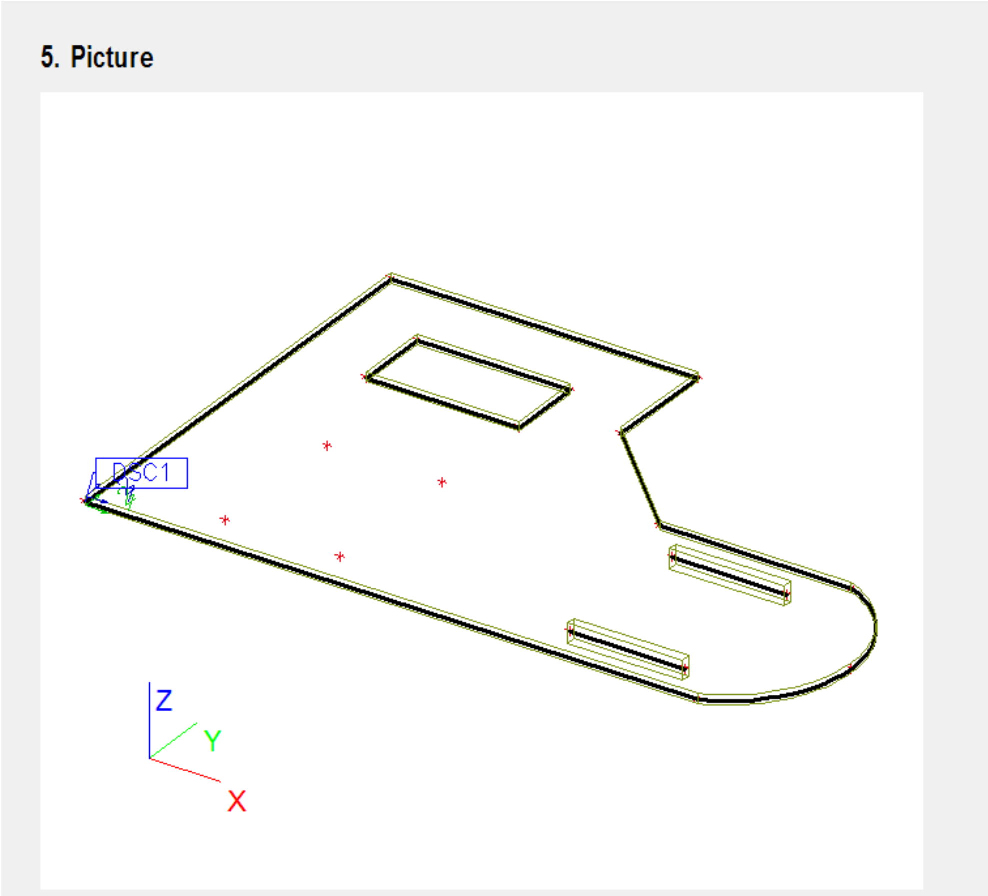
3. The **Caption** field is changed to **Picture** as the title of this picture.

4. The **Percentage of page height** field is changed to **50** so that the image covers 50% of a page, i.e. half a page.

5. Confirm your input with **[OK]** so that the image is sent to the document.

6. Click  in the button bar to open the **Document**.

7. In the **Document Menu**, click **Picture**. The image is displayed in the Preview of the **Document**.



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

Epilogue

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

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

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