TUTORIAL

CONCRETE FRAME

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.

Table of contents

General Information

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

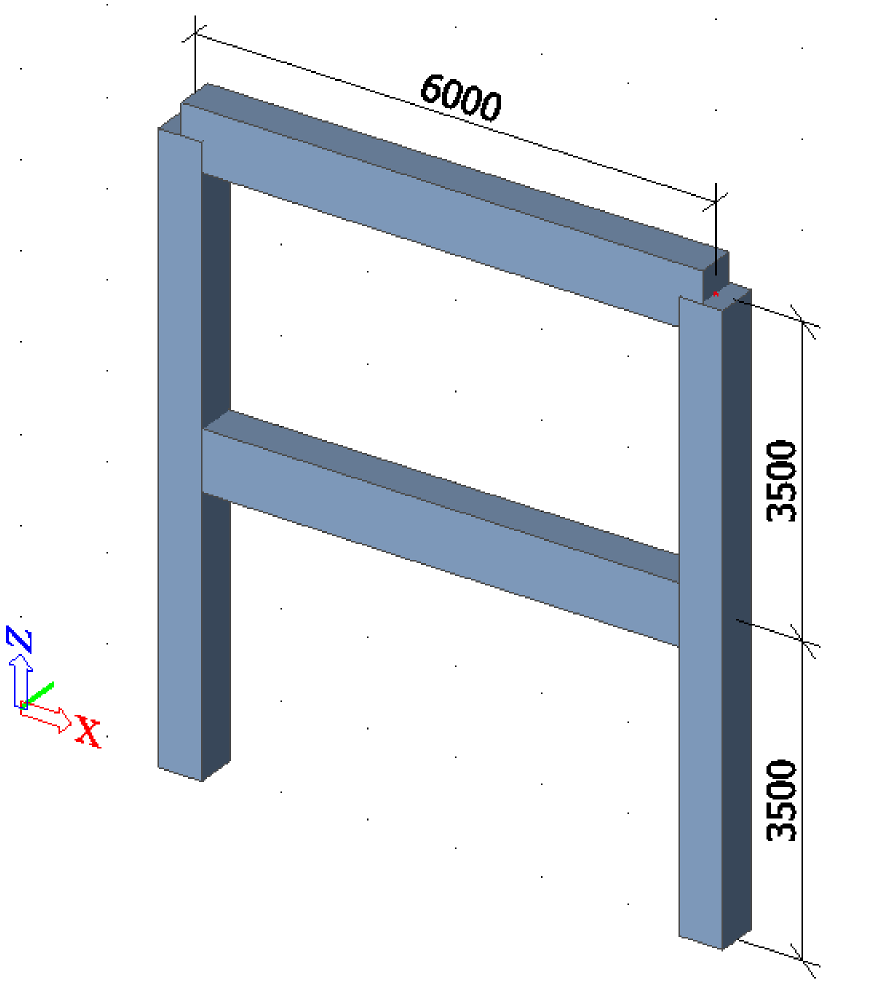
Websites

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 2D concrete frame.
Before you start, you need to be acquainted with the use of your operating system, e.g. working with dialogs (windows), menu bars, toolbars, status lines, the mouse, etc.
First, we will explain how to create a new project and how to model the structure. After the input of the geometry and the loads, the structure will be calculated and you will be able to view the results. Next, the input of the buckling parameters is discussed and then we will proceed with the concrete design and checks. Finally, we will teach you how to make a nice documentation.

The figure below shows the analysis model of the structure that will be modelled:



Getting started

Starting a project

Starting the program

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

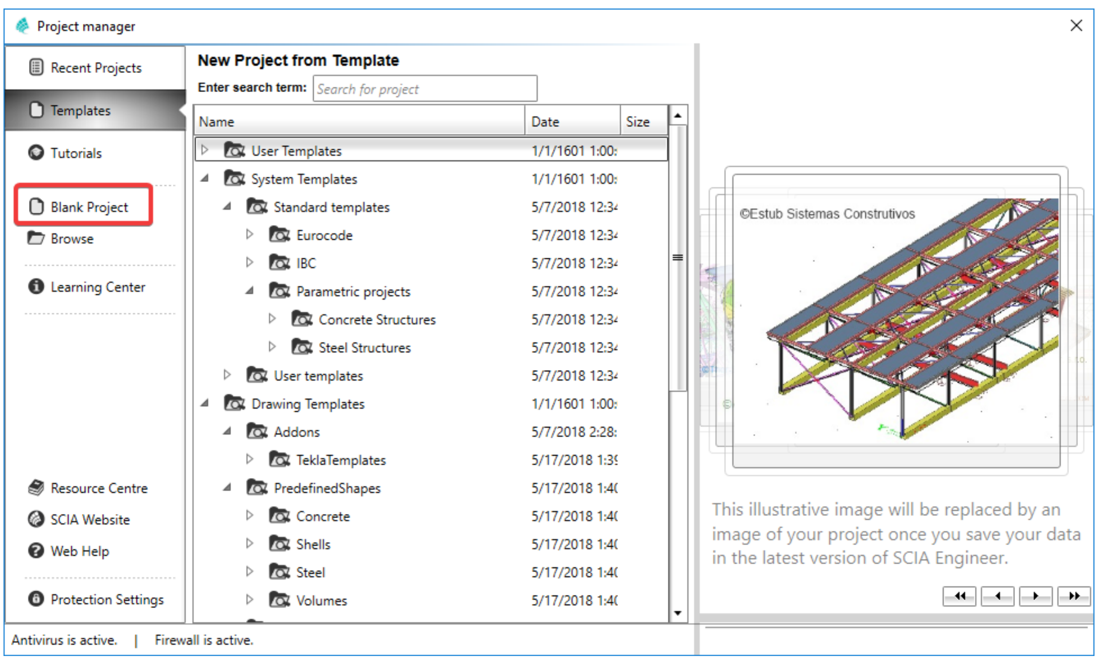
1. Double-click on the SCIA Engineer shortcut in the Windows Desktop, or
2. If the shortcut is not installed, click **[Start]** and choose **All apps > SCIA Engineer 18.0 > SCIA Engineer 18.0**.

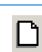
If the program does not find any protection, you will see a dialogue indicating that no protection was found. You are offered to run Protection setup and select appropriate protection type (e.g. try-out), or run the program in Viewer mode.

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

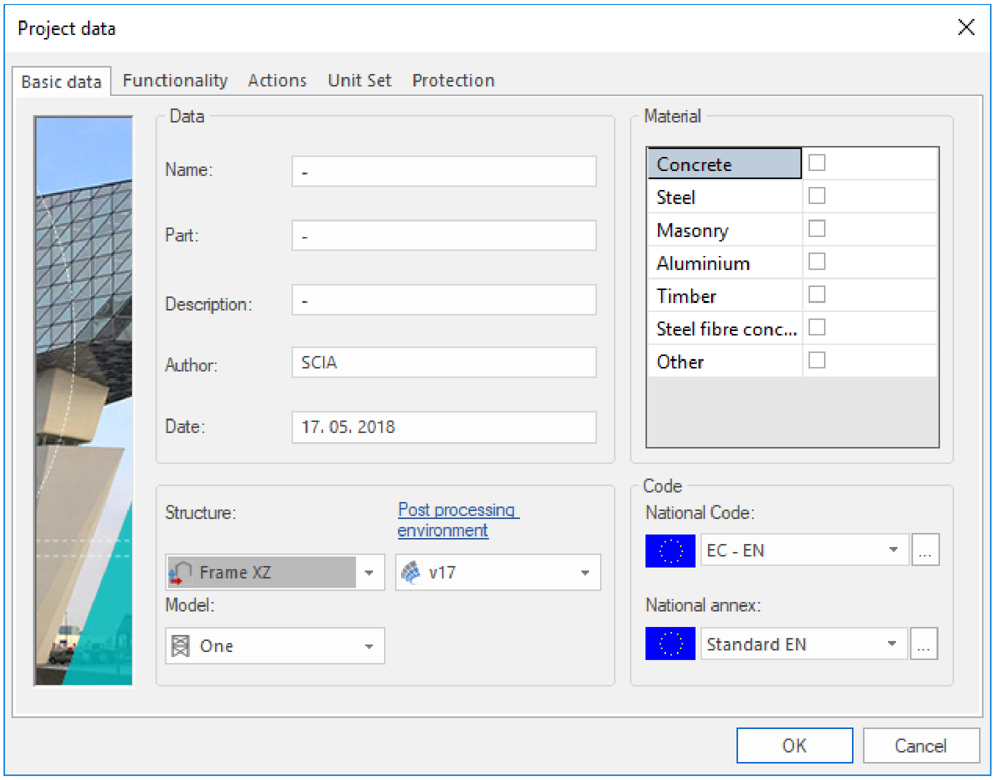
Starting a new project

1. When the **Project manager** dialogue appears, click **Blank project**.



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.



3. In the **Basic data** group, enter your preferred data. These data can be mentioned in the output, e.g. in the report header and on the drawings.
4. Choose the **Structure: Frame XZ** (to limit input possibilities to 1D members in 2D plane only) and **Model: One**.
5. In the **Material** group, tick **Concrete** checkbox.

   Below the item **Concrete** you can choose your preferred concrete grade (select C25/30 here) and reinforcement material (select B 500B here).

   Material is the only required setting to proceed
6. In the Code frame select National Code EC-EN and National annex: Standard EN
7. Confirm your input with **[OK]** button.

*Note: On the **Functionality** tab, you choose the options you need. The non-selected functionalities will be filtered from the menus, thus simplifying the program. We don't need any additional functionality for this tutorial example.*

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**.

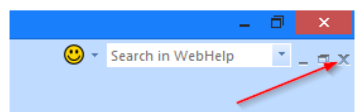
If a project has not yet been saved, the dialog box **Save as** appears. Click on the arrow in the list **Save** to choose the drive you want to save your project in. Select the file in which you want to put the project and click on **[Open**]. Select the subfolders. Enter the file name in **File name**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

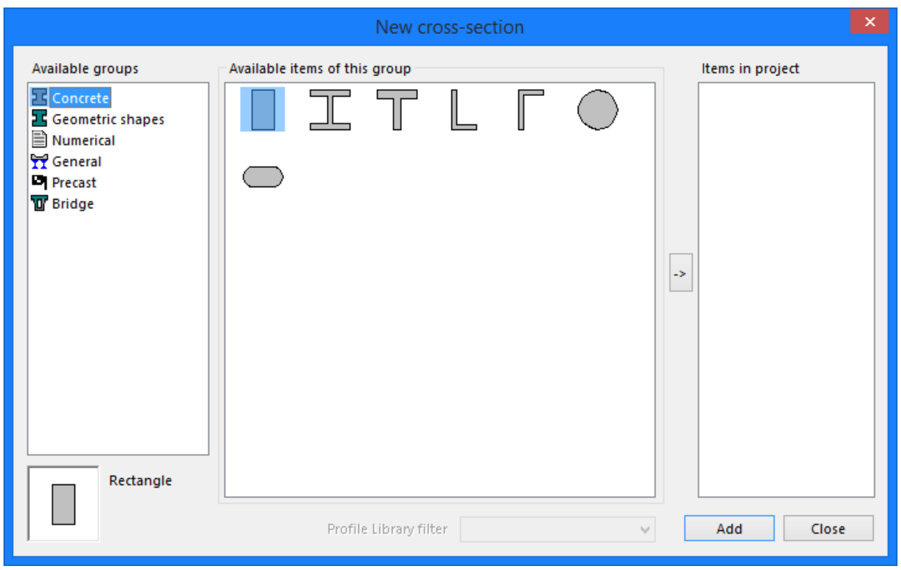
Profiles

When entering one or more 1D structure elements, a profile type is immediately assigned to each member. By default, the active profile type is represented. You can open the profile library to activate another profile type. If you want to add a structure part before a profile type has been defined, the profile library will automatically be opened.

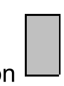
Adding a profile

1. Click on the **Cross-Sections**  icon in the toolbar.

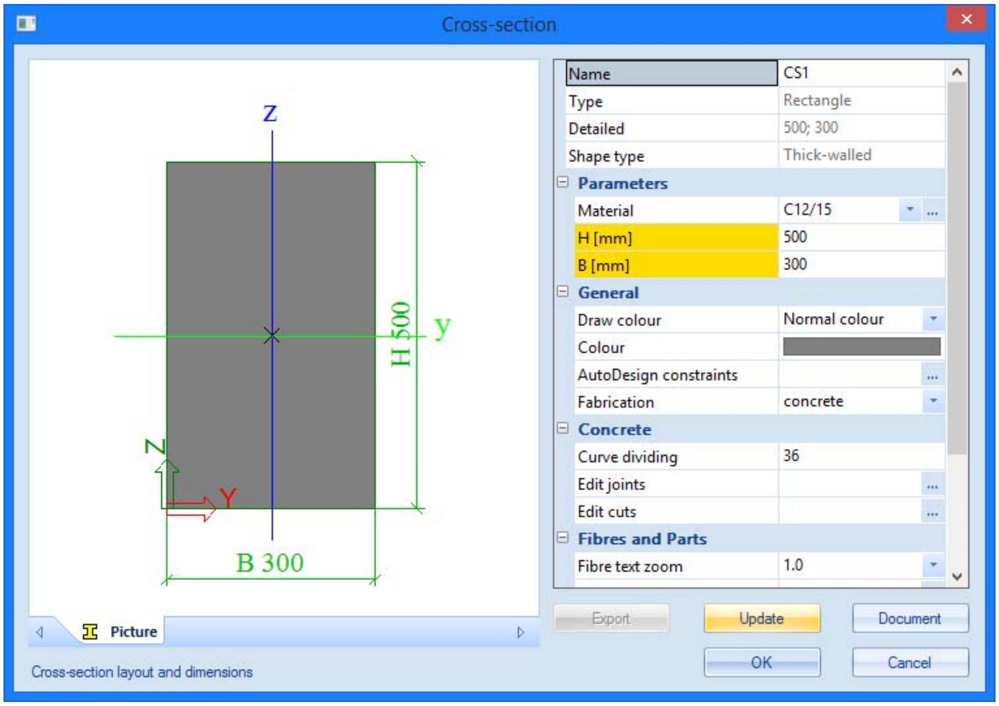
   The cross-sections manager is opened. If no profiles have been entered in the project, the **New cross-section** window will be automatically opened.



2. Click **Concrete** in the left frame **Available groups**.

3. In the **Available items of this group**, you can choose a rectangular section  .

4. Click **[Add]** or  to add the profile to the project. The **Cross-section** window appears.



5. In this window, you can change the properties of the rectangular section. Enter **500 mm** for height **H** and **500 mm** for width **B**.

6. Click **[OK]** to confirm, the profile is added to the **Items in project** frame.

7. A second rectangular section with height **H = 700 mm** and width **B = 450 mm** is added in a similar way.

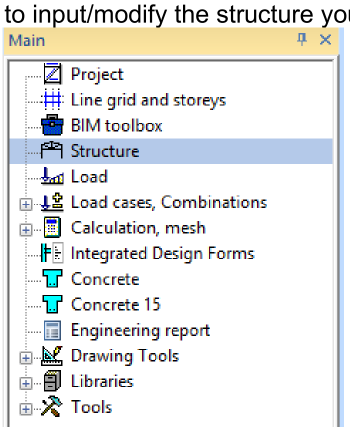
8. Click **[Close]** in the **New Cross-Section** window, the **Cross-Sections** Manager appears.

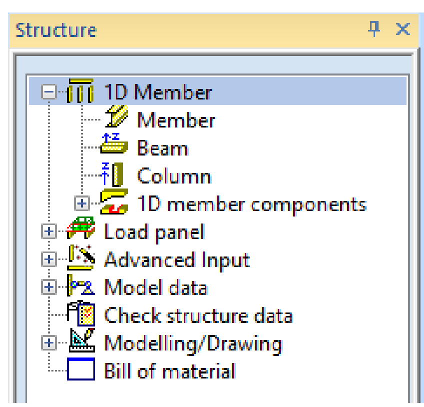
9. Click **[Close]** to close the **Cross-Section** Manager and to return to the basic modelling view.

Geometry

Structure menu

1. When a new project is started, the **Main** tree is automatically opened on left hand side. If you want to input/modify the structure you must double-click on **Structure** in the **Main** window.



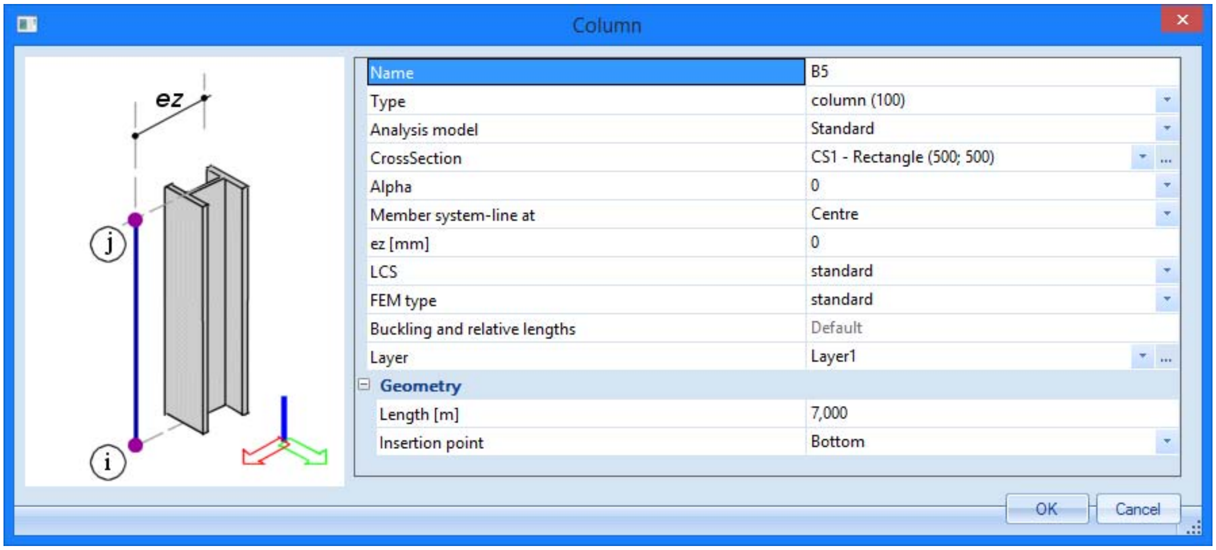


2. In the **Structure menu**, you can choose different structural elements to enter the structure.

To model the frame you first must enter the columns. Next, the beams on top and in the middle of the columns are added.

Entering a column

1. Use the **Column** command in the **Structure** service to enter a new column.



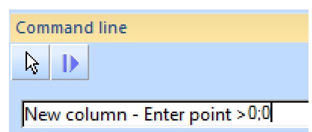
2. In the CrossSection row choose the first entered section **CS1 - Rectangle (500, 500)**.

3. The column length is **7 m**.

4. The insertion position is by default set to **Bottom** so that the bottommost point determines the position of the column.

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

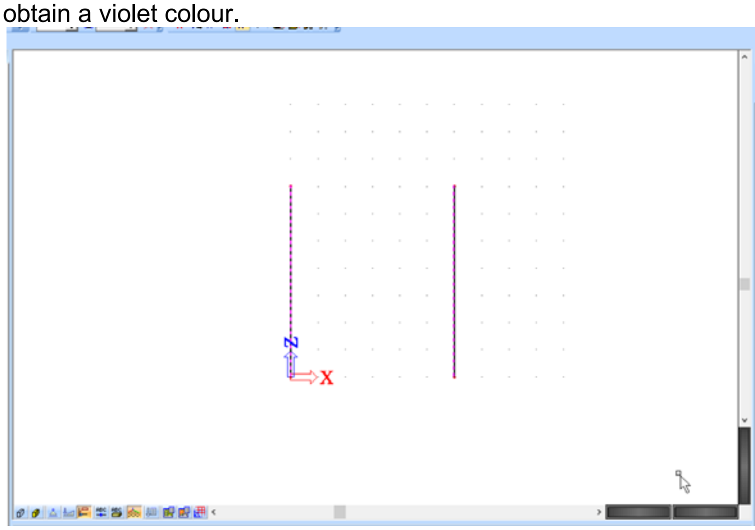
6. The column is positioned in the origin of the coordinate system. To this end, you must enter the coordinates **0;0** in the **Command line** and then press **<Enter>** to confirm your input.



7. A second column is entered in a similar way at position **6;0**

8. End the input using the **<Esc>** key.

9. After input of an entity in SCIA Engineer, the entity is always selected. The columns for instance obtain a violet colour.

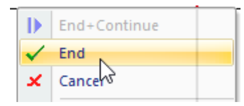


To cancel the selection, press the **<Esc>** key once more.

Notes:
*The properties of selected elements are shown and can be modified in the **Properties window**.*

*If no section has been defined in the project, the **New cross-section** window will automatically appear, as soon as you will try to enter a structural element (column, beam…).*

*You can end your input by pressing either the **<Esc>** key or the right mouse button and pressing **End**.*

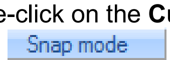


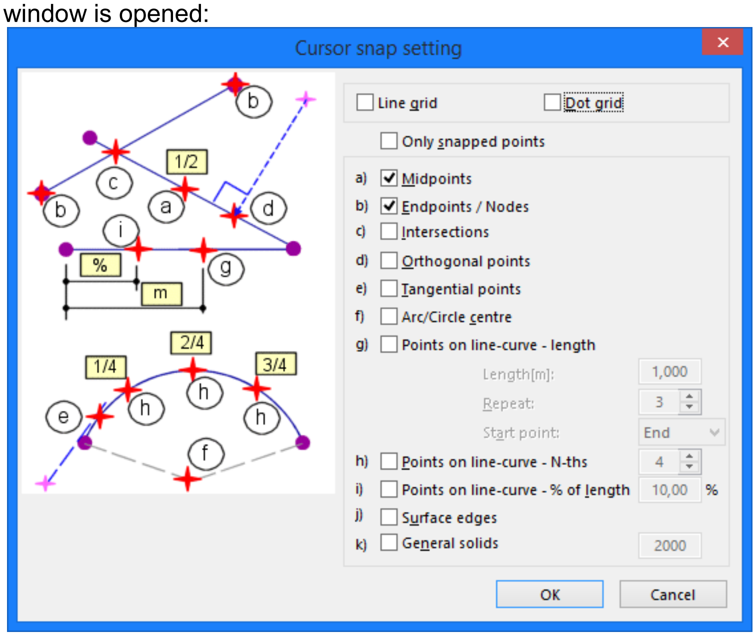
*With **Zoom All***  *in the toolbar or mouse-wheel double-click you can visualize the entire structure.*

To define coordinates in command line you can use semicolon or just leave one space.

When both columns are entered, you can start entering the beams. The start and end points of the beams are already known, i.e. the centre and the end point of a column. Therefore, the beams must not be entered through coordinates, but you can use the **Cursor Snap Settings.**

Cursor snap settings

1. Double-click on the **Cursor snap settings** icon  in the Command line or click on the button  at the lower right corner of the screen. The **Cursor snap settings** window is opened:



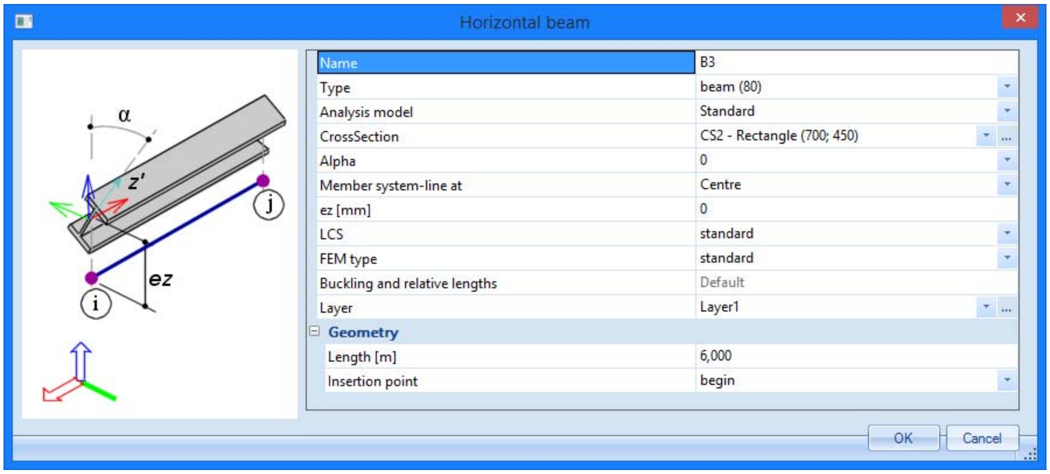
2. Activate the options **a) Midpoints** and **b) Endpoints/Nodes** to pick the desired locations on members in this project.
3. Click **[OK]** to confirm.

Now, you can enter the beams.

Entering a beam

1. To enter a new beam, you use the **Beam** option in the **Structure** service.

2. In the **CrossSection** row choose the second section **CS2 - Rectangular (700, 450)**.

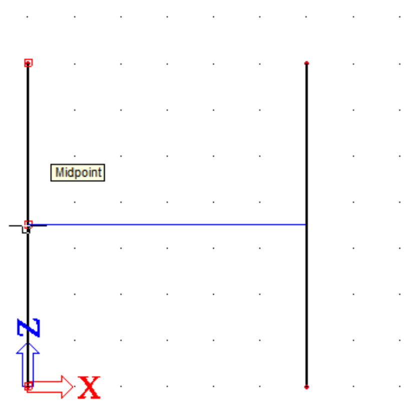


3. The beam length is **6 m**.

4. The insertion position is by default set to **Begin** so that the left point determines the position of the beam.

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

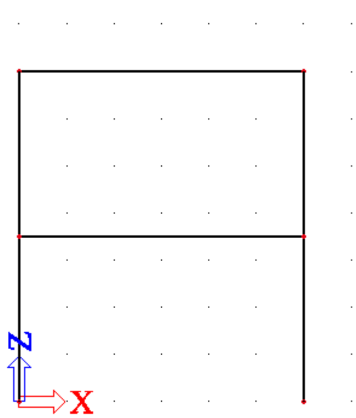
6. Now click with your mouse on the midpoint of the left-hand side column to enter the beam:



7. The upper beam is entered in a similar way by clicking the top node of the left-hand side column.

8. Press **<Esc>** to terminate the input.

9. Press **<Esc>** once more to cancel the selection.

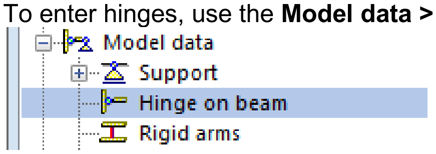


Hinges

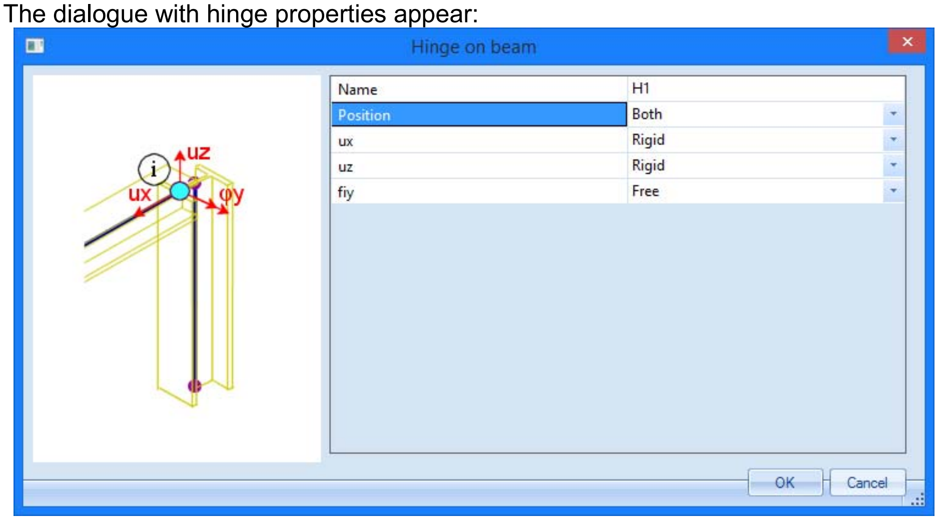
In this project, the beams are connected to the columns in a hinged way. As you have chosen the Frame XZ construction type, this means that by default the members will be connected to each other with fixed ends. Therefore, hinges must be manually entered. Note that hinges are not entered to frame corner nodes but to beam end/ends.

Entering hinges

1. To enter hinges, use the **Model data > Hinge on beam** option in the **Structure menu**.



2. The dialogue with hinge properties appear:



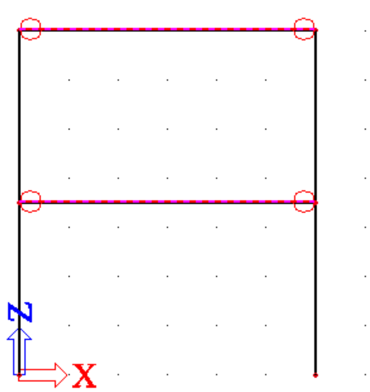
3. The hinges are entered on both sides of the beams, so that you have to choose **Both** for **Position**.

4. To obtain a hinge that does not transfer bending moment from one member to another, the rotation **fiy** is taken **Free**, the translations remain **Rigid**.

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

6. Hinges are added by clicking with the left mouse button on both beams.

7. Press **<Esc>** to terminate the input.



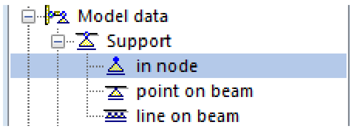
8. Press **<Esc>** once more to cancel the selection.

Supports

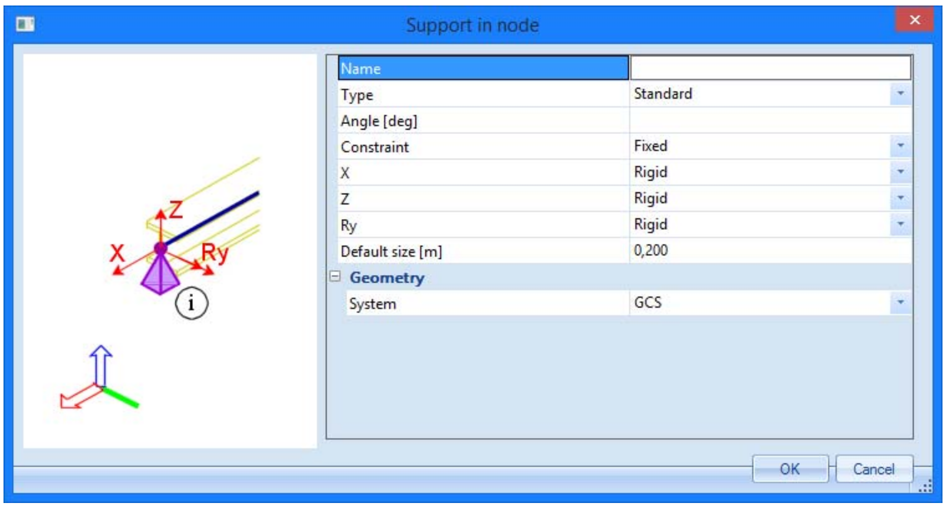
The geometry input can be completed with supports. Both column bases are modelled as fixed supports.

Entering supports

1. To enter supports, use the **Model data > Support > in node** option in the **Structure menu.**

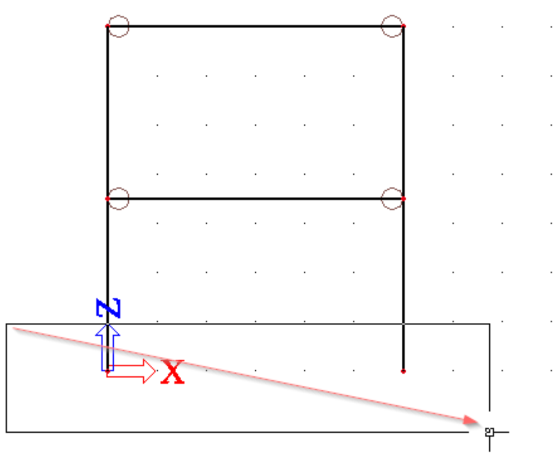


2. To model the fixed ends, both the translations and the rotation are taken **Rigid**. You can also easily select **Contraint** to **Fixed** which makes the definition of below end conditions.



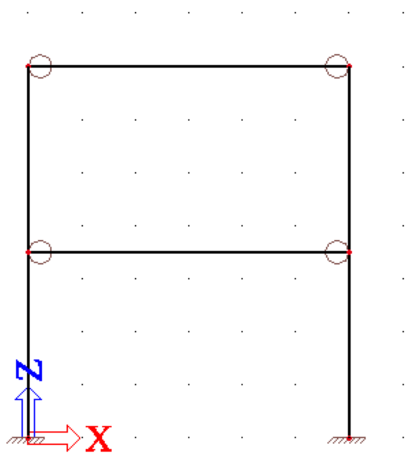
3. Confirm your input with **[OK]**.

4. You can simply select both bottom nodes by drawing a box with the mouse from the left-hand side to the right-hand side:

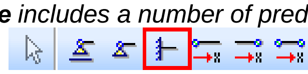


5. Press **<Esc>** to terminate the input.

6. Press **<Esc>** once more to cancel the selection.



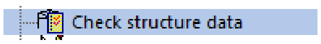
Notes:
If you draw the box from the left-hand side to the right-hand side, only entities, which are completely in the rectangle, will be selected. If you draw the rectangle from the right-hand side to the left-hand side, the entities, which are completely in the rectangle, as well as the entities that intersect with the rectangle will be selected.

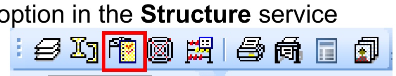
The ***Command line*** *includes a number of predefined supports. For this project, you could have used the* ***Fixed Support***  *icon.*

Check Structure data

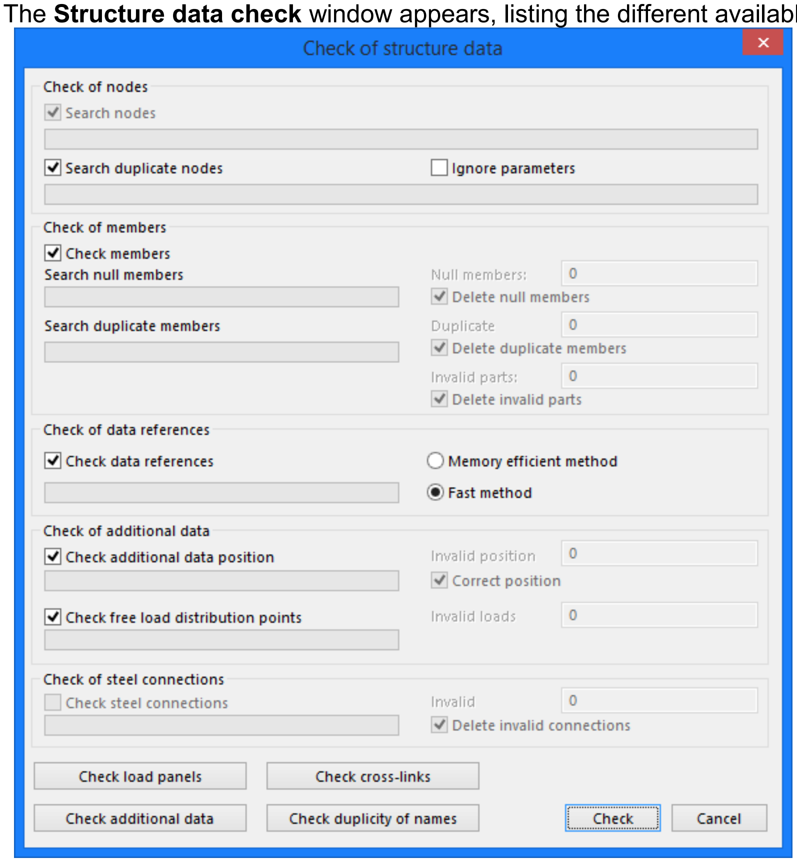
After input of the geometry, the input can be checked for errors by means of the option **Check Structure data**. With this tool, the geometry is checked for duplicate nodes, zero beams, duplicate members, wrong references of hinges or supports etc. However, this tool does not check if the structure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

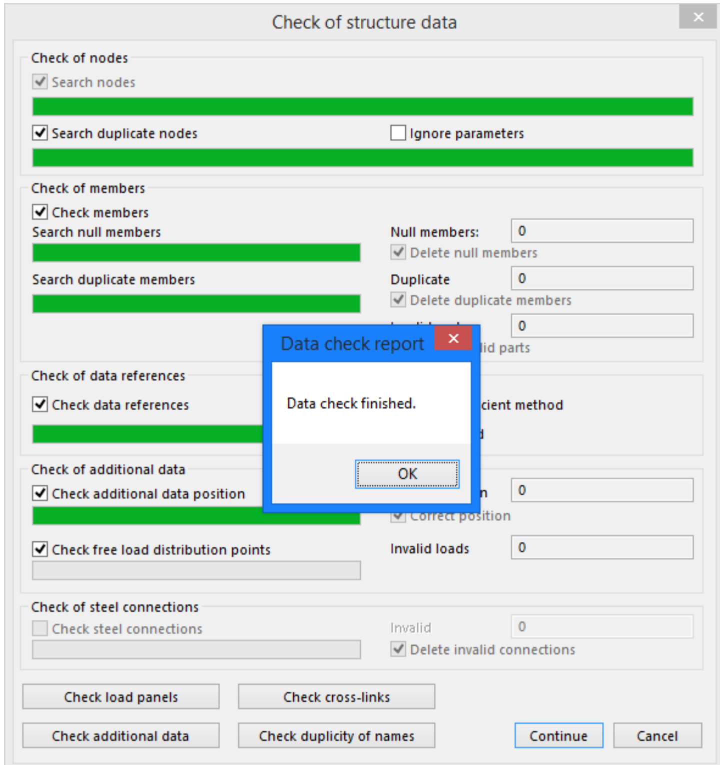
 or click on the  icon in the toolbar.

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



3. Click **[Check]** to perform the checks.

4. The **Data Check Report** window appears, indicating that no problems were found.



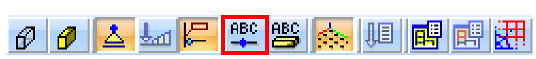
5. Close the check by clicking **[OK]**.

6. In case of any problem SCIA Engineer can automatically correct the structure data (delete duplicated entities, correct wrong reference, etc.)

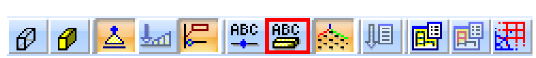
Connecting entities

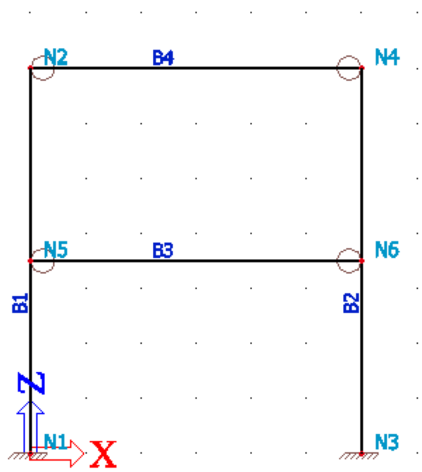
Begin and end node of the top beam is also an end node of a column. Therefore, this beam is automatically connected to the columns.
The beam in the middle of the columns is not modelled in end nodes of other entities, but in midpoints. The end nodes of the beam are located in the internal of the column and therefore are not yet connected to the columns. In this paragraph, we will explain how to connect the members to each other in case they don't have common end nodes. It might be especially important for future editing and smooth calculation.
To display the names of the bars and nodes, you can activate the labels by means of the buttons in the **Command line**.

Activating node labels

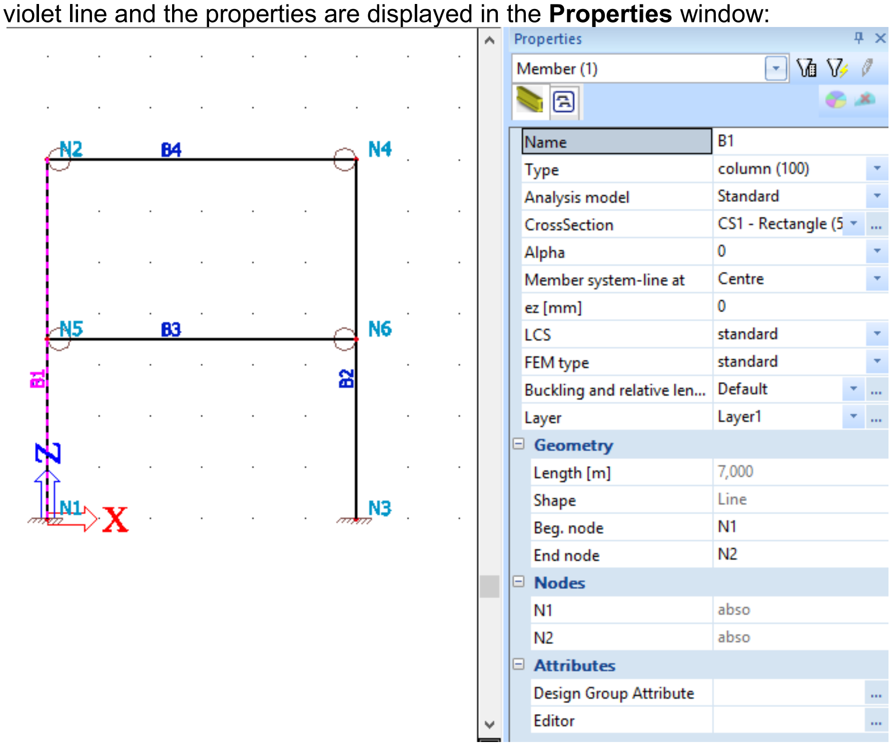
Node labels are activated by means of the  icon at bottom of the modelling window.

Activating member labels

Member labels are activated by means of the  icon at bottom of the modelling window.

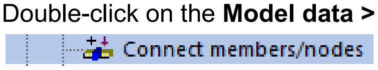


When you select for example column **B1** with the left mouse button, it becomes highlighted by dashed violet line and the properties are displayed in the **Properties** window:

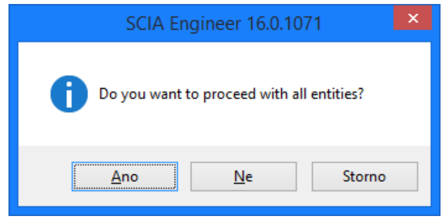


This window indicates that the Begin node is **N1** and the End node **N2**. Node **N5** is not part of the column. To connect beam **B3** to the columns, you must use the command **Connect members/nodes.**

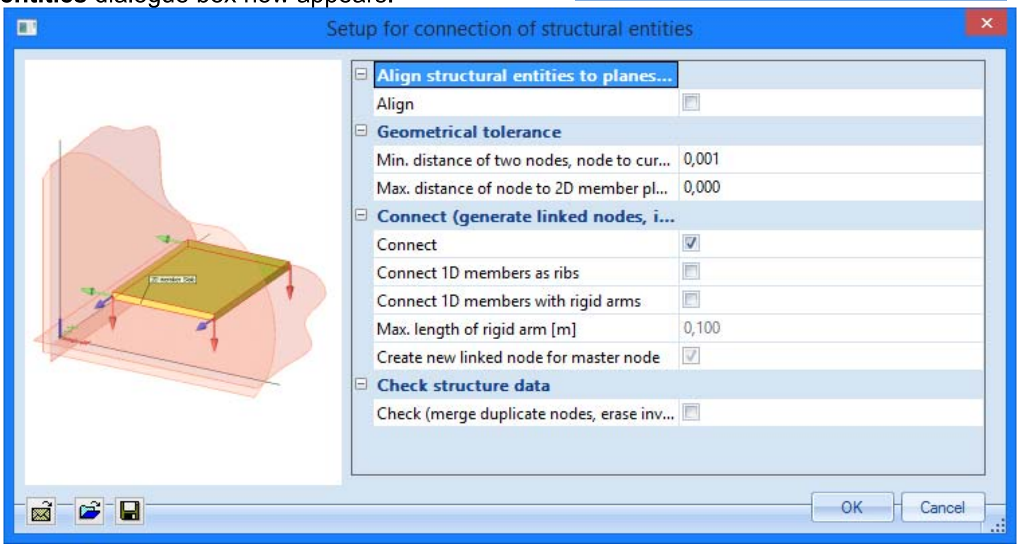
Connecting entities

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** service  or click the  icon in the toolbar.

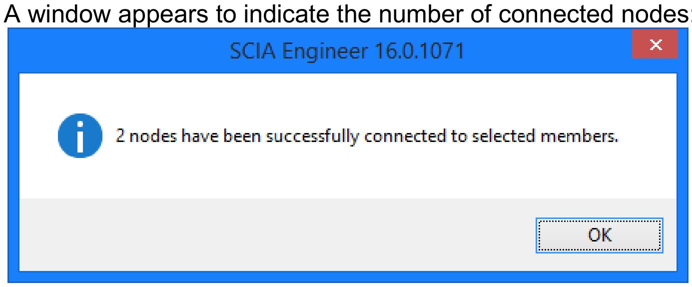
3. A dialogue asks if all nodes must be connected to bars:



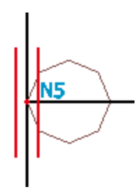
4. Click **<Yes>**.
5. The **Setup for connection of structural entities** dialogue box now appears.



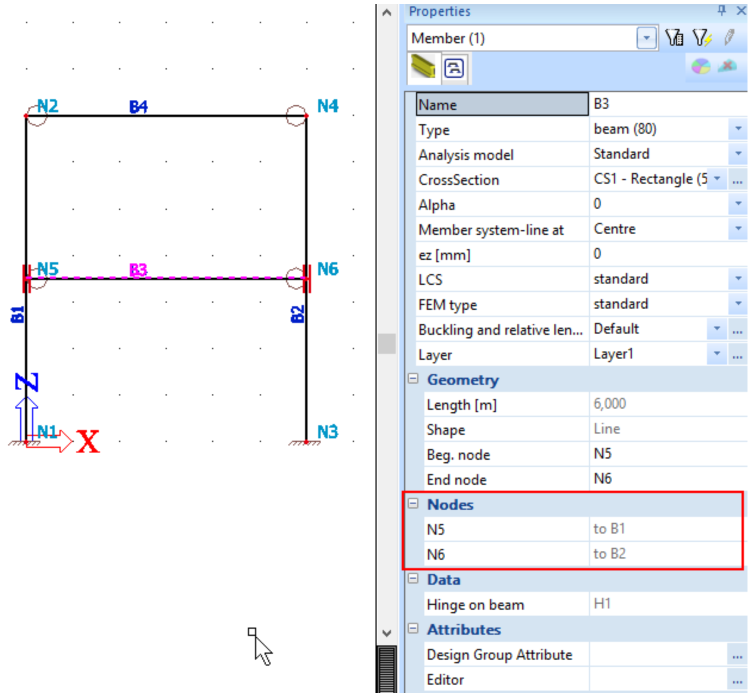
6. Confirm the settings by clicking **<OK>**.
7. A window appears to indicate the number of connected nodes:



8. Connected nodes are represented in the graphical screen by means of double red lines:



If you select for instance beam **B3**, the **Properties** window shows that node **N5** connects the beam to column **B1** and that node **N6** connects the beam to column **B2**.



Note:

*If a possible active selection is not deactivated when the **Connect members/nodes** command is used, the program will only search the nodes to be connected in this selection and not on the entire model.*

*It is also possible to perform the two previous operations at once. Therefore you have to check the option **Check (merge duplicate nodes, erase invalid entities)** in the **Setup for connection of structural entities** dialogue box.*

9. Click **[Close]** at 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 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.*

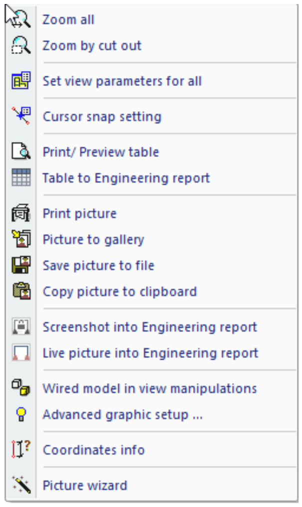
The magnifier

- 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:



Remark:

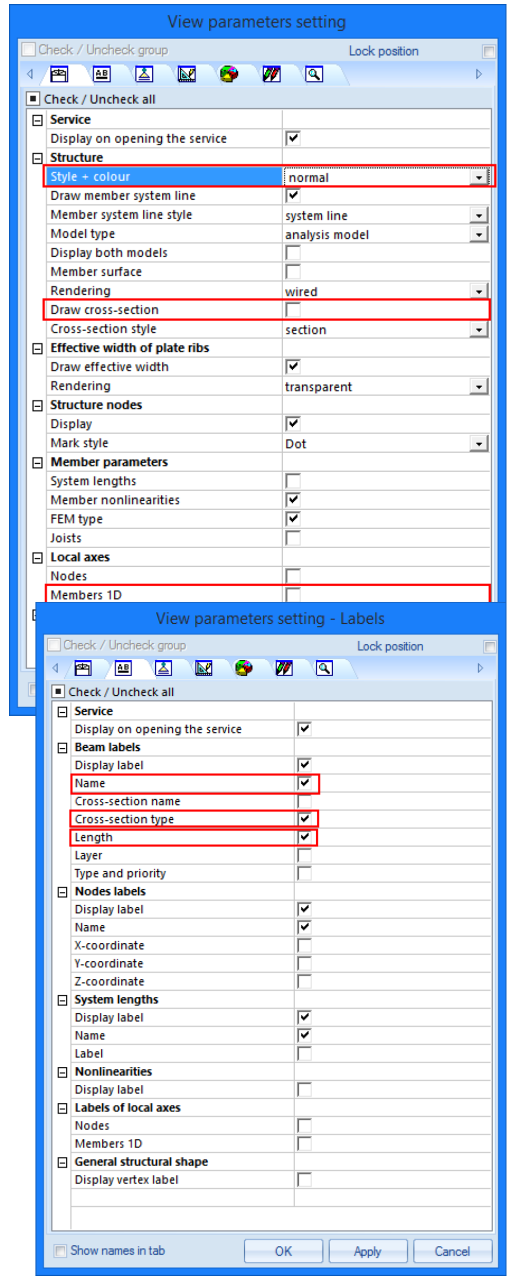
If an entity was selected previously, you can define a setting that only applies to the selected elements. (An adapted shortcut menu appears).

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 – Structure

Through the tab Structure the representation of the different entities can be adapted. In the group **Structure** the following items are important for this project:

- **Style and colour:** You can display the colour per layer, material, cross-section, structural type or design group.
- **Draw cross-section:** With this option checked the symbol of the cross-section is displayed on every 1D member.
- **Local axes:** With this option the local axes of the elements are activated.



View parameters – Labels

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:

- **Name:** Show the name of the cross-sections in the label (e.g. CS.)
- **Cross-section type**: Show the cross-section type in the label (e.g. Rectangle (500; 500)).
- **Length**: show the length of the member in the label (e.g. 6,000 m).

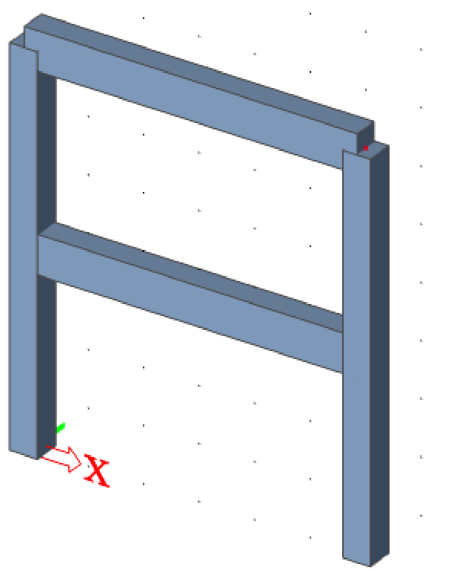
View parameters – shortcuts

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):



Loads 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: Permanent load case: Self weight of the members + weight of the floor and weight of the roof
LC2: Variable load case: Service load on the floor

Defining a Permanent Load Case

1. Double-click on 

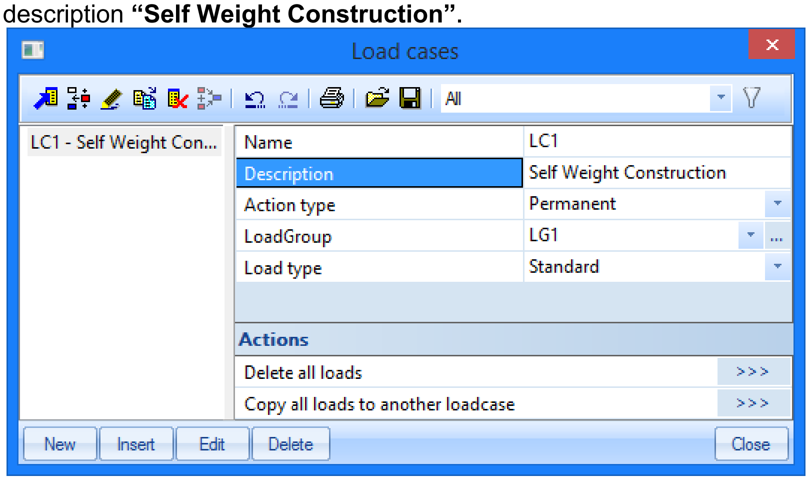
 in the **Main tree**.

2. Before you can define loads, you must enter load cases first. Since this project does not contain any load cases yet, the **Load cases** manager will automatically appear.

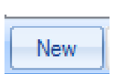
3. By default, the load case **LC1** is created. This load is a permanent load of the **Self Weight** load type. The self weight of the structure is automatically calculated, although not graphically displayed.

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 type the description **"Self Weight Construction"**.



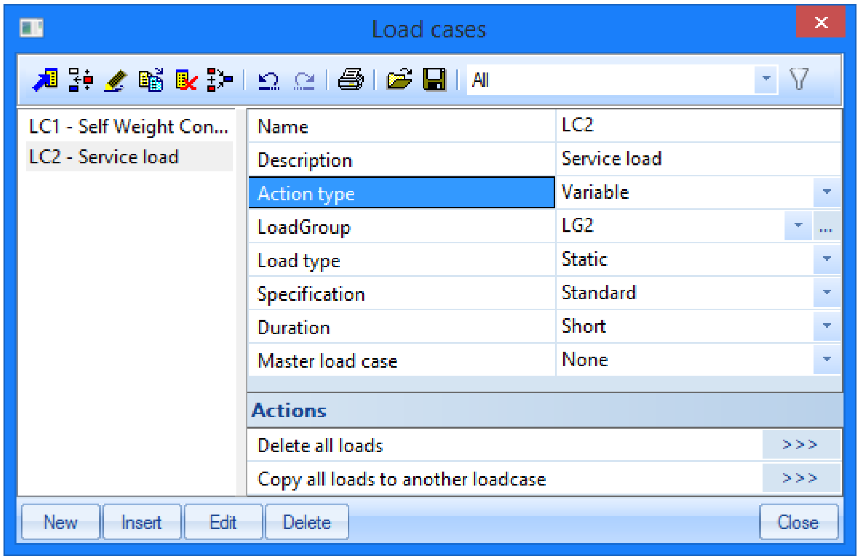
Defining a Variable Load Case

1. Click 

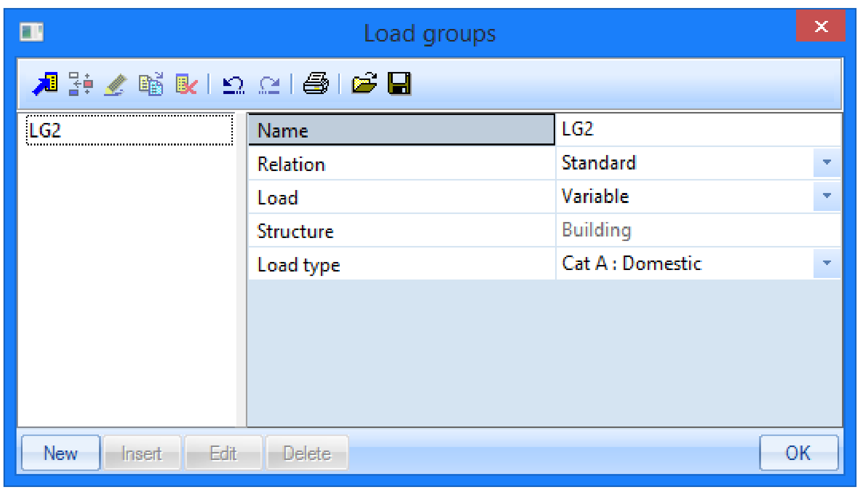
 or  to create a second load case.

2. Enter the description **"Service 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 Load type determines the composition factors that are attributed to the load cases in this load group when Eurocode combination is requested. In this project, choose **Cat A: Domestic.**

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**.

Remark:
Load groups

Each load is classified in a group. These groups influence the combinations that are generated as well as the code-dependant coefficients 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 EN 1991). 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.

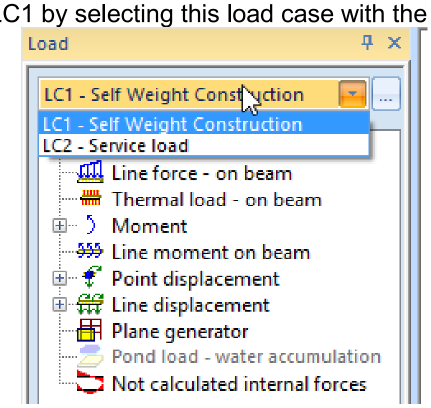
Loads

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

The first load case (**LC1**) includes two loads:
- Self weight of the members
- Roof weight + Floor weight

Switching between load cases

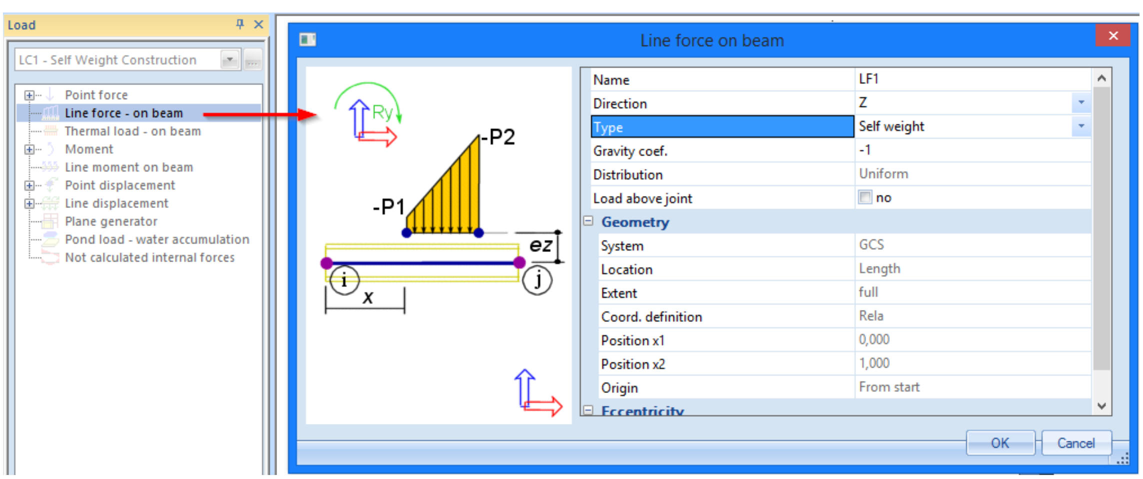
Activate LC1 by selecting this load case with the mouse pointer in the combo-box:



Entering the self weight as linear load

1. Cancel any possibly active selection of entities by pressing **<ESC>**.

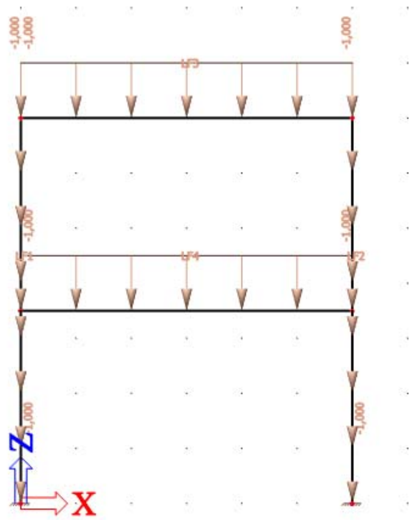
2. Click on **Line Force - on beam** in the **Loads Menu**. The dialogue **Line Force on beam** appears.



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.

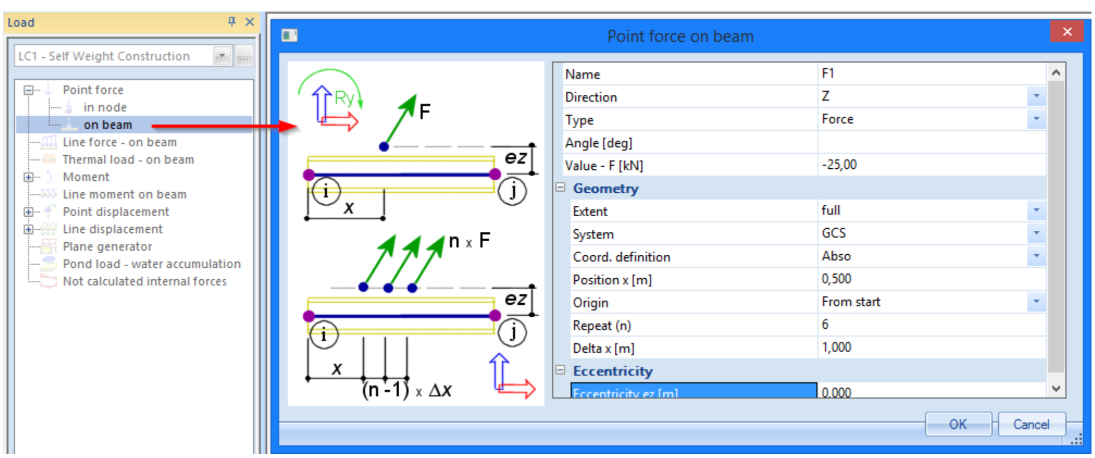
The self weight load is represented in brown:



Now, the floor and roof weight is entered as a series of concentrated loads:

Entering a series of concentrated loads

1. Click on **Point force on beam** in the **Loads Menu**. The dialogue **Point force on beam** appears.



2. The floor beam is charged with 6 concentrated loads of **25 kN** with a distance of **1 m**. The first concentrated load of the series is at **0.5 m** from the starting node of the beam.
3. The Value of the point load is changed to **–25 kN**.
4. The coordinate definition is set to **Abso** (meaning absolute).
5. The starting position “Position x” is changed to **0.5 m**.
6. The series consists of 6 concentrated loads so that the Repeat (n) field is set to **6.**

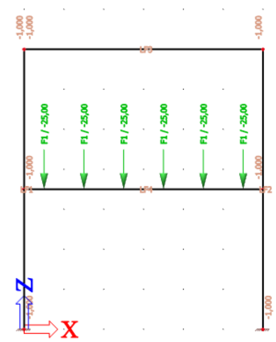
7. The clearance Delta x between the concentrated loads is **1 m**.

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

9. Select beam **B3**.

10. Press **<Esc>** to terminate the input.

11. Press **<Esc>** once more to cancel the selection.



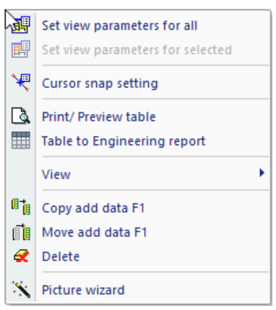
The roof beam is similarly loaded with concentrated loads of **12.5 kN**. Therefore, the load of the floor beam can be copied to the roof beam and adapted according to the steps below.

Note:
Loads, supports, hinges etc. are considered as additional data, i.e. data that are additionally added to entities such as nodes, bars...

Copying loads

1. Select one of the point loads on the floor beam with the left mouse button. As this point load is part of a series, the entire series is automatically selected.

2. Press the right mouse button, a popup menu appears:



3. Choose the option **Copy add data F1**.

4. Select the bar where this load should be copied - the roof beam B4.

5. Press **<Esc>** to terminate the input.

6. Press **<Esc>** once more to cancel the selection.

Now, the value of the roof load can be changed.

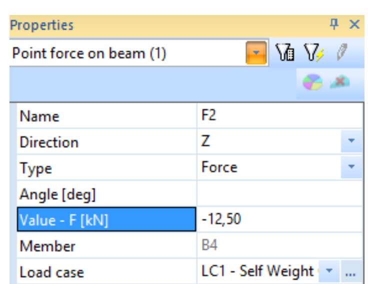
Adapting a load

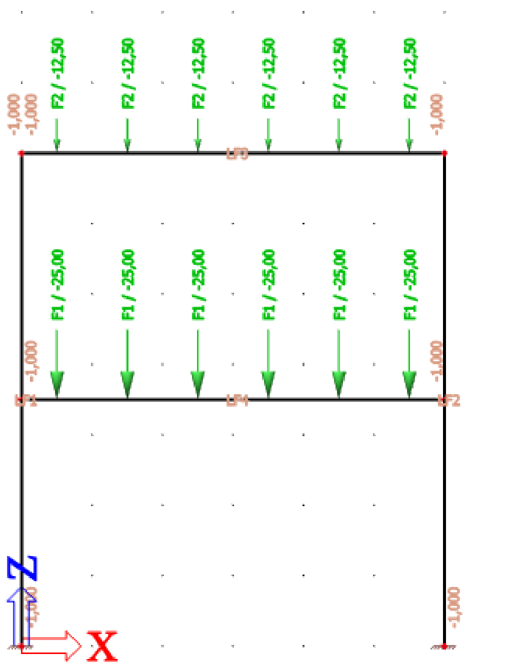
1. Select one of the point loads on the roof beam. As this point load is part of a series, the entire series is automatically selected.

2. The properties of the series are displayed in the **Property Window**.

3. Change the value from **–25 kN** to **–12.5 kN**.

4. Confirm the change by pressing **<Enter>**. Below is the resulting view:

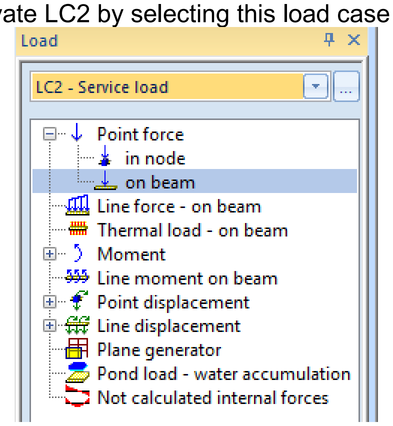




After input of the loads in the first load case you can enter the service load into the second one. The floor beam is loaded with a service load of 10 kN/m.

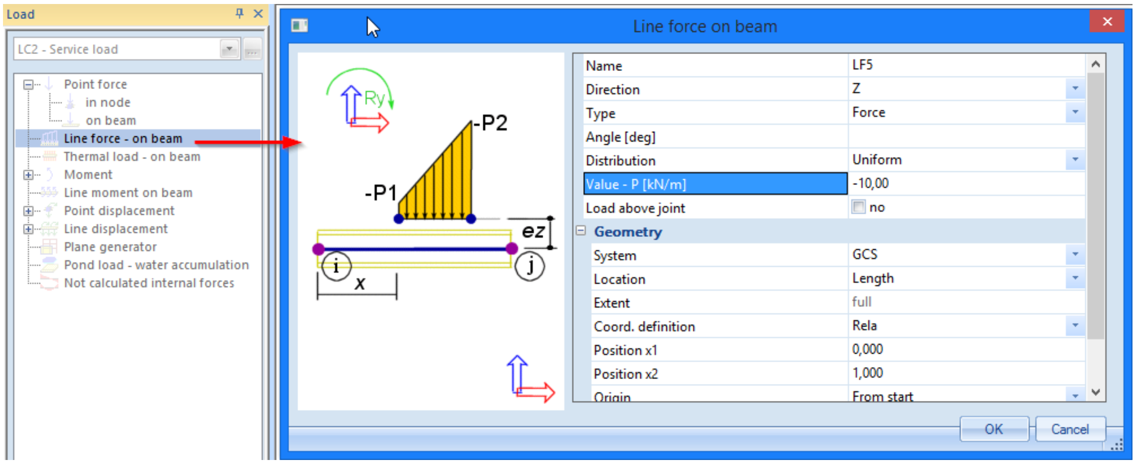
Switching between load cases

Activate LC2 by selecting this load case with the mouse pointer in the combo-box:



Entering a linear load

1. Click on **Line force – on beam** in the **Loads Menu**. The dialogue **Line force on beam** appears.
2. Change the type to **Force** and the value to **-10 kN/m**.

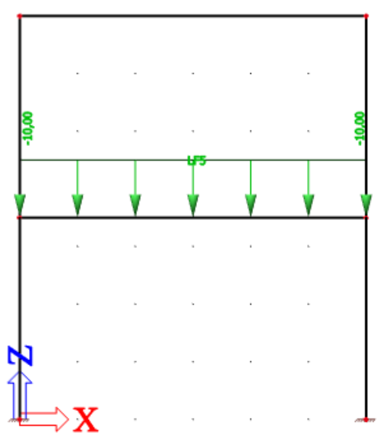


3. Confirm your input with **[OK]**.

4. Select the bar on which this load must be positioned: floor beam **B3**.

5. Press **<Esc>** to terminate the input.

6. Press **<Esc>** once more to cancel the selection.



7. Click **[Close]** to leave the **Loads service** and to return to the **Main tree**.

Note:

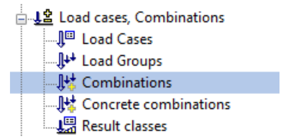
*The **Command line** includes a number of predefined loads*  *, which enable a fast and simple input of usually used type of loads.*

Combinations

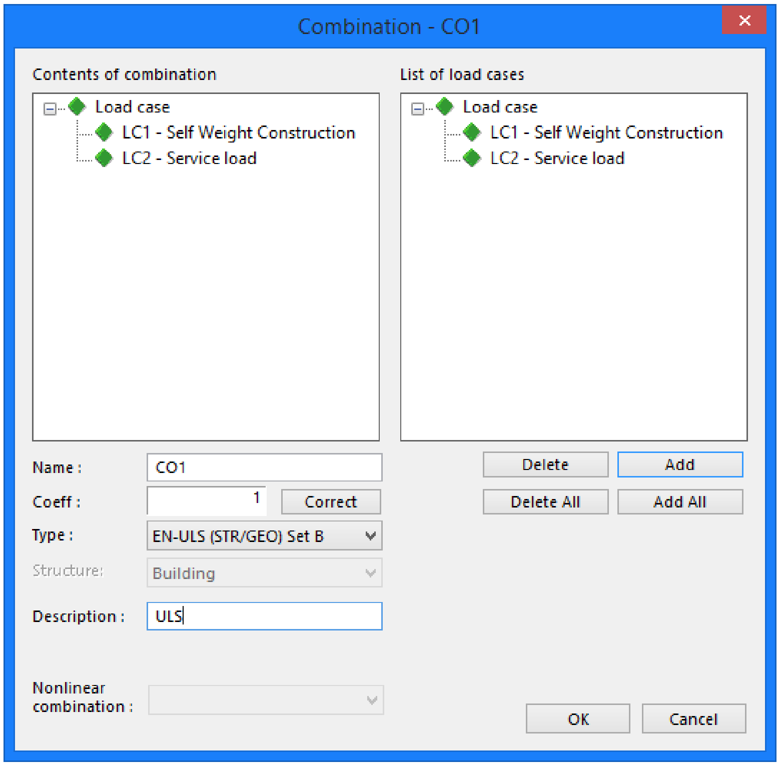
After input of loads and load cases, the latter can be grouped in combinations. In this project, two code combinations are created, one for the Ultimate Limit State and one for the Ultimate Serviceability State.

Defining Combinations

1. Double-click on Combinations in the **Main tree**.

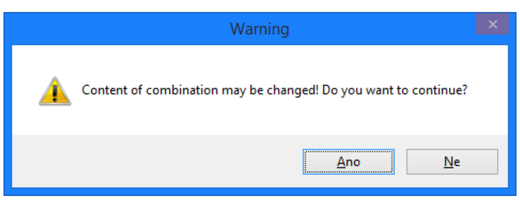


2. Since no combination has been entered yet, the window to create a new combination automatically appears.



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.

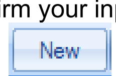
4. A warning message that controls the content of code combinations with respect to load type may appear. Close it with **[Yes]**



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.

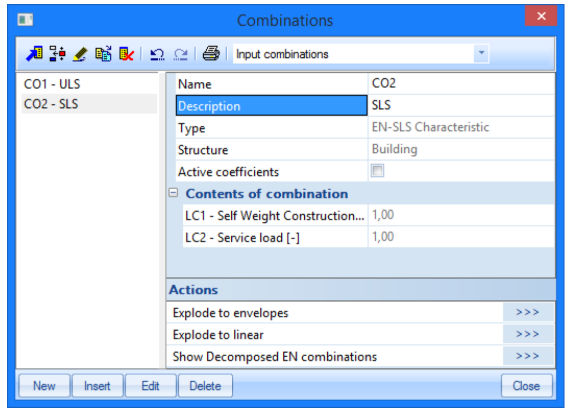
8. Click 

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**.



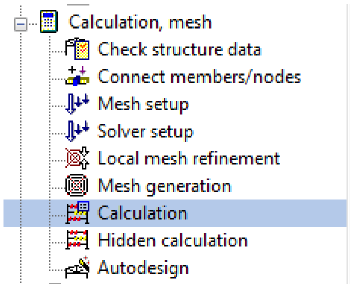
Calculation

Linear Calculation

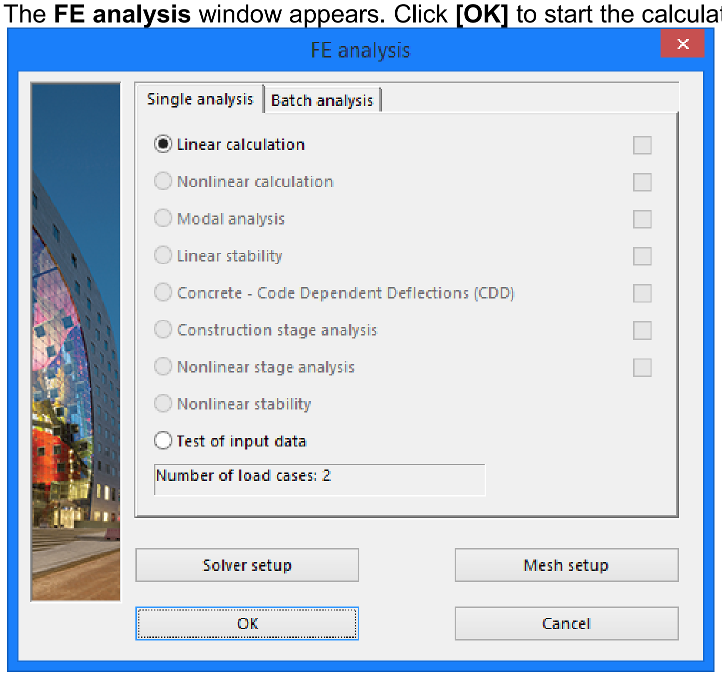
As the calculation model is completely ready, you now can start the calculation.

Executing the Linear Calculation

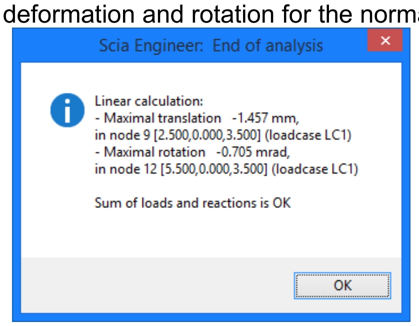
1. Double-click on Calculation in the **Main window,** or use identical icon in toolbars.



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

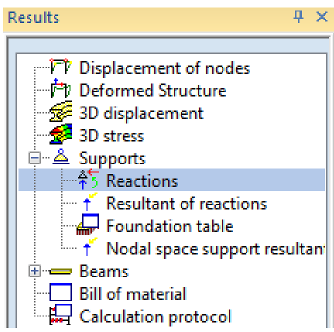
When the calculation is completed, results can be viewed. New service appears in the **Main** tree and also Properties window announces that Linear calculation is finished.

Viewing the Reaction Forces

1. Double-click on 

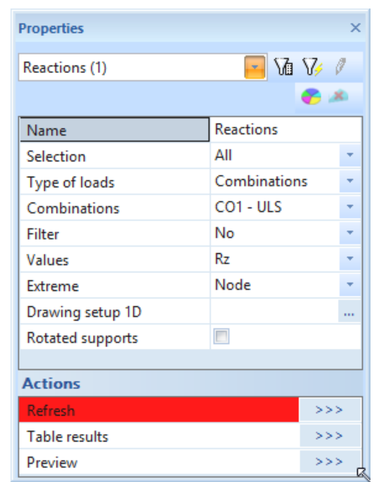
in the **Main tree**. The **Results menu** appears.

2. Below **Supports**, click **Reactions**.



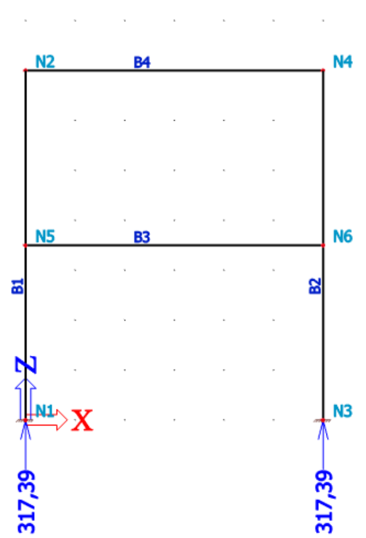
3. The options in the **Property Window** are configured in the following way:

- The Selection field is set to **All**.
- The Load type is set to **Combinations** and the Combination to **CO1 - ULS**.
- The **Values** are wanted for **Rz**.
- The **Extreme** field is changed to **Node**.

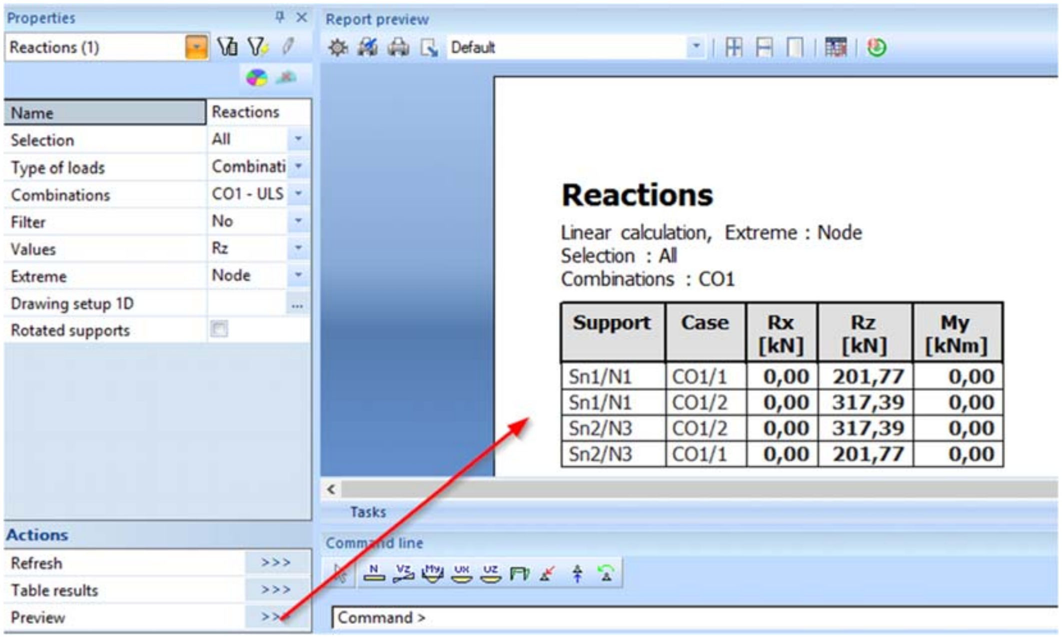


4. The action button **Refresh** has a red highlight, i.e. the graphical screen must be refreshed. Click on the 

button next to **Refresh** to display the results in the graphical screen in accordance with the options above.



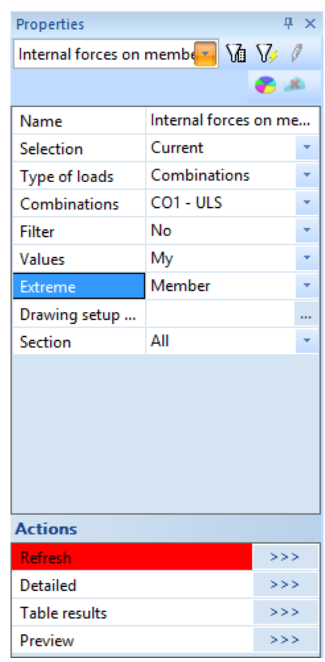
5. To display these results in a table, the **Preview** action is used. Click on the  next to **Preview** to open the Report preview.



Support	Case	Rx [kN]	Rz [kN]	My [kNm]
Sn1/N1	CO1/1	0,00	201,77	0,00
Sn1/N1	CO1/2	0,00	317,39	0,00
Sn2/N3	CO1/2	0,00	317,39	0,00
Sn2/N3	CO1/1	0,00	201,77	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 beam

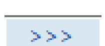


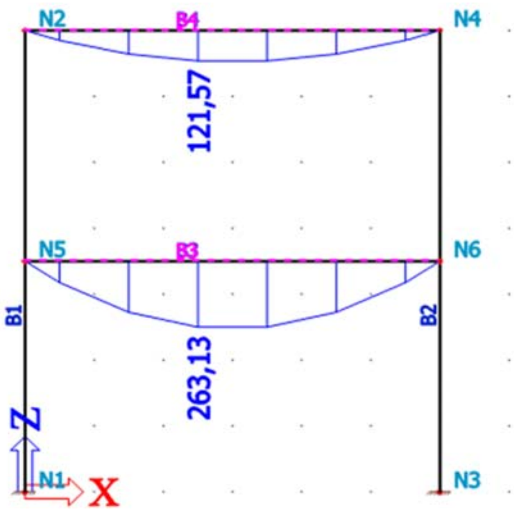
1. In the **Results** menu, open the **Beams** group and select **Internal forces on beams.**

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

   - The Selection field is set to **Current**.
   - The Load type is set to **Combinations** and Combination to **CO1 - ULS**
   - The Values are wanted for **My**.
   - The Extreme field is changed to **Member**.

3. Select the two beams **B3** and **B4** using the left mouse button.

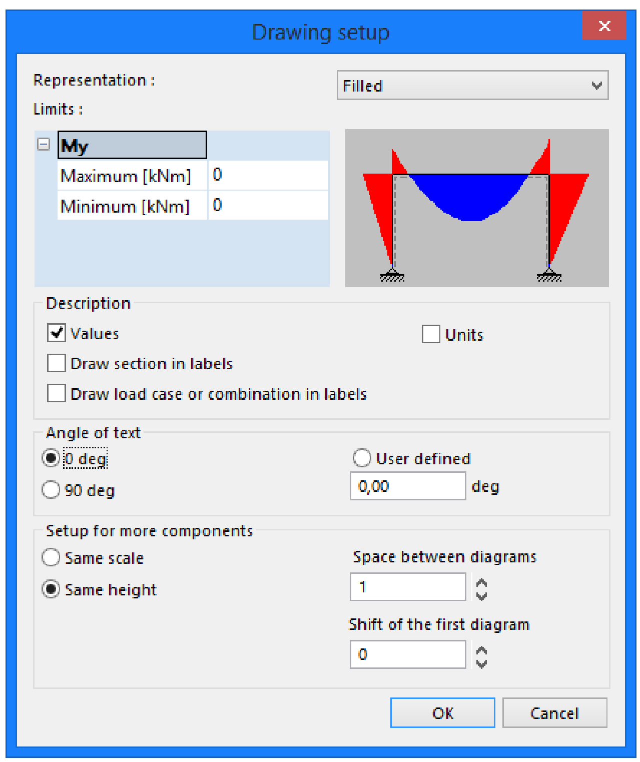
4. Click on the  button next to **Refresh** to display the results on the graphical screen in accordance with the set options.



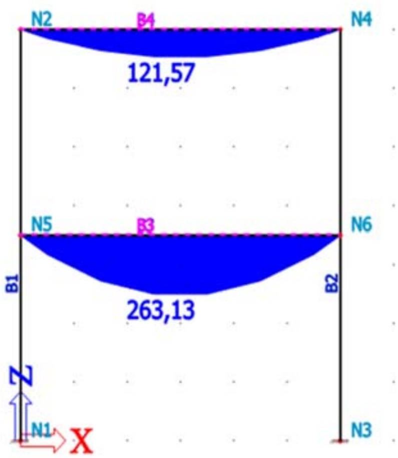
To change the display of the results, the settings of the Graphical screen can be adapted.

Configuring the Graphical Screen

1. In the **Properties window**, click the icon next to **Drawing setup 1D**. Possibilities for various graphical representations are opened.



2. In the **Representation** combo-box choose **Filled**.
3. The **Angle of text** is set to **0 deg**.
4. Click **[OK]** to confirm your input.
5. In the **Property Window**, click the  button next to **Refresh** to display the results in the graphical screen in accordance with the above options.



6. Then click **[Close]** to leave the **Results service**.
7. Press **<ESC>** to cancel the selection.

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.*

Code check

Concrete modules of SCIA Engineer contain a number of powerful tools to execute the concrete calculations in accordance with the chosen design code.
The possibilities are:

- Input of advanced concrete data
- Calculation of the slenderness
- Reduction of bending moment and shear force above supports
- Design of theoretically needed reinforcement amount
- Input of the Practical reinforcement
- Capacity-response check
- Stress limitation
- Crack width analysis
- Calculation of physical non-linear (PNL) deformation
- and more...

In this tutorial, we will only explain the basics of the concrete design. For more information regarding the advanced concrete calculations we refer to the Concrete course (training) document.
Before you can start the concrete design you must first check the buckling parameters of the bars. By means of the view parameters, the buckling lengths of the bars can be displayed on the structure.

Buckling parameters

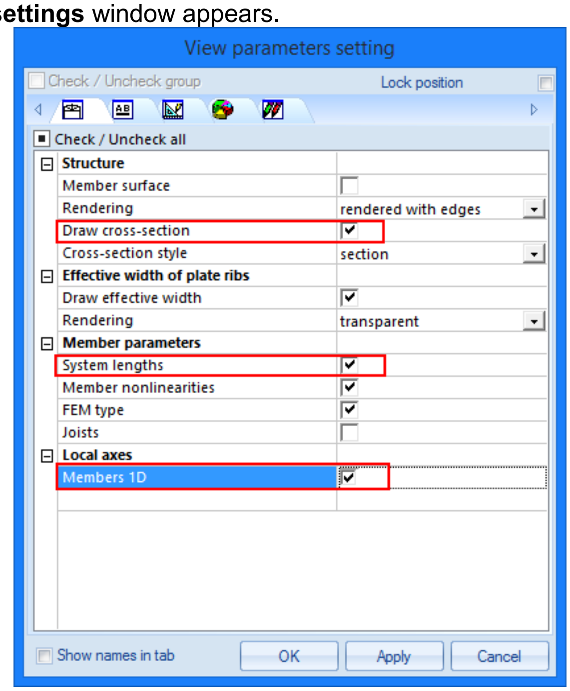
Displaying the system lengths

1. Select column **B1** with the left mouse button, the left column of the first frame.

2. Click the right mouse button at an arbitrary position in the workspace. Context menu lists the possibilities for the selected entity.

3. In this menu, hit the 

option. The **View parameter settings** window appears.

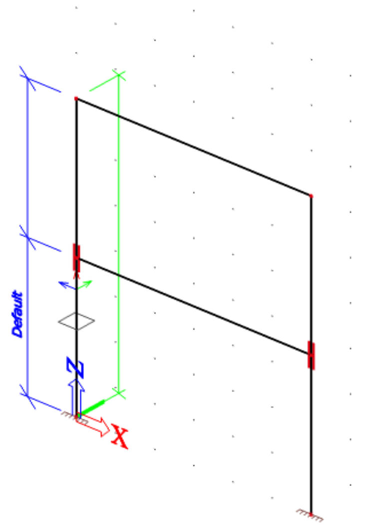


4. Activate the **System lengths** and **Draw cross-section** options to display the reference lengths and the section of the bar.

5. Activate the **Local axes – Members 1D** option to display the local coordinate system of the bar.

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

7. Press **<ESC>** to cancel the selection.



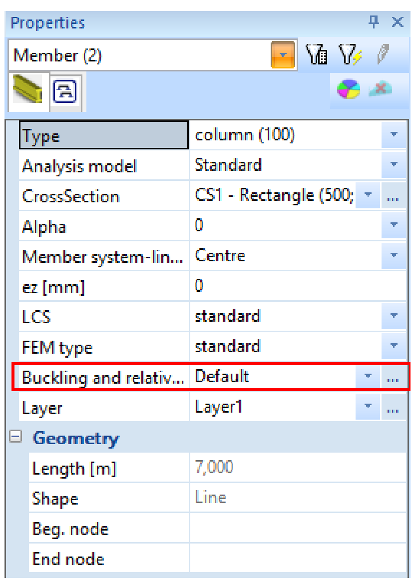
The figure shows that system length (Ly) for buckling around the strong axis (y-y) is 3,5 m and Lz for buckling around the weak axis (z-z) is 7 m. The beam in the middle of the column therefore supports the column for buckling around the strong axis.

To modify the buckling data of a member use the option **Buckling and relative lengths** in the **Property window** of the selected member.

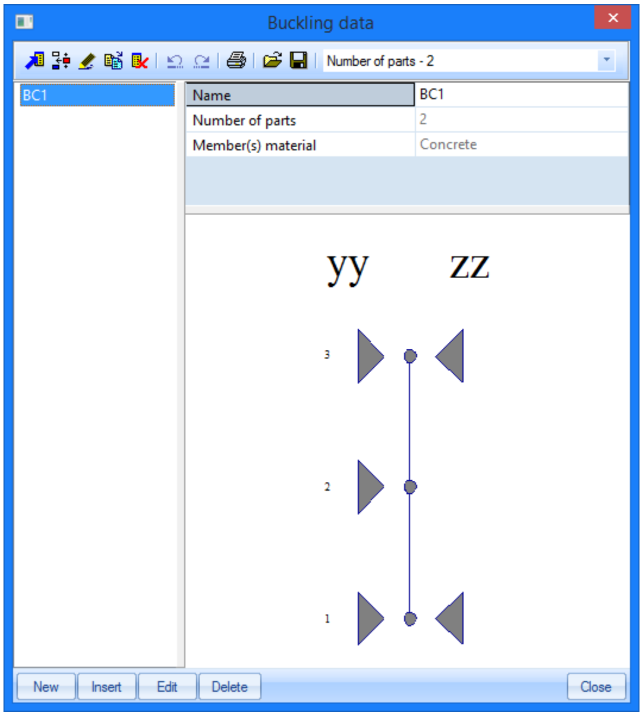
Setting the Buckling Parameters

1. Select both columns with the left mouse button.

2. The **Properties window** shows the common properties of both entities. The **Buckling and relative lengths** are set to **Default**.

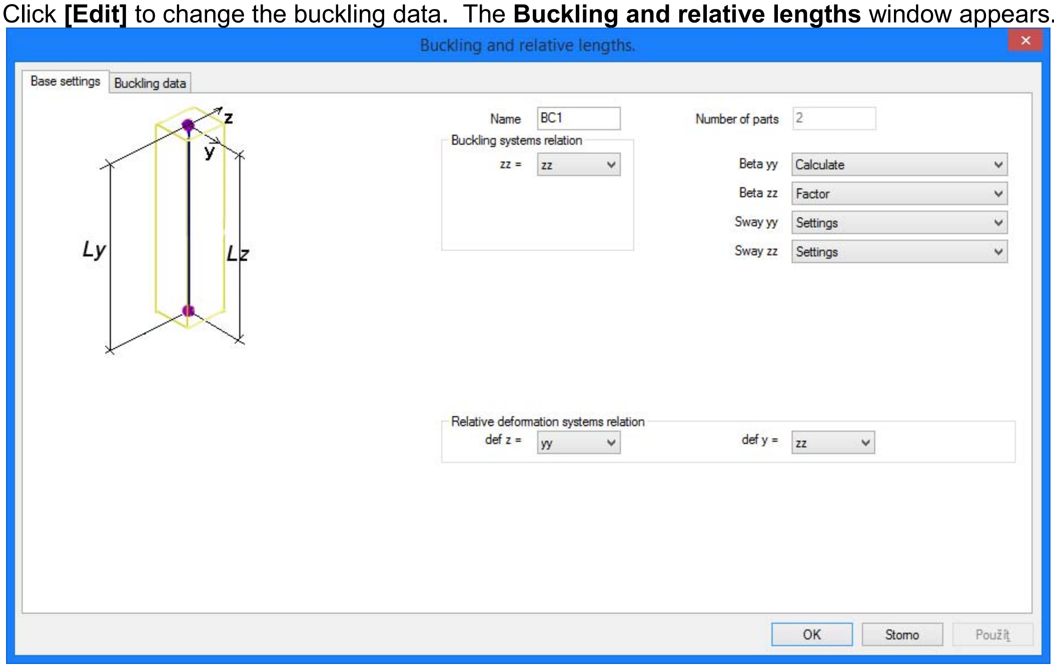


3. Click the ... icon next to **Buckling and relative lengths**. The **Buckling data** window appears.



This window shows that the column is supported in the middle for buckling around the strong axis (Y-Y) but not for buckling around the weak axis (Z-Z).

4. Click **[Edit]** to change the buckling data. The **Buckling and relative lengths** window appears.



5. On the **Base Settings** tab several data can be changed.

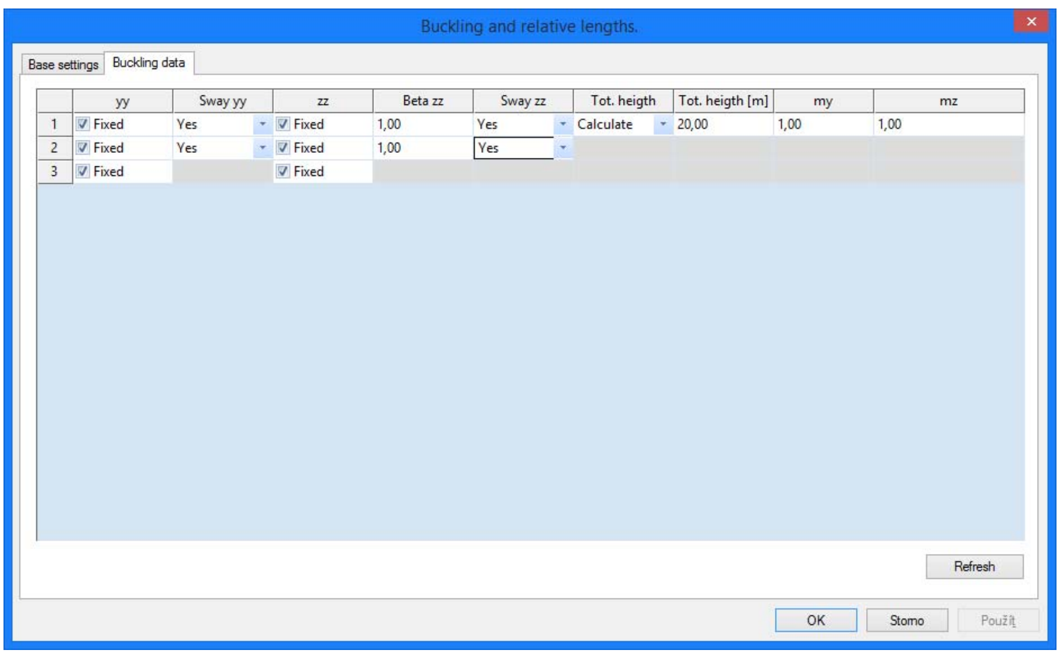
- The **Name** field contains the name of the buckling parameter, in this case **BC1**.
- **Beta yy** and **Beta zz**: in these combo-boxes you can indicate if the program should calculate the buckling factor round the particular axis or if you prefer entering this factor manually. A third option allows for a manual input of the buckling length, instead of buckling factor. The Support option can be used to determine the buckling factor in accordance with the model column from the Eurocode.

- **Buckling system relation zz**: in this field, you can indicate the system length to be used for the weak axis, whether it is dependent on the system for strong axis (zz=yy) or if it is independent (zz=zz).
- **Sway yy** and **Sway zz**: in these fields, you can indicate if the member is braced or not in the particular direction. When you choose the Settings option, the default settings (from the general concrete setting) are used.
  *The default settings for the buckling parameters are displayed in **Concrete 15 > Concrete setting (structure) > Design defaults > Default sway type**. By default, both directions are unbraced for a concrete calculation. Thus for concrete calculation, no wind bracings are assumed.*
- **def z** and **def y**: in these fields, you can indicate the buckling system to be used for the relative deformations – the length to be used to detect relative deformation check.

6. On the **Buckling data** tab you can change the parameters in detail. As the columns consist of 2 components, 3 positions are available: (1) at the beginning, (2) in the centre of the floor beam and (3) at the end.

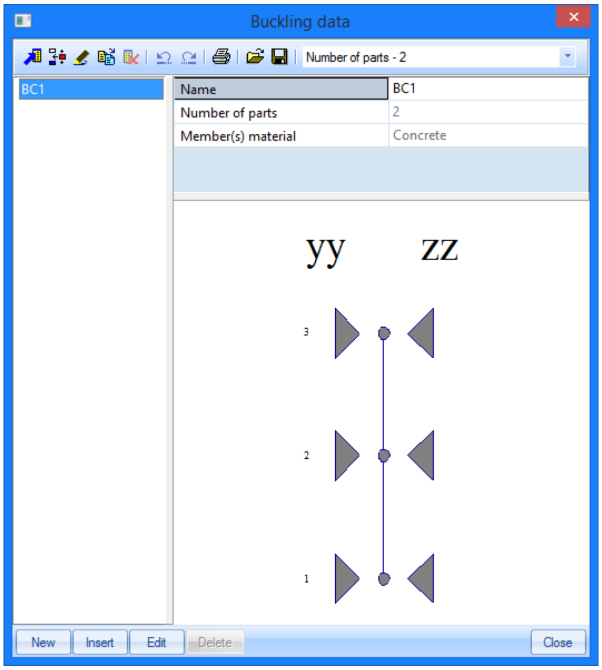
   - For this project, we assume that the columns are not braced in both directions. Therefore, the **Non-braced** property can be set to **Yes** both for the strong and the weak axis. This could also be left on settings, since default settings are sway for concrete calculation.

   - For this project, we also assume that the column is supported in the middle for buckling around the weak axis. The **zz** property at position (2) therefore can be set to **Fixed**.



7. Click **[OK]** to close this window.

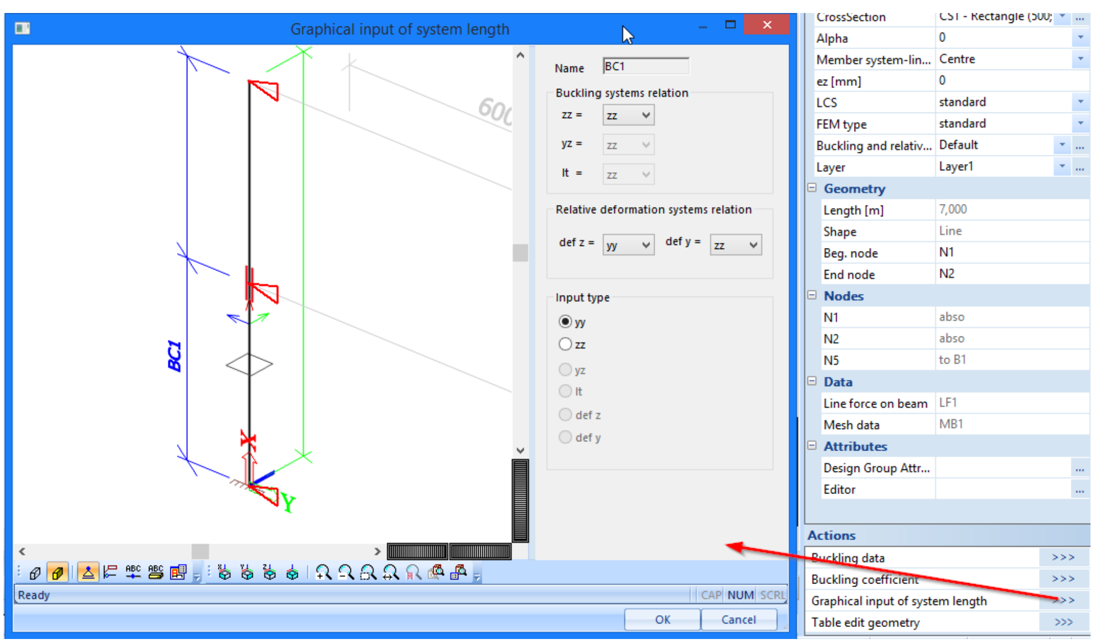
8. The **Buckling data** window re-appears and displays the changed buckling data. Click **[Close]** to close this window.



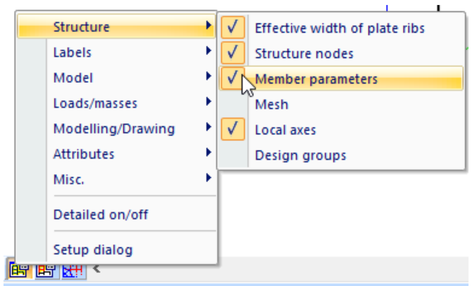
9. The **Properties window** indicates that the buckling parameter **BC1** is used for both columns.

10. Press **<Esc>** to cancel the selection.

Remark:
You can double check the buckling system setting by action button Graphical input of system length. Here you can also change free nodes to fixed (by clicking the red triangles at specific locations at the members) and the other way round, or change relations between buckling systems.



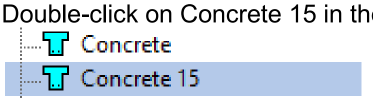
When the buckling parameters are set, you can continue with the concrete design. Before proceeding, deactivate the display of system lengths through Fast adjustment of view parameters on whole model button.



When the buckling data are adapted, you must recalculate the project. See chapter Linear calculation.

Concrete design

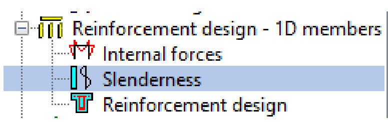
Double-click on Concrete 15 in the **Main tree** to open the **Concrete menu**.



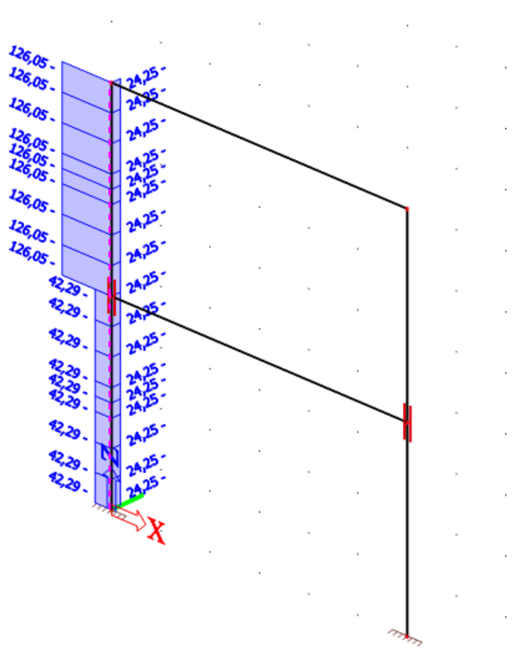
There are two services for concrete design in version SCIA Engineer 15 and 16. The old one called simply Concrete won't be discussed in this tutorial because the new one called Concrete 15 replaces it.

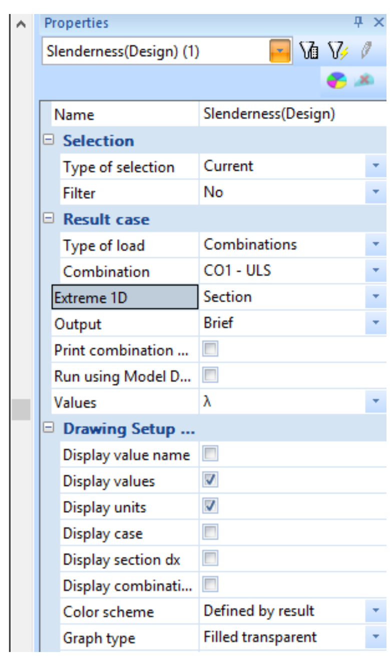
Displaying the Slenderness and the Buckling Lengths

1. Click the Slenderness icon in the **Concrete 15** menu under chapter Reinforcement design – 1D members

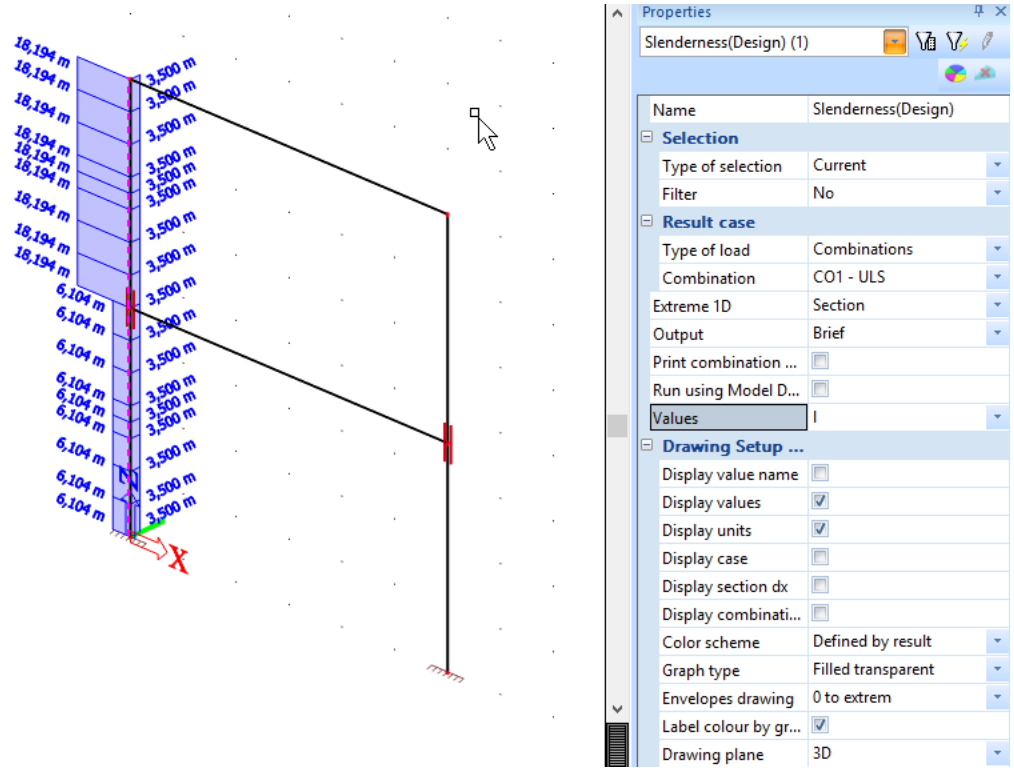


2. The options in the **Properties window** are configured in the following way:
   - **Type of selection** field is set to **Current**.
   - **Type of load** is set to **Combinations** and Combination to **CO1** - **ULS**.
   - **Values** are set to **λ**, i.e. the slenderness lambda in both directions.
   - **Extreme 1D** field is modified to **Section**.
3. Select column **B1**, the left column of the frame and hit the button **Refresh**



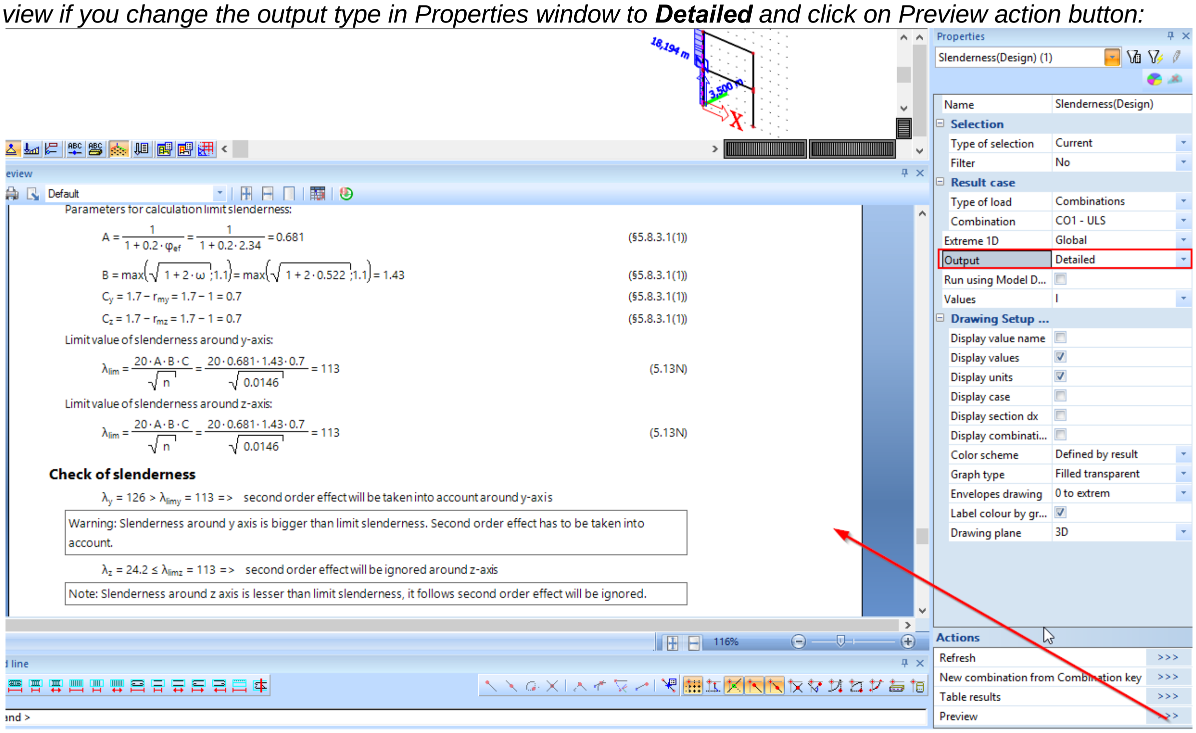


4. Change the **Values** field to **l** to display the buckling length for buckling around both axes. Hit again the **Refresh** action button to regenerate the view.



As already indicated in the buckling parameters, the buckling length is 3,5 m.

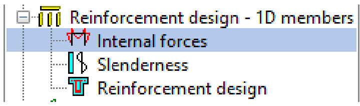
Notes:
*The buckling length **ly/lz** is determined from system length **Ly/Lz** multiplied with buckling factor **βy/βz**. The calculation of slenderness is important for reinforcement calculation in columns. It will determine whether a second order moment has to be taken into account. For a detailed explanation reference is made to the Concrete Course Document.*
*Detailed calculation including formulas and code references can be however displayed in Report preview if you change the output type in Properties window to **Detailed** and click on Preview action button:*



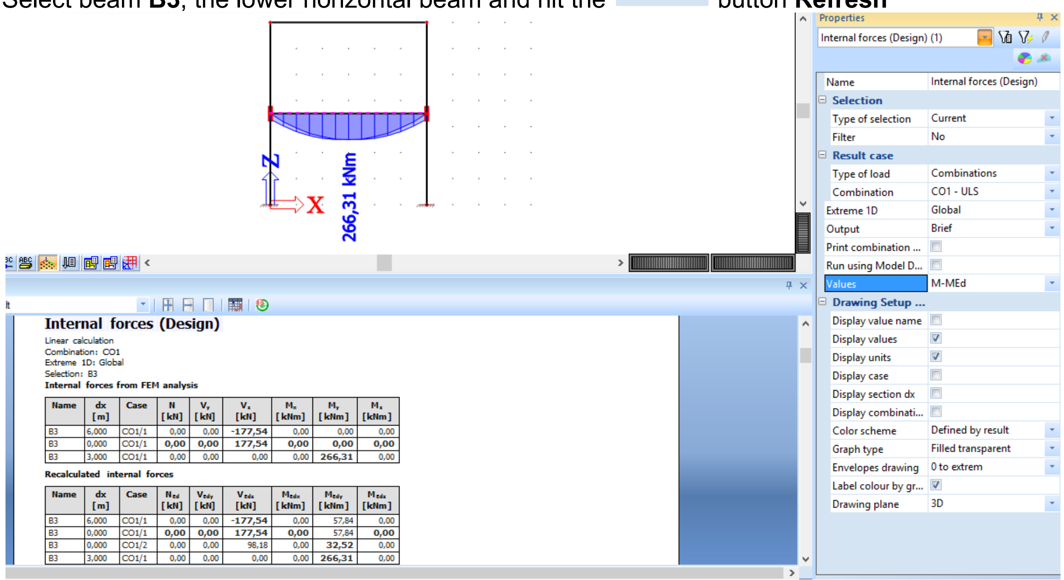
Internal forces for reinforcement design

Internal forces in service Results are purely values obtained by finite element analysis. For concrete design we need to recalculate these forces into design values, using various code regulations. Although this command is identically named in service Results and Concrete 15, here we show both basic and recalculated values that will be used to get required reinforcement areas.

1. Click the Internal forces in the **concrete 15** menu under chapter Reinforcement design – 1D members



5. The options in the **Properties window** are configured in the following way:
   - **Type of selection** field is set to **Current**.
   - **Type of load** is set to **Combinations** and Combination to **CO1 - ULS**.
   - **Values** are set to **M-MEd**, i.e. comparison of basic and design bending moment diagram.
   - **Extreme 1D** field is modified to **Global**.
6. Select beam **B3**, the lower horizontal beam and hit the button **Refresh**



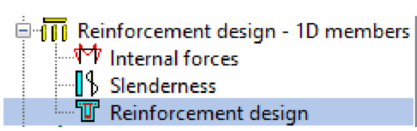
7. The brief output in **Report preview** shows in two tables the difference between internal forces from Fem analysis and recalculated internal forces.

Theoretically required reinforcement

The reinforcement design will only be illustrated for the beam B3. Obviously, the reinforcement calculation for all the other members is identical.
For the background of this calculation we refer to the Concrete Course Document.

Longitudinal reinforcement As

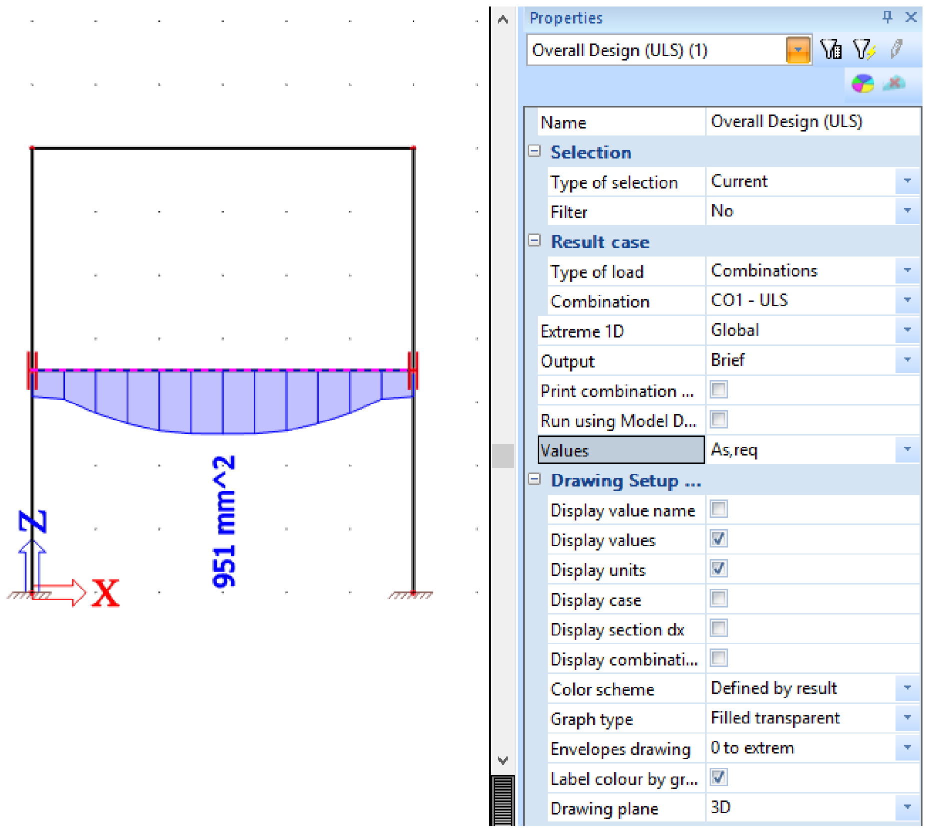
1. In the **Concrete 15** menu, go to **Reinforcement design** under chapter Reinforcement design – 1D members.



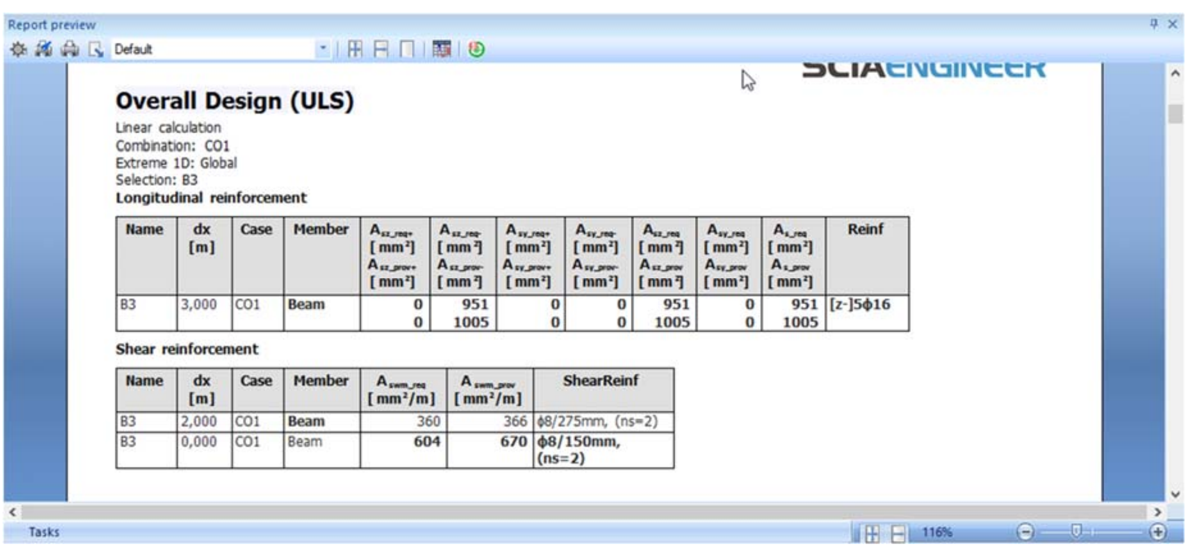
2. The options in the **Property window** are configured in the following way:

- **Type of selection** field is set to **Current**.
- **Type of loads** is set to **Combinations** and Combination to **CO1 - ULS**.
- **Values** are wanted for **As,req**.
- **Extreme** field is changed to **Global**.

3. Select beam B3 with the left mouse button Click the >>> button **Refresh** to display the results on the graphical screen in accordance with the options above.



4. Click the >>> button next to **Preview** to display the results on the table form in accordance with the options above.



Overall Design (ULS)

Linear calculation
Combination: CO1
Extreme 1D: Global
Selection: B3
Longitudinal reinforcement

Name	dx [m]	Case	Member	A_{sz_req+} [mm²] A_{sz_prov+} [mm²]	A_{sz_req-} [mm²] A_{sz_prov-} [mm²]	A_{sy_req+} [mm²] A_{sy_prov+} [mm²]	A_{sy_req-} [mm²] A_{sy_prov-} [mm²]	A_{sz_req} [mm²] A_{sz_prov} [mm²]	A_{sy_req} [mm²] A_{sy_prov} [mm²]	A_{s_req} [mm²] A_{s_prov} [mm²]	Reinf
B3	3,000	CO1	Beam	0 0	951 1005	0 0	0 0	951 1005	0 0	951 1005	[z-]5ϕ16

Shear reinforcement

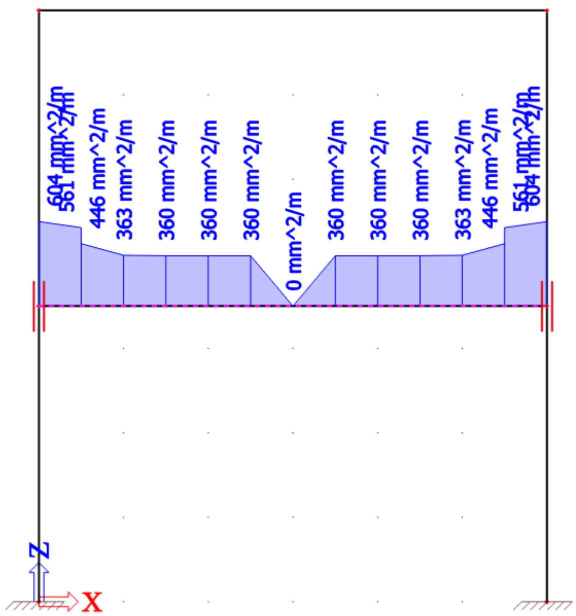
Name	dx [m]	Case	Member	A_{swm_req} [mm²/m]	A_{swm_prov} [mm²/m]	ShearReinf
B3	2,000	CO1	Beam	360	366	ϕ8/275mm, (ns=2)
B3	0,000	CO1	Beam	604	670	ϕ8/150mm, (ns=2)

Note:

- *Detailed analysis, pictures and explanation of all calculation steps can be seen if the Output type is changed to **Detailed**. Instead of a table with a few rows (**Brief** output) this can be extended to many pages.*
- *Alternatively – compromise solution is to display **Standard** output which summarizes the design on one page A4 approximately.*

Shear reinforcement Aswm,req

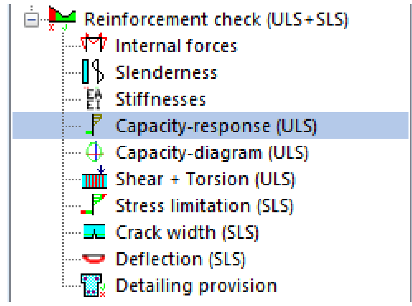
1. Now change the **Values** field to **Aswm,req** to change the diagram in graphical screen so that it displays required shear reinforcement.
2. Set **Extreme 1D** to **Section**.
3. Click the >>> button **Refresh** to display the results on the graphical screen in accordance with the options above. Optionally, click the Preview action button to show text output too.



Reinforcement check (ULS)

Once you know the required reinforcement amount you can start modelling so called practical reinforcement. This means that every beam and column gets graphically clear representation of longitudinal bars and stirrups along whole length of the member. This procedure will be described later in this tutorial.
In case that knowing reinforcement layout in cross-section at particular sections along the member is sufficient, there is no need to use practical reinforcement. Instead, Section check tool can be used to design and immediate check of cross-section in particular position of the beam or column.
In this tutorial we will demonstrate this tool by checking capacity-response diagram on one beam and checking interaction diagram on one column.

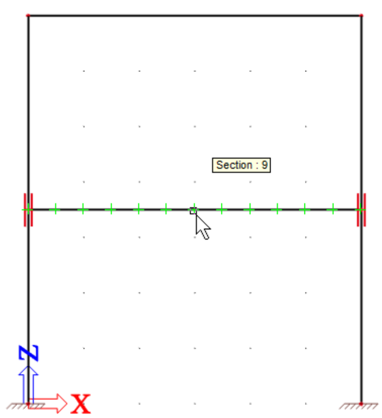
Section check – response check



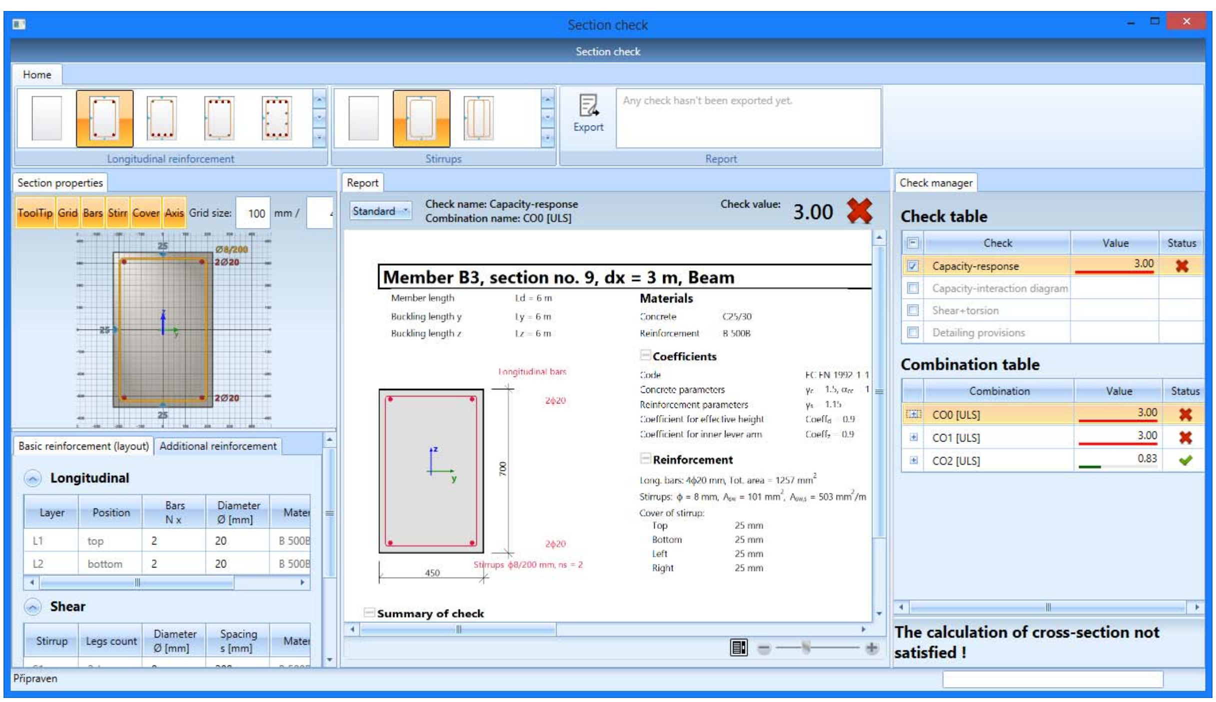
1. Select **Capacity-response (ULS)** command in the **Concrete 15** menu under chapter Reinforcement check (ULS+SLS).

2. Adapt the setting in Properties window:
   - **Type of load** change to **Combinations**
   - **Combination** set to **CO1 - ULS**

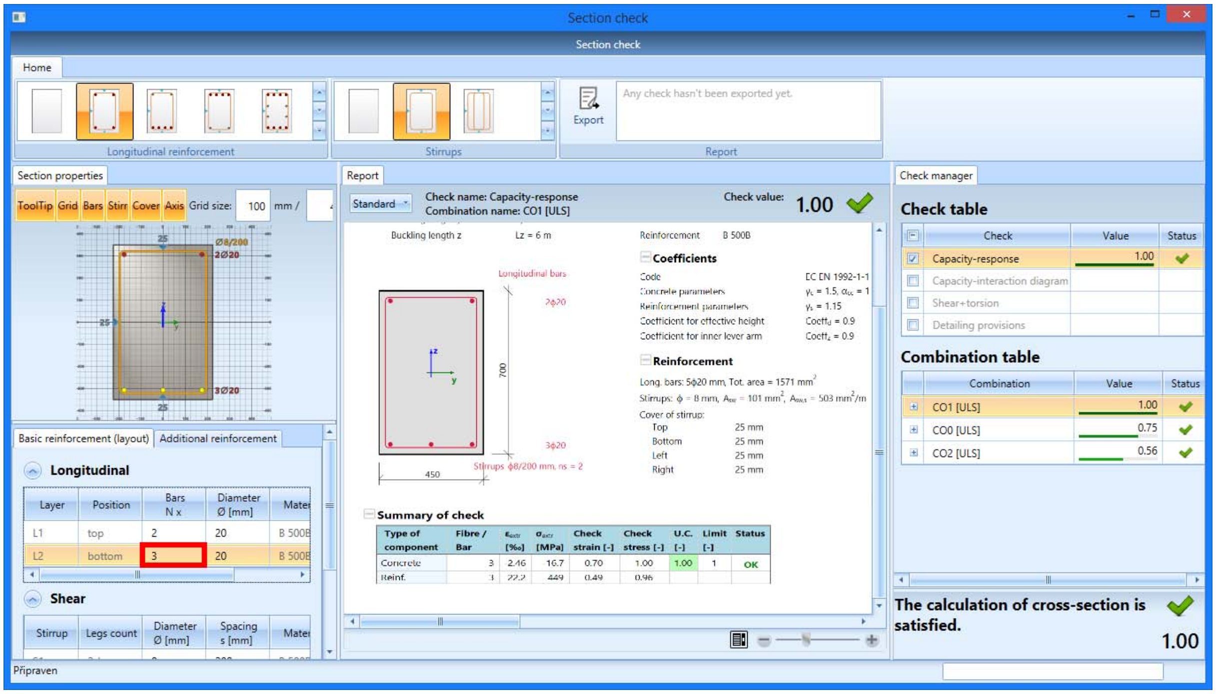
3. Click on >>> button next to **Section check** in Actions and select beam B3 with mouse cursor. At this moment a number of green crosses appear on the selected member, representing various sections along the beam where the check can be performed. Click on the middle one, **Section: 9**.



4. A new dialogue **Section check** opens. Here you can define reinforcement template in that particular section, edit diameter and number of bars and directly see the unity check value. Number 3,00 with the red cross means that the current reinforcement does not satisfy.

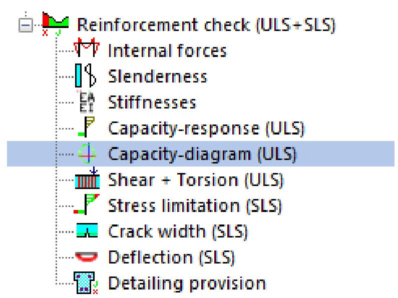


5. Change number of bars for longitudinal reinforcement at bottom layer to 3, press **<Enter>** and you immediately see the recalculated value of unity check. With this new reinforcement the check satisfies for all linear combinations from the code envelope CO1.



6. Close the **Section check** dialogue with red cross icon at top right corner.

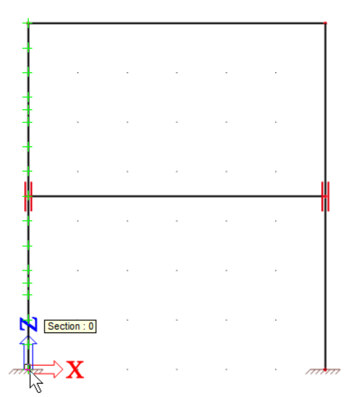
Section check – capacity diagram



7. Select **Capacity-diagram (ULS)** command in the **Concrete 15** menu under chapter Reinforcement check (ULS+SLS).

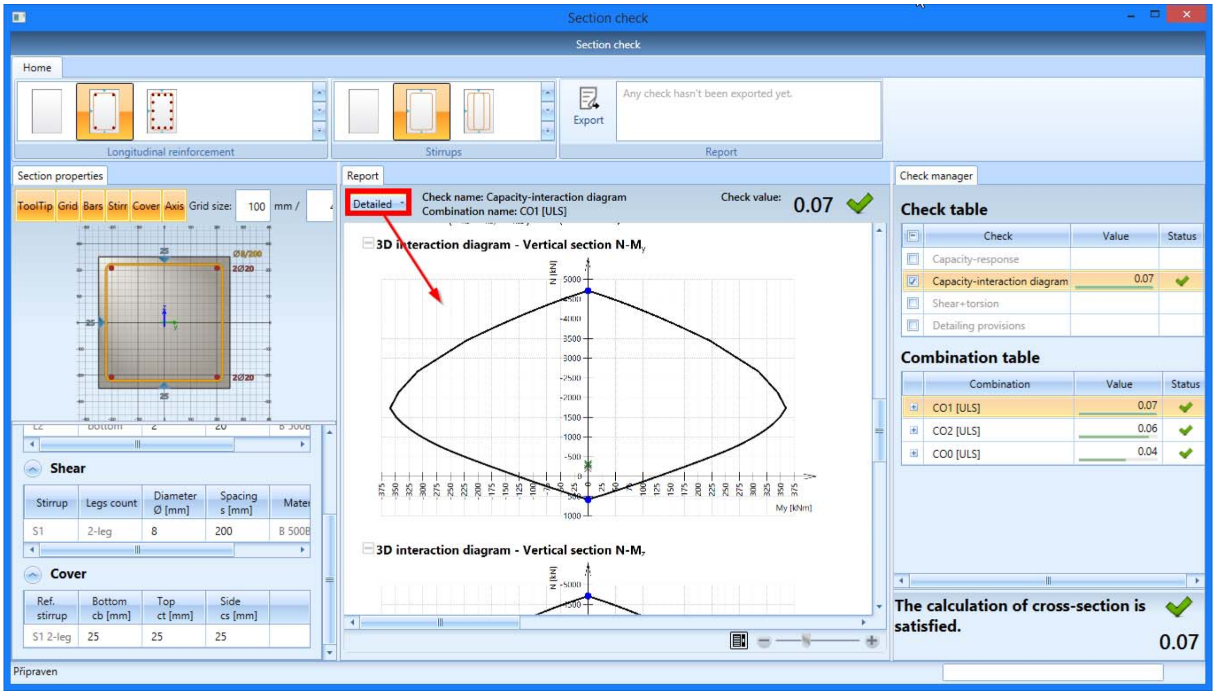
8. Adapt the setting in Properties window:
   - **Type of load** change to **Combinations**
   - **Combination** set to **CO1 - ULS**

9. Click on button next to **Section check** in Actions and select column B1 with mouse cursor. At this moment a set of green crosses appear on the selected member, representing various sections along the beam where the check can be performed. Click on the middle one, **Section: 0**.



10. The dialogue **Section check** opens. It clearly shows that the column satisfies with the current reinforcement.

11. Switch the report type from **Standard** to **Detailed** and scroll-down the page in the middle of the dialogue to see the N-M interaction diagram.



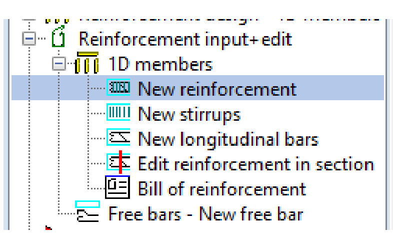
12. Close the **Section check** dialogue with red cross icon at top right corner.

Practical reinforcement

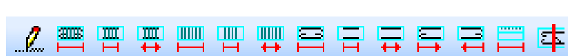
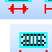
Previous chapter can give you an insight on the reinforcement amount you need to input in particular beam of column. You may need to represent it graphically to make bill of reinforcement or detailing drawings.
But you can also input practical reinforcement without previous design and do the checks directly, to see if the provided reinforcement area is sufficient.

1. Press **<Esc>** to cancel any the selection.

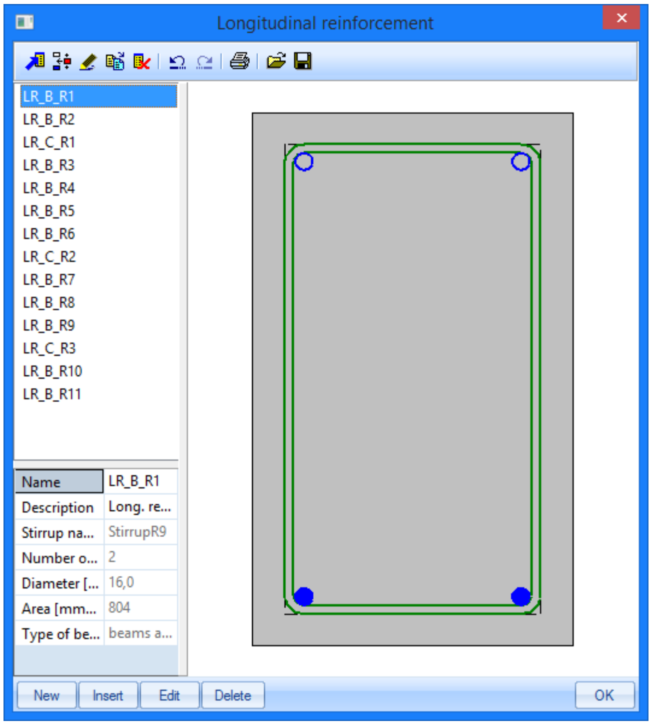
2. To enter practical reinforcement you can open the group **Reinforcement input+edit** in **Concrete 15** menu and select the item **New reinforcement**.



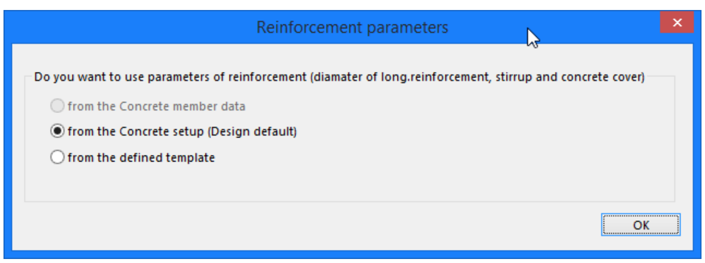
3. Select beam **B3** with the left mouse button. You will be directly asked (as can be seen in the Command line) to select the first point of the reinforcement zone and the last point. Click on node N5 and node N6 consecutively.

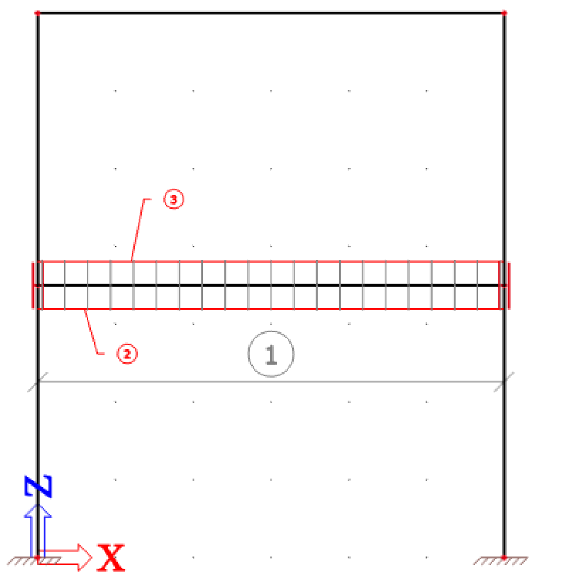
Alternatively, you can use the shortcuts  in the **Command line**. Select the first one **Add reinforcement on whole beam** .

4. The **Longitudinal reinforcement** manager will appear. Choose the first template (LR_B_R1) from the list and click OK.



5. **Reinforcement parameters** dialogue now appears, just before the reinforcement is inputted. Select the middle option "from the Concrete setup (Design default)" to specify the diameters of bars and stirrups according to concrete design defaults and hit **OK** button.



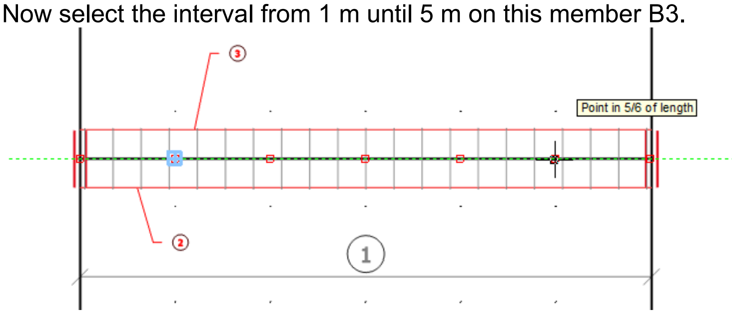


6. We have calculated that some additional reinforcement was required in the middle of this beam. Therefore we will add a new bar of reinforcement in the zone from 1 m to 5 m.

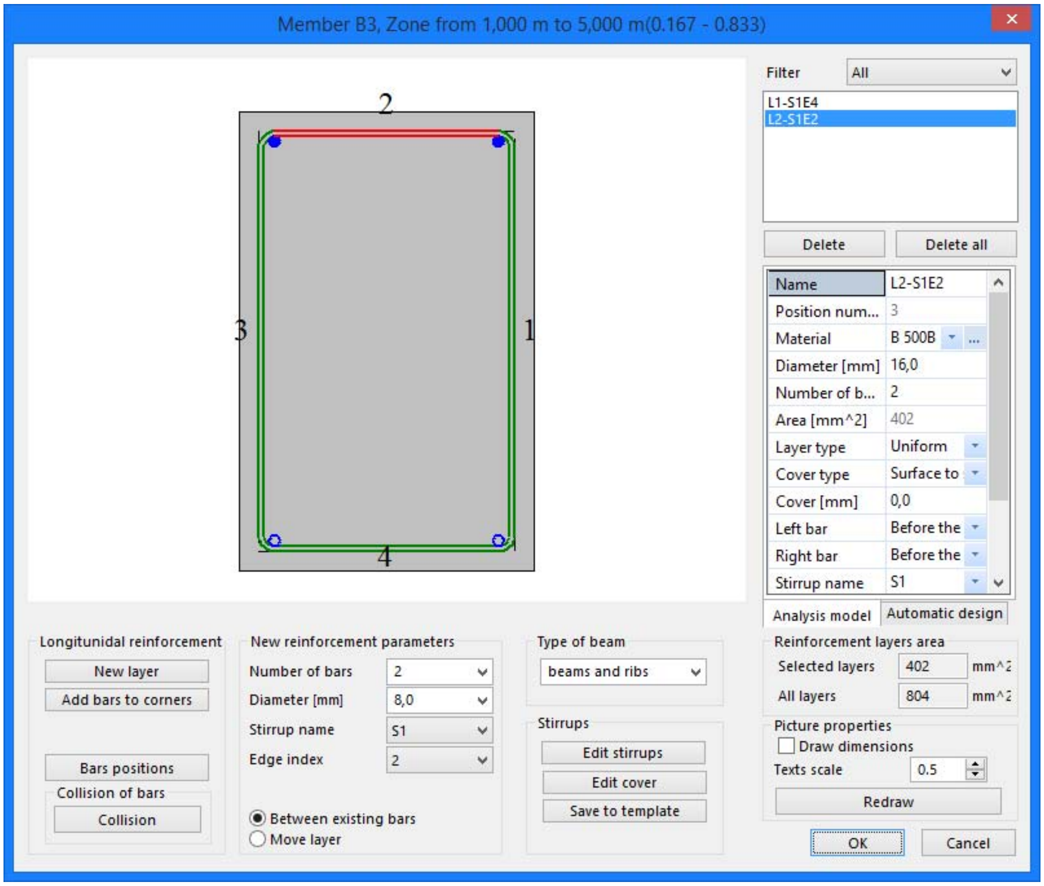
   To enter a layer of reinforcement on a selected interval you could either use command New longitudinal bars which can be found in the **Concrete 15** menu under or the shortcut in the command line to add longitudinal reinforcement on selected interval .

   When you start the command to add longitudinal reinforcement on a selected interval, you first have to select the beam B3.

   After the beam is selected you have to define the interval.
   To define the interval it can be helpful to use snap points. Therefore select the cursor snap setting in the command line .
   In the cursor snap setting you activate option h) to create six snap points on each beam.
   On beam B3 you will have now one snap point every one meter.
   Now select the interval from 1 m until 5 m on this member B3.



7. The next screen shows us the reinforcement layers in the zone from 1 m to 5 m of member B3.



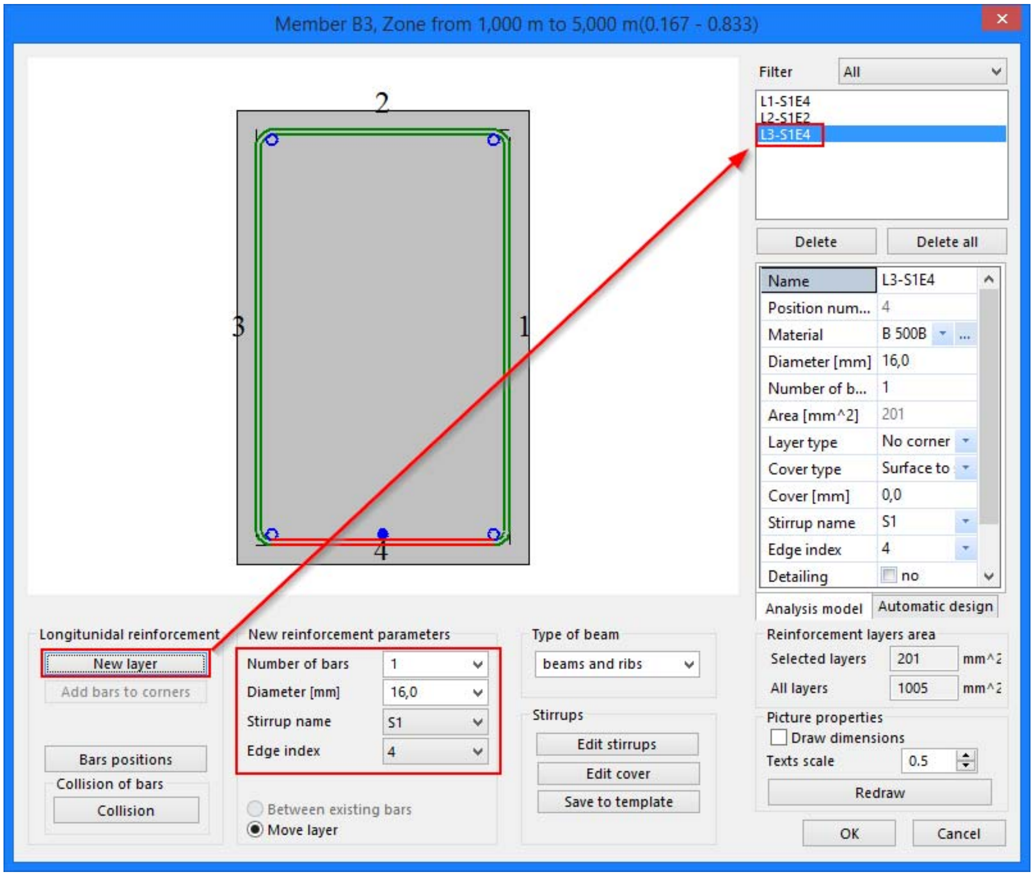
Here you can create new layers by using the icons in the left bottom corner.

Create a new layer of 1 bar of 16 mm in the middle of the lower edge.

To do this you set the new reinforcement parameters:

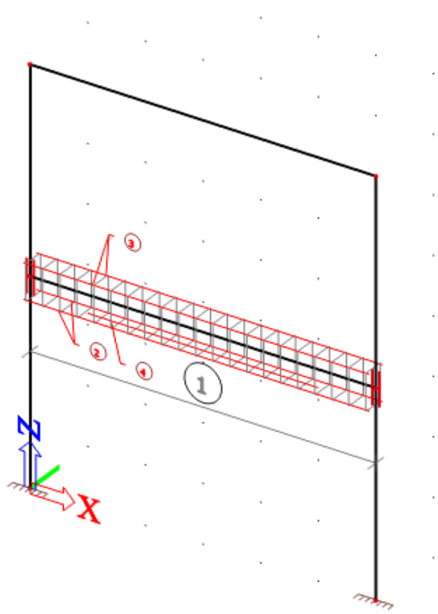
- Number of bars: 1
- Diameter (mm) : 16
- Stirrup name S1
- Edge index : 4

If these new reinforcement parameters are set, click on  button.



8. Confirm the input by **OK** button. Confirm reinforcement parameters “from the Concrete setup (Design defaults)” in the next dialogue by **OK** button again.

You can choose axonometric view by button  in View toolbar to see the new reinforcement bar



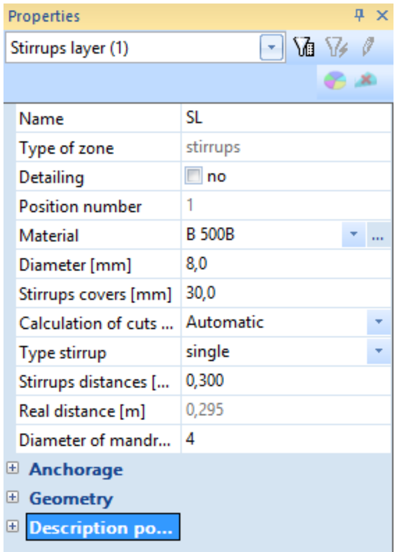
After the input of practical reinforcement layers, these layers are additional data on beam. They can be copied easily to other beams.

Properties of a reinforcement layer can be viewed and adapted in the Properties window.

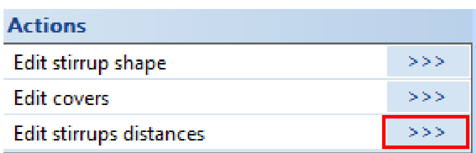
9. We will now adapt the layer which contains the stirrups to change the stirrup distances.

a) Select the **Reinforcement layer** of the stirrups by clicking with the left mouse button on the circled digit 1  above grey dimension line below the beam.

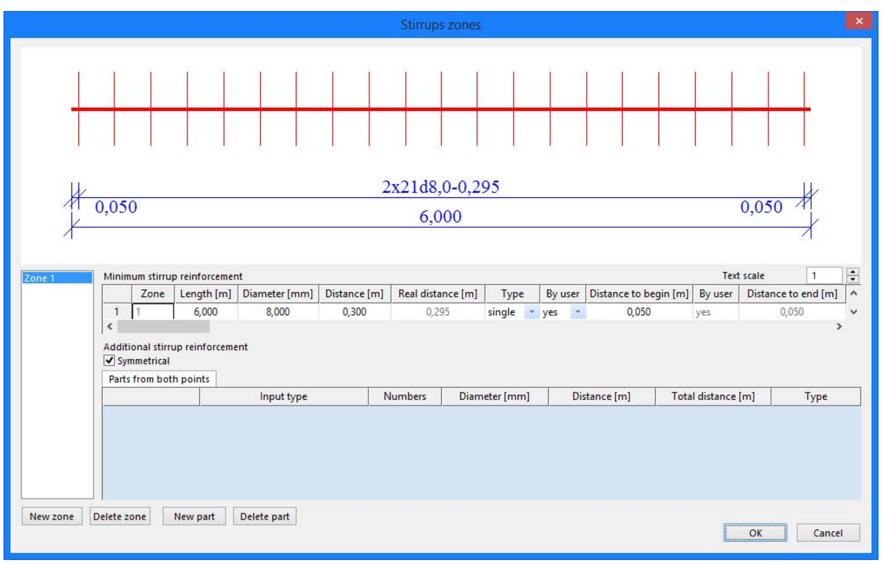
b) Activate the properties of this **Reinforcement layer** in the **Properties Window** by selecting this entity with the mouse pointer in the combo-box:



c) Click the Action button **Edit stirrup distances** at bottom right corner



d) The window **Stirrup zones** appears

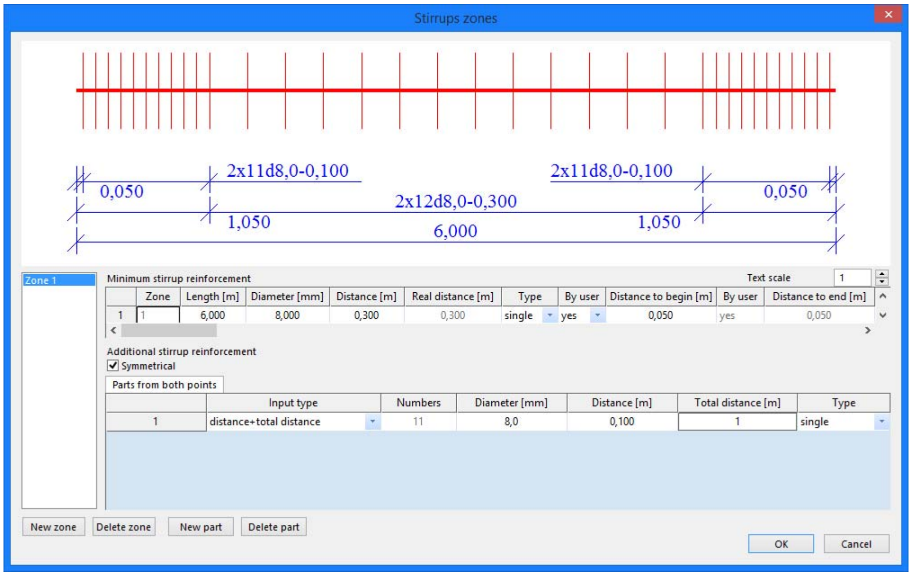


e) Click on the **[New Part]** button to enter a new part.

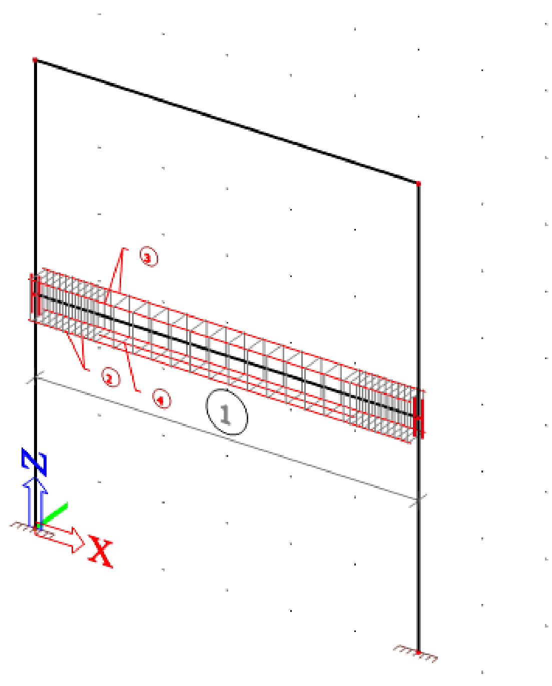
f) Change the Input type to **distance + total distance.**

g) Change the **Distance [m]** field for this part from **0.300** into **0.100** and take **1** as **Total distance [m]**.

Note that the stirrups are put closer because the parts are automatically defined symmetrically from both ends.

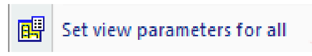


h) Click **[OK]** to confirm your input.



10. The graphical representation of the practical reinforcement can be changed so that a 3D view of the reinforcement for so far can be obtained. To do this, follow the steps described below:

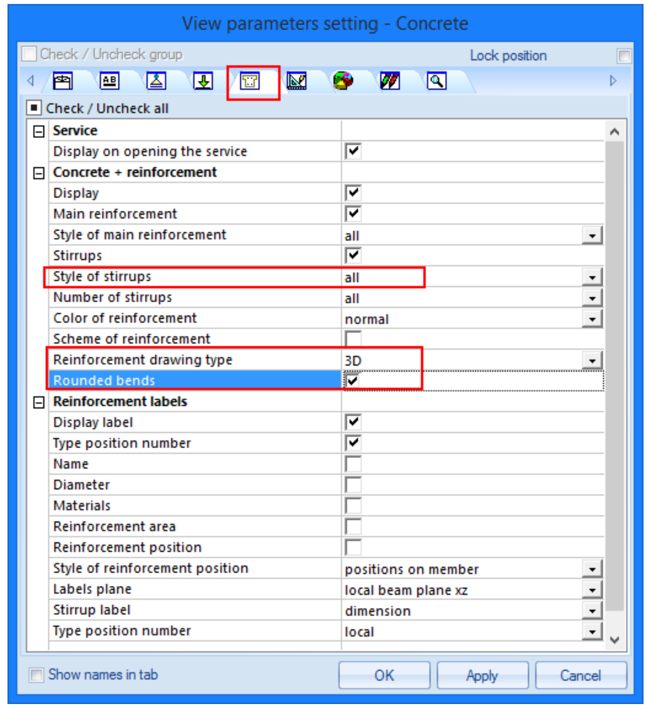
a) Right-click on an arbitrary position in the workspace.

b) In the context menu that is opened, select the 

option. The window **View parameter setting** appears.

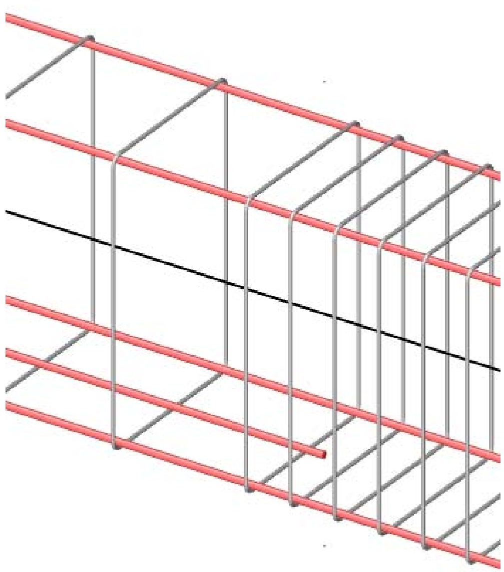
On the **Concrete** tab, the reinforcement data are displayed under the heading **Concrete + Reinforcement**. Change the options in the following way:

- Set the option **Style of stirrups** to **All**.
- The option **Reinforcement drawing type** is set to **3D**
- The option **Rounded bends** is set ticked on.

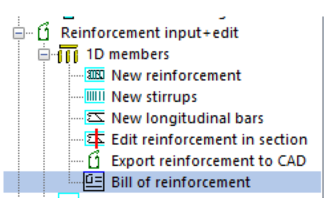


c) Confirm your input by **[OK]**.

Below is shown a picture of a part of the practical reinforcement.



11. It is also possible to show a bill of reinforcement which contains a list of all practical reinforcement in the project. The command can be found in the **Concrete 15** menu in chapter **Reinforcement input+edit**.



Report preview, opened by action button

, is shown below.

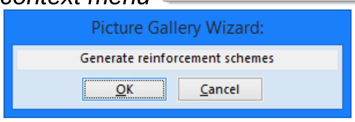
SCIAENGINEER

Bill of reinforcement

Selection : All
The lengths of longitudinal reinforcement bars and stirrups are calculated without rounded bends.
Type of position number : Global

Member	#	Ø [mm]	Material	Length [m]	Number of bars	B 500B length [m]	B 500B weight [kg]
B3	1	8	B 500B	2,900	34	98,600	38,9
B3	2	16	B 500B	6,000	2	12,000	18,9
B3	3	16	B 500B	6,000	2	12,000	18,9
B3	4	16	B 500B	4,000	1	4,000	6,3
		8	Total for diameter			98,600	38,9
		16	Total for diameter			28,000	44,2
			Total for material			126,600	83,1
			Total			126,600	83,1

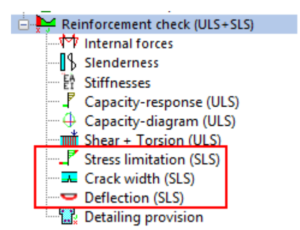
Note
Automatic picture generator creates individual picture with reinforcement scheme for every single beam or column. User right-mouse-click in the modelling window, click option Picture wizard at the bottom of context menu Picture wizard *and then click [OK] to generate reinforcement schemes.*



Reinforcement check (SLS)

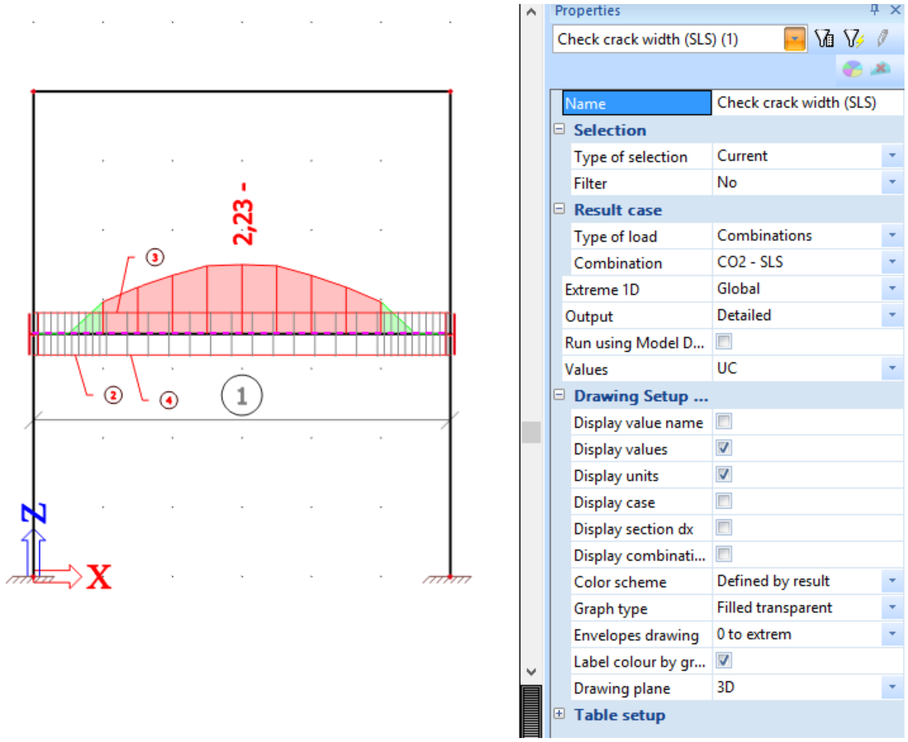
There are several more checks in SCIA Engineer that controls also the serviceability limit state:

- Stress limitation
- Crack width
- Deflection



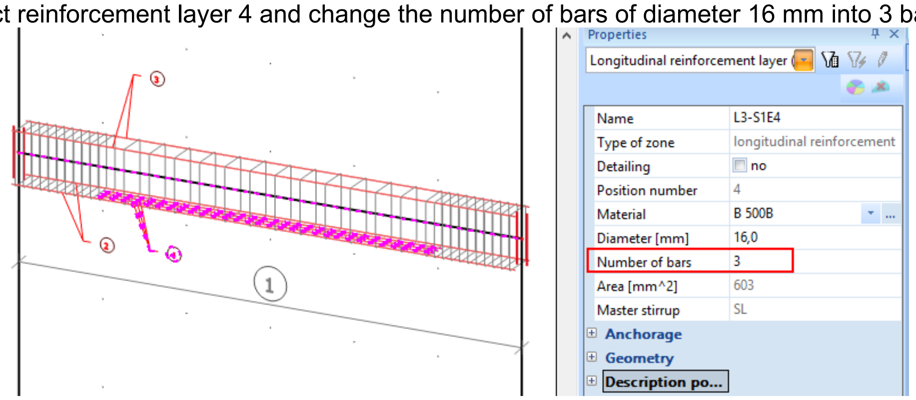
Crack control

1. In the **Concrete 15** menu select **Crack width (SLS)** check function.
2. Press **<Esc>** to cancel the selection.
3. Options in **Properties window** are configured in the following way:
   - **Type of selection** is set to **Current**.
   - **Type of loads** is changed to **Combinations** and second combination named **CO2 - SLS** is chosen. Program automatically filters only combinations for serviceability limit state here.
   - **Values** are set to **UC** (meaning Unity Check) .
4. Select horizontal beam **B3** with the left mouse button.
5. Click the >>> next to **Refresh**, highlighted by red background. The crack control is carried out and the unity check is then displayed on the screen.

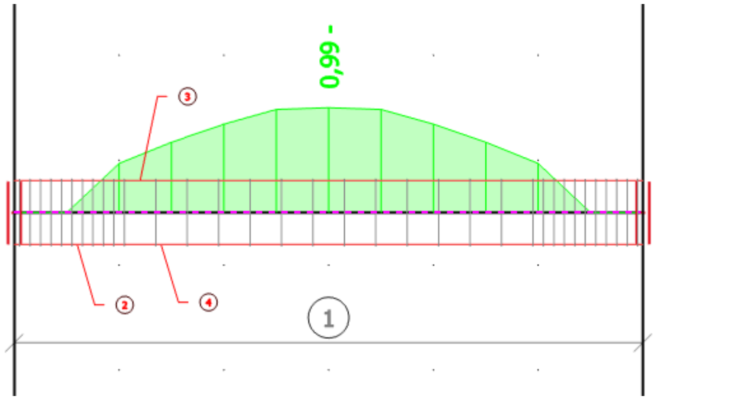


Note
In Report preview you can open Brief, Standard or Detailed output to see the calculation process that gives this unity check value and stress and strain of uncracked and cracked section pictures.

6. It is clear from this result that the currently provided reinforcement does not satisfy the crack control check. Therefore we will make some adaptations to the practical reinforcement.
   Select reinforcement layer 4 and change the number of bars of diameter 16 mm into 3 bars.



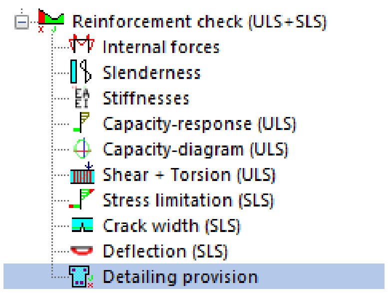
7. If we repeat the steps 1 to 5 we can check again if crack control is satisfied.



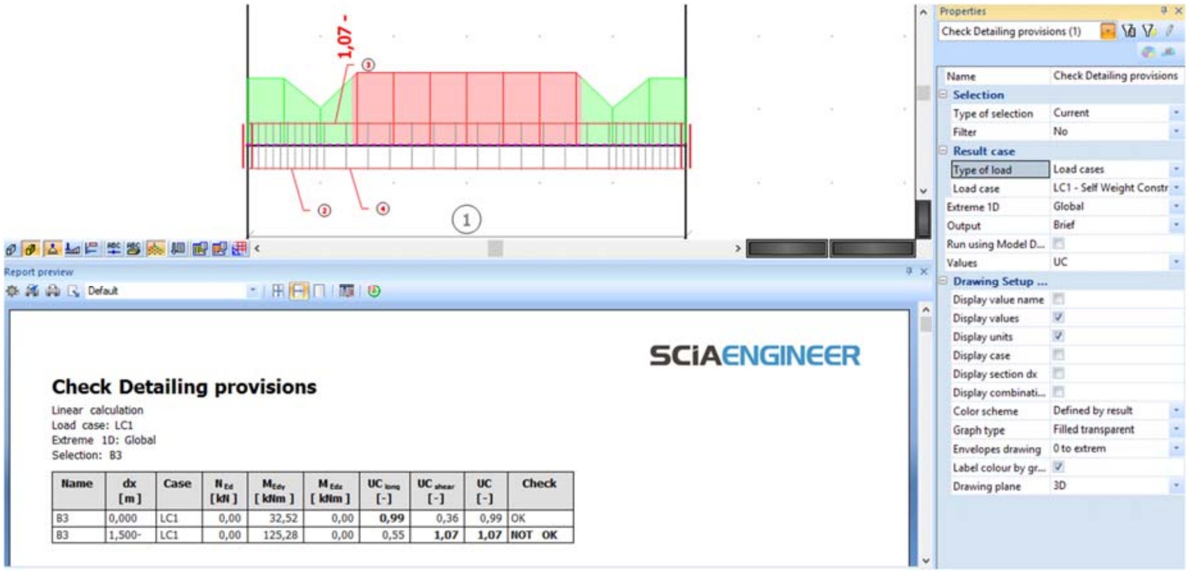
It is clear that now the crack control is satisfied because we have reinforcement bars with shorter distance between them.

Detailing provisions

1. In **Concrete 15** menu, choose the command **Detailing provisions**.



2. **Properties window** is configured in the following way:
   - **Selection** field is set to **Current.**
   - **Type of load** is irrelevant for the check of detailing provisions.
   - **Output** define as **Brief**.
   - **Values** set to **UC** (meaning Unity Check).
3. Select floor beam **B3** with the left mouse button if not selected from previous check.
4. Click the >>> next to **Refresh** and afterwards **Preview** in Actions**.**



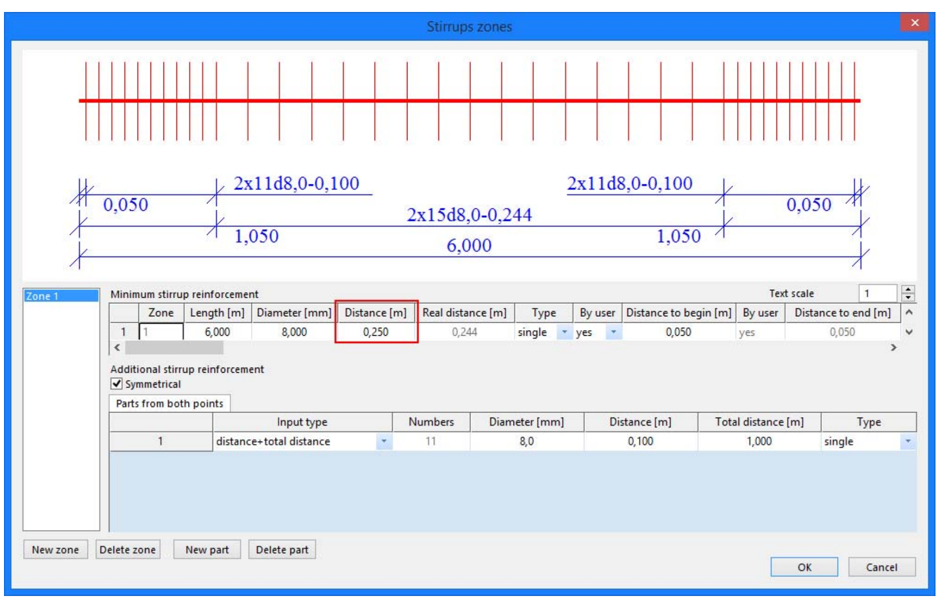
5. Shear reinforcement check shows value higher than 1,00 at particular span. We need to shorten the distance of stirrups to satisfy the detailing provisions.

a) Select the **Reinforcement layer** of the stirrups by clicking with the left mouse button on the circled digit 1  above grey dimension line below the beam.

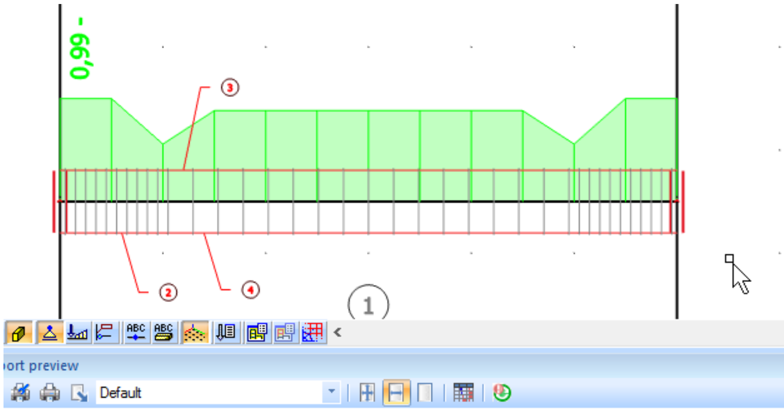
b) Activate the properties of this **Reinforcement layer** in the **Properties Window** by selecting this entity with the mouse pointer in the combo-box:

c) Click the Action button **Edit stirrup distances** at bottom right corner

d) The window **Stirrup zones** appears. Change **Distance (m)** value of the zone 1 from 0,3 m to 0,25 m and hit **[OK]** button.



6. If you repeat the Detailing provisions check you can see al values green with maximum UC = 0,99.



Check Detailing provisions

Linear calculation
Load case: LC1
Extreme 1D: Global
Selection: B3

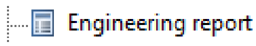
Name	dx [m]	Case	N_{Ed} [kN]	M_{Edy} [kNm]	M_{Edz} [kNm]	UC_{long} [-]	UC_{shear} [-]	UC [-]	Check
B3	1,500-	LC1	0,00	125,28	0,00	0,55	**0,87**	0,87	OK
B3	0,000	LC1	0,00	32,52	0,00	**0,99**	0,36	**0,99**	**OK**

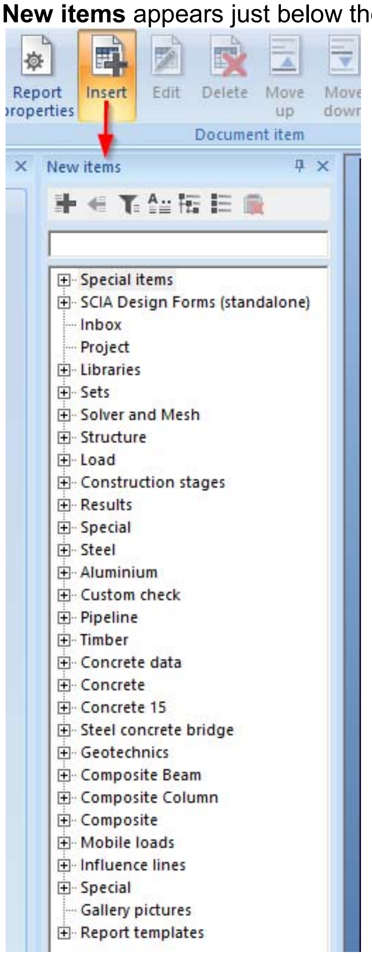
Note
The detailing provisions are checked according to Eurocode and its thorough setting can be viewed in the Concrete setting (structure).

Document

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

Engineering Report

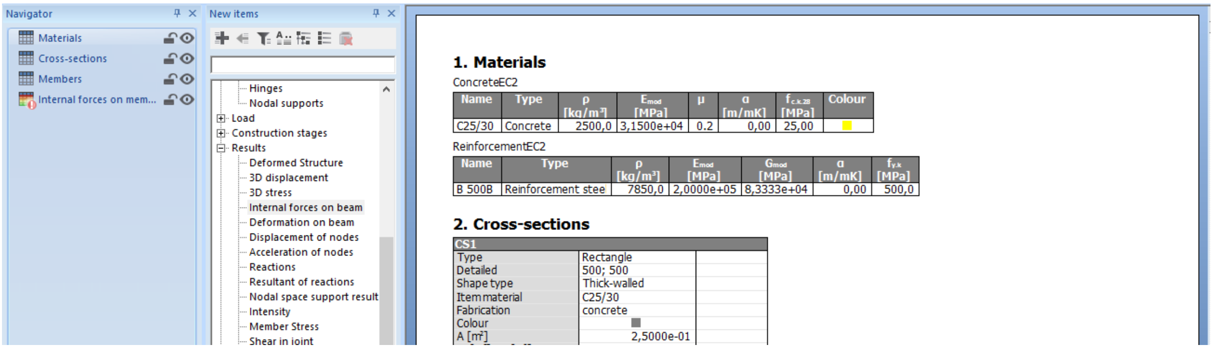
1. Double-click  in the **Main Window** or click  in the toolbar. Because no report was created before, **Report_1** directly appears as a new application. This application is in a certain way independent on the SCIA Engineer application. That is significant also in the Windows main bar


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 **Members.**
   - Open the **Results** group and click **Internal forces on beam.**

4. You can directly see these items in the Navigator and on the paper preview as well:

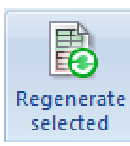


Drag the items with the mouse to change their order.

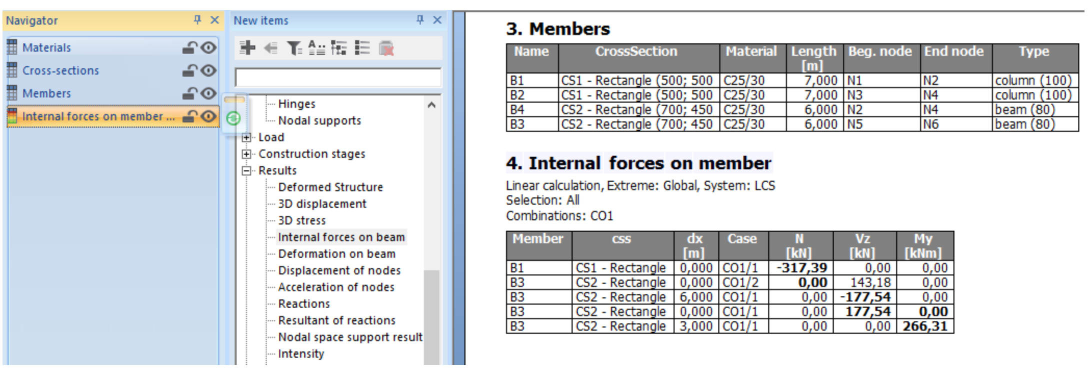
Displaying results in the document

1. In the **Navigator** click **Internal forces on beam**. 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**.
   - **Values** are set to bending moment **My**.
   - **Extreme** field is changed to **Global**.

2. Click the 

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

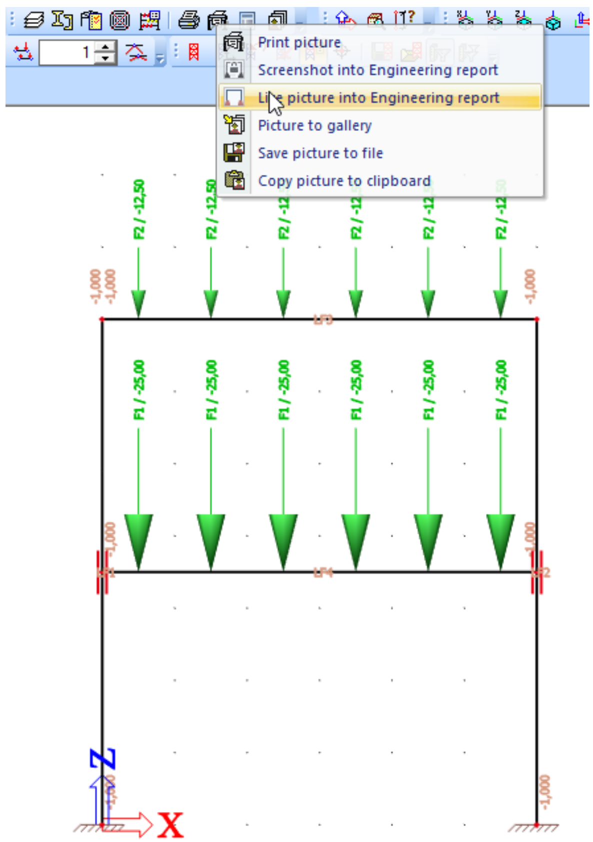


Adding an image to the Report

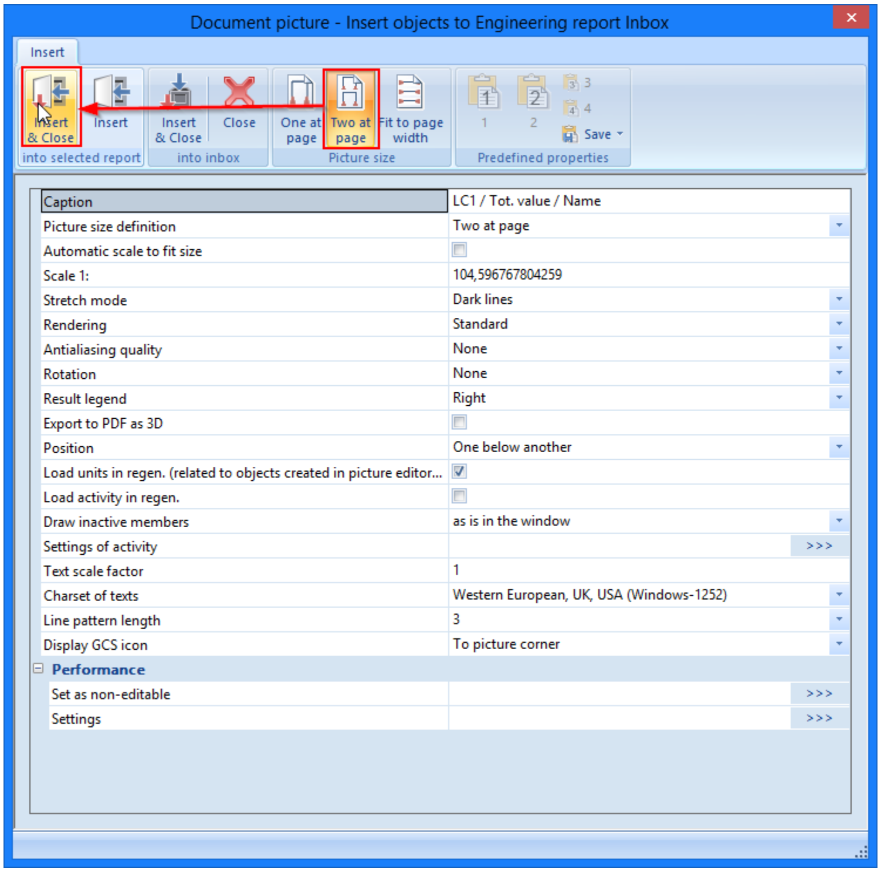
1. Any picture from SCIA Engineer application can be set to Engineering Report. Either as print-screen (that is unchanged for ever) or as live picture (that can be regenerated and is always up-to date).

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:

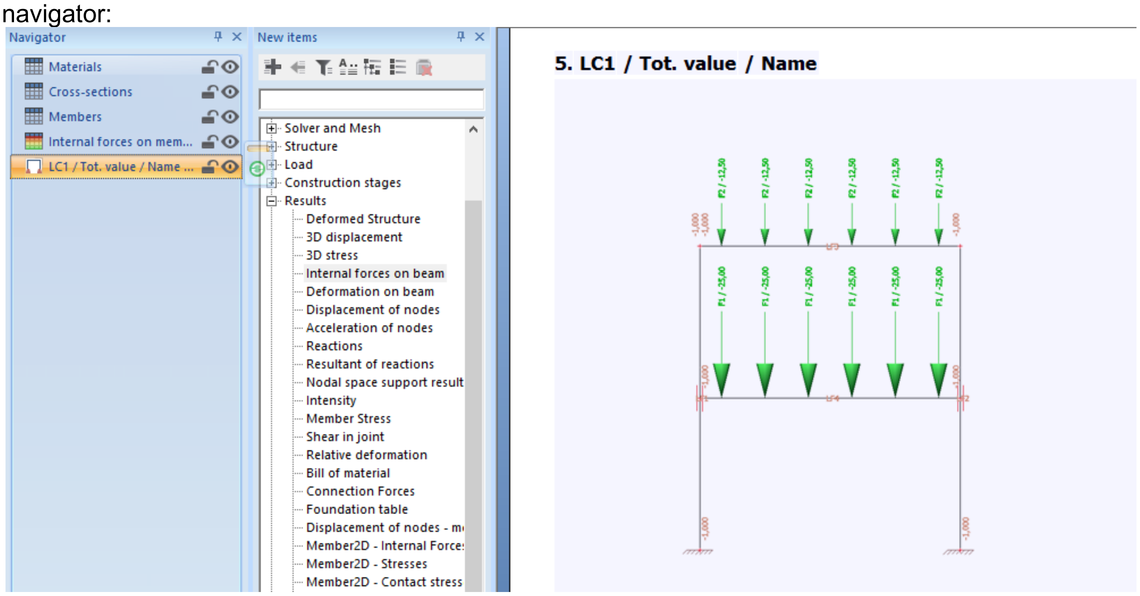

3. Click on button **Print Picture in toolbars** and select **Live picture into Engineering Report**



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

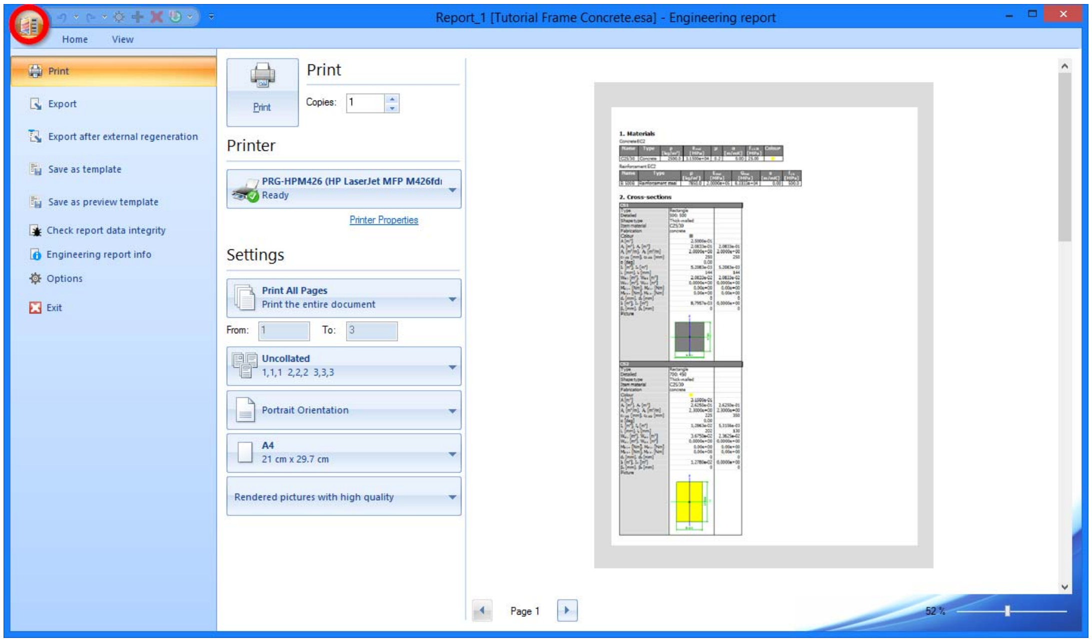


5. Switch to Engineering Report application again and see the picture as the very last item in the navigator:



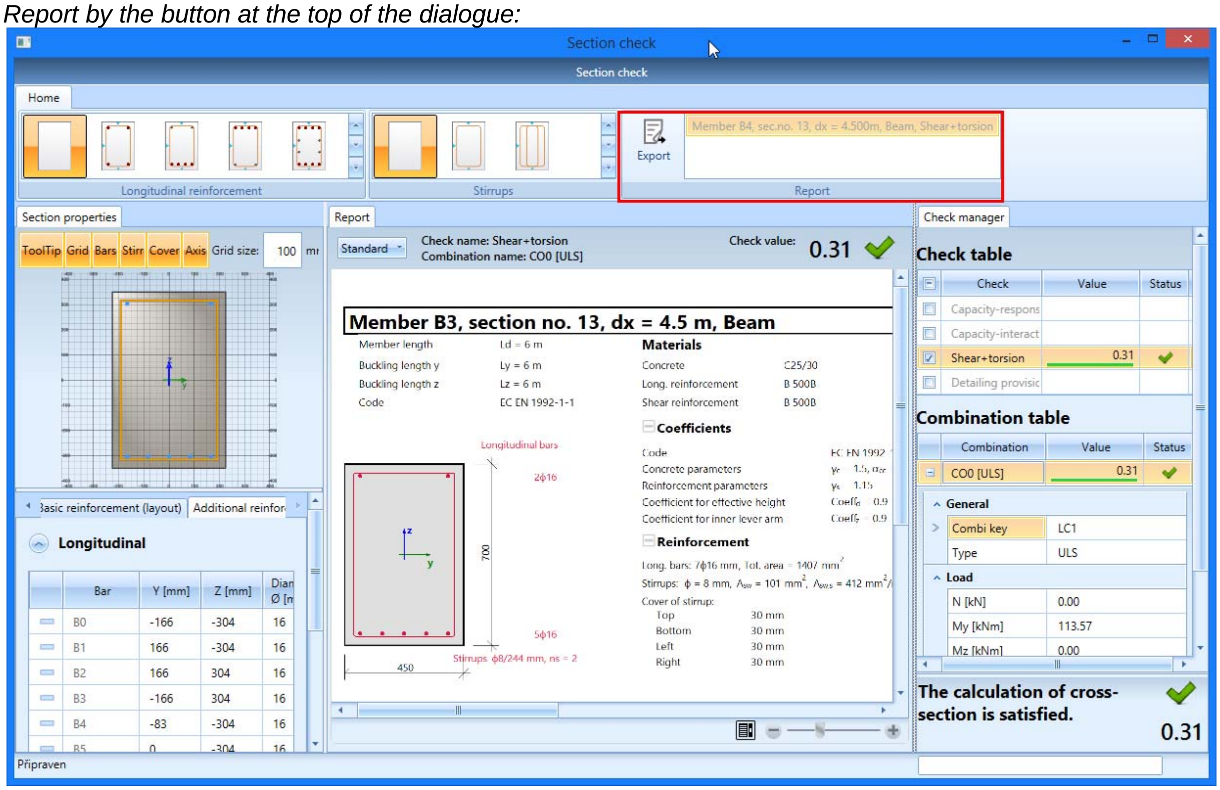
Printing Engineering Report

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



Note:

Any output, standard or deatiled, from the Section check in Concrete 15 can be easily send to Engineering Report by the button at the top of the dialogue:



Epilogue

In this syllabus, the basic functionalities of SCIA Engineer for the input of a concrete structure including the concrete calculation were introduced by means of a simple example.

After reading the text and executing the example the user should be able to model and calculate simple structures consisting of concrete beams and columns.

For more detailed information about concrete calculations we refer to the Advanced Concrete Training documentation or the dedicated Web help chapters.