



# Scia Engineer

## Concrete for Starters

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 2013 Nemetschek Scia nv in cooperation with Joren Severy and Yoshi Vemeiren. All rights reserved.


## Table of Contents

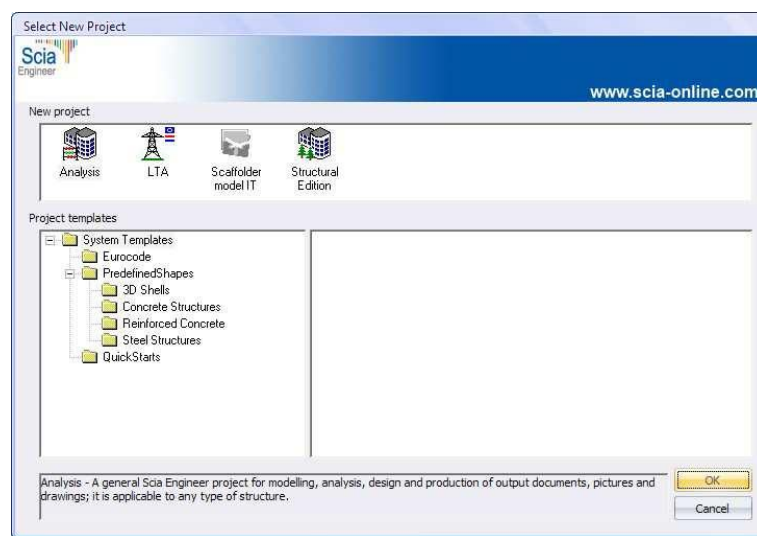
Starting a project.....	4
Defining the construction .....	5
Applying the loads.....	10
Materials library.....	16
Linear calculation.....	17
Results .....	19
Concrete – general.....	23
Concrete – theoretical reinforcement.....	25
Concrete – AMRD.....	27
Concrete – checks .....	31
Concrete – Bill of reinforcement .....	36
Document.....	37

# 1DBeam

## Starting a project

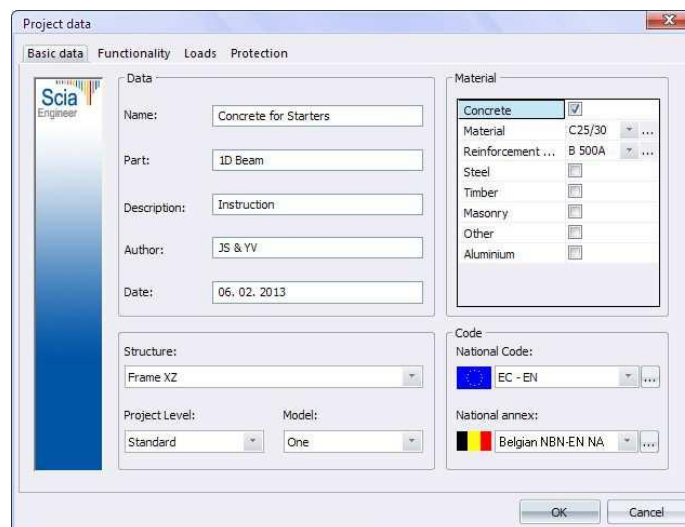
### Starting a new project

To start a new project, one has to click on New , which can be found in the toolbar. A new project will open in a new screen, namely Select New Project. Because calculations will have to be made, choose for the option Analysis.



Next the dialog box Project data will appear.

### Entering the project data



In this dialog box, the general data of the project has to be entered (e.g. name, author, etc.). As construction type, choose for the option Frame XZ, this leads to a

structure of which all nodes are fixed. Consequently, the user will be able to introduce forces on the elements themselves (Truss XZ implies structures with hinged nodes, where forces can only be applied in these nodes).

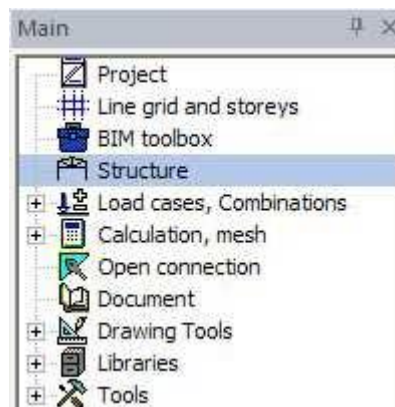
Additionally, the user can further specify the used materials. In this example is opted for Concrete C25/30 and Reinforcement steel, type B 500A.

Finally, the calculations will be executed according to NBN-EN NA. Click on the OK button to confirm the project data.

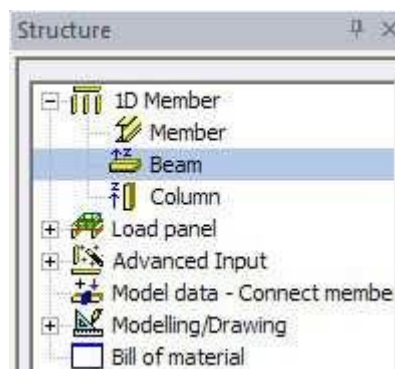
## Defining the construction

### The 1D-beam

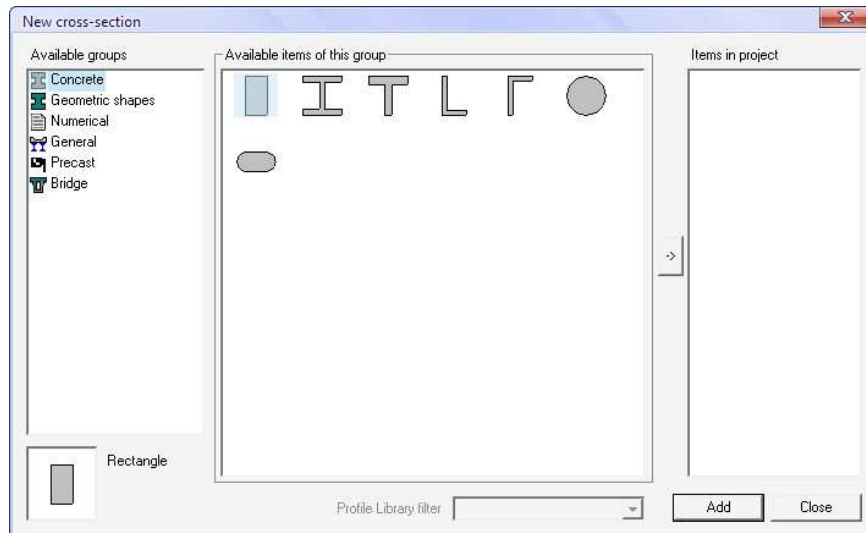
To determine the geometry of the structure, double-click on Structure in the Main menu option.



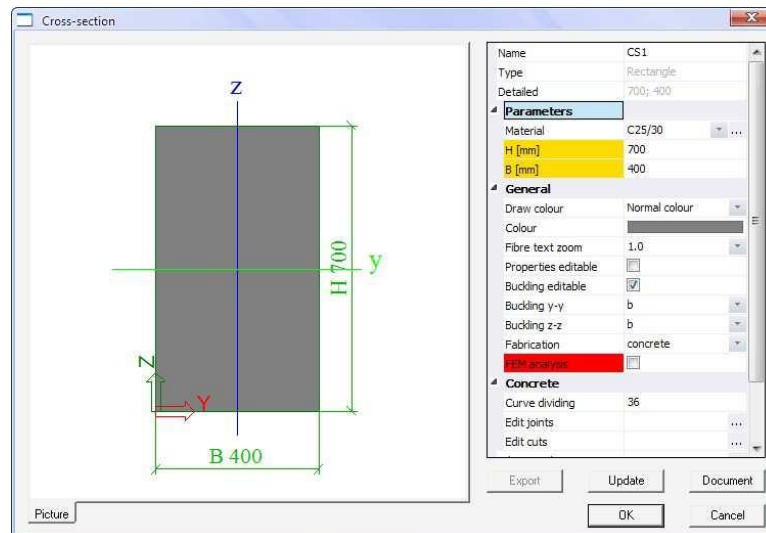
The selected menu will open. In this project, there is chosen for a 1D beam. This can be done through the submenu 1D member > Beam.



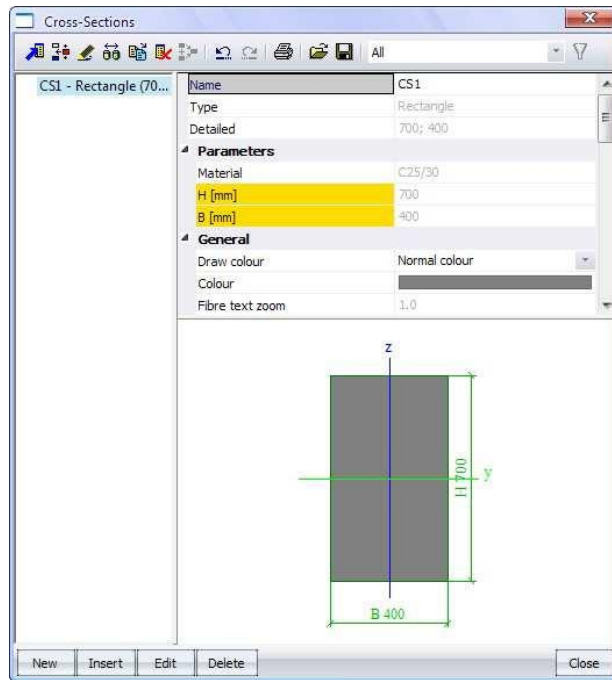
Because there is no section that has been defined yet, the window New cross-section will open up automatically. The beam in this project has a rectangular cross-section. To create this cross-section, one has to select Rectangle and click the Add button.



Now the dimensions of this cross-section have to be entered. Choose the following input parameters:

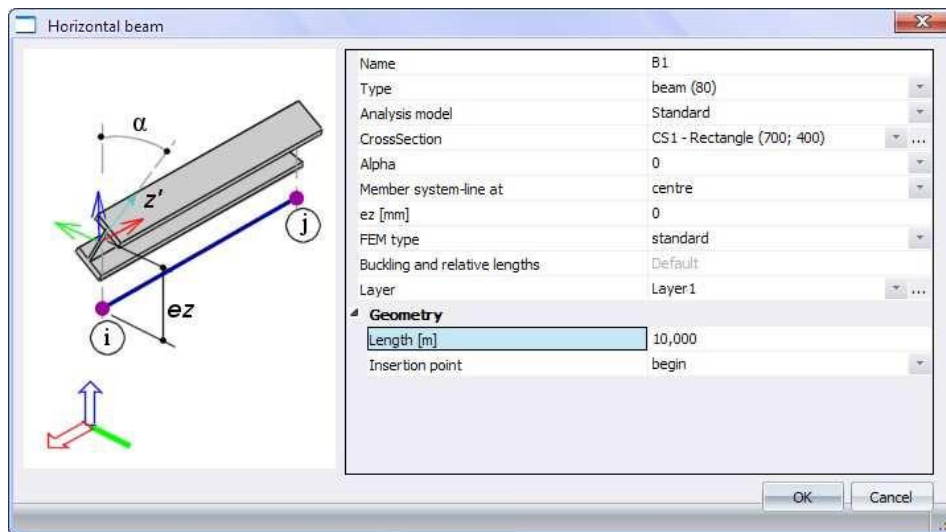


After entering the data, click on the OK button. As a result, this cross-section will become available for the user in the Cross-section window.

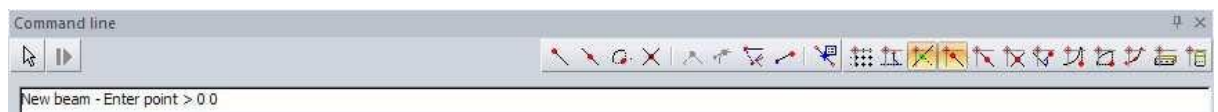


Then click Close.

Now it is possible to adjust the specifications of the beam. In this case, one has to choose for a length of 10 meters.

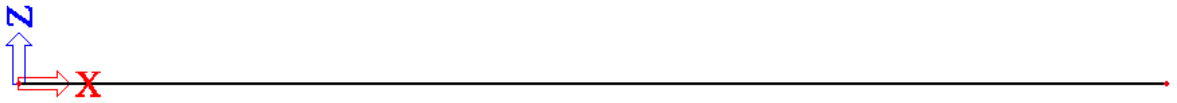



Confirm the input by clicking the OK button. In the command line appears New beam – Enter point. The insertion point here, should be the origin of the GCS (Global Coordinate System), this can be done by entering the x,z-coordinates as follows:



The coordinates can also be entered, separated by a semicolon (;). To complete the input, click the Esc key. Finally, to deselect the beam, click the Esc key again.

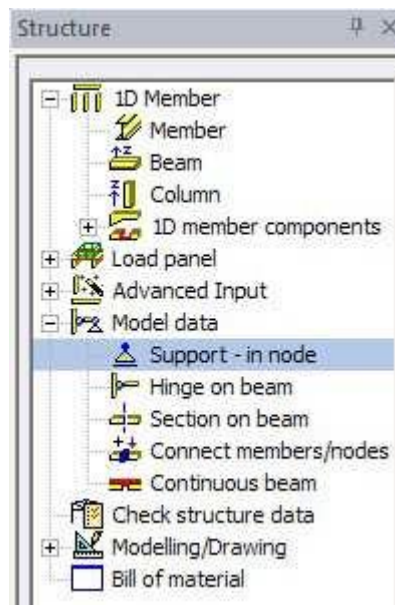
The beam will be displayed as follows:



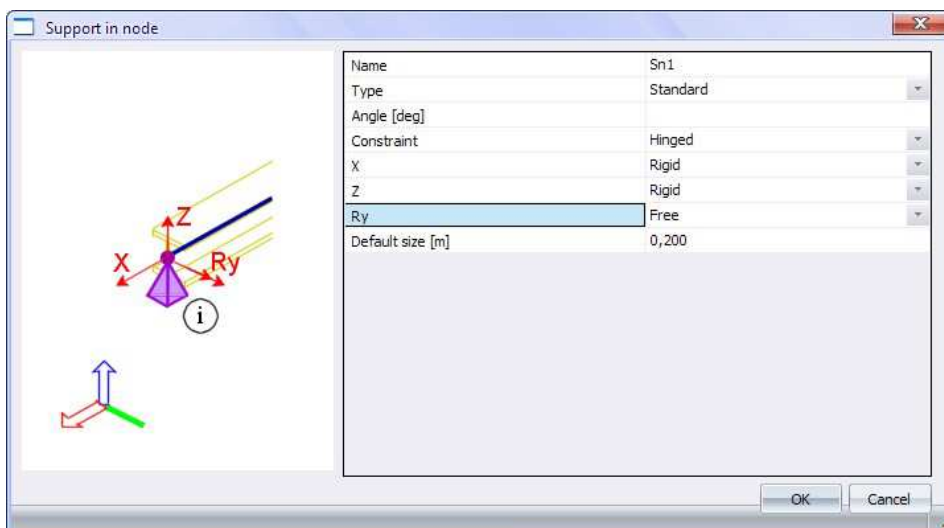
Remark: through the  button, other cross sections can be defined.

### Adding the supports

Now the supports of the structure have to be added. This is done by using the option Model data > Support – in node in the Structure menu that is already active.

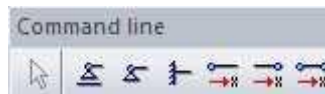


In the dialog box Support in node, one can adjust the details of the supports. At the leftmost point of the beam, the beam is supported by a hinge, so the following entry must be confirmed with the OK button.

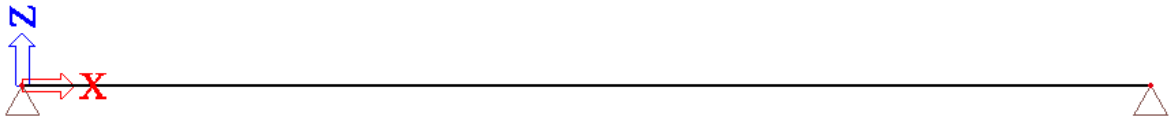




Select the appropriate node to add the previously defined support. Then deselect by double-clicking the Esc key twice. To add a sliding support at the right most point of the beam, use the same method (use other parameters) or click on the icon Sliding support.




Finally, the following output will we obtained:

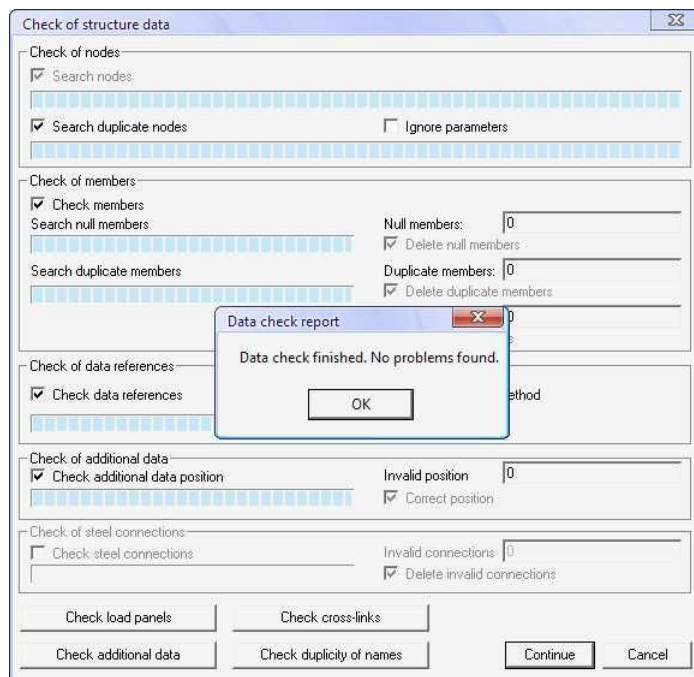



**Remark:** With the buttons, located above the Command line, the representation of the structure can be adjusted.



## Checking the construction

In order to check the construction, one can use the Check structure data button . By clicking the Check button in the dialog box Check of structure data, the construction check will be executed. When there are no problems to be found, the following output will appear:



The structure check can be closed, through the OK button. Next click the Close button in the submenu Construction to return to the Main menu. It is also possible to use the following icon .

## Applying the loads

### Creating the load cases

The loads can always be divided into permanent and imposed loads. In this project there are 3 permanent loads and 1 imposed load, these are:

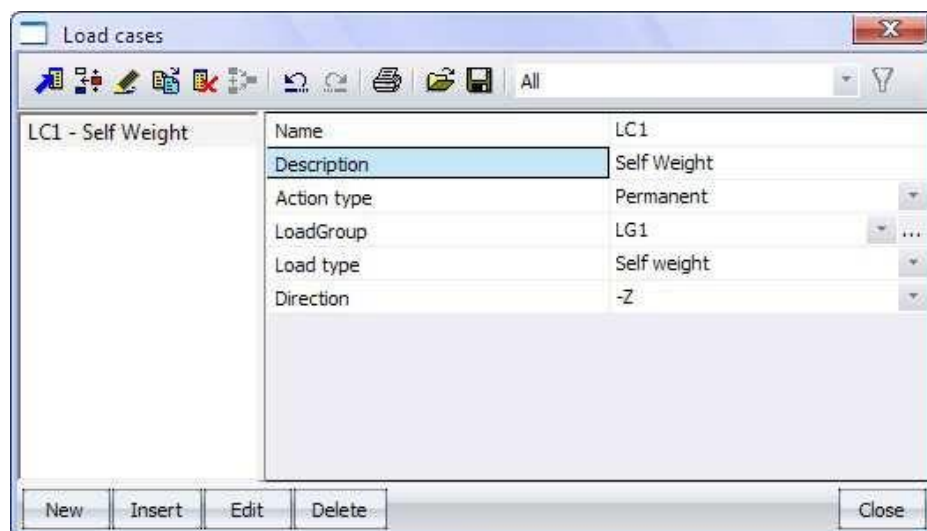
#### Permanentloads

- Self weight of the beam
- Weight of the slab, located on the beam
- Weight of the finishing layer

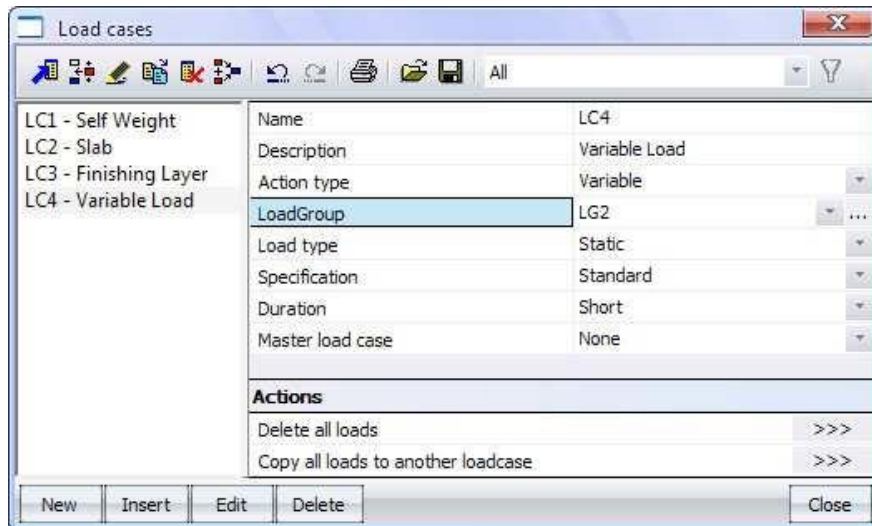
#### Imposedload

Category B (offices)

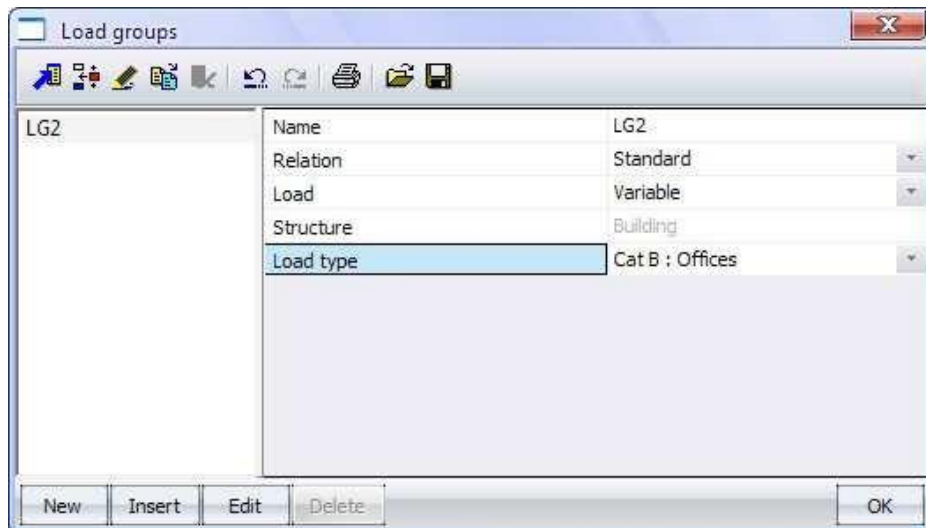
To apply the loads, double-click the option Load cases, Combinations in the Main menu. Because there are no loads yet defined, the dialog box Load cases will open automatically. Here, the user can define the characteristics of the loads. First enter the self weight of the beam as follows:



By clicking the New button, the remaining permanent loads can be entered as well. Be sure to select Standard as Load type, when creating the permanent loads, other than the self weight. Finally, to add the variable load, insert following parameters.



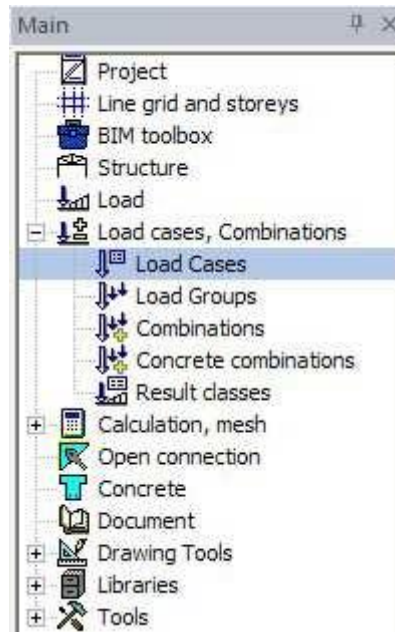
This example concerns an office (category B), therefore it is necessary to change the load type by clicking . Choose for the option Cat B: Offices.




Click the OK button to return to the dialog box Load cases. If the necessary loads are created, one closes the entry by using the Close button.

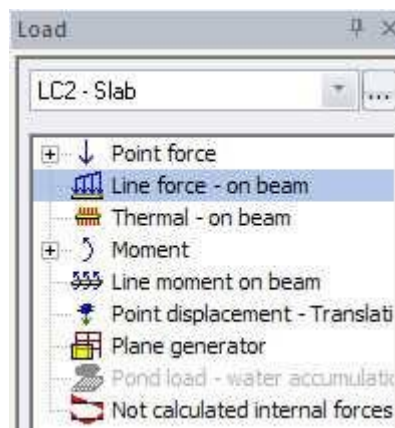
**Remark:** Load groups determine the way the individual loads will be combined with each other, when making load combinations.

Remark: Load cases can be adjusted through the option Load cases, Combinations < Load Cases.

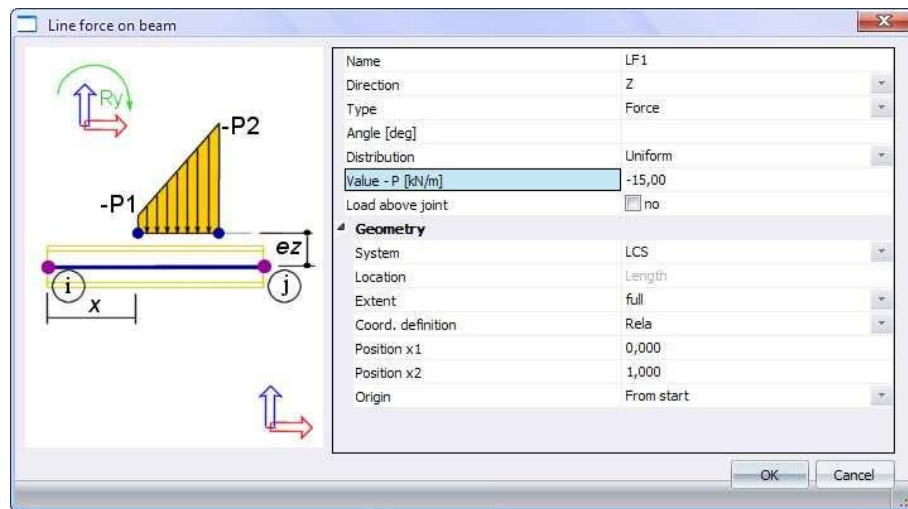


### Applying the loads

The submenu Load will open automatically. Within this menu, all the different loads can be applied. By clicking , the user can choose the proper load case. Scia Engineer will automatically calculate the self weight of the structure, therefore no further attention needs to be spend on this load case. The weight of the slab (LC2 – Slab) on the beam can be represented as a Line force of 15 kN/m. This can be done by double-clicking Line force – on beam.

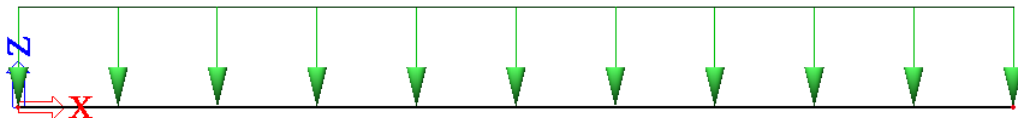


In the dialog box Line force on beam, the following parameters have to be entered:



Note that the value of the load is negative, this is because the load disposes of a negative sense in the Z-direction. The System is set to LCS (Local Coordinate System), this means that all parameters are related to the coordinate system of the structure. Choose Rela as Coord. definition, hereby the values of Position x1 and Position x2 will be applied, relative to the beam. The value 0,000 for Position x1 will therefore propose the starting point of the beam, the value 1,000 for Position x2 the endpoint.

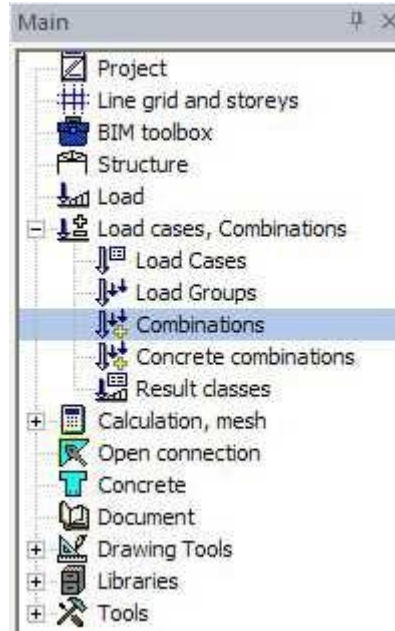
After clicking the OK button, one obtains a graphical representation of the applied load.



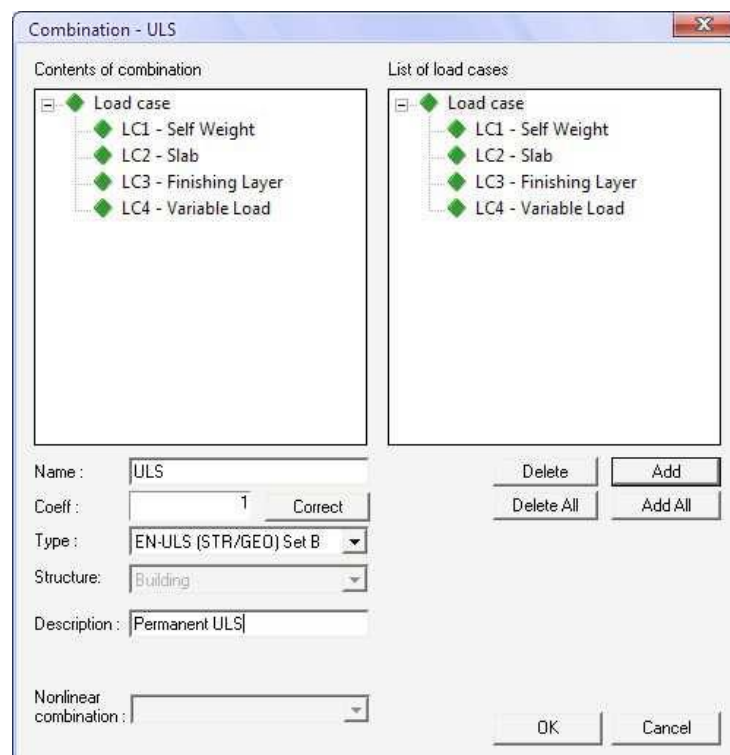
In this example, only Line forces occur. This means all loads can be applied as described above. LC3 – Finishing layer has a value of -6,9 kN/m and LC4 – Variable load a value of -9 kN/m.

## Creating the load combinations

The load combinations can be created with the option Load cases, Combinations > Combinations, these will be generated according to the national annex NBN-EN NA.

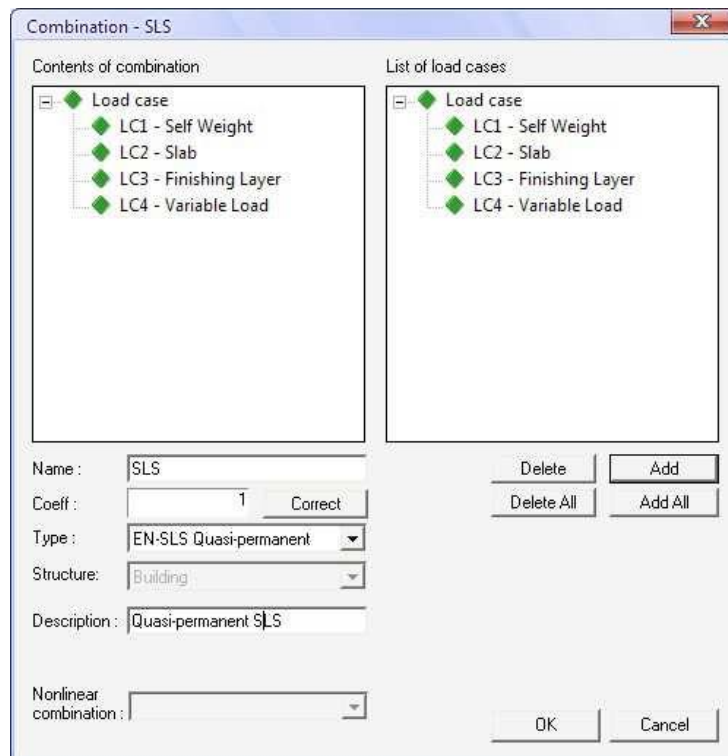


First, one has to create the combinations in the Ultimate Limit State (ULS) as follows:

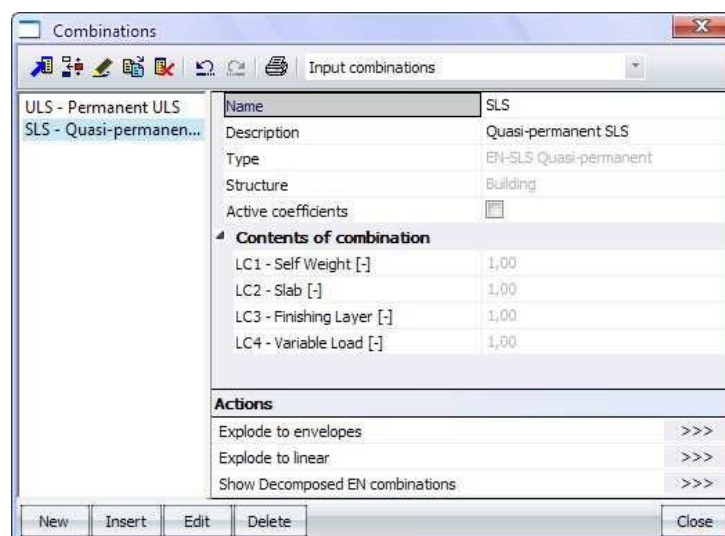


This ULS-combination contains all created loads, one can add this to Contents of combination through the Add All button. The user also has the possibility to add an additional Description. Choose the type EN-ULS (STR/GEO) Set B, as this is the permanent situation without any geotechnical effects. Confirm with the OK button.

Next, a combination for the Serviceability Limit State (SLS) has to be created. This can be done by clicking the New button en choosing the type EN-SLS Quasi-permanent.



The created combinations can always be consulted in the dialog box Combinations.



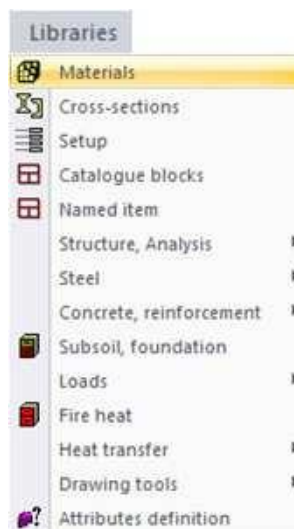
Click the Close button to close this window.



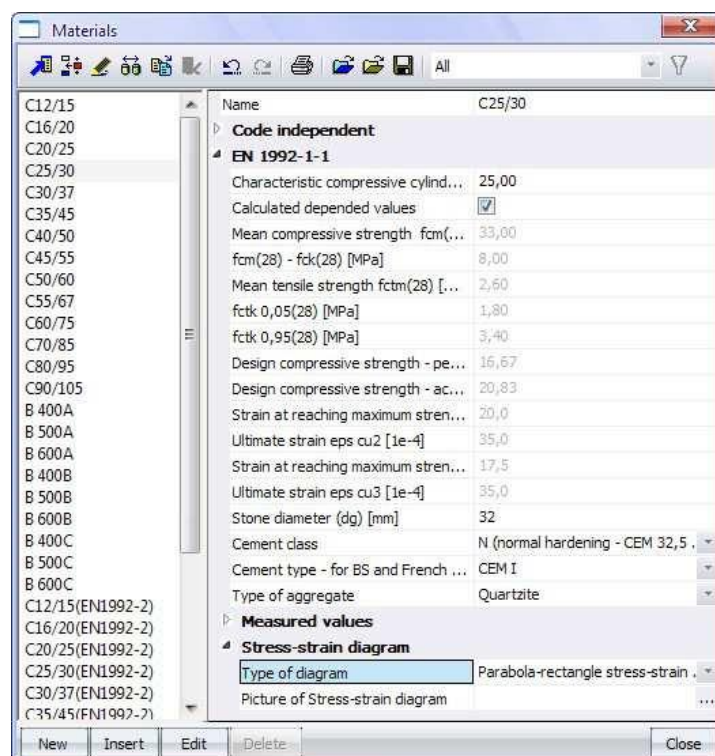
Remark: through the **>>>** behind Explode to linear in the dialog box Combinations, all possible linear combinations will be generated from the selected combination, in order to perform any checkups.

## Materials library

The characteristics of the used materials can be consulted and modified in the materials library. This option is located in the menu bar.




The dialog box Materials will open. All materials are listed on the left and the specific characteristic on the right.




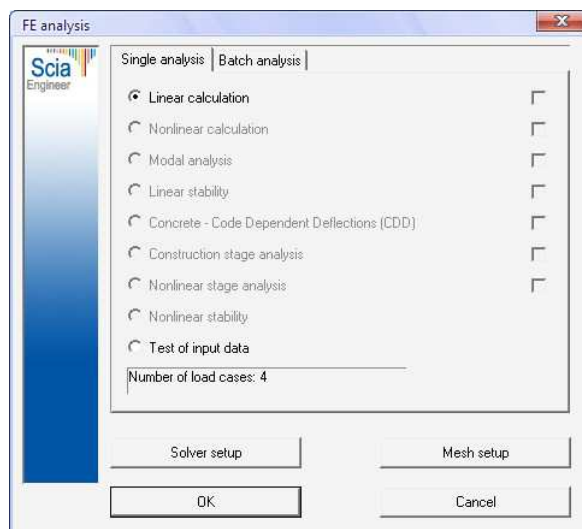


Any further calculations will be performed according to a parabola-rectangle stress-strain diagram as Type of diagram for the material C25/30.

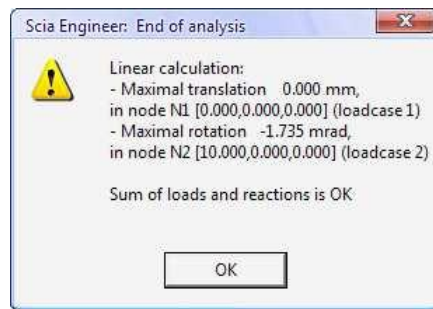
Choose also for a Bi-linear without an inclined top branch for steel B 500A. To obtain a graphical representation, click  after Picture of Stress-strain diagram.

## Linear calculation

After creating and applying the loads, the construction needs to be calculated. This is done through . A new dialog box, FE analysis, will appear. Perform a linear calculation by clicking the OK button.



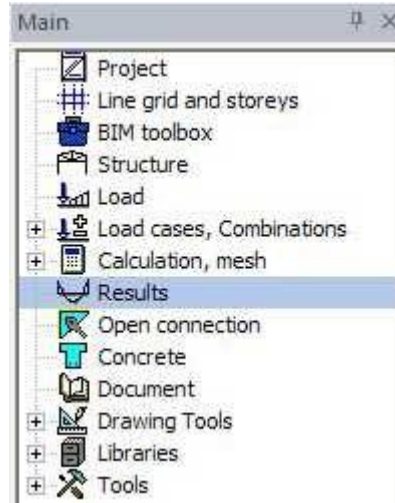
After performing this linear calculation, Scia Engineer reports the end of the calculation:



Click the OK button to close this window.

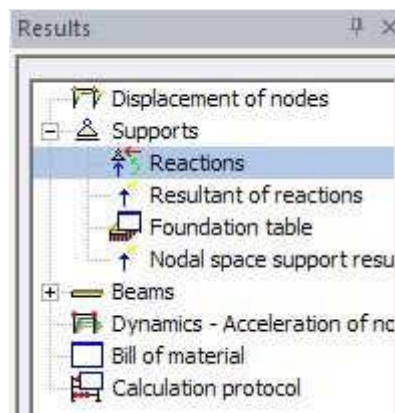
## Results

After completing the calculation, the user gains access to the Results menu.

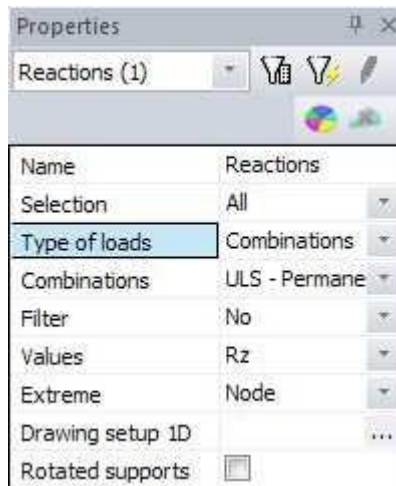


### Reaction forces

Through Results > Supports > Reactions, the reaction forces at the supports can be determined.



In the Properties window, the user can adjust any settings. To display the reaction forces in the Z-direction, choose as Values for  $R_z$ . Be sure Combinations are set to ULS-Permanent.



Next, to visualize the reaction forces, according to the chosen properties, click **>>>**, located behind Refresh.



The result will be shown as follows:

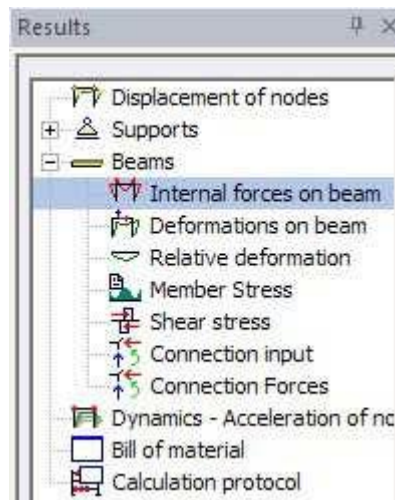


The results in SLS, following result will be obtained:

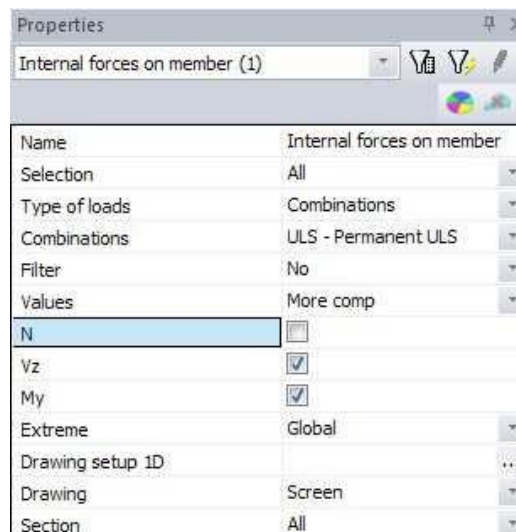


## Internal forces

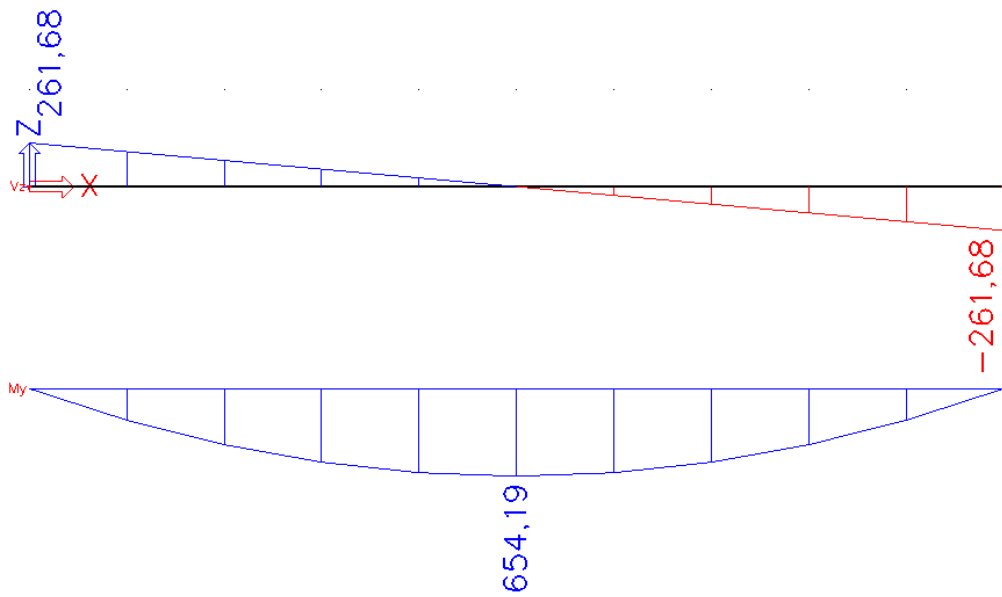
Through Results > Beams > Internal forces on beam, the internal forces can be requested.



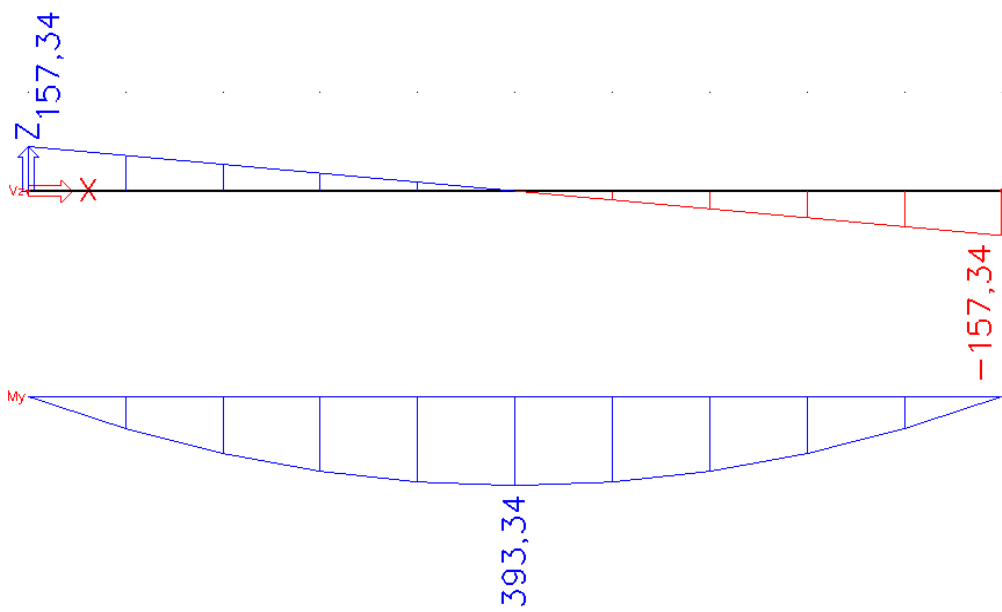
In the Properties, the user has to select the appropriate properties:



The results will be visible after clicking **>>>**, behind Refresh. Following internal forces will be given, in case of the ULS:

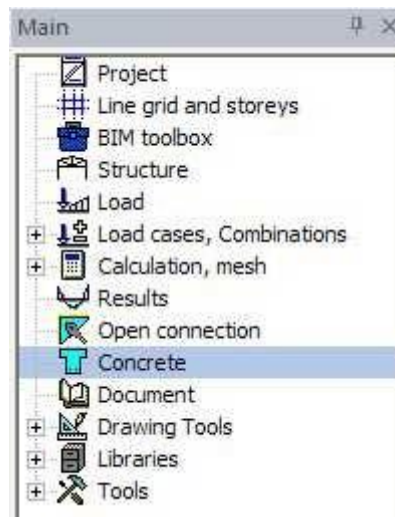


And for the SLS:

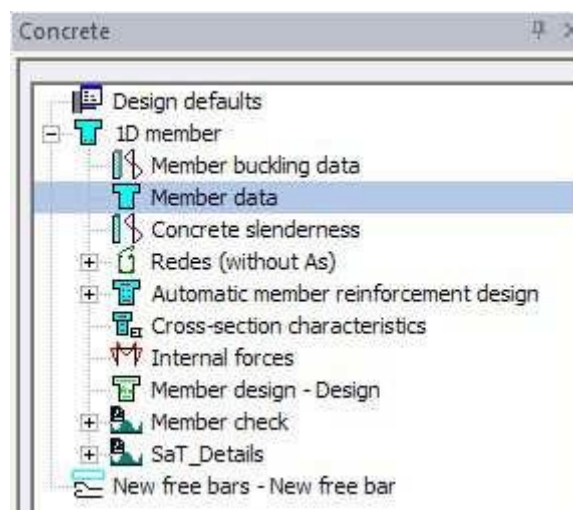


## Concrete – general

Through the Concrete menu in Main menu, the user has the ability to perform specific concrete related checkups.



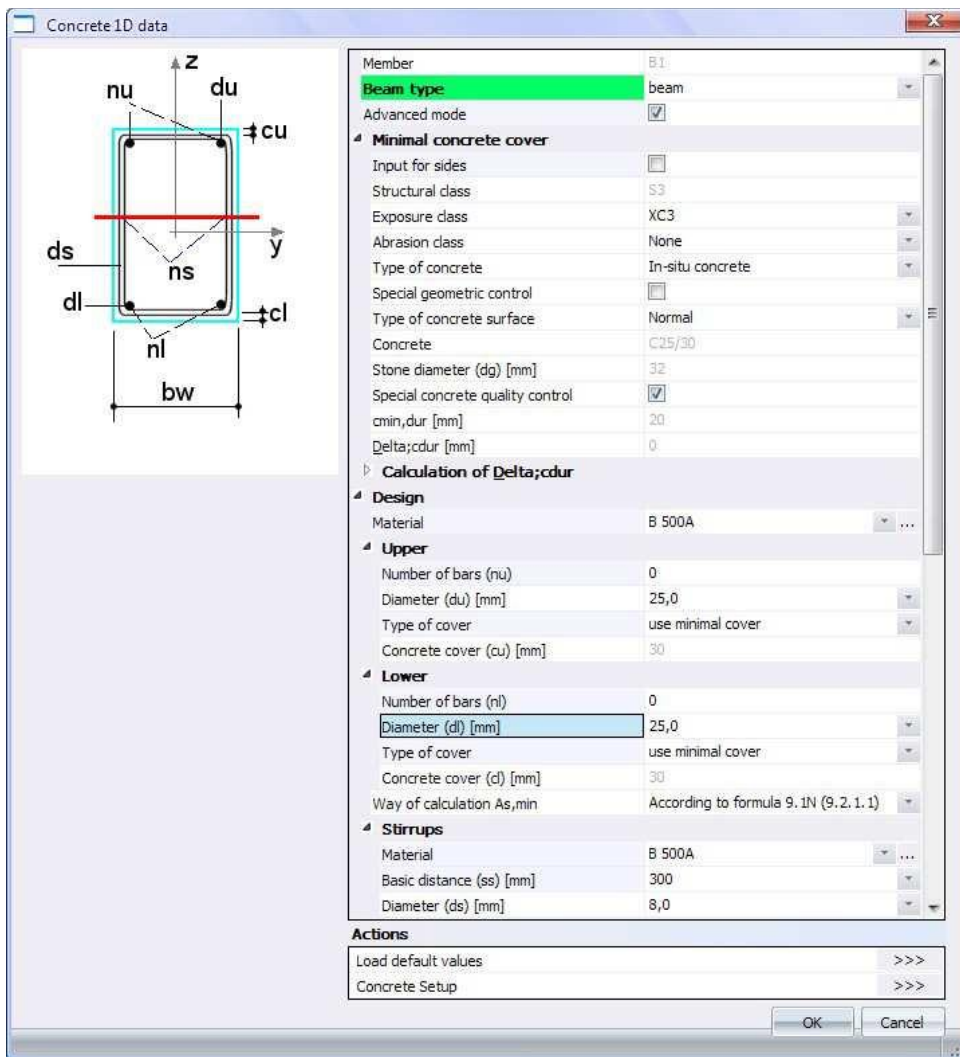
To adjust the properties of a concrete element, consult the option 1D member > Member data.



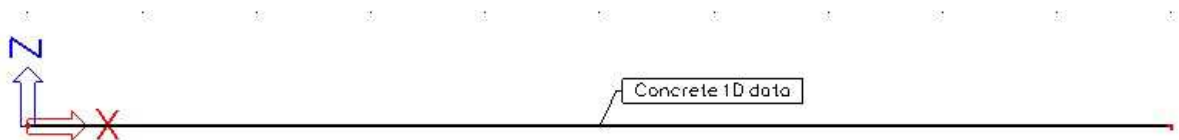
The concrete element, of which the properties need to be changed, has to be selected. This is indicated in the command line.



The dialog box Concrete 1D data will open. After ticking the Advanced mode, the user can perform several adjustments concerning the characteristics of the selected element. These adjustments need to be done as follows:



After clicking the OK button, a label will appear next to the concrete member. By selecting this label, the properties can be consulted or changed.

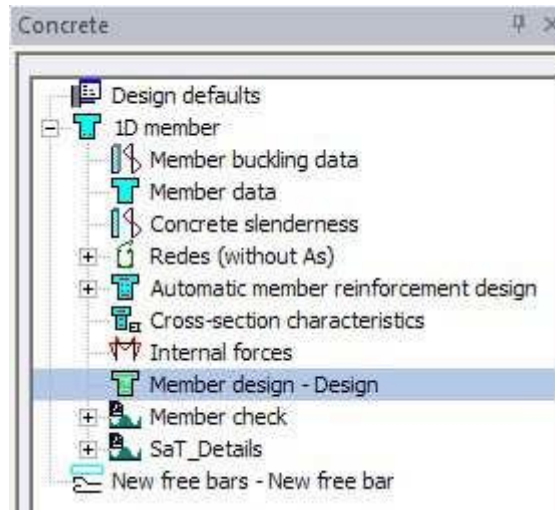


**Remark:** the user is also able to change the overall settings of all the different elements through the Design defaults. However these changes will not affect the elements that have been provided with member data.

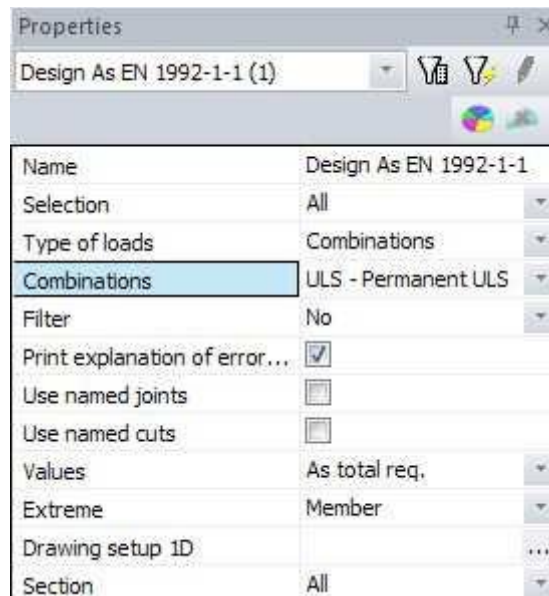


## Concrete – theoretical reinforcement

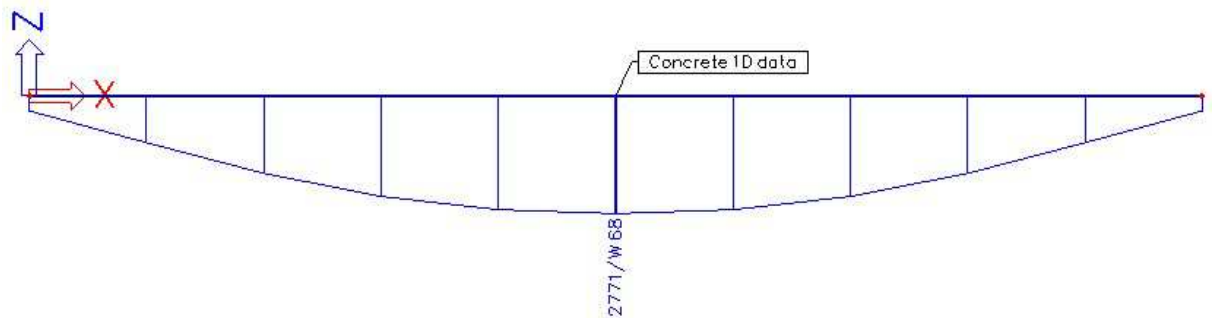
Through the option 1D member > Member design – Design, several aspects of the theoretical reinforcements can be calculated.



Choose the following options in the properties window:



By choosing  $A_s$  total req. and clicking **>>>**, Scia Engineer will calculate the reinforcement that is needed to resist the internal forces [mm<sup>2</sup>].



By clicking **>>>** behind Preview, more details will be shown in a table.

Preview

120 % default default

### Design As EN 1992-1-1

Linear calculation, Extreme : Member  
 Selection : All  
 Combinations : ULS

**Main lower reinforcement for selected beams**

Member	$d_x$ [m]	Case	$N_d$ [kN]	$M_{yd}$ [kNm]	$x_u$ [mm]	$d$ [mm]	$A_{s,reg}$ [mm <sup>2</sup> ]	Reinf.[no.]	W/E
B1	5,000	ULS/1	0,00	654,19	264	650	2771	6x25,0(2945)	68

**Explanation of warnings and errors**

68	The bar distance for the lower reinforcement is too small.
----	------------------------------------------------------------

Ready [en]

Remark: by ticking Print explanation of errors, any errors (E) or warning (W) will be explained.

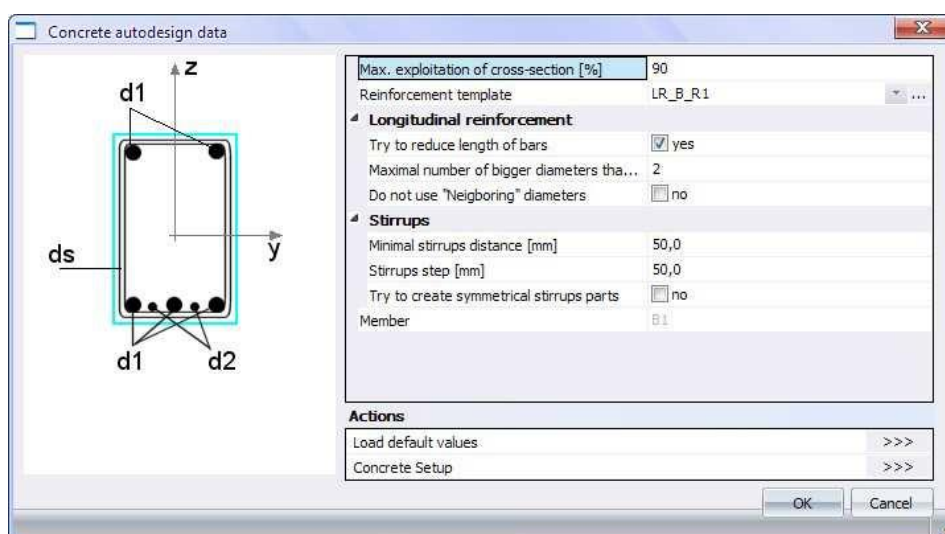
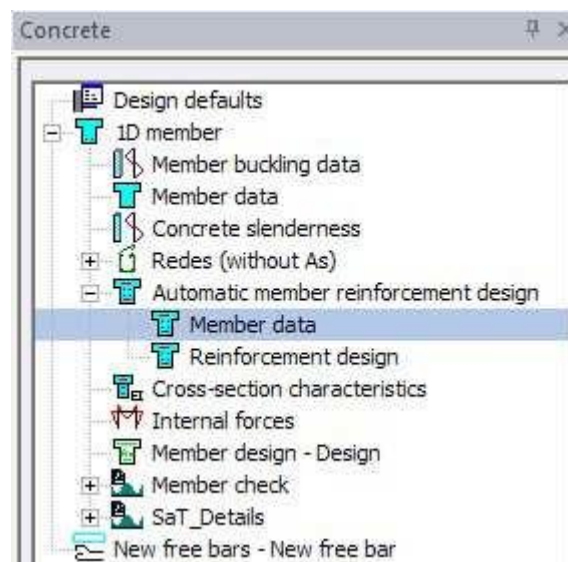
Remark: by selecting  $A_s$  user defined or  $A_s$  add. req., respectively the reinforcement added by the user and the extra reinforcement that is needed will be given.

## Concrete – AMRD

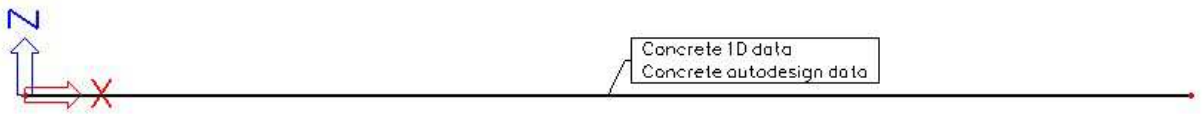
The user is also able to calculate and apply automatically the practical reinforcement.

### Member data

Within 1D member < Automatic member reinforcement design < Member data, the user is able to determine the Maximal numbers of bigger diameters than the default diameter. In this example, the value for this option is equal to 2. Because there is already chosen for a diameter of 25,0 mm, Scia Engineer will try to implement reinforcement bars with diameters varying from 25,0, 28,0 and 32,0 mm.

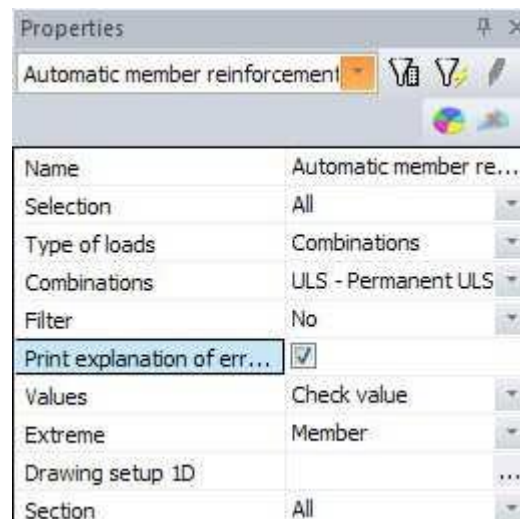
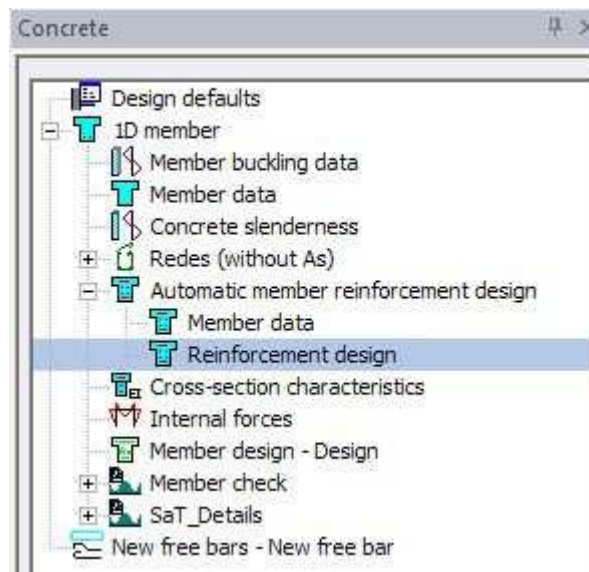


After confirming the properties, the selected element will obtain another label.

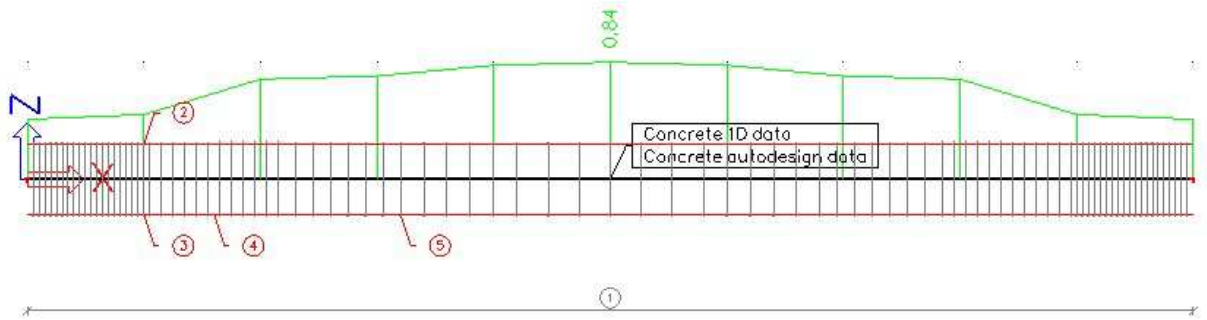


## Reinforcement design

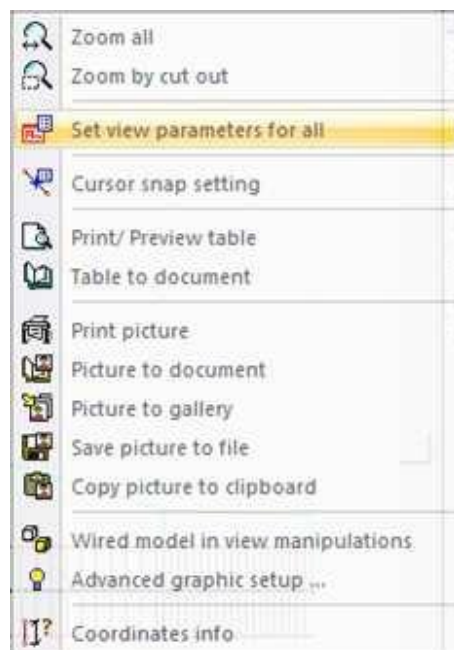
Through 1D member < Automatic member reinforcement design < Reinforcement design, one can calculate the actual reinforcement, after entering the correct parameters and confirming these by clicking the Refresh button.



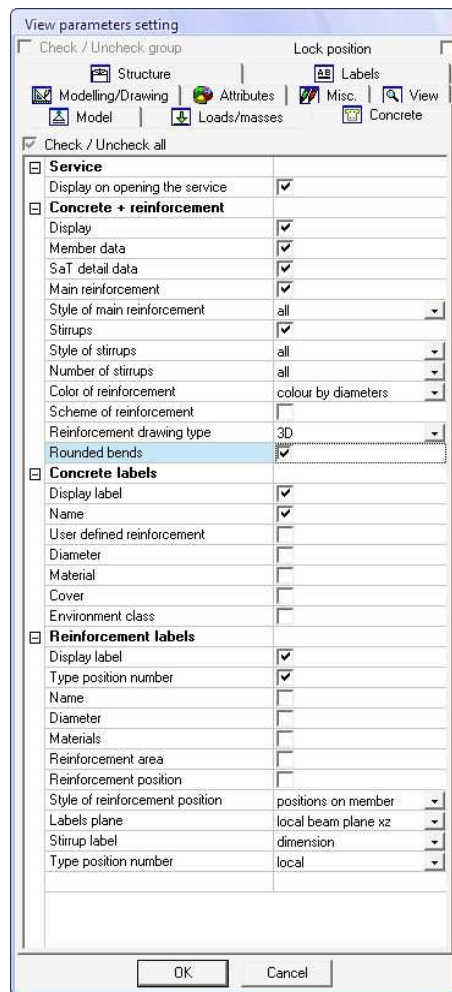
The following output will be displayed:



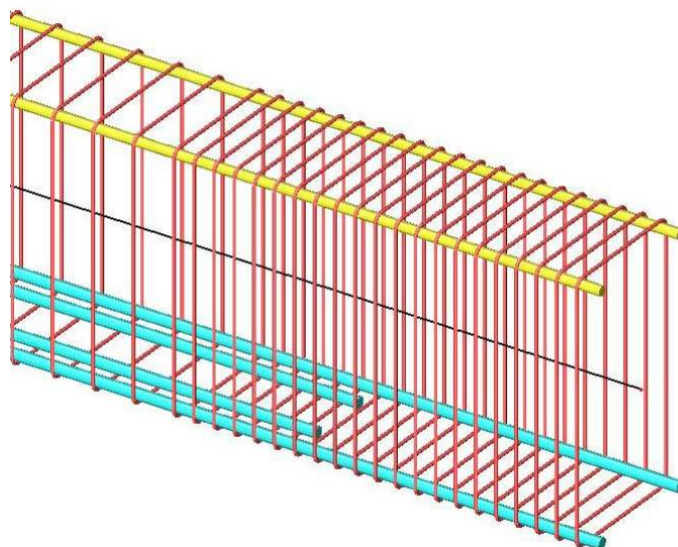
Remark: It is possible to display the reinforcement bars in a more realistic manner by changing the properties in the Set view parameters for all option.




More specifically by changing the settings in the Concrete – tab as follows:



Eventually, the reinforcement bars will be displayed like this:



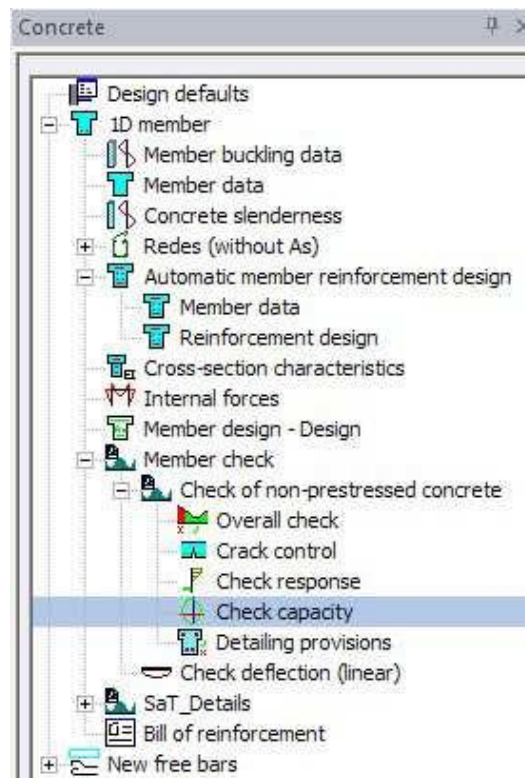
Remark: by using the following buttons , the user is able to change the view or to zoom in and out.



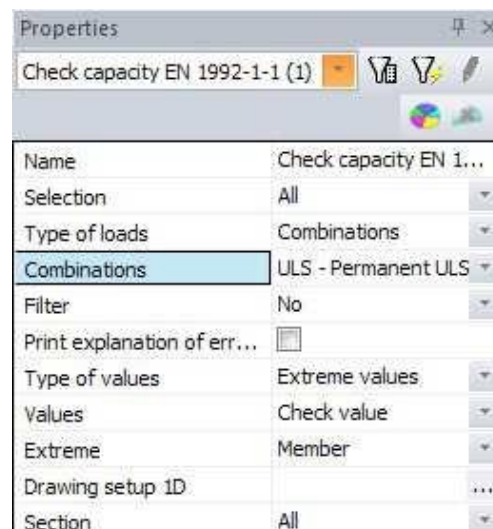
## Concrete – checks

### Capacity check (ULS)

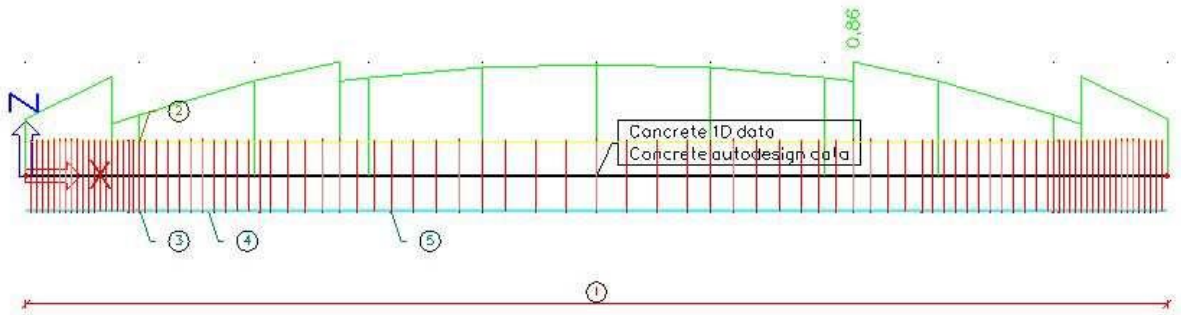
The capacity check is performed in order to check whether the interaction between the normal force  $N$  and the bending moments  $M_y$  and  $M_z$  is located within the fixed boundaries. This check determines the efficiency of the added reinforcement and is executed through the option 1D member < Member check < Check of non-prestressed concrete < Check capacity.



Choose for the properties as shown below:

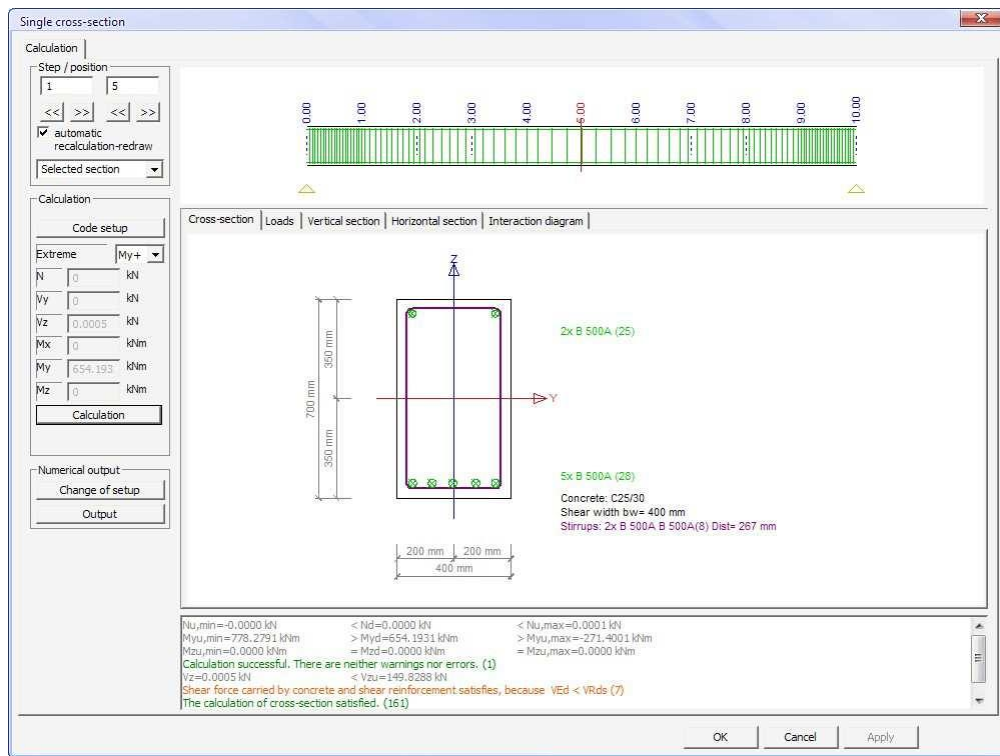


After refreshing, the output will be as follows:



**Remark:** the output will be the maximum value, which has to be smaller than 1 in order to be correct. A correct output will be displayed in green.

The results of each cross section can be observed by clicking **>>>**, behind Single Check. The command line will ask the user to select the member of which the cross section will be analyzed.

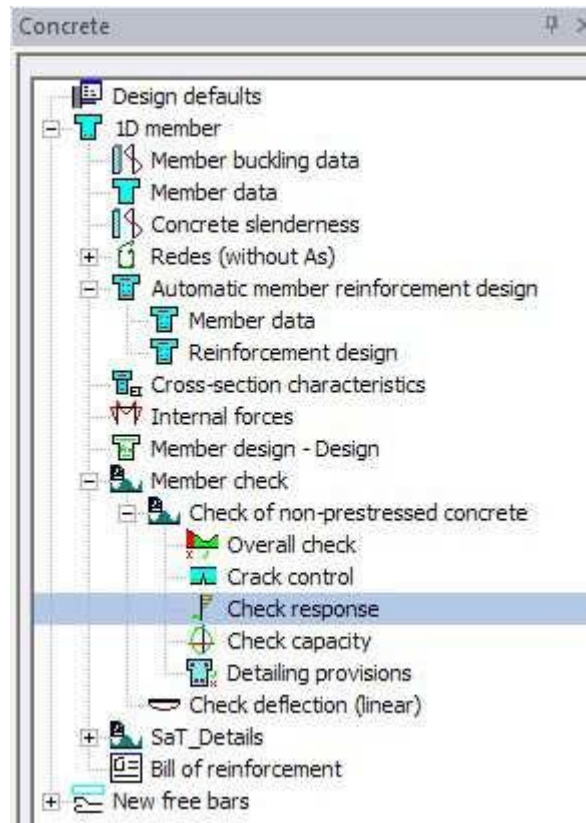


**Remark:** it is necessary to click Calculation, in order to obtain the output as shown above. The user is also able to choose the exact position of the cross section, as well as to scroll through the results by using the tabs.

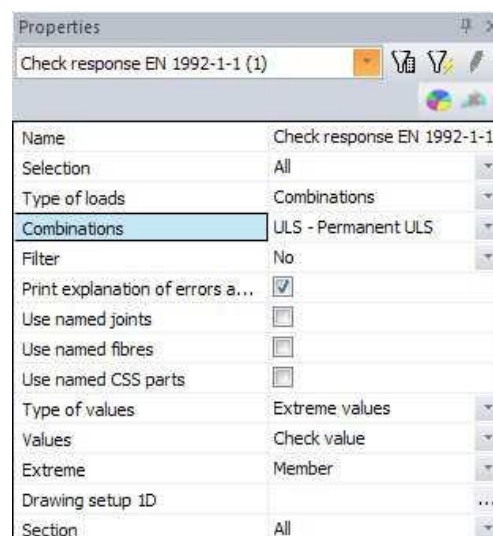


## Response check (ULS)

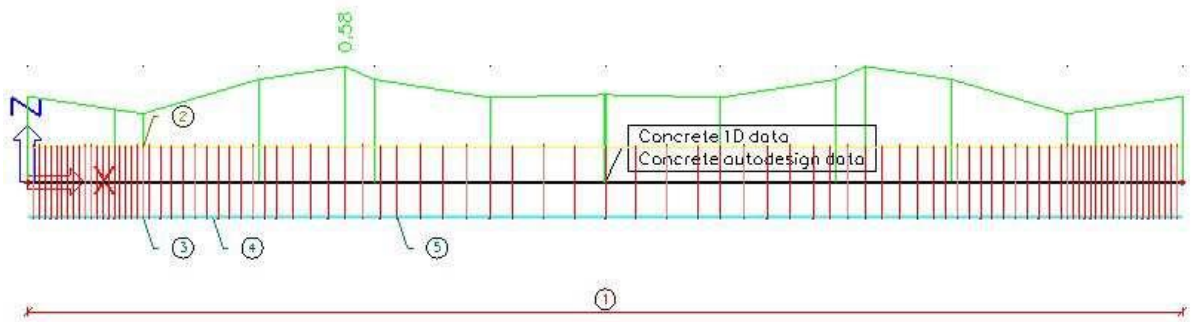
This check is executed in order to make sure the stresses, in both concrete and reinforcement steel, do not exceed their limit values. The check can be performed through 1D member < Member check < Check of non-prestressed concrete < Check response.



Select the following settings in the properties window:



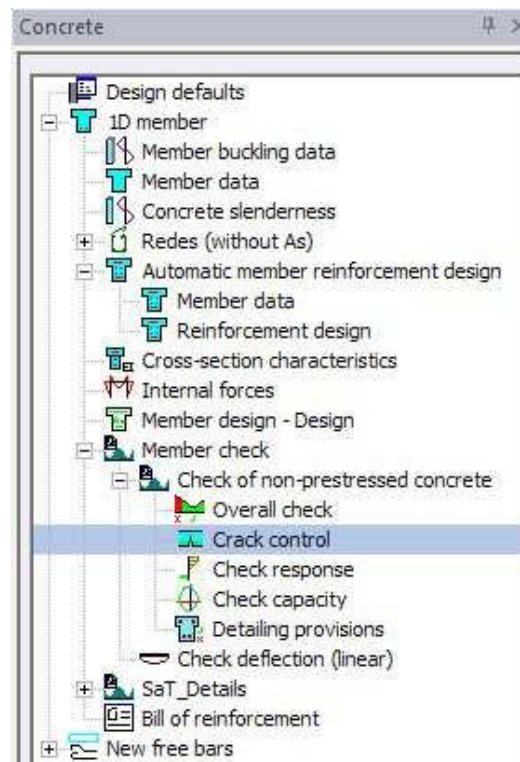
After clicking the Refresh button, the following output will be obtained:



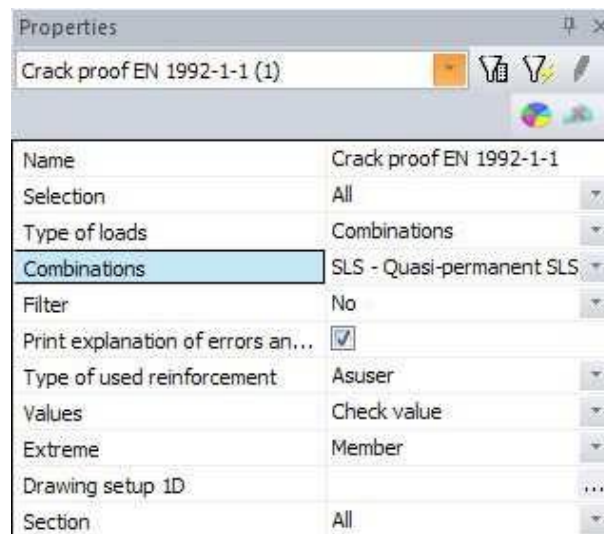
The user is also able to request a Preview and to perform a Single Check.

### Crack control (SLS)

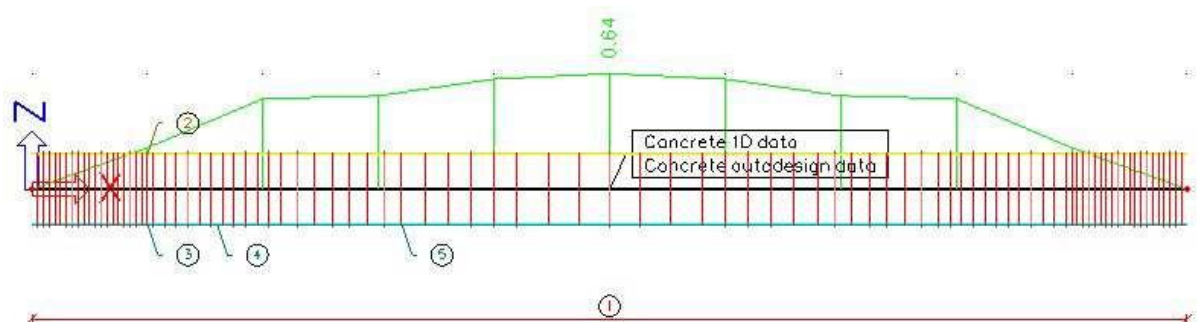
Scia Engineer also offers the opportunity to check the construction for possible cracks. This is done with the option 1D member < Member check < Check of non-prestressed concrete < Crack control.



The properties should be changed as shown below:



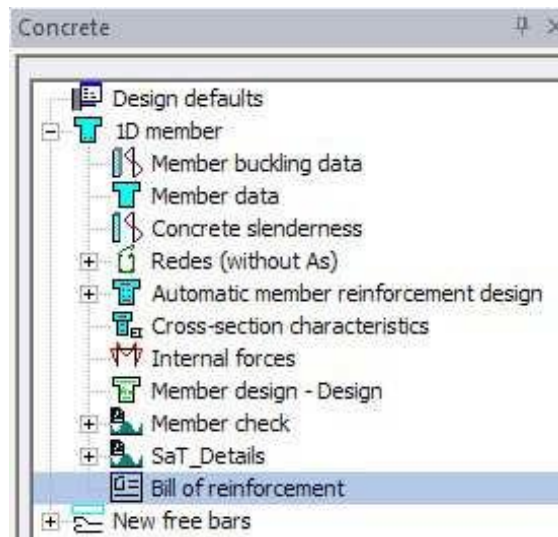
Make sure to choose the option Asuser, this value is equal to the previously added reinforcement during the AMRD. After clicking the Refresh button, the output will be as follows:



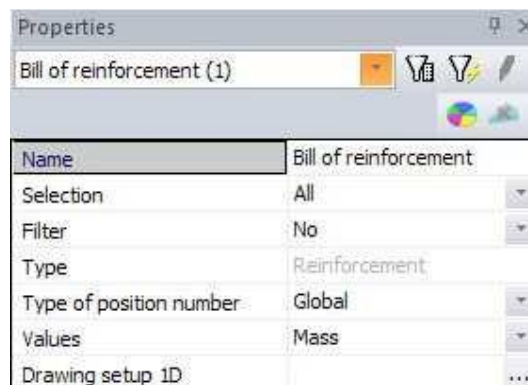
The user is also able to request a Preview and to perform a Single Check.

## Concrete – Bill of reinforcement

The length and mass of the added reinforcement can be requested through the option 1D member < SaT\_Details < Bill of reinforcement.



After refreshing the following properties, the user will be able to view the output:



Preview

150 %

default

### Bill of reinforcement

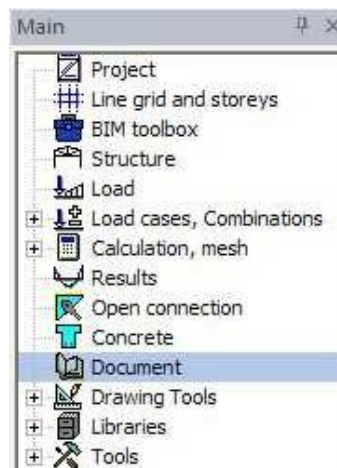
Selection : All  
The length of reinforcement and stirrups are calculated without rounded bends.  
Type of position number : Global

Member	Position number	Diameter [mm]	Material	Length [m]	Number of bars	B 500A length [m]	B 500A weight [kg]
B1	1	8	B 500A	2,800	93	260,400	102,7
B1	2	25	B 500A	10,000	2	20,000	77,1
B1	3	28	B 500A	10,000	2	20,000	96,7
B1	4	28	B 500A	8,500	2	17,000	82,2
B1	5	28	B 500A	4,500	1	4,500	21,8
			8			260,400	102,7
			25			20,000	77,1
			28			41,500	200,6
			Total for material			321,900	380,4
			Total			321,900	380,4

Ready [en]

## Document

All required results can be put together in one document. This function can be activated in the Main menu.



By clicking  , the required results can be selected. These will be included in the Document. In the dialog box New document item, all items are summed up. These can be added by the user by clicking  . By adding for example Results < Internal forces on member, the following output will be obtained:

## 1. Internal forces on member

Linear calculation, Extreme : Global, System : LCS

Selection : All

Load cases : LC1

Member	Case	dx [m]	N [kN]	Vz [kN]	My [kNm]
B1	LC1	0,000	0,00	34,34	0,00
B1	LC1	10,000	0,00	-34,34	0,00
B1	LC1	5,000	0,00	0,00	85,84

Remark: in order to gain more insight, it is useful to add the Combination key.

